Introduction to ENCY

Introduction to ENCY

Welcome to ENCY, a powerful and intuitive CAD/CAM software designed to optimize your design and manufacturing processes. Whether you're a beginner or a seasoned professional, ENCY offers a full suite of tools that cater to a wide range of industries, from automotive to aerospace, and from consumer goods to industrial machinery.

What is CAD/CAM?

CAD (Computer-Aided Design) and CAM (Computer-Aided Manufacturing) are essential technologies in modern engineering and manufacturing. CAD software lets you create precise 2D and 3D models of products, while CAM software automates the manufacturing process, translating these designs into instructions for CNC (Computer Numerical Control) machines.

With ENCY, you can:

- Design with Precision: Create complex geometries and detailed designs using a comprehensive range of drawing and modeling tools.
- Simulate and Analyze: Run simulations and analyzes to ensure your designs meet the required specifications and performance criteria.
- Automate Manufacturing: Generate accurate toolpaths and G-code for CNC machines, optimizing production efficiency and minimizing errors.

Applications

ENCY supports virtually the entire spectrum of manufacturing technologies, offering versatility and adaptability in your production processes. From 2-axis turning and 4-axis wire EDM to multi-axis milling, mill-turn operations, and Swiss-type turning centers, ENCY equips you with the tools needed to tackle even the most complex tasks with ease.

Additionally, ENCY is capable of creating projects for additive and hybrid manufacturing, enabling you to explore and implement advanced production techniques seamlessly.

Interface

ENCY features a user-friendly, minimalist interface specifically designed to speed up your workflow. The streamlined design ensures that tools and functions are easy to access, reducing the time spent navigating menus and boosting overall productivity.

ENCY includes a dark mode for a more comfortable work environment, helping to reduce eye strain during extended use.

Ecosystem

ENCY Software offers a complete ecosystem for digital manufacturing, bringing together a range of specialized software products that cover every aspect of modern production. From the powerful CAD/CAM capabilities of ENCY to the advanced offline robot programming (OLP) solutions in ENCY Robot, our ecosystem delivers an integrated and seamless experience. This includes tools for G-code simulation and NC program editing with ENCY Tuner, real-time (hybrid) robot programming with ENCY Hyper, AI-driven workflow optimization with ENCY AI, and cloud-based collaboration and storage with ENCY Clouds.

Each product in the ENCY ecosystem is designed to work together harmoniously, giving you everything you need to bring your manufacturing projects to life with maximum efficiency and precision.

Getting Started

This help section is designed to guide you through the various features of ENCY, providing the knowledge and skills you need to fully leverage the software's capabilities.

Let's start your journey with ENCY. Go beyond CAD/CAM.

System requirements

The minimum and recommended PC configurations for ENCY:

Requirements	Minimum (small and medium projects)	Recommended (large projects)
OS *, **	Windows 10 64-bit	Windows 11
CPU	Intel® or AMD® 64-bit processor, 2.4 GHz or faster	Intel® i7/i9 Rocket Lake or later or AMD® Ryzen 7®/Ryzen 9® Zen 3 or later
Memory	8 GB RAM	32 GB RAM
Video	OpenGL-capable graphics card (OpenGL 1.5) with 1 GB memory	Nvidia GeForce® with 4 GB dedicated memory or higher
Storage	Solid State Drive (SSD) with at least 5 GB free space	NVMe drive with at least 5 GB free space
Monitor	1280×960 or higher	1920×1080 or 4K, dual monitors
3D mouse	—	3Dconnexion 3D mouse-compliant
Internet	—	100 Mbit internet connection

Note: The recommended configuration of the computer much depends on the complexity of the models to be machined and machining quality. The more complex the model is, and the higher the machining quality required, the greater number of calculations will be needed to perform, in order to generate the tool movement toolpath. Therefore, the higher the specification of the computer, the faster the calculations will be performed.

*ENCY is not supported on Apple Macintosh® – based machines. Some customers have shown success in running ENCY in a Virtual Windows environment on Mac computers using Boot Camp or any Windows emulator (Wine, Parallels Desktop, VirtualBox, VMware Fusion, CrossOver, etc.). While the end user may choose to run Windows on a MAC®, this is not supported by ENCY.

** Antivirus Software: Most computers today have some kind of antivirus software to protect from unwanted malware. In some cases, these have been found to interfere with applications such as ENCY which are running on the PC. ENCY software does not recommend particular products, but if you see

unexpected issues, it may be a clash with antivirus software. Our software are checked by Kaspersky Antivirus service, Avast company, and we are trusted for our software. Try temporarily disabling the antivirus software or setting an exception for ENCY.

Configurations and options

ENCY configurations

Premium Configurations

Premium configurations include: 2D sketching, 3D modeling, dimensioning and geometry transformation; import of CAD formats such as IGES, STL, VRML, PostScript, DXF, Rhinoceros, Sprut Models, Parasolid, STEP, Sprut geometry format, SW, SE, PLY, AMF, JTOpen, PLMXML, 5DC, and OSD; add-ins for integration with popular CAD systems; high-end 3D machining simulation with G-code verification support; setup sheet generator; support for probing operations; tool library; equipment library; MachineMaker for importing, assembling, and configuring digital twins of machines and robots; postprocessor generator; a set of basic postprocessors and interpreters.

Bound Configurations

Bound license is tied to a specific machine or robotic cell and includes all the features of the Premium configurations, except for: MachineMaker, the postprocessor generator and the set of basic postprocessors and interpreters.

Applications

Cutting

Cutting 2D: 2D cutting (Hole machining, FBM Mill, 2D contouring, 2.5D contouring, Jet cutting). Nesting included.

Cutting 5D: Multi-axis cutting (3D contouring, 6D contouring, Chamfering, Jet cutting 5D), includes all features of Cutting 2D; 3+2 indexed multi-axis machining with local coordinate systems, TCPM supported. Nesting included.

Cutting 5D Advanced: Multi-axis cutting, includes all features of Cutting 5D plus the extended 6D contouring operation: Feature Recognition (Circle, Rectangle, Rounded Rectangle, Slot, Polygon, Keyhole), NC cycle output and Support Generator. Nesting included.

Milling

2.5X Mill: 2.5-axis milling (Hole machining, FBM Mill, 2D contouring, Engraving, Pocketing, 2.5D contouring, Face milling, Roughing waterline limited to machining pockets with vertical walls only, Flat land, Chamfering, Jet cutting). 3+2 indexed multi-axis machining with local coordinate systems. Nesting included.

3X Mill: 3-axis milling (Roughing waterline, Finishing waterline, Roughing plane, Finishing plane, 3D contouring, Combo Finish offering Plane and Waterline strategies and their combination, without Scallop

machining), includes all features of 2.5X Mill. 3+2 indexed multi-axis machining with local coordinate systems. Nesting included.

3X Mill Advanced: 3-axis advanced milling (Scallop, Morph, 3D Helical, 5D surfacing (limited to generate only 3-axis toolpath), Undercut waterline, Combo Finish, Optimized plane, Complex), includes all features of 2.5X Mill, 3X Mill. 3+2 indexed multi-axis machining with local coordinate systems. Combo Finish is a 3D finishing operation offering Plane, Waterline, and Scallop machining strategies, as well as their combinations, for efficient machining of complex surfaces. Nesting included.

Rotary: 4-axis continuous milling (Rotary waterline, Rotary roughing, Rotary finishing, 4D surfacing, Morph 4D, 4D contouring), includes all features of 2.5X Mill, 3X Mill, and 3X Mill Advanced. 3+2 indexed multi-axis machining with local coordinate systems. Nesting included.

5X Mill: 5-axis continuous milling (Surfacing, 6D contouring, 5D by meshes, Morph 5D, Jet cutting 5D), includes all features of 2.5X Mill, 3X Mill, 3X Mill Advanced, and Rotary. 3+2 indexed multi-axis machining with local coordinate systems, TCPM supported. Adaptive Tilting and Nesting included. Adaptive Tilting automatically avoids holder collisions by tilting the tool along two axes. It is available in 3-axis operations; converting the toolpath into a 5-axis path requires a 5X Mill license.

5X Mill Advanced: 5-axis advanced continuous milling (Swarf, 5D Roughing waterline, Turbomachinery PRO), includes all features of 2.5X Mill, 3X Mill, 3X Mill Advanced, Rotary, and 5X Mill. 3+2 indexed multi-axis machining with local coordinate systems, TCPM supported. Turbomachinery PRO (Impeller Machining and Blade Machining operations), Adaptive Tilting and Nesting included.

Turning

Lathe: 2-axis turning (Lathe facing, OD roughing, ID roughing, OD finishing, ID finishing, OD grooving, ID grooving, Face grooving, OD adaptive turning, OD threading, ID threading, Profile threading, Lathe part-off, Part-off with takeover, Lathe hole machining), bar feeding and part transfer operations (Bar feeding, Sub spindle working, Turn take over).

WireEDM

Wire EDM: 2-axis, tilted, and 4-axis wire EDM

Feature extensions and Options

Configurations and machine match table:

Option	Description	Availability
Turn XZCY	Mill-turn machining with C and Y axes	Any milling configuration

Option	Description	Availability
Turn XZCYB	Mill-turn machining with C, Y, and B axes (4-axis Turning)	Any milling configuration
Wire EDM	2-axis, tilted, and 4-axis wire EDM	Any milling configuration
EquiMill	Proprietary high-speed roughing strategy, formerly named Adaptive. Full reference	Any milling configuration except 2.5X Mill
SOLIDWORKS Reader	Separate paid module for opening SOLIDWORKS files and supporting future SOLIDWORKS versions. Free of charge until the end of 2026; continued support for new SOLIDWORKS releases requires an active Software Maintenance Contract (SMC).	Any configuration
Robot+	Industrial robot programming. See ENCY Robot specification for details Full reference	5X Mill and 5X Mill Advanced milling, Cutting and Cutting 5D configurations
Multichannel	Support for multiple program channels with synchronization for machines and robots	Available for mill-turn or milling configurations with Robot+
Nesting	Automatic layout of parts on sheet or stock material to maximize utilization and minimize scrap. Arranges multiple contours across the available area with configurable spacing and rotation for laser, plasma, waterjet and oxy-fuel machines. In 3.0 included in all milling and cutting configurations at no extra	Any milling and cutting configuration (not available for Lathe)

Option	Description	Availability
	charge; remains orderable as a separate option.	
G-code based operations	Insertion and use of G-code in the machining project Full reference	Any configuration except Wire EDM
Additive Planar 3D	3D additive manufacturing (Area cladding, Curve cladding, Cladding 3D, Planar slicing) Full Reference	Any milling configuration
Additive Non-planar 5D	5D additive manufacturing (Cladding 5D, Non-planar slicing). *Note: Additive Planar 3D operations are NOT included.* Full Reference	Any multi-axis (Rotary and above) milling configuration
Disc cutting 2D	2D cutting with a disc tool	Any milling configuration
Disc cutting 6D	Multi-axis cutting with a disc tool	5X Mill and 5X Mill Advanced milling configurations
Disc Roughing	Rough machining with a disc tool	Any multi-axis (Rotary and above) milling configuration
Knife cutting 2D	2D cutting with a knife Full reference	Any milling and cutting configurations
Knife cutting 6D	Multi-axis cutting with a knife Full reference	Any milling and cutting configurations
Welding	Automated welding (Point welding, Welding 6D) Full Reference	Any configuration with Robot+
Spraying/Painting	Automated spraying/painting (Contour spraying, Surface spraying, Morph spraying, Rotary spraying) Full reference	5X Mill and 5X Mill Advanced milling configurations

Option	Description	Availability
Robotic turning	Turning operations on an industrial robot	Any robot application
Turbomachinery PRO	Advanced 5-axis turbomachinery machining package featuring dedicated Impeller Machining and Blade Machining operations	5X Mill milling configuration (included in 5X Mill Advanced)
Laser Cutting PRO	Extended 6D contouring operation: Feature Recognition (Circle, Rectangle, Rounded Rectangle, Slot, Polygon, Keyhole), NC cycle output and Support Generator.	Cutting 5D, 5X Mill and 5X Mill Advanced configurations (included in Cutting 5D Advanced)

ENCY eXpand – An ecosystem of integrated ENCY CAD/CAM software products and online services

ENCY Clouds

Digital workspace: collaboration and remote support tools; built-in chat interface for messaging, file sharing, and machining project synchronization; mobile application. Online library containing ready-to-use machining projects from the vendor, dealers, and users; private cloud storage for the user's machining projects.

ENCY Clouds available for all users with active maintenance.

Robot Components library

Online library containing ready-to-use machining projects from the vendor, dealers, and users, with toolpath preview; private cloud storage for the user's machining projects.

A free resource available to all users.

AI SuperAgent

AI assistance for CAM workflows. Delivered via Extension Store

Extensions Library.

DMC - Digital Machine Center

Digital machine schemas and postprocessor access.

Selected packages included in maintenance.

Support Generator

Automatic generation and nesting of supports, integrated for CNC and robot cutting. Delivered through the Extensions Library / Delivered via Extension Store

License settings

Type	Description	Availability
Netlock	Local license server. Supports floating licenses. Useful for customers with restricted internet access	Not available with Online protection type
Maintenance	Software Maintenance Contract includes software updates, technical support from the dealer, and access to ENCY X services: ENCY Clouds for messaging, file sharing, and cloud-based project synchronization; Online library containing private cloud storage for the user's machining projects.	1,2,3,4,5 years. Allows setting a unified maintenance end date across all licenses, ensuring consistent support periods and simplifying contract management.
Block geometric models export	Disables all functions that allow exporting any geometric models to external files. Useful for customers with strict security restrictions.	Free for all license types
Block MachineMaker	Disables the ability to launch MachineMaker	Free for all license types
Block NC output	Disables the ability to output NC programs. Used for educational institutions to prevent the possibility of using educational licenses for commercial purposes.	Free for all license types

Type	Description	Availability
Block internet activity	Disables all online features, no data exchange with any servers. Important: Excluding work with network licenses and third-party modules.	Free for all license types
Enable Virtual PC	Enables the ability to run ENCY software on a virtual machine, for example when using Parallels on an Apple MacBook	Free for all license types

License Types

Type	Description
Trial	Trial license, issued for a period from 2 weeks to 3 months. Not for commercial use.
Commercial	Commercial license
Educational	Educational license, for educational institutions and students. Not for commercial use. Issued for a period from 2 weeks to 1 year.
Educational PRO	Paid educational license, for educational institutions and students. Not for commercial use.
NFR	Temporary license for dealers and partners to demo and promote ENCY software. Issued for 6–12 months, it's used for non-commercial evaluation, pilot projects, and pre-sales support.

License protection types

Type	Description
Dongle	A hardware security dongle is required. Supported dongle types include HASP, Matrix, and Guardant.
Online	Access is provided through a username and password. An internet connection is required.

Type	Description
Software	Local License File; an internet connection is not required to run the software; the license file is non-transferable. Delivery and Activation Process: - If the customer has an internet connection, the license file is downloaded automatically after the customer is authorized in the License Manager. - If the customer does not have an internet connection, the license file can be delivered via email or external storage. Offline activation is supported.

License durations

Perpetual: Perpetual license (license without time limitations)

Subscription: Software subscription for a period from 3 months to 1 year. After three years of active subscription, the user is entitled to convert the subscription into a perpetual license. The cost of maintenance is not included and must be paid separately.

Standard package

The boxed package of ENCY ® includes:

1. Physical media containing the ENCY ® system.
2. Documentation (PDF file).
3. Electronic key to prevent unauthorized copying (optional – depends on configuration and protection method).
4. License agreement.

The e-package of ENCY ® includes:

- Executable file for installing the ENCY ® system.

Program installation and launch

ENCY uses the Internet for installation. The setup program scans the ENCY web server for available modules and downloads them. You can install only required modules, skipping others. Subsequent runs use previously downloaded modules to accelerate the process. There is a "local" installer based on the same technology, which uses local modules for installation.

There is also a "local" installer based on the same technology, which uses local modules for installation.

ENCY installation on a computer with Internet connection:

1. Download an installable file from the Website.
2. Run the installation program on Windows. In the Run dialogue, type the Path and click OK or hit Enter.
3. The Installation program will guide you through various dialogue boxes, which will require some input during the installation process, including:
4. End user license confirmation.
5. Choose default, minimal, or custom installation (can exclude example project files, importable files, and other auxiliary files).
6. Specify the installation directory.
7. Specify the main menu directory for shortcuts.

To proceed to the next step, click the Next button.

After installation, a confirmation window will appear. Close it by clicking the Done button. If you have a CAM system with electronic key protection, insert the key into a USB port before running the system.

ENCY installation from a physical media:

1. Insert the physical media.
2. Run the installation program from it. In the Run dialogue, type the `Drive_letter` and click OK or hit Enter.
3. The Installation program will guide you through various dialogue boxes, which will require some input during the installation process, including:
4. End user license confirmation.
5. Choose default, minimal, or custom installation (can exclude example project files, importable files, and other auxiliary files).
6. Specify the installation folder.
7. Specify the main menu folder for shortcuts.

To proceed to the next step, click the Next button.

After installation, a confirmation window will appear. Close it by clicking the Done button. If you have a CAM system with electronic key protection, insert the key into a USB port before running the system.

Note 1: The ENCY installation must be performed with administrator rights. This version requires administrator rights for add-ins setup. If you don't have these rights, create a link to run ENCY with administrator rights or consult support.

Note 2: I n some cases, when installing ENCY with an electronic key, the user may need to additionally install the electronic key driver. To do so, after the installation process, select in Program menu key drivers' setup.

Note 3: If "Electronic key not found!" appears, check that the key is connected. If this occurs on first run, install the electronic key driver.

System files

Main executable files of a system:

- ency.exe — executable system module.
- InP.exe — executable Postprocessors generator module.
- SCUpdater.exe — executable program updater module.

Files of this type can be imported into ENCY:

- *.igs, *.iges — Initial Graphics Exchange Specification file format;
- *.dxf — Drawing Exchange Format by Autodesk™;
- *.stl — native stereo lithography CAD file format;
- *.vrl — Virtual Reality Modeling Language file format;
- *.ps — PostScript file format;
- *.eps — Encapsulated PostScript file format;
- *.x_t, *x_b — Parasolid™ kernel file format;
- *.sldasm, *.sldprt — native SolidWorks™ project file format;
- *.asm, *.par, *.psm, *.pwd — native SolidEdge™ project file format;
- *.3dm — native Rhinoceros™ project file format;
- *.step, *.stp — STEP, a modern standard for 3D data exchange;
- *.stcp — system project files containing all information about the project, such as geometrical models, machining operations, toolpaths, etc. The previously used extension *.stc is also supported;
- *.stcx — XML format project files. The project is also saved in the *.stcp format;
- *.sto — files that contain information about operations, including the structure and parameters of operations within a project;
- *.dxf, *.eps, *.ps — files containing 2D geometry information;
- *.stl — files containing simulation results;
- *.xml — files containing toolpath information for external simulators;

- *.spp, *.inp, *.sppx — postprocessor tuning files for various CNC systems, used by ENCY and the Postprocessors generator.
- *.stfc — a zip-container file that can contain any other system files (postprocessors, machine schemes, projects, etc.). Content can be encrypted and digitally signed;
- *.snci — interpreter tuning files for various CNC systems, used by ENCY in Simulation of machining by NC program mode and in G-code based operations.
- *.dsk, *.cfg — files containing data about the system screen settings. These are generated automatically if they are missing.

Other files and folders created during installation are essential for the proper functioning of the program. Modifying or deleting these files may prevent the program from functioning correctly. Substituting files should only be done during version upgrades and must be performed in strict accordance with the provided update instructions.

Technical support

Technical support of the software is provided by the dealer or one of the representatives of ENCY Software Ltd.

Find your nearest ENCY dealer: encycam.com/find-dealer/

If you have any questions or comments regarding ENCY ® please refer to

- The WEB-page: encycam.com
- Technical support service: support@encycam.com

Postal address:

ENCY Software Ltd.

9, Aiolou and Panagioti Diomidous 3020 Limassol Cyprus

For frequent notification about updates of the current version, and release of new versions of ENCY ®, we recommend registering on our company's website.

Brief and to the point

This section introduces ENCY ideology, fast familiarization with the system, and new features in ENCY 3.0.

Ideology of ENCY

ENCY® is a system with high levels of automation and a multitude of advanced functions.

The machining process creation procedure consists of the following actions. At first, it is necessary to define a model of the produced part, an initial workpiece and fixtures. Next, it is needed to create a machining sequence operation by operation. A technologist should define what and how must be machined for each operation. Usually, it's usually any part of the full item or the entire part and general requirements for the machining process, such as the height of the scallop, maximum cut angle, approach methods, etc., are used. Then the system then calculates the optimal toolpath according to the specified parameters. As default, every next operation uses the same part and fixtures as the previous operation, but this one takes a material that remained after the previous operation machining as a workpiece. Therefore, the intermediate workpiece form is changing operation by operation from the initial workpiece to the produced part.

The procedure for NC program for the CNC machine is generally reduced to a sequence of actions:

- to import a geometrical model;
- to define a model of the produced part, the initial workpiece and fixtures for the root node of the operations list;
- to create a sequence of machining operations, to set their parameters and calculate them;
- to generate NC program.

When creating a new machining operation, the system automatically sets the default values for the entire set of operation parameters, taking into account the method of machining and geometrical parameters of the part. Thus, the toolpath is ready for calculation, and there is no need to fill in many parameters beforehand. Modification of the order of the machining operations and editing their parameters is possible at any stage of the machining process designing.

ENCY® always observes the rule: the part must never be "gouged" for any circumstances, whether it is a work pass, transition, approach, cut, or drilling. That does not depend on the tool type, or on the type of machining or parameters entered. A technologist defines the machining method and the system generates a NC program that removes the workpiece material from outside the part.

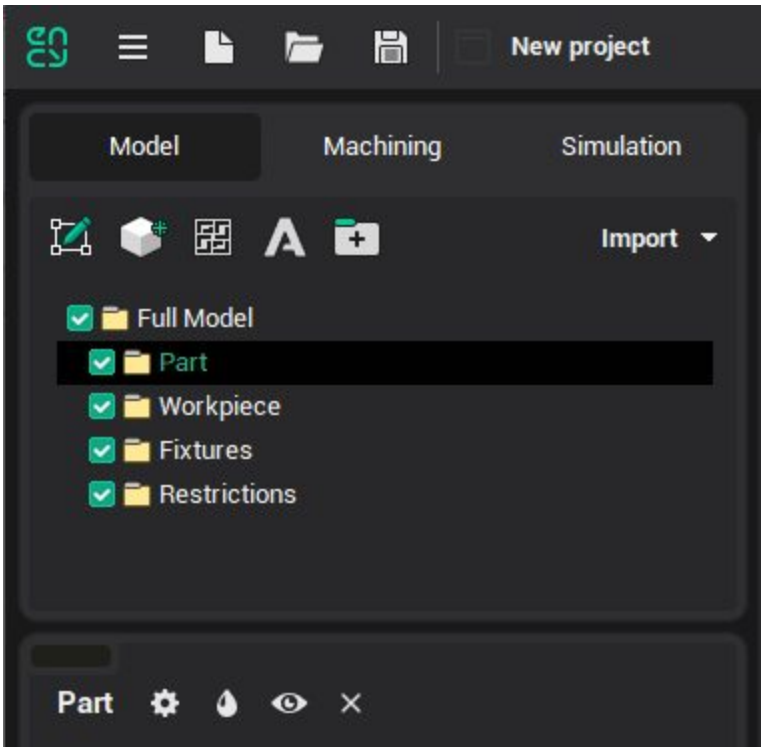
The interface of ENCY® allows the customer to alter any parameter in any order that is required by them. The "active graphics" in many of the dialogues are automatically updated when a customer selects different operations or parameters. This allows for significantly reducing the time needed to get to know the system and the time spent reading the documentation.

ENCY software is always looking at how to improve their software products to make them convenient to use and boost your profits. Therefore, the technical support department will be glad to answer any of your questions and will be thankful for your suggestions and wishes.

Fast familiarization with the system

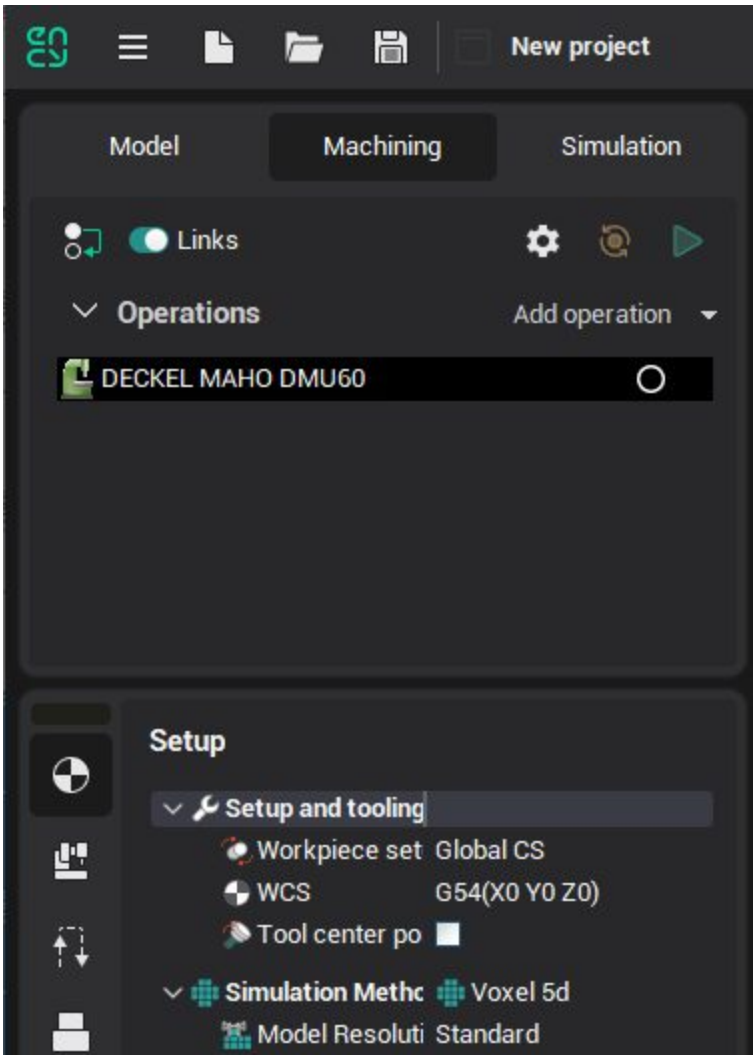
Do the following actions to compute a toolpath and generate an NC program in a short time. The typical action sequence allows you to quickly get to know the CAM system.

- Set the working mode to Model by selecting the tab sheet with the same name on the main panel.



- Import a model from the geometrical data exchange file. To do so, just click the Import button and select the file you want in the window. The model will be imported into the active folder. Therefore, before the import starts, please activate the `Full Model/Part` folder for the part model import, `Full Model/Workpiece` for the workpiece and `Full Model/Fixtures` for fixtures etc. It is recommended to make all modifications to the geometrical model before defining the machining sequence, as the default parameter values are chosen according to the part and workpiece at the moment of operation creation.

- Set the working mode to Machining by selection of the corresponding tab sheet on the main panel:



- Check the machine type and its parameters; edit them if it is necessary. The available operation set depends on the machine type. For example, lathe operations only are available for the lathe machine, milling operations only – for the milling machine, but both operation types can be applied for a turning and milling center.

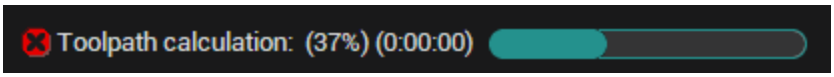
- Form the Part the Workpiece and Fixtures for the root node of the operations list if the model definition demands of some addition constructions. It is possible to make prisms and bodies of revolution that are based on curves and references items (box or cylinder that is circumscribed on the part for example) in addition to the imported models.

- Create a new operation (or several operations). To create an operation, press the Add operation button and select the new operation type in the opened window. The newly created operation will be added into the Machining folder of the root node. The new operation is set as current, and it is ready for editing and execution.

- Define the operation's parameters. When creating an operation, the program sets Part and Fixtures as references to the models of the same name from the previous operation. The references for the first operation are to the root node models. Workpiece is set as a reference to the material that remained after the previous operation of machining. Job assignment is the full part as default. CAM

system automatically defines all other operation parameters according to the foregoing geometrical data. In many cases, the new operation can be immediately executed. You can check or alter operation parameters on the panel in the bottom-left side of the main window or in the windows that are opened by the Parameters button click.

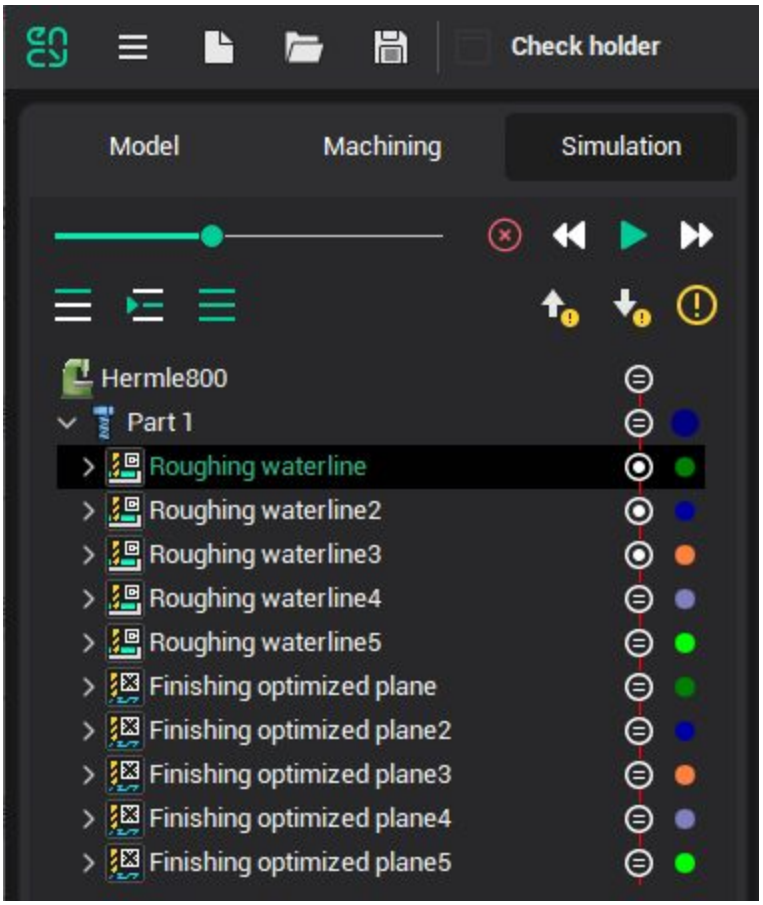
- Start operation execution by clicking on the  button. On the process indicator, you will see how much of the calculation has already been completed. By clicking within the indicator area, you can stop the process.



The toolpath calculating rule is to machine the Job assignment by the defined strategy with control of the Part and Fixtures. Roughing operations remove all covering material of the Workpiece to do that. The workpiece checking is optional for all finishing operations. If the workpiece is taken into account then toolpath appears for the job assignment areas where the rest material of the workpiece presents else, this one appears for the whole job assignment.

- You can estimate the machined piece by enabling Machining result visibility  on Visibility panel.

- Check the obtained toolpath block by block in Simulation mode. Select the corresponding tab sheet on the main panel and click on the  button to start the simulation process. If it is necessary, then change operation parameters and recalculate them.



- Run the Postprocessor by the 

keystroke at Machining mode. The postprocessor window will be opened. Select a CNC system (choose postprocessor file with the *.sppx extension) and click the Run button. You can see the NC program name in the Output file field. As default, the file has a name of the current project and an extension that is set in the selected postprocessor.
- Save the project by clicking on the  button of the main toolbar. You can select the project save mode by the slider position. The project size will be bigger if the saved data is more detailed.
- You can load a previously created project by clicking on the  button on the main toolbar.

What's new in ENCY 3.0

- General improvements
  - Interface.
    - The Home Page has been updated.
    - The context menu of the inspector has been update.
    - The Machine limits group has been added.
    - Toolpath recalculation has been optimized.
    - User defaults have been implemented in the inspector.
    - Redesigned the Utilities button of main panel for quick access to built-in and custom tools.
  - Technology updates
    - New tool assemblies have been added.
    - A new Combo Finish operation has been added.
    - A new Blade Machining operation has been added.
    - A new Contour Polishing operation has been added.
    - A new option has been added to the Check Holder: Adaptive tilting, Check links.
    - The Recognize features option has been added for the 6D Contouring operation.
    - The Smart Motion option has been improved.
    - The Feeds/Speeds calculator has been redesigned and enhanced.
    - Improvings in Corner control.
    - Pullback option has been added to the Welding 6D.
    - New precise format for exporting simulation result
    - Support Generator has been added.
  - Additive manufacturing updates
    - Improvings in Non-planar slicing operation
      - Radial slicing strategy
    - Gap to prevent overlapping
    - Custom angle sequence for infill.
    - Zig Zag infill pattern.
    - Power function
    - Maximum layer time
  - Design module enhancements
    - Design in 3D mode.
    - Added a Coil.
    - A new option has been added for Helic: Start Runout and End Runout.
    - A new option has been added for splines: Closed.
  - New Nesting Functionality
  - 3D View Controls
    - Added an interactive linear axis ruler.
    - Updated Machine Display Settings in the 3D view.

    - Updated Toolpath Display Settings in the 3D view.
    - Updated Display Mode Settings in the 3D view.
- Other updates
    - Extension Store.
    - Added interactive fork steering for Loop edge selection - Custom Loop.
    - A new Macro option has been introduced.
    - Added a new navigation mode for view rotation.
    - Gimbal functionality has been added to Intersect.

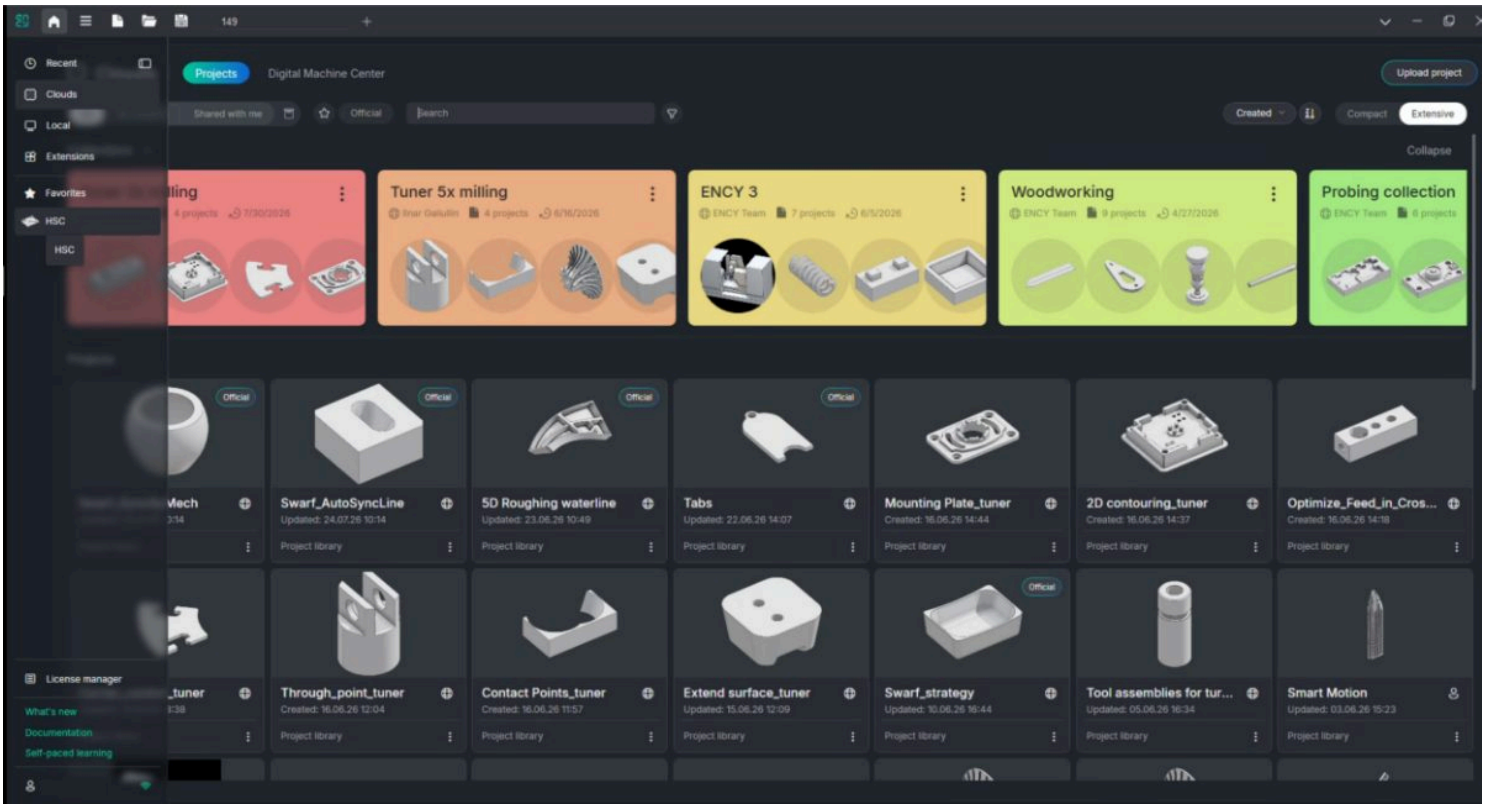
General improvements

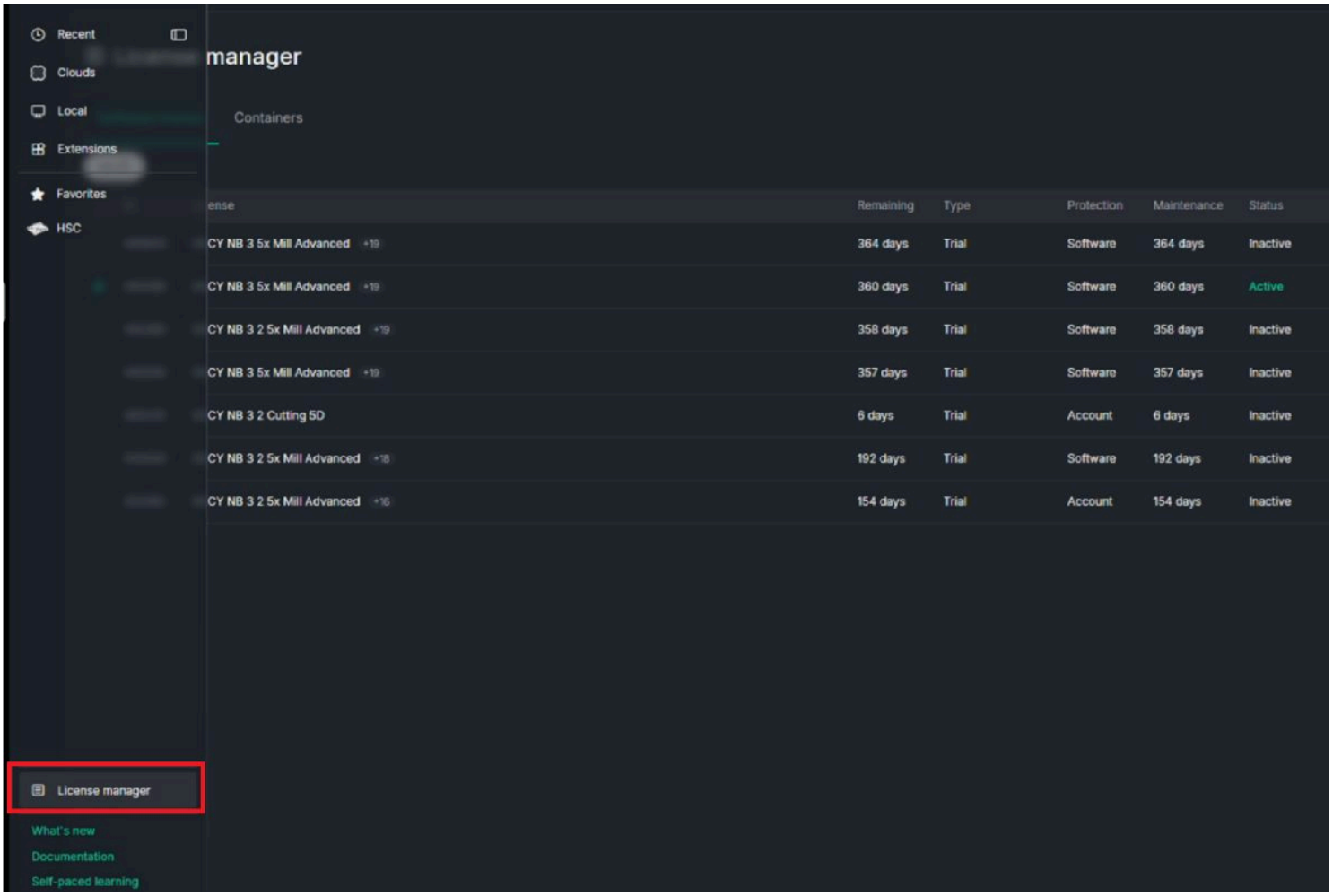
Interface.

The Home Page has been updated.

The appearance of the Home Page has been revised. You can now organize your projects into collections — grouping related projects together as large color-coded cards at the top of the page, while individual project cards are shown below in a redesigned grid. The left navigation panel has been reorganized for quicker access to Recent, Clouds, Local, Extensions, Favorites sections. Details

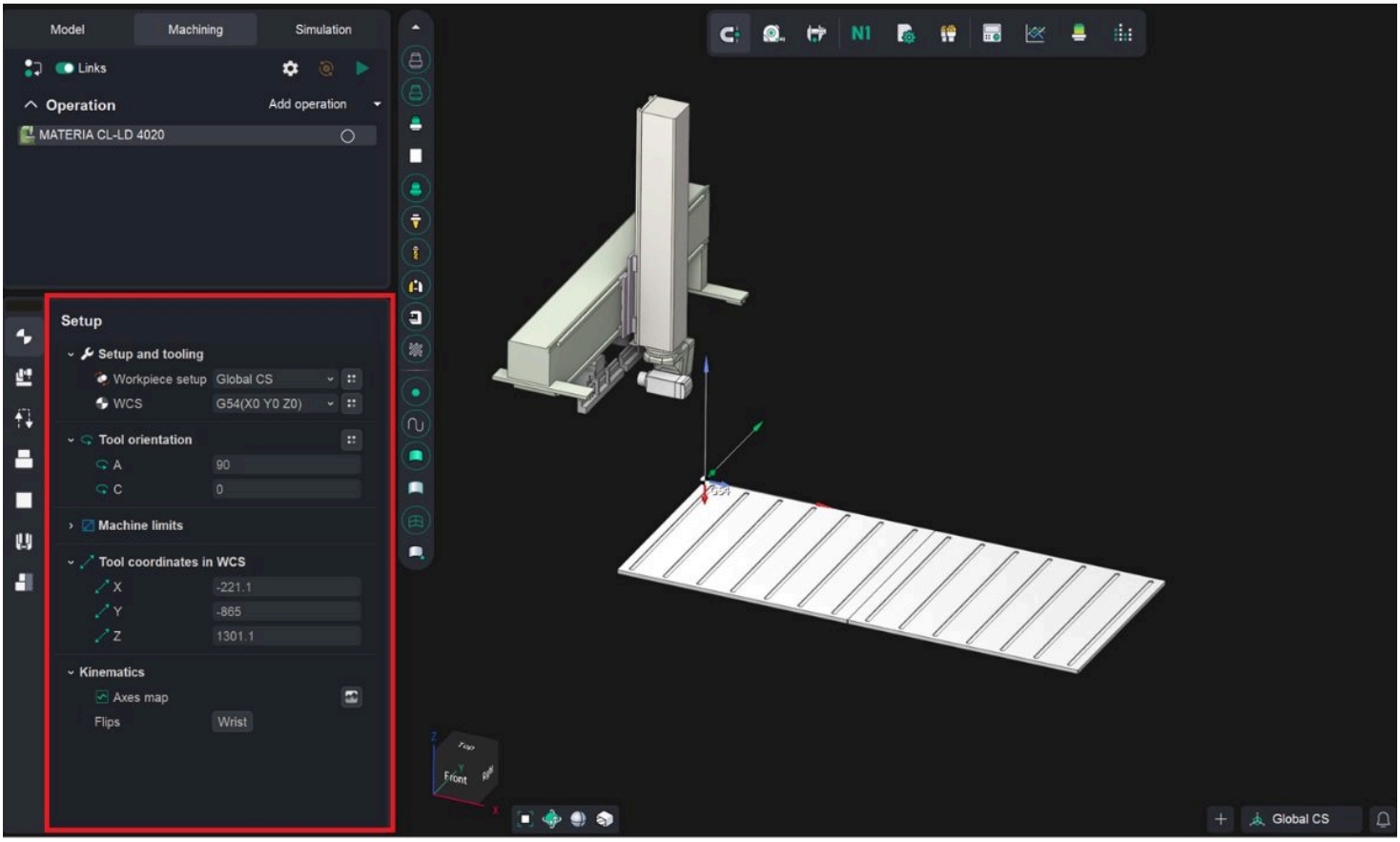
In addition, the License manager is now built directly into the Home Page window instead of opening as a separate window.





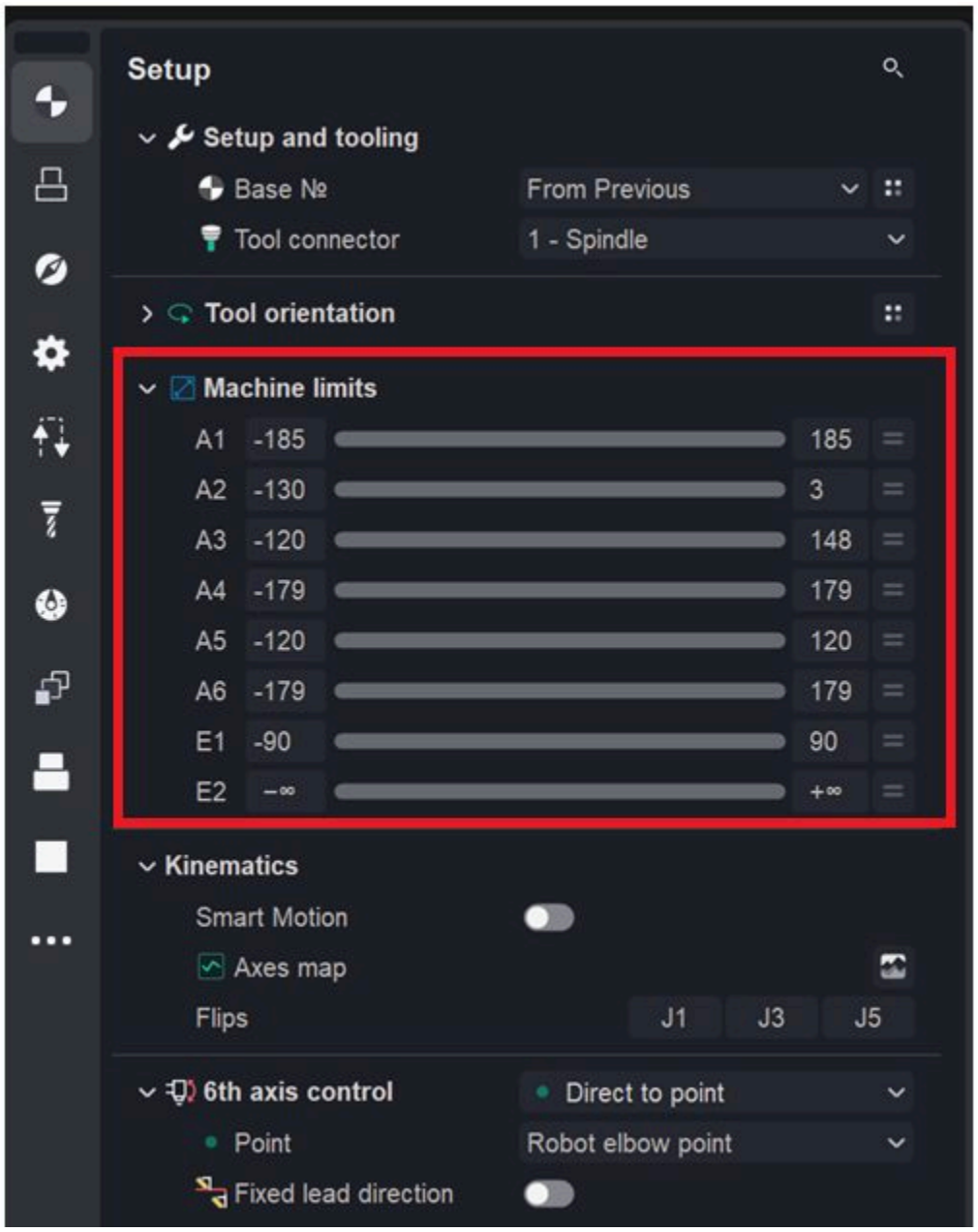
The context menu of the inspector has been update.

The appearance of elements in the inspector (including dropdown lists, switches, etc.) has been revised. Additionally, parameters on the Setup tab for machines and robots have been reorganized. Details



The Machine limits group has been added.

The Machine limits group belongs to the Setup panel and defines reduced per-axis travel limits that are applied when the toolpath and maps are calculated. These values narrow the axis ranges taken from the machine to a smaller working envelope, so that motion is planned strictly within the range acceptable for the current operation. Details



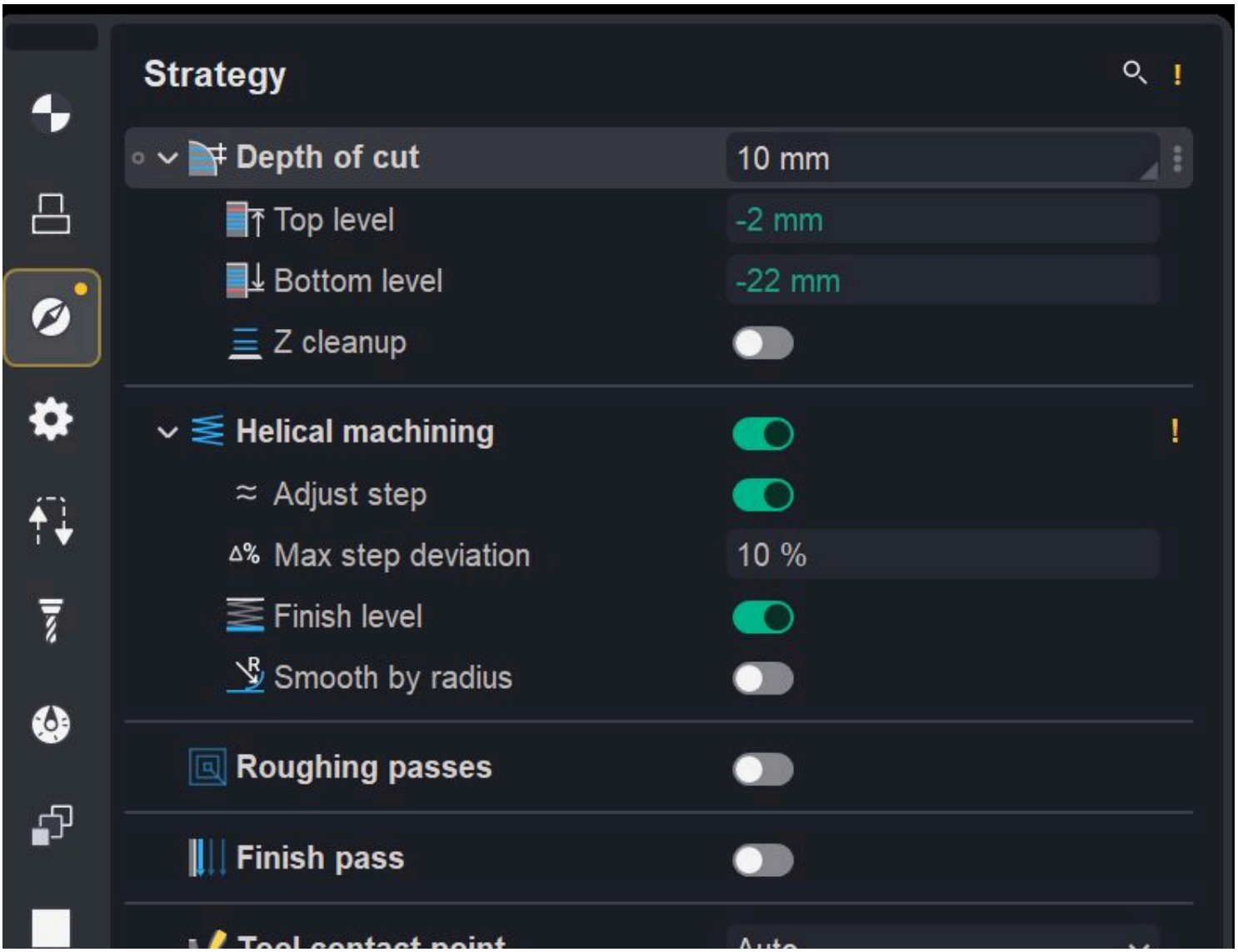
Toolpath recalculation has been optimized.

When parameter changes do not require a full recalculation, the calculated toolpath is preserved. Recalculation updates only the parts affected by the changes, such as transitions, approaches, and retracts.

Changed parameter indicators have been added. The CAM system saves the operation's parameter state when the toolpath is calculated. If subsequent edits cause the current settings to differ from those used for the calculated toolpath, the operation receives a yellow warning status. The inspector marks changed parameters, their groups, and the corresponding pages, making it easier to locate changes that require recalculation.

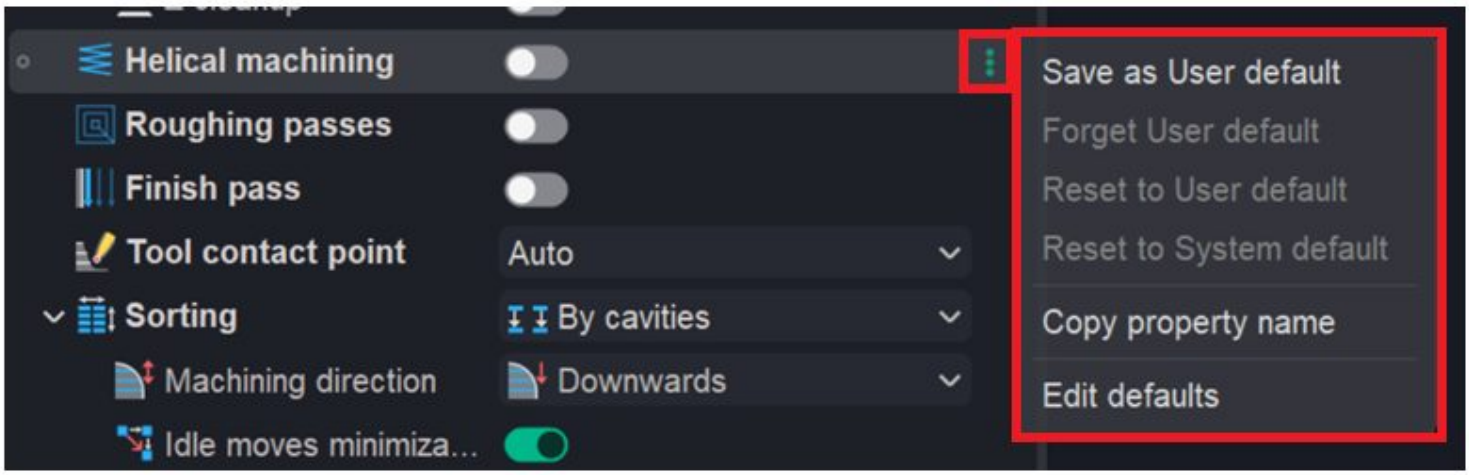
Before recalculating, the parameter values used to calculate the current toolpath can be restored.

To hide the hints, open System setup → Additional → Auto-reset toolpath. Details



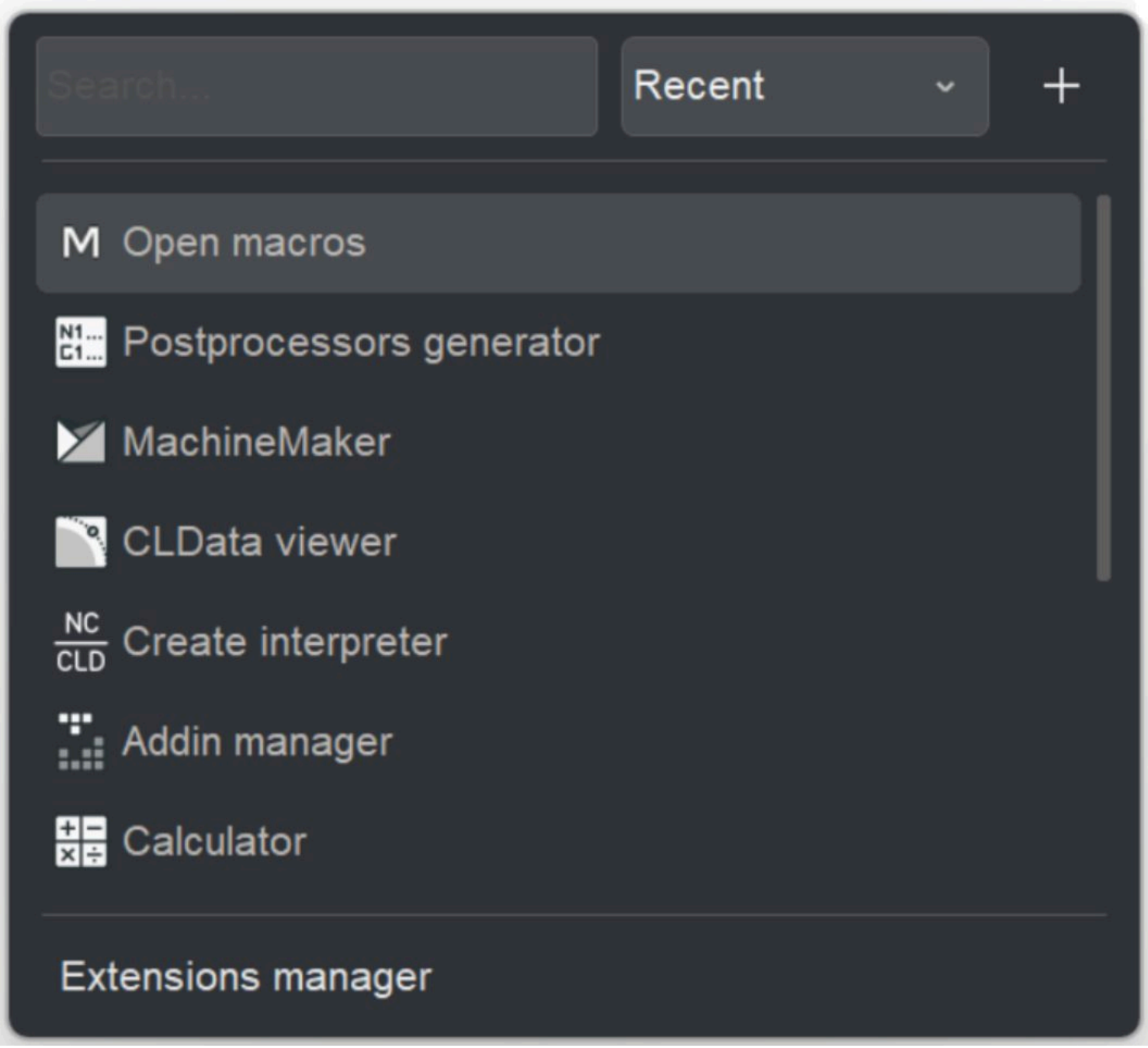
User defaults have been implemented in the inspector.

The inspector's context menu (opened via the three-dot button) now includes the following menu items: Save as User default, Forget user default, Reset to User default, Reset to System default, Copy property name, and Edit defaults. [Details]



Redesigned the Utilities button of main panel for quick access to built-in and custom tools.

The Utilities button of main panel provides quick access to built-in and user-defined tools directly from the main application window. It combines search, sorting, launching, management of custom utilities, and access to the extensions manager in a single panel. Details

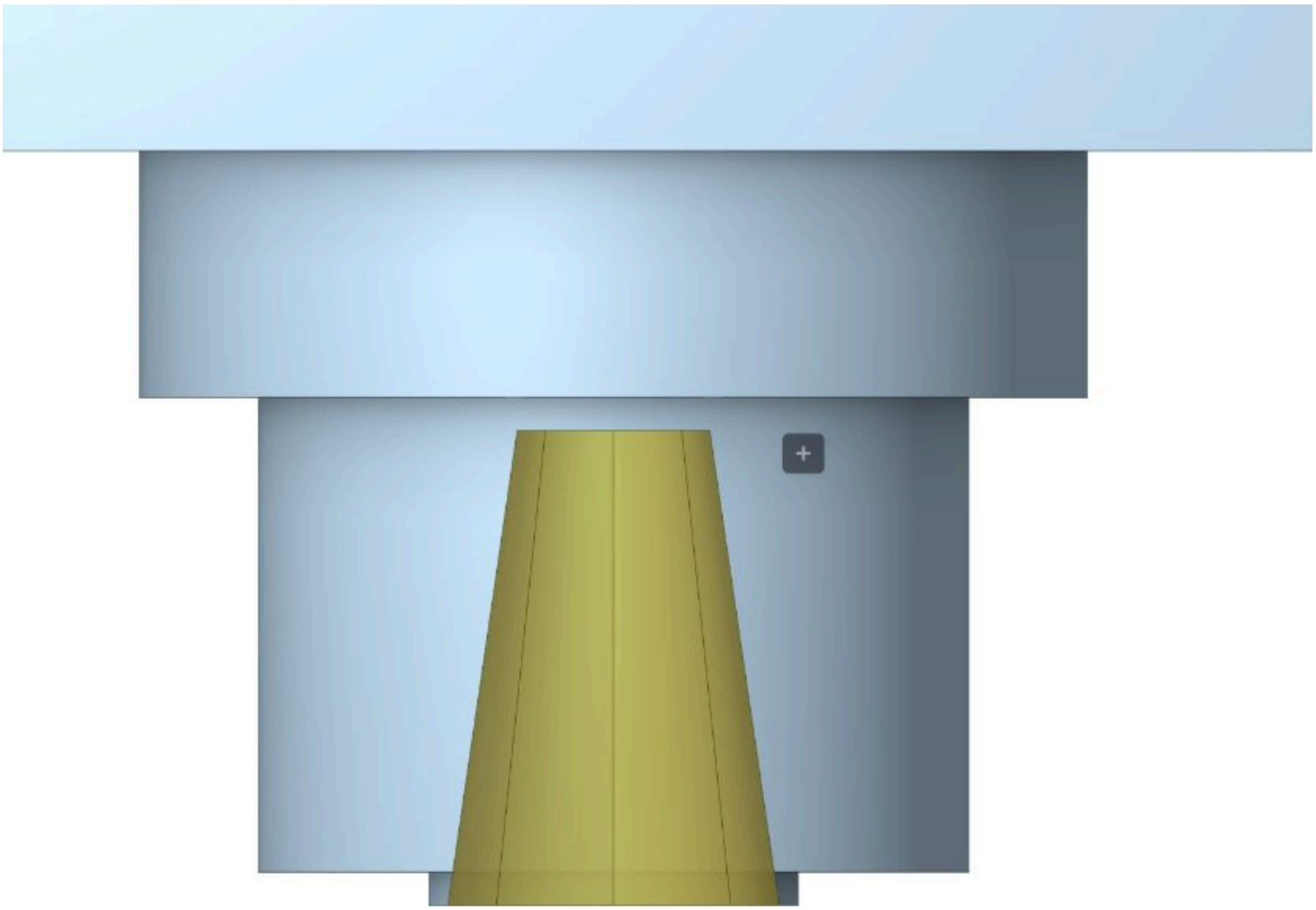


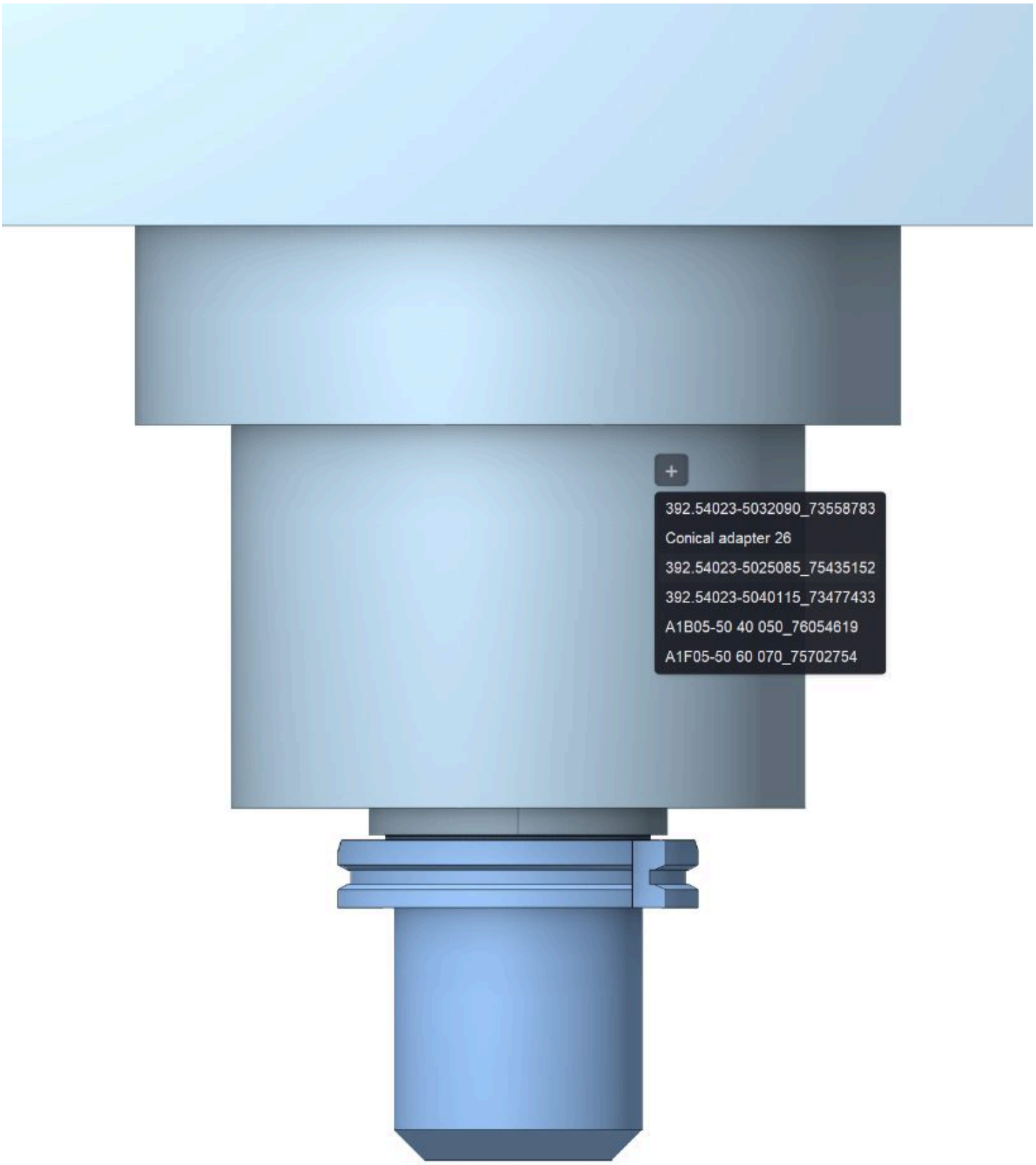
Technology updates

New tool assemblies have been added.

A new mode has been introduced for working with both parametric and 3D tools. It allows you to interactively build a tool assembly in setup mode on the Machining tab — from the mounting seat in the

spindle or another machine mounting system to the tool, including various toolholders. Details




392.54023-5032090_73558783
Conical adapter 26
392.54023-5025085_75435152
392.54023-5040115_73477433
A1B05-50 40 050_76054619
A1F05-50 60 070_75702754

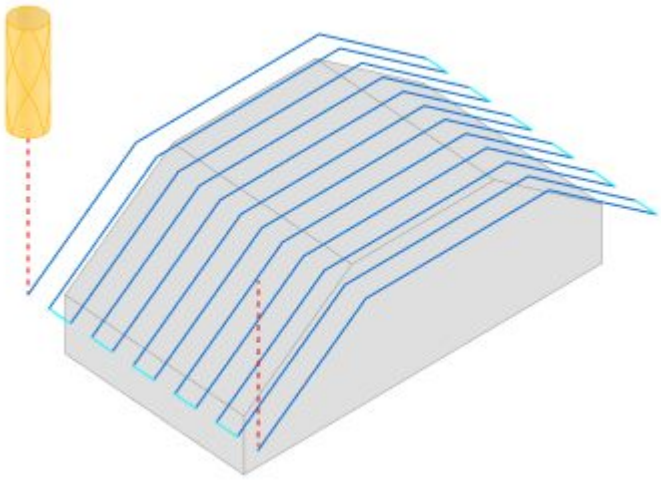

MH20-R025A25-08M_75501240
R216F-25A25S-045_75606597
DS20-D1800L25-04_77094836



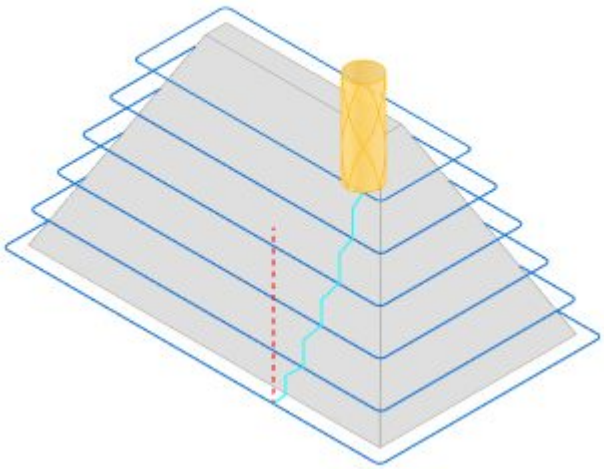
A new Combo Finish operation has been added.

Strategies:

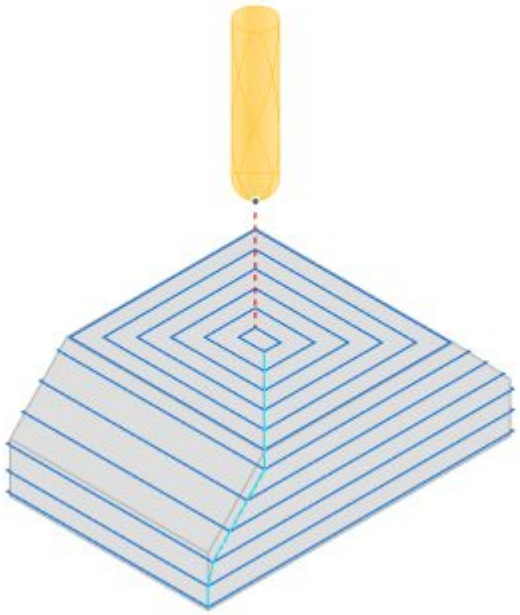
- Plane (Finishing);



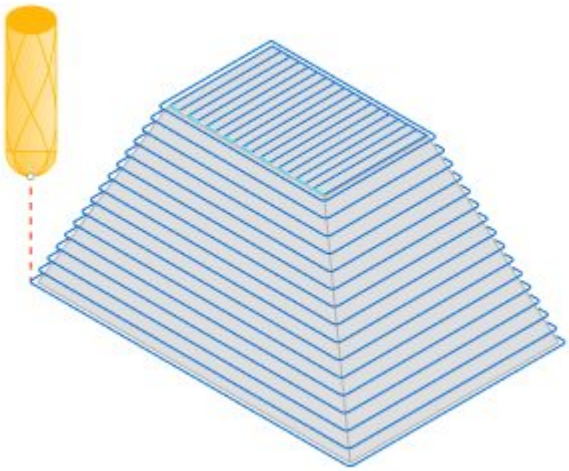
- Waterline;



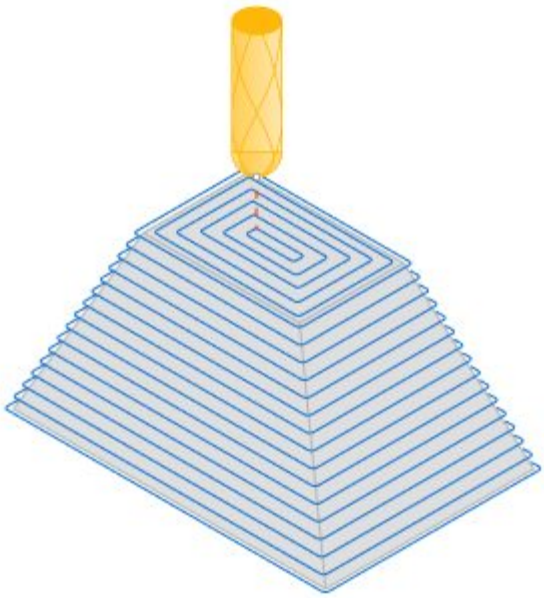
- Scallop;



- Plane + Waterline;



- Scallop + Waterline.



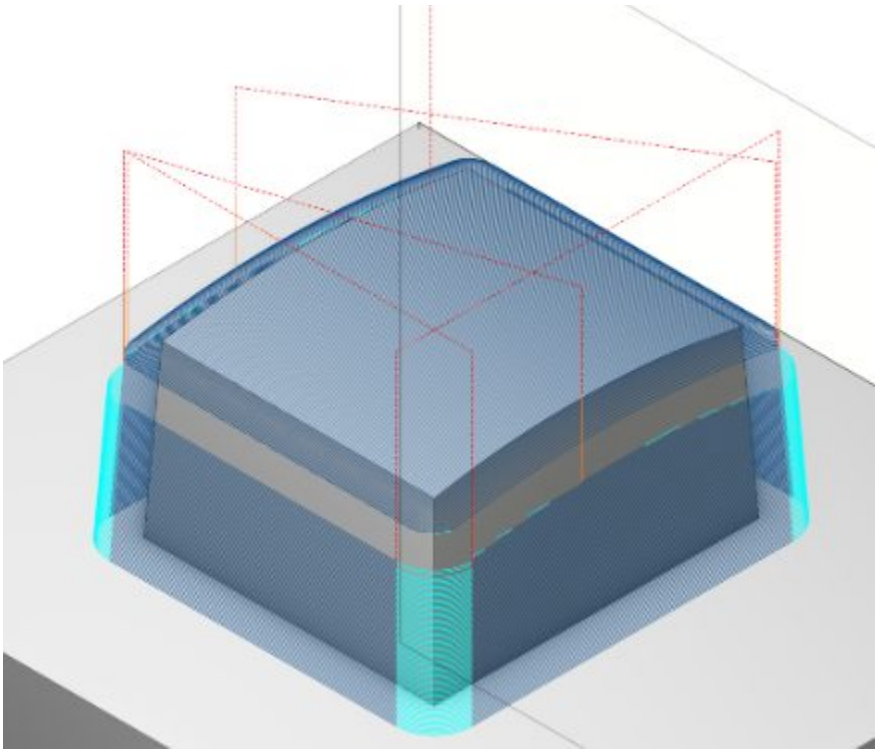
Advantages:

- Support for generating helices for waterline passes (helical machining).
- In combined strategies, gentle and steep areas are processed using different strategies. This ensures a uniform scallop pattern on the surface and improves cutting conditions.
- Combined strategies feature smoothing/merging of machining zones based on the Smooth distance parameter. This helps produce a more solid toolpath with fewer unnecessary passes and transitions.
- Intelligent toolpath merging for combined strategies has been implemented.
- The calculation process is fully parallelized.

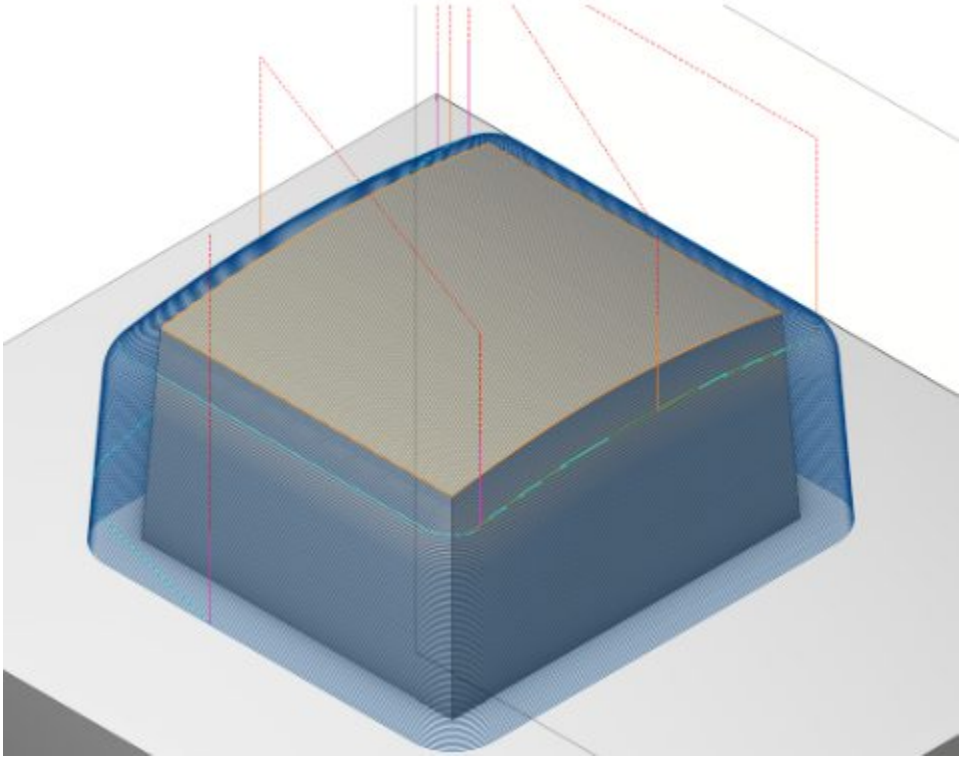
This operation can replace outdated legacy operations and address associated issues. Details

The Combo Finish operation with the Plane+Waterline strategy replaces the legacy Complex operation, offering improved surface quality and reduced machining time.

Complex

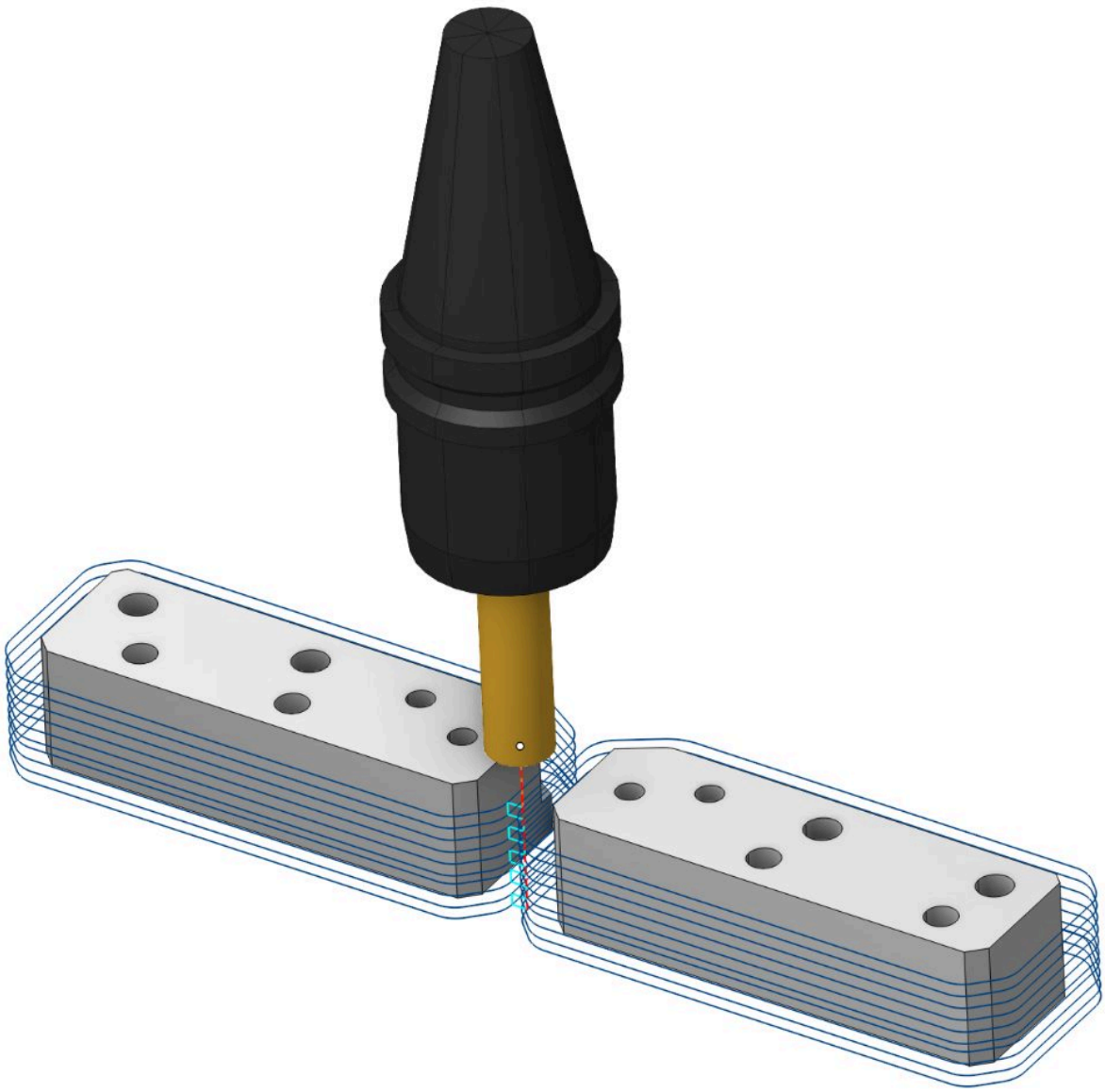


Combo Finish (Plane+Waterline strategy)

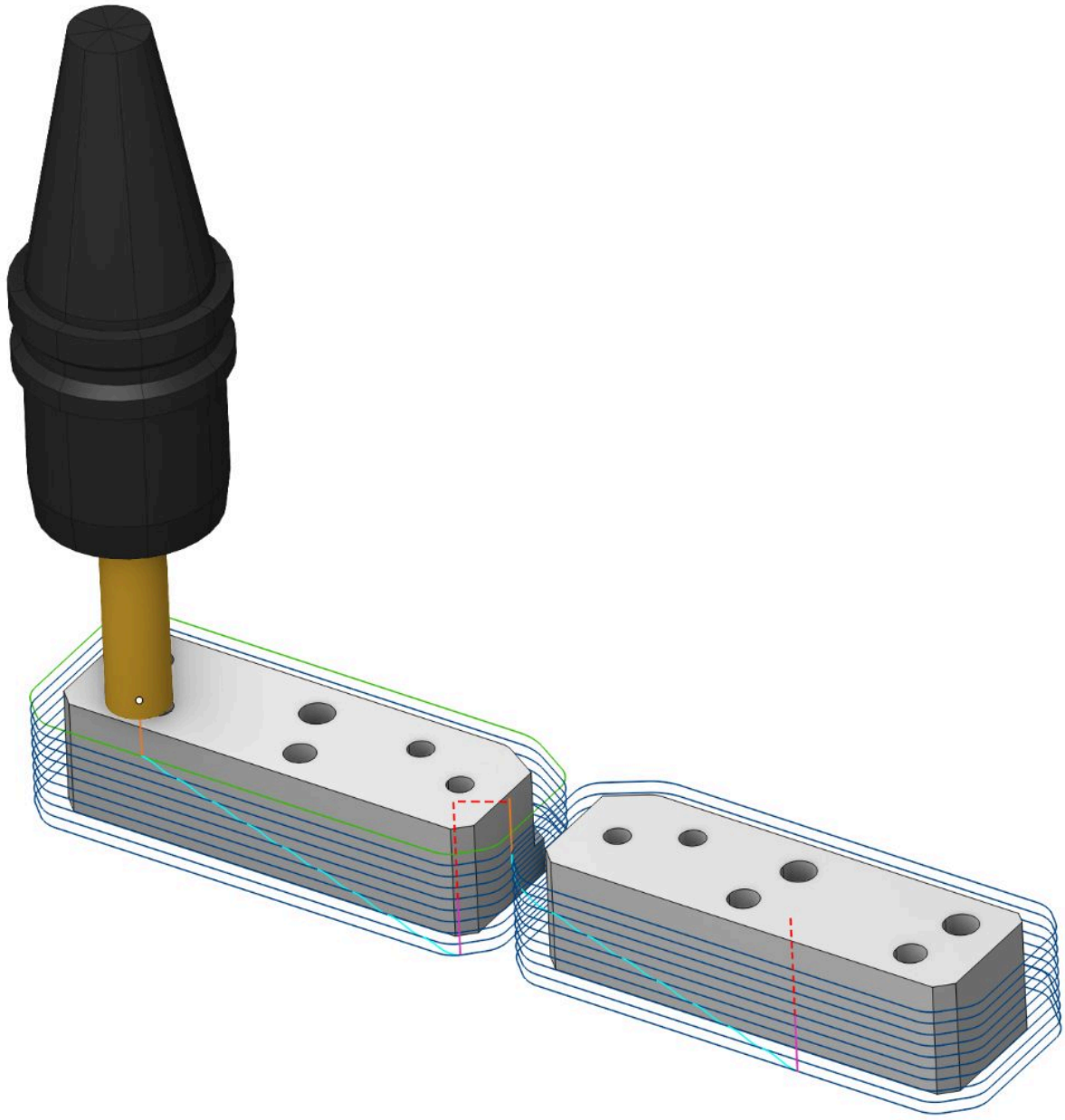


The Finishing Waterline operation lacks a defined start point, which may cause tool collision or gouging between adjacent parts. The new Combo Finish operation now includes a user-definable start point in all strategies, enabling more flexible and controlled machining.

Finishing Waterline (Unsatisfactory tool entry into the workpiece)

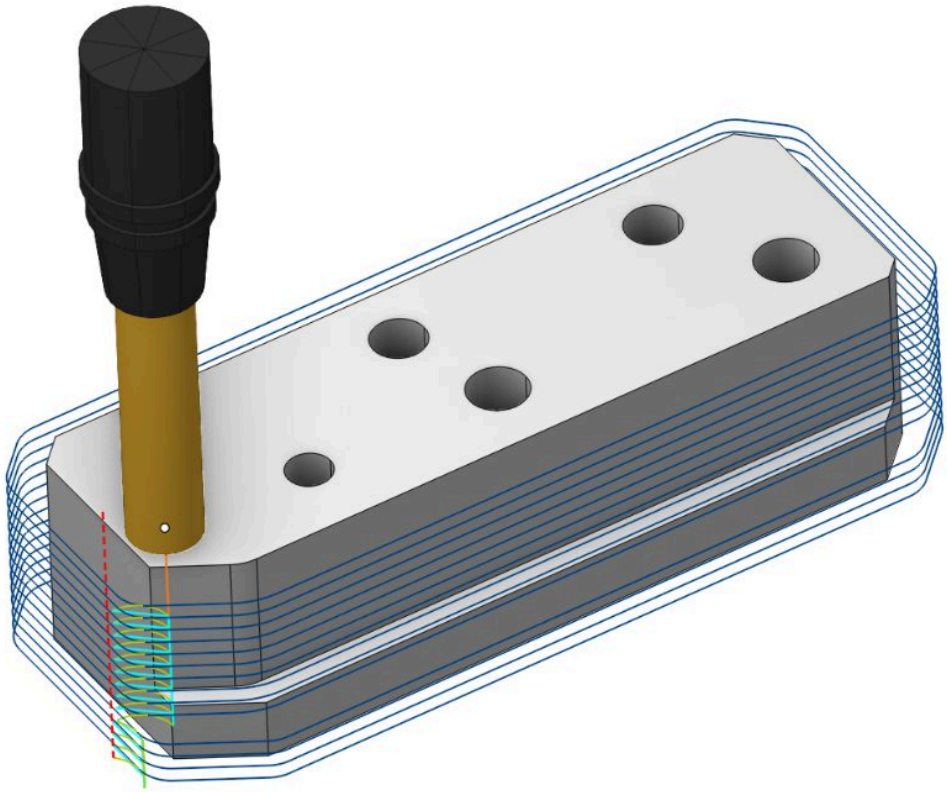


Combo Finish (Allows shifting the start point to ensure smoother tool engagement between two parts.)

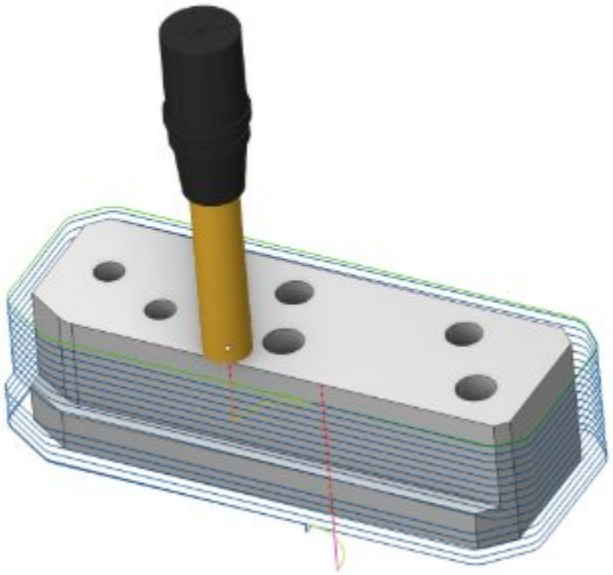


The Finishing Waterline operation currently lacks Spiral machining capability. As a result, it is not possible to perform the intended machining process using this strategy.

Finishing Waterline (Spiral machining is not supported)

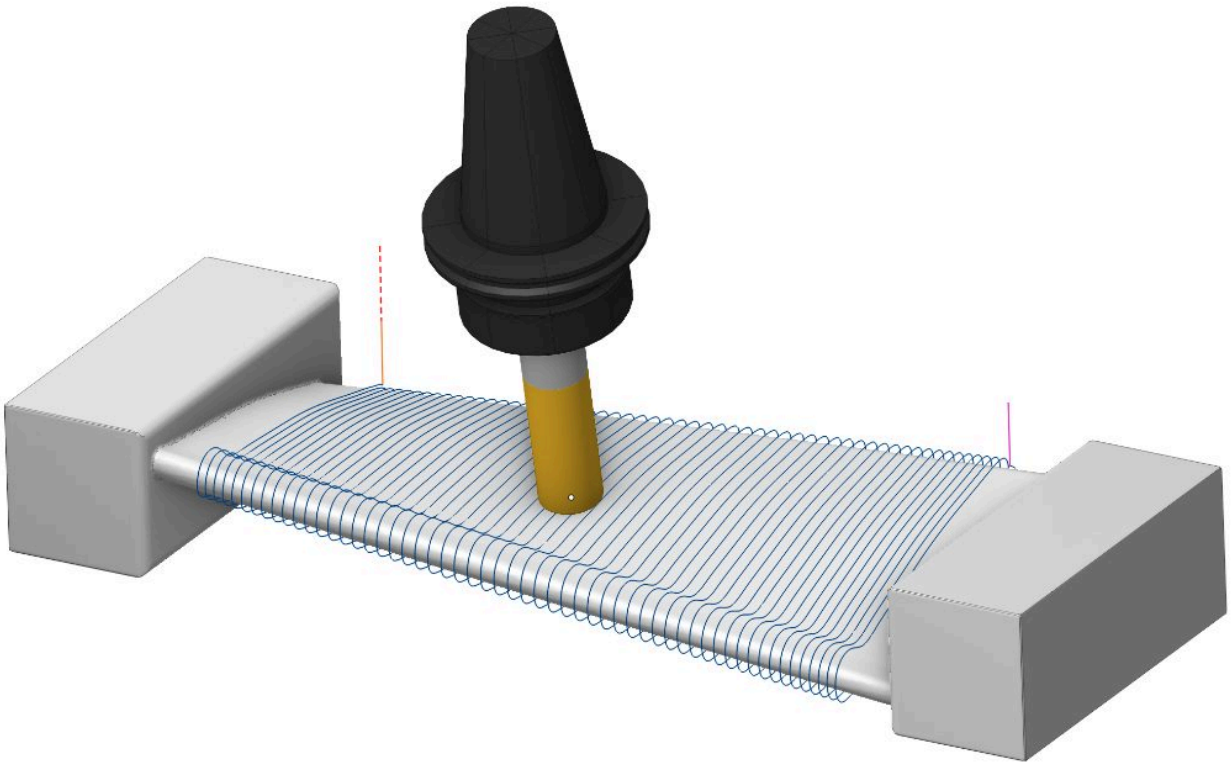


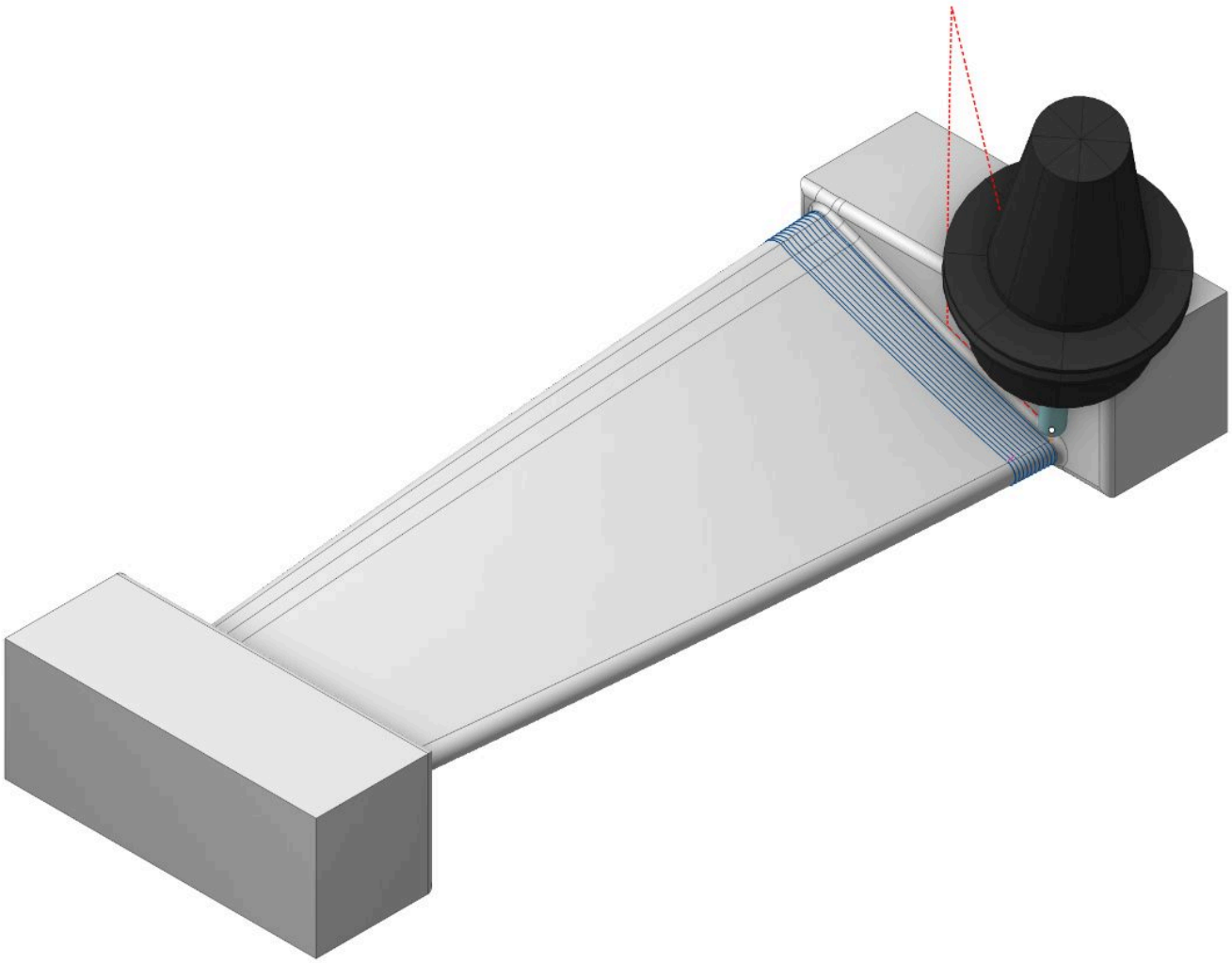
Combo Finish (Allows spiral machining)

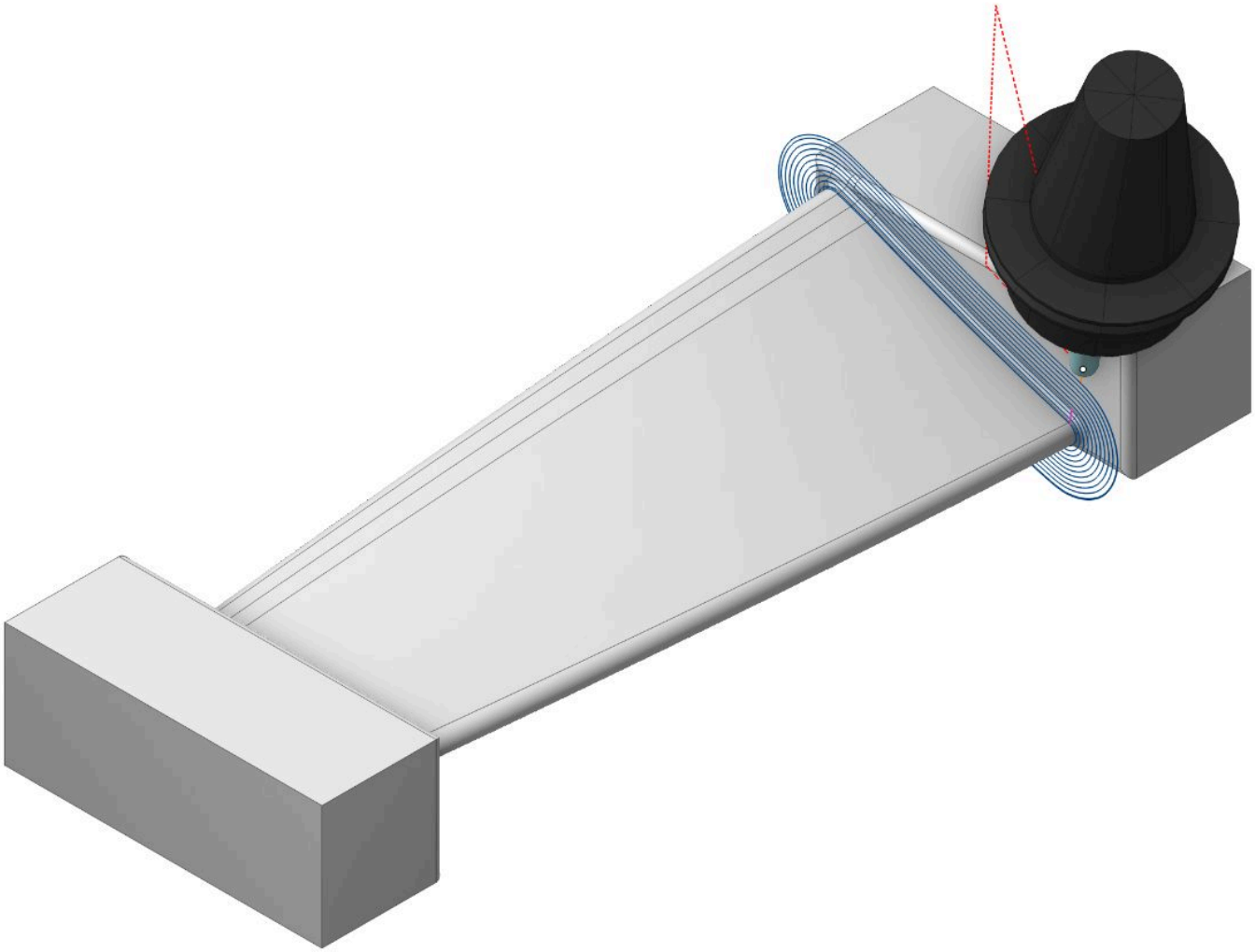


A new Blade Machining operation has been added.

The specialized blade machining module introduces dedicated processing strategies for key structural elements: Blade, Root zone of blade, Hub. This enables optimized toolpaths and improved surface quality for each critical area. Details

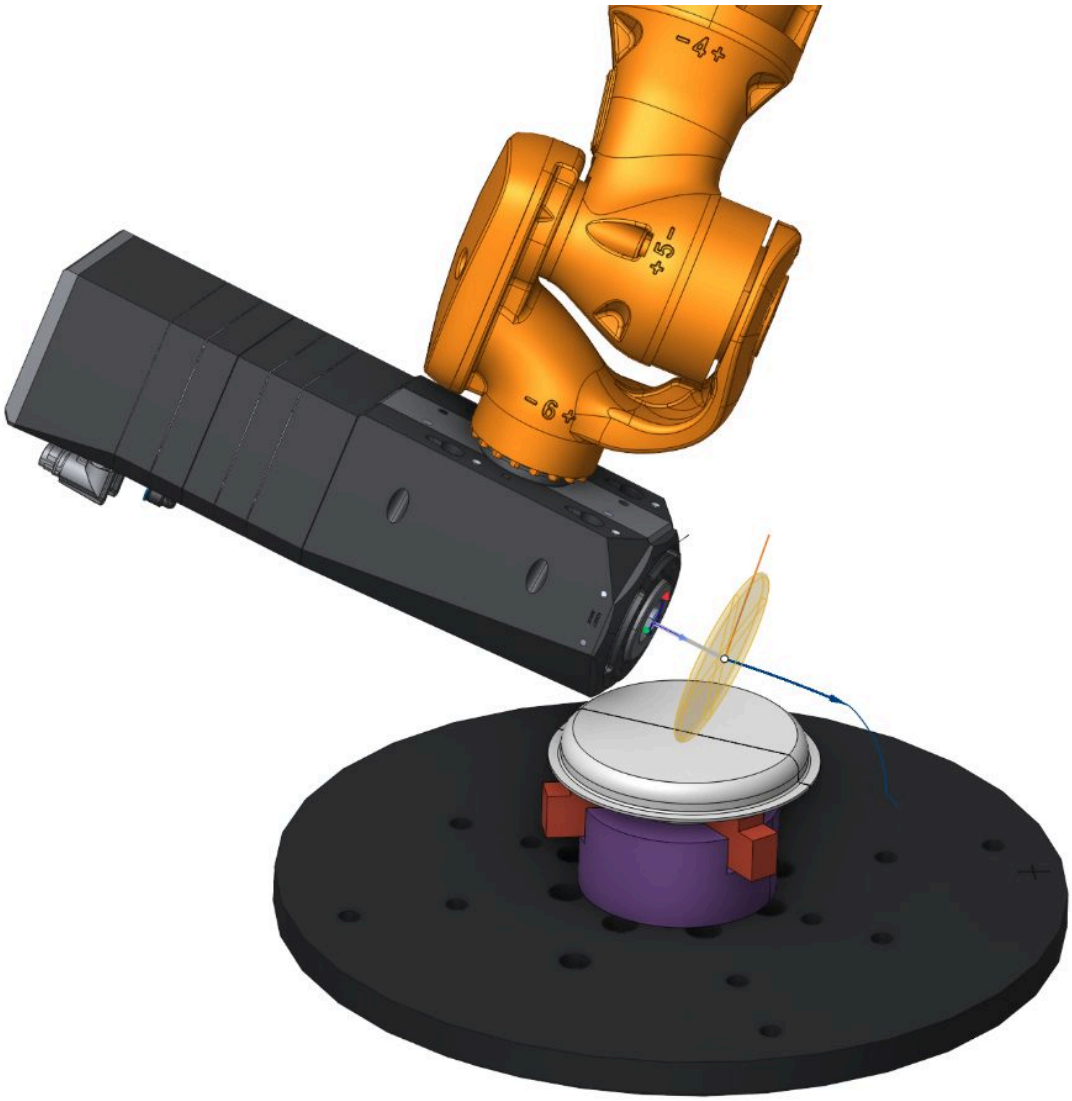






A new Contour Polishing operation has been added.

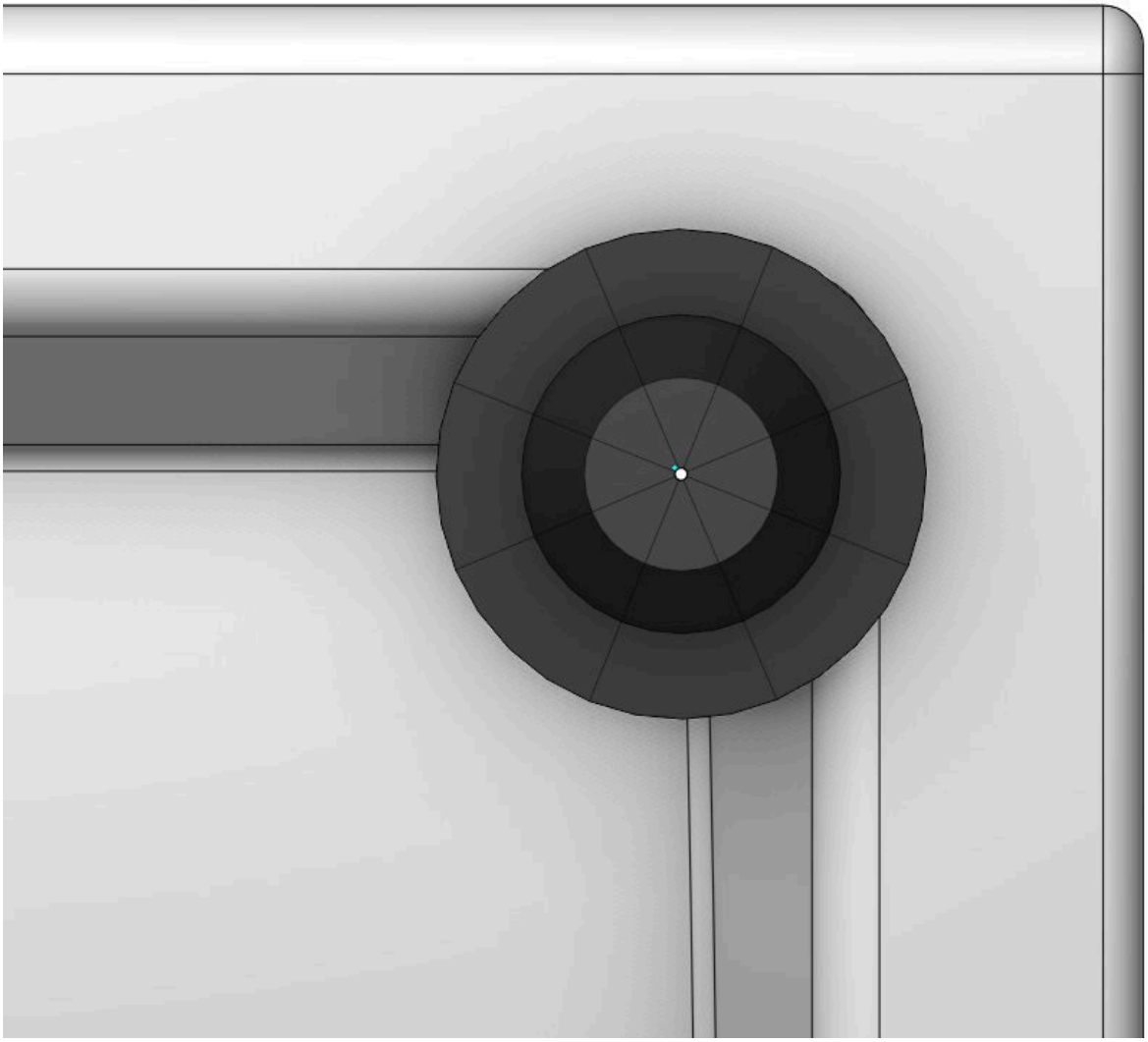
The operation finishes part walls and edges along defined contours using the side of a disc tool. Details

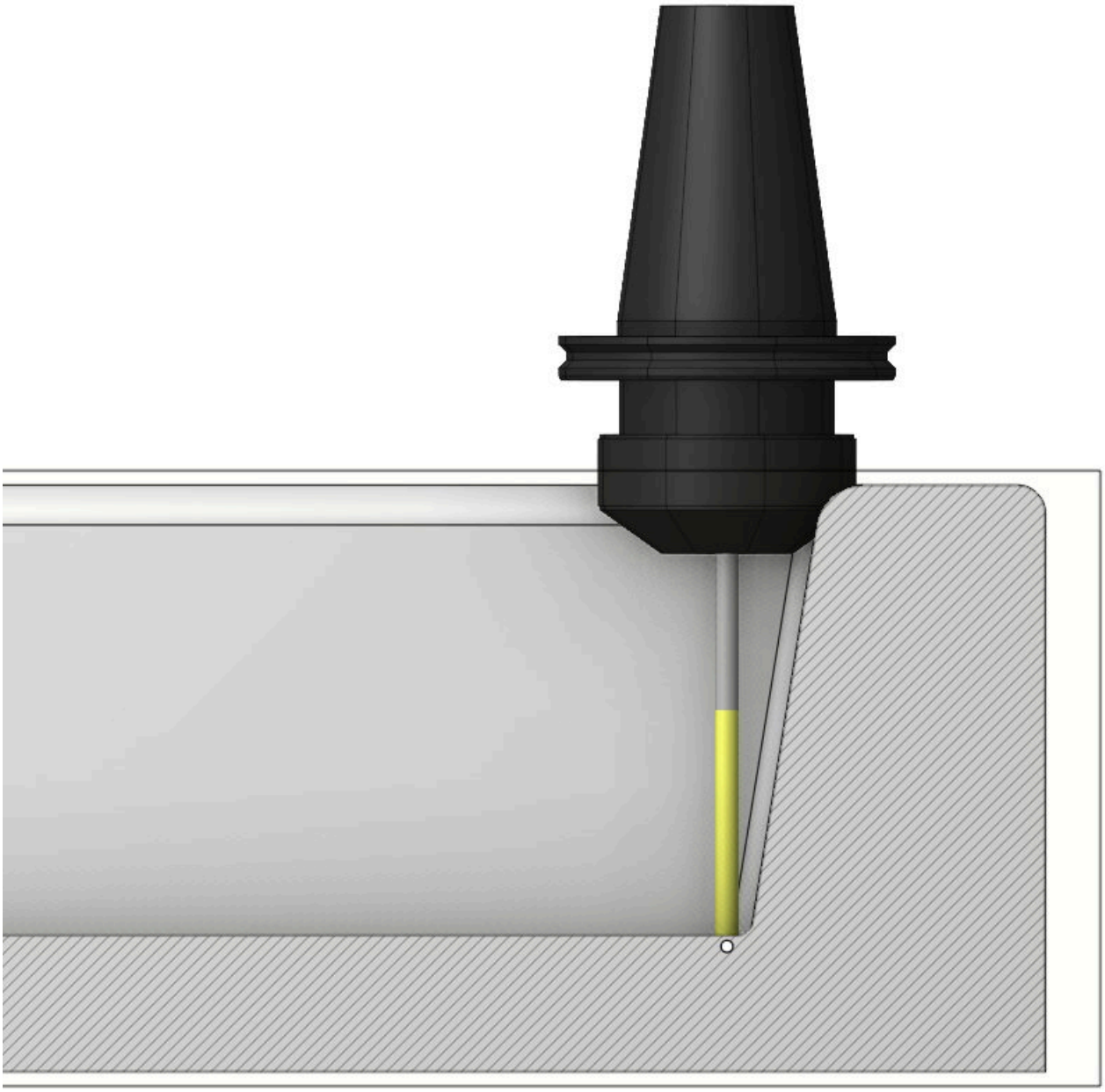


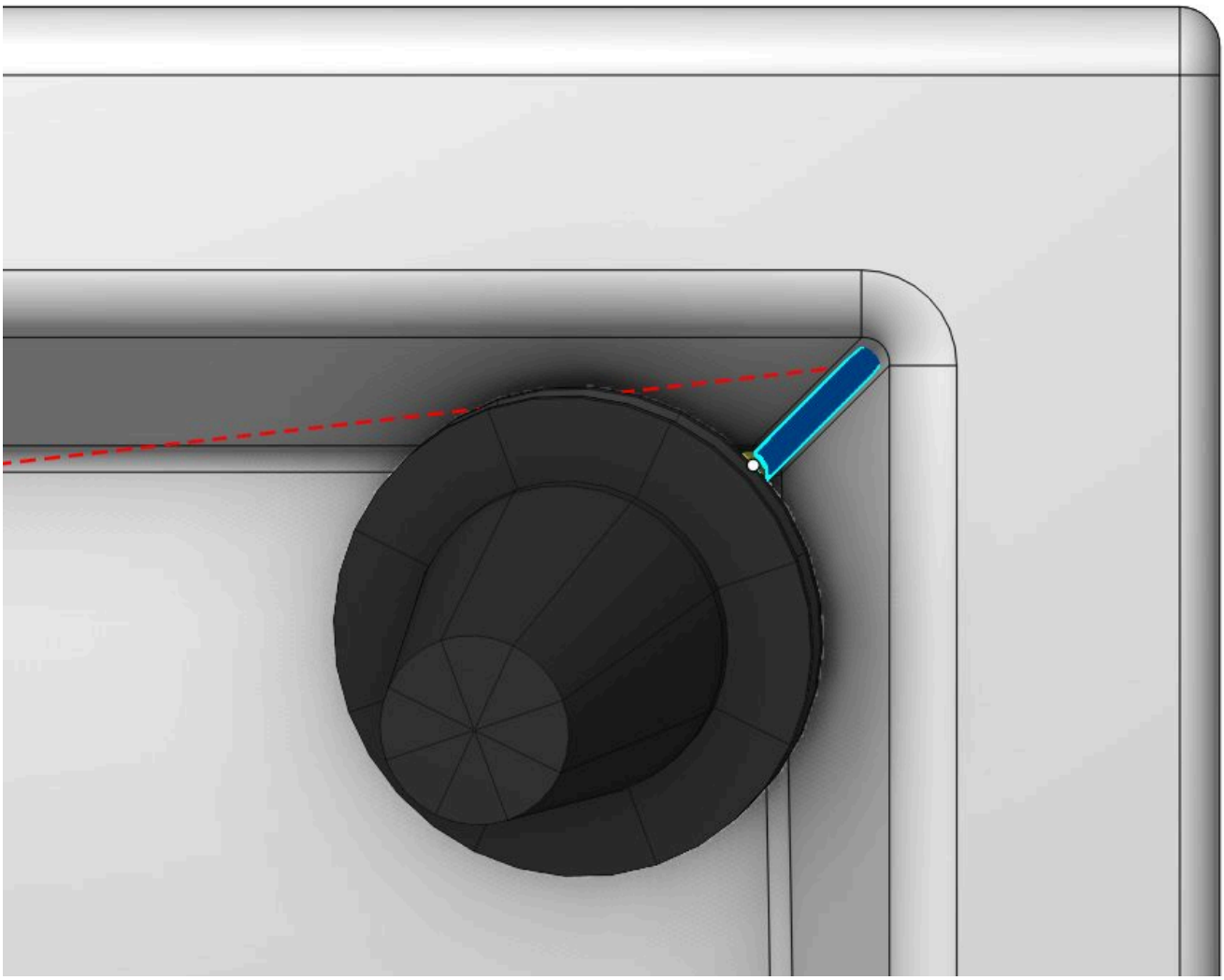
A new option has been added to the Check Holder: Adaptive tilting, Check links.

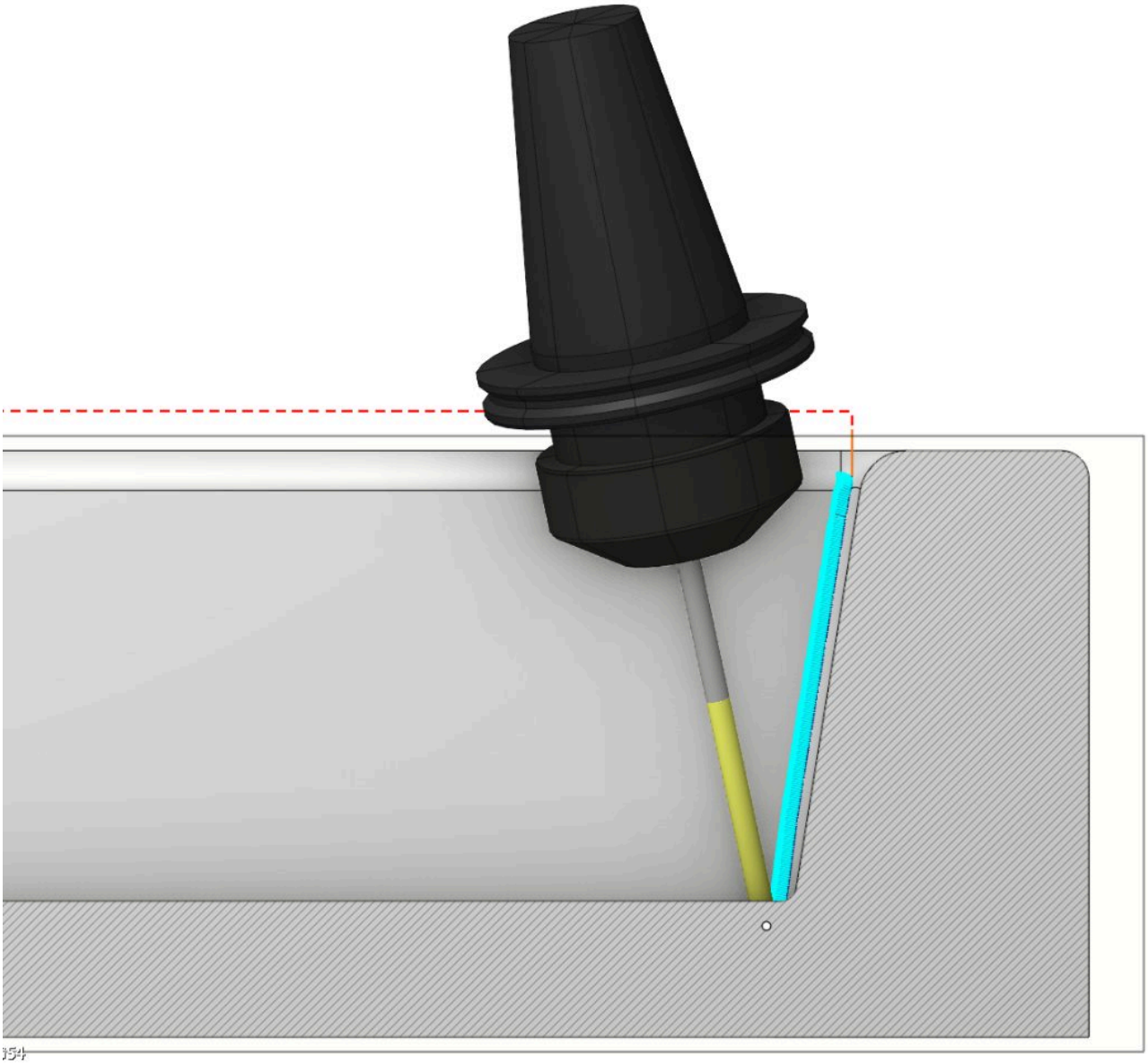
Unlike the Side Tilting option, this feature enables simultaneous angular deviation along two axes. This capability allows for the conversion of 3-axis machining to 5-axis machining at various angles. Details

The Check Links function enables angular deviation during transitions between machining (cutting) passes.



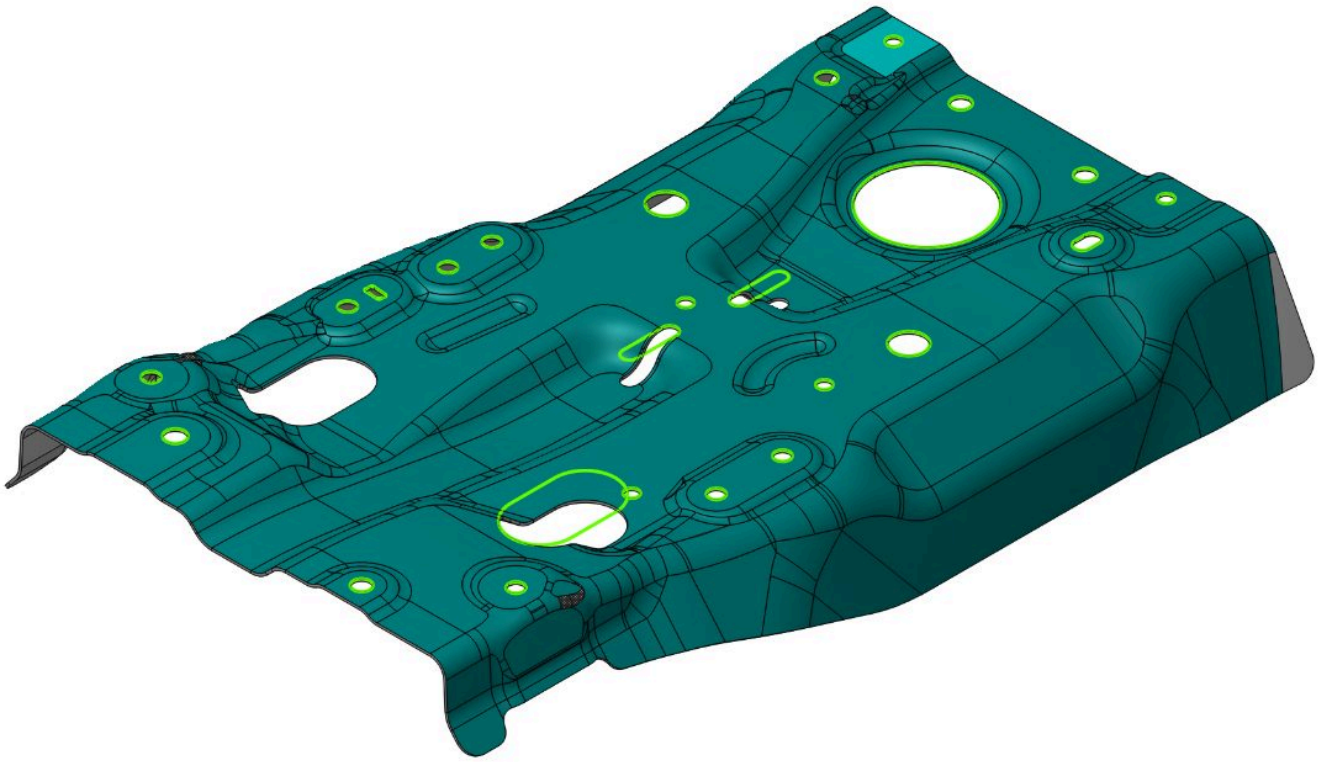






The Recognize features option has been added for the 6D Contouring operation.

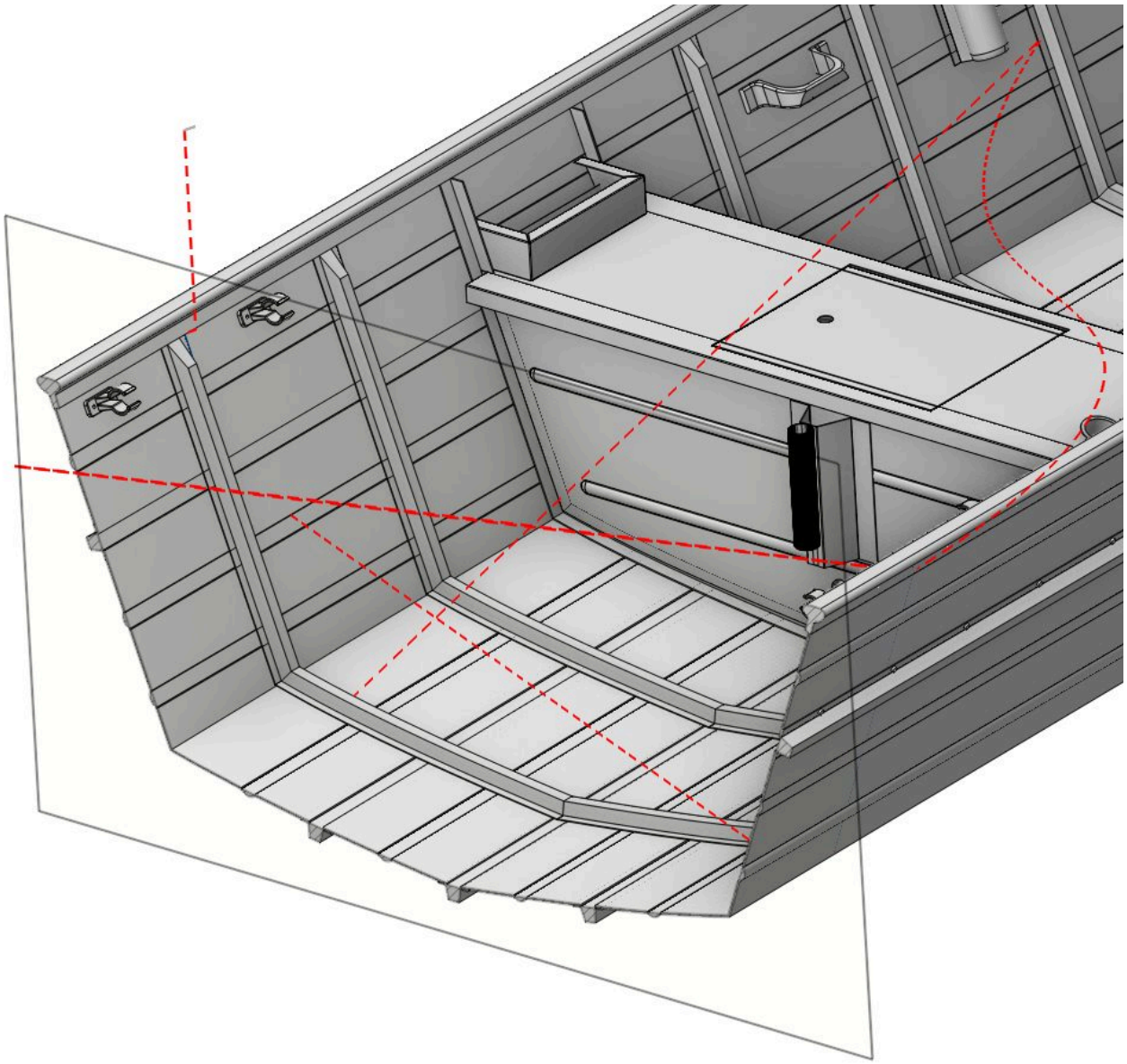
The Recognize features option has been added for the 6D Contouring operation (Details). It allows recognizing the following constructive elements: Circle, Rectangle, Rounded Rectangle, Slot, Hexagon, and Keyhole. This feature replaces primitive motion commands (line segments, arcs) in NC programs with high-level constructive elements to improve code readability, compactness, and enable advanced post-processing features. Details

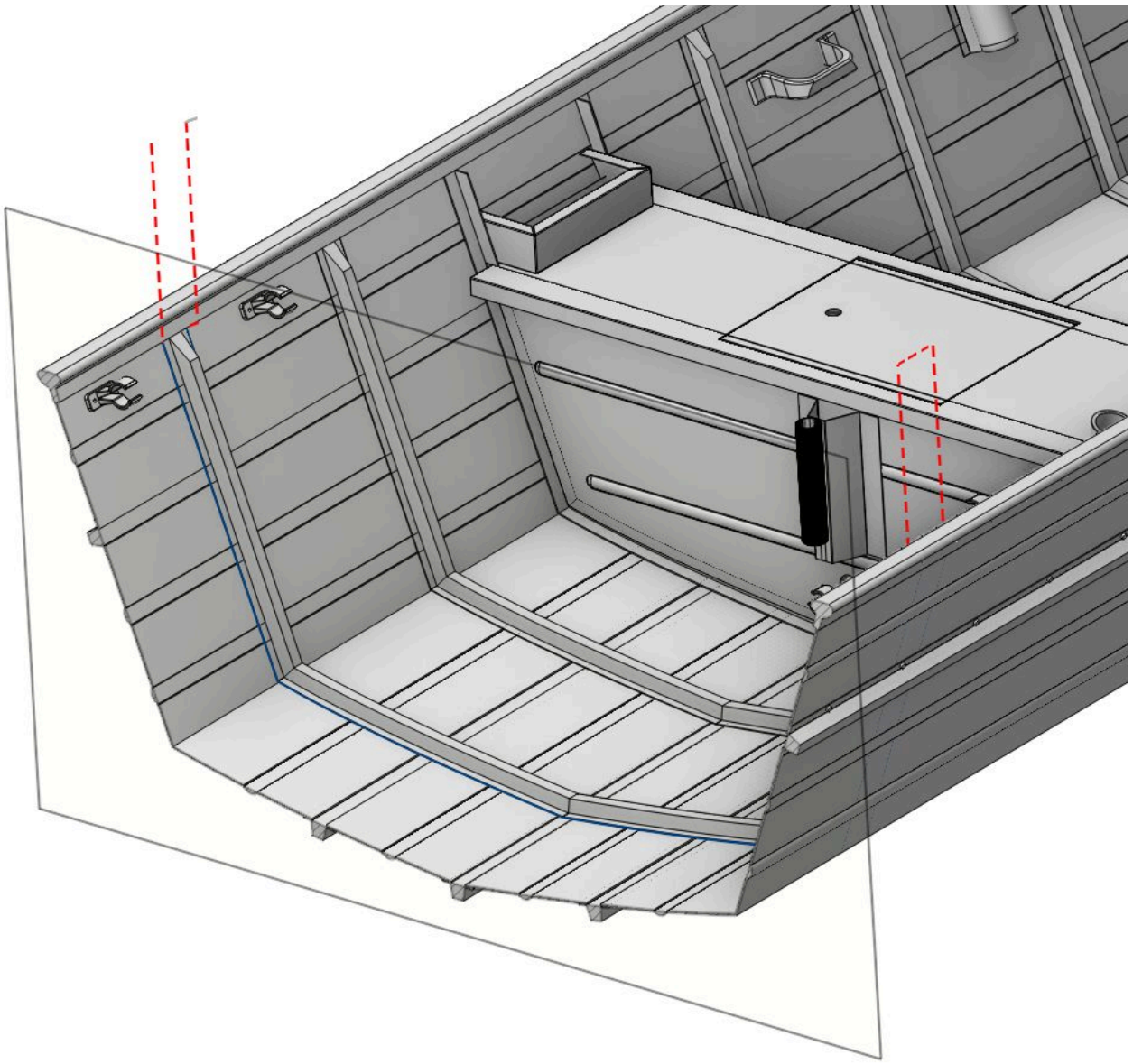


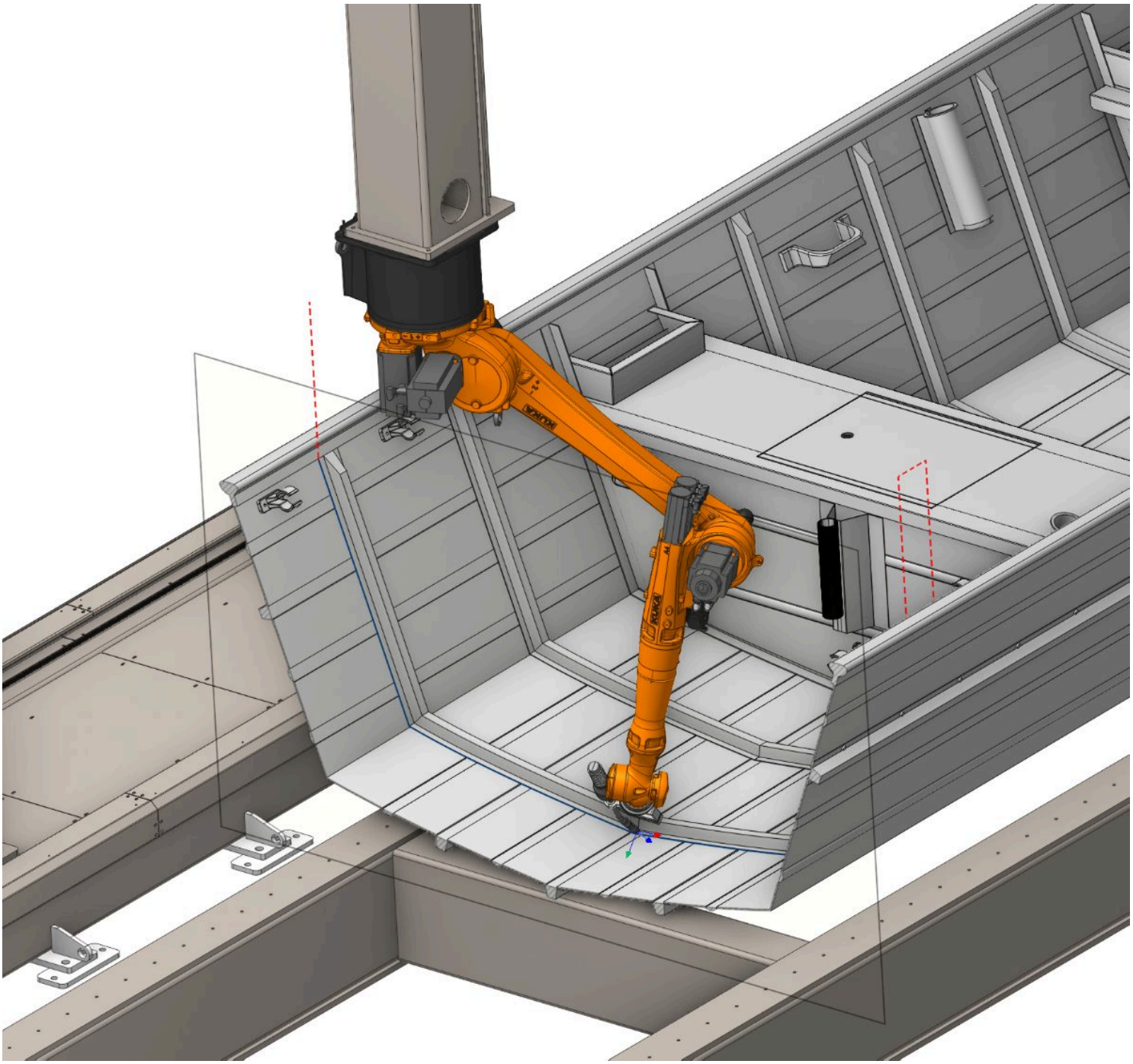
- Circle(Radius(7.2))
- KeyHole(Width(7.11), Head Radius(4.4), Neck Radius(2.7))
- Polygon(Width(7.27))
- Polygon(Width(8.43))
- Polygon(Width(16.63))
- Rect(Width(25), Height(14.2), Radius(0))
- Rect(Width(31.4), Height(17.6), Radius(1.4))
- Slot(Width(22), Height(5.6))

The Smart Motion option has been improved.

Smart Motion enables robot trajectory generation for 5D and 6D points without requiring a robot axes map. The resulting trajectory stays within axis limits and includes singularity and collision avoidance. Additionally, the trajectory can be fine-tuned using the robot axes map.







The Feeds/Speeds calculator has been redesigned and enhanced.

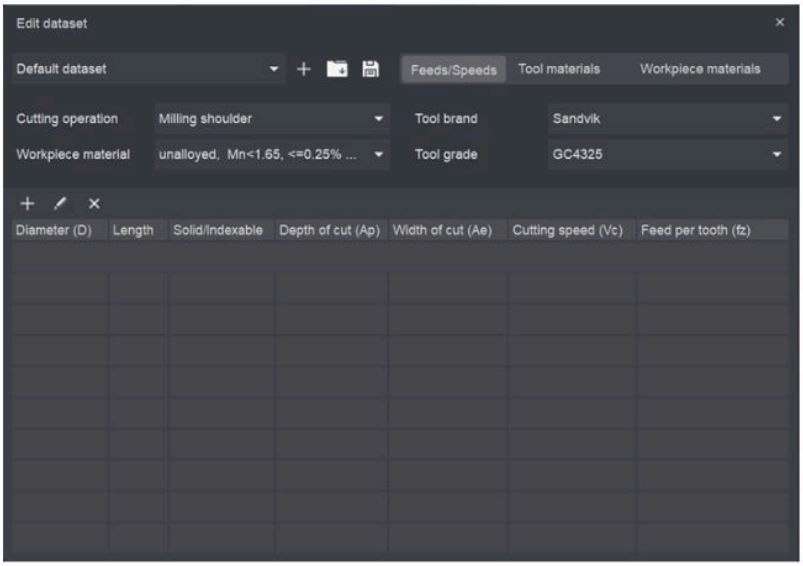
In previous software versions, the calculator provided basic automatic calculation of cutting conditions. For instance, users could input the cutting speed and feed per tooth to automatically derive the spindle speed and table feed rate. The new version of the calculator introduces a comprehensive suite of tools for automated cutting condition calculation. Details

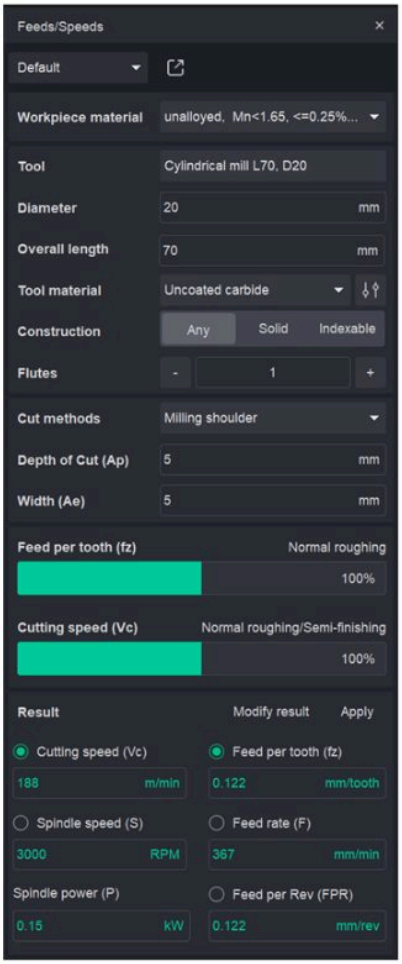
The system now accounts for critical machining parameters, including:

- tool type (Solid or Indexable);
- workpiece material;

- and other relevant factors.

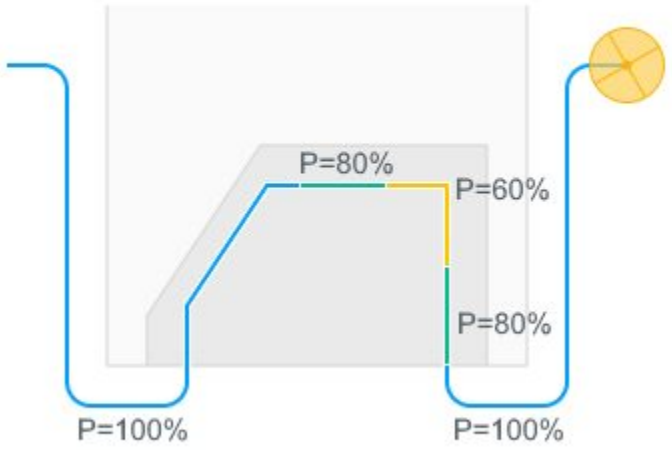
This enhanced functionality enables users to build and maintain a custom library of cutting conditions. By default, calculations are performed using established industry databases from leading manufacturers: Walter, Kennametal, and Sandvik.





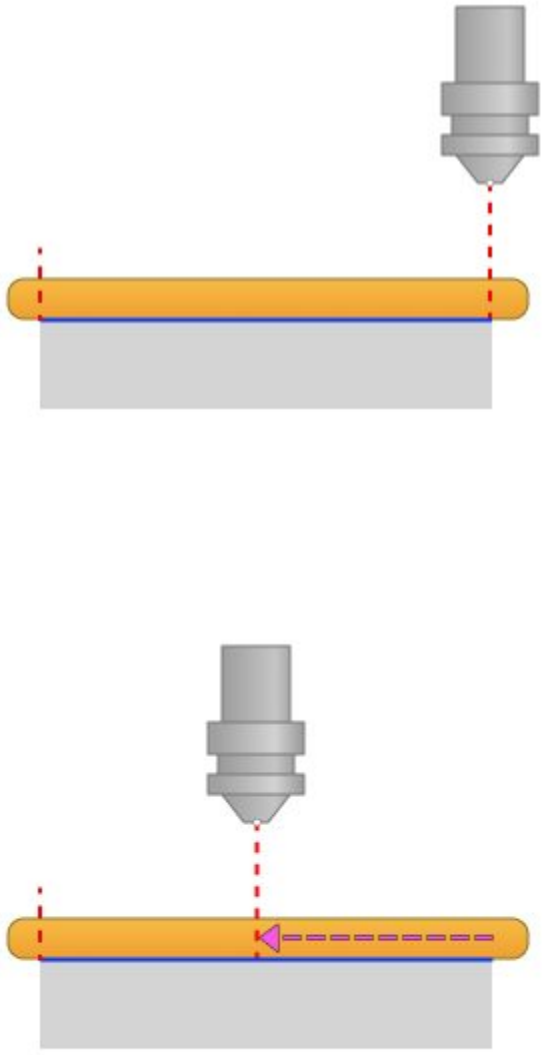
Improvings in Corner control.

Added new Action Type options: Adjust Feed, Adjust Power, and Adjust Feed & Power. Feed adjustment now supports values above 100%, enabling corner acceleration. Power can be controlled independently or alongside feed. Details



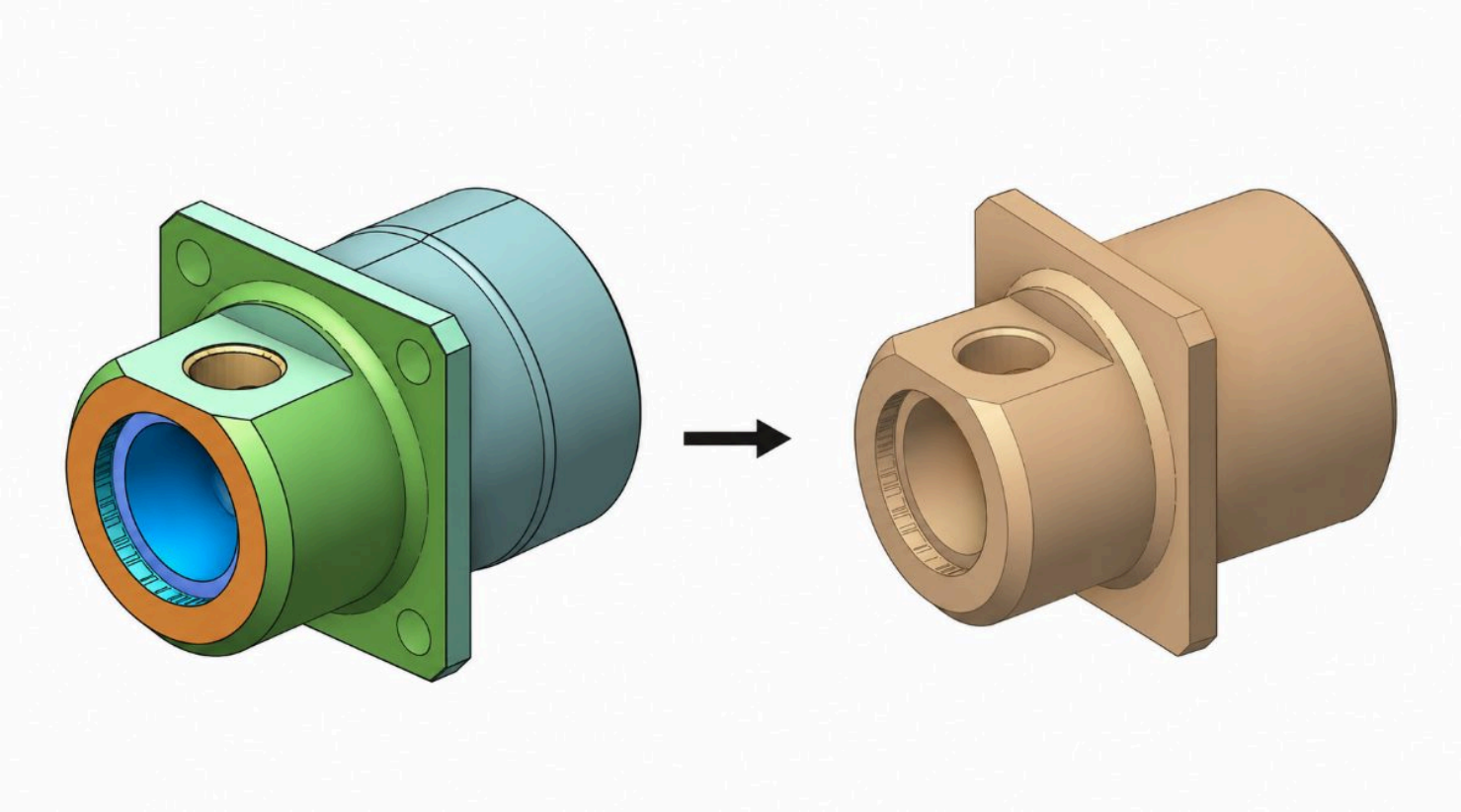
Pullback option has been added to the Welding 6D.

Available for several welding types. When enabled, the tool travels back along the finished weld by a configurable distance before it lifts off, so residual energy stays over already-welded material and does not mark the clean surface — useful for laser welding, where the beam may not switch off instantly. Details



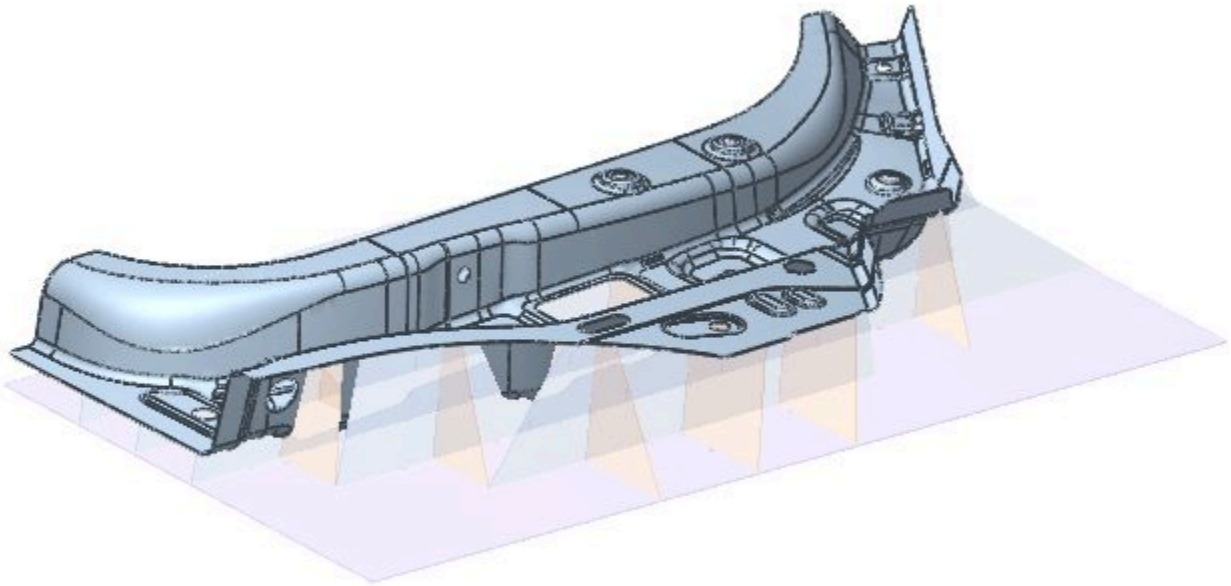
New precise format for exporting simulation result

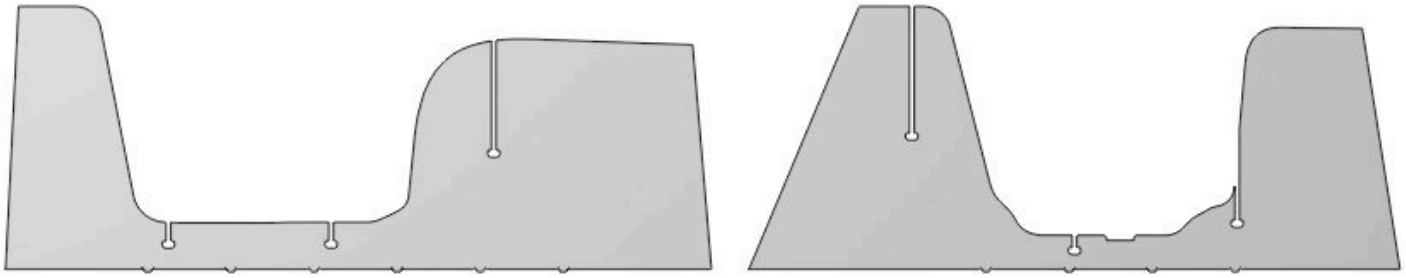
Simulation results can now be saved in the new precise.camsr format and loaded into another project as a workpiece, preserving the exact model instead of losing detail through STL. Find it in the simulation result export dialog and via the new Sim result button on the Workpiece tab. Details



Support Generator has been added.

Support Generator is a new extension that builds custom fixtures for holding parts during machining, straight from a 3D model. Choose a build plane, tune the rib grid interactively, and watch the fixture take shape in a live preview. A single click turns it into ready-to-manufacture geometry — complete with the flat-sheet nesting layout for laser cutting. Available in the ENCY Extension Store.





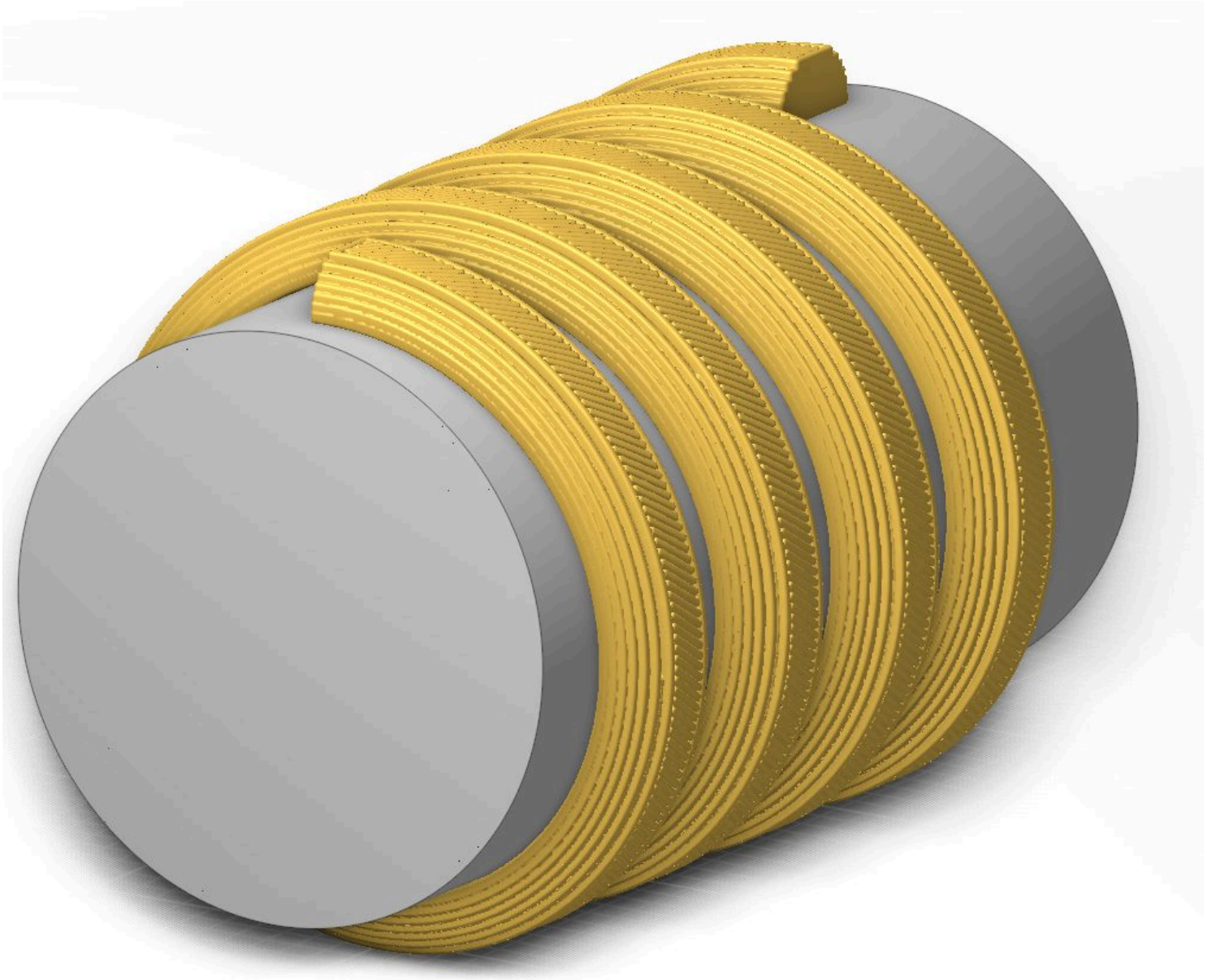
Additive manufacturing updates

Improvings in Non-planar slicing operation

Radial slicing strategy

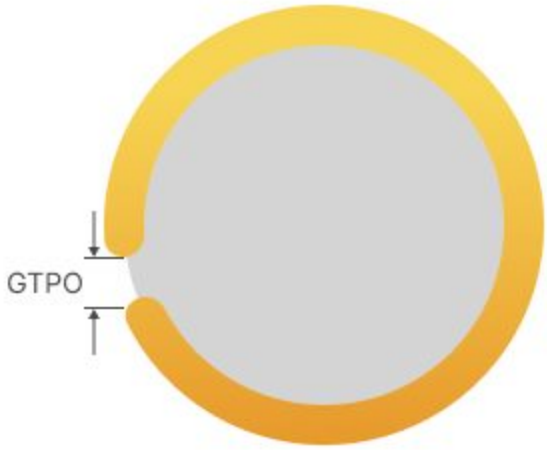
Added a new Radial slicing strategy — slicing is performed by cylindrical surfaces around a specified rotary axis with a constant radial step between layers. The strategy supports parts with any angular extent around the rotary axis:

- fully closed (360°) — rings, cylindrical sleeves, shells;
- with an angular gap (<360°) — turbine blades, propellers, impellers;
- helical (>360°) — augers, spiral parts.



Gap to prevent overlapping

In a Non-planar slicing operation, you can now specify a gap at the seam of closed contours to prevent line overlap, using Gap to prevent overlapping. Details



GTPO - Gap to prevent overlapping.

Custom angle sequence for infill.

Added the By sequence of angles option to Alternate lines direction, which lets you define a custom sequence of angles for infill direction across layers. Details

Example: (0,45,90,135) means layer 1 = 0°, layer 2 = 45°, layer 3 = 90°, layer 4 = 135°, layer 5 = 0° and so on:



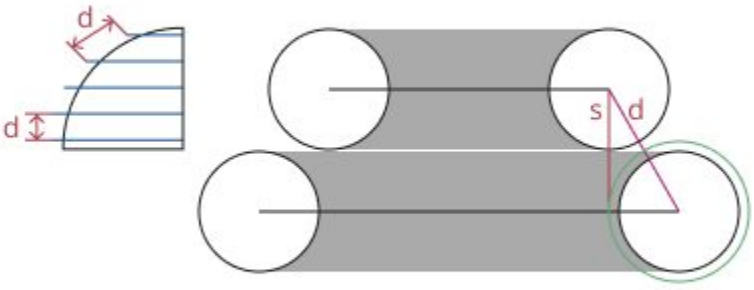
Zig Zag infill pattern.

Added a new Zig Zag infill pattern. Each model layer is divided into regions that can be filled with a continuous zigzag toolpath. A separate toolpath is generated for each region: a contour containing five regions produces five zigzag toolpaths, while a single-region contour produces one continuous toolpath. Details



Power function

Added a Power function option to the Non-planar slicing operation. It works the same as in the Cladding 5D operation. Details



Maximum layer time

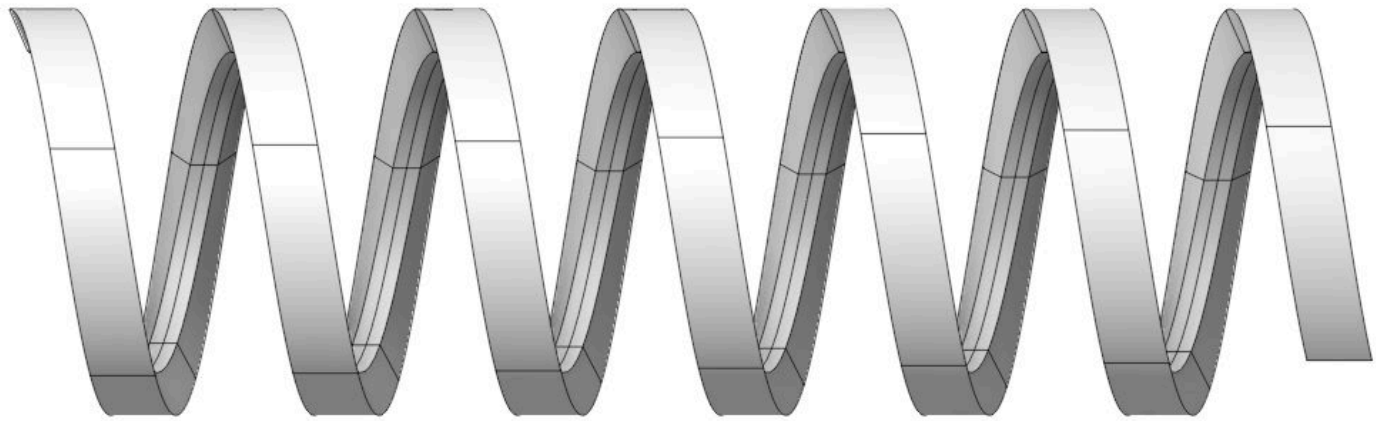
Added a Maximum layer time parameter for additive operations that automatically scales up the working feed when a layer takes too long — useful for concrete 3D printing to avoid cold joints. Details

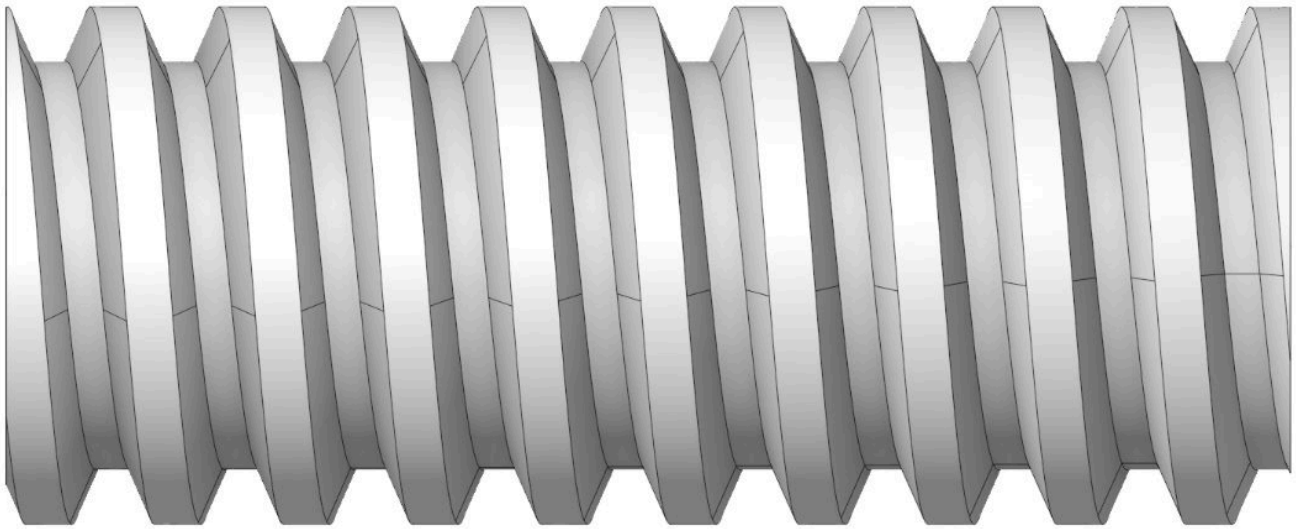
Design module enhancements

Design in 3D mode.

Added a Coil.

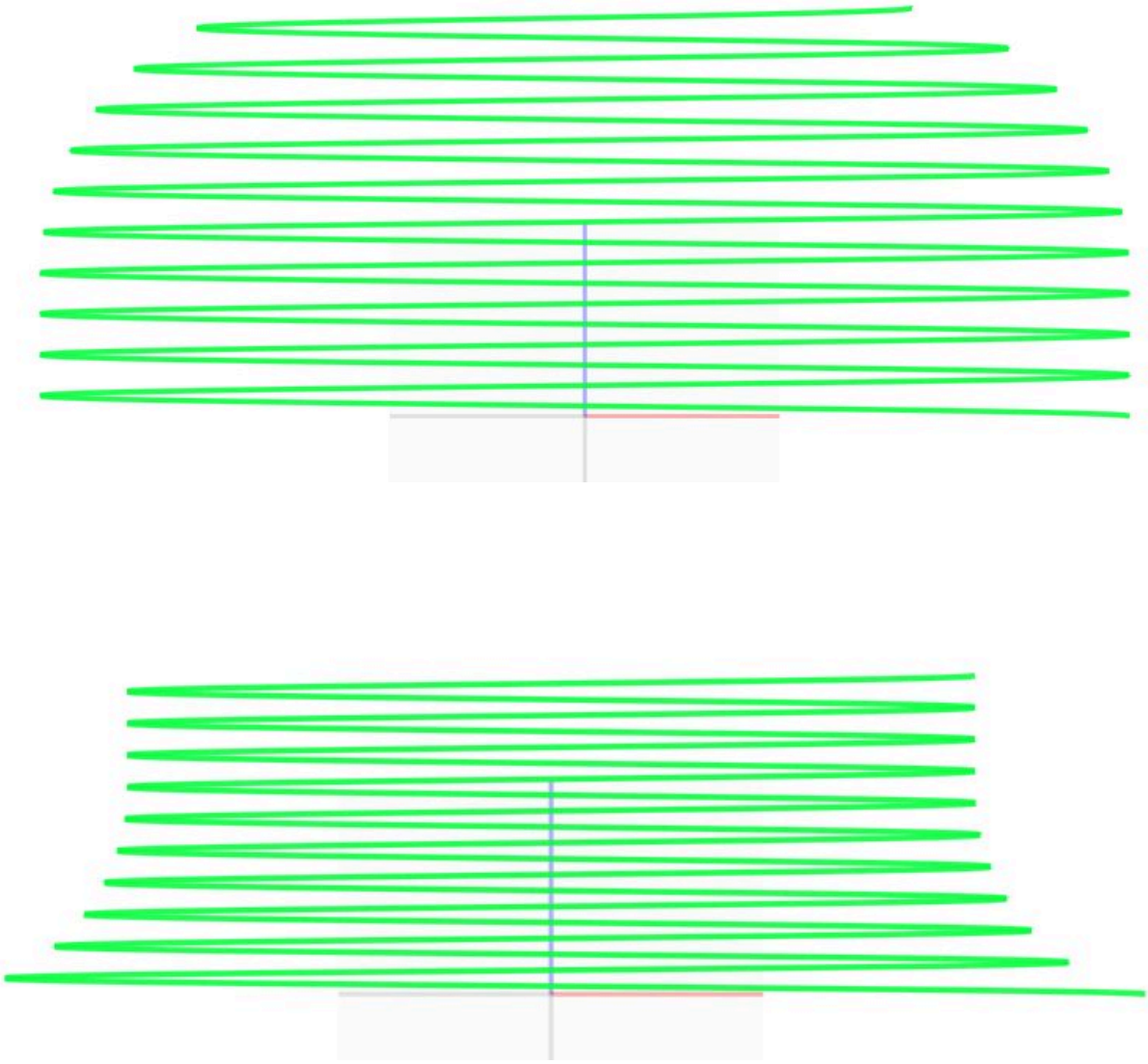
Generation of a 3D solid by extruding a sketch along a helical trajectory (unlike the Sweep operation, explicit creation of a helical guide curve is not required). Details



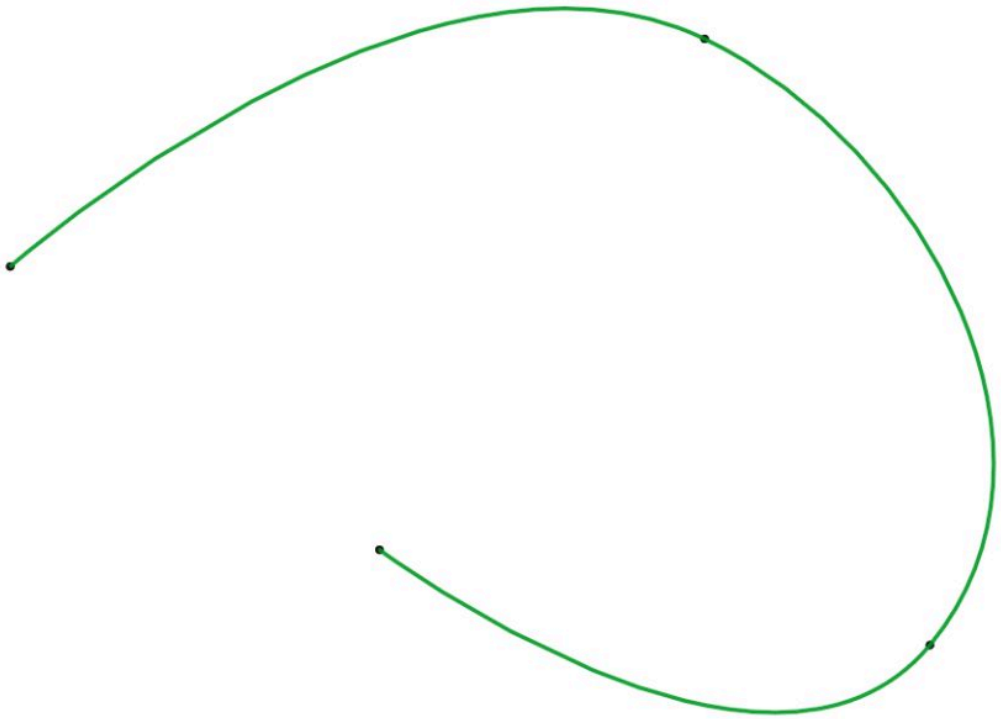


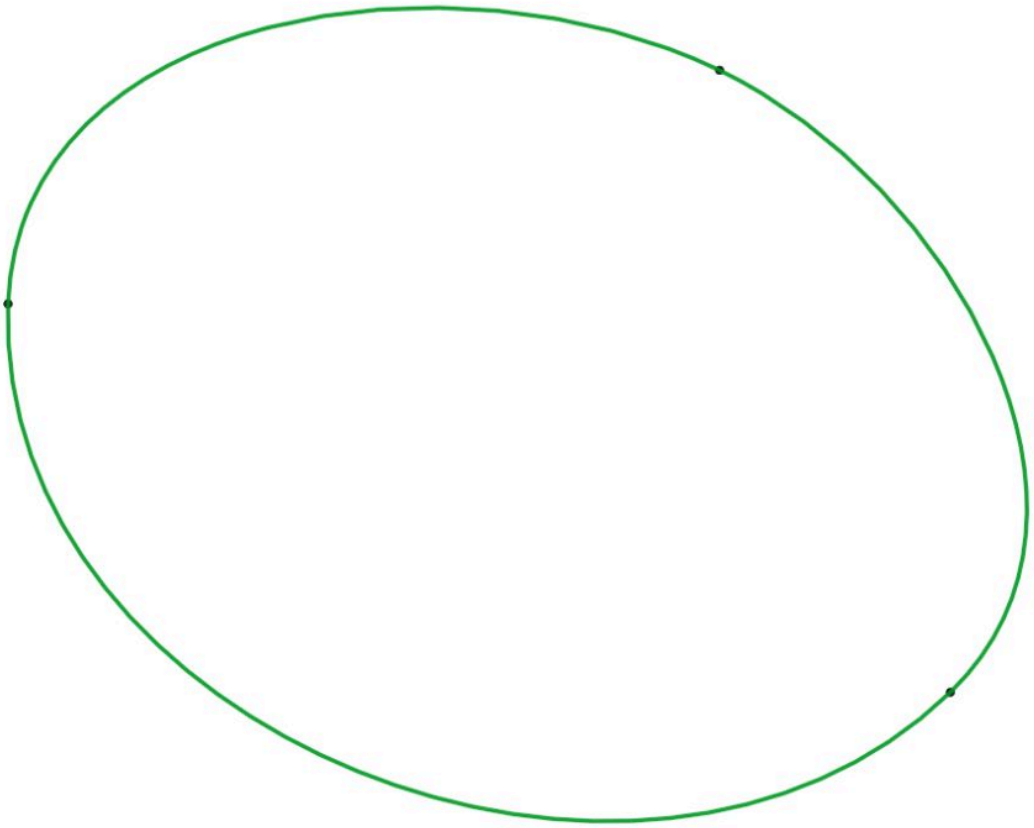
A new option has been added for Helic: Start Runout and End Runout.

Radius Exit for the Start diameter / End diameters. Details



A new option has been added for splines: Closed.

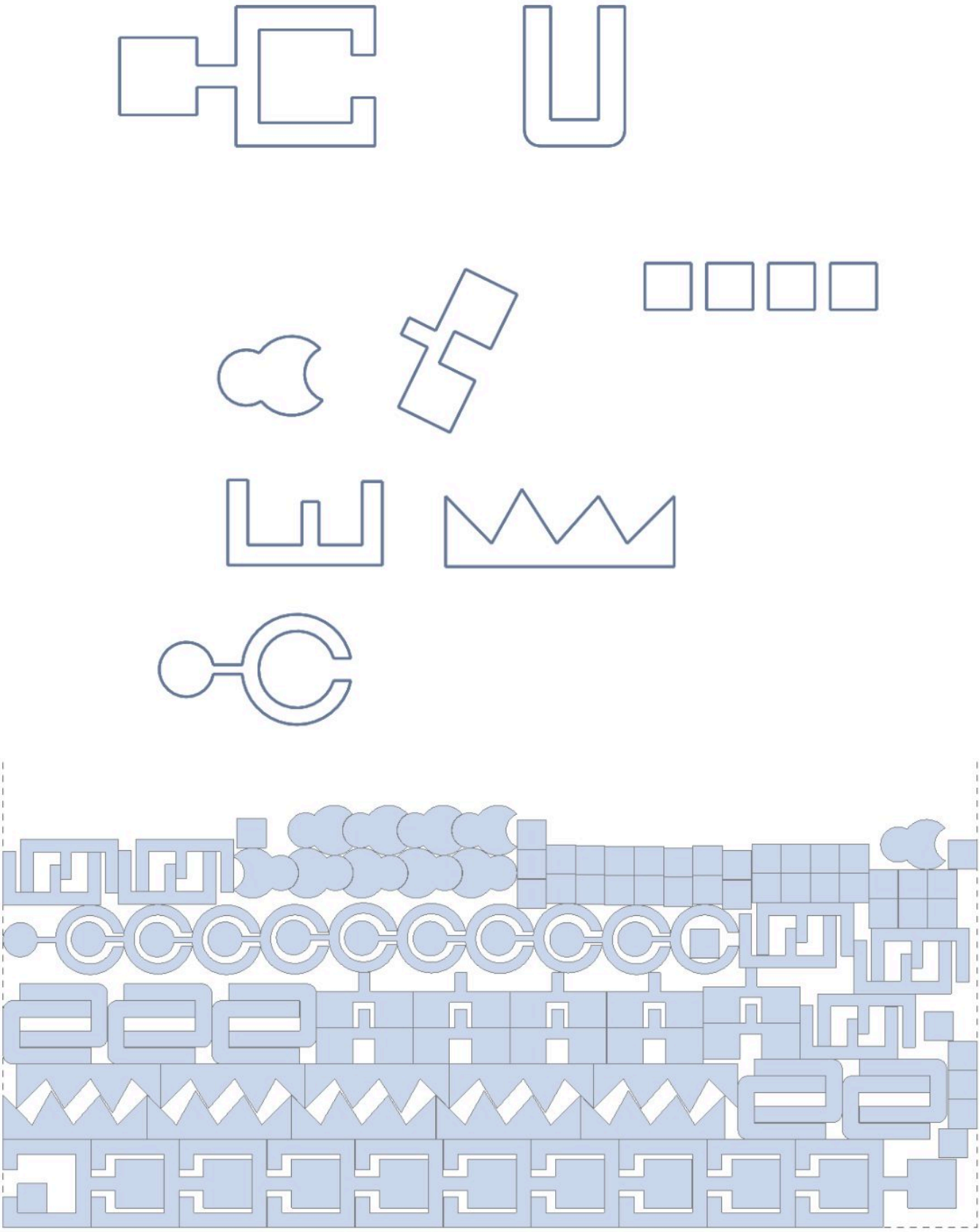




New Nesting Functionality

The new Nesting tool is available from the Model tab. Right-click a sheet and select Create nesting to automatically arrange selected parts and optimize material usage.

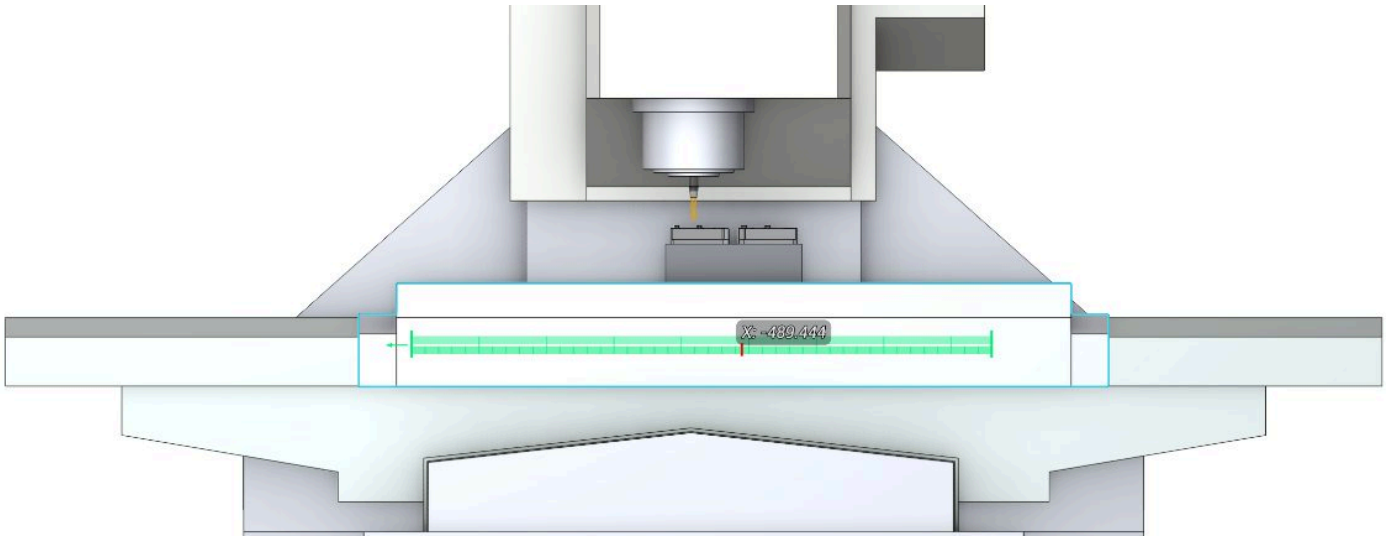
The Nesting tool automatically arranges selected parts on a sheet to optimize material usage and reduce waste. Configure sheet dimensions, spacing, margins, part copies, rotation, mirroring, grain direction, and placement rules before generating the layout. Details



3D View Controls

Added an interactive linear axis ruler.

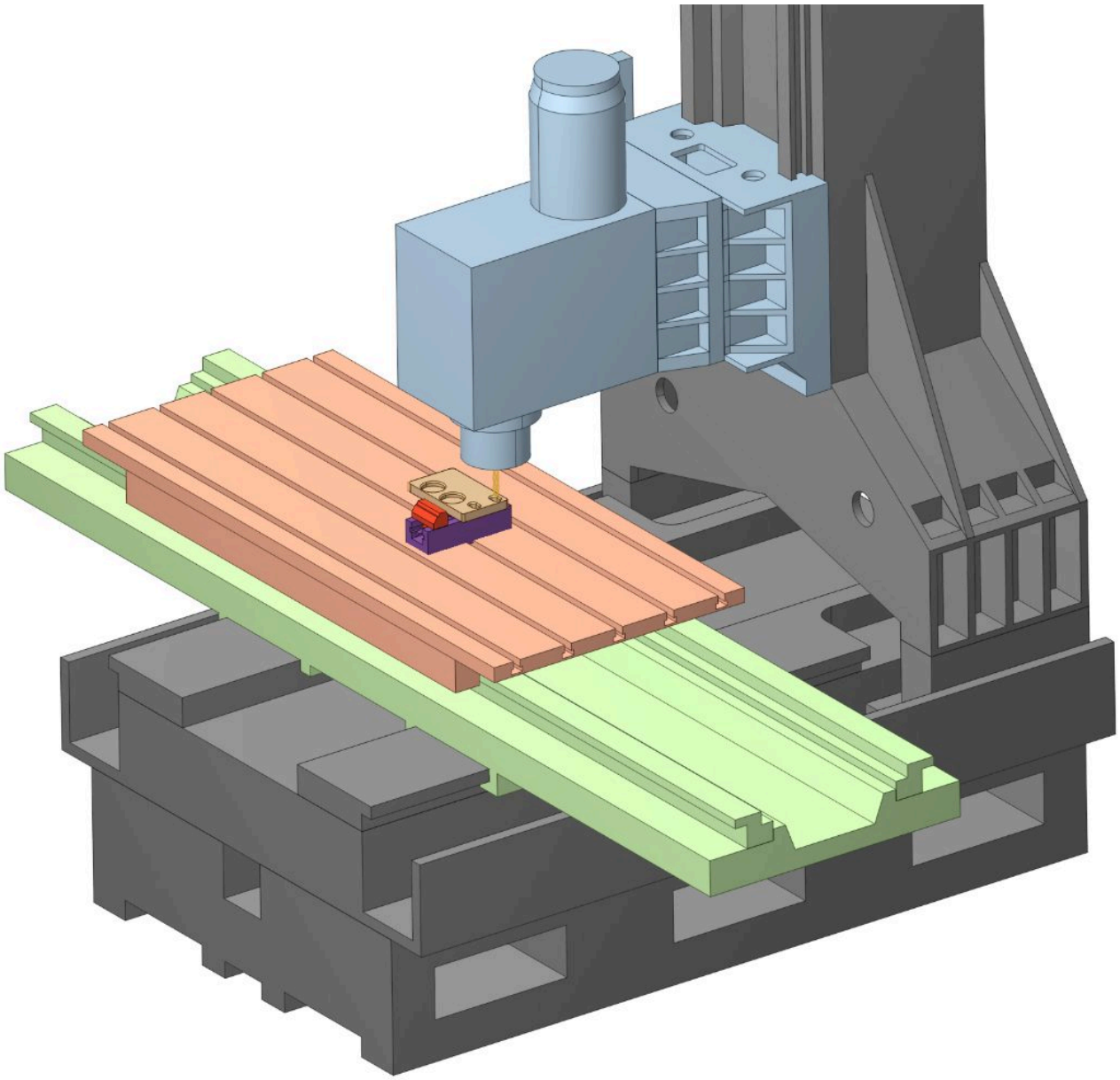
It appears when you drag a machine node along a linear axis in the 3D view. The ruler displays the current axis coordinate, its travel limits, and provides precise position control. Details

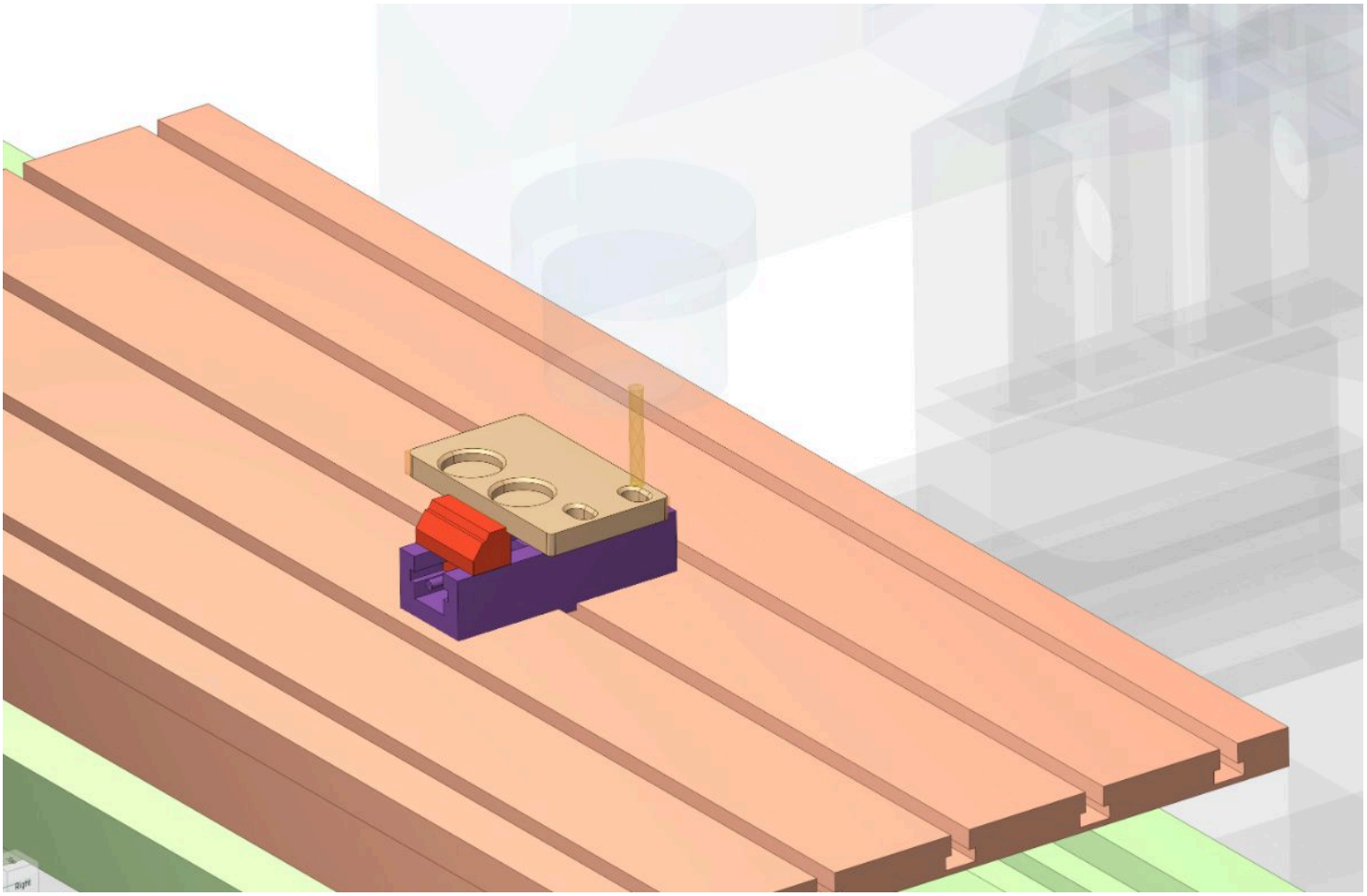


Updated Machine Display Settings in the 3D view.

The Machine Visibility Settings popup helps keep the working area visible and convenient during setup and simulation. It lets you select a machine surface material, such as Gunmetal or Industrial, and keep a selected machine component visually stationary while the remaining components move relative to it. You can adjust the opacity of machine nodes to reveal internal mechanisms or the machining area, and control the visibility of individual nodes in a tree where hovering over a node highlights it in the 3D view. Details

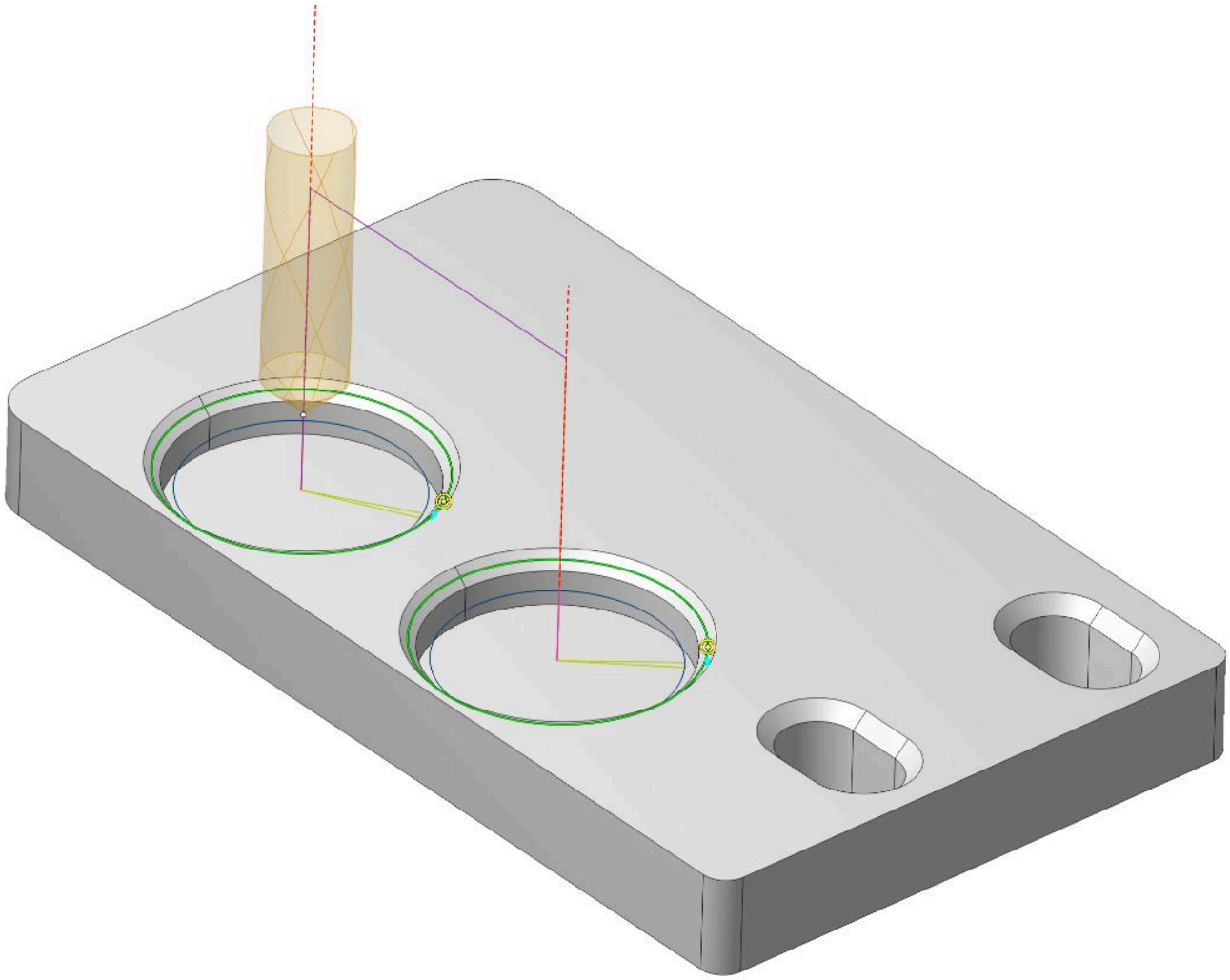
Interactive Machine is available when Expert Tools (Additional tab of the settings) is enabled. Details

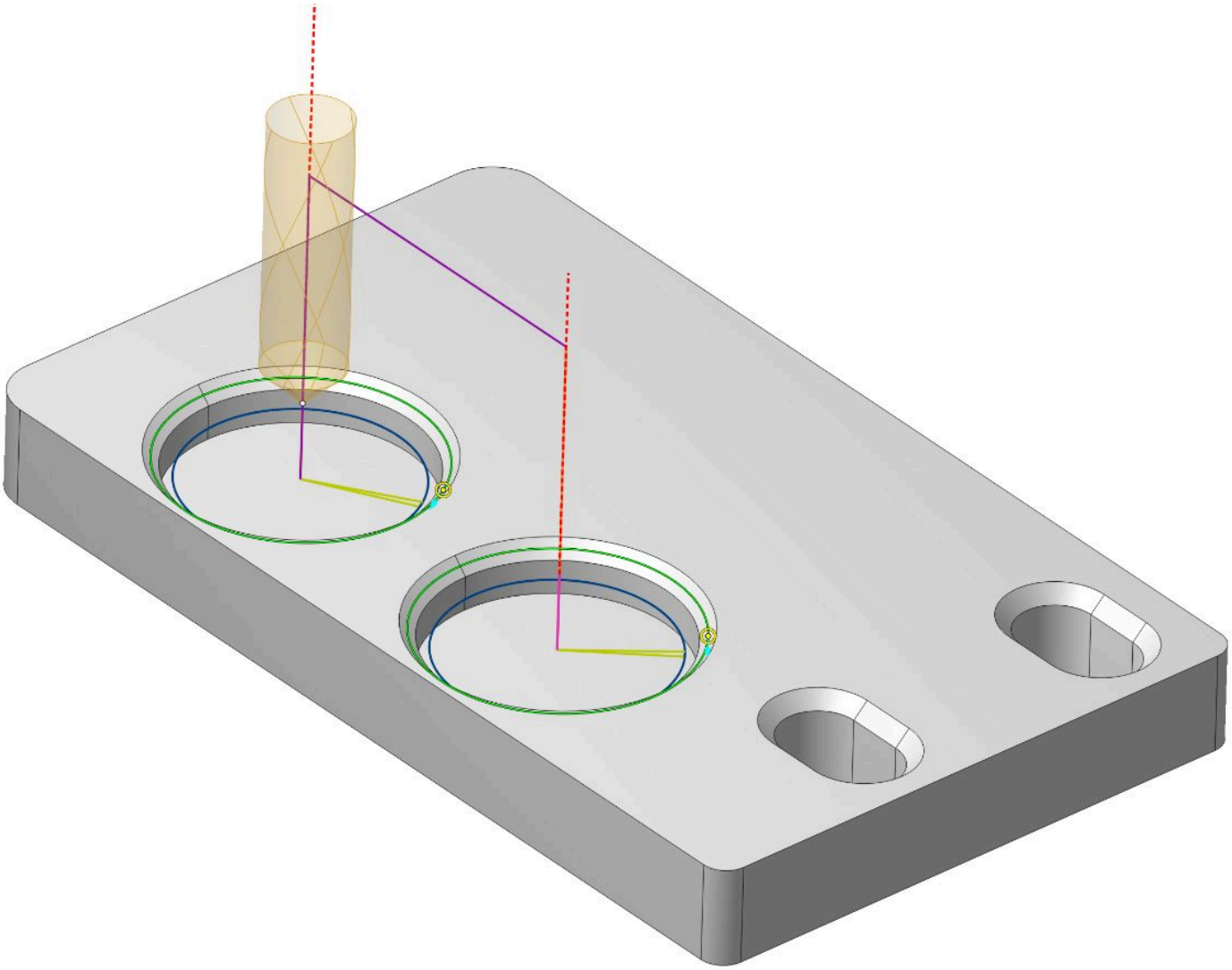




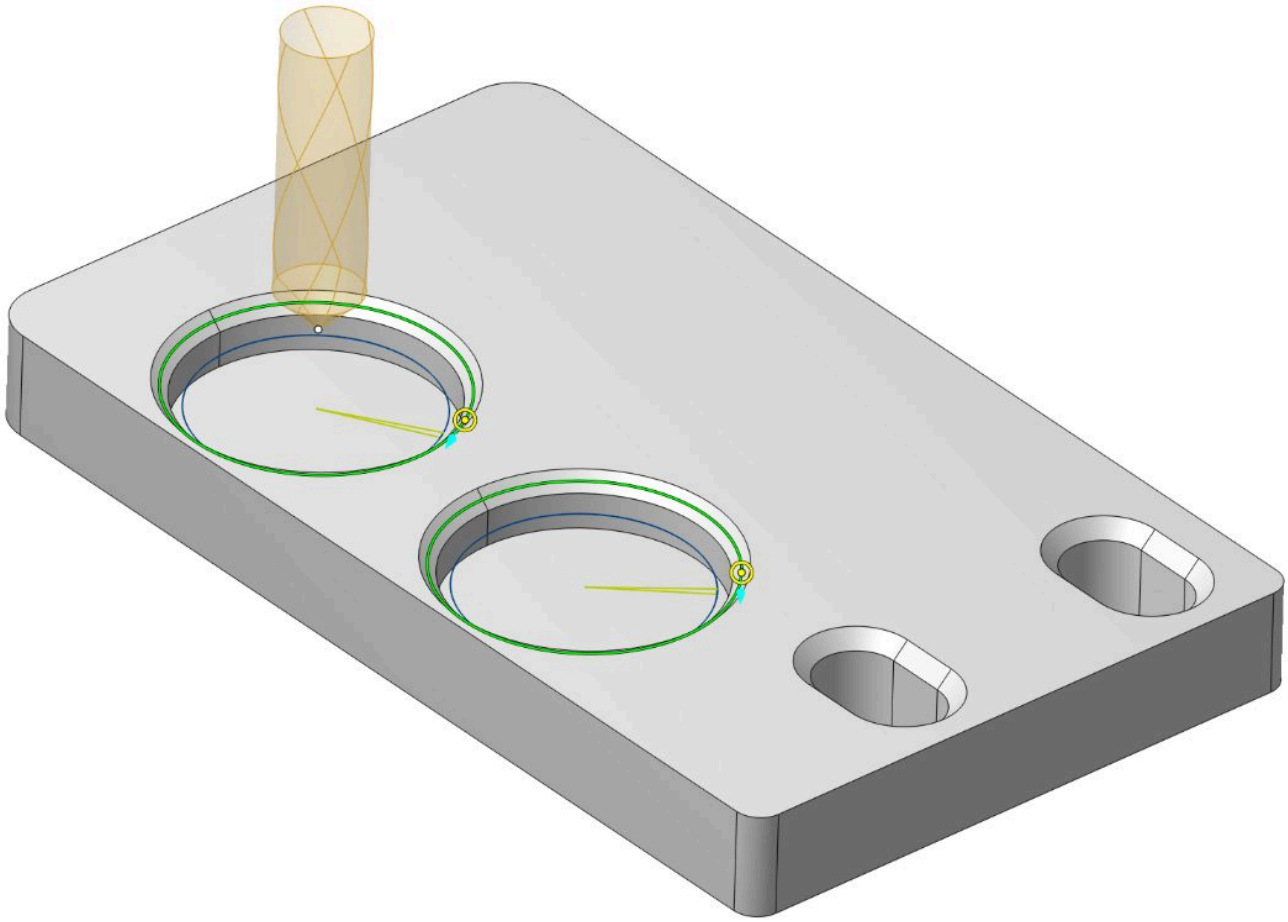
Updated Toolpath Display Settings in the 3D view.

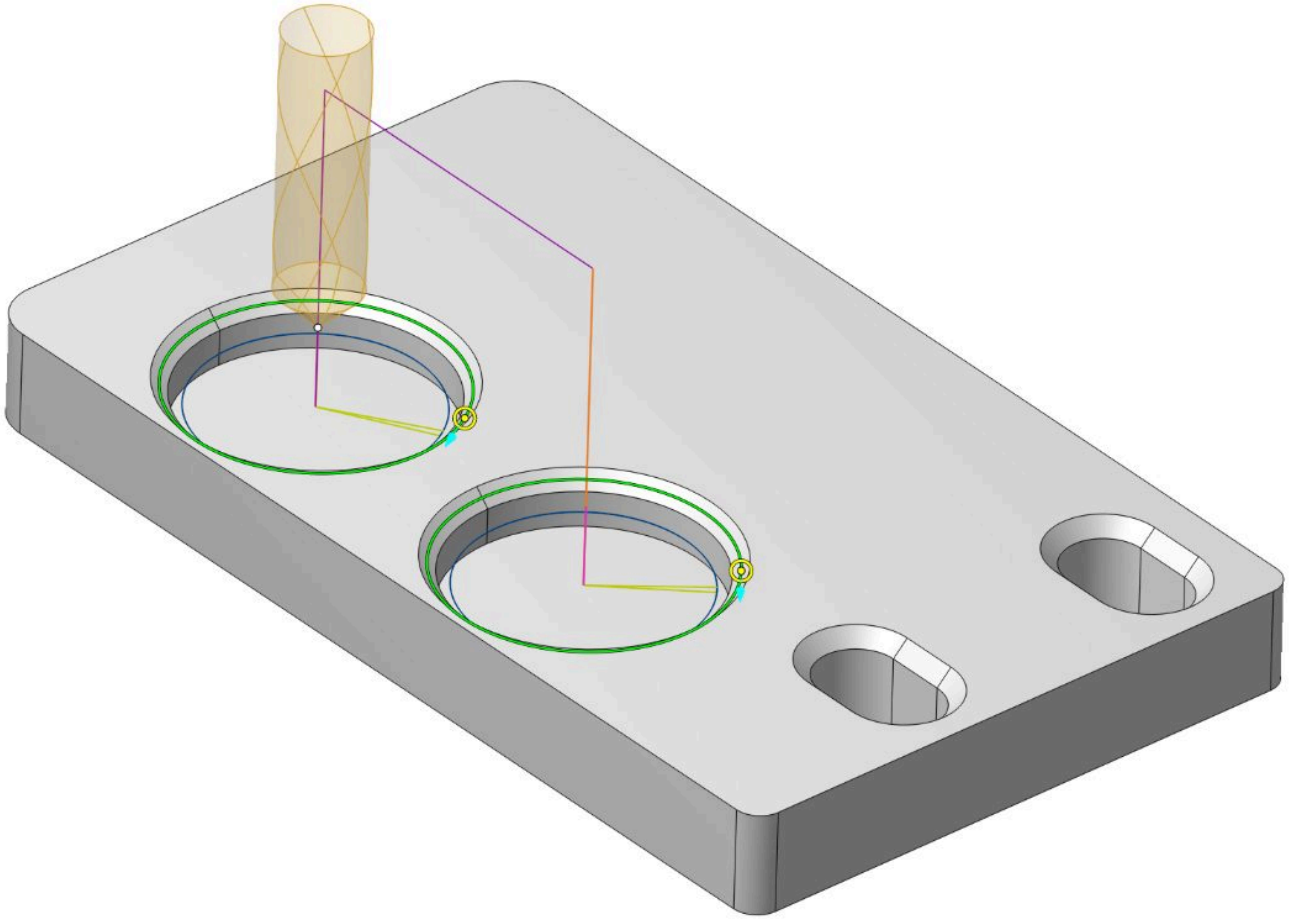
The new Toolpath Display Settings popup makes the machining trajectory easier to inspect during preparation and simulation. You can enable feed-type coloring or use a single toolpath color, and adjust the line width to keep the trajectory readable at any zoom level. Details

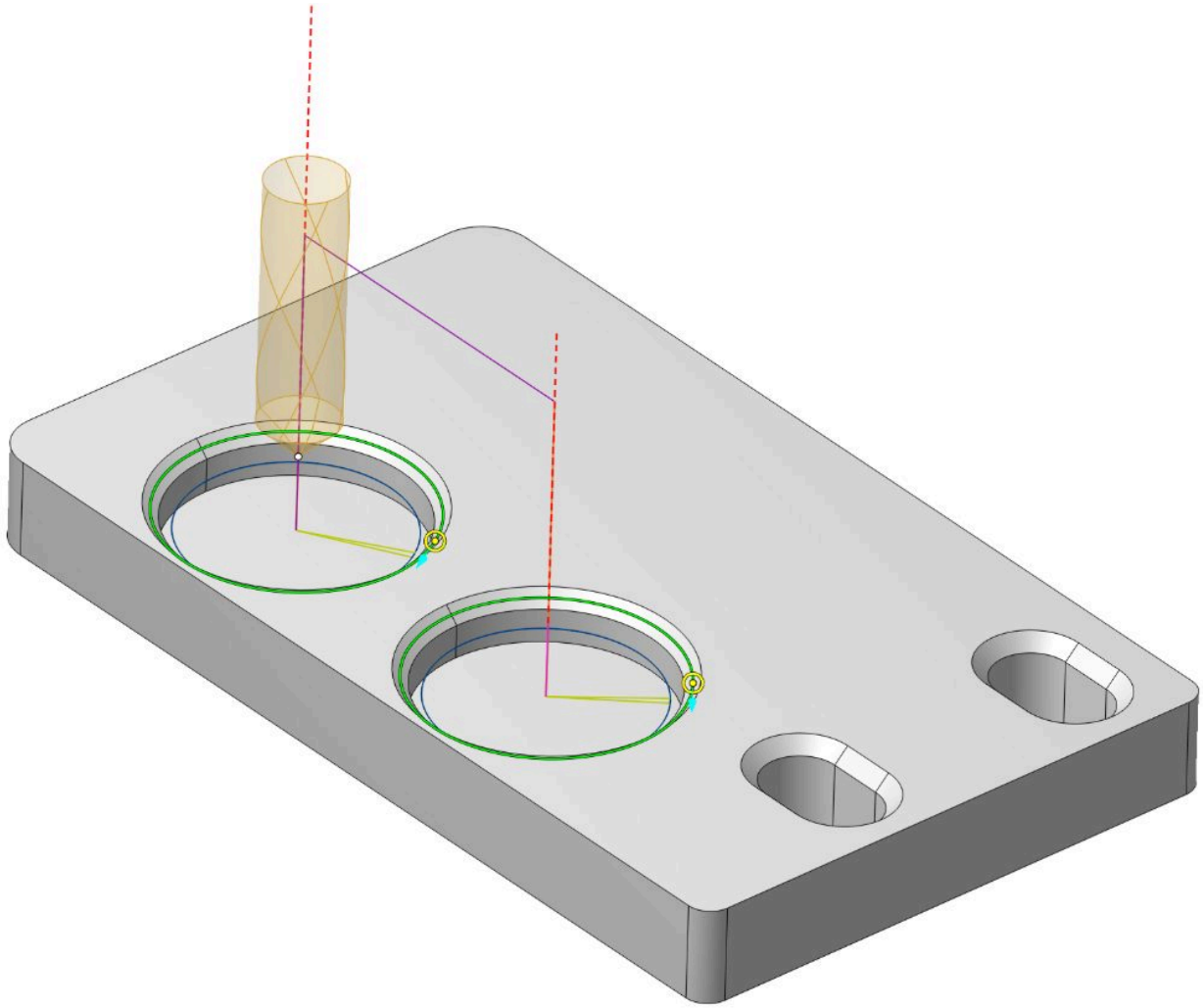




During simulation, select which part of the current operation's toolpath is shown: the full trajectory, the path already completed, the remaining path, or a configurable segment before, after, or around the current tool position. The rest of the program remains visible as a ghost layer. You can also hide rapid, approach/return, and link moves to focus on cutting passes. Surface normals and toolpath point markers can be displayed to verify the orientation and structure of multi-axis toolpaths. These overlays follow both the selected move visibility settings and the current playback trace.



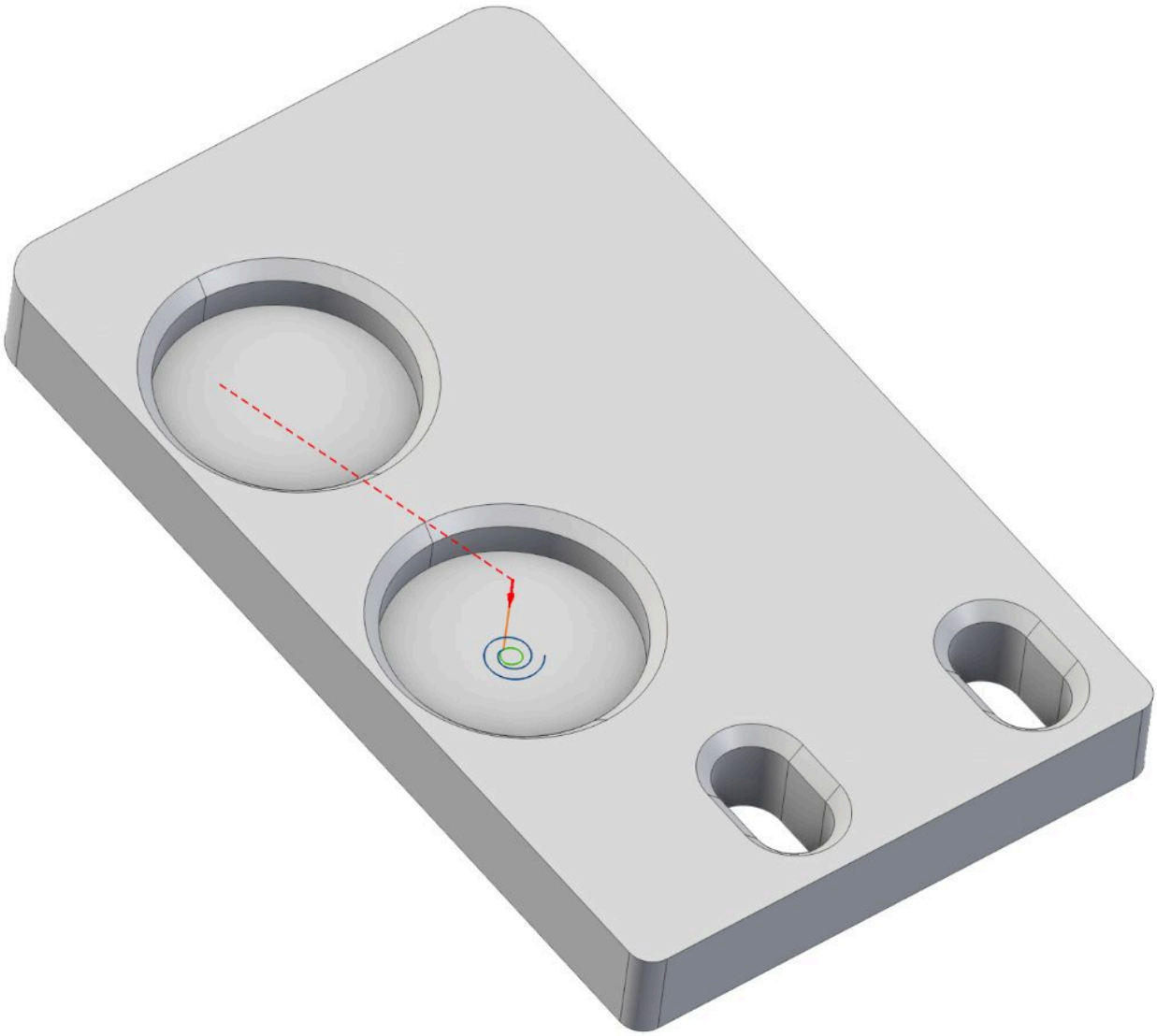


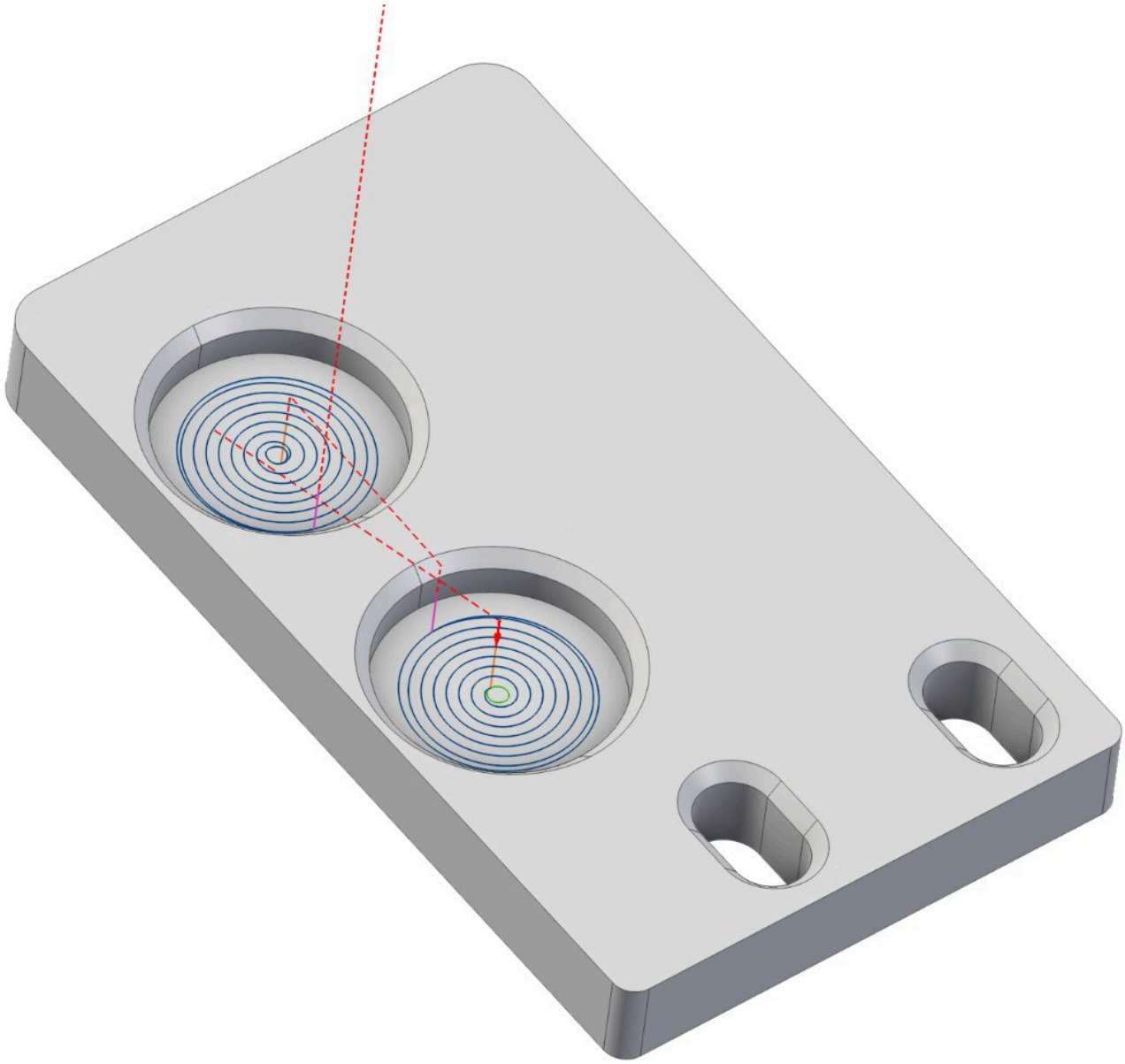


The Simulation tab now includes additional options for visualizing the toolpath. Configure which parts of the trajectory are displayed to focus on the current machining stage and inspect the operation more clearly.

- Current layer — shows the current toolpath layer.
- Full toolpath — displays the entire toolpath.
- From start — shows the toolpath from the beginning to the current tool position.
- To end — shows the remaining toolpath from the current position to the end.
- Tail — displays a limited toolpath segment around the current position.
- Tail ahead — displays a segment ahead of the current tool position.
- Tail both ways — displays segments before and after the current tool position.



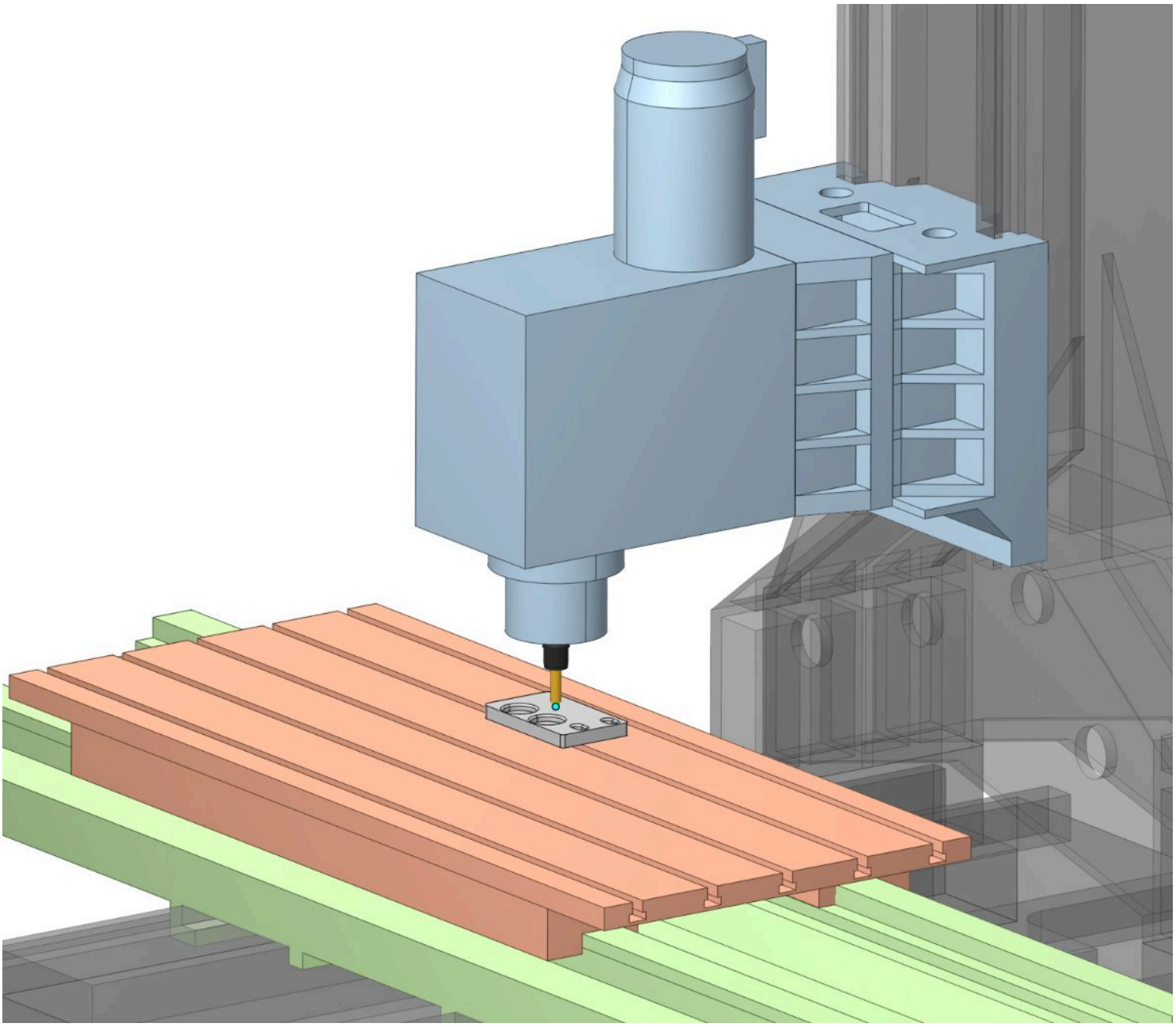


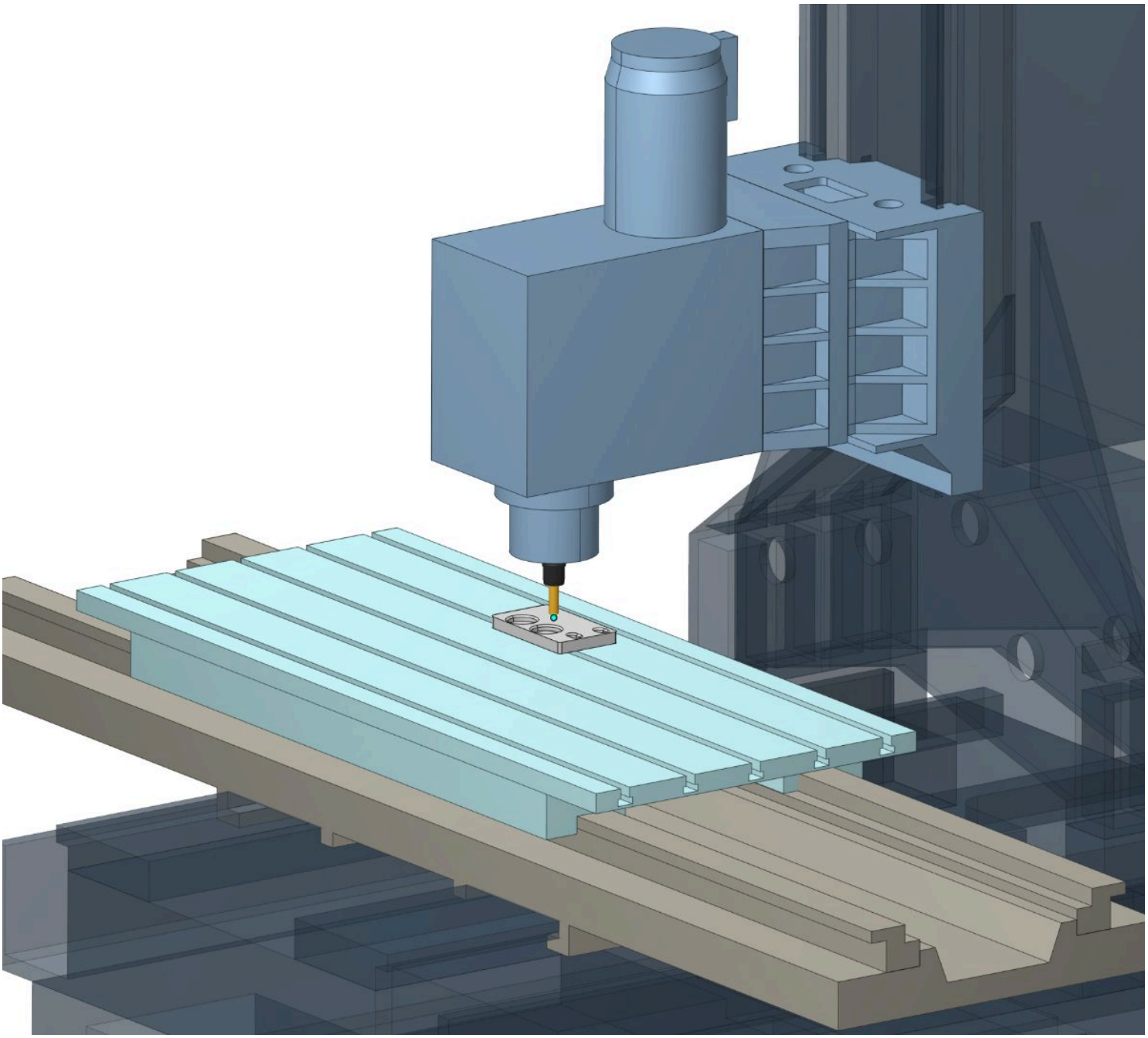


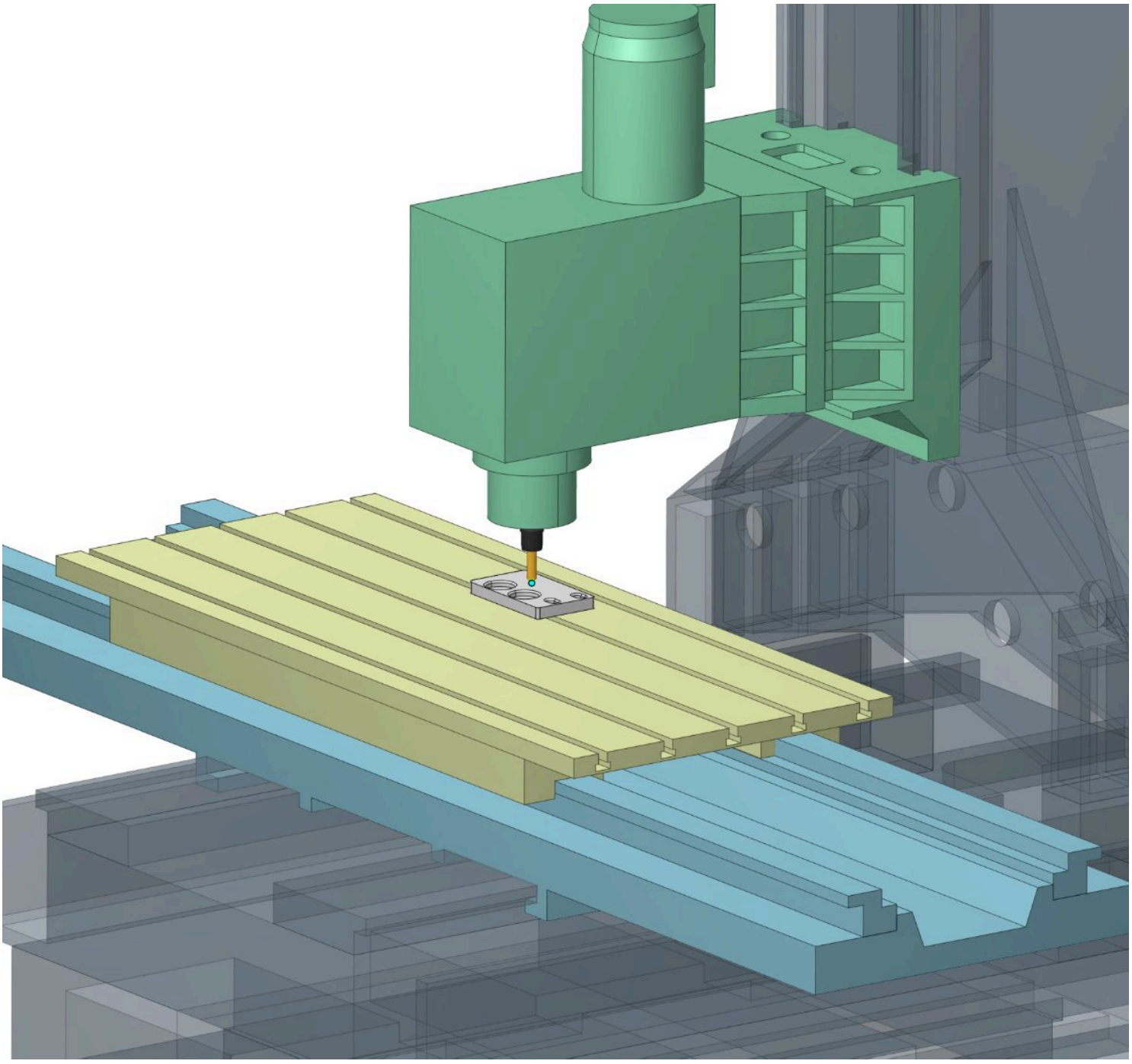
Updated Display Mode Settings in the 3D view.

The Display Mode popup provides quick control over the visual presentation of the 3D scene. You can switch between shaded, shaded-with-edges, and wireframe rendering modes to choose the most convenient way to inspect the machine, part, and surrounding geometry.

The popup also lets you enable ambient occlusion for improved depth perception and a ground plane for clearer scene orientation. Equipment color schemes can be applied to highlight the machine kinematics: each kinematic link receives a distinct contrasting color, while static components remain neutral. Project Colors preserves the original model colors, while Gunmetal and Restrained provide alternative industrial palettes. Robot manipulators retain their native manufacturer colors in every scheme. Details





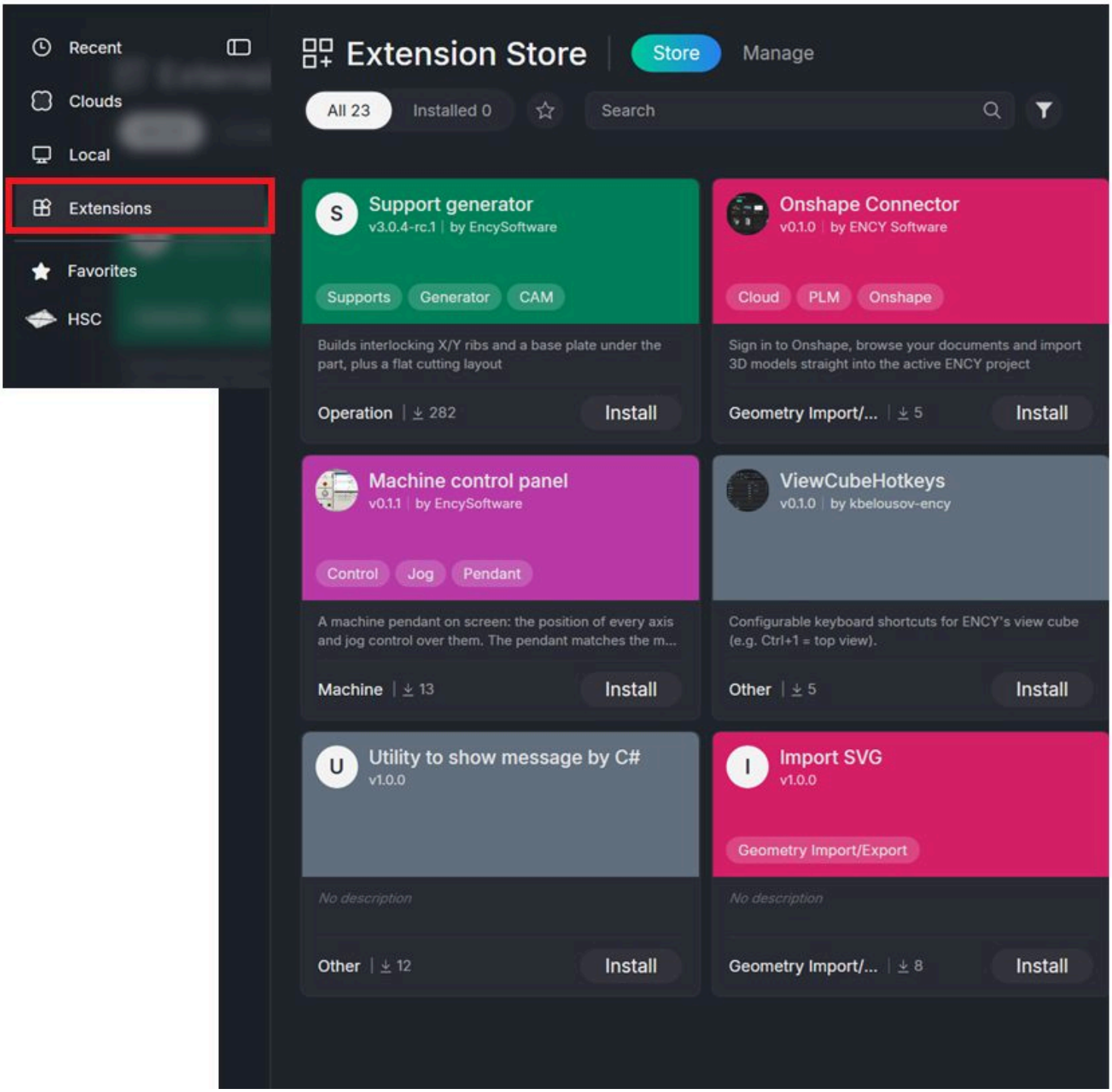


Other updates

Extension Store.

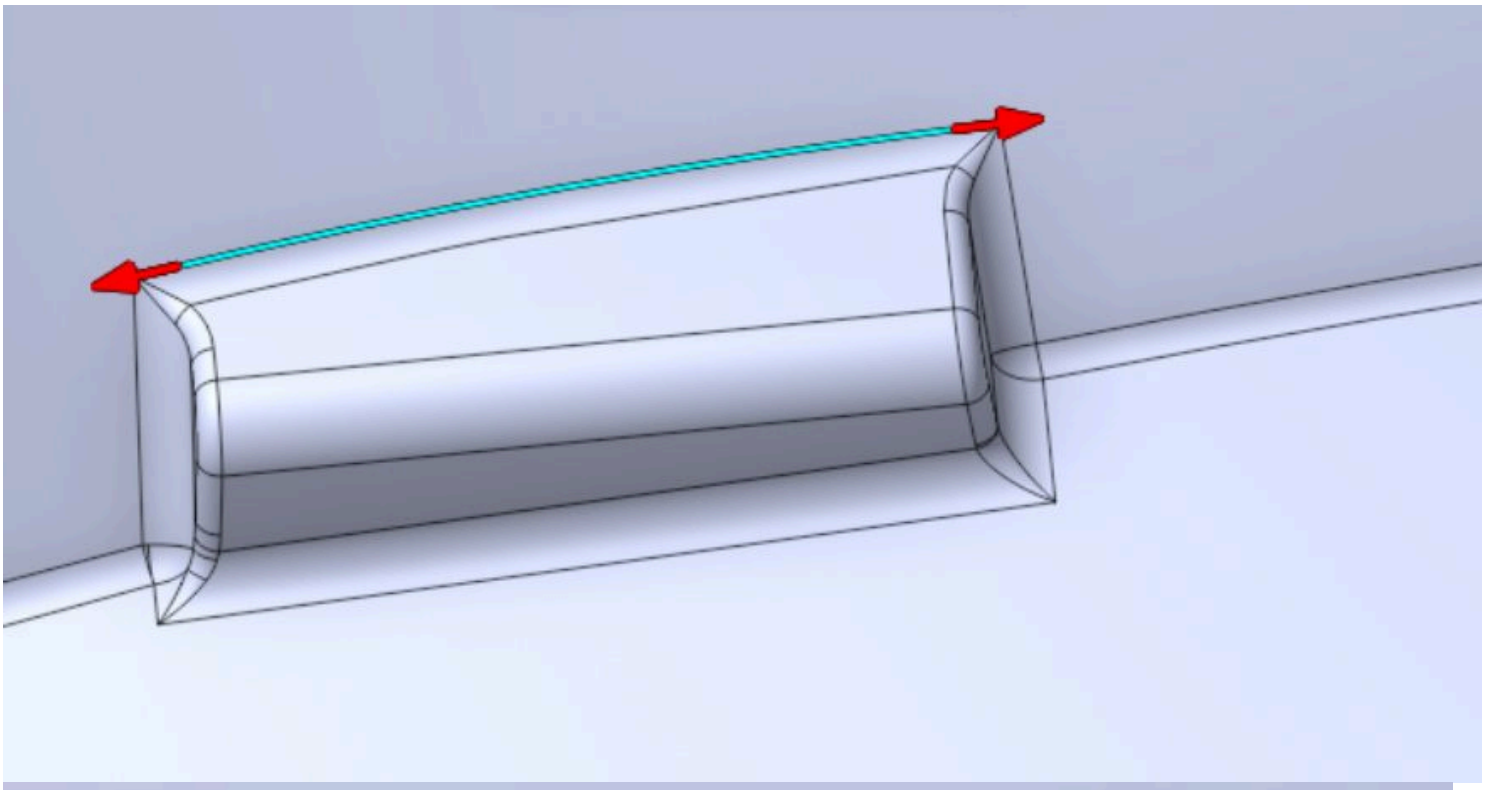
The Extension Store is the place where optional add-ons for the CAM system are discovered, installed, updated, and removed. It connects a web catalog with the CAM application: the web storefront lists extensions and can start an installation, while the in-application tab performs the actual download, registration, and removal on the local machine. Details

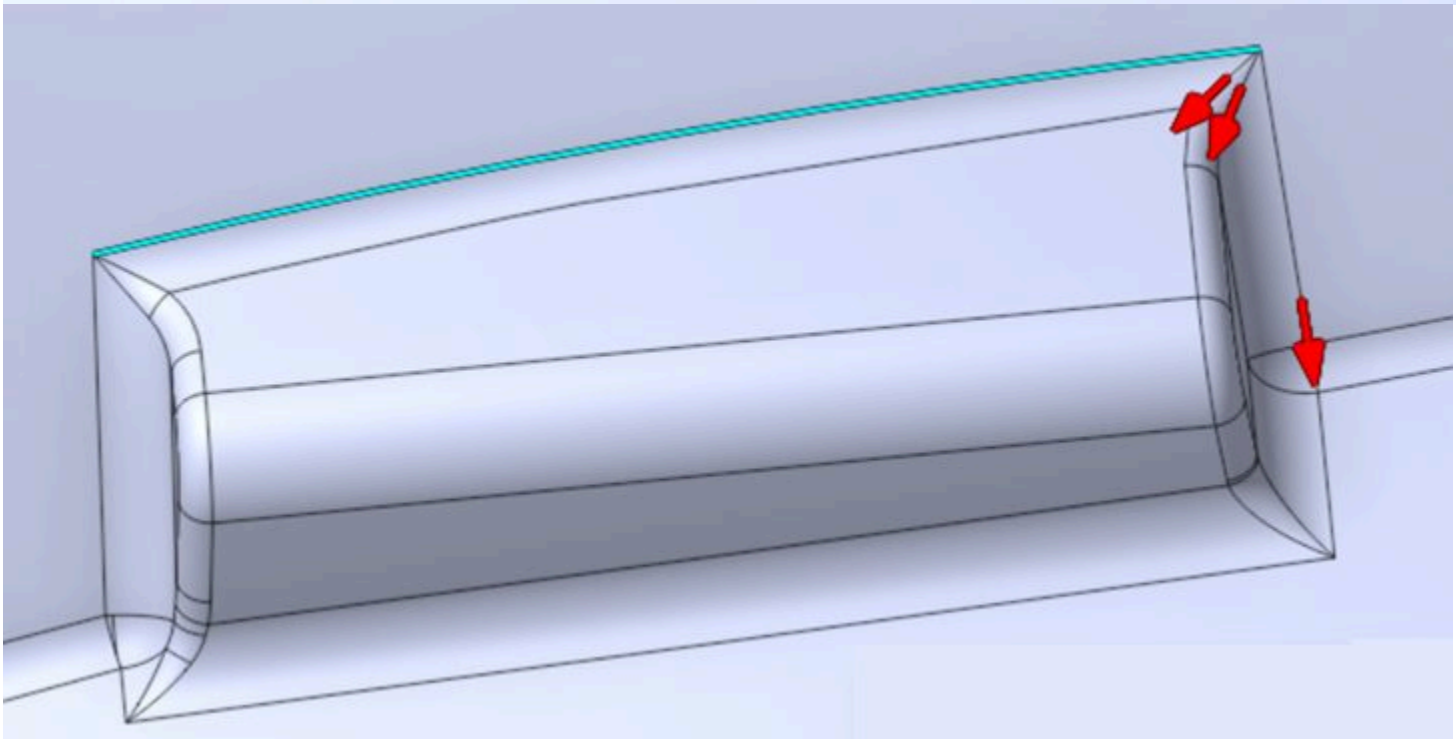
The Extension Store is located on the Home page.

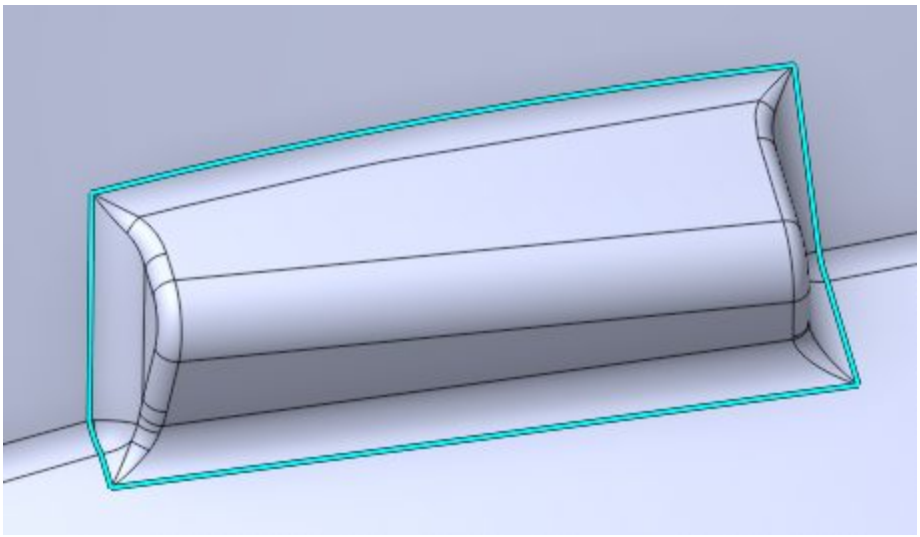


Added interactive fork steering for Loop edge selection - Custom Loop.

It lets the operator control, edge by edge, the direction the loop takes at every branch point (fork) it meets while propagating along the model, using the ◀ / ▶ keys. Details



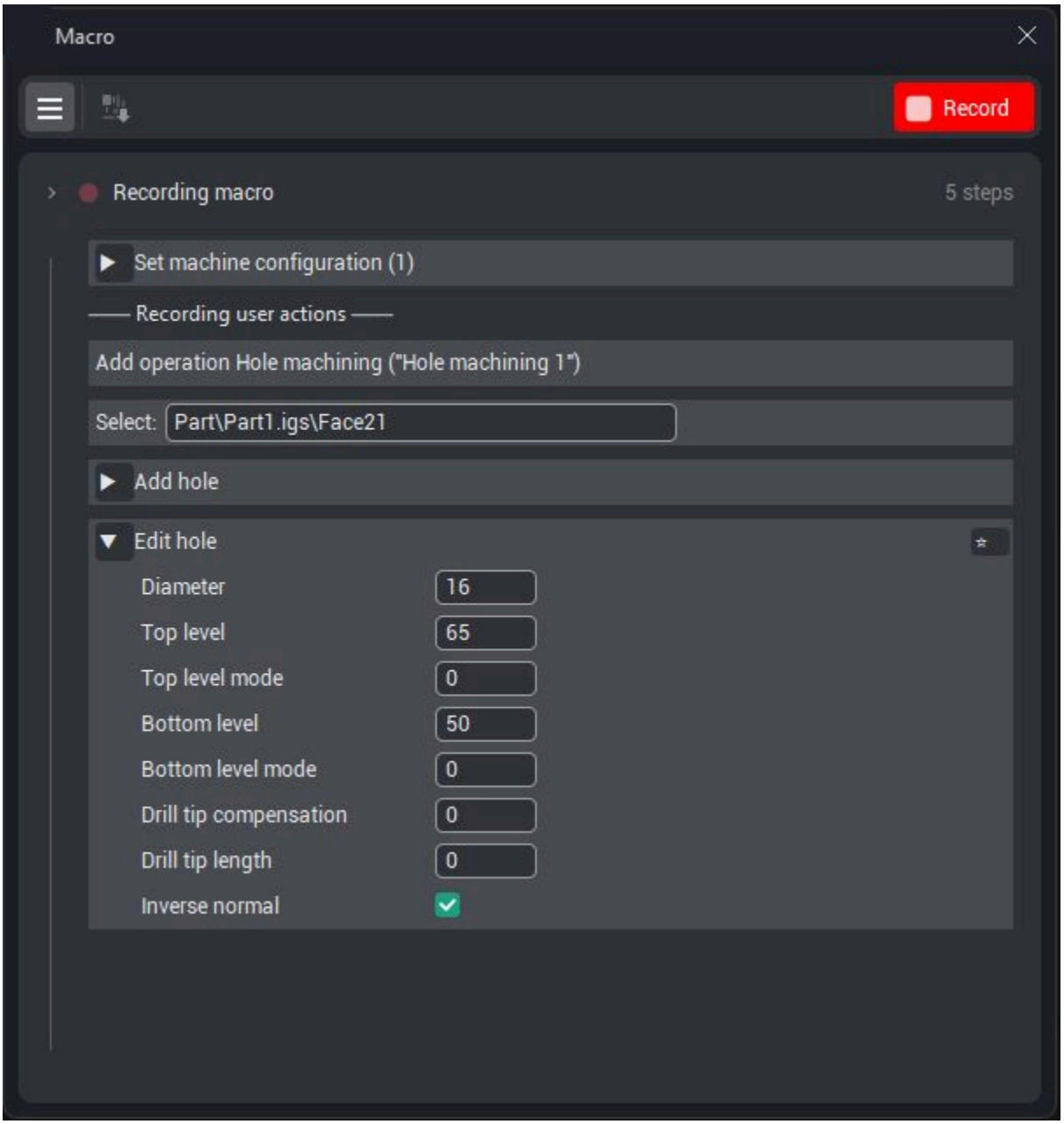




A new Macro option has been introduced.

It enables recording user actions, which can later be opened in VS Code. These macro can be saved and replayed as needed. Details

It is also possible to export all macro parameters to use them for working with another part, i.e., to simulate a standard technical process.

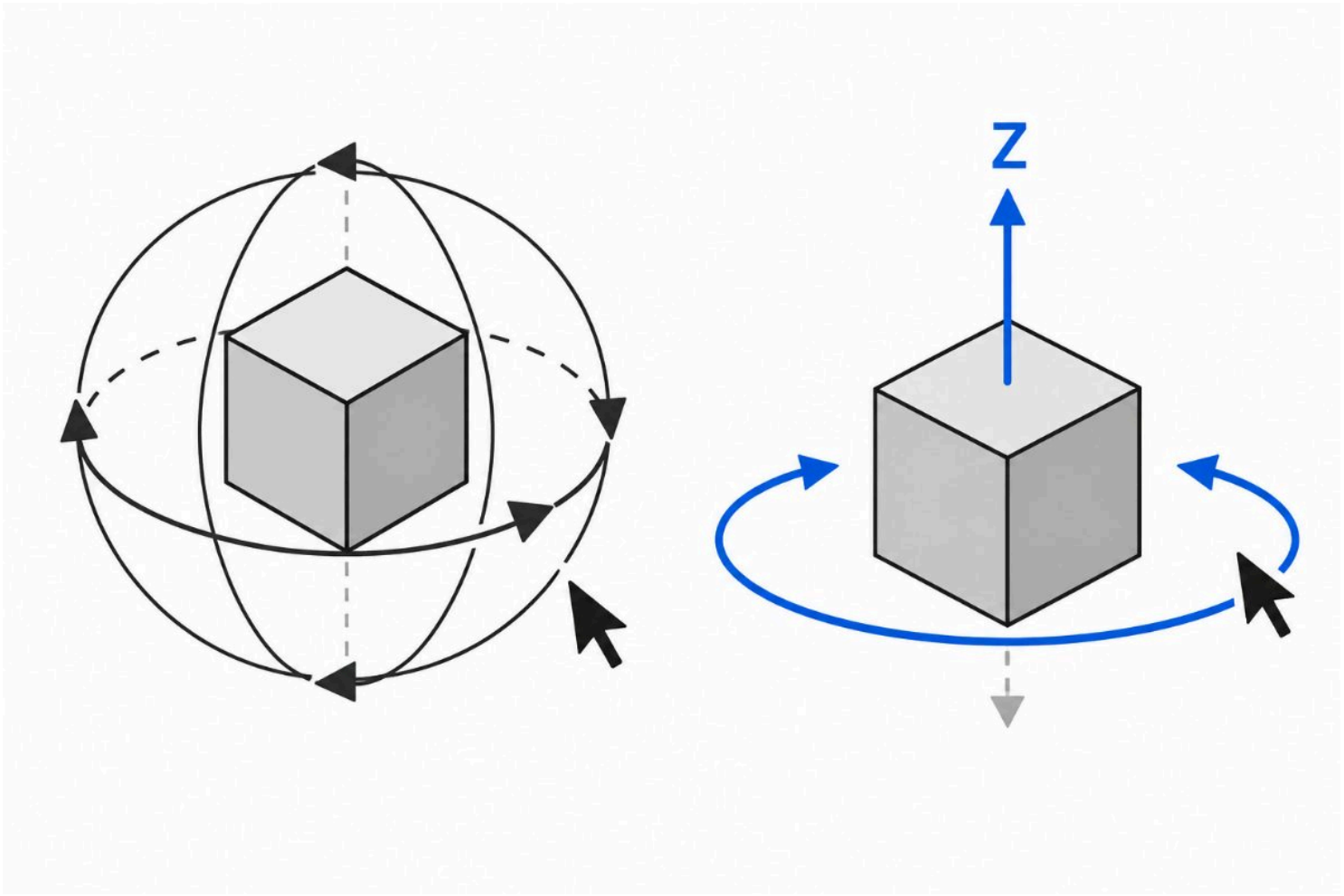


Added a new navigation mode for view rotation.

Rotation Mode — a new toolbar button that switches between free Trackball rotation and constrained Orbit rotation, in which a chosen local axis (X, Y or Z) is kept vertical on screen.

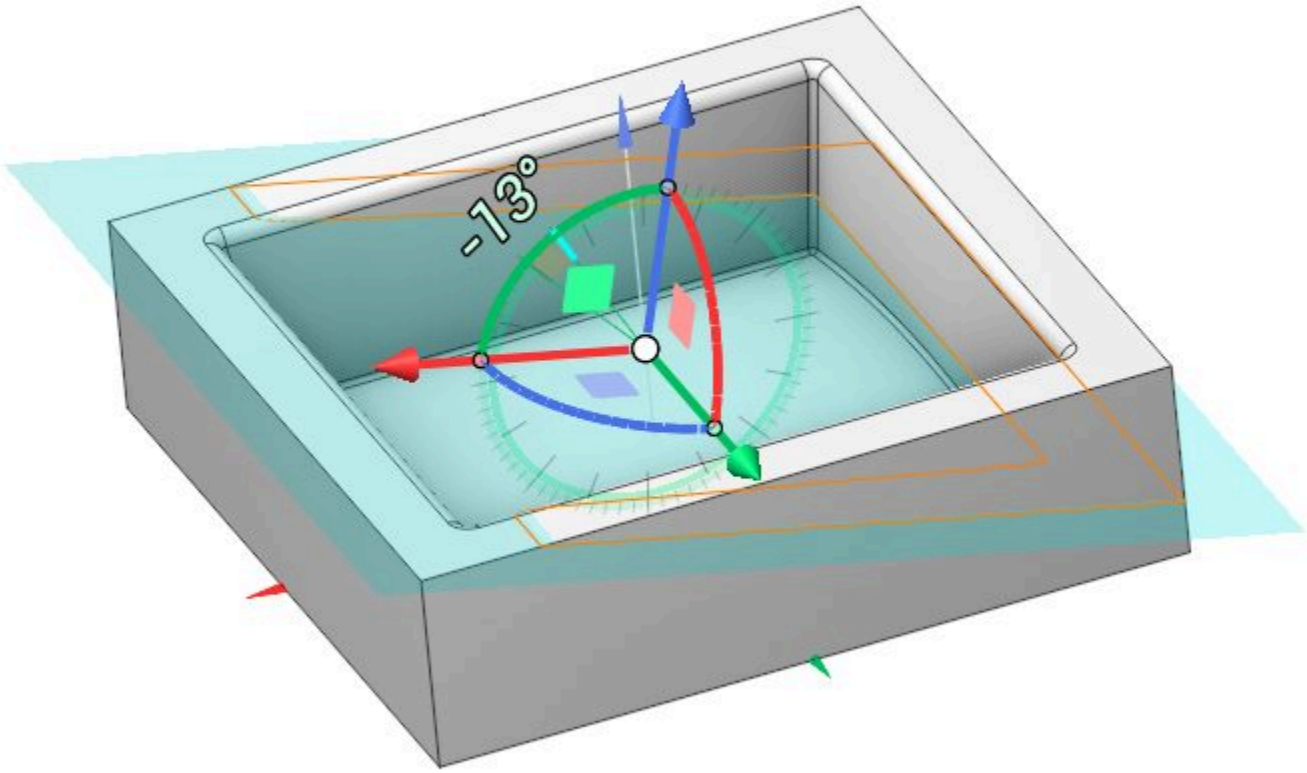
When Orbit mode is enabled, the view smoothly snaps so the selected axis becomes vertical, and horizontal mouse movement rotates the scene around it.

The button is located in the lower part of the screen on the View Control toolbar. Details



Gimbal functionality has been added to Intersect.

The gimbal (Details) provides an interactive way to precisely control the position and orientation of section planes. Details

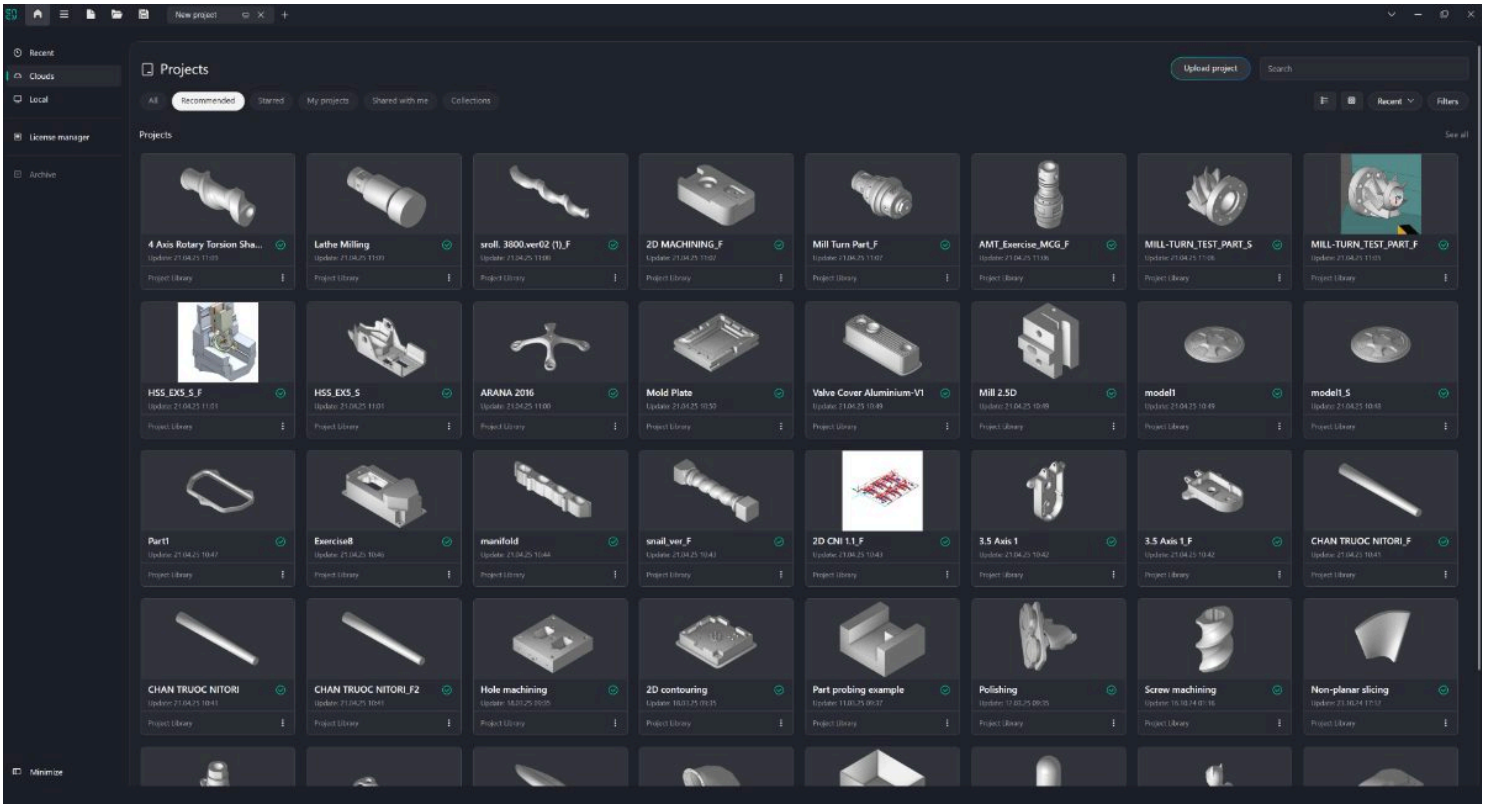

-13°

General information

This section introduces the CAM system interface and the tools used for everyday project work. It describes the Home Page, the main window, graphics view controls, object selection, and system settings.

The topics also cover project and data files, software updates, license management, system logs, and cloud collaboration. Use this section to become familiar with the working environment and find the common commands and settings needed to prepare and manage machining projects.

Home Page

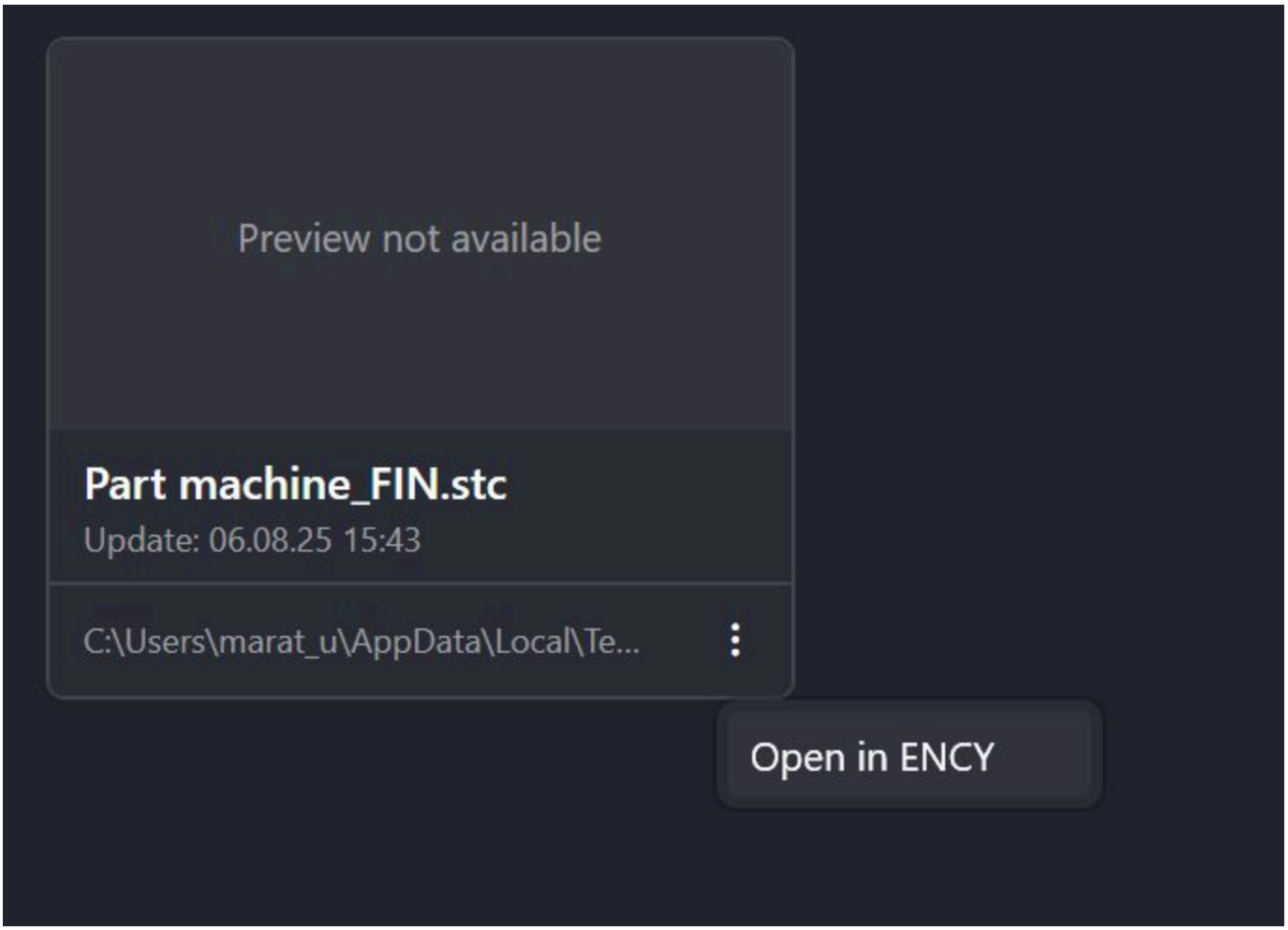


Application Area:

The Home Page allows you to quickly and conveniently browse previously opened projects and use the project library, accelerating the start-up process and increasing developer productivity.

Recent.

The Recent tab allows you to see recently opened projects for quick access to them. Clicking on the three dots allows you to open the project in the last active window of the system.



Cloud.

The Project Library allows you to find and open example projects from our online library. Details

Local.

It acts as a file explorer. Now there's no need to search for a project through Windows Explorer. You can do it on the system's Home Page, which greatly simplifies and speeds up project navigation.

Through the three dots, you can also open the project in the CAM system or open its location in File Explorer.

License manager.

The license manager contains functions for working with licenses for CAM system and licenses for its modules. Details

Minimize.

It allows you to hide the tab labels, which increases the workspace.

Recent

Clouds

Local

License manager

Archive

Projects

All | Recommended | Starred

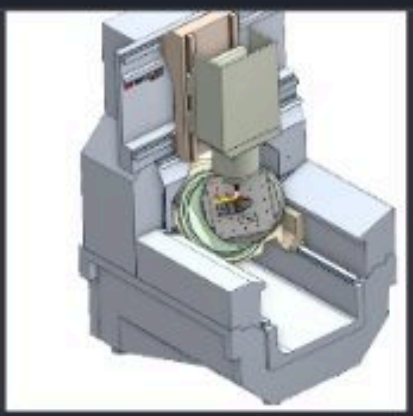
Projects



4 Axis Rotary Torsion Sha...

Update: 21.04.25 11:09

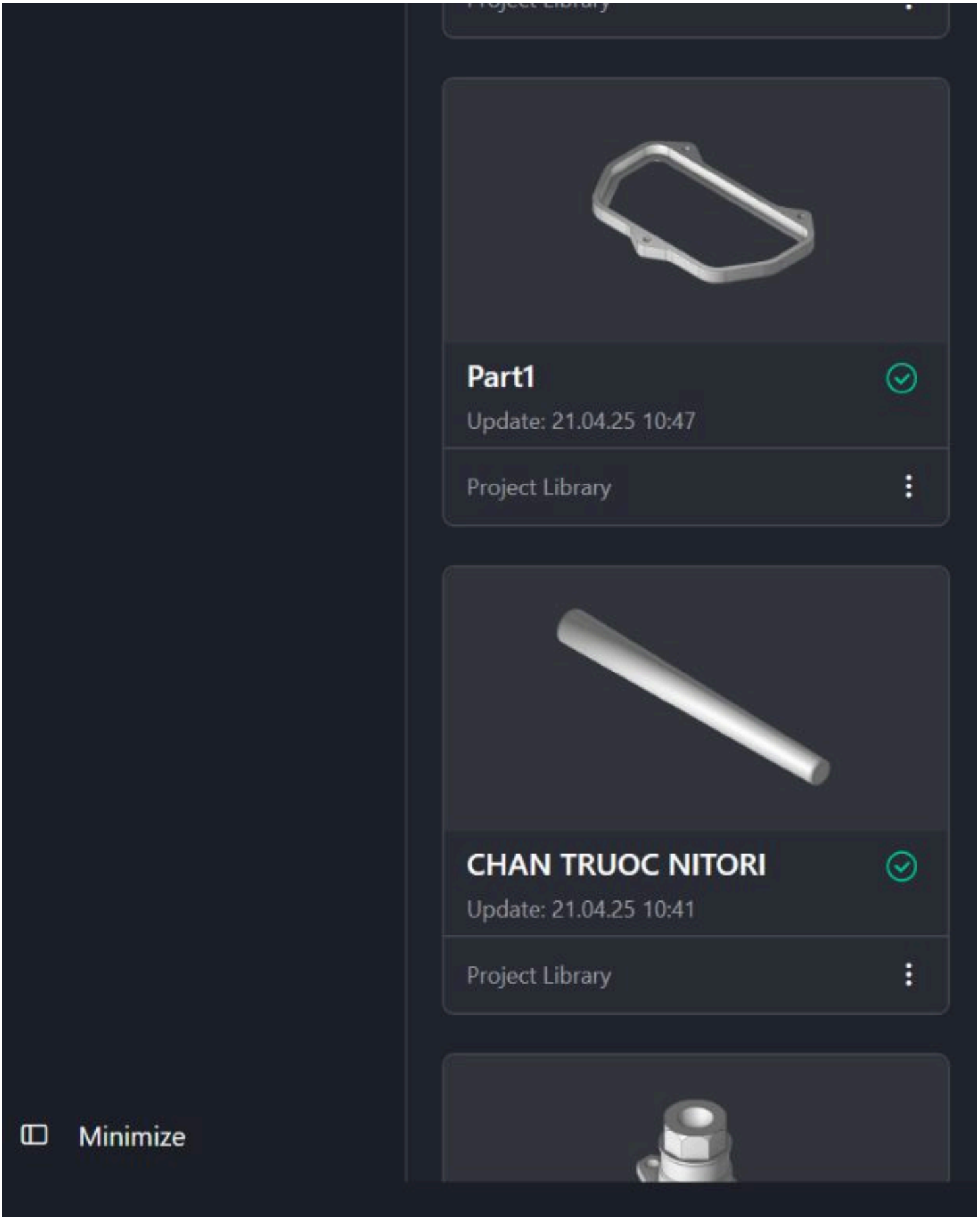
Project Library

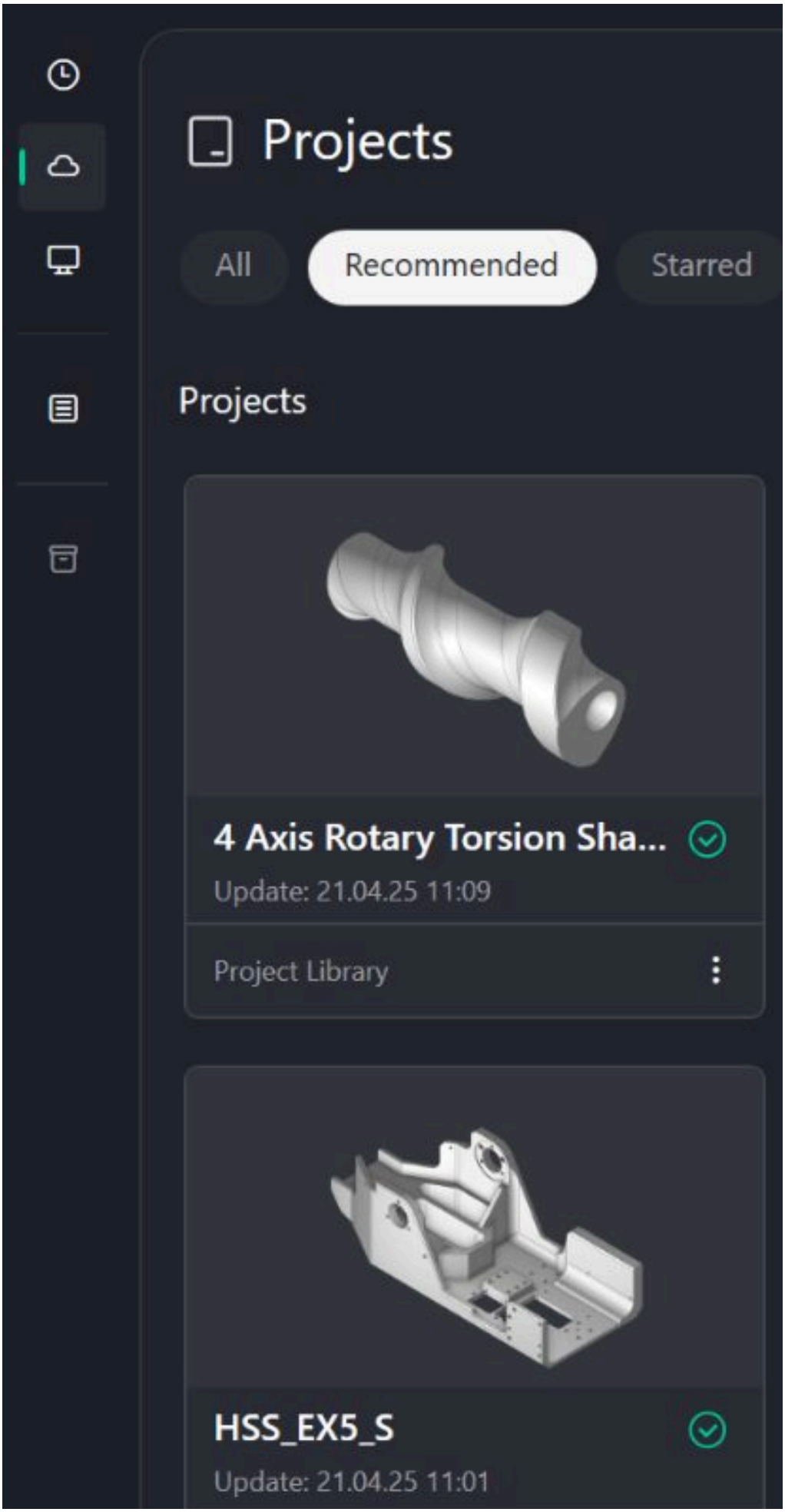


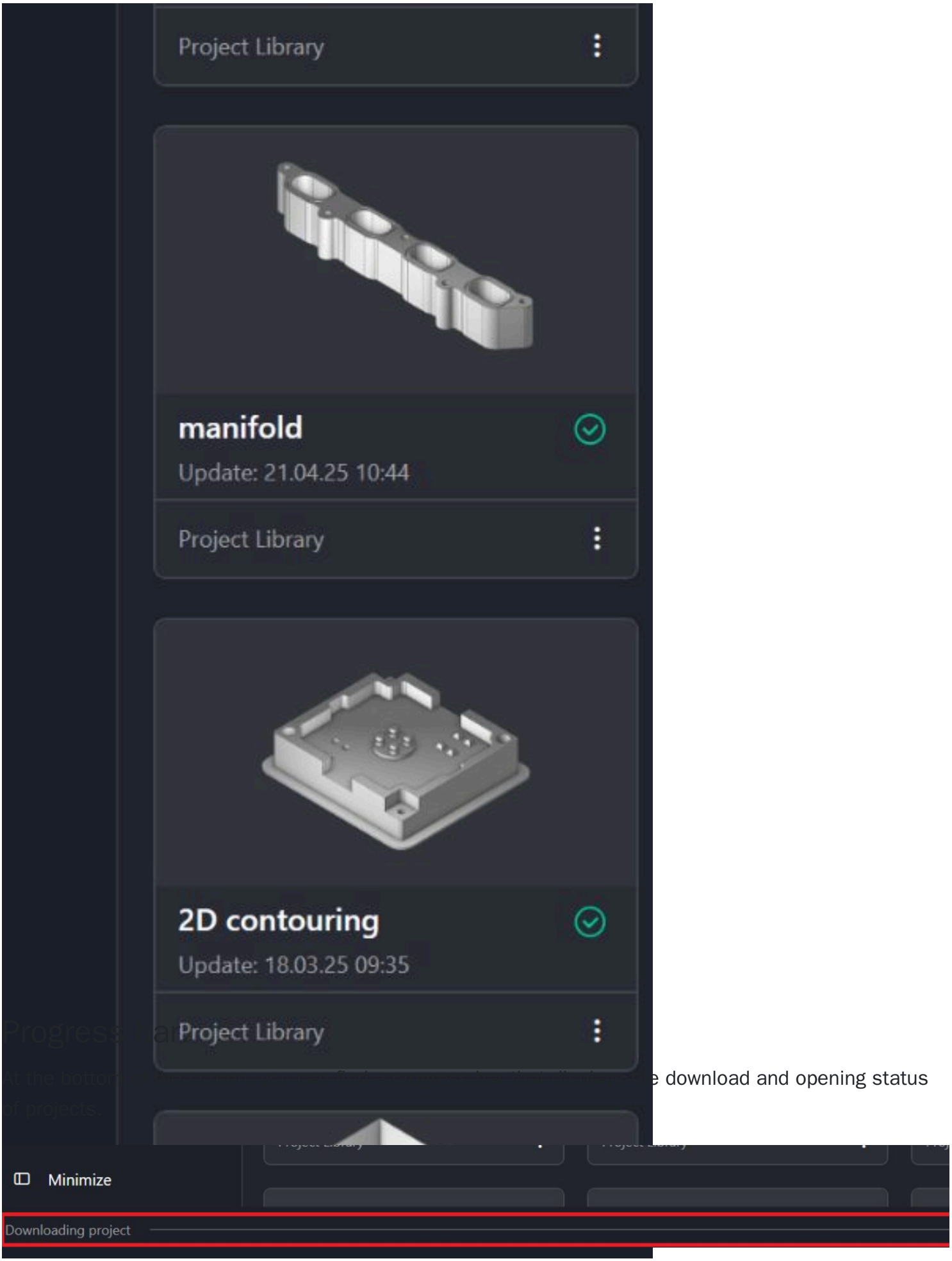
HSS_EX5_S_F

Update: 21.04.25 11:01

Project Library


Part1
Update: 21.04.25 10:47
Project Library
CHAN TRUOC NITORI
Update: 21.04.25 10:41
Project Library
Minimize

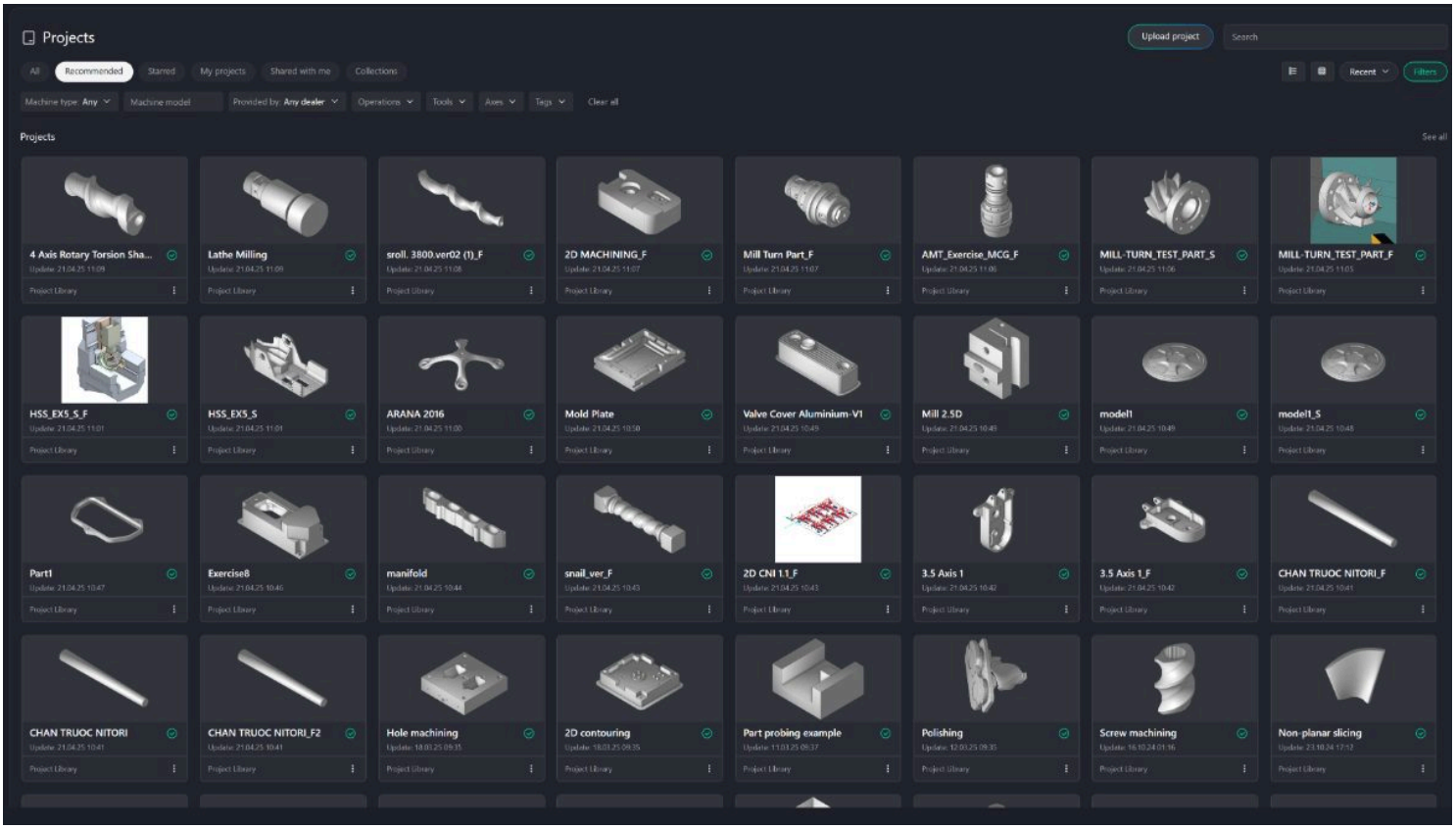

Projects
All
Recommended
Starred
Projects
4 Axis Rotary Torsion Sha...
Update: 21.04.25 11:09
Project Library
HSS_EX5_S
Update: 21.04.25 11:01



e download and opening status



Project library

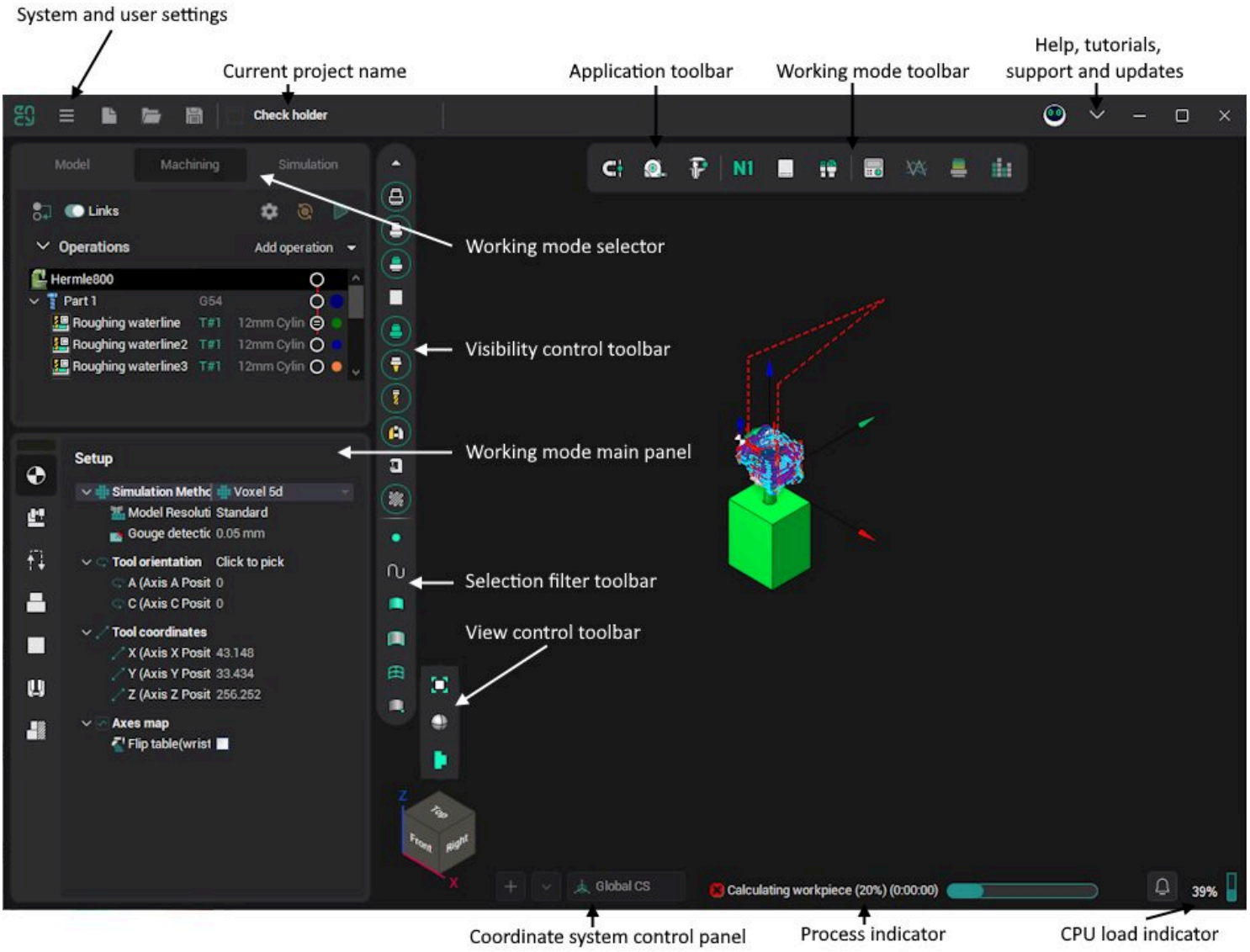


Application Area:

The Project Library allows you to find and open example projects from our online library.

System's main window

The system's main window has the following view:



The graphics window is in the center. Depending on the current working mode, it displays geometry models, machine and technology, or machining simulation.

Along the edges of the window are the main toolbars. There are drop-down menus associated with many buttons on these toolbars. Just hold the mouse pointer over the button or click on it so that the drop-down menu is displayed.

The application toolbar is in the top-left corner of the window. It contains buttons to manage projects and control global features like Smart Snap.

The working mode toolbar is at the center of the window's top bar. It contains features useful in current system mode.

The project title bar shows the name of the current project.

Utilities and Help buttons are located on the left and right sides of the window's toolbar, respectively. The utilities drop-down menu, located on the left, provides system and user utilities, such as the Postprocessors generator. Use the Help drop-down menu, located on the right, to access help, tutorials, and get support.

The working mode main panel is on the left. Depending on the current system mode, it shows geometry model items, the technology tree, the properties inspector, or the toolpath tree (CL-data). Click on one of the tabs (Model, Machining, Simulation) to change working mode.

Use the view control toolbar to change the view vector and visualization properties of the current mode.

Use the visibility control toolbar to change which objects should be visible in the current mode.

Use the selection filter toolbar to change whether objects of the appropriate type should be selected from the screen. These settings also control which objects are visible in the list view in Model mode.

The process indicator displays the current calculation progress.

Use the Coordinate System control panel to add, remove, and modify coordinate systems.

The CPU indicator displays the current overall load on the CPU.

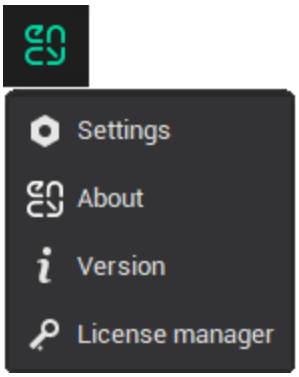
Sometimes, pop-up notifications of the application may appear in the lower right corner of the main window. A panel with a general list of such notifications can be opened by clicking the corresponding icon.

Application Toolbar

The Application Toolbar is positioned on the top panel of the main window. Several buttons within this toolbar include associated drop-down menus, allowing for expanded functionality.



Application Button Menu

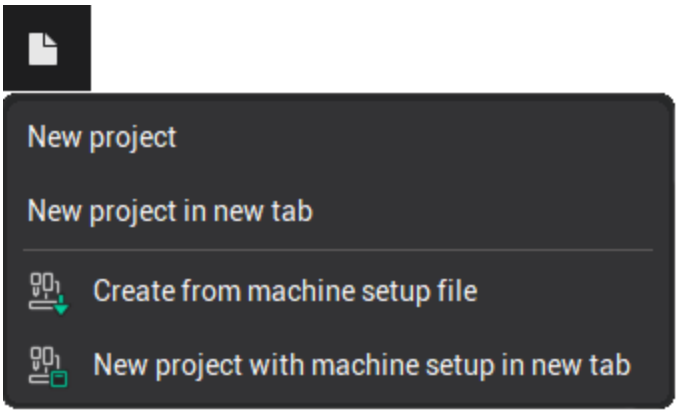


- Settings: Opens a window for editing system settings.
- About: Displays the splash screen of the application.



- Version: Opens the window that provides version information for the application.
- License Manager: Opens the License Manager window.

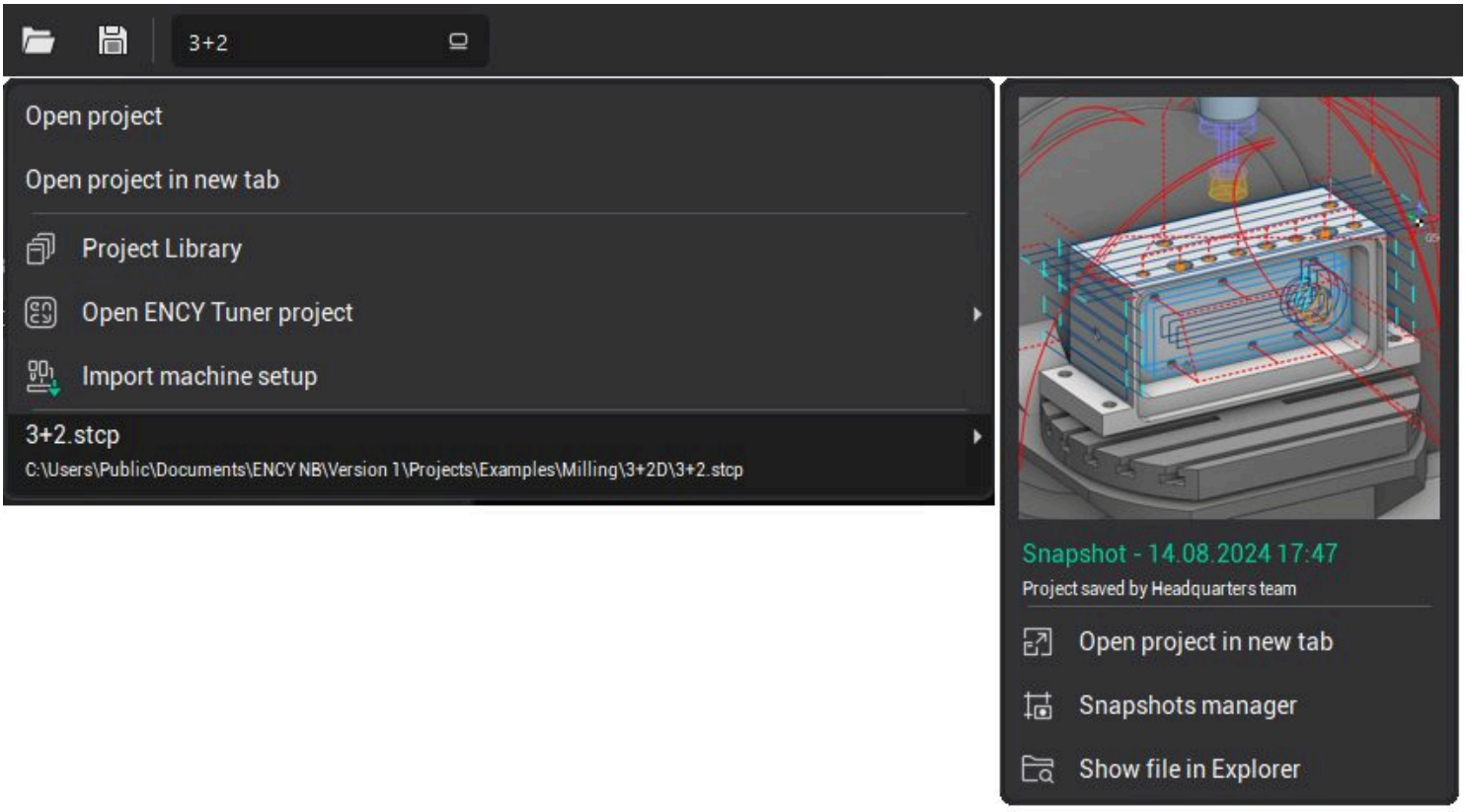
New Project Button Menu



The New Project button menu includes the following options:

- New Project: Closes the current project and initiates a new project in the current tab.
- New Project in New Tab: Opens a new tab and creates a new project inside it.
- Create from Machine Setup File: Closes the current project and creates a new one from a machine setup file. You can choose from recently opened machine setup files or browse your drives to select a file.
- New Project With Machine Setup In New Tab: This option allows you to start a new project in a separate tab using a machine setup file. You can select from your recently used setup files or browse your drives to select a file. The current project will remain open in its original tab, while the new project is created in a new one.

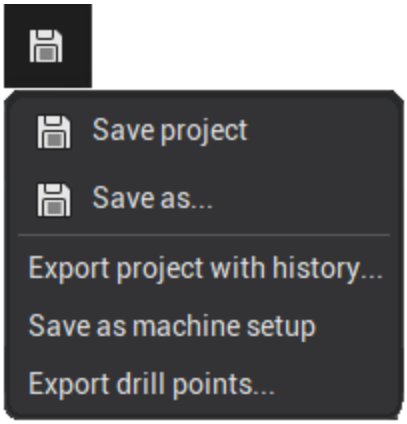
Open Project Button Menu



The Open Project Button provides several options:

- Open Project: Loads an existing project via a standard file selection dialog.
- Open Project In New Tab: Opens an existing project in a new tab, allowing you to keep your current project open while working on another one simultaneously. You can select the project file via a standard file selection dialog, making it easy to switch between multiple projects without closing your current workspace.
- Project Library: Opens a window to search for and open sample projects from around the world.
- Open ENCY Tuner Project: Loads a project created within ENCY Tuner. Holding the cursor over this item reveals the option to open the project in a new tab.
- Import Machine Setup: Imports data from a machine setup file into the current project.
- Additionally, a list of recently loaded projects is available. Each project in this list has a drop-down menu offering further actions, such as:
  - Load Recent Backup Versions (Snapshots): Quickly load backup versions of the project.
  - Open Project in New Tab: Opens the selected project in a new tab.
  - Snapshots Manager: Opens the snapshot management window to manage all snapshots (backup versions) of the project.
  - Show File in Explorer: Opens Windows Explorer and selects the project's file.

Save Project Button Menu



The Save Project Button offers the following options:

- Save Project: Saves all changes to the current project. If the project has not been saved before, the system will request a new name.
- Save As...: Saves the project under a new name via a save dialog.
- Export Project With History: Exports the project along with its full modification history, including snapshots made throughout the project lifecycle.
- Save as Machine Setup: Saves the current project template to a machine setup file.
- Export Drill Points: Opens a window to export drill points of the current operation (if a Holes page exists in the properties) to a DXF file. This file can be imported into another project's model page.

Additional Tools and Buttons

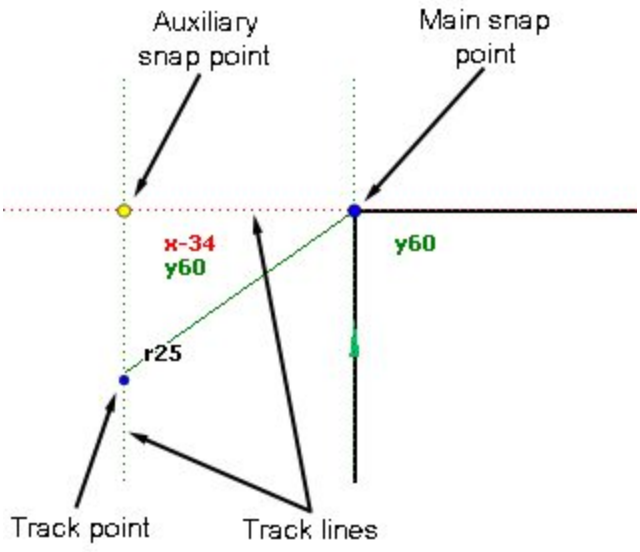
Several additional tools and buttons are available for further functionality:

-  Smart Snap: Enables or disables snapping to objects in the graphics window.
-  Geometry Measuring: Activates the measurement mode for objects within the graphics window.
-  Measure Tool: Allows for measuring distances between two arbitrary points on the screen.
-  Help: Provides access to the help menu.

Smart Snap

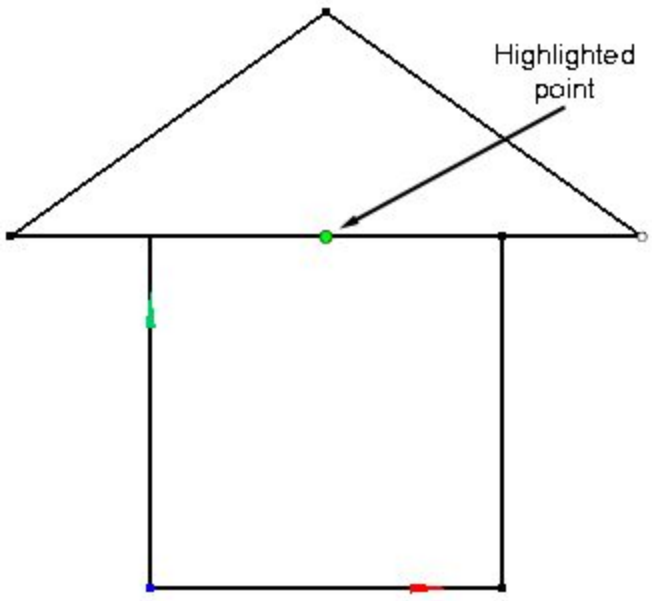
The Smart Snap  button enables or disables snapping to objects in the graphics window. This feature is useful when creating new geometric objects or editing existing ones, such as points, lines, contours, and coordinate systems.

For example, by using the Smart Snap feature, you can easily create precise, well-constrained drawings on the fly.

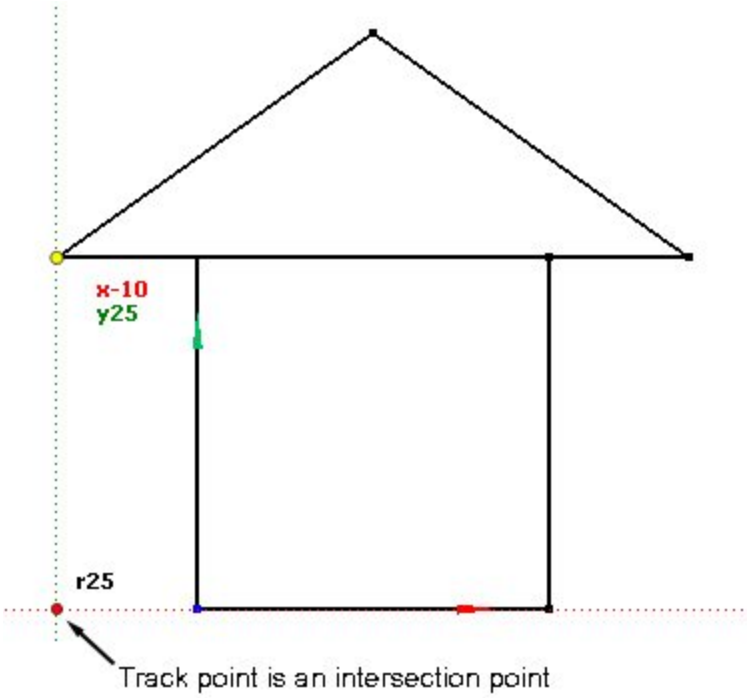


The main building blocks of Smart Snap are:

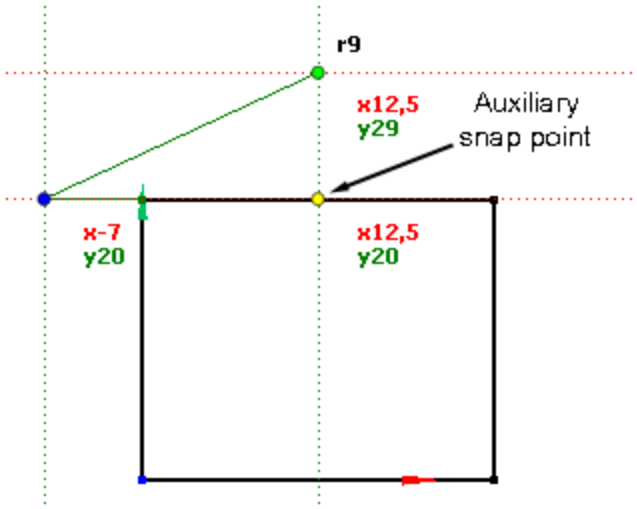
- Highlighted points: When you move the cursor over geometric entities, the points you can snap to are highlighted in lime color. These include terminating points of cuts and arcs, distinct points, the center points of arcs and circles, the middle points of cuts, intersection points, etc.



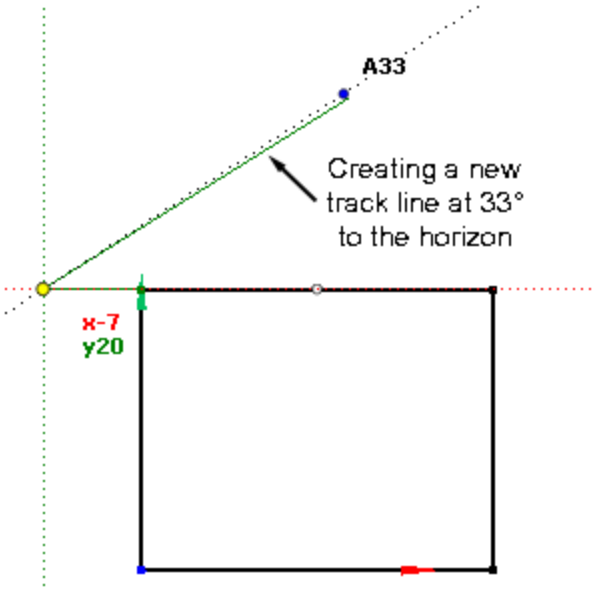
- Main snap point: Whenever you click the left or middle mouse button on a highlighted point or a track point (see below), you create a main snap point with an associated set of track lines. The main snap point is persistent, meaning it doesn't disappear when you move the cursor away. It is labeled with the point coordinates.
- Main track lines: These are lines originating from the main snap point. Some lines are aligned with the current Local Coordinate System, while others are aligned with neighboring geometric entities (e.g., if the main snap point is the termination point of a cut, the main track lines include lines parallel to and perpendicular to the direction of the cut).
- Current track point: The track point is the point on a track line. When you move the cursor over a track line, the track point moves as well. It is labeled with the distance from the current snap point, and may also represent the intersection of a geometric entity or another track line with the current track line. In this case, the track point is colored red.



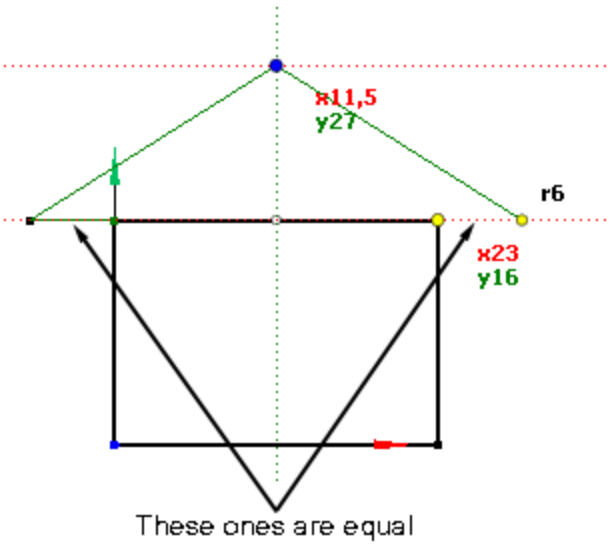
- Auxiliary snap point: When you hover over a highlighted point or track point for some time, the point becomes an auxiliary snap point. It behaves like the main snap point but is not persistent, meaning it disappears when you move the cursor away from it and its track lines. The auxiliary snap point has its own set of track lines, allowing you to construct intersection points between the main and auxiliary track lines on the fly. This is a very useful feature.



- Dragging track lines with the middle mouse button: You can press the middle mouse button over a track line and, while holding the button, drag the line. This action creates a new track line at an angle relative to the source track line. While dragging the line, you can see the actual angle value on the screen.



- Keeping track of the last distances: When drawing, you may want to make cuts of equal length. To help with this, Smart Snap remembers the last three track distances you accepted when creating the drawing. When you move the cursor over the track line, the track point may be colored yellow, indicating that the distance from the track point to the snap point matches one of the last three distances.

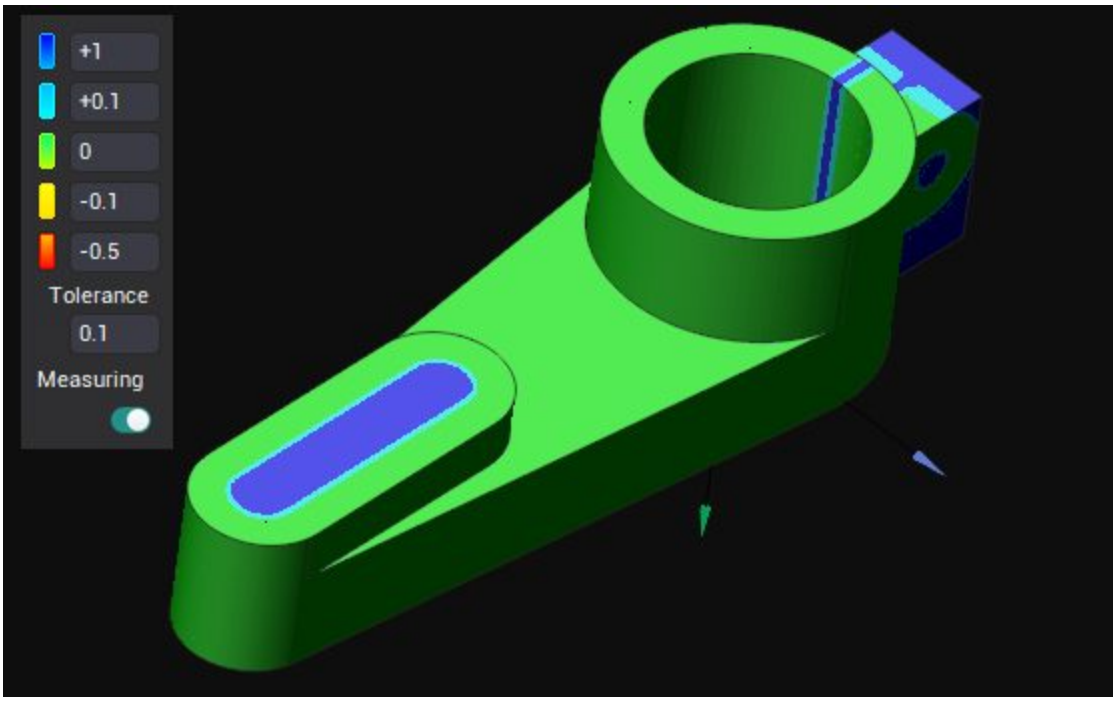

x11,5
y27
r6
x23
y16
These ones are equal

Measurement and analysis

The CAM system includes tools for conveniently measuring geometrical parameters of a model and analyzing the results of simulated machining.

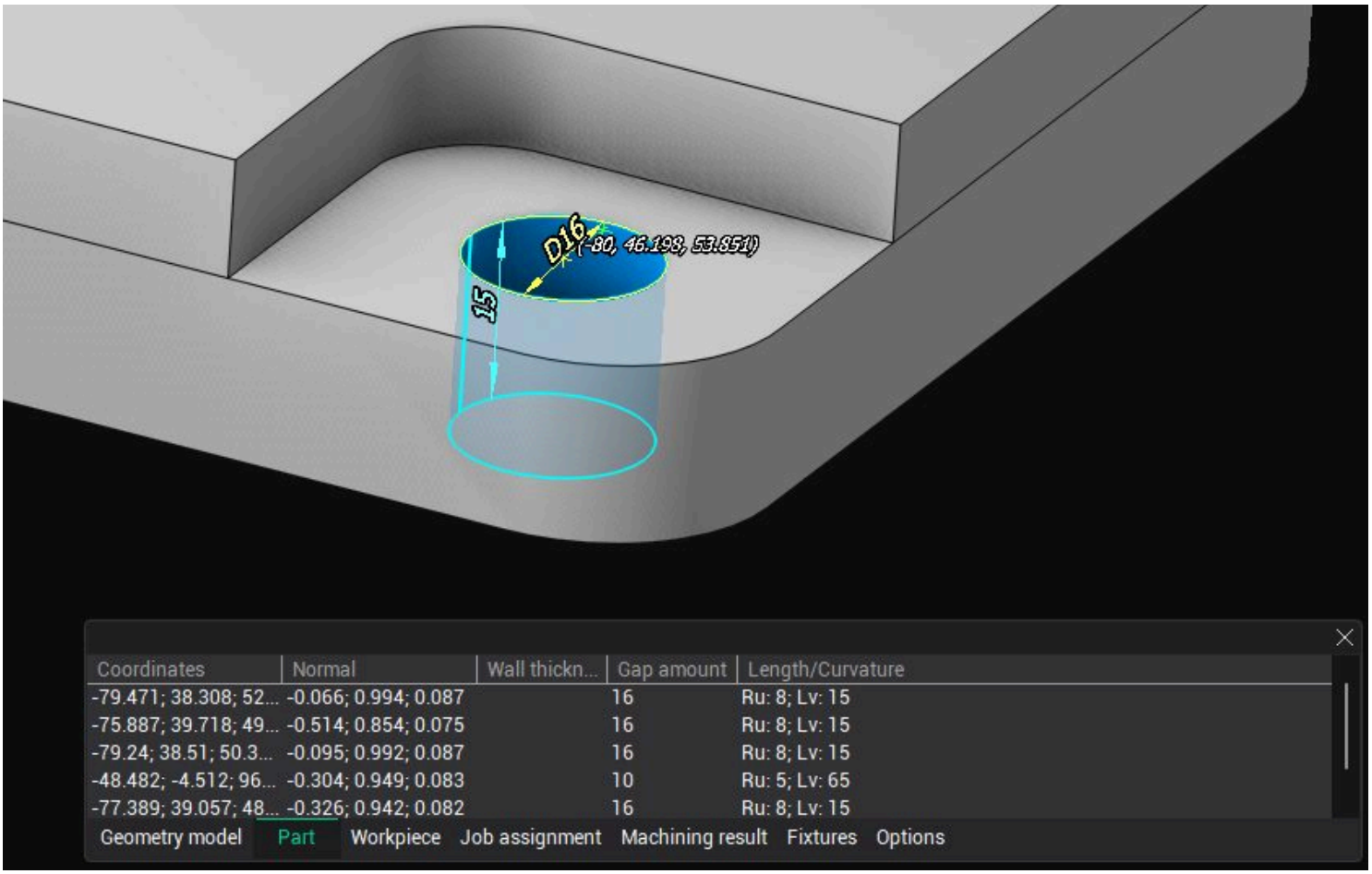
Verify Compare

The Verify Compare  feature can be activated in either Machining or Simulation mode. This tool visually displays the differences between the original part and the machining result using a color scheme, where each color represents a specific deviation range between the compared elements.



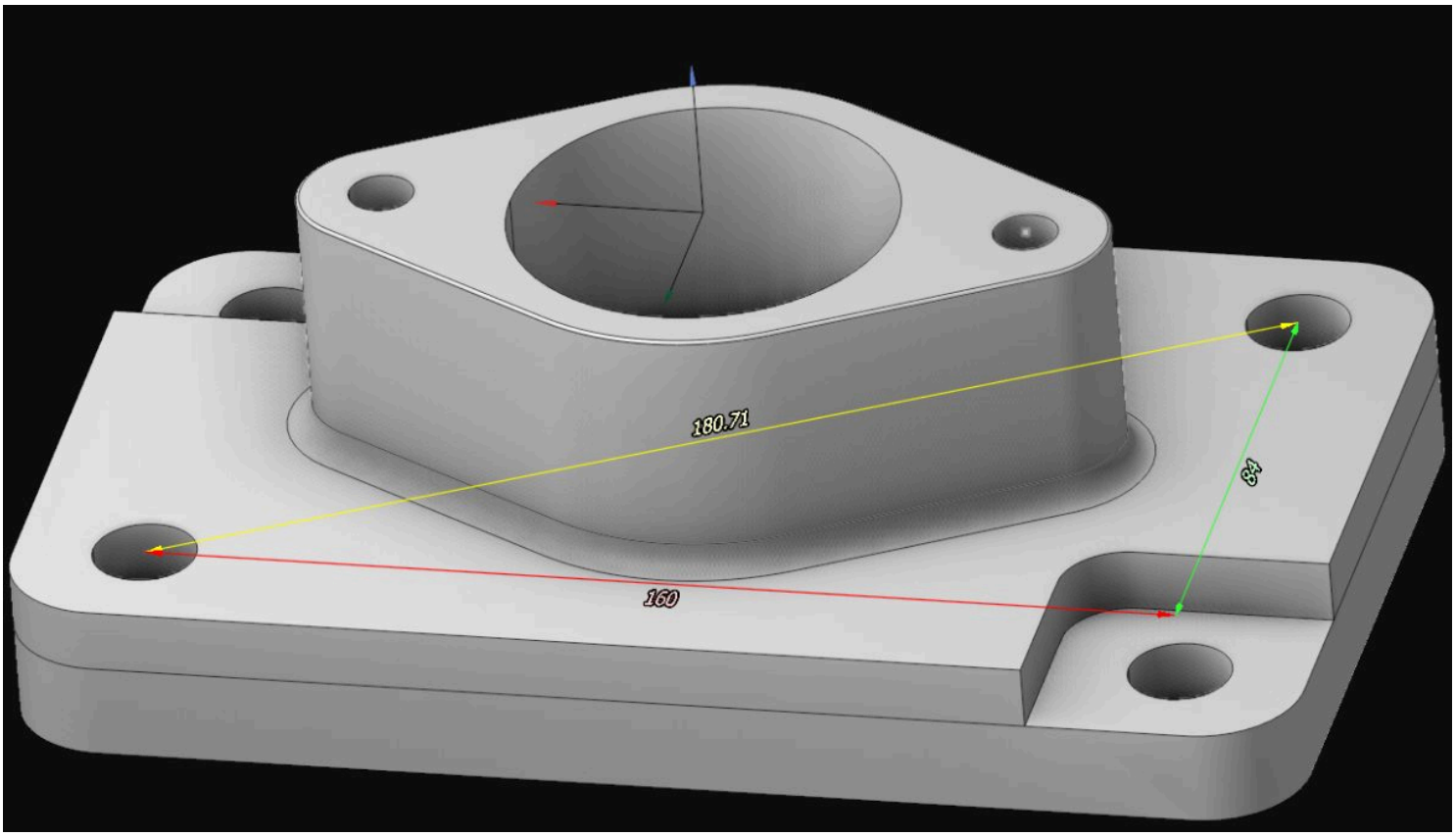
Geometry Measuring

The Geometry Measuring  tool helps you determine the primary geometric dimensions of the selected object in the graphics window. Upon activating this mode, an additional window appears, allowing you to choose the type of object being measured (e.g., part, workpiece, machining result, etc.) and configure measurement parameters. When you select an object, its key dimensions are displayed in the graphics window, and a row with the main parameters related to the selected point is added to the measurement window. Some dimensions may depend on the active geometrical coordinate system.



Measure Tool

The Measure Tool button  enables the mode for measuring distances between two arbitrary points on the screen. After activation, select any two points in the graphics window. Three orthogonal sizes and one diagonal will appear on the screen. The orientation of these dimensions depends on the active geometric coordinate system. You can continue selecting additional pairs of points to measure the distances between them.


180.71
64
160

Autosaves and Project snapshots

The program features an automatic backup system for projects, allowing users to revert to a previous version of any project if unforeseen issues arise.

The key concept in this system is a snapshot, which represents the complete state of the project at a specific point in time. Each time a project is saved, the system generates a snapshot. The most recent snapshot is always stored within the project's *.stcp file. Typically, the system retains several backup versions at once, with older snapshots placed in a backup file located in the history subfolder next to the project: `projectfolder\__history\projectname.stcp.~back`.

- projectfolder refers to the folder containing your current project.
- projectname refers to the name of your current project.

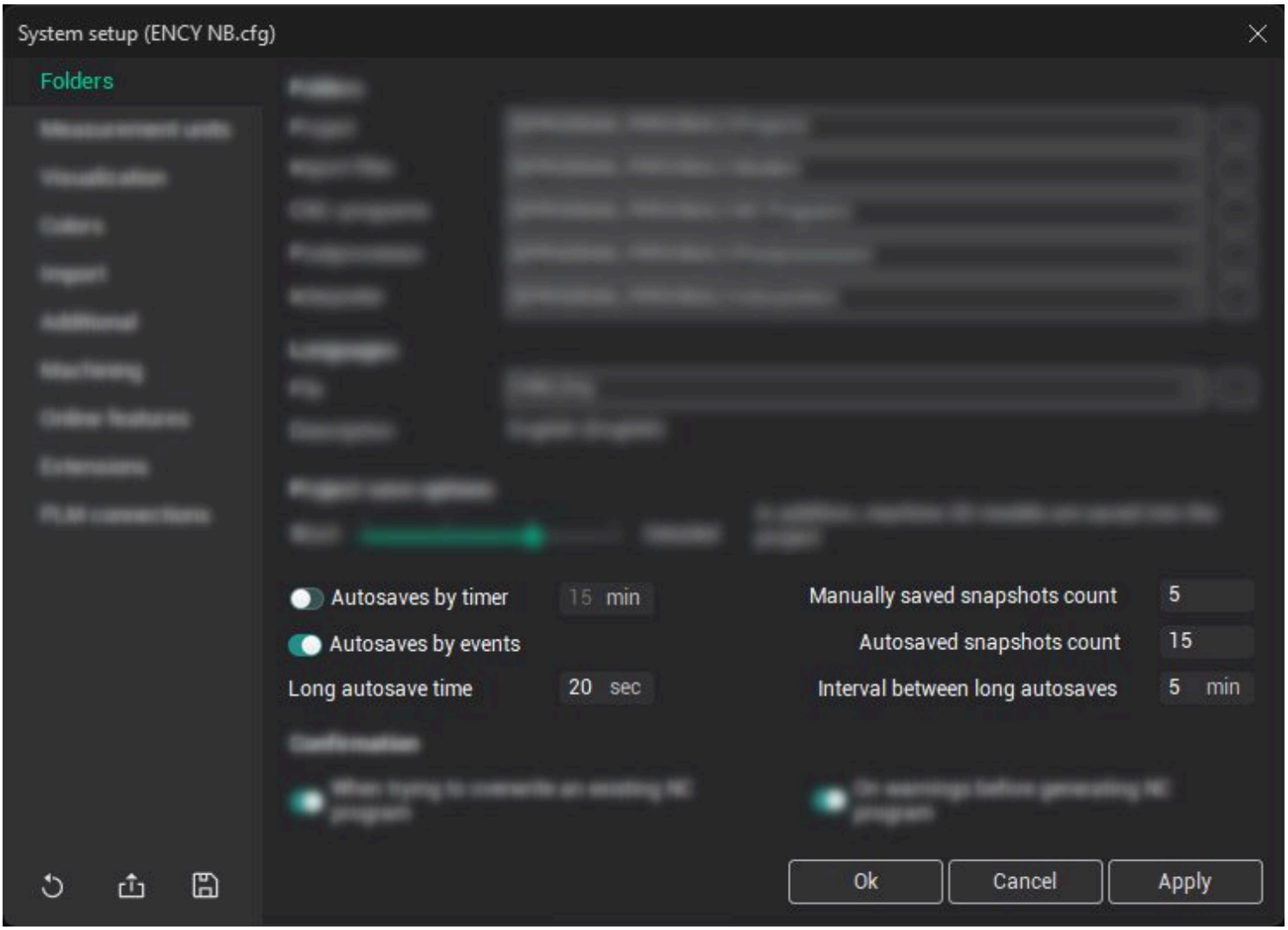
Although a snapshot captures the full project state, this doesn't mean the size of two snapshots equals the size of two full projects. When saving each subsequent snapshot, the system only saves the data that has changed since the previous snapshot, reducing disk space usage.

New snapshots are created in two main cases:

1. When the user manually saves the project (manual snapshots).
2. When the system automatically saves the project (autosaved snapshots).

These two types are managed separately, and automatic saves never overwrite manually saved snapshots. Both are stored in parallel within the snapshot list.

You can configure autosave and snapshot options in the system settings.



The autosave process is optional and can occur in two ways:

- By a timer at specified intervals (using the interval in minutes setting if the Autosaves by timer option is enabled).
- By specific system events, such as before deleting a technological operation, before toolpath calculation, and immediately after toolpath calculation, etc. (using the Autosaves by events option in the system settings)

During extended project work, numerous snapshots may accumulate. To avoid disk overflow, the program automatically deletes snapshots if their number exceeds the specified limit in the system settings. Manually saved snapshots are considered more important by the system, so two different snapshot limits can be set:

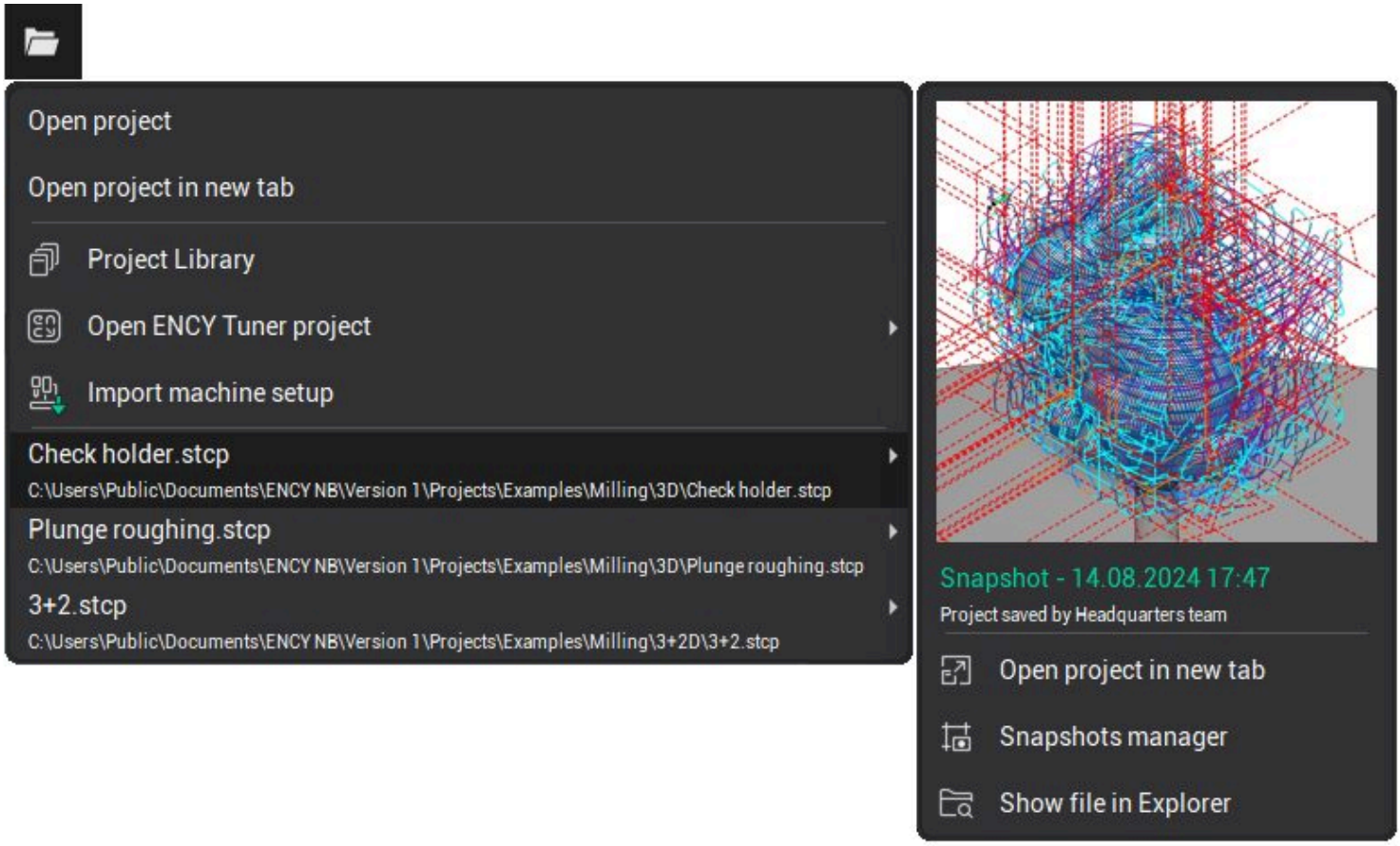
- Manually saved snapshots count: The maximum number of user-saved snapshots (default is 5).
- Autosaved snapshots count: The maximum number of automatically saved snapshots by timer or events (default is 15).

You can also mark any snapshot as persistent, meaning it won't be deleted even if the maximum number of snapshots is exceeded. Persistent snapshots can only be deleted manually, allowing you to maintain

alternative versions of the same project.

For large projects, saving can take time. To reduce interruptions, the system performs automatic saves no sooner than a specified period after the previous save — set by the Interval between long autosaves option (default is 5 minutes). The system considers a project large if the last save took longer than the duration specified in the Long autosave time parameter (default is 20 seconds). For example, if the previous snapshot took more than 20 seconds to save, the next autosave will not occur until at least 5 minutes later.

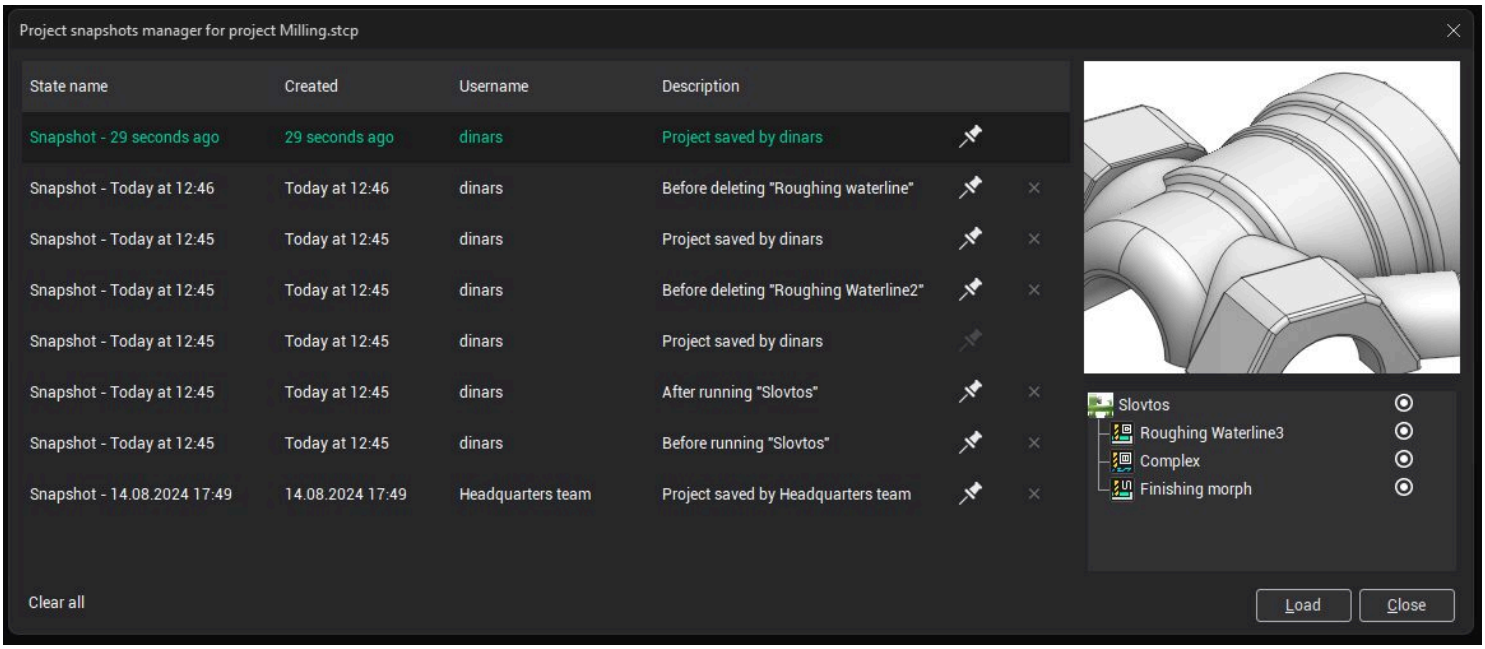
The user interface allows quick loading of saved project states. The list of snapshots is displayed in a drop-down menu next to the project name in the recent projects list on the main toolbar. Snapshots are shown in reverse chronological order, with the most recent at the top. The current snapshot is highlighted in green. The Pin button lets you mark a snapshot as persistent, preventing automatic deletion.



To keep the menu concise:

- For the current project, up to 10 of the most recent snapshots, plus all persistent ones, are shown.
- For non-current projects, only persistent snapshots are displayed.

To view more snapshots, you can open the Project Snapshots Manager window.



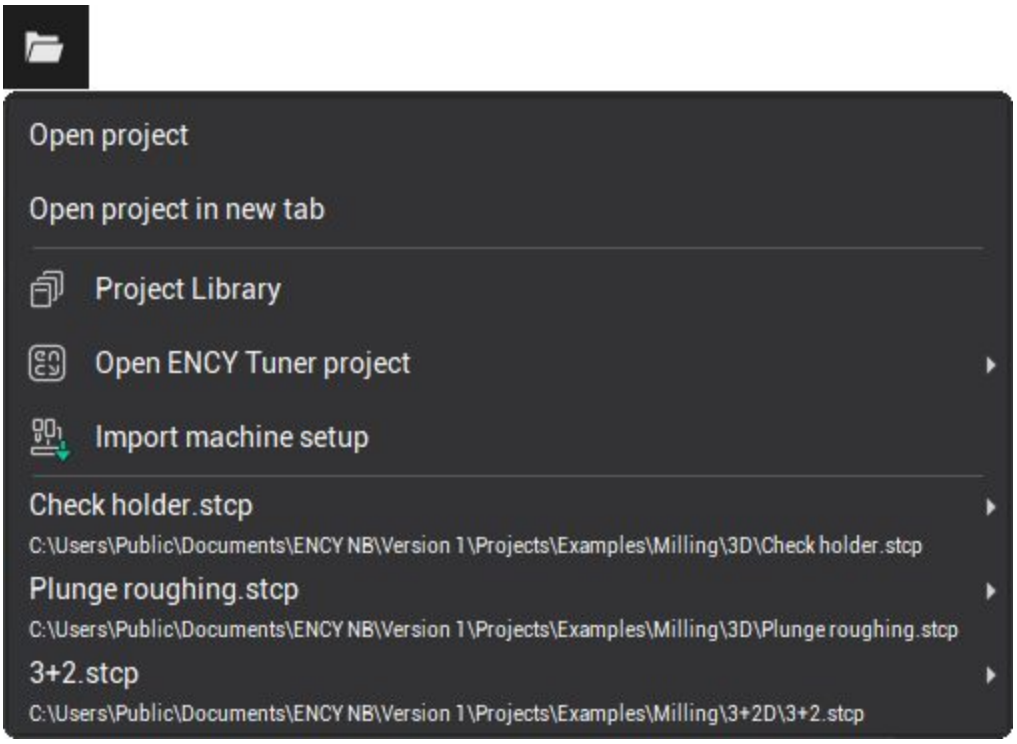
In the Project Snapshots Manager, you can:

- View the full list of snapshots.
- Edit their names or descriptions.
- View a thumbnail.
- See the user who saved the snapshot.
- Review a list of operations associated with each snapshot.
- Delete snapshots or mark them as persistent.
- Load a snapshot.

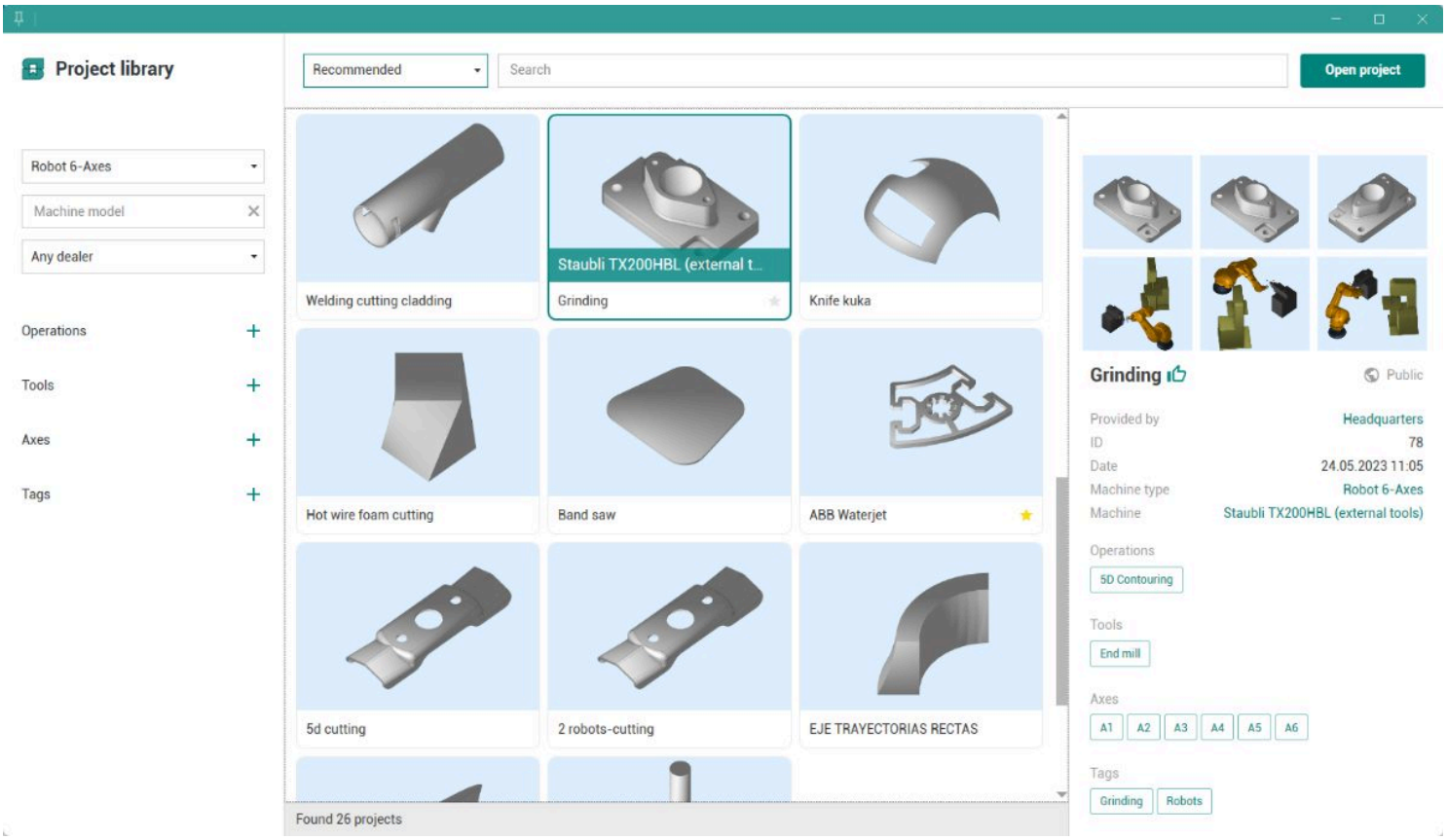
If you hover over the creation date, a tooltip will display the full date.

Project library

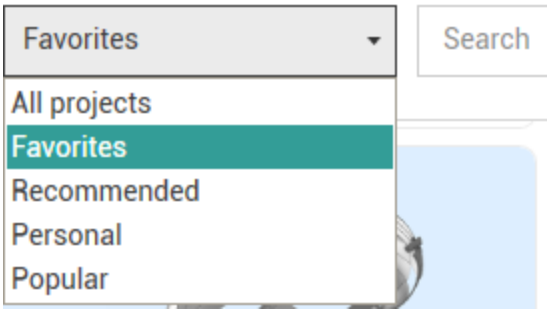
The Project Library allows you to find and open example projects from our online library. You can access the Project Library directly from the CAM system's "Open Project" menu.



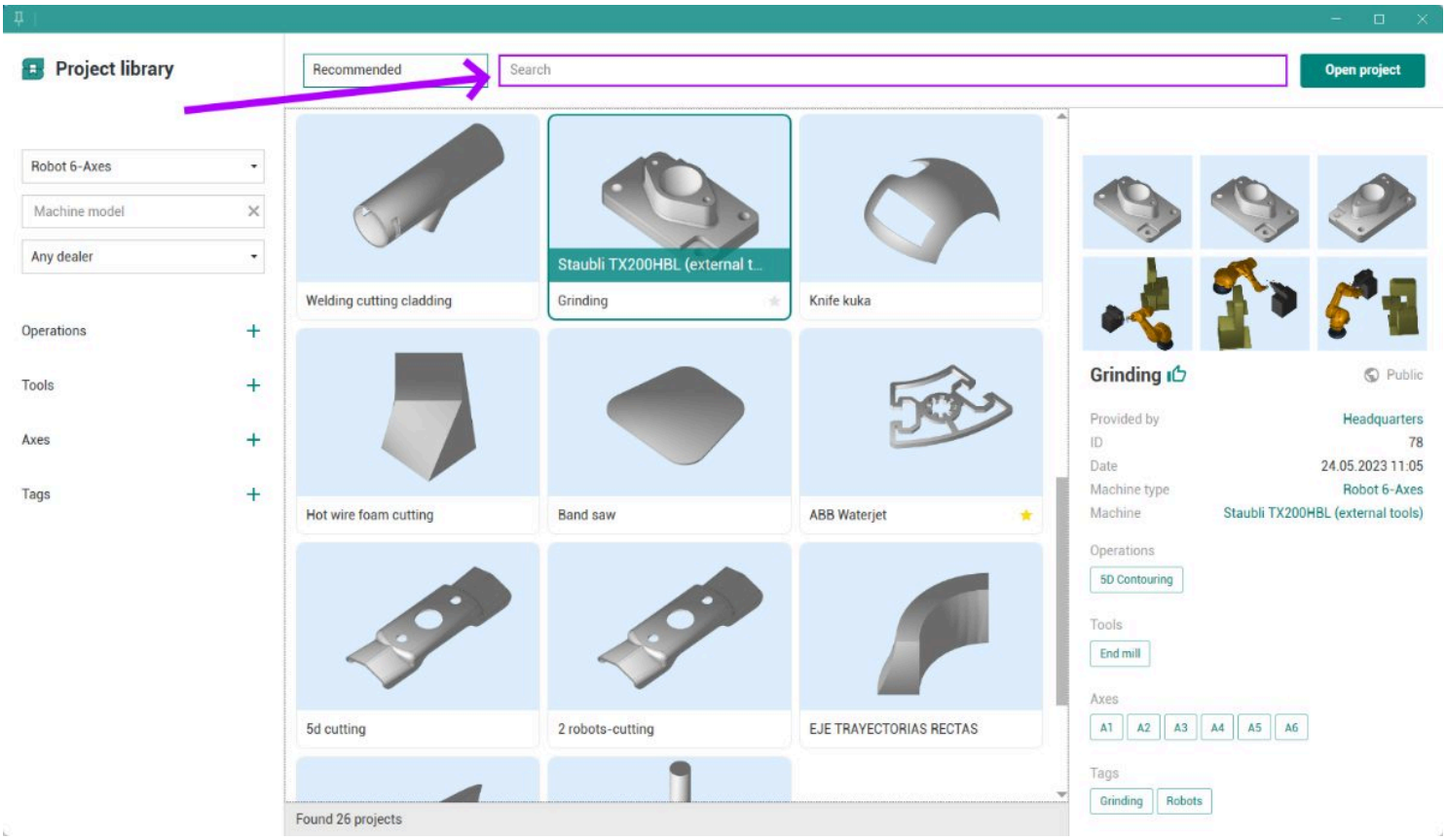
In the main window of the Project Library, all available projects that can be loaded into the CAM system are displayed.



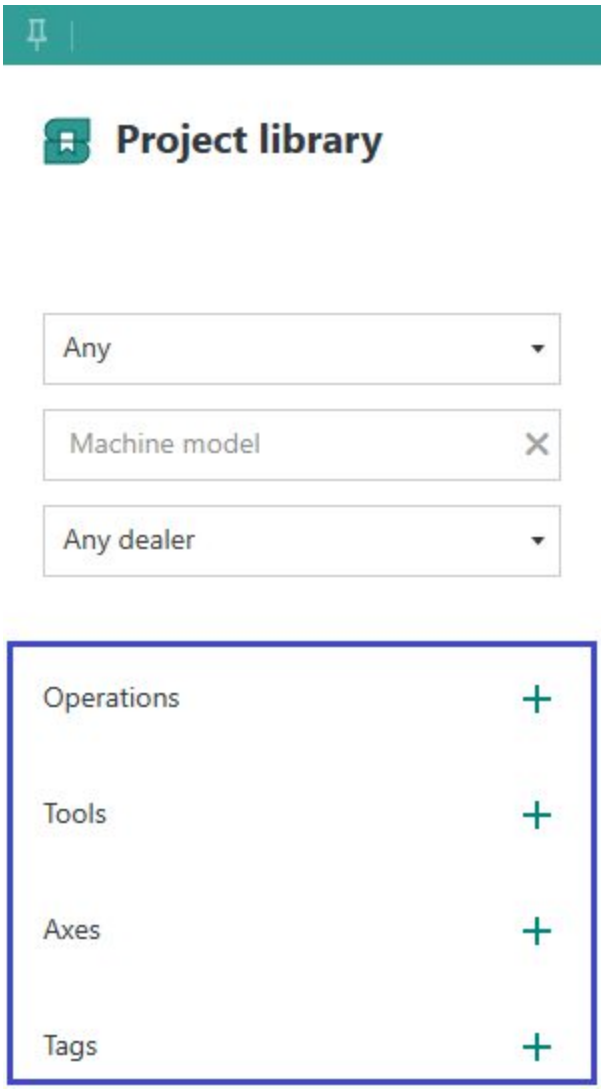
You can use the projects source selector to view All projects, Favorites, Recommended, Personal, or Popular projects.



Projects can be searched by ID, Name, Machine model, Machine type, Operations, Tools, etc. Filters on the left side of the window allow for more refined searches.

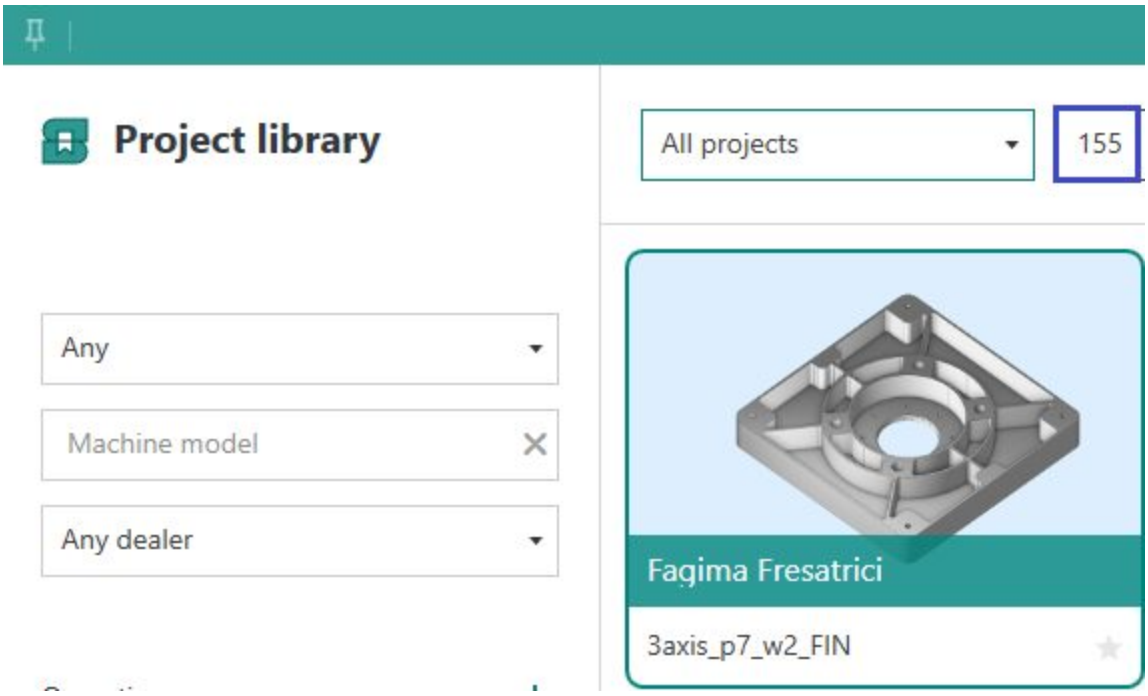


You can also search for projects using the search menu by Operations, Tools, Axes, or Tags.



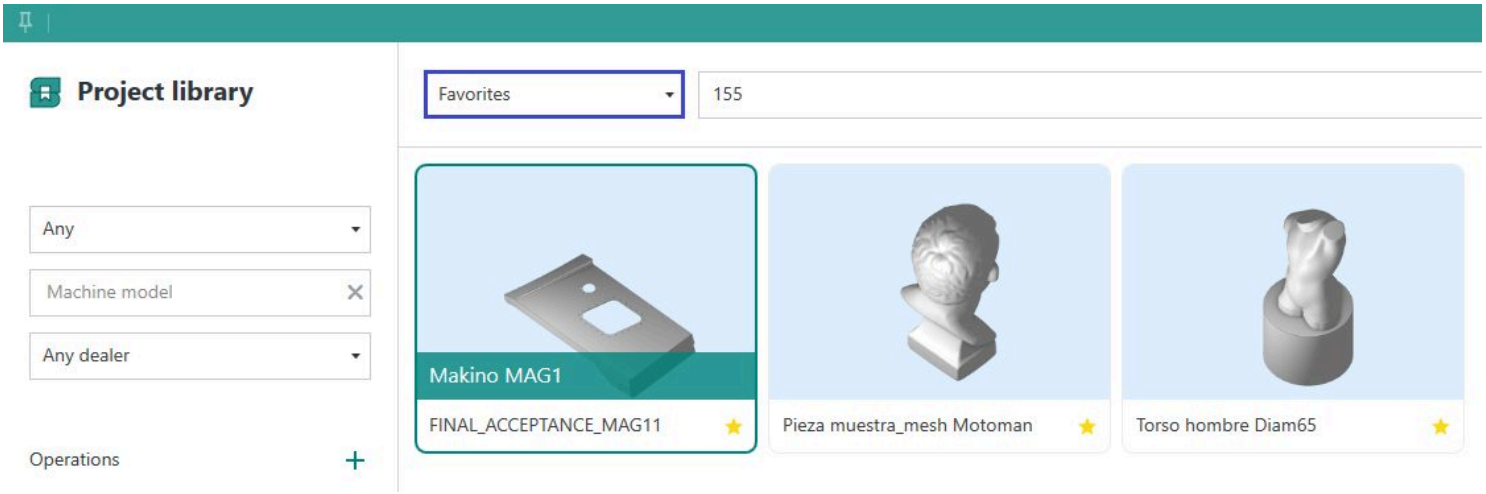
To copy the project ID to the clipboard, click button. The project ID is useful for future searches.



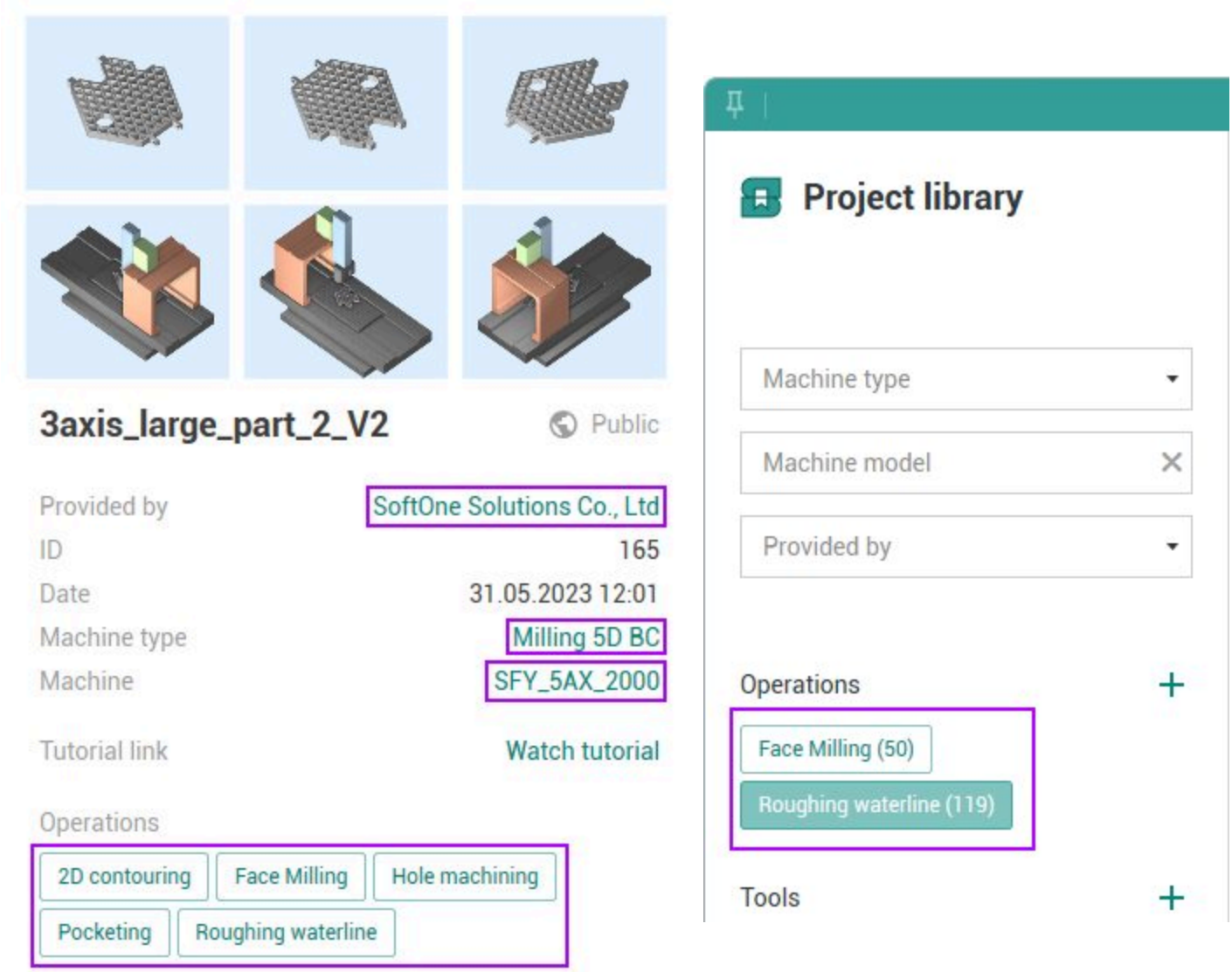


To save a project to your favorites list, use ☆ icon.





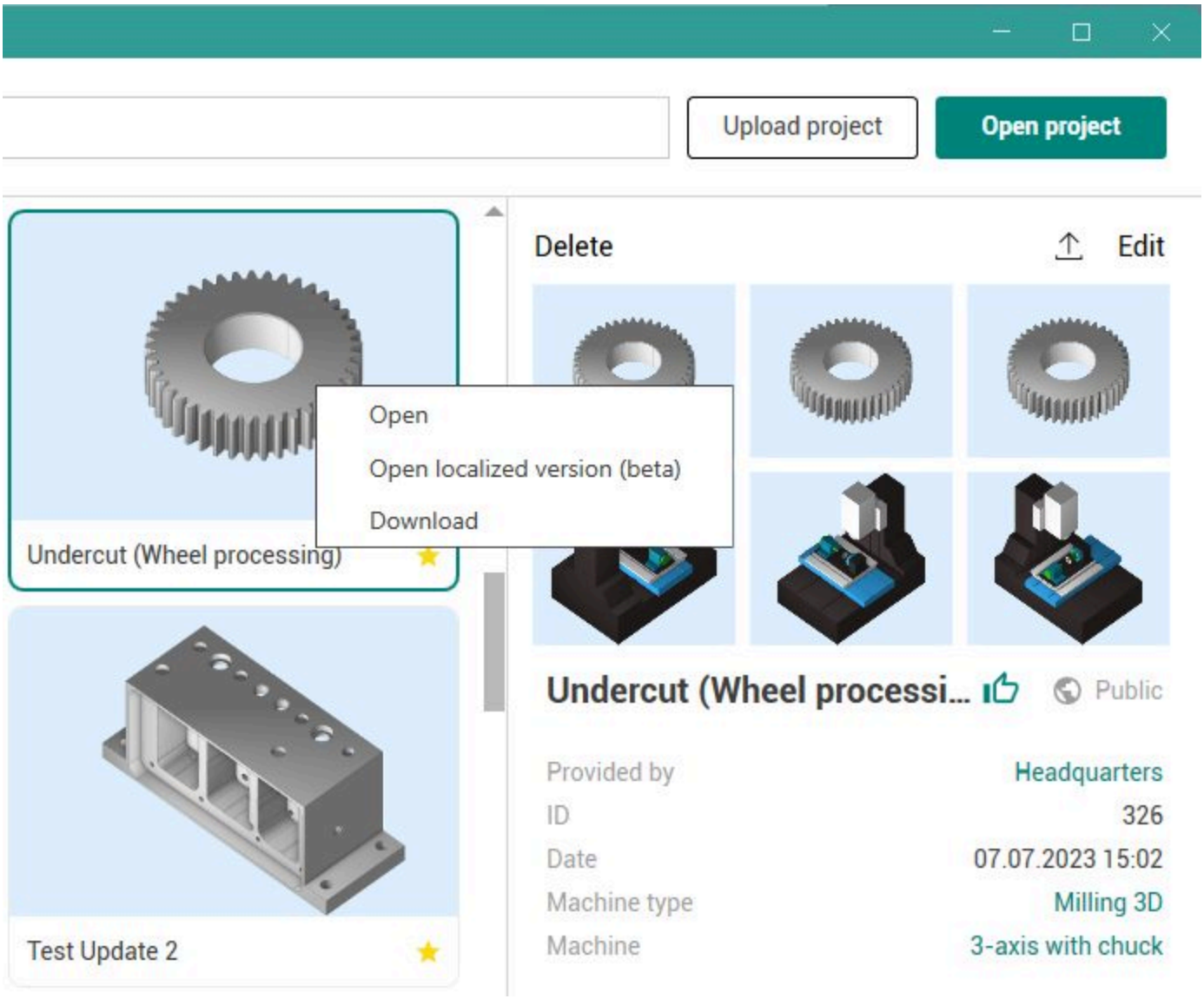
All green values in the project details panel are clickable. Clicking a value will add it to the filters list, and clicking it again in the filters panel will remove it.



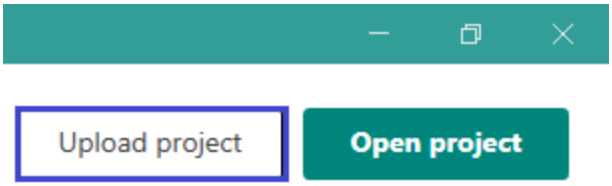
Right-click on a project to open the list of actions. From this menu, you can:

- Open the project
- Open a localized version of the project
- Download the project file

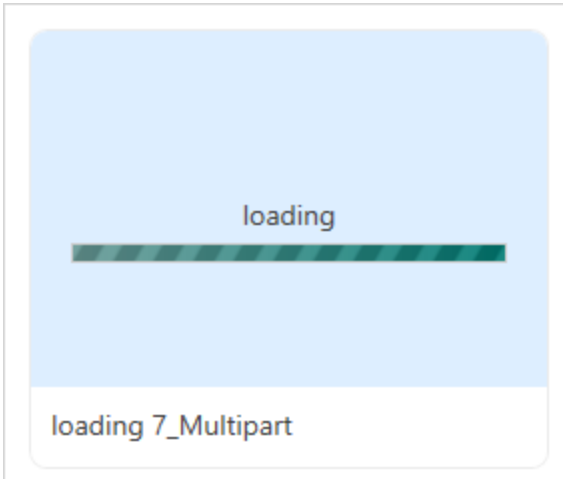
You can also open a project by clicking the Open Project button or by double-clicking on the project.

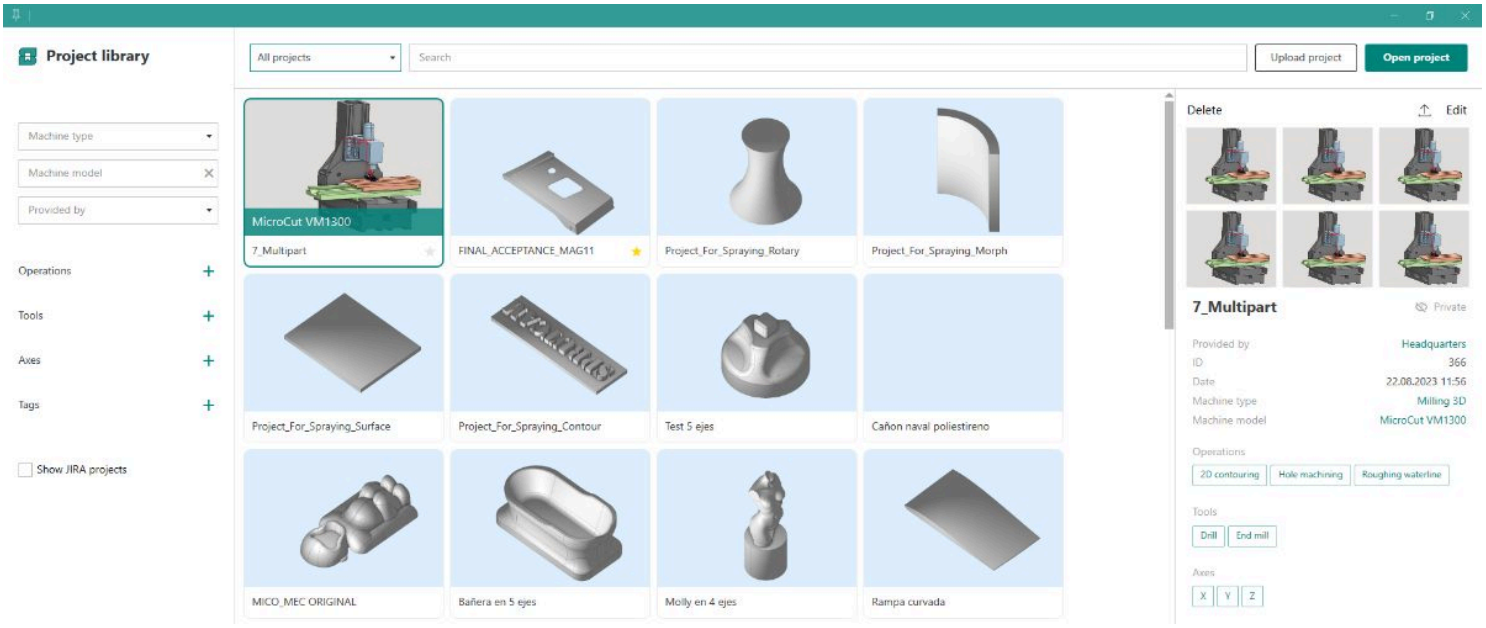


To upload your own project, click the Upload Project button.



Once you have selected a project, you will see it loaded into the Project Library.

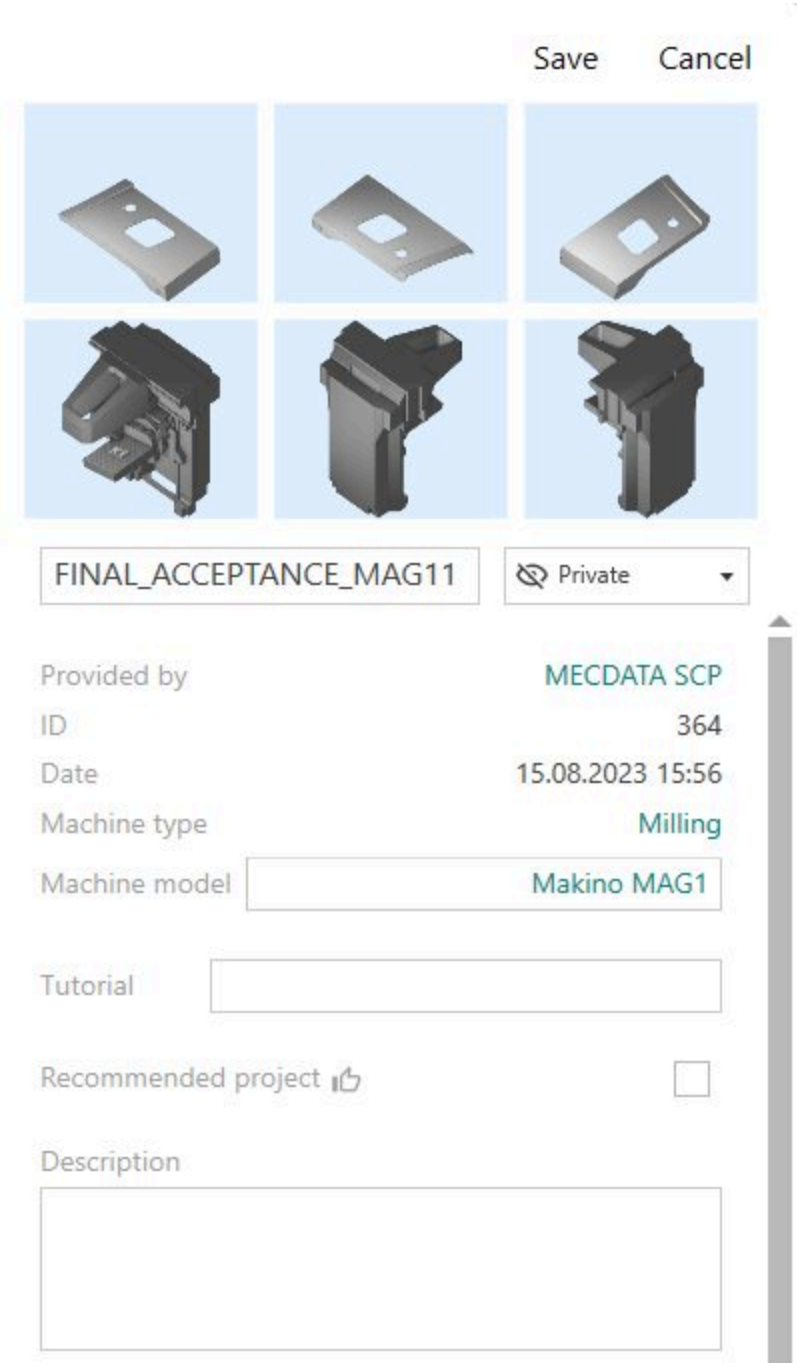




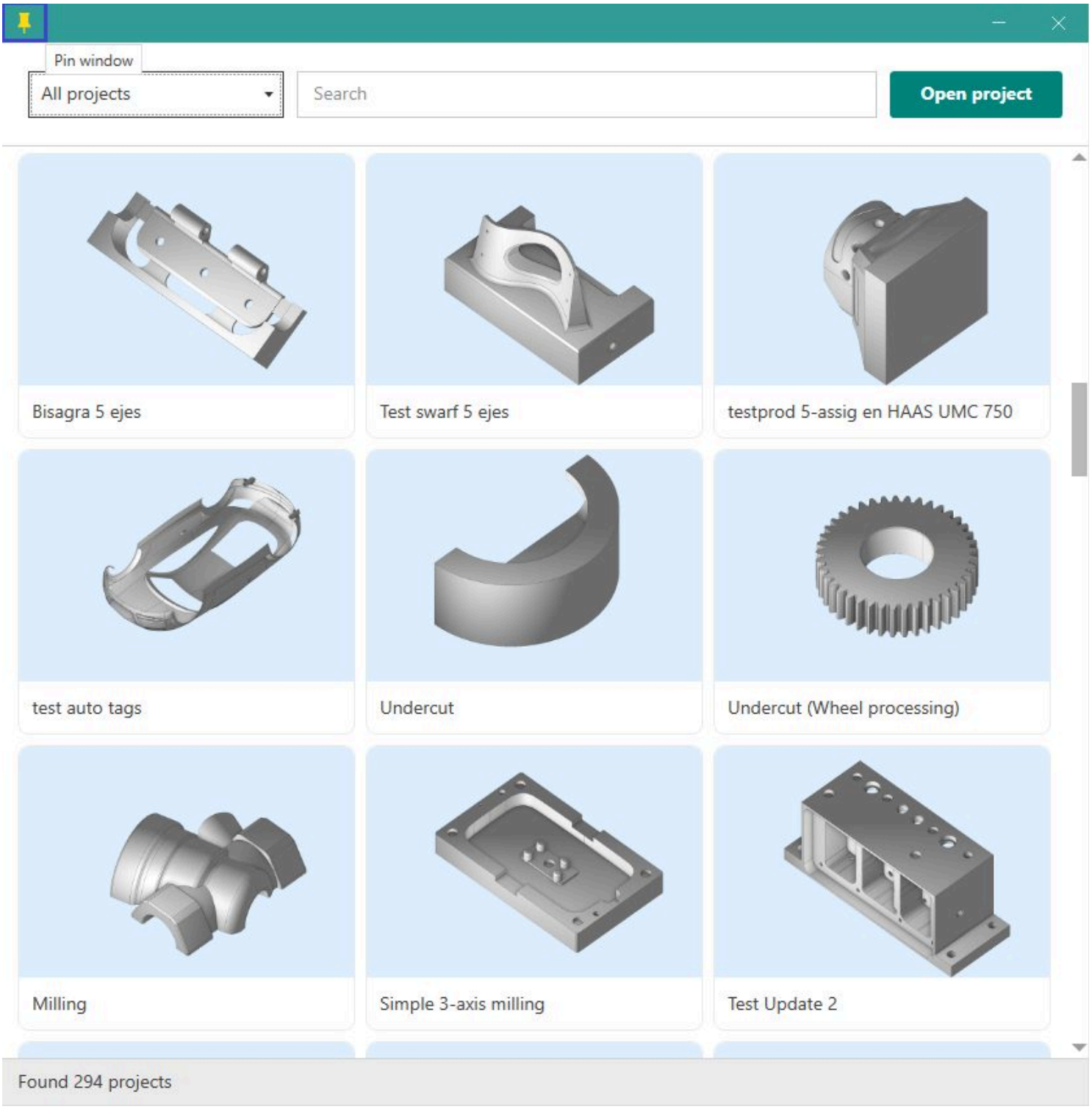
You can delete or edit your projects.



When editing, you can add a description or change the project name.



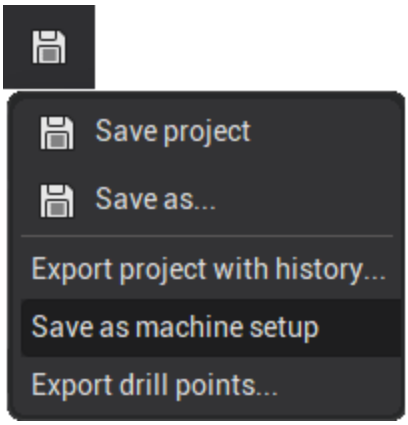
The Pin window button in the top-left corner of the window switches the Project Library to compact mode and keeps the window on top.


Pin window
All projects
Search
Open project
Bisagra 5 ejes
Test swarf 5 ejes
testprod 5-assig en HAAS UMC 750
test auto tags
Undercut
Undercut (Wheel processing)
Milling
Simple 3-axis milling
Test Update 2
Found 294 projects

Machine setup (project template)

The machine setup file is designed to help you quickly create new projects from a template.

You can save a project as a machine setup file (*.stms) by selecting the Save as Machine Setup option from the drop-down menu of the project save button on the application toolbar.

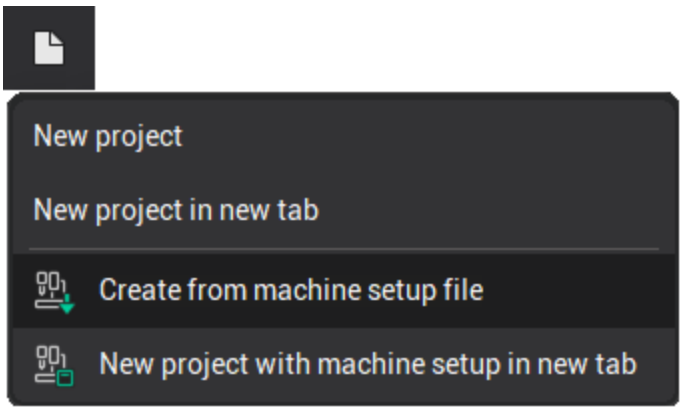


This allows you to efficiently create new projects using the saved setup file as a template.

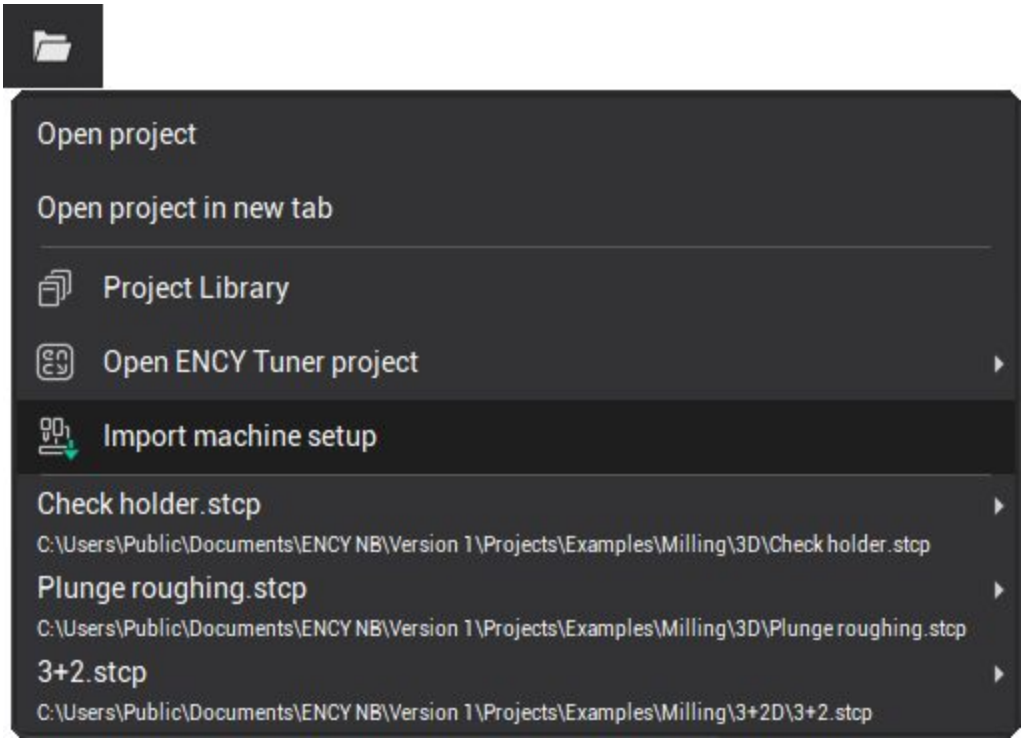
The following data is stored in the machine setup file:

- Machine
- Stages and part (as a group of operations)
- Fixtures (including their position)
- Tools
- Approaches/returns
- Workpiece coordinate systems
- Types of tool blocks and their placement in the turret

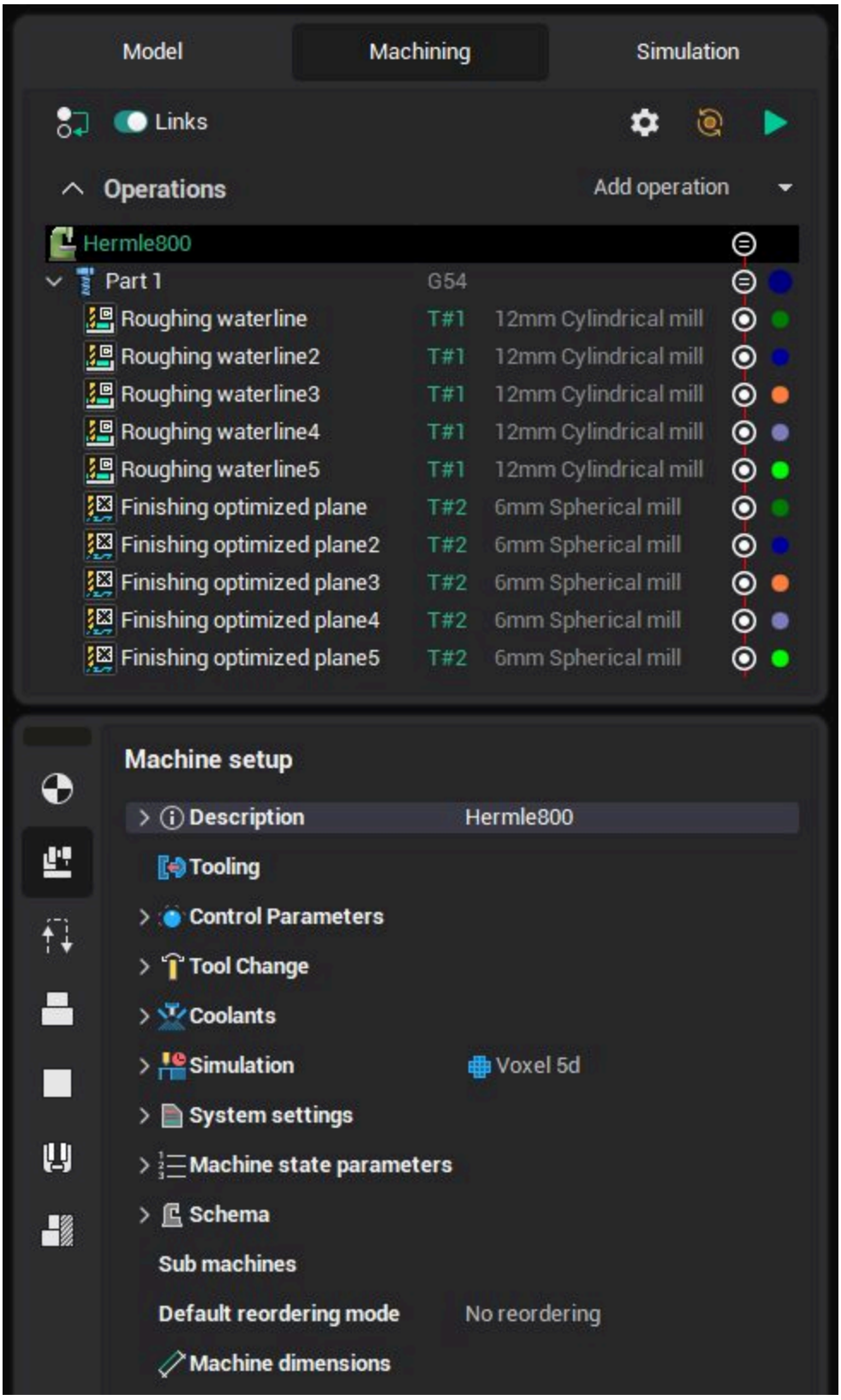
A new drop-down menu will appear under the New Project button on the application toolbar, allowing you to select one of the recently used machine setup files. Alternatively, you can choose a different setup file by selecting the Create from Machine Setup File option.



You can also import a machine setup file into a current project. To do this, use the Import Machine Setup option from the drop-down menu of the open button on the application toolbar.



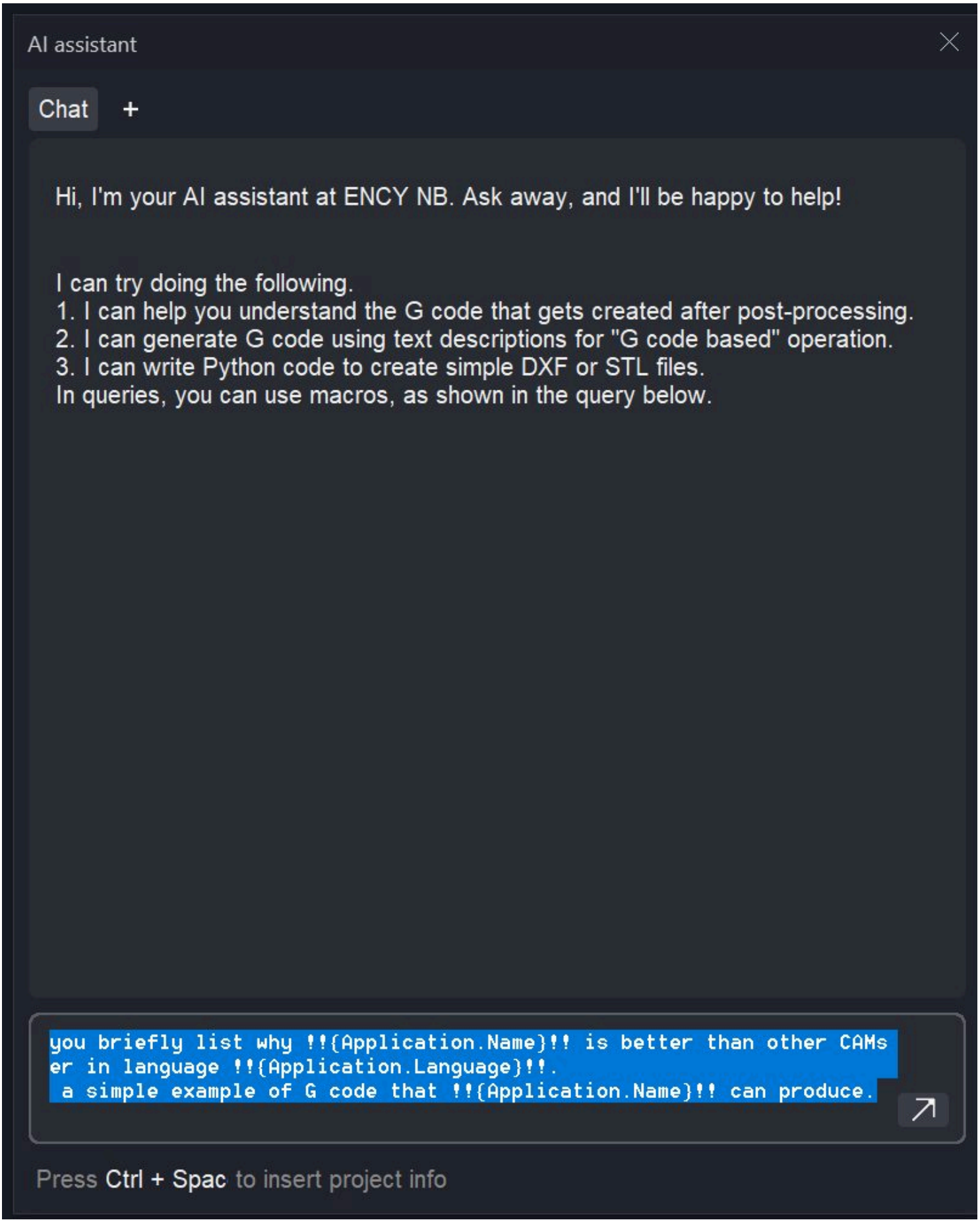
During import, all data from the machine setup file will be added to the current project, except for the machine itself. If the project's machine matches the machine in the setup file, the machine parameters will be updated from the file. If the machines are different, the Machine dimensions and Tooling parameters will be updated, if applicable.


Model
Machining
Simulation
Links
Operations
Add operation
Hermle800
Part 1 G54
Roughing waterline T#1 12mm Cylindrical mill
Roughing waterline2 T#1 12mm Cylindrical mill
Roughing waterline3 T#1 12mm Cylindrical mill
Roughing waterline4 T#1 12mm Cylindrical mill
Roughing waterline5 T#1 12mm Cylindrical mill
Finishing optimized plane T#2 6mm Spherical mill
Finishing optimized plane2 T#2 6mm Spherical mill
Finishing optimized plane3 T#2 6mm Spherical mill
Finishing optimized plane4 T#2 6mm Spherical mill
Finishing optimized plane5 T#2 6mm Spherical mill
Machine setup
Description Hermle800
Tooling
Control Parameters
Tool Change
Coolants
Simulation Voxel 5d
System settings
Machine state parameters
Schema
Sub machines
Default reordering mode No reordering
Machine dimensions

AI assistant

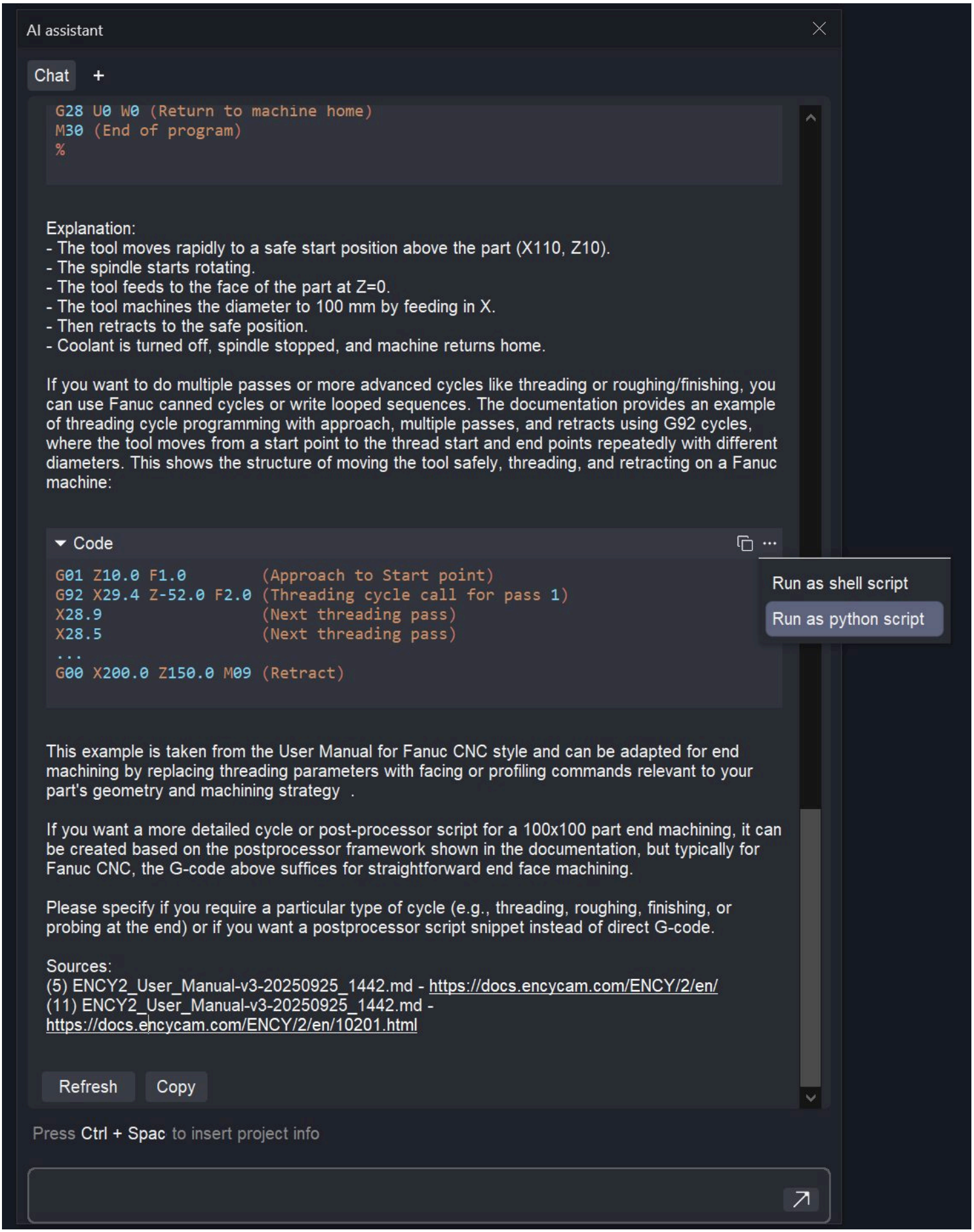
AI virtual assistant helps users with various aspects of the CAM workflow. It is based on ChatGPT technology and is called AI assistant.

AI assistant greets users with the message: "Hi, I'm your AI assistant. Ask away, and I'll be happy to help!" Powered by the OpenAI API, AI assistant uses deep learning models to understand and generate natural language.


AI assistant
Chat
+
Hi, I'm your AI assistant at ENCY NB. Ask away, and I'll be happy to help!
I can try doing the following.
1. I can help you understand the G code that gets created after post-processing.
2. I can generate G code using text descriptions for "G code based" operation.
3. I can write Python code to create simple DXF or STL files.
In queries, you can use macros, as shown in the query below.
you briefly list why !!{Application.Name}!! is better than other CAMs
er in language !!{Application.Language}!!.
a simple example of G code that !!{Application.Name}!! can produce.
Press Ctrl + Spac to insert project info

You can communicate with AI assistant via text commands in multiple languages, and it will respond appropriately. Additionally, you can create requests using macros that are automatically tailored to the task at hand. The "Ctrl + Space" key combination in the prompt edit box shows possible macro values.

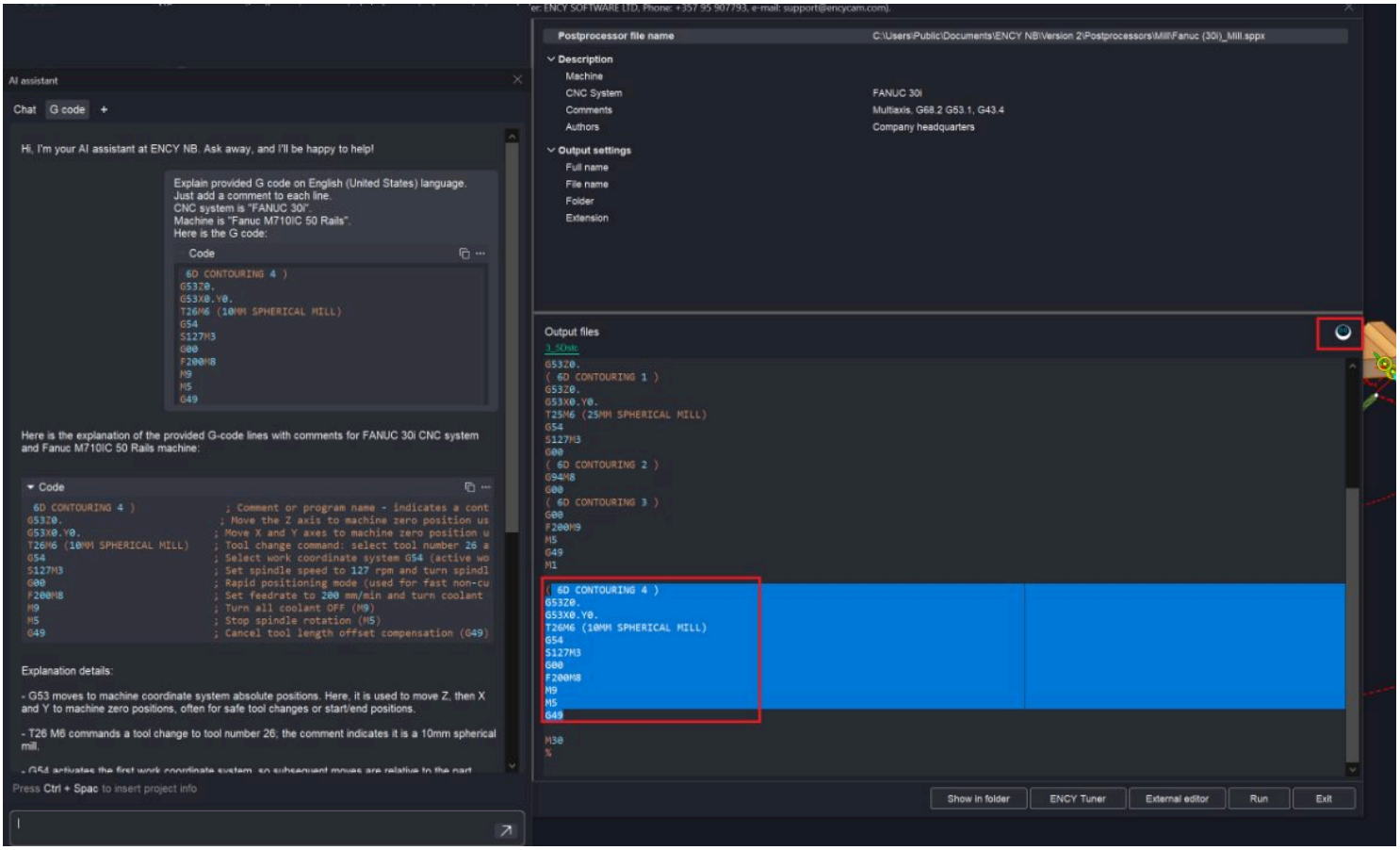
You can also perform actions on blocks of code that AI assistant displays. Clicking the three-dot button in the upper right corner of the code block opens the list of actions.


AI assistant
Chat
+
G28 U0 W0 (Return to machine home)
M30 (End of program)
%
Explanation:
- The tool moves rapidly to a safe start position above the part (X110, Z10).
- The spindle starts rotating.
- The tool feeds to the face of the part at Z=0.
- The tool machines the diameter to 100 mm by feeding in X.
- Then retracts to the safe position.
- Coolant is turned off, spindle stopped, and machine returns home.
If you want to do multiple passes or more advanced cycles like threading or roughing/finishing, you can use Fanuc canned cycles or write looped sequences. The documentation provides an example of threading cycle programming with approach, multiple passes, and retracts using G92 cycles, where the tool moves from a start point to the thread start and end points repeatedly with different diameters. This shows the structure of moving the tool safely, threading, and retracting on a Fanuc machine:
Code
G01 Z10.0 F1.0 (Approach to Start point)
G92 X29.4 Z-52.0 F2.0 (Threading cycle call for pass 1)
X28.9 (Next threading pass)
X28.5 (Next threading pass)
...
G00 X200.0 Z150.0 M09 (Retract)
Run as shell script
Run as python script
This example is taken from the User Manual for Fanuc CNC style and can be adapted for end machining by replacing threading parameters with facing or profiling commands relevant to your part's geometry and machining strategy .
If you want a more detailed cycle or post-processor script for a 100x100 part end machining, it can be created based on the postprocessor framework shown in the documentation, but typically for Fanuc CNC, the G-code above suffices for straightforward end face machining.
Please specify if you require a particular type of cycle (e.g., threading, roughing, finishing, or probing at the end) or if you want a postprocessor script snippet instead of direct G-code.
Sources:
(5) ENCY2_User_Manual-v3-20250925_1442.md - https://docs.encycam.com/ENCY/2/en/
(11) ENCY2_User_Manual-v3-20250925_1442.md - https://docs.encycam.com/ENCY/2/en/10201.html
Refresh
Copy
Press Ctrl + Spac to insert project info

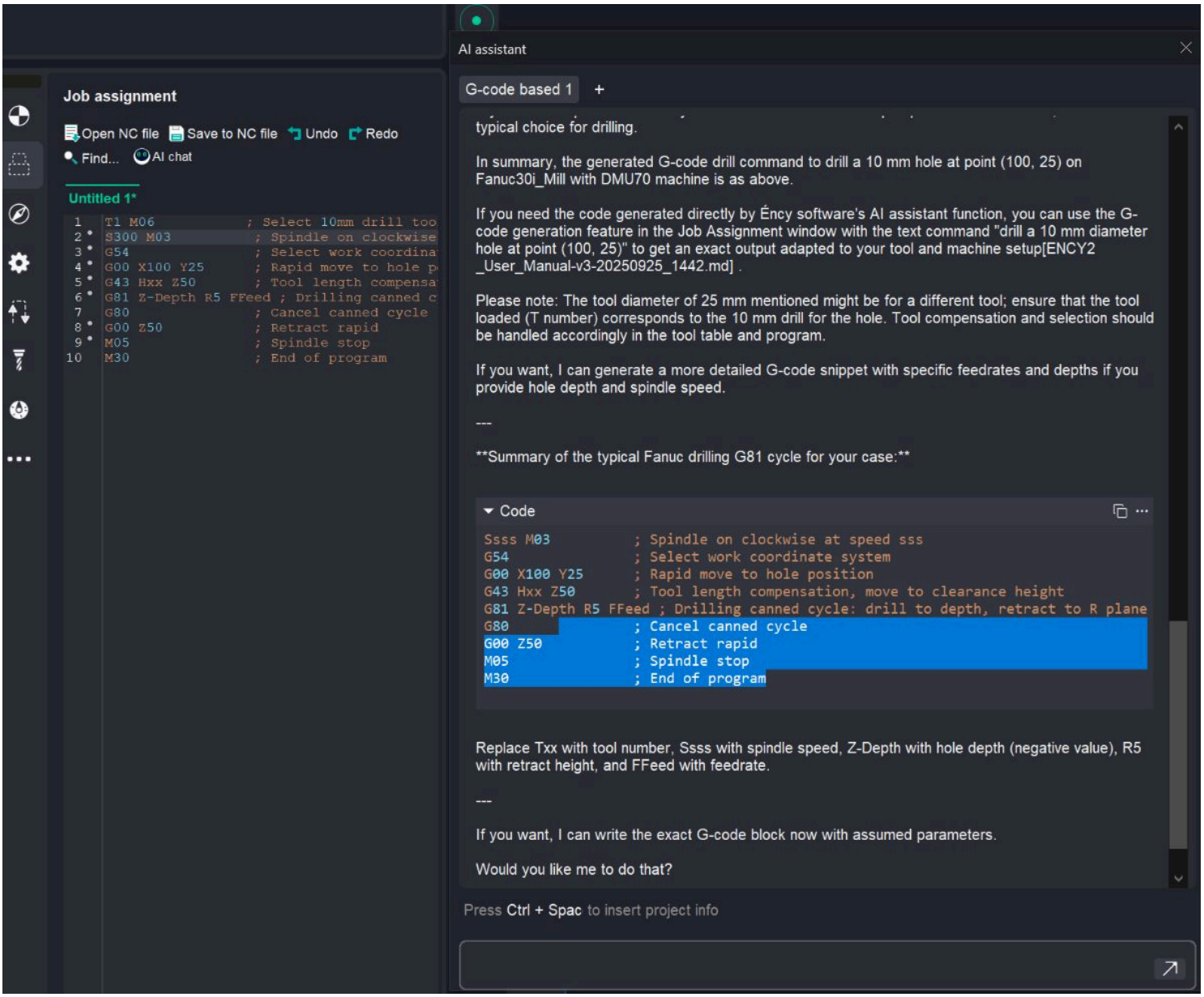
The list of available actions and macros depends on the context from which each chat tab was opened. For instance, if the chat was opened from the G-code generation window, the *!! {GCodeWindow.SelectedText}!!* macro will be available, substituting selected text from the window in the request.

AI assistant can perform the following tasks:

- Explain G-code: AI assistant can clarify any G-code line or command. After selecting the text, click the "AI Assistant" button inside the G-code generation window.



- Generate G-code: Using text descriptions, AI assistant can generate G-code. For example, typing "*drill a 10 mm diameter hole at point (100, 25)*" will produce the corresponding code. In the Job Assignment window of the G-code-based operation, click the AI Assistant button. In this case, elements will appear in the list of code block actions that allow you to quickly apply the generated block to the operation, calculate it, and immediately see the result.



- Write Python code: AI assistant can create.dxf or.stl files using Python. Just ask it to generate Python code to create a.dxf file in the form you need, then click Run as Python Script in the actions list of the generated code block. If the script is correct, the resulting file will be saved in the "*My Documents*" folder, under the $(ApplicationName)\AIActions subfolder. You can import it in the standard way. If the script is incorrect, copy the error message and ask the assistant to resolve the problem.

AI assistant

Chat +

1. I can help you understand the G code that gets created after post-processing.
2. I can generate G code using text descriptions for "G code based" operation.
3. I can write Python code to create simple DXF or STL files.
In queries, you can use macros, as shown in the query below.

Generate Python code to create a .dxf file

Here is a simple example of Python code that creates a .dxf file with basic geometry (lines, circle) using the popular `ezdxf` library, which is commonly used to generate DXF files in Python:

It looks like you want a simple Python example to create a .dxf file with basic geometry such as lines and a circle using the ezdxf library. I can provide you a sample code snippet for that. Here it is:

Code

```
# Create a new DXF document
doc = ezdxf.new(dxfversion='R2010')

# Add a modelspace to the document
msp = doc.modelspace()

# Add a line from point (0, 0) to (10, 0)
msp.add_line((0, 0), (10, 0))

# Add a circle with center at (5, 5) and radius 3
msp.add_circle(center=(5, 5), radius=3)

# Save the DXF document to a file
doc.saveas("example.dxf")
```

This code initializes a new DXF document, adds a modelspace where graphical entities can be placed, then adds a line and a circle to this modelspace. Finally, it saves the file as "example.dxf".

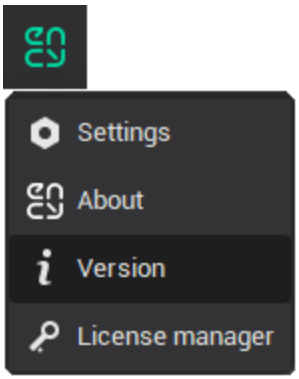
If you want, I can also help you with more advanced examples or explain how to add other shapes or layers. Just let me know!

Press Ctrl + Spac to insert project info

- Provide reference information: AI assistant offers information on industrial robots or CNC machines when creating kinematic schemes in MachineMaker. You can ask the AI assistant for any information about the robot or CNC machine you are using, such as its dimensions, specifications, capabilities, limitations, and more.

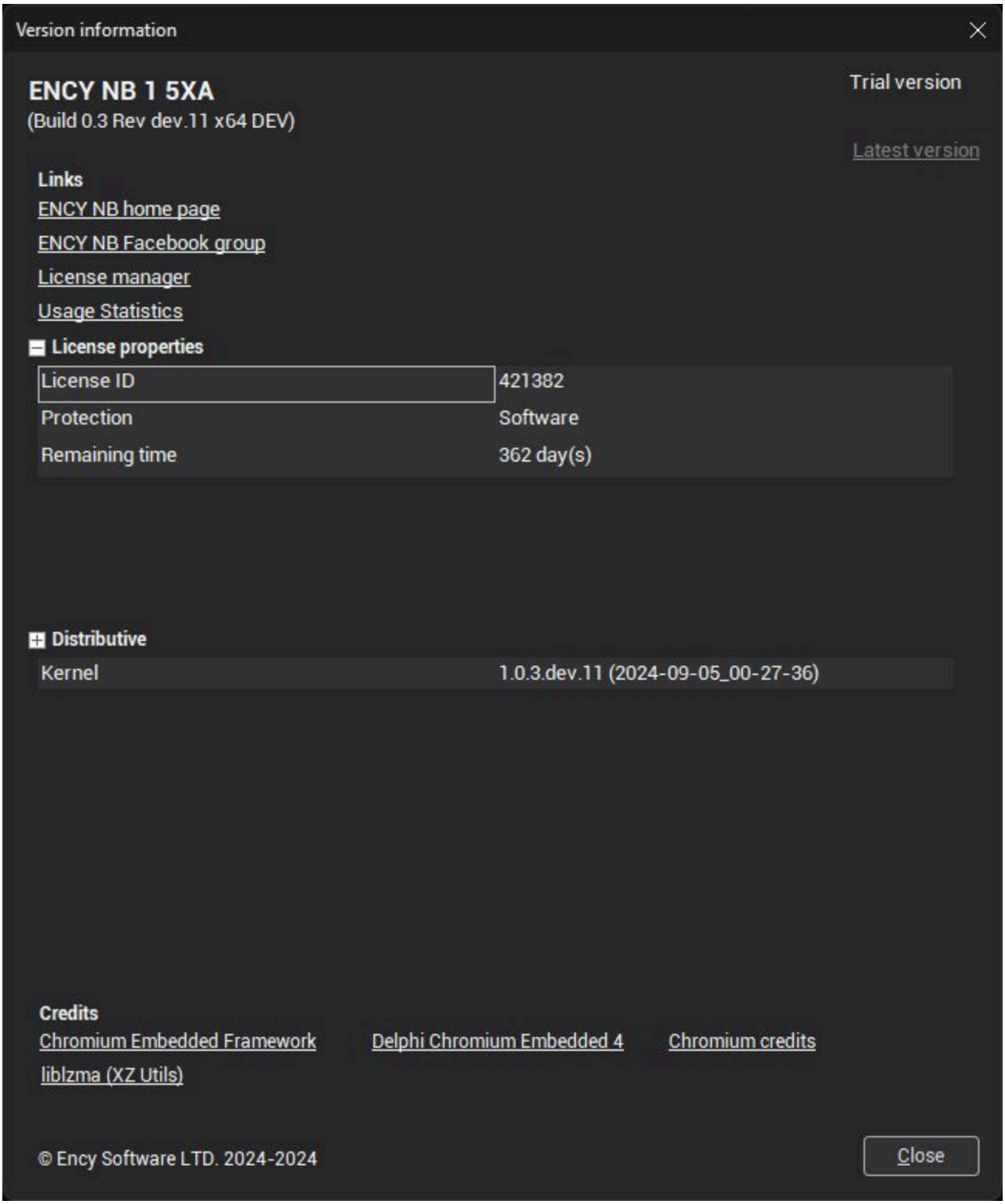
Version info window

The Version information window can be accessed from the first button on the top of the application toolbar.



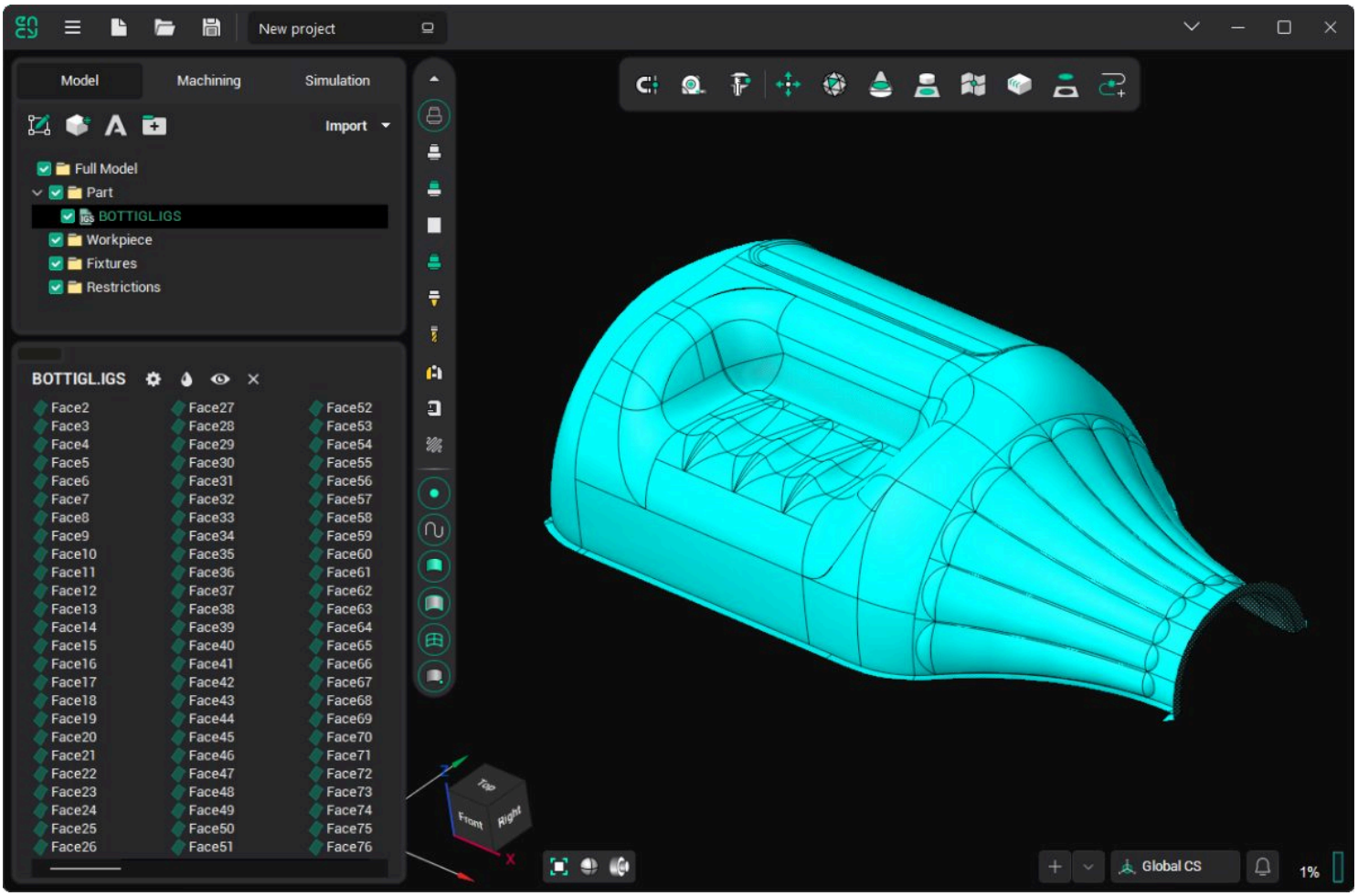
In this window, you can view:

- The application name.
- The application build identifier.
- Useful links to our web resources.
- Basic license properties.
- A version list for each application module.
- Information about the software dealer.
- Copyright information.


Version information
ENCY NB 1 5XA
(Build 0.3 Rev dev.11 x64 DEV)
Trial version
Latest version
Links
ENCY NB home page
ENCY NB Facebook group
License manager
Usage Statistics
License properties
License ID
421382
Protection
Software
Remaining time
362 day(s)
Distributive
Kernel
1.0.3.dev.11 (2024-09-05_00-27-36)
Credits
Chromium Embedded Framework
Delphi Chromium Embedded 4
Chromium credits
liblzma (XZ Utils)
© Ency Software LTD. 2024-2024
Close

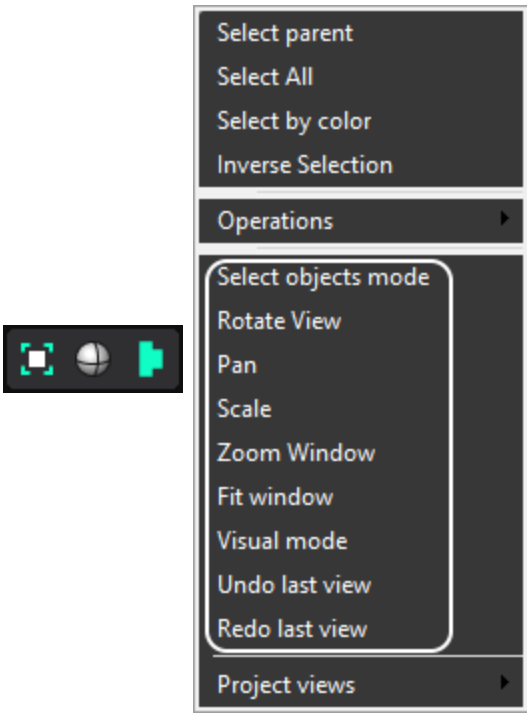
Graphic window and visualization control

The main part of the screen is occupied by a graphic window. It can display objects such as geometric models, machines, workpieces, toolpaths, etc.



Some visualization control hotkeys can be changed in the System settings window on the Visualization tab.

Visualization parameters can be changed using the View Control toolbar on the left side of the main window or via the context menu of the graphic screen:



The first four elements switch the actions performed by the left mouse button:

- Select objects mode: Left-click will select objects from the screen (this is the default state).
- Rotate view: Moving the mouse while pressing the left button will rotate the view.
- Pan: Moving the mouse while pressing the left button will shift the view.
- Scale: Moving the mouse while pressing the left button will scale the view.

However, all these actions can be easily performed without changing the mode:

- To rotate the view, move the mouse with the right button pressed.
- To shift the view, move the mouse while pressing the middle button.
- Rotate the mouse wheel to zoom.

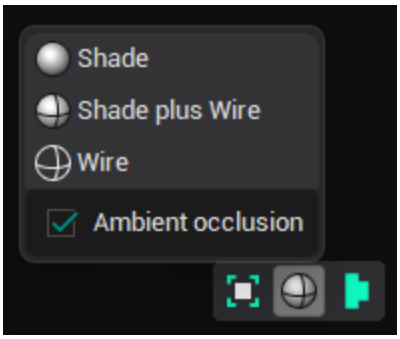
There are additional methods for changing the view using hotkeys. See the following topics: Window zoom, Interactive rotate, Interactive pan,Interactive zoom in-out, Zoom extents.

For quick scaling to display all visible objects, use the button or the Fit window menu item.



The view cube and standard views are described on the View cube page.

Use the Undo/ Redo menu items to quickly return to the previous or next view.



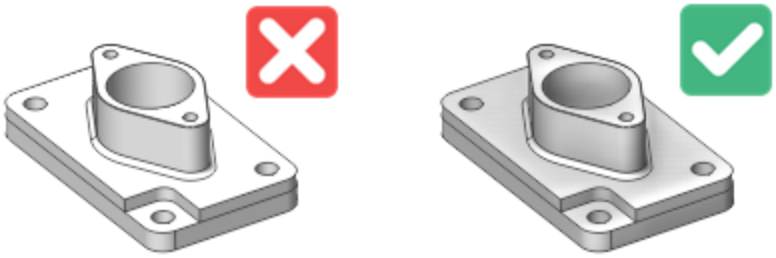
Display mode settings:

The  button with the drop-down menu allows you to switch the display mode of three-dimensional objects:

- Shade: shaded without edges
- Shade plus Wire: shaded with edges
- Wire: edges only

Clicking on the button itself toggles these modes alternately.

Ambient occlusion is a shading and rendering technique used to calculate how exposed each point in a scene is to ambient lighting. See the example below.

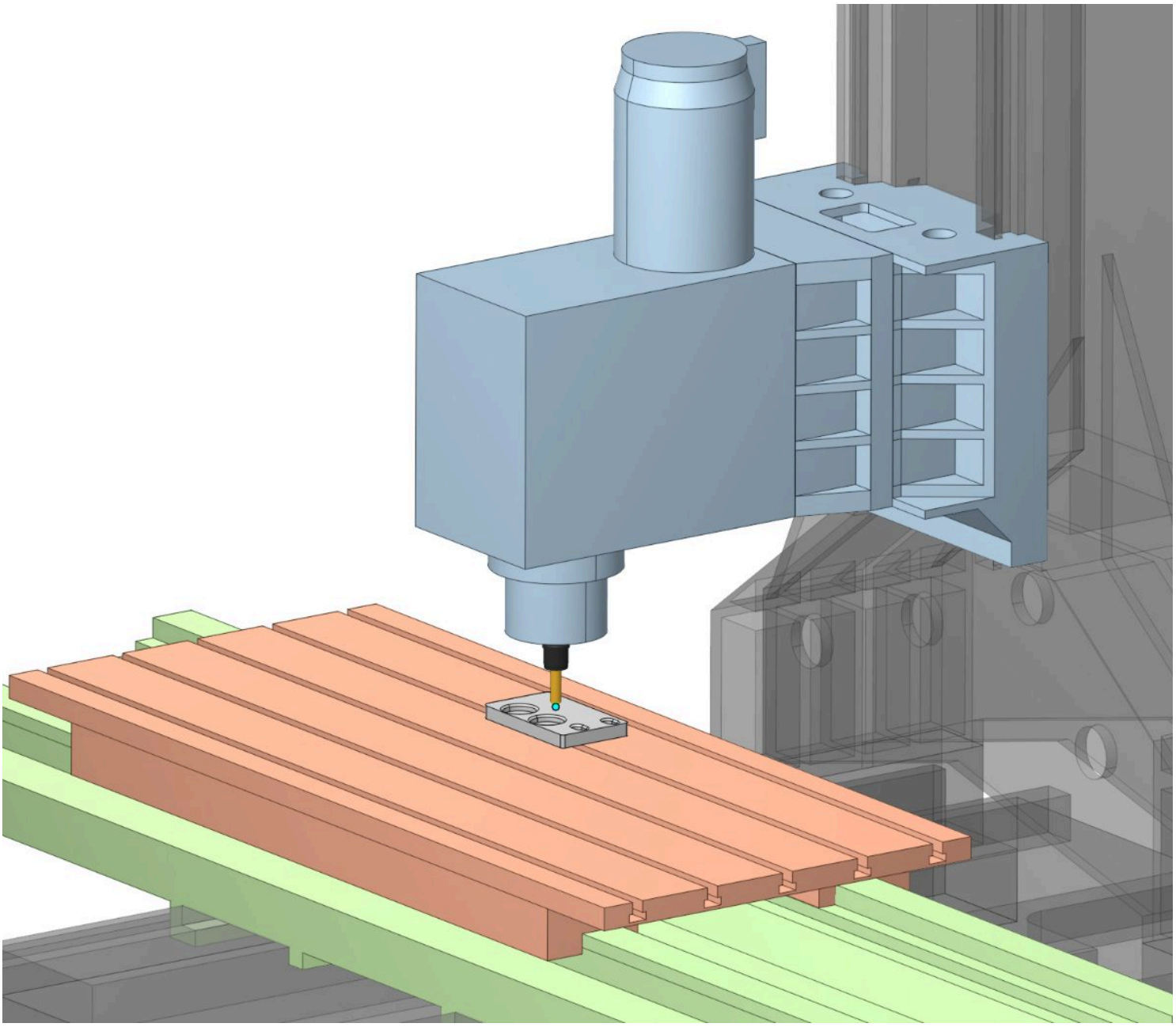


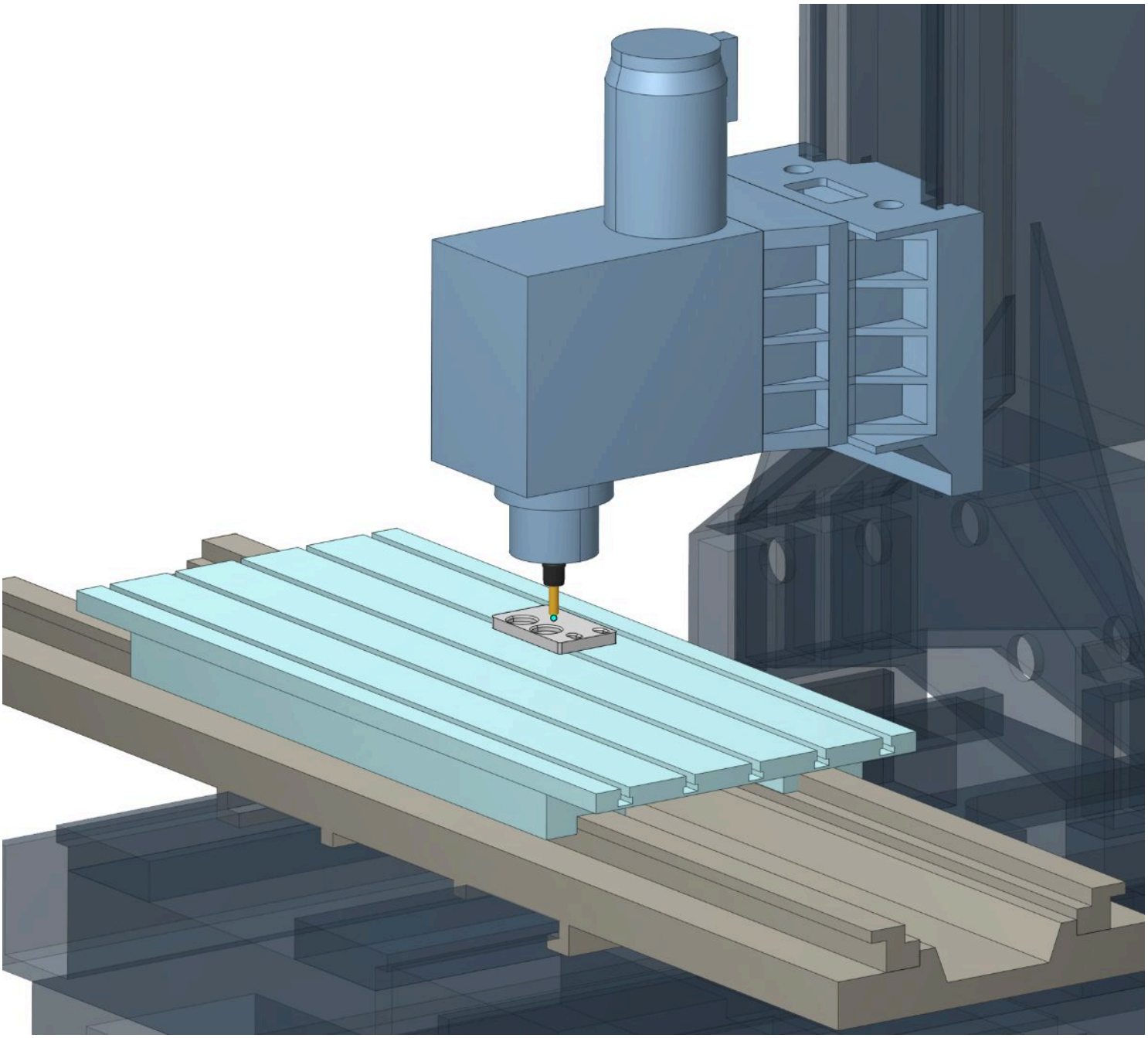
Ground plane. See View cube menu.

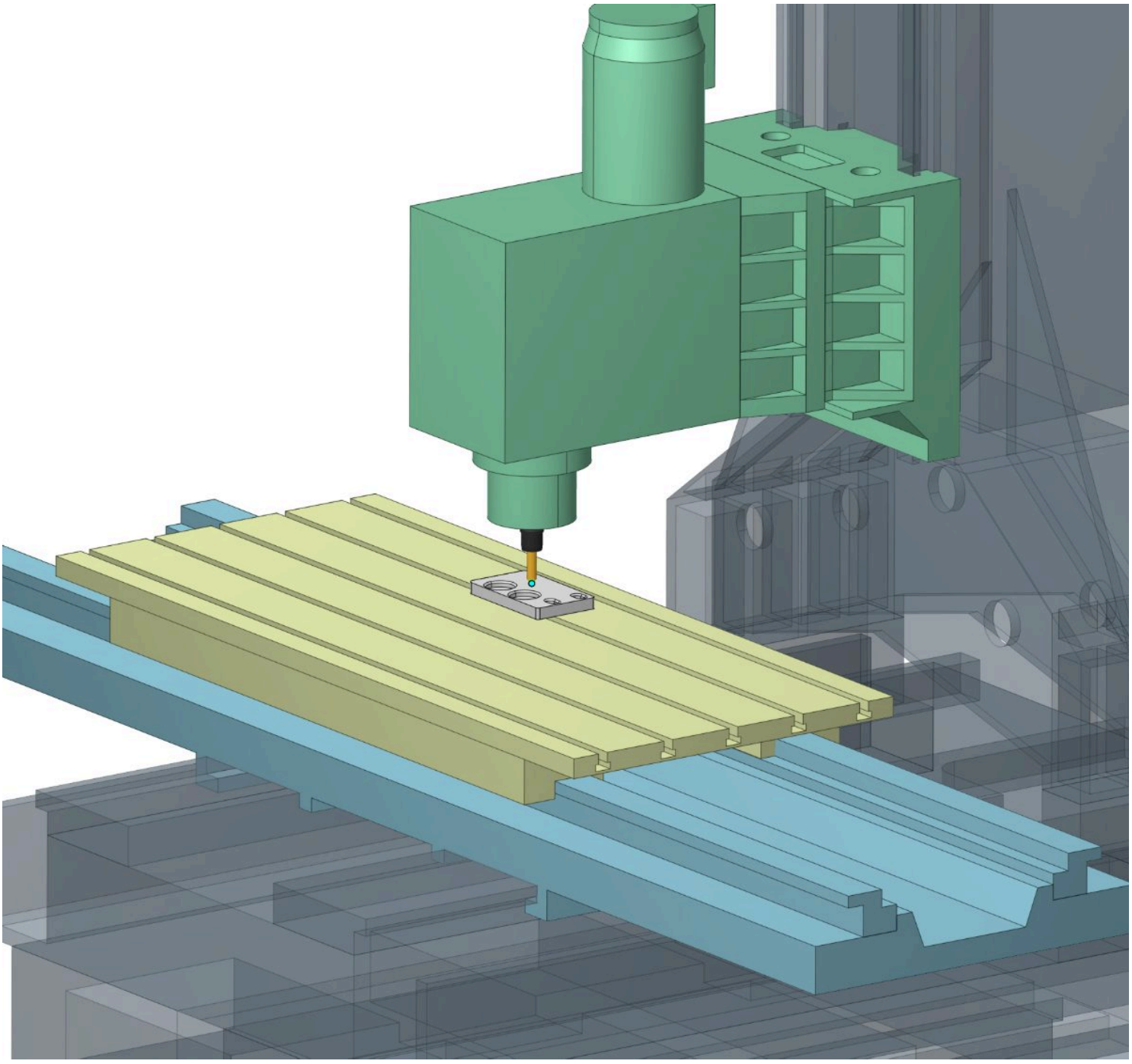
Equipment color schemes. Control the colors used to display the machine. Contrasting colors distinguish kinematic links, while static components remain neutral.

- Project Colors. Preserves the original model colors.
- Gunmetal. Applies an industrial color palette.
- Restrained. Applies an alternative industrial color palette.

Robot manipulators retain their native manufacturer colors in every scheme.



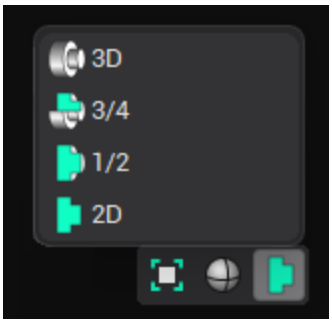




Examples of equipment color schemes in the 3D view.

Display settings for individual objects are available on the Visibility panel. Details

Turn mode:



The  button above defines a visualization mode for revolution bodies only:

- 3D
- 3D without a quarter
- Half of 3D
- 2D axis section.

Interactive linear axis ruler:

The interactive linear axis ruler displays the current coordinate and travel limits when a machine node is dragged along a linear axis in the 3D view. Details

View cube



The standard views panel in the main window may vary depending on the type of machine, such as turning or milling, etc. When one of the buttons is selected, the corresponding view vector is set in the graphic window. If the view vector in the graphic window is changed by another method, the selected button on the panel will automatically release.

Clicking the [Middle Mouse Button] (Mouse Wheel) sets one of the nearest standard views.

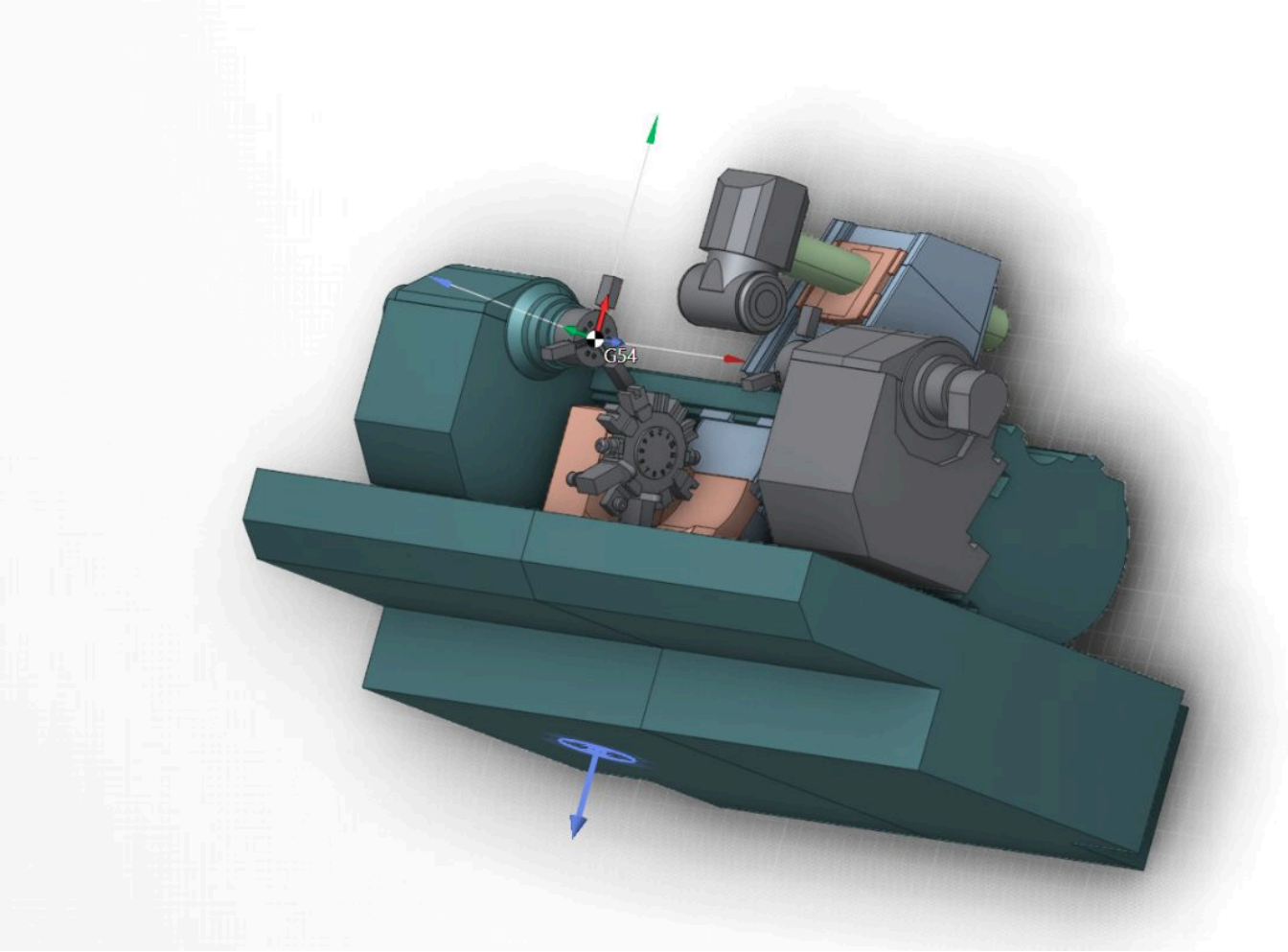
View cube menu

When the cursor hovers over the view cube, the ⋮ button appears in its upper-left corner. The button opens a menu that turns the scene environment on or off and sets the position of the machine on the floor.

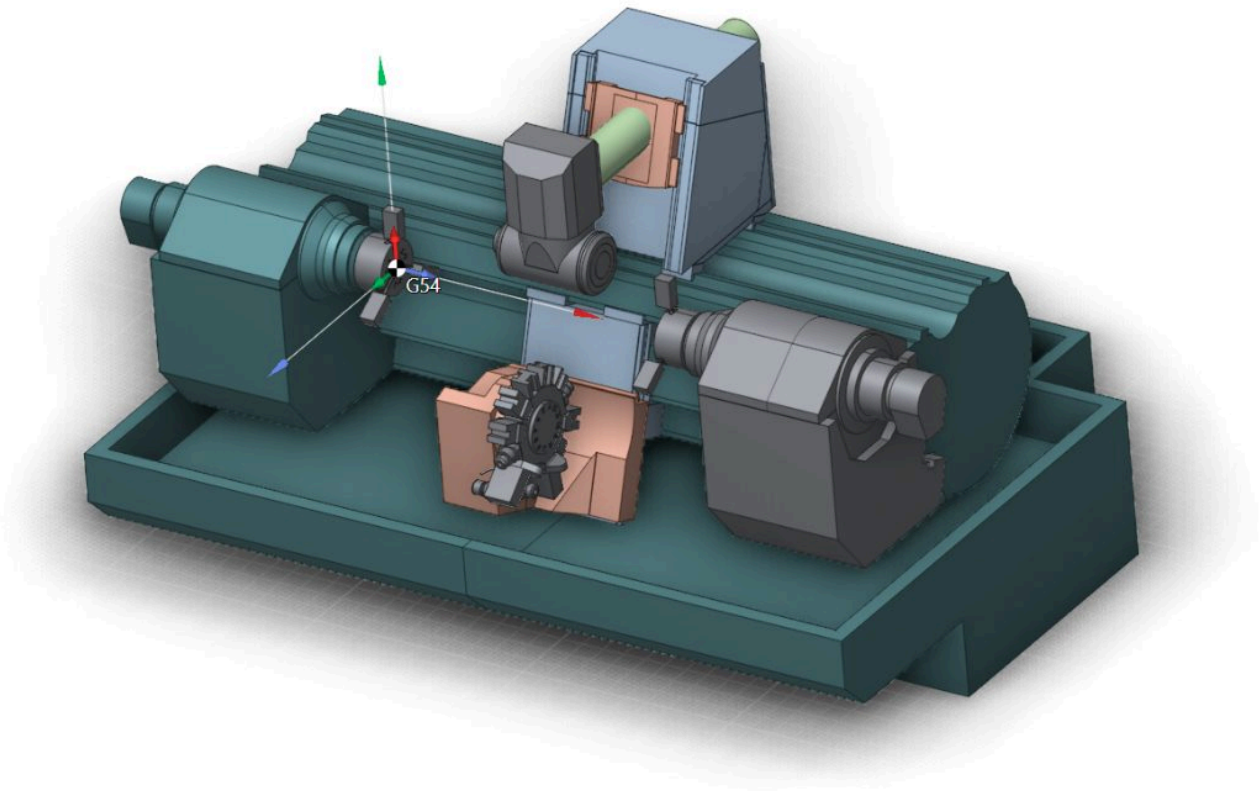
Environment. Shows or hides the floor in the graphic window. In the Studio theme, the studio walls and the shadows on the floor are displayed together with the floor. The selected state is kept between sessions. The item is unavailable while the sketch grid is turned on.

The floor commands are available when Environment is turned on and the machine is displayed in the graphic window.

Define floor. Places the machine on the floor by the selected surface. After selecting the command, click the surface of the machine that stands on the floor. A marker under the cursor shows the point and the normal direction of the surface. To cancel the selection, press Esc, click the right mouse button, or click outside the graphic window.



Machine after defining a custom floor.



Machine with the automatic floor position.

Rotate 90°. Rotates the machine on the floor by 90° about the vertical axis.

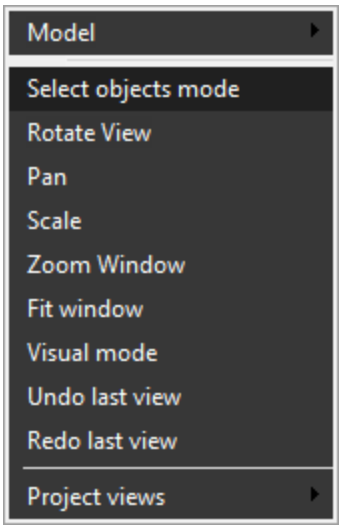
Reset. Restores the automatic floor position defined by the bottom of the machine. The command is available when the floor position has been changed with Define floor or Rotate 90°.

The floor position set with the menu commands is saved in the project. The automatic position is not written to the project.

In an overall view of the machine, the faces of the view cube follow the floor. When the view is zoomed in on the part, the cube returns to the scene coordinate system.

Objects selection mode

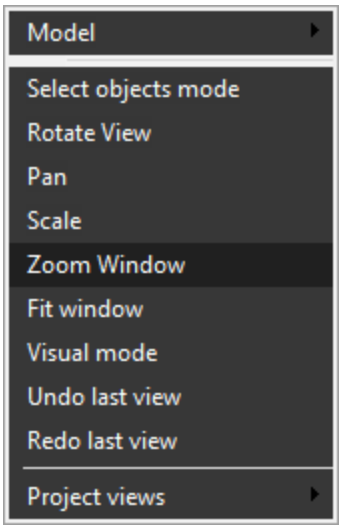
Enable the "Select Objects Mode" option in the drop-down menu of the viewport to activate object selection in the graphical window (this mode is enabled by default). The mouse pointer will change to its usual form . After that, objects can be selected with the mouse cursor in the graphical window. When you move the pointer over the screen, the object beneath it will be highlighted if it can be selected. A [Left Mouse Button] click will select the highlighted object.



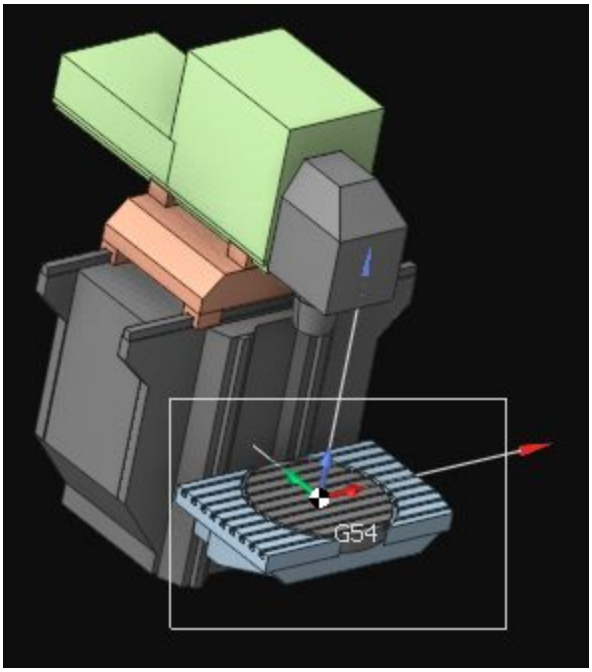
See the Objects selection topic for more advanced methods of selecting objects from the screen.

Window zoom

Zooms to display an area defined by two opposite corners of a rectangular window. Select the Zoom Window item from the pop-up menu of the graphic window to activate the window zoom mode.



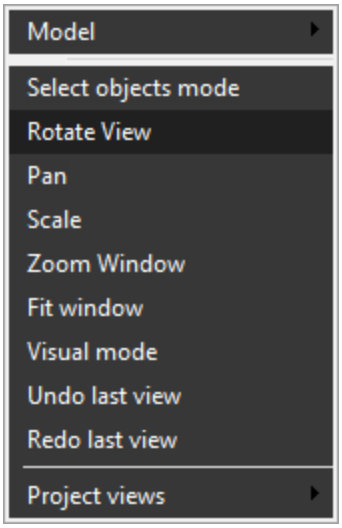
Specify the first corner of the window, hold the left mouse button, and specify the second corner. After releasing the mouse button, the area within the rectangle will be magnified to fit the viewport, and the mode will be canceled.



You can also use [Alt] + [Right Mouse Button] or the Left and Right Mouse Buttons simultaneously to achieve the same result without changing the mode.

Interactive rotate

Press and hold the [Right Mouse Button], then drag on the main visualization screen to rotate. Alternatively, use the Rotate View item from the pop-up menu of the graphic window to activate the rotate view mode.



Drag up and down to rotate around the horizontal axis; drag left and right to rotate around the vertical axis.

Press and hold the [X], [Y], or [Z] key, and drag the mouse to rotate around the X, Y, or Z axis of the active coordinate system.

Hold [Space] while rotating to loop the rotation and animate it continuously. Press the [Esc] key to exit the continuous rotation mode.

Changing the view vector is also possible via the Standard views button.

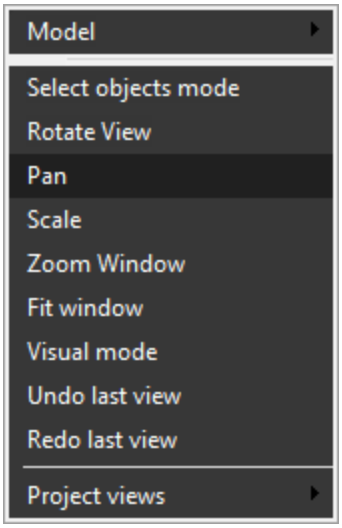
Clicking the [Middle Mouse Button] (Mouse Wheel) in the graphic window performs a complete rotation of the current view vector to the nearest standard view.

The software also supports various 3D mouse devices, such as the SpaceNavigator.

Interactive pan

Press and hold the [Middle Mouse Button] (Mouse Wheel), then drag on the main visualization screen to move graphic objects to a new location.

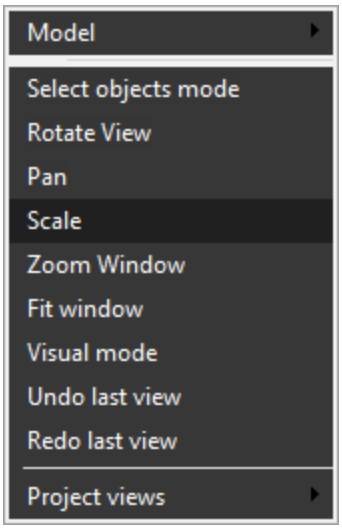
Alternatively, use the [Left Mouse Button] with the Pan mode enabled, which can be activated from the pop-up menu of the graphic window.



Interactive zoom in and out

Rotate the [Mouse Wheel] to zoom. You can also use [Ctrl] + [Right Mouse Button], then drag on the main visualization screen to zoom.

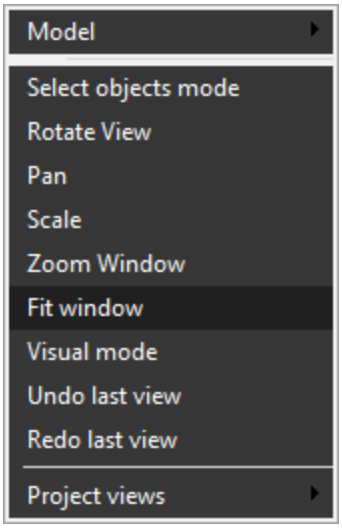
An alternative method is to activate Scale mode from the context menu of the graphic window.



Then, click and hold the [Left Mouse Button] while moving the pointer vertically in the graphic window to dynamically zoom in and out. Moving the pointer horizontally in this mode does not affect the image.

Zoom extents

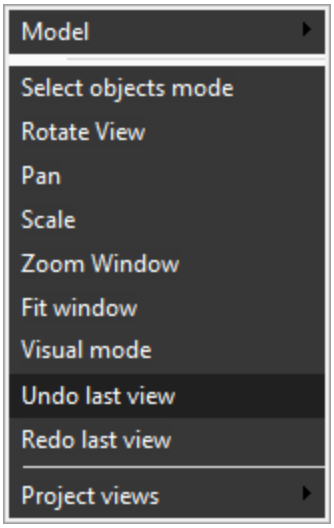
For quick scaling of the view to display all visible objects, use the  button or the Fit Window item from the pop-up menu of the graphic screen.



This function can also be activated by double-clicking the [Middle Mouse Button], but it will also activate the closest standard view.

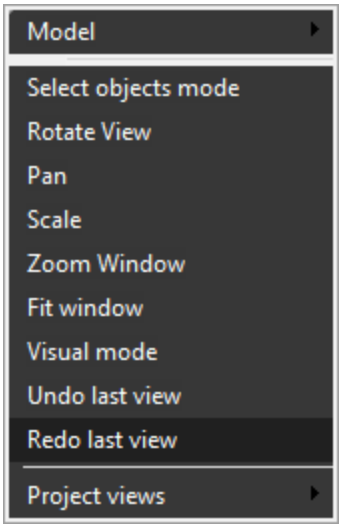
Undo view

The Undo View feature allows the user to restore the parameters of the active viewport (scale, visualization vector, etc.) to their previous state. Use the pop-up menu of the graphic window to activate it.



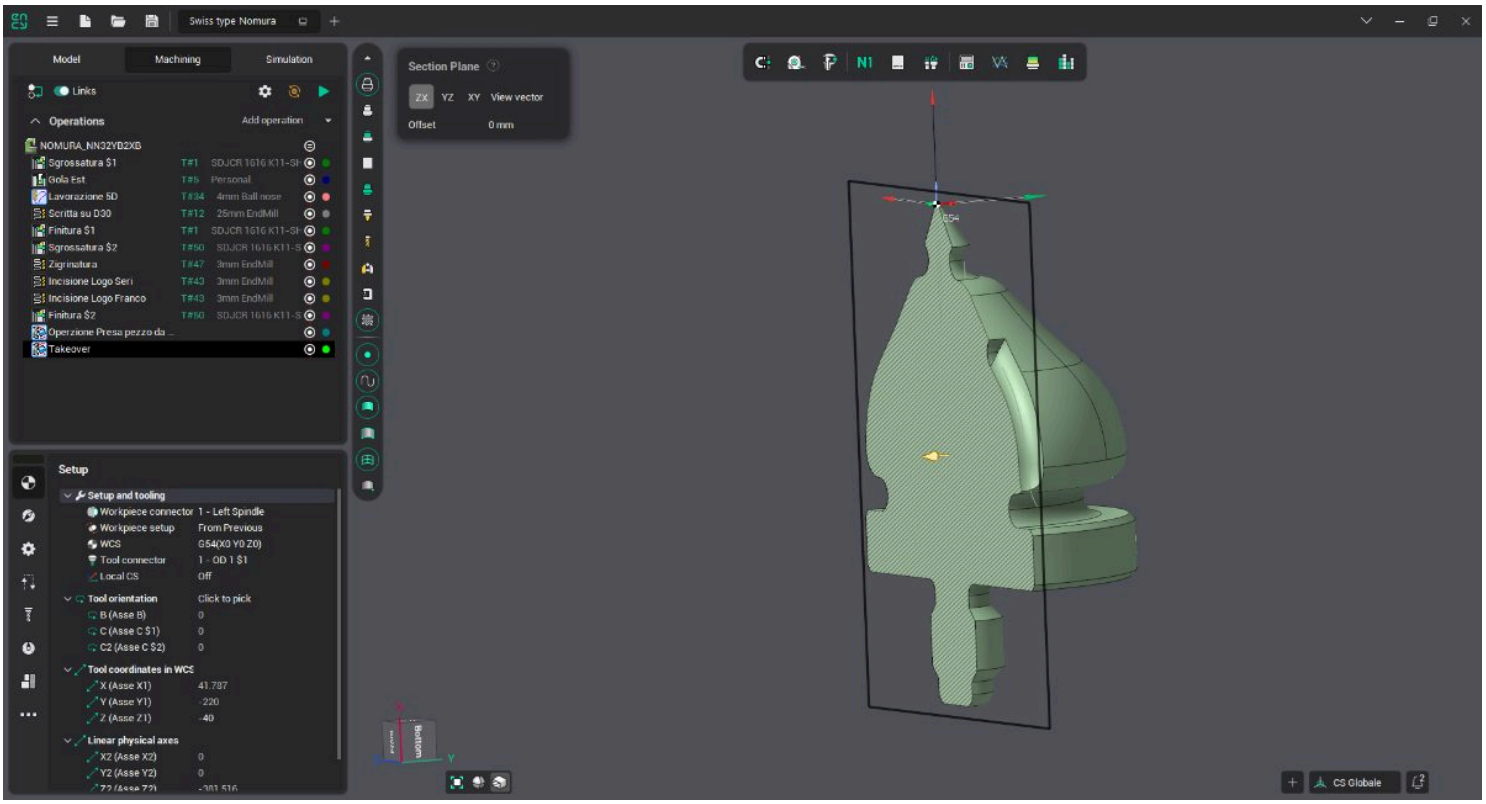
Redo view

The Redo View feature allows the user to restore the parameters of the active viewport (scale, visualization vector, etc.) to the state that was previously discarded by the Undo view feature. Use the pop-up menu of the graphic window to activate it.



Section plane

The Section  button enables (or disables) a special mode for displaying the internal structure of an object. When turned on, an infinite plane is created, at the intersection with which the cut-off part is marked with bar lines.

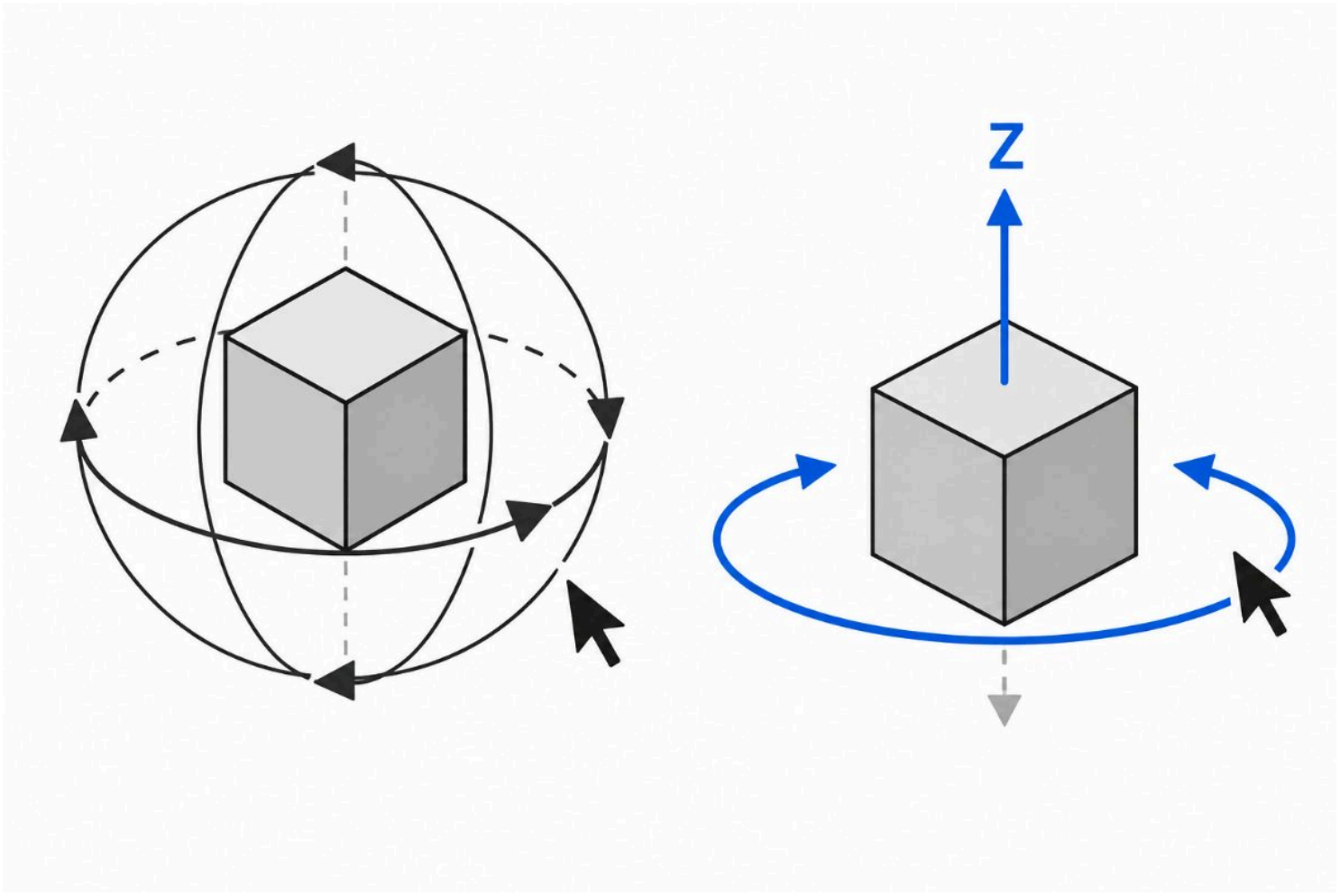


If the section plane is selected as the active object, then its settings will be available in the upper-left corner of the scene.

- The [ZX, YZ, X Y, Vector view] buttons allow you to select the direction of the section along the base axis [ZX] [YZ] [XY] or along the view vector [Vector view].
- The [Offset] field shows the offset of the section plane from the origin. You can change the value either by specifying it directly in the field itself, or by moving the *slider*.

It is also possible to build a cross section on the first selected face. To do this, select the face and click on the "Section" button.

Rotation Mode



Application area:

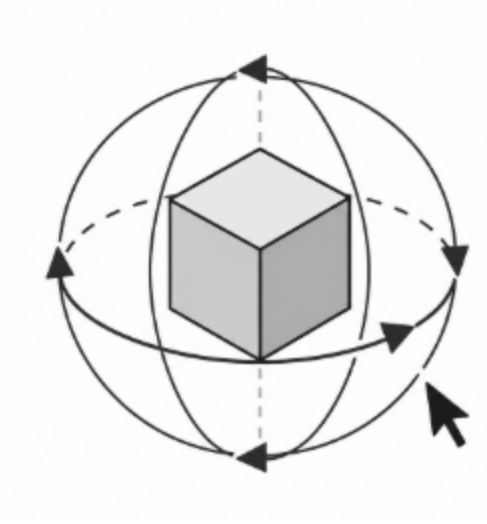
Rotation Mode has been added as a camera navigation tool for controlling viewport rotation behavior in 3D scenes.

It allows switching between free Trackball rotation and constrained Orbit rotation. In Orbit mode, a selected local axis (X, Y or Z) remains vertical on screen, providing a predictable and stable camera orientation during navigation.

Trackball.

Trackball mode provides unrestricted viewport rotation.

The scene can be rotated freely in all directions, allowing the user to inspect the model from any angle.

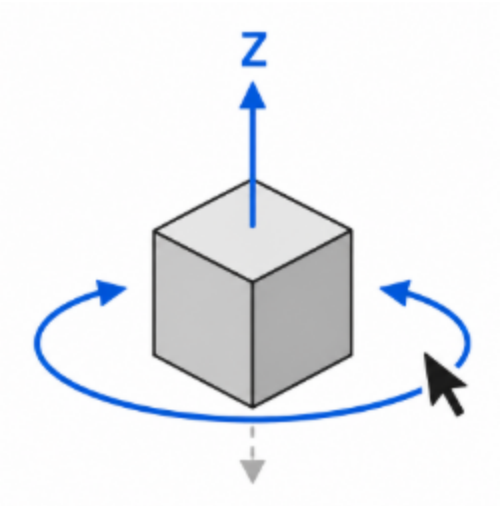


Orbit.

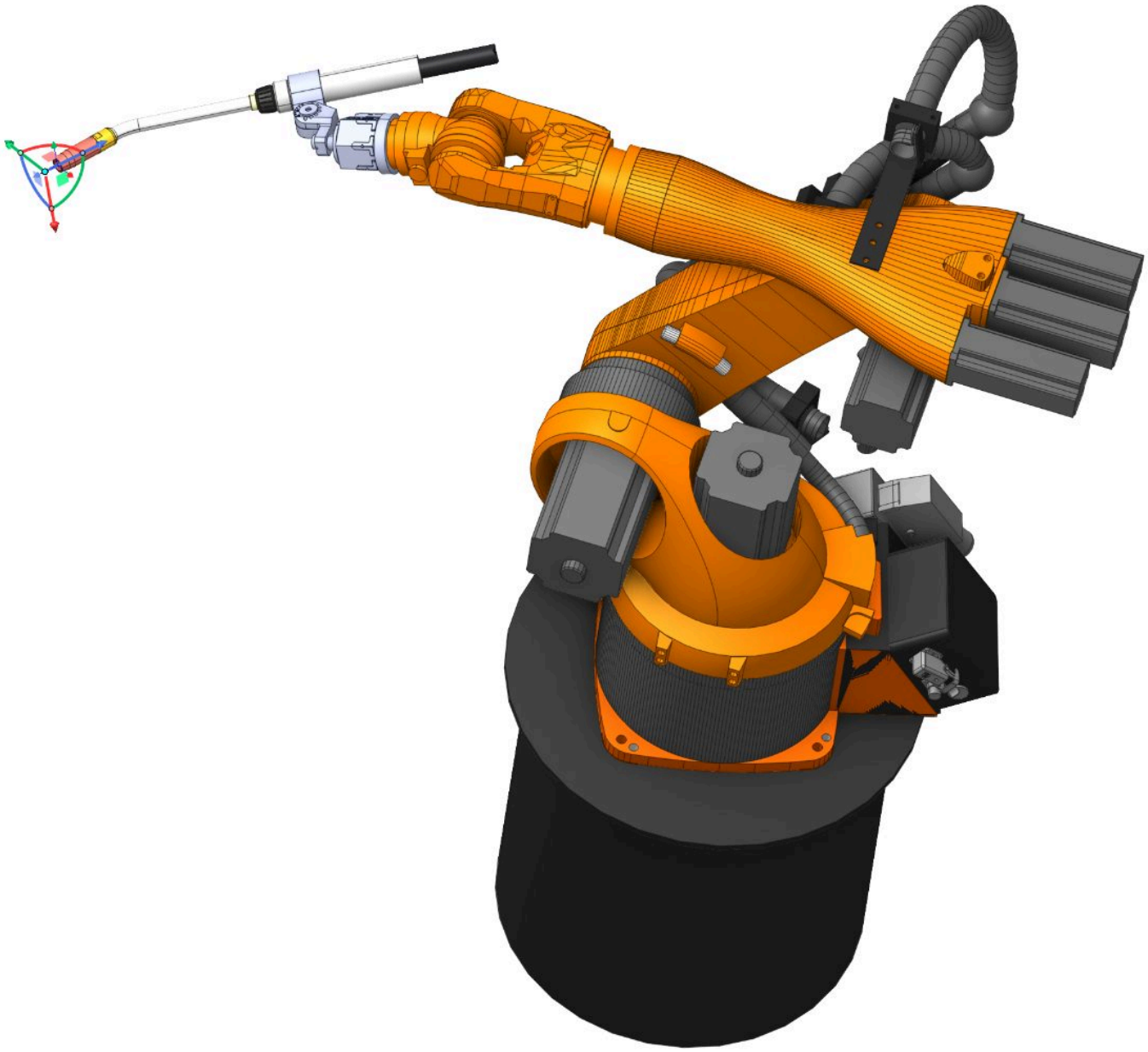
Orbit mode constrains viewport rotation around a selected local axis.

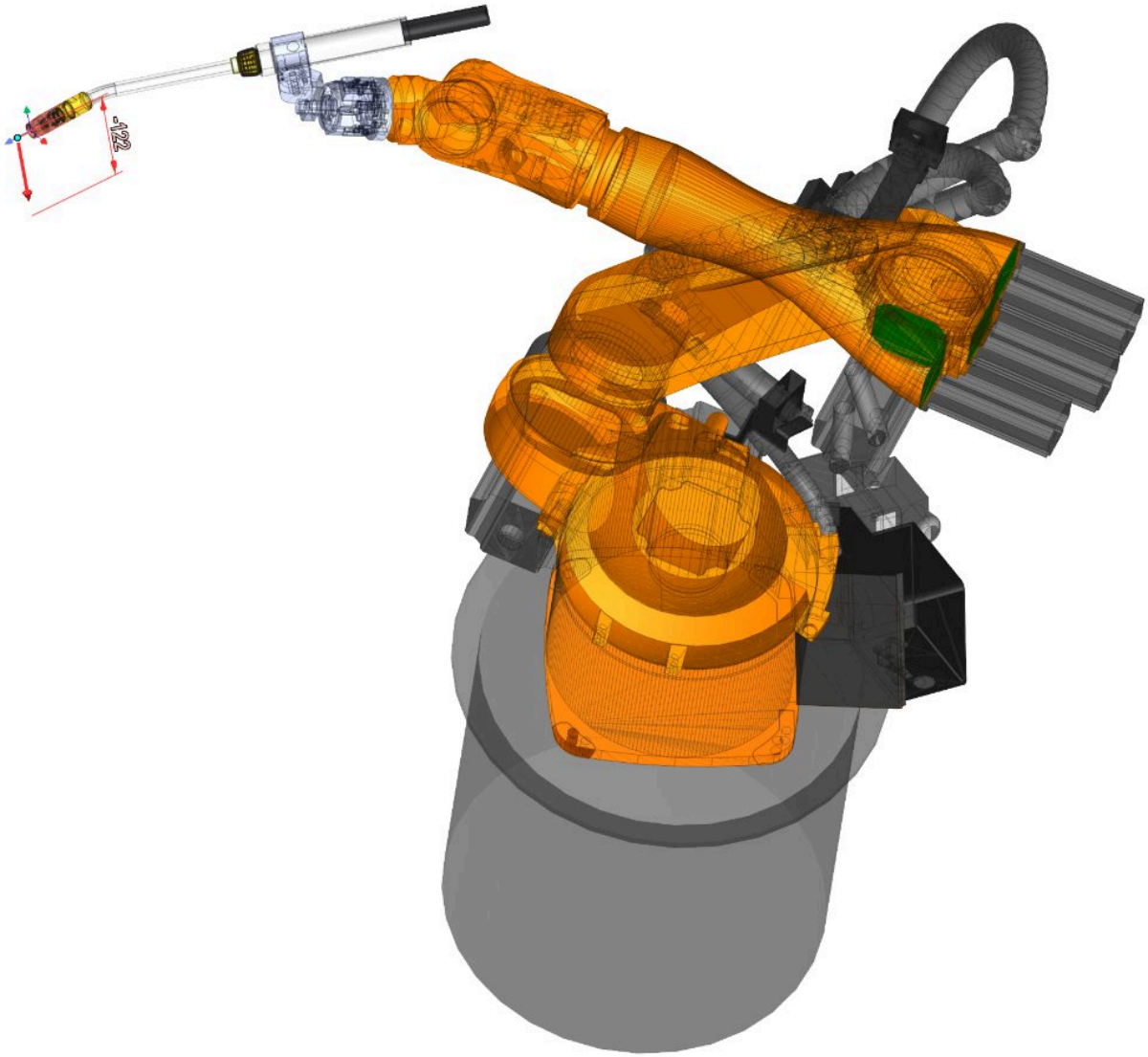
When Orbit mode is activated:

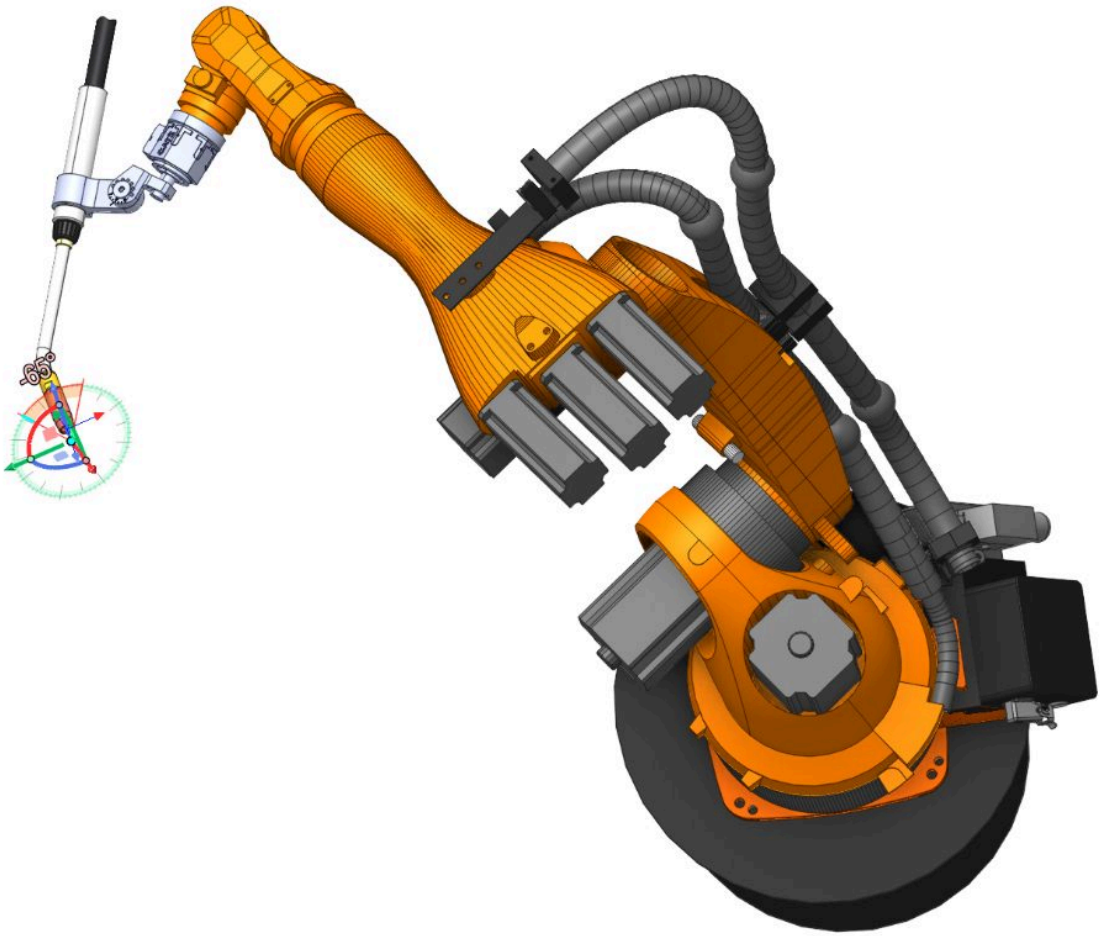
- The selected local axis (X, Y or Z) is automatically aligned vertically on the screen.
- The view smoothly snaps to the new orientation.
- Horizontal mouse movement rotates the scene around the selected axis.
- The selected axis remains vertical throughout navigation.



Gimbals




-122



Application area:

Gimbals have been added as a visual tool for manipulating (moving, rotating) objects in 3D space. They provide an interactive way to precisely control the position and orientation of objects.

Gimbals have been added for use in the following cases:

- Robot TCP movements
- Defining 6th axis control via custom point
- Defining a point when controlling the tool axis via point
- Tool axis orientation
- Fixture placement

Interactive linear axis ruler

Application area:

The interactive linear axis ruler displays the position of a machine node while it is dragged along a linear axis in the 3D view. It shows the current axis coordinate and travel limits, helping position the node and assess its location within the available travel range.

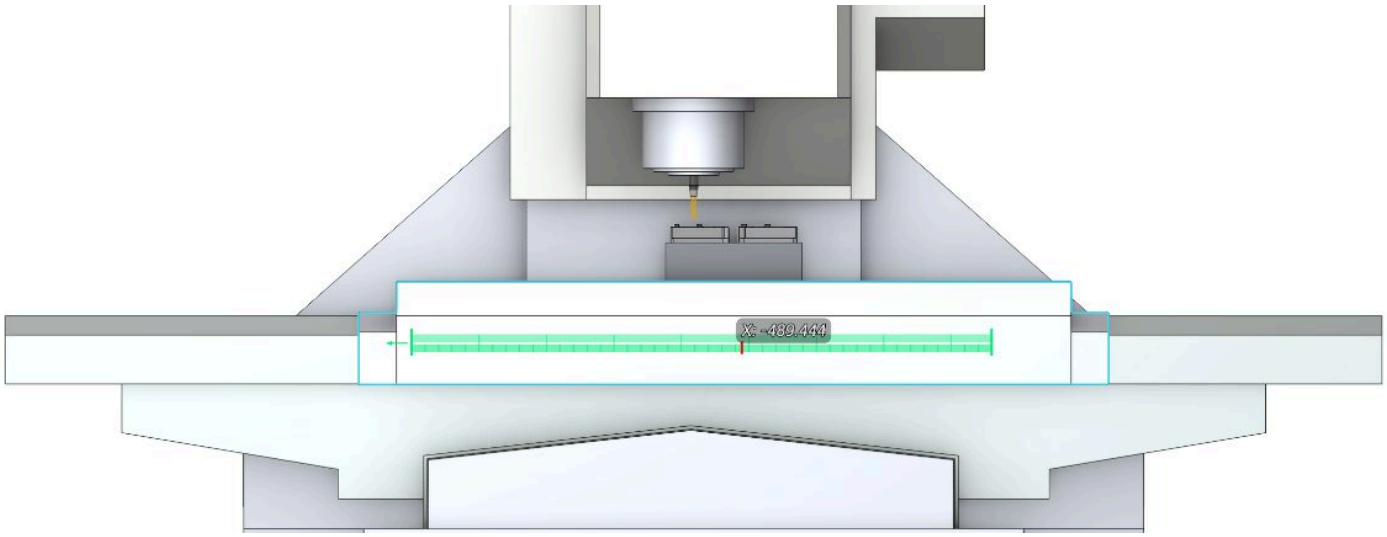
The ruler is used with Interactive Machine when adjusting the machine position directly in the graphic window.

Ruler elements:

Axis scale. Represents the travel range along the linear axis being moved.

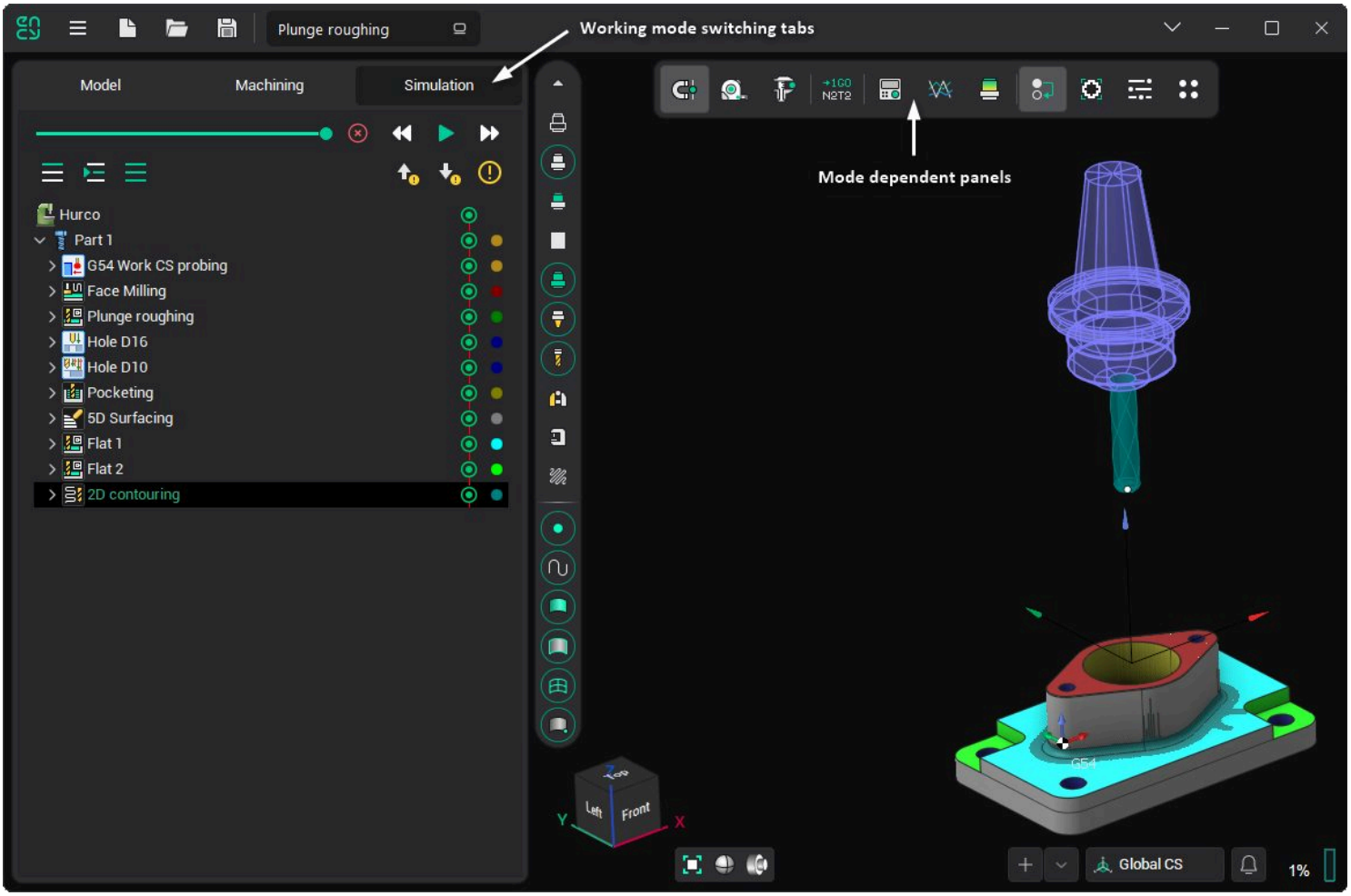
Current axis coordinate. Displays the axis name and its current coordinate beside the position marker. In the illustration, the label X: -489.444 identifies the axis and its position.

Travel limits. Mark the ends of the axis travel range, allowing the current position to be compared with the limits.



Linear axis ruler displayed while moving a machine node along the X axis.

Work modes



For convenience, the system interface is divided into several working modes. Tabs are used to switch modes in the main window. When you change a tab, the contents of the panels in the main window change:

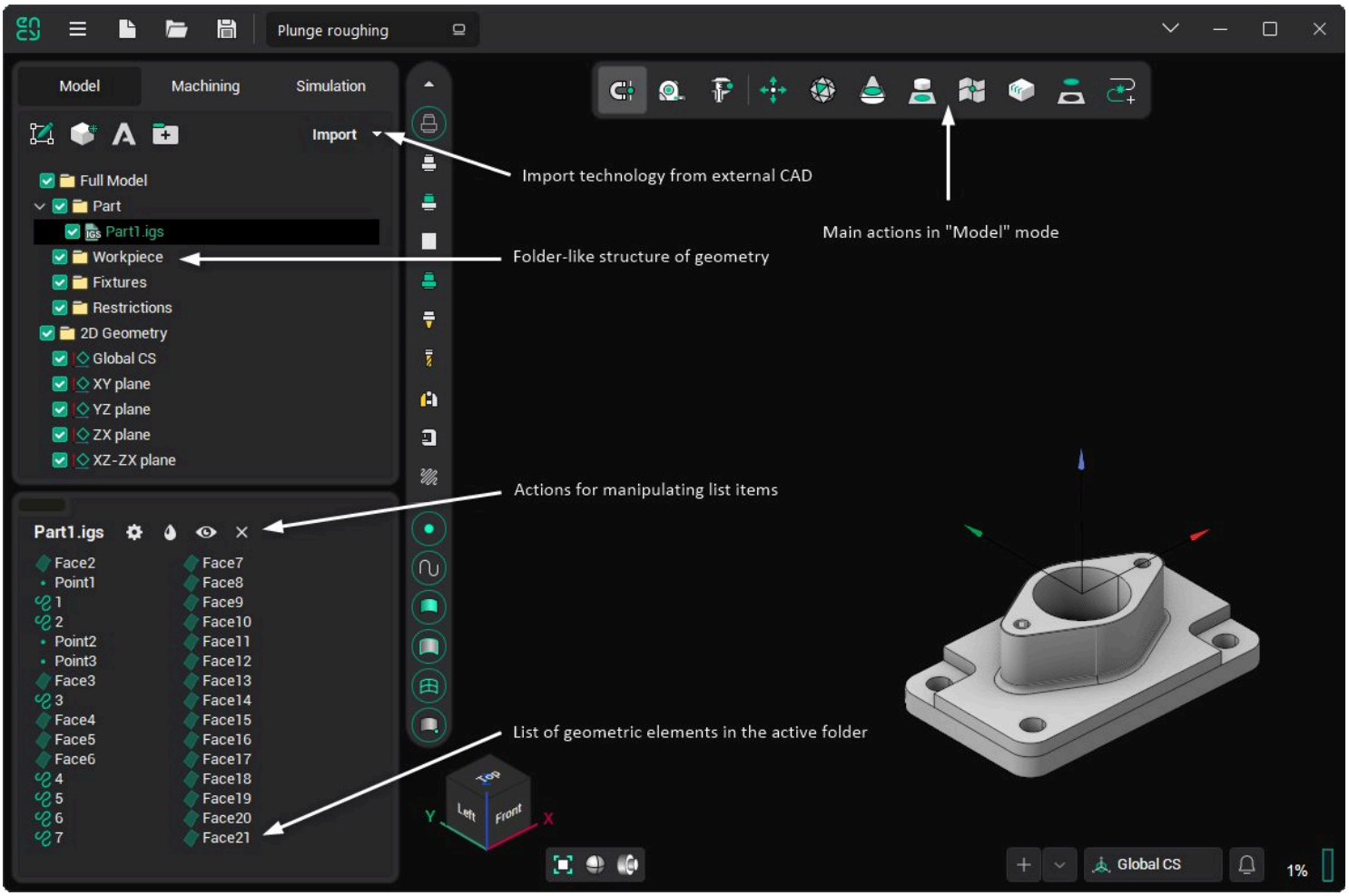
- Working mode content page on the left side of the window.
- Working mode toolbar in the middle of the top window pane.

As a result, the number of buttons displayed simultaneously on the screen is significantly reduced, simplifying the interface and expanding the space for primary work. The sequence of tabs approximates the order of basic user actions when working on a project, though the relationship is not strictly linear.

The following briefly describes the purpose and appearance of the window for each mode.

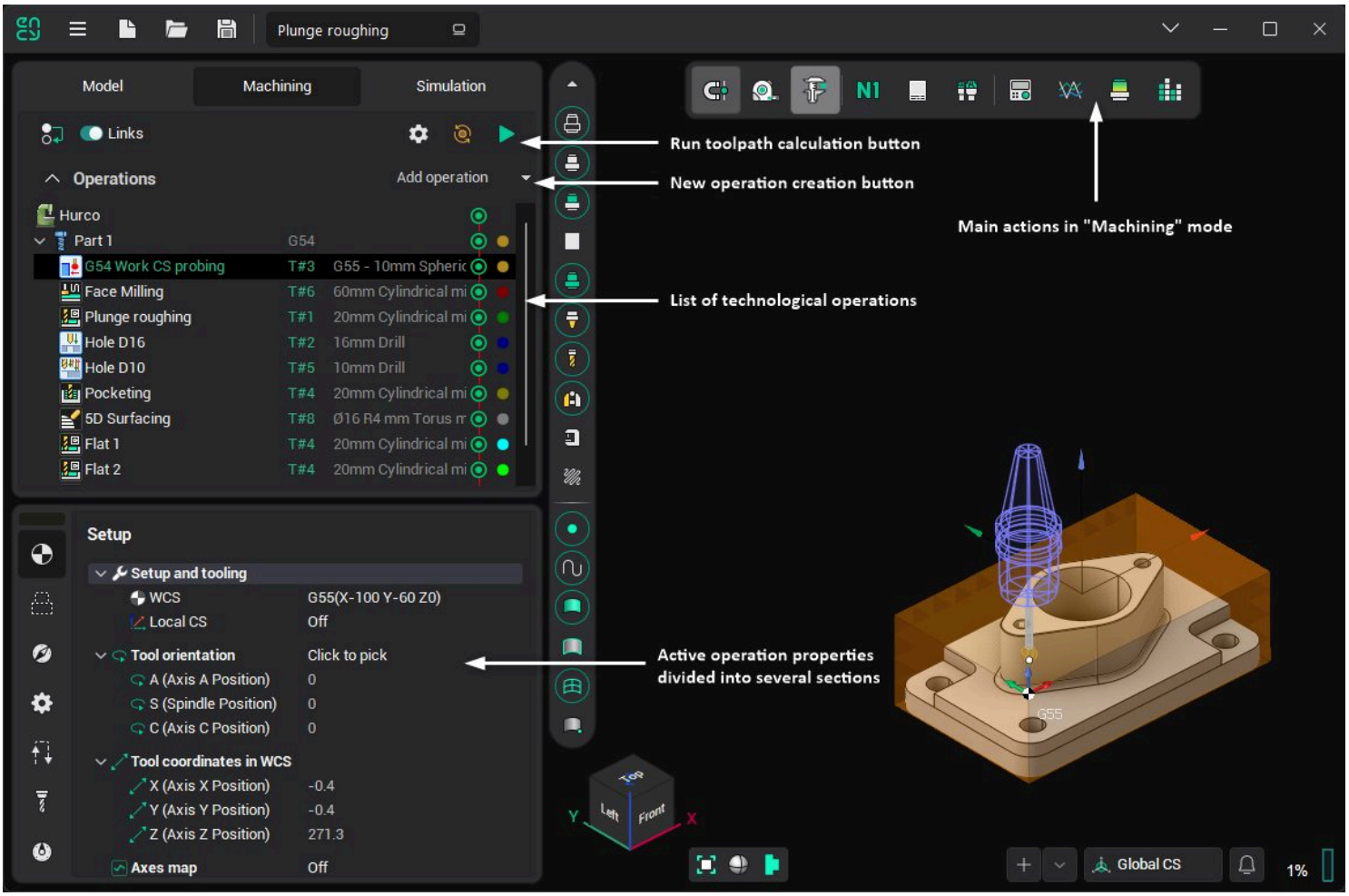
Model

In Model mode,the user can: import geometrical data (CAD) files, modify (cut, delete etc.) the structure of a geometrical model, spatially transform (move, rotate etc.) objects, generate new elements (copy, draw, intersect, triangulate, etc.) from existing ones, and manage the object's visual properties.



Machining

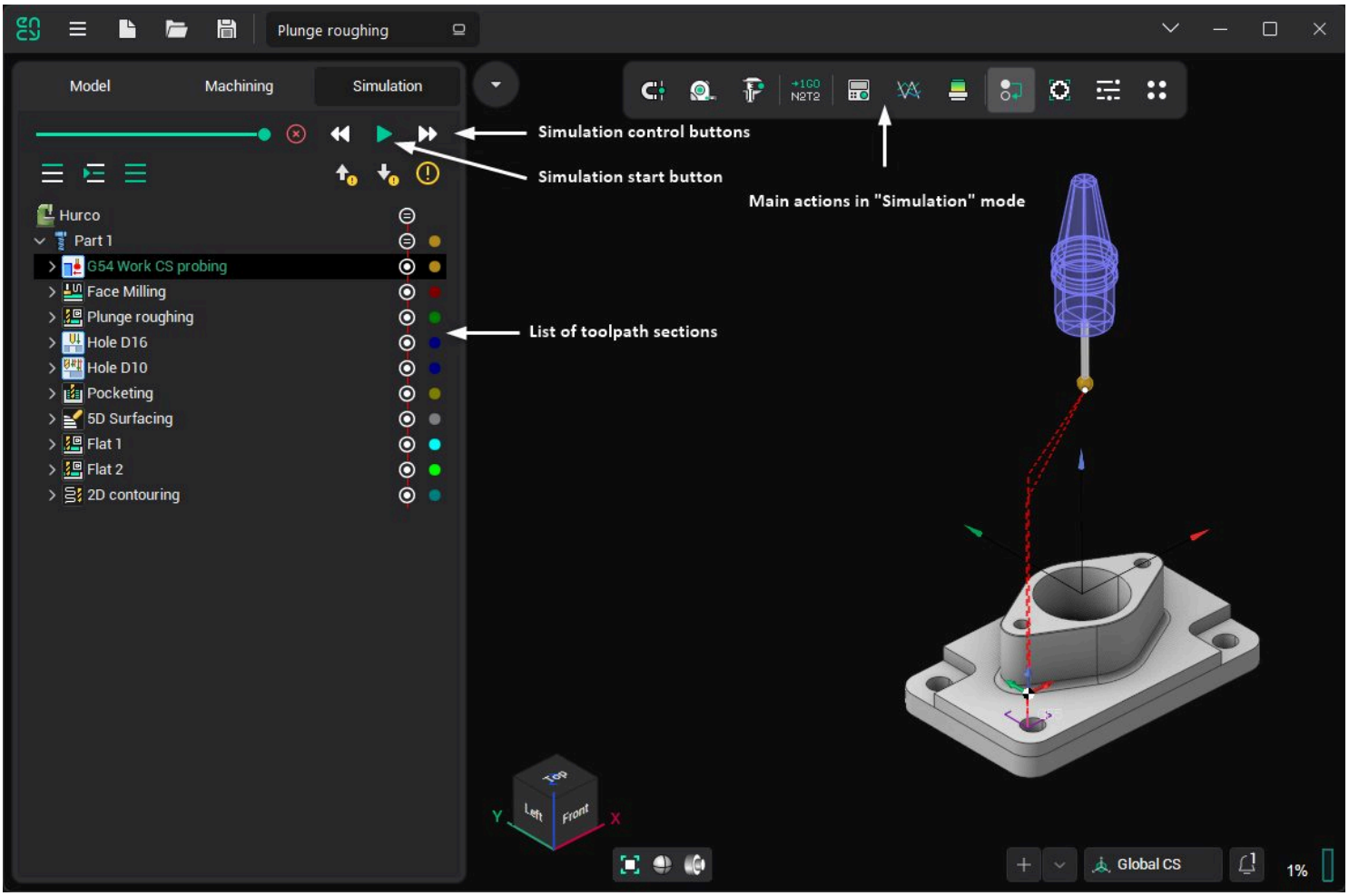
In Machining mode, the user creates the machining process for the part, choosing from a list of available technological operations. The operation determines the processing strategy and type of toolpath. Fine-tuning of all machining operation parameters and calculation of the toolpath can also be performed here. After receiving the toolpath, it can be viewed on the screen, along with the preliminary result of machining. After debugging the process, you can run the postprocessorto generate the NC program andmachining report.



The structure of the Machining mode can be studied in this section.

Simulation

In Simulation mode, the user has access to integrated machining simulation tools that allow controlling material removal, machine collisions, and visualization parameters dynamically, and checking the generated toolpath by blocks.



Several buttons are common to both Machining and Simulation modes.

- Machine Control Panel: allows you to view and modify the current values for all machine/robot coordinates.
- Graph of the axes: displays changes in each machine axis over time during the toolpath execution of the current operation. This helps identify unfavorable toolpath segments with bouncing that could lead to vibrations.
- Verify Compare: allows you to display the difference between the original part and the machined result using a color scheme, where each color represents a specific deviation range between the compared elements.
- Tool Reach Inspector: this panel allows viewing zones of the part where the toolholder does not have collisions and determining the best angle for part processing.

Selection filter

The object selection filter toolbar is located at the bottom of the right pane of the main window.

When choosing among the many geometric objects in the graphics window, it is useful to limit the selectability of objects depending on their type (point, curve, mesh, surface, etc.). Filter parameters can be set by pressing the corresponding buttons on the toolbar. By [Left Mouse Button] clicking the chosen button, the corresponding filter is toggled on and off.

- – Allow/Restrict selection of points
- – Allow/Restrict selection of curves
- – Allow/Restrict selection of surfaces
- – Allow/Restrict selection of edges
- – Allow/Restrict selection of meshes
- – Allow/Restrict selection of vertices

Visibility panel

Application area:

The Visibility panel controls which objects are shown in the 3D view and how they are displayed. Visibility and display settings are defined separately for the Model, Machining, and Simulation working modes.

The panel provides settings for the Geometry model, Part, Job assignment, Workpiece, Machining result, Holder, Tool, Fixtures, Machine, and Toolpath.

Opening display settings:

- Left mouse button. Click an object button to switch its visibility ON or OFF.
- Right mouse button. Click the same button to open its display settings popup. The available controls depend on the object type.

Common display settings:

The following controls are shared by several object types. Their availability is listed in the next section; Machine and Toolpath have separate sets of controls.

Color. Sets the display color. Click the color swatch to select a color. For Geometry model, Machining result, and Tool, the popup also includes a switch for enabling the selected color.

Material. Selects the surface appearance used to display the object. The material list contains Default, Polished, Gunmetal, Mirror, Plastic, Matte, Paint, Ceramic, Industrial, and Satin. This setting controls visual appearance, rather than the workpiece material used for machining calculations.

Transparent. Switches transparent display ON or OFF. Adjust the transparency level with the slider or enter a percentage in the numeric field.

Display mode. Selects wireframe, shaded, or shaded-with-edges display. Default follows the display mode selected on the visualization control toolbar.

Turn mode. Selects how bodies of revolution are displayed: 3D, 3/4, 1/2, or 2D. Default follows the turn mode selected on the visualization control toolbar.

Line width. Adjusts the width of displayed edges and outlines.

Motionless. Keeps the corresponding object visually stationary while the other components move relative to it. This changes the reference used to display motion in the 3D view.

Settings by object type:

All object types in the table provide Material, Transparent, and Display mode. The table lists the availability of the other common controls.

Object	Color	Turn mode	Line width	Motionless
Geometry model	Switch and swatch	Available	Available	—
Part	Swatch	Available	Available	Available
Job assignment	Swatch	Available	—	—
Workpiece	Swatch	Available	Available	Available
Machining result	Switch and swatch	Available	Available	Available
Holder	Swatch	—	Available	—
Tool	Switch and swatch	Available	Available	Available
Fixtures	—	—	Available	—

Restrictions color. An additional color swatch in the Job assignment popup sets the color of restricted machining areas.

Machining result. In Simulation mode, this button controls the display of the workpiece being machined. In other modes, it controls the machining result of the current operation, or the entire technological process if no operation is selected.

Machine display settings:

The Machine popup provides controls for the appearance and visibility of machine nodes.

Material. Selects the surface appearance of the machine, such as Gunmetal or Industrial.

Motionless. Keeps the machine reference visually stationary while other components move relative to it.

Machine node opacity. Adjusts the opacity of machine nodes. The current state is shown beside the control, such as Show all.

Machine nodes visibility:

Machine nodes visibility. Expands a tree of machine nodes. Expand or collapse branches to access individual components, and use the checkboxes to show or hide them. Hovering over a node highlights the corresponding component in the 3D view.

Opacity. Adjusts the opacity of the selected machine node.

Colors from schema. Uses the colors defined in the machine schema for the selected node.

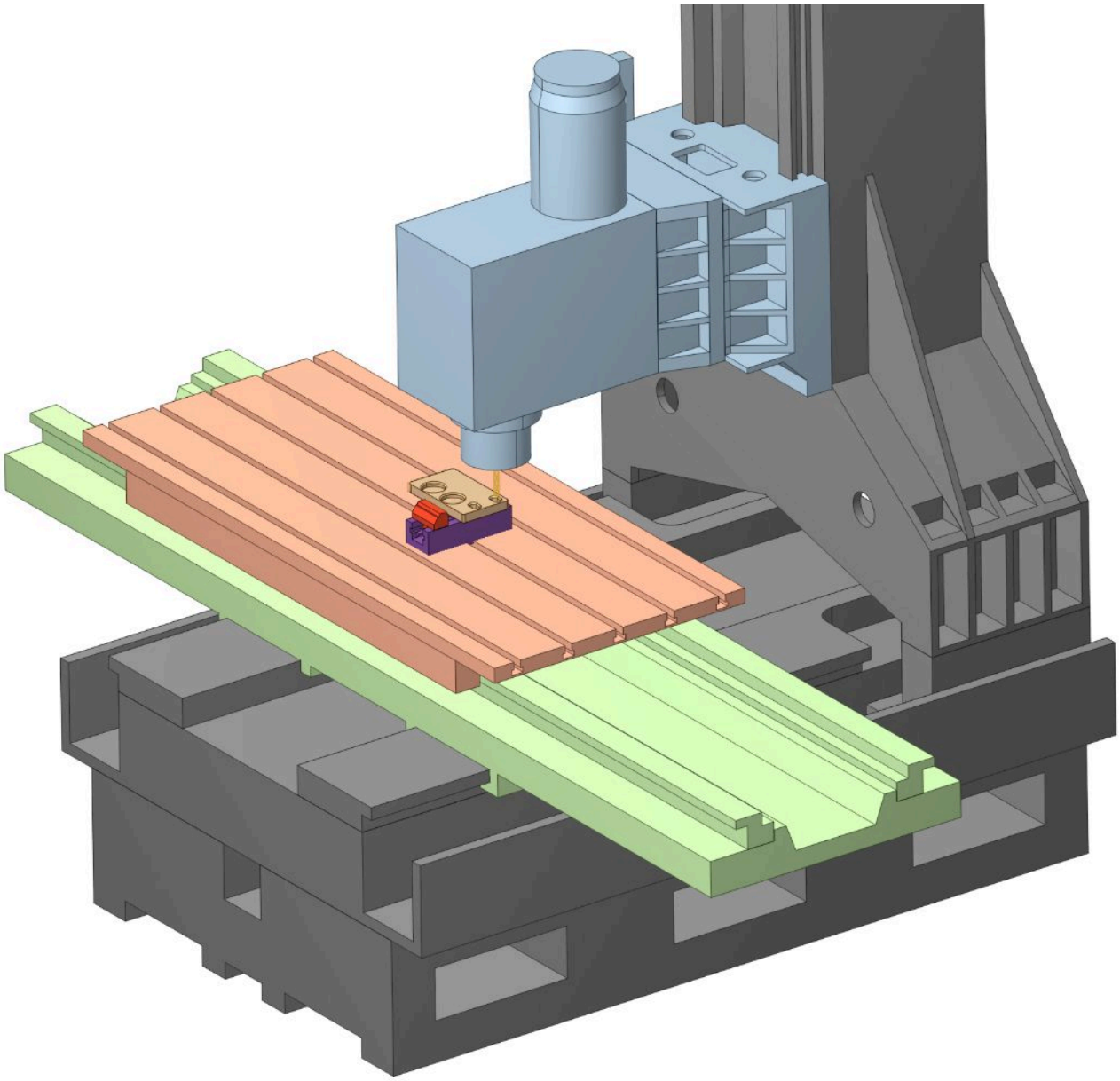
Custom color. Sets a custom color for the selected node when the control is available.

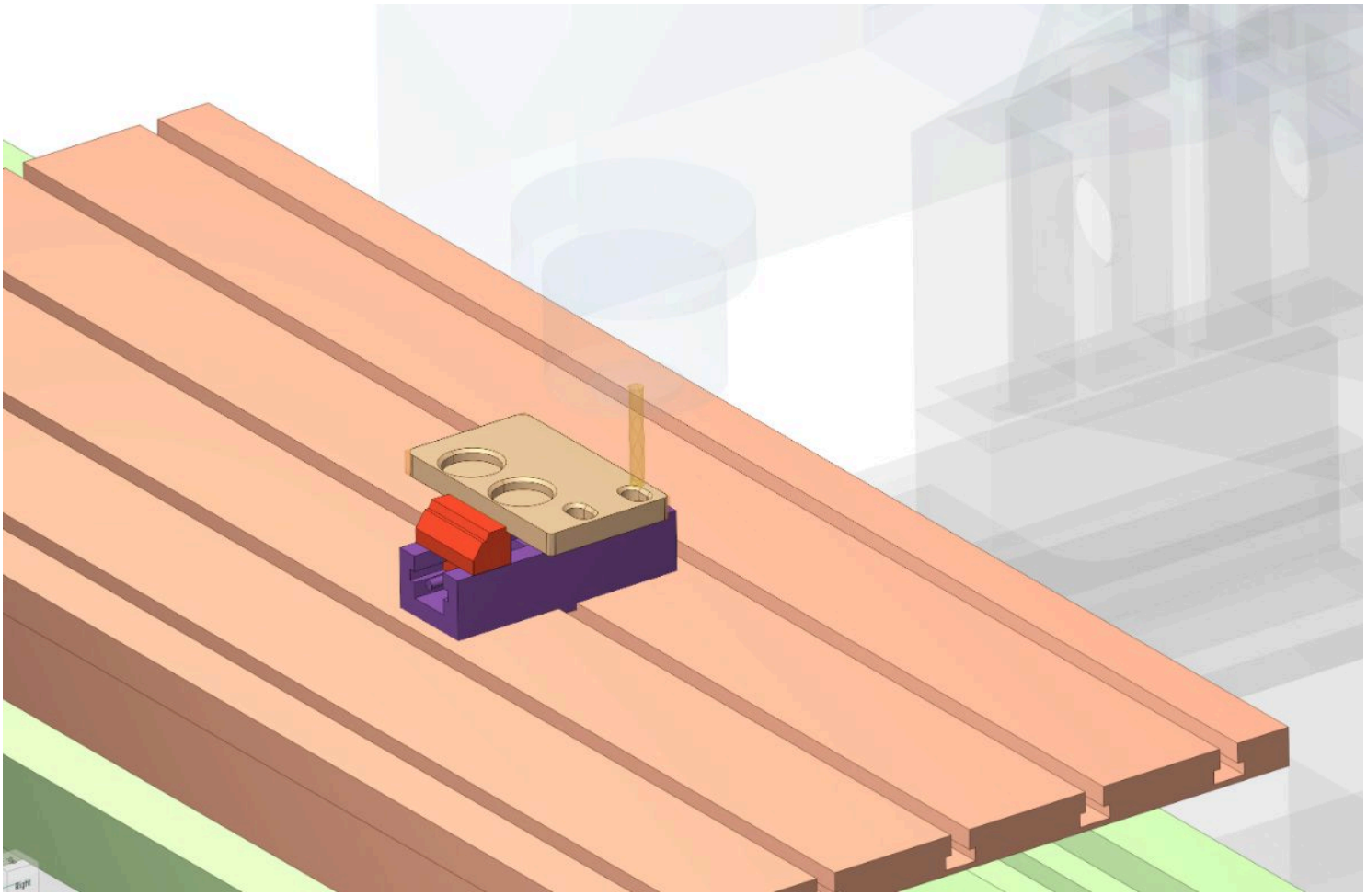
Selection and transparency:

Snap to machine nodes. Enables snapping to machine nodes in the 3D view.

Click through transparent. Allows selection through transparent machine components.

Auto transparency. Switches automatic transparency of machine components ON or OFF.





Automatic transparency reveals the tool and machining area behind machine components.

Sensitivity. Adjusts the sensitivity of automatic transparency.

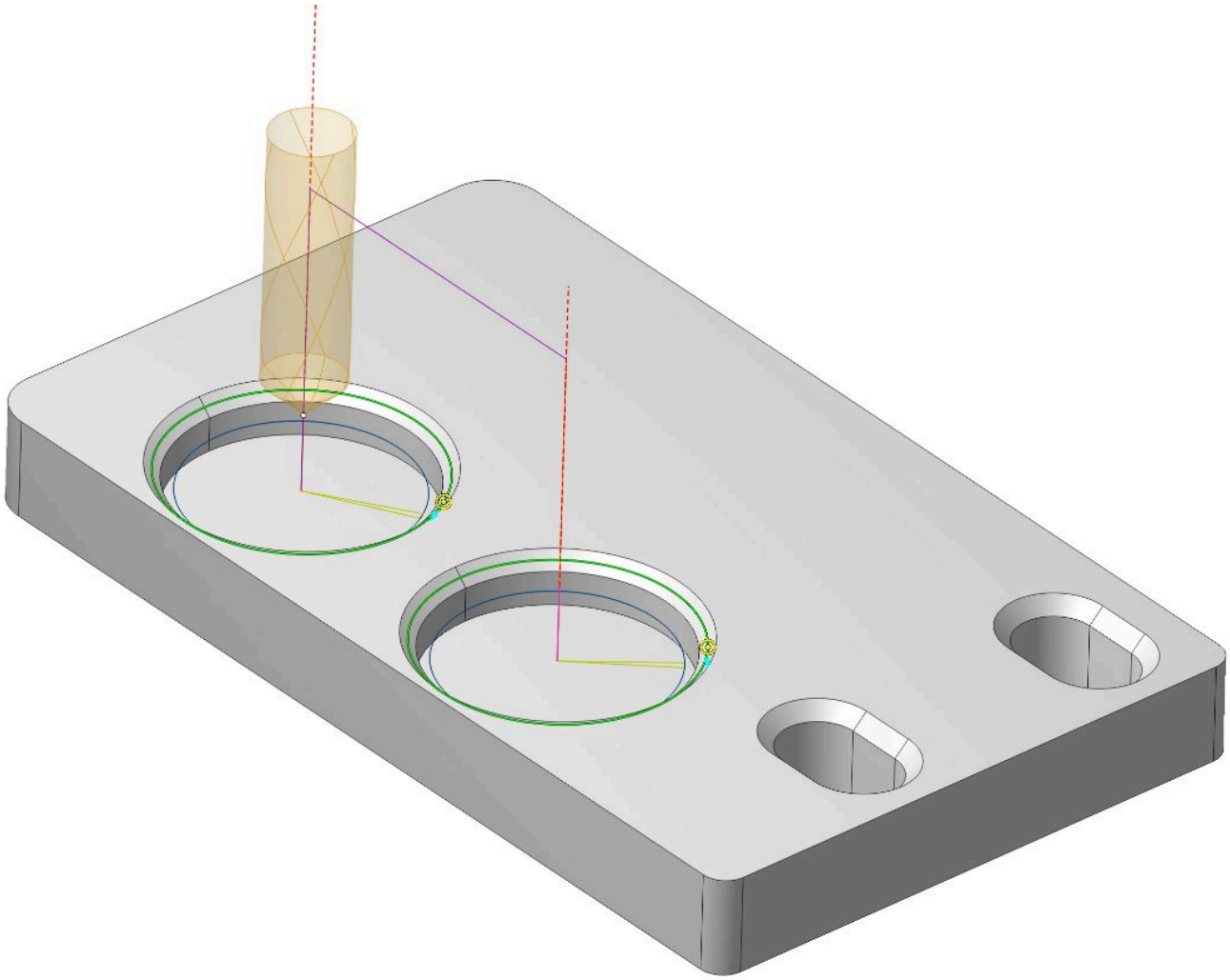
Interactive positioning of machine nodes is described in Interactive Machine. Details

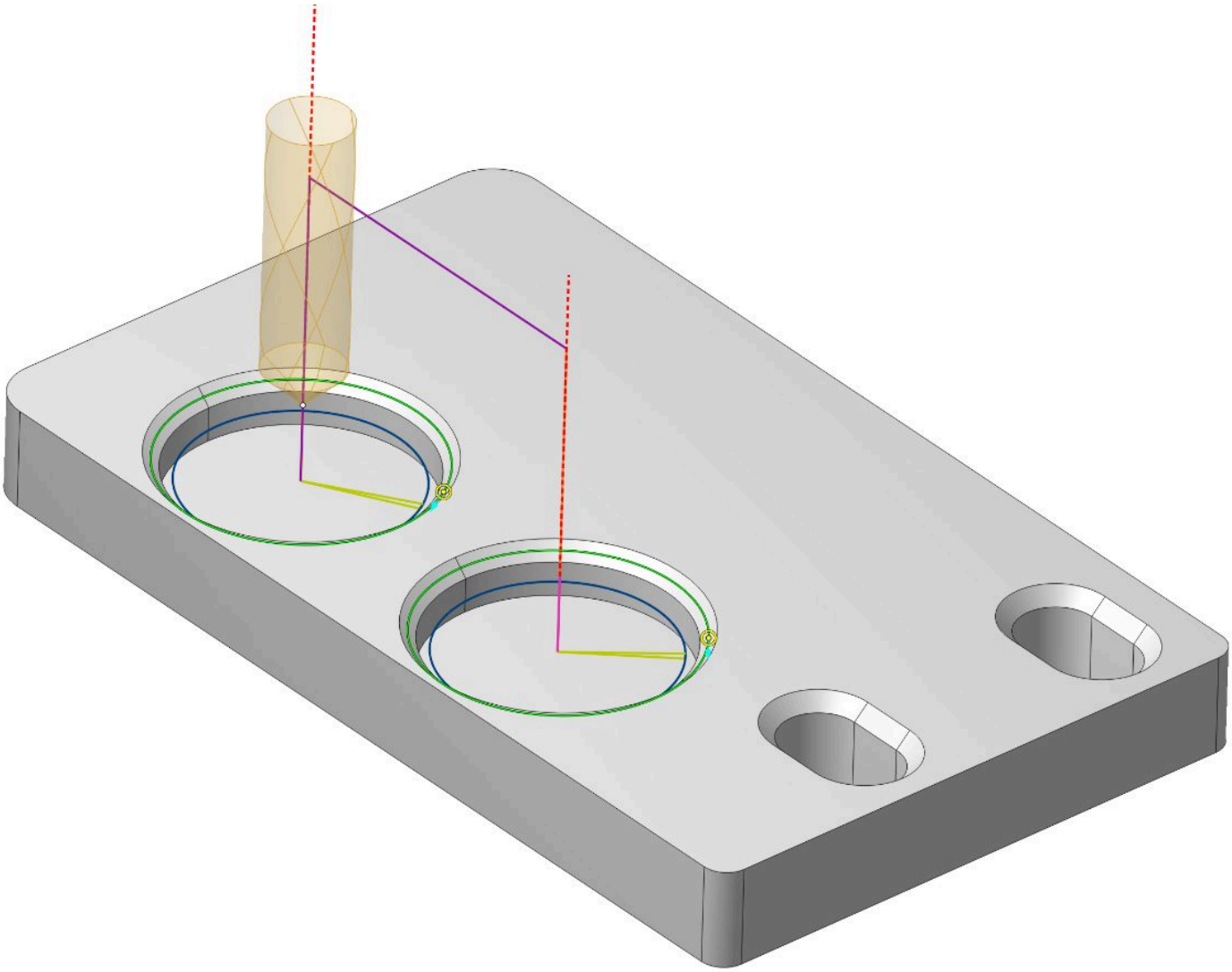
Toolpath display settings:

The Toolpath popup controls the appearance of the trajectory and which movements are shown.

Color. Switches toolpath coloring. The toolpath can be displayed using feed-type colors or a single color.

Line width. Adjusts the width of toolpath lines.





The same toolpath displayed with different line widths.

Playback trace. In Simulation mode, selects which portion of the current operation's toolpath is displayed:

- Current layer. Displays the current toolpath layer.
- Full toolpath. Displays the entire toolpath.
- From start. Displays the toolpath from its beginning to the current tool position.
- To end. Displays the remaining toolpath from the current tool position to its end.
- Tail. Displays a limited toolpath segment near the current tool position.
- Tail ahead. Displays a segment ahead of the current tool position.
- Tail both ways. Displays segments before and after the current tool position.

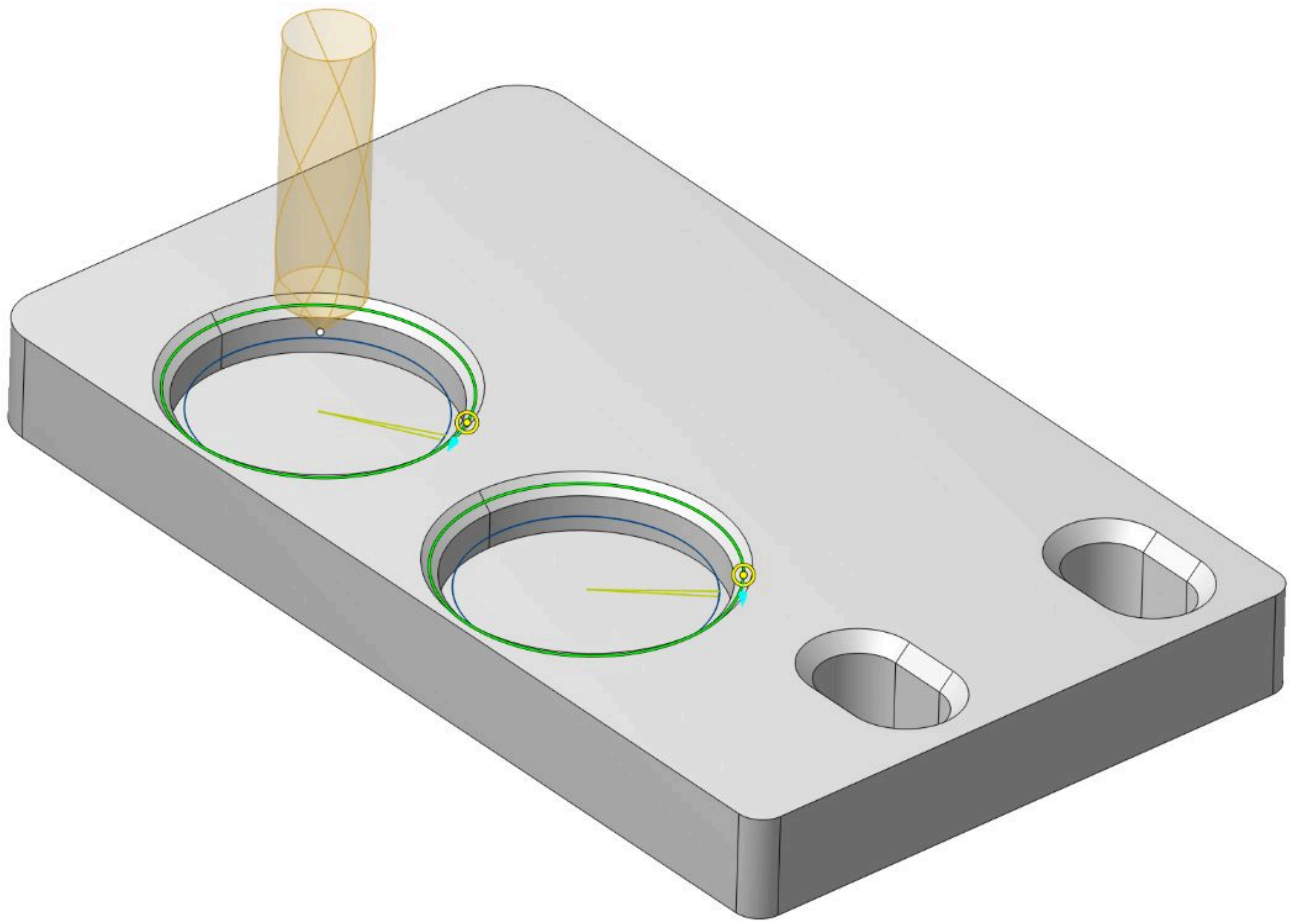
The rest of the program remains visible as a ghost layer.

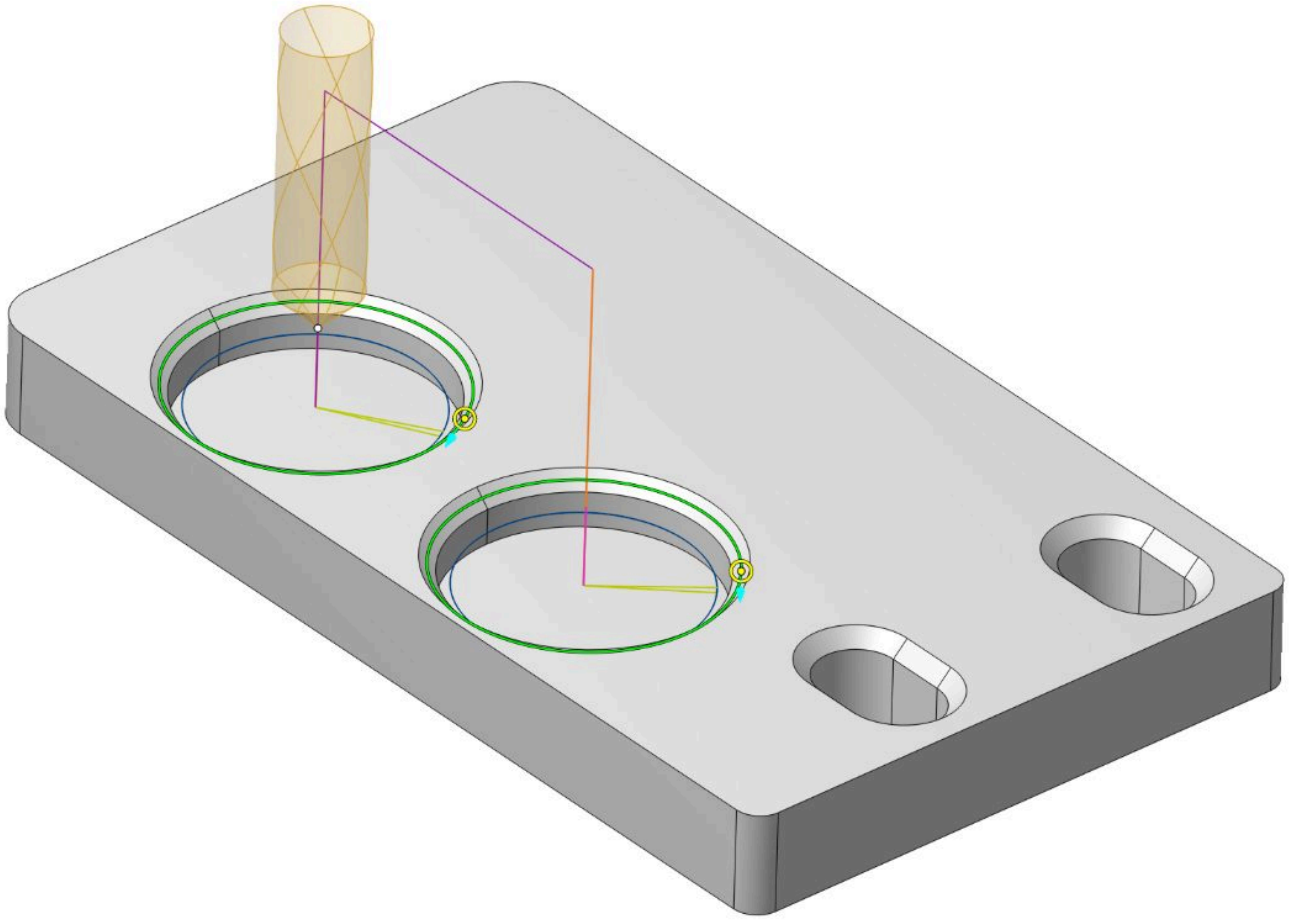
Toolpath visibility. Groups the switches for displaying rapid, approach/return, and linking movements. These controls work together with Playback trace to determine which movements are visible during simulation.

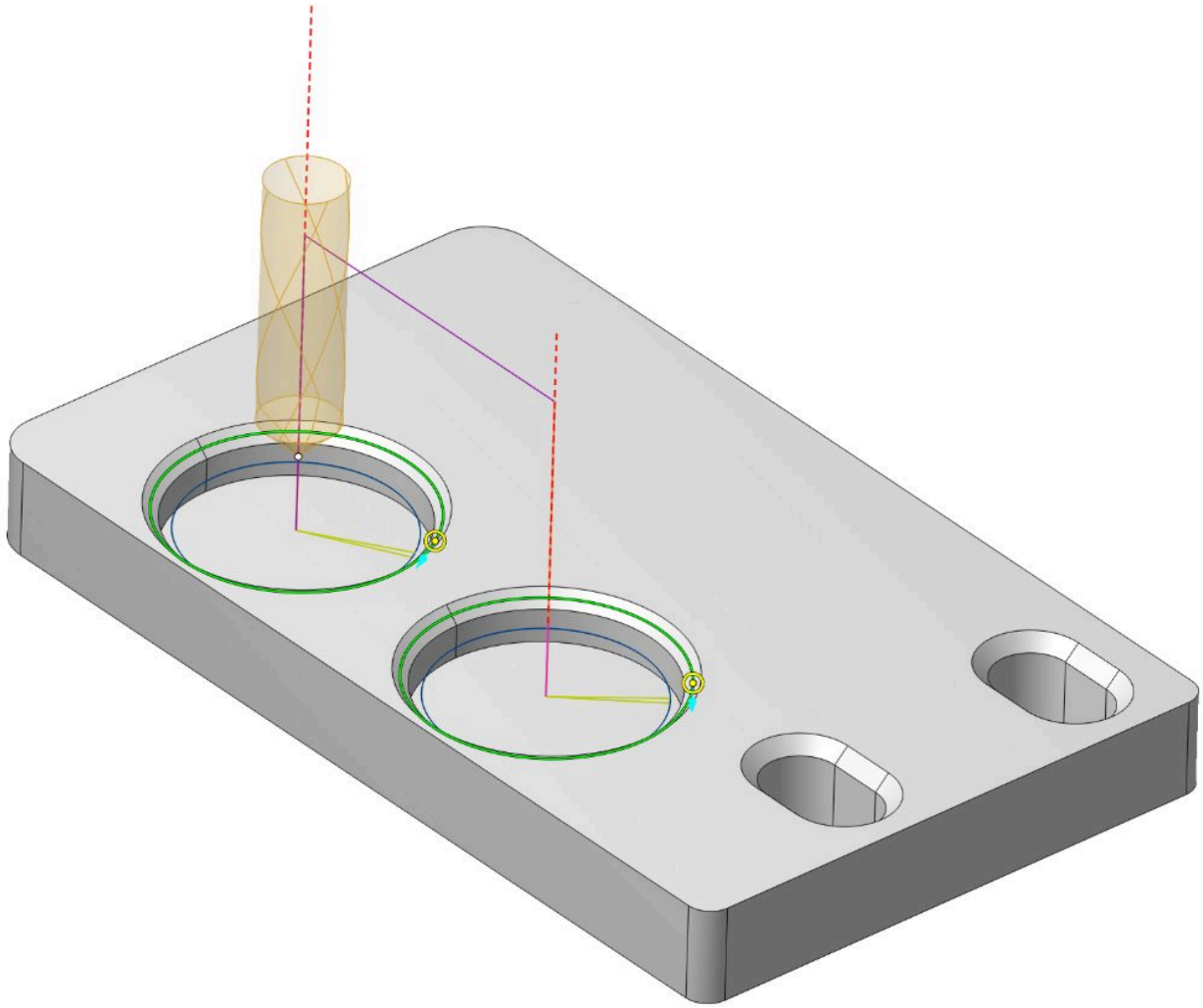
Rapid moves. Shows or hides rapid movements.

Approach/return moves. Shows or hides approach and return movements.

Link moves. Shows or hides linking movements.







Toolpath display with different movement visibility settings.

Show normals. Displays direction markers for the tool orientation at toolpath points.

Show points. Displays markers at toolpath points.

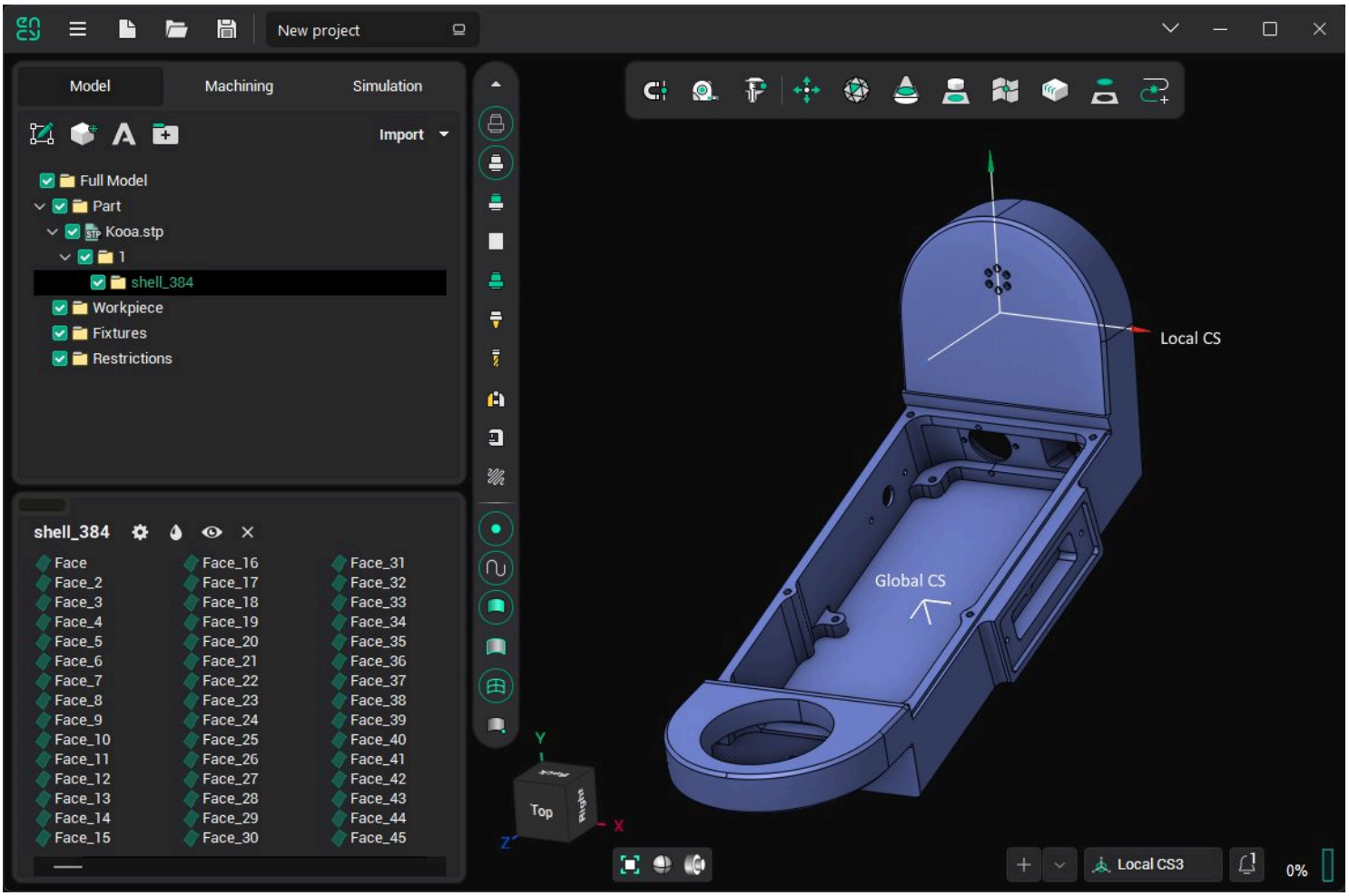
The point and direction markers follow the selected movement visibility and the displayed portion of the toolpath.

Saving settings:

The object display settings are saved in the CAM system configuration and restored when the application starts. Each working mode retains its own visibility and display settings.

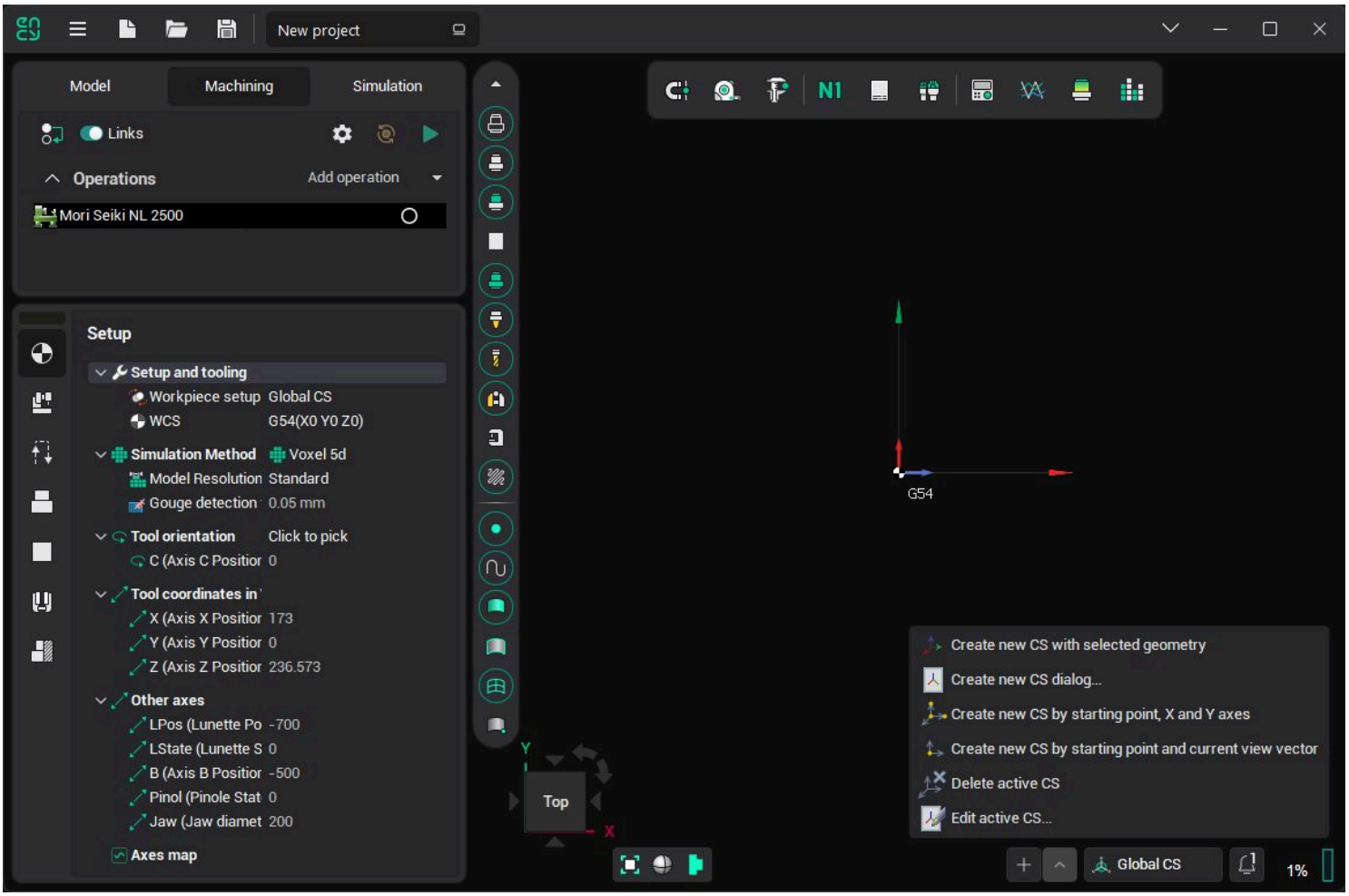
Geometrical coordinate systems

It is possible to create any number of local geometrical coordinate systems (CS) to make the machining process generation more convenient.



These systems can be useful when working with geometry, for example, if you need to draw a contour in an arbitrary plane or check the dimensions of an object. They are also helpful when setting operation parameters (Workpiece CS, Local CS, rotary axis, toolpath multiplying plane), etc.

The Coordinate Systems Management Panel located in the lower side of the main window



There is a concept of an active coordinate system. Only one CS can be active at any given time, and its name is displayed on the panel. Some system features take the active coordinate system into account, such as displaying the overall dimensions of geometric objects in this CS, or initializing a new sketch plane based on the active CS.

Click on the name of the CS in the panel to open the list of CS and change the active one.

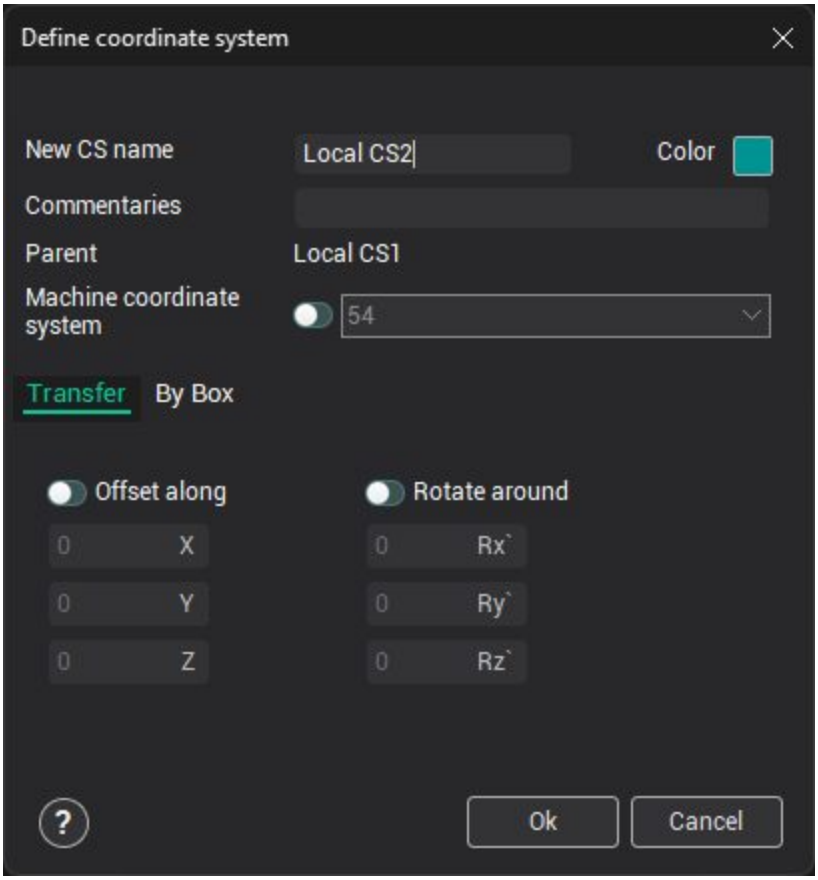
The button next to the CS name has a drop-down menu that contains possible actions for coordinate systems:

- Create new CS dialog...: Opens a window where you can directly set the parameters of a new CS. If any geometric element is selected at the time of the click, the window will not open, and the smart CS creation procedure will start instead.
- Create new CS by starting point, X and Y axes: Defines the new coordinate system interactively by selecting the origin point and two leading vectors, X and Y.
- Create new CS by starting point and current view vector: Defines the new coordinate system interactively by selecting the origin point and using the current view vector.
- Delete active CS: Deletes the active coordinate system. All coordinate systems listed below it will shift up by one level. The global coordinate system cannot be deleted.

-  Edit active CS: Opens the parameters window for the active coordinate system.

Creation of the coordinate system via dialogue window

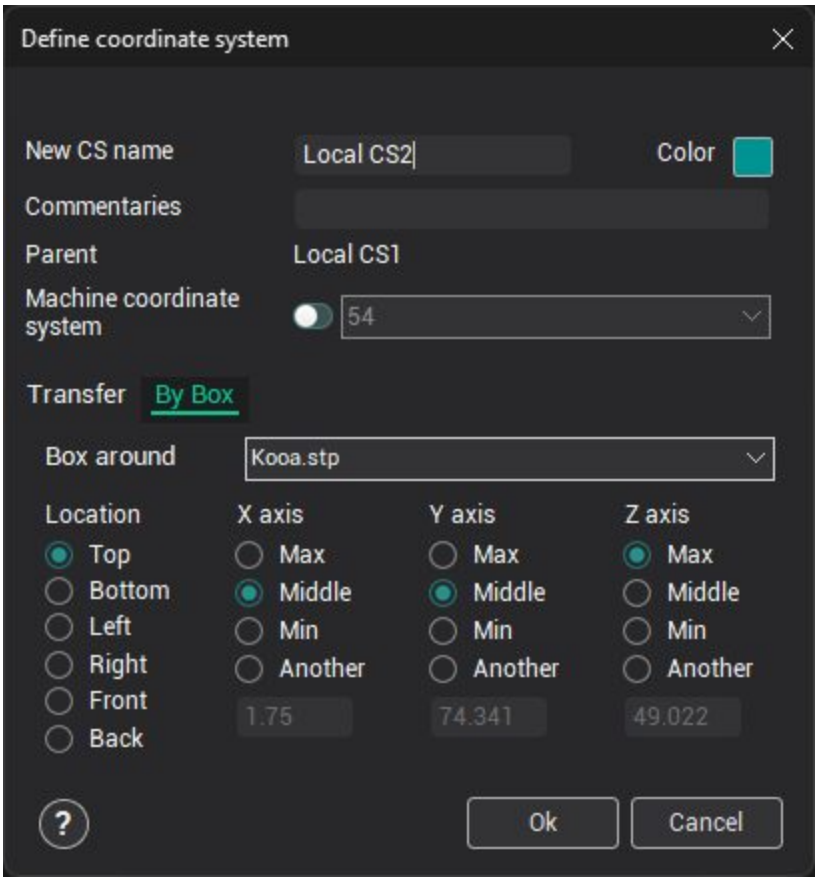
The window for creating a new coordinate system opens by pressing the menu item on the coordinate systems panel. If, at the time of the click, any geometric element is selected, the window does not open, and the smart CS creation procedure starts instead.



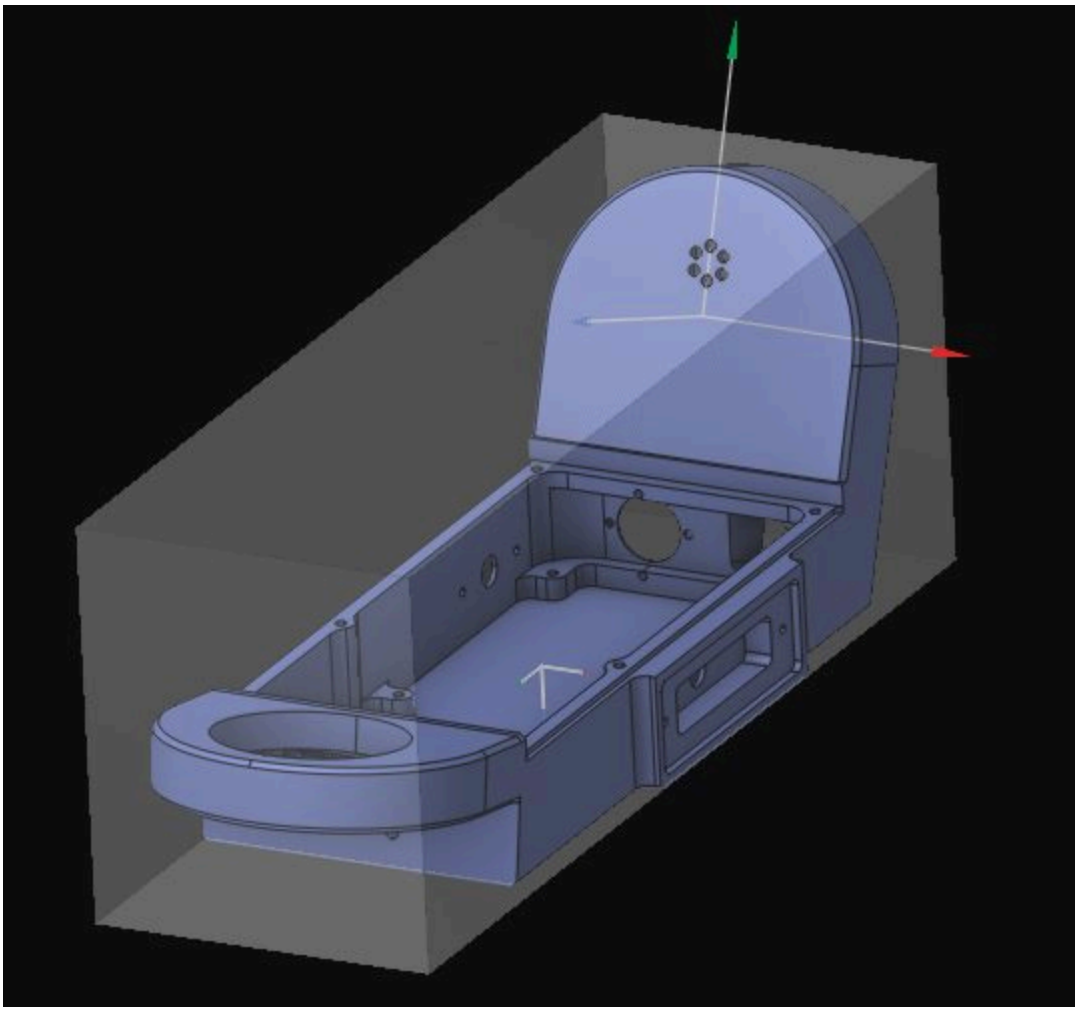
In this window, you assign the New CS name, its Color, and Commentaries.

There are two methods available for defining the position:

1. Transfer – All transformations are performed relative to the active coordinate system. The newly created system can be displaced and/or rotated relative to the parent coordinate system.
2. By Box – The newly created system is set on the external dimensions of the group selected in the Box Around list. In Location, select how the coordinate system will be positioned, and provide the origin point. The coordinates of the point on the corresponding axes can be selected as Max, Middle, or Min relative to the box of the group, or in absolute coordinates (Another).



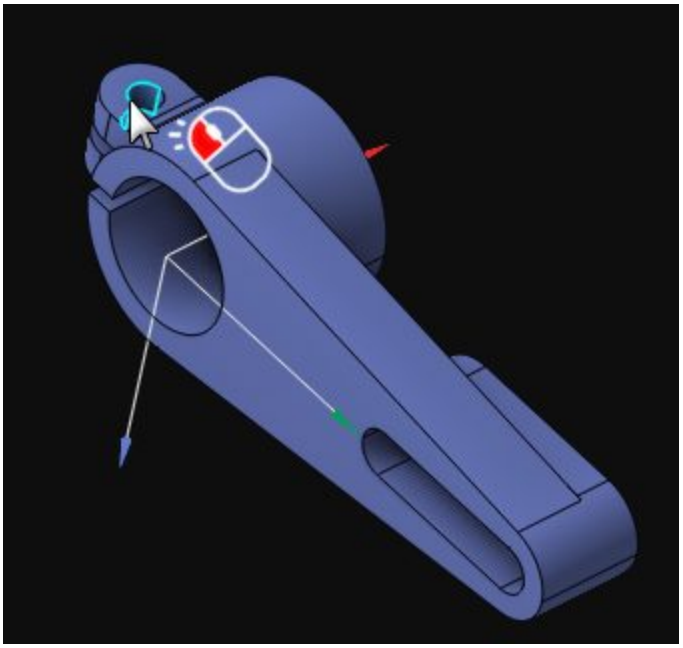
All changes will be previewed in the graphic window immediately.



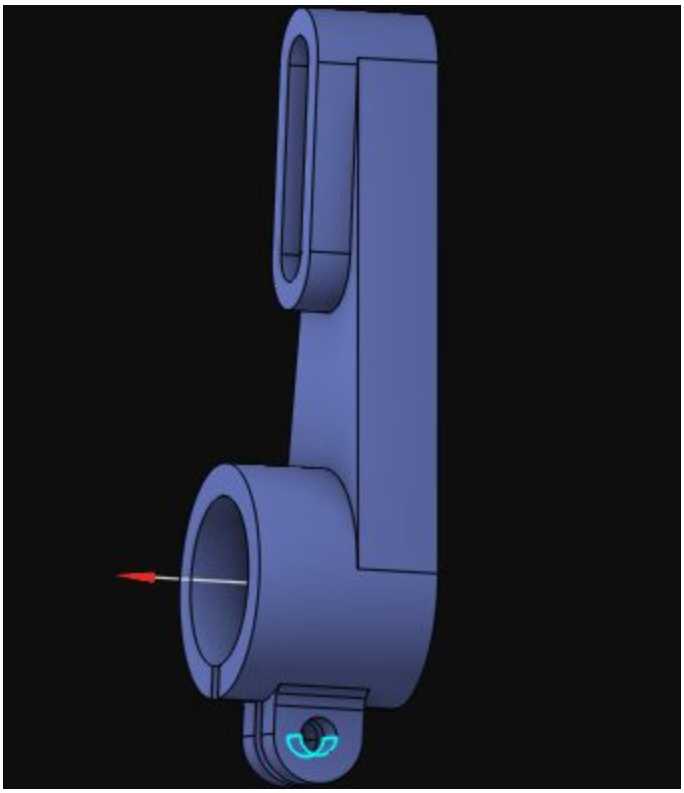
Smart LCS creation

You can easily create a coordinate system with approximately two mouse clicks. It is very straightforward.

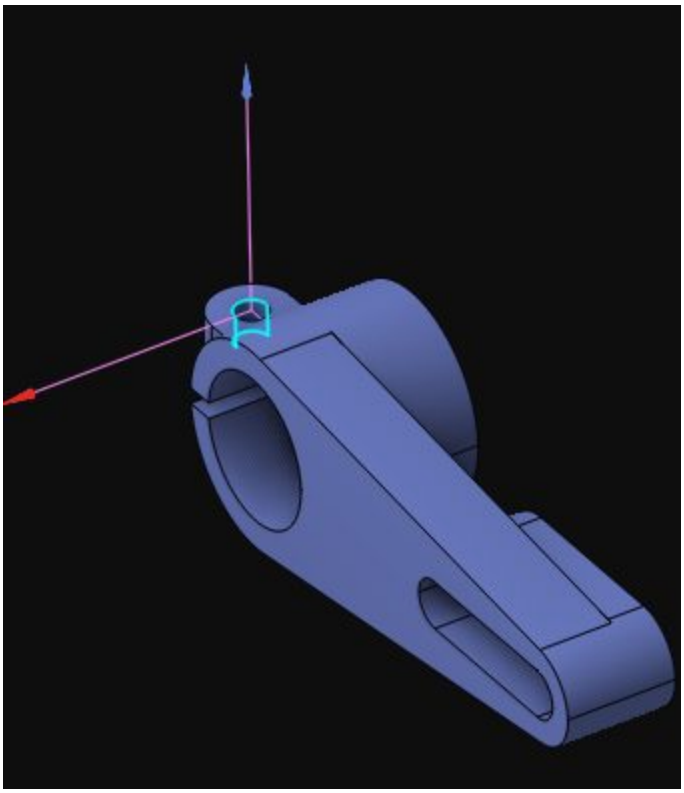
1. First, select in the graphic view a design feature that defines the location of the new coordinate system. You can select any entity that has an origin, an axis, or both. For example, a circular arc, a line, a revolution surface, or a plane defines both the origin and the axis. A point defines only the origin, while a vertex of a solid model defines both the origin, the Z axis (taken from the neighboring surface), and the X axis direction (taken from the neighboring edge). A straight edge of a solid model defines the origin placed in the middle of the edge, with the Z axis direction taken from the neighboring surface and the X axis direction taken from the edge itself.



2. Since no geometry entity defines a local coordinate system completely, the CAM system must decide how to select the missing parameters, such as the Z or X axis direction of the new LCS. You can easily guide the CAM system by rotating the view with the mouse. This way, the CAM system selects the missing parameters for the new LCS based on the orientation of the view, similar to how you orient the view using the smart middle mouse click.

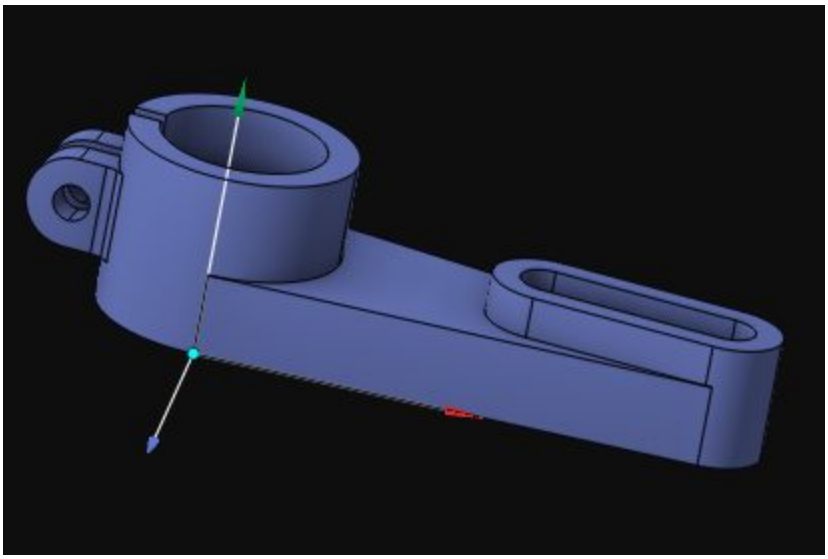


3. The final step is a simple click on the menu item on the coordinate systems panel.

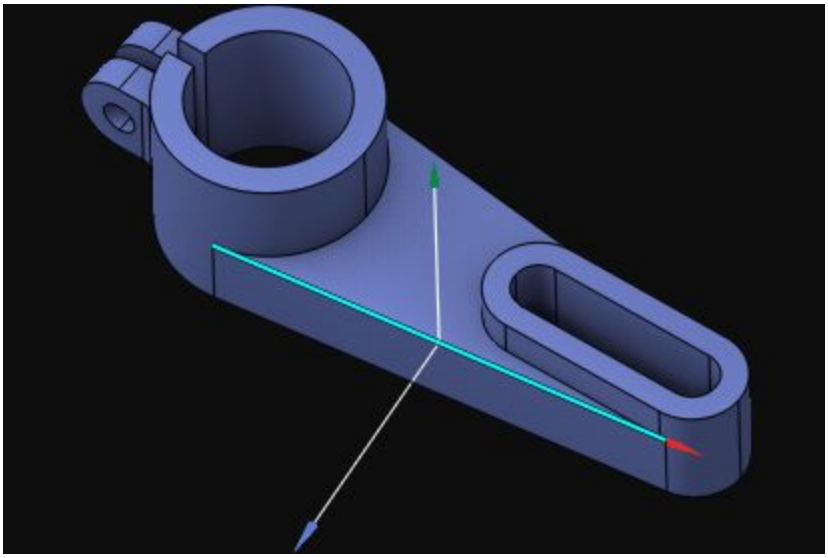


Examples of Smart LCS creation:

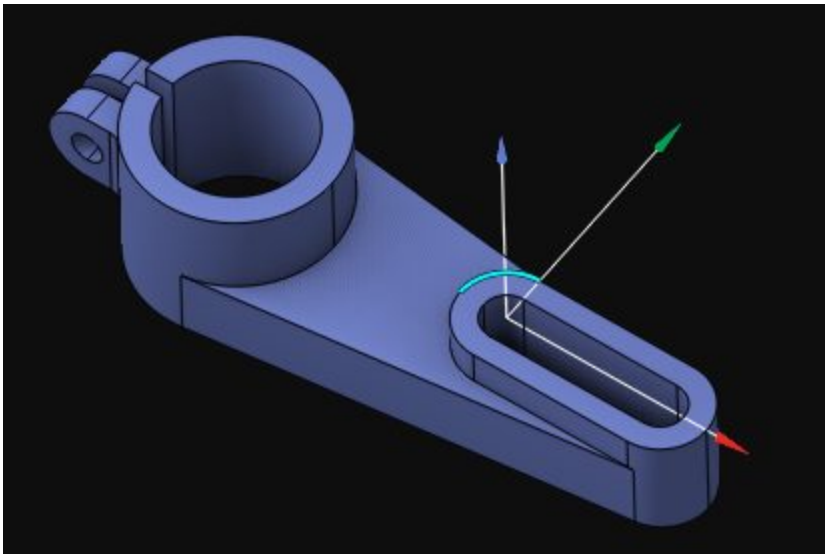
- LCS by a solid vertex:



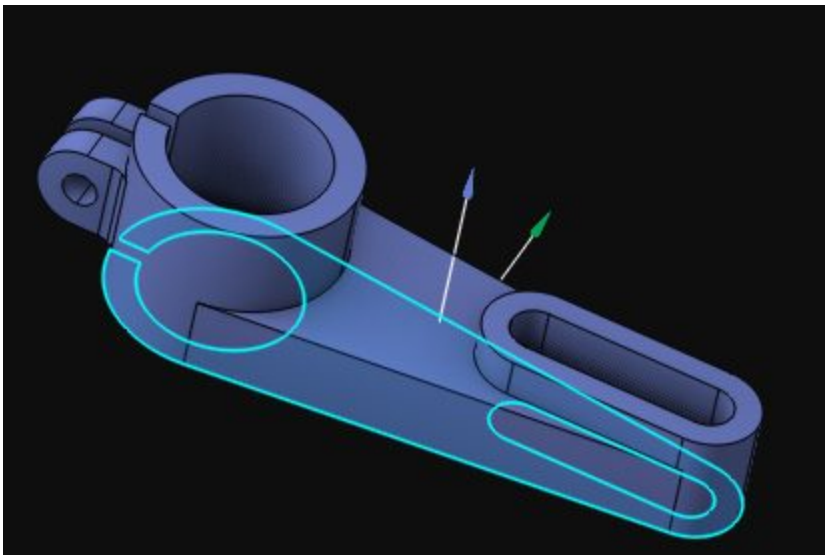
- LCS by a solid straight edge:



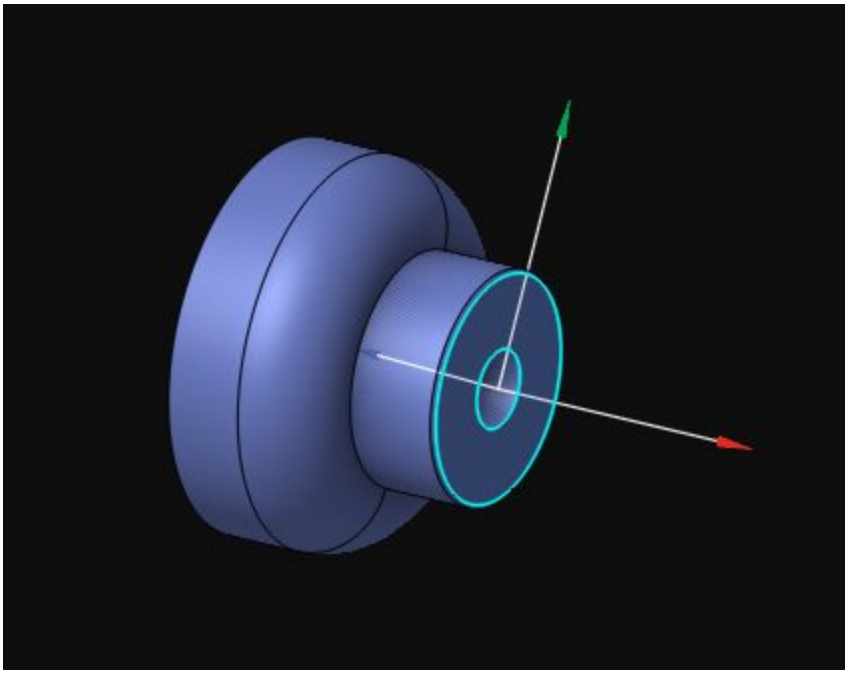
- LCS by a solid circular edge:



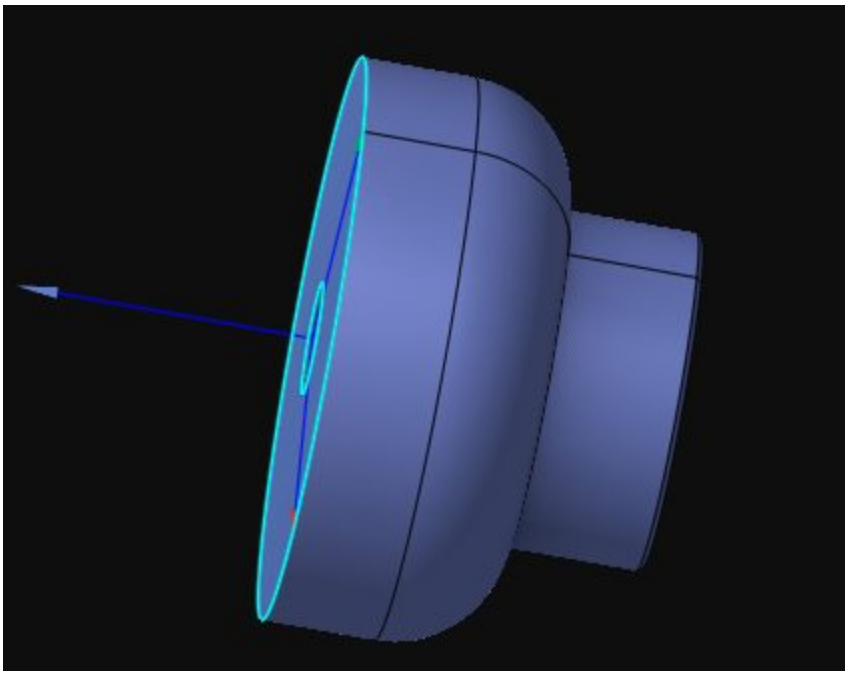
- LCS in the center of a planar face with an inverted Z axis direction:



- XOY LCS in the center of a flat face:



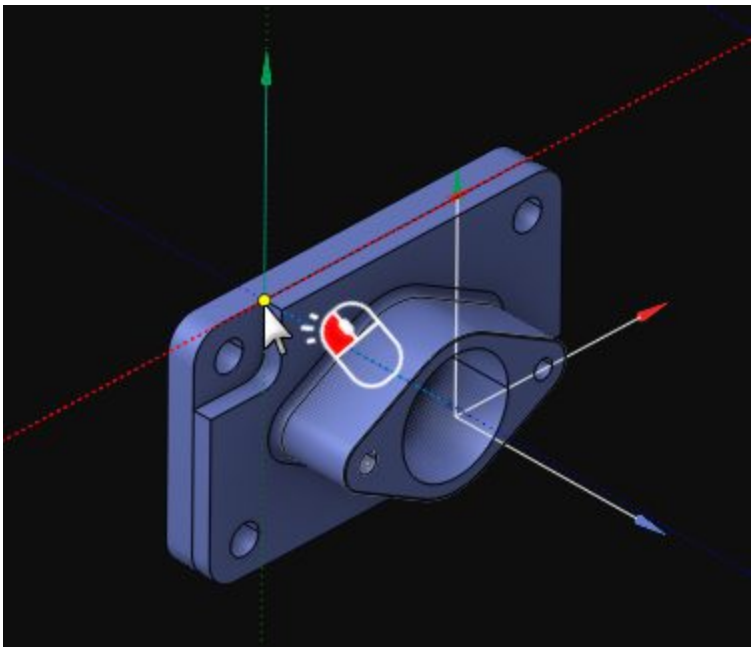
- XOY LCS in the center of a flat face with an inverted X axis direction:



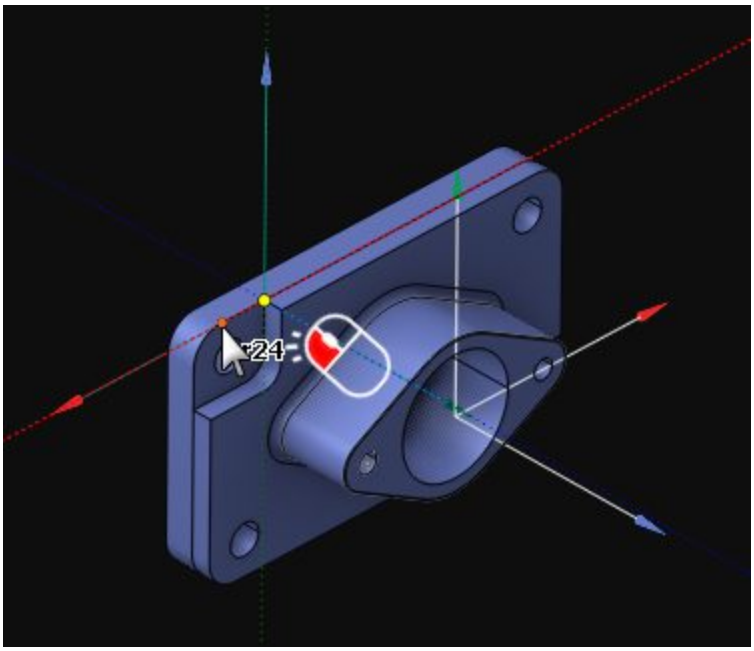
Creation of the coordinate system by the origin point and two leading vectors

Creating a new coordinate system in this way is activated by pressing the menu item on the coordinate systems panel.

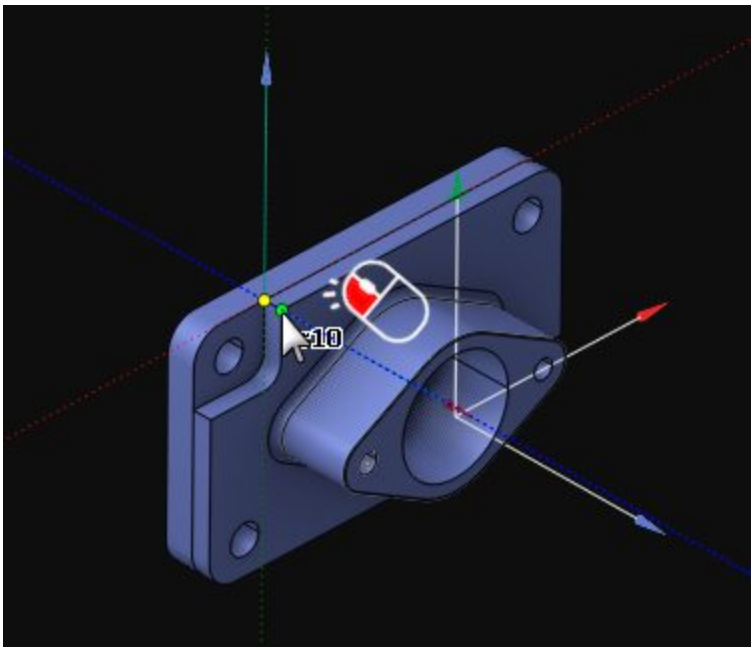
The new coordinate system is defined by the origin point and two leading vectors, X and Y. The zero point must be assigned by moving the cursor in the graphic window to the desired point. If it is a valid point to be used as the origin of the coordinate system, it will be highlighted. The selection is confirmed by left-clicking on it:



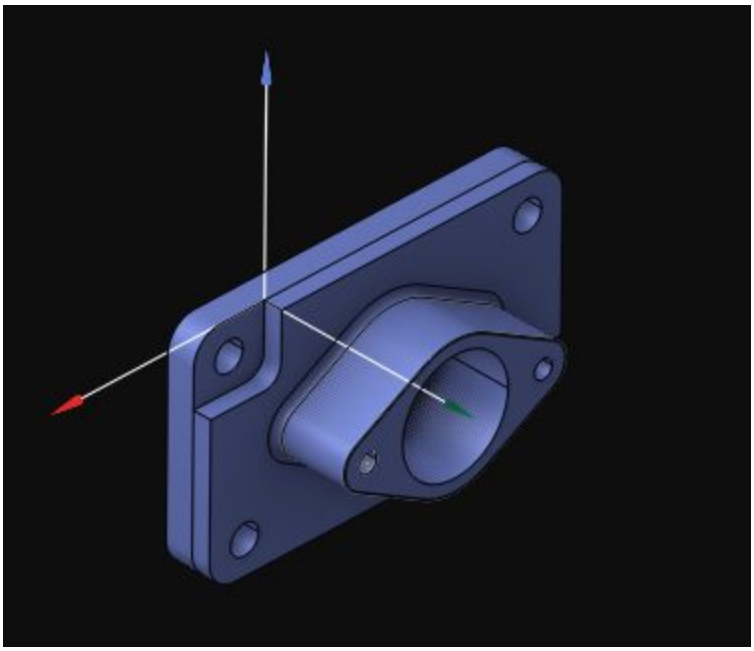
After that, the direction of the X-axis must be specified by choosing a point on the screen through which the X-axis will pass and left-clicking the mouse:



Then, the process is repeated for the Y-axis:



Afterward, the newly created coordinate system becomes active:

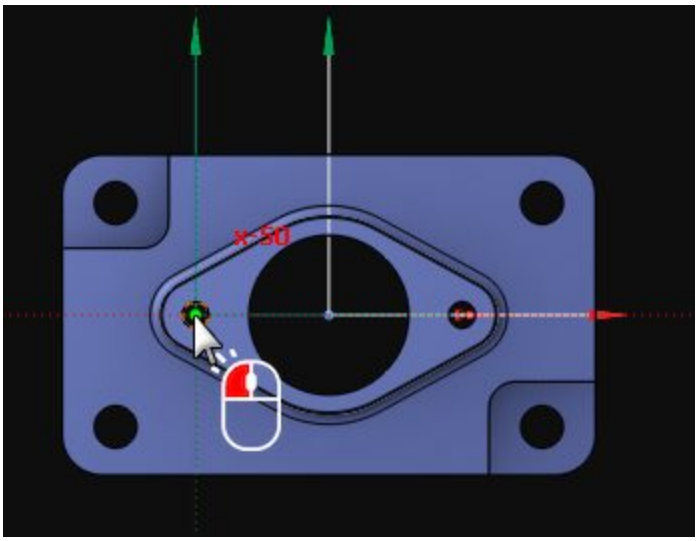


At any time, the name of the coordinate system, its color, and its origin point can be changed.

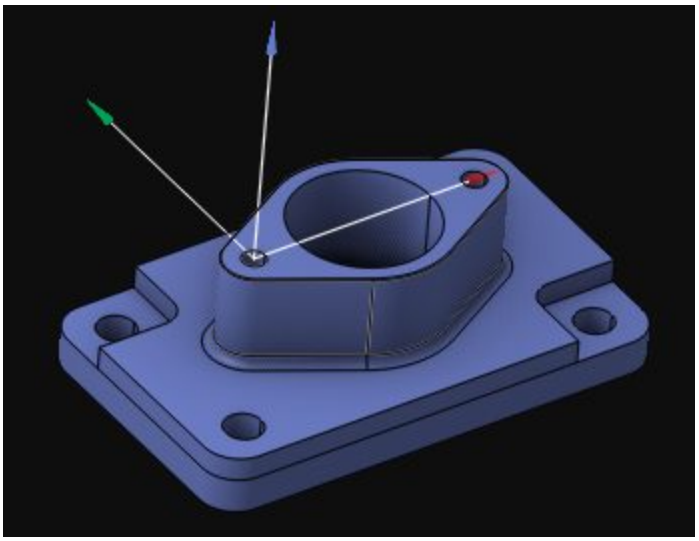
Creation of the coordinate system by the current view direction

Creating a new coordinate system in this way is activated by pressing the menu item on the coordinate systems panel.

When creating the coordinate system by this method, the required view must first be set, or one of the standard views can be selected. Next, an origin point must be assigned by moving the cursor to the desired point in the graphic window. If it is a valid point that can be used as the origin of the coordinate system, it will be highlighted. The selection is confirmed by left-clicking on it:



After that, the newly created coordinate system becomes active:

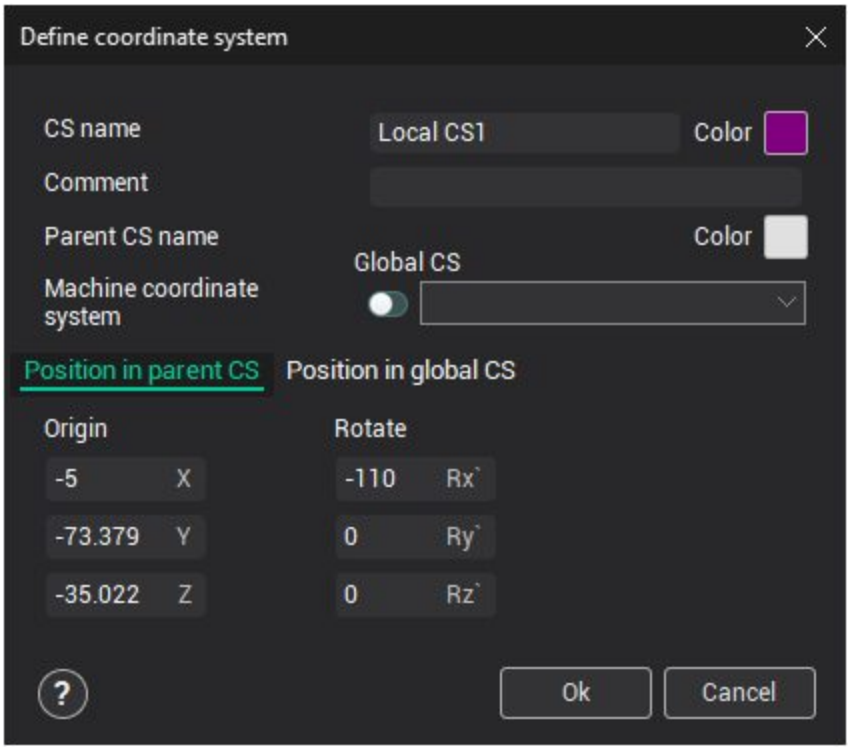


At any time, the name of the coordinate system, its color, and its origin point can be changed.

Changing properties of an existing coordinate system

To open the coordinate system properties window, press the Edit active CS... menu item on the coordinate systems panel.

The coordinate system properties window is shown below:

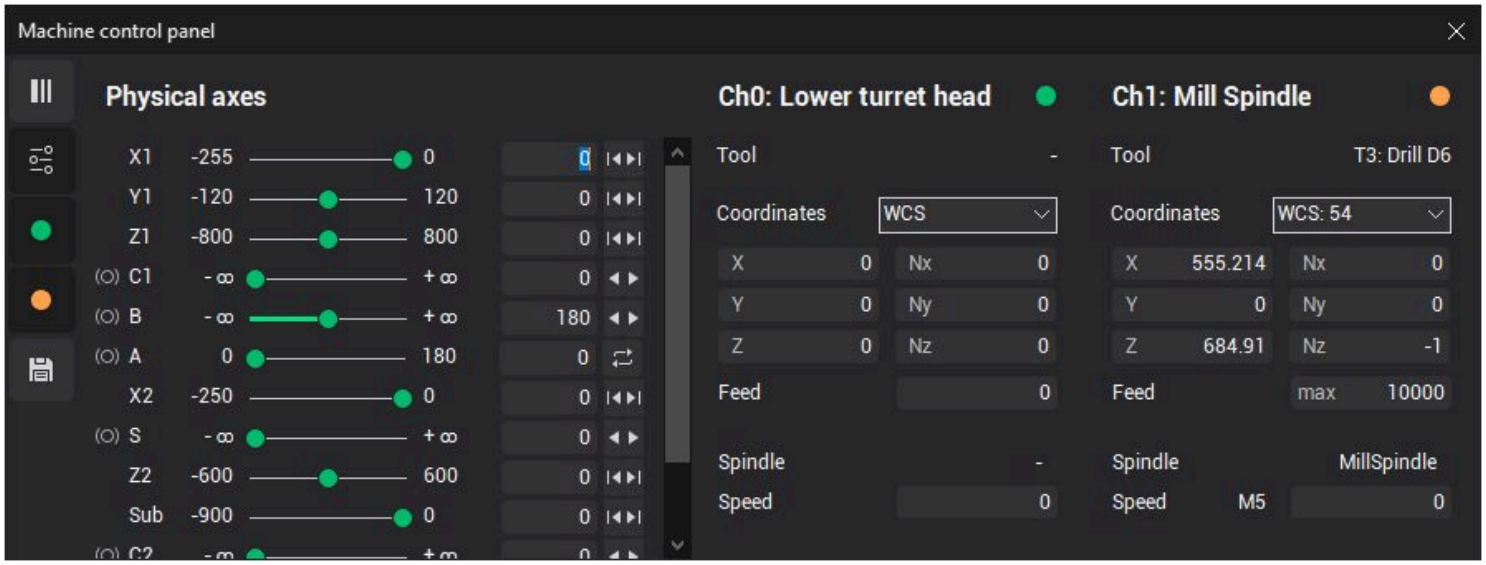


In this window, the CS name can be changed, as well as its Color and Comment.

To move or rotate the coordinate system, define the displacement value for the X, Y, and Z axes or the corresponding rotation angles. These values are relative to the Parent or Global coordinate system, depending on the active tab.

Machine axes control panel

The Machine Axes Control window allows you to monitor and modify the current values for all machine/robot coordinates. To display this window, click the  corresponding button on the main toolbar. It is helpful when analyzing toolpaths in simulation mode, constructing manual approaches, or determining the initial machine position to calculate the technological operation.



Visually, the window is divided into several areas. The number of panels and their content may vary depending on the particular machine configuration. The toolbar at the top or left side controls the visibility of these panels.

-  Switch layout: vertical/horizontal: toggles the layout of panels between vertical and horizontal.
-  Physical axes panel visibility: shows/hides the panel with the list of physical machine axes.
-  Channel panel visibility: shows/hides the panel with information about the machine's control channels. There may be several such buttons, depending on the number of channels a machine has. The icon color indicates the unique color associated with each channel. All related information will be displayed in this color when referring to that channel.
-  States panel visibility: shows/hides the panel that displays a list of stored machine states.

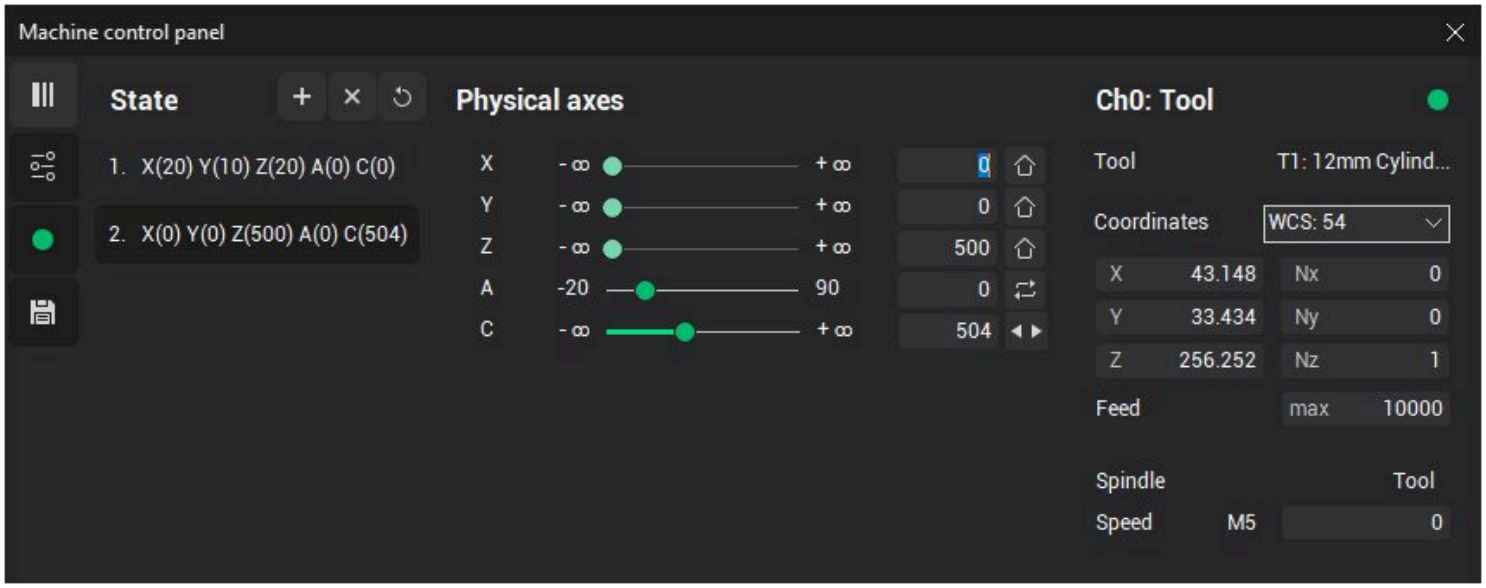
The Channel panel displays the following values:

- Color, index, and name of the channel
- Active tool number and name of the channel

- Active WCS number combo, where you can select additional coordinate systems to display tool coordinates
- Tool coordinates and orientation angles, shown as vectors or quaternions in the specified spatial CS. The type of orientation angles can vary depending on the machine schema settings.
- Current feed value and measurement units (type of feed, e.g., mm/min, mm/rev or "max" for rapid feed).
- Current spindle name, speed, rotation direction (M3, M4, M5), and rotation mode (e.g., rev/min for RPM or m/min for CSS)

The Physical axes panel displays information for each machine axis, including:

- Axis name
- Current physical value of the axis, which does not depend on the current WCS, but rather on how the axis is defined in the machine schema. This value may turn red if the axis is out of range.
- Minimum and maximum limits of the axis
- Axis brake status, if applicable:
  - (○) Brake is off
  - (○) Brake is on
- Quick modification buttons for the axis value, based on axis properties:
  - ⌂ Returns the axis to its home position
  - ⏮ Moves the axis to its minimum limit
  - ⏭ Moves the axis to its maximum limit
  - ◂ Subtracts one period for a rotary periodic axis (if within limits)
  - ▸ Adds one period for a rotary periodic axis (if within limits)
  - ⇄ Switches the machine axes to an alternative solution if the same tool-workpiece relative position can be achieved in multiple ways



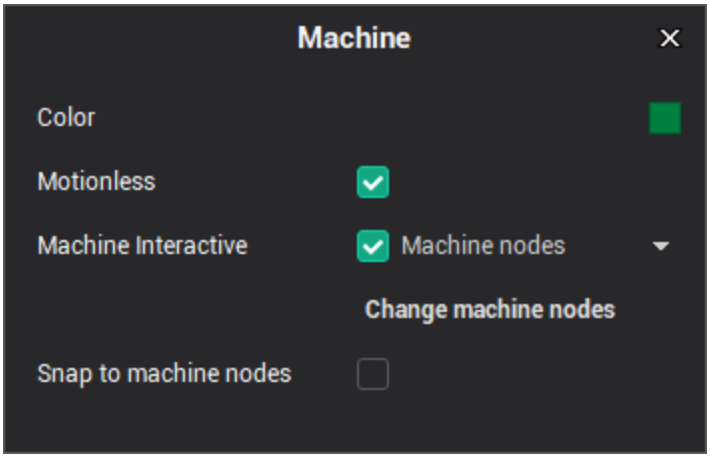
The State panel shows stored machine states and includes the following buttons:

- + Add new state: Saves the current machine state to the list
- × Delete selected state: Removes the selected state from the list
- ↺ Delete all states: Clears the entire list of stored states

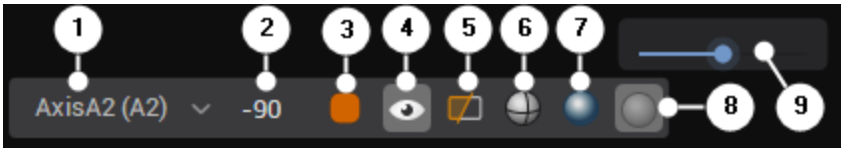
You can double-click on a state in the list to quickly switch the machine to that state. Using the Remember state feature is helpful when constructing manual transitions on machines with complex axis configurations, such as 6-axis articulated robots.

Interactive Machine

The system provides the ability to interactively control the machine and change its visualization parameters. To enable or disable this feature, use the special item in the pop-up menu of the machine visibility button.



Machine components under the cursor will be highlighted. A dotted line shows the rotation axis for rotary axes. After clicking on any machine node, the visual parameters window will be displayed:



1 — Node name

2 — Current position of the axis: You can enter the desired value or use the mouse wheel to set it. This field is shown only for movable machine nodes.

3 — Machine node color: Click to open the Select Color dialog.

4 — Machine node visibility

5 — Switch to Ghost mode: In this mode, only invisible nodes will be shown

6 — Machine node painting mode (Wire, Shade, Edge-Shade)

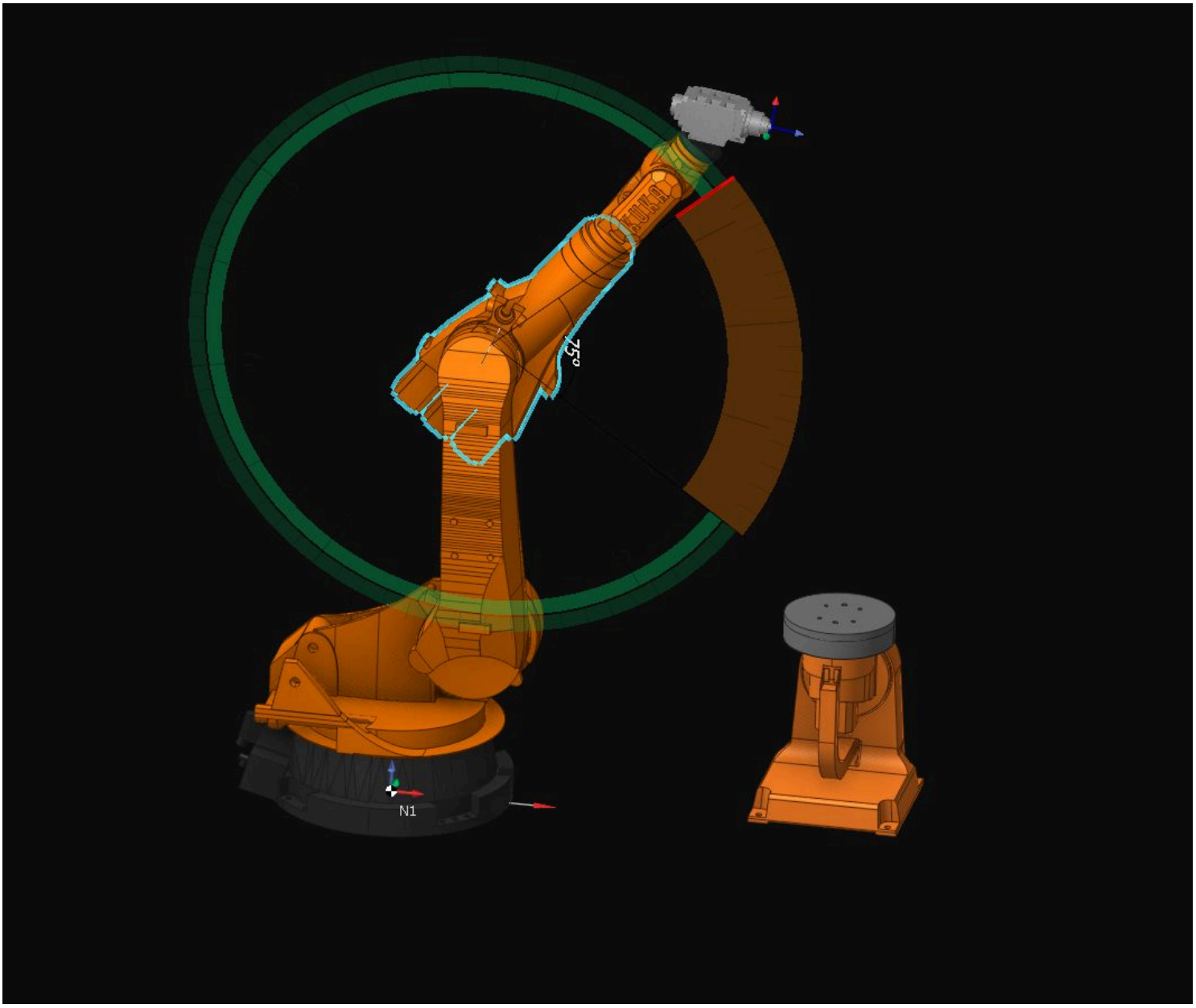
7 — Turn on metallic mode

8 — Machine node transparency

9 — Machine node transparency level

You can move the machine nodes using drag-and-drop. If a node is not movable, the nearest movable parent node will be moved. For rotary nodes, an auxiliary circle showing the rotation plane will be

displayed. The radius of the circle equals the distance from the axis of rotation to the point where the machine node is clicked.

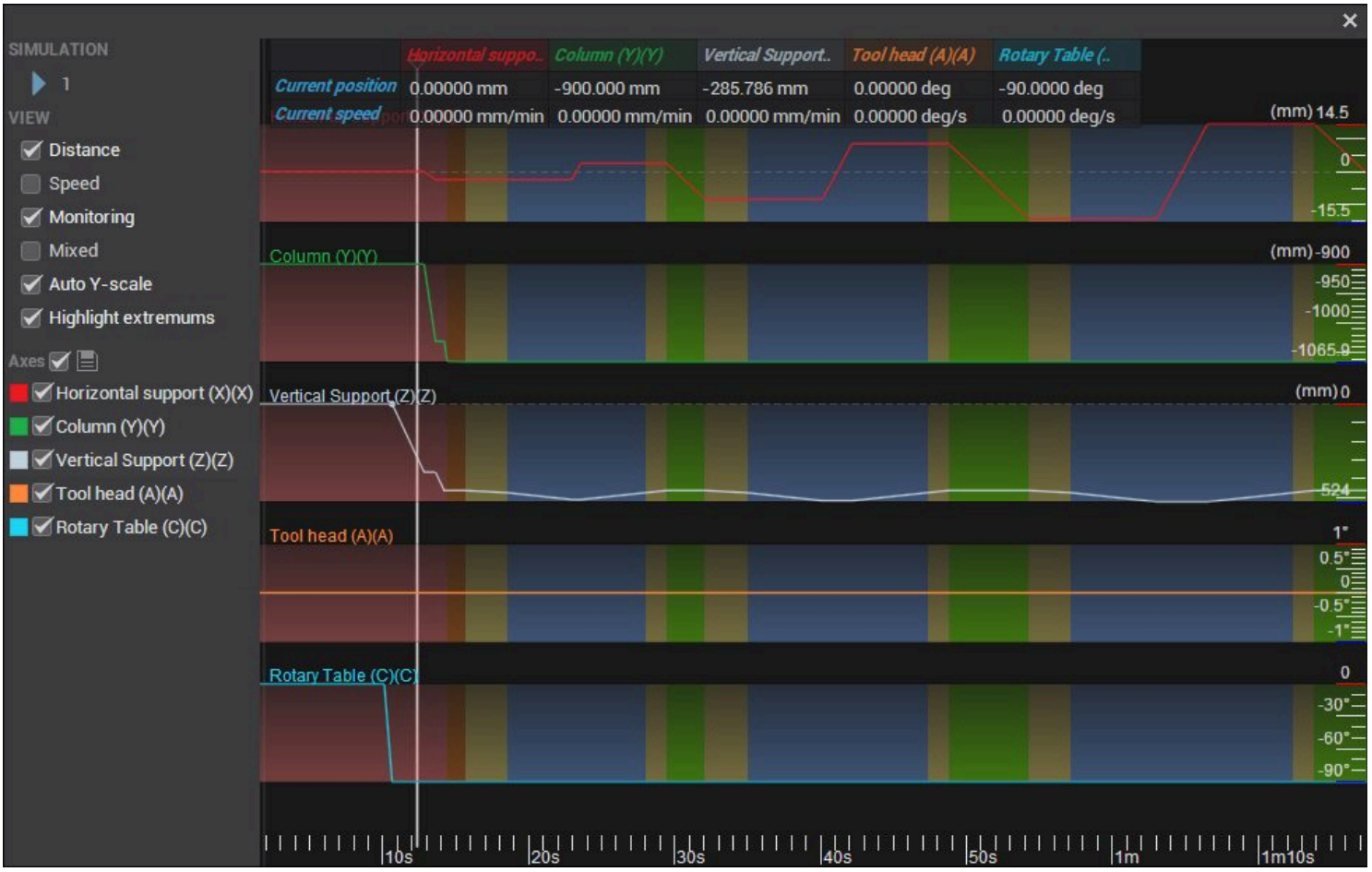


It is also possible to set the entire machine's position by defining the desired tool position. The machine will adjust the nodes' positions to achieve the specified tool position. Drag-and-drop the tool to use this feature. If the machine does not have an active tool, a yellow point will be shown instead. The tool moves along the view plane, so set the view vector along the X, Y, or Z axes for precise control.

Graph of the machine axes window

This feature allows you to view the change in the values of the machine axes over time as the tool travels along the path for the current operation, displayed as a set of graphs. To generate a graph, select the calculated operation from the list of operations, and click the corresponding button  on the main toolbar.

The window has the following layout:



SIMULATION:

Simulation. Starts tool and machine axis motion synchronously in the main graphics window and on the graph. A moving marker indicates the current time. Left-click the graph to place the marker at any point in time.

Change the simulation speed by scrolling the mouse wheel over the speed field or entering a value in the field 1 : 1.

VIEW:

Graph types:

Distance. Shows the position of each machine axis as the tool moves along the toolpath. Linear axis positions are expressed in millimeters or inches; rotary axis positions are expressed in degrees.

Speed. Shows the first derivative of axis position with respect to time: the rate at which the axis coordinate changes. This is linear speed for linear axes and angular speed for rotary axes.

Material removal rate. Shows how the rate of material removal changes over time. The volume removed during each calculation interval is divided by the duration of that interval. The graph allows tool load to be compared along different sections of the toolpath. Selecting this graph displays its calculation settings and controls.

Display:

Monitoring. Displays a table at the top of the window with the current position and speed of each visible axis. Linear axis positions use the machine's length units, typically millimeters or inches, and linear speeds are shown in mm/min or in/min. Rotary axis positions use the machine's angular units, typically degrees, and rotary speeds are shown in deg/s. The values update as the time marker moves.

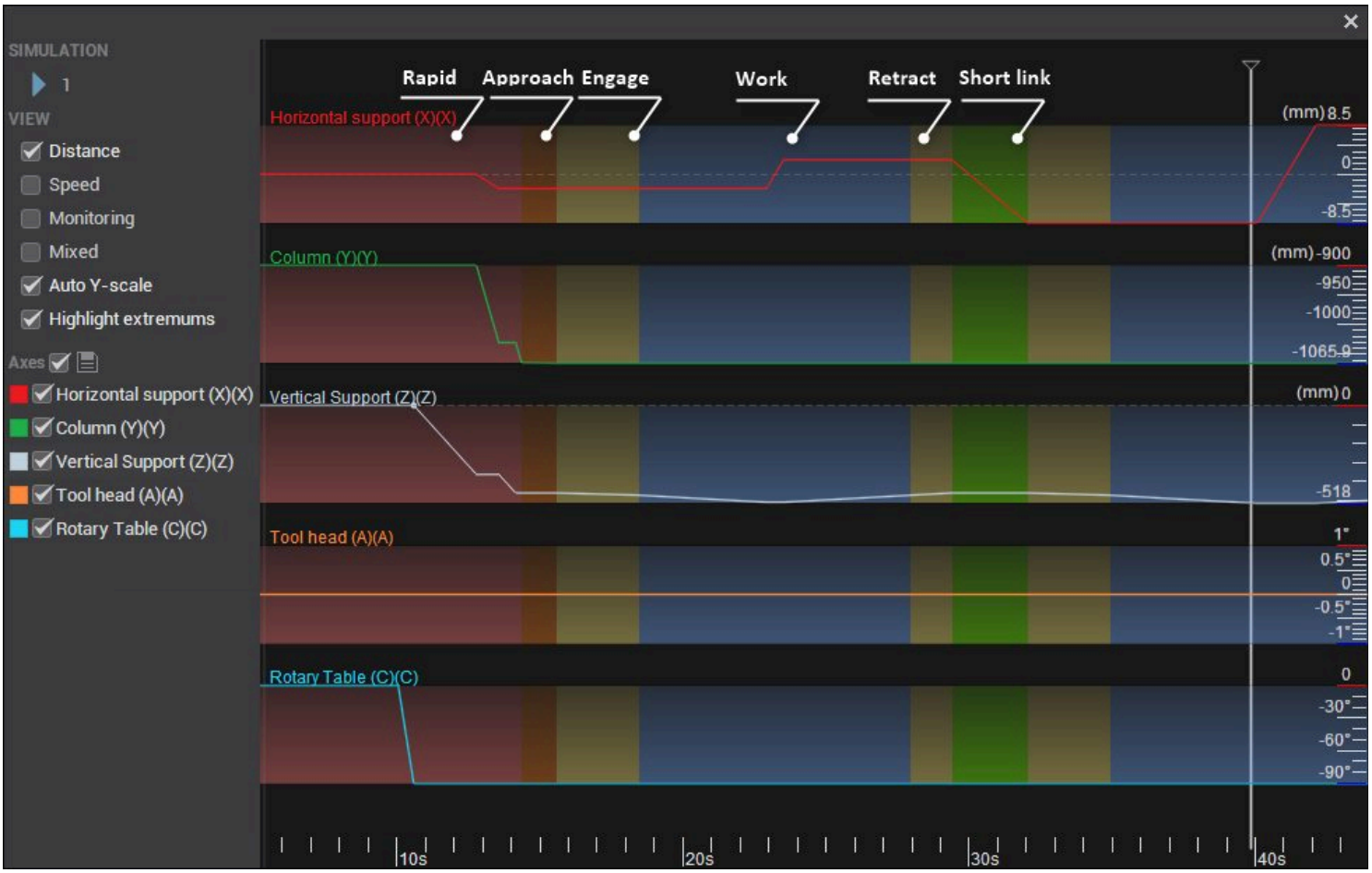
Mixed. Displays all selected axes on a single graph. When disabled, the axis graphs are displayed separately.

Auto Y-scale. Automatically adjusts the vertical scale to the minimum and maximum graph values. When the axes are displayed separately, this allows each axis graph to fill the same height. Rotate the mouse wheel with the pointer over the graphics area to change the horizontal scale.

Highlight extremums. Marks points where the direction of change of an axis coordinate reverses. This option is available only for the Distance graph. It is unavailable for Speed and Material removal rate.

Axes. Allows individual axes to be hidden and their colors to be changed by clicking the colored square.

The background color beneath the axis graphs indicates the feed type: working, rapid, approach, return, and so on.



Material removal rate calculation:

Tool load threshold. Sets a threshold for the material removal rate. Graph sections with high tool load are highlighted relative to this threshold. Enter a value and press Enter to apply it. The threshold can also be adjusted by dragging its line on the graph.

Calculation step. Sets the maximum step along the toolpath when calculating the graph. Click Recalculate graph to apply a new step to the graph.

High model resolution. Uses a high-resolution simulation model for the calculation. When disabled, the resolution specified in the simulation settings is used. Recalculate the graph after changing this option.

Recalculate graph. Starts calculation using the specified step and model resolution. The previous graph data is cleared at the start of the calculation.

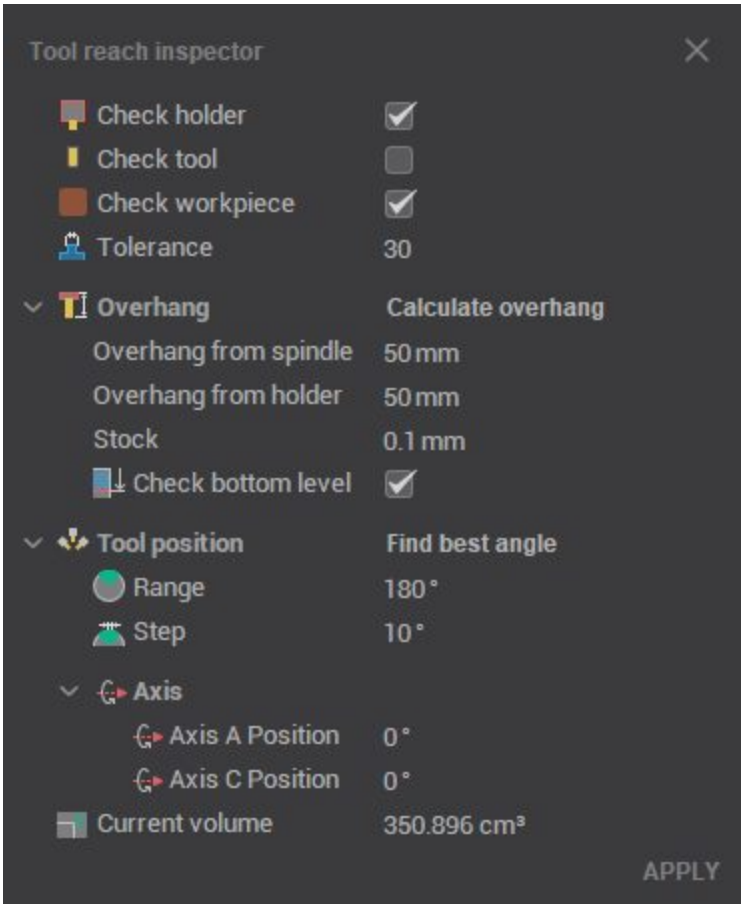
Break process. Replaces Recalculate graph while a calculation is running and stops the calculation when clicked.

Holder occlusion check utility

The Holder Occlusion Check panel allows you to view zones of the part where the toolholder has no collisions, and to determine the best angle for part processing.

This feature is helpful in preparing the technical process during part manufacturing.

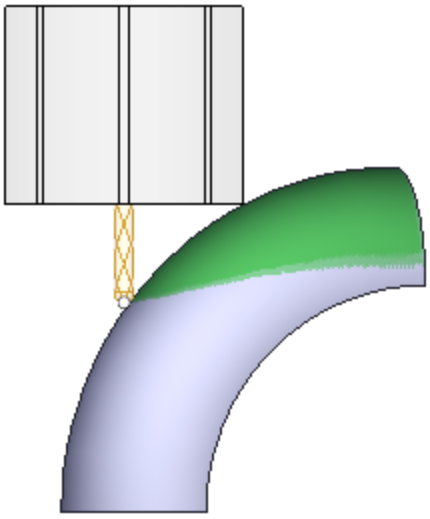
To display this window, click the corresponding button  on the Control tab of the main panel.



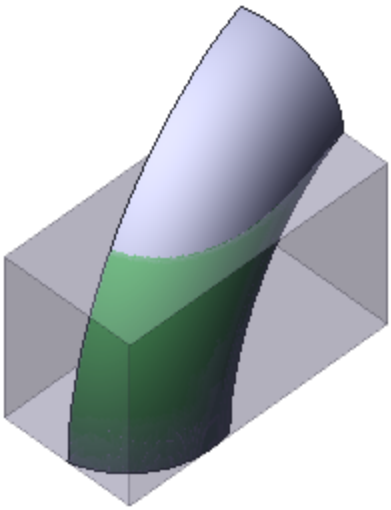
Visually, the window is divided into two areas. The content of these areas may vary depending on the specific machine configuration.

The top panel displays the main operation parameters that affect the safe zone size for the toolholder.

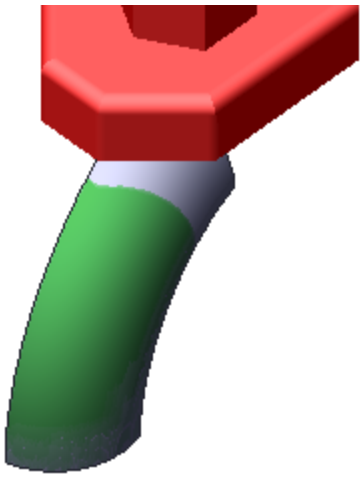
- Check Holder: Detects segments of the toolpath where the toolholder collides with the part, and modifies those segments according to the specified strategy. This feature helps prevent collisions by adjusting the toolpath to avoid contact between the toolholder and the part, ensuring safe and efficient machining even in complex areas.



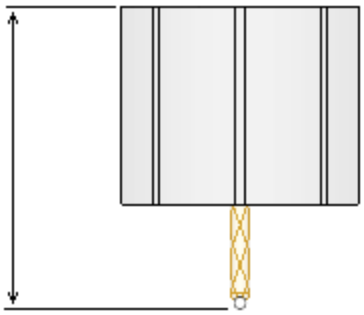
- Check Workpiece: Detects collisions between the tool and the workpiece during machining. Using this feature helps ensure that the tool can safely access the necessary areas of the part without damaging the workpiece, especially when working with complex geometries.



- Check Fixtures: Checks for potential collisions between the tool and any fixtures during machining. Using this feature ensures that the tool avoids collisions with clamps, chucks, or other fixtures, which is essential for preventing damage to both the part and the machine setup during complex operations.

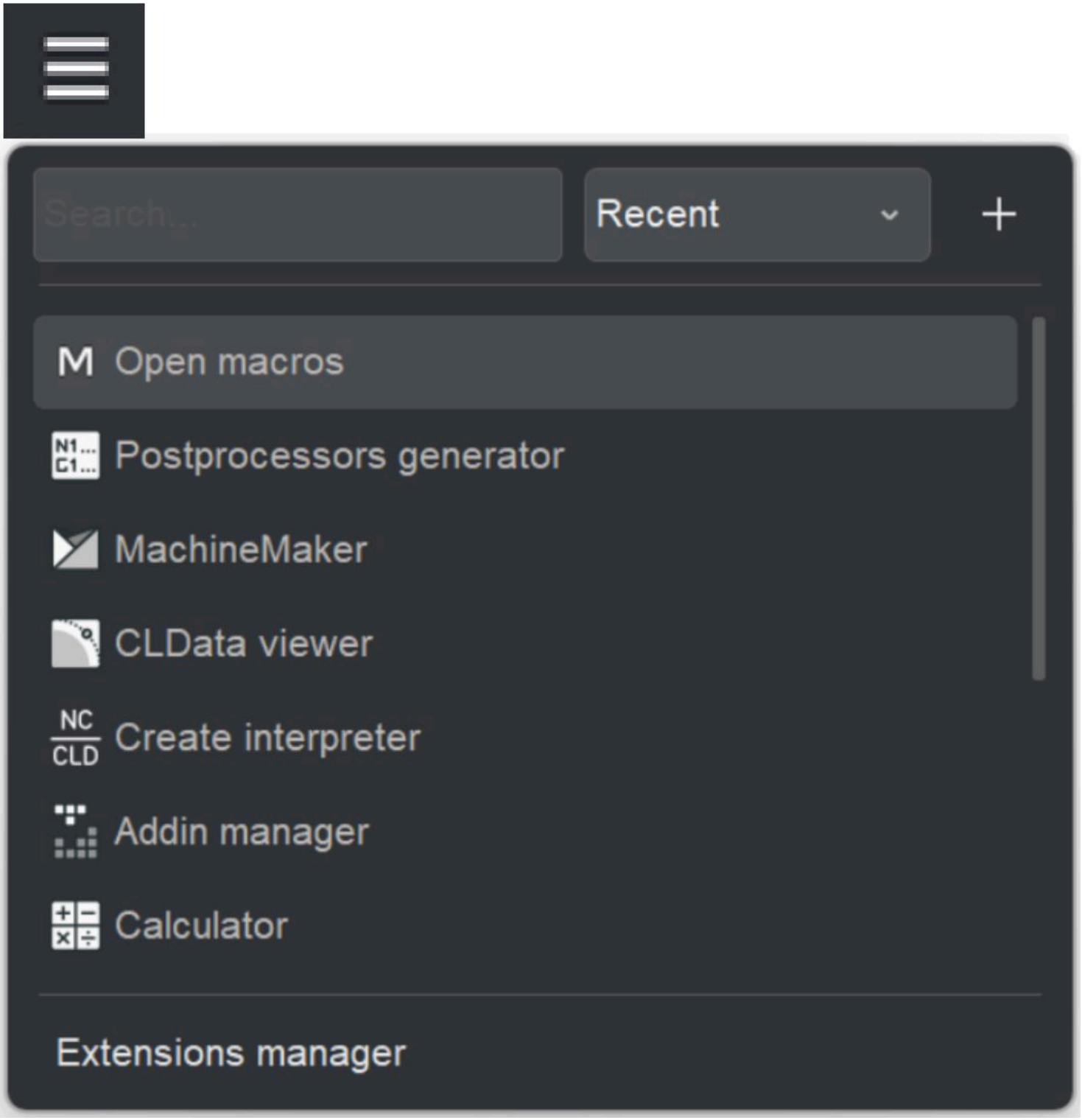


- Overhang: Allows you to set the length of the toolholder extending from the spindle. Setting the correct overhang value is crucial for ensuring the toolholder has enough clearance to avoid collisions and optimize machining performance by minimizing deflection and tool vibration.



At the bottom of the window, the Rotary Axis panel displays information about the current machine axis status and provides a tool for finding the optimal tool angle for processing.

Utilities button of main panel



Application area:

The Utilities window provides quick access to built-in and user-defined tools directly from the main application window. It combines search, sorting, launching, management of custom utilities, and access to the extensions manager in a single panel.

The window is opened with the hamburger button (three horizontal lines) on the main panel.

The window is used to locate and launch auxiliary tools without leaving the main application.

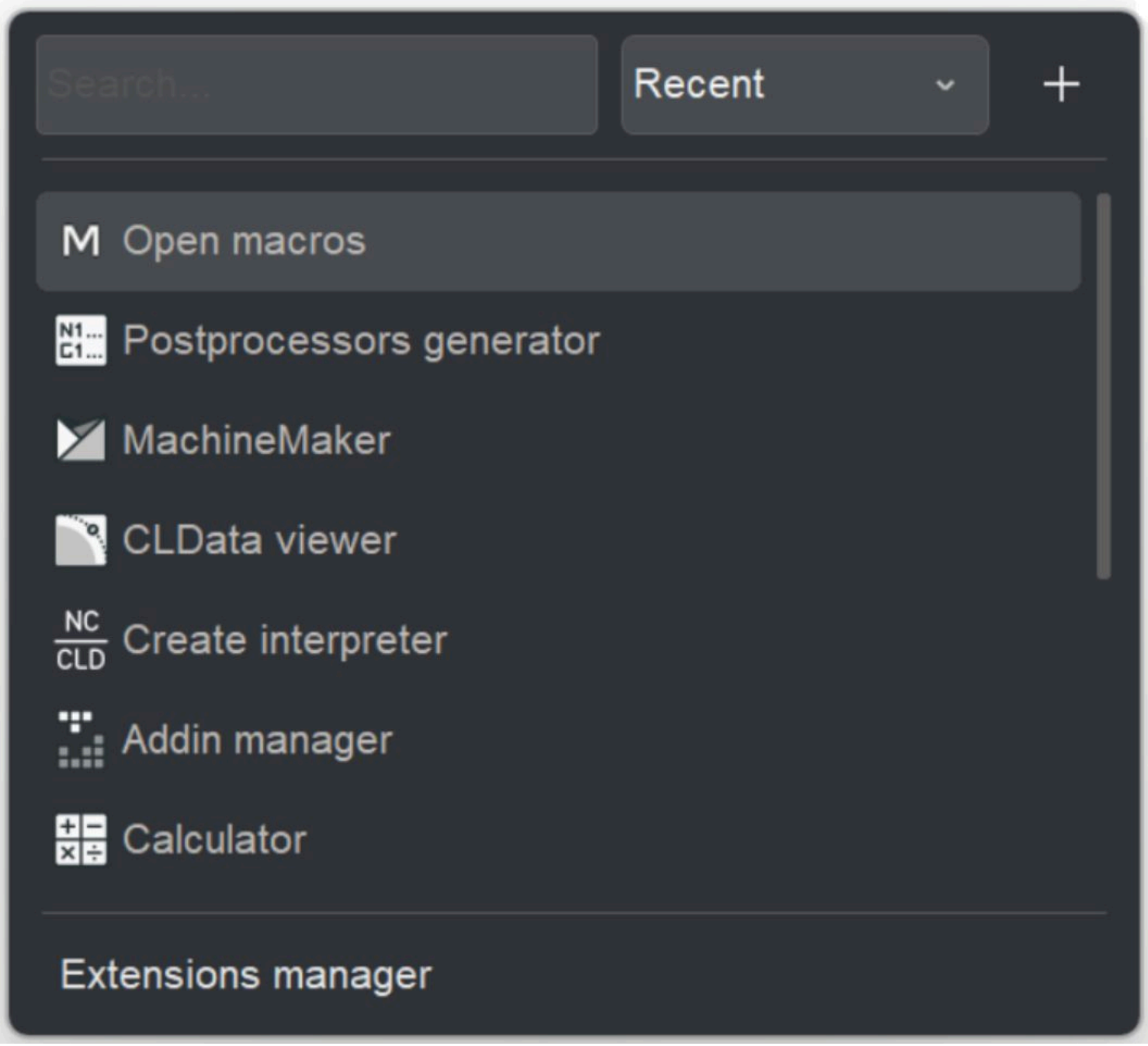
Typical scenarios:

- Starting a built-in tool — Postprocessors generator, MachineMaker, CLData viewer, Create interpreter, Addin manager, Calculator, Open macros and etc.
- Registering and launching an external program, library, or script as a custom utility.
- Opening the Extensions manager to manage installed extensions.

Window layout.

The window consists of a top control bar and the utilities list.

- Search field. A text field on the left of the control bar for filtering the list.
- Sort selector. A drop-down list for choosing the sort order.
- Add button (+). A button on the right of the control bar that opens the New Utility dialog.
- Utilities list. The scrollable list of all available utilities, each shown with an icon and a name.
- Extensions manager. A link at the bottom of the window. Details



Launching a utility.

Launch. Each item in the list represents one utility. To run a utility, click it with the mouse or select it and press *Enter*. The window closes automatically after launch.

Searching.

Search field. The search field at the top of the window filters the list by utility name.

- The search is case-insensitive and matches by substring — any part of the name.
- Typing any printable character automatically moves the focus to the search field, so the search can be started without clicking the field first.

To clear the filter, delete the text in the search field. The full list is then restored.

Sorting.

Sort selector. The drop-down list to the right of the search field switches the sort mode of the list:

- Recent — by usage frequency, most frequently used utilities first.
- A – Z — in alphabetical order by name.
- Z – A — in reverse alphabetical order.

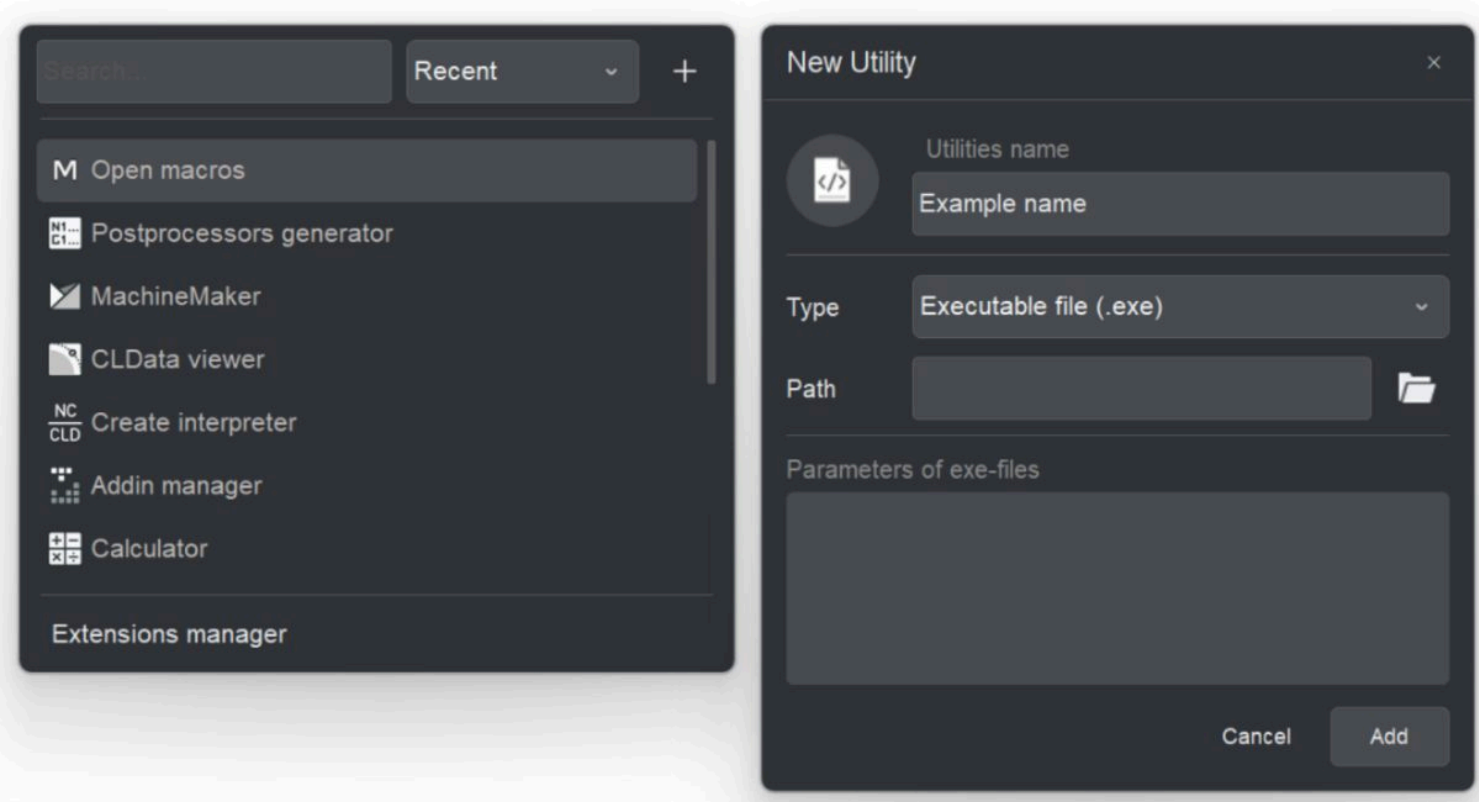
Adding a custom utility.

Add button (+). The + button opens the New Utility dialog for creating a custom utility.

The dialog contains the following fields:

- Icon. The icon shown for the utility in the list. Click the icon area to assign a custom image.
- Utilities name. The display name of the utility in the list.
- Type. The type of the utility. The supported types are:
  - Executable file — applications with the extensions.exe.
  - Library —.dll.
  - Script —.spr.
- Path. The path to the utility file. Click the folder button to the right of the field to select the file through the standard file-open dialog.
- Parameters of exe-files. Optional command-line parameters passed to the utility on launch. The field is available for all utility types and does not change when the type is switched.

Click Add to create the utility and add it to the list, or Cancel to close the dialog without saving.



Editing and deleting custom utilities.

When a custom utility is hovered in the list, two controls appear on the right of its row:

- More button (⋯). Opens the Utility parameters dialog, where the name, type, path, parameters, and icon can be changed.
- Delete button (×). Removes the custom utility from the list.

> Built-in utilities are protected from modification. The editing and deletion controls are available for user-defined utilities only.

Built-in tools.

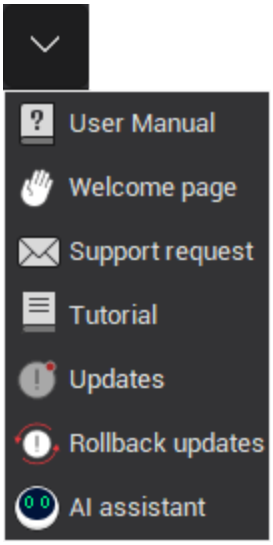
The Utilities menu contains items for running additional internal tools and external user utilities. It can be customized with the Configure utilities menu item. Some items may be hidden depending on the Show expert tools option in the system settings.

Button	Description
	Configure utilities: Opens the Utility manager window.

Button	Description
	MachineMaker: Starts the application to create Machine schemas.
	Calculator: – Runs the internal calculator.
	System logs: Opens a window displaying a list of events, that have occurred in the application since launch.
	Postprocessors generator: Launches the Postprocessors generator, an application for generating postprocessor tuning files.
	Create script operation: Opens a dialog to create a scriptable operation.
	Create interpreter: Opens the Create interpreter tool.
	Addin Manager: Opens the Addin Manager window.
	Operations manager: Starts the manager that allows enabling/disabling external operations, whose parameters are described in separate XML files andstfc-containers.
	CLData viewer: Starts the CLData viewer, an application that helps in creating C#-based postprocessors.
	Scripts IDE: Opens a window for writing and debugging CAM system script files.

Help button of main panel

The Help menu consists of actions that allow you to find answers to any questions that may arise.

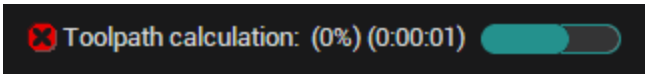


Button	Description
	User Manual: Shows the CAM system help.
	Welcome page: Opens a page with up-to-date information from the CAM system website.
	Support request: Prepares a message to the technical support service of ENCY SOFTWARE LTD., e-mail: support@encycam.com.
	Tutorial: Runs the CAM system tutorial.
	Updates: Checks for available updates.
	Rollback updates: Rollback to a previous version of the software, if necessary.
	AI Assistant: A virtual assistant that helps users with various aspects of the CAM workflow.

Process indicator

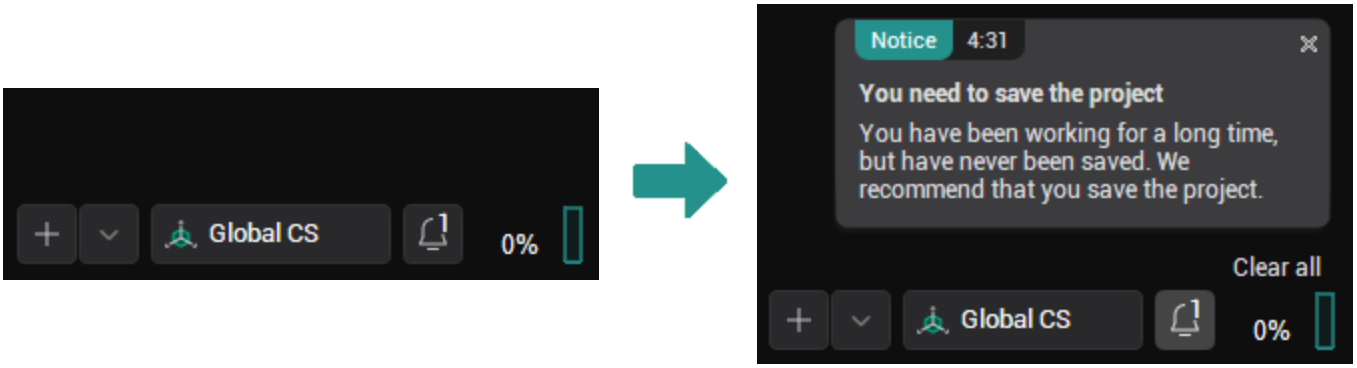
The Process indicator runs when the system performs lengthy operations, such as geometrical model import, toolpath calculation, machining simulation, etc. Left-clicking on the indicator cancels these operations, but only after a confirmation prompt.

The Process indicator is located at the bottom of the main window.



Application events notifications

There is a mechanism to inform the user about events in the application that require special attention — pop-up notifications. When an event occurs, a small window pops up in the lower right corner of the main window, briefly describing the event. The icon in the corner also displays the total number of such notifications. By clicking on it, you can open a panel with the complete list of notifications.



Multiproject interface

Multiproject allows you to combine multiple windows as "tabs" for simultaneous work on multiple projects. You can use the multiproject interface to attach and create new projects in separate instances. However, you can still choose to work with only one window, without using the multi-project interface.

Options for opening an additional window:

1. Opening an additional window from the current system window: The workflow will look as follows:



+ User Actions =


- With an open system window, click on the New Project in New Tab button. Hover over New Project  and select New Project in New Tab from the drop-down list. A new tab will appear in the project list.
- With an open system window, click on the New Project in New Tab button. Hover over Open Project  and select Open Project in New Tab from the drop-down list.
- You can also open a project by hovering over a specific project in the Open Project  drop-down list and selecting Open Project in New Tab from the second drop-down list.
- To open a project with machine setup, click the New Project with machine setup in a New Tab button. Hover your cursor over the New Project  and select New Project with machine setup in new tab  from the drop-down list. A new window will appear in the project list.

2. Opening an additional window through external systems:

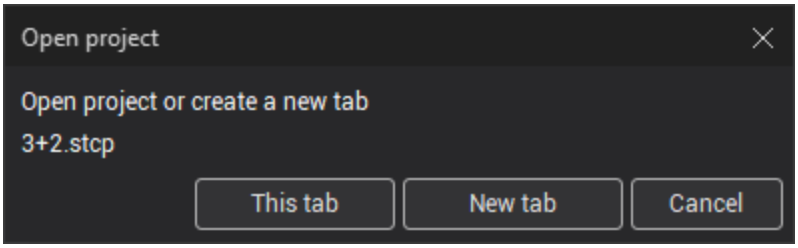
- With the system window open, click on the Export from External CAD System option in the CAD system window.
- With the system window open, click on the Export from Machine Maker option.
- With the system window open, click on the Export from Project Library option.
- Launch a project by double-clicking the project icon or by using the drag-and-drop function.

The workflow will look as follows:



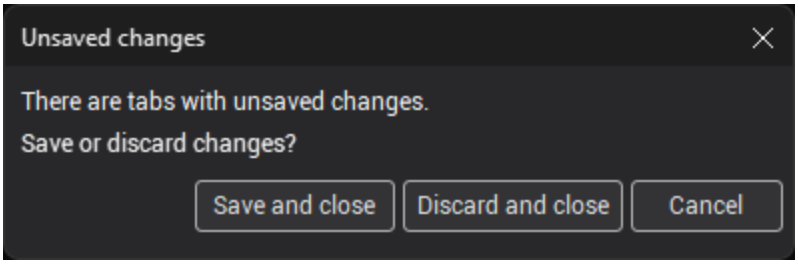
Important points when opening an additional window through external systems:

1. If the window is empty (no operations, parts, etc.), imported files will load into this window.
2. If there are changes in the current window (and it's not empty), the system will prompt you to choose into which window the import should be performed.



Therefore, the user decides how they want to work, either in single-tab mode or with multiple tabs simultaneously (multi-project mode).

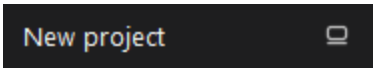
When exiting the system while working in multiple tabs with unsaved changes, the system will suggest saving them. It will initially prompt to save the project from the current tab, and this will continue until all projects in all tabs are closed or saved.



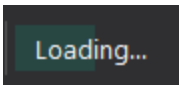
Project Navigation:

Working with the project list:

- The project name panel displays the name of the project file open in the current tab



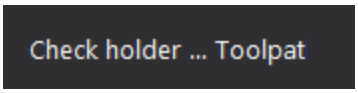
- The number of tabs corresponds to the number of projects combined in the current system window.
- An animated loading progress bar appears when a new tab is created and added to the existing ones



- If there are multiple tabs, the title of the active project is highlighted with a darker color compared to the others. Clicking on the title of an inactive project will make it the active one. Additionally, hovering over the project name panel of an inactive tab will reveal a close button.



- If a calculation is running in a tab (e.g., toolpath calculation, machining simulation), a brief description is shown.

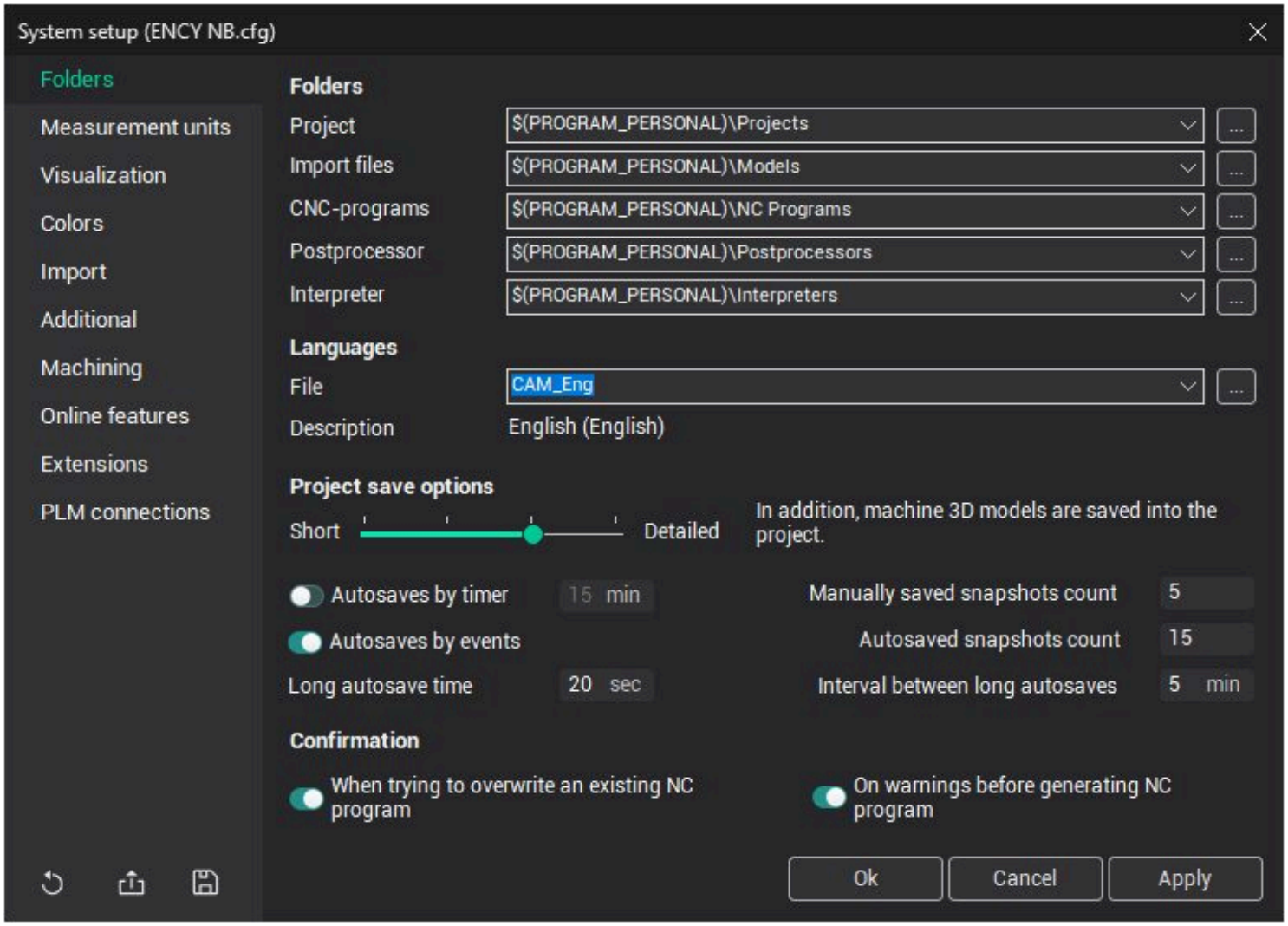


You can also copy operations between different tabs using [Ctrl+C] and [Ctrl+V] or the Copy and Paste commands.

In multiproject mode, it's not possible to open two windows with the same name. In such cases, the system will generate an error message.

System settings window

The System Settings window can be opened by pressing the button on the main panel.



All default parameters are saved in a .cfg file. User-defined parameters can be saved or loaded using the buttons on the right-hand side of the window. Users can create as many configuration files as needed.

Buttons in the System setup window:

- Save defaults: Saves the settings into a user-defined configuration file and closes the window. The saved file becomes the current configuration, and the settings are applied for the current session.
- Load saved defaults: Loads settings from the selected configuration file and makes them the current settings. The loaded file will be used as the default in the next launch.
- Default settings: Resets all user-defined settings back to their default values.
- OK: Saves the settings into the current configuration file, applies them for the current session, and ensures they are applied in future sessions. Closes the window.

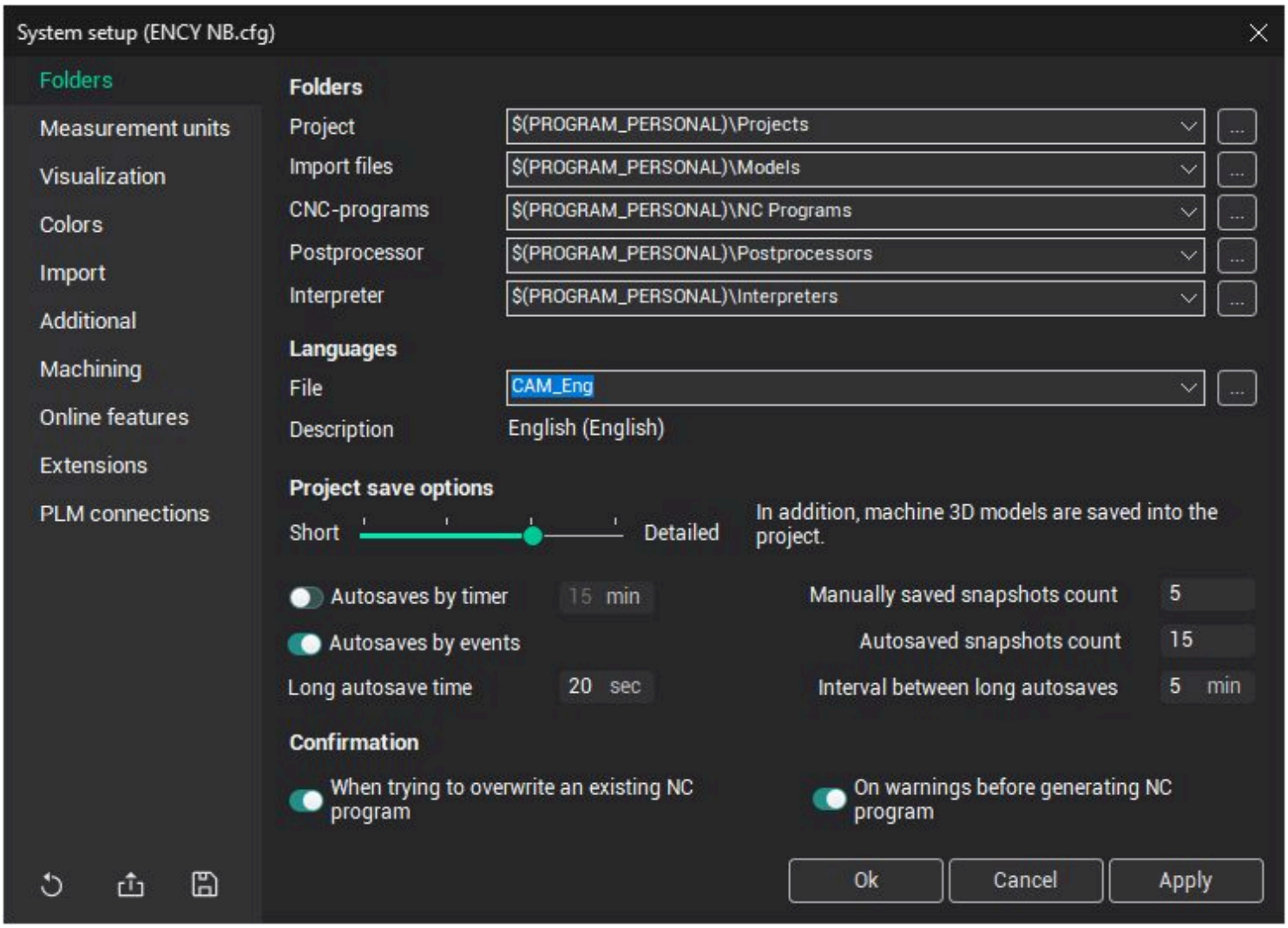
- Cancel: Closes the window and discards any changes made.
- Apply: Applies the new parameters for the current session only, without saving them for future sessions.

If, when launching the CAM system, the current configuration file is missing, a new configuration file will be created automatically, and all system settings will be reset to their default values.

The window contains the following tabs:

- Folders
- Measurement units
- Visualization
- Colors
- Import
- Additional
- Machining
- Online features
- PLM extensions
- PLM connections

Folders tab



In this window, the user can set default paths for the CAM system files.

- Project: The default folder for loading and saving projects.
- Import files: The default folder for loading geometrical model files (IGES, DXF, etc.).
- **`CNC-programs`**: The default folder for saving NC-programs generated by the postprocessor.
- Postprocessor: The default folder for loading postprocessor files.
- Interpreter: The default folder for loading interpreter files.

The paths can be entered manually or by using the path selection dialog, which can be accessed by clicking the ... button.

Pre-defined variables for defining directories:

- **`$(ENCYDIR)`**: The directory from which the CAM system was launched.
- **`$(PRJDIR)`**: The directory defined in the Project field.
- **`$(PROGRAM_PERSONAL)`**: The directory created in the user's personal documents folder My Documents during installation.

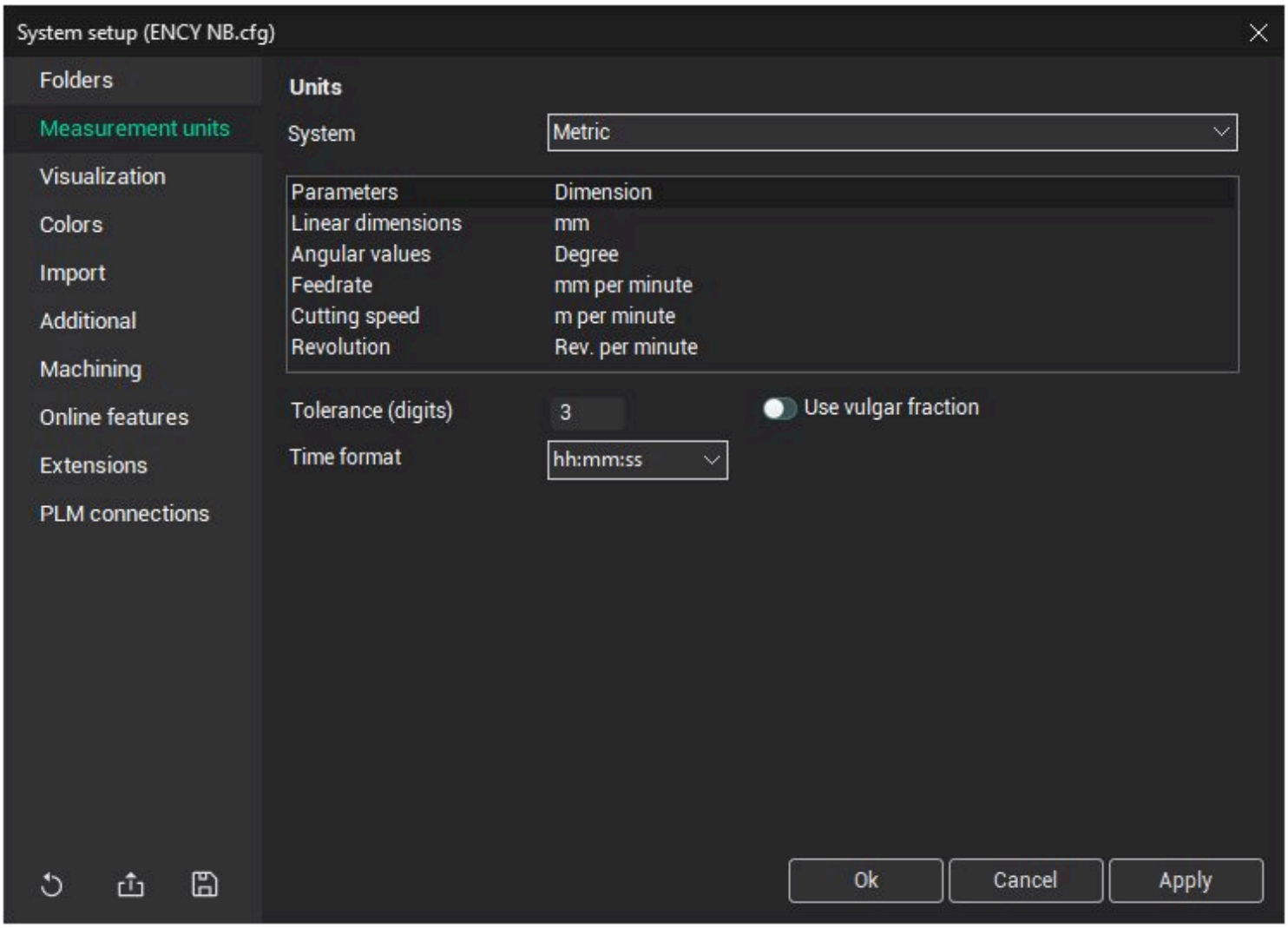
- **`$(PROGRAM_COMMON_DOCUMENTS)`**: The directory created in the public documents folder Public Documents on the local computer during installation.

When defining the actual directory paths used by the CAM system, these variables will be replaced by the corresponding full paths at system start-up or based on user-defined settings.

Additional settings:

- System languages can be changed on the Languages panel. The language change will be applied to the current session.
- Project save options: These options control the size of the project files. The more detailed the information saved, the larger the project file. The following methods are available, listed from saving the minimum necessary information to the most complete:
  1. Only source data (geometrical models, coordinate systems, operations, and their parameters) are saved.
  2. In addition to the above, the toolpaths of operations are saved.
  3. In addition to the above, 3D models of the machine schema files and tools are saved.
  4. In addition to the above, intermediate workpiece states are saved.
- Autosaves and snapshots: A group of options that allows configuring the creation of backup versions of projects, including Autosaves by timer, Autosaves by events, Manually saved snapshots count, Autosaved snapshots count, Long autosave time, and Interval between long autosaves. See the Autosaves and project snapshots article for more details.
- Confirmations: It is possible to disable the confirmation message during NC file generation when overwriting an existing file on the Confirmation panel. You can also configure confirmations on warnings before generating NC programs, providing additional control over critical steps in the workflow.

Measurement units tab



The Measurement Units settings allow you to define the measurement units for the system.

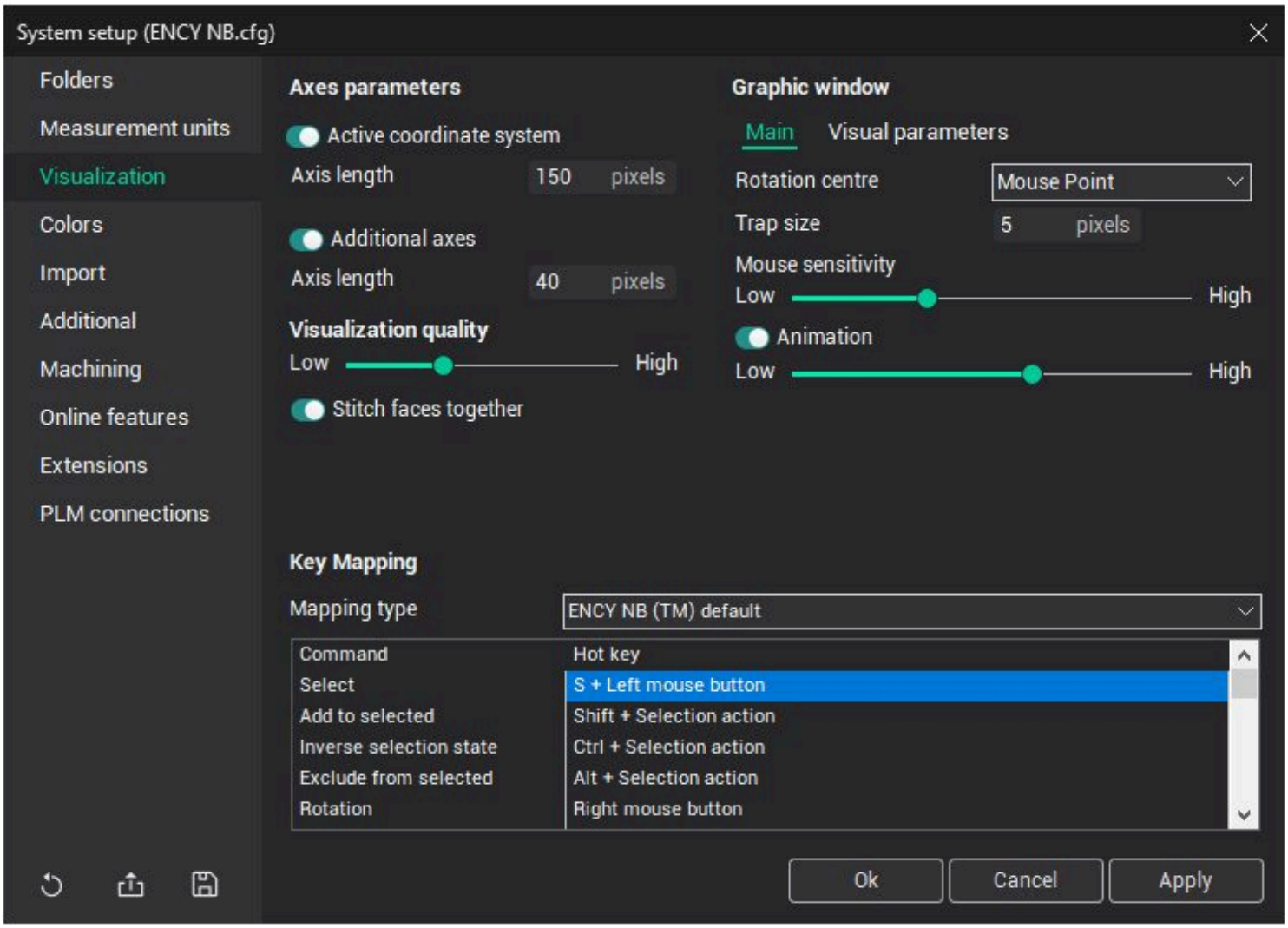
Measurements are based on the units used in the imported model. The output data (NC) is created using the same units. Consequently, to generate an NC program for a CNC milling machine in millimeters (or inches), all measurements of the model must be in millimeters (or inches).

Angular measurements are given in degrees with decimals.

Additional Settings:

- **`Tolerance (digits)`**: Defines the number of digits after the decimal separator for values output into the NC program. It is recommended to set this value at least one digit higher than the maximal NC machine tolerance (or digits count after the decimal separator in the NC program).
- Time format: Specifies the format for time output, such as for machining time.
- Use vulgar fraction: This option allows the system to use vulgar fractions where appropriate.

Visualization tab



The Graphical Window Settings control the visualization and display parameters.

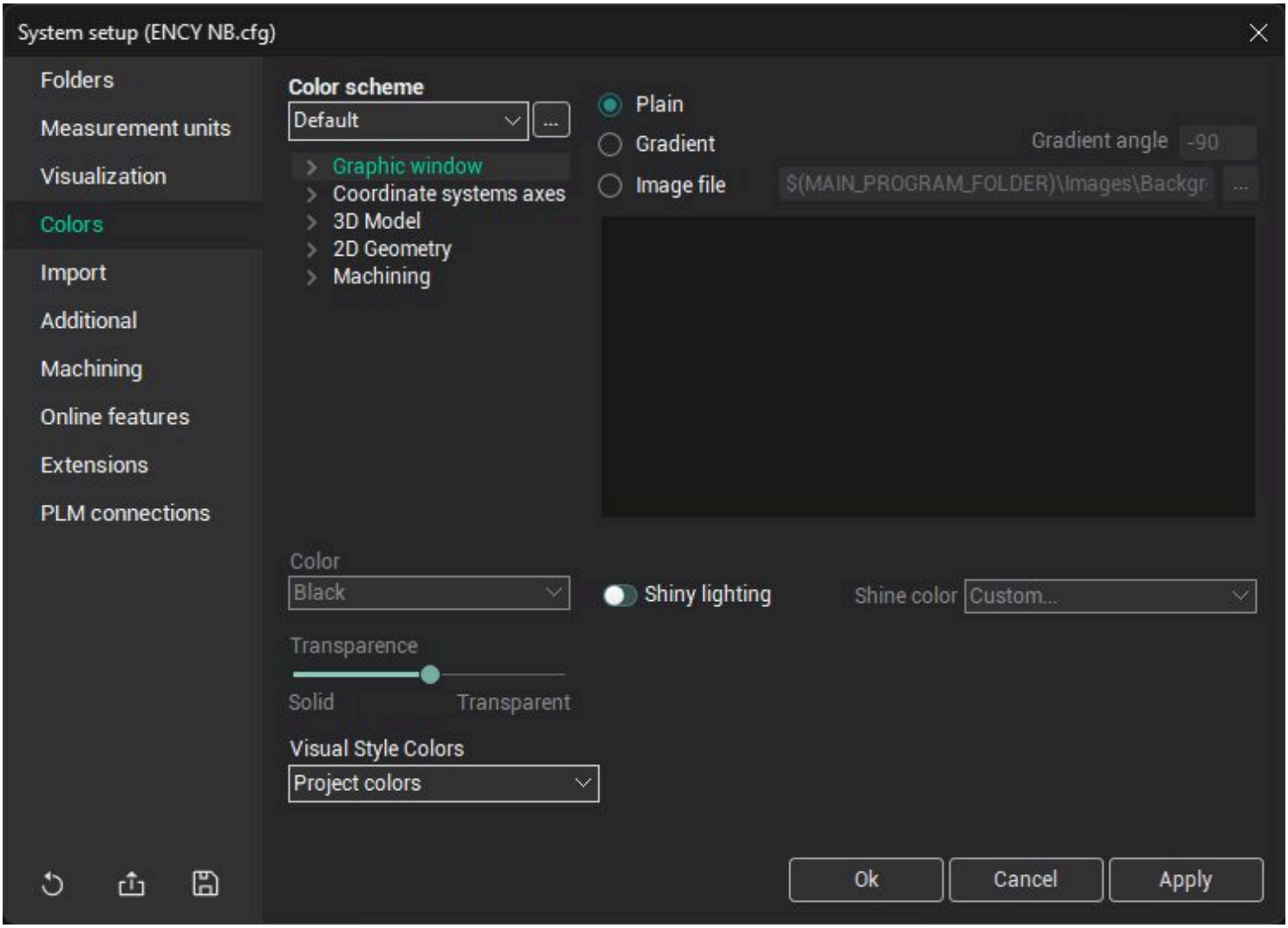
- Axes parameters: This panel allows you to adjust the visibility of coordinate systems and the length of their axes. The Active coordinate system is displayed in real space, while Additional axes are drawn on the left side of the graphical window.
- Visualization quality: You can set the default visual quality of objects on this panel. Moving the slider at the bottom will adjust the visual accuracy—the higher the quality, the more computer resources are required. This accuracy setting will apply to all newly imported objects. *Note: It is not recommended to set high accuracy on low-specification computers, as it may negatively affect performance.*
- Stitch faces together: When triangulating surfaces for visualization, this option ensures that surfaces are stitched together, avoiding gaps between them.

Graphic Window Settings:

- Main

    - Rotation centre: Defines the center point of rotation for the graphical window.
    - `Trap size (px)`: Defines the trap size in pixels for selecting and highlighting objects.
    - Mouse sensitivity: Controls the speed of rotation and scaling when using the mouse.
    - Animation: Specifies the number of intermediate frames when the view parameters of the graphical window change instantly.
    - Perspective: Determines the degree of perspective distortion applied to objects during visualization.
- Visual parameters:
    - Rendering mode: Offers three options:
    - 1. Hardware Advanced: Shader-based rendering.
      2. Hardware Standard: Classic hardware-accelerated OpenGL rendering without shaders.
      3. `Software (very slow)`: Software-based rendering with OpenGL, not utilizing the video card.
    - Ambient occlusion: Enables object shading for added depth and realism.
- Key Mapping: Defines hotkeys for visualization control. The CAM system offers four different key mapping schemes, designed for users familiar with other systems. On this panel, you can select the key mapping scheme, but hotkeys themselves are read-only.

Colors tab



In the Color scheme field, it is possible to select one of the predefined color schemes. By pressing the ... button, it is possible to restore the default color values or, if the Another color scheme is selected, to load values from another color scheme.

Below the color scheme, a tree-like structure shows the color values for individual elements. These elements are grouped as follows: Graphic window, Coordinate systems axes, **`3D Model`**, **`2D Geometry`**, Machining. By expanding the necessary group, you can edit the color values for any element. The color of a selected element is set in the Color field by choosing the desired value from the drop-down list. The Transparence slider allows you to adjust the transparency of the selected element.

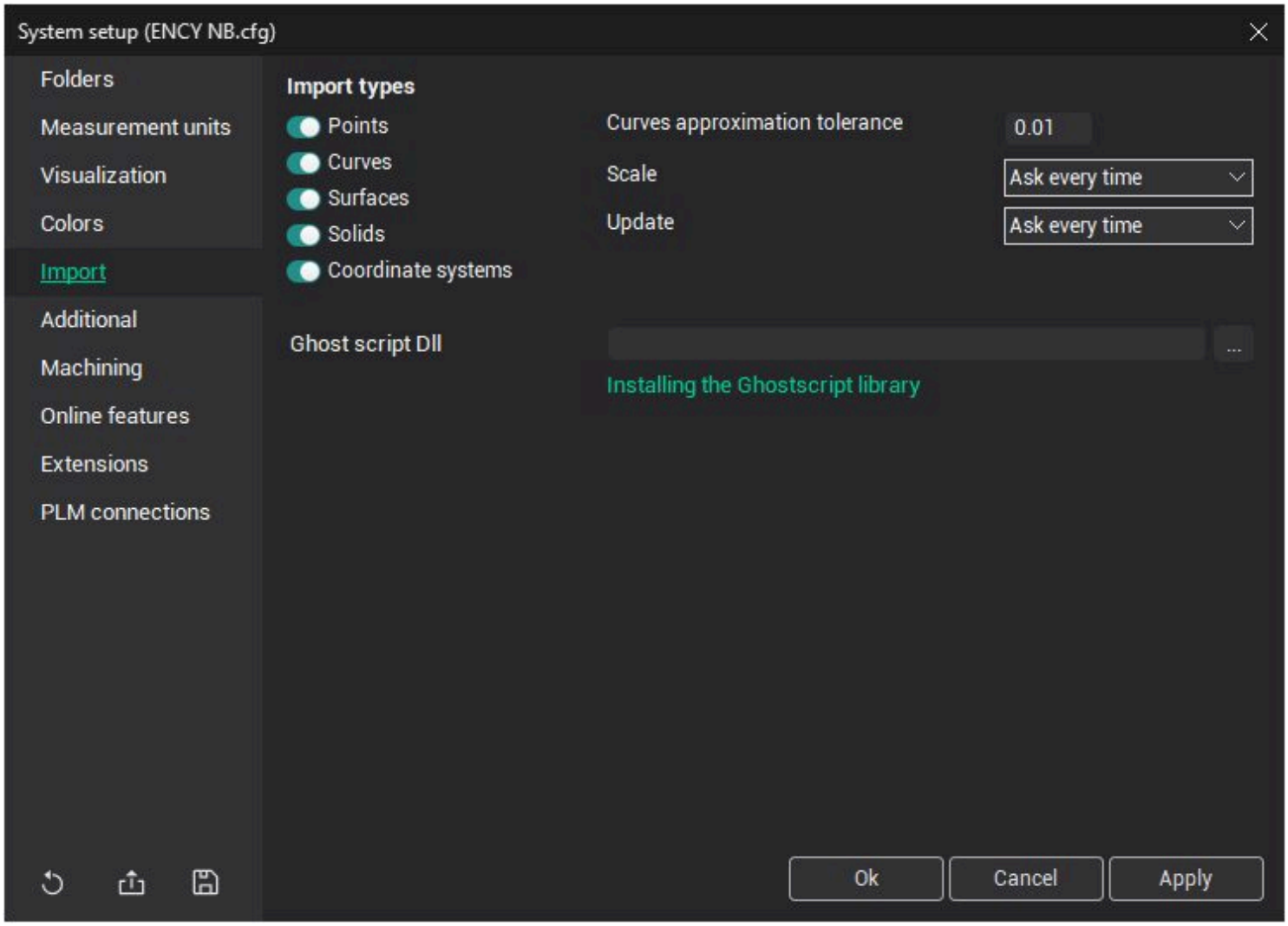
You can also choose the background of the graphic window:

- Plain: A solid background. The color is set in Graphic window -> First background color.
- Gradient: A gradient background. The colors are set in Graphic window -> First background color and Second background color. The Gradient angle field allows you to set the angle of the gradient relative to the vertical axis.
- Image file: Use a graphic image as the background. Supported formats are BMP and JPEG.

- Shiny lighting: Enables an additional light source for enhanced visuals.

Import tab

In this section, you can set the general import parameters.

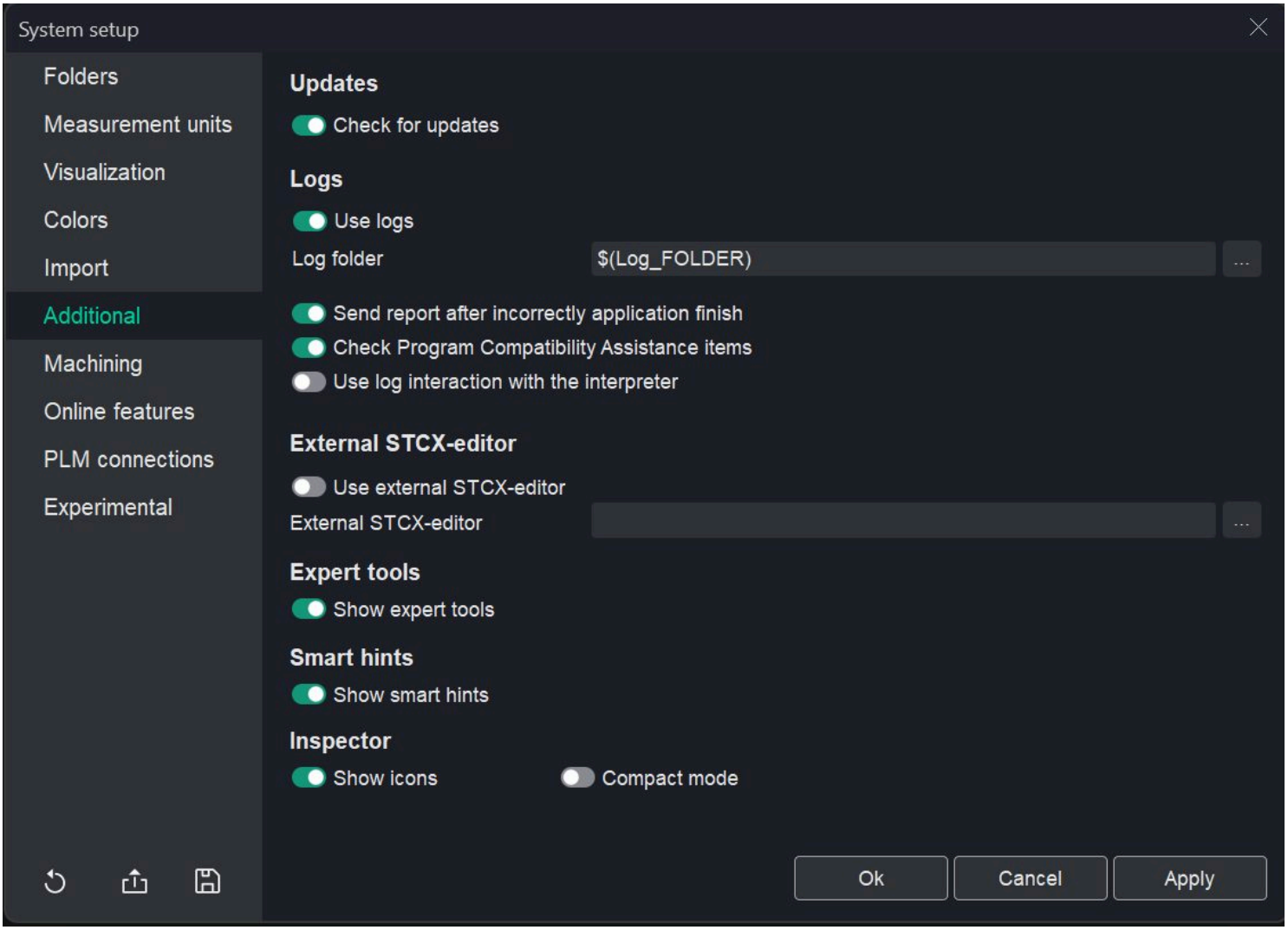


- In the Import types objects group, the user can define the types of geometrical objects that can be imported from CAD files. If the box next to a specific type is not selected, the corresponding object will be ignored during the import process.
- If a curve needs to be transformed during import, approximation will be performed using the value specified in the Curves approximation tolerance field.
- The Scale option is used to control the appearance of imported objects.
- Ghost script dll is used for importing PostScript files.

Additional tab

To configure system event logging and update settings, use the Additional page.

Logs are useful when you encounter issues which are difficult to explain in words or which occur only when a specific consequence of actions is executed.



Update Settings:

- To enable automatic update checks, switch on the Check for updates option.

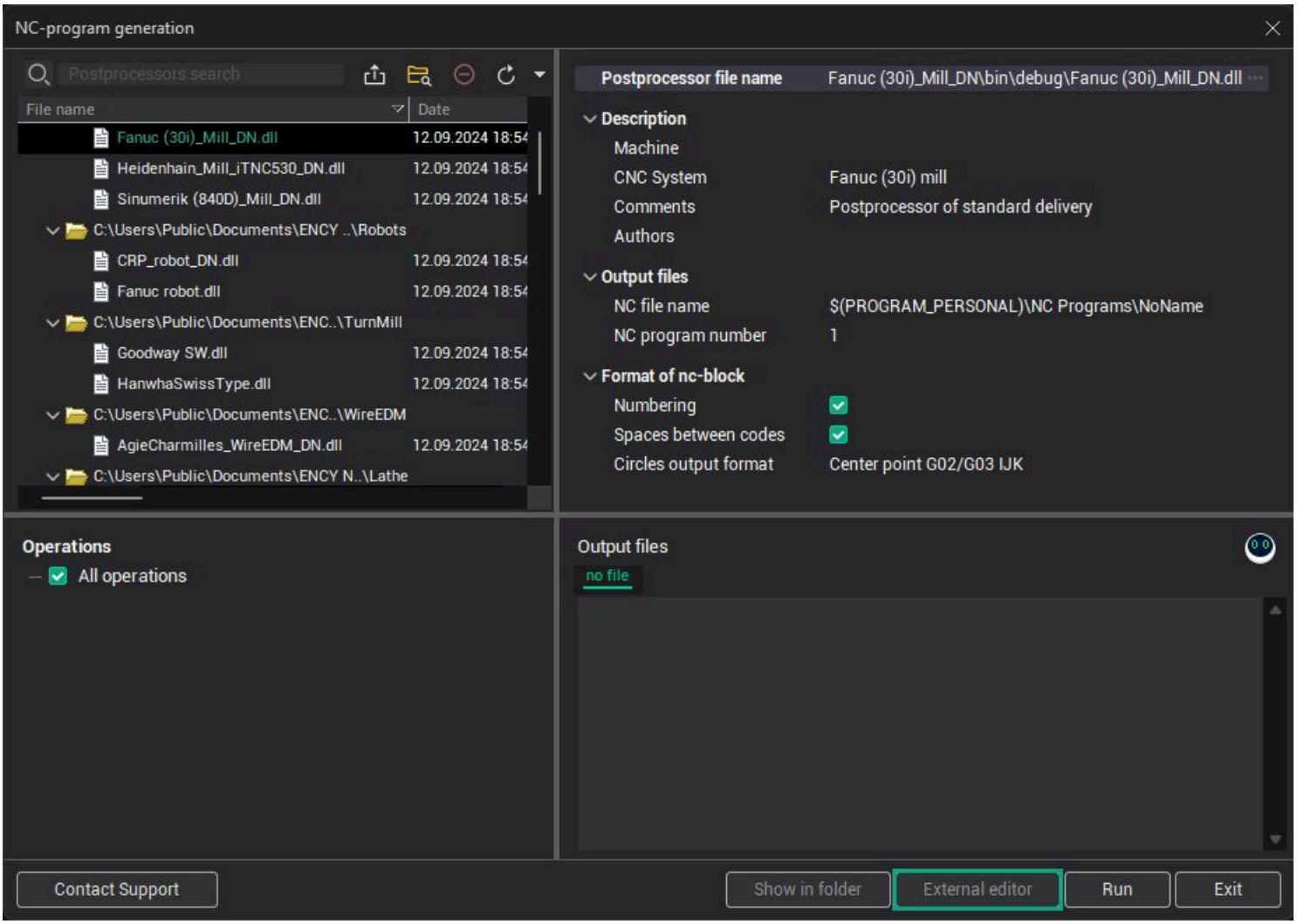
Logging Settings:

- To enable logging, check the Use logs option. Logs are useful for diagnosing issues that are difficult to describe or that occur only during a specific sequence of actions.
- The Log folder field allows you to specify the location where the system will save log files.
- To automatically send error reports, check the Send report after incorrect application finish option.
- The Check Program Compatibility Assistance items option excludes the system from compatibility checks with other programs by the Program Compatibility Assistant.

- To log the system's interaction with the interpreter (used in Simulation of machining by NC program and G-code based operations), enable the Use log interaction with the interpreter option.

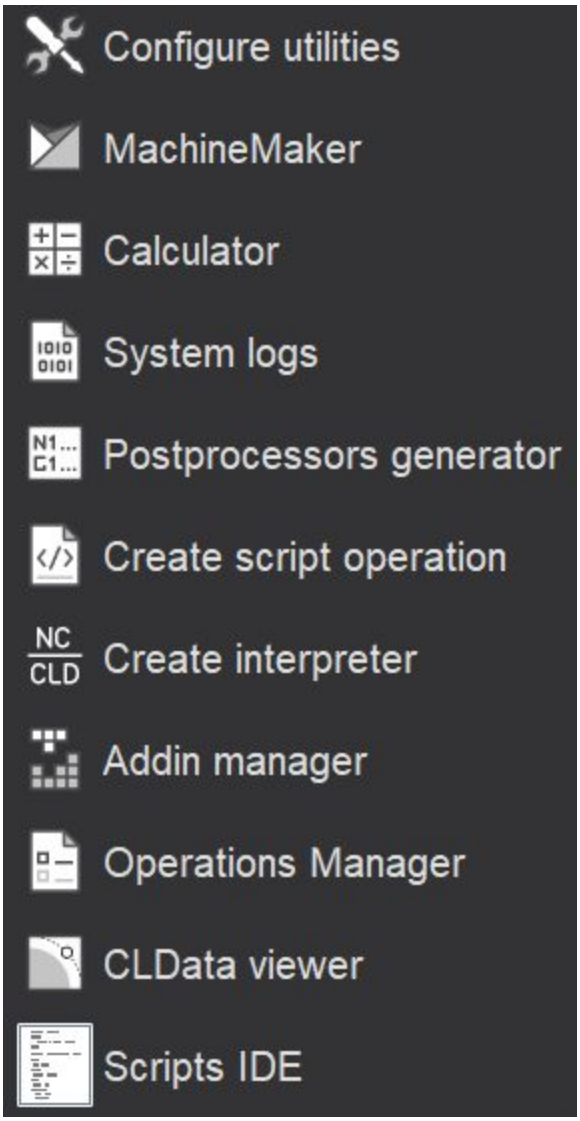
External Editor Settings:

The **`External STCX-editor`** panel affects the NC-program generation window and allows you to select an external editor to modify NC programs.



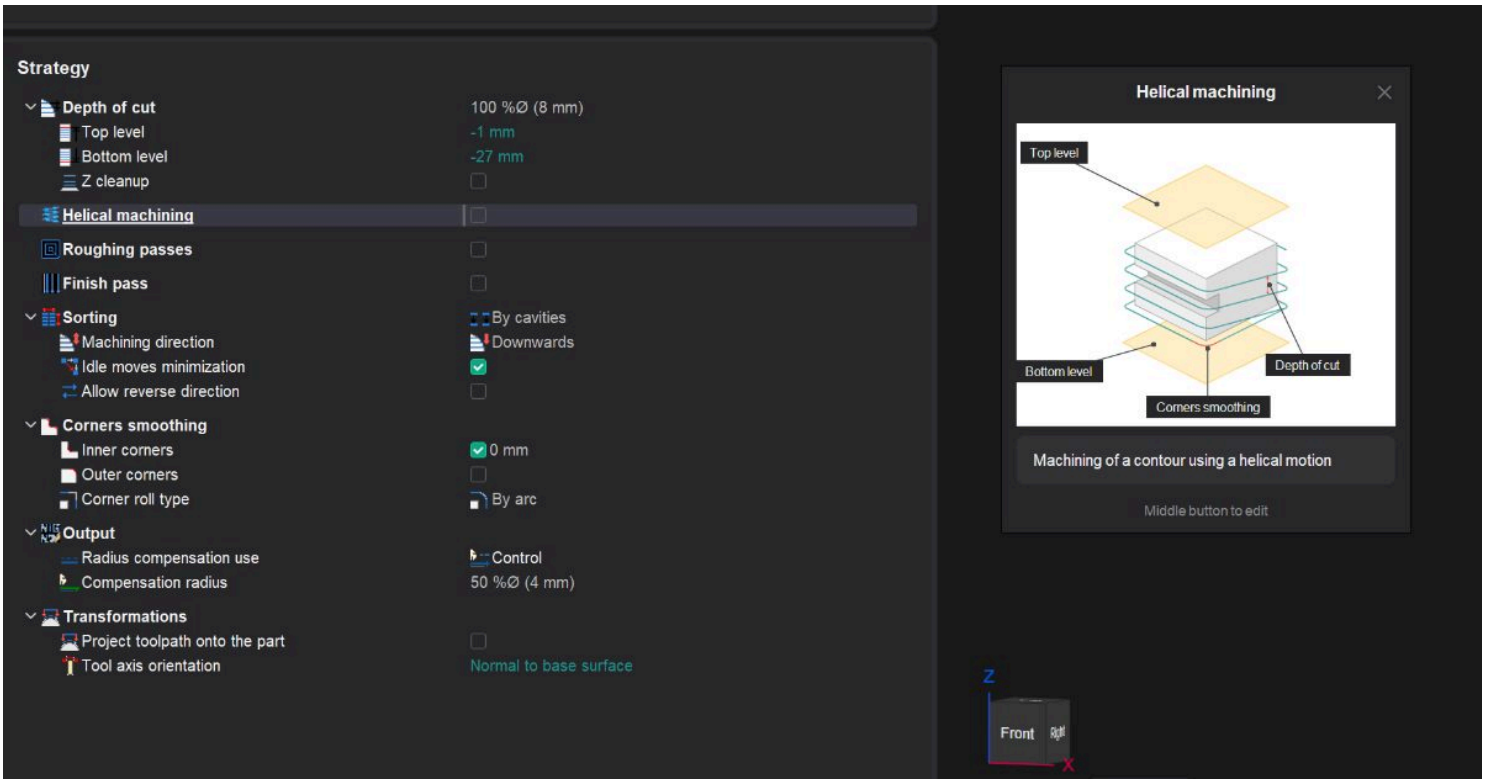
Expert Tools:

The Show expert tools option makes advanced tools visible for expert users.

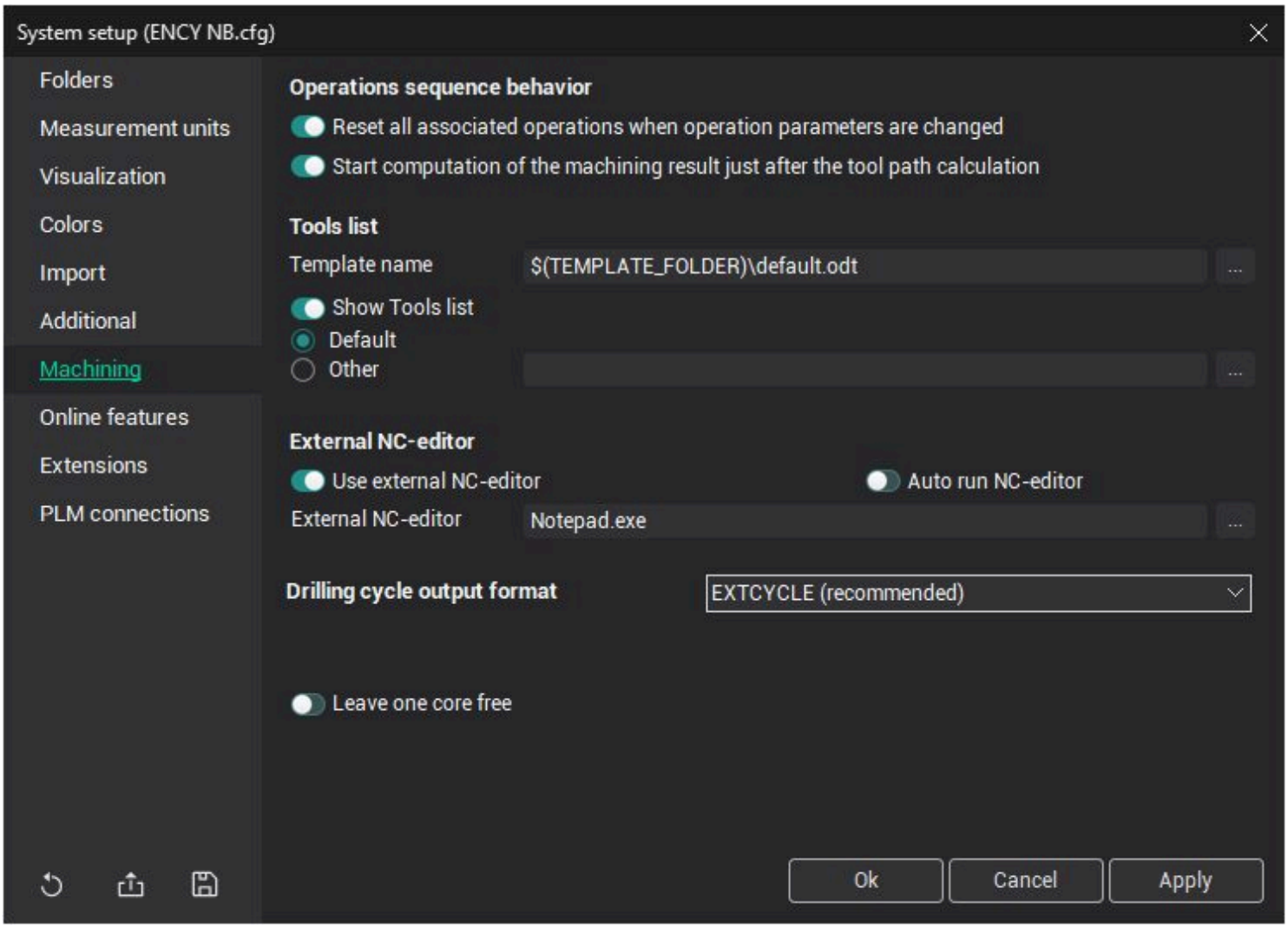


Smart Hints:

The Show smart hints option toggles the visibility of smart hints in the interface.


Strategy
Depth of cut
100 %Ø (8 mm)
Top level
-1 mm
Bottom level
-27 mm
Z cleanup
Helical machining
Roughing passes
Finish pass
Sorting
By cavities
Machining direction
Downwards
Idle moves minimization
Allow reverse direction
Corners smoothing
Inner corners
0 mm
Outer corners
Corner roll type
By arc
Output
Radius compensation use
Control
Compensation radius
50 %Ø (4 mm)
Transformations
Project toolpath onto the part
Tool axis orientation
Normal to base surface
Helical machining
Top level
Bottom level
Depth of cut
Corners smoothing
Machining of a contour using a helical motion
Middle button to edit
Z
Front
X

Machining tab



Operations Sequence Behavior:

The Operations sequence behavior panel allows you to control how operations behave when changing their status to calculated/reset:

- Reset all associated operations when operation parameters are changed: Automatically resets the toolpath of operations when input data from previous operations in the process changes.
- Start computation of the machining result just after the toolpath calculation: Automatically starts a background simulation immediately after the toolpath is calculated.

Tools list:

Tools list is used for setting up the parameters of the tools list generation.

The tools list can be created in ODT and PDF format.

When generating a tools list, the system uses templates. Several templates are included in the installation package, from which you can choose the desired one. It is also possible to create new templates in any text editor with ODT format support. For more information, contact the support desk.

Once created, document can be saved immediately, or viewed and edited if necessary. To save a tools list without viewing, deselect Show tools list.

To view the created tools list file, you can use either the default program for that file type or a user-defined program. To assign a different program as the editing tool, specify the full path to the application file.

External NC-Editor Panel:

The **`External NC-editor`** panel is used to specify an external application for viewing NC programs. The system recognizes specific command-line parameters:

- **`$NCOUTPUTNAME$`**: The name of the NC program file generated by the postprocessor.
- **`$CD.**ItemName**$`**: Custom data keywords that will be replaced by the appropriate item value.

If the **`Autorun NC-editor`** checkbox is selected, the specified NC-editor will be automatically launched after the NC program is generated by the postprocessor.

Postprocessor Compatibility Settings:

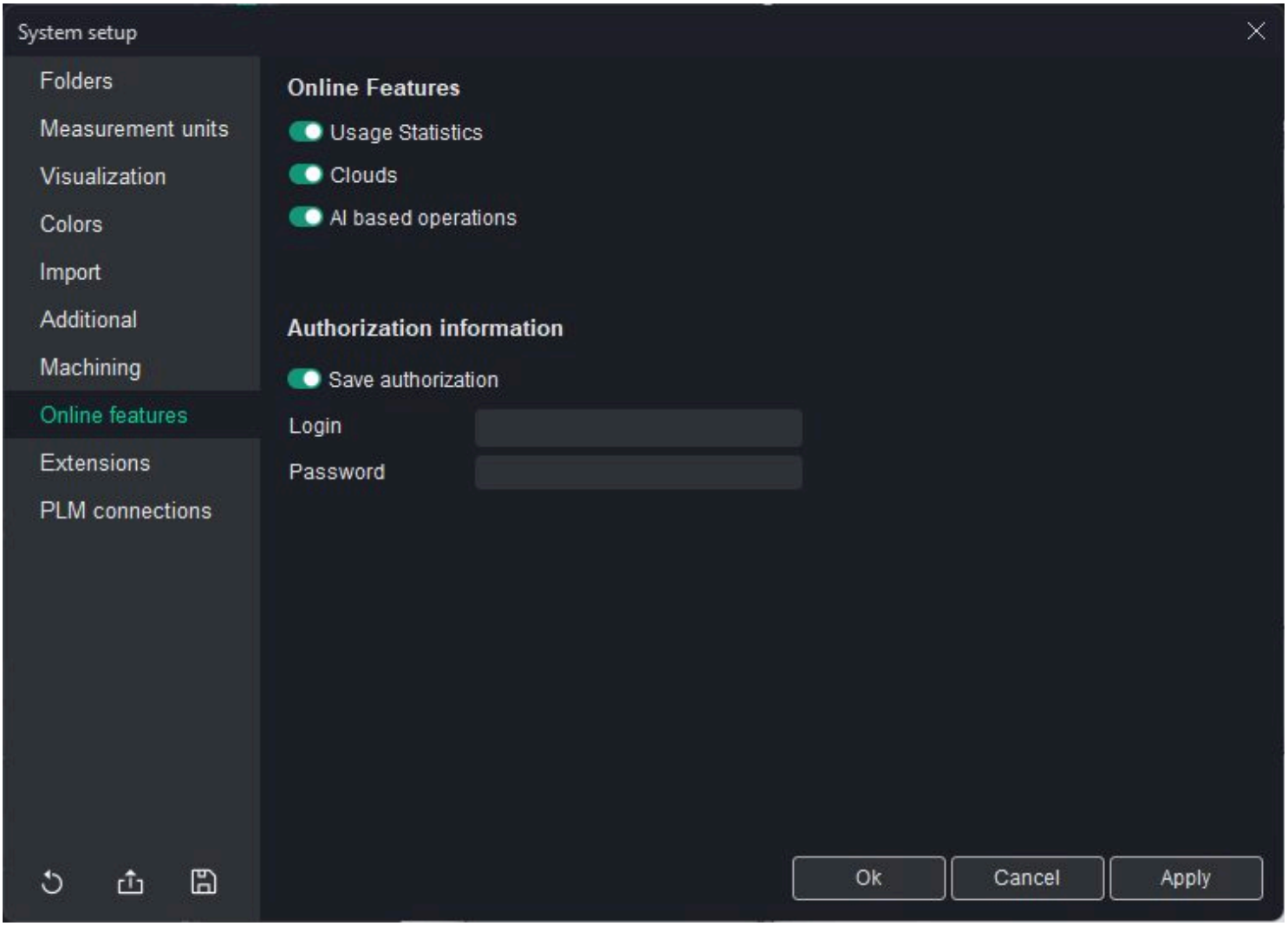
For compatibility with older versions of postprocessors, the system provides the ability to change the output format of the drilling cycle (when the non-expanded toolpath output method is used for hole machining operation). The Drilling cycle format parameter can have the following values:

- **`EXTCYCLE (recommended)`**: This is the new format that includes an advanced set of parameters, supporting all machining strategies implemented in the system and allowing realistic simulation of tool movements according to the chosen strategy.
- **`CYCLE (for old postprocessors)`**: This old format is required for compatibility with older postprocessors. It does not support some strategies (e.g., hole pocketing or spiral machining) and simulates any strategy as a simple movement to the hole's lower level.

Performance Options:

Leave one core free: Allows one core to remain free during multi-core computing (requires restart).

Online features tab



Online Features:

- Usage Statistics option allows you to enable or disable the collection of anonymous usage statistics. When enabled, the system gathers data on how features are used to help improve the software. No personal or sensitive information is collected, and the data is used solely for enhancing the user experience and optimizing system performance.
- Clouds. When this option is enabled, it grants access to the Clouds feature. Details.
- AI-based operations. When this option is enabled, user can create operations based on artificial intelligence. When it is disabled, this option does not appear in the list of operations available for creation..

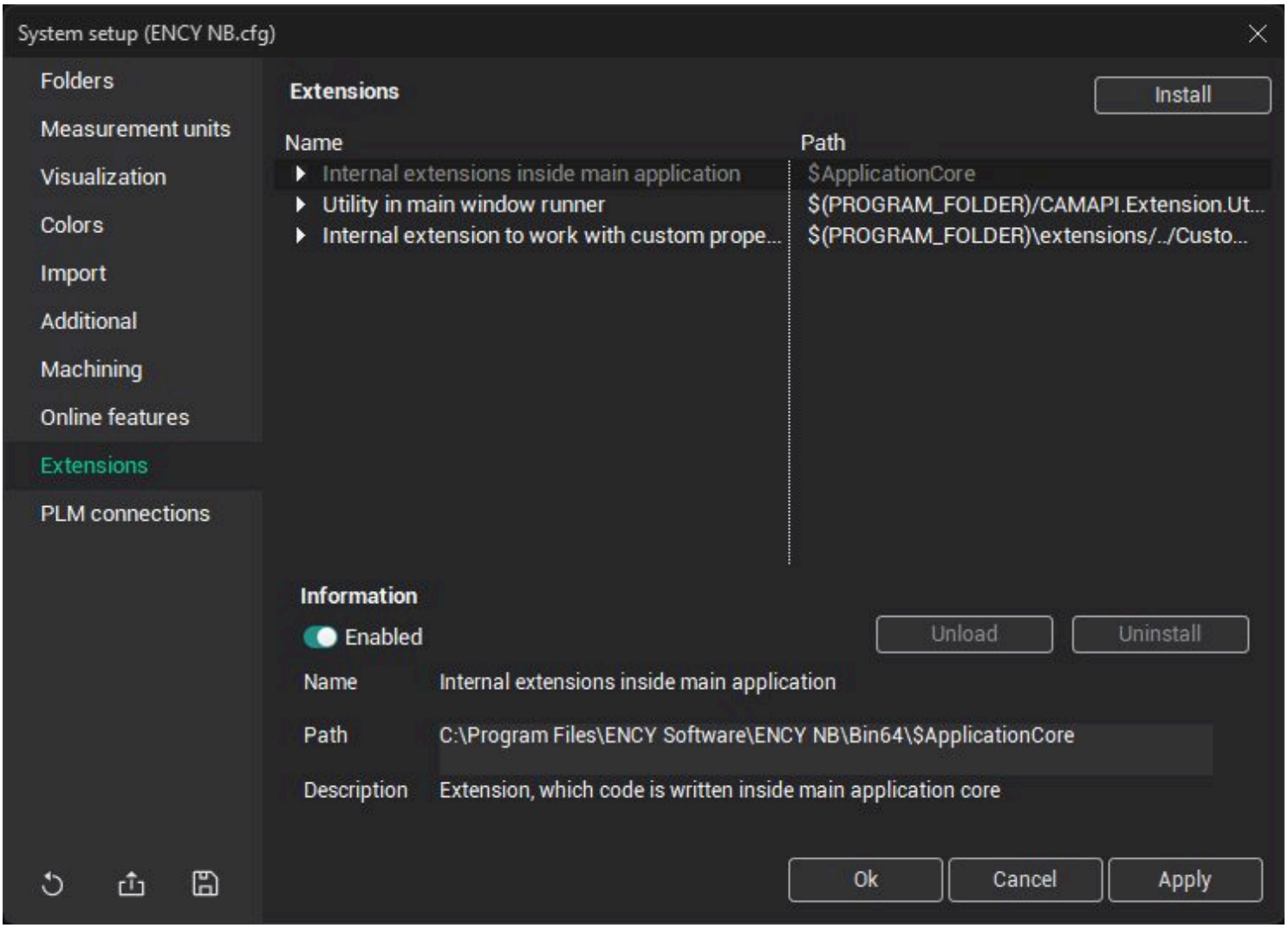
Authorization information:

- Save authorization. This option saves the user's login credentials (login and password) to the License Manager. When enabled, the system will remember the user's details, providing quicker access to online services.

- Login: This field is used to input the user's login required to access online features such as cloud collaboration or external integrations.
- Password: This field is for entering the password associated with the user's login credentials, allowing secure access to the system's online services.

Extensions tab

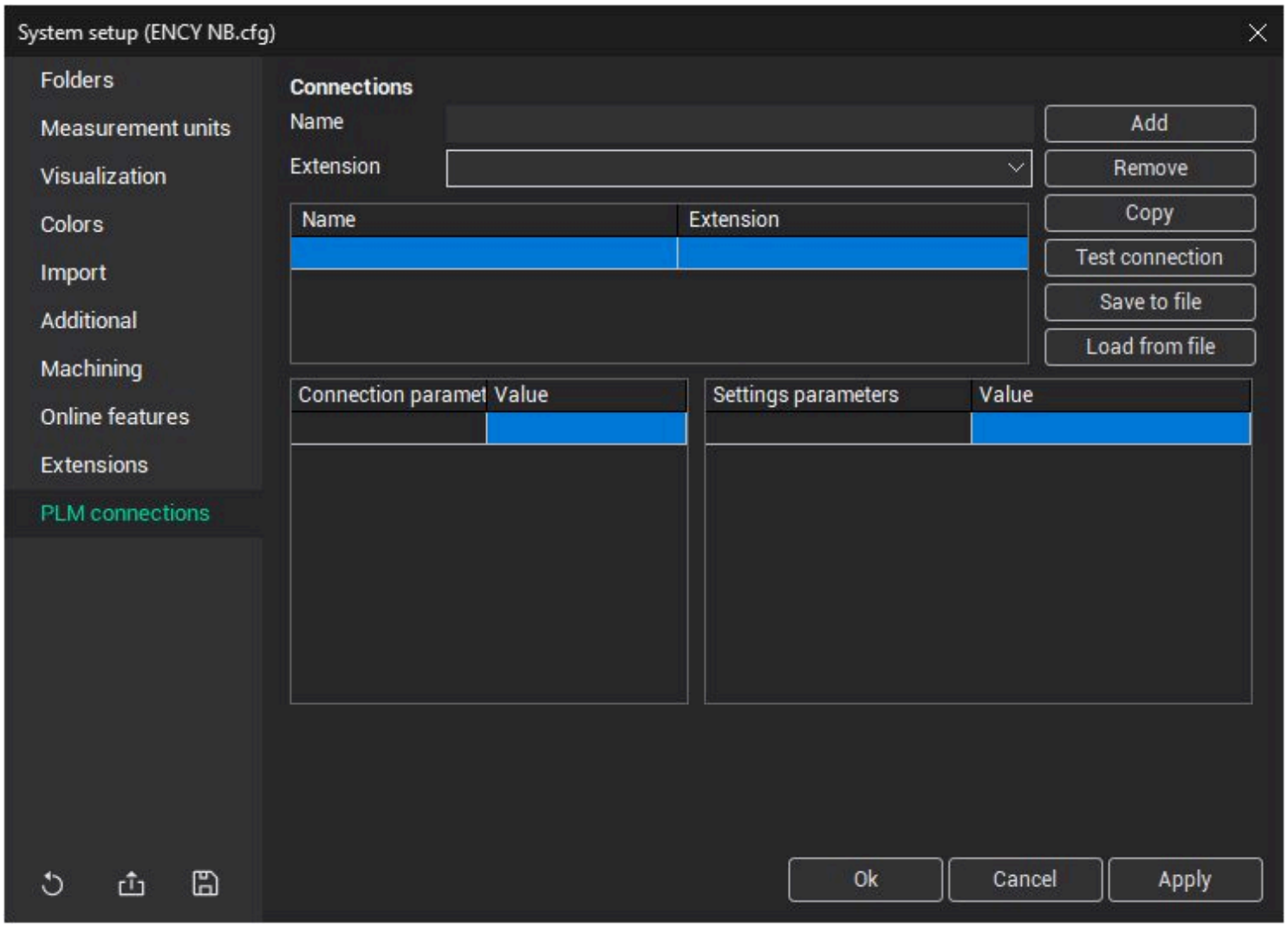
This tab is used to install extensions for the CAM system. See more at [Extensions of the CAM system].



PLM connections tab

This tab is used to configure PLM connections for different PLM systems. Configuration features are described in the PLM connection setup.

For detailed information, please refer to the documentation of the integration module with the corresponding PLM system.



Exchange files

This section describes project files, data import and export, postprocessor tuning files, NC programs, interpreters, machine schemas, encrypted containers, and machine setup files.

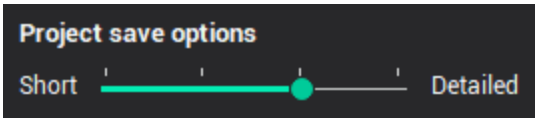
Projects files

A project is a file with the extension ".stcp" saved on disk (previously used the extension ".stc"). It stores inside itself the information that the user generated while working in the system, when creating a program for a specific part machining, and which is minimally necessary for its subsequent restoration in the application later. This is for example information such as

- Geometrical models of the part and workpiece.
- Machining technology - the list of operation in a defined order. Each operation contains all the necessary information in order to machine a separate region of the part - links to the geometrical curves or faces, machining strategy parameters (sequence flags, steps, tolerances and so on).
- Information on the necessary equipment: machine settings, tool list.
- The resulting toolpath in the form of machine independent CLDATA.

The data storage format inside the project's *.stcp file is a kind of multi-file archive. Thanks to this, it is possible to save some files inside the project (in addition to the basic information listed above) that are needed to fully work with the project, but which are stored separately on the computer. As a result, the project can be freely transferred and opened on any other computer. These can be files such as machine schema files, files of 3D models of tools. During normal user work on his computer, the project refers to the corresponding external files, and a copy of them is written inside the *.stcp file. However, if the project is then transferred to another computer and the desired external file does not appear, then the system switches to working with the version of the file that is stored inside the project. Due to the fact that storing additional files within a project can increase its size, this feature can be disabled in the system settings.

The saving mode can be changed in the System setup window.



- Short mode. Minimal source data only saved - geometrical model, coordinate systems, technological operations and their parameters. Any information which can be recalculated is not saved.
- First intermediate mode. Calculated toolpath is saved in addition to the previous data.
- Second intermediate mode. The external files that the project uses are copied inside it: 3D models of used equipment and so on.
- Detailed mode. Intermediate states of the workpiece and geometry cache data are saved additionally to the all modes above.

Use Save and Open project buttons on the main toolbar to create/restore the project file. Projects can also be opened by using the drag and drop function, i.e. simply by dropping the project file (*.stcp) onto the main window of the application. A project file can be opened also if the file name is added as a parameter into the shell command line.

Importable files

In CAM system there is the ability to import a geometrical model from any draftsman's or designer's systems (CAD/modeling) via data exchange files formatted as:

- IGES (*.igs, *.iges);
- STEP (*.step, *.stp);
- DXF (*.dxf);
- PostScript (*.ps, *.eps);
- STL (*.stl);
- PLY (*.ply);
- AMF (*.amf);
- VRML (*.vrl);
- SW (*.sldasm, *.sldprt);
- SE (*.asm, *.par, *.psm, *.pwd);

or directly from

- Rhinoceros (*.3dm);
- Parasolid (x_t; x_b).

The number of importable files depends on the system configuration and can be changed optionally.

It is possible to extend the import format list by add-ins supplement. There is Addin Manager to do that. The add-ins are aimed to tune up a collaboration of Cam system with different CAD-systems. Add-ins permit generally CAM system to open directly project files of the CAD-systems.

The geometrical model of a machined part, workpiece, machining equipment can be prepared in any CAD/modeling system and imported into CAM system using any of the supported formats. CAM system can be integrated with any CAD-system. The internal model supports different representations of solid, surface, mesh and curve geometrical objects. Therefore, the representation of the geometrical information in CAM system does not differ from the internal representation of geometry in many CAD-systems, which is very useful for avoiding "damaged" models during transmission from one system to another.

DXF export

The geometrical objects of CAM system can be exported into the DXF format file. Curves and points export only is allowed in the current version. The splines are used to save the text that was created in CAM system. Contours of 2D geometry are saved in the XY-plane irrespective of the real orientation of the local coordinate systems.

Postprocessor tuning files

Postprocessor tuning files to specific CNC system have the SPPX (*.sppx) extension. There is a unique tuning file for each CNC system. Tuning file contains all data regarding the CNC unit and subprograms of machining programs processing. The file is required by the postprocessor for transformation of machining commands into an NC program for this control. The files are used by the postprocessor but they are created and altered by Postprocessors Generator.

Postprocessor files previously had the extension *.spp. These postprocessors are supported by the current version of the system and, if necessary, can be converted to *.sppx format.

Earlier versions of CAM system work with postprocessors which are designed in two linked files with identical name but different extension (*.inp, *.ppp). All existing ppp-files can be used to generate NC-programs by the current version. But if the postprocessor musts be edited then the associated inp-file is necessary. The pair files can be opened by the Postprocessors Generator and converted into the new sppx-format. The converted file can be modified by the same application.

Postprocessor files can be placed inside encrypted zip-containers with the.stfc extension.

NC program files

NC programs are created by the postprocessor by conversion of the machining commands sequence into a sequence of commands for the CNC unit following the rules described in the postprocessor tuning file (*.sppx).

NC code is output by the postprocessor into a standard text file. The name can be defined in the postprocessor before generation. The extension of the output file is defined in the postprocessor tuning file (*.sppx). NC programs for different CNC controls can have different extensions therefore, different tuning files are used.

Transfer of the NC program from the computer, where CAM system is installed, to the CNC unit can be performed by any available method.

Interpreter files

Interpreter files to specific CNC systems have the SNCI (*.snci) extension. There is a unique tuning file for each CNC system. The tuning file contains all data regarding the CNC unit and subprograms for machining program processing. The configuration file contains information about the CNC system and a link to the software library that implements the interpretation of the NC program into a set of machining commands for this system.

The interpreter file is used when the Simulation of machining by NC program mode is enabled and in G-code based operations.

Interpreter files can be placed inside encrypted zip-containers with the.stfc extension.

Machine schema files

The CAM system uses machine schema to describe the kinematic model of the equipment (CNC machine, robot, etc.), its name, CNC system, using postprocessor file and some other settings. Typically, the scheme consists of a main xml file and several additional files that contain a 3D model for each of the machine nodes. They can have the extension.osd and.stl and be located either next to the xml file or in a subfolder. Machine schema files can also be placed inside encrypted zip-containers with the.stfc extension or inside the project.stcp file.

Use Machines library window to select desired machine schema file.

Encrypted containers .stfc

An encrypted container is a file with the.stfc extension, which is a zip archive. Some files in this archive can be encrypted. To decrypt and use them, you usually need an individual license for each container.

The container may contain any files that may be needed to work with the system, for example:

- postprocessors,
- interpreters,
- machine schemas,
- additional operations' parameters descriptor files.

Contact your dealer to obtain a license for the desired container. Use the license manager to activate the container license.

Machine setup files

The machine setup file is a file with an expansion.stms. Inside himself, it contains information about machine setup for a certain processing and is designed to quickly create a project on the template. The format of storage of data inside the machine setup file is similar to the format of the project.

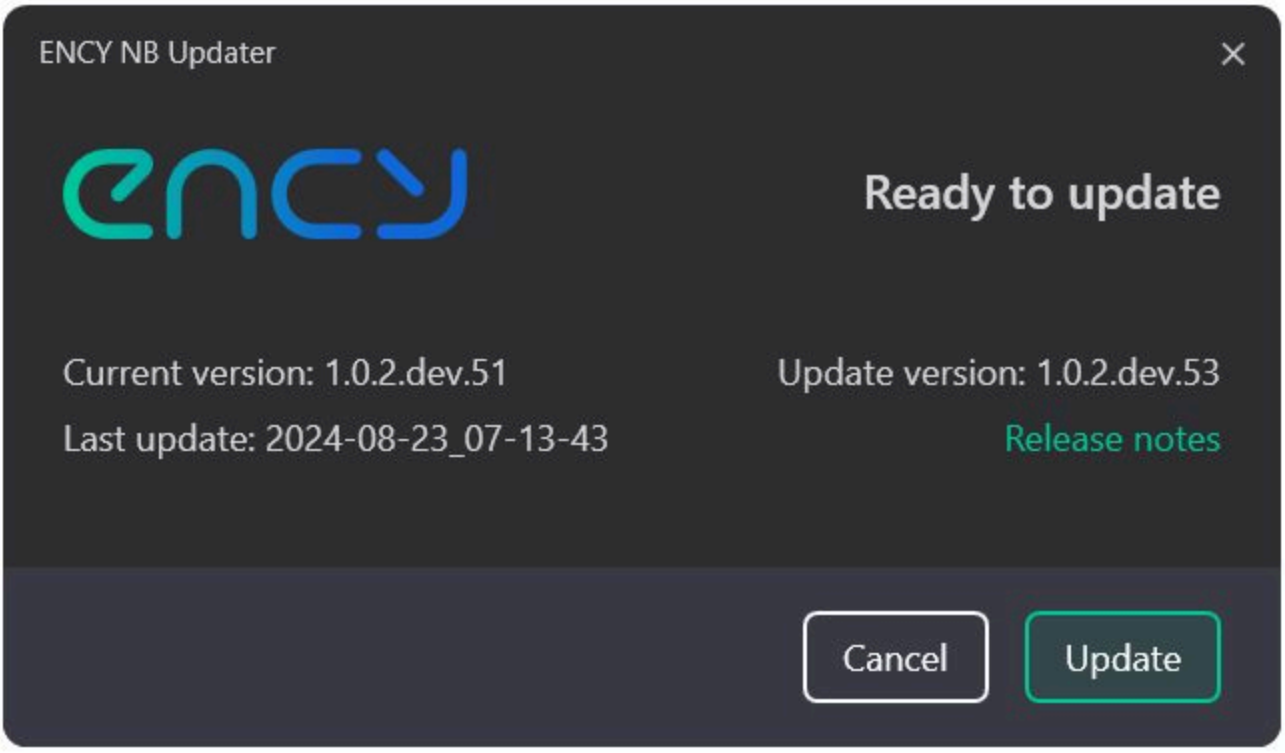
To control the machine setup files, use the "Save as a machine setup", "Import from the machine setup file" and "New project from the machine setup file" on the main panel.

Read more about machine setup files here: Machine setup (project template).

Updating

CAM system checks for updates at each start. The system will automatically download all necessary files and check them. It is not necessary to keep CAM system open – background updates checking will be processed even after CAM system closes.

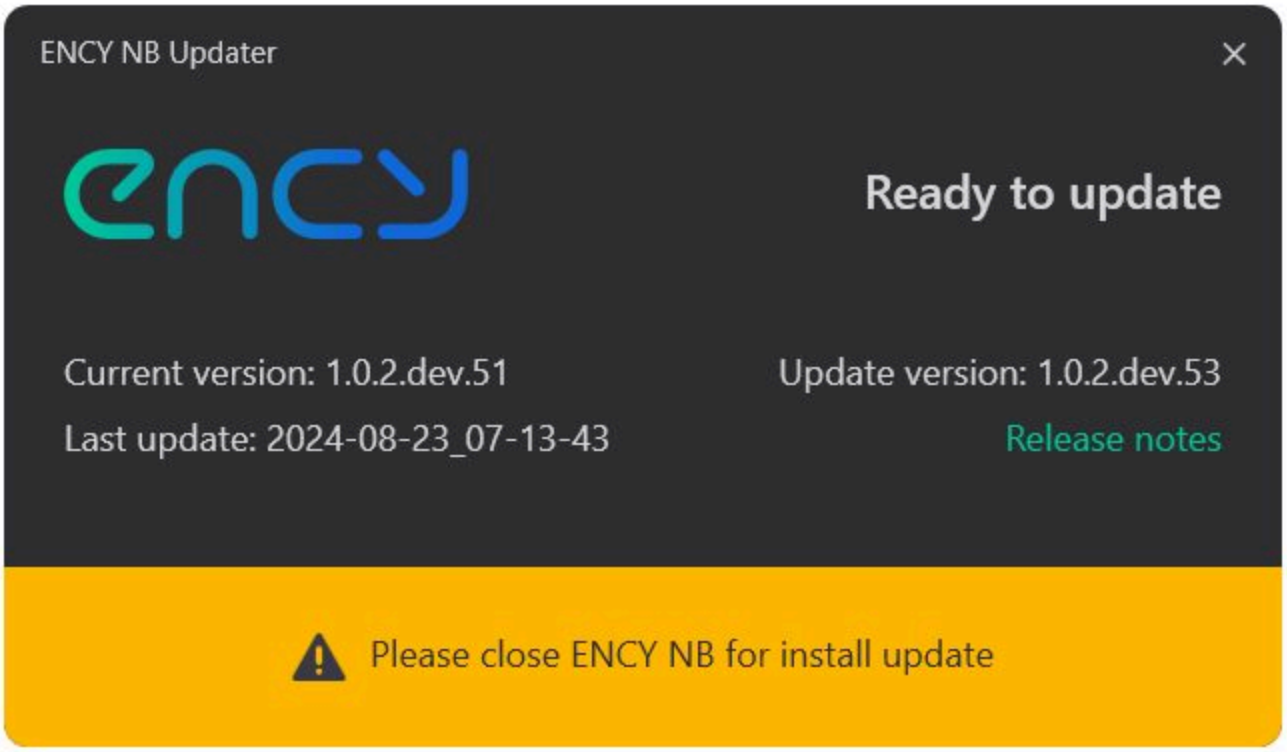
When updating systems find and successfully download modules, an updating prompt will be shown:



Press the Update button to run the updating process.

Updating process

The CAM system updating process includes three steps:



- The Close CAM system — system waits while the user saves all projects and closes CAM system manually. CAM system launching will be blocked during the update. It is possible to cancel the updating process in this step by closing this updating window.
- The Updating — system starts updating. CAM system can ask an administrative account to complete this step.
- The Finish — system deletes all temporary data, used for update.

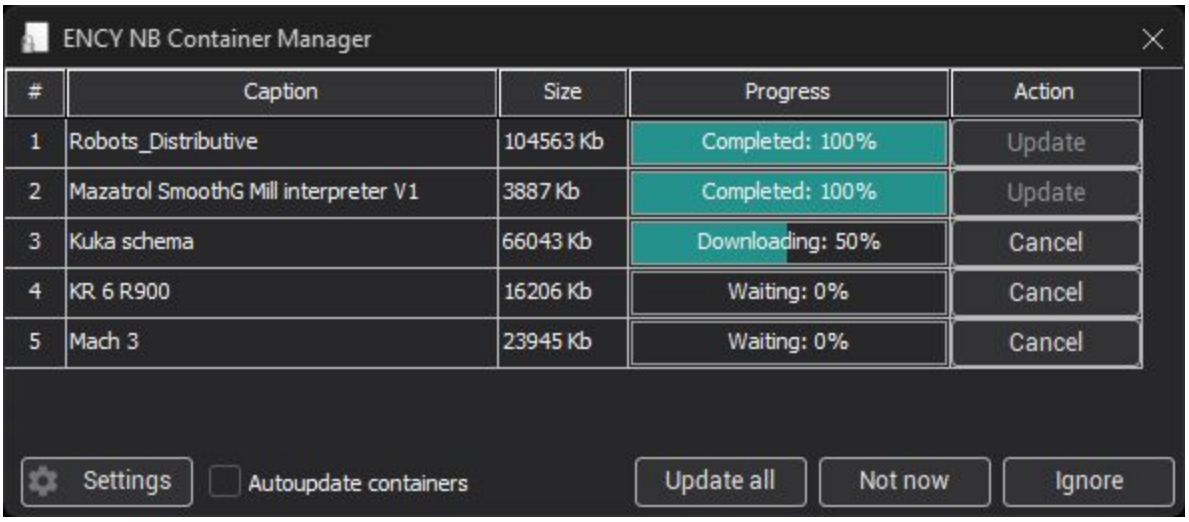
After these steps, the updating window will be closed and CAM system will be started automatically.

Container manager

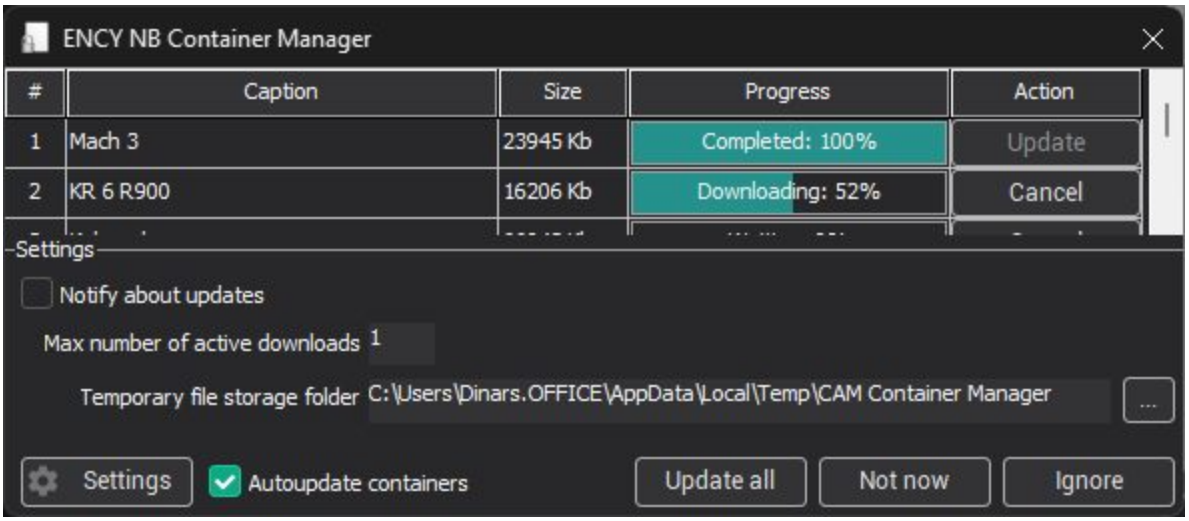
A container manager has been added to the CAM system. It automatically downloads and updates container files (.stfc), provided that licenses for the respective containers are available.

The system checks for uninstalled containers and container file updates each time the CAM system is launched and during license installation through the license manager.

When you click on a message or an icon in the tray, a window with a list of containers available for download or update appears:

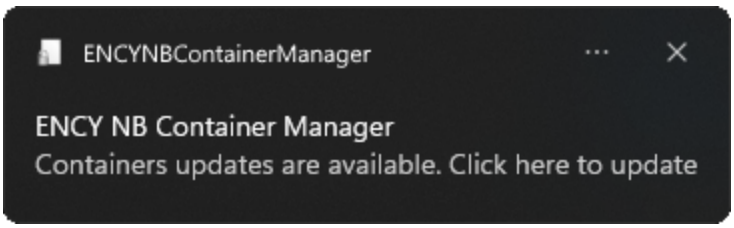


- The Update button initiates the download and update process for the selected container.
- The Update all button initiates the download and update process for all containers in the list.
- The Not now button cancels the download/update process and minimizes the container manager to the tray. The system will prompt you to update next time.
- The Ignore button cancels the download/update process and minimizes the container manager to the tray. The system will no longer show update notifications until the Notify about updates option is enabled in the container manager settings panel.
- The Autoupdate containers option enables automatic container downloads/updates without user notification.
- The Settings button displays or hides the container manager settings panel:



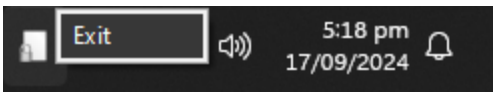
- The Notify about updates checkbox enables or disables notifications about container updates. Clicking the Ignore button removes this flag.
- The Max number of active downloads field sets the number of simultaneous container downloads. If you have a slow internet connection, it is recommended to set the value to "1".
- The Temporary file storage folder field sets the directory for temporary storage of downloaded container files.

When the download process begins, container files are saved to the temporary directory specified in the settings panel. After the download is complete, the container manager will wait for the CAM system to close before moving the downloaded files to the system's container storage directory.



At the end of the process, the CAM system container manager will close automatically.

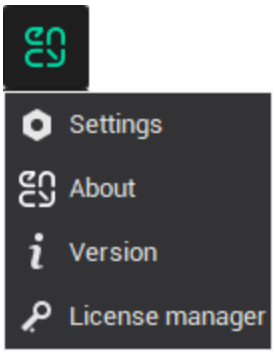
To force close the container manager, right-click on the tray icon and select Exit from the context menu:



If the container manager is forcefully closed, all active downloads will be terminated, and previously downloaded containers will not be moved to the CAM system container storage directory. The next time the system starts, it will automatically re-download any containers whose update was interrupted.

Licence manager

The license manager contains functions for working with licenses for CAM system and licenses for its modules.



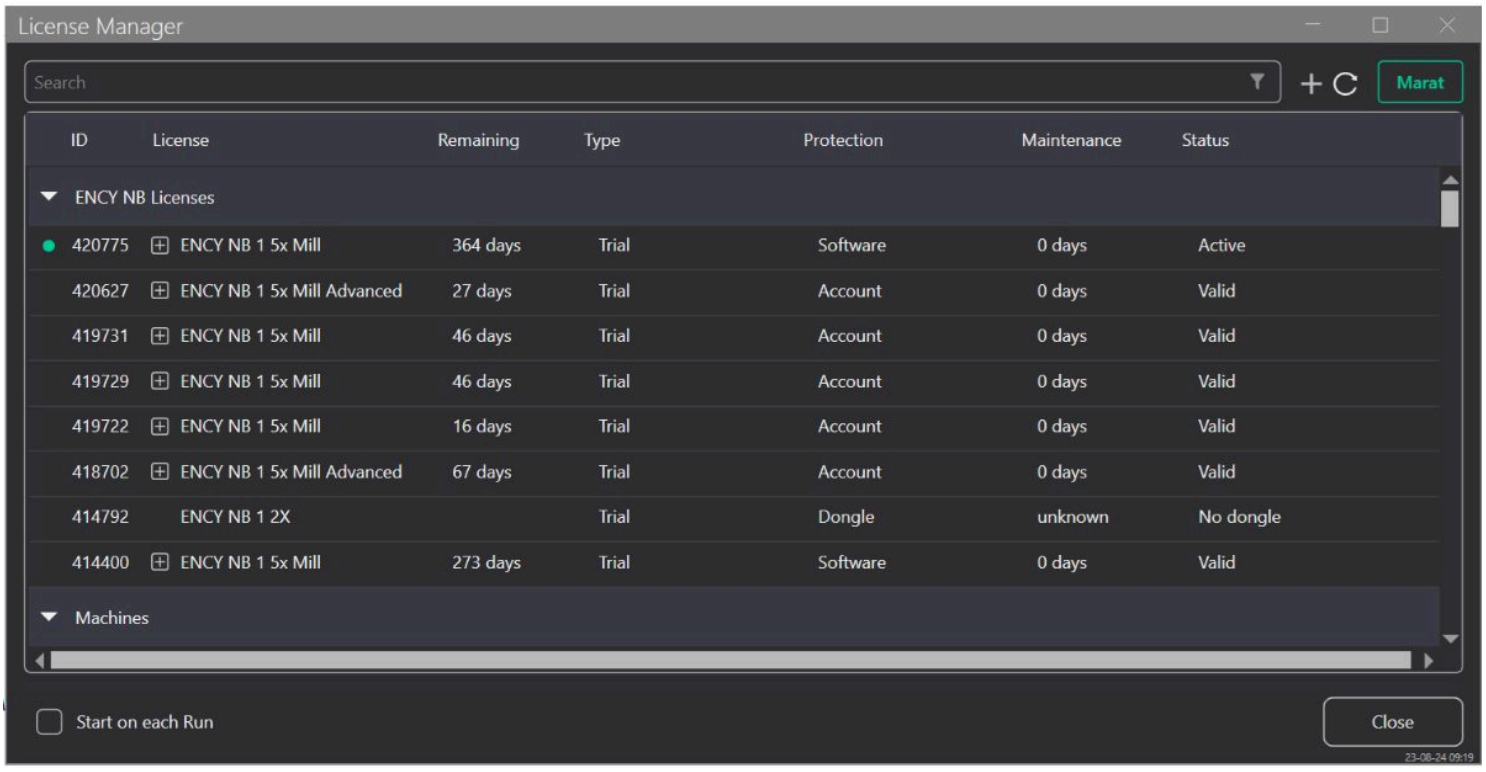
To start the manager, use the License Manager item in the drop-down list of the main menu.

Information: The license manager opens automatically if there are no licenses available or if the option Start on each run is enabled.

Each CAM system customer has a personal account in which all available licenses and functions are located.

Information: Access to your personal account is provided by login / password and requires access to the Internet. To obtain data for authorization should contact your CAM system dealer.

The main manager window is presented below:

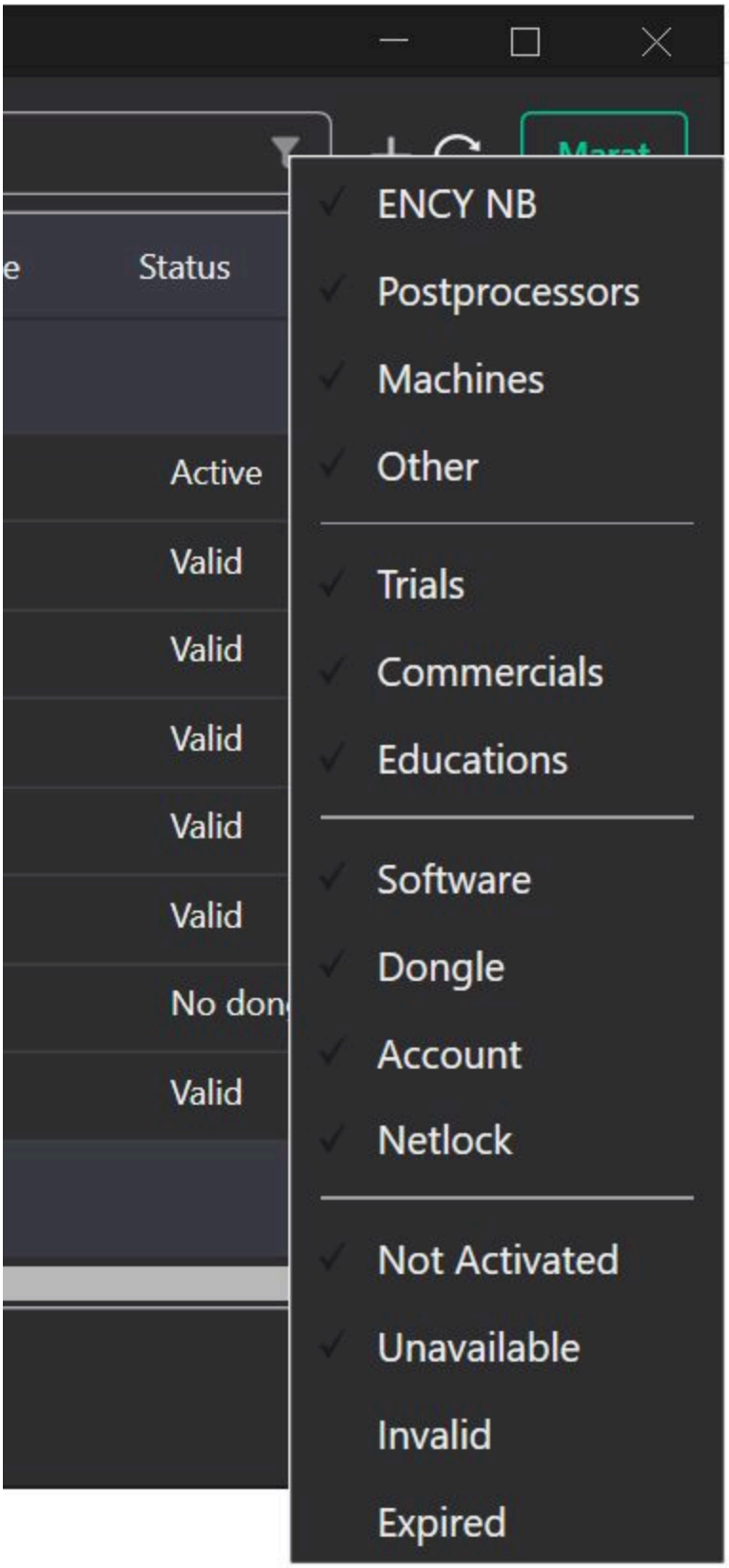


The upper part of the window is occupied by the list of CAM system licenses already available on the computer, their status and brief information on the composition and the remaining working time.

At the bottom of the window on the right is a help button and button to close the license manager.

At the top of the window places search box and licenses filter. All licenses are divided by:

1. CAM system licenses.
2. Postprocessors.
3. Machines
4. Other containers licenses
5. Licenses type (Trials, Commercials, Educations)
6. Protection type (Software, Dongle, Account, Netlock)
7. License status (Not activated, Unavailable, Invalid, Expired)



When you hover over the line with the license, the area with 3 buttons is shown on the right:

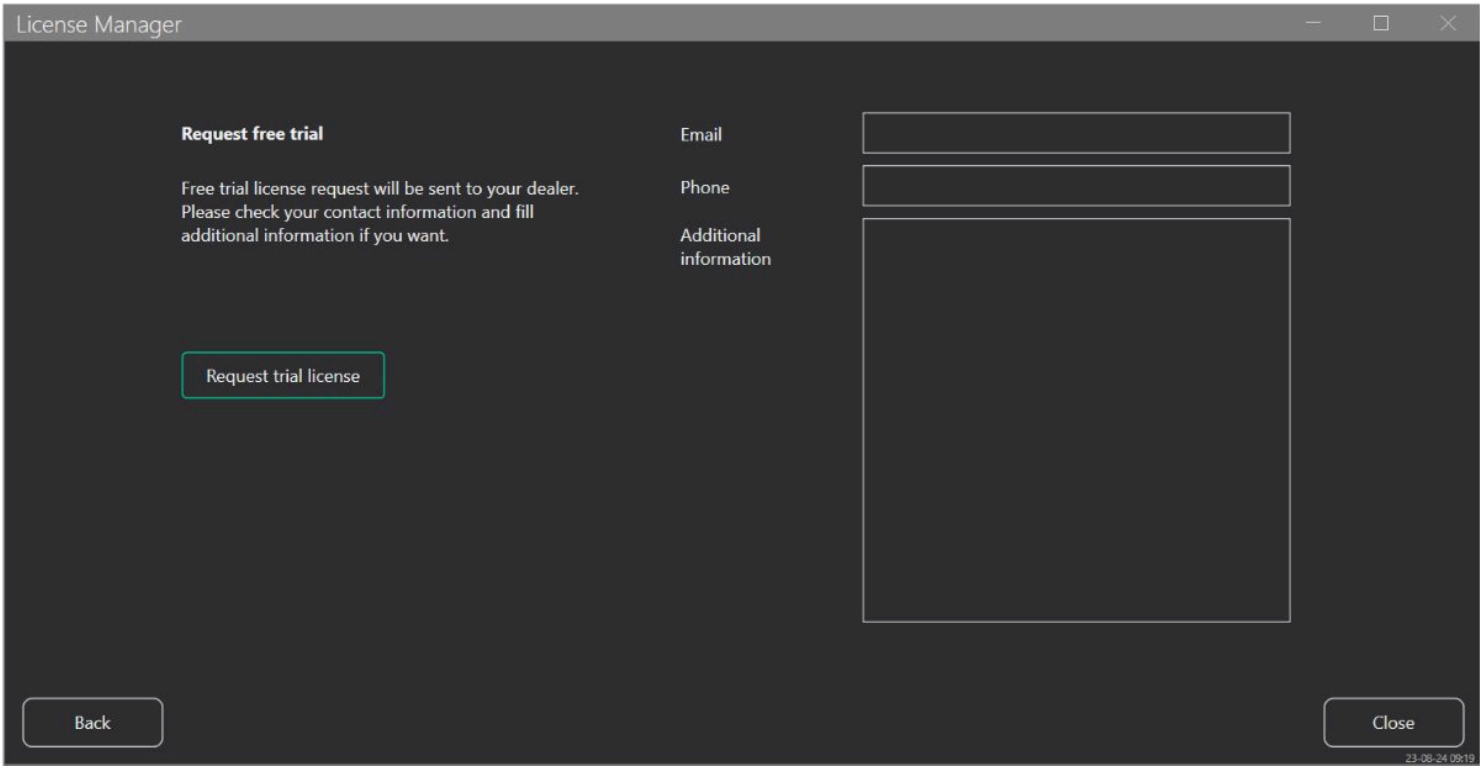


Activate - Performs license activation.

Refresh - Updates information on the current license.

Deactivate - Deactivate the current license.

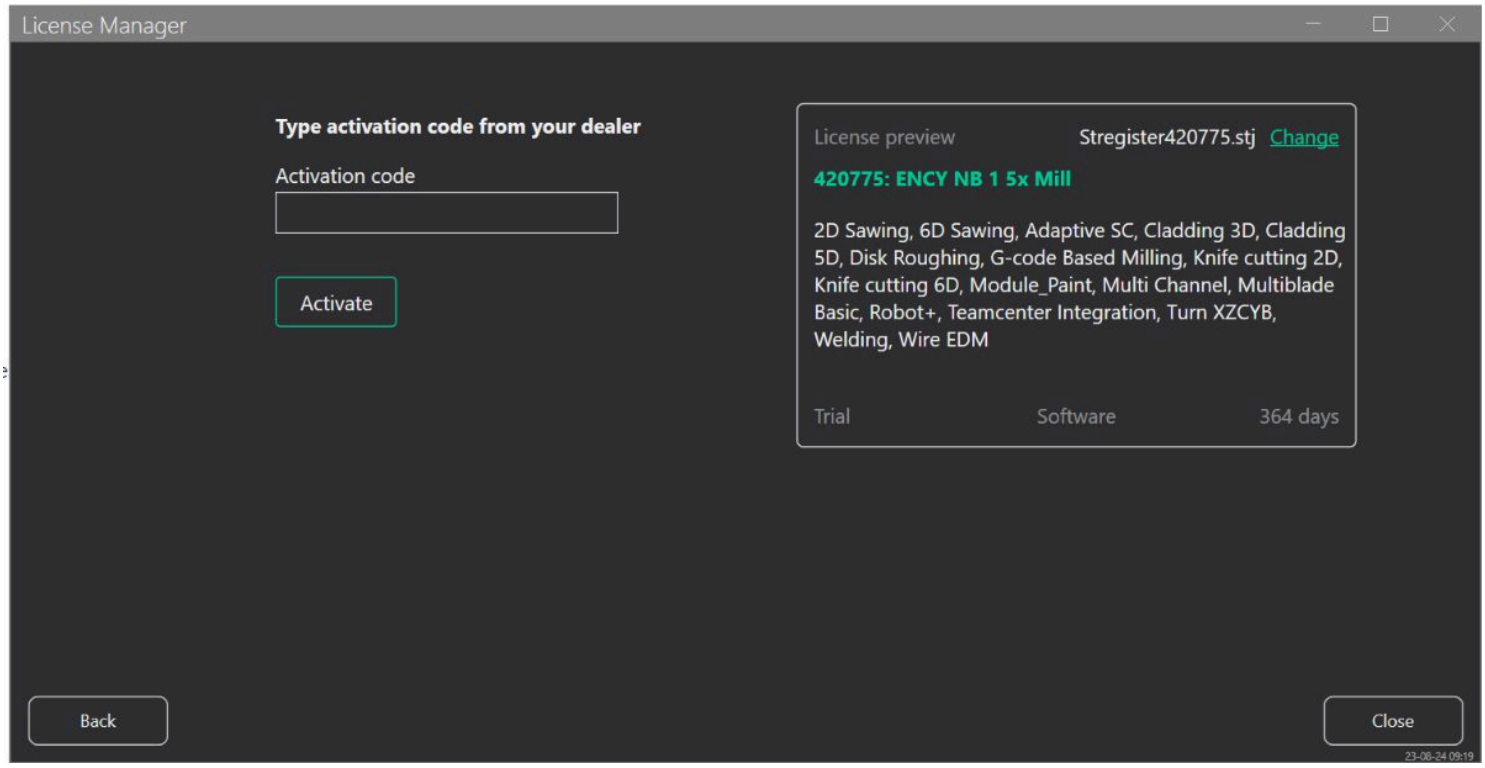
At the bottom of the window on the left is a button Add, by clicking opens this page:



Request for new license

If you do not have an Internet connection, when you click the Request a new license button, a window opens with the ability to receive a CM code to activate an offline license:

I have license file - Open the license file selection dialog. This menu item is useful when installing CAM system on a clean computer using an online installer.



Start on each Run - Turns on/off the start of the license manager every time CAM system starts.

User settings - Opens a window with user data settings

Refresh licenses - Updates the list of licenses.

Sign out - Logs out of the current authorized user account

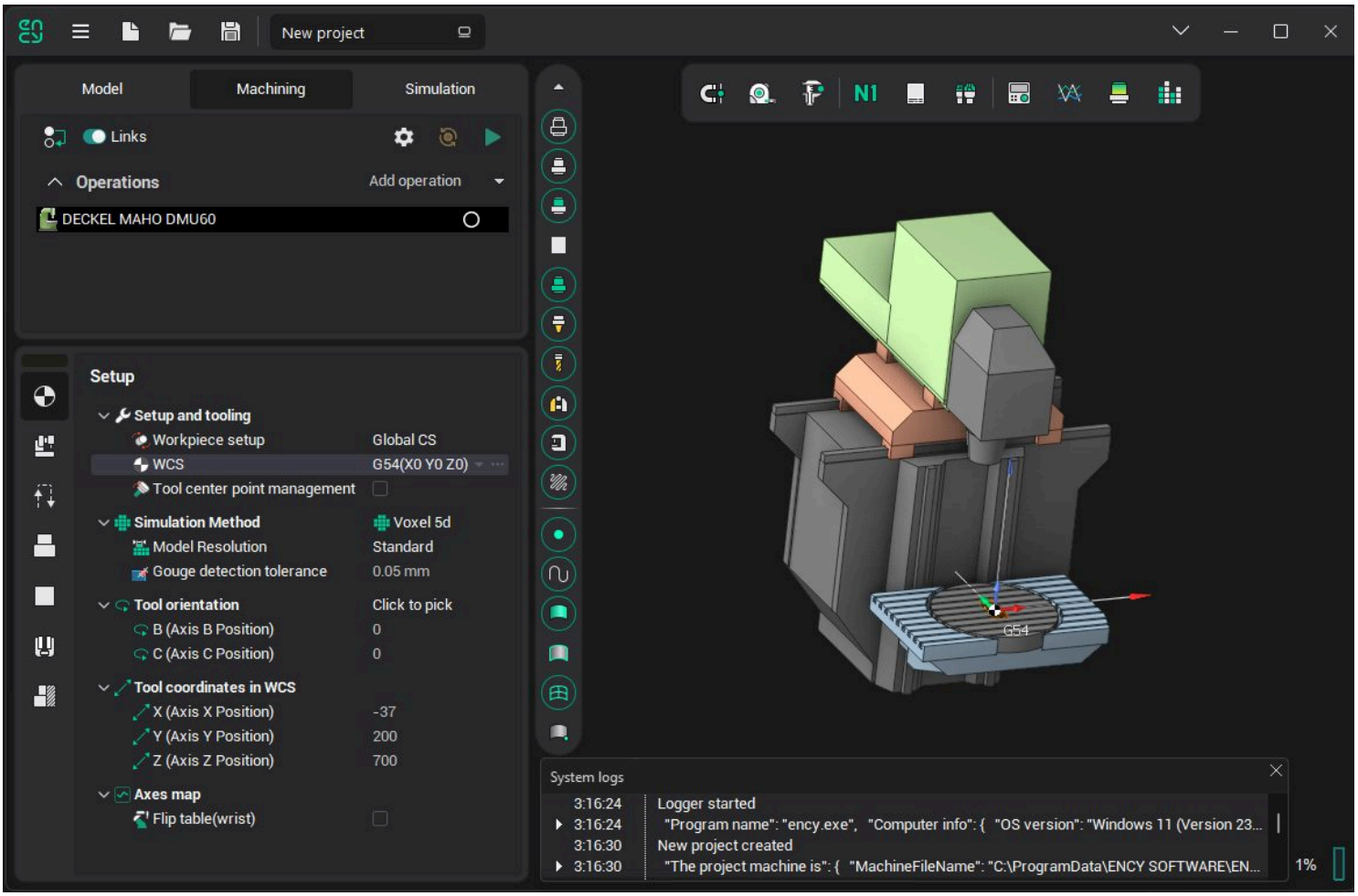
The page allows you to: enter your personal account, restore a forgotten password, register a new user, as well as the possibility of authorization through social networks.

After authorization, a list of licenses from the server will be available.

System logs

The system provides the ability to view a list of main events that have occurred since the application was launched.

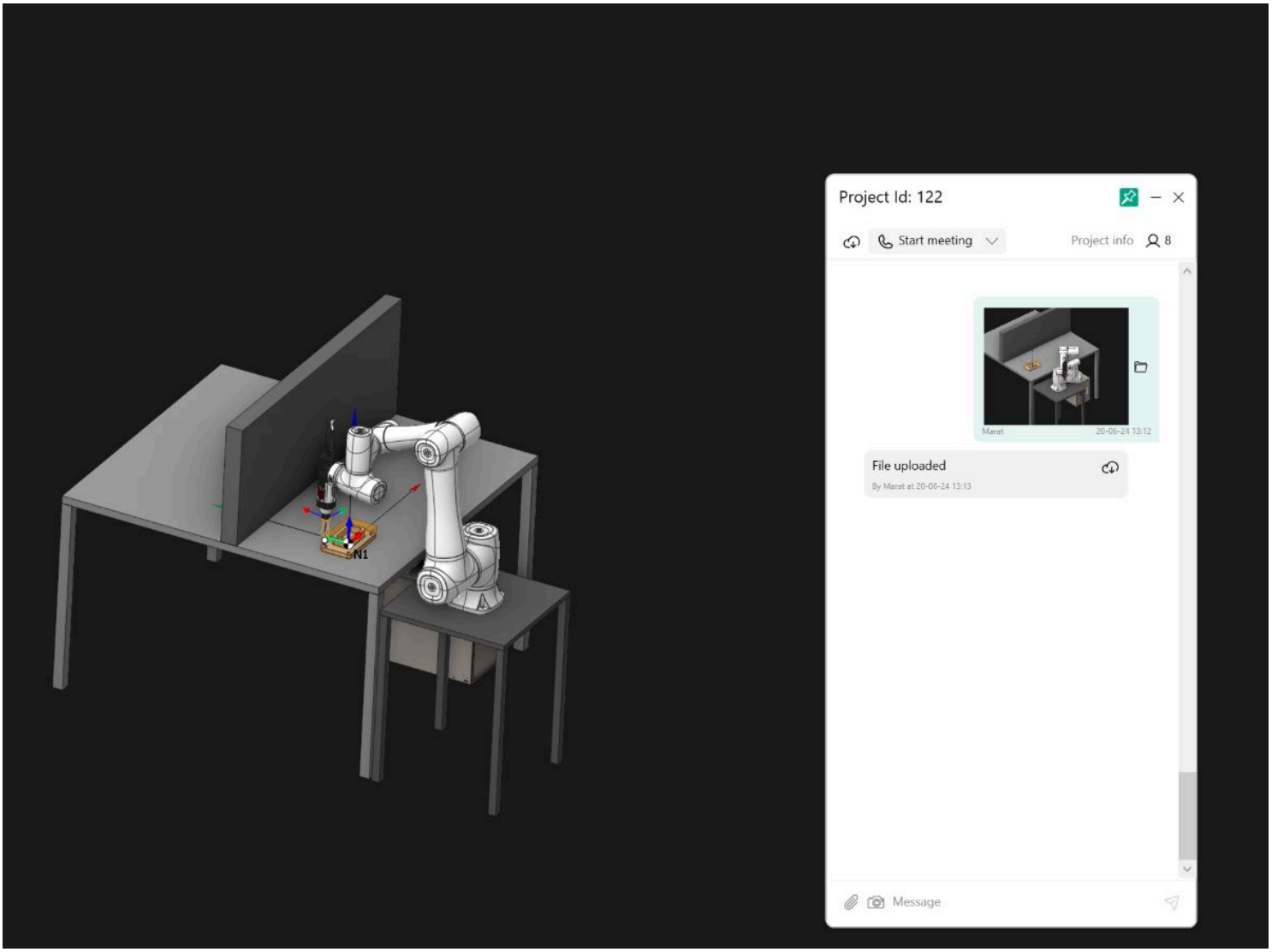
Events are displayed in a separate window, accessible via the Utilities System logs menu.



Each event includes the time of occurrence and a description. If the description is too long to fit on one line, an "Expand" button will appear at the beginning of the row. Events are color-coded based on their type: red for errors, yellow for warnings, and white for normal behavior.

This window can be particularly useful for troubleshooting, as it allows you to track event details and potentially identify the cause of issues.

Work with a team in the cloud



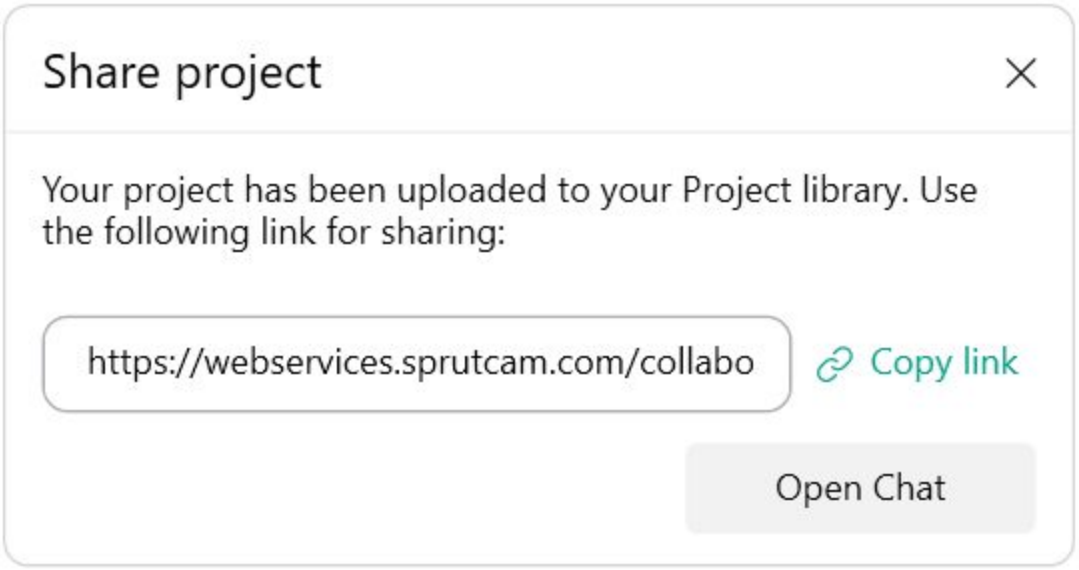
Application area:

Working with a team in the cloud, emphasizing the chat, shared project access, and cross-device compatibility. Imagine a workspace where distance is irrelevant. Where teams, geographically dispersed or simply in different rooms, can seamlessly collaborate on projects, share ideas in real time, and stay connected no matter where they are. This is the power of cloud-based collaboration.

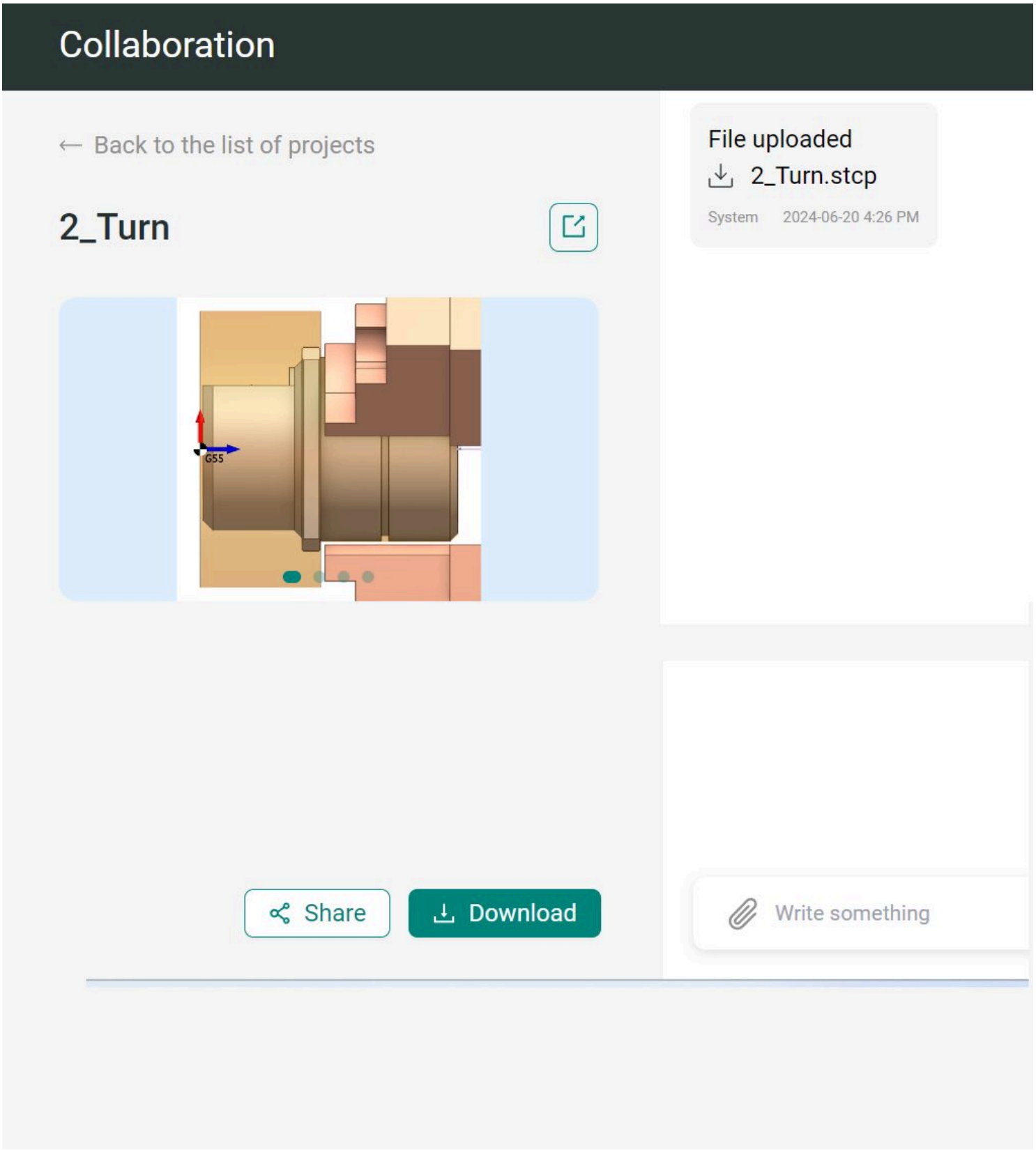
The peculiarities of working with a team in the cloud:

1.Shared you project. When you open a project, you can see a tick in its name to share your project with another person.

You'll be presented with a window to share your project. Send the link to your team



This link will grant your team access to a downloadable file on our website. The file provides information on the processing technology, and they can access it after authentication



2. Discuss things, share screenshots and videos, and work together to get things done.

The system supports data sharing through chat. Explore the various features available to optimize your workflow:


Project Id: 125
4
Start meeting
Project info
1
6
5
3
File uploaded
By Marat at 20-06-24 16:26
2



1. The chat supports the sharing of a wide variety of data, from project files to images and videos.
2. The chat history feature allows you to easily reference past discussions and stay informed about previous conversations.
3. You have the ability to see who has access to your collaboration.
4. The chat window can be pinned to ensure it remains visible above all other windows.
5. You can give your conversation partner a call.
6. Saving your project to the cloud enables your team to access and view the intermediate results. This action is automatically executed when saving the project within the CAM system.

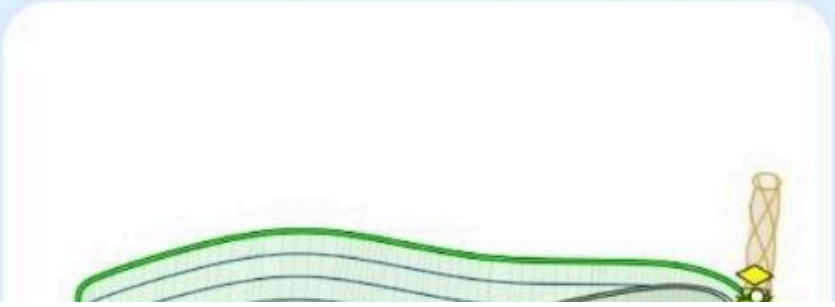
3.Maintain communication with others regardless of location through the use of this application.

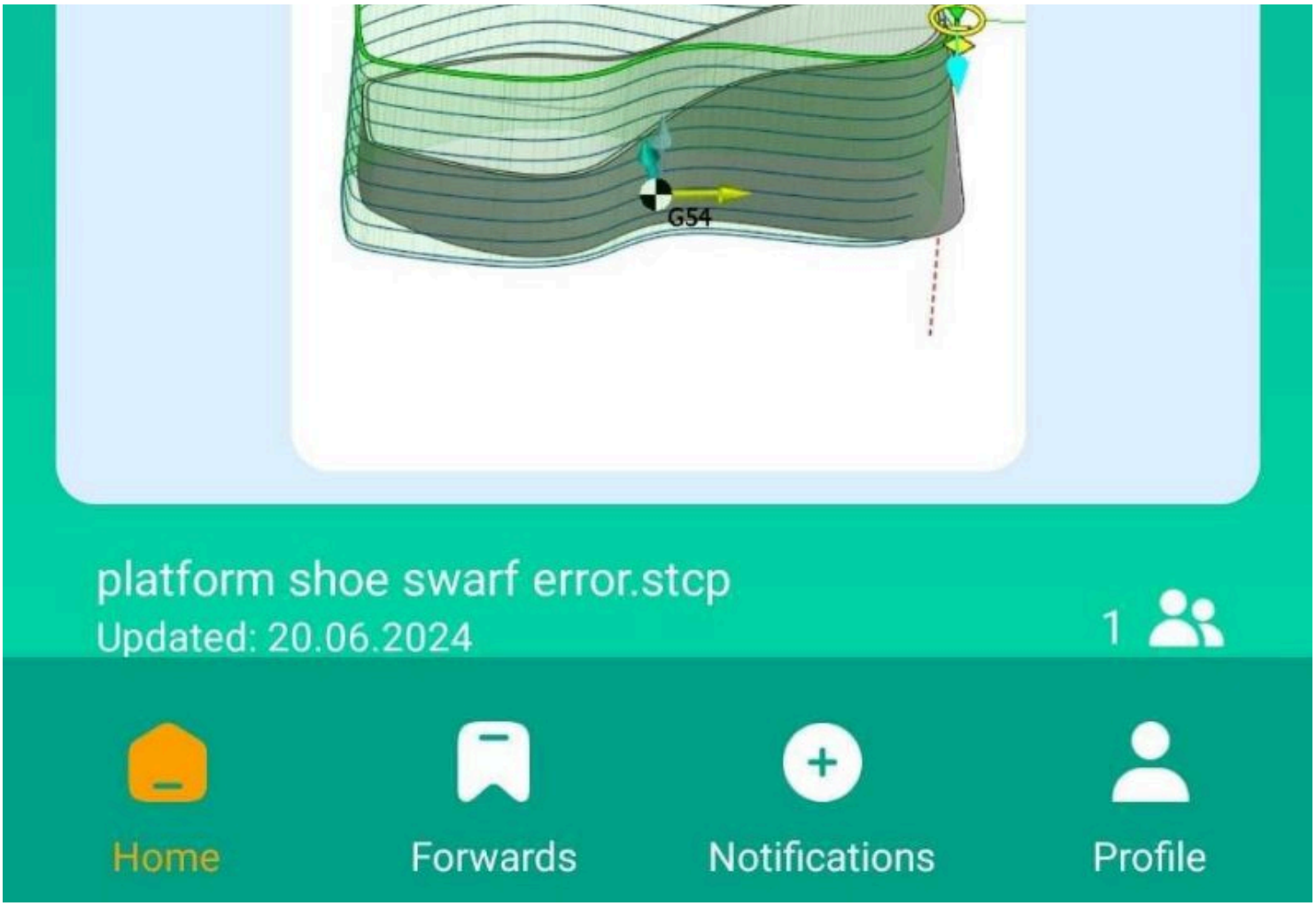
Search a project



Project171836474414025529460050171143...
Updated: 20.06.2024

8




G54
platform shoe swarf error.stcp
Updated: 20.06.2024
1
Home
Forwards
Notifications
Profile

Geometrical model preparation

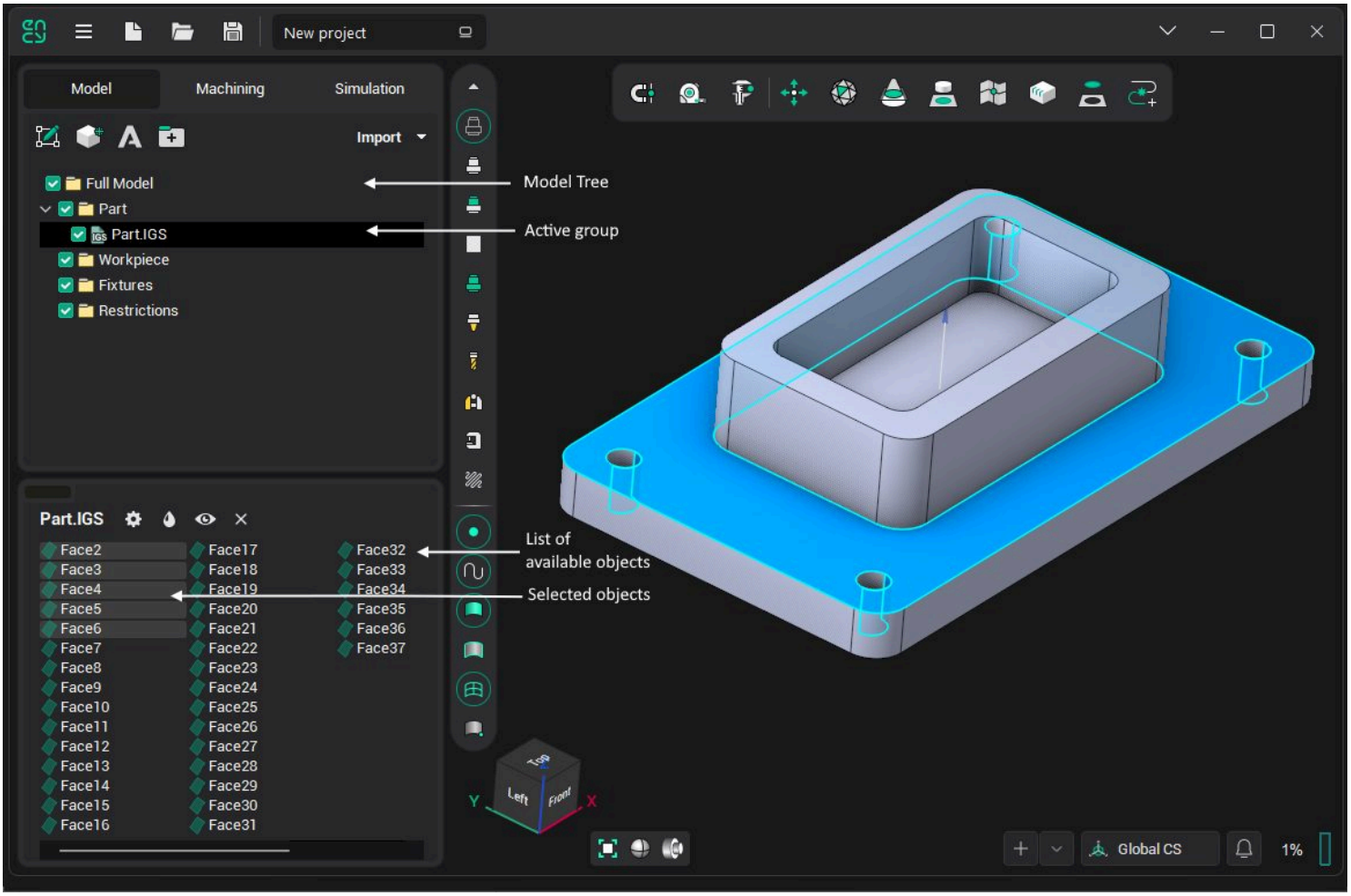
The geometrical model preparation mode becomes activated by the mouse click onto the Model tab in the system's main window.

In the Model mode one can:

- to import geometrical data (CAD) files;
- to correct the structure of the geometrical model;
- to perform spatial transformations of the geometrical objects;
- to generate new elements from existing ones;
- to alter the object's visual properties;

Access to elements of the model is performed both from the model tree window and interactively on the screen. Different functions can be launched via the pop up menu in the graphical window and the model tree window. Buttons for frequently used functions are put on the toolbar of the Model tab.

Geometrical model structure



A geometrical model in CAM system is represented as a tree of folders. Different geometrical objects are grouped in the folders. A working with the geometrical model structure is similar to the working with the files structure which is used in the Windows operating system.

Note: In keeping with most file systems, all objects inside one group must have different names. The presence of several objects with the same name is not allowed.

Most functions for the model editing are found in the geometrical model structure window. When creating a new project, the main folders of the geometrical model are generated automatically. Full Model is contained inside these predefined folders: Part, Workpiece, Fixtures.

The Part folder is the produced part for the full machining sequence by default. Therefore, it is recommended to import and transfer into the model group those geometrical objects, which define the final produced part.

The freeform workpiece model should be placed in the Workpiece folder. If using simple-form workpieces (box etc.), this folder should be left empty.

The Fixtures folder is aimed for models of clamps, supports, vices and other machining fixtures and equipments.

Geometrical objects types

This section describes geometrical object types: faces, meshes, curves, points, groups, active folders, and PMI.

Faces, meshes, curves and points

The geometrical model is presented as aggregate of geometry objects. Every object is a whole element and cannot be divided into smaller parts. These are the objects that affect the path of the tool movement toolpath in machining operations, which are formed from them (the part, workpiece, restrictions etc). The object have unique name, that includes objects type and ordinal number by default. The name can be changed by the user in the geometrical model structure window.

There is some geometry objects types:

- Face
- Mesh
- Curve
- Point
- Edge is a special type of geometry objects. This is the curve, that formed by ranges of meshes and faces. Edges is parametrical objects.
- Vertex is a point that marks the end of edge. Vertices is parametrical objects as well as edges.

Parametrical objects has some features. They keep permanent connection to source object. Consequently there is some limitations on available actions with them and they not listed in available objects in geometrical model structure window. However this objects can be selected in graphical window and can be used as Job assignment, Part, Workpiece, Fixtures, etc in Machining mode.

Note: Edges and Vertices is available for sewed faces only.

For more convenience, geometrical objects can be joined into groups.

Groups (folders)

A Group (folder) is an element of the geometrical model. It is aimed for grouping of geometrical objects which are similar by destination or by any other affinity. The folders are very useful to make adaptive projects. It is possible to define once the folders as geometrical parameters of operations. Next it is enough to change the folders contents to tune up the project for the modified geometry. In this case a part, workpiece, fixtures and a job assignment of operations will be updated automatically when the associated folders content is changed.

Note: When importing, groups can be automatically placed in special folders, if you add one of the following keys to the name:

- sc_f_ - the group will be moved to the Fixtures folder;
- sc_w_ - the group will be moved to the Workpiece folder;.

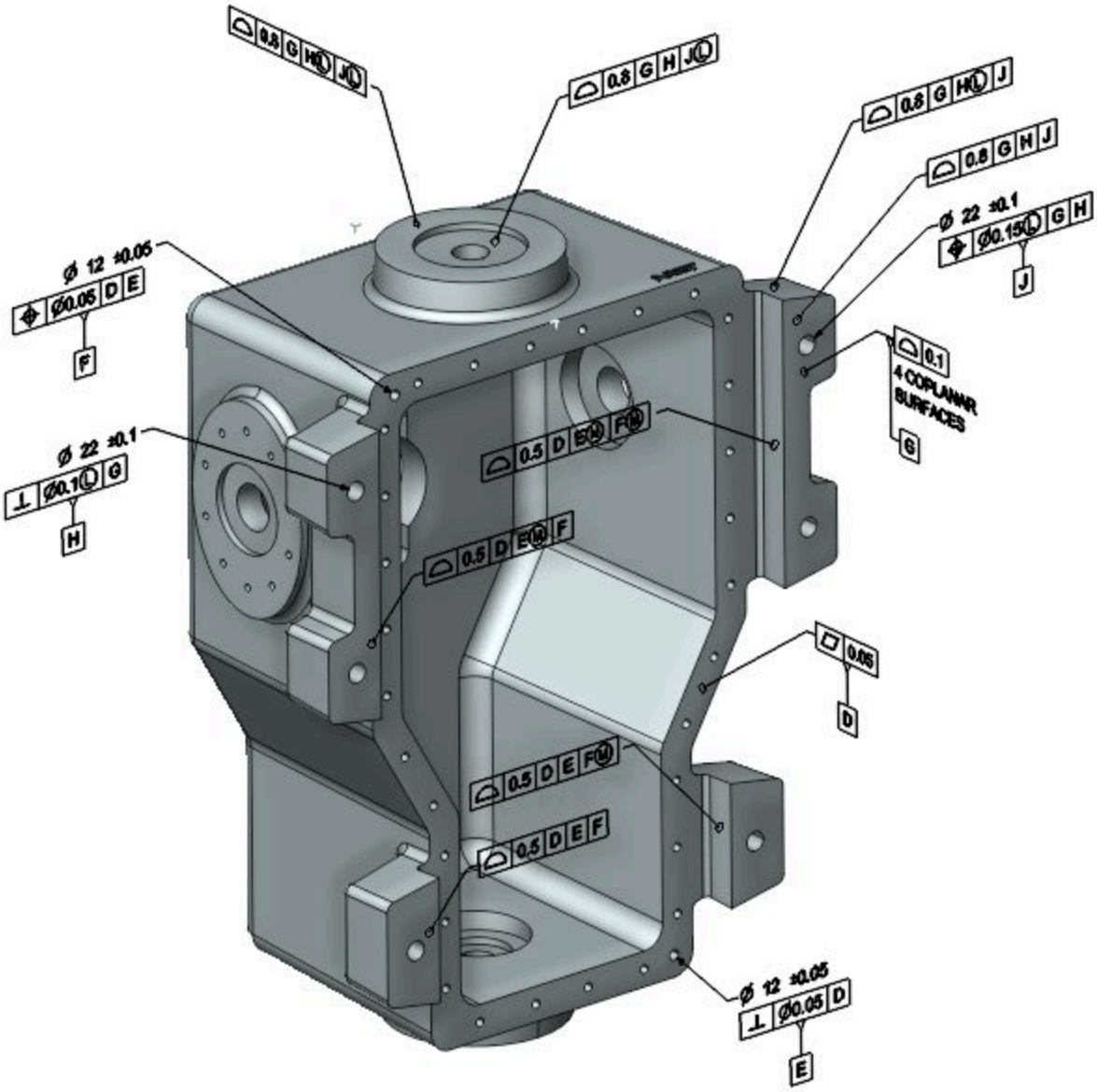
Active folder



Only one group can be active at one time. Work (import, detailing, transformations and etc.) is possible only with the objects located in the currently active group. The group, which is active is treated as a single object, and is considered indivisible. In order to work separately with elements located inside the group, you will have to "open" or activate it. These rules are similar to working with any file system (e.g. Windows). It is only possible to work with a folder or a file after having "entered" (opened) the folder containing it.

The currently active folder is shown in the model structure window. In the window one can also find the list of geometrical and structural elements, which are inside the active folder. Selection of these elements can be performed both from the list directly and from the graphic window. Selection of the active group is performed by selecting the corresponding tree-link in the model structure window or by selecting an object in the graphical window.

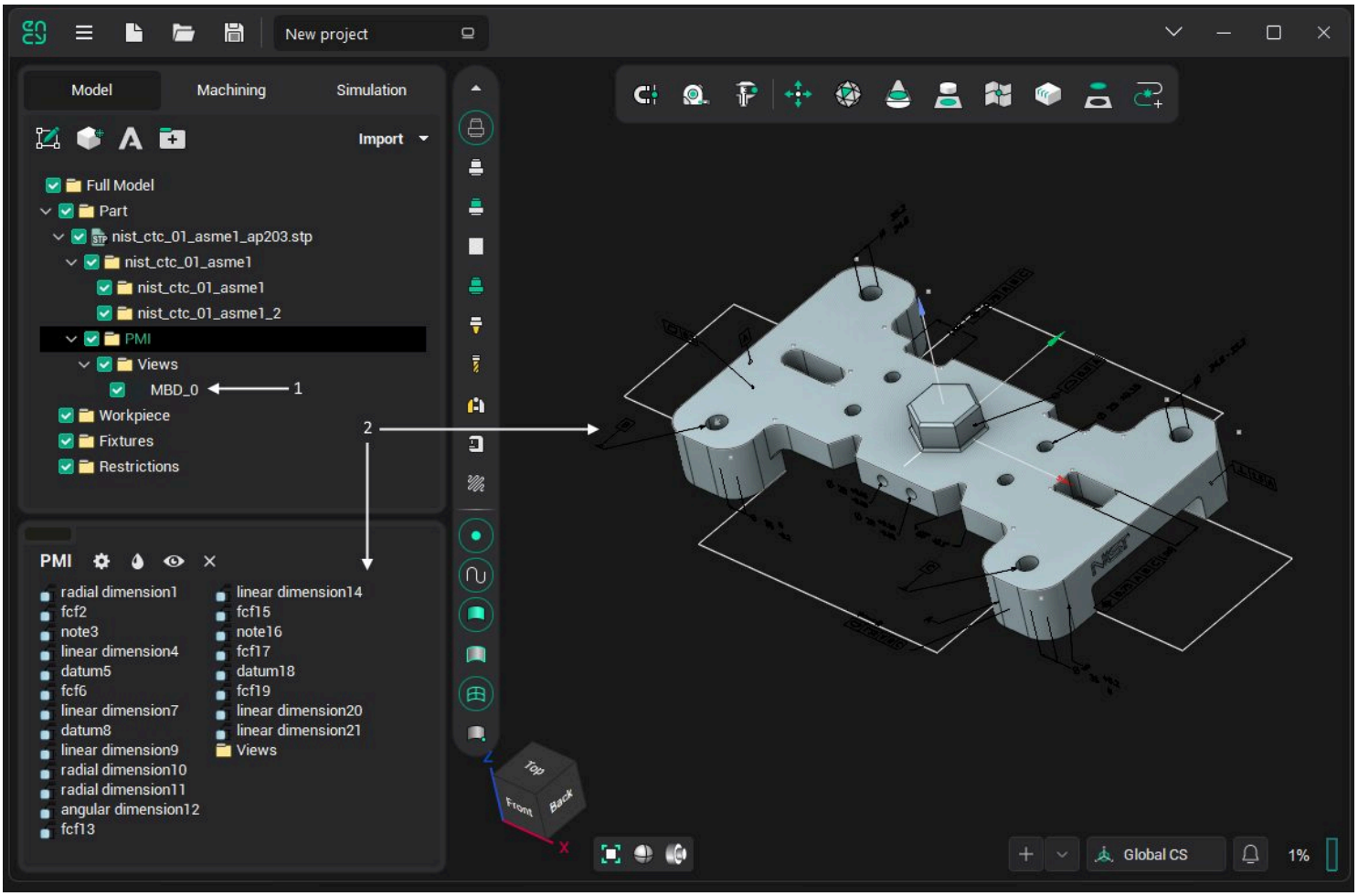
PMI



PMI -

Product and manufacturing information. These are various dimensions and notes related to the specific 3D model elements.

PMI presented as curves is imported with the model and stored within the model folder.



In the Geometrical model structure window, PMI is represented by the following objects:

1. Views
2. PMI nodes

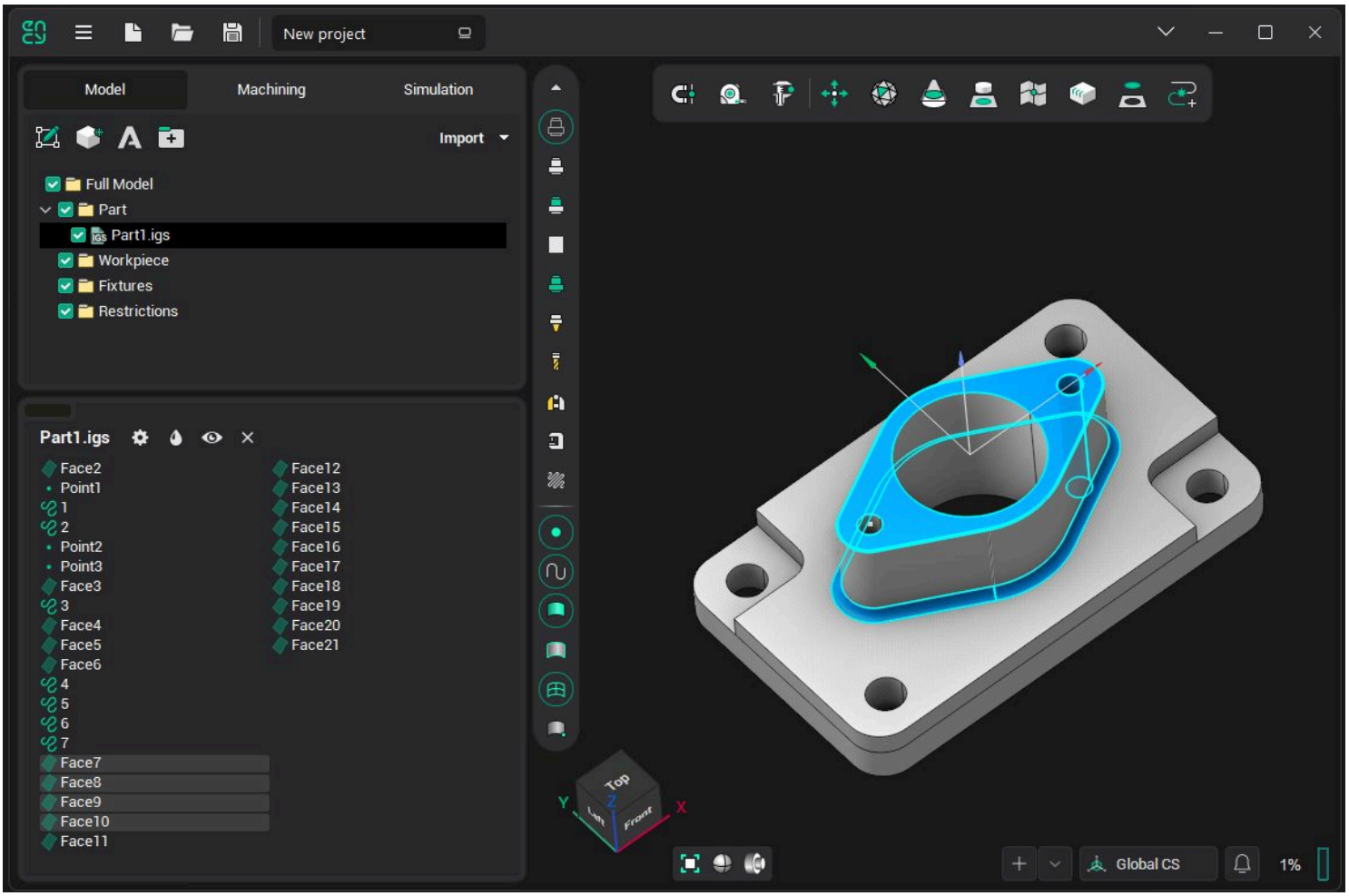
Some PMI nodes have connections to 3D model objects.. When you select a PMI node, they are also highlighted.

When selecting the View model rotates in accordance with the coordinate system of View. If you change the visibility of View, the visibility of the associated PMI nodes also changes.

PMI import works for Step, JT, Prt(NX)

Geometrical model structure window

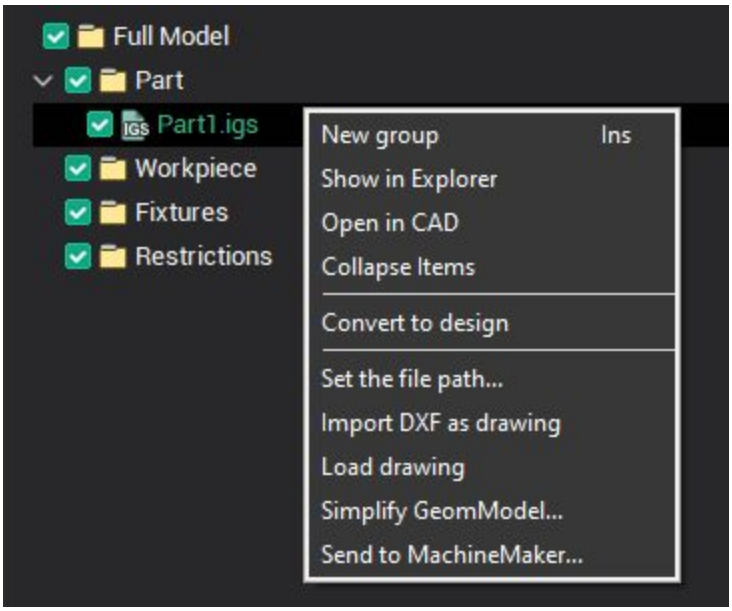
The model structure window consists of three panels: the tools panel, the model tree and a list of available objects.



In the model tree panel above, the structure of the whole model is displayed. Three nodes make up the groups of the model, which are located at different levels. The active groups are highlighted. When selecting an inactive group using the mouse or keyboard, the group becomes active and the list of available objects changes accordingly.

> Below, commands from pop-up windows will be considered, repeated commands will be skipped.

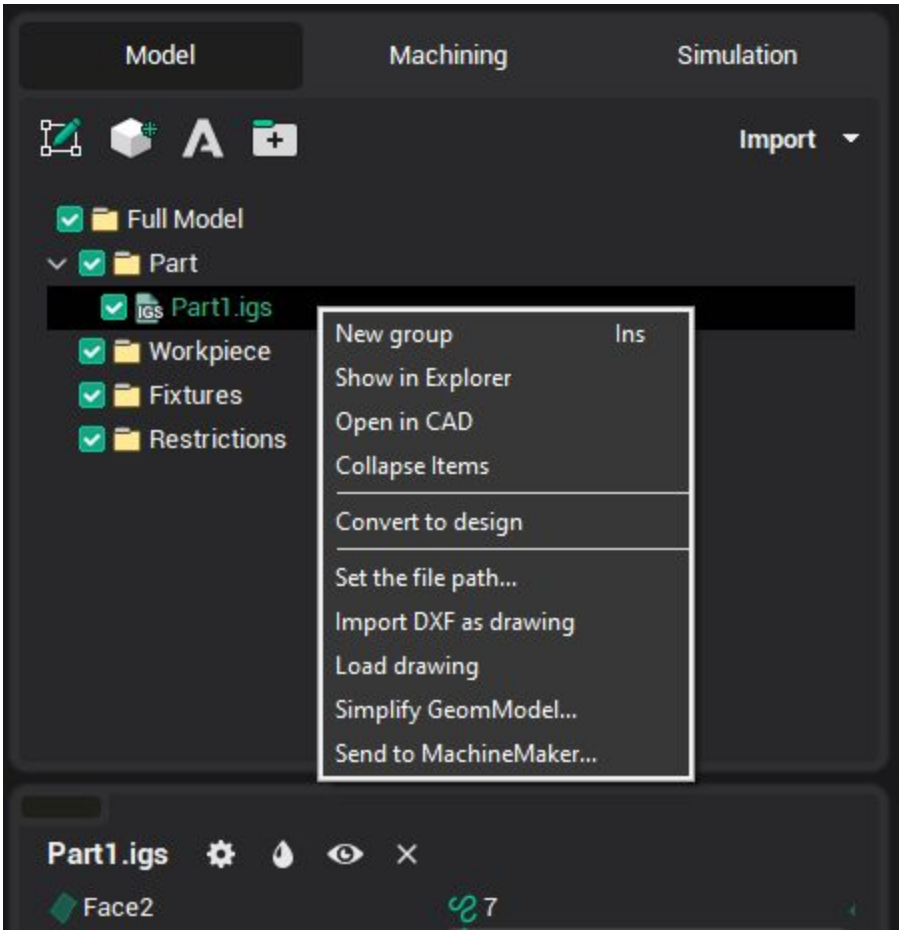
Clicking the right mouse button on a normal group in model tree brings up the following pop-up window:



Commands:

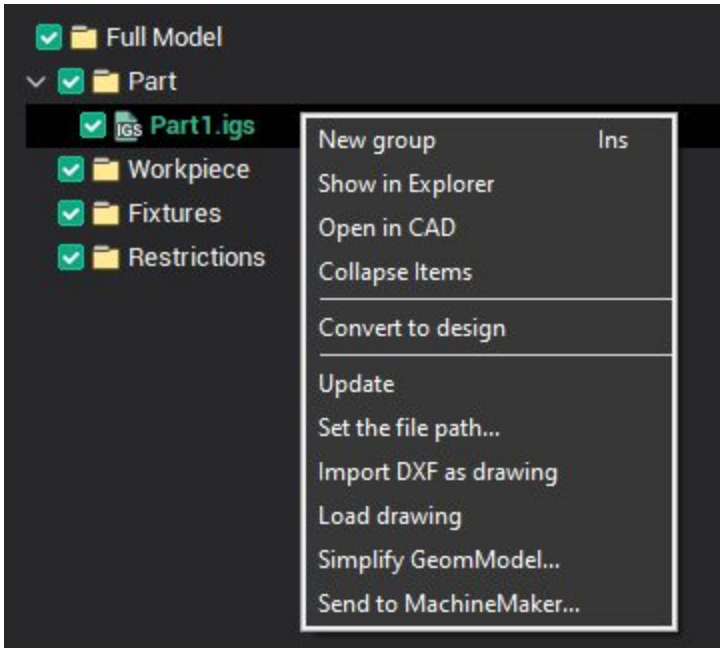
- New group - adds a new object the active group.
- Import DXF as sketch, Load sketch - commands for work with sketches.

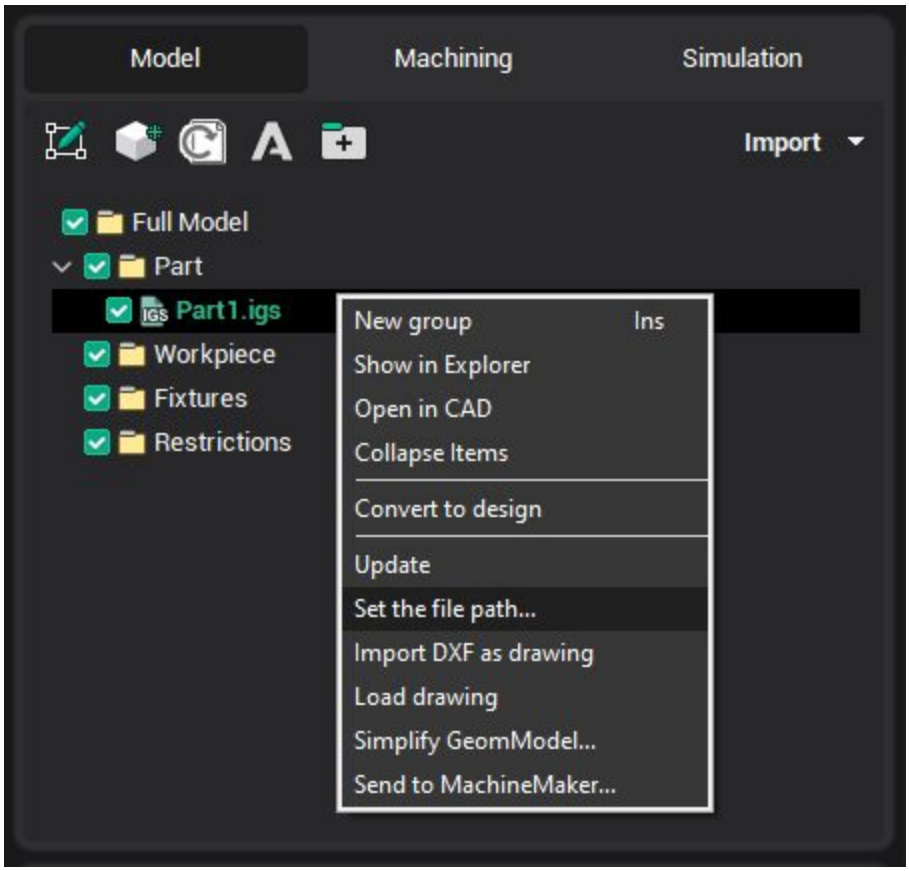
Clicking the right mouse button on the node of the imported model in the model tree causes the following pop-up window:



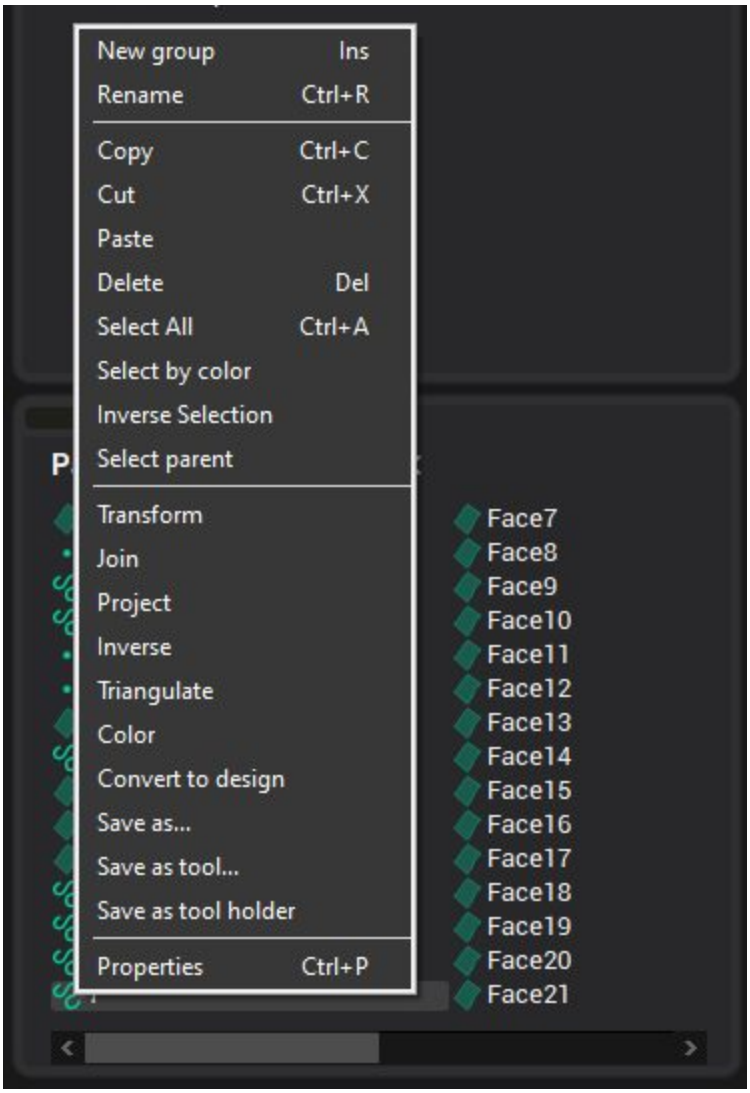
Commands:

- Show in Explorer - will open the folder, where contains the imported file;
- Open in CAD - file will be opened in the linked CAD, if not, then window selection appears.

> If the imported file has been changed, the name of the group will be allocated bold, the toolbar button appears "Model files update manager" , and the Update (update only selected group) command will appear in a pop-up window:
>
> 
>
>
> If the imported file has been renamed, moved or deleted, the Set File Path... (allows you to specify the new location of the imported file) command will appear in a pop-up window.



Clicking the right mouse button on any group in the list of available objects causes the following pop-up window:

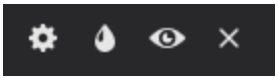


Commands:

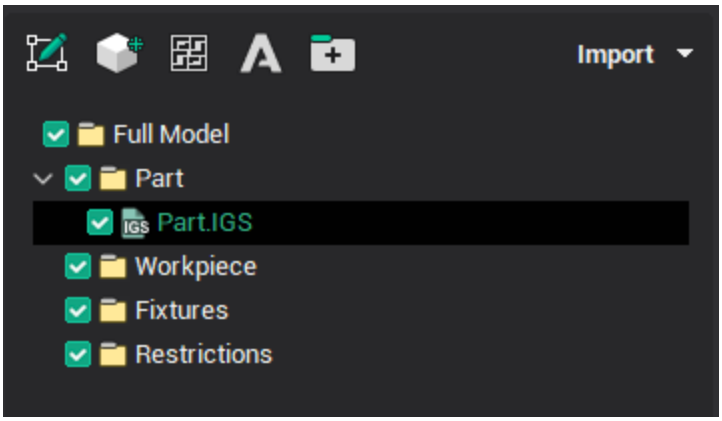
- Rename - allows you to edit the name of the selected object;
- Copy, Cut, Paste - working with the exchange buffer;
- Delete - deletes selected objects;
- Select All - selects all geometric objects;
- Select by color - selects all objects in the tree that have the same color as the selected one:
- Inverse Selection - selects all other objects, and the selection of current cancels;
- Transform, Join, Project, Triangulate - commands are duplicated on tool panel and described below;
- Inverse - inverting normals of surfaces;
- Color - allows you to change the color of the selected object;
- Save as... - offers to save the selected geometric objects in one of the supported formats;
- Save as tool... - saves the selected object in the form of an arbitrary shaped tool;
- Save as tool holder - saves the selected object in the form of an arbitrary holder;
- Properties - calls the properties window of geometric objects.

In the list of available objects, all groups and geometrical objects, which are a part of the active group, are displayed. That is, the objects which are currently available for selection and modification. Single left

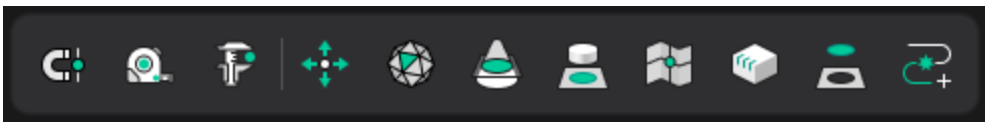
mouse clicking on any of the listed objects, selects that object.



- – Viewing and editing properties of selected objects.
- – Setting color of selected objects.
- – Set selected surfaces visible/invisible.
- – Deletes selected objects.



- – Starts the CAM system CAD drawing.
- – Creating texts.



- – Spatial transformations of selected objects.
- – Surface triangulation.
- – Getting curves as section of 3D model by plane.
- – Surfaces boundary projection on plane.
- – Sew faces.
- – Triangulation of selected curves or patching holes in meshes.
- – Joining curves.

Object selection

All visible elements of the geometrical model can be easy selected from the screen. Elements of the model, that are available for selection, are highlighted when you hover the mouse pointer on them. The element selecting is performed by the left mouse button click. At any time, one of folders of the geometrical model tree is active. When a geometrical object choosing the parent folder of the object automatically becomes active. Transition to parent group (change of the active folder level) is possible by double-clicking the left mouse button in the graphics window. Thus, the possibility of effective navigating through the model tree without need to resort to the structure of the window realized.

All object modification operations are only performed on selected objects.

To select a part of the model or separate elements it is necessary to activate the group, which includes these geometrical objects, or groups. Then select the desired objects from the list of available objects in the graphic window. If selecting in the graphic window, the objects selection mode must be switched on in drop-down menu of the view port or [S] button on keyboard must be hold.

An element in the graphic window can be selected either by single left mouse clicking or by dragging a rectangular area. To use rectangular area selection method, press the left mouse button, hold, and move the mouse. If selecting by clicking in the current view, there can be more than one object selected. The other method is to select the object in the list of available objects in the model structure window.

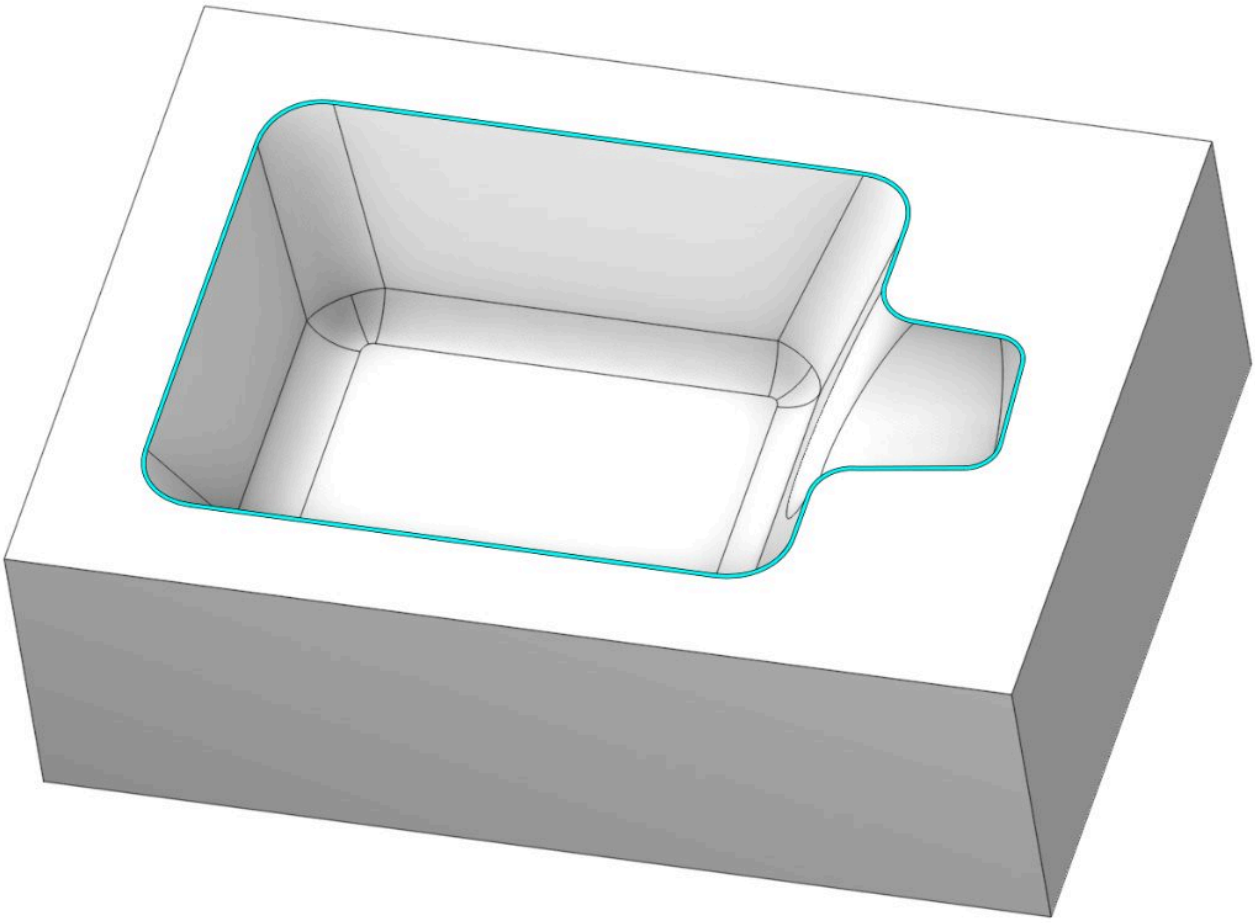
The normal selection method described above, allows user to define objects singly. To select multiple objects, press and hold the [Ctrl] key. Doing this the newly selected object will be selected but if selected object is selected again then it becomes de-selected. Alternatively, by holding the [Shift] key, the selected object(s) will always be selected.

Another convenient ways to select multiple objects:

- press and hold [Shift] key and move the mouse with pressed left button above objects you want to select.
- use [Ctrl]+[Shift]+Double click to select faces that are connected smoothly.

In order to select objects of one type, one should use the Object selection filter. With this, only the objects of the required type will be displayed in the model structure window and be selectable in the main graphic window.

Intellectual object selection



Application area:

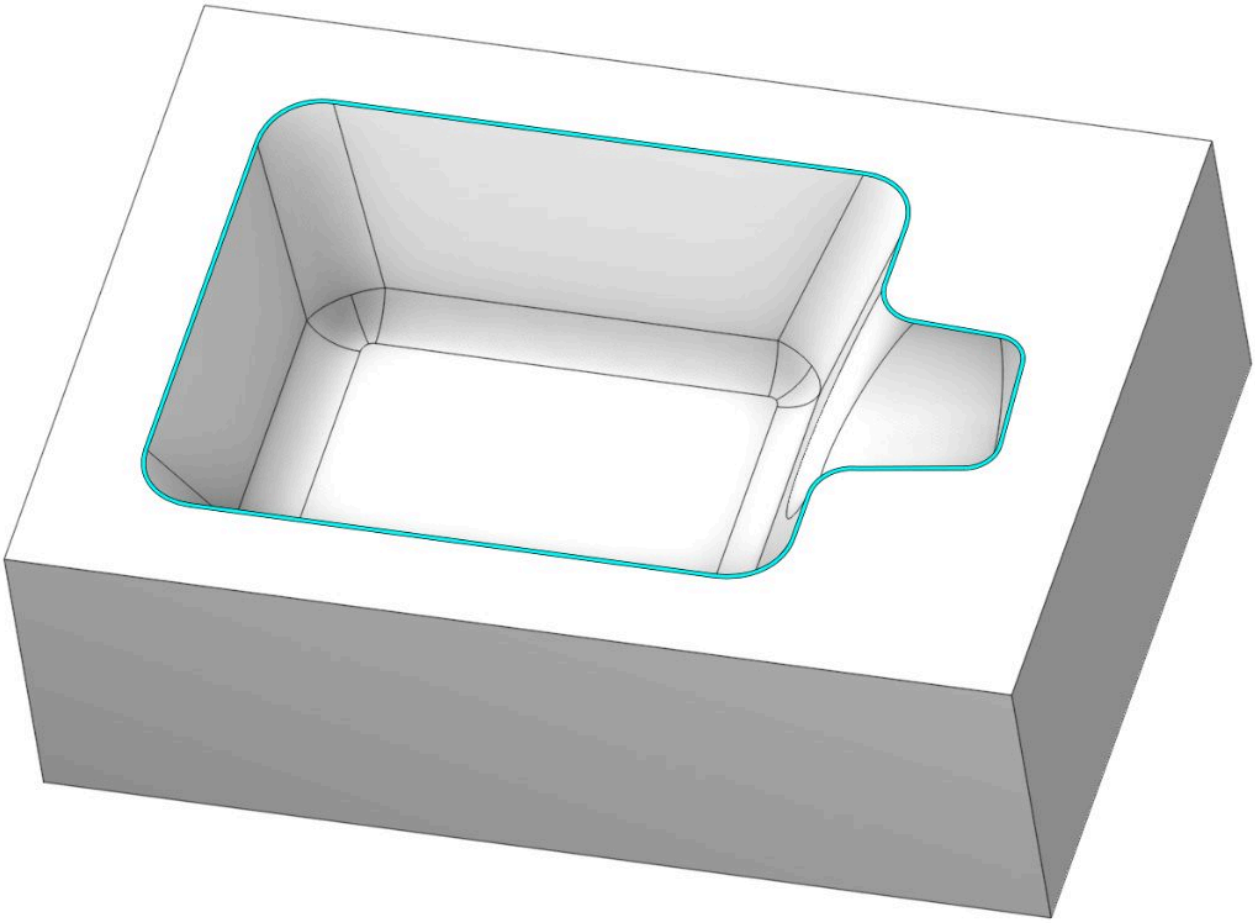
The functionality allows you to save time when selecting various parts elements due to certain rules.

To activate a particular algorithm, you need to use different hot keys:

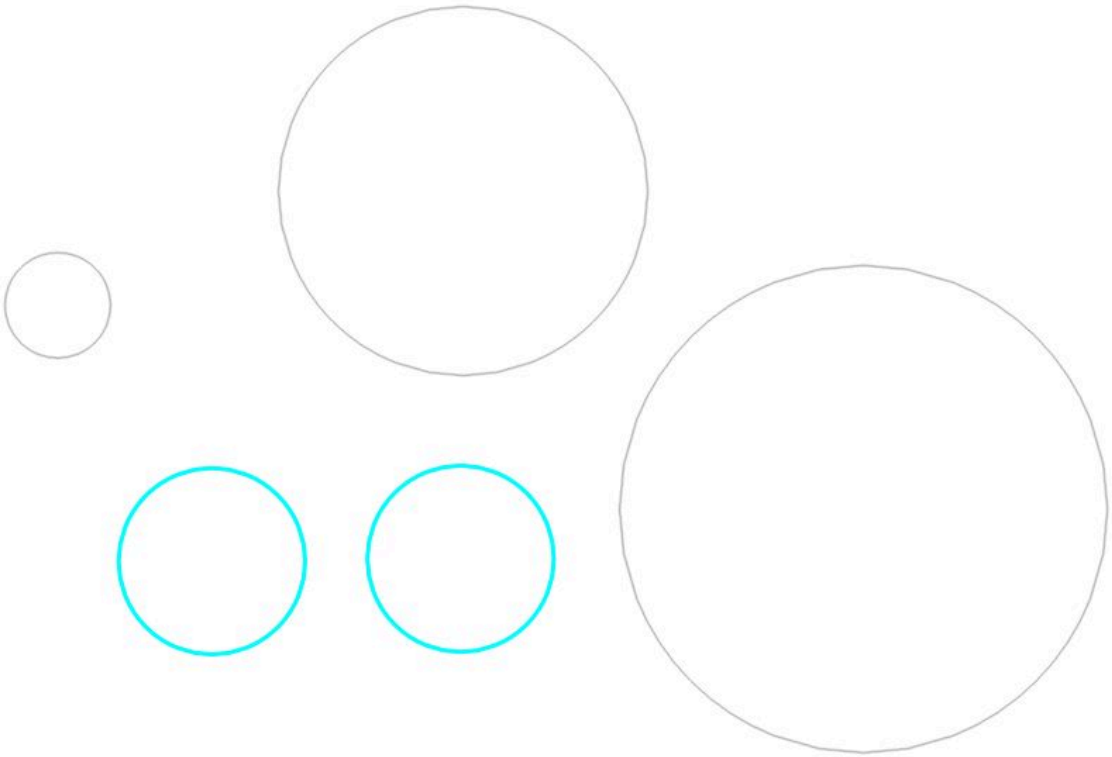
Double click on a 3d model entity.

▼ Click here to expand..

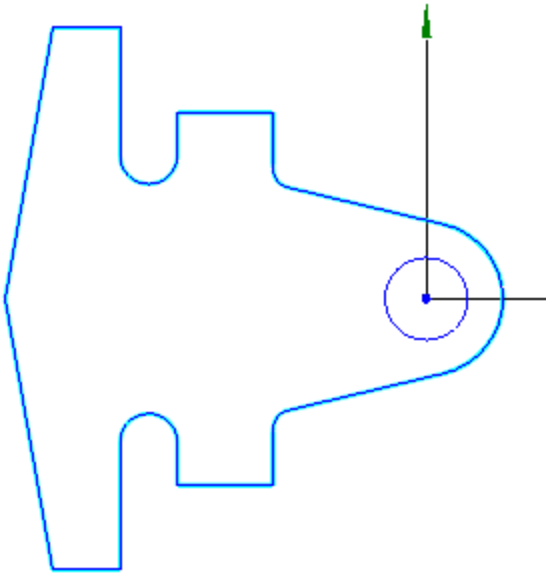
- Double click on a 3d edge automatically selects all tangent edges.



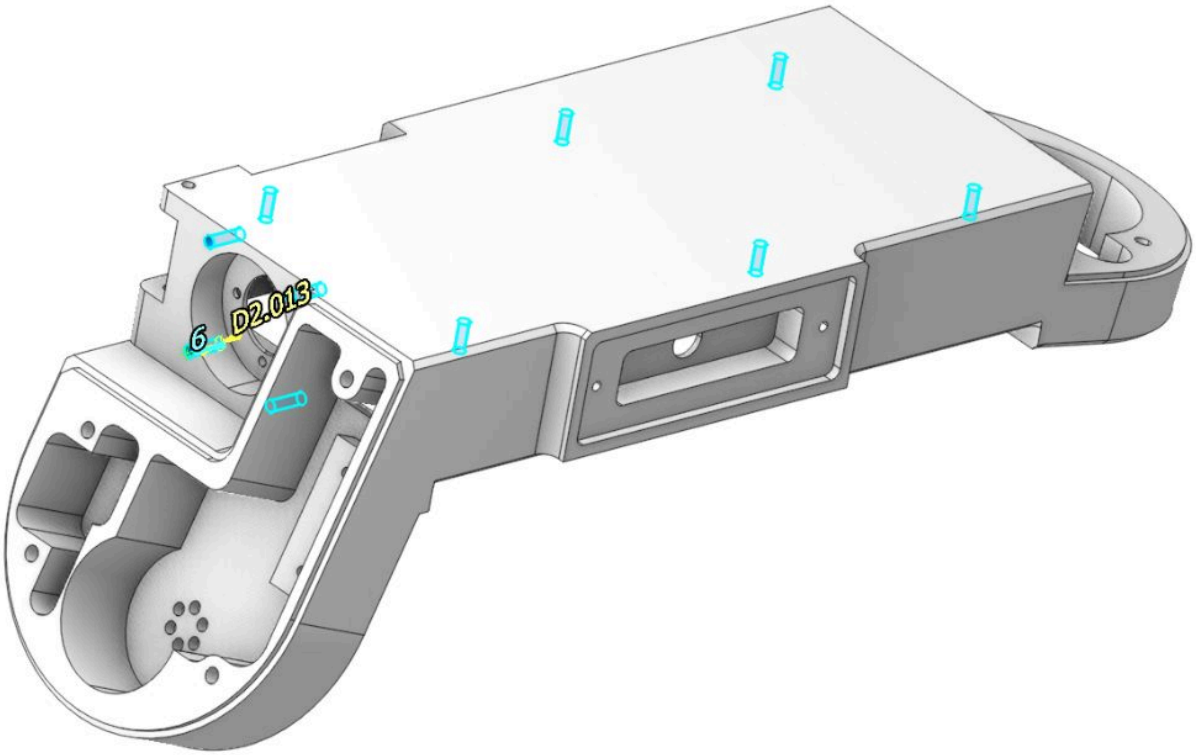
- Double click on a circle selects all the circles with the same diameter.



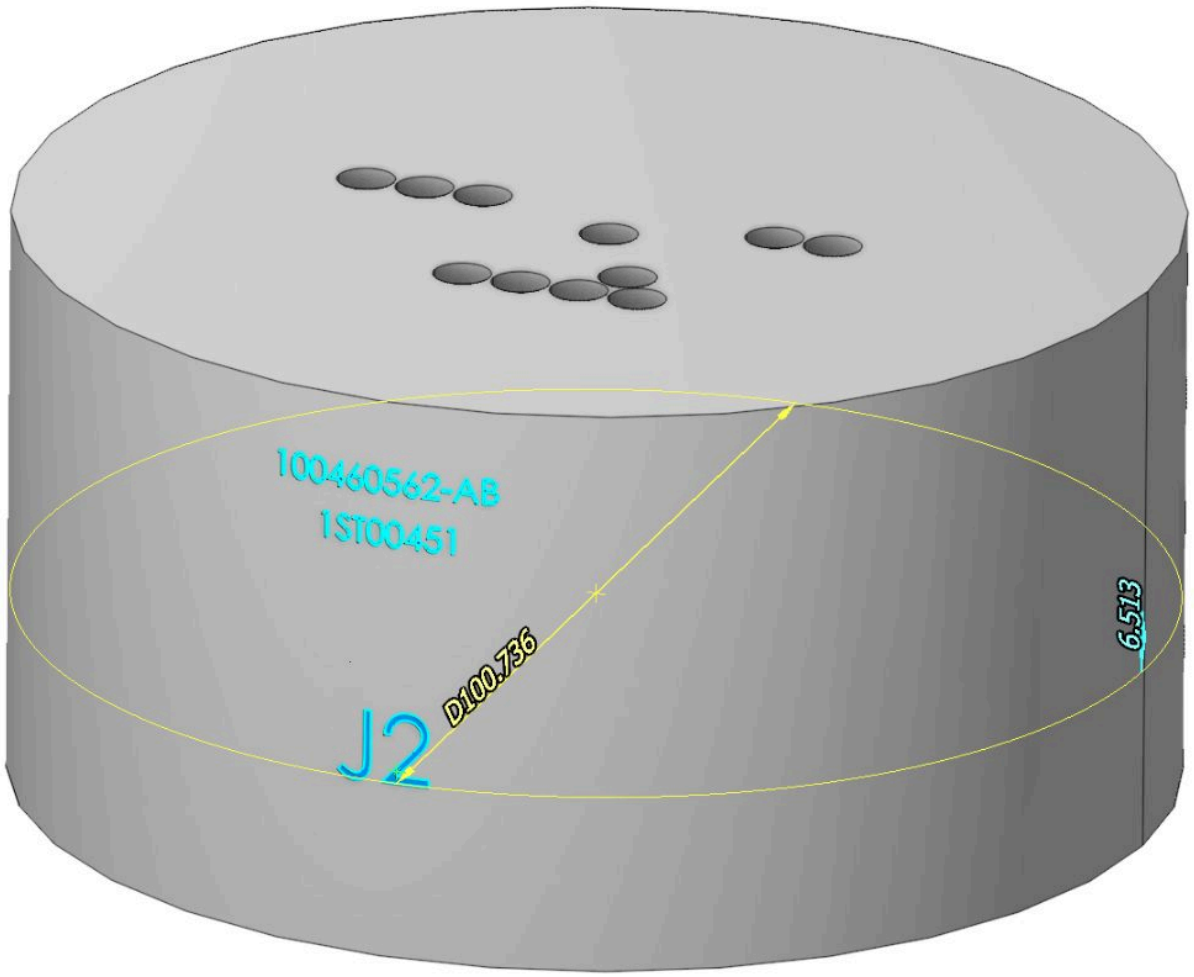
- Double click on a segment of a contour automatically selects all segments of the contour.



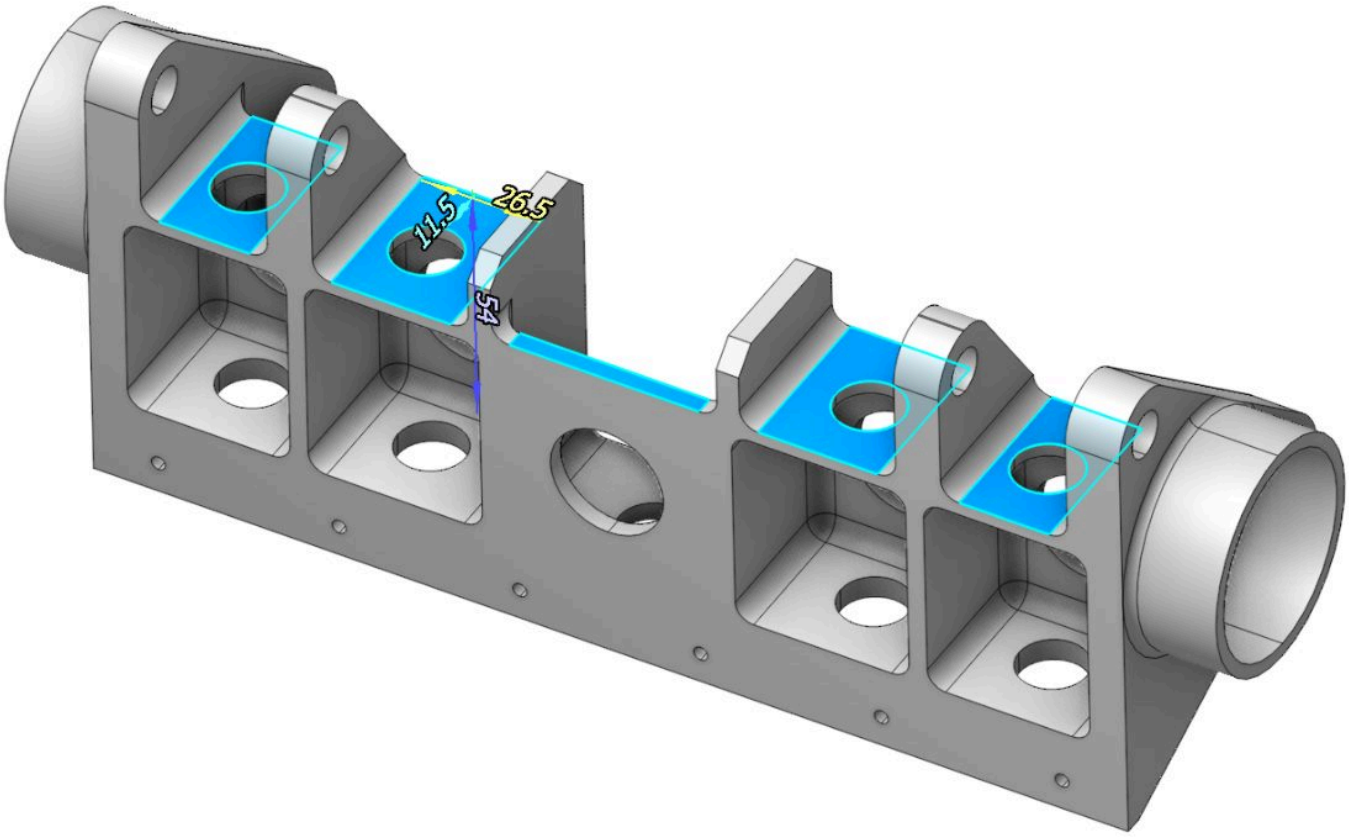
- Double click on an internal cylindrical face or hole automatically selects all the inner cylinders with the same diameter.



- Double click on an external cylindrical face automatically selects all cylindrical faces with the same diameter and same axis.



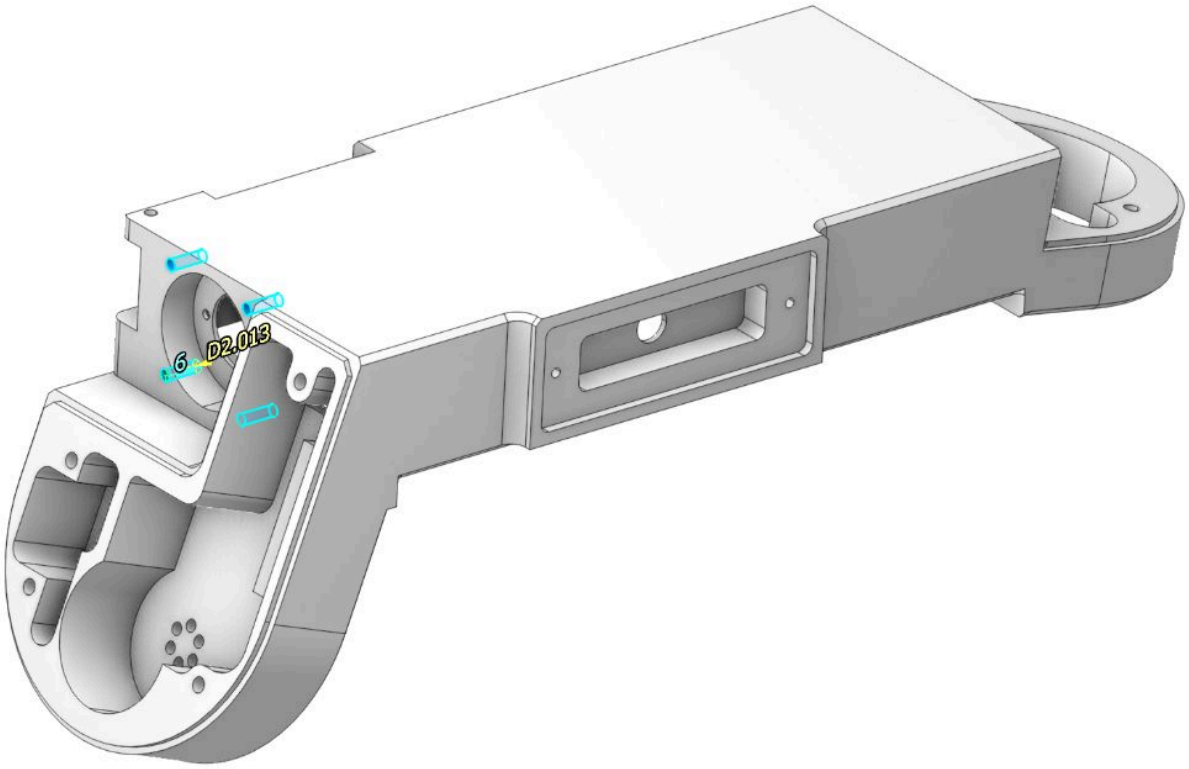
- Double click on a planar face automatically selects all coplanar faces.



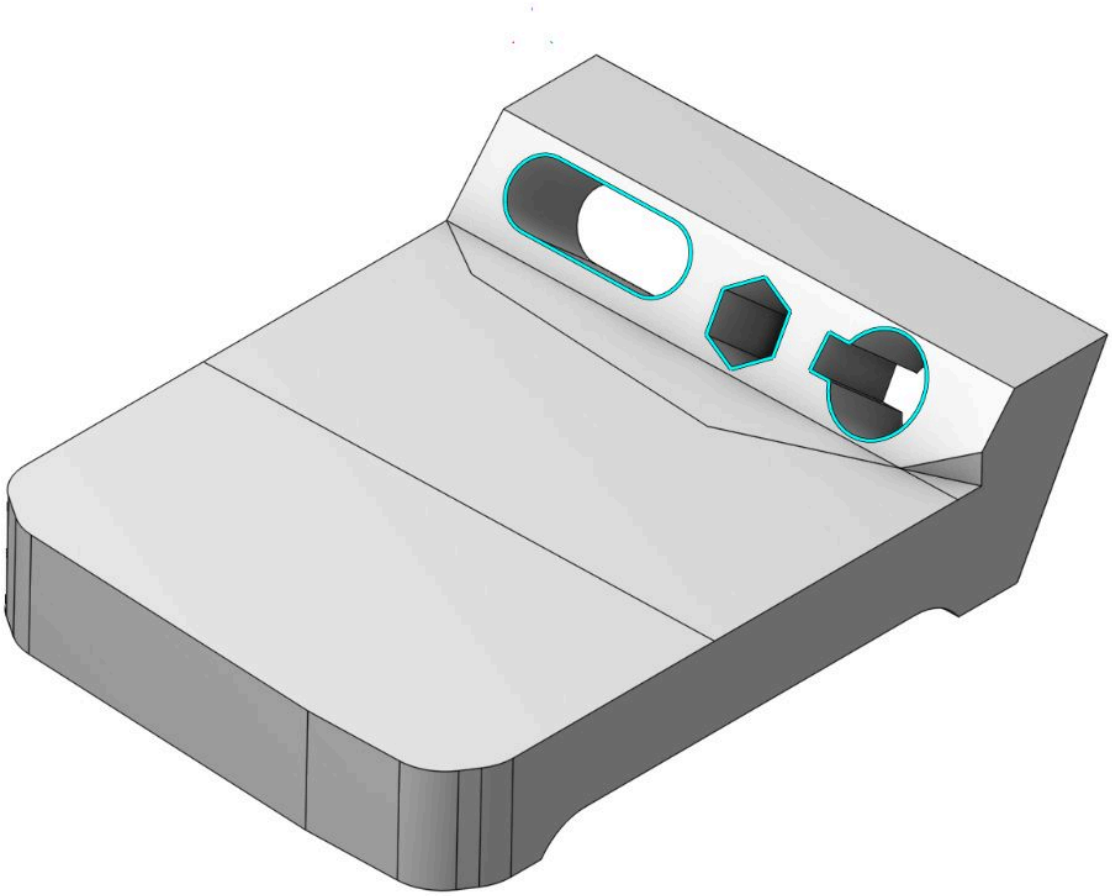
Double click + shift on 3D model object.

▼ Click here to expand..

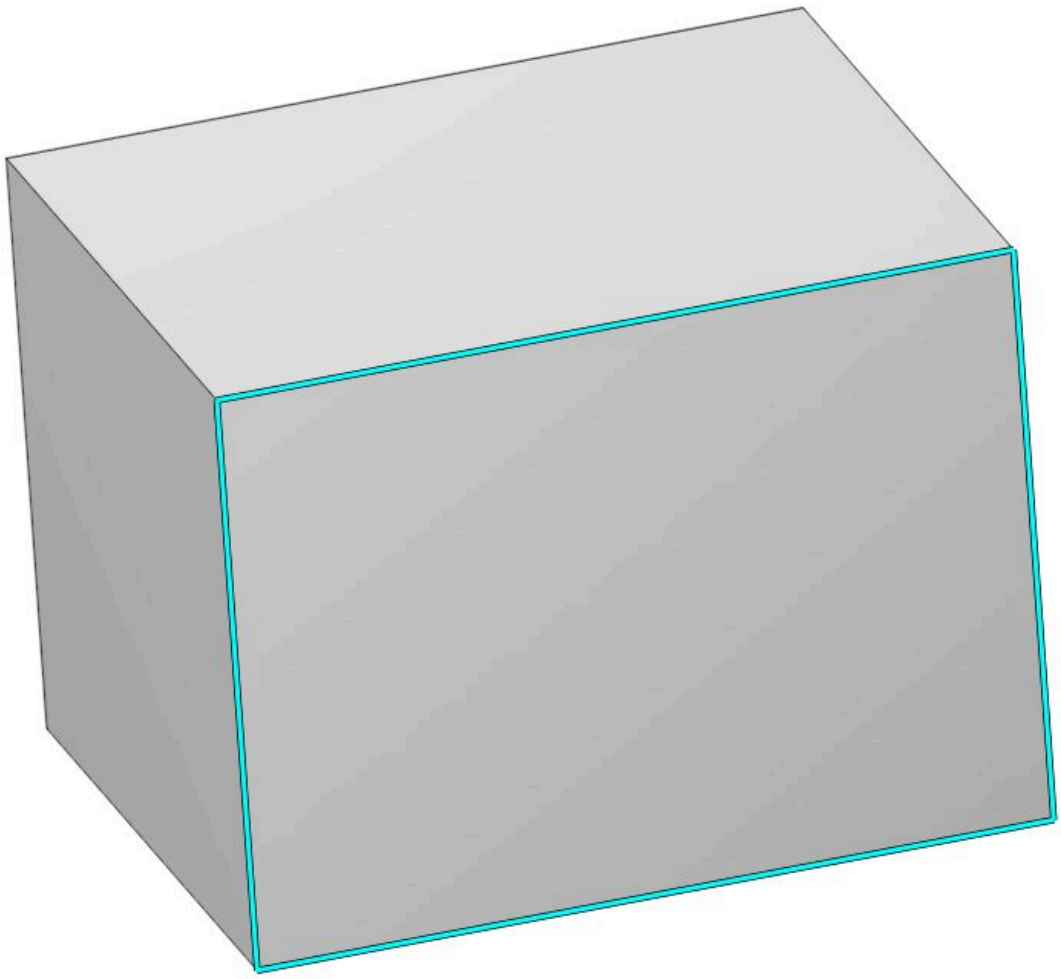
- Selects all internal cylinders with the same diameter in a given plane.

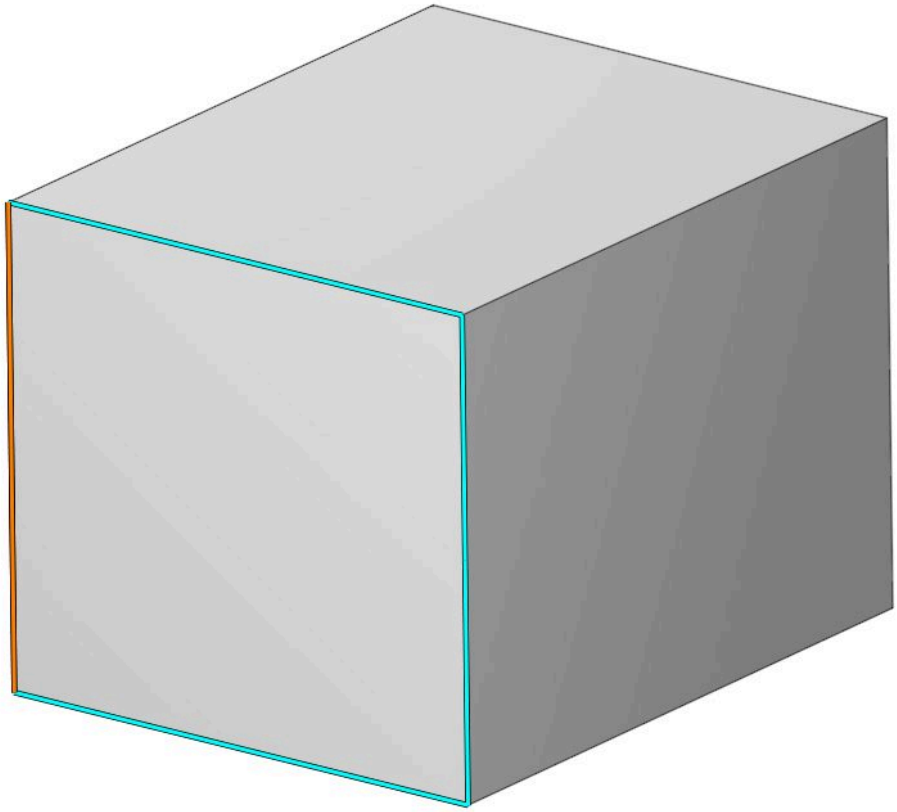


- In the Job assignment 6D contour allows you to select edges similar in size



- Allows you to select non-tangential edges (edges are selected depending on the position of the part in the graphics window)

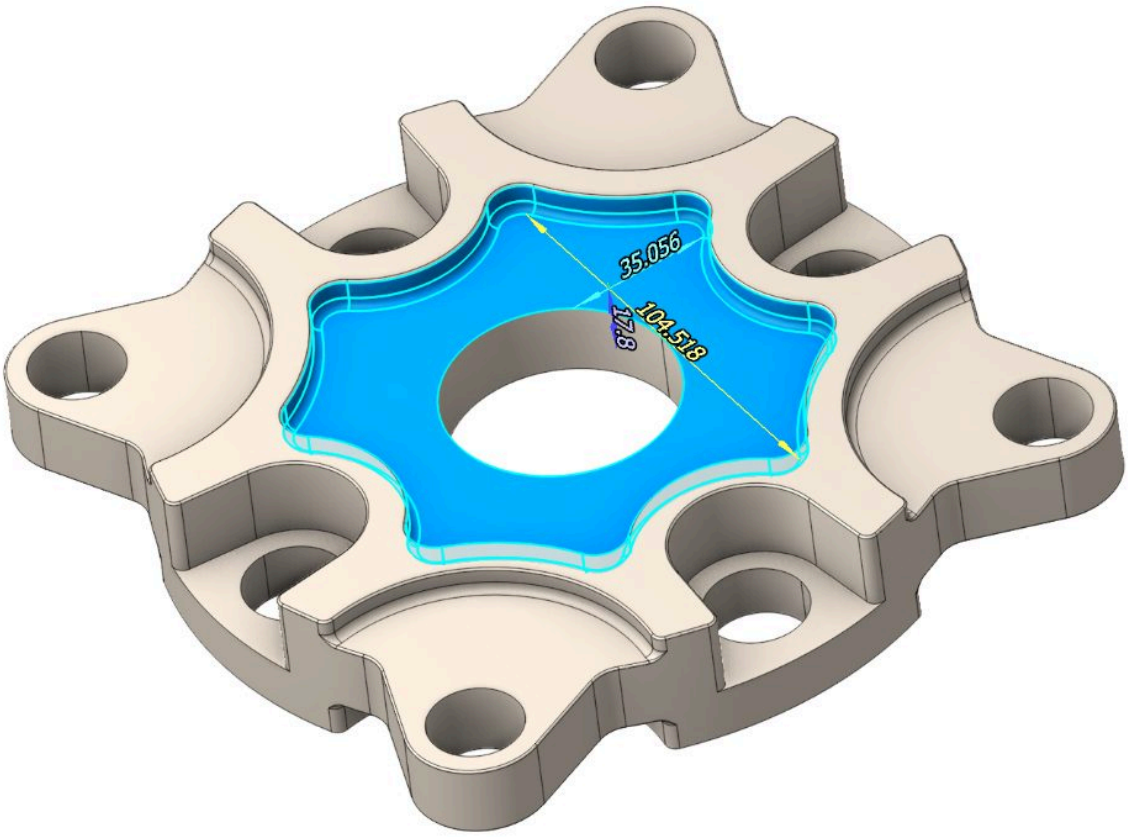




Double click + shift + ctr on 3D model object.

▼ Click here to expand..

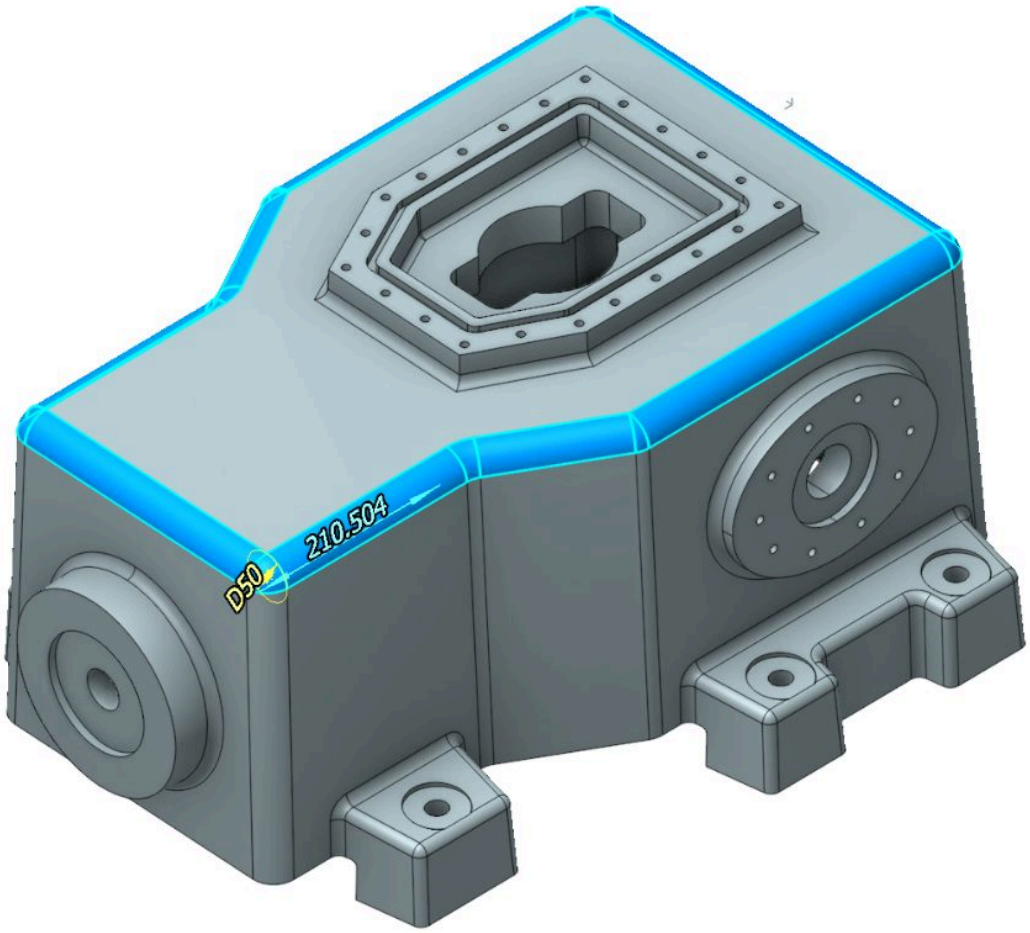
Allows you to select smoothed surfaces.



Right-click the surface to activate the mode (Propagation)

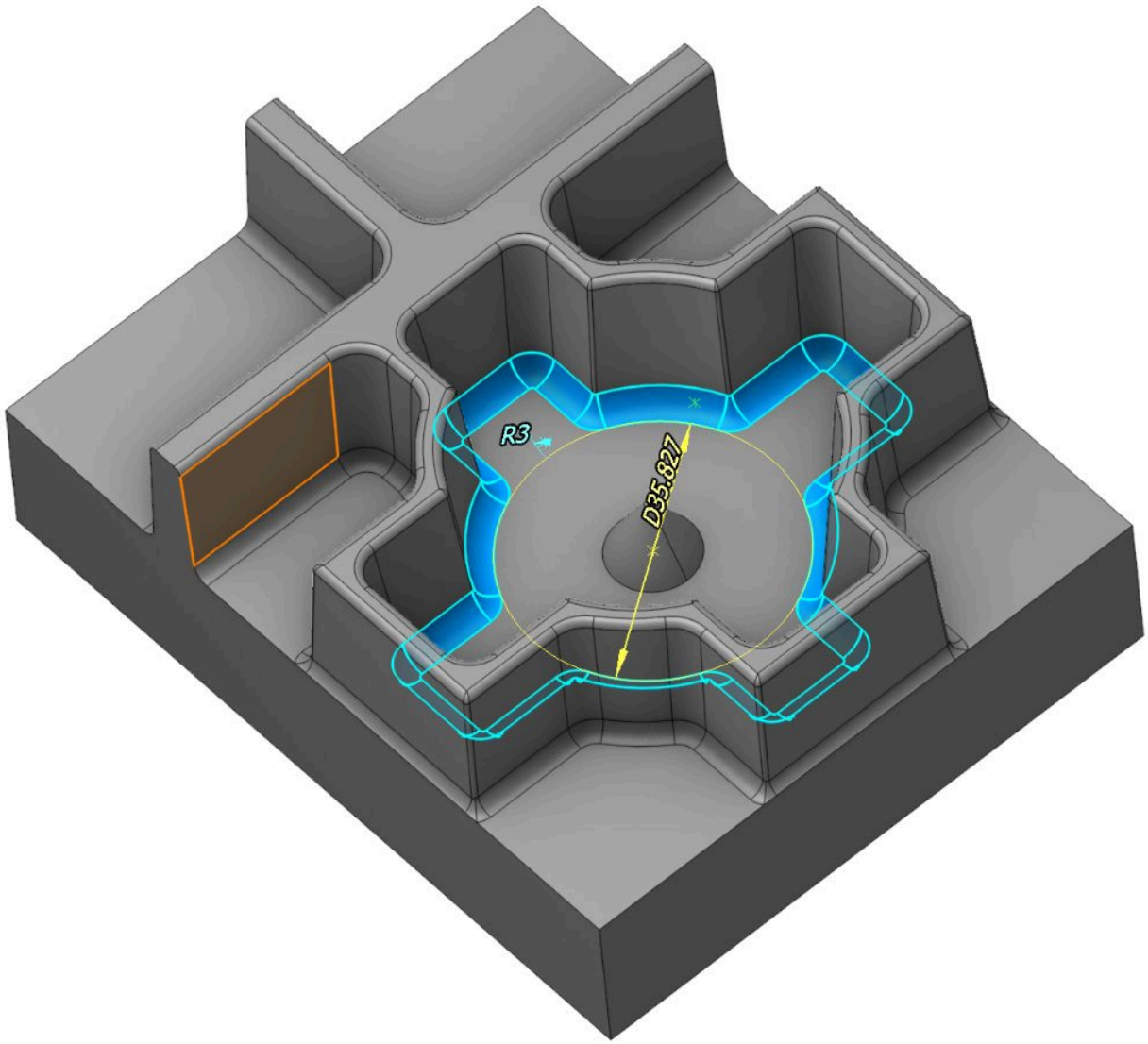
▼ Click here to expand..

In addition to the selected elements, a set of context filters is available for more flexible use of the functionality.

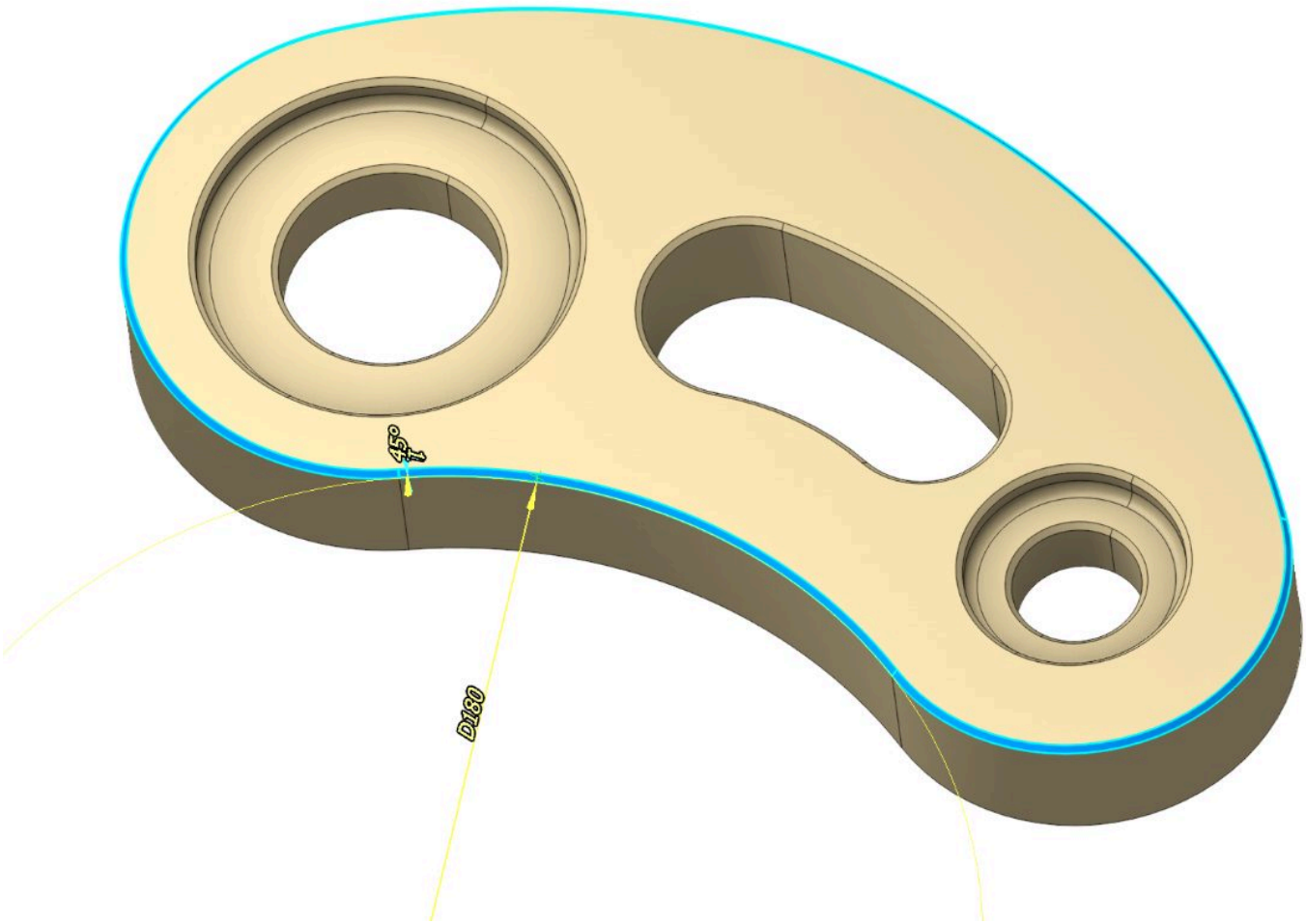


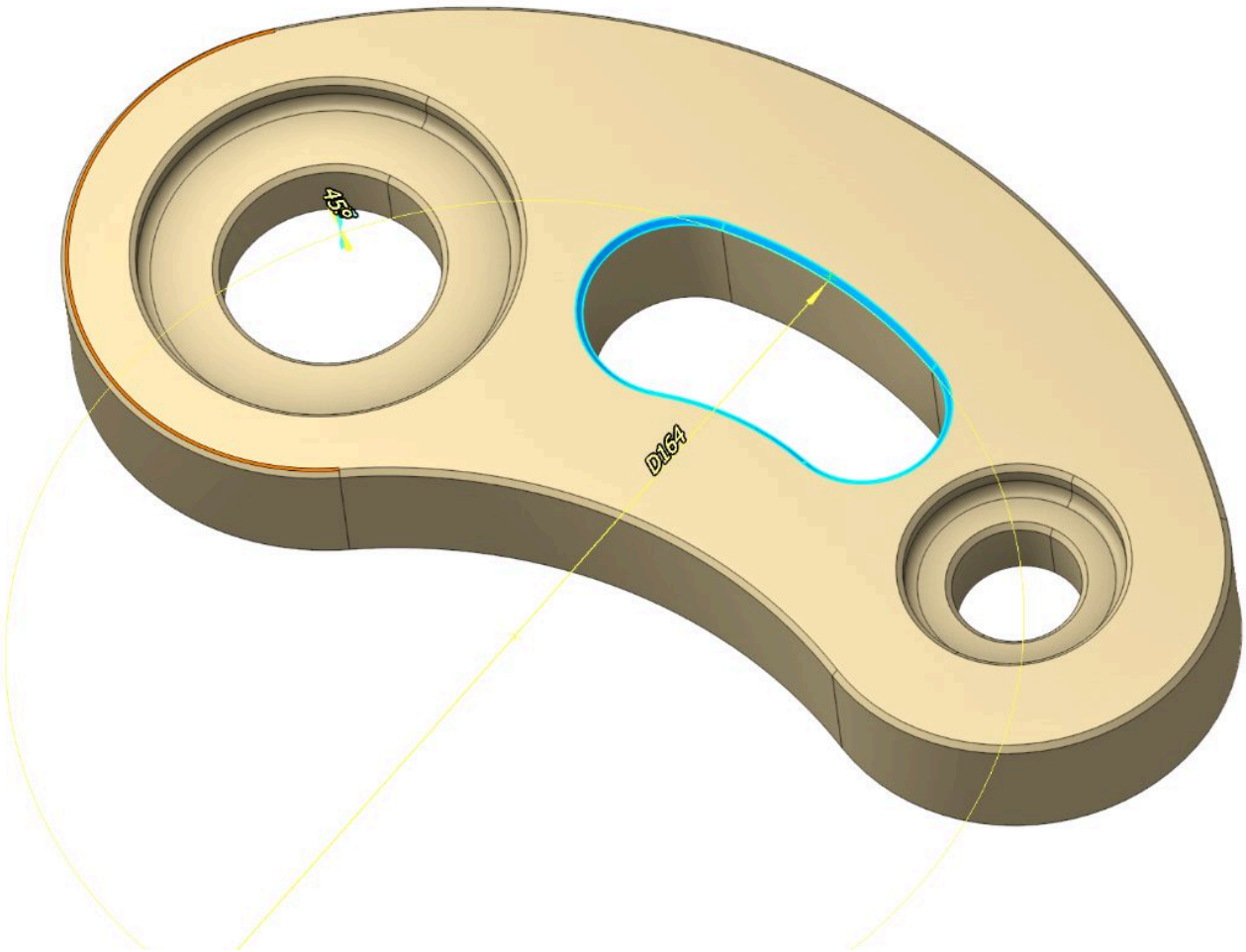
Basic geometric elements of selection:

- Fillet

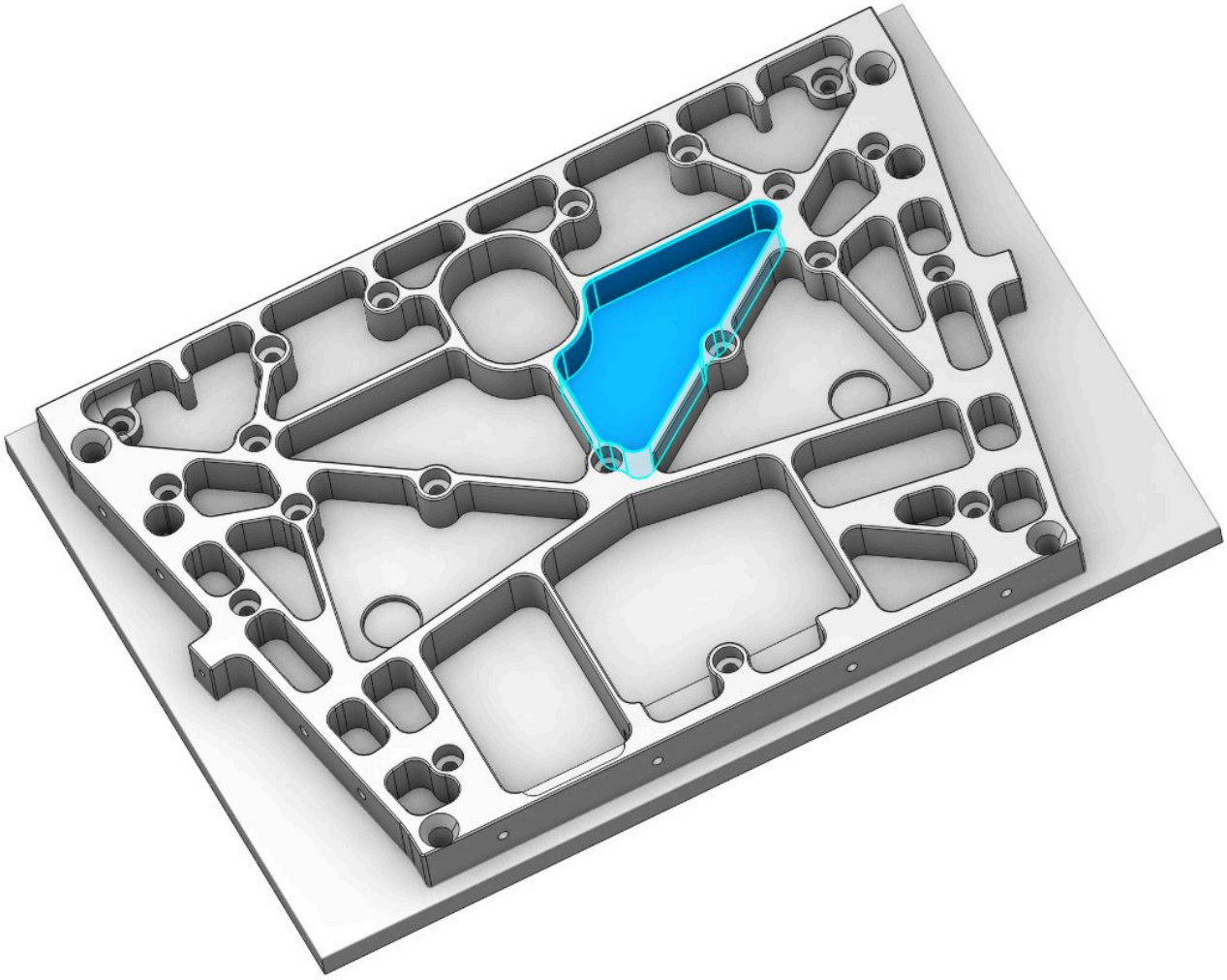


- Chamfer

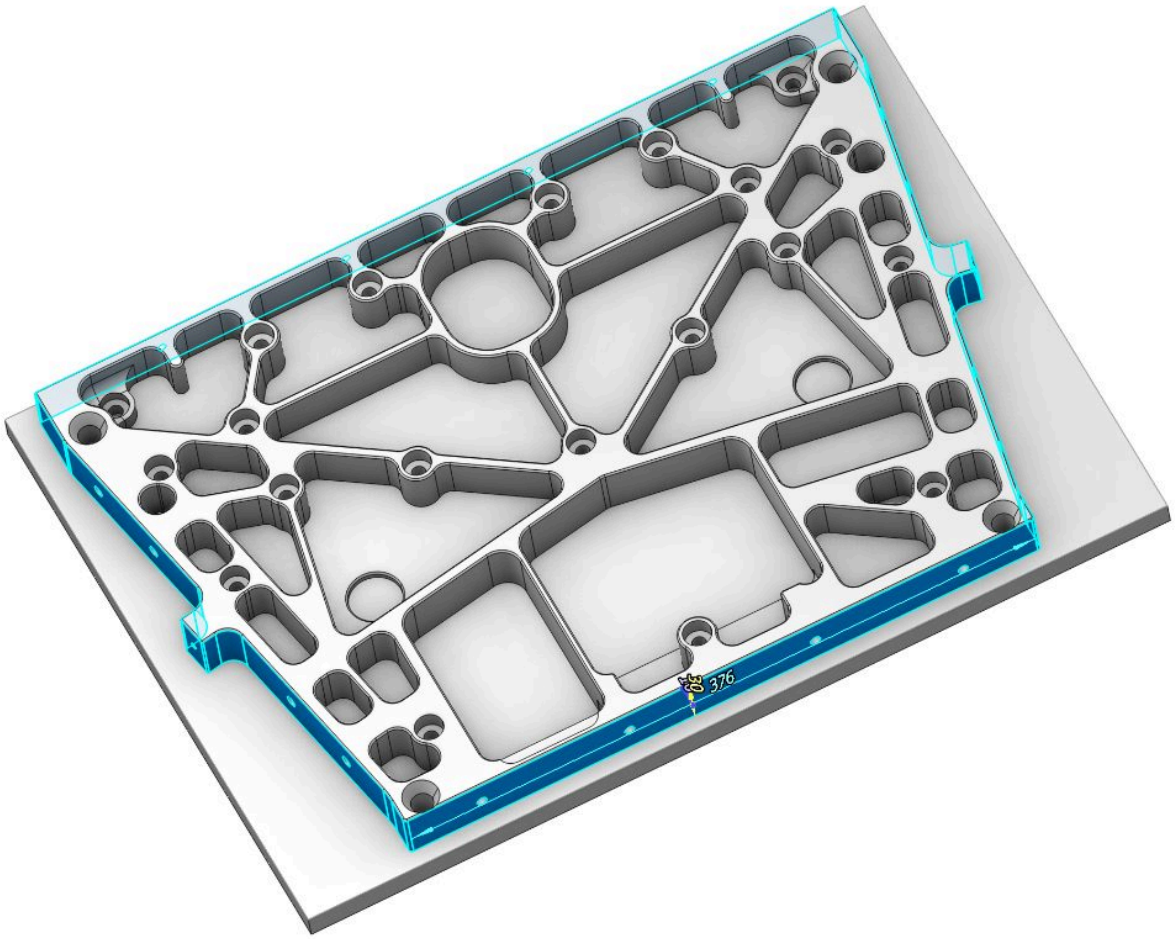

45°
D160



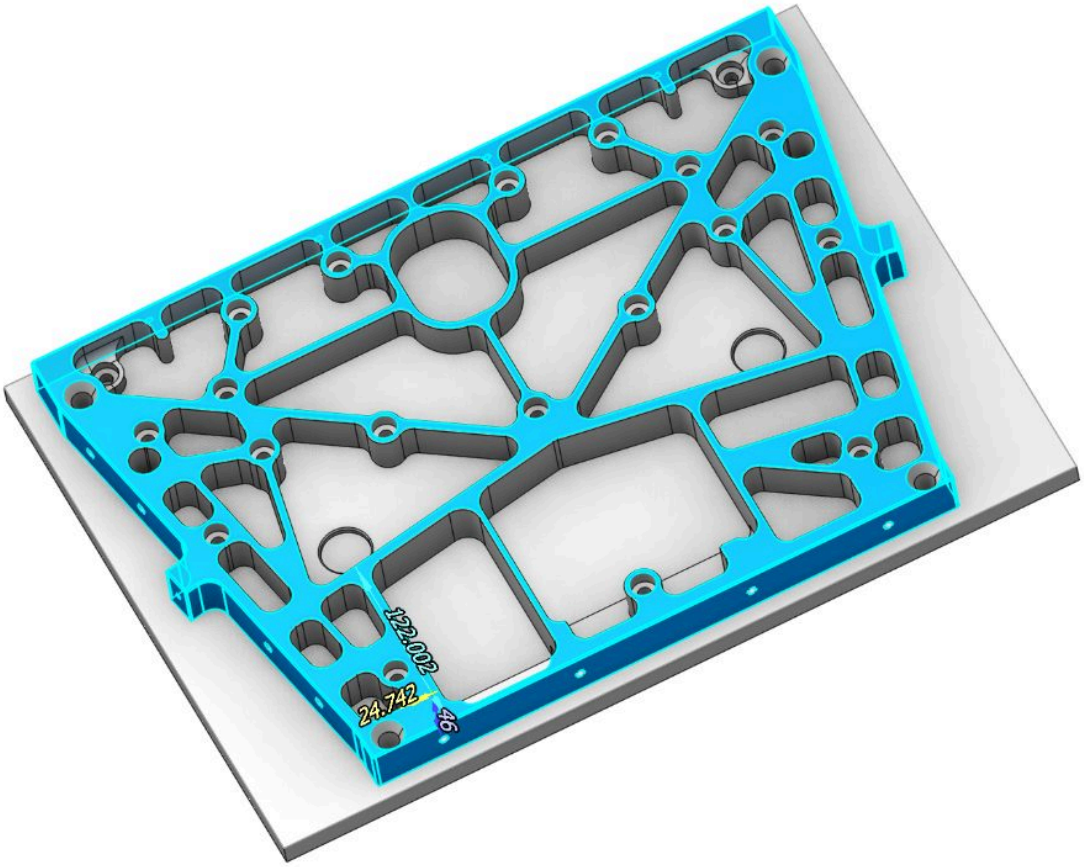
- Pocket (You need to select the bottom of the pocket)



- Side

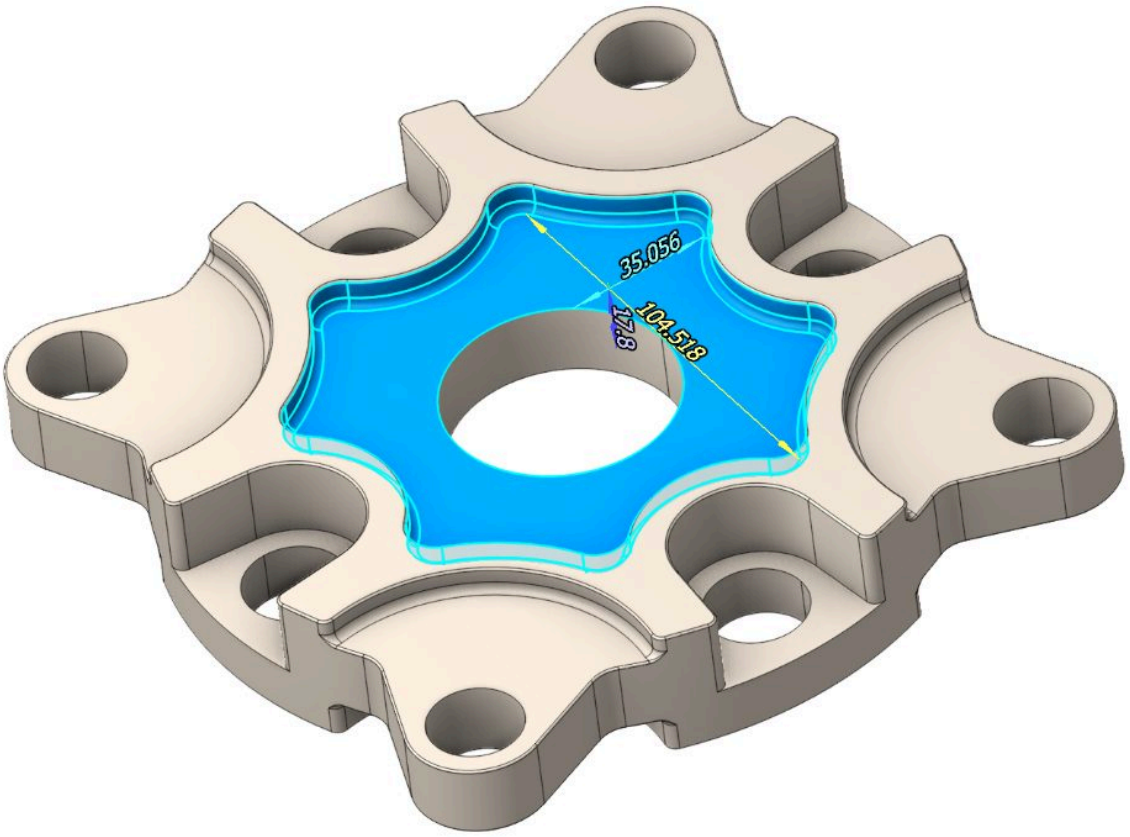


- Boss (You need to select the top plane of the boss)

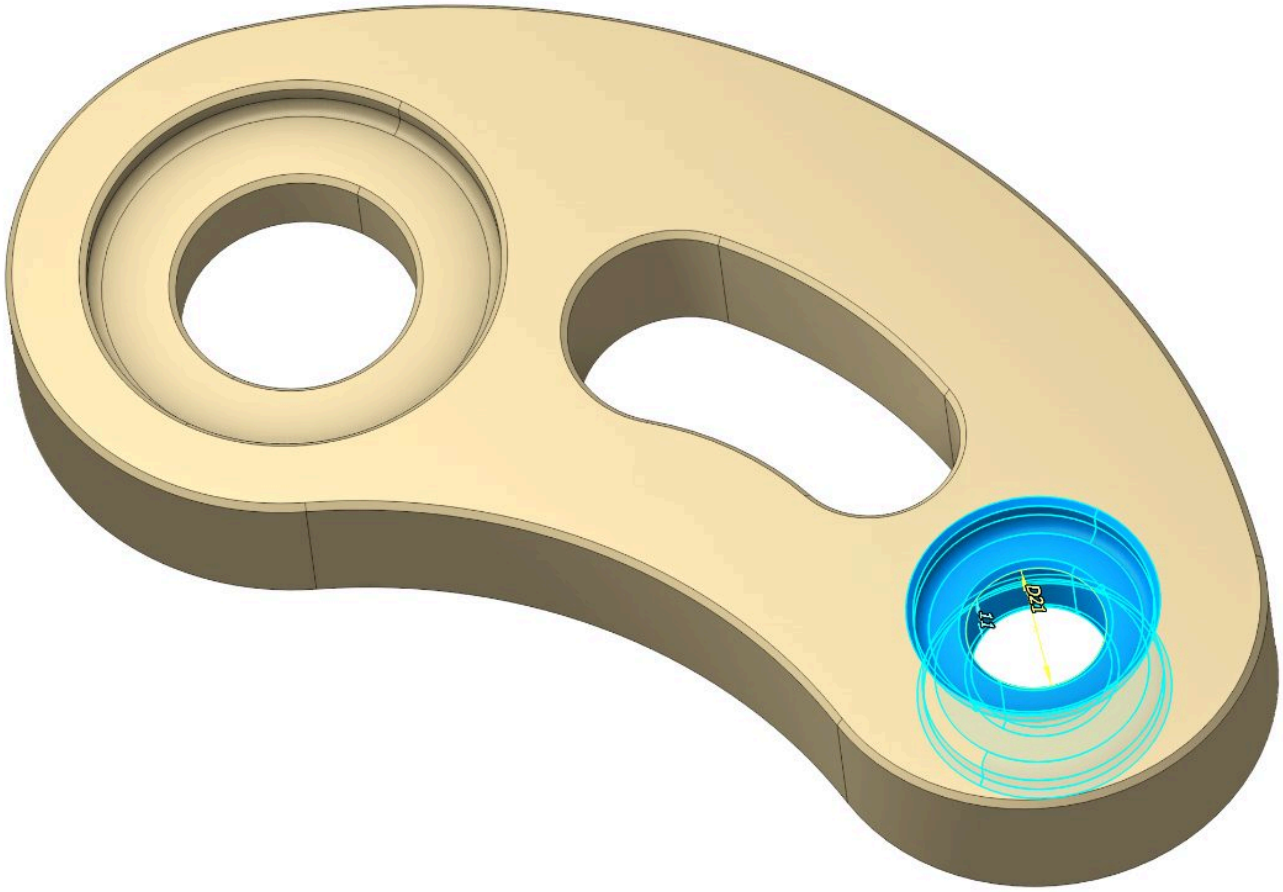


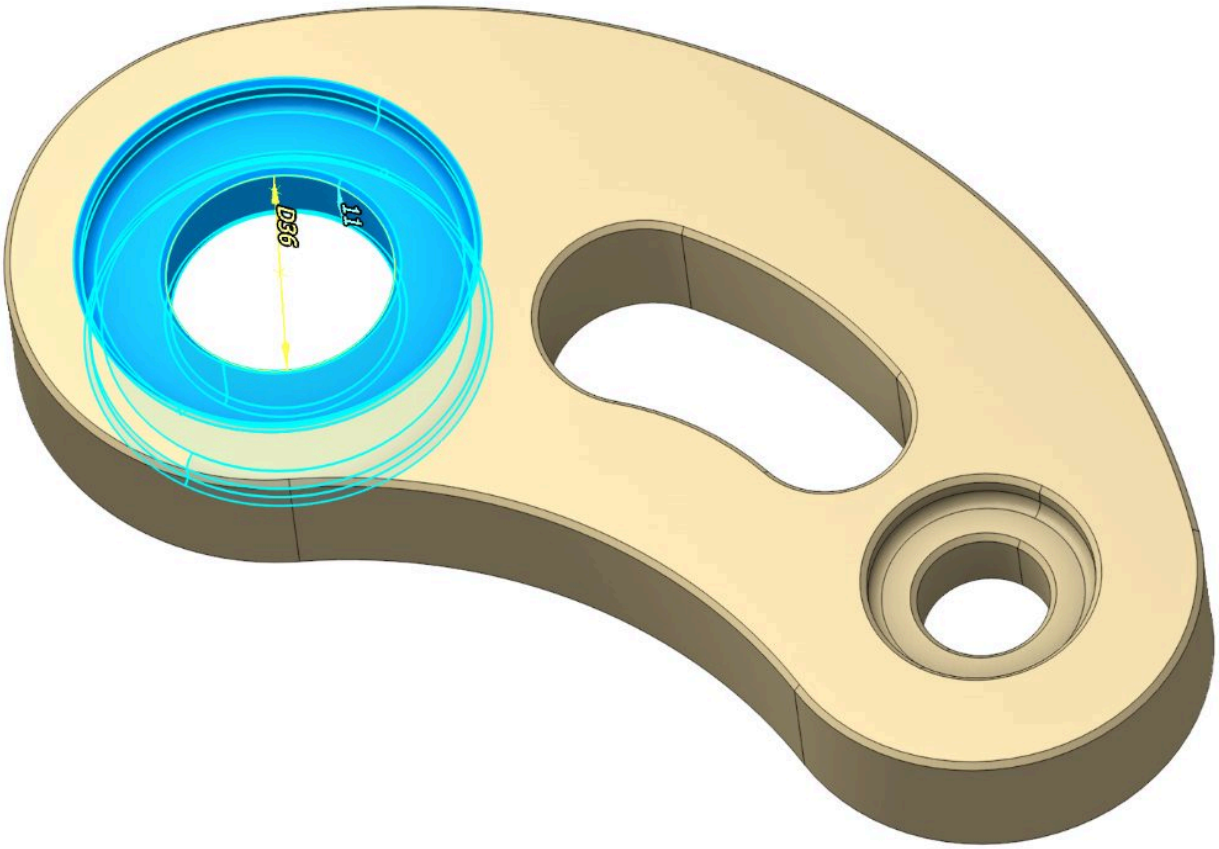
- Smooth (Smoothly mating surfaces)


136.749
210.856
17.00



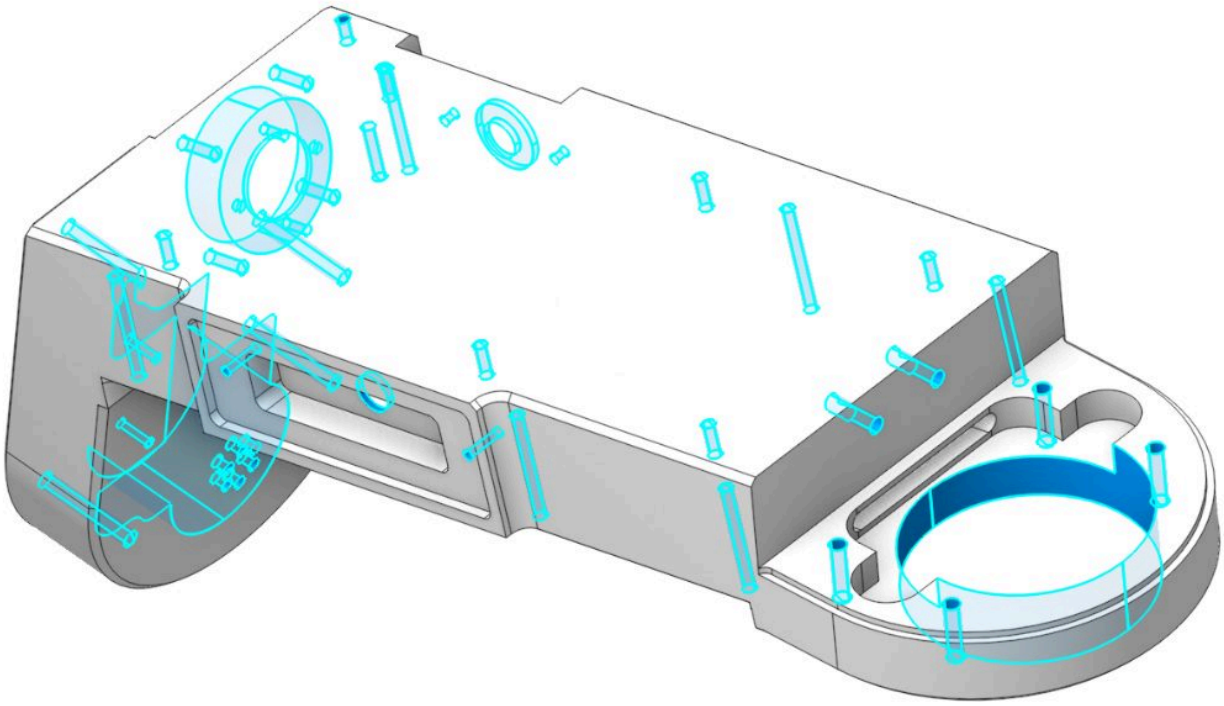
- Hole


D21
11

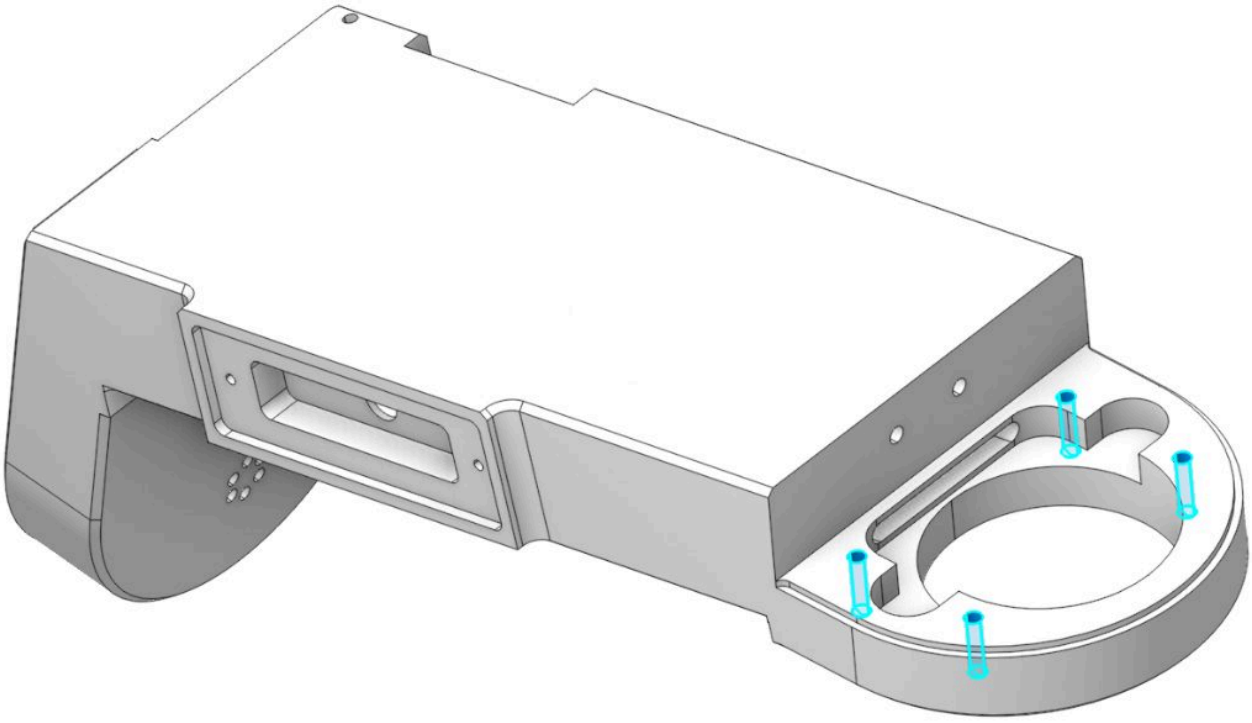


When choosing holes, there are several filters:

- By shape.

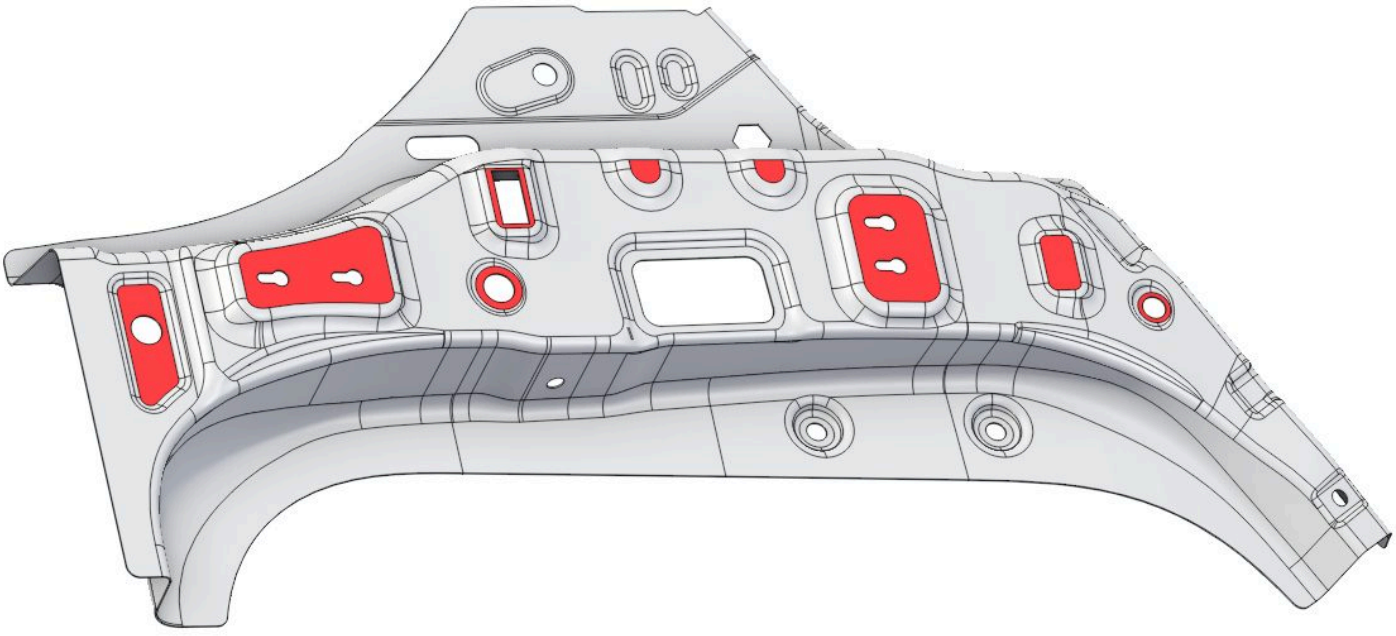


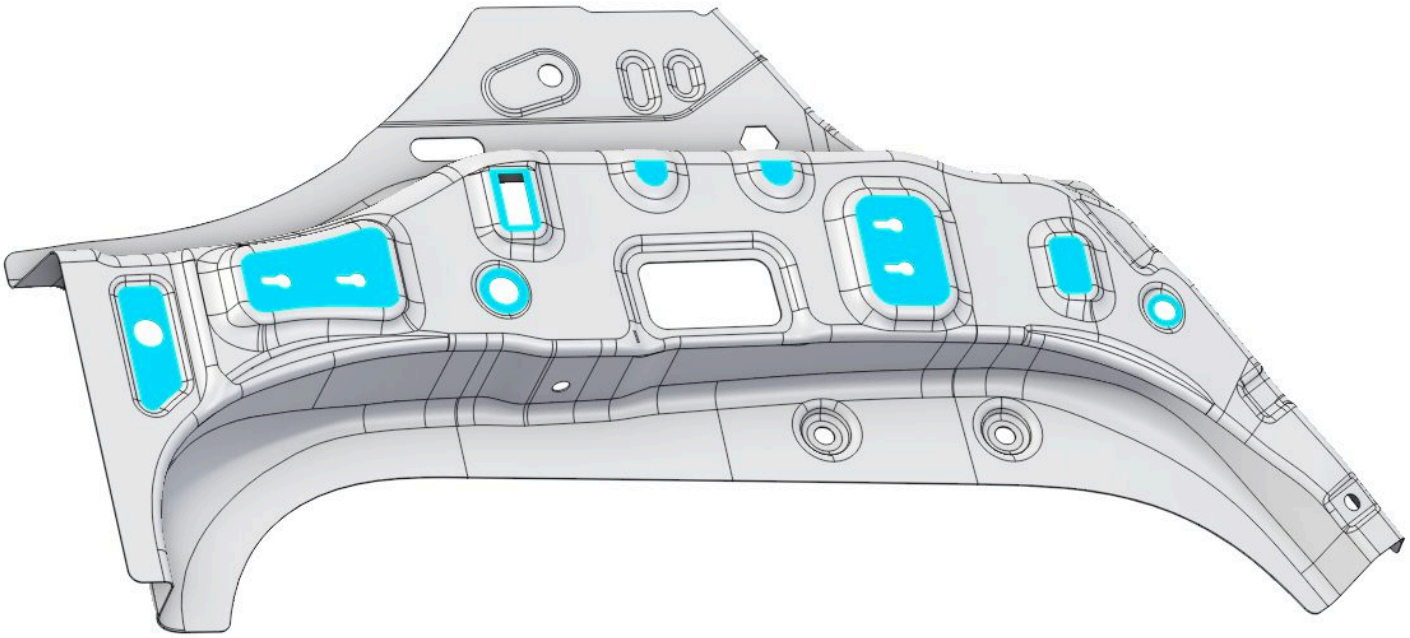
- Plane and shape.



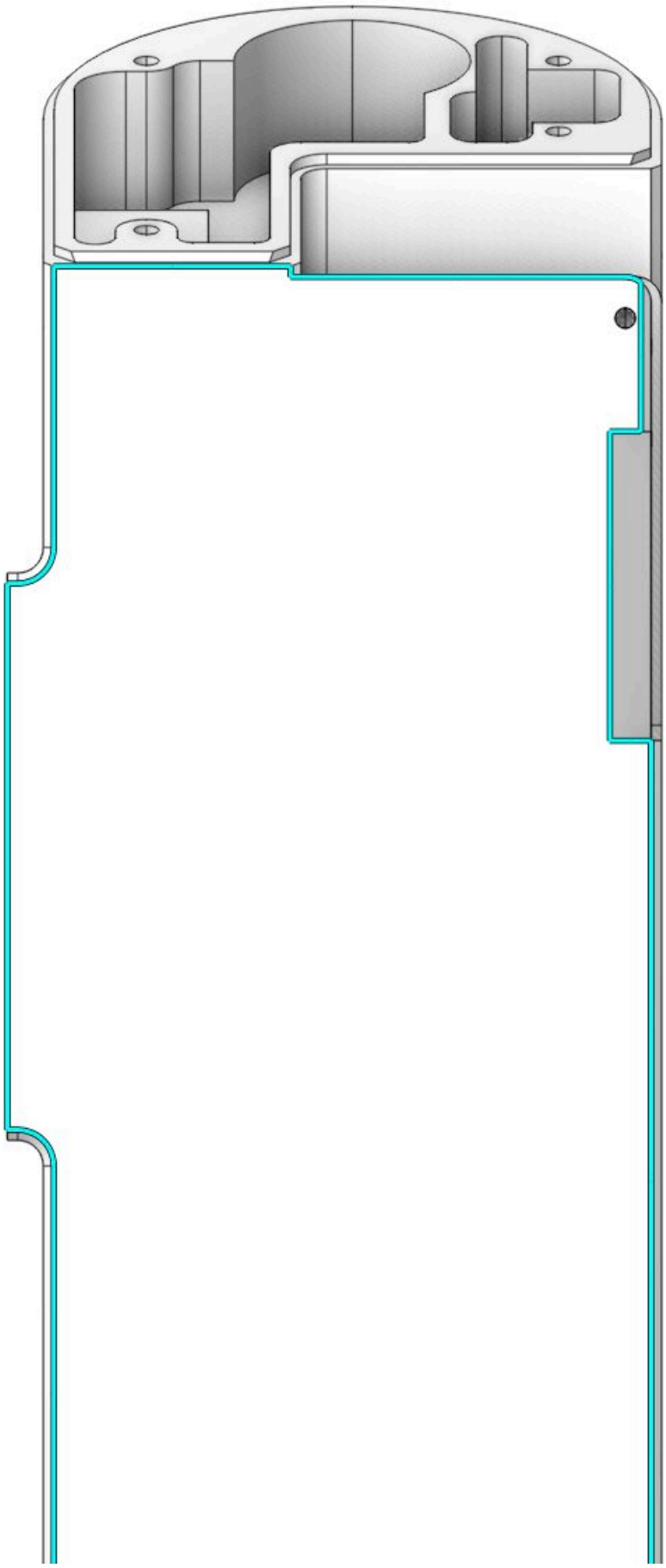
When selecting surfaces, you can use the following filter:

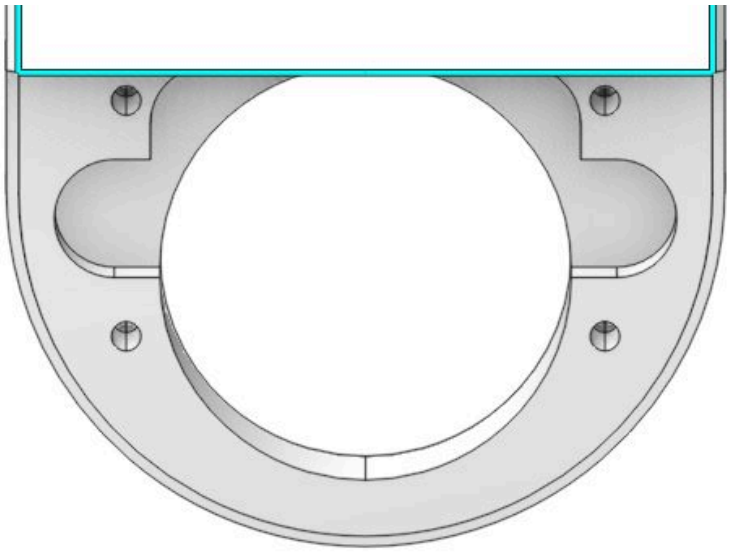
- By Color. Select a surface, click Propagate, and enable By Color. The system selects all surfaces in the 3D model with the same color.



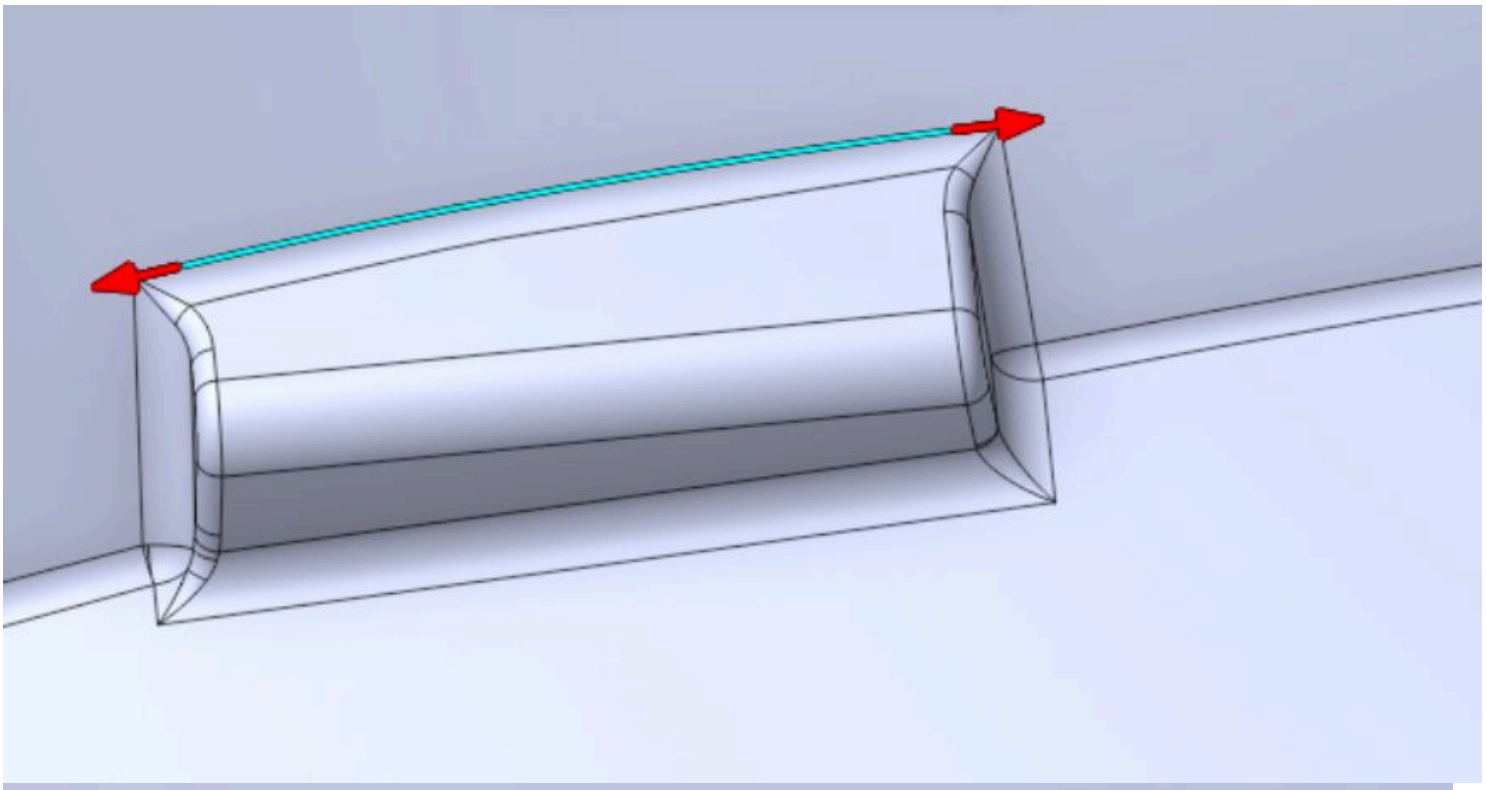


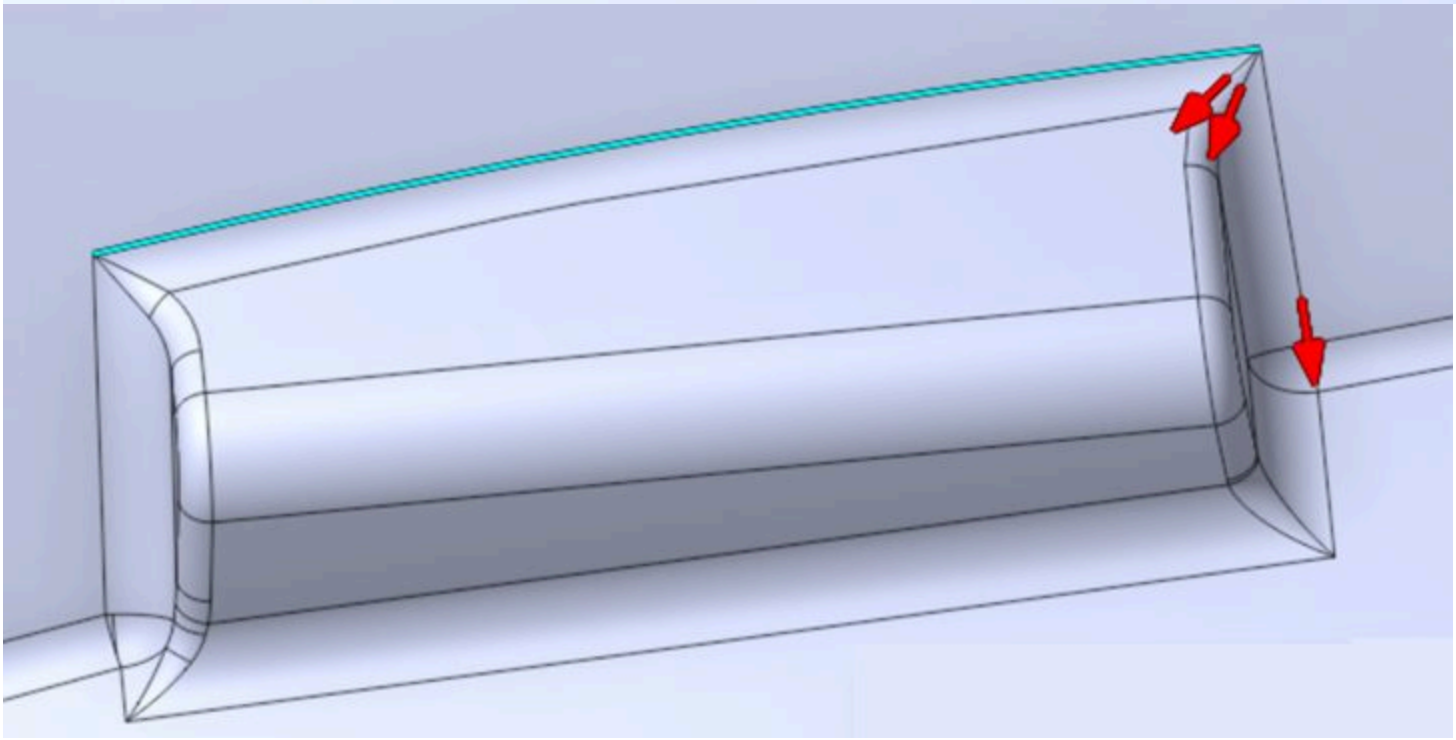
- Outer loop. This is a function that allows for the automatic selection of external edges belonging to a chosen surface.

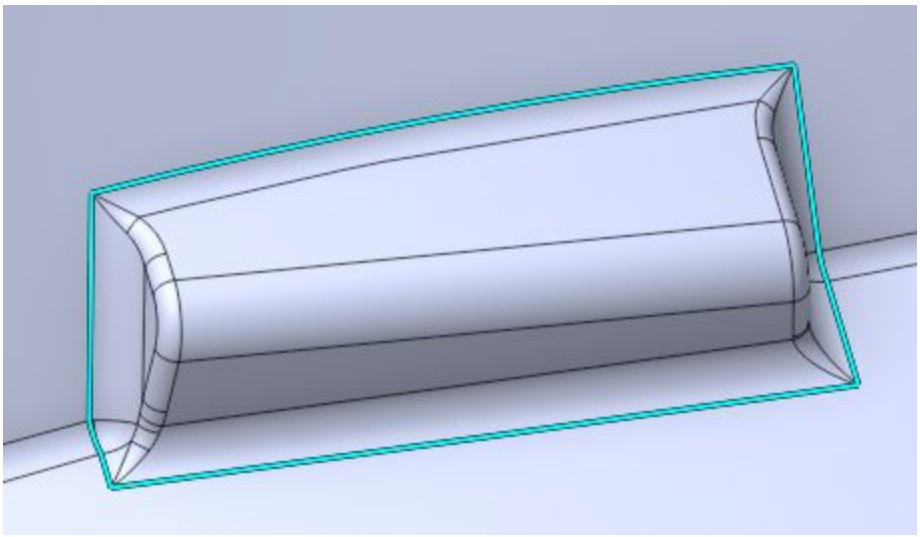




- Custom Loop. It lets the operator control, edge by edge, the direction the loop takes at every branch point (fork) it meets while propagating along the model, using the ◀ / ▶ keys.



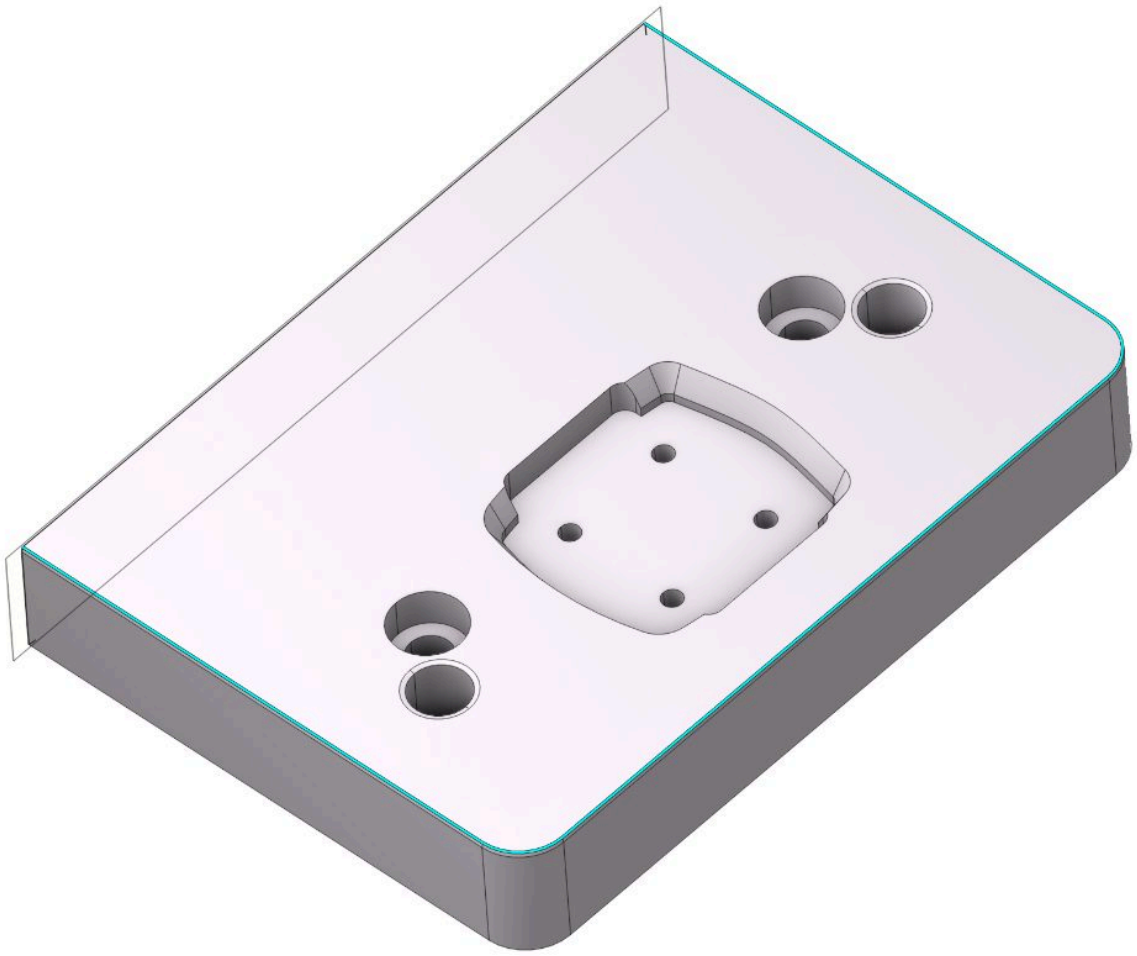




Right-click on the edge to activate the mode (Propagation)

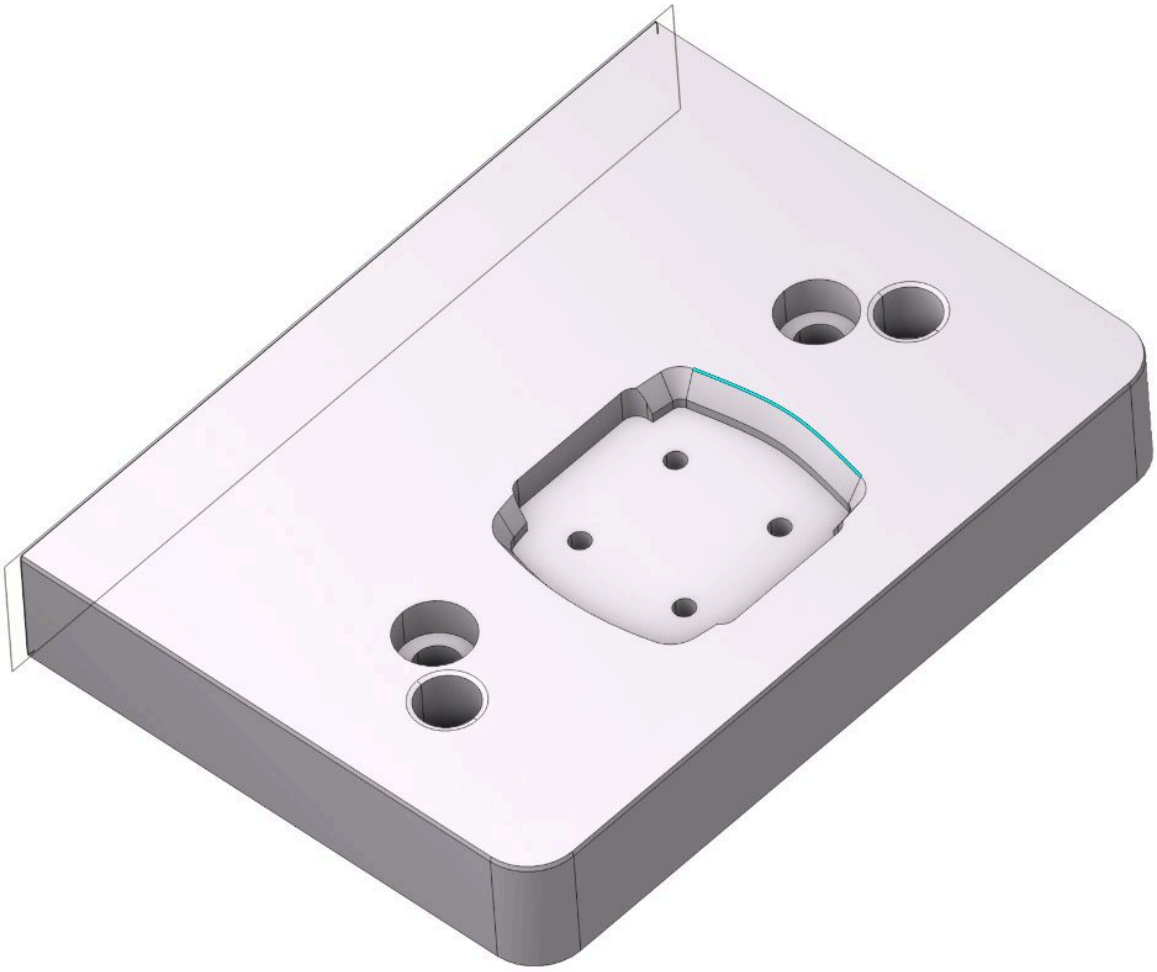
▼ Click here to expand..

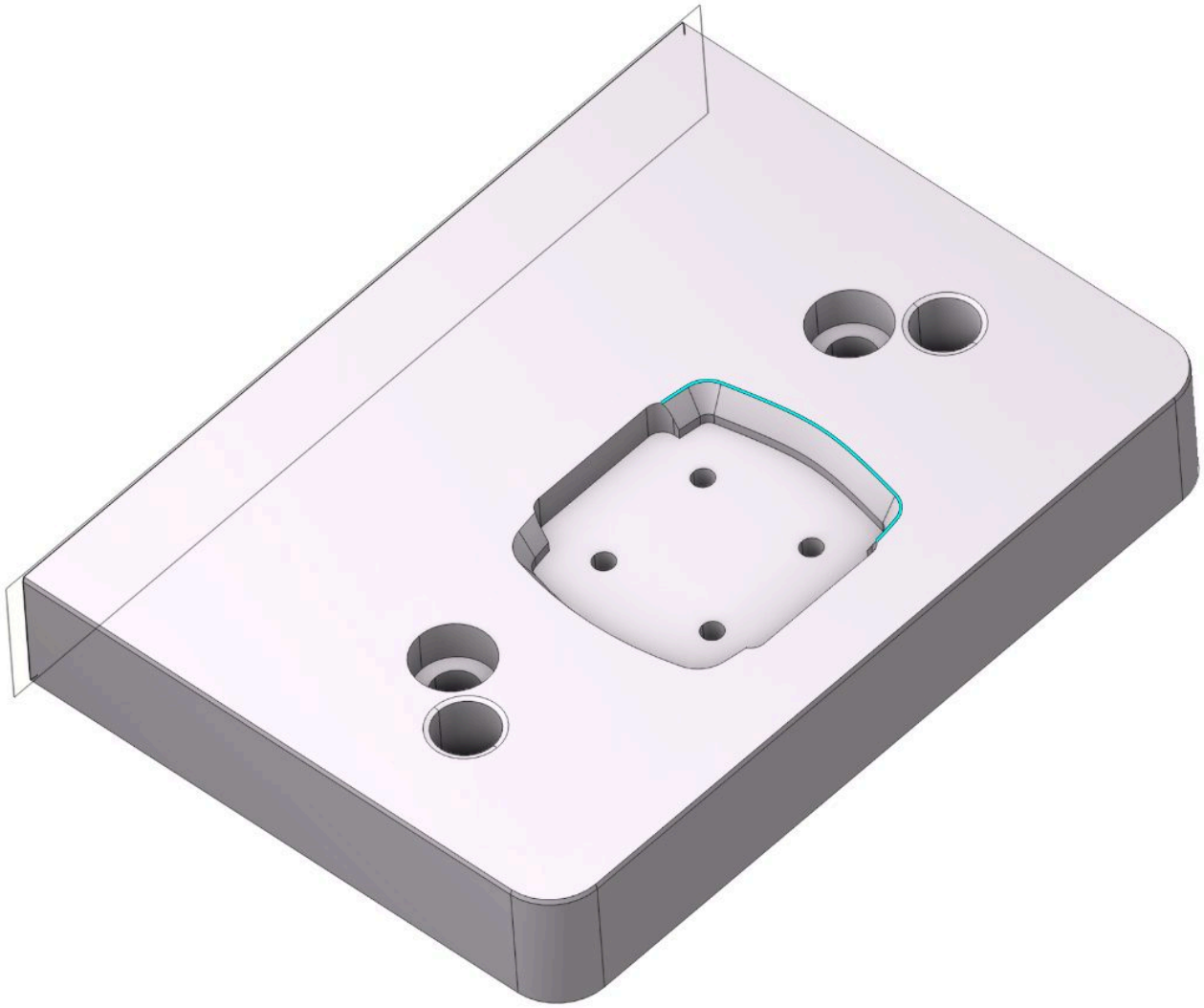
In addition to the selected elements, a set of context filters is available for more flexible use of the functionality.



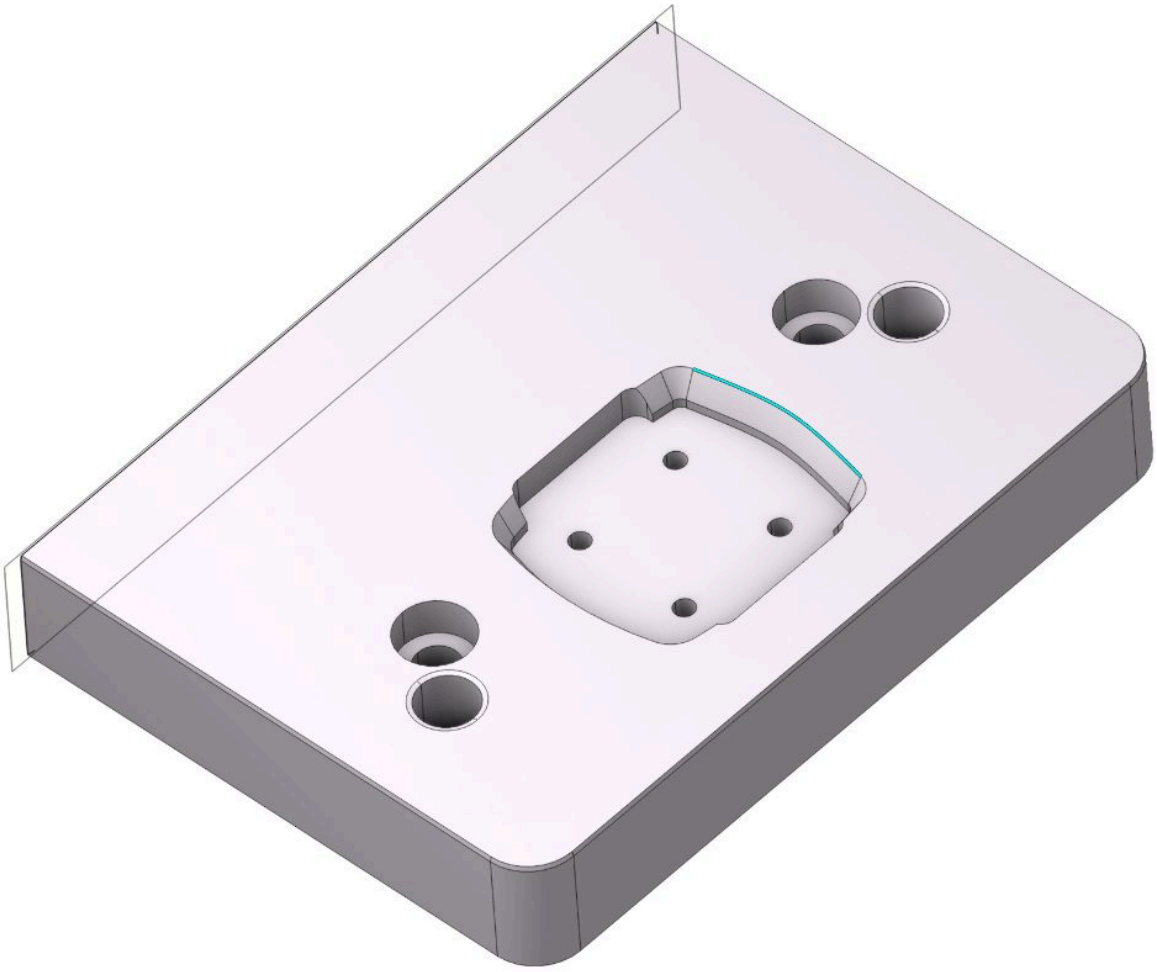
Basic geometric elements of selection:

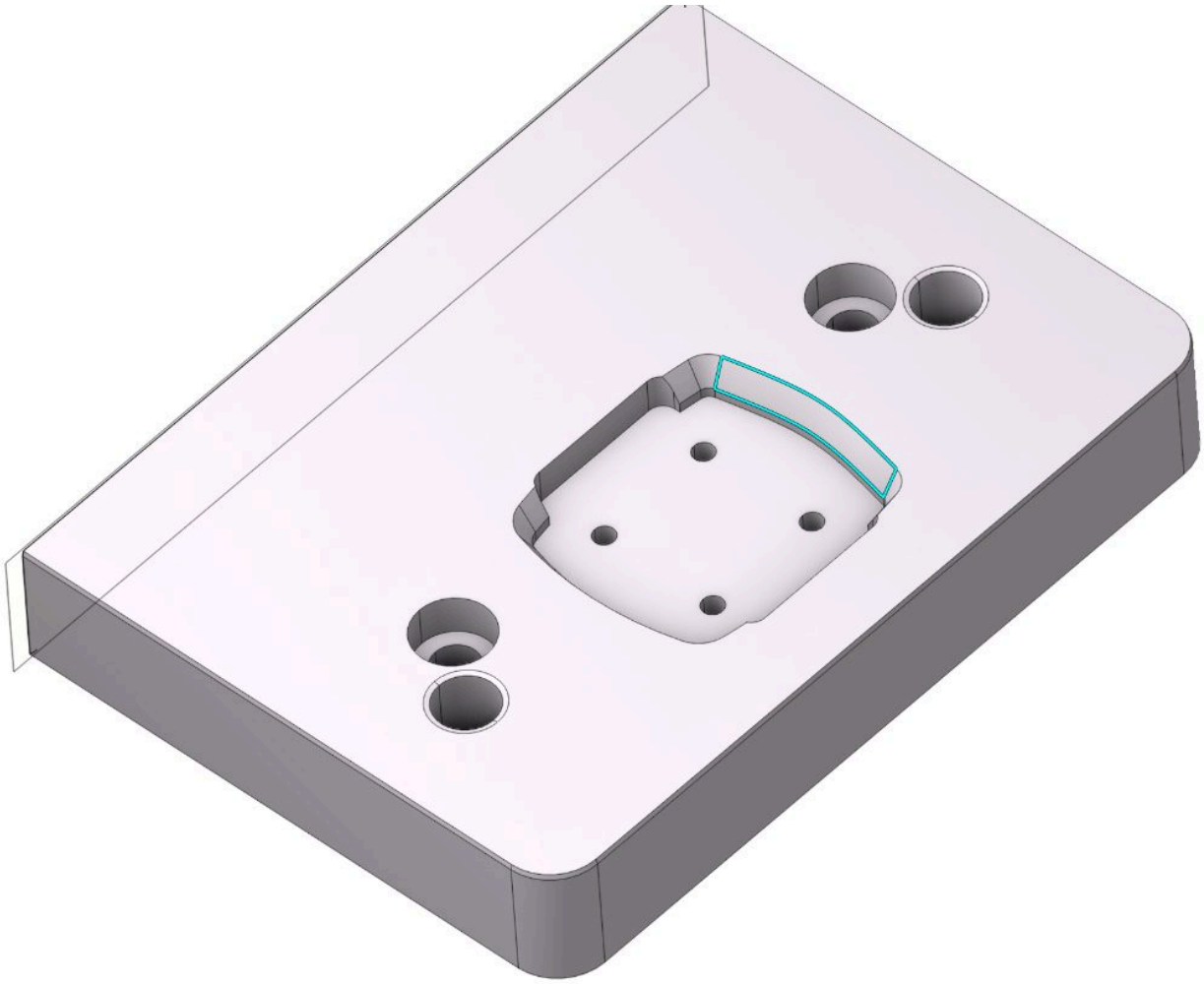
Tangent Edges. Feature designed to automatically identify and select edges that are in tangential contact with each other in a 3D model.



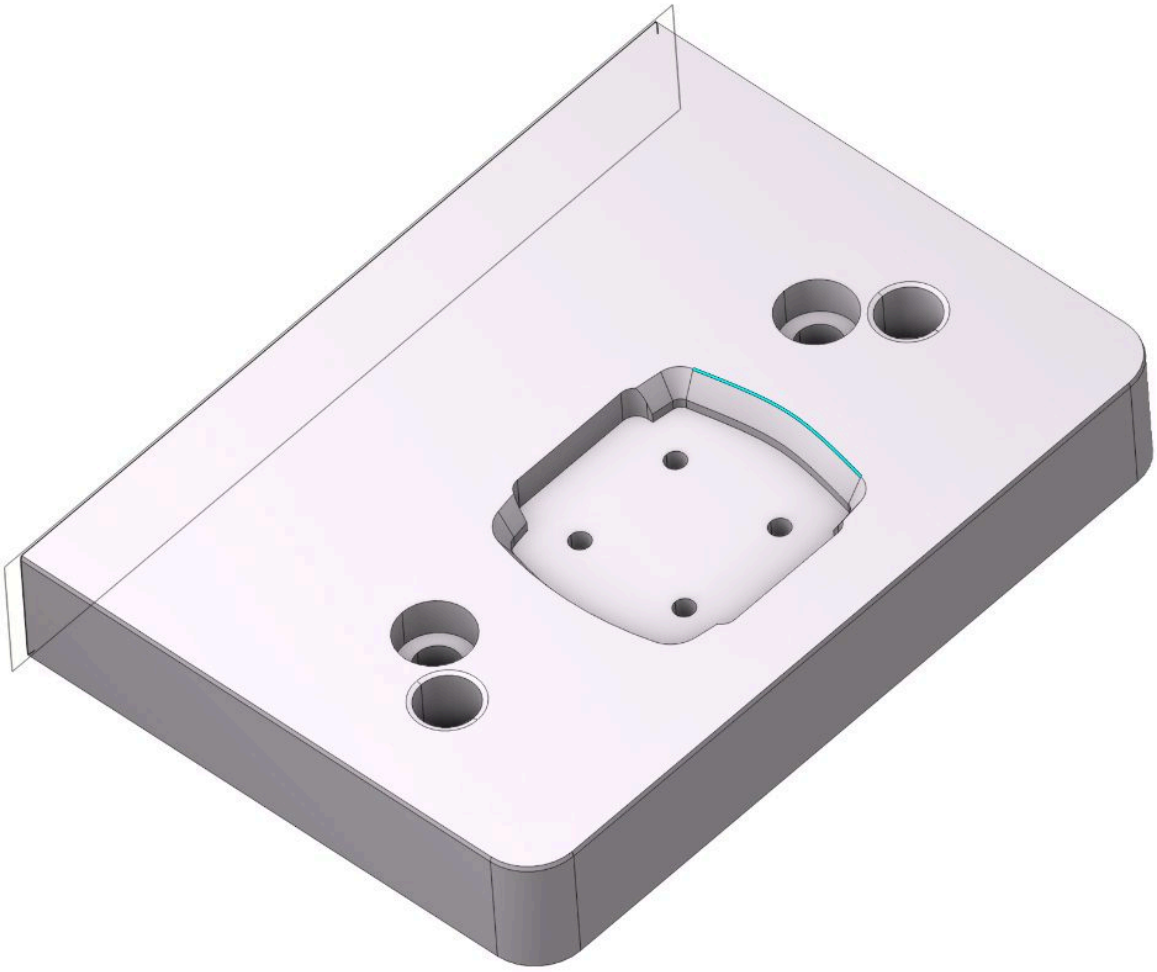


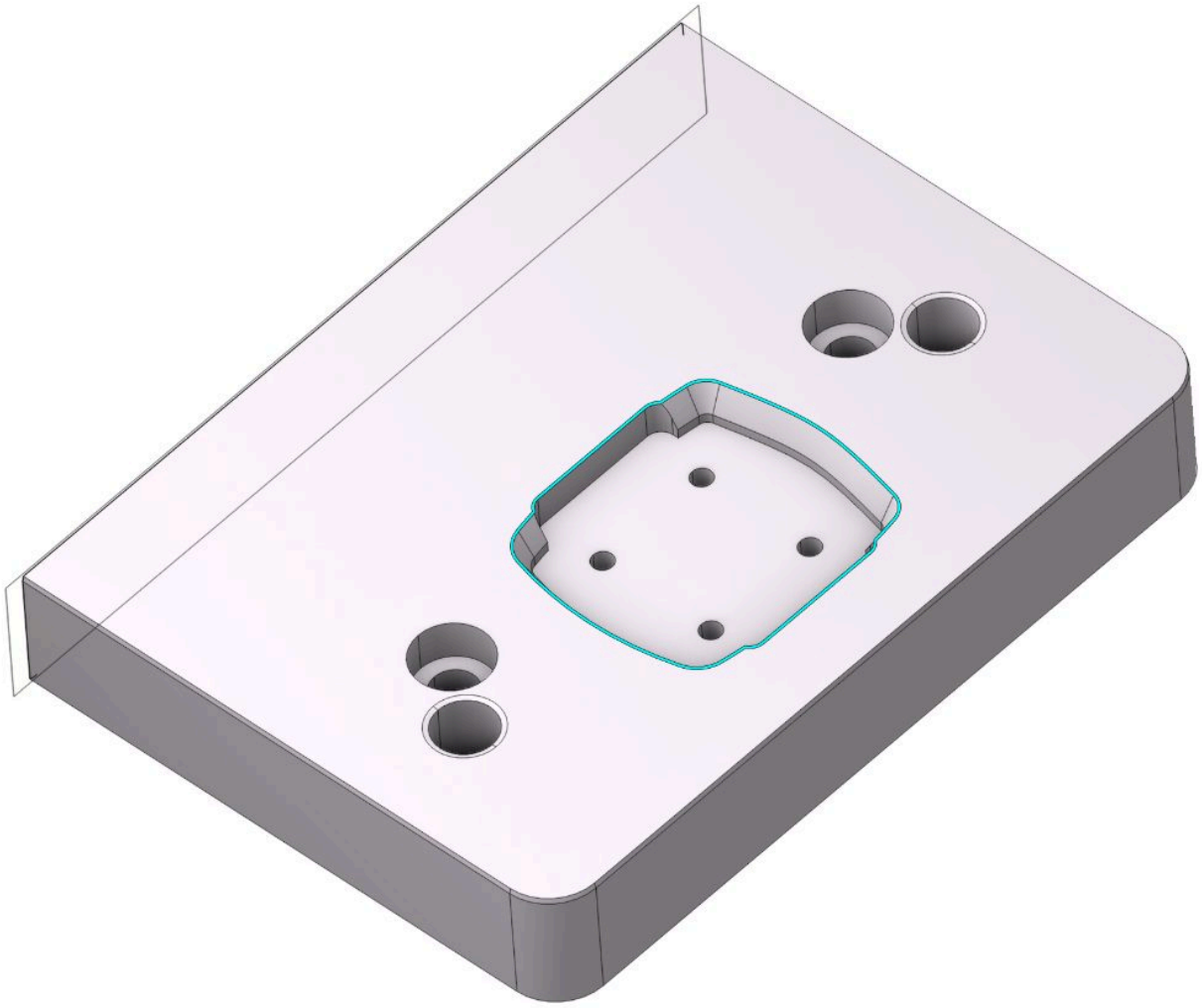
Loop1. Function for selecting consecutive (closed) chains of edges. It works according to the first option



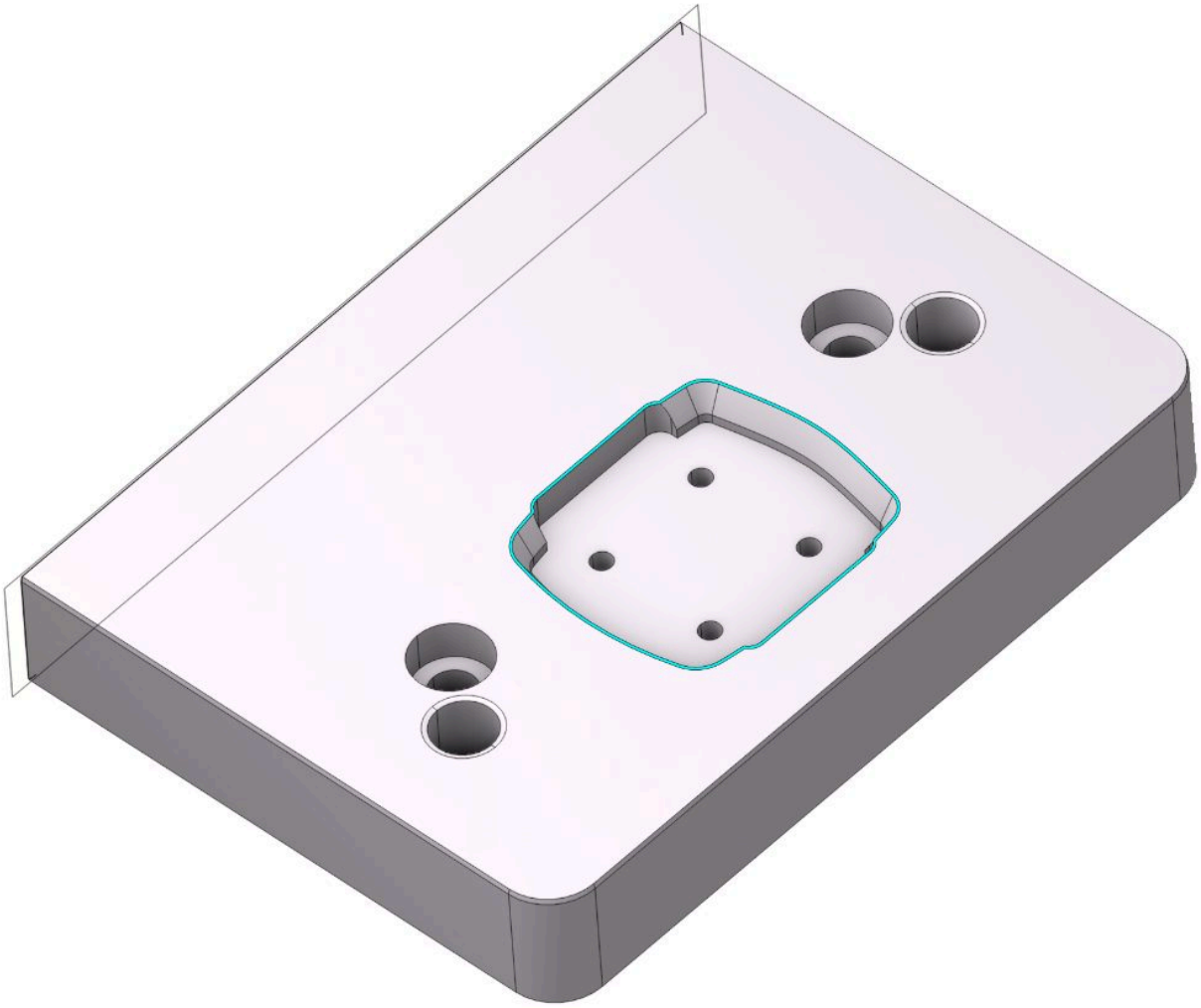


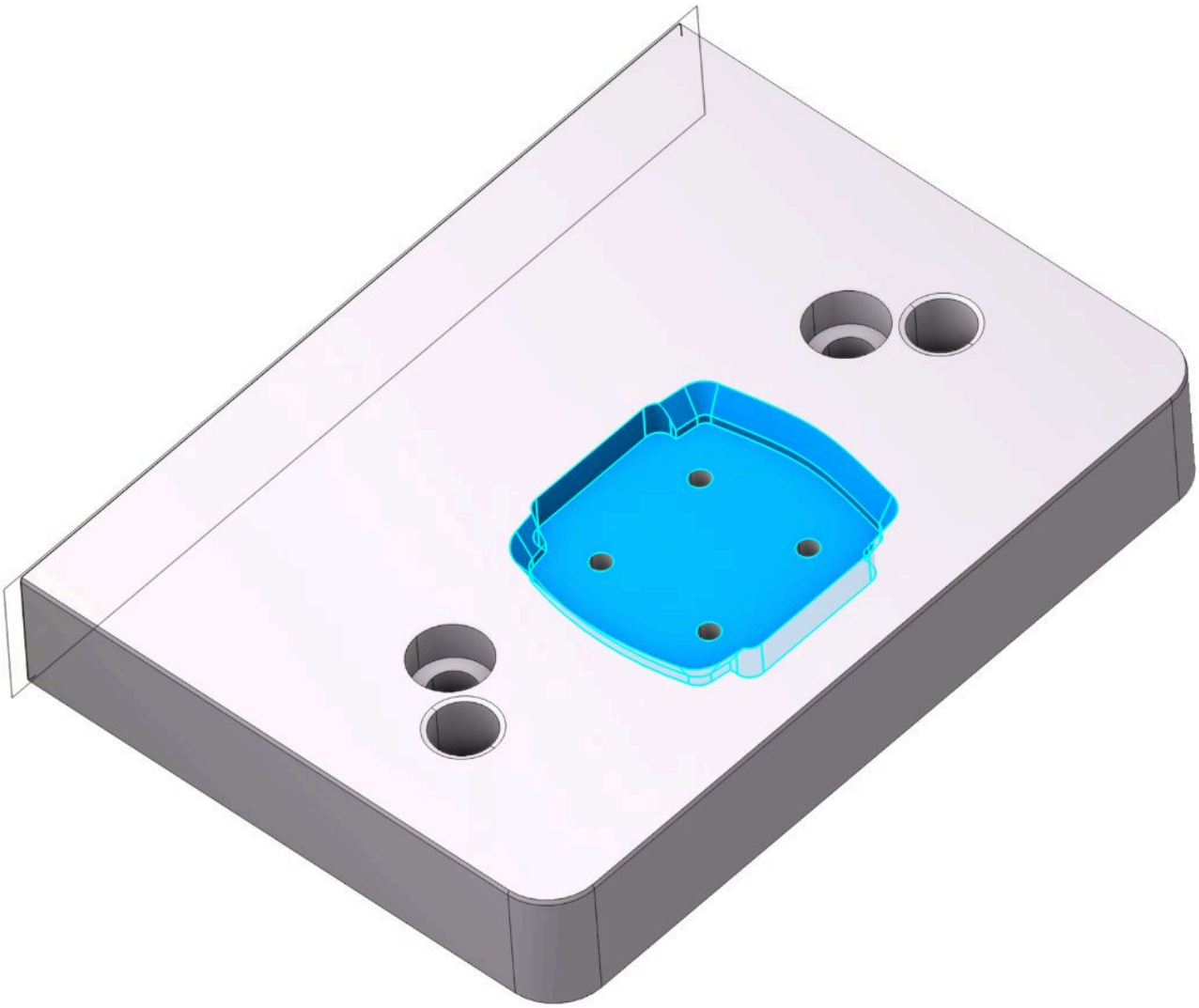
Loop2. Function for selecting consecutive (closed) chains of edges. It works according to the second option



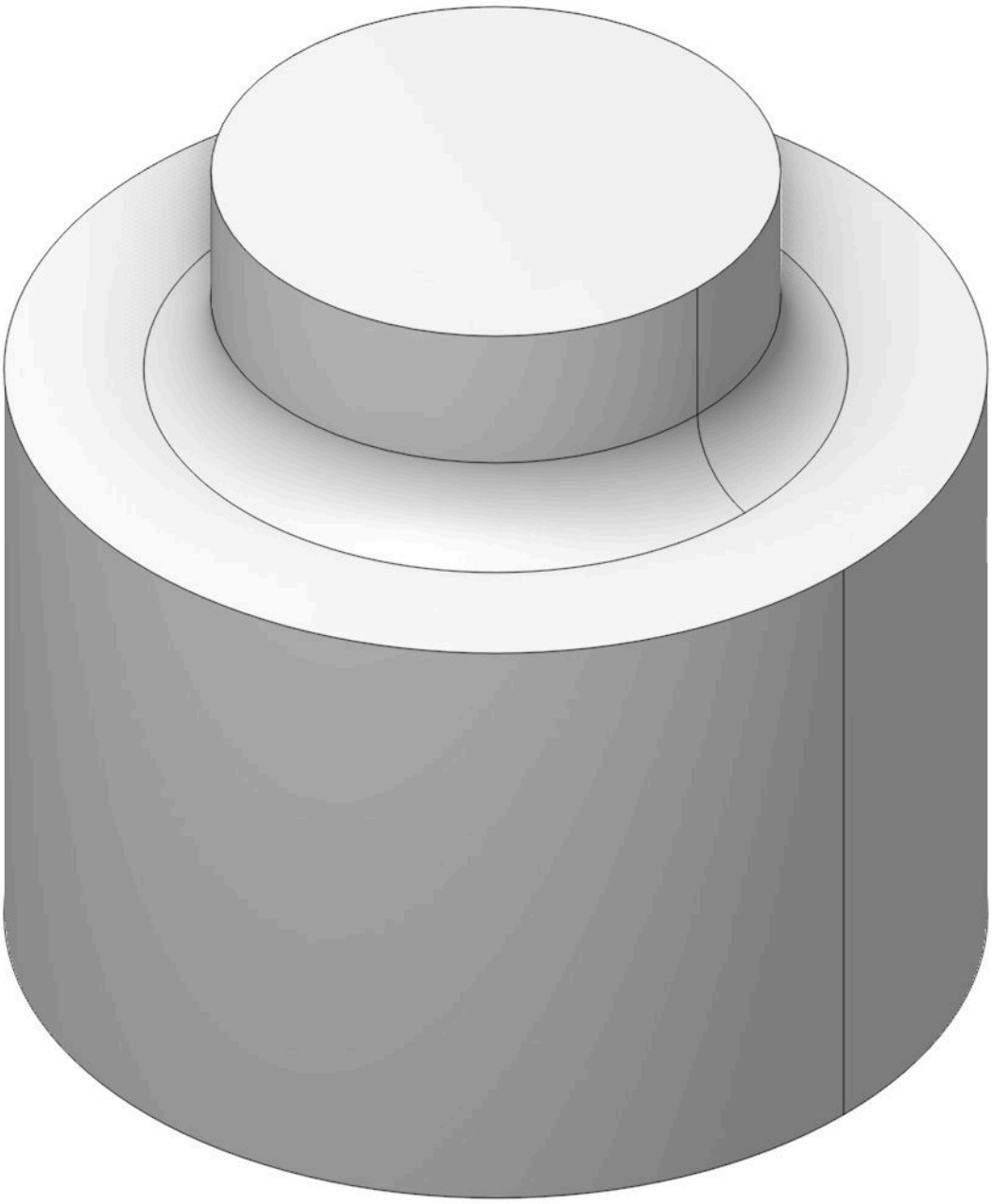


Enclosed Faces. Allows selecting surfaces within a closed contour





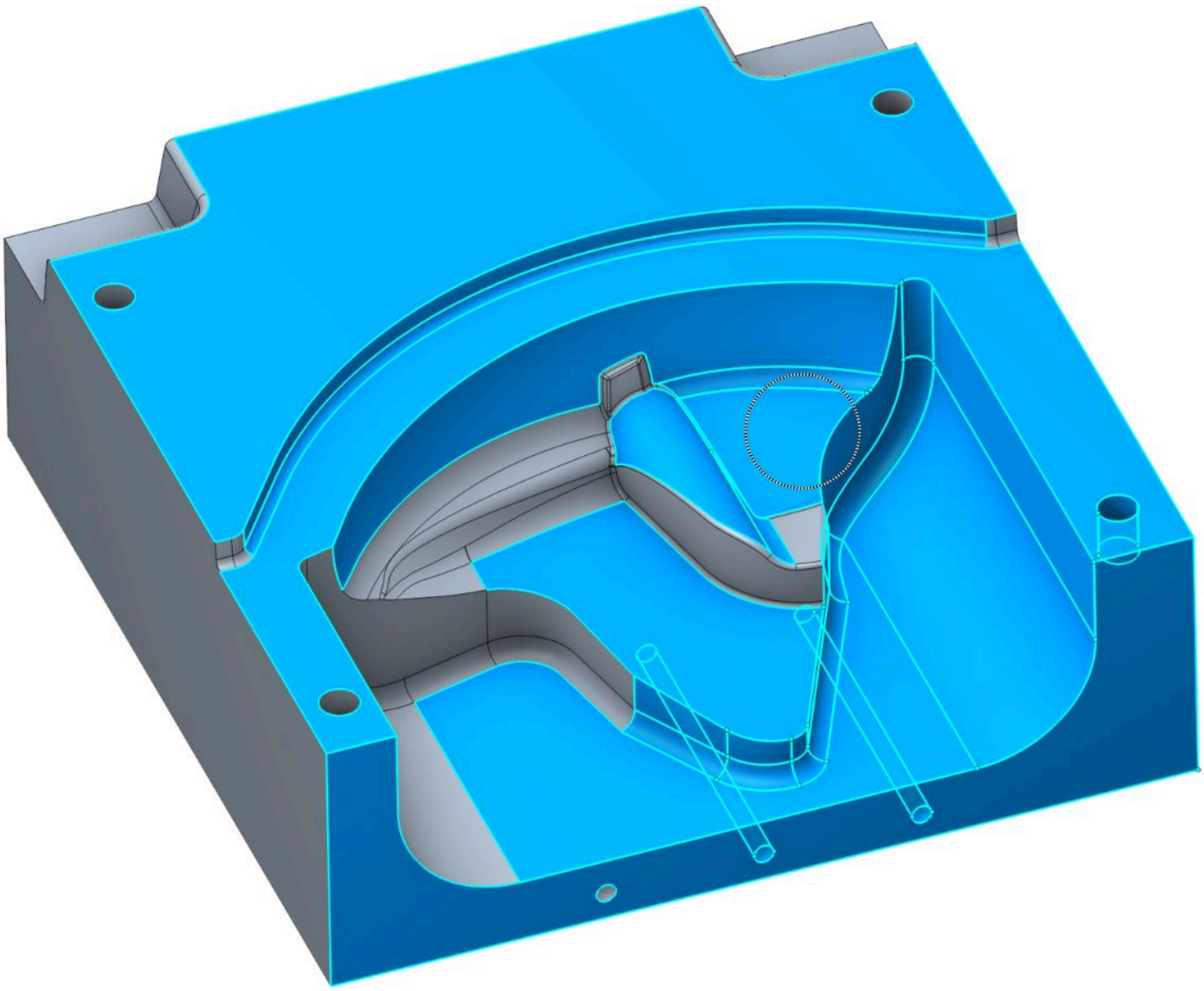
Outer Adjacent Faces. Allows you to select surfaces from outside a closed contour.



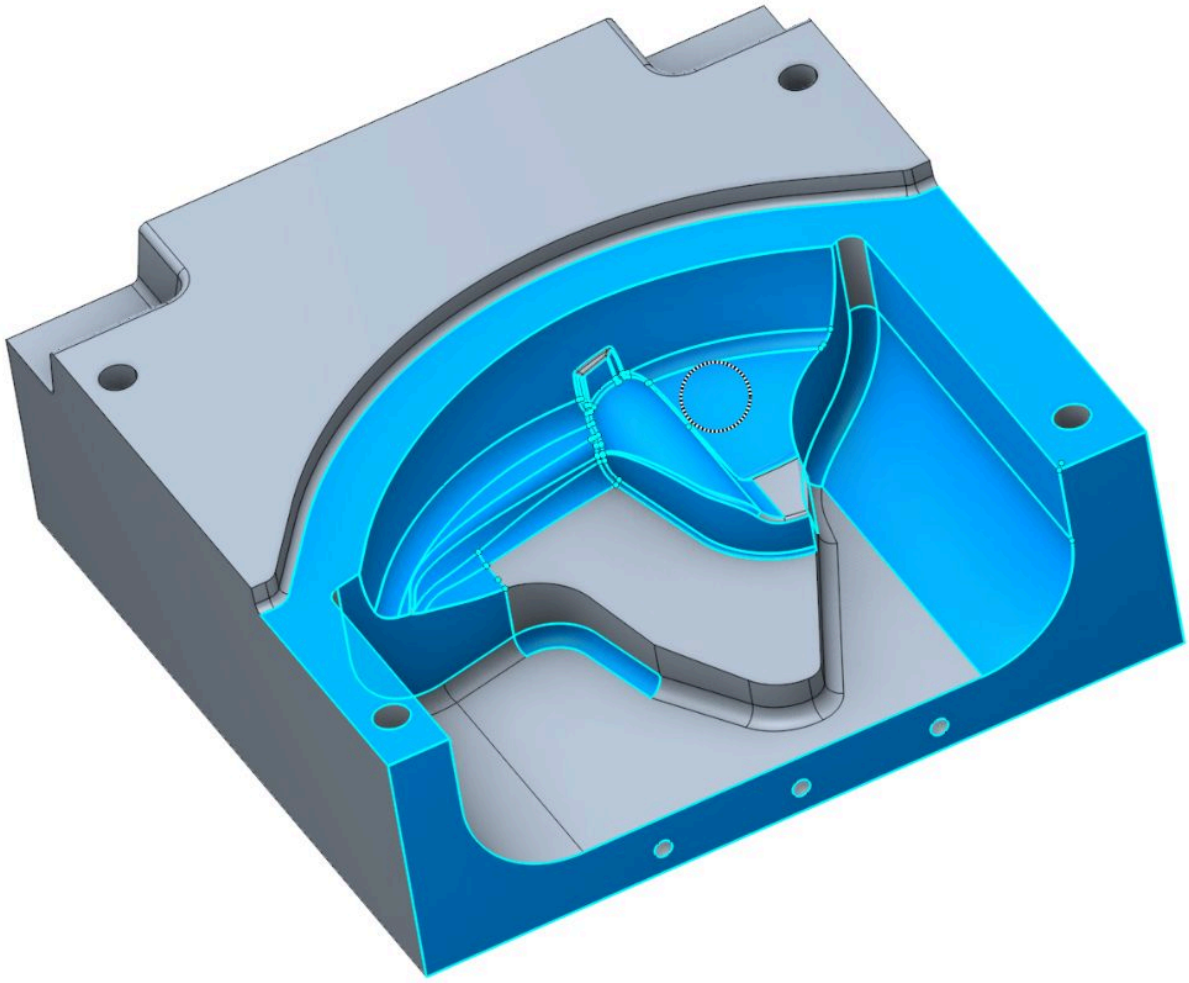


Left-click + Ctrl+Alt the surface to activate the mode (Surface selection via painting).

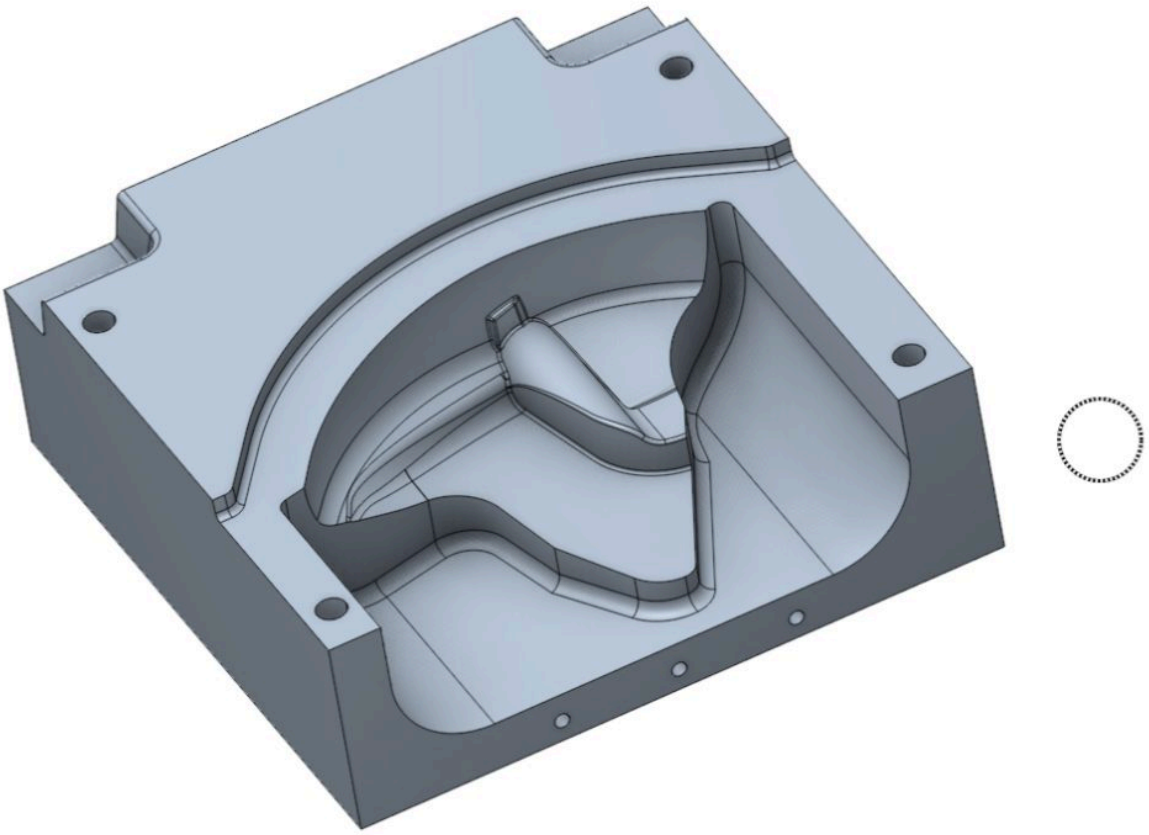
▼ Click here to expand..

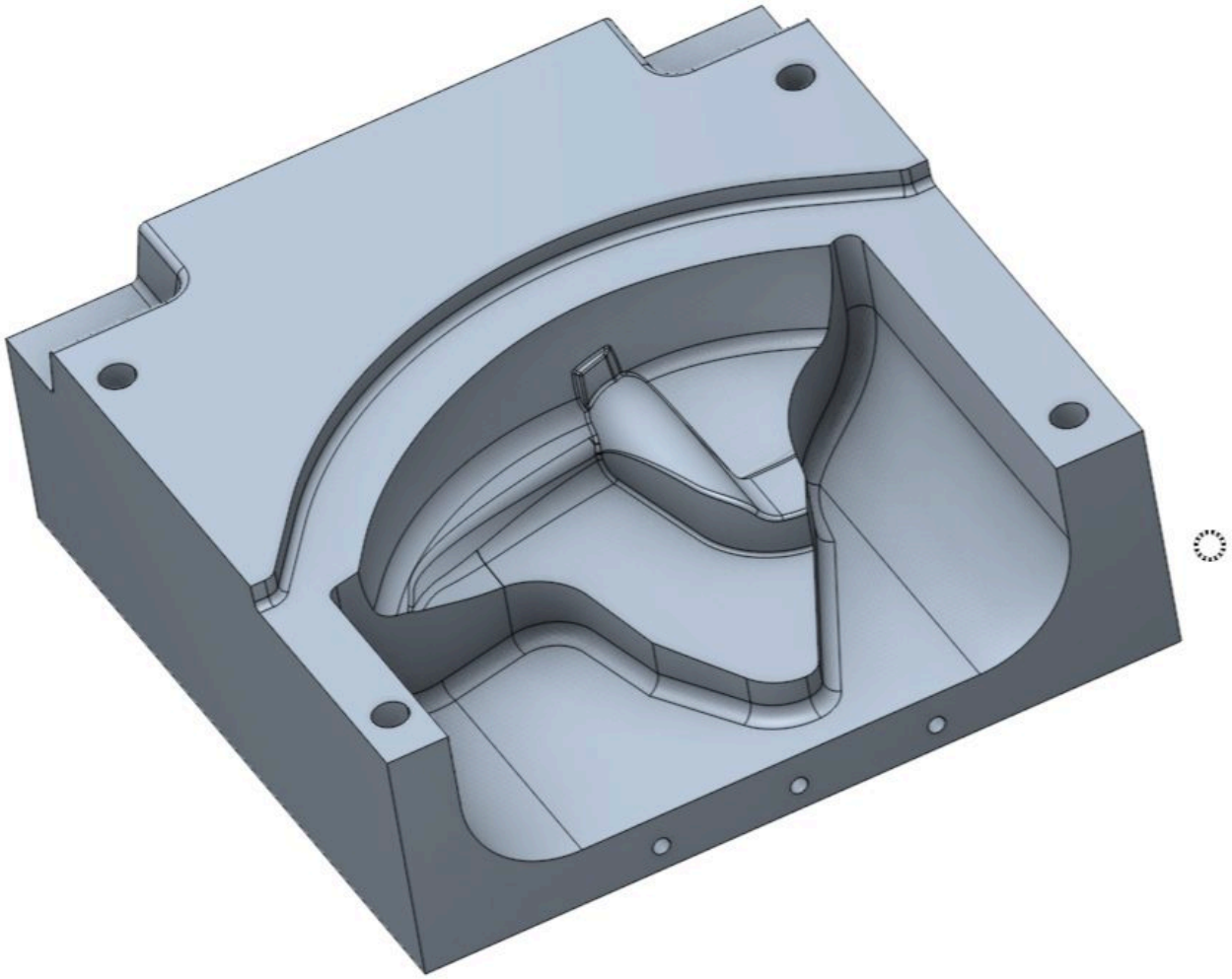


To disperse the surfaces, hold Ctrl+Alt + Left-click.

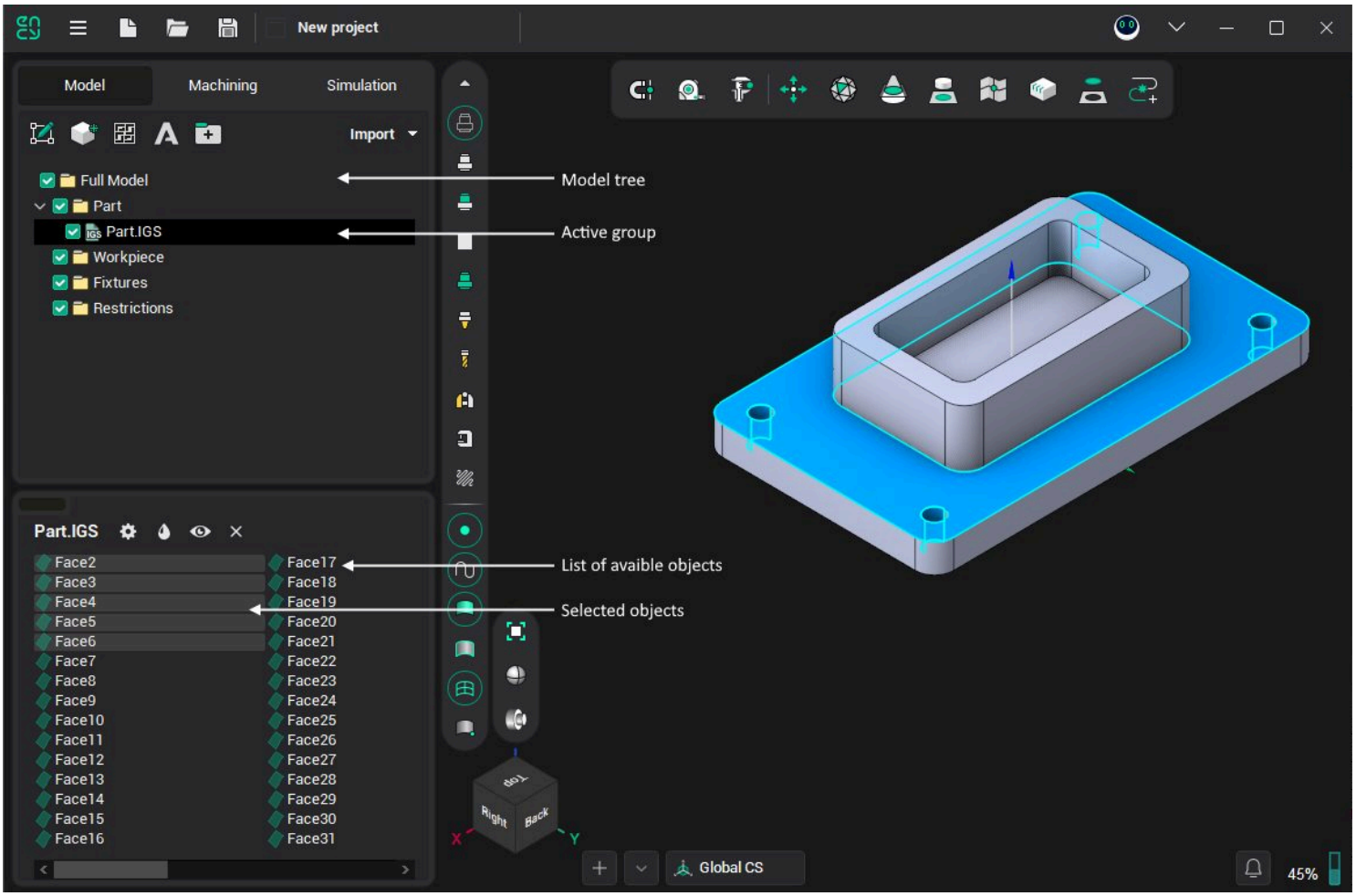


You can change the brush radius. To do this, hold down Shift + Alt and rotate the mouse wheel up or down.





Geometrical model structure editing



When editing the model structure one can create new groups (model structure tree nodes), delete, cut/copy to the exchange buffer or paste geometrical and structural objects (surfaces, meshes, curves, points and groups) from the exchange buffer.

Predefined groups (Full model, Part, Workpiece, Fixtures, `2D Geometry`), and all objects inside `2D Geometry` group cannot be deleted or cut into the exchange buffer. However copying the objects into the exchange buffer is possible without any limitations. Objects copied from `2D Geometry` will lose their connection with that environment, and if any subsequent changes made in the 2D geometry mode, these objects will remain unaltered.

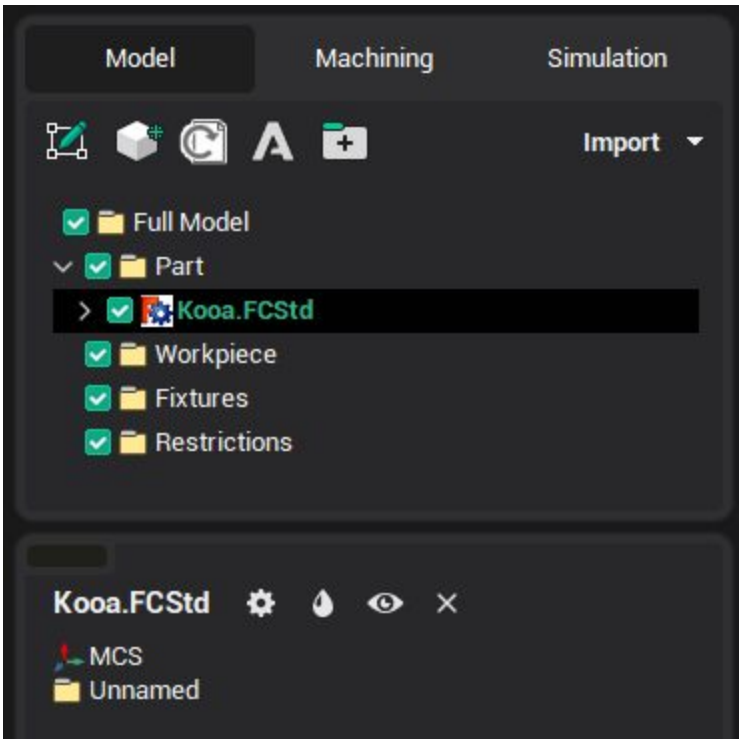
Creating a new group

The  button in the geometrical model structure window creates a new group. The new group will be created inside the active one.

Geometrical model structure updating

When updating, the old groups in the model will be replaced with new ones, this will retain all of spatial transformations that took place earlier and optionally created by the user (for example, section), groups. Checking that the file has changed is performed when the CAM system window becomes active.

If the imported file has been changed, the name of the group will be allocated bold, the toolbar button appears "Model files update manager" , and in the pop-up window will be added to another team Update.



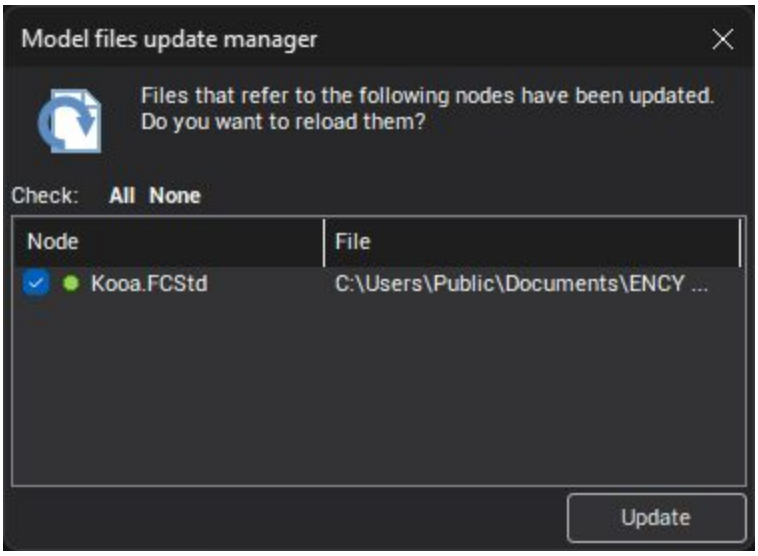
There are two options for the update:

- from CAM system:
    - via Model files update manager;
    - via context menu Update.
- from CAD system in which installed a CAM system addin.

Updating from CAM system

Model files update manager

After importing the model file, the CAM system will periodically check the modification date of this file. Once the change is detected, the following window appears:

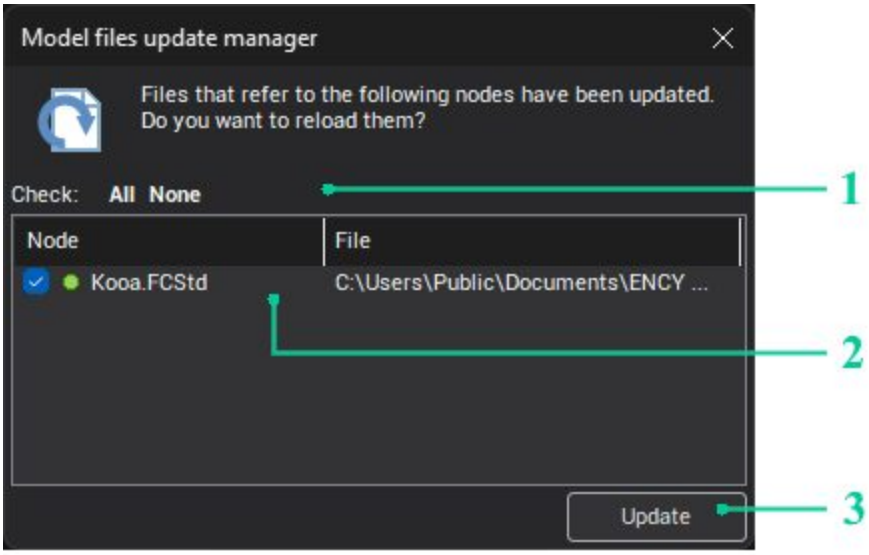


> If you just close the Model files update manager window, the next time it appears only when a new change is detected.

This window contains a list of imported models, files that have been changed. Here you can update them.

> Model files update manager can also be opened by using the button , if it is available on the panel.

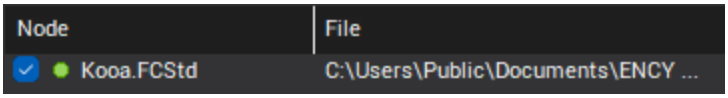
Description of the window Model files update manager



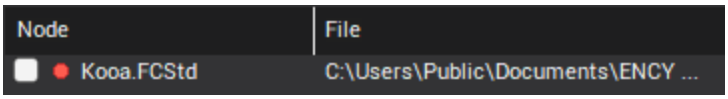
1. All button select all files, None button deselect all marks.
2. There is a list of all the imported files that were changed in this section. Set checkbox beside the file indicates that it is marked and after the pressing the Update button it will be updated.

> Only the selected files will be updated.

The green round mark shows that the file is available for upgrade:



The red round mark shows that file updating is impossible (the reasons can be different: file was displaced, supplement in Addin Manager is disabled, CAD system is not available, etc.). In this case, it can not be set (tick the box)

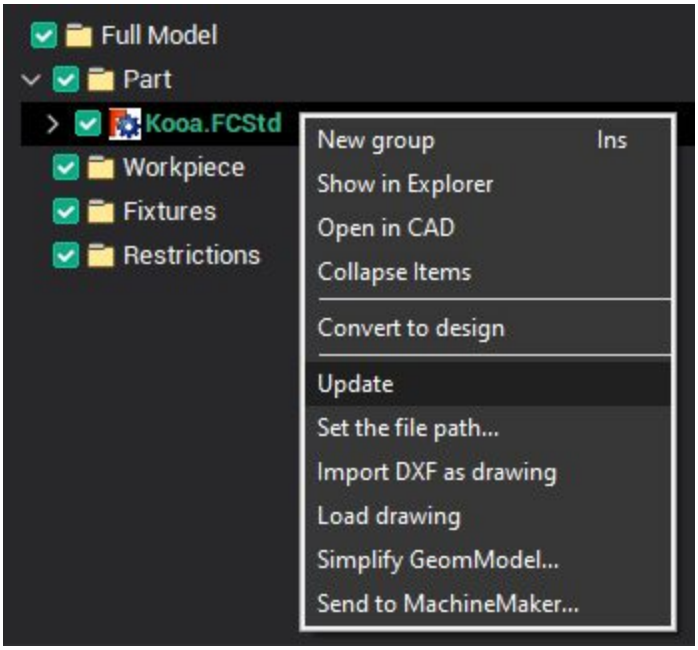


> Double-click on the line will open a folder with the file.

3. Update / Close button works as follows:

- if at least one file is selected, it shows the Update button, otherwise Close button;
- when you click on Update — all selected files will be updated, and the window will close;
- when you click on Close — the window will be closed (works the same as clicking on the X).

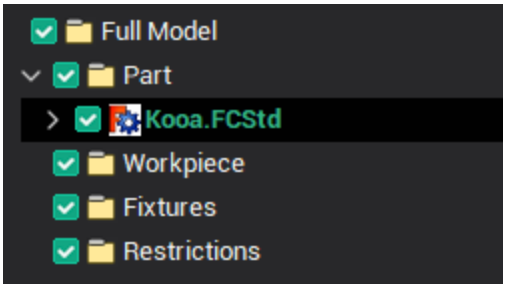
Context menu Update



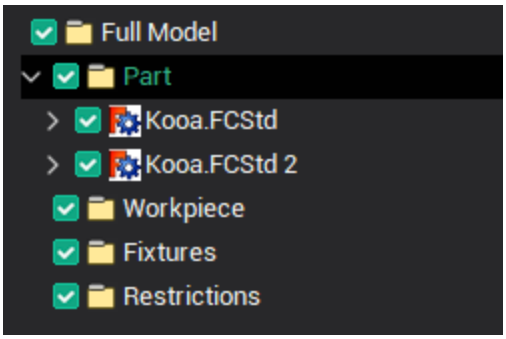
Updating from CAD system

Importing files is always in the active group, so CAM system's behavior depends on which group will be active:

- If you want to update, imported earlier model, this group needs to make active or go to CAM system and update from it;



- If you want to re-add imported earlier model, it is necessary to make active any other node.



Working with the exchange buffer

On the tool panel in the model structure window you will also find the buttons for working with the exchange buffer (cut, copy & paste).

-  – Move selected objects to the clipboard.
-  – Copy selected objects to the clipboard.
-  – Insert the objects from the clipboard to the current group.

Copied or cut objects can be repeatedly inserted into the currently active folder.

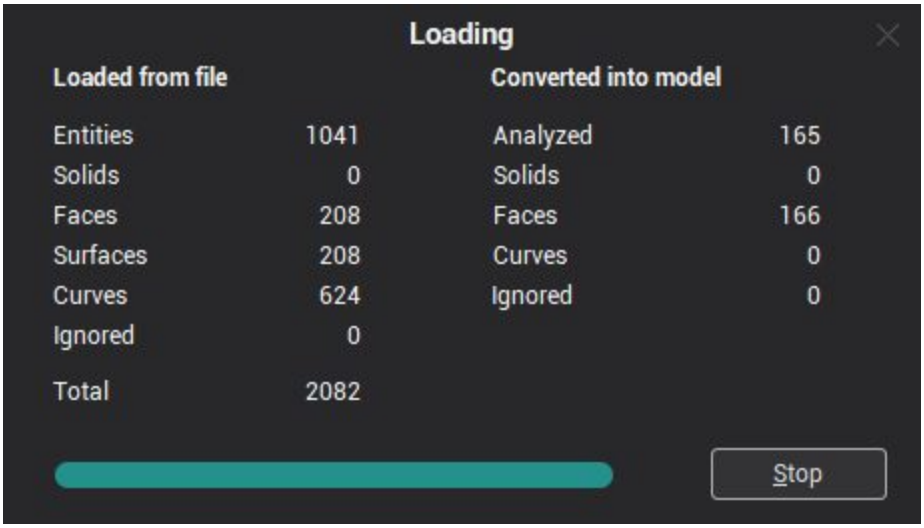
The functions are duplicated in the context menu (right mouse click) in the graphic window. Access to the functions is also possible by using the shortcut keys: [Ctrl+X], [Ctrl+C], [Ctrl+V].

Geometrical objects import

Import of models from external CAD files is performed by clicking the Import button or by simply dragging the model file into the application's window. Function available also from main menu File –> Import, and by pressing hotkey [Ctrl+I].

In the file selection window, it is possible to specify file extension filters. The set of supported file formats depends on the configuration of CAM system and installed addins.

During the import process of external CAD files, the current information concerning the progress of the file reading process and creation of geometrical objects is displayed in the window. A process indicator reflects the percentage of import function completed. When importing from files with a simple structure, the system uses a one-pass algorithm i.e. reading the file and the formation of the geometrical model is performed simultaneously. When importing from files with a complex data structure the system uses a double-pass algorithm. In the first stage, the system reads the file and analysis the model structure, and in the second, creates the geometrical model.



The Loaded from file panel shows statistical data about reading the file:

- Entities – loaded entities counter;
- Solids – loaded solids counter;
- Faces – loaded faces (bounded surfaces) counter;
- Surfaces – loaded surfaces counter;
- Curves – loaded curves counter;
- Ignored – ignored (insignificant, incorrect or not supported) entities counter;
- Total – loaded entities total number.

The Converted into model panel displays statistical data on conversion of the read data into the inner model:

- Analyzed – the converted entities counter;

- Solids – the converted solids counter;
- Faces – the converted faces counter;
- Curves – the converted curves counter;
- Ignored – the ignored entities counter.

All topological references between objects are analyzed exactly at the stage of creation of the inner model, also, out of a huge number of components, the complex objects are formed (Solids, Faces). All simple objects (curves/points etc.) within the more complex ones are additionally duplicated in the form of independent objects. Therefore, the total number of loaded from file objects is actually more than those converted into a model.

For example, a face consists of an unlimited surface and several restricting curves. When reading the file, the face and all its contents are counted in the appropriate fields of the counter. Moreover, when creating the inner model all these elements are counted as one face.

Clicking the Stop button during import will stop the loading process.

During the import process, the system analysis the imported model and if errors or any inappropriate records or unsupported data types occur, a corresponding report message is created. Error messages are displayed in an auxiliary window, which opens when clicking the Errors button. The button becomes available only if there were errors during the import process. Should fatal errors occur, file loading terminates. If errors occur, it is advised to study more closely the particular import features of this file format, and comply with recommendations on how to avoid such errors.

If the box marked Close the window automatically is ticked, then if there are no errors, the window will automatically close after the completion of the import process. If the checkmark is switched off, or there were errors during the import, then the system awaits pressing the Ok button to proceed.

Note: Only those types of objects will be imported, which are defined in the system settings window on the Import tab. Elements of other types are ignored.

Note: If it is necessary to transform types of curves, the maximum deviation during approximation will be less than the value defined in the Curves approximation tolerance field in the same window.

Importing objects from IGES files

Geometrical data exchange files in the IGES format normally have an *.igs or *.iges extension. IGES format allows the transfer of a multitude of different types of geometrical objects. This is why one can achieve data transfer between different systems with virtually no distortion. The IGES format is widely used, especially in areas where high data transmission accuracy is required due to very complex three-dimensional geometrical models.

There is build-in module for import IGES at CAM system.

Requirements for IGES-file

The IGES import module has been developed based upon the specifications of IGES version 5.3. The system imports only IGES files in ASCII format. This means that IGES files, created in compressed ASCII-format or in binary format, will be evaluated by the system as incorrect. The system automatically defines the type of text file (DOS-type or UNIX-type, use different indications of the string end) and correctly loads both file types.

Types of importable objects

All objects as defined in the IGES standard are divided into groups. Listed below are the IGES groups and objects, currently importable by the system.

The following types are imported from the Curves and Surfaces group:

- Circular Arc, type 100;
- Composite Curve, type 102;
- Conic Arc, type 104;
- Copious Data, type 106;
- Plane, type 108;
- Line, type 110;
- Parametric Spline Curve, type 112;
- Parametric Spline Surface, type 114;
- Point, type 116;
- Ruled Surface, type 118;
- Surface of Revolution, type 120;
- Tabulated Cylinder, type 122;
- Transformation Matrix, type 124;
- `NURBS-curve Rational B-Spline Curve`, type 126;
- `NURBS-surface Rational B-Spline Surface`, type 128;
- Offset Curve, type 130;
- Offset Surface, type 140;
- Boundary, type 141;
- Curve on a Parametric Surface, type 142;
- Bounded Surface, type 143;
- Trimmed Surface, type 144;

The following types are imported from the `B-Rep Solids` group:

- Face, type 510;
- Loop, type 508;
- Edge, type 504;
- Vertex, type 502;

This allows the program to work with `Manifold Solid B-Rep Object`, type 186 as with the set of bounded surfaces.

From the Annotation Entities group no type is imported. These entities are not significant for machining purposes.

From the Structure Entities group only the Color Definition, type 314 entities are imported. This means that model colors in CAM system are identical to the colors used in the modeling program.

Recommendations on how to adjust IGES-export module in your modeling program

Virtually all modern systems of 3D modeling have an export module in IGES format. This module normally has options for tuning and configuring. Here are some recommendations that you are advised to use when preparing IGES files.

- If in the IGES export tuning menu there is an accuracy control, it should be set relatively high. This will allow the system to link surfaces forming the model more accurately, and consequently more accurately process it. It is not recommended to export the model with accuracy less than the required machining tolerance.
- It is advised to substitute objects types that cannot be imported by the system into those that can. For example, the draughtsman, working in the modeling program, may use a solid-sphere entity. Importing this into the system as an IGES-entity Sphere (type 158) is impossible, but it is correct to change the entities of that type to a combination of entities of other types (e.g. 144, 143, 510).
- If sets of boundary curves (bound, loop) are defined in modeling 3D-space, then corresponding boundary curves in parametric space (UV) will be automatically created during import. This theoretically can bring additional errors into the model. Therefore, the presence of parametric boundary curves is required. This concerns Boundary (type 141) and Loop (type 508) entities. If there are problems during import of geometrical objects, then try allowing parametric bounds generation in the IGES-export options in the modeling program. If that is impossible, then change Bounded Surface (type 143) and Face (type 510) entities to Trimmed Parametric Surface (type 144).

Importing objects from STEP files

STEP - Standard for Exchange of Product model data. The format of a STEP-File is defined in ISO 10303-21 (part 21 - geometric shape representation). STEP format allows the transfer of a multitude of different types of geometrical objects. This is why one can achieve data transfer between different systems with virtually no distortion. The STEP format is widely used, especially in areas where high data transmission accuracy is required due to very complex three-dimensional geometrical models.

Supports the following model types

- wireframe;
- surface;
- solid bodies.

Schema files

- CONFIG_CONTROL_DESIGN;
- AUTOMOTIVE_DESIGN.

The standard extensions of files are.*step*;.*stp*.

Importing objects from DXF files

DXF format is used for transmission of flat drawings and vectored images. Transferring volumetric models is supported, but depends on the version used. These files have a *.dxf extension.

Limitations: The section HEADER must be present in DXF file. A file without a header is considered faulty.

Currently, only geometrical objects can be imported from DXF files. Object geometry has considerable affect on the machining technology, and such features, as thickness and style of the objects are not required, and therefore are ignored.

Note: The current version does not import text (object TEXT). To be able to import text, it must be first converted into curves.

Types of importable objects:

- POINT;
- LINE;
- CIRCLE;
- ARC;
- POLYLINE;
- SPLINE;
- BLOCK, INSERT – all above types will be imported without blocks (exploded).

Importing objects from PostScript files

PostScript format allows transferring flat vectorial and raster figures. Files normally have *.ps or *.eps extension (Encapsulated PostScript). The format is used widely in publishing and when transferring information to printers, supporting the PostScript-interface.

Limitations: CAM system imports from PostScript-files only vector information. It does not import raster images inserted into PostScript file. The current version does not import text. For correct import of text it must be converted into curves.

GhostScript library is used for the import (installation instructions). The first time you import will be prompt to load a library, and specify its location. Also, it can be done in the System Settings window, the Import tab.

Recommendations on how to export into PostScript file in CorelDraw:

- Switch Curves mode in the Export text as group, with this CorelDraw will generate sets of curves, which correspond to every figure of the text in the PostScript file.
- Switch on the Include header option in the Image header group.

Installing the GhostScript Library

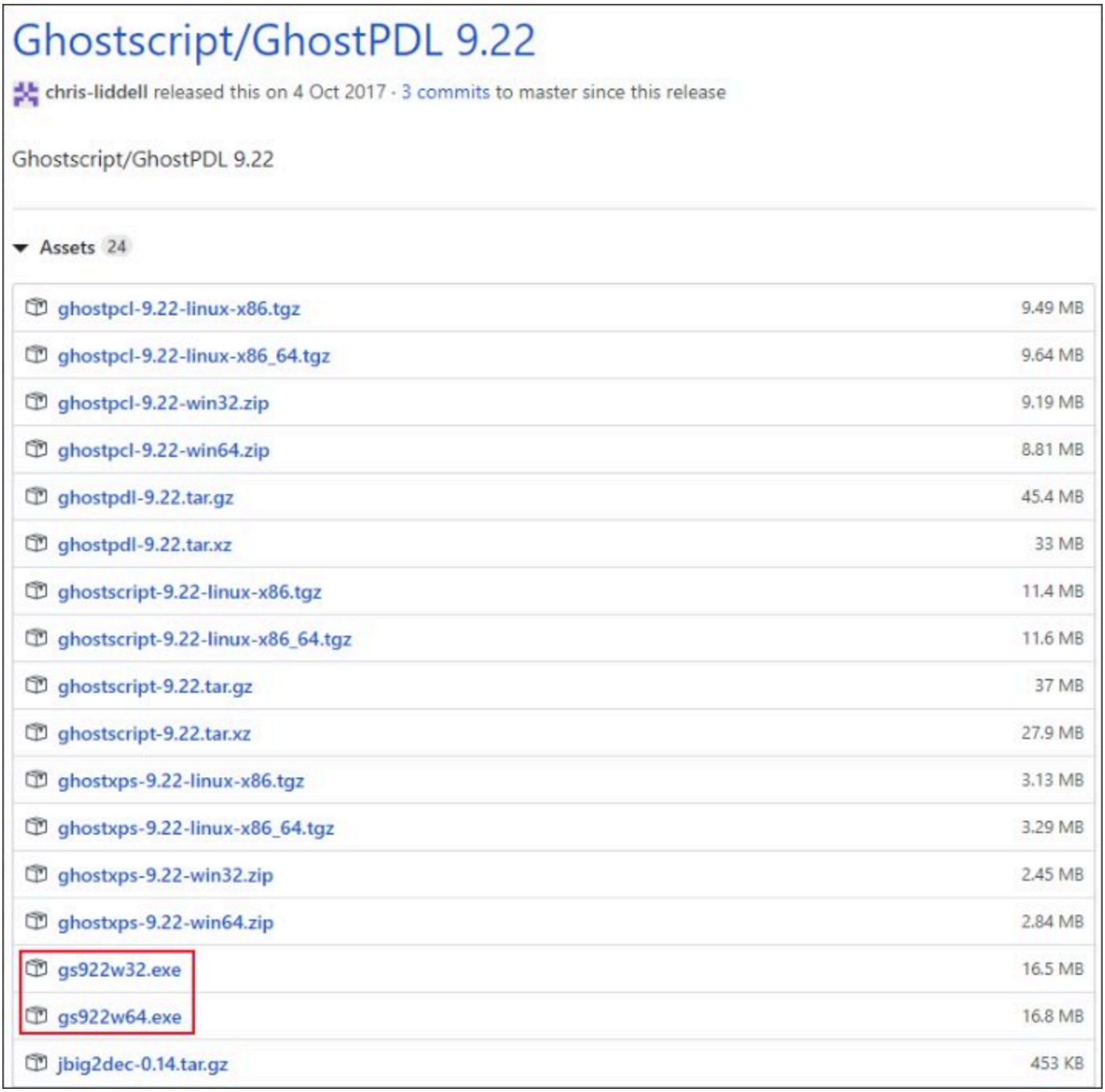
Ghostscript software is required to import PostScript files. This library was developed by Artifex Software, Inc. and is not included in the CAM system distribution due to licensing restrictions. But you can download and install it for personal use.

CAM system supports the Ghostscript library of version not older than 9.22.

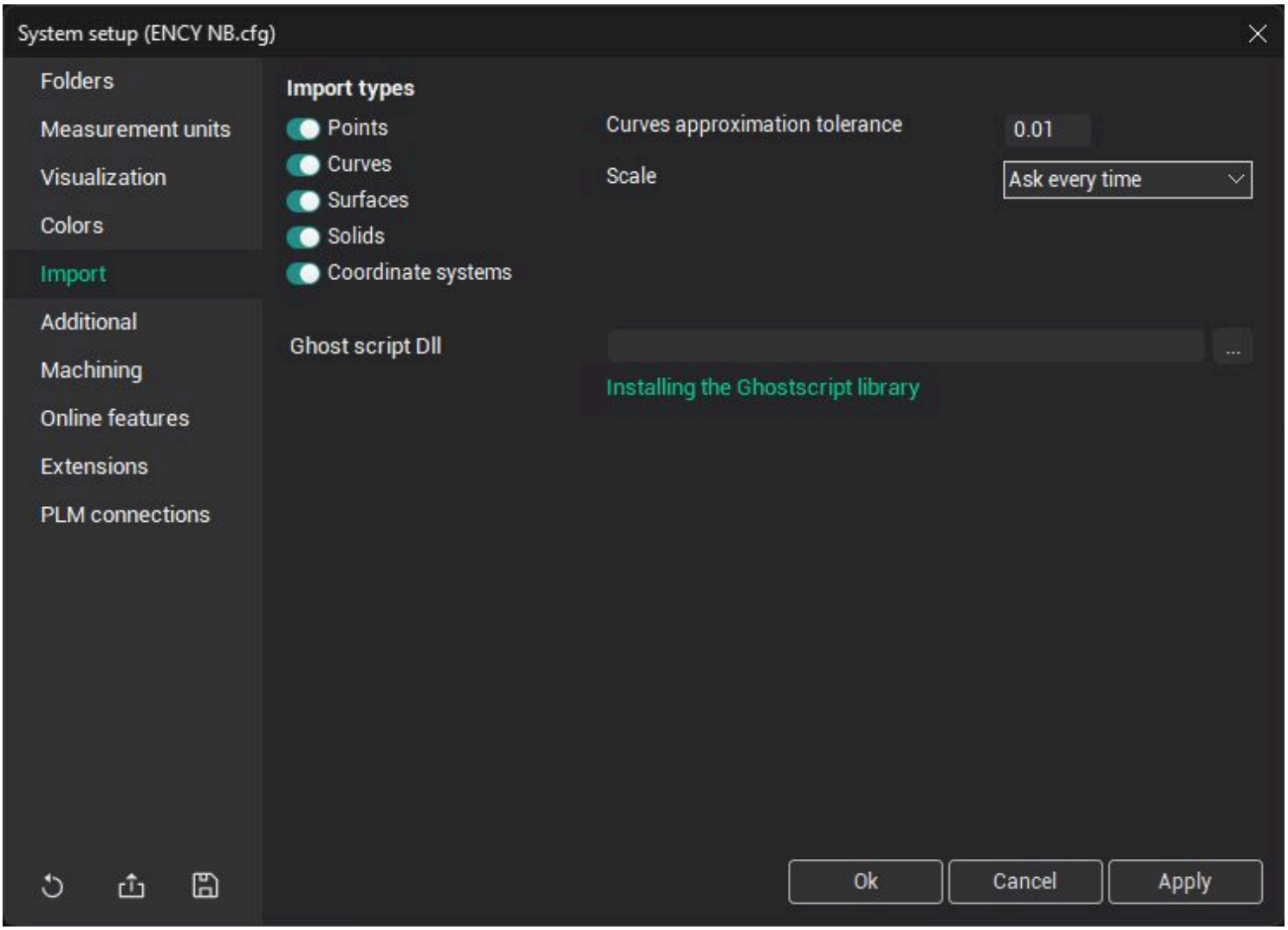
Download Ghostscript 9.22

The GitHub repository window will open - select compatible windows version (64 bit):

Note: You must select - Ghostsript 64 bit.



After installation, you must specify the path to the library: gsdll32.dll or gsdll64.dll (32 or 64 bit). It is located in the directory where you installed the software Ghostscript, in the bin folder. This can be done either by using the System setup tab Import:


System setup (ENCY NB.cfg)
Folders
Measurement units
Visualization
Colors
Import
Additional
Machining
Online features
Extensions
PLM connections
Import types
Points
Curves
Surfaces
Solids
Coordinate systems
Curves approximation tolerance
0.01
Scale
Ask every time
Ghost script Dll
...
Installing the Ghostscript library
Ok
Cancel
Apply

Importing objects from STL files

The format allows transferring volumetric models, represented using flat triangles. The files normally have an *.stl format. The system imports both, binary and text formats.

There are no limitations on the type of importable objects. Should there be problems with importing files in binary format, try importing via text format, for it is platform-independent.

Note: A model transferred via STL format is approximated by many triangles. Therefore, transmission without accuracy loss is only possible for some geometrical model types. To transfer models, it is a commonly held view that you set approximation accuracy, when exporting that is not less than the required machining tolerance, or, to use a more accurate format (e.g. IGES).

Importing objects from PLY files

PLY is a computer file format known as the Polygon File Format or the Stanford Triangle Format. It was principally designed to store three-dimensional data from 3D scanners.



Importing objects from AMF files

Additive Manufacturing File Format (AMF) is an open standard for describing objects for additive manufacturing processes such as 3D printing. The official ISO/ASTM 52915:2013 standard is an XML-based format designed to allow any computer-aided design software to describe the shape and composition of any 3D object to be fabricated on any 3D printer.



Importing objects from VRML files

VRML (Virtual Reality Modeling Language) – this is a file format for interactive three-dimensional objects and virtual worlds. The VRML format is designed for use on the Internet (and is basically the 3D graphics standard on the Net). VRML files are also used in local systems.

Each VRML file is a description of 3D space containing graphical objects. The scene can be dynamically changed using different language mechanisms.

To design a machining technology, information concerning the object geometry is required, and such attributes as light source, background color, transparency or smoothing angle, animation elements and event processing are irrelevant and therefore will be ignored.

In the current version the following geometrical objects are supported:

- Box;
- Cone;
- Cylinder;
- ElevationGrid;
- Extrusion;
- IndexedFaceSet;
- IndexedLineSet;
- Sphere;

Information about a geometrical objects color, their location and spatial transformations is imported. Named objects supported too.

Limitations:

- Cannot import object TEXT.
- Ignores block Inline for using data from other files and the Internet.
- It is not recommended to use VRL files with PROTO and EXTERNPROTO sections, due to their partial support.

Requirements for VRML files:

- Files are imported in `VRML 2.0` format.
- Presence of a file header `#VRML V2.0 utf8` is obligatory, otherwise the file will be regarded as in an incorrect format.
- A packed file must first be unpacked.

Importing objects from 3dm files (Rhinoceros™)

CAM system performs direct reading of project files from the Rhinoceros CAD system versions 4.0 - 7.0 (*.3dm).

All types of geometrical data are imported. Information regarding elements layers and visual properties, except for color, are ignored.

Note: it is not need installed software Rhinoceros for import Rhinoceros file.

Note: it is possible import 3dm files with CAM system addin "Rhinoceros Toolbar & import addin".

Importing objects from Parasolid™ files

Parasolid™ is the core of a geometrical modeling format which supports the following model types:

- wireframe;
- surface;
- solid bodies;
- finite elements.

The data transmission format of Parasolid™ allows the user to transfer data not only about the model, but also the relations between models.

The standard extensions of files are x_t; x_b.

To design a machining technology, information concerning the object geometry is required, and such attributes as light source, background color, transparency or smoothing angle, animation elements and event processing are irrelevant and therefore will be ignored.

CAM system supports the Parasolid™ data transmission format up to 35.1 versions.

Importing objects from SolidWorks™ files

ENCY allows you to import project files SolidWorks™.

SolidWorks™ project files contain a section of Parasolid™, we extract and import it.

The standard extensions of files are SLDASM; SLDPRT.

> Starting in July 2026, a separate license will be required to use the SolidWorks module (see Changes to ENCY–SOLIDWORKS Integration Licensing – ENCY CAD/CAM software),

> At the moment, imports files without configuration impossible.

Importing objects from SolidEdge™ files

ENCY allows you to import project files SolidEdge™.

SolidEdge™ project files contain a section of Parasolid™, we extract and import it.

The standard extensions of files are ASM; PAR; PSM; PWD.

Importing objects from SGM files

3D models in from the CAM system (*.sgm) is fully supported by ENCY, without any limitations.

Importing objects with CAM system addins

Addin is component or small program, which may be attach to CAM system.

They provide the ability to import project files CAD.

Common rule for use addins it is installed CAD system to PC.

Importing objects from 5DC files

5DC format allows loading a splines as a set of points(xyzijk). The main features:

- maintained only text format UTF-8 encoded
- allows to contain multiple splines: as a separator, use the symbol ";" (semicolon) on a new line. After the symbol, you can specify the name of the curve.
- minimum number of spline points(lines) should not be less than 2
- the first line of the file instead of dots may contain settings

If the first line of the file does not contain any settings, then for each line including the first one:

- coordinate numbers are separated by a space
- point is provided as 6 numbers: "X Y Z NX NY NZ", when (X,Y,Z) - coordinates of a point, (NX,NY,NZ) - normal vector

If the first line of the file contains the settings, then for each line except the first one:

- the number of coordinates is separated by the separator specified in the settings line
- the point is represented as multiple numbers, according to the format specified in the settings line

If the settings line is set, it must contain three mandatory parameters and may contain six optional parameters. The parameters should be separated by a space.

Mandatory parameters:

- Point number format:

	Format	Description	Number of points in a line
1	tffNormalVector	Coordinates of the point and normal	6
2	tffQuaternion	Coordinates of the point and quaternion X Y Z W	7
3	tffEulerXYZ	Coordinates of the point and Euler angles Rx Ry Rz with rotations around the movable axes	6

	Format	Description	Number of points in a line
4	tffEulerXZY	Coordinates of the point and Euler angles Rx Rz Ry with rotations around the movable axes	6
5	tffEulerYXZ	Coordinates of the point and Euler angles Ry Rx Rz with rotations around the movable axes	6
6	tffEulerYZX	Coordinates of the point and Euler angles Ry Rz Rx with rotations around the movable axes	6
7	tffEulerZXY	Coordinates of the point and Euler angles Rz Rx Ry with rotations around the movable axes	6
8	tffEulerZYX	Coordinates of the point and Euler angles Rz Ry Rx with rotations around the movable axes	6
9	tffEulerXYX	Coordinates of the point and Euler angles Rx Ry Rx with rotations around the movable axes	6
10	tffEulerXZX	Coordinates of the point and Euler angles Rx Rz Rx with rotations	6

	Format	Description	Number of points in a line
		around the movable axes	
11	tffEulerYXY	Coordinates of the point and Euler angles Ry Rx Ry with rotations around the movable axes	6
12	tffEulerYZY	Coordinates of the point and Euler angles Ry Rz Ry with rotations around the movable axes	6
13	tffEulerZXZ	Coordinates of the point and Euler angles Rz Rx Rz with rotations around the movable axes	6
14	tffEulerZYZ	Coordinates of the point and Euler angles Rz Ry Rz with rotations around the movable axes	6
15	tffFixedABC	Coordinates of the point and Euler angles Rx Ry Rz without rotations around the movable axes	6
16	tffFixedXZY	Coordinates of the point and Euler angles Rx Rz Ry without rotations around the movable axes	6

	Format	Description	Number of points in a line
17	tffFixedYXZ	Coordinates of the point and Euler angles Ry Rx Rz without rotations around the movable axes	6
18	tffFixedYZX	Coordinates of the point and Euler angles Ry Rz Rx without rotations around the movable axes	6
19	tffFixedZXY	Coordinates of the point and Euler angles Rz Rx Ry without rotations around the movable axes	6
20	tffFixedZYX	Coordinates of the point and Euler angles Rz Ry Rx without rotations around the movable axes	6
21	tffFixedXYX	Coordinates of the point and Euler angles Rx Ry Rx without rotations around the movable axes	6
22	tffFixedXZX	Coordinates of the point and Euler angles Rx Rz Rx without rotations around the movable axes	6
23	tffFixedYXY	Coordinates of the point and Euler angles Ry Rx Ry without	6

	Format	Description	Number of points in a line
		rotations around the movable axes	
24	tffFixedYZY	Coordinates of the point and Euler angles Ry Rz Ry without rotations around the movable axes	6
25	tffFixedZXZ	Coordinates of the point and Euler angles Rz Rx Rz without rotations around the movable axes	6
26	tffFixedZYZ	Coordinates of the point and Euler angles Rz Ry Rz without rotations around the movable axes	6
27	tffTwoAngAB	Coordinates of the point and two angels AB	5
28	tffTwoAngAC	Coordinates of the point and two angels AC	5
29	tffTwoAngBA	Coordinates of the point and two angels BA	5
30	tffTwoAngBC	Coordinates of the point and two angels BC	5
31	tffAxisAngleRad	Coordinates of the point and axis-angle in radians	6

	Format	Description	Number of points in a line
32	tffAxisAngleDeg	Coordinates of the point and axis-angle in degrees	6

- The angles are specified in degrees: True (degrees)/False (radians). Relevant for the formats in paragraphs 3-30.
- Separator-Always one character.

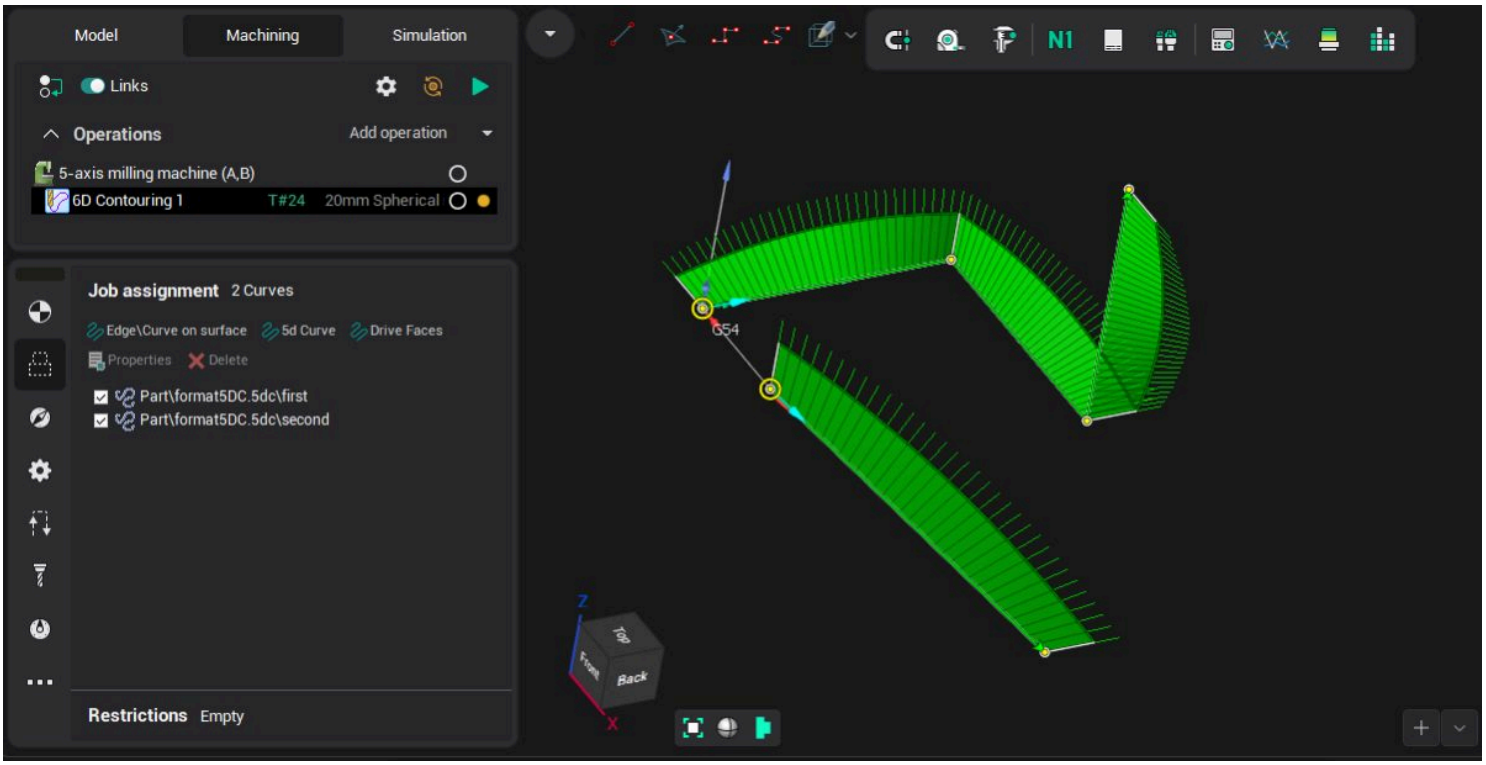
Optional parameters (additional transformation):

- Transformation along the X-axis. X=N, where N is offset value
- Transformation along the Y-axis. Y=N, where N is offset value
- Transformation along the Z-axis. Z=N, where N is offset value
- Rotation to a A-angle. A=N, where N - rotation angle value
- Rotation to a B-angle. B=N, where N - rotation angle value
- Rotation to a C-angle. C=N, where N - rotation angle value

Example of a file that does not contain settings line:

```
                    format5DC.5dc
; first
0.000000 0.000000 0.000000 -1.000000 0.000000 0.000000
0.000000 50.000000 0.000000 0.000000 0.000000 1.000000
50.000000 50.000000 0.000000 0.000000 1.000000 0.000000
50.000000 50.000000 50.000000 1.000000 0.000000 0.000000
; second
25.000000 0.000000 0.000000 0.000000 0.000000 1.000000
50.000000 50.000000 -50.000000 0.000000 1.000000 0.000000
```

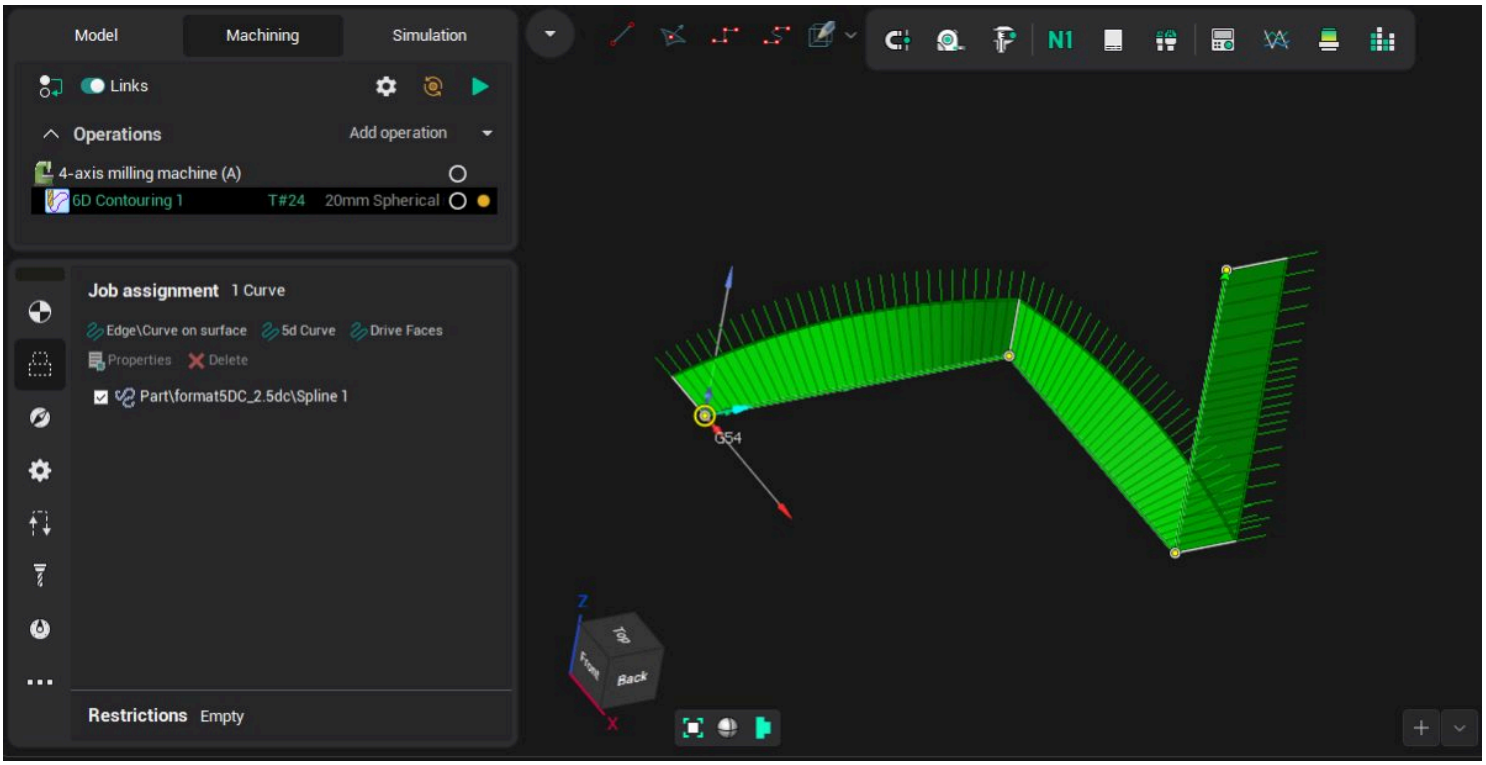
The result of adding a curves in Job assignment by the button 5d Curve in 5D Contouring operation is shown on the following image:



Example file, with settings line:

```
                    format5DC_2.5dc
tffEulerXYZ true , A=0.00 B=0.00 C=0.00
0.000,0.000,0.000,0.000,-90.000,0.000
0.000,50.000,0.000,0.000,0.000,90.000
50.000,50.000,0.000,-90.000,0,0.000
50.000,50.000,50.000,-90.000,0,0.000
```

The result of adding such a curve in Job assignment by the button 5d Curve in 5D Contouring operation is shown on the following image:


Model
Machining
Simulation
Links
Operations
Add operation
4-axis milling machine (A)
6D Contouring 1
T#24
20mm Spherical
Job assignment
1 Curve
Edge\Curve on surface
5d Curve
Drive Faces
Properties
Delete
Part\format5DC_2.5dc\Spline 1
Restrictions
Empty
G54
N1
Top
Front
Back

Addins for CAM system

Application area:

Addin is a component or a small program, which may be attached to the program. It may be used to support extended CAM system abilities. (For example, importing a 3D model from an external CAD system).

Usually, it is dynamic library (DLL), macros or VBA/JS-scripts, COM-objects, etc.

Addin can be used to import geometry from CAM or to expand the CAD program user interface using the activation toolbar or the menu.

Supported version of Addin`s:

Alibre Desing. 2012 - V26

CADBro. 2016 - 2023

Inventor. 2019 - 2024

IRONCAD. 2013 - 2024

FreeCAD. 0.14 - 1.00

KeyCreator. 18.0 - 19.0

MegaCAD. 2014 - 2016

NX. 8.5, 11, 1926

OnShape. All versions

Rhinoceros. 4 - 8

SolidCAM. 2021

SolidEdge. ST2 - 2022

SOLIDWORKS. 2013 - 2026

SpaceClaim. 2017 - 2021

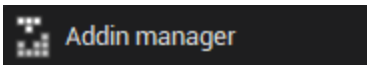
ZW3D. 2013 - 2023

The required CAD system must be installed on your computer for this option to work.

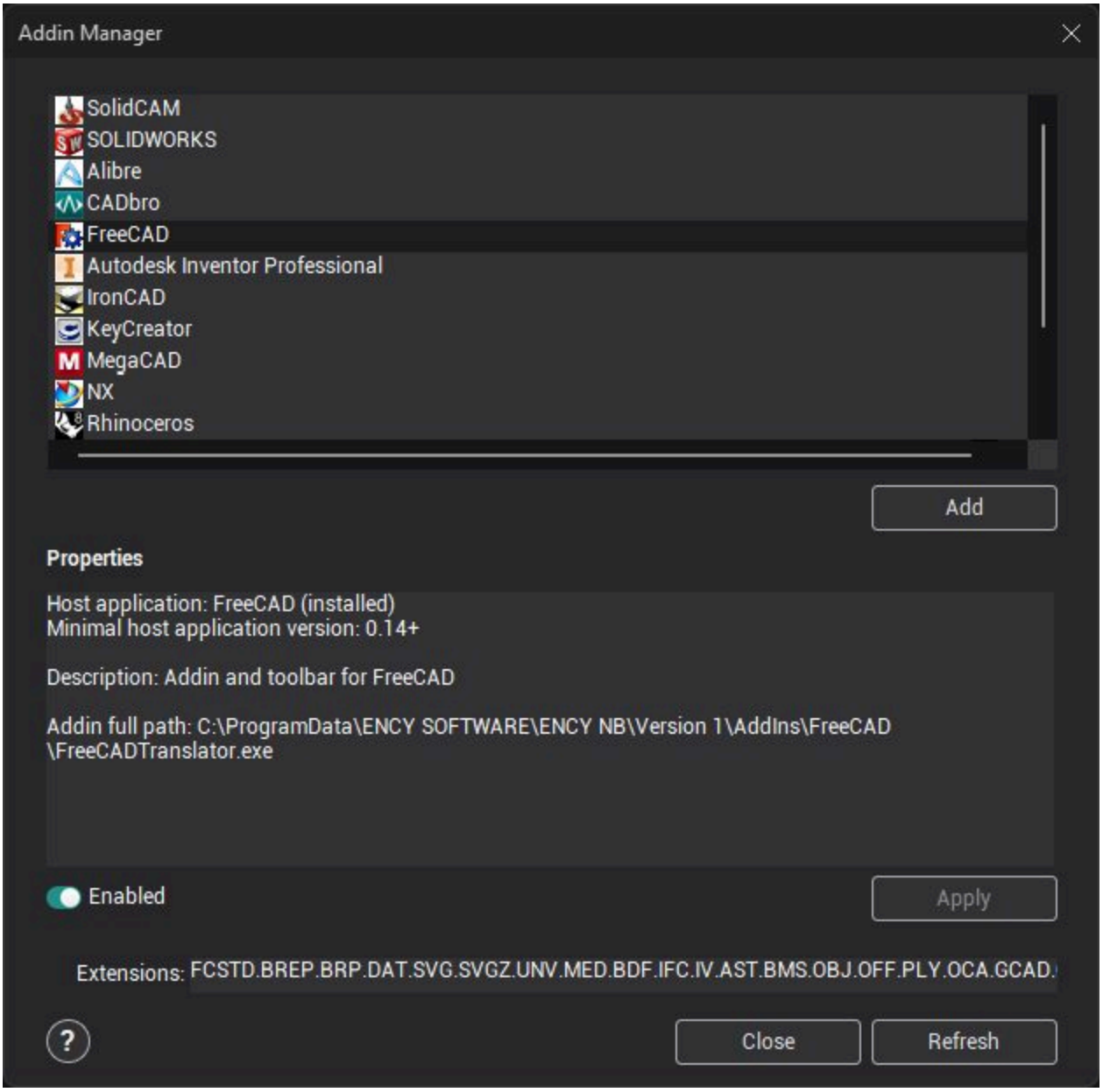
Addin Manager

Addin Manager is a program designed to control CAM system addin's and external CAD translators.

You can run Addin's Manager from CAM system- use menu item on main panel.



Addin Manager window:



Addin's

Work sequence to install addin:Select necessary addin from the top list. Press Install button. Edit parameters (custom). Press Apply button to save changes (custom). Press Close button (custom).

Work sequence to uninstall addin:

Select necessary addin from the top list. Press Uninstall button. Press Close button (custom).

Before installing\uninstalling the toolbar must be closed system for which this panel will be installed, this will inform the installer. If you do not, then the installing/uninstalling will be canceled.

Addin's are listed in the top list. When one of them is selected then additional information on the Parameters panel is displayed. Usually, it is a name and version of application addin works with; description of addin abilities, etc. Editable parameters (if exists) are on this panel too.

If addin was already installed then it can be switched on/off without changes lost of it's default parameters by the Enabled checkbox (sequence of buttons Uninstall Install will reset all changes and as a result addin will be installed with parameters by default).

Field Extensions illuminates if only a selected addin supports misc manipulation with files with specific extensions (types), for example file import. In this sample, the user could edit file types which will be imported by addin selected.

Default fills properties, by default, for the selected addin. For example if file extensions were changed then press of Default resets its as default values from addin.

Apply saves changes for the selected addin.

Install / Uninstall – registers/unregisters selected addin for a work.

Close – closes CAM system Addin Manager. If there were changes then `Save/No/Cancel` dialog will be shown.

Refresh – rereads all addins. Note: all changes will be lost.

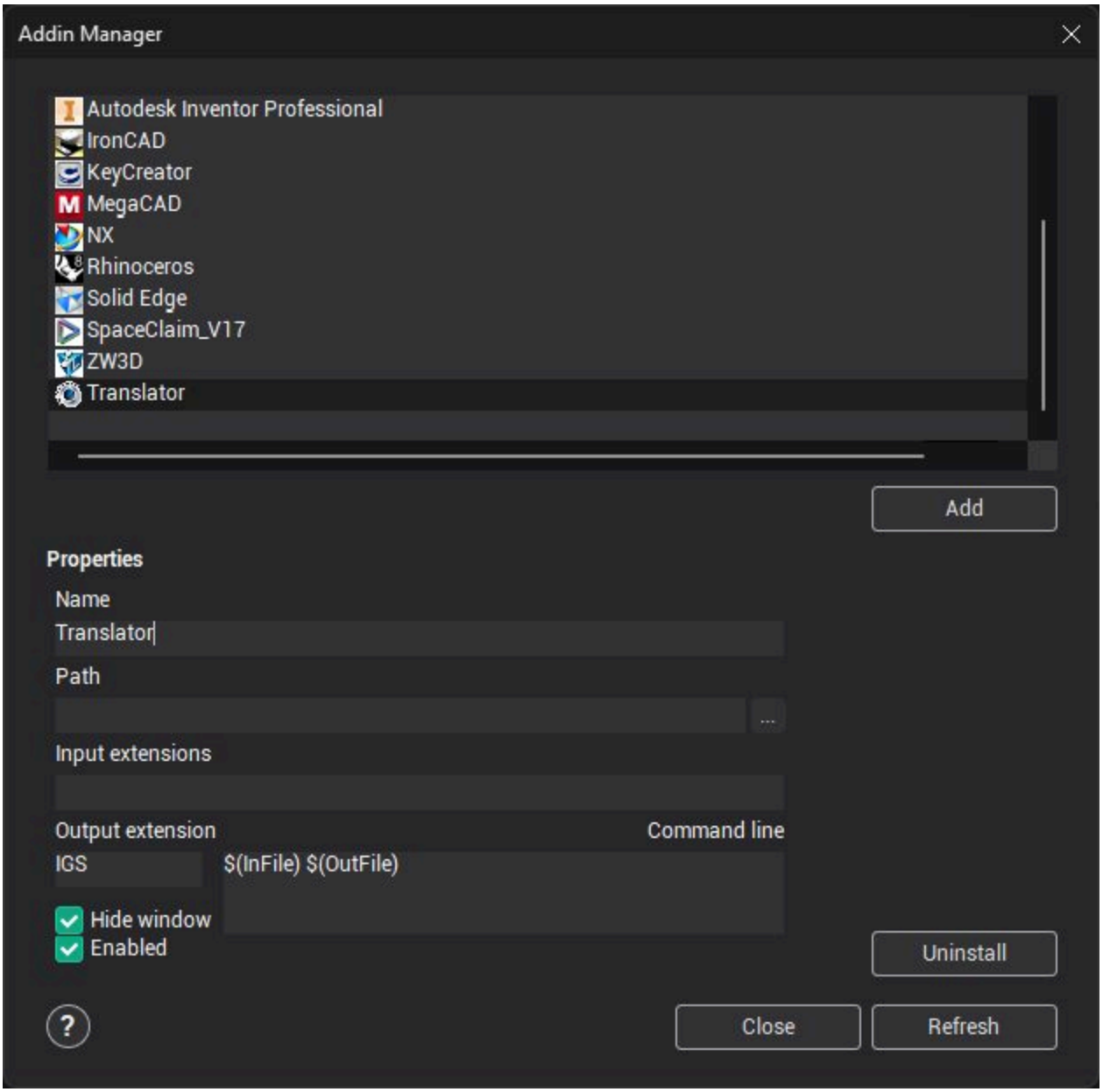
Help – displays this help. In some cases there is a need to reload application (addin host) when addin properties are changed. For example after installation of the CAM system toolbar for SOLIDWORKS® ' addin SOLIDWORKS® should be reloaded. All addin's must be in the 'Addins' folder (the folder for CAM system Addin Manager) and/or its subfolders. CAM system Addin Manager scans those folders to look for addin's and lists its. That is enough to place any addin (from CAM system or from any other source) to those folders and it will be accessible for CAM system Addin Manager (press Refresh button if CAM system Addin Manager is run).

External CAD translators

You can use the Addin Manager to plug external CAD translators. A CAD translator is a program which is able to convert one file format (e.g. CATIA, Pro/E, NX etc.) into another native CAM system formats (e.g. IGS, STL, 3DM etc.) and supports the command line interface (e.g. you can type convert.exe Model.CATIA Model.IGS in a command line and the translator will generate an IGS file from the CATIA file). After

registering a cad translator in the Addin Manager you will be able to seamlessly import the cad files supported by the translator into CAM system.

So, to add an external CAD translator into CAM system you shouldOpen the Addin Manager Click on the Add button. Fill the parameters of the translator.



The parameters are the following:Name - This name will be appeared in the Import dialog in the list of supported formats (e.g. Translator1 (external) (.model.ipt) Path - This is the full path to the translator executable. You can fill this field by pressing the ellipses button next to the text box and navigating to the executable using the standard windows file explorer. Input extensions Here you should enter the input file extensions supported by the translator. The extensions must be dot separated. E.g. model.ipt.step.sat. Output extension - In this field you have to specify the output file extension of the translator. It must be one of the CAM system natively supported formats, e.g. IGS, 3dm, STL, STEP, X_T. Command line - In this box you should specify the command which will be passed to the translator. Use the $(InFile) macro

instead of the input file name, and the $(OutFile) macro instead of the output file name. E.g. -i $(InFile) -o $(OutFile)

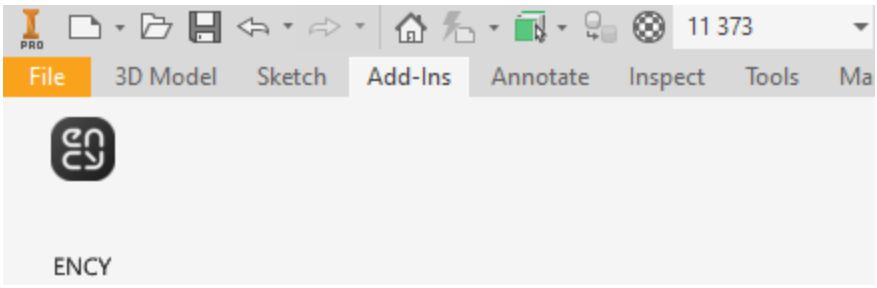
Addin's list

The program includes the following addin's:Alibre Design™ toolbar & import addin; Autodesk Inventor™ toolbar & import addin; IronCAD™ toolbar; CADbro™ toolbar; CAXA 3D™ toolbar; FreeCAD™ toolbar & import addin; KeyCreator™ toolbar; NX toolbar & import addin; Rhinoceros™ toolbar & import addin; SolidCAM™ toolbar & import addin; SolidEdge™ toolbar & import addin; SOLIDWORKS™ toolbar & import addin; SpaceClaim™ toolbar & import addin; ZW3D™ toolbar & import addin; Onshape™ connector plugin.

Autodesk Inventor™ toolbar & import addin

Autodesk Inventor Professional

The toolbar allows you to export geometric data from Autodesk Inventor Professional™. You can view the version of the supported CAD system. Details



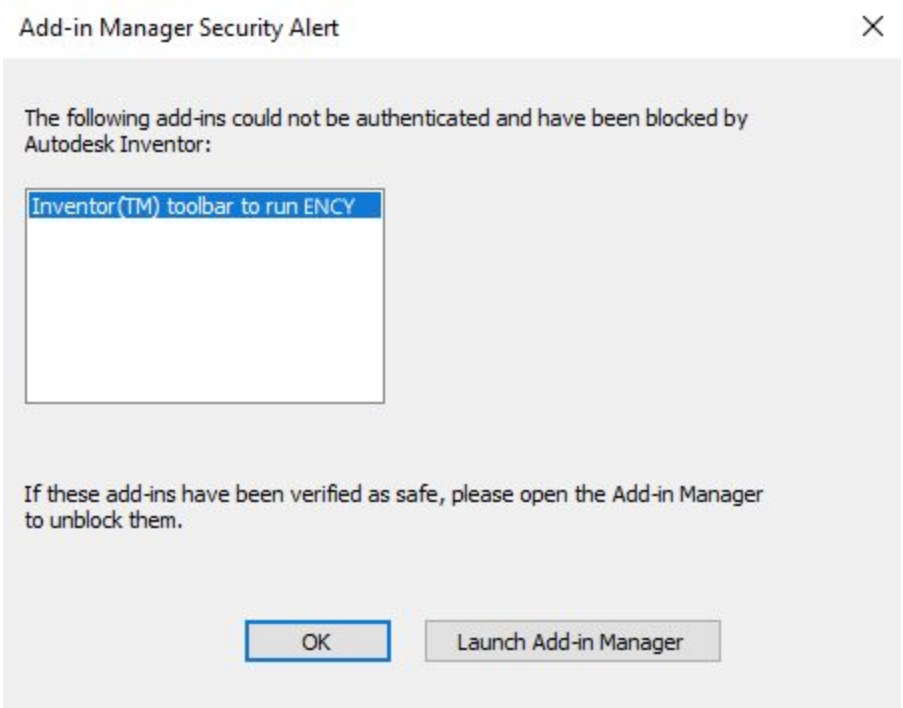
The addin allows you to import Autodesk Inventor Professional™ project files.

Supported file extensions: IAM, IDW, IPT, IPN, IDE, PRT, ASM, SAT, STE, STEP, DWG, DXF, IGES, IGS.

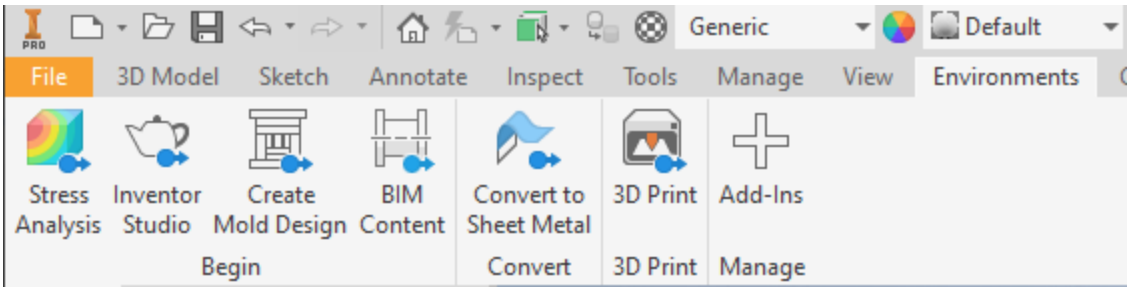
The required application (Autodesk Inventor Professional™) must be installed on your computer for this option to work.

Autodesk Inventor Professional™2021 may have problems installing the toolbar:

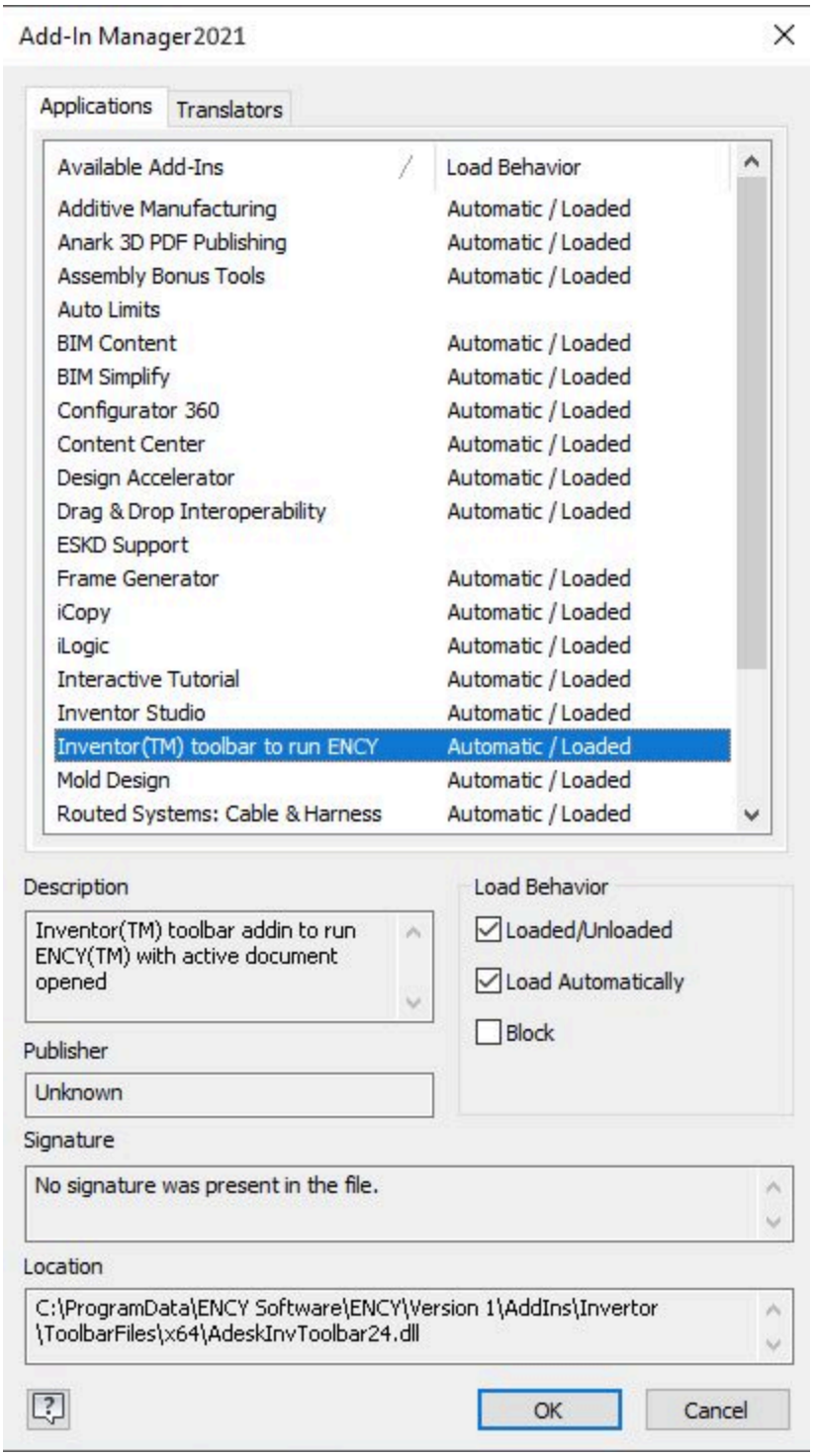
If at startup the following window appears:



you must manually enable the toolbar in Autodesk Inventor Professional™2021. To do this, start the Add-in Manager in this window, or find the button on the ribbon panel:



In the window that opens, you need to find the CAM system toolbar and set the parameters as shown in the following screenshot:

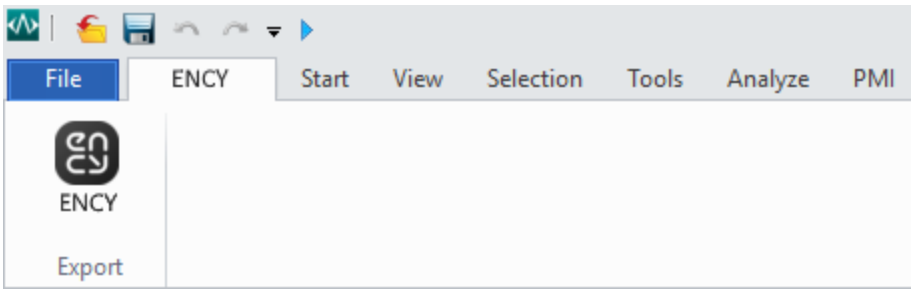


Then the CAM system toolbar should appear on the ribbon panel.

CADbro™ toolbar

CADbro

The toolbar allows you to export geometric data from CADBro™. You can view the version of the supported CAD system. Details

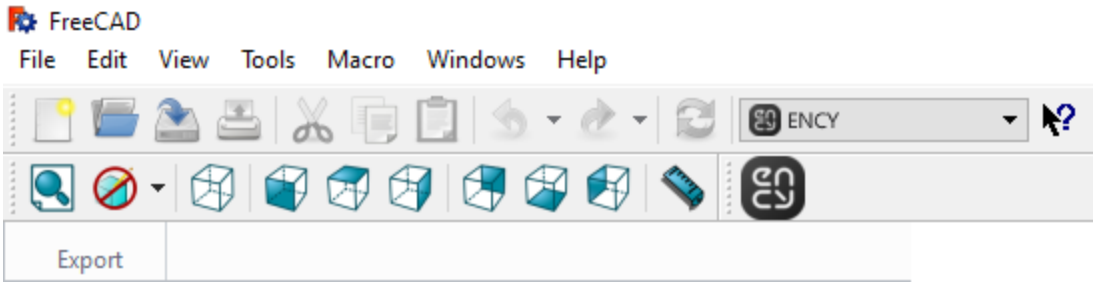


The required application (CADbro ™) must be installed on your computer for the correct work of this option.

FreeCAD™ toolbar & import addin

FreeCAD

The toolbar allows you to export geometric data from FreeCAD™. You can view the version of the supported CAD system. Details



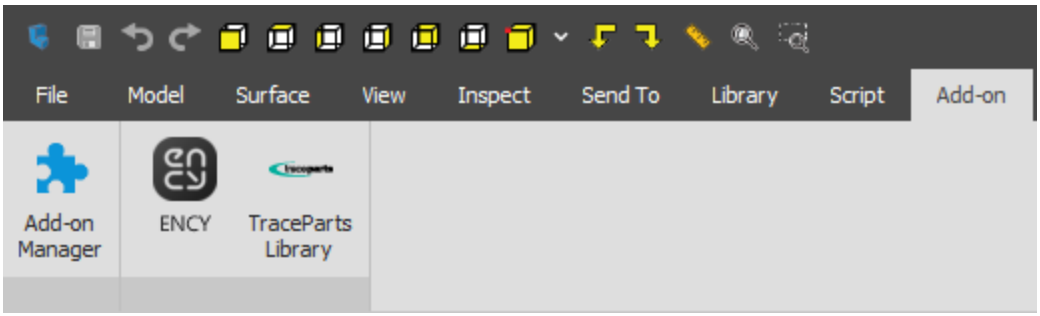
The addin allows you to import FreeCAD™ project files.

Supported file extensions: FCSTD, BREP, BRP, DAT, SVG, SVGZ, UNV, MED, BDF, IFC, IV, AST, BMS, OBJ, OFF, PLY, OCA, GCAD, CSG, ASC, POV, INC. The required application (FreeCAD™) must be installed on your computer for this option to work.

Alibre Design™ toolbar & import addin

Alibre

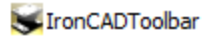
The toolbar allows you to export geometric data from Alibre Design™. You can view the version of the supported CAD system. Details



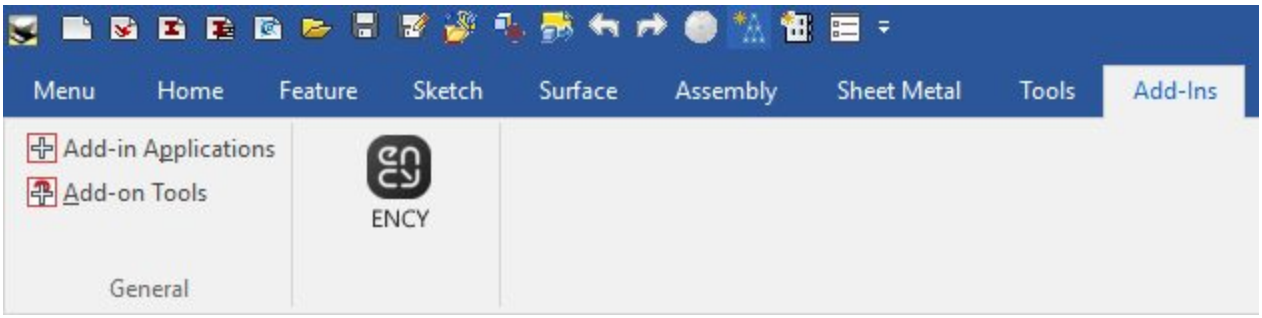
The addin allows you to import Alibre Design™ project files.

Supported file extensions: AD_PRT, AD_ASM, AD_SMP. The required application (Alibre Design™) must be installed on your computer for this option to work.

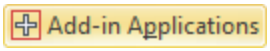
IronCAD™ toolbar



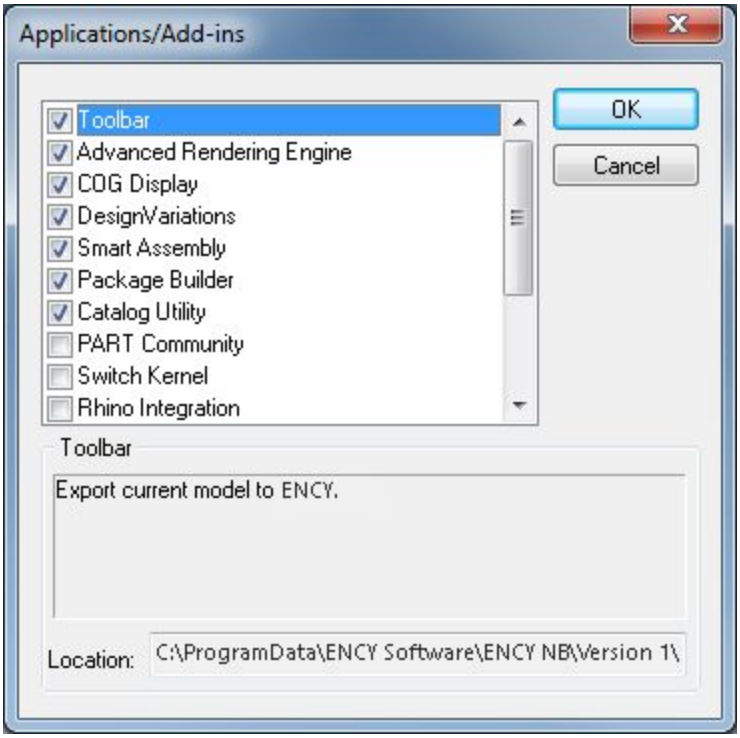
The toolbar allows you to export geometric data from IronCAD™. You can view the version of the supported CAD system. Details



Once installed, the toolbar may not appear in IronCAD™. Then you need to press



in the dialog box to activate it:

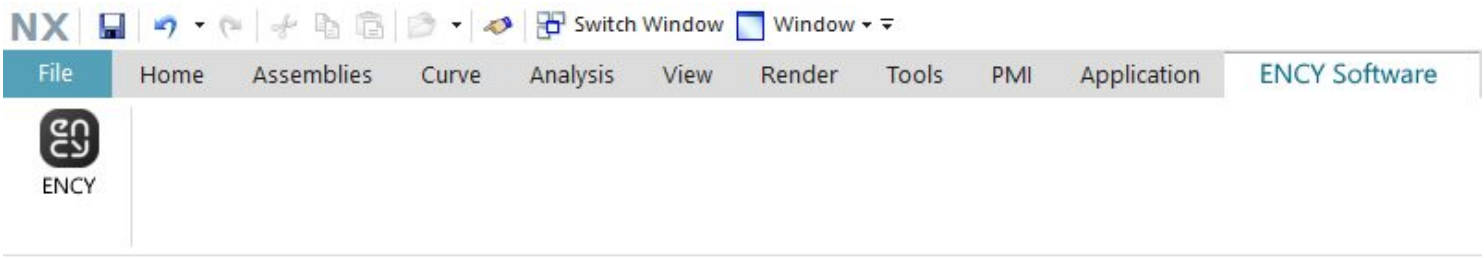


Supported file extensions: ICS, IC3D, ICSW, ICD, EXB. The required application (IronCAD™) must be installed on your computer for this option to work.

NX™ toolbar & import addin

NX

The toolbar allows you to export geometric data from NX™. You can view the version of the supported CAD system. Details



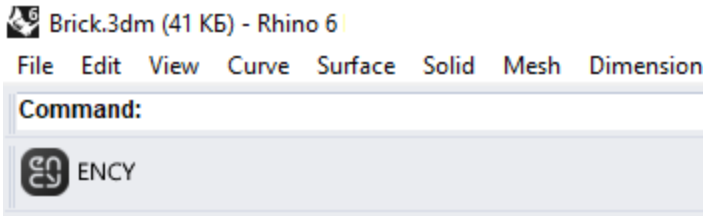
The addin allows you to import NX™ project files.

Supported file extensions: PRT. The required application (NX™) must be installed on your computer for this option to work.

Rhinoceros toolbar & import addin

Rhinoceros

The toolbar allows you to export geometric data from Rhinoceros™. You can view the version of the supported CAD system. Details



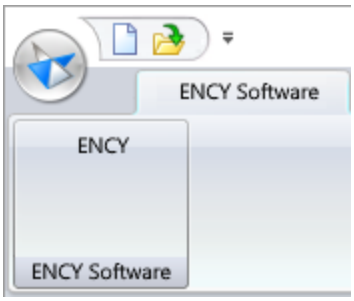
The addin allows you to import Rhinoceros™ project files.

Supported file extensions: 3DM, RWS, 3DS, STP, STEP, RAW, WRL, VRML, AI, EPS, LWO, SPL, VDA, DWG, DXF, DGN, SLDPRT, SLDASM. The required application (Rhinoceros™) must be installed on your computer for this option to work. The Rhinoceros™ application must be running and closed at least once in order for the data to be recorded in the registry. Otherwise, the toolbar will not be able to install.

SolidEdge™ toolbar & import addin

Solid Edge

The toolbar allows you to export geometric data from SolidEdge™. You can view the version of the supported CAD system. Details

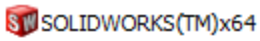


The addin allows you to import SolidEdge™ project files.

Supported file extensions: ASM, DFT, PAR, PSM, MDS, PWD, DGN, DXF, DWG, PRT, SAT, STP, STEP, X_B, X_T.

The required application (SolidEdge™) must be installed on your computer for this option to work.

SOLIDWORKS™ toolbar & import addin



The toolbar allows you to export geometric data from SOLIDWORKS™. You can view the version of the supported CAD system. Details



The addin allows you to import project files SOLIDWORKS™.

Supported file extensions: SLDASM, ASM, SLDPRT, PRT, SLDDRW, DRW, X_B, X_T, STP, STEP. Starting in July 2026, a separate license will be required to use the SolidWorks module (see Changes to ENCY–SOLIDWORKS Integration Licensing – ENCY CAD/CAM software), The required application (SOLIDWORKS™) must be installed on your computer for the correct work of this option.

SpaceClaim™ toolbar & import addin

SpaceClaim

The toolbar allows you to export geometric data from SpaceClaim™. You can view the version of the supported CAD system. Details



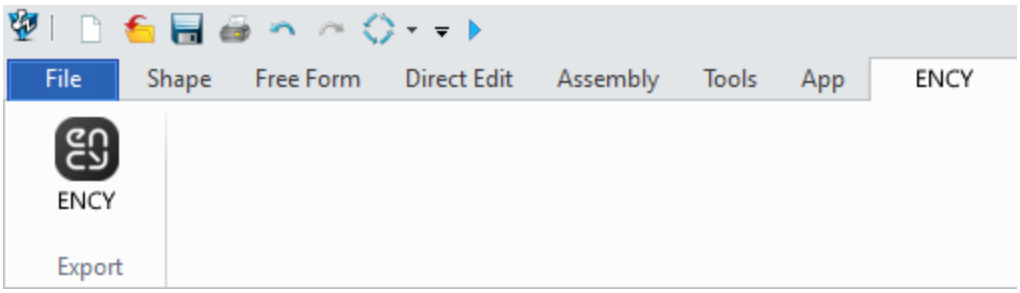
The addin allows you to import SpaceClaim™ project files.

Supported file extensions: SLDASM, ASM, SLDPRT, PRT, SLDDRW, DRW, X_B, X_T, STP, STEP. The required application (SpaceClaim™) must be installed on your computer for the correct work of this option.

ZW3D™ toolbar & import addin

ZW3D

The toolbar allows you to export geometric data from ZW3D™. You can view the version of the supported CAD system. Details



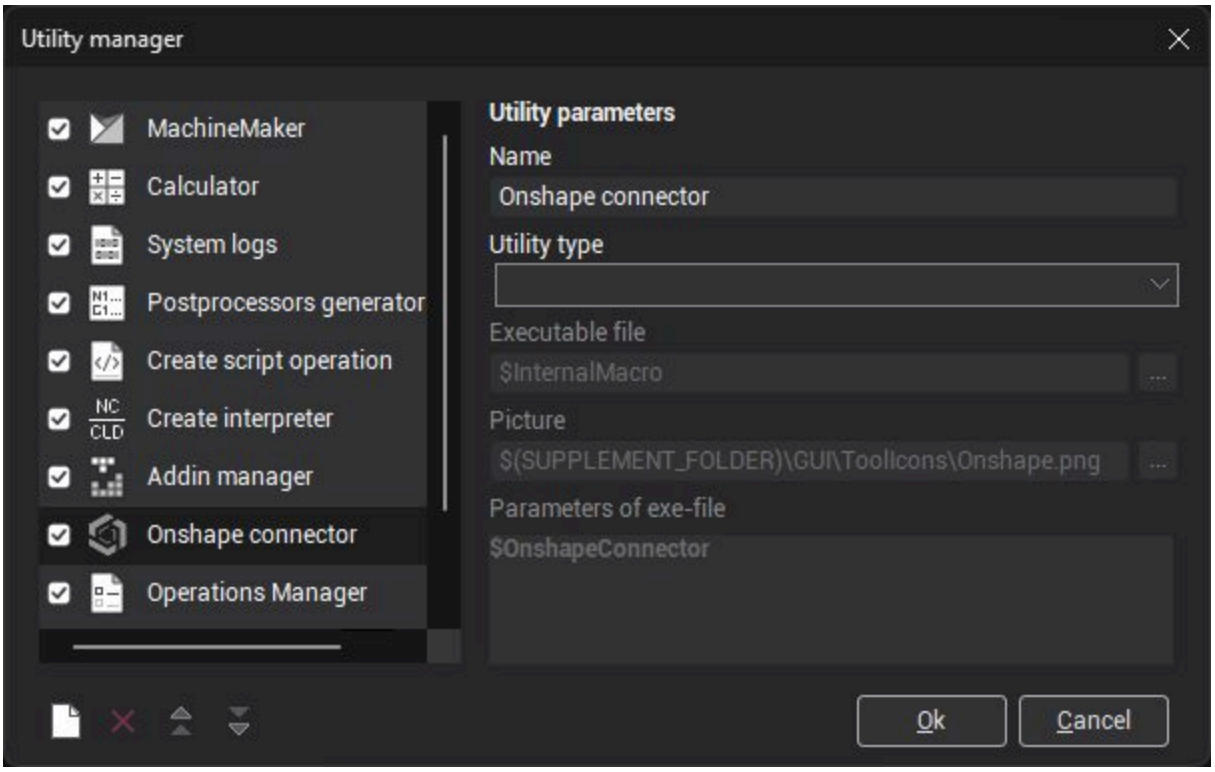
The addin allows you to import ZW3D™ project files.

Supported file extensions: Z3. The required application (ZW3D™) must be installed on your computer for this option to work.

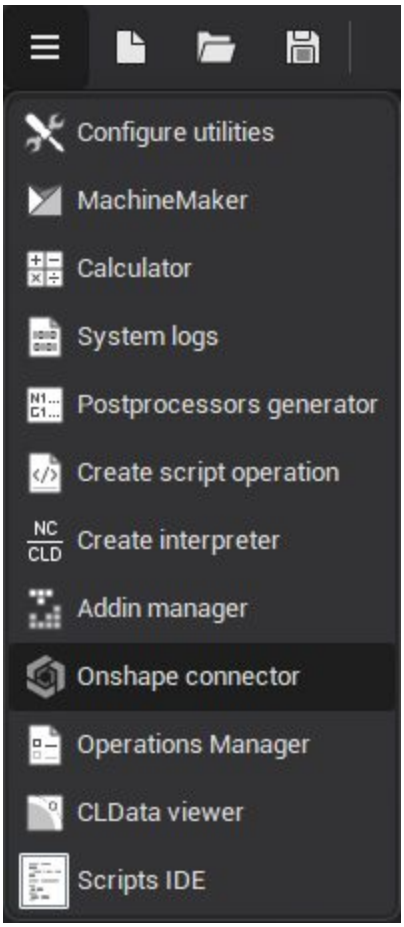
Onshape™ connector

Onshape™ is a new generation of full-cloud CAD designed specifically for modern agile design teams. You can view the version of the supported CAD system. Details

To use the Onshape connector plugin, it must first be enabled in the Configure utilities window.



Once enabled, the plugin can be launched by clicking the Onshape icon on the toolbar.

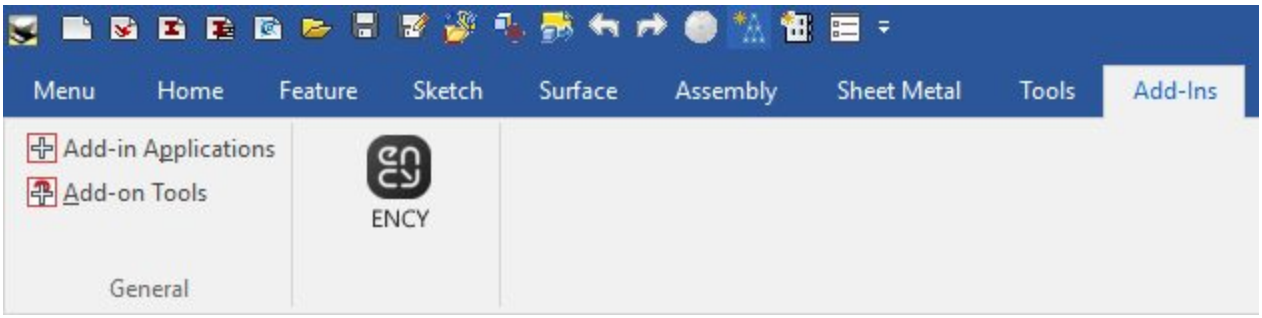


After signing into your Onshape account, the plugin will display a list of available models. Selecting a model from this list will initiate the import process. If the imported model is later updated in Onshape, the connector plugin will prompt you to reimport the model. The Onshape connector plugin maintains model associativity, ensuring updates are reflected in your workspace.

CAXA 3D™ toolbar

CAXA 3D

The toolbar allows you to export geometric data from CAXA 3D™ to CAM system. You can view the version of the supported CAD system. Details



Once installed, the toolbar may not appear in CAXA 3D™. Then you need to press

Add-in Applications

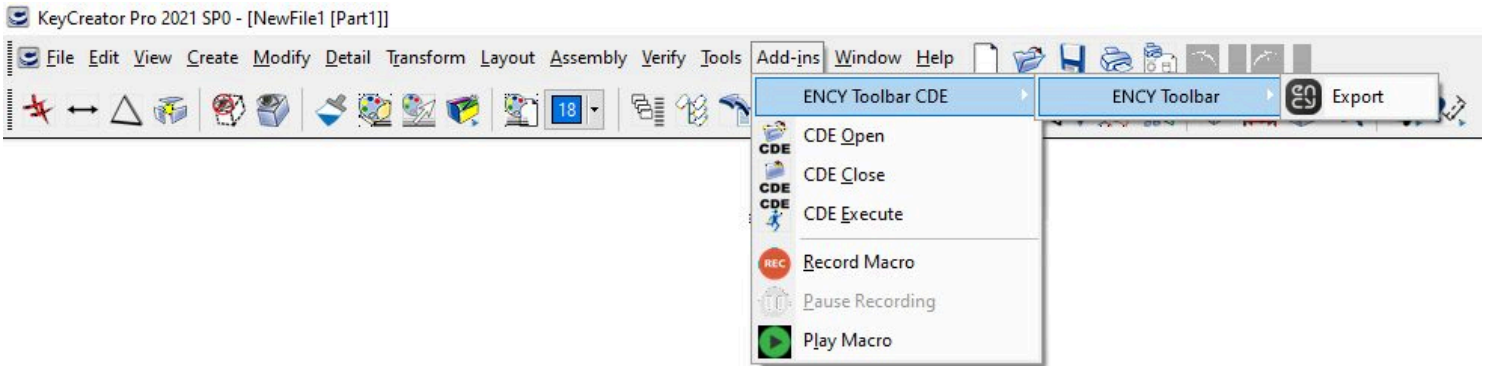
in the dialog box to activate it:

Supported file extensions: ICS, IC3D, ICSW, EXB. The required application (CAXA 3D™) must be installed on your computer for this option to work.

KeyCreator™ toolbar

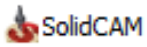
KeyCreator

The toolbar allows you to export geometric data from KeyCreator™. You can view the version of the supported CAD system. Details

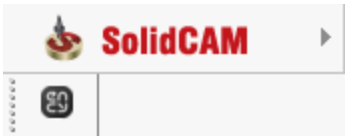


Supported file extensions: CKD. The required application (KeyCreator™) must be installed on your computer for this option to work.

SolidCAM™ toolbar & import addin



The toolbar allows you to export geometric data from SolidCAM™. You can view the version of the supported CAD system. Details



The addin allows you to import project files SolidCAM™.

Supported file extensions: SLDASM, ASM, SLDPRT, PRT, SLDDRW, DRW, X_B, X_T, STP, STEP. The required application (SolidCAM™) must be installed on your computer for the correct work of this option.

Associativity for Addin`s

According to the new changes, associativity means that during the model updating by addin's, the job assignment is also updating.

Note NX™ ✓ Onshape™ ✓ SOLIDWORKS™ ✓ SpaceClaim™ ✓

Editing geometrical model

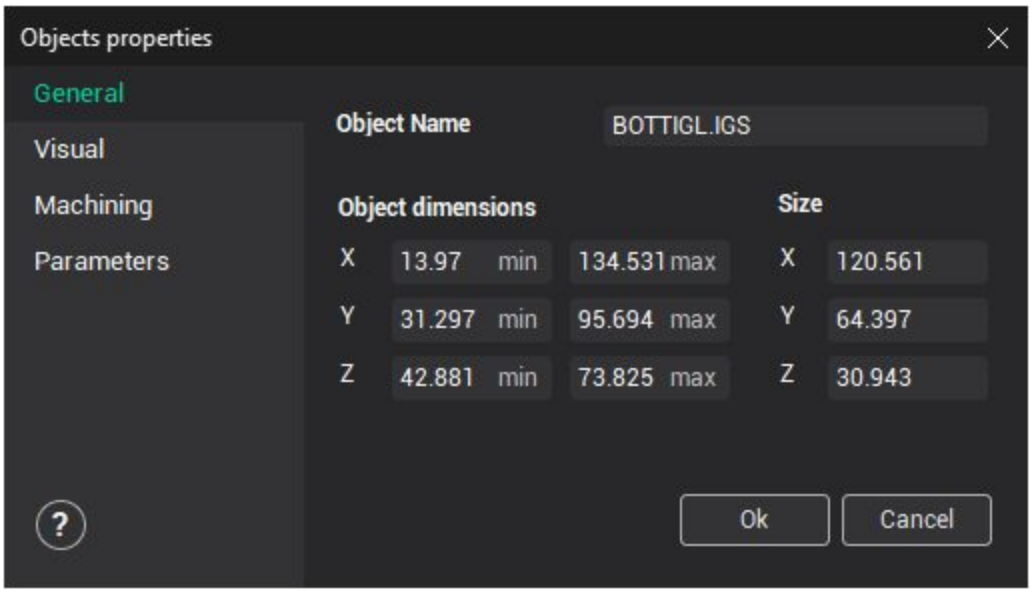
This section covers geometrical model editing: object properties and display, deletion and transformations, boundary projection, curve joining, triangulation, text creation, intersection and face sewing, 3D model export and simplification, hole patching, isoline extraction, spline editing, and surface extension.

Geometrical object properties

The Properties window is opened by pressing the ⚙ button on the Model tab or from the context menu in the graphic window or model structure window This window allows the viewing of general properties and to change visual and machining properties of objects. The window consists of four tabsheets:

- General;
- Visual;
- Machining;
- Parameters.

General tab

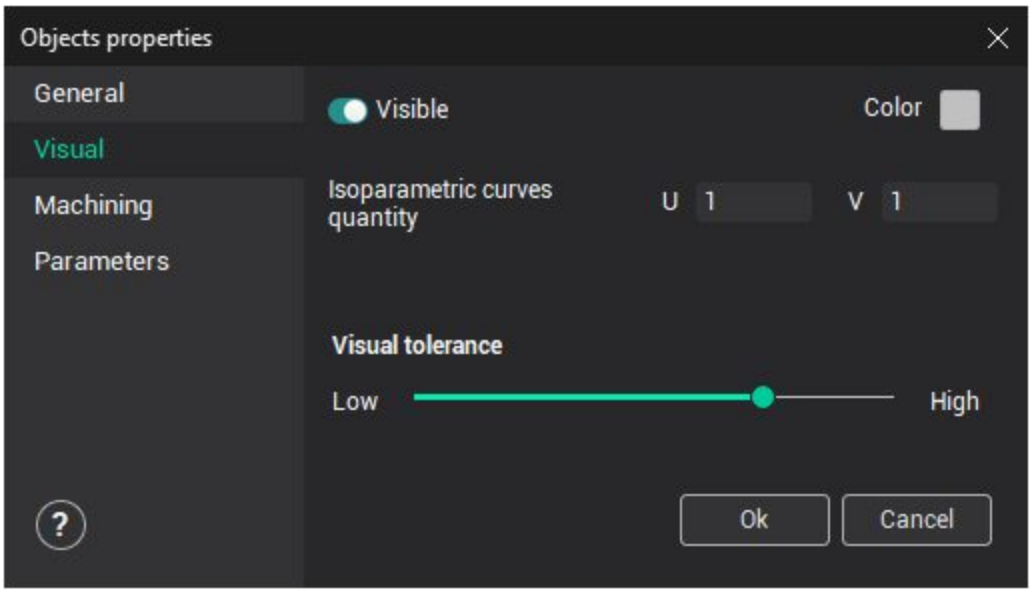


On the general properties tab, if an object is selected, its name can be changed. Displayed also are the minimum and maximum coordinates of the selected objects along each of its axes.

Object Name – name of the selected object. If several names are selected, then the field will be empty.

Object dimensions – overall dimensions of the selected object.

Visual tab



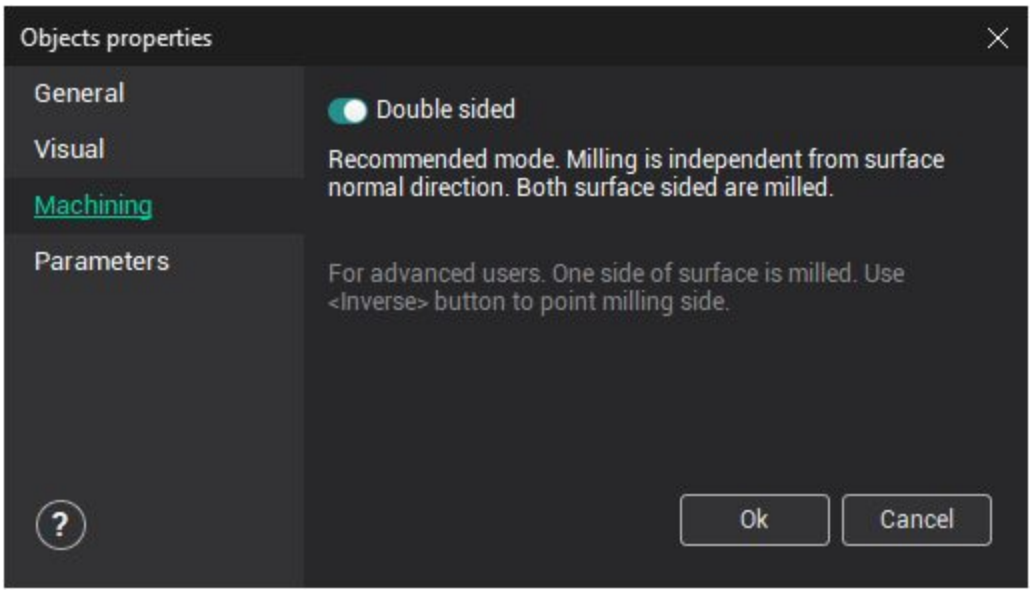
Access to the visual properties of objects is duplicated on the visual properties tab: visibility and color. It is also possible to assign the number of isoparametric lines and the object's visual tolerance in the graphic window.

- Visible – if unchecked, then the selected object will not be displayed.
- Color – allows changing the color of the selected object.
- Isoparametric curves quantity – when displaying surfaces it is sometimes necessary to define the number of displayed isoparametric curves. On this tab it is possible to define the number of curves by adjusting U and V parameters. When the value is zero, isoparametric curves are invisible, when it is one – surface borders are visible, when it is two – every surface segment is divided by two etc. When displaying curves and surfaces on the screen, the system approximates the curve by using lines and surfaces by flat polygonal edges.
- Visual tolerance – allows the user to set the visual quality of 3D objects, or to find a compromise between satisfactory visualization quality and computer speed. The tolerance is adjusted using the slider bar control. Tolerance in this case is the maximum approximated deviation of sections used when drawing the curves on the screen. The higher the visualization tolerance of 3D objects, the more memory resources will be taken to draw them on the screen.

Note: It is not recommended to set high visual tolerance on slower computers. Computer performance may be affected.

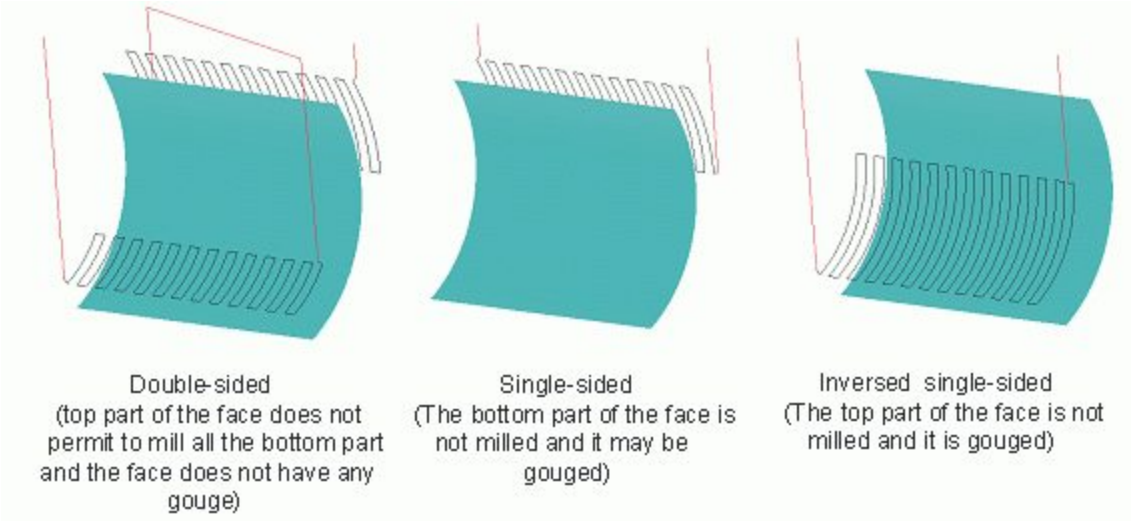
The original visual tolerance, when loading the model, is defined in the system settings window.

Machining tab



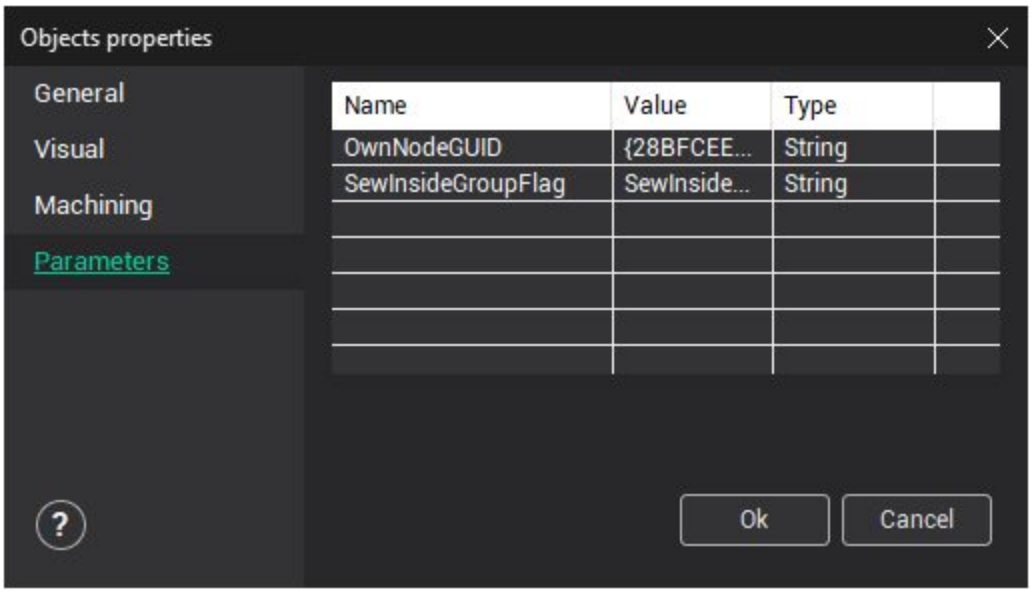
Double sided – allows the user to define the surface type. When loading a geometrical model all surfaces are set as "double sided". This means that surface machining will be performed independently from the normal vector direction – from both sides. Thus, the side of the surface being machined is defined only by its spatial position – the top side will be machined. This mode is recommended to use for surface models. This has very little effect on the calculation time.

The user can also define the side to be machined. To do so, the tick in the Double sided field must be unchecked. In this case the system will allow machining only on the side that the surface normal vector is pointing to. The side to be machined is selected using the invert function. When surface machining in single side mode, the calculation of toolpaths is performed faster than when machining a double sided surface, but it might cut a part of the surface, where the normal is pointing downwards. This mode is recommended for use with 'solid' models, where all normals are pointing outwards or with models with a small number of surfaces.



Attention: Incorrect direction of the normal vectors for non "double-sided" surfaces may cause faulty results during execution of machining operations.

Parameters tab



On the Parameters tab other additional parameters are displayed. The parameters are imported or defined during designing.

Changing visual properties

The button allows the user to manage the visibility of the selected objects on the screen. When pressed, if a group is selected and at least one object of the group is visible, then all subgroups and geometrical objects of the group become invisible.

The  button allows the user to redefine the color of the selected objects. When this button is pressed, the standard color selection dialogue opens.

Visual properties of objects can be also changed in the properties window.

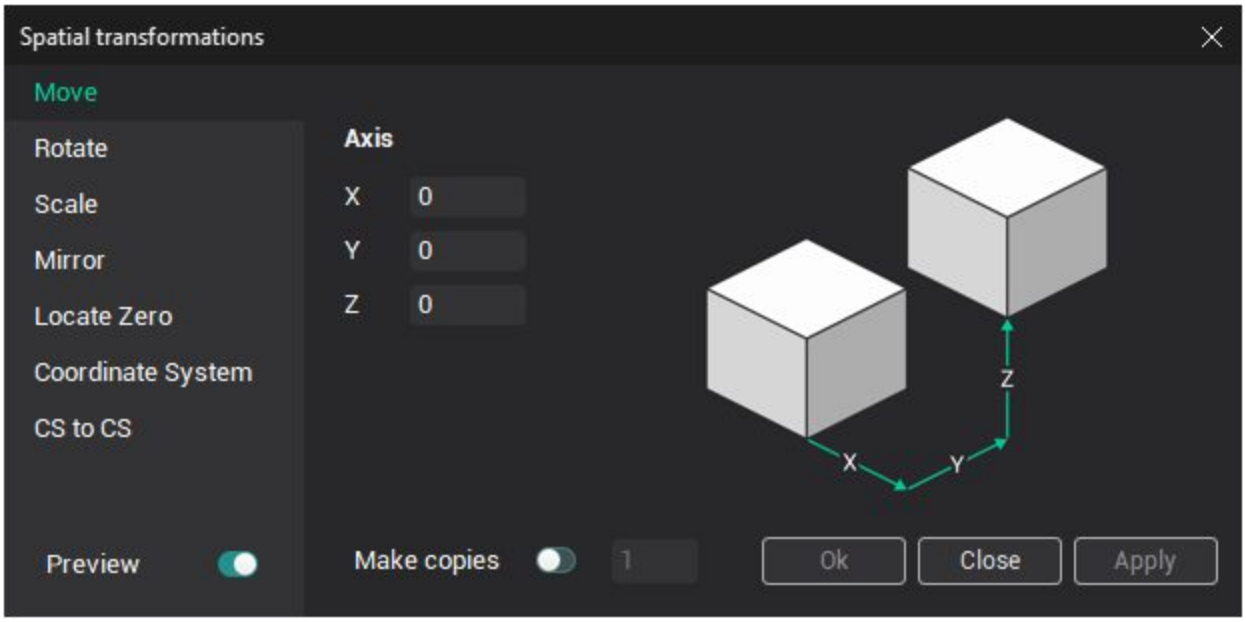
Delete

The ✕ button deletes the selected objects. If the objects to be deleted are used in a machining operation, then confirmation will be requested. The function available on [Del] key too.

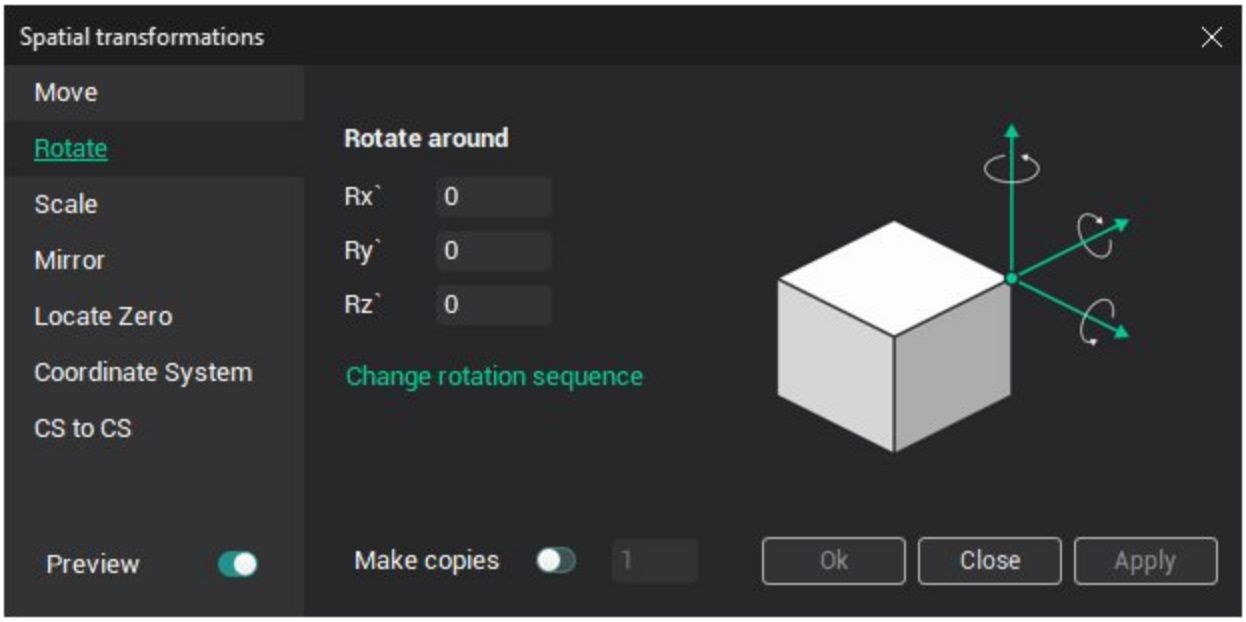
Spatial transformations

A wider range of transformation methods of selected geometrical objects is available in the Spatial transformations window. The window opens when the  button is pressed.

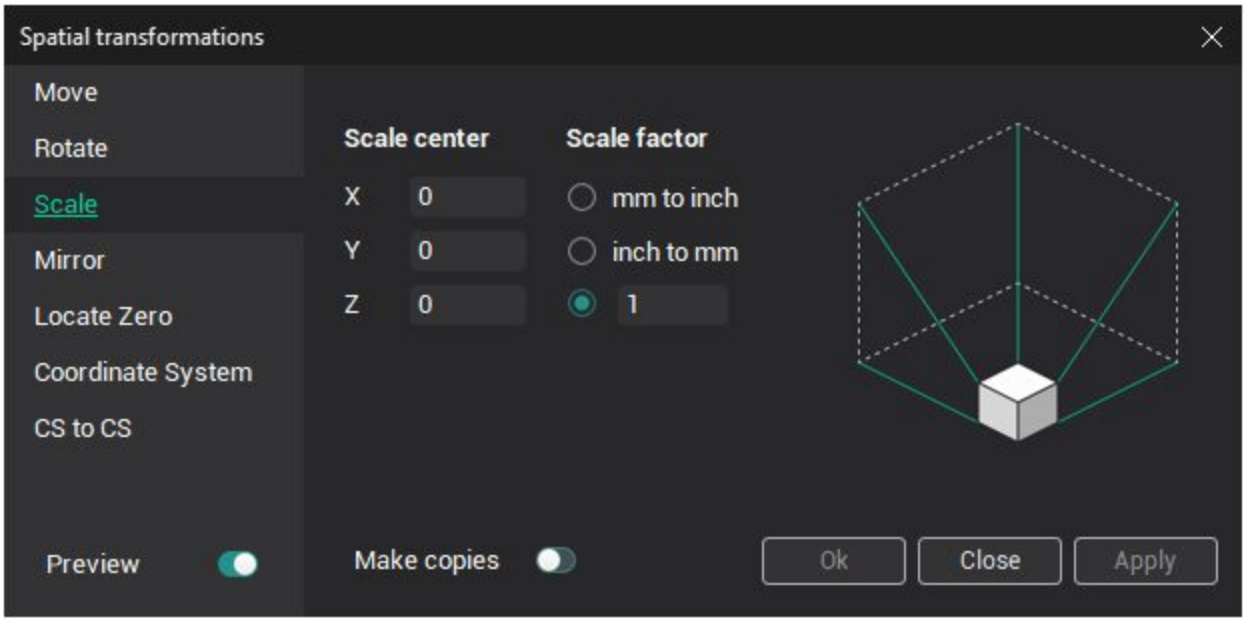
- On the Move page, the user can define the parallel transition of an object. In fields X, Y, Z setting shift values by axes. If there is no checkmark in the field Make copies, then the selected object will be transferred by the defined distance along each of respective axes. If the checkmark in that field is set on, then the selected object will be copied to the defined place. It is possible to assign a number of copies. For instance, if the number of copies is set as two, then the second copy will be created at the transition distance from the first one.



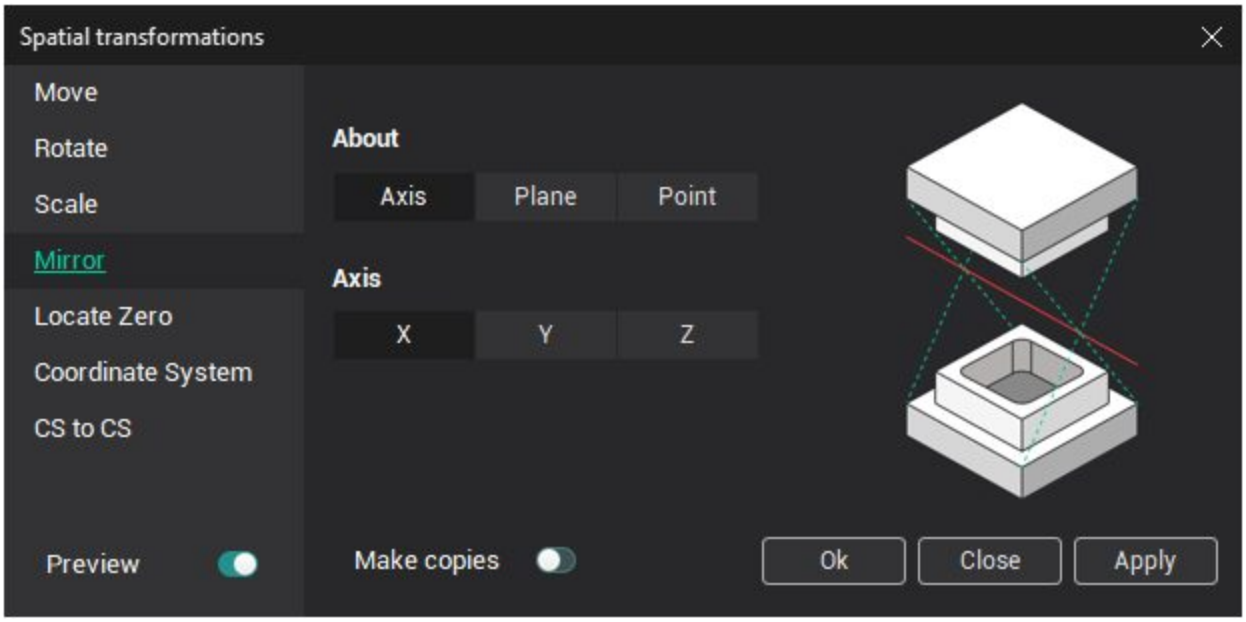
- On the Rotate page the user can rotate selected objects round the selected axis to the defined angle. The angle is assigned in degrees. Working with copies is incremental, that is, every subsequent copy is obtained by rotation of the previous one around the defined axis to the defined angle.



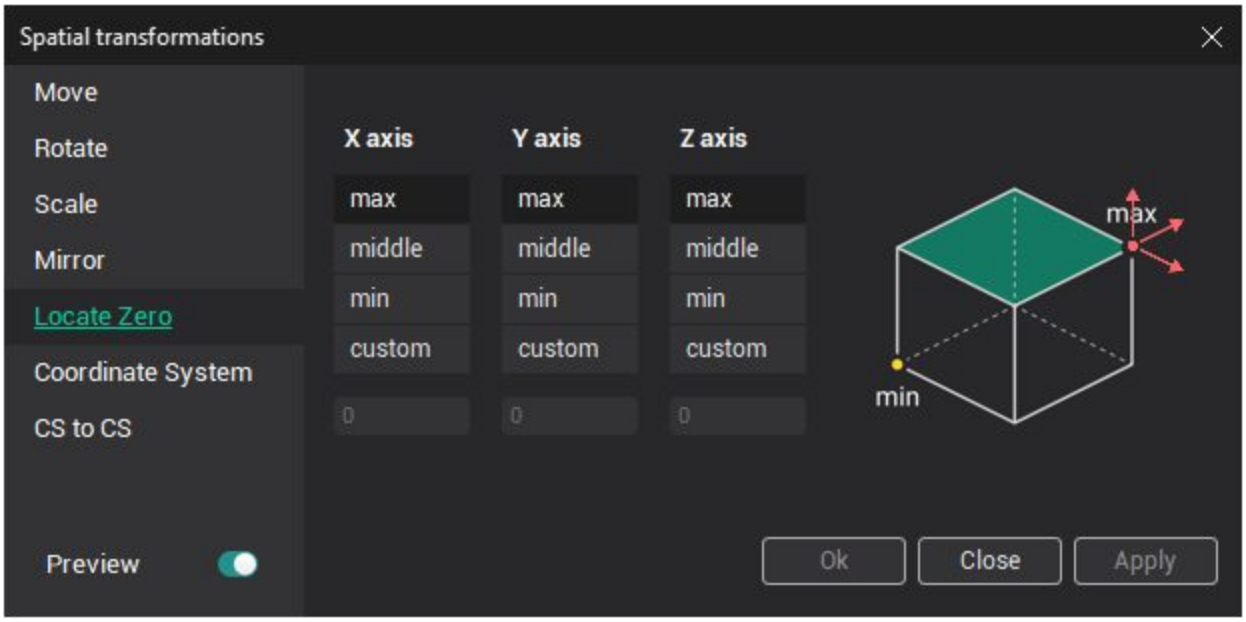
- On the Scale page the user can enlarge or decrease selected objects. In the field Scale center, the coordinates of the center point of scaling are defined. Scale factor can be one of two default: mm to inch or inch to mm, or arbitrary value: Other. When assigning a coefficient of scaling of more than one, then objects will be enlarged. If a coefficient of scaling is less than one, then objects decrease in size accordingly.



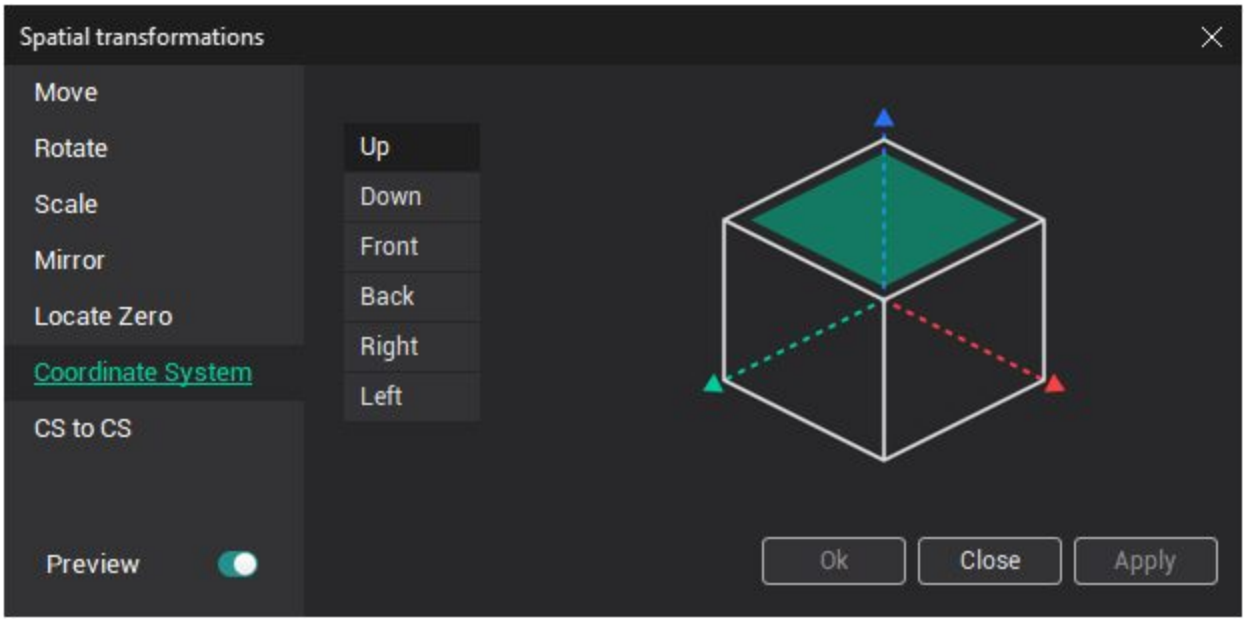
- On the Mirror page the user can obtain an object symmetrical to the selected one relative to an axis, plane or point.



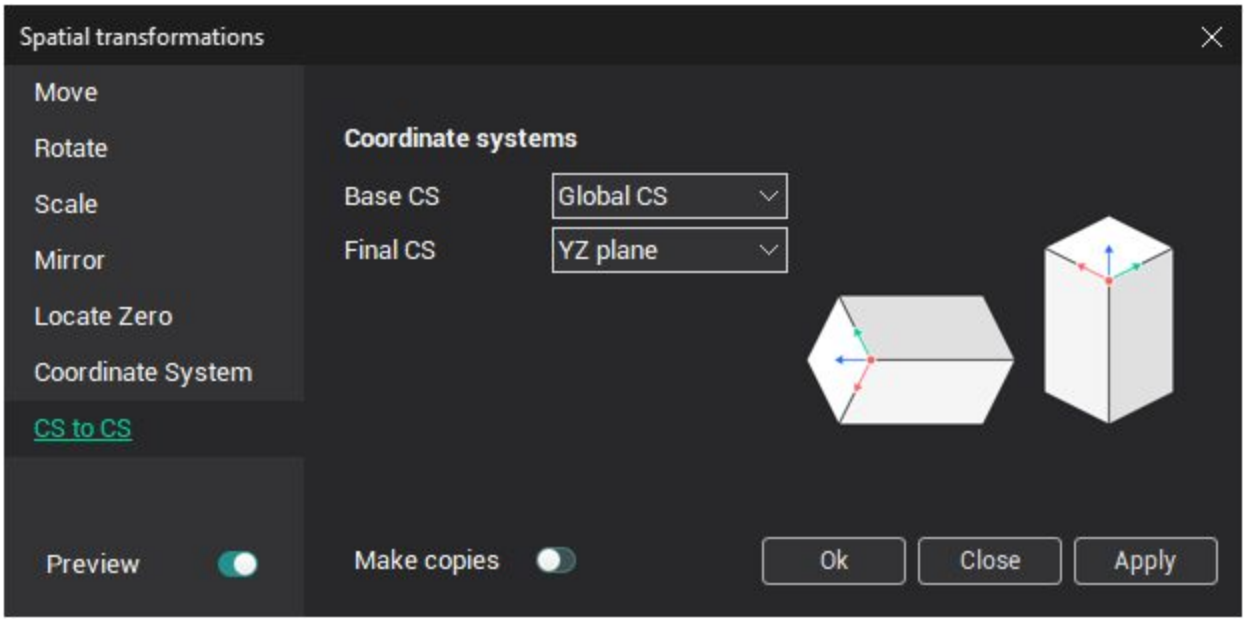
- On the Locate Zero page the user can perform a parallel transition of an object according to its spatial dimensions. The selected geometrical object will be shifted for such values along axes to have the transformed object relatively the current coordinate system according to the defined parameters.



- On the Coordinate System page the user can turn an object, so that the selected edges are on the top. Also the model can be transformed by the way that constructed plane will be combined with XY plane of global coordinate system.



- On the CS to CS page the user can transform an object located in one Base CS coordinate system into another Final CS coordinate system.



Upon pressing the OK button, the selected transformation will be applied and the window closes automatically. Upon pressing the Cancel button, the window closes without applying the transformations made. Upon pressing the Apply button, all transformations will be applied to the selected objects, but the window will remain active.

Inversion

Inverting normals of surfaces can be executed by selecting Inverse from popup menu on selected objects in geometrical structure window.

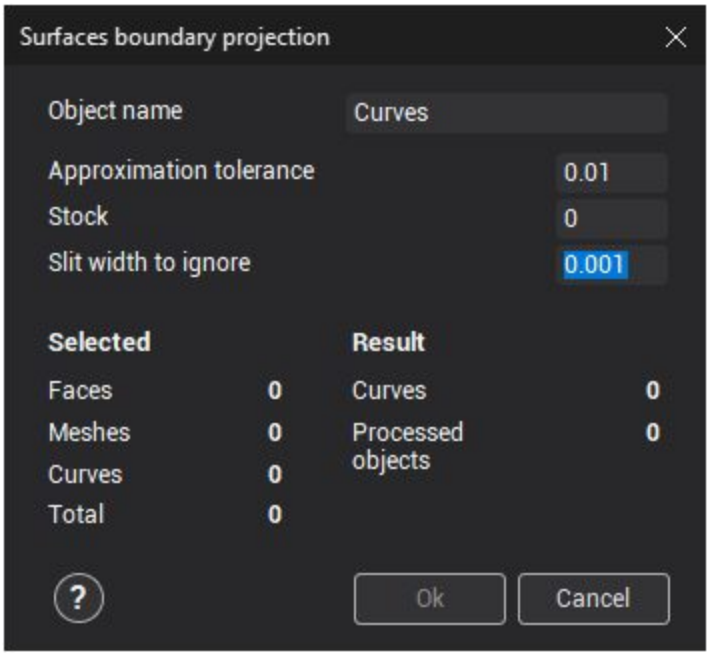
For non "double sided" surfaces (the Machining page of the Object properties window), the side of the surface to be machined can be defined by this function. The surface will be machined from that side only, where the normal vector is pointing. To create the correct NC program it is necessary that normal vectors of all its elements have the direction from the detail.

For "double sided" faces (all faces are "double sided" after the import process as default), the direction of the normal is unimportant. Use of the inversion function does not affect the tool movement toolpath.

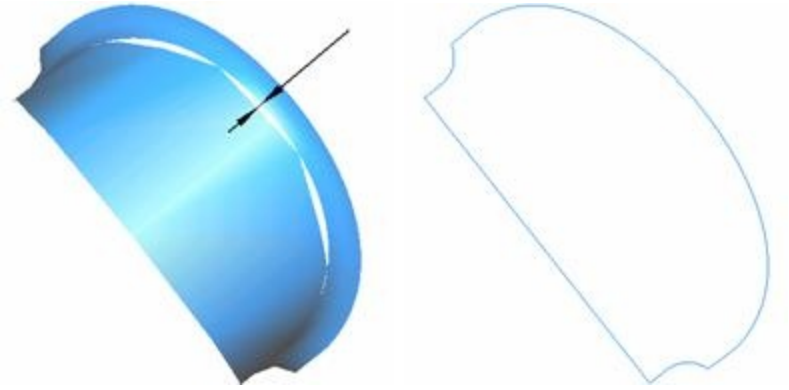
Attention: Incorrect direction of the normal vectors for non "double-sided" surfaces may cause faulty results during execution of machining operations.

Surfaces boundary projection

The  button opens the Surfaces boundary projection window. The function is used for the construction of outer enveloping projections of the selected surfaces and meshes onto the XY plane of the current coordinate system. In addition to surfaces borders the function allow to project ordinary curves on the plane. The curves it creates can be used when assigning parameters for machining operations.



- Object name – the name of the resulting curve. If several curves are created as the result of projection, then they will be put into a new group with the name defined in this field. The new curve or group will be created in the currently active group.
- Approximation tolerance – maximum outer deviation of the resulting curves from the surface borders. For ordinary curves this value used as approximation tolerance by arcs.
- Stock – offset value for the resulting curves away from the surface borders. Positive value gives an outwards offset from the surface, negative – inwards (equivalent to equidistant curve projection). For ordinary curves will built offset on stock value.
- Slit width to ignore – maximum value of gaps between surfaces which will be ignored. If surfaces are joined with high accuracy, then the surfaces contour will be projected, if with low – then any gaps between neighboring surfaces will be included in the projected curve.



The panel Selected displays the type and the number of object selected for the boundary projection.

- Faces – number of surfaces selected.
- Meshes – number of meshes selected.
- Curves – number of curves selected.
- Total – total number of geometrical objects selected.

The panel Result shows the number of objects selected for the projected boundary operation, and the number of curves that will be created (if the projected objects are very complex, projection calculation may take some time).

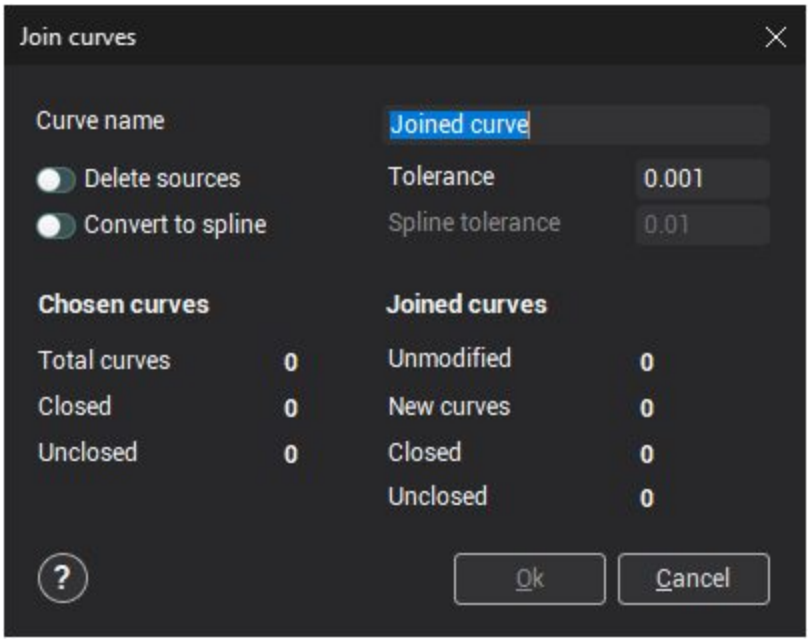
- Curves – number of obtained curves.
- Processed objects – total number of processed objects.

When changing any projection parameters, the values in the Result panel will automatically be recalculated.

If the results of the defined parameters is correct, then the window should be closed using the Ok button. The boundary projection of the selected objects will be put into the active group. To cancel the projection function, press the Cancel button.

Join curves

The  button opens the Join curves window.



Sometimes, when importing curves, the file contains non-joined curves, but the contour is split into several separate sections. When working with these contours, the separate sections require to join. The curve joining function allows users to obtain a joined curve by linking neighboring curve sections.

- Curve name or Folder name – name of the new curve or folder. If as the result of curve joining several curves will be obtained, then they will be put into the newly created folder with the name defined in this field. The new curve or group will be created in the currently active group.
- Delete sources – a tick in this field means that when the joining operation is completed, all source objects will be deleted.
- Tolerance – maximum distance between ends of neighboring curves which can be joined. By altering the tolerance value one can achieve the desired result for joining (the ends of imported curves are often not coincident with each other).
- Convert to spline - will allow with a given tolerance(by parameter Spline tolerance) convert joined curves to spline.

The panel Chosen curves shows the number and type of source curves.

- Total curves – total number of selected curves.
- Closed – number of closed curves.
- Unclosed – number of unclosed curves.

The panel Joined curves shows the number and type of obtained curves.

- Unmodified – number of curves left without modification.

- New curves – total number of new curves created.
- Closed – number of obtained closed curves.
- Unclosed – number of obtained unclosed curves.

When the joining tolerance is changed, the field values in Joined curves will be automatically recalculated.

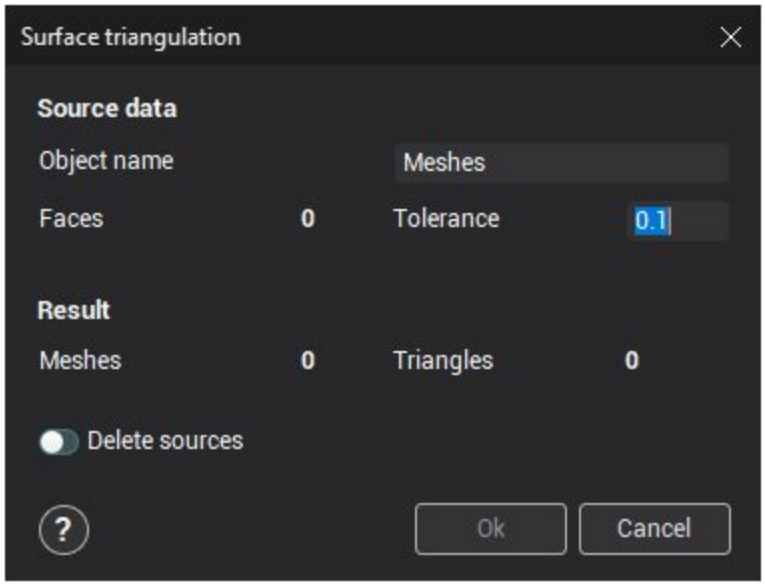
If the result of the defined parameters is correct, then the window should be closed using the Ok button. The joined curve or a group of curves will be put into the active group. Source curves will be deleted if Delete sources was selected.

To cancel performing the joining function, press the Cancel button.

Surface triangulation

The Surface triangulation window opens from the pop up menu of the graphic window or when the  button is pressed.

The function is designed for the alteration of selected surfaces tolerance. Used in cases when a machining operation is performed with a tolerance that is smaller than the tolerance of the surface itself, or when it is impossible to machine the detail due to problems arising because of incorrect model construction – spiral transitions, needle surfaces etc.



The Source data panel shows the type and number of objects selected for triangulation and other source data.

- Object name – name of the resulting surface mesh. If as the result of triangulation several surfaces meshes are obtained, then they will be put into a newly created group using the name defined in this field. The new surface or group will be created in the currently active group.
- Faces – number of selected surfaces.
- Tolerance – maximum deviation of the resulting surface from the source one. By altering the tolerance value, one can achieve the desired surface tolerance.

The Result panel shows the number and type of newly created objects. Triangulation may take some time should the selected objects be considerably complex; therefore job completion percentage may be shown.

- Meshes – number of obtained meshes.
- Triangles – total number of triangles in meshes.

When changing the triangulation tolerance, the field values in the Result panel will automatically be recalculated.

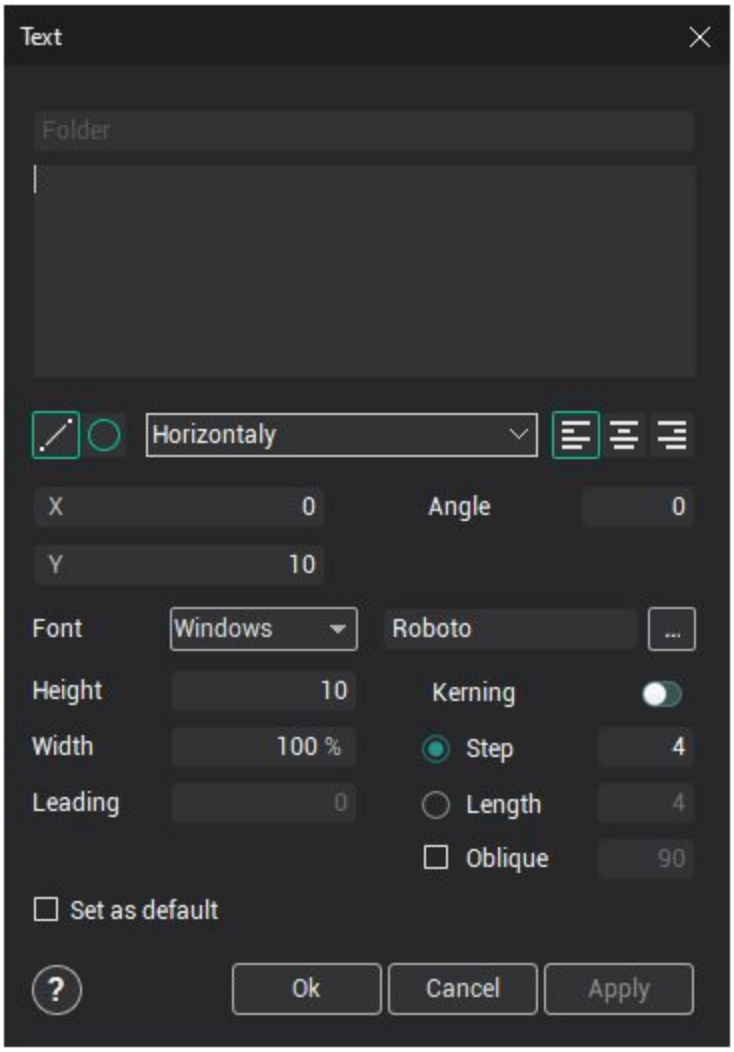
Delete sources – a tick in this field means that when finishing the triangulation operation, all source objects will be deleted.

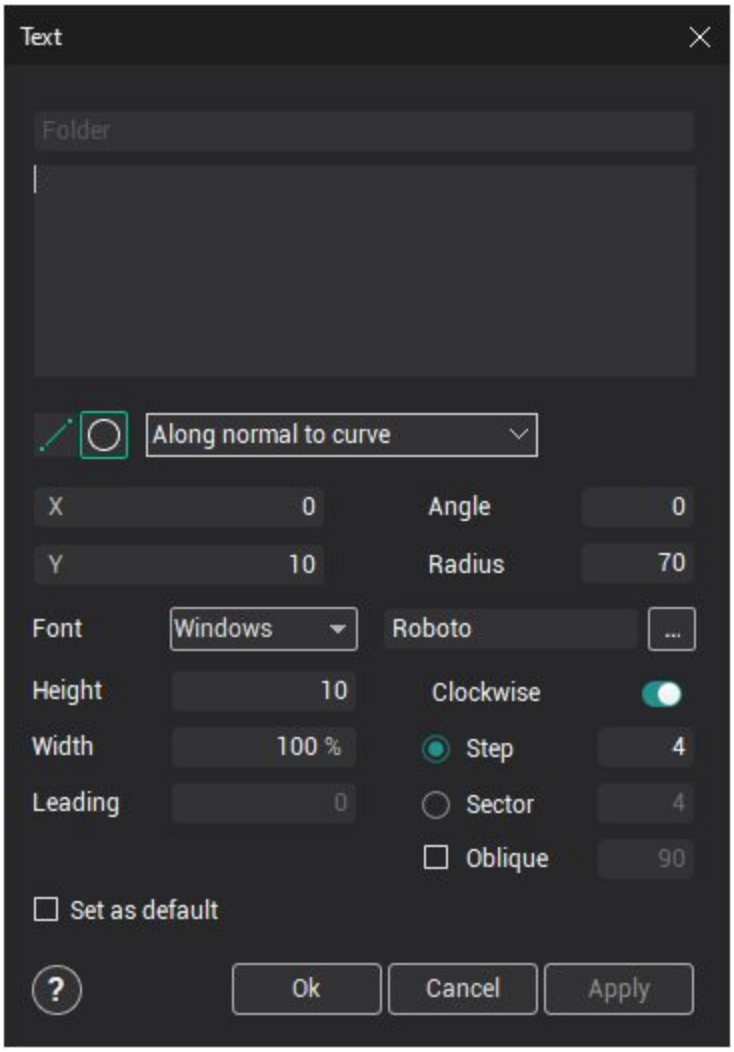
If the triangulation results are correct, then the window should be closed using the Ok button. The obtained surface or group of surfaces will be put into the active group. The source objects will be deleted if Delete sources has been selected.

To cancel performing triangulation, press the Cancel button.

Creating text

The text creation window can be opened either by pressing the  button on the Model panel, or by using the pop up menu in the graphic window or on the geometrical structure window.





Text can be typed along a line  (default), or along a circle .

- 

define letter orientation.
-  - Text alignment is define text placement from the start point.

To change text font it is necessary use the font open dialog button  or font type combo box.

X,Y - define start point position.

Angle - define angle between horizon and text direction line for placement text along line or define the start angle for place text along circle.

Radius - define circle radius.

Height - define letters height.

Width - allow to change letter width in %.

Leading - define vertical space between text rows.

Kerning - allow to use kerning tables from the selected font for calculate distance between letters.

Step - define distance between letters ignore kerning information.

Length - define common width of text.

Oblique - define slope angle of the letters in degrees.

Set as default - store parameters as default for a new text.

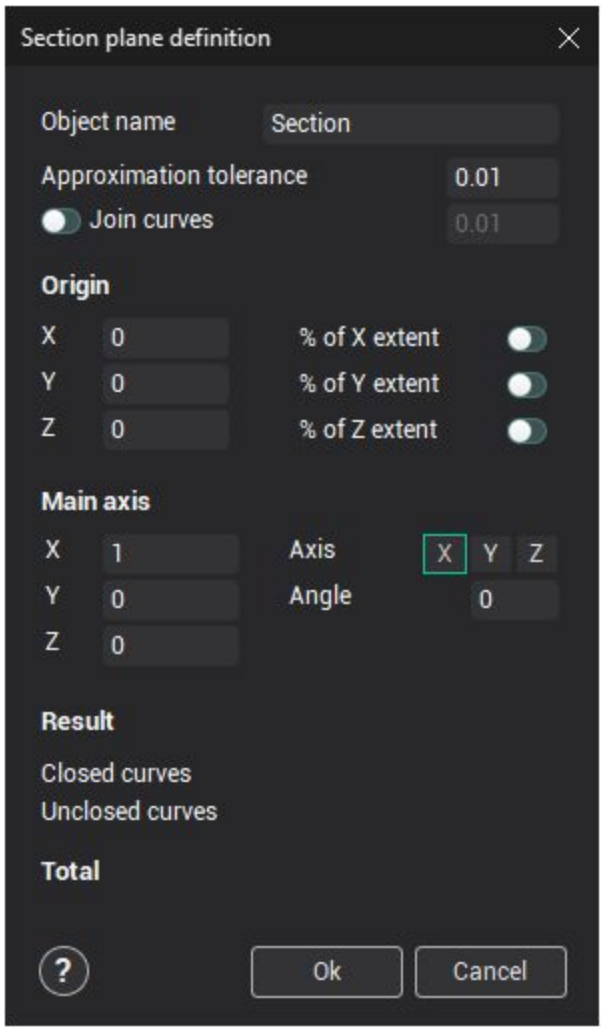
To preview the results press the Apply button. If all parameters assigned were correct, the text will be displayed in the graphic window. If required, the text parameters can be corrected.

Having assigned the text parameters presses the Ok button. At folder with the name defined in the Folder field will be created in the model tree.

Intersect

A section is a parametrical element of the geometrical model. The section object is added into a folder of the geometrical tree to automatically receive intersection curves of all faces and meshes from the folder with the defined section plane.

Press the  button in the Model mode to create the section object for the active folder. After that the section plane definition mode will be activated and the plane parameters window will be shown.



The section plane is defined by the origin point, main axis direction and rotation angle around the axis.

Coordinates of the origin point can be defined on the Origin panel by absolute coordinates or in percentage of overall dimensions of the intersected model. The associated fields must be checked for the second case.

The origin point can be defined interactively in the graphical window. To do that it is necessary to click once by the left mouse button on the small box of the origin point and to move it into a new place. Click once more on the same mouse button to fix the origin point there.

The main axis direction can be defined by the second point coordinates on fields of the Main axis panel or by axis choosing on the same panel. The main axis direction can be defined interactively too. The interactive method likes to the origin point definition.

The angle value of rotation around the main axis is defined on the Main axis panel or interactively. It is possible to drag the section plane into a new position by one of axes or by any point of the plane.

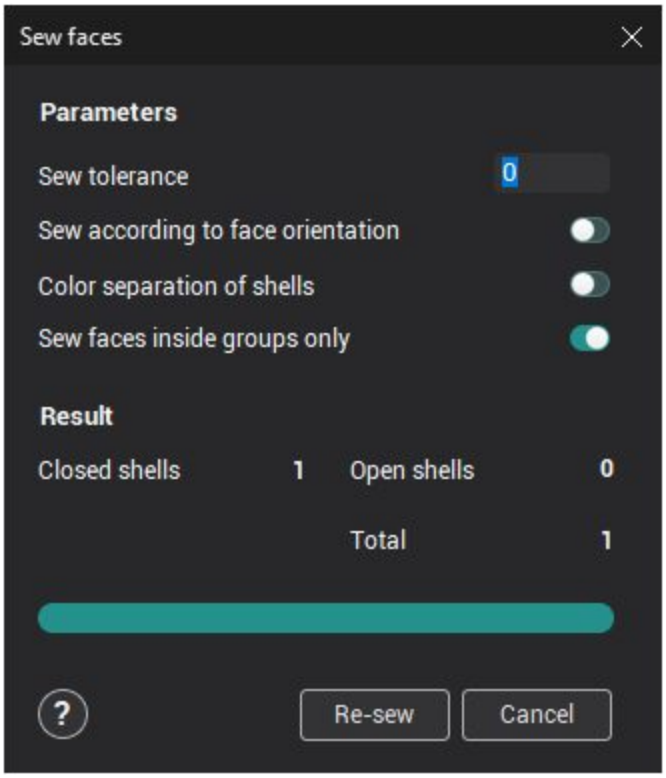
The Result panel shows count of the intersection curves for the current parameters. The curves are calculated with the Approximation tolerance value. If the Join curves option is checked then the system tries to join all result curves with defined tolerance.

The designed section represents as a group of curves. The curves can be modified, copied into another folder or deleted. However the curves will be regenerated when an object from the associated folder is changed.

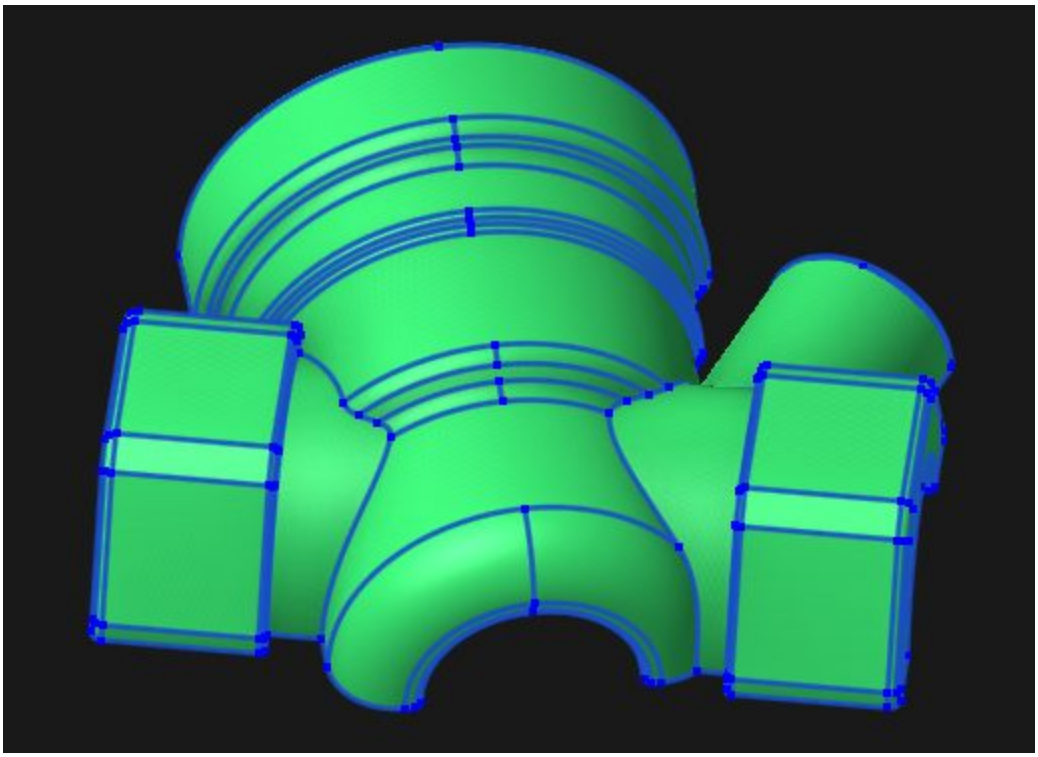
The section parameters can be edited. It is necessary to select the section and choose the Modify Plane item in the popup menu of the Model mode.

Sew Faces

The Sew faces window can be opened either by pressing the  button on the Model panel, or by using the pop up menu in the graphic window or on the geometrical structure window. This function assigned for sewing faces on 3D model together. At that in geometry model added edges and vertices of selected objects.



As result of sewing in geometry model becomes available special parametric objects – edges and vertices. Edges is special curves, that forms borders of faces and meshes. Vertices is points, that comes out from limits of edges.

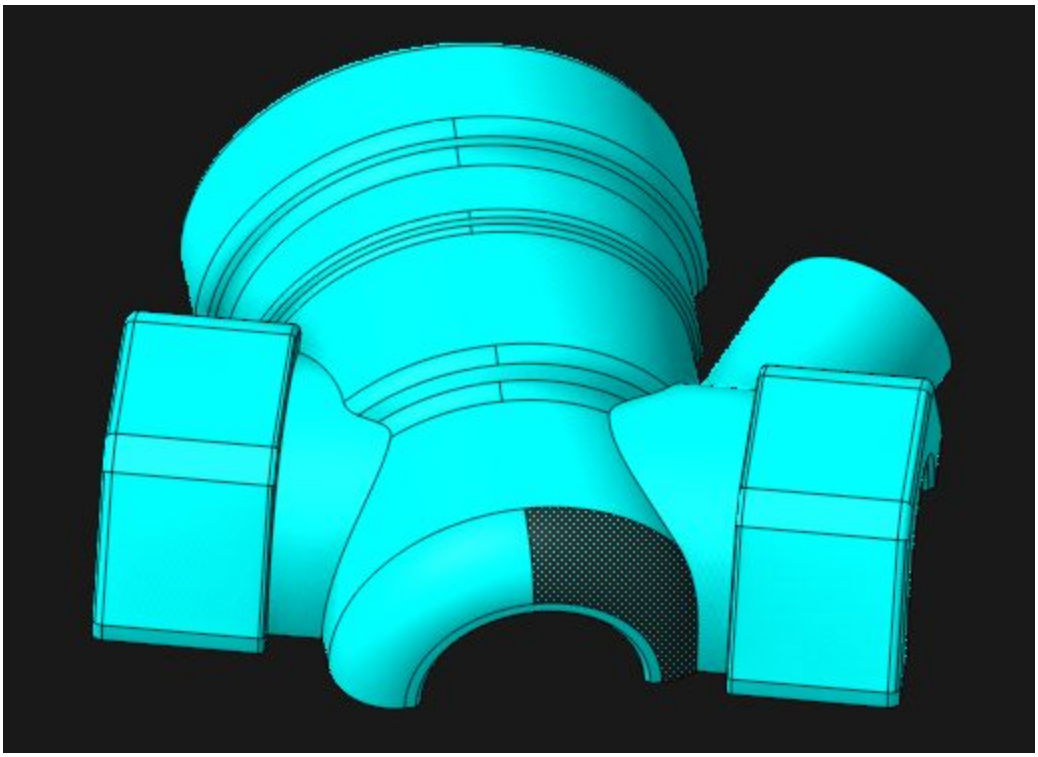


Parametric objects has some peculiar properties. They keeps permanent connection with source object. When this faces is modified, that edges and vertices will be reconstructed too. In consequence there is some limitations on actions with them. Its cannot be copied, deleted, transformed, etc. Therefore this objects is not listed in available objects. However this objects can be selected in graphic window. So, edges, series of edges and vertices can be used in Machining mode as Job assignment, Part, Workpiece, Fixtures in all cases, when allowed addition of edges and vertices.

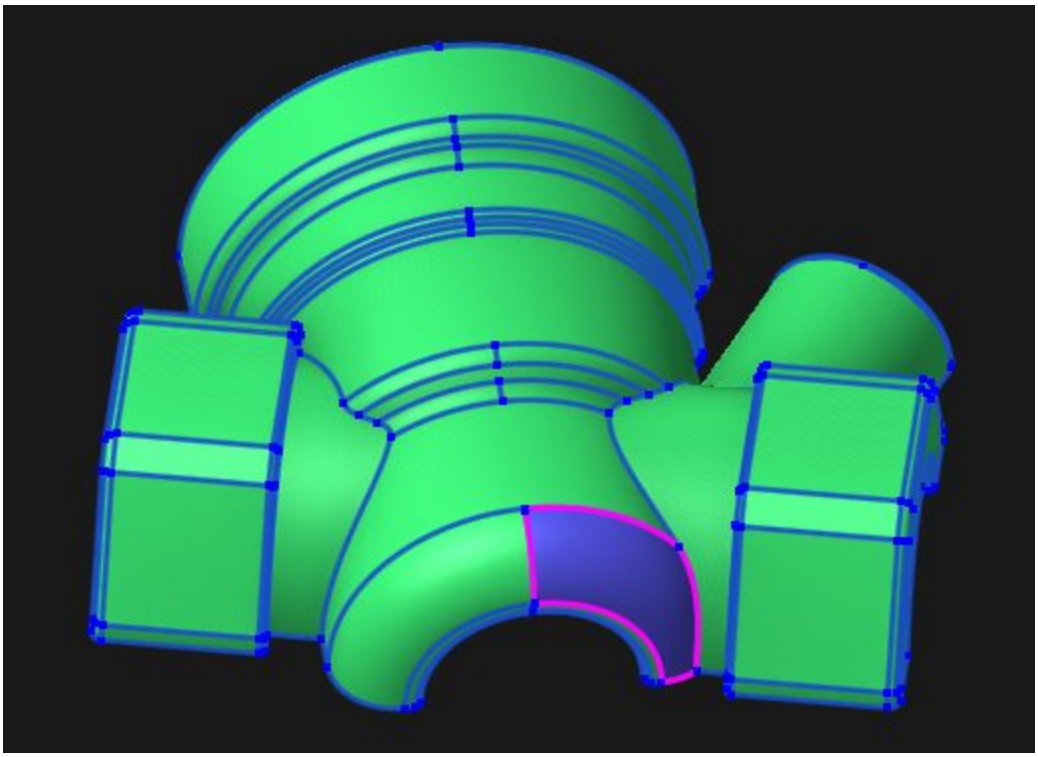
In Sew tolerance field assigned maximal value of deviation between edges and vertices of nearest faces. When tolerance is 0 this value taken automatically with account to overall dimensions of sewing objects.

The Sew according to face orientatin option allows you to take into account the normal direction of faces when sewing them. The default is off.

In some cases, errors may occur during import due to which the normals of some faces will be directed in the wrong direction:



Such models are not always possible to sew correctly:



To solve these problems, the following tools were added (appear when selecting a face):

-  - Inverts the selected faces.
-  - Inverts all faces of the shell.

-  - Inverts all faces whose normal is directed inside the shell.
-  - Enables color separation of shells mode.

The Sew faces inside group only option sets the boundaries for finding faces when sewing them. It is understood that closed shells must be in separate Groups. Enabled by default. The option is not shown if no other groups with faces are found.

The Result panel shows count of closed and opened shells and its total count.

If results is correct, then the window should be closed using the Ok button. To cancel sewing, press the Cancel button.

Export of 3D Model

The function can be activated from main menu or Save as item in popup menu of geometrical model structure window. This action opens a file dialogue where user can to select one of supported format.

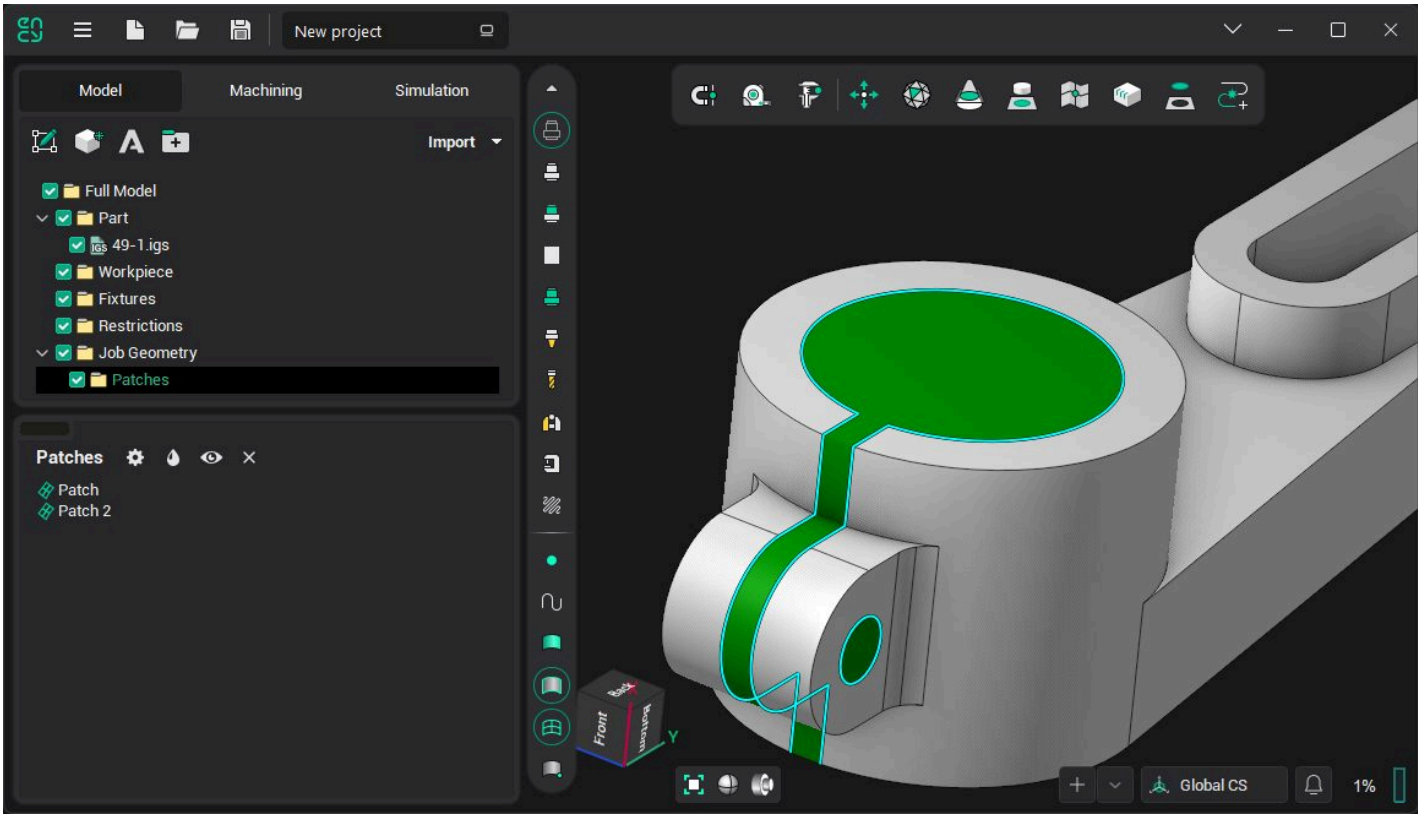
- The STL format allows to deliver volumetric models as collection of flat triangles. Extension of the file is *.stl. The file is saved as text.
- The DXF format basically used to deliver flat drafts and vectorial drawings. Volumetric models can be exported too, but has essential distinctions depending on version. In this moment is supported export of curves and points only. Extension of the file is *.dxf.
- The OpenGL stream data is internal format of CAM system. Its used to save 3D models of single units of machines kinematic schemes. Extension of the file is *.osd.

Patch hole

The patching holes function allows you to generate triangulation of the selected edges or meshes. This function is called by pressing this button - , on the "Model" page. By default, the button is not active. To activate it, you need to select edges or meshes for which you want to patch the holes in the graphics window or in the model tree.

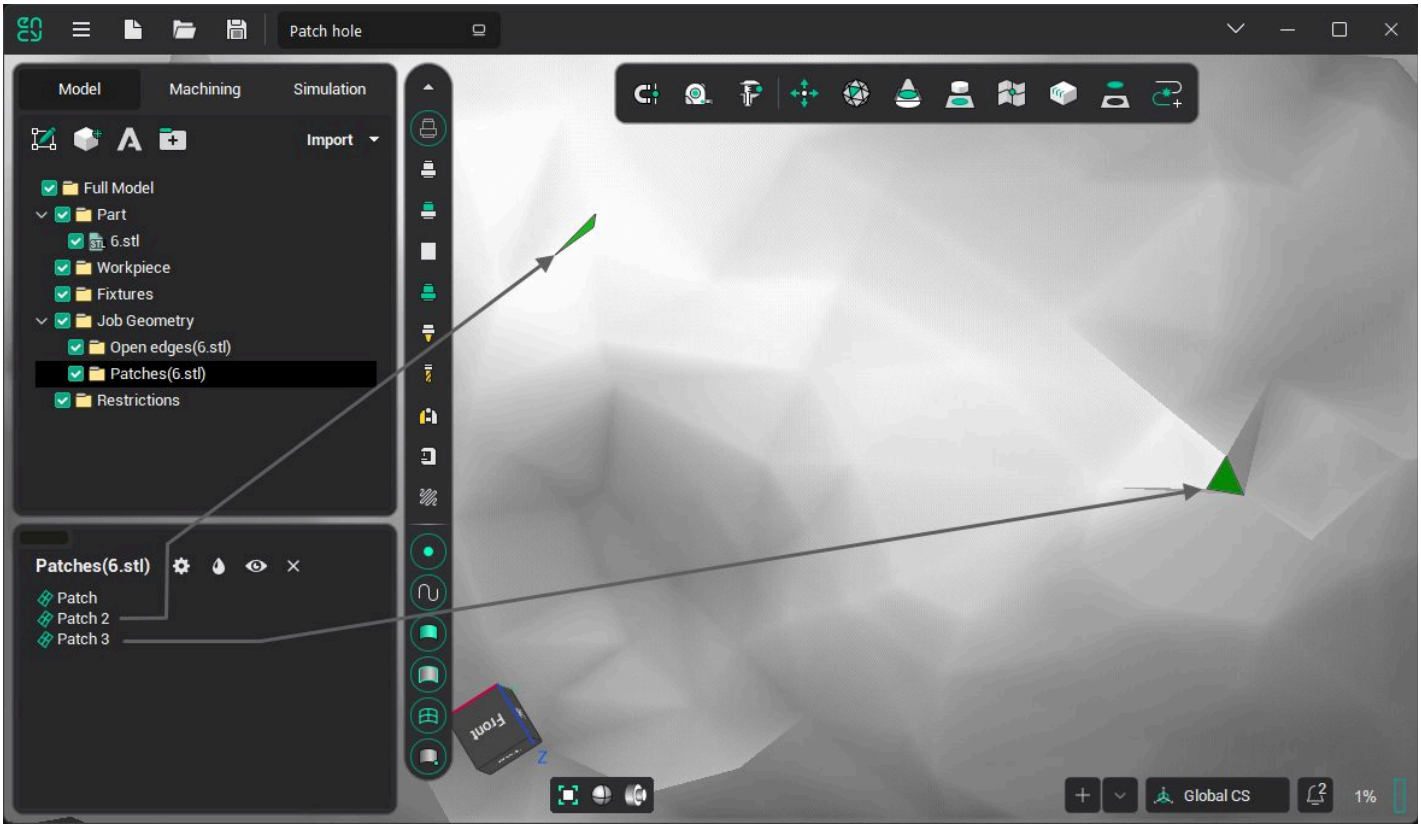
The work of the function for edges and meshes is different:

1. Edges. First, closed and open edges are determined. For closed ones, triangulation is generated immediately. And based on all the other edges a closed one is formed, for which triangulation is also generated.

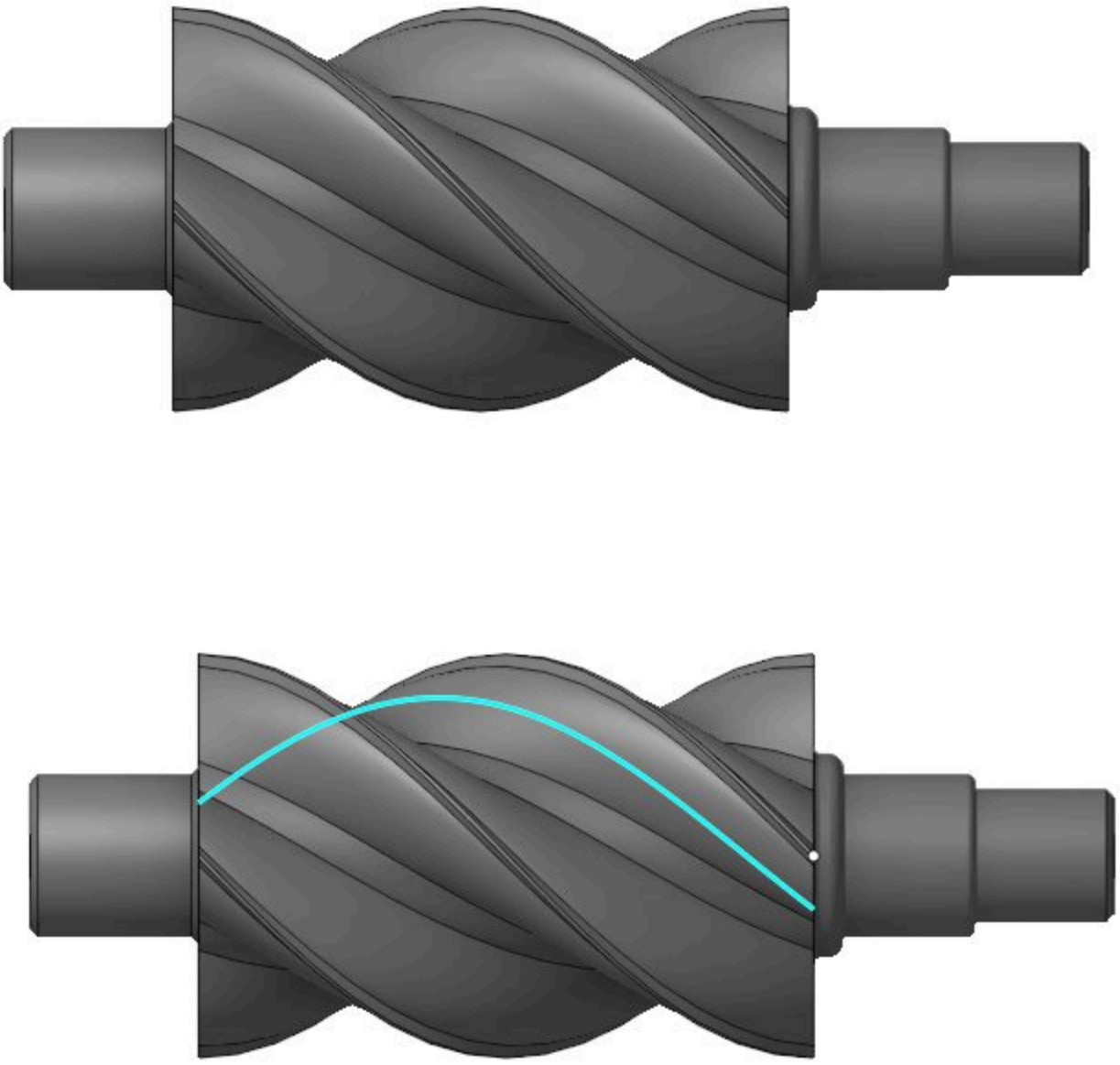


2. Meshes. In meshes, first, a topology check is performed. If open edges were found, then closed contours are formed. Then triangulation is generated for all contours. Unlike the first option, here, in addition to new meshes, the closed contours along which the triangulation was generated are added

to the model tree.



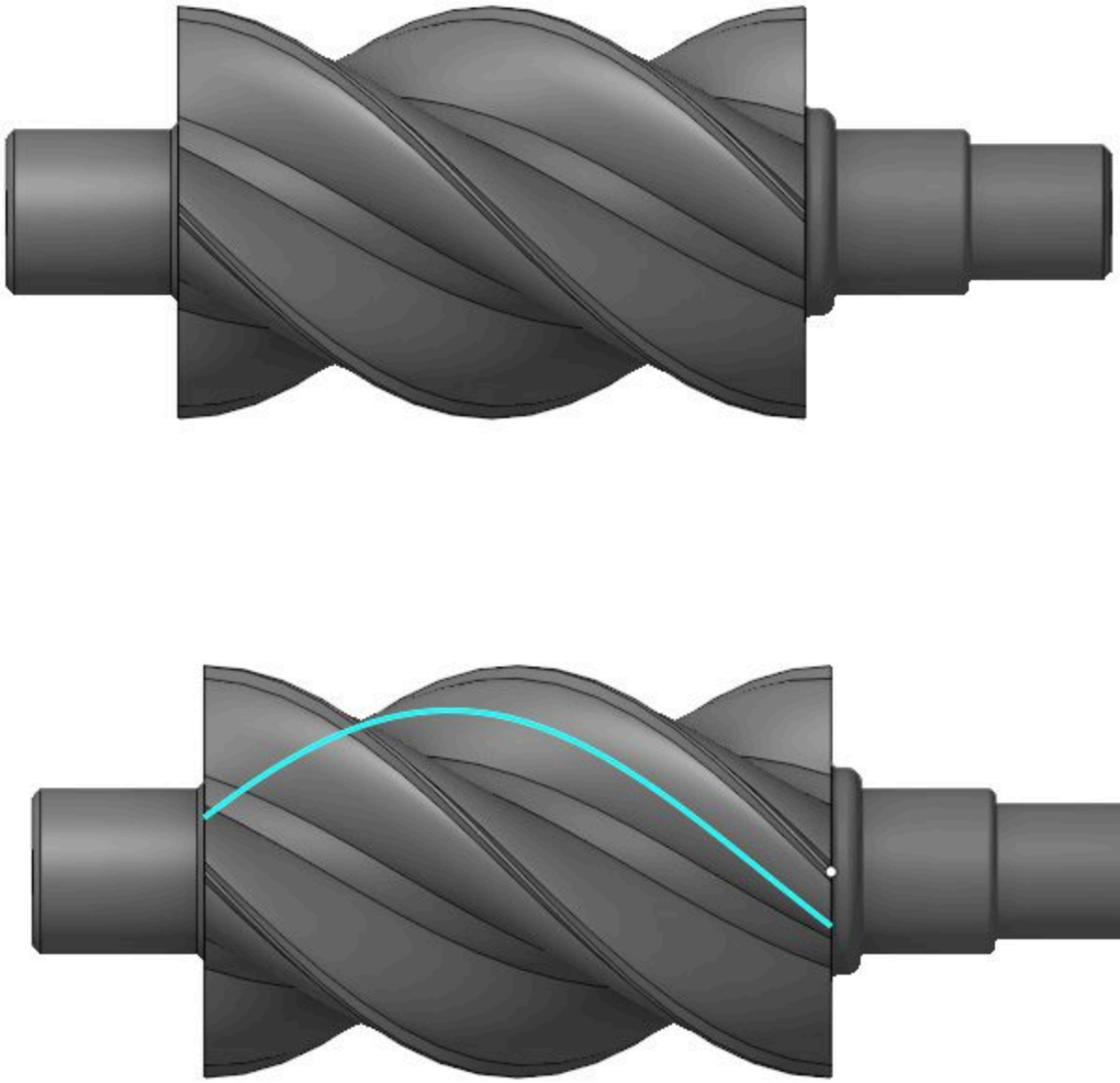
Extract isolines



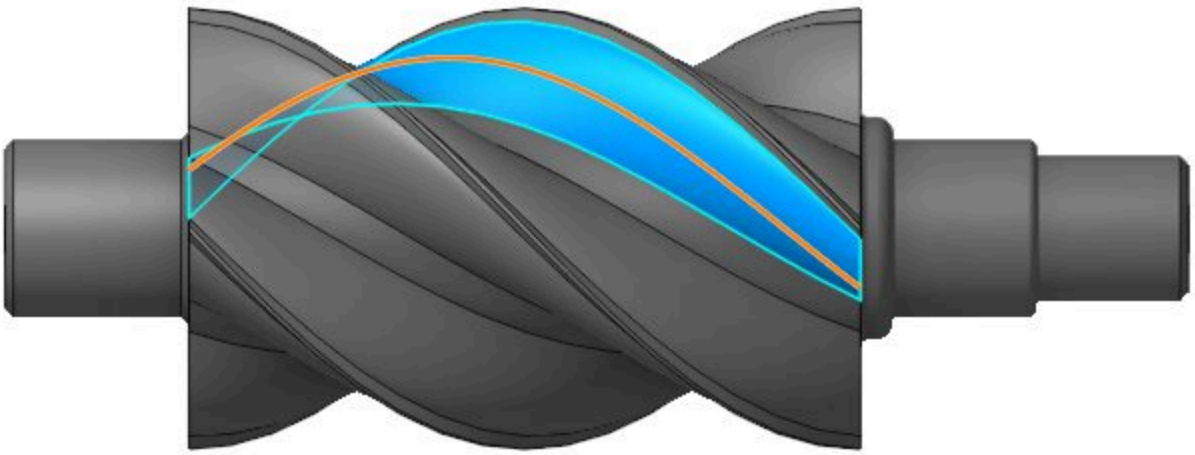
Application area:

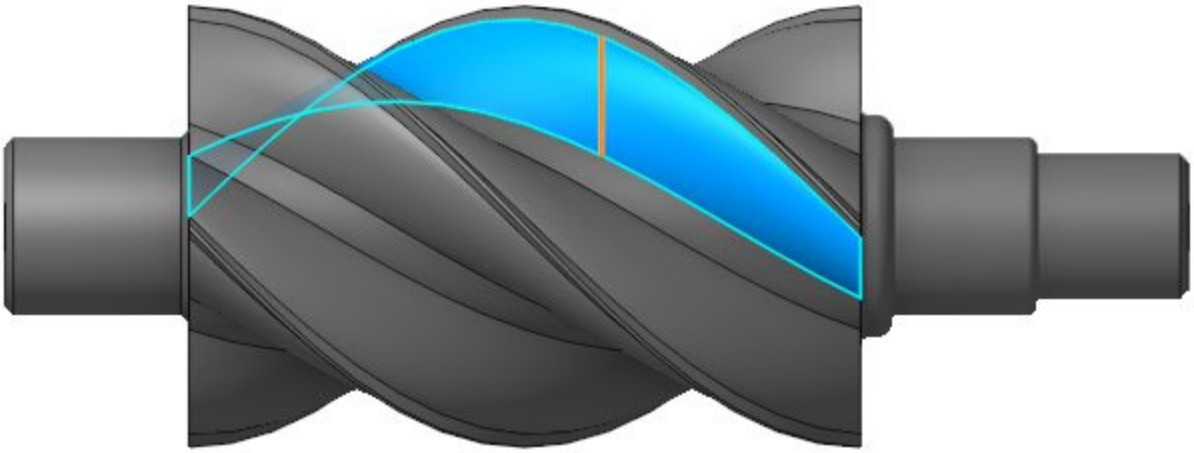
The Extract Isolines tool extracts isolines from a surface. It activates after selecting a face on the Model tab. The editing panel lets you create a spline, change the isoline direction, or split the face. Use the Measure tool for precise spline positioning.

- to form a spline from the isoline.

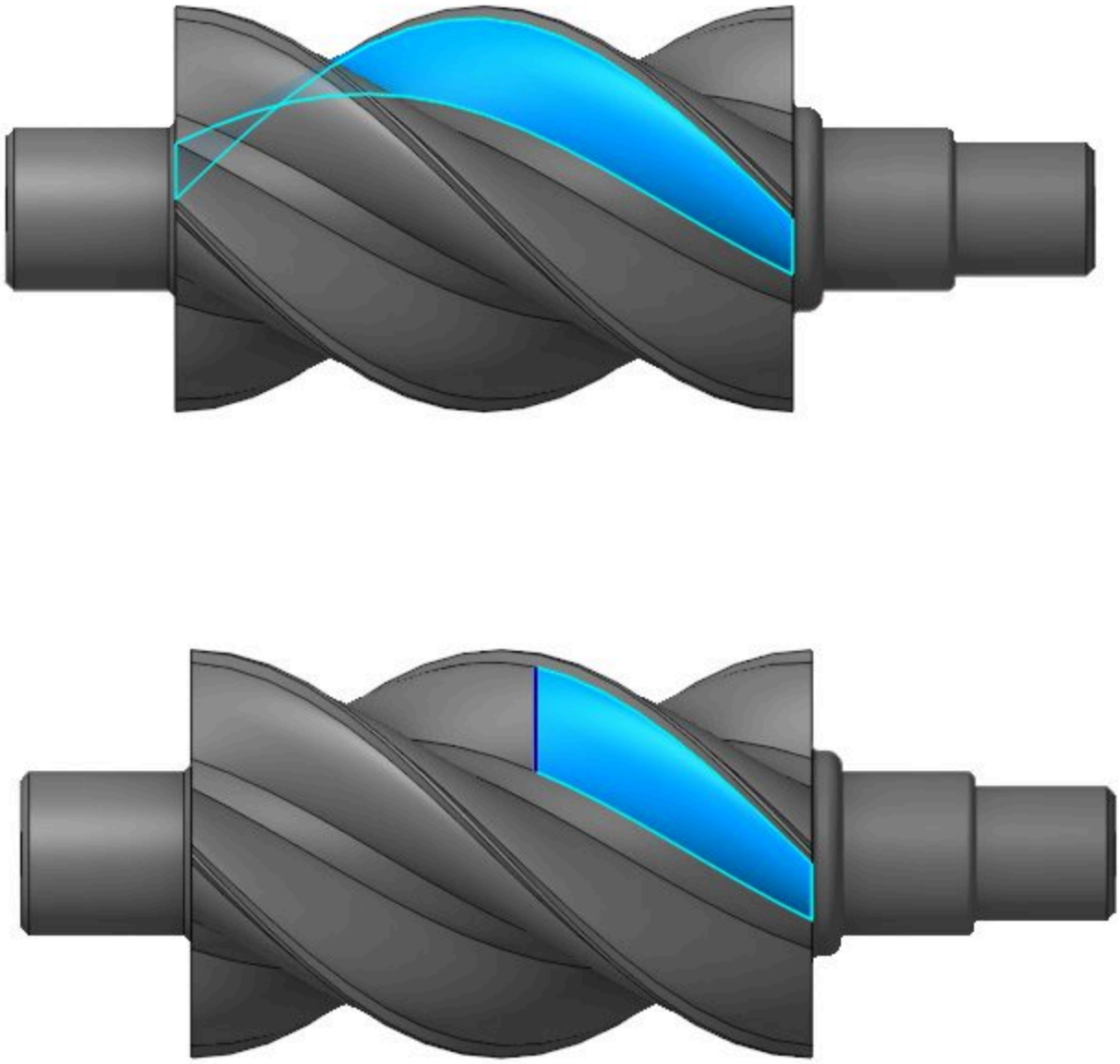


- to change a isoline direction.



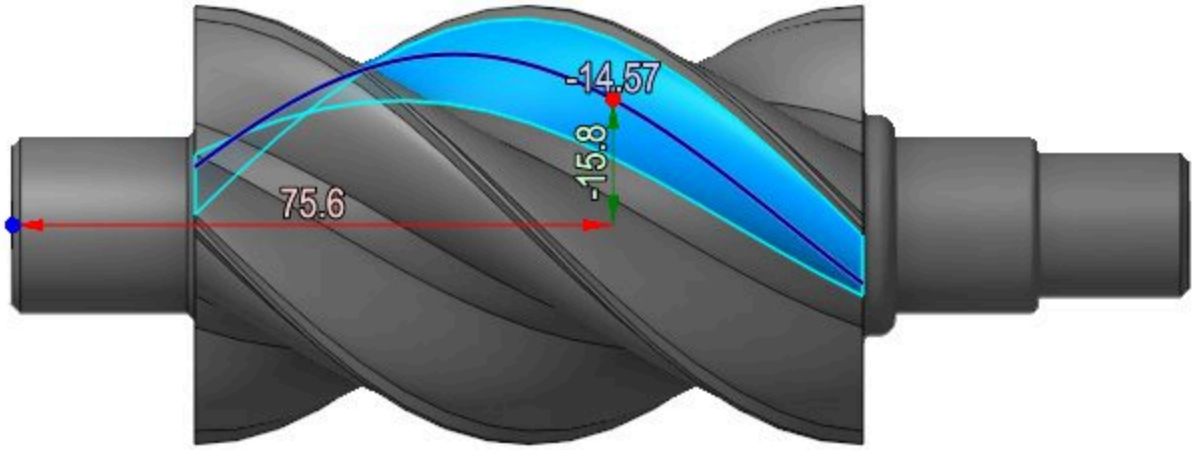


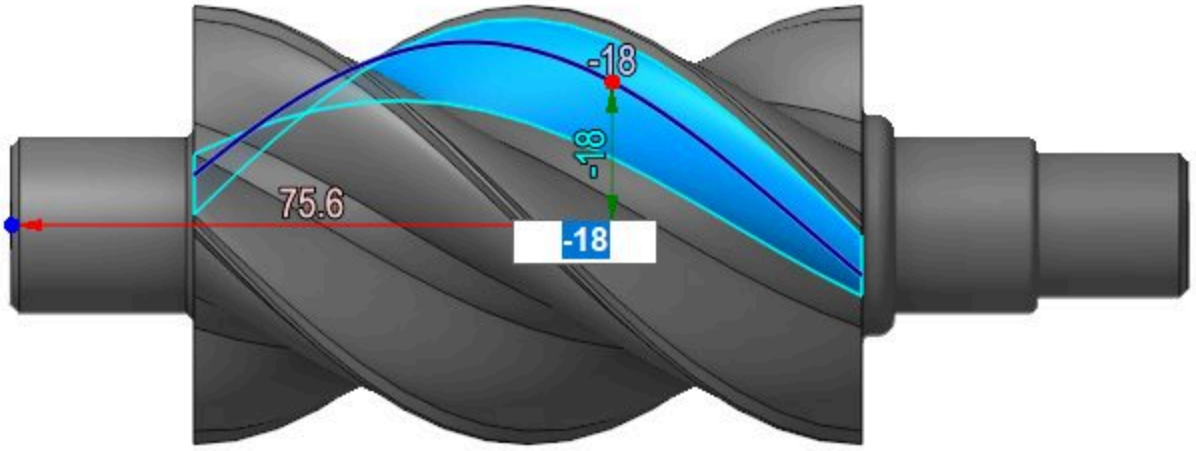
- split one face into two, with or without preserving the common (not split) face



- Use the Measure tool to accurately define a spline. You can move the point to any location on the part to bind it to a specific position. Once the endpoint is anchored, you can modify the spline's position.

.

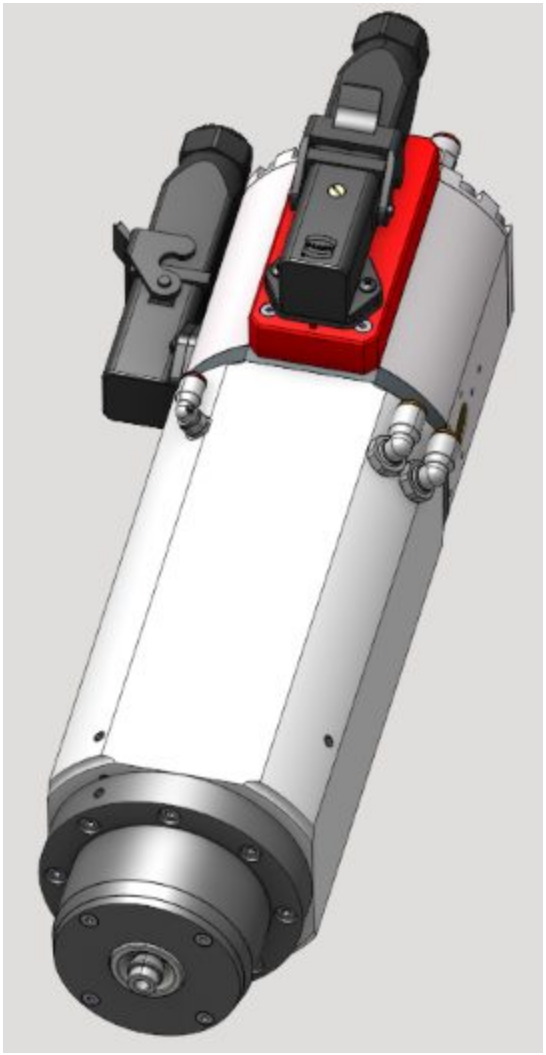

-14.57
-15.8
75.6

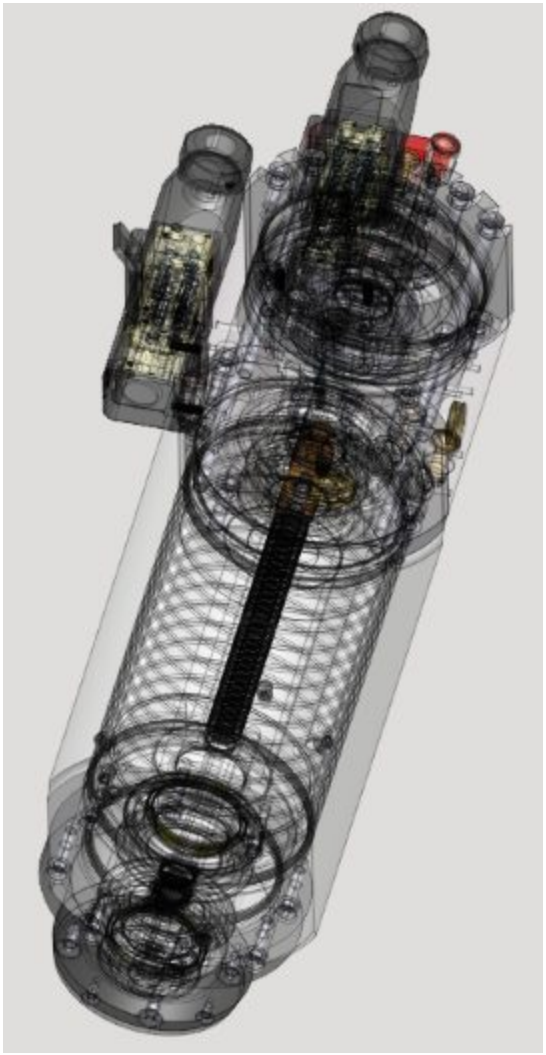

-18
-18
75.6
-18

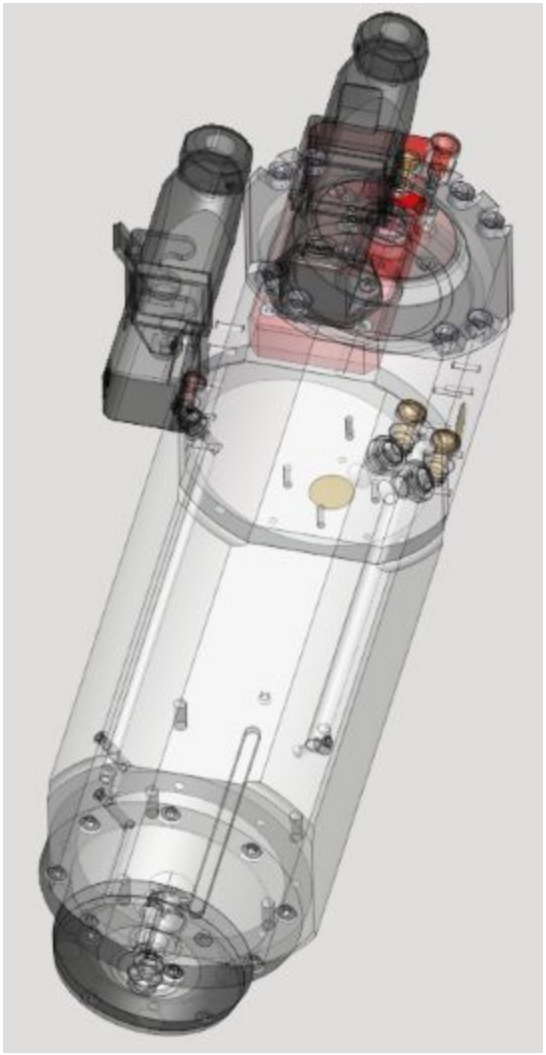
Simplifying geometrical model

The Simplifying geometrical model function can be called from the context menu of the graphic window.

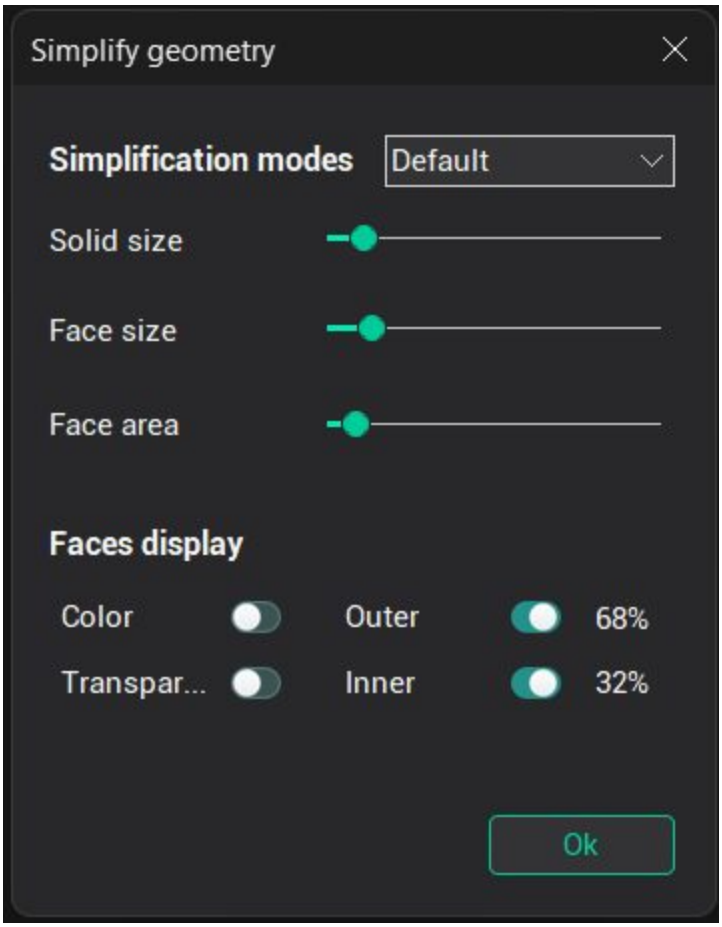
This function is intended only for removing internal faces from a closed shell of a geometric model. This can be useful if the project uses a very detailed model, which can affect both the speed of the program and the final size of the project.







As a result of the function, with a certain accuracy, a hollow shell is obtained.



By default, the simplification parameters are optimally set, but if you wish, you can use both ready-made sets of parameters or set them manually.

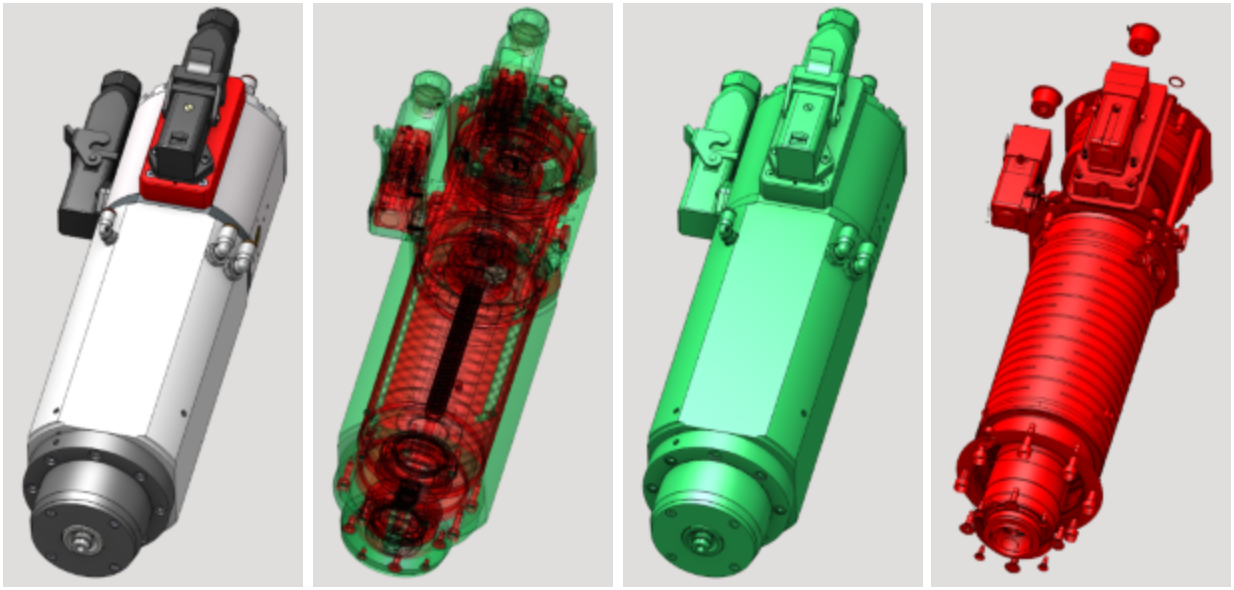
The Simplification modes block sets the parameter sets.

Solid size - Sets the minimum size of shells that will participate in the calculation. Anything less is automatically marked as an inner object.

Face size - Sets the minimum size of individual faces that will participate in the calculation. Anything less is automatically marked as an inner object.

Face area - Sets the minimum inner face area at which they will be considered inner.

The Faces display block is responsible for the face display modes.



Color - Shows the original colors of the geometry. By default, the outer faces are green and the inner ones are red.

Transparent - Shows a transparent geometry model.

Outer - Shows outer faces.

Inner - Shows inner faces.

Working with splines

Ways to select multiple points

1. Use box selection.
2. Hold Shift and click on a point to select the continuous range of points from the last selected point to the clicked point.
3. Hold Ctrl and click to select multiple points individually

Delete points

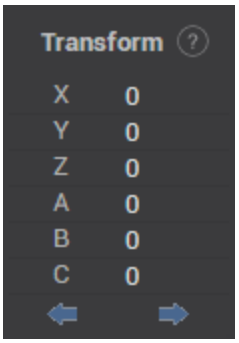
Right click on a point or press the DELETE key on keyboard to delete all the selected poitns

Undo and Redo

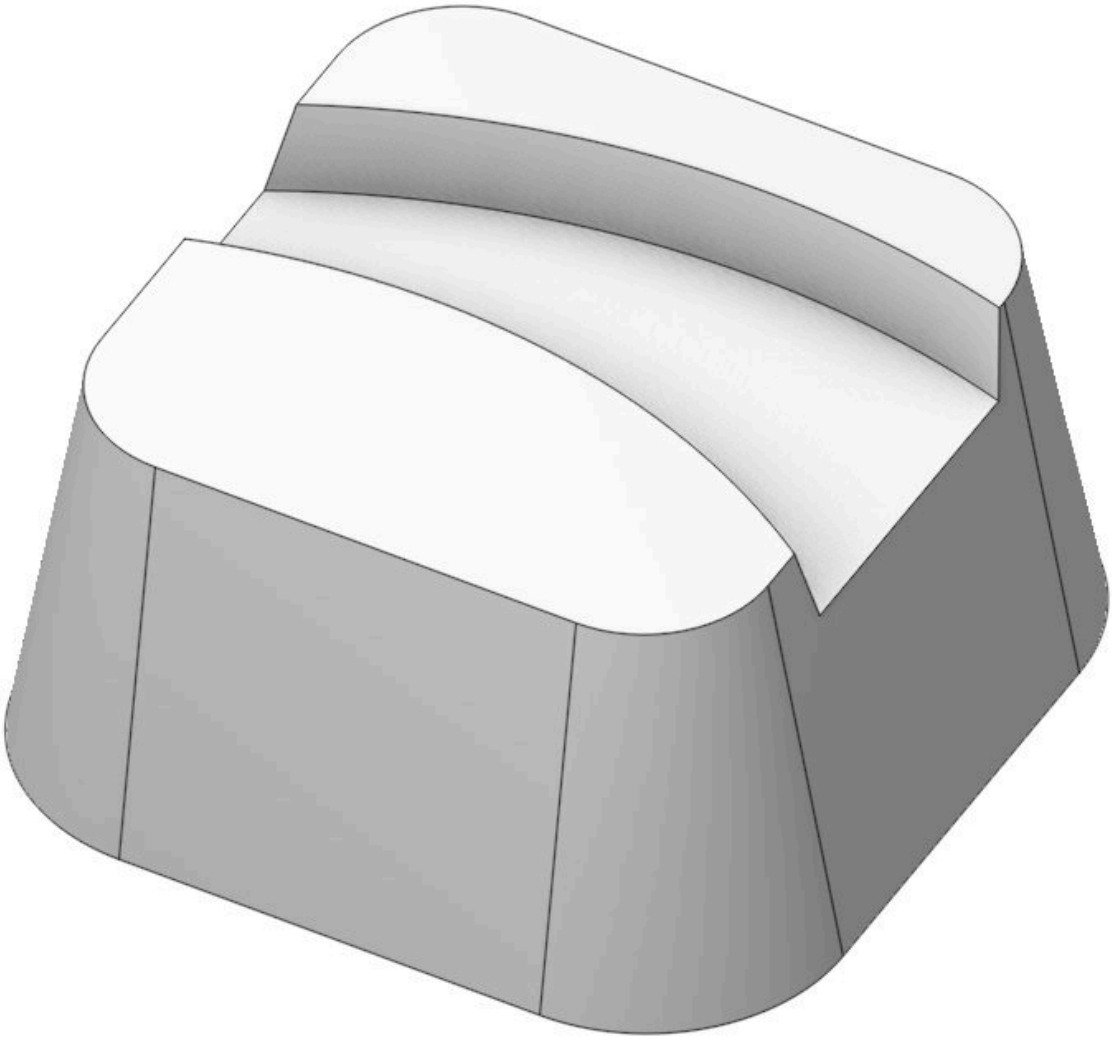
Press `Ctrl+Z` to undo the last action, press `Shift+Ctrl+Z` to redo the last action.

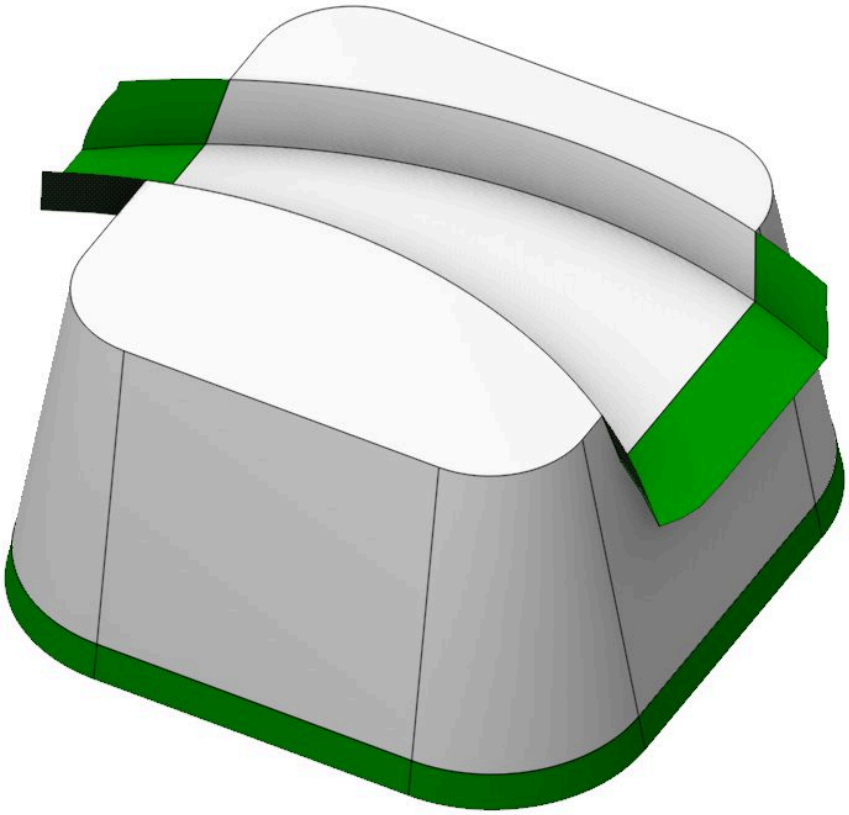
Transform the spline or the selected points of the spline

With the transformation panel you can move or rotate the spline or its selected points by the specified offset distance along/around the XYZ axes of the current coordinate system.



Extend surface

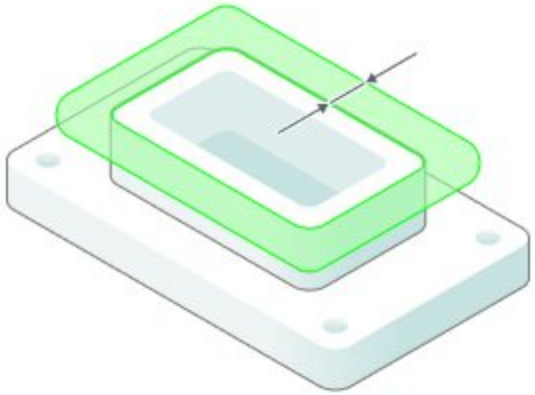




Application area:

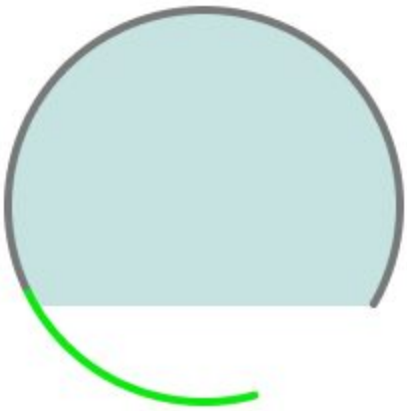
The technology allows to expand the original surfaces and expand the edges along which the surfaces are completed. Available in model mode on the top panel. Select the edges of the surfaces you want to extend.

Extension distance. The value to which it is necessary to extend

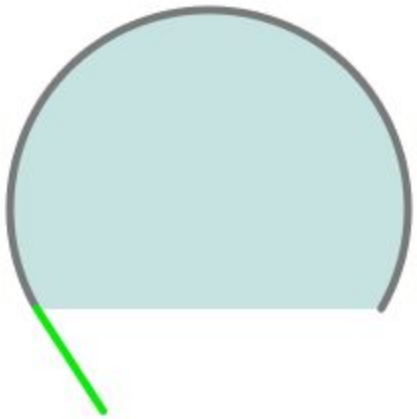


Extension type. Extension method

Equidistant to edges. The original surfaces are extended (due to the fact that some surfaces are twisted, extension is impossible)

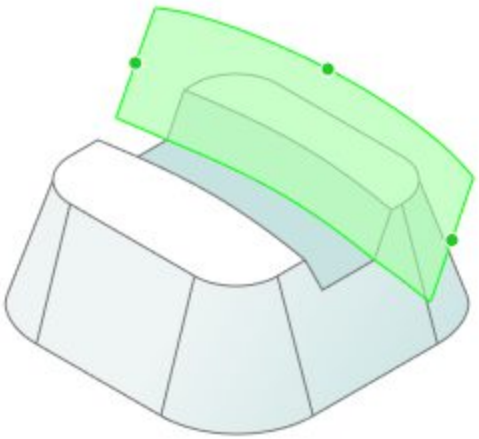


Interpolation of tangent edges. First the edges are extended, then the surface is completed along them.

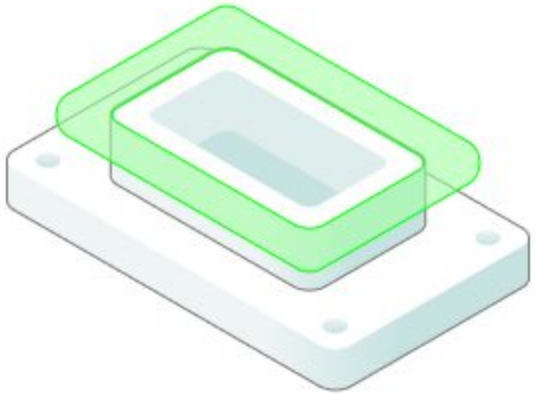


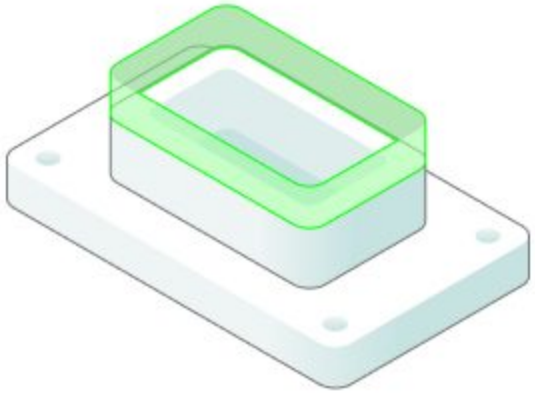
Surface UV Box. Allows you to create UV space of the surface. This surface has interactive elements to change its area. It is necessary to select a surface.

Options not available: Alternate from side, With source face.

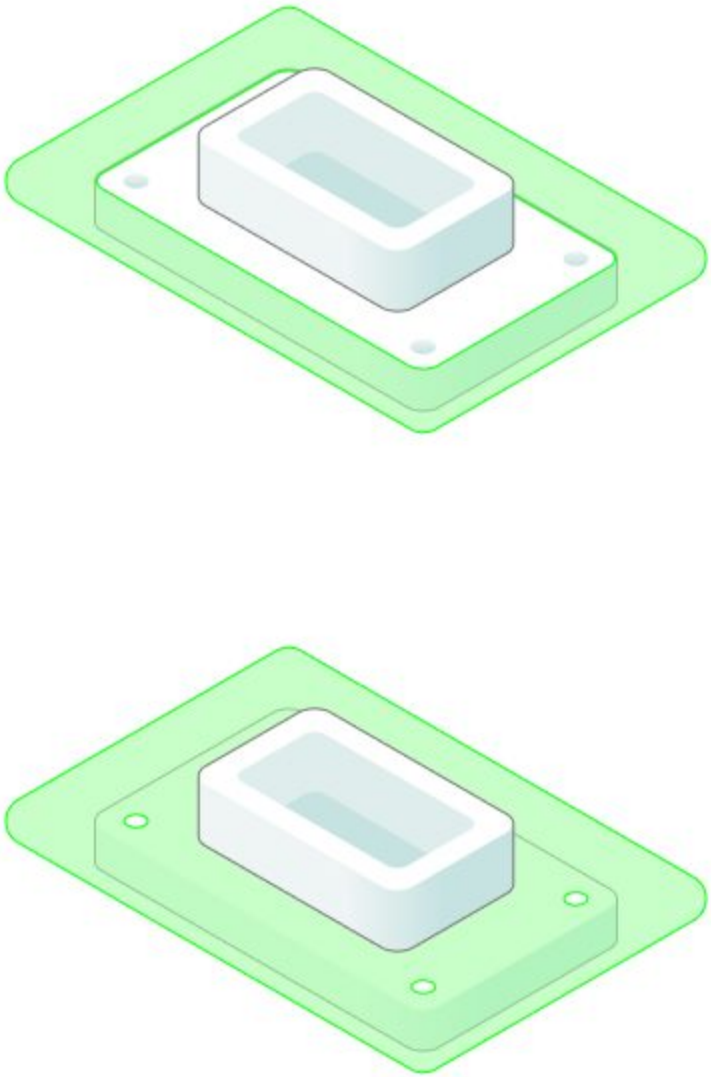


Alternate front side. Choice of which side of the rib to extend the surface.

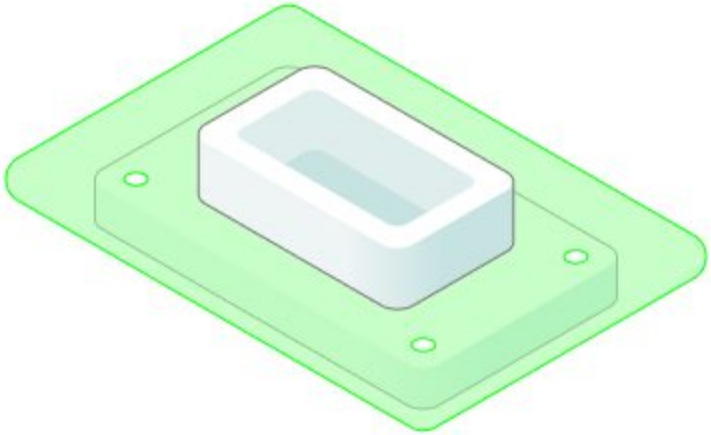


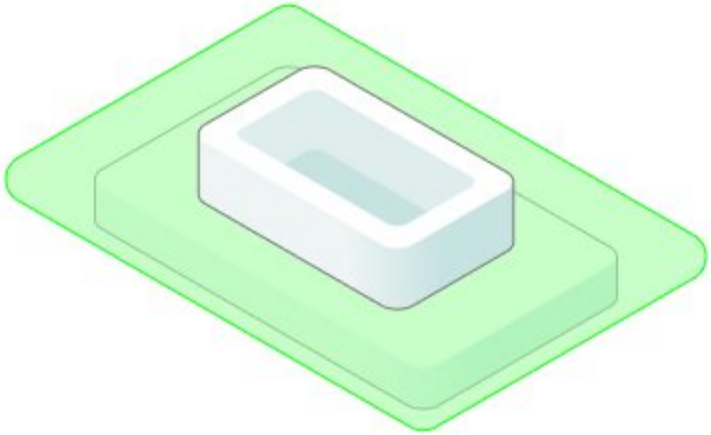


With source face. The extended surface forms a single surface with a copy of the original surface



Hole capping. Closes holes with a "radius" below a given value on the original surface





2D Geometry sketcher

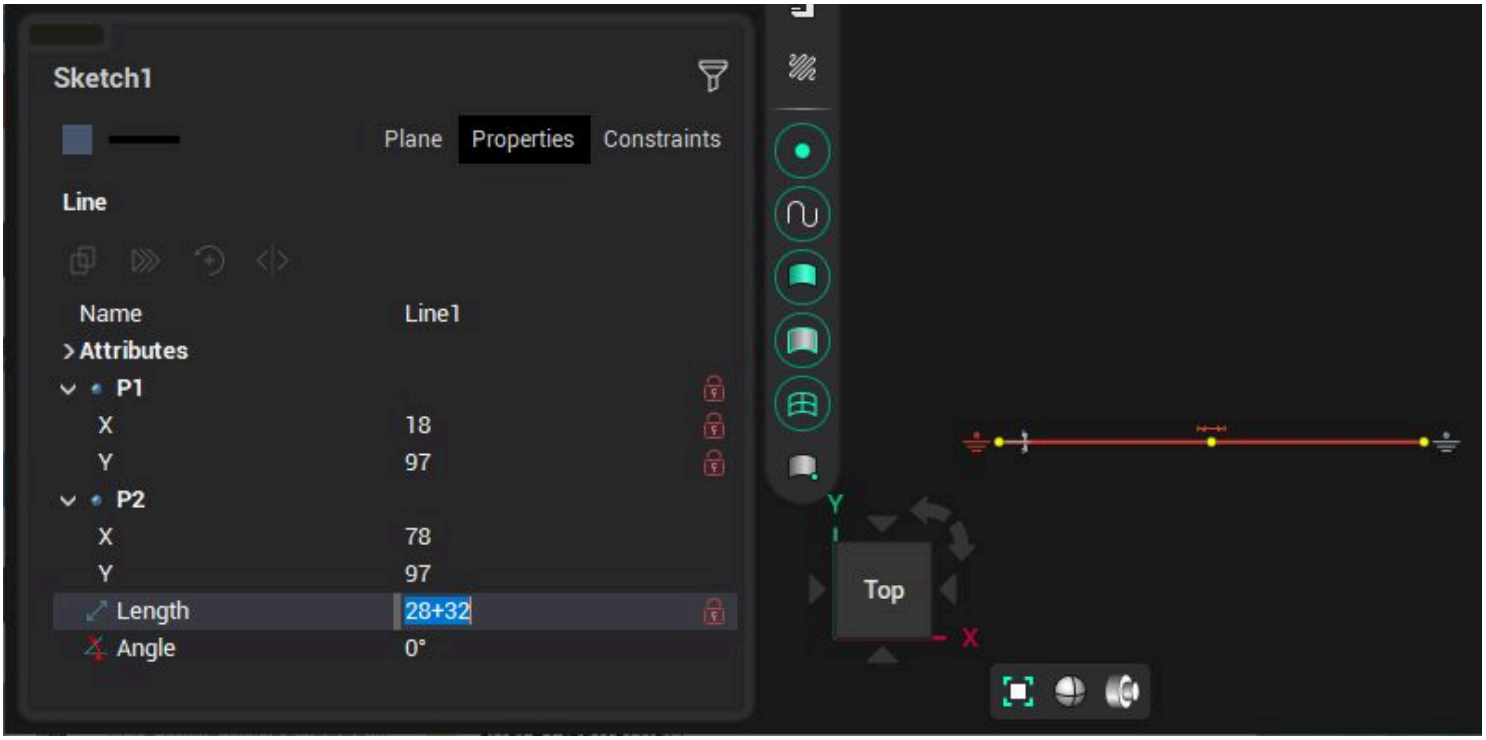
The 2D Geometry sketcher provides tools for creating and editing planar geometry in the CAM system. It supports lines, arcs, circles, rectangles, and contours, together with tools for adding chamfers, rounding corners, offsetting, trimming, and splitting elements.

This section explains the common building and editing workflow and the settings of individual sketching tools. Details It also covers dimensions, geometric constraints, transformations, and reusable blocks. Use these topics to construct and modify drawings and contours, either interactively or by entering coordinates and parameter values.

Building and editing elements

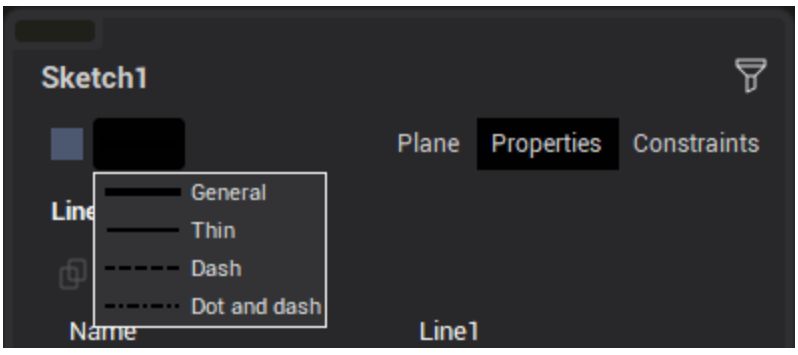
Almost all elements are built according to a typical scenario: the first nod point of the object is indicated, then the next ones and subsequent ones. When the last point is built, the construction of the object ends and the construction of the next object of the same type begins. In order to stay in the element editing mode, you must press the [Shift] key and only then specify the last point.

An alternative way of building is to enter point coordinates or parameters in the Property inspector. Entering numerical values supports the calculation mode. It means, that you can enter a value in the form of a simple formula:



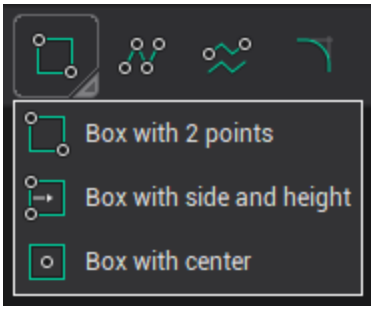
Pressing the [Esc] key at any time will cancel the element building. The same result is obtained by pressing the element building button again.

When building and editing an element, you can set its color and line type. These parameters are controlled by the buttons located above the inspector.



Also, the line type can be changed with a keyboard shortcuts [Alt+1], [Alt+2], [Alt+3], [Alt+4].

Some objects allow you to build them in different ways. To select an alternative method, you need to click on the triangle in the lower right corner of the button, or right-click and select the desired item in the drop-down menu.



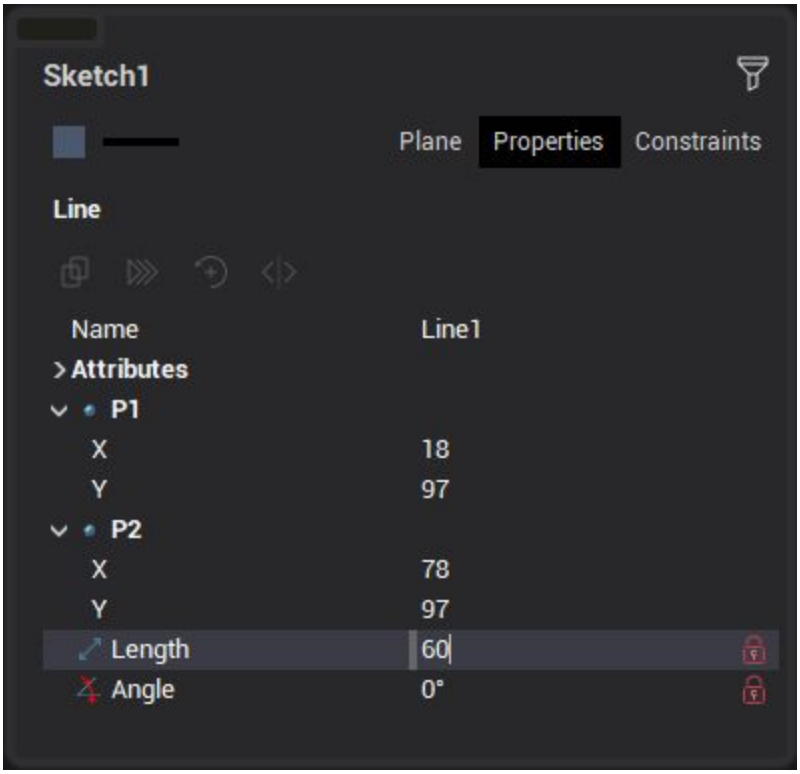
Exceptions are such elements as: chamfers, fillets, dimensions and hatching, which will be indicated in the description of these elements.

Line

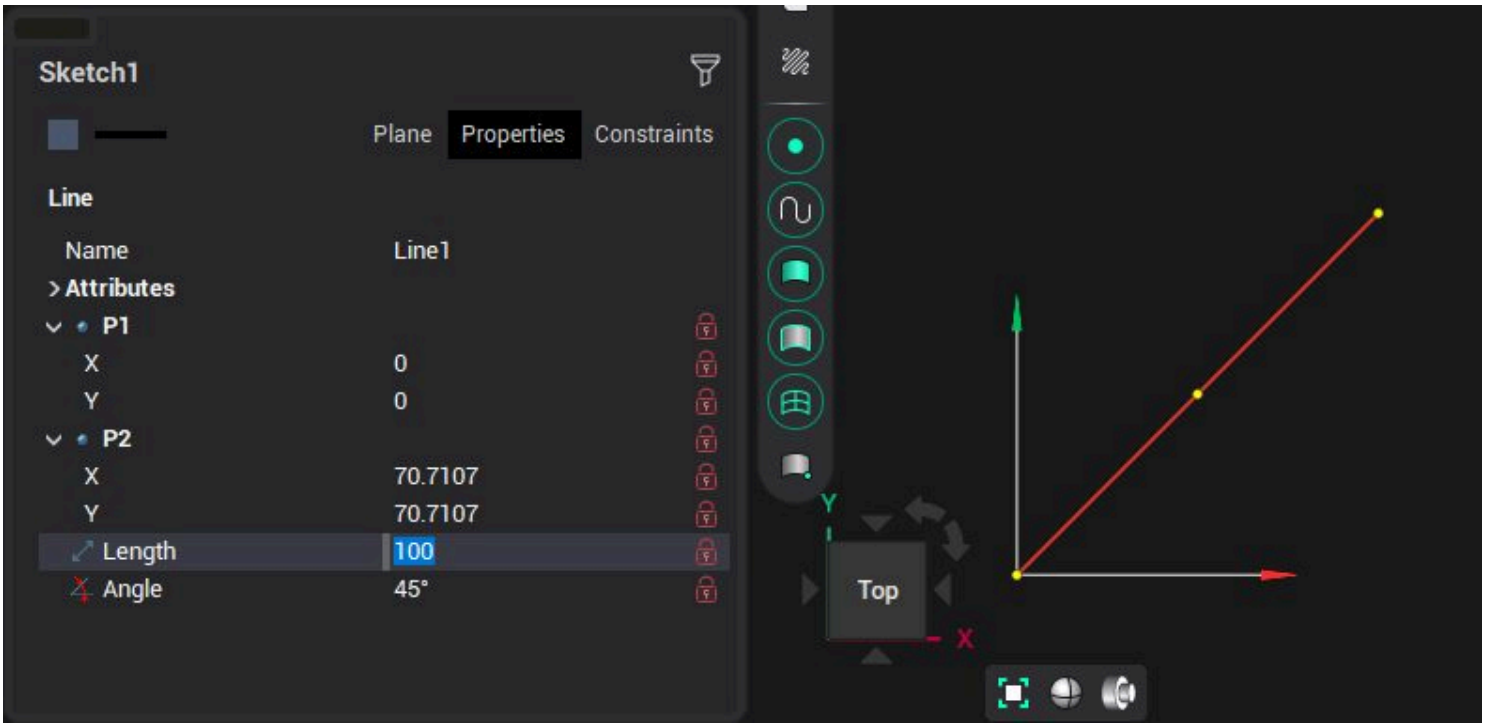


Line is built on two points and has a point lying on the line and divides it in half. The line is constructed by specifying the coordinates of the end points of the line in the Property inspector. Also, using the mouse, you can move the line beyond its midpoint without changing its length and rotation angle. When the cursor captures the end points of a line, it is possible to dynamically change its length and rotation angle. If you press the [Shift] key, the line is moved without changing its length and angle. Double-click on a circle or arc will create axis lines for that arc or circle.

Line parameters in the Property inspector window:

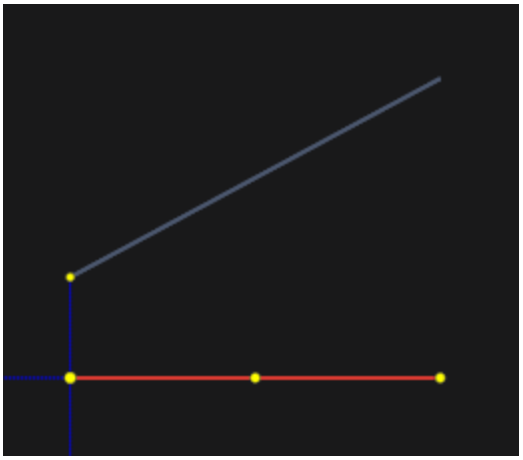


When editing a line, all these parameters can be fixed by clicking the "lock" symbol in the inspector, or the appropriate icon on the line itself.

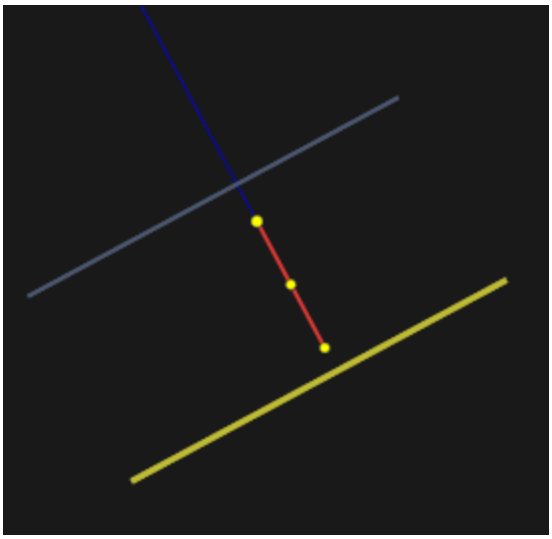


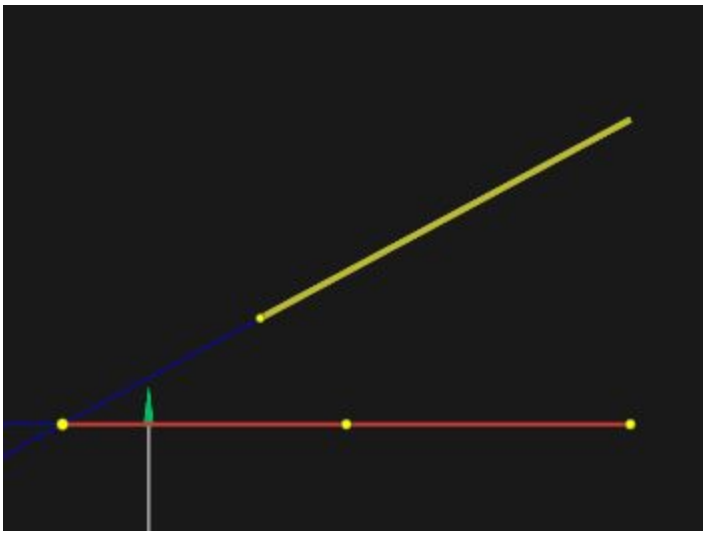
When building and editing a line, it is possible to use the help of guide lines. By default, guides are offered at 0°, 45°, 90° and the angle at which the segment was previously created in edit mode.

In addition, you can create guides by hovering over a point on a previously created object:

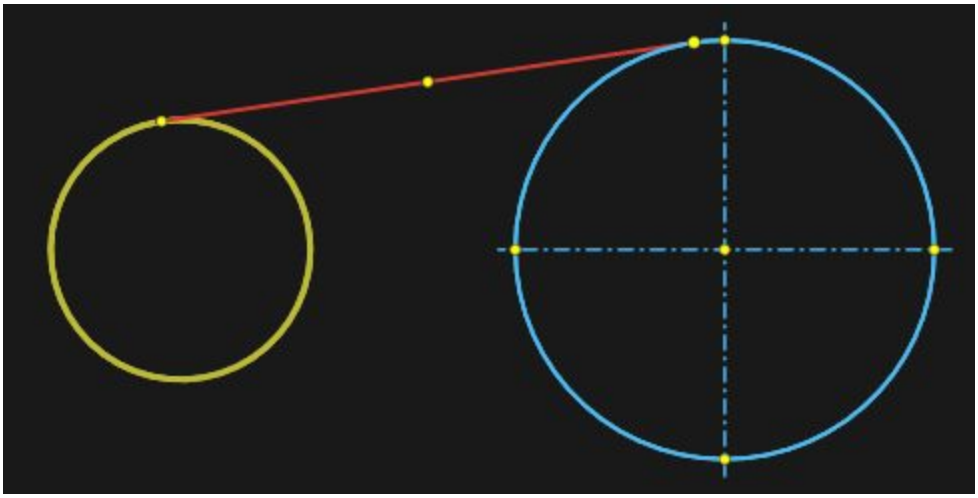


If you press the [Ctrl] key while drawing a line and hold the cursor over a previously created line, then perpendicular and parallel guides to this line, as well as a line that is a continuation, will be additionally proposed.





If an arc or circle was specified with the [Ctrl] key pressed, then a line tangent to the given arc/circle will be constructed. This applies to both the first and second points of the line.



Arc

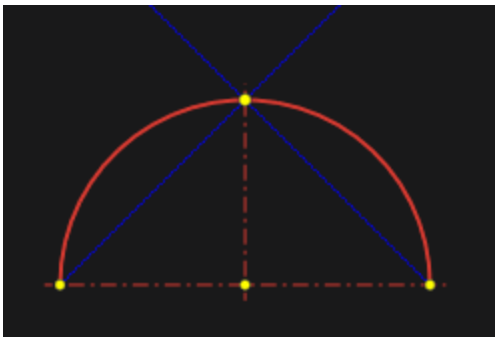
There are 3 construction options available for an arc:

- Two points and radius
- Center, starting point and angle
- By three tangents

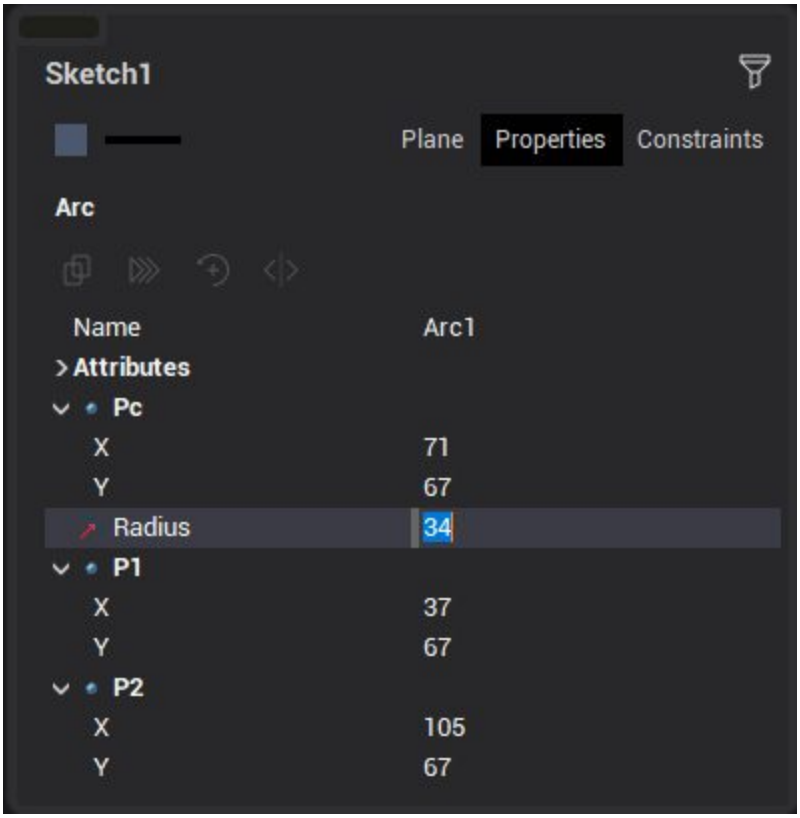
Two points and radius



To construct an arc, you must sequentially specify the first, second, and midpoint of the arc.



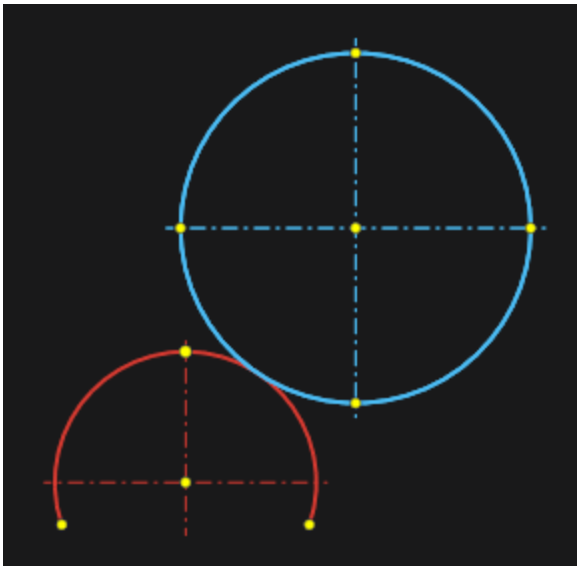
Arc parameters in the Property inspector window:



When defining an arc, it is possible to use the help of guide lines. By default, guides are offered from the first or second point at angles of 0°, 45°, 90° and relative to the center and midpoint of the arc in editing mode.

In addition, you can create guides by holding the cursor over a point on a previously created object, like building or editing a line.

If during the construction in the last step a segment, arc or circle was specified with the [Ctrl] key pressed, then an arc tangent to this element will be constructed.



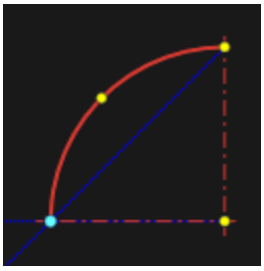
Center, starting point and angle



To construct an arc, you must consistently specify the center, first and second points of the arc.

Arc parameters and their fixation are completely similar to the first construction method.

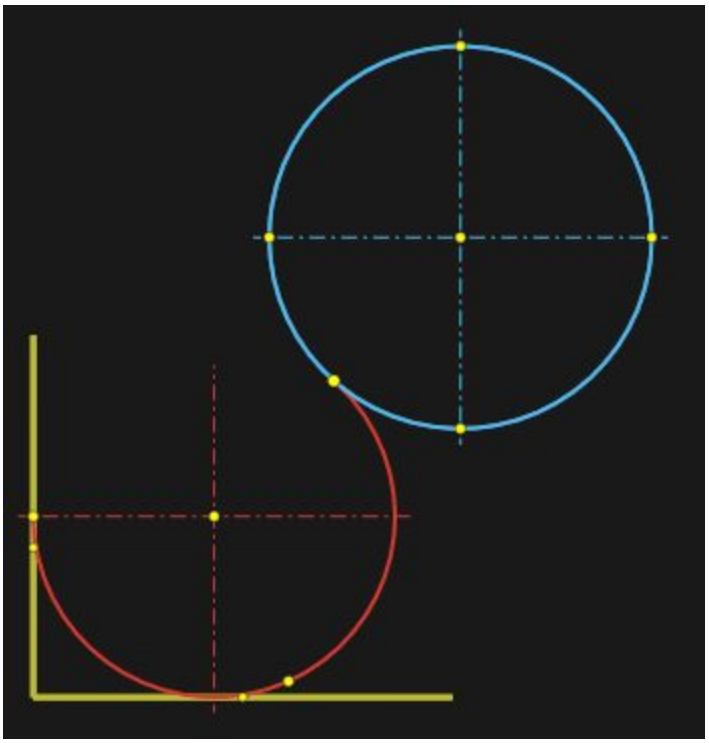
Guides are offered from the center (after specifying it), and then additionally from the first point of the arc.



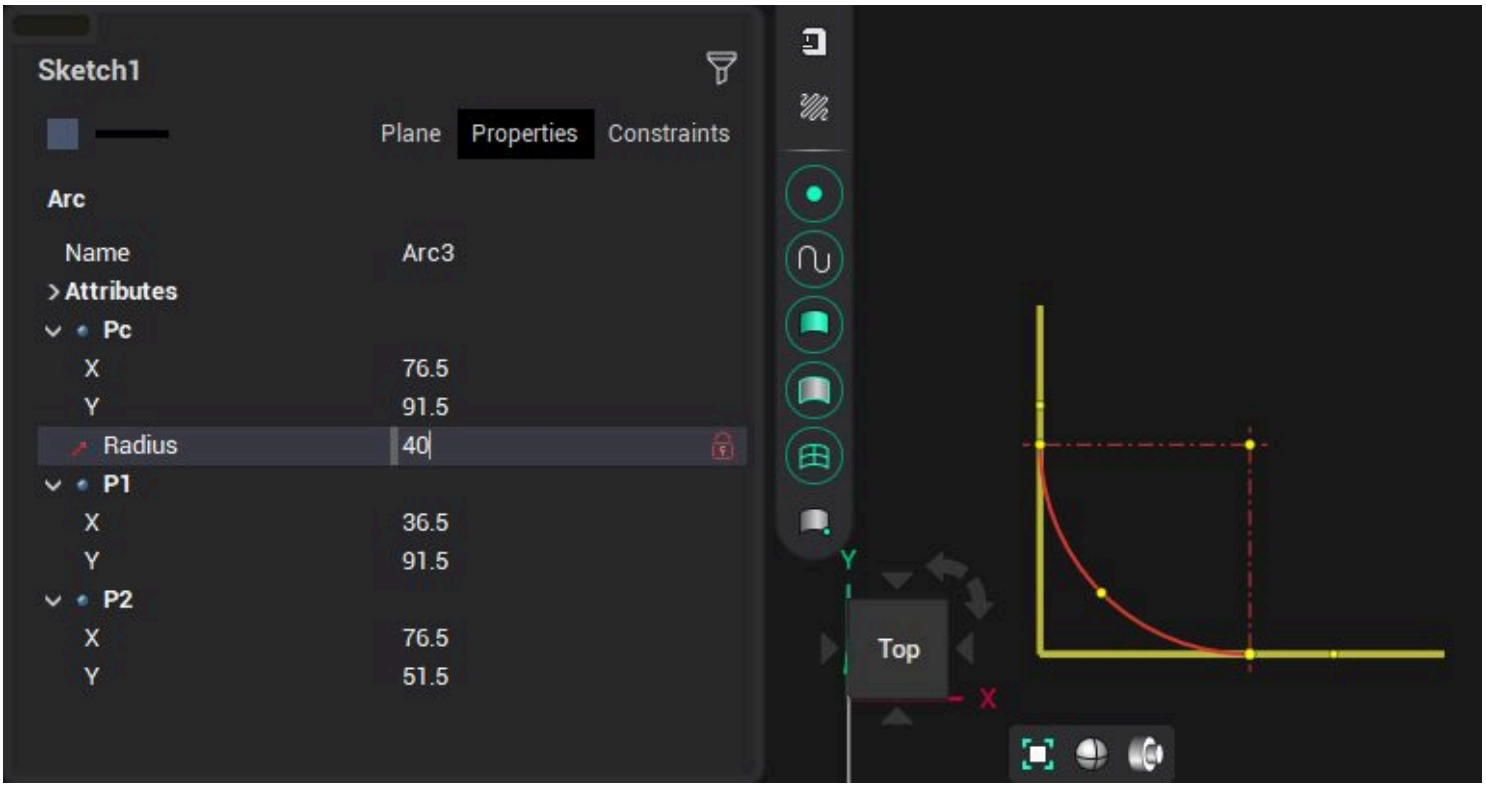
By three tangents



To construct an arc, you must sequentially specify three objects to which the arc should be tangent, with the [Ctrl] key pressed



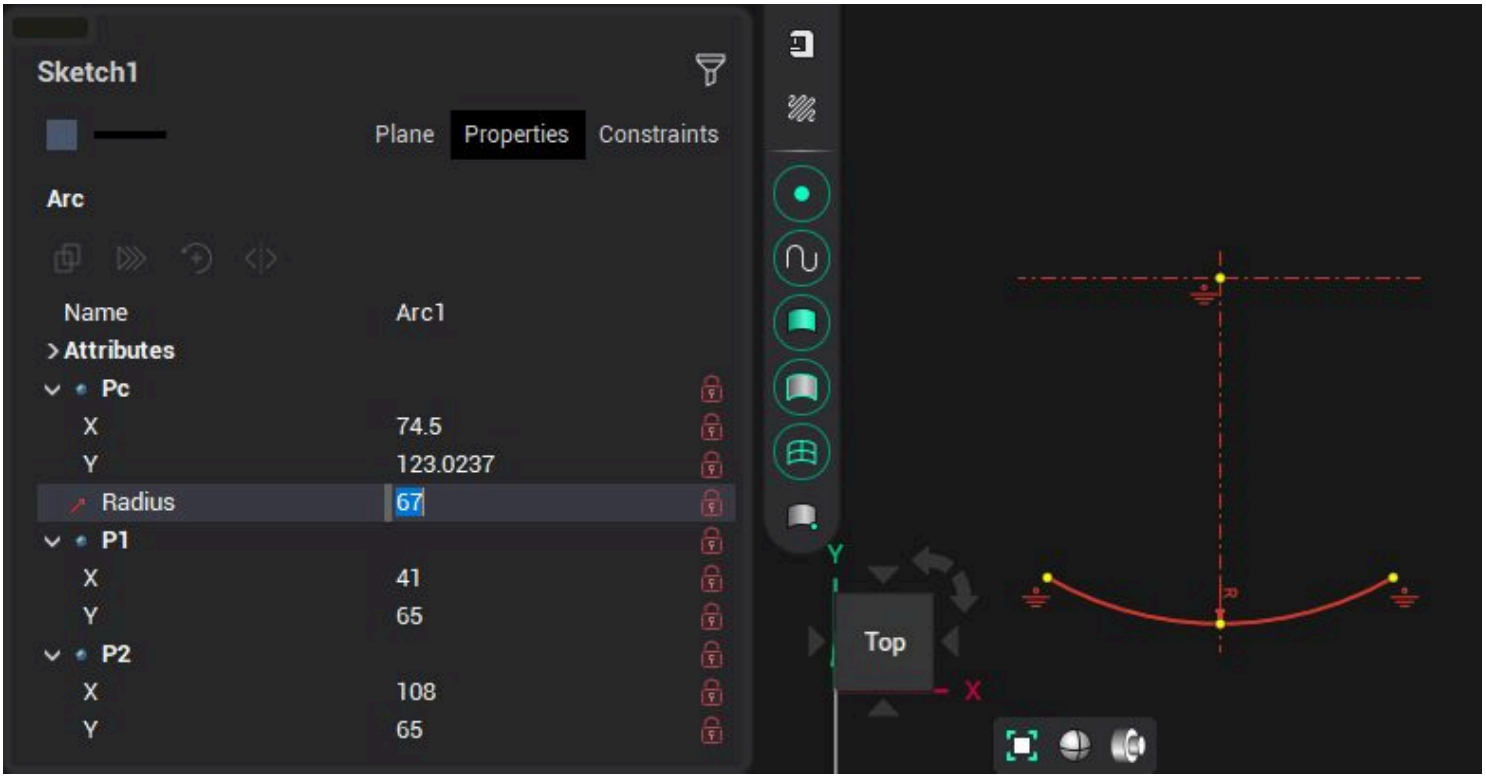
Or specify two objects and enter the radius of the arc:


Sketch1
Plane
Properties
Constraints
Arc
Name
Arc3
Attributes
Pc
X
76.5
Y
91.5
Radius
40
P1
X
36.5
Y
91.5
P2
X
76.5
Y
51.5
Top
Y
X

Edit arc

Selected arc object can be modified by directly moving points in graphical window or by changing parameters in inspector panel.

When editing an arc, all parameters can be fixed by clicking the "lock" symbol in the inspector, or the corresponding icon on the arc itself. Axis lines for the arc can be added by selecting Line and double-clicking on the arc.



When editing an arc, it is possible to use the help of guide lines. By default, guides are offered from the first or second point at angles of 0°, 45°, 90° and relative to the center and midpoint of the arc in editing mode.

In addition, you can create guides by holding the cursor over a point on a previously created object.

Circle

There are 2 construction options available for a circle:

- Build Circle with Center and radius. Edit circle
- Circle by three tangents

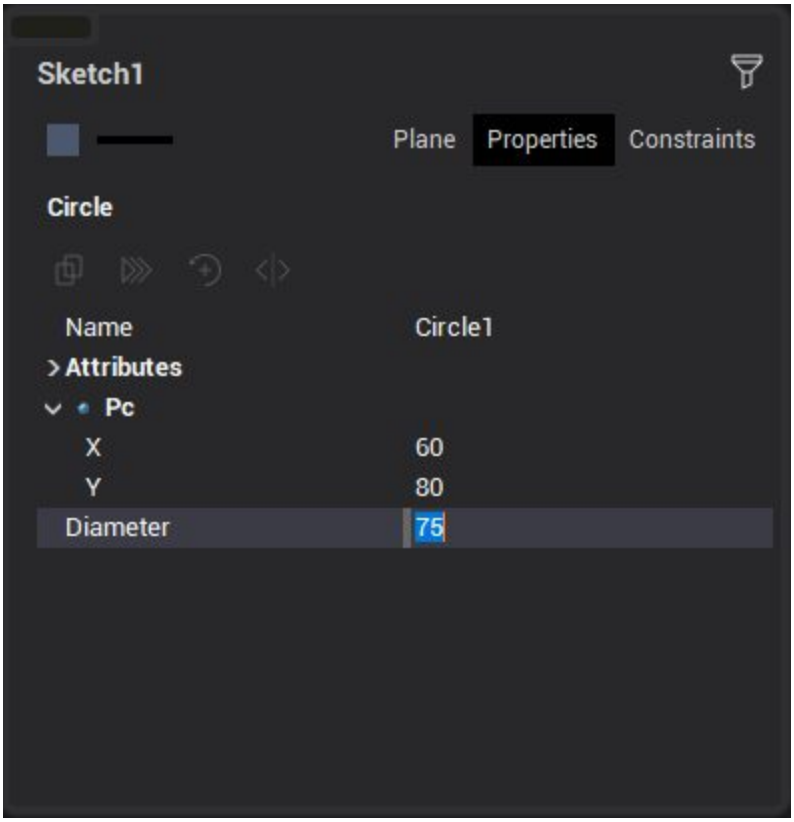
Build Circle with Center and radius. Edit circle



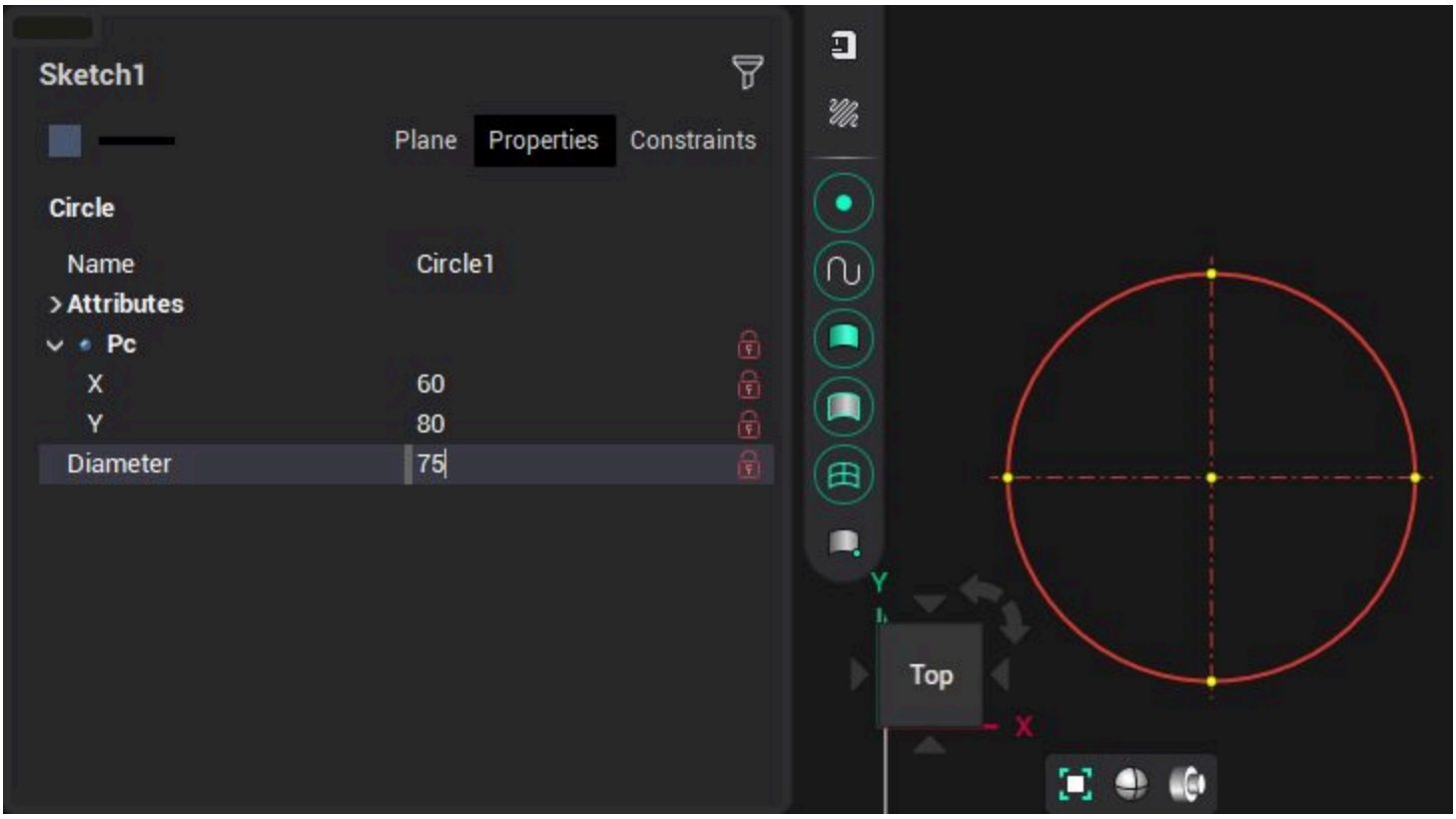
To build a circle, you need to specify the center point with the cursor, or enter the coordinates in the inspector. Specify the radius, in the same way, interactively, or by entering the diameter value in the inspector. You can also press [R] to switch to radius input mode or [D] to switch back to diameter input mode.

When editing a circle you can modify it directly in graphical window by moving circle points or change circle properties in inspector panel. Axis lines for the circle can be added by selecting Line and double-clicking on the circle.

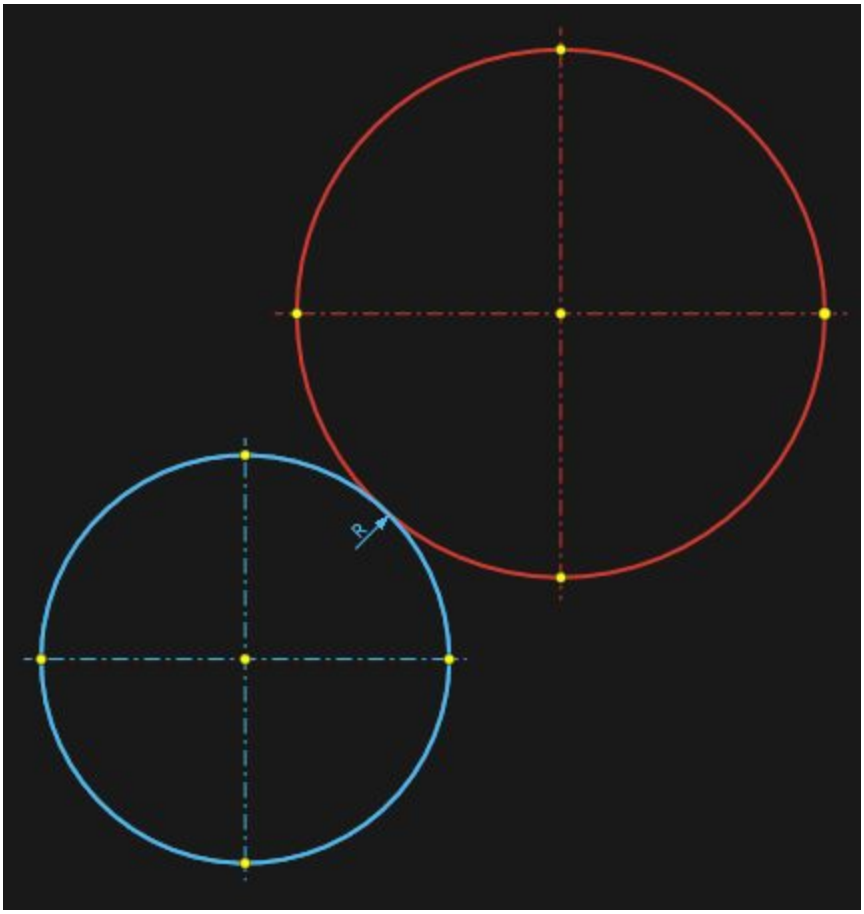
Circle parameters in Property inspector:



Circle parameter values can be fixed by clicking "lock" button in the editor field or by clicking constraint icon in the graphic window.



If it is necessary to construct a circle with a given center and a tangent to another object, then press the [Ctrl] key when specifying this object.

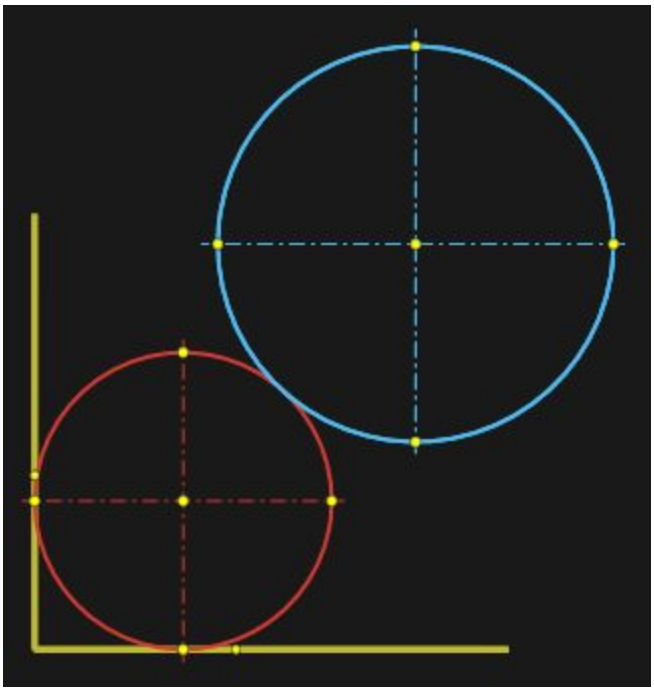


When defining or changing a circle, you can use the help of guide lines. By default, guides are offered at angles of 0°, 45°, 90° from the edited point.

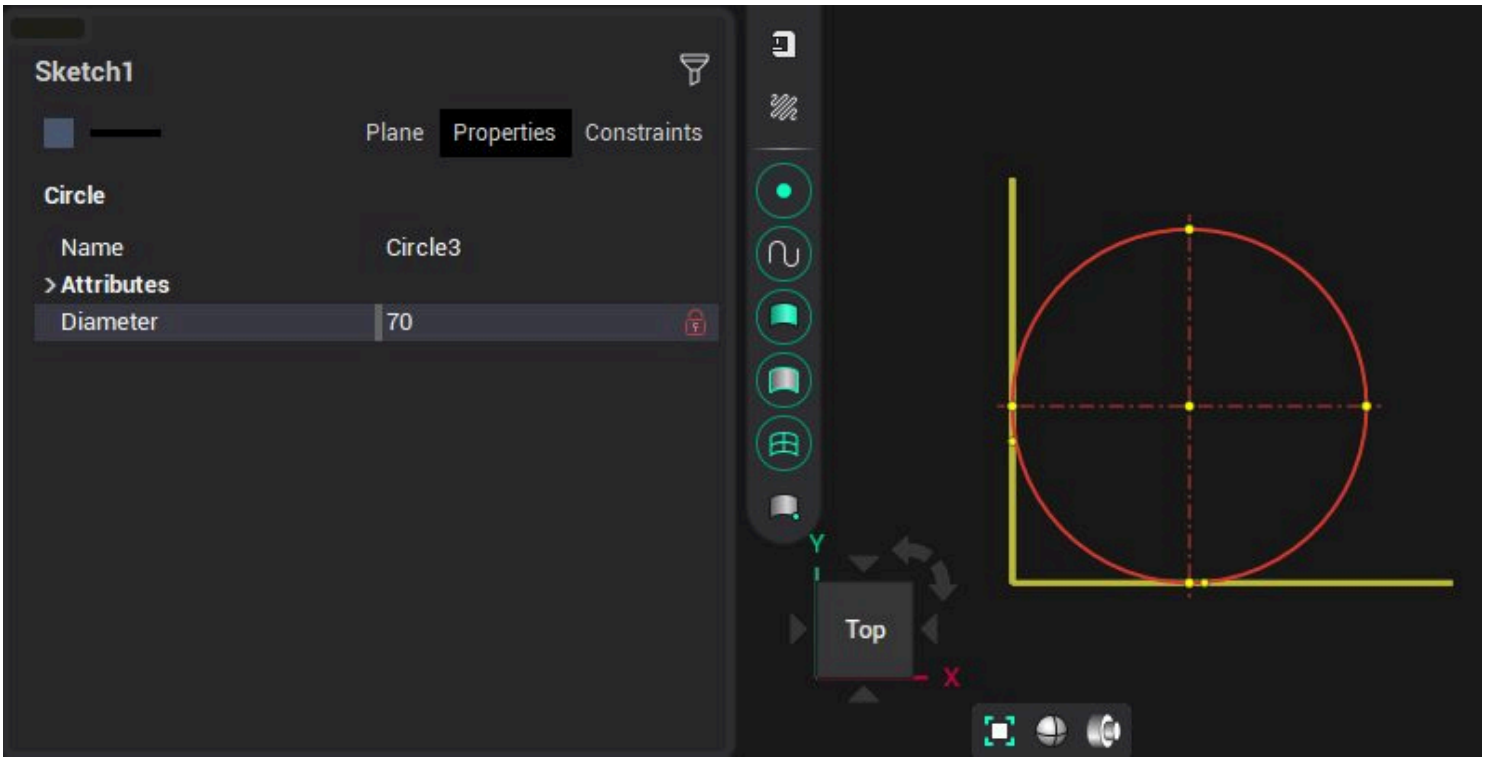
Circle by three tangents



To construct a circle, you must sequentially specify three objects to which the circle must be tangent with the [Ctrl] key pressed.



Or specify two objects and enter the radius of the circle:



Rectangle

There are 3 construction options available for a rectangle:

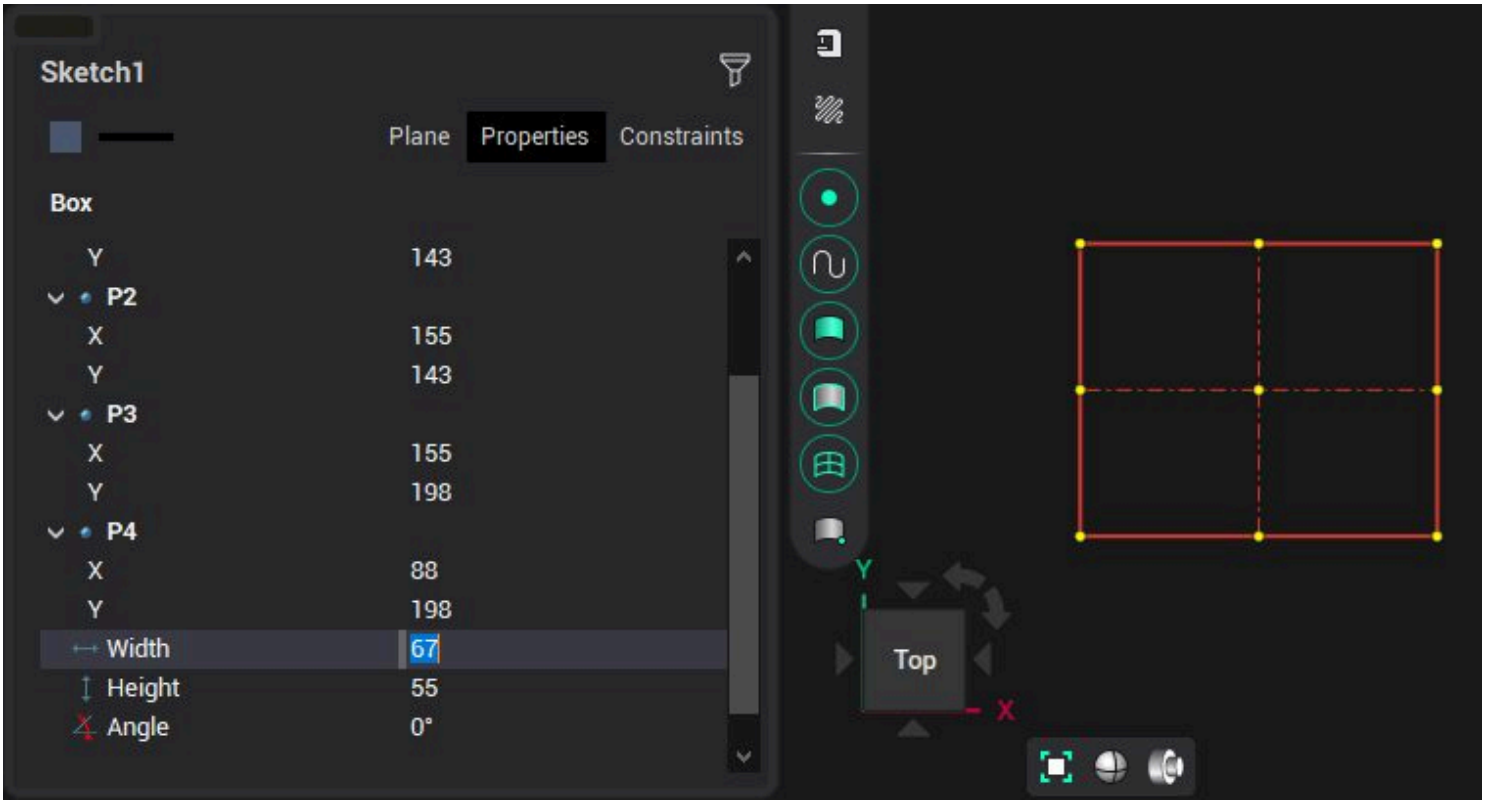
- By two points
- By side and height
- By center and point

By two points



The rectangle is built by specifying two diagonal points interactively, or by specifying the coordinates of the first point and the width/height of the rectangle in the inspector. You can rotate a rectangle only by changing the angle of rotation in the inspector.

Rectangle options in the Property inspector window:

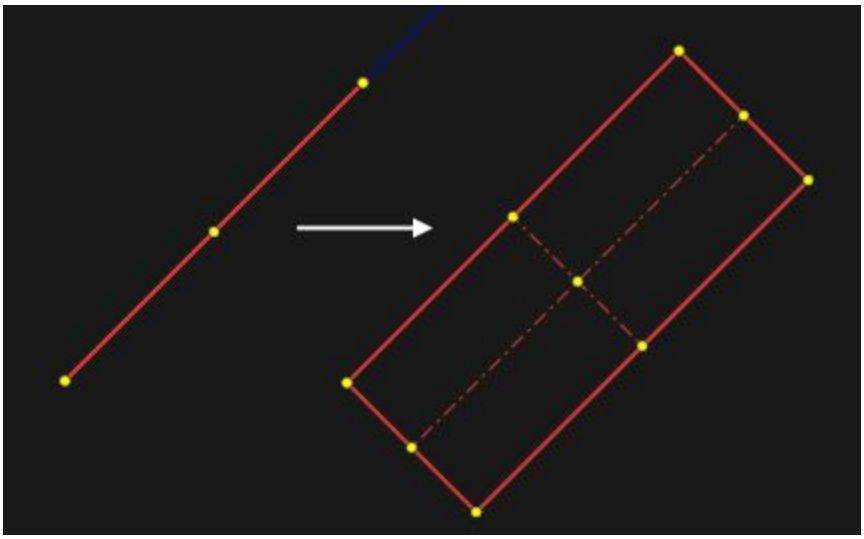


When defining a rectangle, you can use the help of guide lines. By default, guides are offered from the point being edited and the opposite point at angles of 0°, 45°, 90° and along a straight line passing through these points.

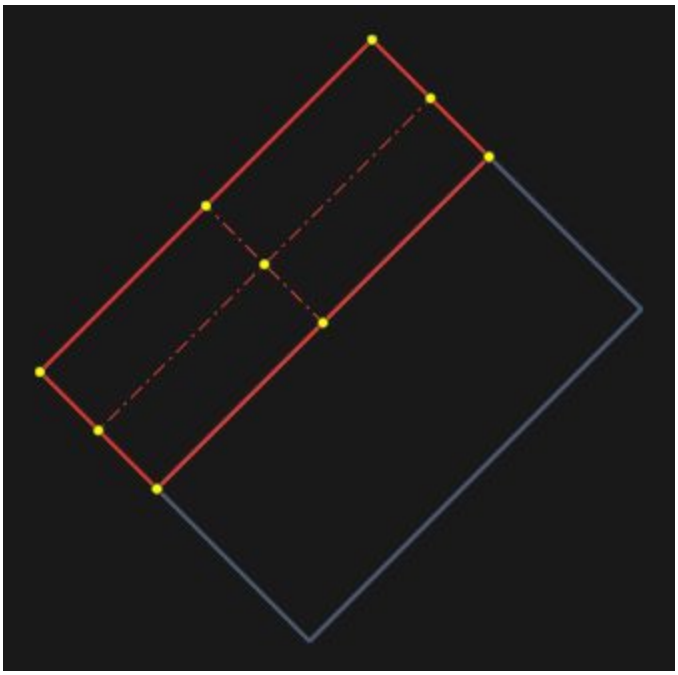
By side and height



First, one side of the rectangle is specified, and then it is stretched in height. This allows you to build rectangles at the angle you need.



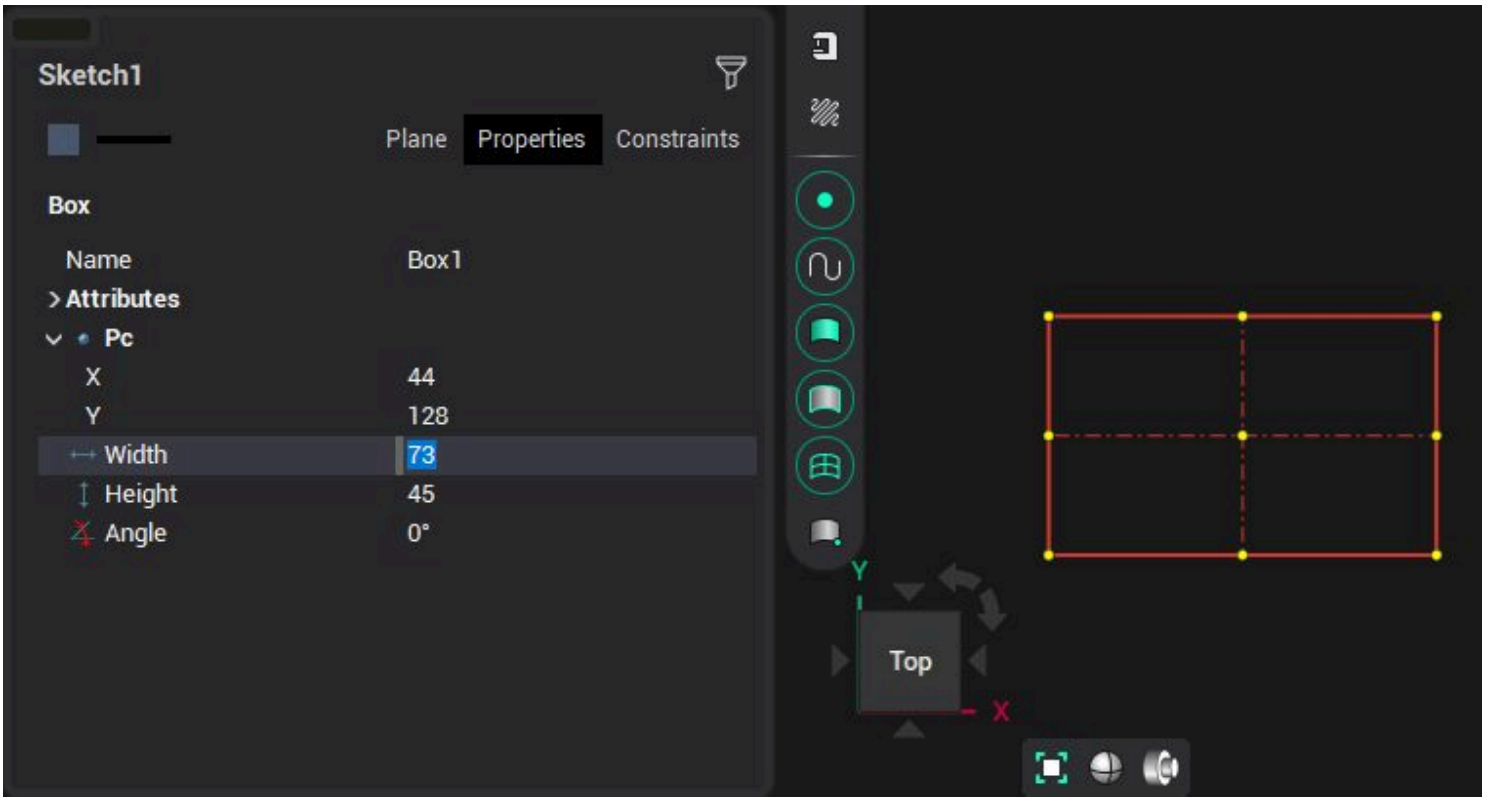
You can also specify an existing line and the side will be copied from this line.



By center and point



In this method, the center of the rectangle is specified first, and then one of the points of the rectangle.

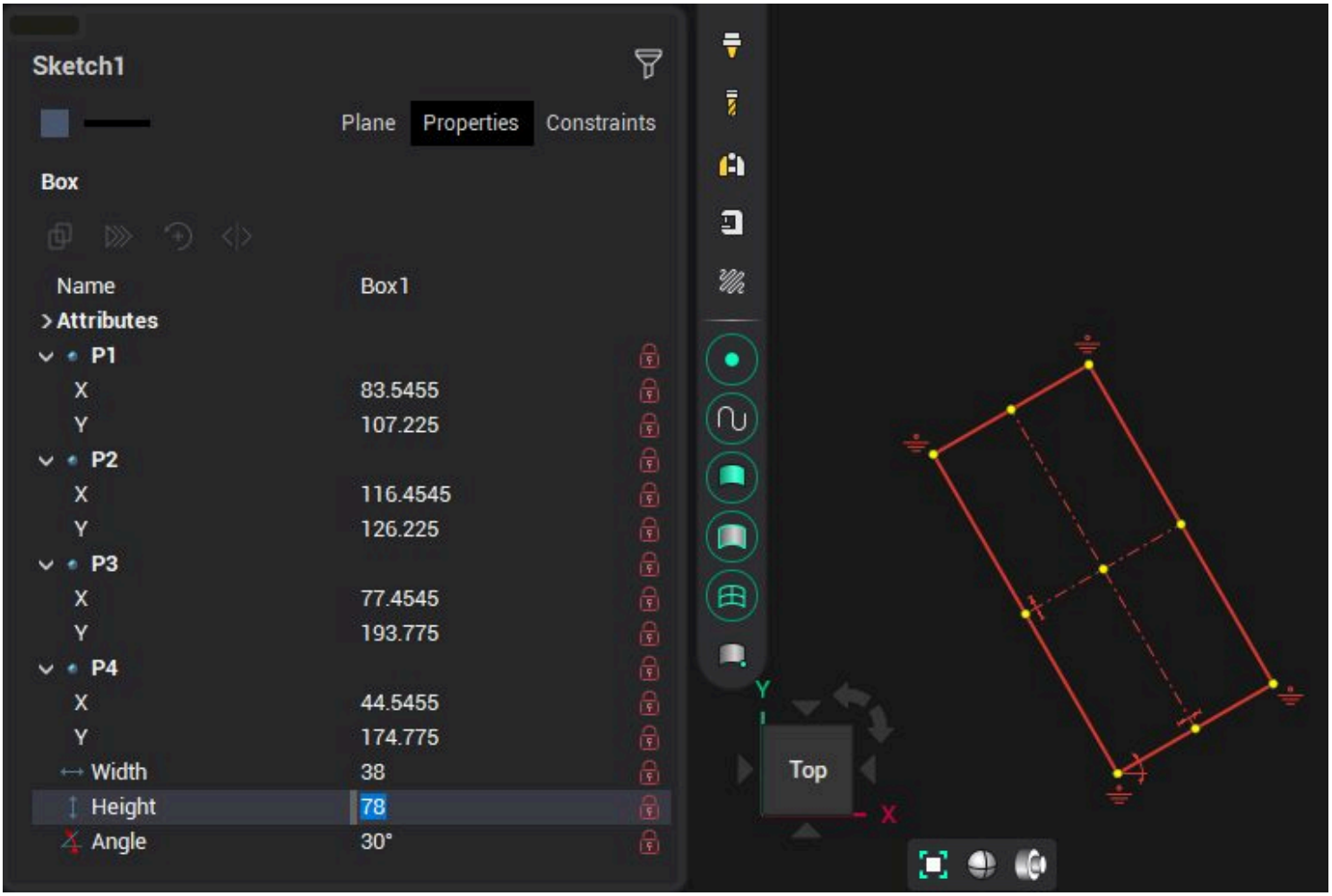


The inspector displays this center, as well as the width, height, and rotation of the rectangle.

Edit rectangle

Selected rectangle object can be modified in graphical window or you can change it's parameters in the inspector.

When editing a rectangle, all these parameters can be fixed by clicking the "lock" symbol in the inspector, or the appropriate icon on the rectangle itself.



When editing a rectangle by moving it's points, you can use the help of guide lines. By default, guides are offered from the point being edited and the opposite point at angles of 0°, 45°, 90° and along a straight line passing through these points.

You can select a line that is part of rectangle by holding the [Ctrl] key while clicking that line in the graphic window. If you "tear off" the line from the rectangle, the rectangle will be split into lines.



Contour

A contour is a set of lines and/or arcs connected by extreme points. There are two options for constructing a contour:

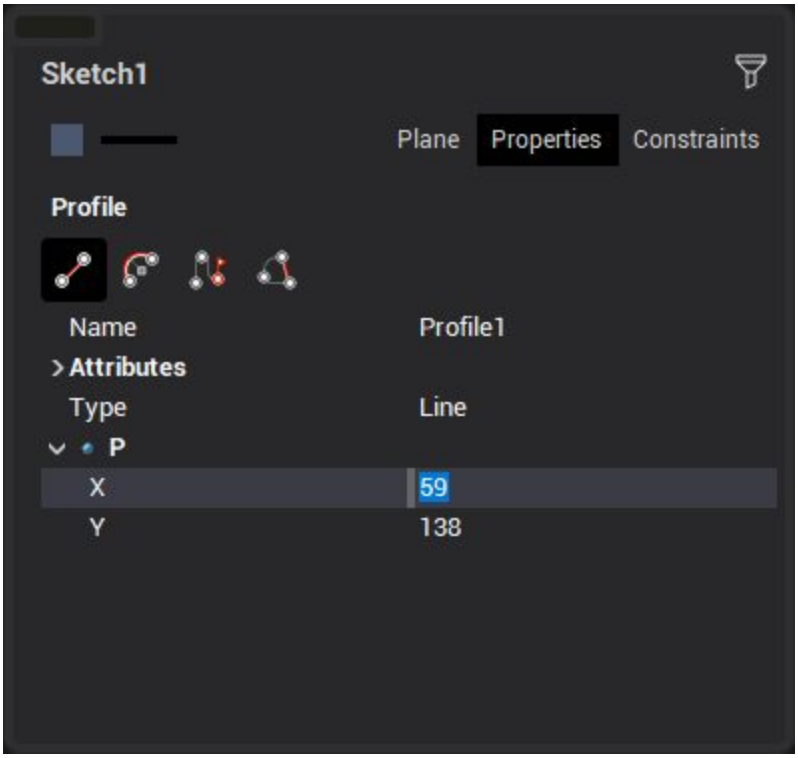
- Build by element
- By zone or border

Build by elements

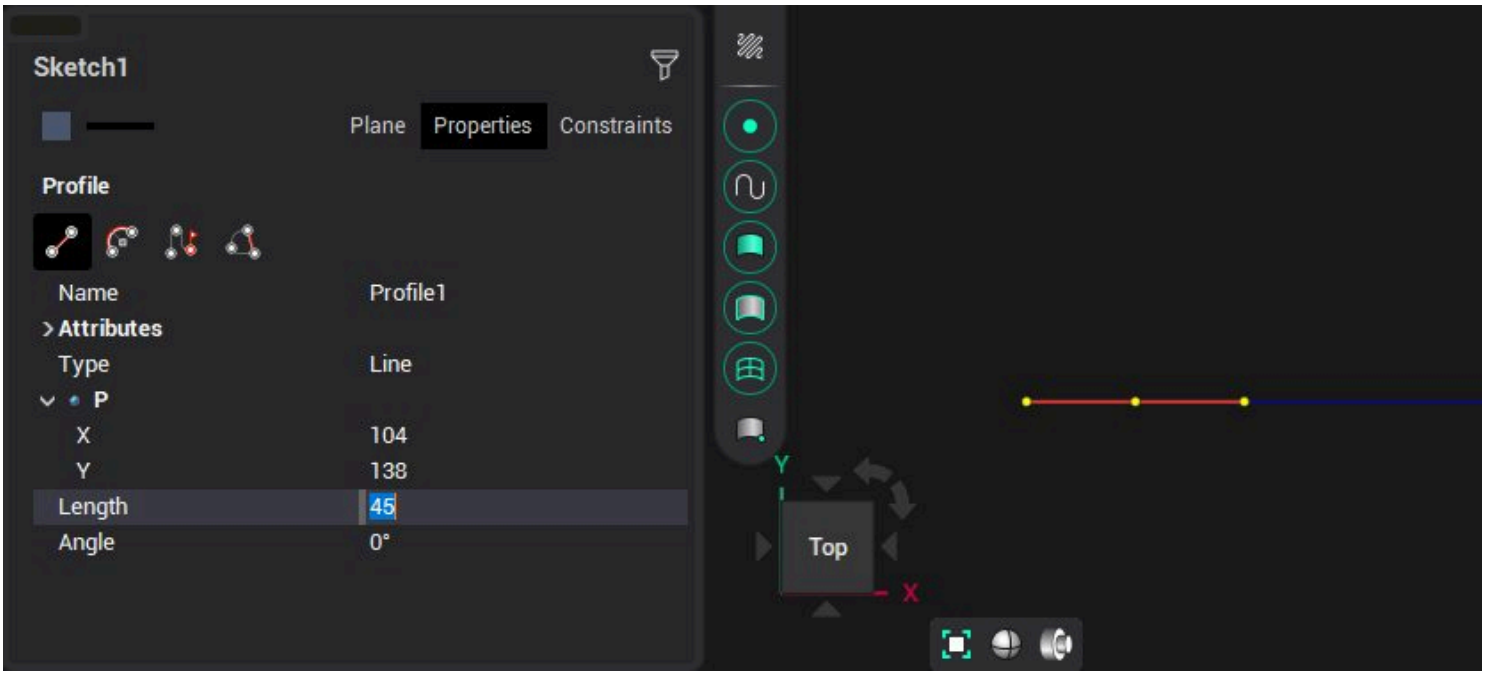


The building is carried out by successive construction of lines/arcs by specifying points interactively or through the inspector.

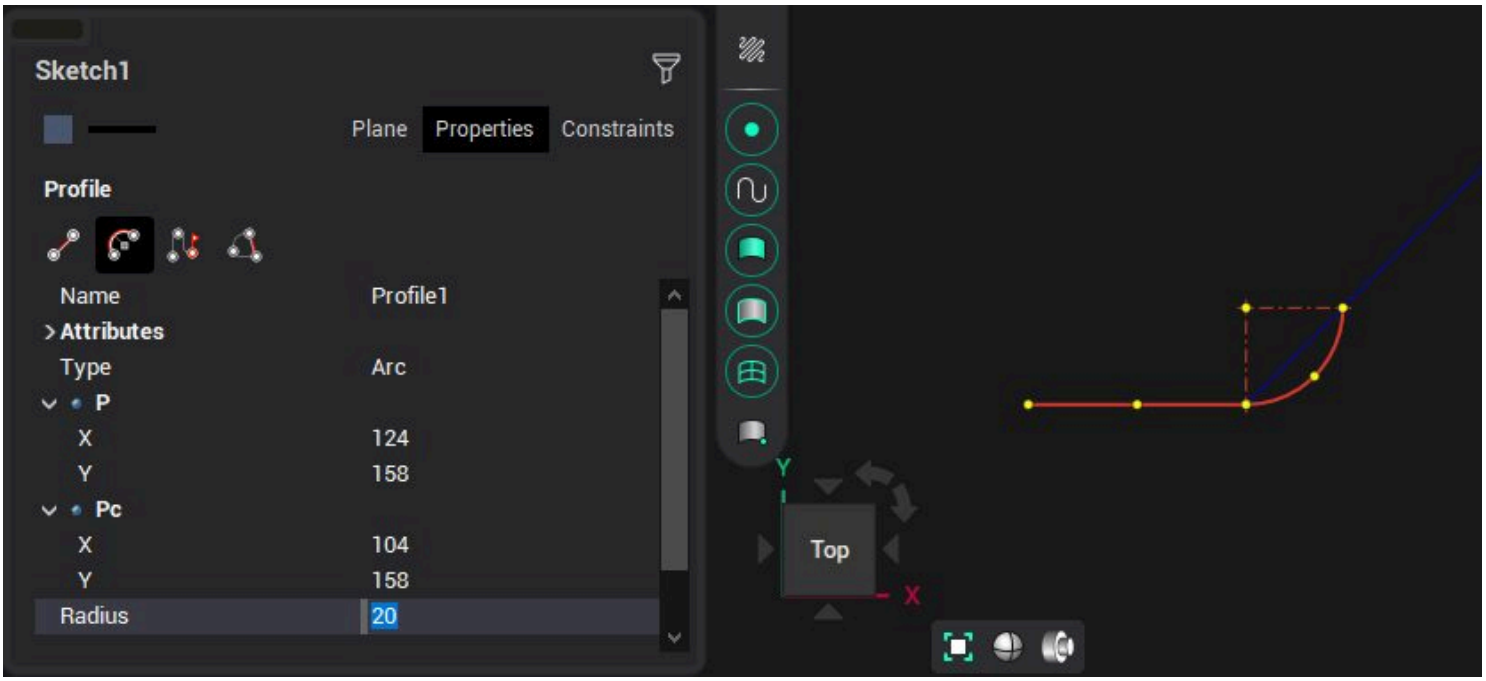
At the first step, you need to specify the starting point of the contour, or enter its coordinates in the inspector.



Next, you also need to specify the next point, or enter the length and angle of the line.



Or specify an arc. The coordinates of the next point and the center point of the arc, its radius and angle to the tangent to the previous element will be available in the inspector.



To switch between a segment and an arc (and vice versa), press the appropriate button:



Or by pressing [Ctrl+Space].

Complete defining contour by double-clicking at final point or click one of the buttons:



 ends the contour by the last confirmed element,  creates a closed contour by connecting last confirmed point to first with current selected element type.

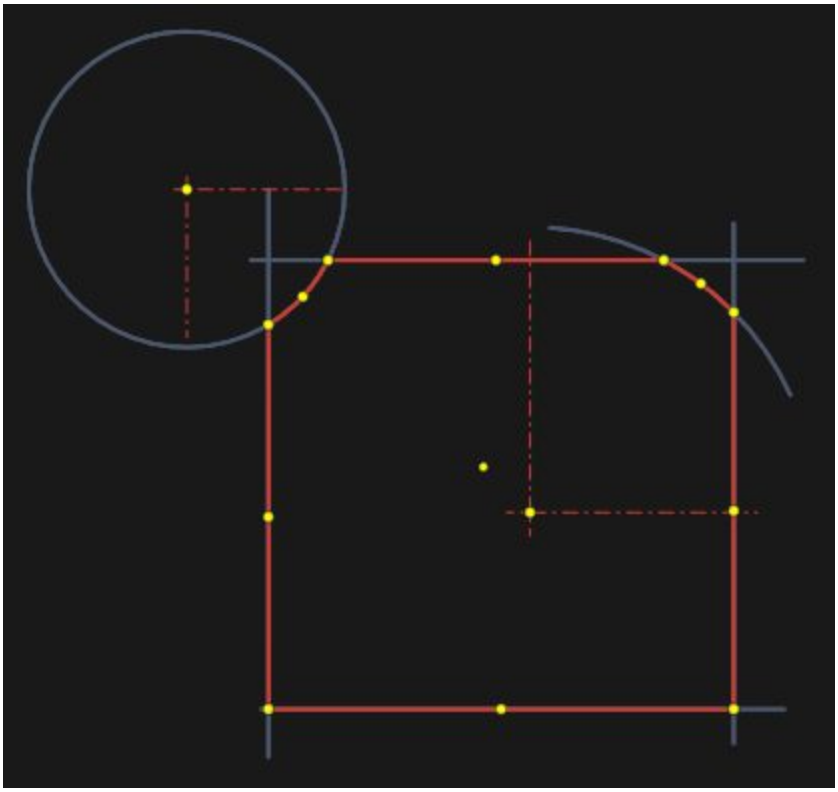
You can also end the construction by double-clicking the mouse button when building the last element

By zone or border

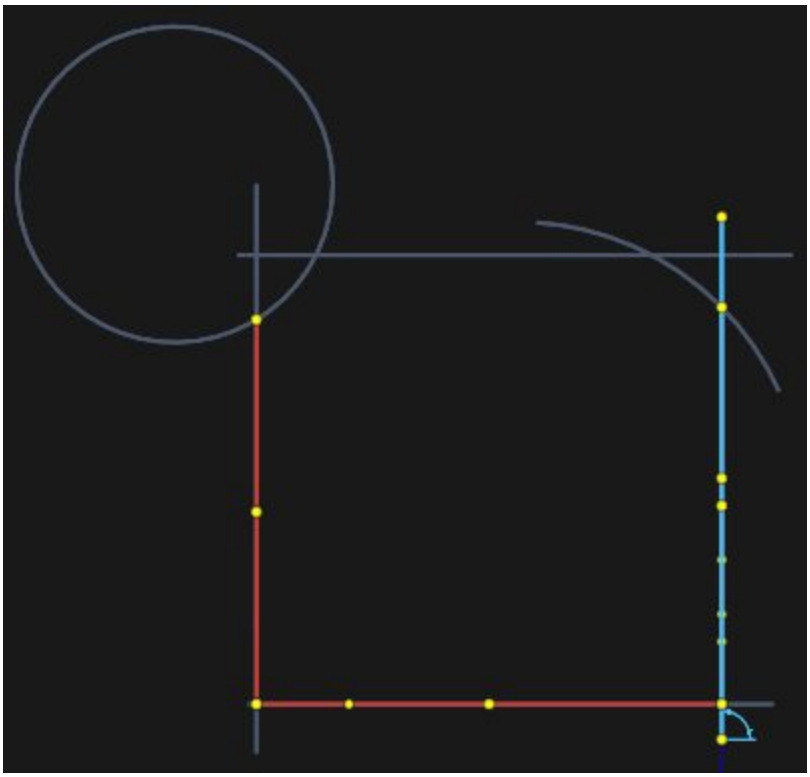


In this method, the contour is built on the already existing elements.

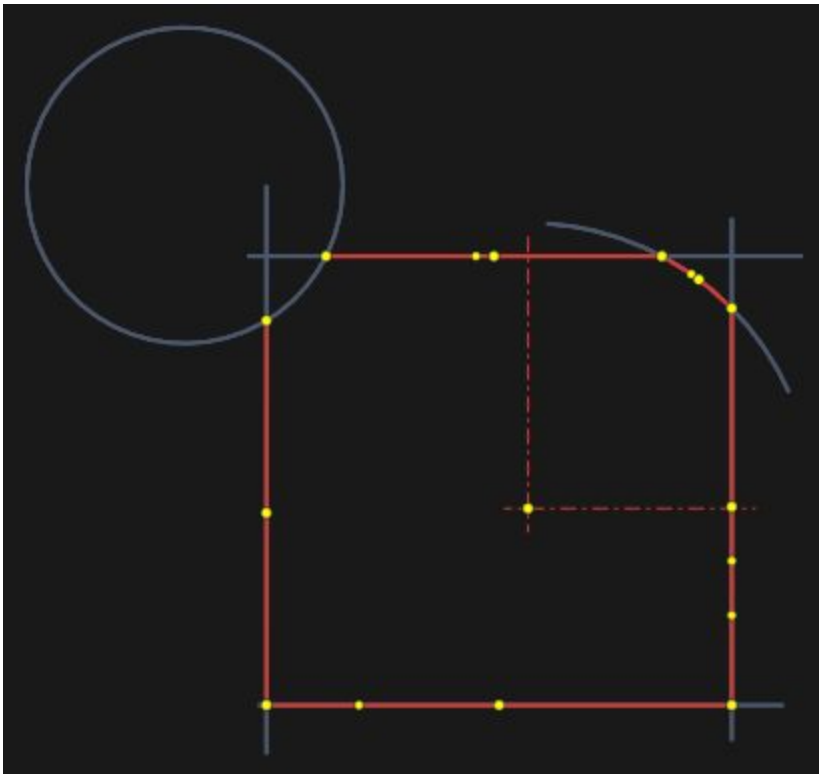
When specifying a point inside a group of elements, a cont will be created. The components of this contour are located in this zone.



When selecting the contour boundary, its elements will be created automatically along the chain to the nearest intersection with another object.



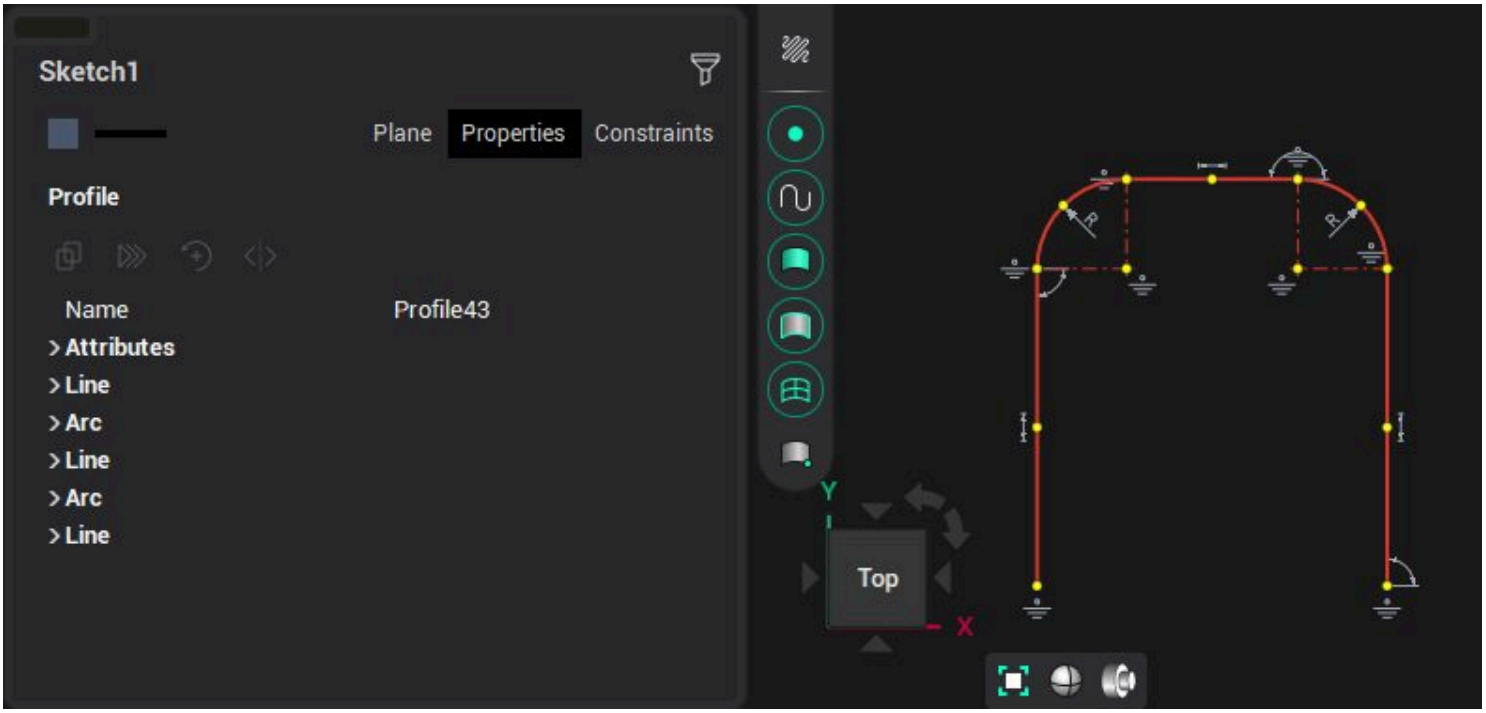
To continue the contour, select its next element. If the specified element is not connected to the previously built one, the system will try to complete it along the chain.



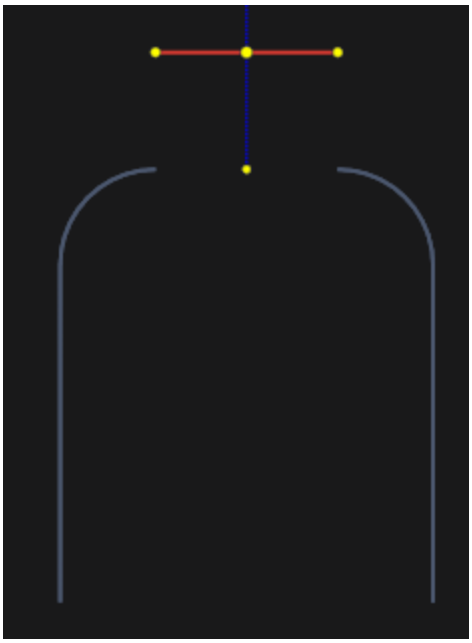
Edit contour

Selected contour can be modified by changing it's points in graphic window. You can change contour elements properties in the inspector.

In edit mode, the contour will be presented as a set of lines and arcs.



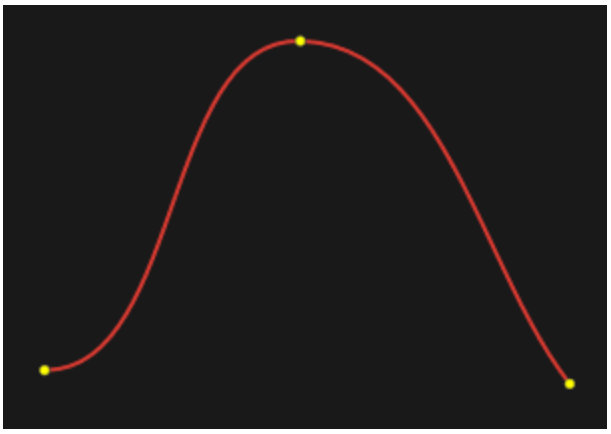
If you select the side of the contour with pressed Ctrl, only the highlighted contour element will be selected. If you separate an element from a contour the contour will be split into elements.



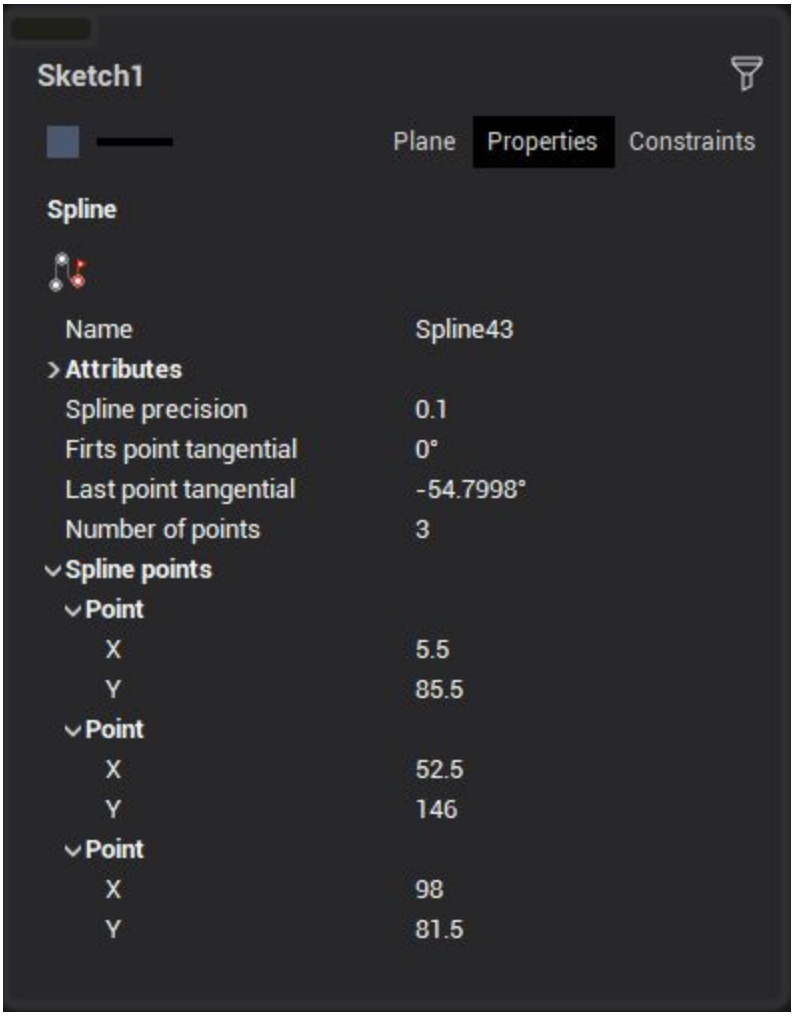
Spline



Spline is a piecewise curve interpolation with lines. Spline passes through given points and is tangential to given start and end angles.

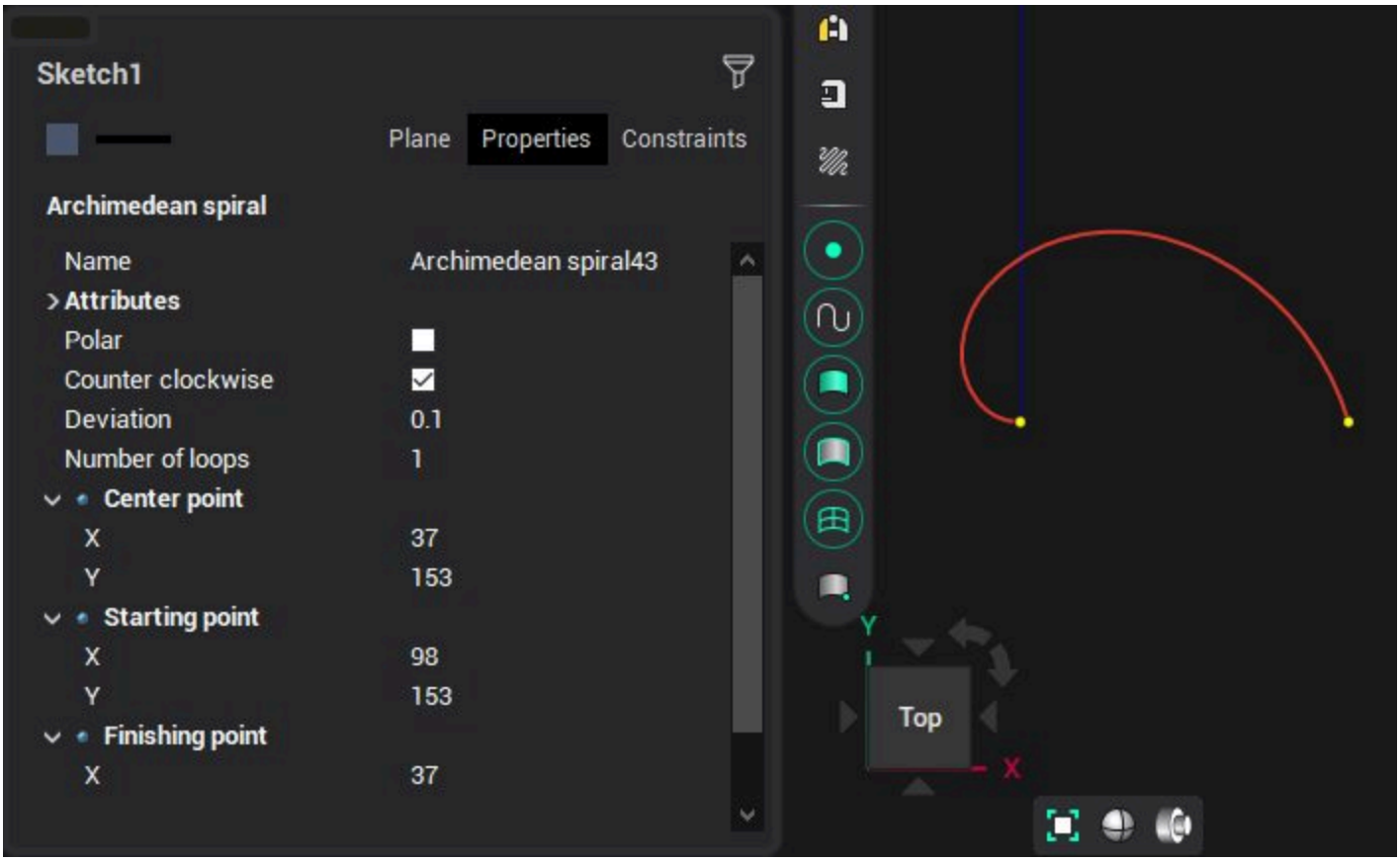


It is possible to input spline points directly in the graphic window. Otherwise the left inspector can be used to input number of points and point coordinates.



Click  or double-click at last point to finish spline.

Archimedean spiral

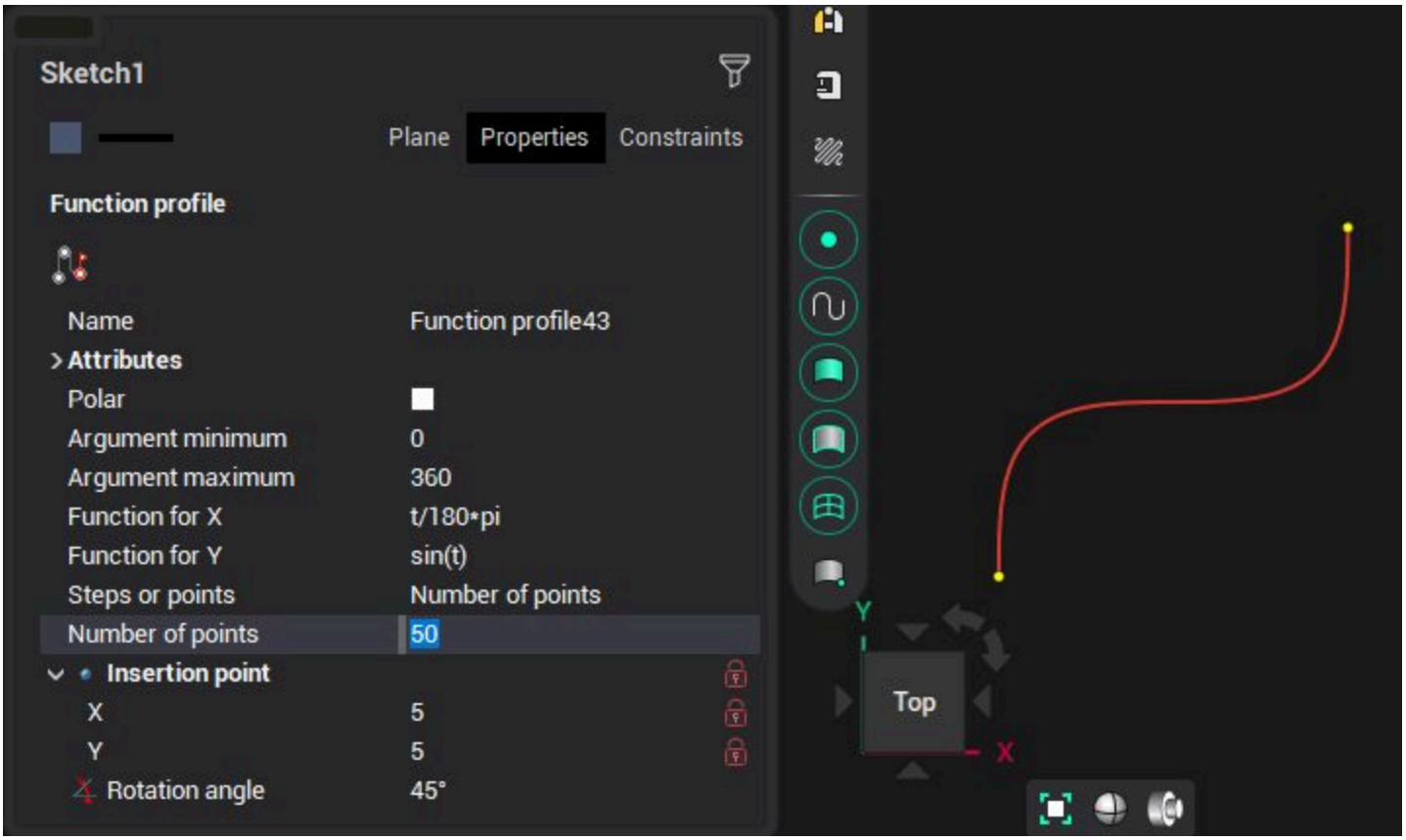


Create Archimedean curve defining center point, start point and end point. You can set the points interactively, or provide coordinates in inspector panel.

Function profile



Create curve that passes through points whose coordinates are calculated by expression provided by user.



Points are calculated by substituting t with values from Argument minimum to maximum into expressions for X and Y. It is possible to set curve insertion point and add rotation around that point.

Click the  button or click in the graphic window to finish curve definition.

Expressions recognizes common mathematical functions:

- sin(t) - sine of t
- cos(t) - cosine of t
- tan(t) - tangent of t
- cotan(t) - cotangent of t
- arctan(t) - arc-tangent of t

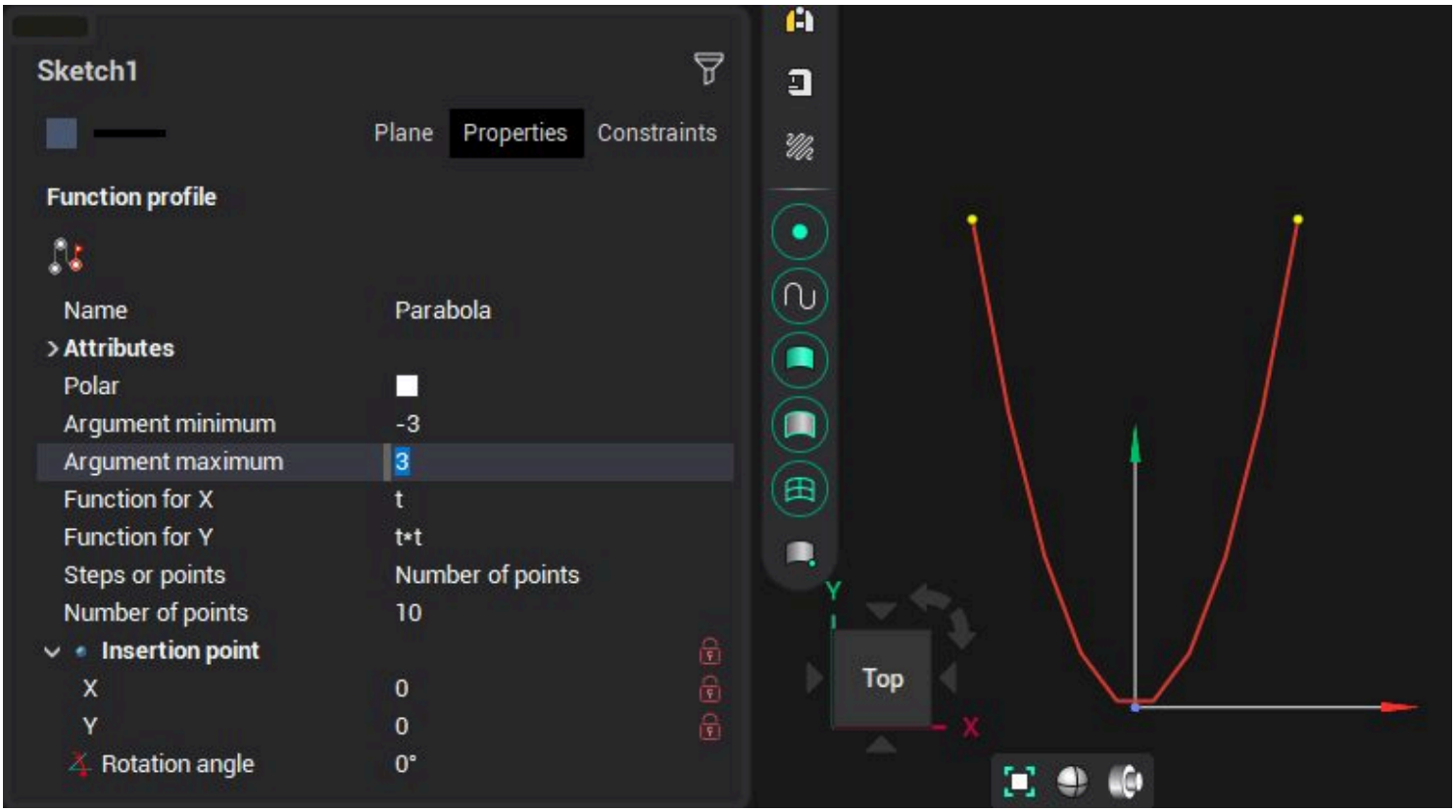
For trigonometrical functions the argument is in radians. Pi is the π number.

- exp(t) - e raised to the power of t

- ln(t) - natural logarithm of t
- sqrt - square root of t

Examples of function profiles

Quadratic function (parabola): Y = X2



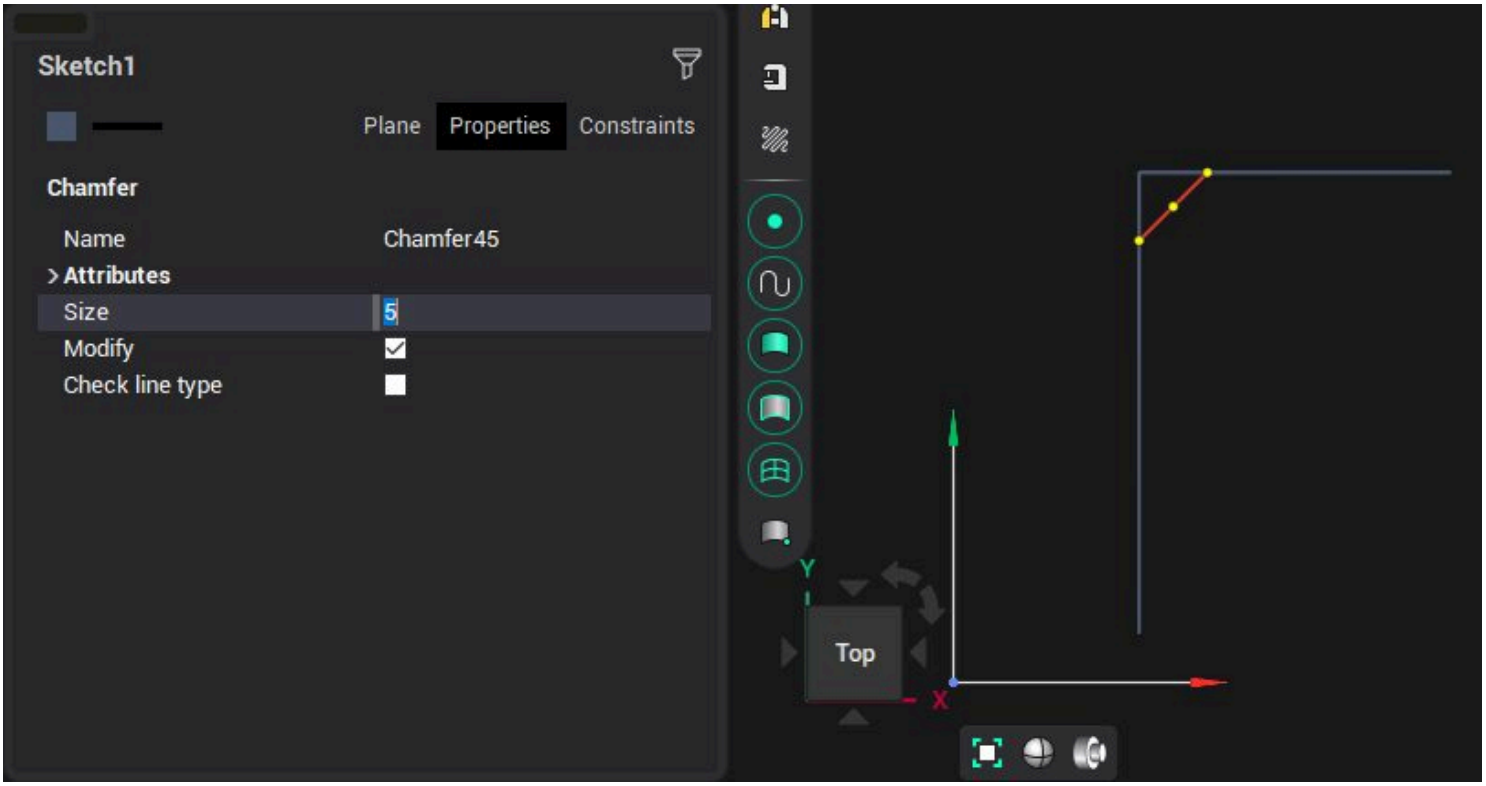
Exponential: Y = 10x

Chamfer and Rounding

These two objects are built automatically. You just need to move the cursor to the intersection of two objects, enter the size value and click the mouse button to complete the construction. If for some reason the system cannot build the required element, then the elements must be sequentially selected.

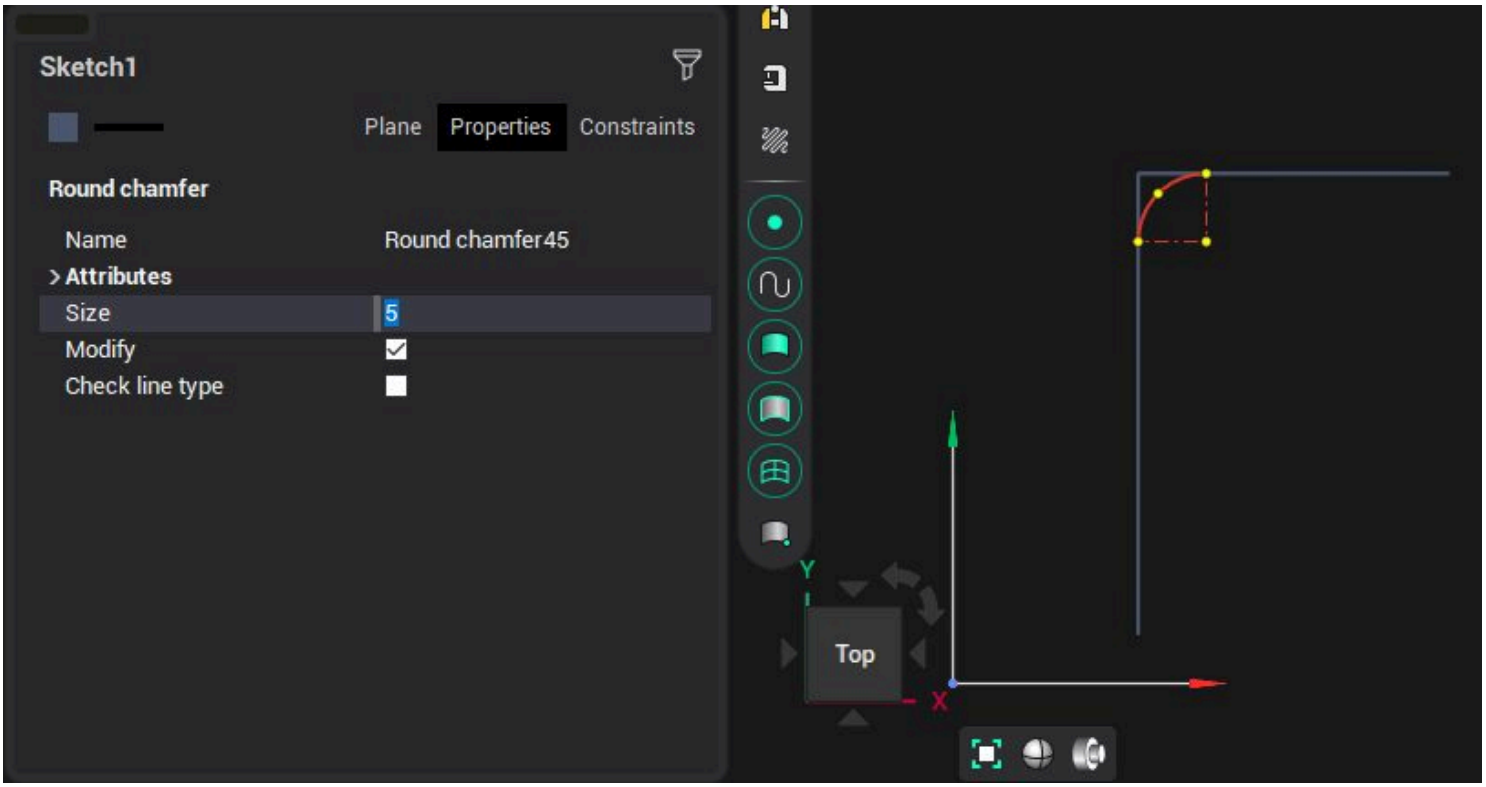
Chamfer



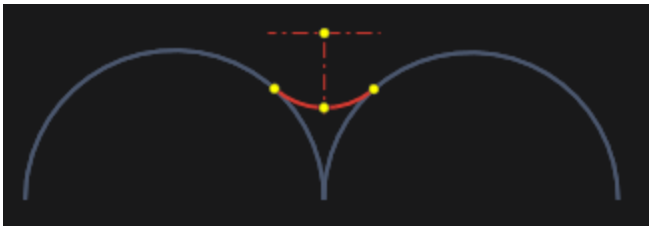


Rounding

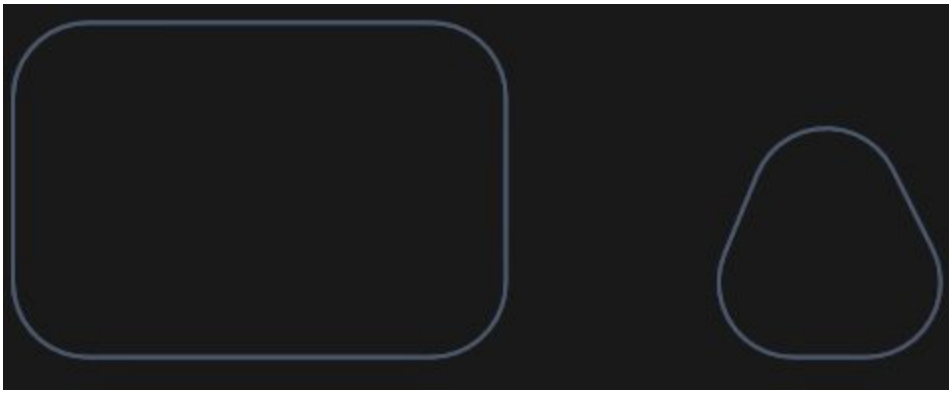




It is also possible to build chamfers and roundings between arcs.



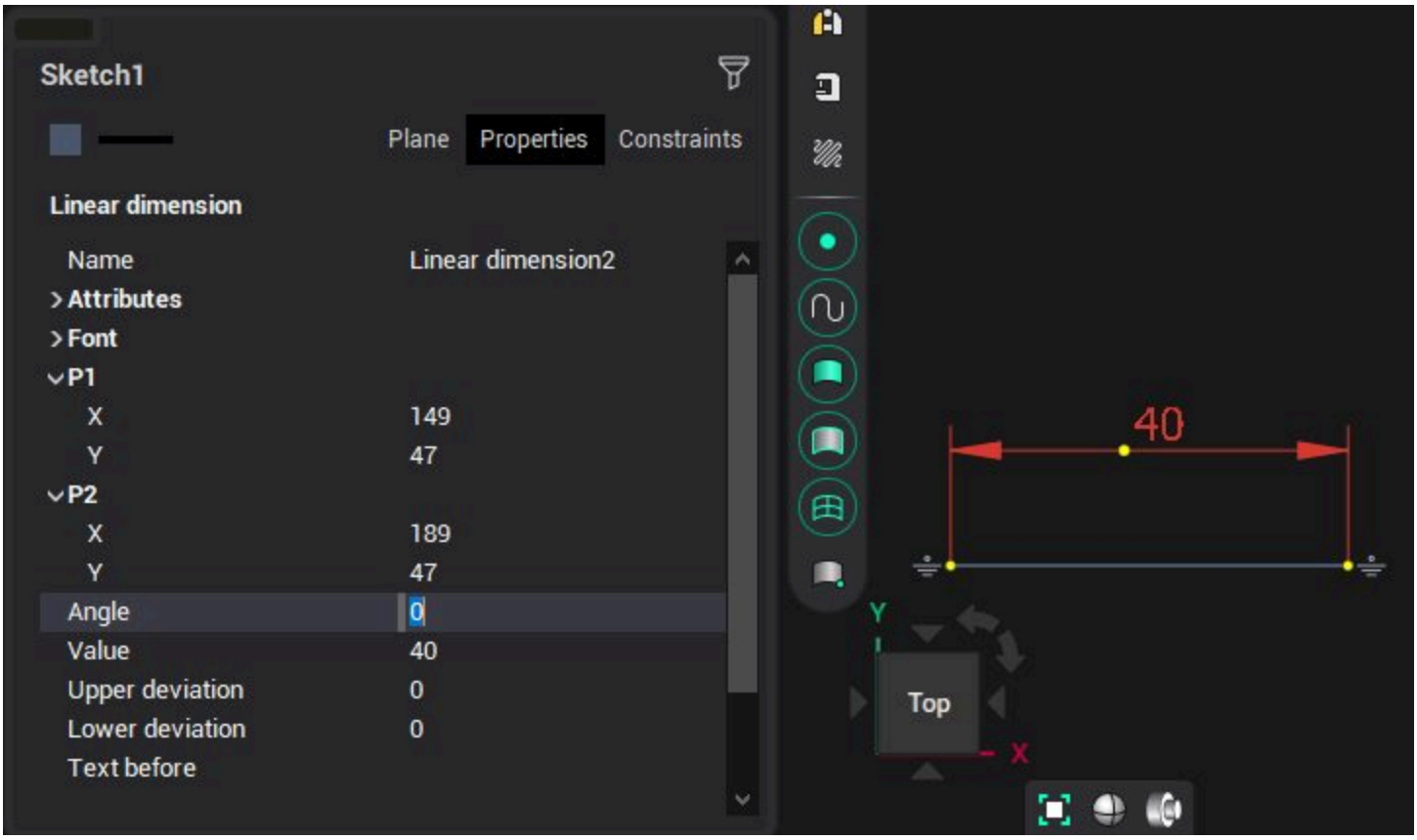
If you double-click on a rectangle or contour, then all corners of such element will be rounded.



Dimension

There are three types of dimensions for dimensioning drawings: linear, radial and angular.

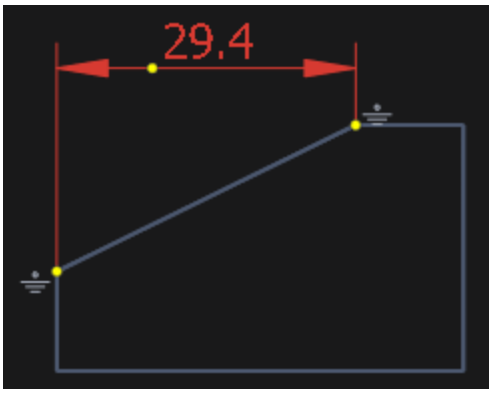
Linear dimension



When constructing a linear dimension, you must specify the start and end points, the size of the leader, and the offset of the beginning of dimension text relative to the first extension line. The dimension value is calculated automatically. The inspector sets the slope of the dimension line relative to the x-axis, the values of the upper and lower deviations, the text before and after the dimension value, and the text font.

It is also possible to create a linear dimension by specifying an existing line. In this case, you do not need to specify the start and end points, and the system will have a link between the dimension and the line. Subsequent editing of a line or dimension will change both objects.

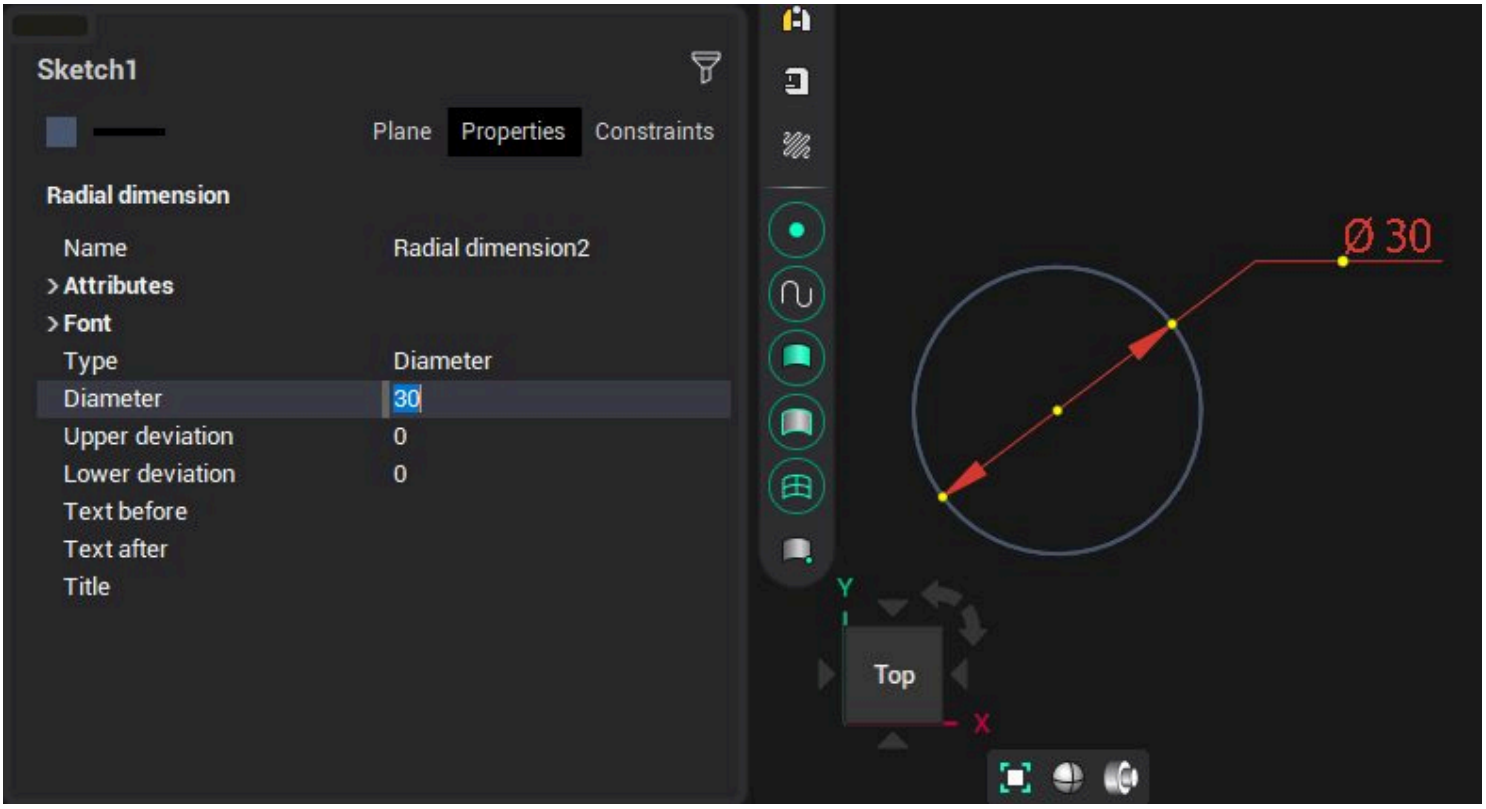
If the dimension line is not horizontal or vertical and needs to be normalized, press [Shift] while constructing the dimension.


29,4

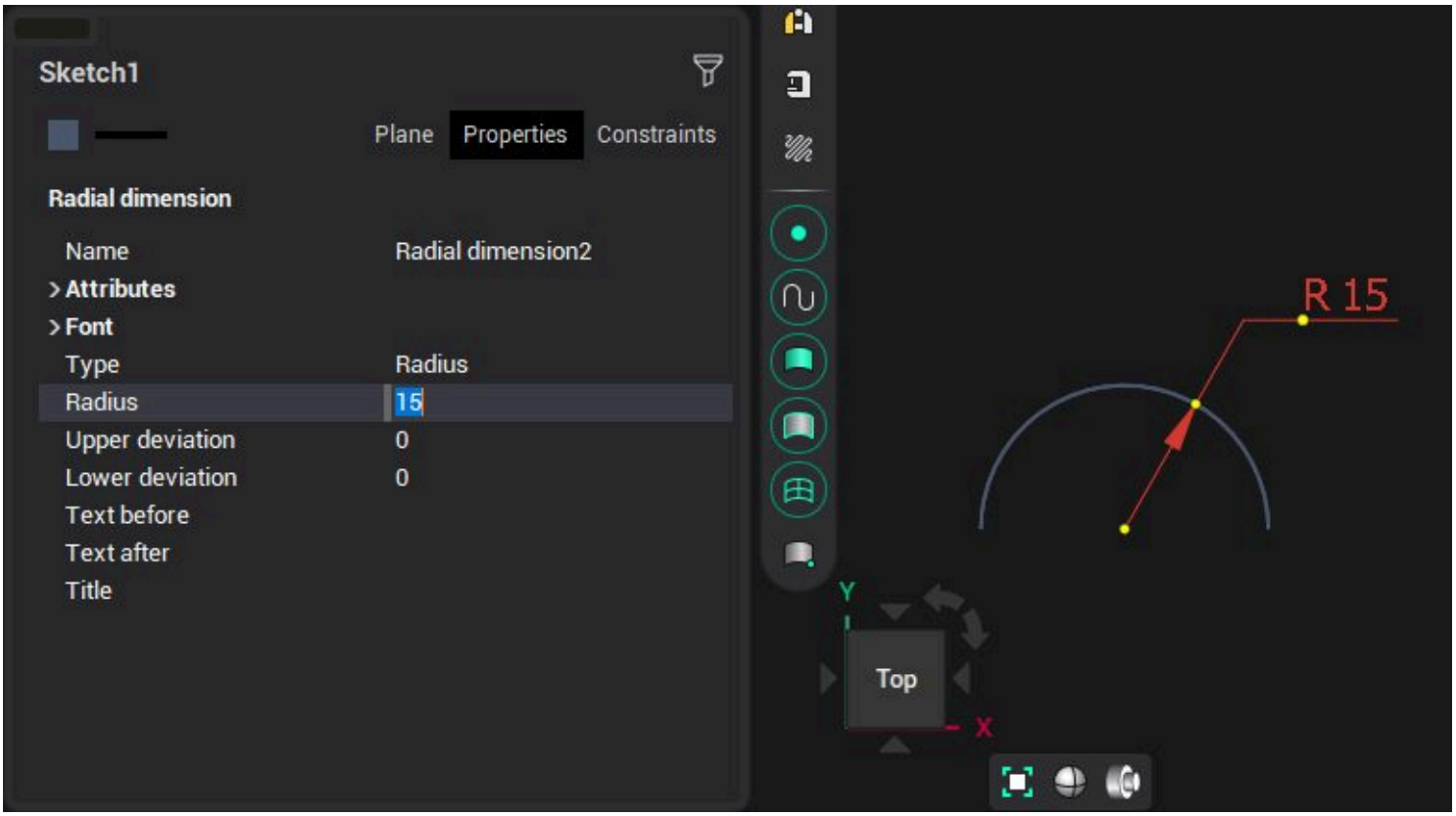
Radial dimension



To construct a radial dimension, you need to specify an existing circle or arc and the point where the text starts. The dimension value will be automatically determined. The inspector sets the top and bottom tolerances, the text before and after the dimension value, and the font of the text.



When dimensioning a circle, by default, a dimension of the Diameter type will be built, as in the picture above. When dimensioning an arc, a dimension of the Radius type will be built by default:



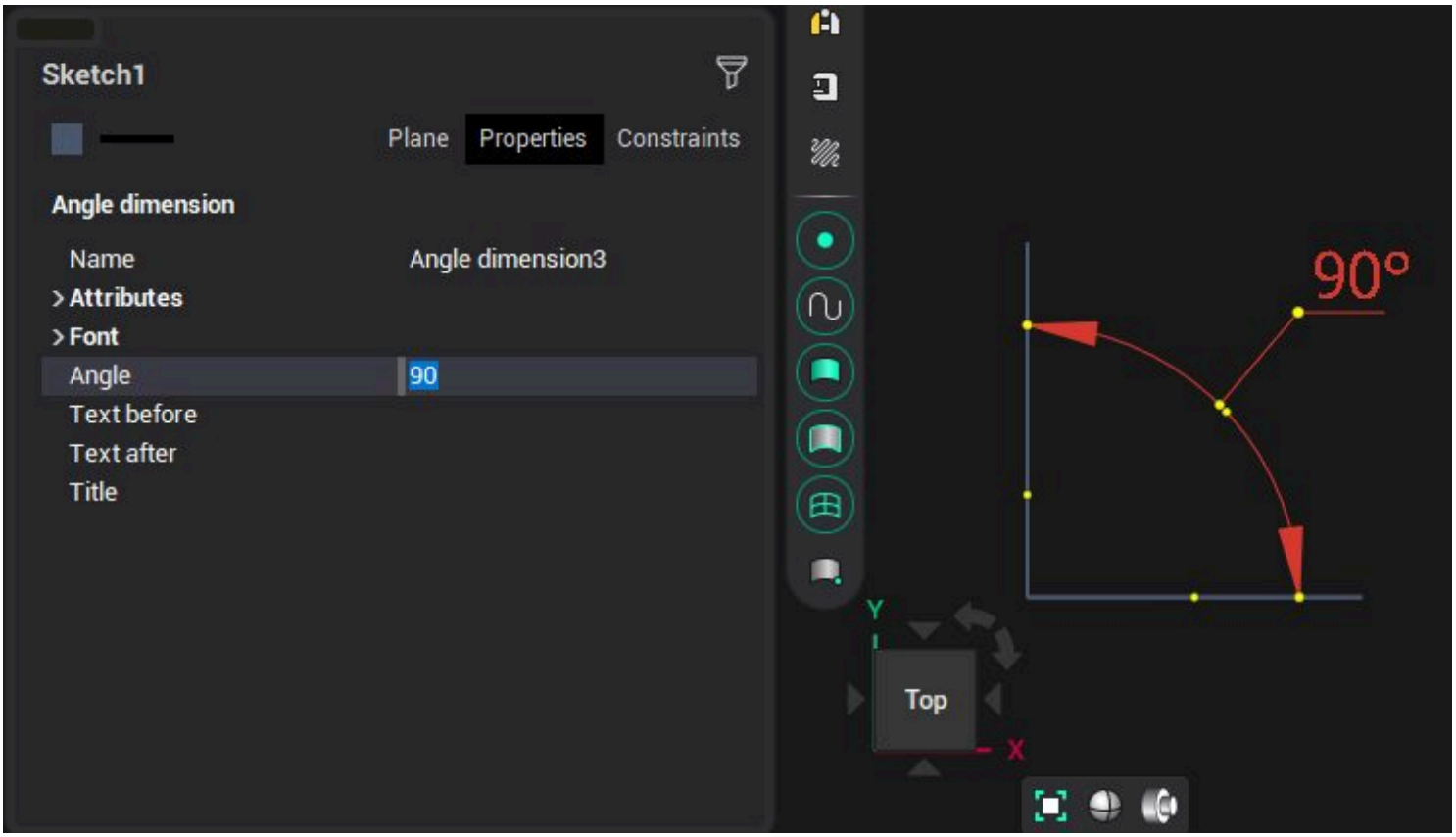
You can switch the type in the field of the same name at any time.

Just as in the case of a linear dimension, the system will determine the connection of these objects and further editing of one of them will change the other.

Angular dimension



To build an angular dimension, you must sequentially specify two lines and the start point of the text. The dimension value will be automatically determined. The inspector sets the text before and after the size value and the font of the text.

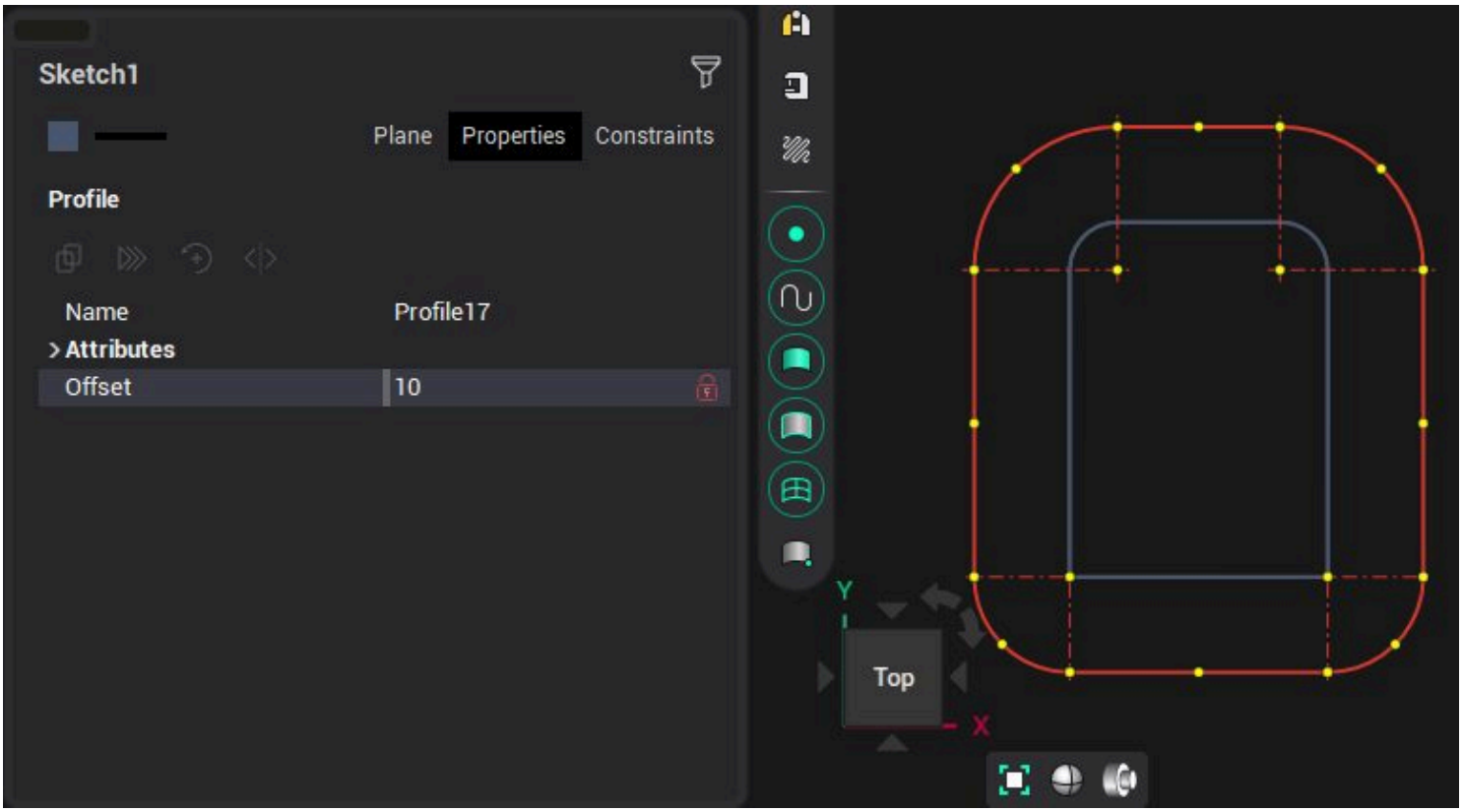


As a result of construction, the system will have a link between the dimension and the lines. Subsequent editing of a line or dimension will change both objects.

Offset

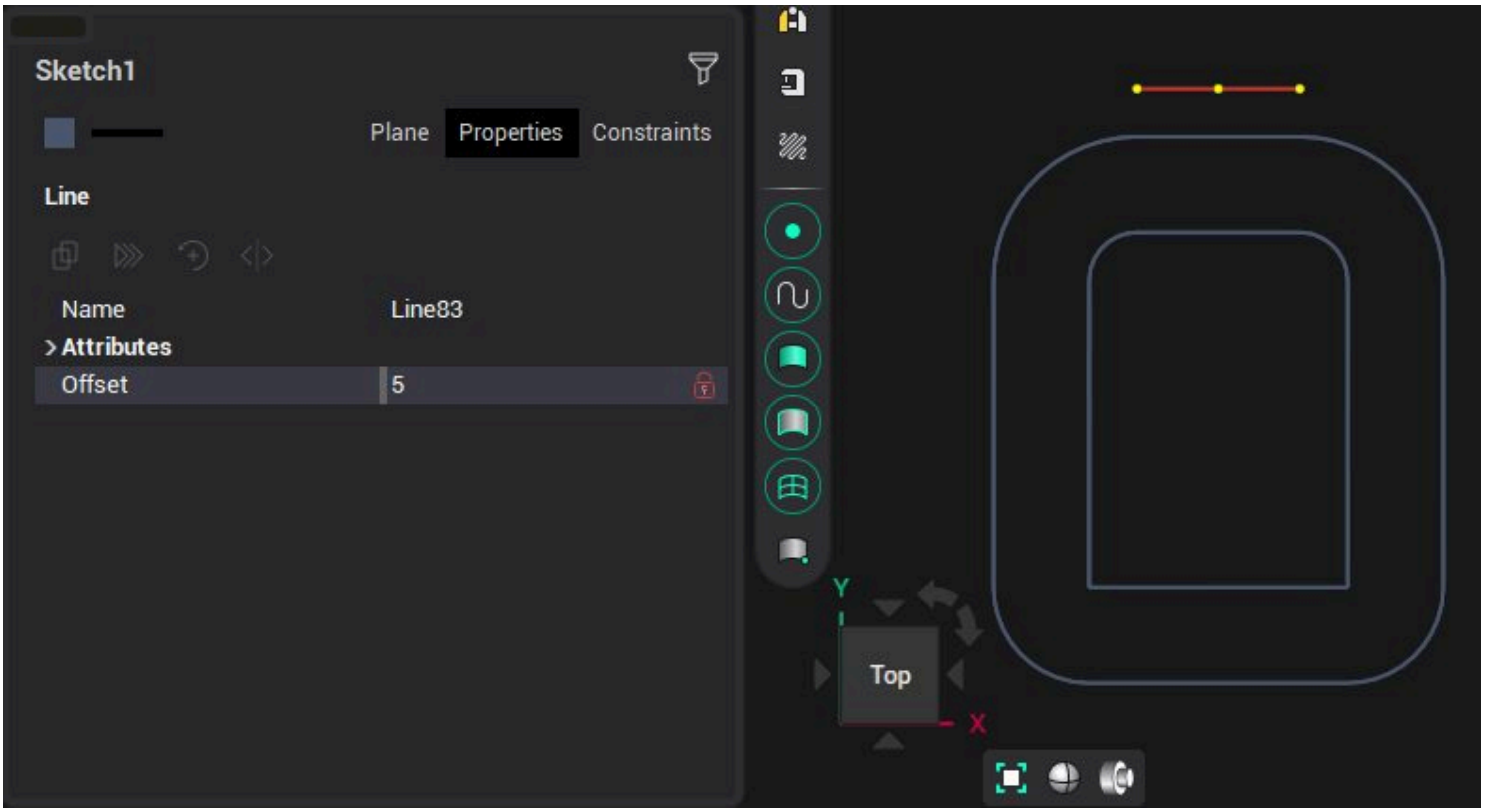


This function creates an offset object to the previously created one.



Offset can be built both interactively and by entering an offset value from the original object. The new object will be the same type as the original.

If Ctrl was pressed when selecting a compound object (rectangle, contour), then only the specified side (line or arc) will be selected to build the offset, and not the entire object.


Sketch1
Plane
Properties
Constraints
Line
Name
Line83
Attributes
Offset
5
Y
Top
X

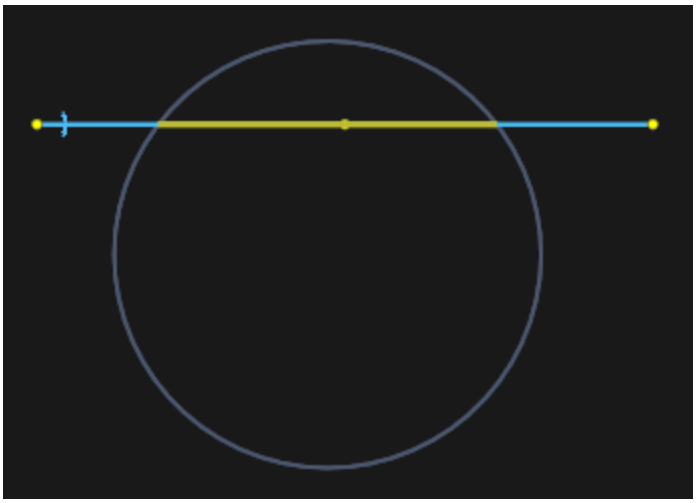
Annotations

-  Text. Allows you to create custom text.
- Hatch. Fills a closed sketch area with hatch lines.
- Surface roughness. Adds a surface roughness symbol.
- Leader. Adds explanatory text with a leader line.
- Geometric tolerance. Adds a feature control frame to specify a geometric tolerance.
- Datum feature. Marks a datum feature for specifying geometric tolerances.

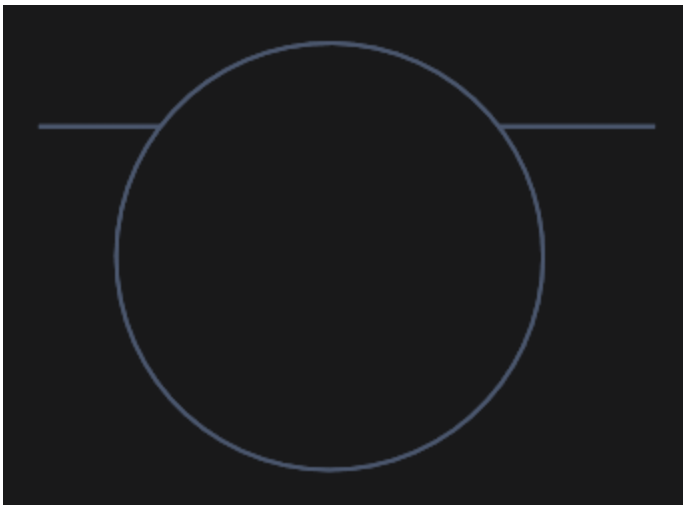
Trim function



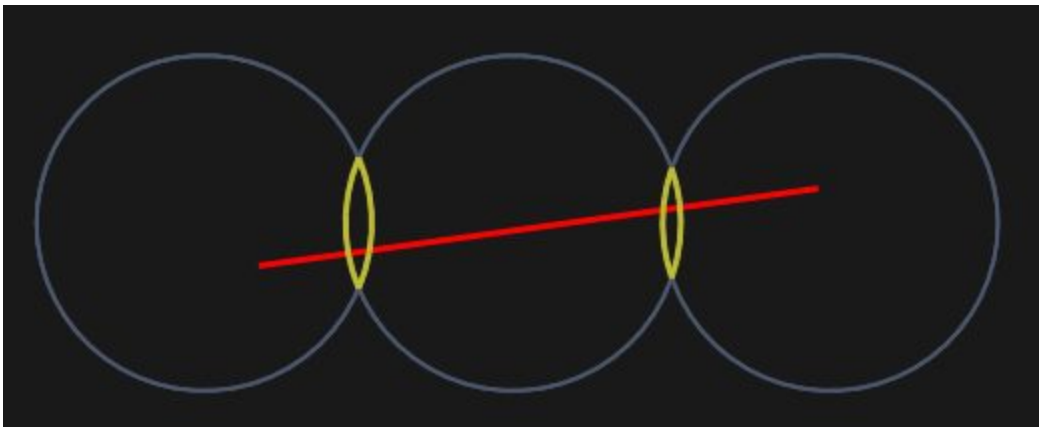
The function finds the intersection points of the specified element with others and highlights the area that will be deleted after confirmation by clicking.



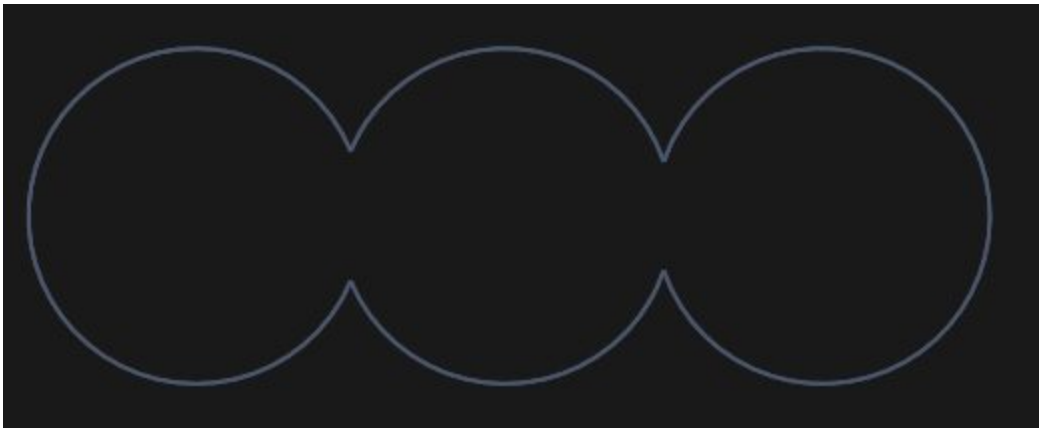
After confirmation:



You can also cut fragments of several elements at once by pressing the mouse button and dragging.



All intersections of objects with this imaginary line will be removed after releasing the mouse button:

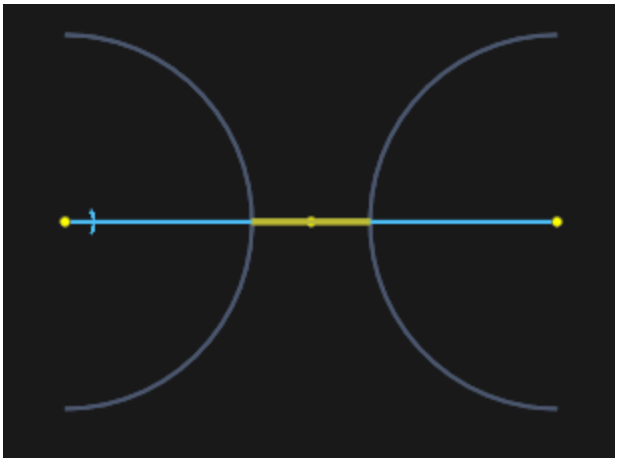


Split object



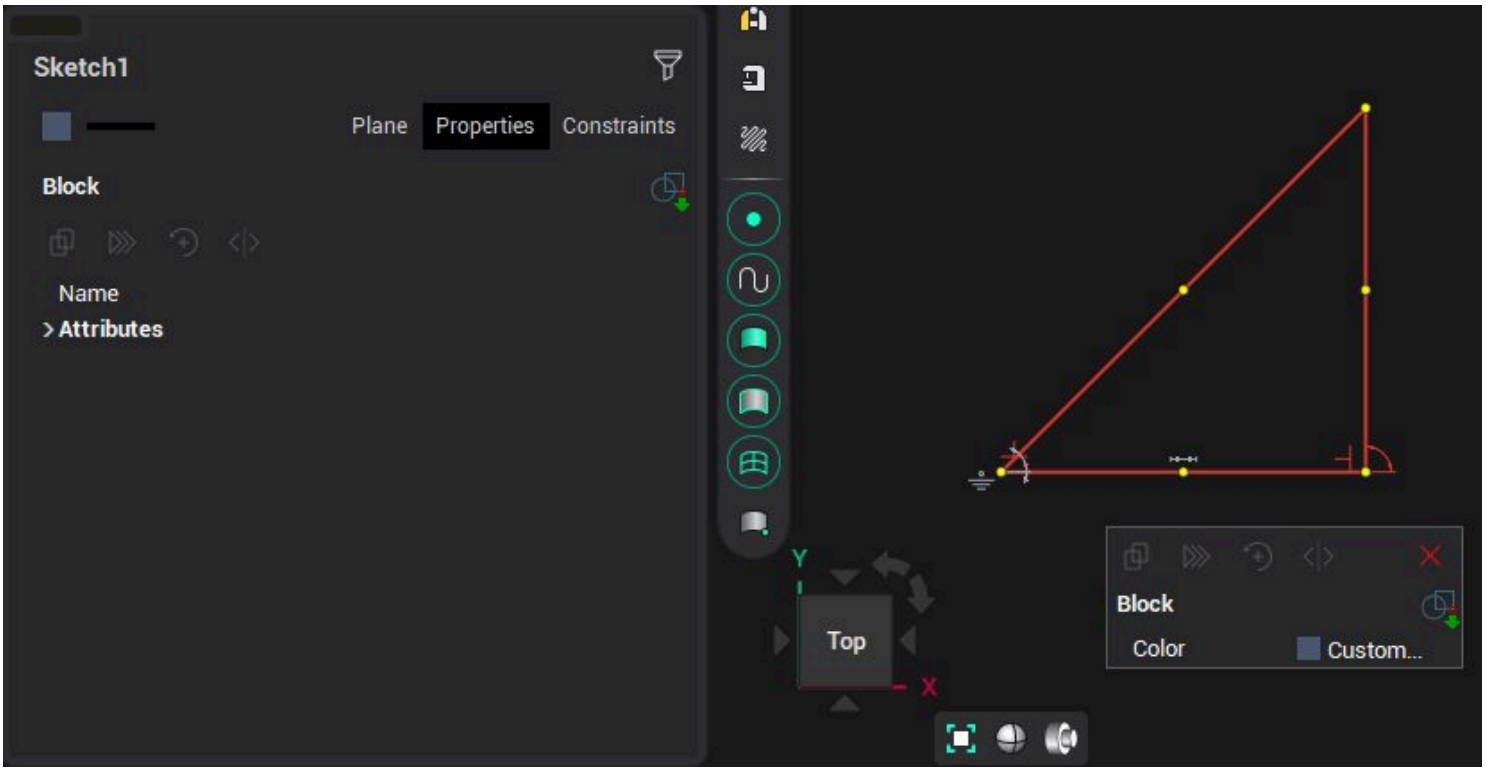
The function converts the specified compound element (rectangle, contour, named block) into its parts. If a block is pointed, it will be split into the elements from which it was built.

If you select an elementary object (line, arc, circle), this element will be split into several at the intersection points with other elements.



Additional functions

Additional functions are available when editing single objects and blocks. Their activation buttons are located in the inspector and the pop-up menu, which is called by the right mouse button.



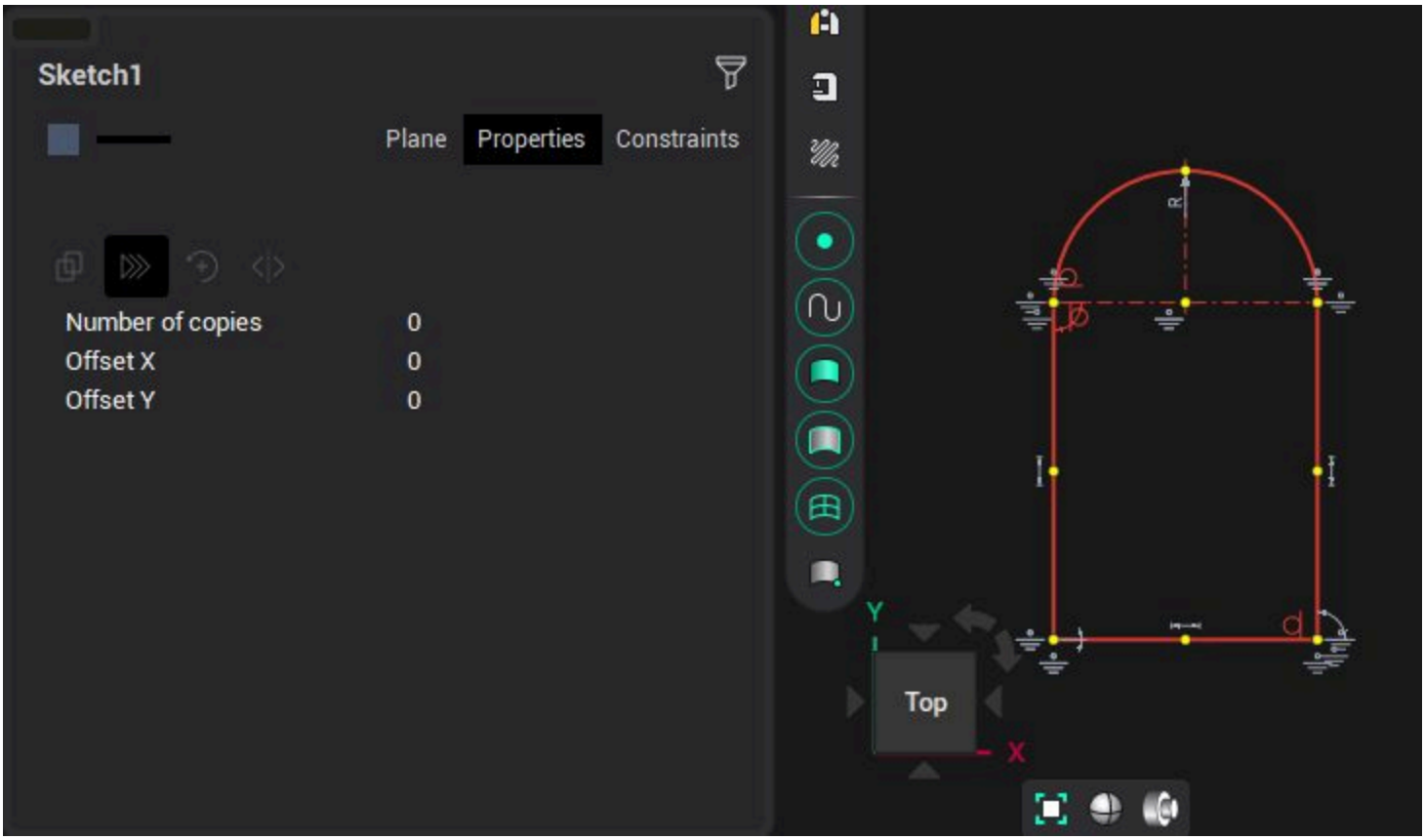
Copy



A copy of the block or single object is created in the same place as the original.

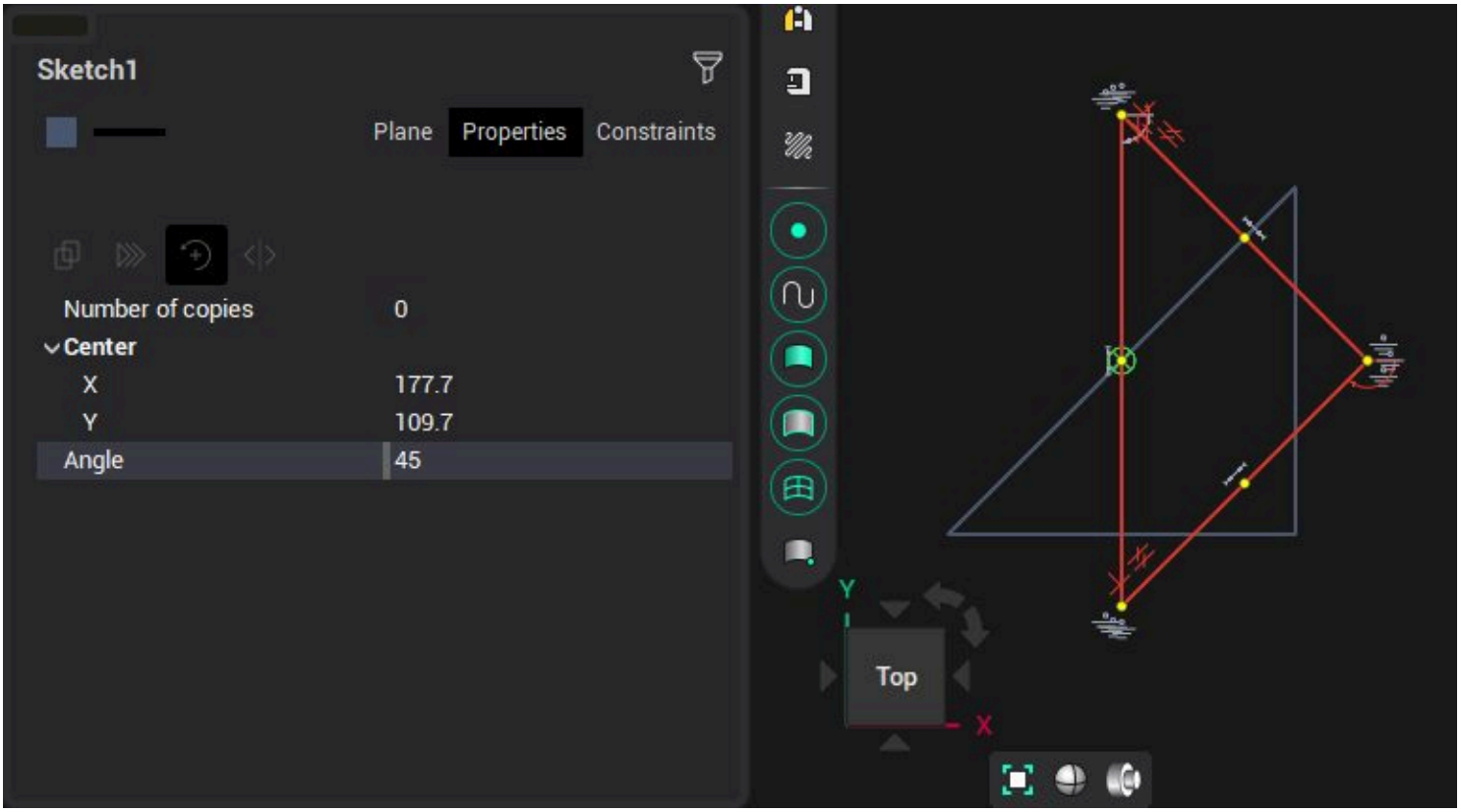
Shift

Shift a group of objects by a specified distance. In addition, you can specify the number of copies and each next copy will be shifted by the specified value from the previous one.



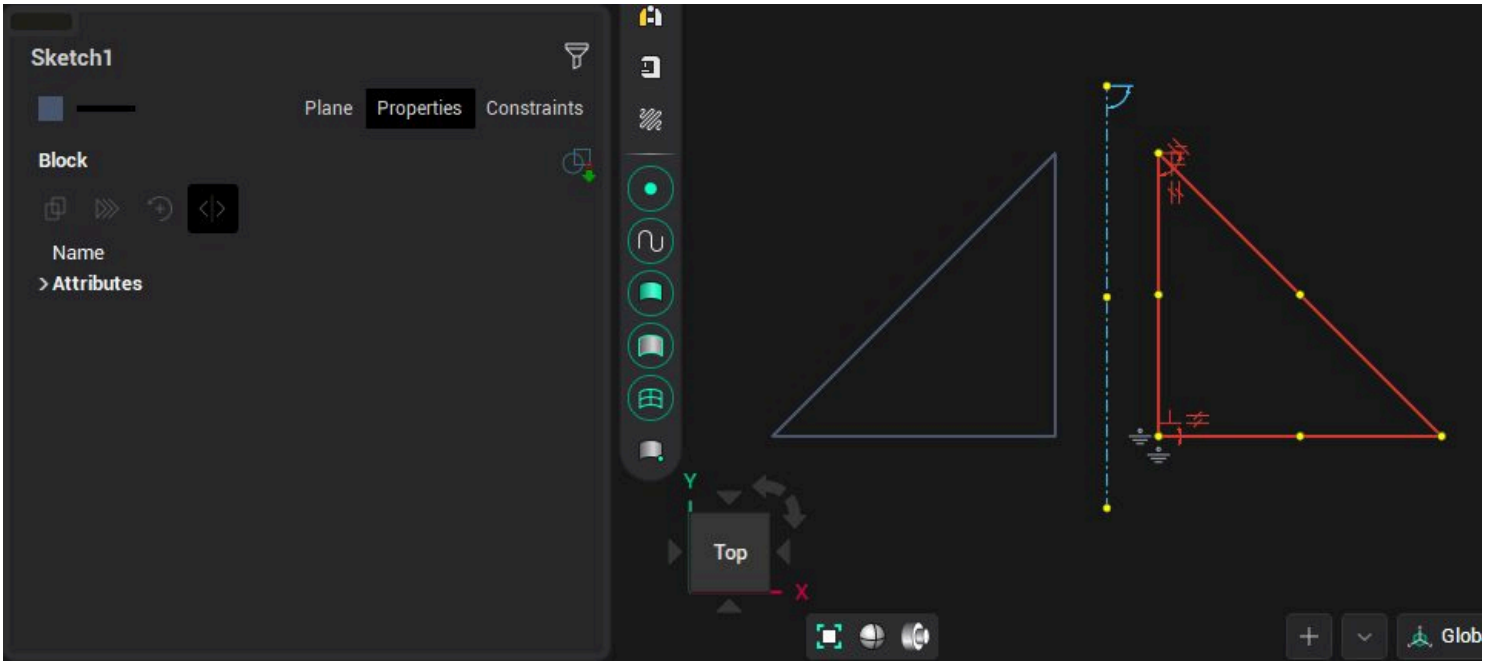
Rotate

The object is rotated around the specified point by the specified angle. Rotate can be done interactively or by specifying the coordinates and angle in the inspector. In addition, you can specify the number of copies, and each next copy will be rotated by the specified value from the previous one.



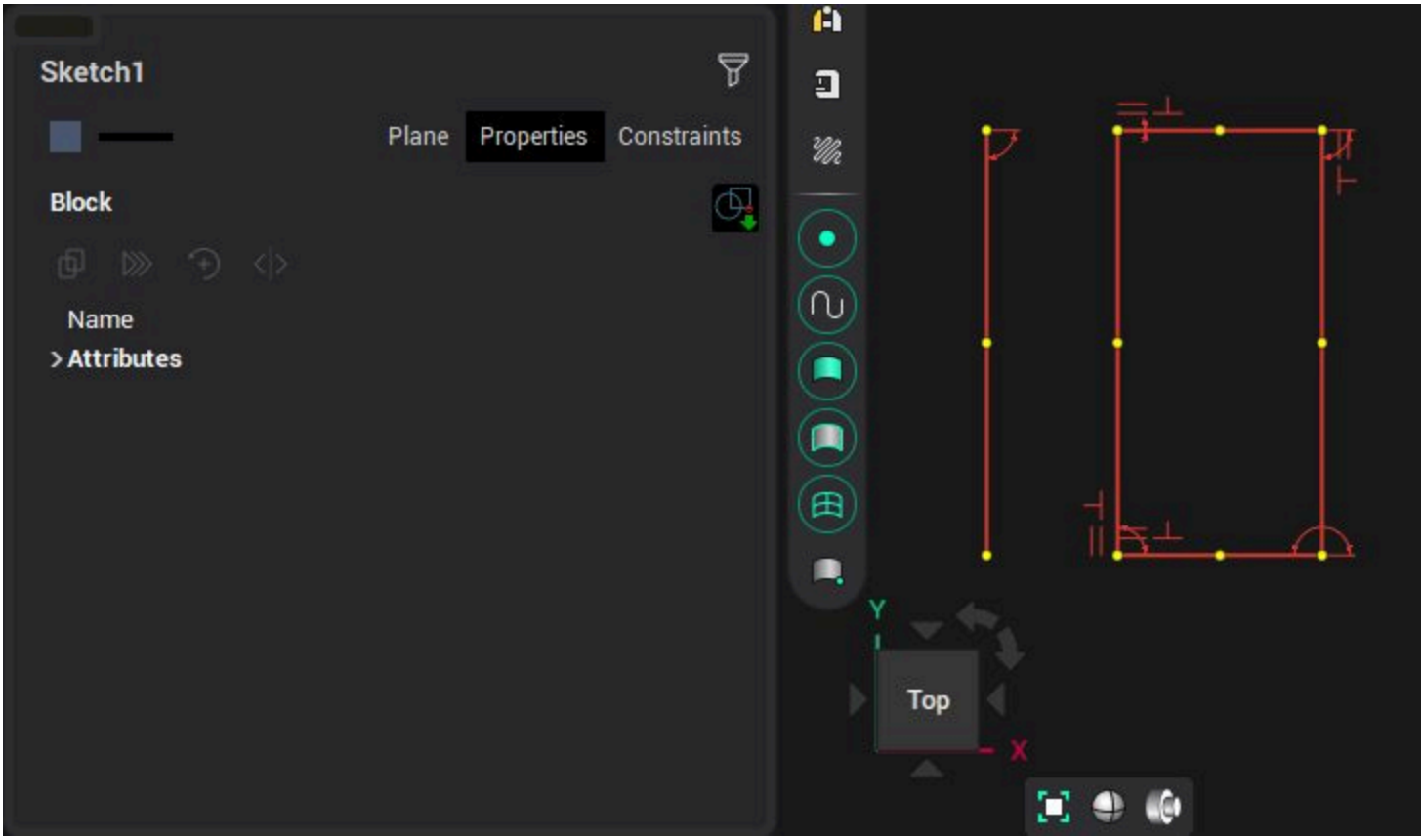
Mirror

For mirror, you must specify the axis relative to which the mirror will be performed.

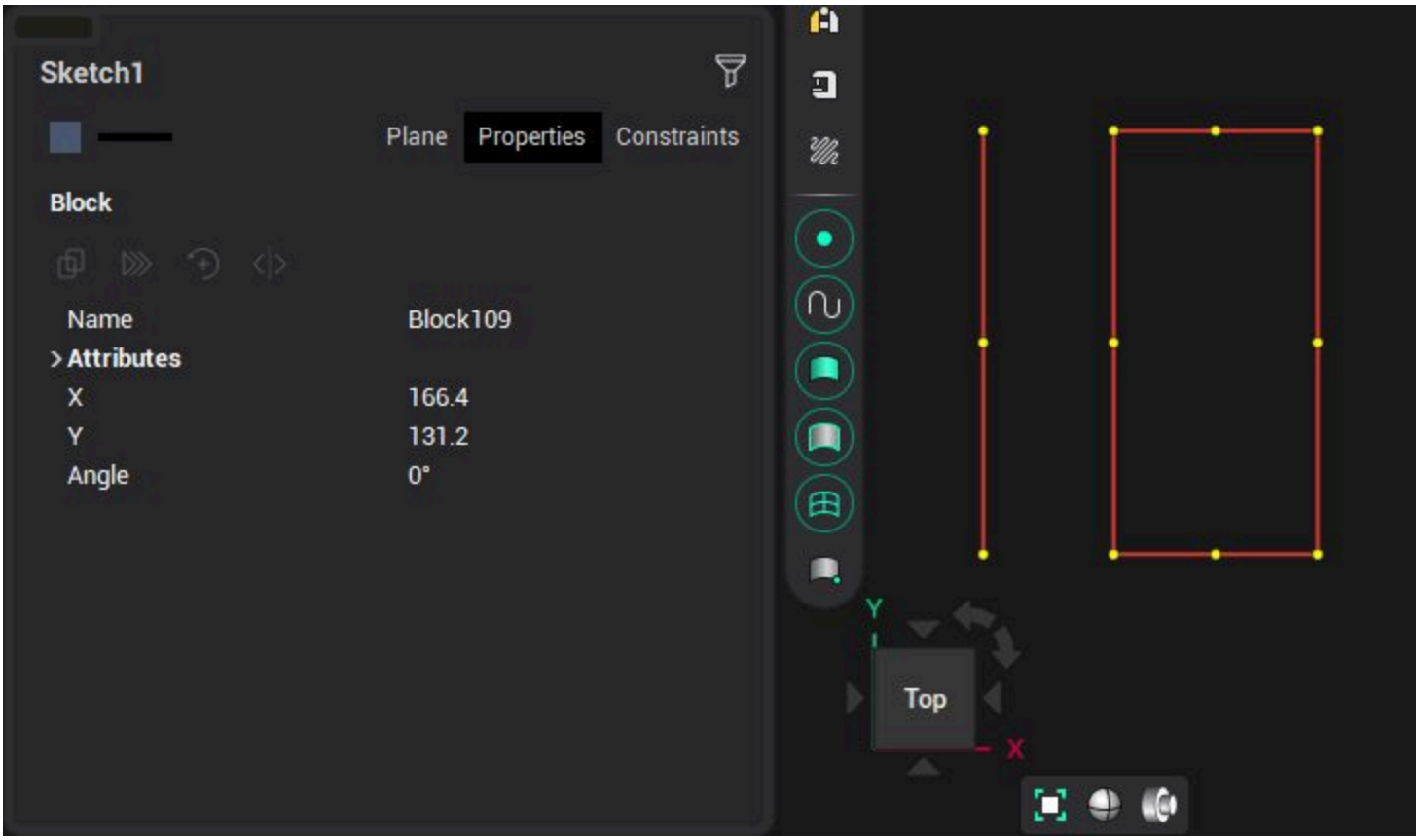


Named block

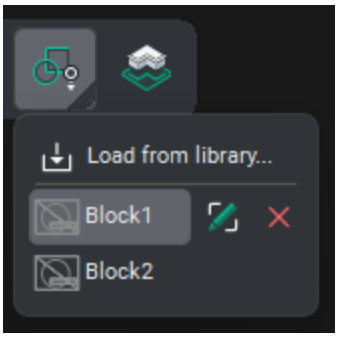
To create a named block, you need to select the necessary elements in the block. To do this, by pressing the mouse button and holding it down, select the elements with a rectangular frame, or sequentially select the elements with the [Shift] key pressed. The exclusion of elements from the block is also performed by specifying these elements with the [Shift] key pressed.



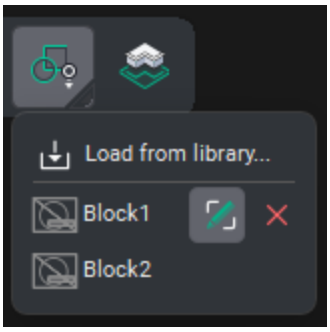
Click the button in inspector. If necessary, you can enter its name or set the rotation angle of the block.



In the future, you can insert this block into the drawing by hovering over the corresponding button and selecting the desired block from the drop-down menu. It is also possible to load a block from the library.

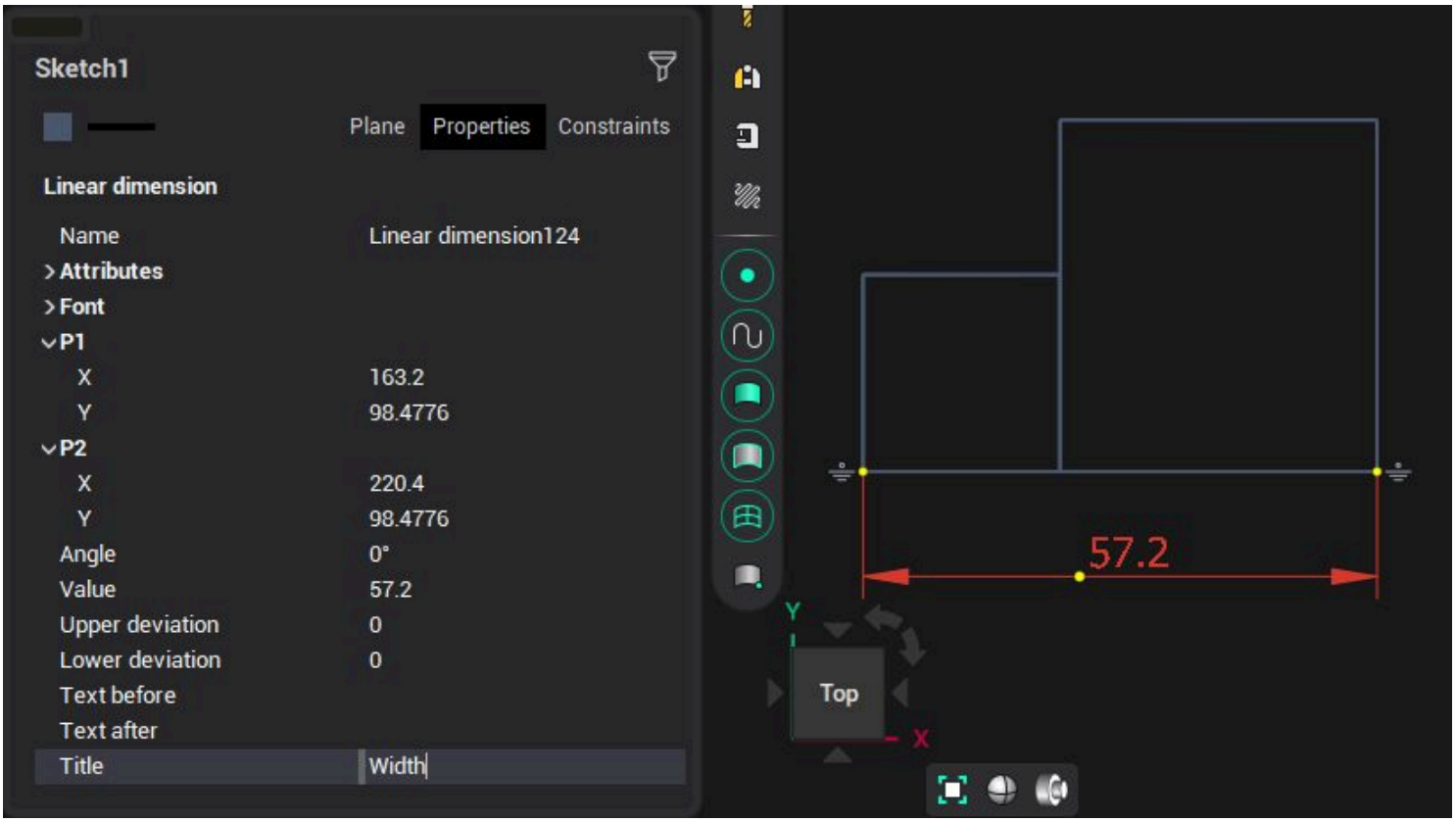


Subsequently, you can edit the original block by clicking the "scissors" in the menu, or delete it. All copies of the edited block will also change.

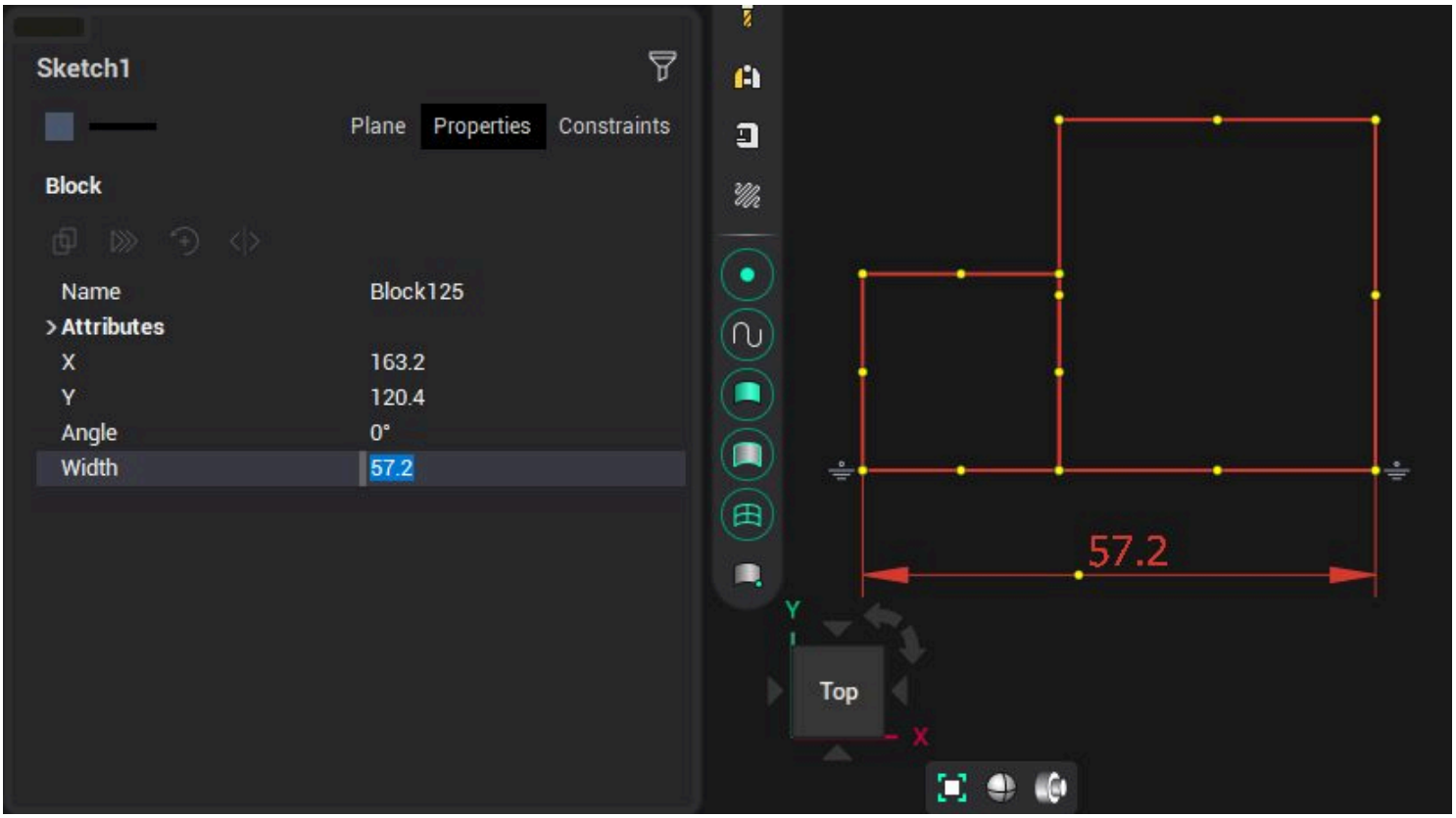


Block parameterization

Before combining the elements into a block, enter the values you need in the [Title] fields of the sizes.



After merging the elements, these parameters will appear in the inspector and will be available for editing.


Sketch1
Plane
Properties
Constraints
Block
Name
Block125
Attributes
X
163.2
Y
120.4
Angle
0°
Width
57.2
57.2
Y
Top
X

Constraints

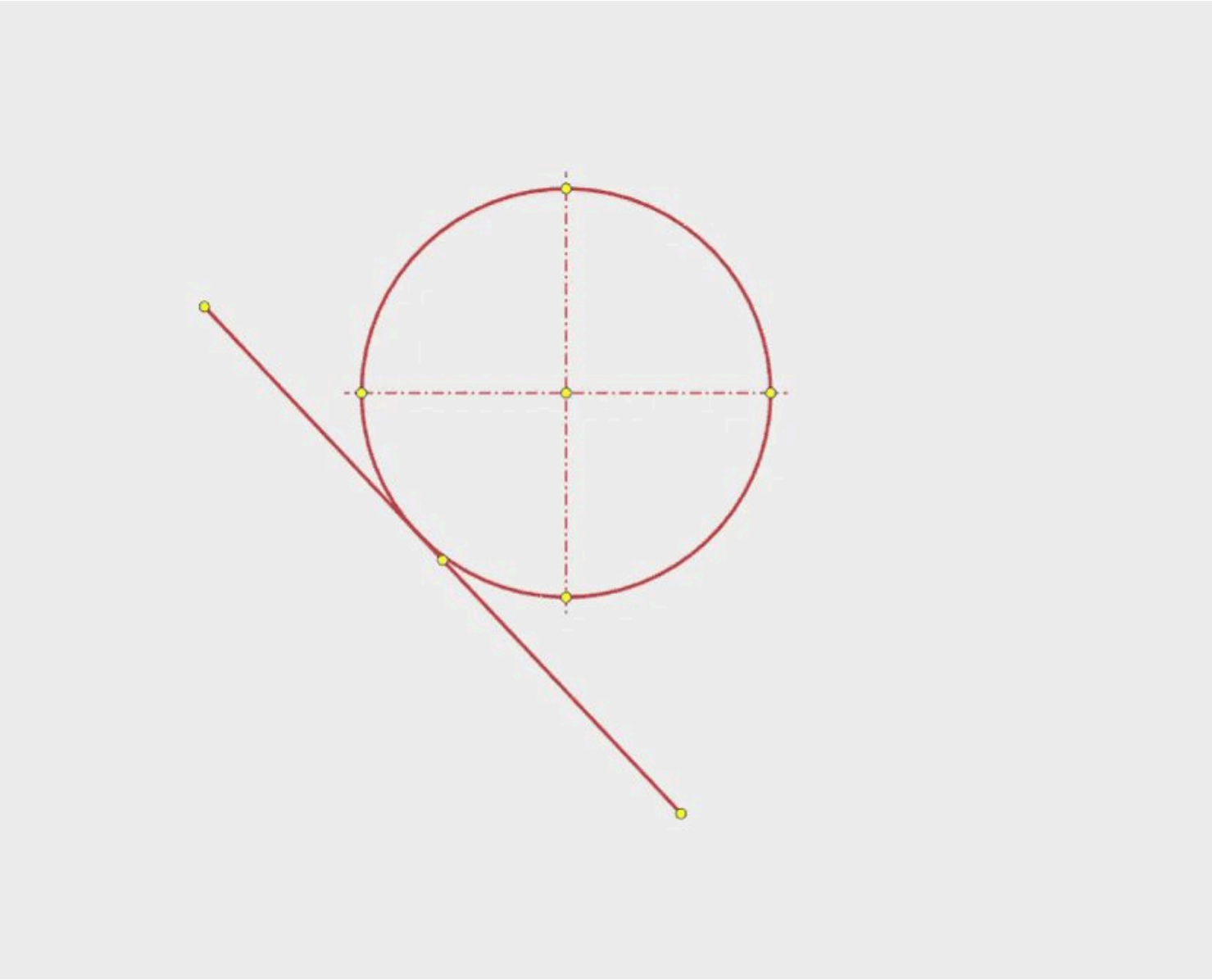
Constrains are created during the build process and are removed during editing when the link conditions are lost. For example, the line has ceased to be parallel or the touch has disappeared.

In the process of editing a block, the c onstrains are preserved and the objects change in accordance with them. That is, tangency, parallelism of lines, etc. is saved.

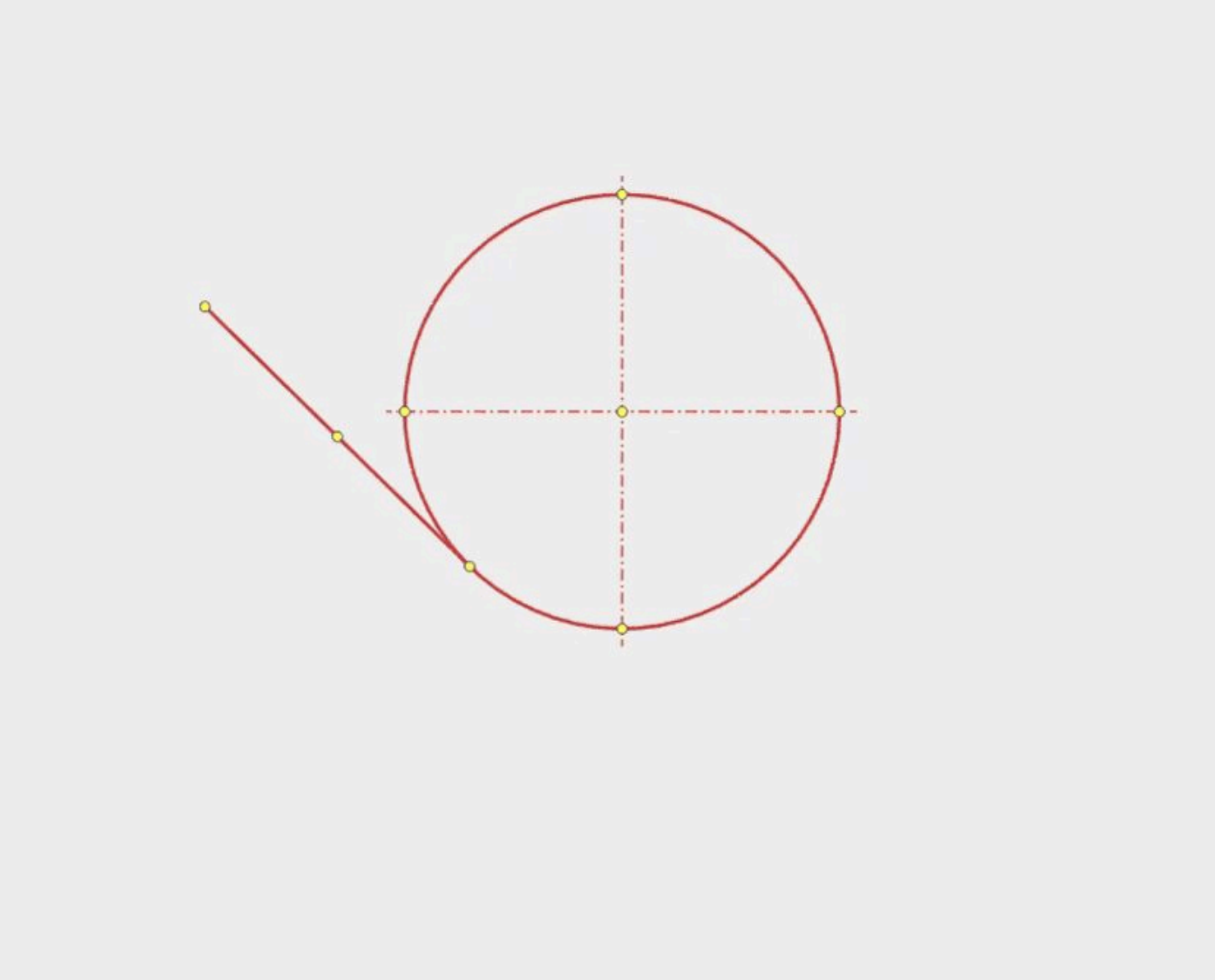
Constrain types:

- Tangency

During construction, while pressing [Ctrl], select the object to which this c onstrain will be applied. For example: a circle touches a line, or a line segment touches a circle or arc.

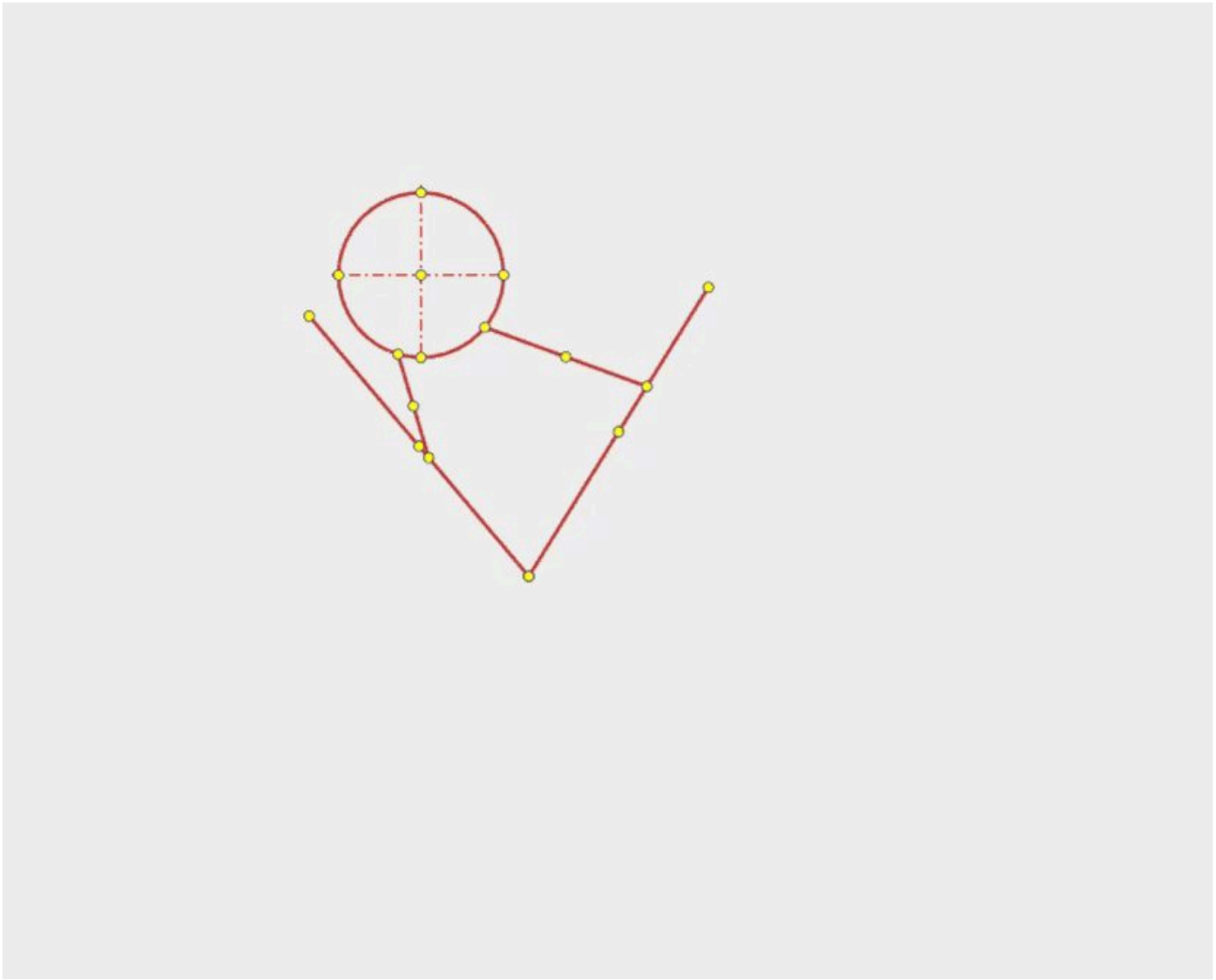


When constructing a line, in addition to tangency, it is set that the point lies on the circle.



- Point on a circle, line, arc.

When constructing, if the point is on the object, then the corresponding connection is created. If the point during construction coincides with the point of another object, then 2 joint connections are created and the point becomes common when editing the block.



- Parallel, perpendicular

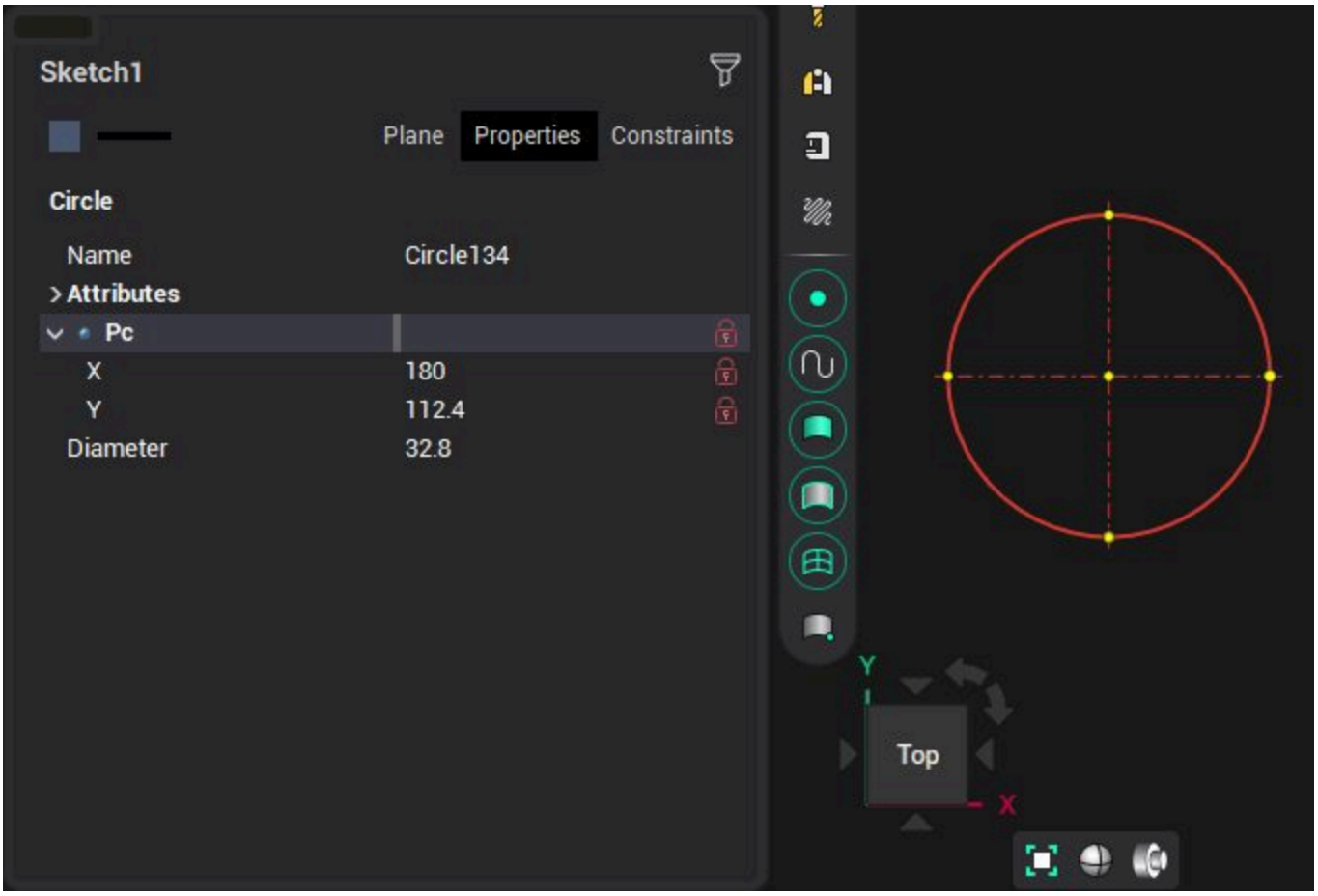
If during the construction process with pressed [Ctrl] move the cursor to the line, then auxiliary lines for construction will appear and a link will be set. You can also set the link in the inspector on the Links tab, and if the lines did not meet this criterion, they will be adjusted.

- Alignment

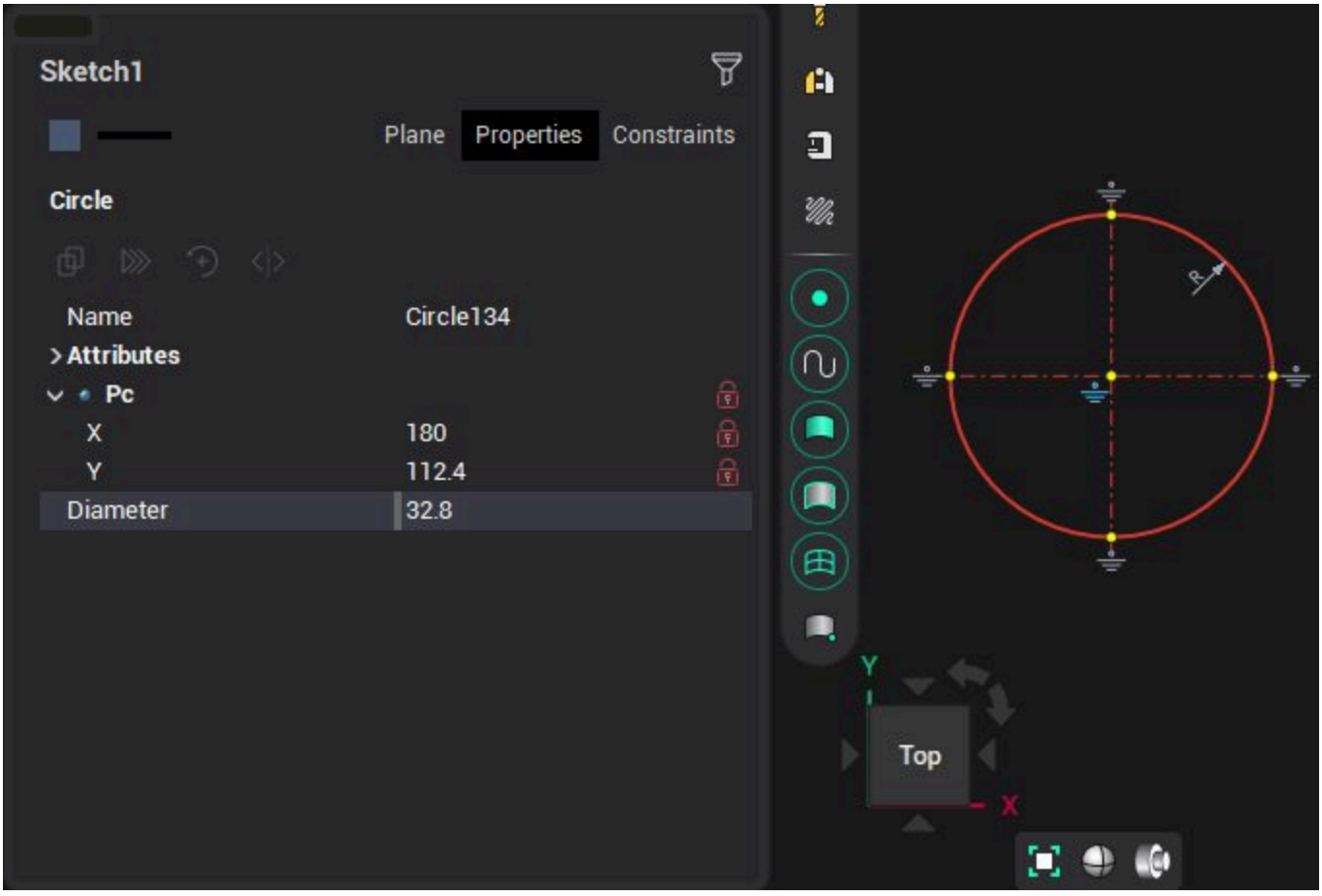
The lines lie on the same straight line. You can get this connection as a result of trimming. You can also set such a link in the inspector on the Links tab.

- Fixing

You can fix points by X, Y coordinates, separately or together. You can fix the length, radius, corners, height and width of the rectangle. You can fix it with the buttons in the inspector:



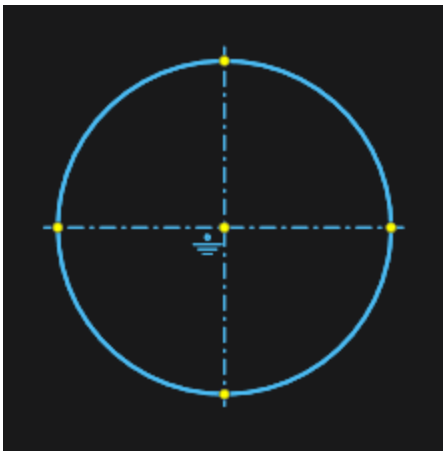
Or by anchors in the drawing:



For dimensions, you can fix the value Value, Radius, Angle in the inspector. This affects how elements behave when edited in block mode - this size does not change.

- Axes

If, when constructing a line, by double-clicking on an arc or a circle, axes will be drawn that will follow the change in this element.



- Equality

The equality of segments, radii of arcs, circles is set when copying with replication. You can also set such a link in the inspector on the Links tab.

- Linear size

A binding of a linear dimension is created when it is assigned to a line. Subsequent editing of the segment will also adjust the size.

- Point coincide

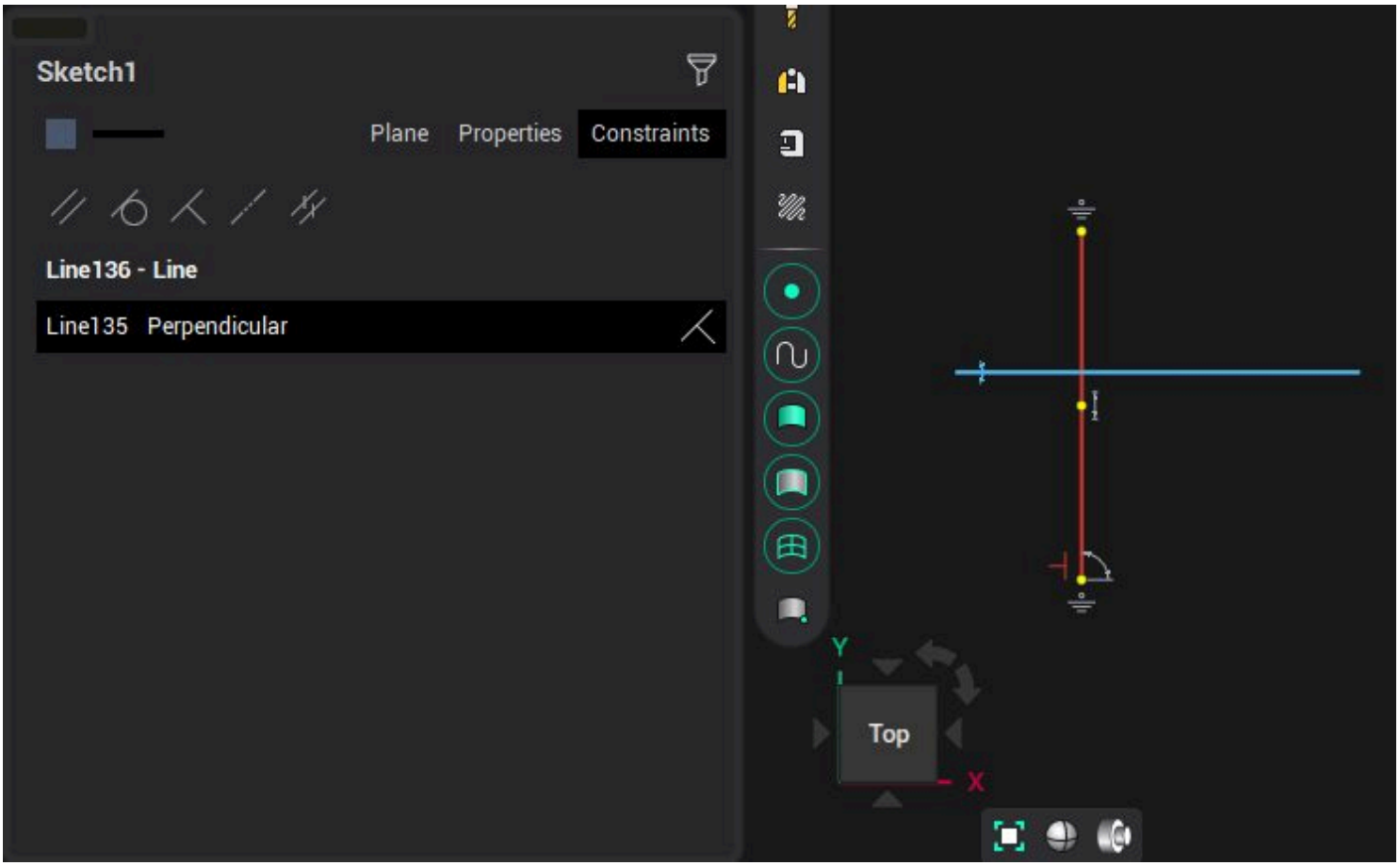
Such a connection is created during construction, when the points of the objects coincide. You can also set such a link in the inspector on the Links tab.

- Object-forming links

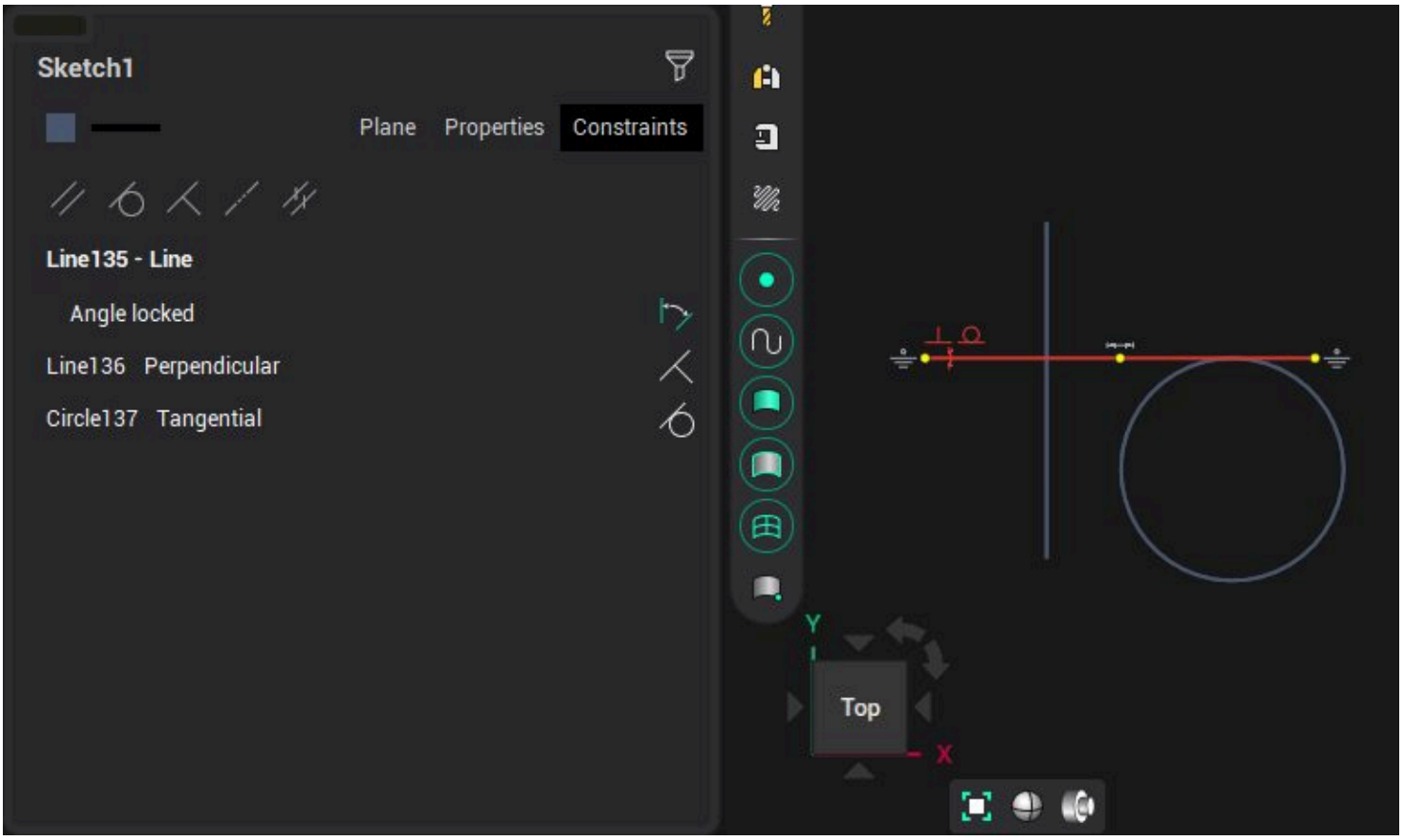
They are used to connect created complex objects, such as a contour, a radial dimension, an angular dimension, a rectangle, a fillet, a chamfer, a script. The link persists as long as the original object exists.

Editing links in a block

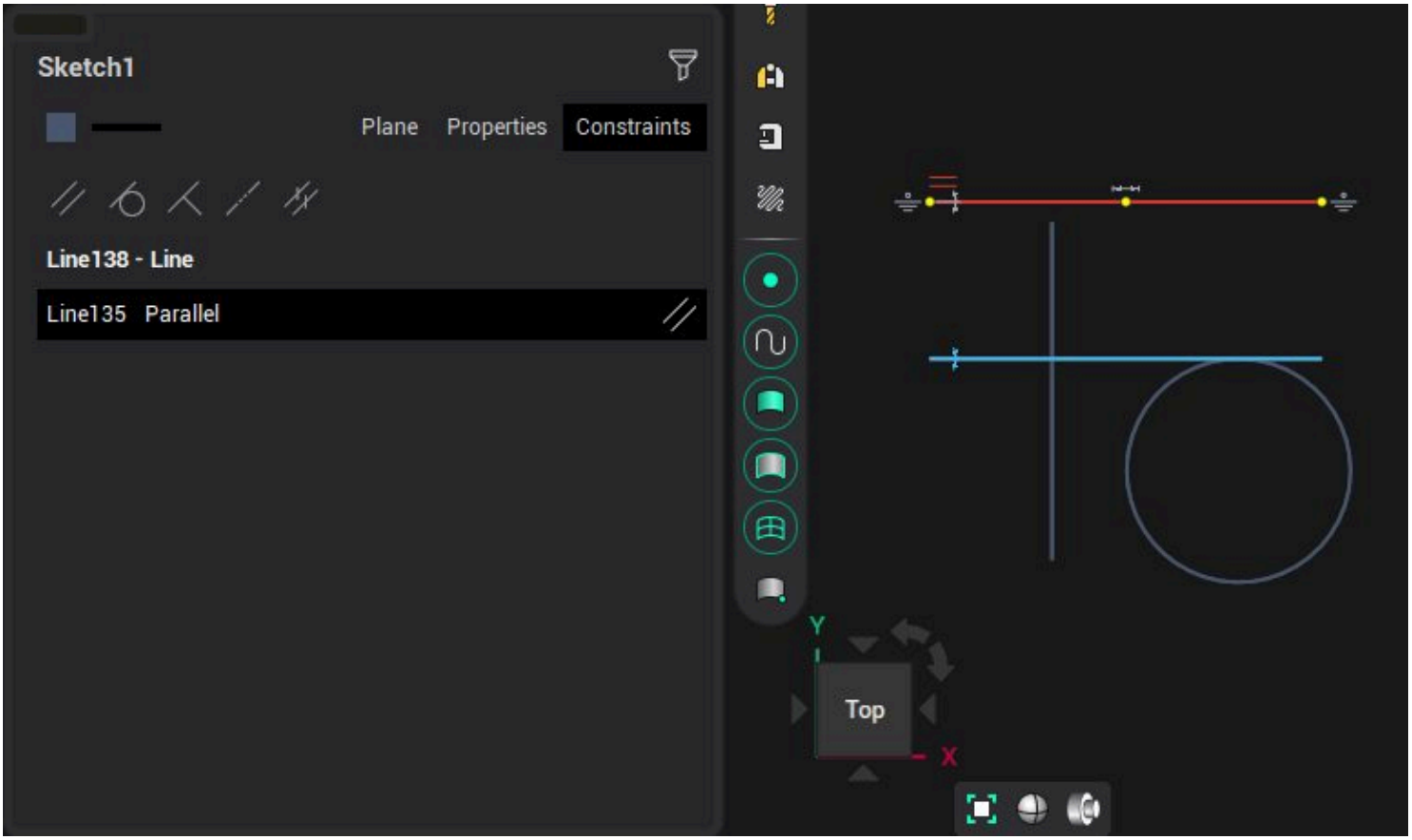
In the inspector, in the Constraints tab, a list of the active element's links is shown.



In block mode, you can disable / enable the restriction by clicking on the link image.



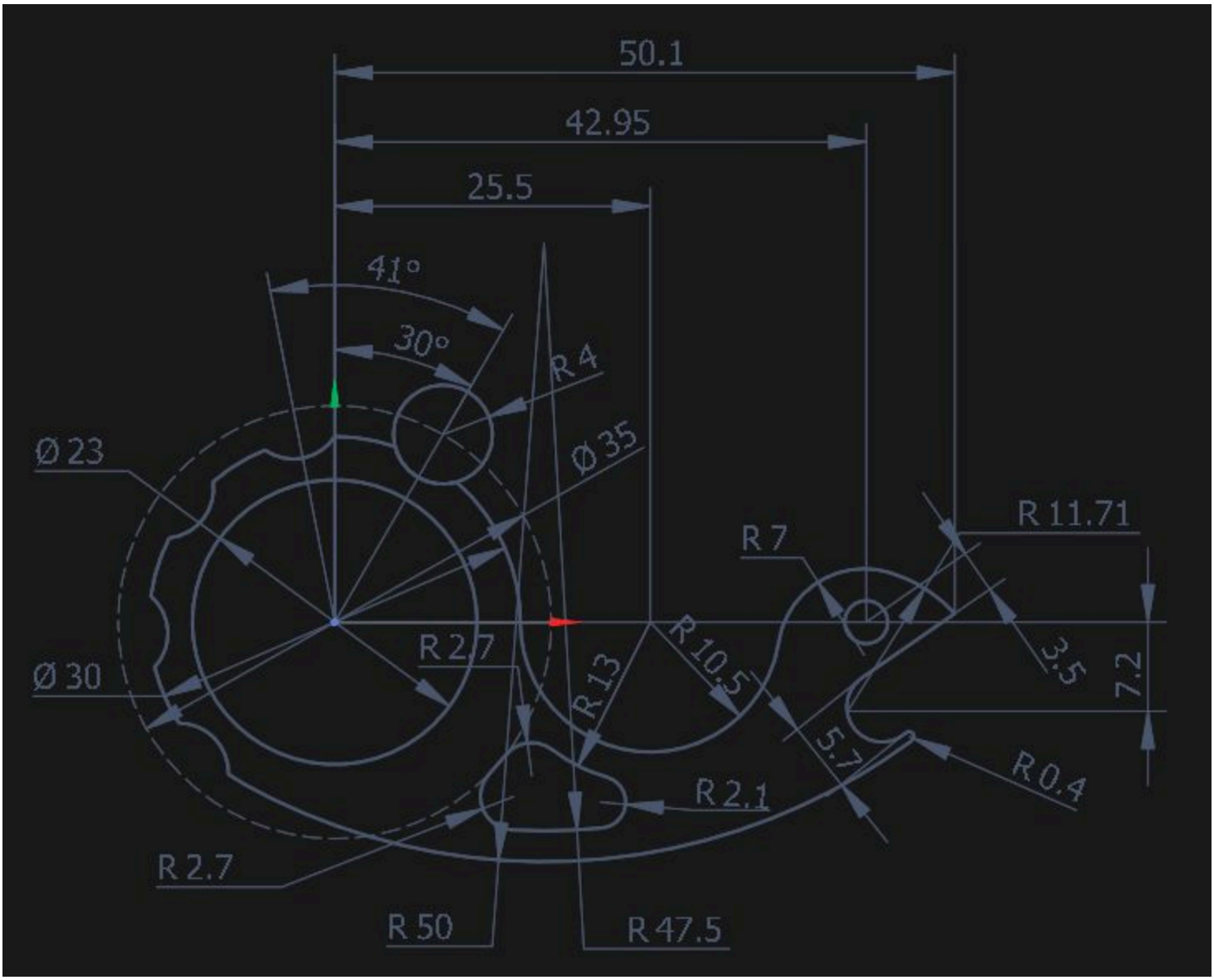
To create links Parallelism, Tangency, Perpendicularity, Coaxiality, Equality in block mode, press the corresponding button and sequentially select a pair of objects.



Releasing the button will return the link enable/disable mode. After exiting the block mode, disconnected links are permanently disabled.

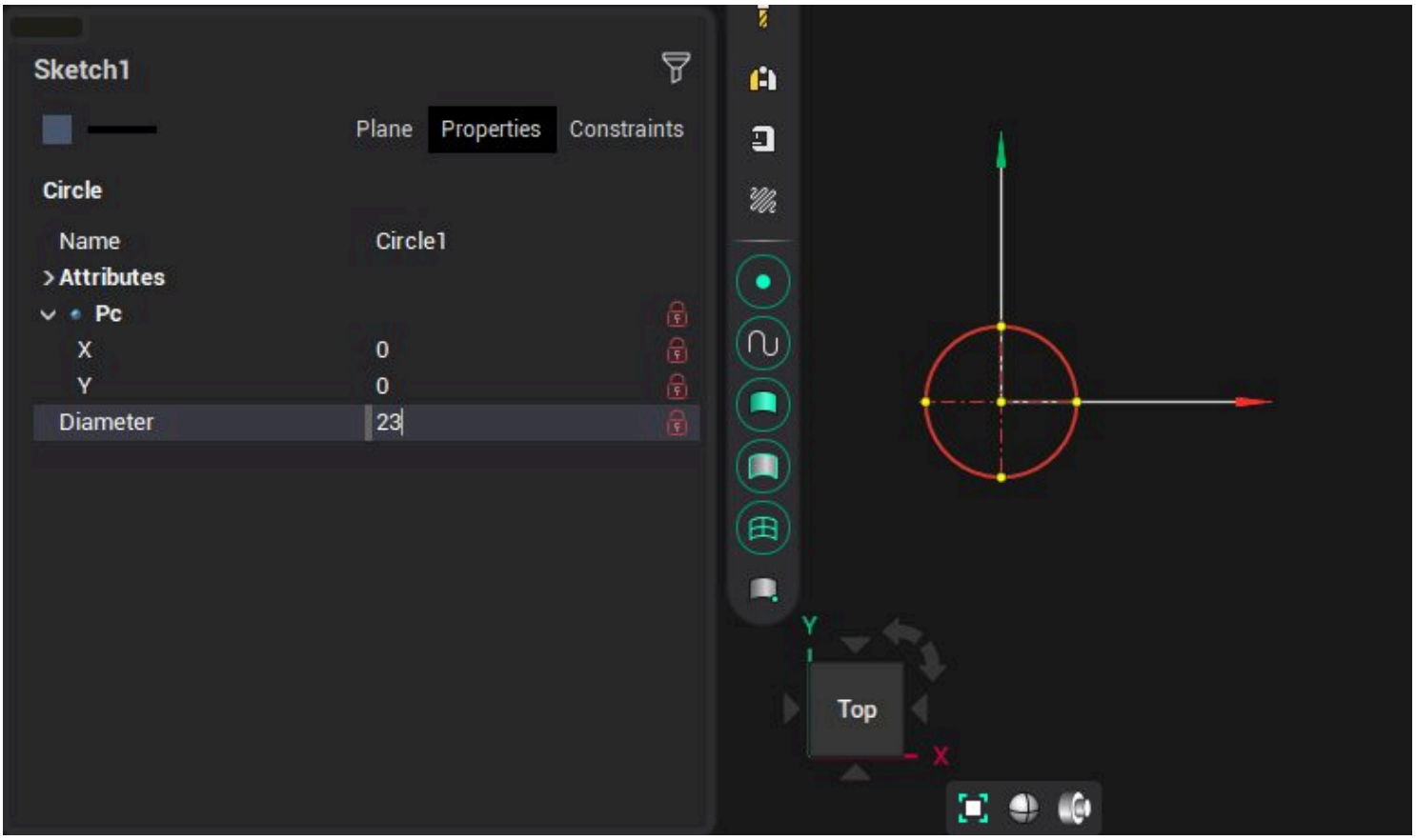
Drawing example

Let's look at the construction using this drawing as an example:

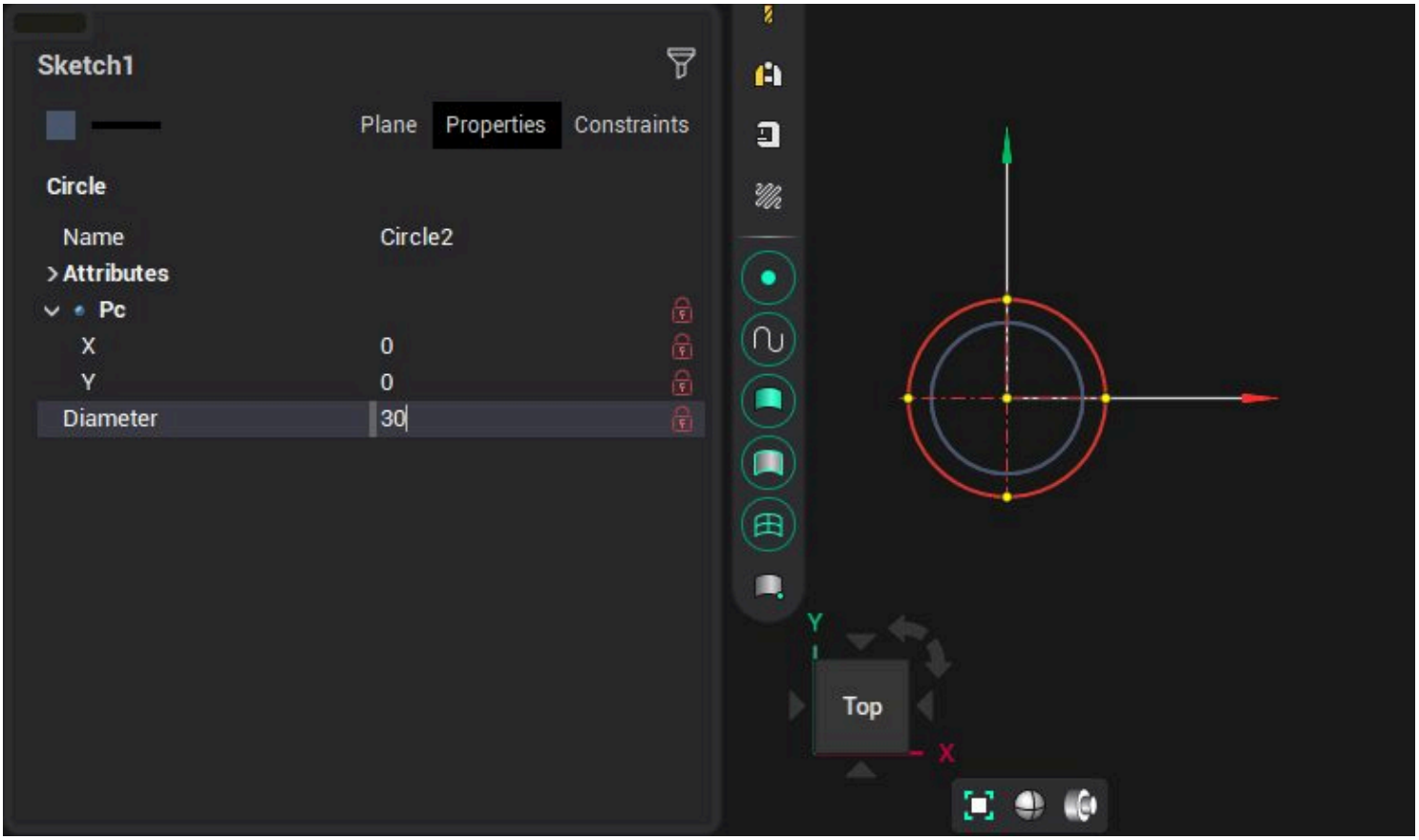


Press the button for drawing a circle 

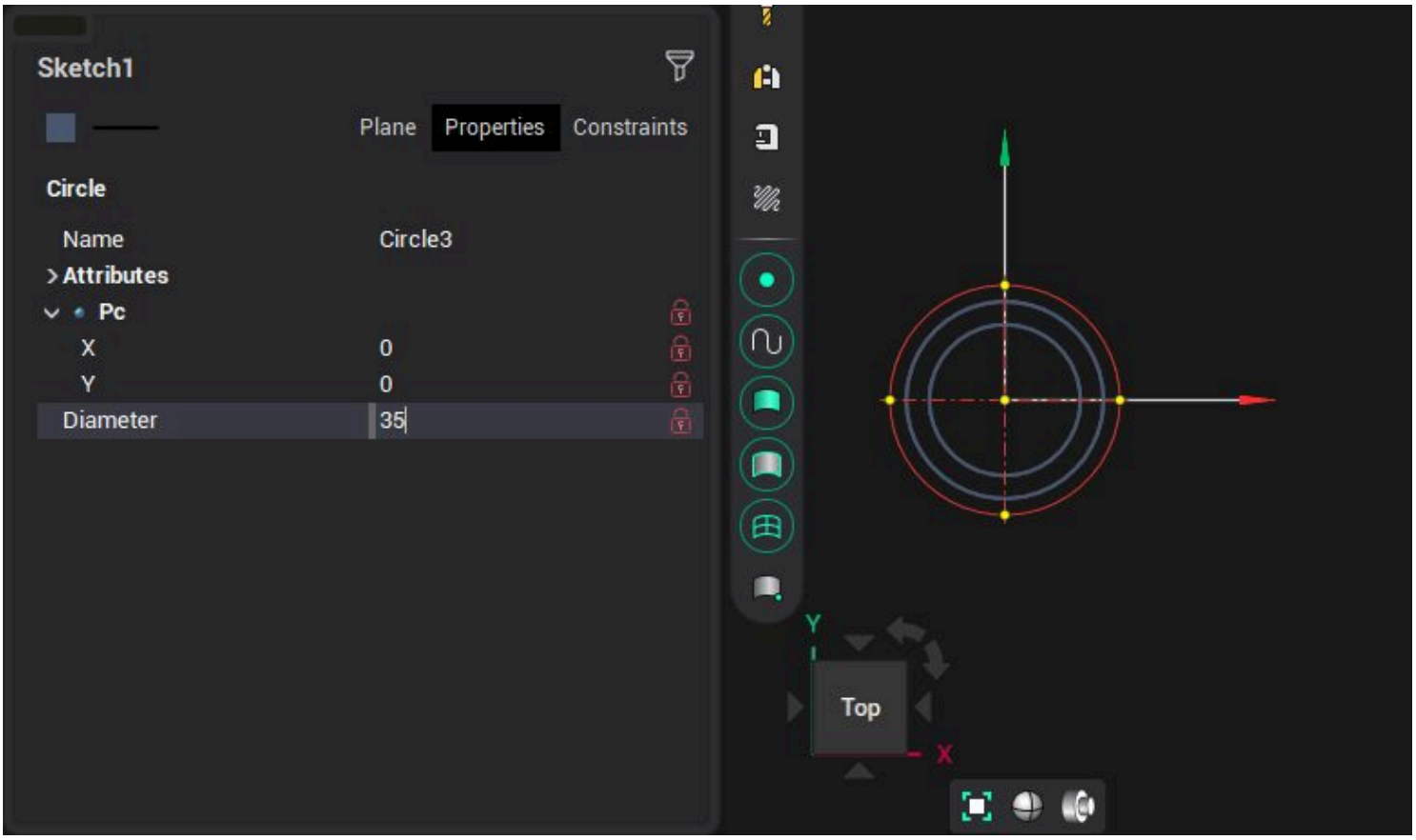
In the inspector, enter coordinates 0, 0 and radius 23.



In the same way, we build the following circle:

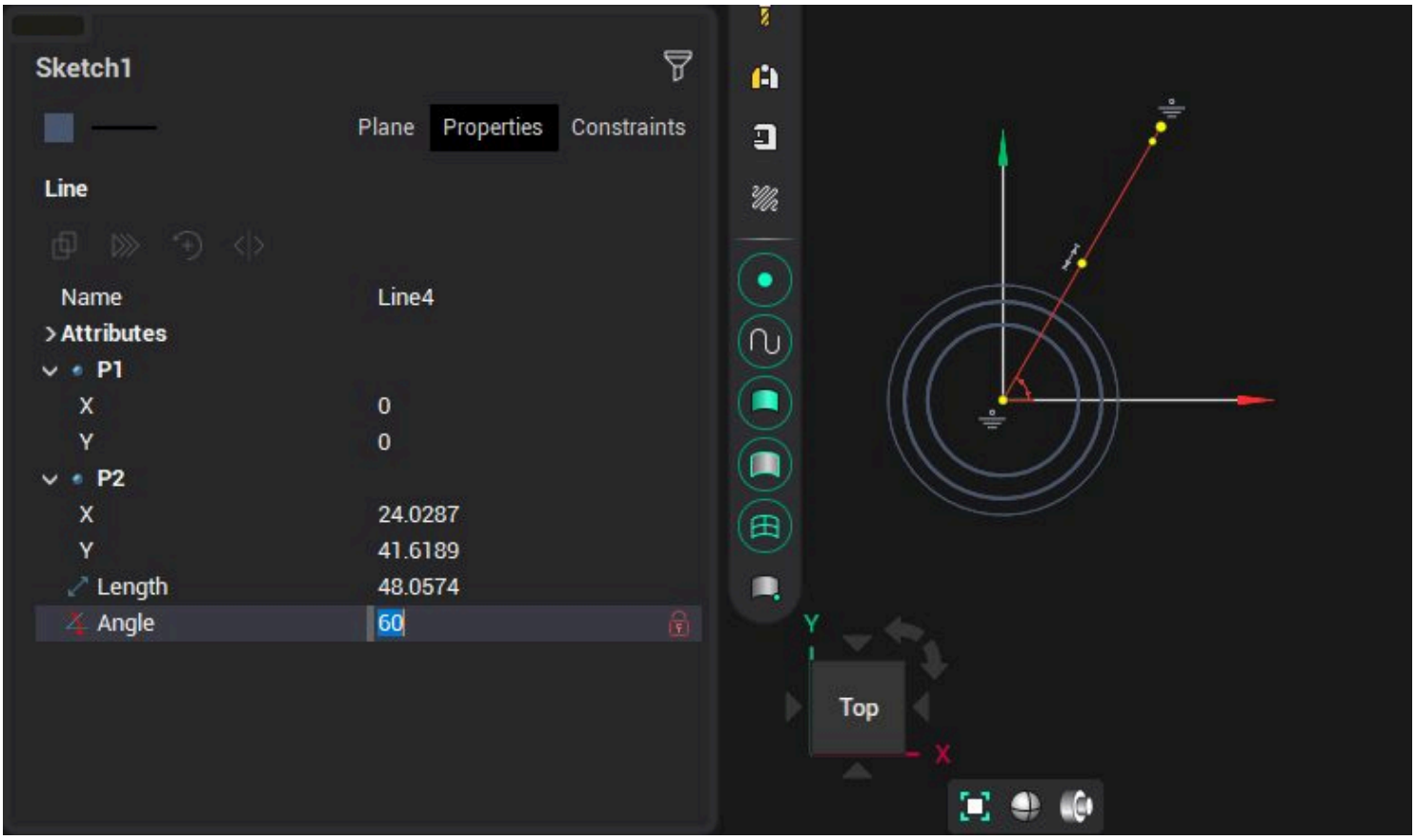


And the third, after pressing [Alt + 2] to switch the line type:

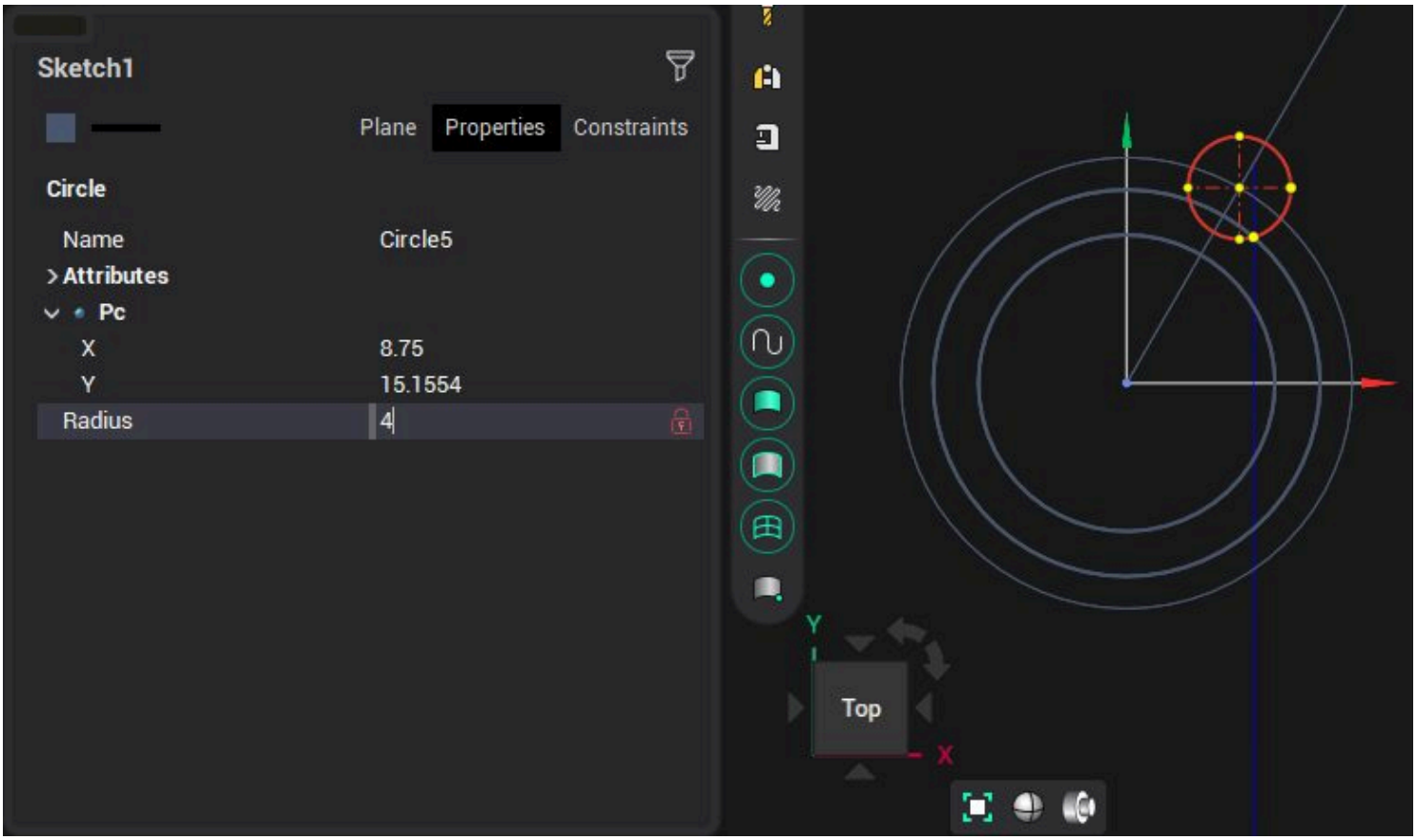


Next, switch to the line construction mode. 

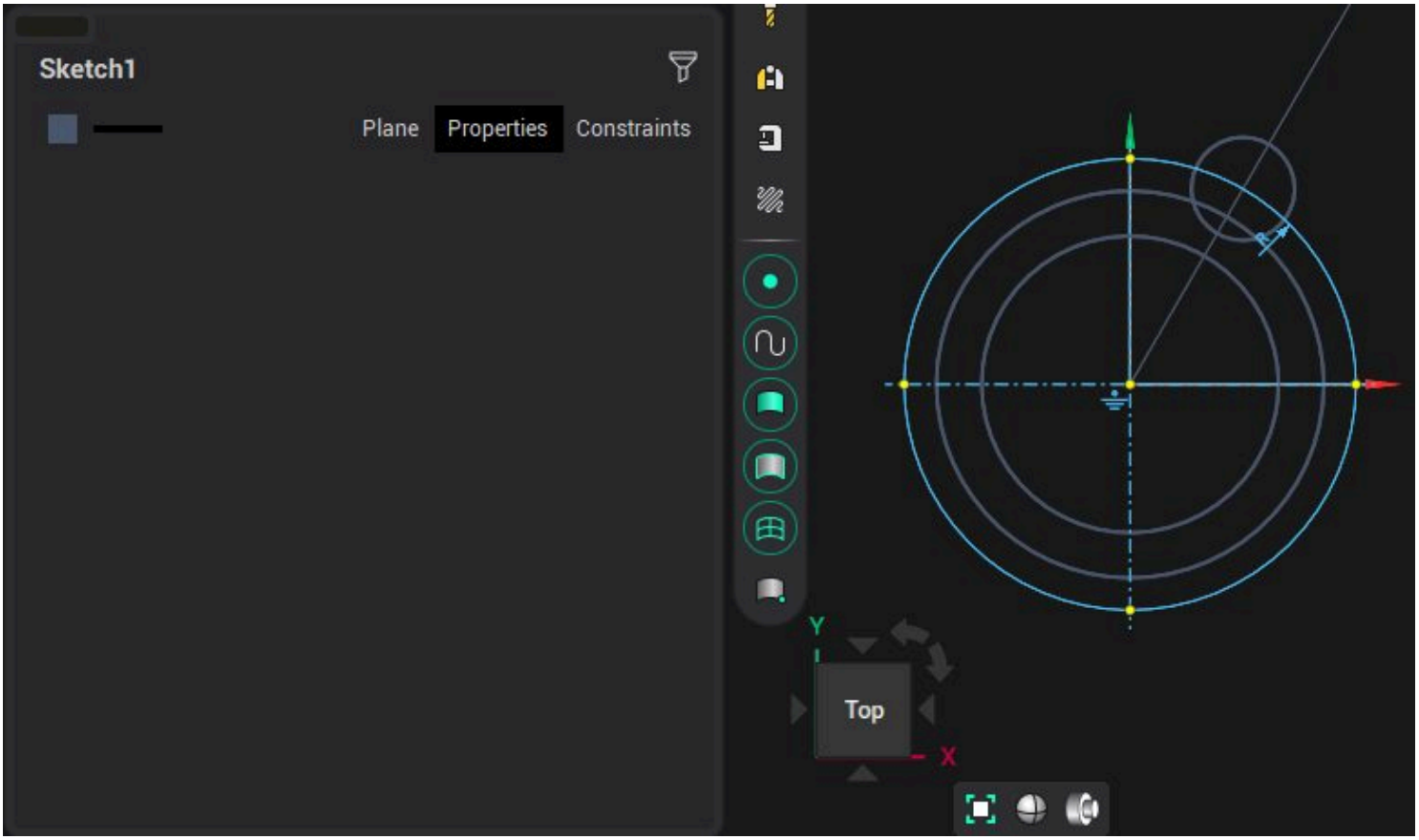
Specify the first point in the center of the circles, then drag it outside the third circle, and in the inspector enter an angle of 60°:



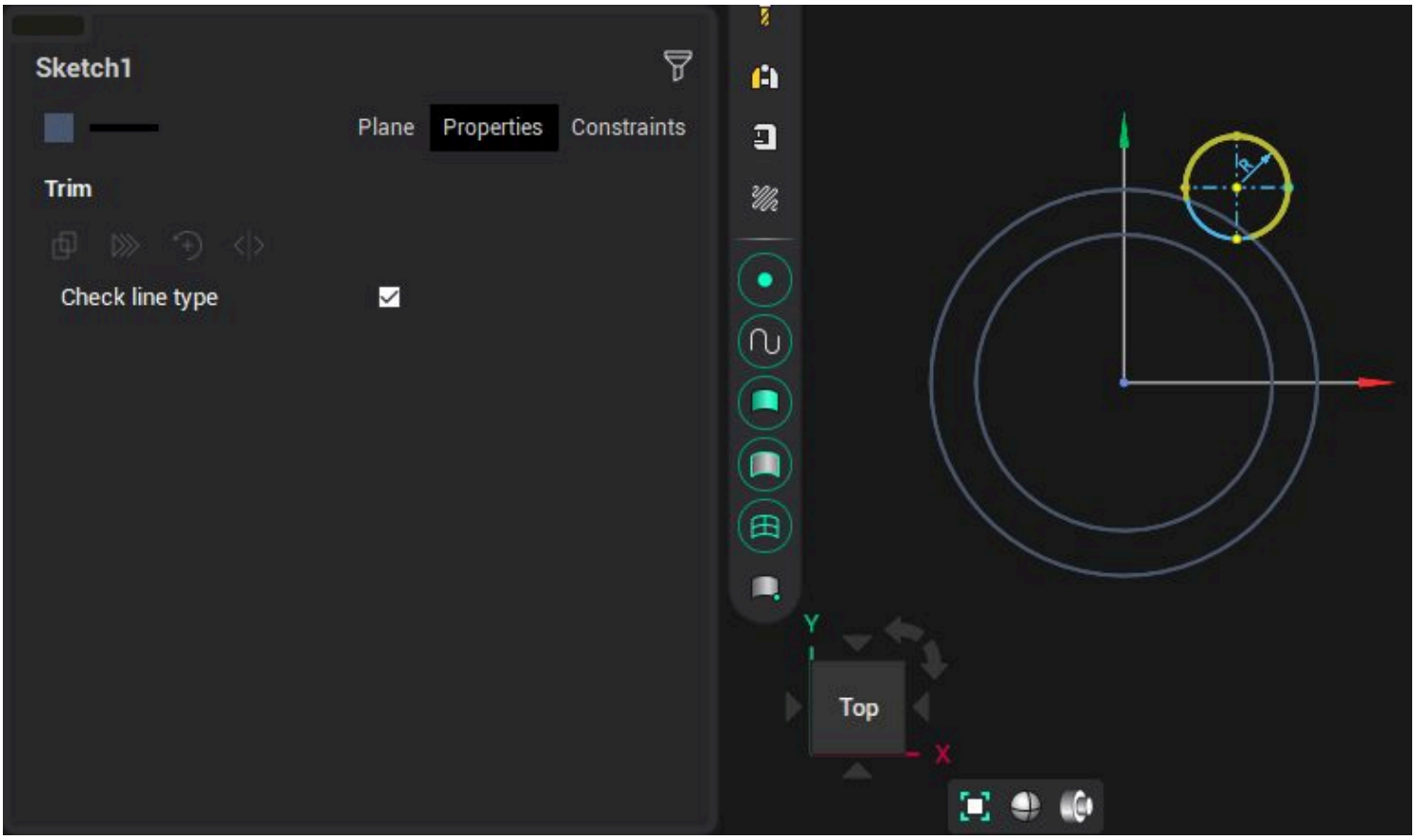
Switch back to the circle construction mode, press [Alt+1] to return to the main line construction mode, specify the intersection point of the segment and the third circle as the center, press the [R] key to switch to the radius input mode and enter 4:



Press the [Esc] key or release the circle creation button to exit this mode. We highlight the segment we no longer need and press the [Del] key to delete it, then delete the outer circle:

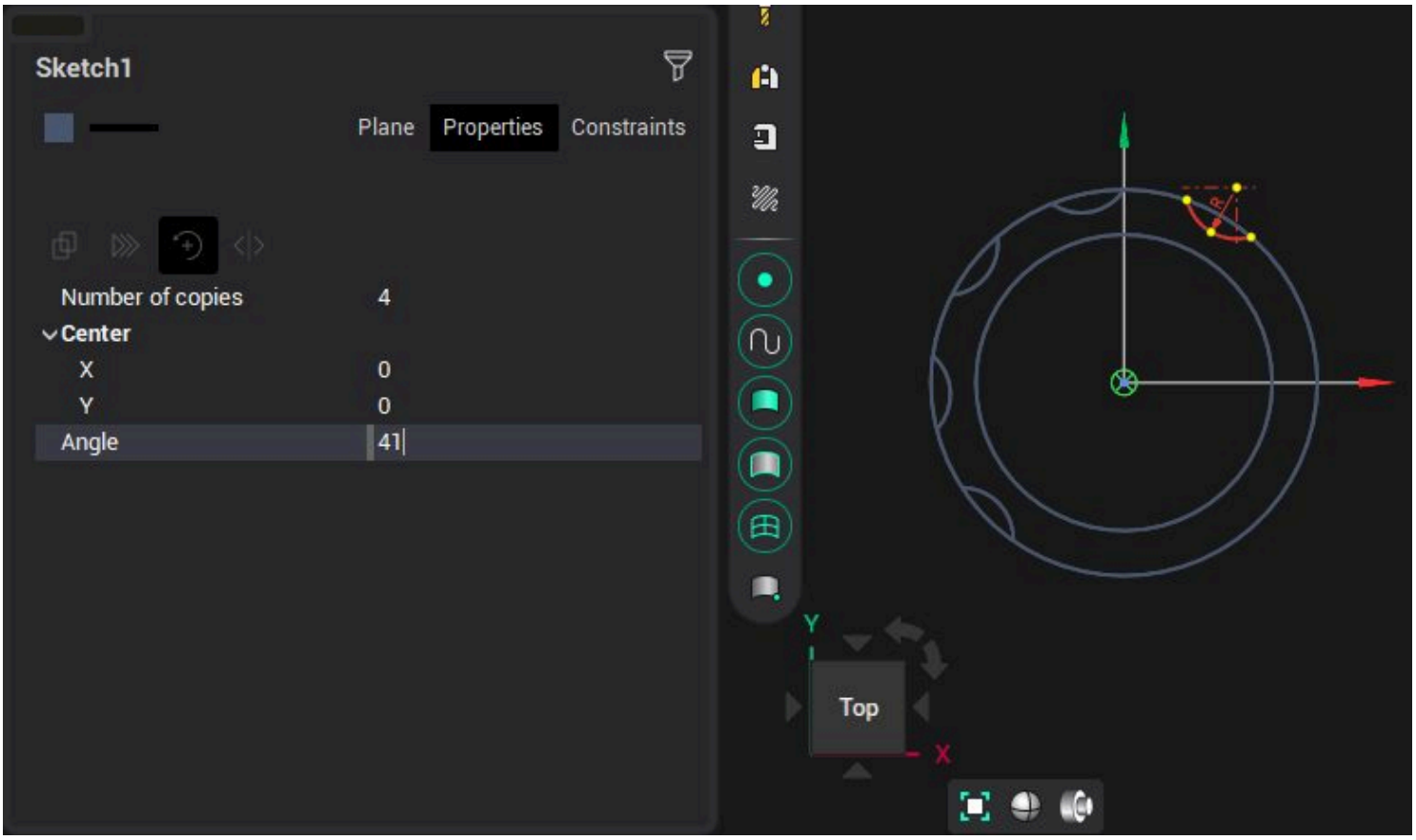


Switching to trimming mode  and delete the unnecessary part of the circle:

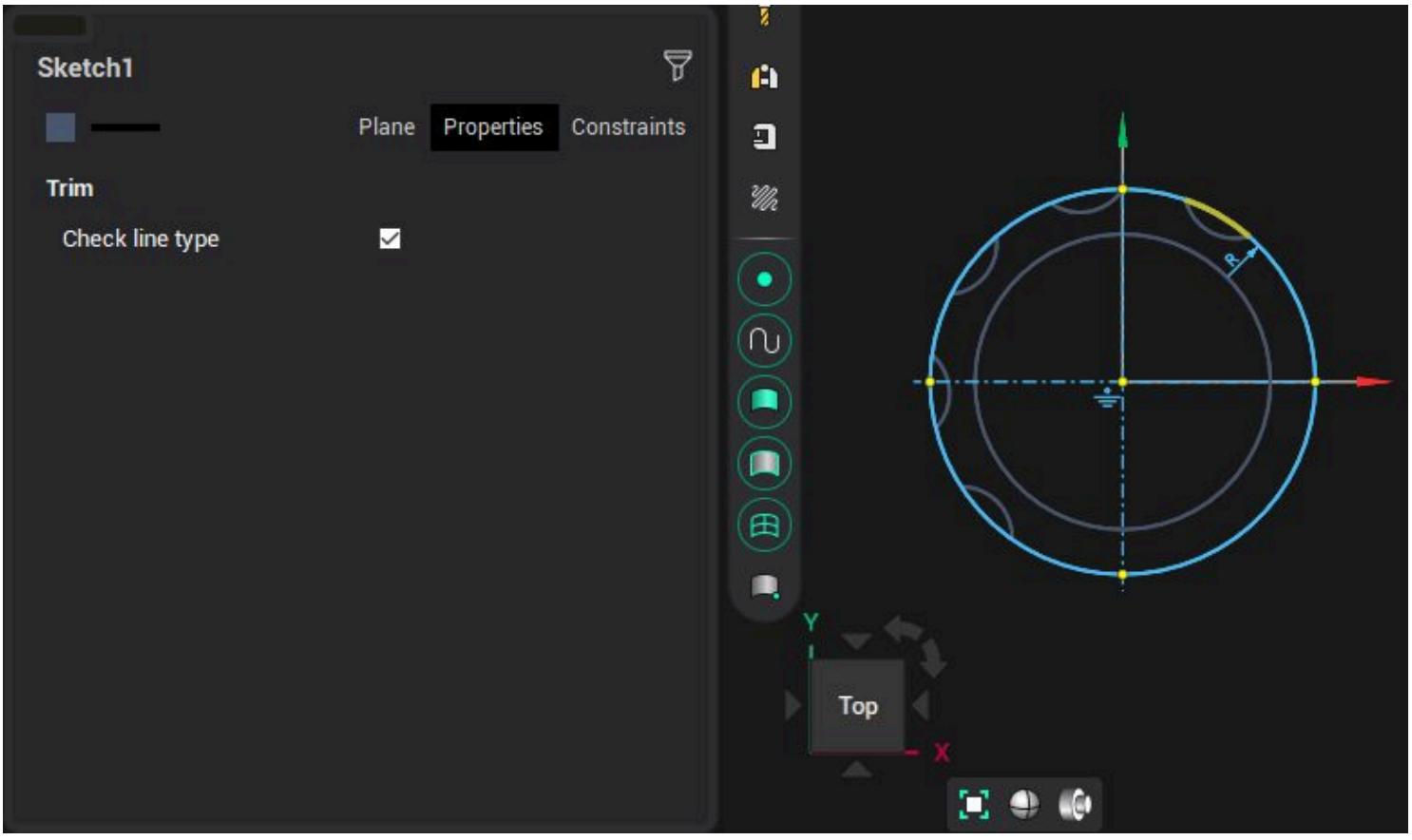


Select the resulting arc, switch to the object rotation mode 

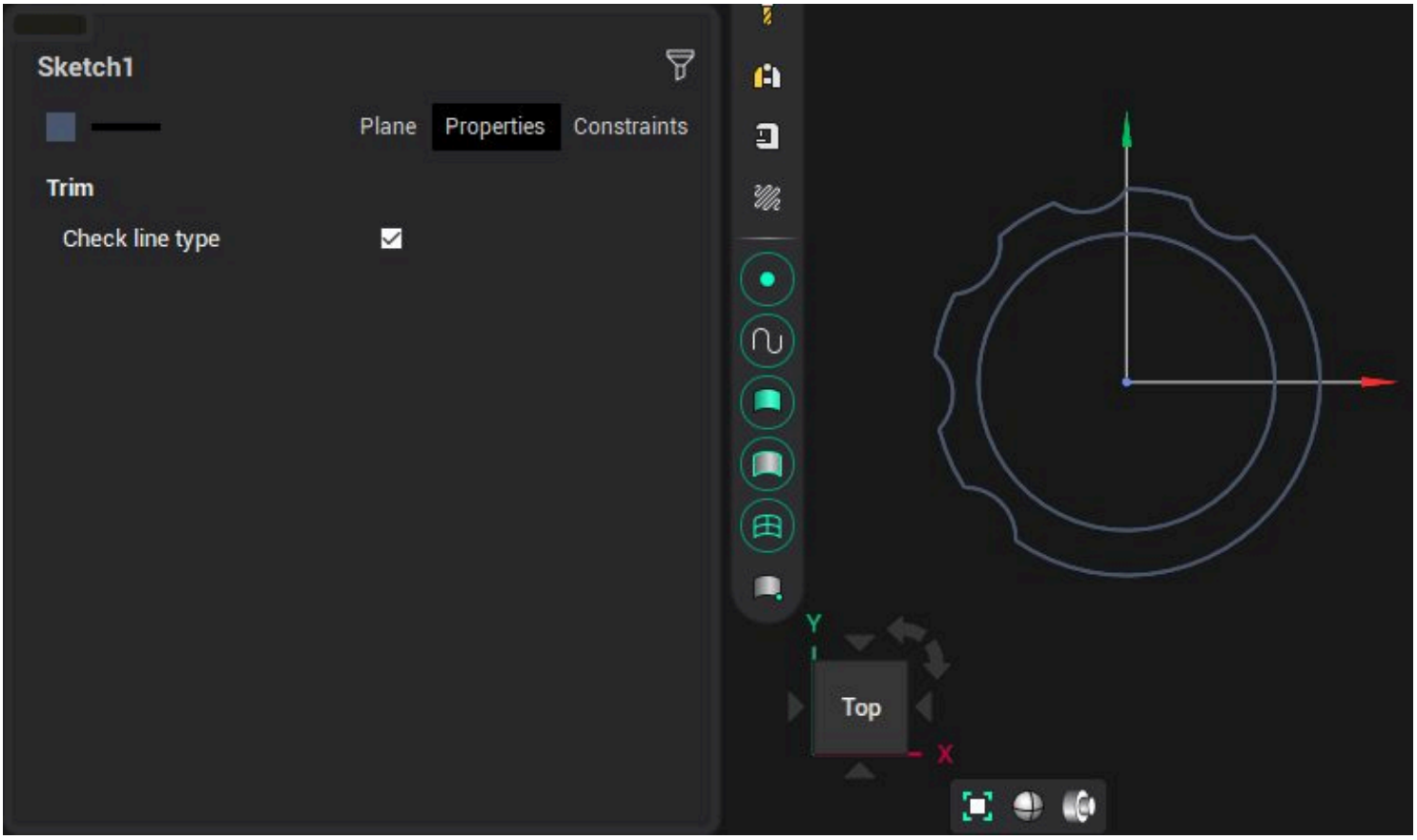
Move the rotation center to the center of the circles, enter 4 in the [Number of copies] field, and 41° in the [Angle] field:



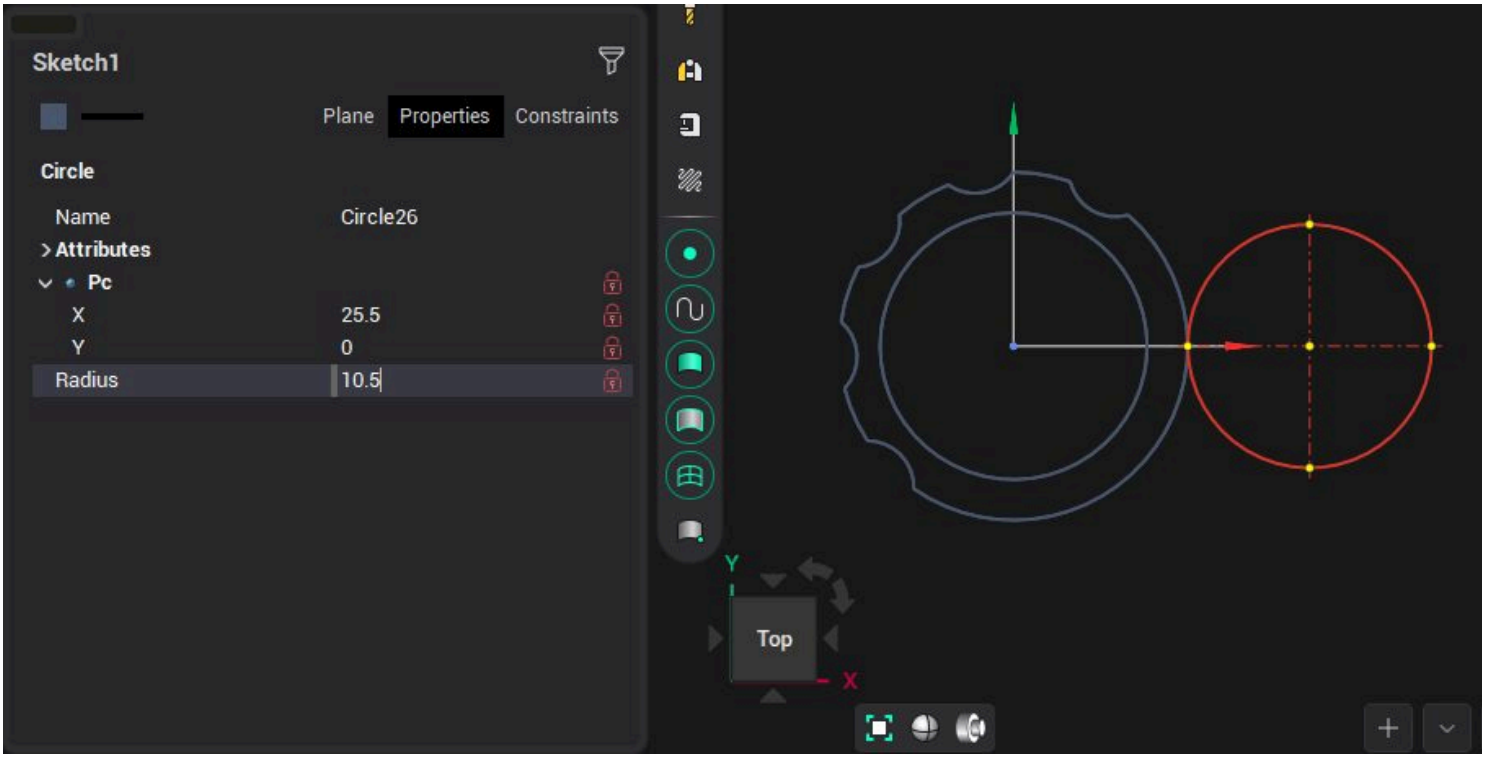
Next, turn on the trim function  and sequentially remove unnecessary sections:



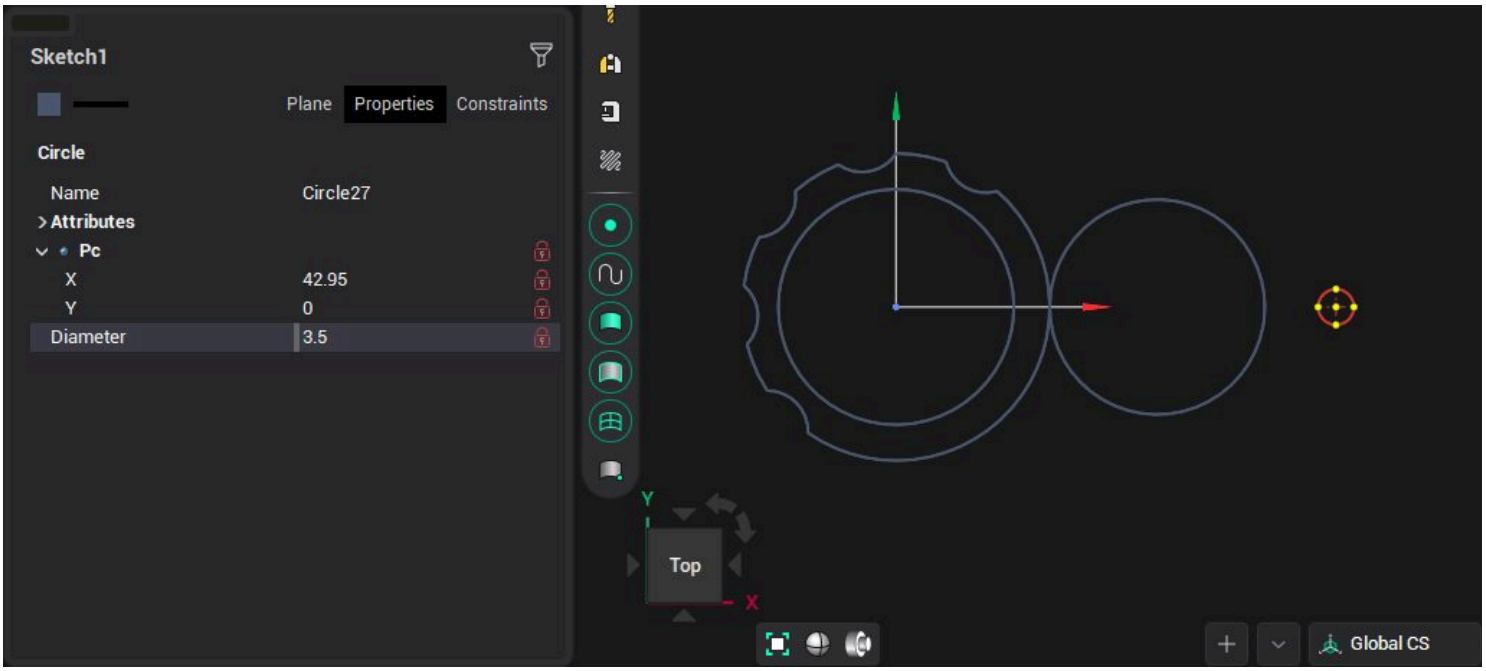
As a result, we get this figure:



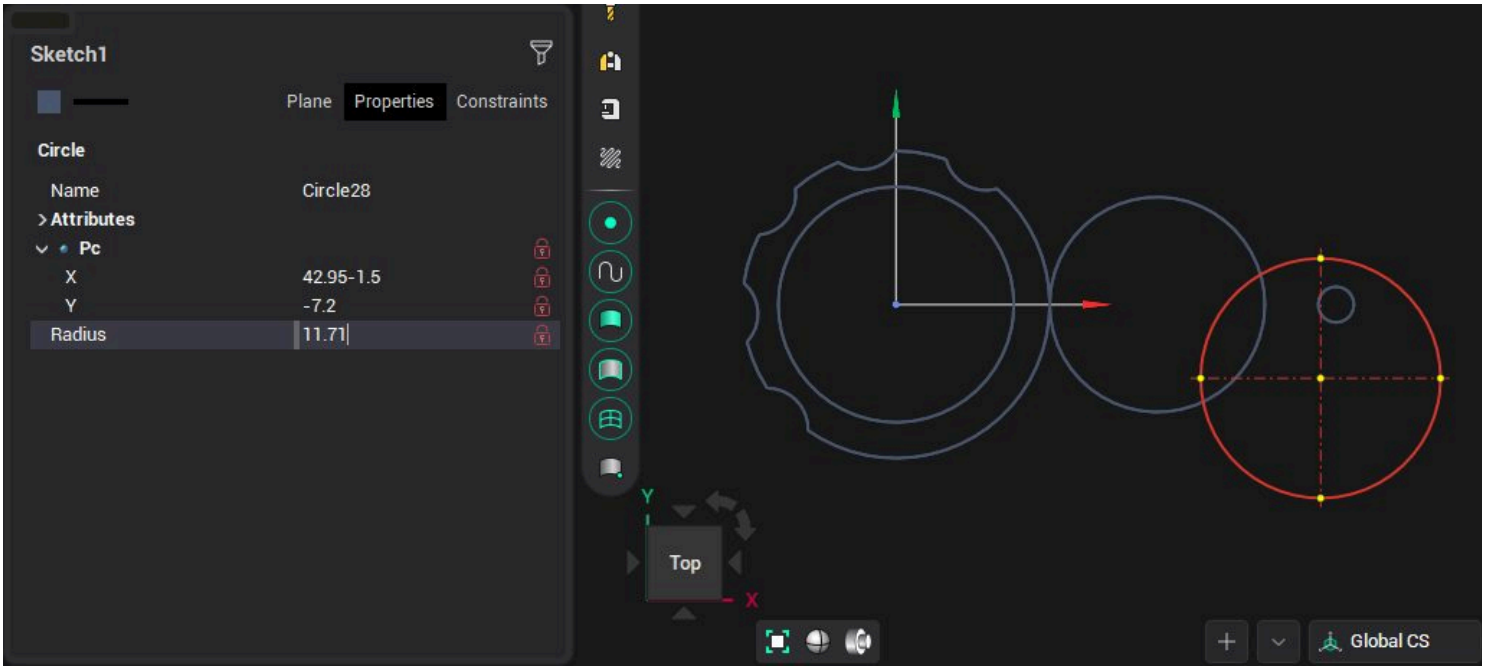
Turn on the construction of the circle again and enter the following values:



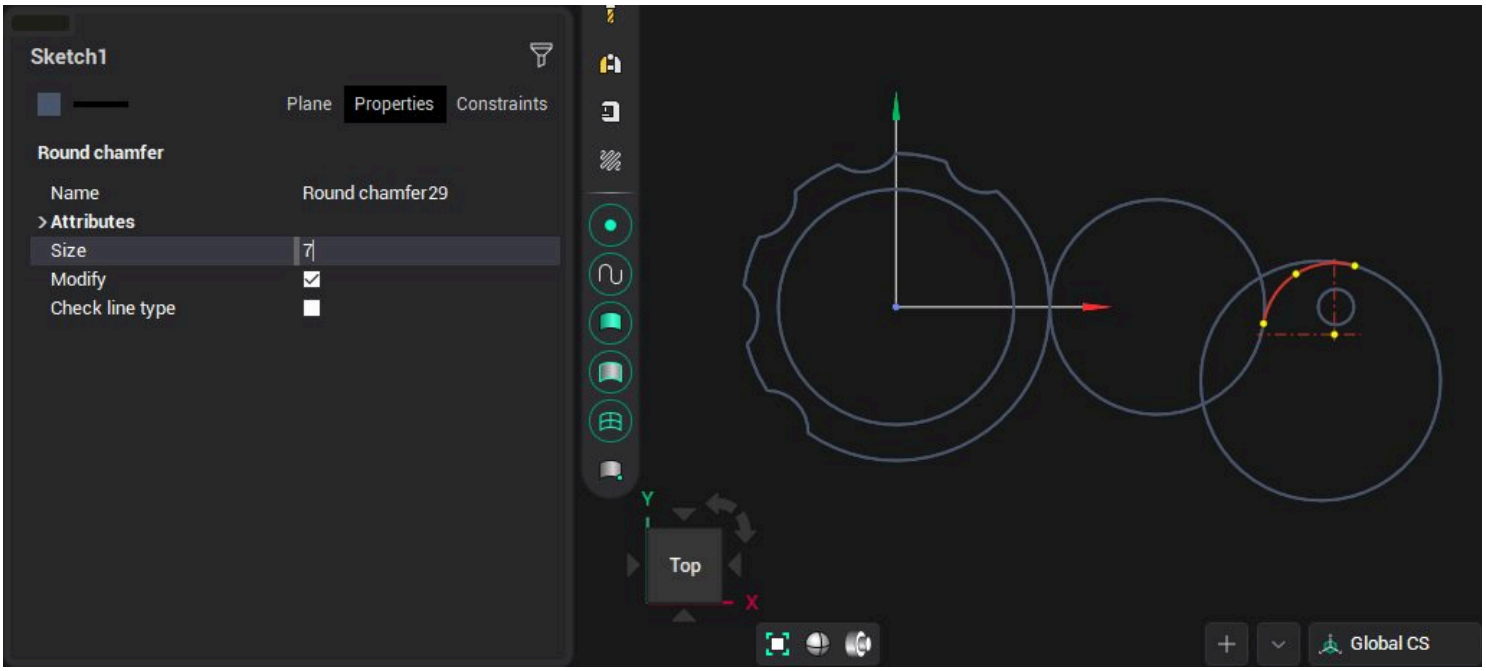
And the following circle:



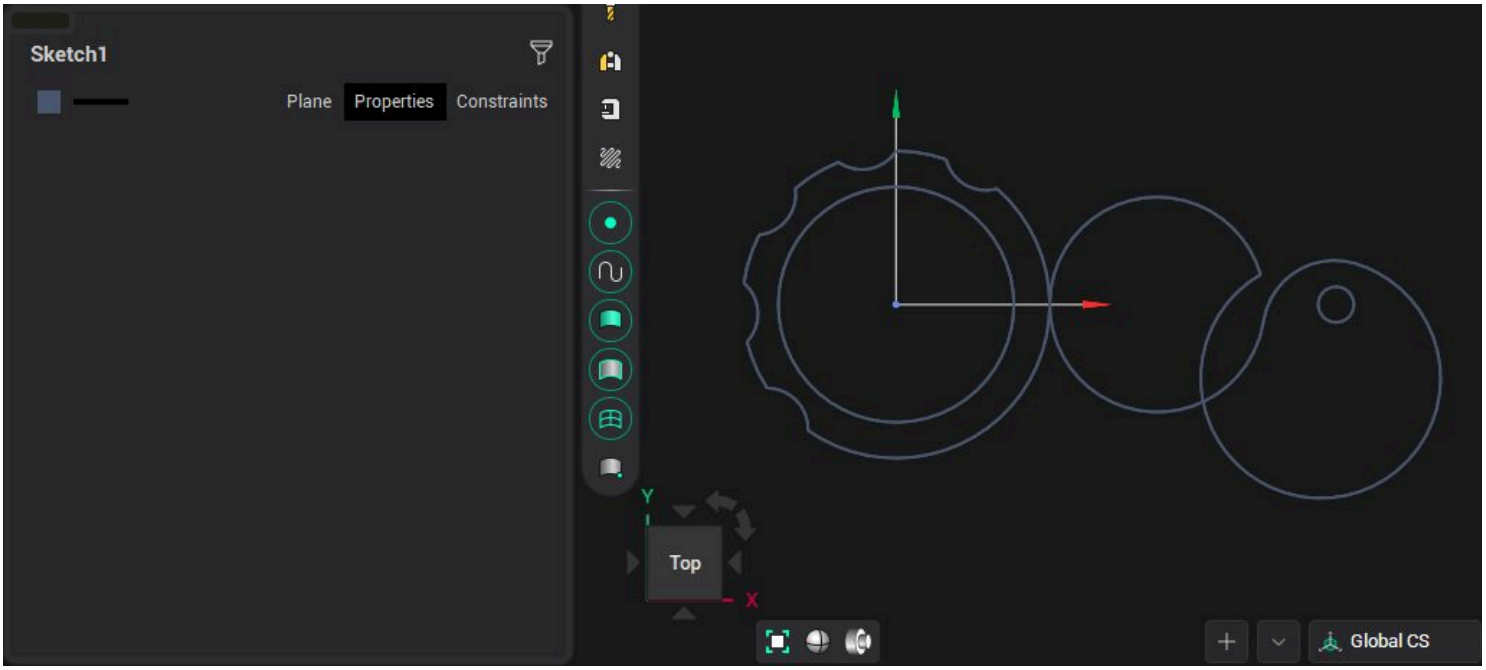
The following circle is the base of an arc with a radius of 11.71:



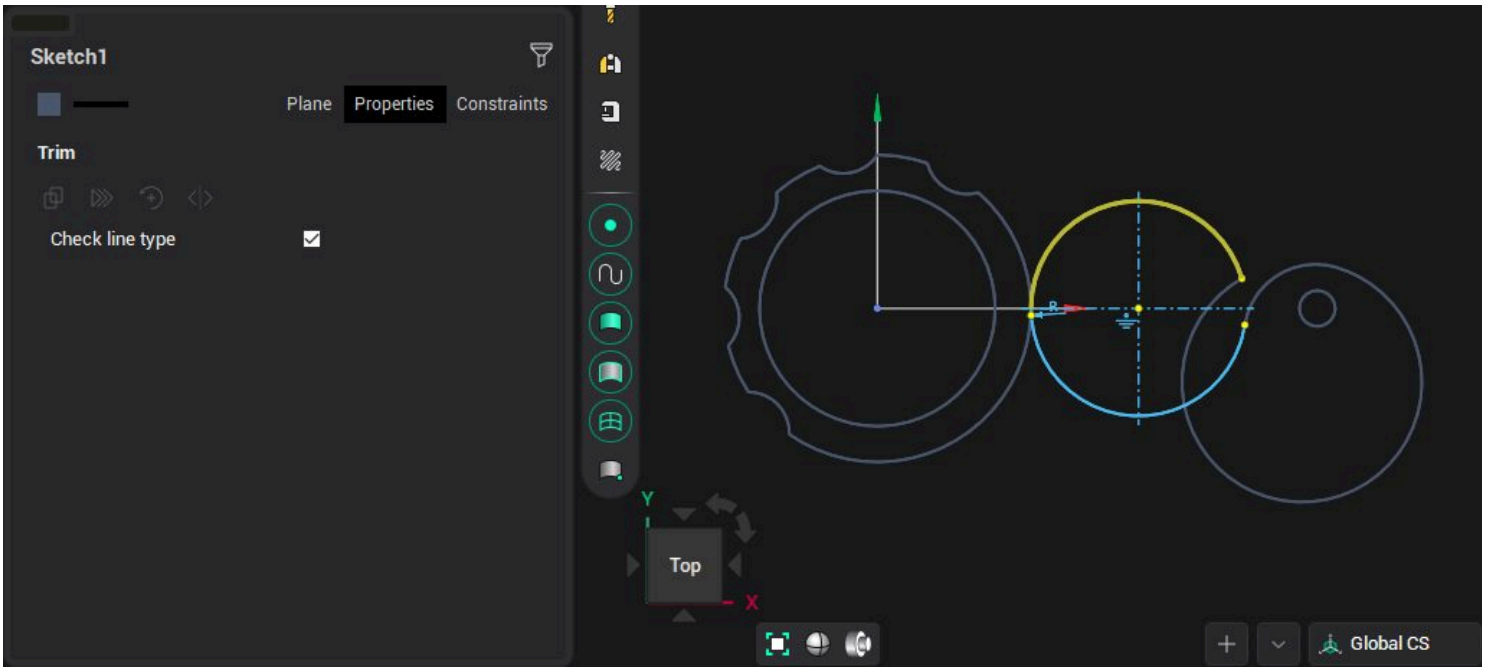
Turn on the construction of the rounding  and build an arc with a radius of 7:



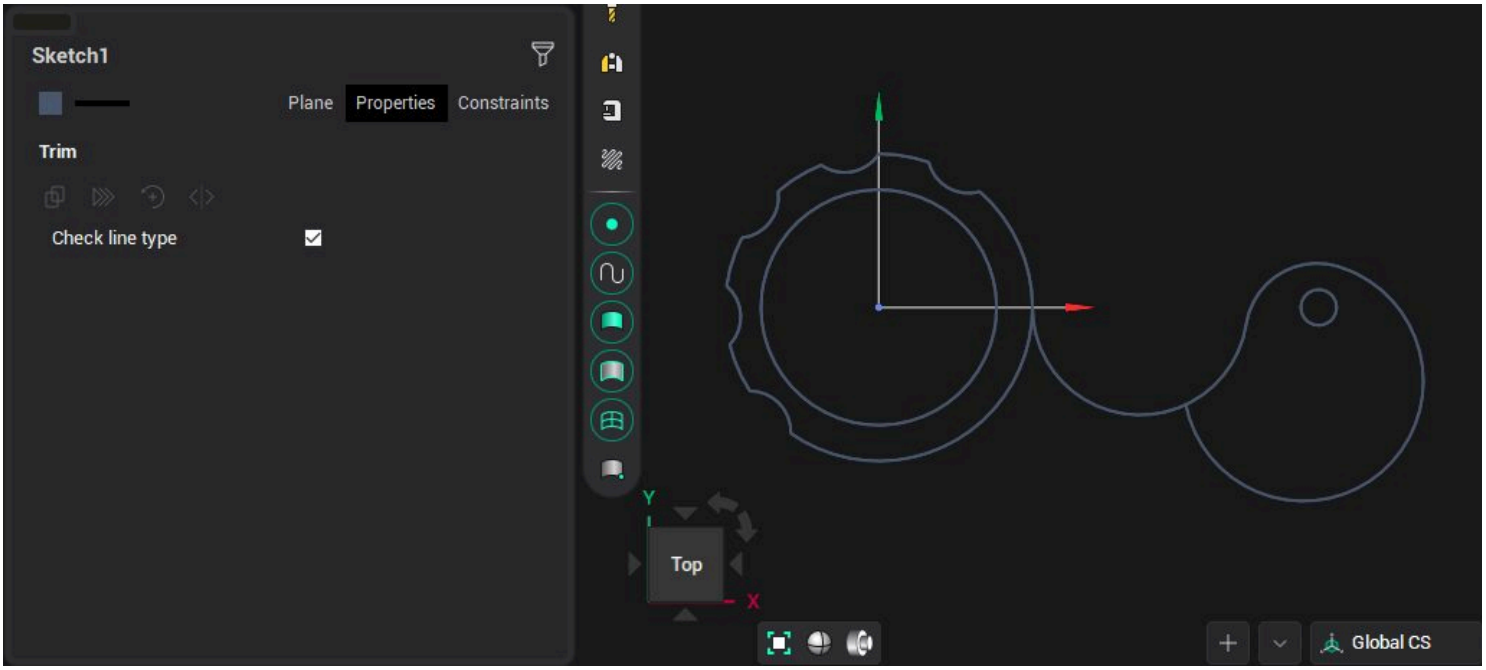
We got this:



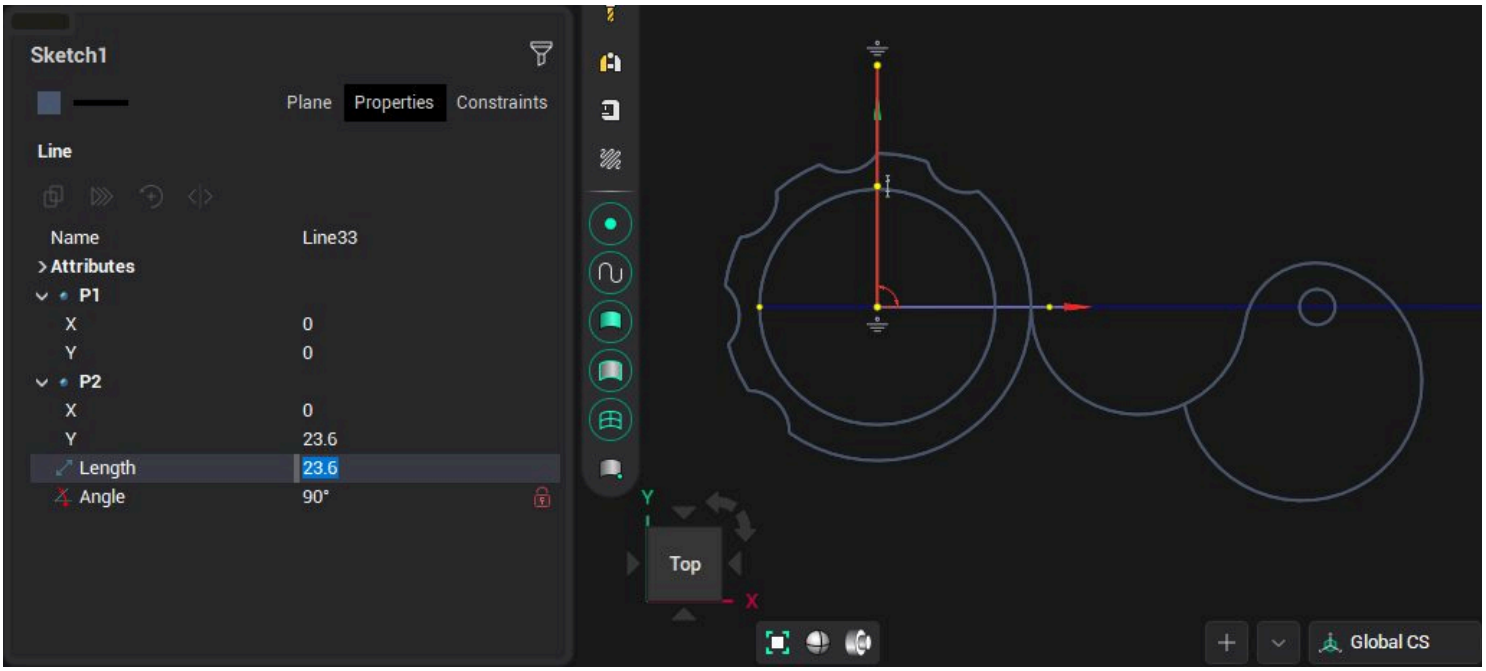
Turn on trim mode and remove two extra fragments:



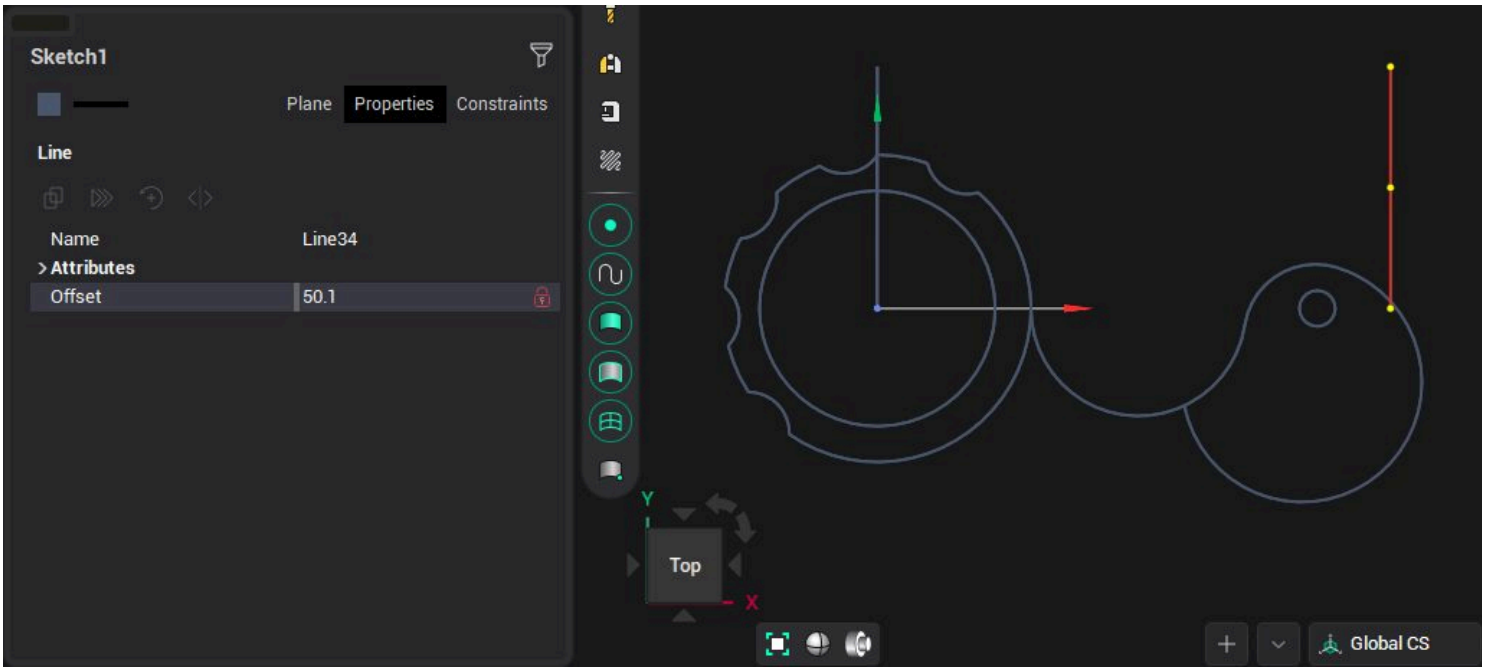
The result will be the following:



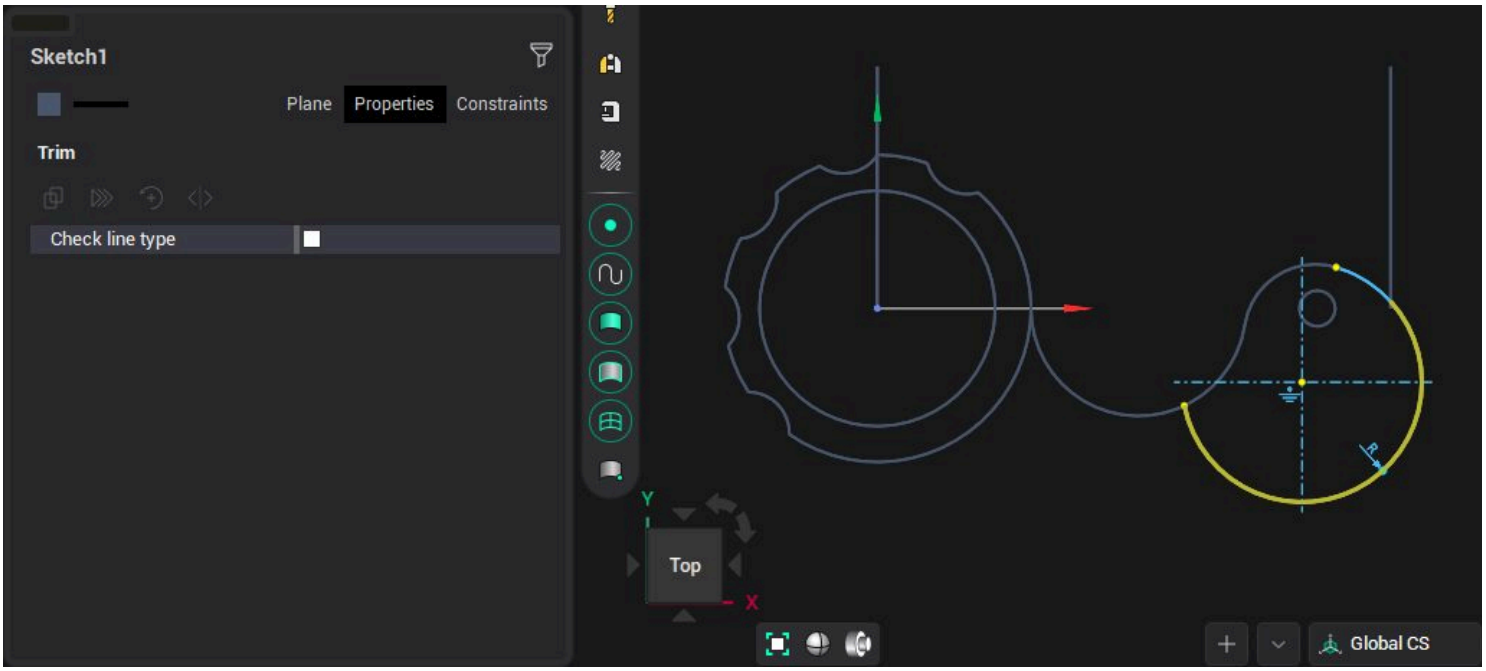
Build a vertical line:



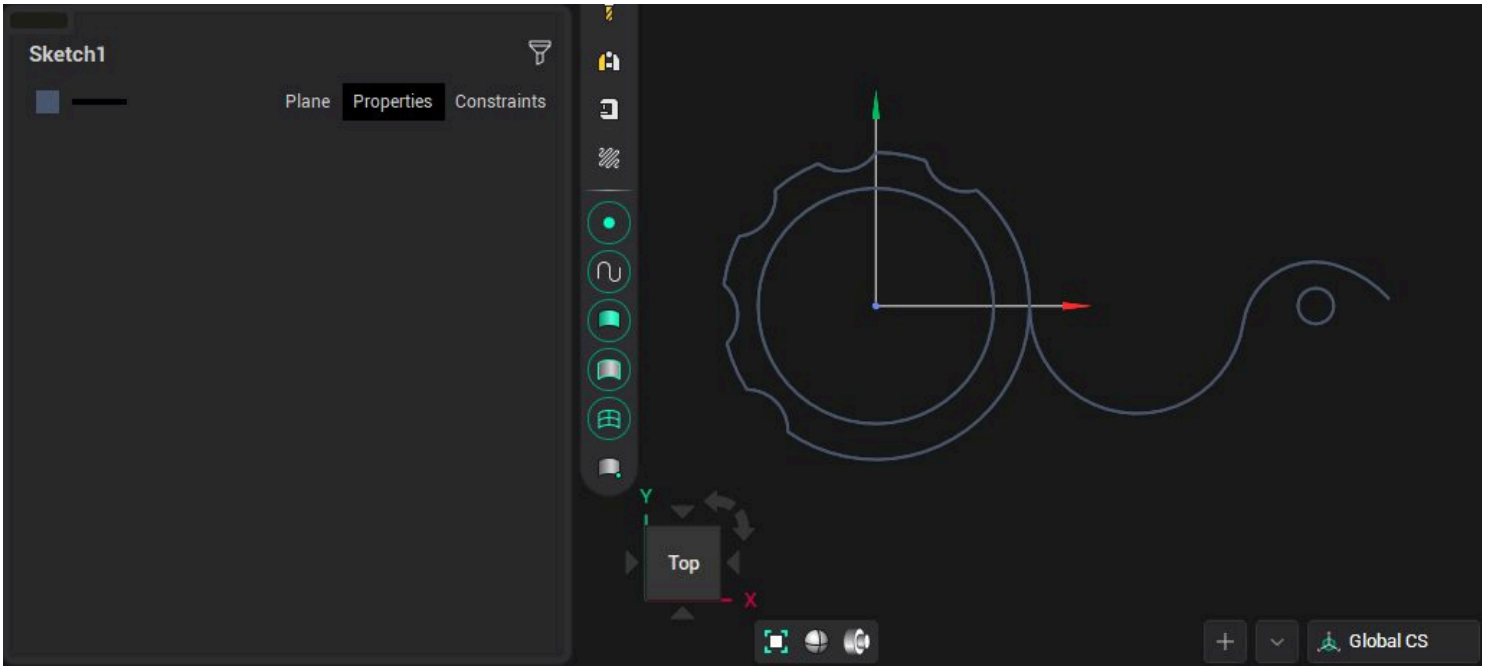
Turn on the construction of equidistant  and build a line parallel to the previous one at a distance of 50.1:



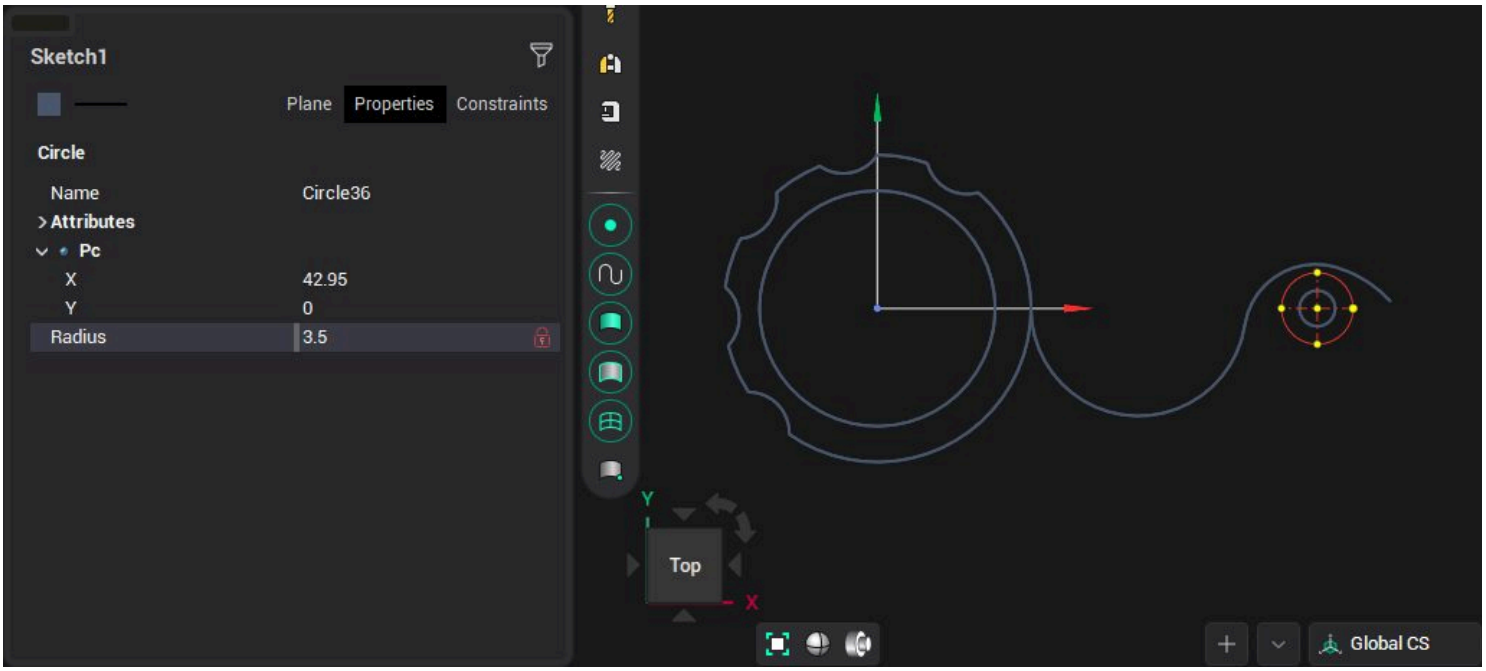
Turn on clipping and remove the excess part of the arc, having previously disabled the Check line type field in the inspector:



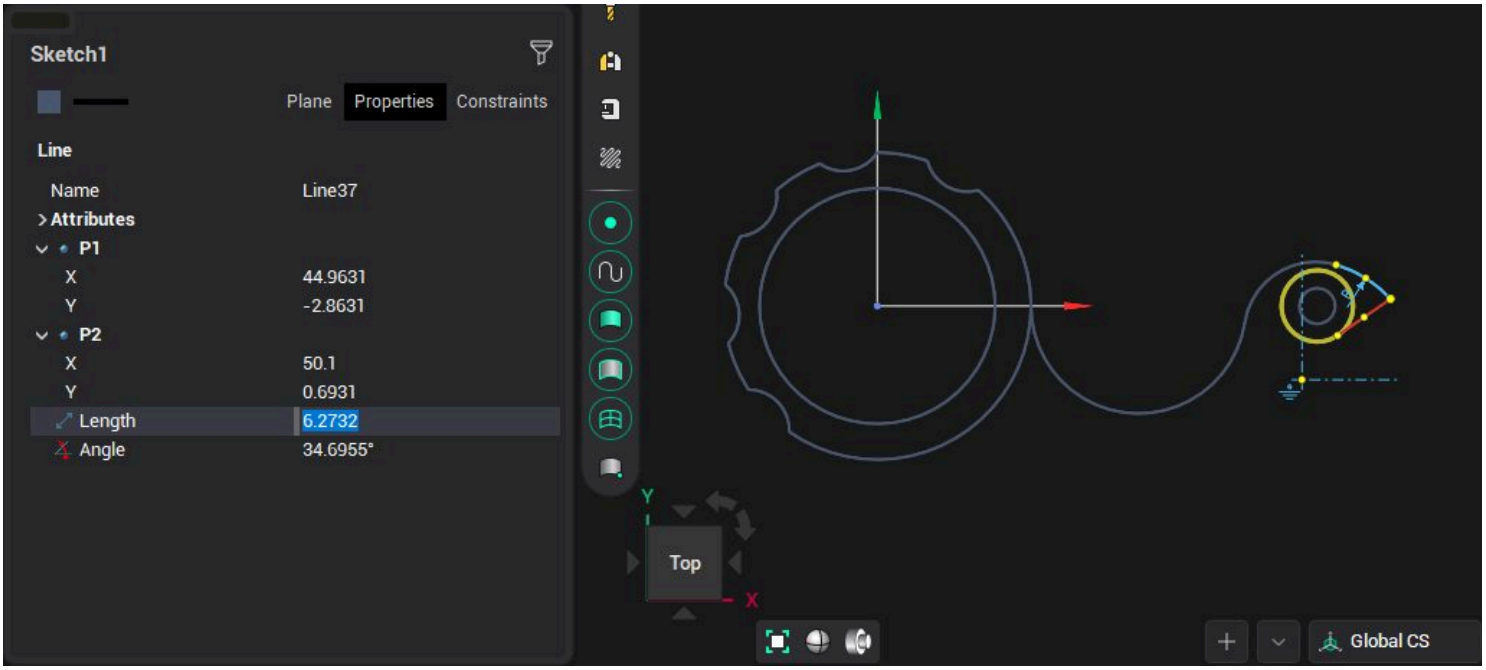
After that, we delete both time lines and we have the following result:



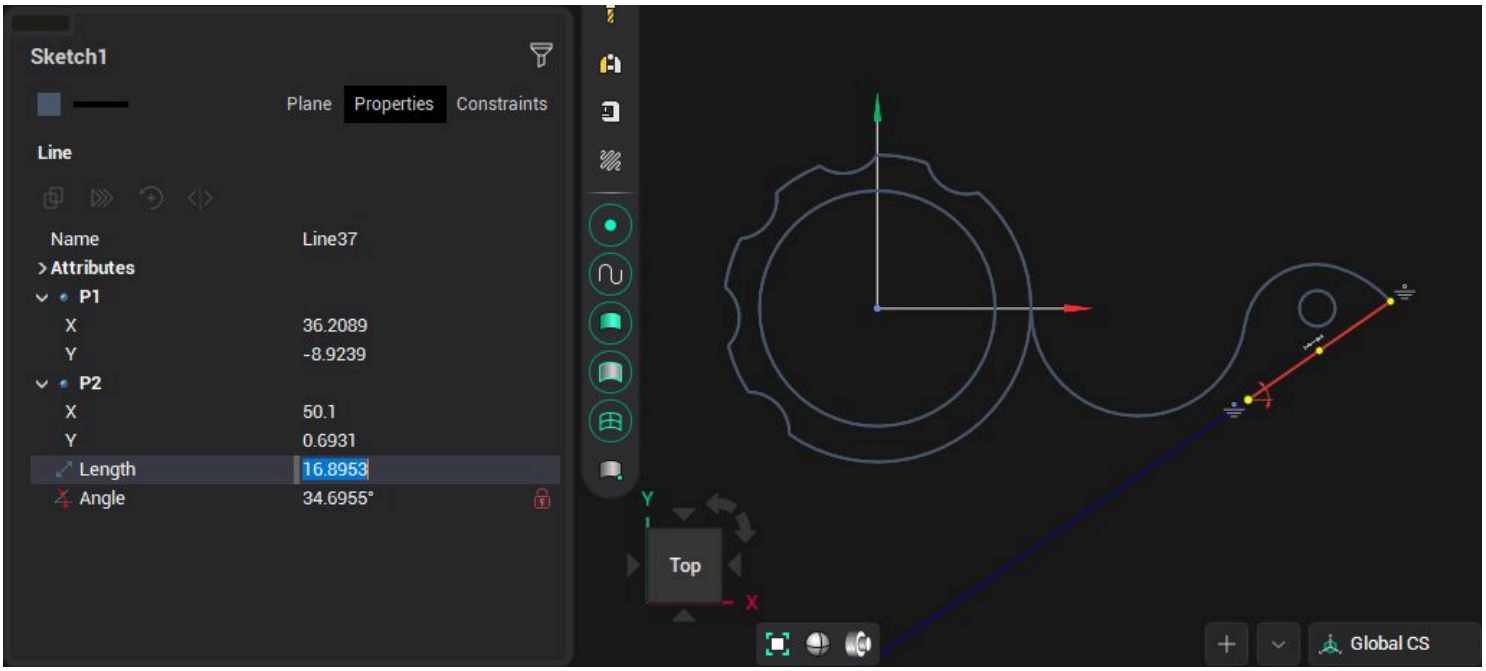
Now we build an auxiliary circle with a center at the point 42.95, 0 and a radius of 3.5:



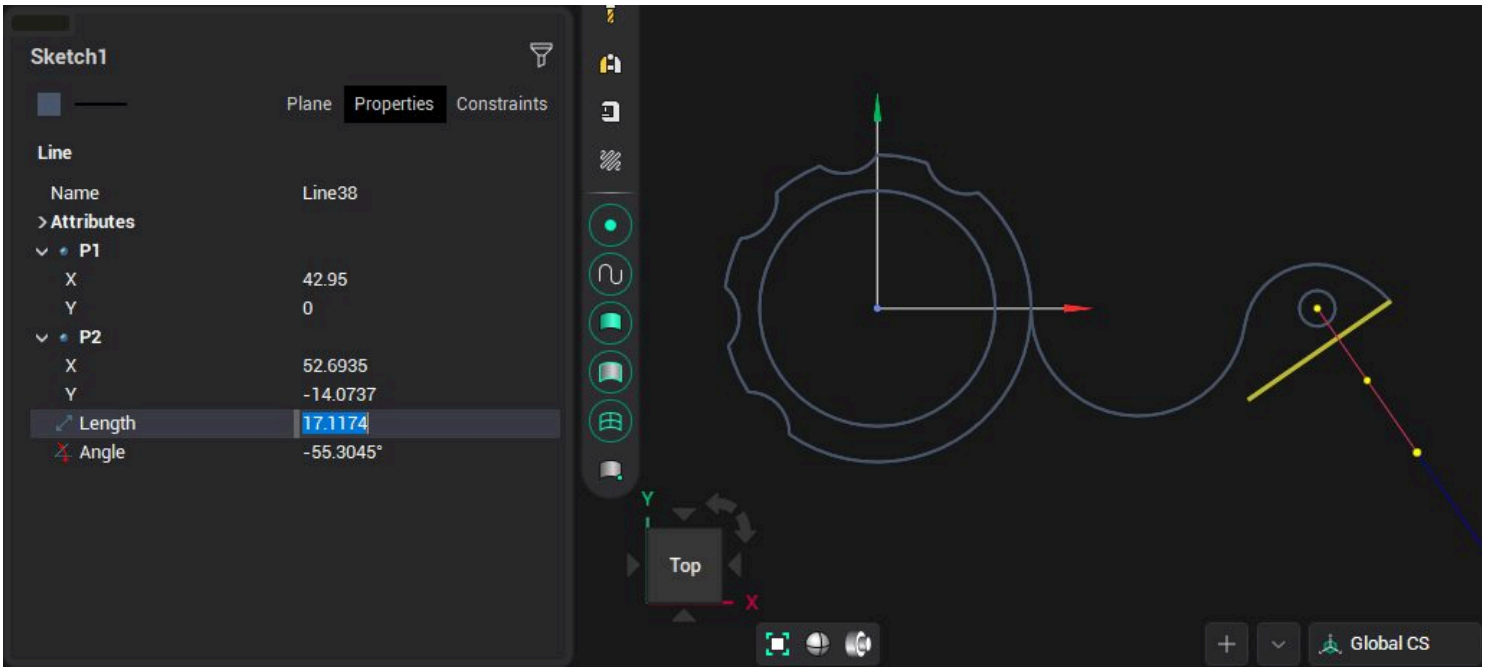
And then a line tangent to this circle, pressing [Ctrl] before selecting and connecting to the extreme arc:



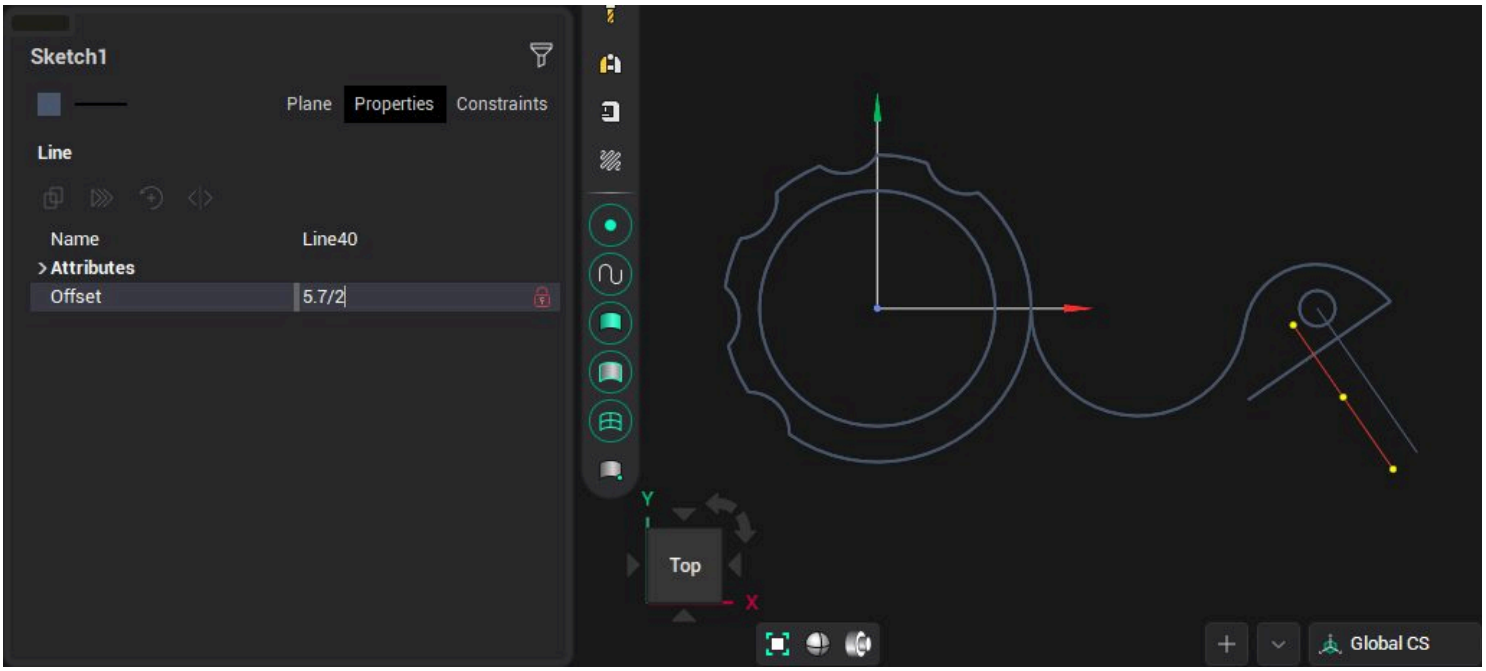
Delete the auxiliary circle, select the resulting line and fix the angle and lengthen it:



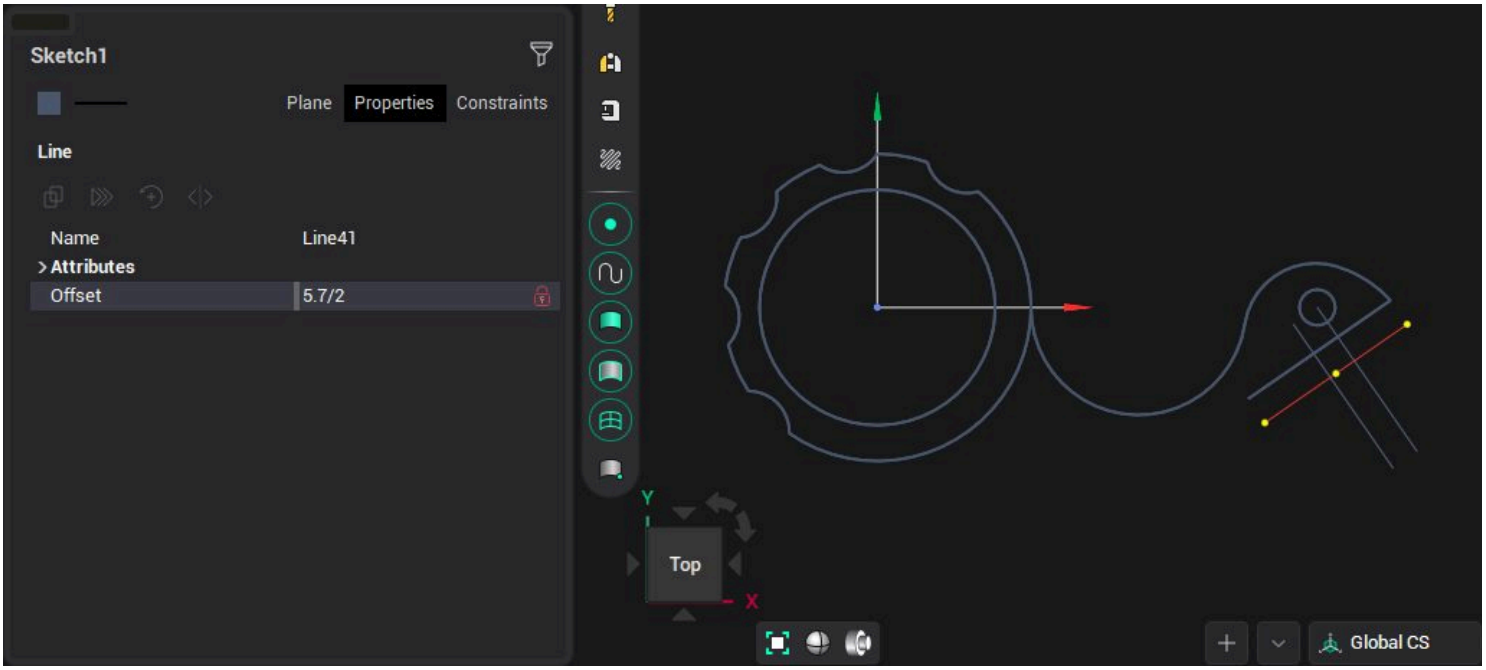
Next, we build another auxiliary line with a center from a small circle and perpendicular to the resulting line, for which we hover over the line with [Ctrl] pressed:



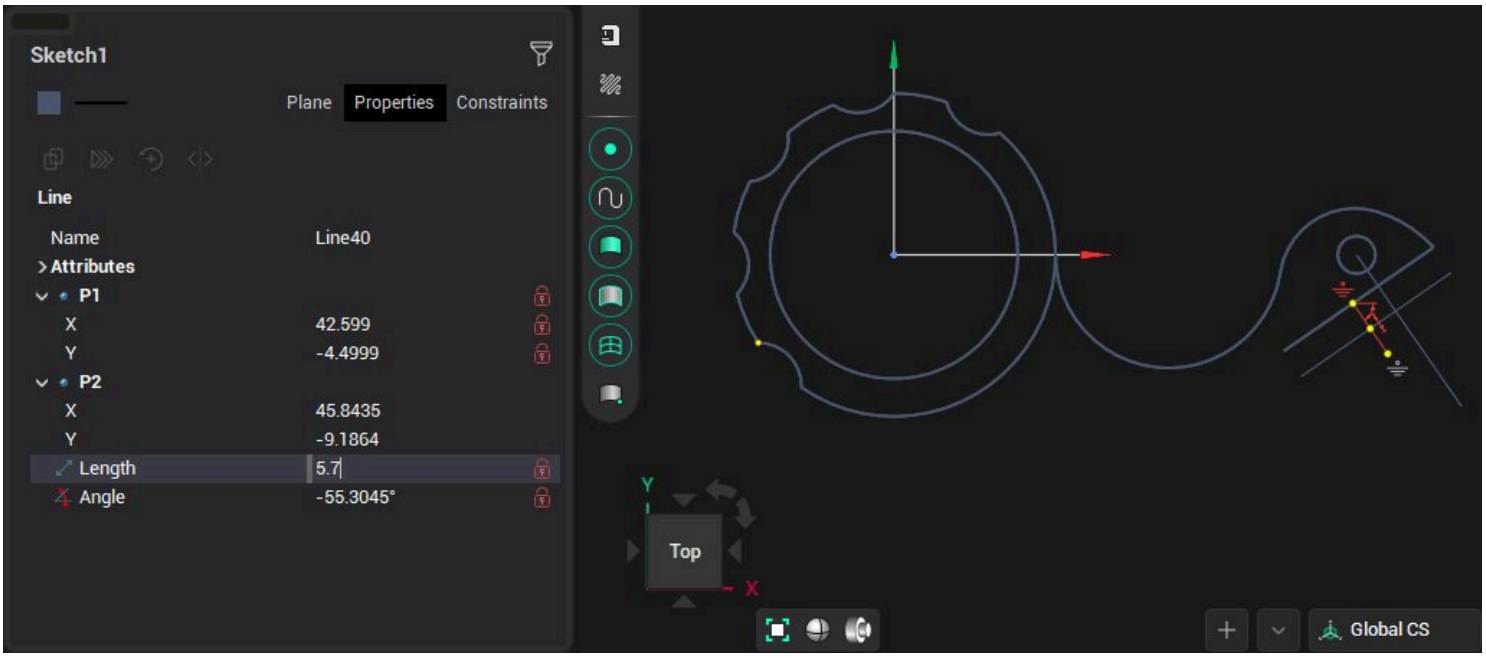
Turn on the construction of equidistant and build a line at a distance of 2.85 from it:



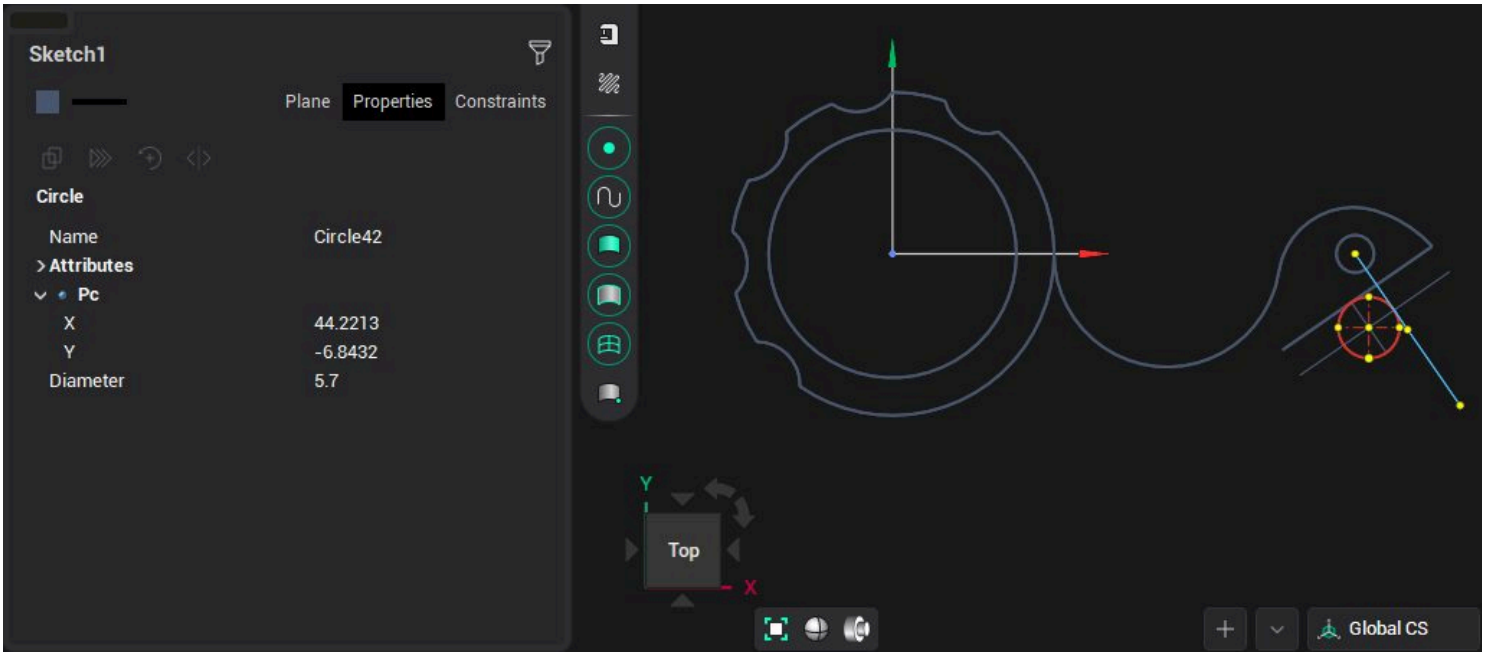
And another one at the same distance:



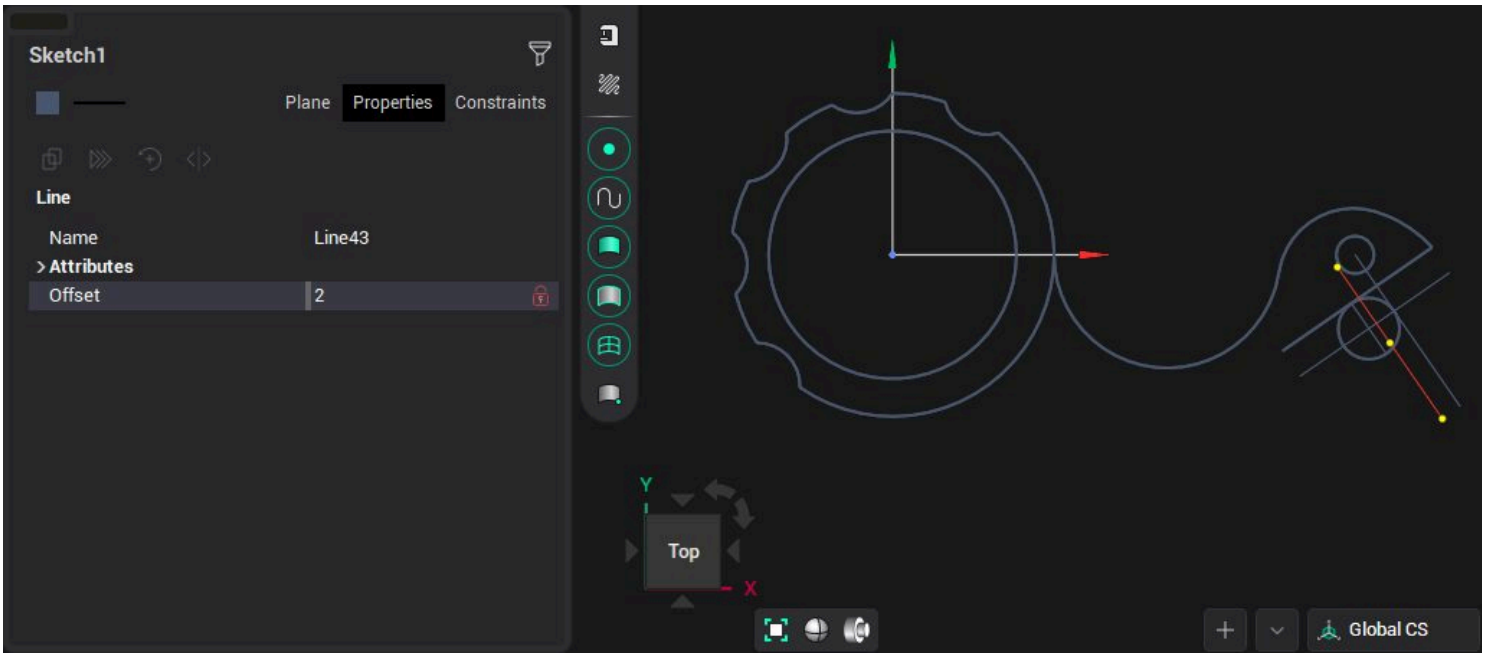
We correct the previous line, after fixing the angle in such a way that it remains on its line and has a length of 5.7:



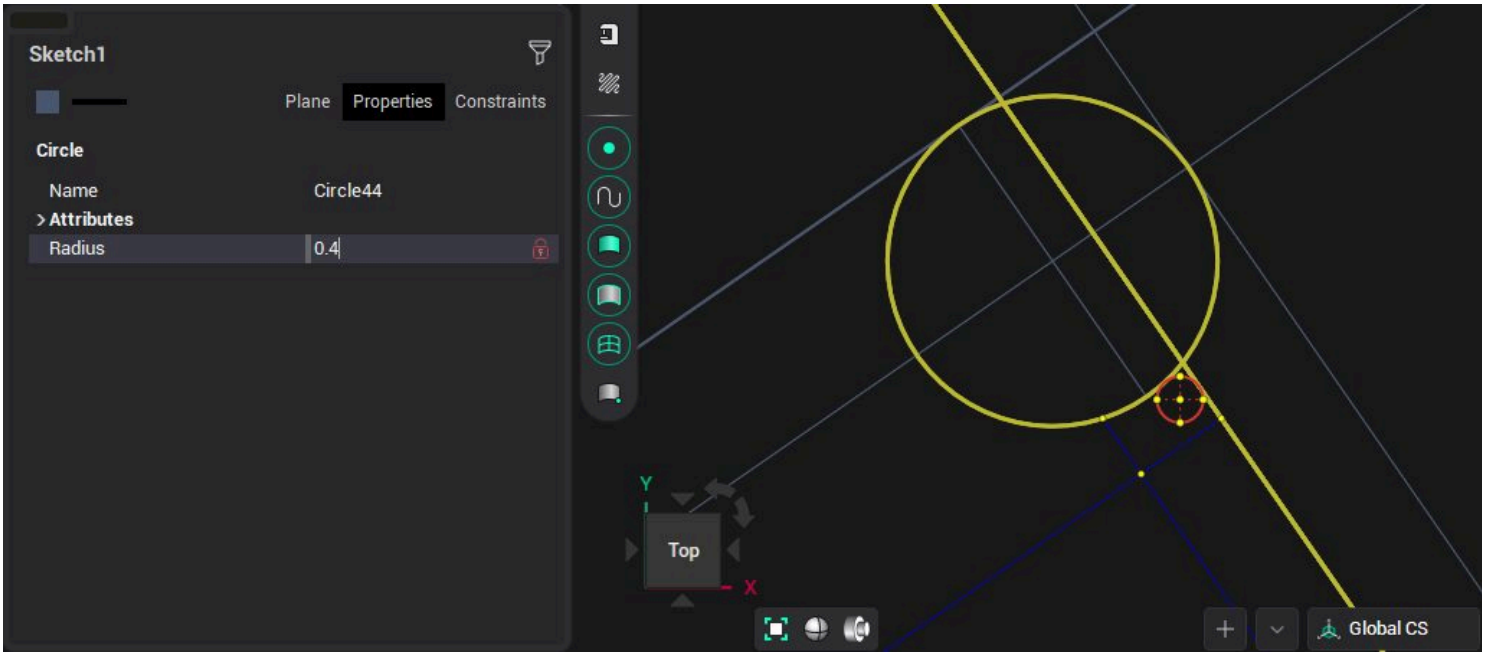
We build a circle according to the received element:



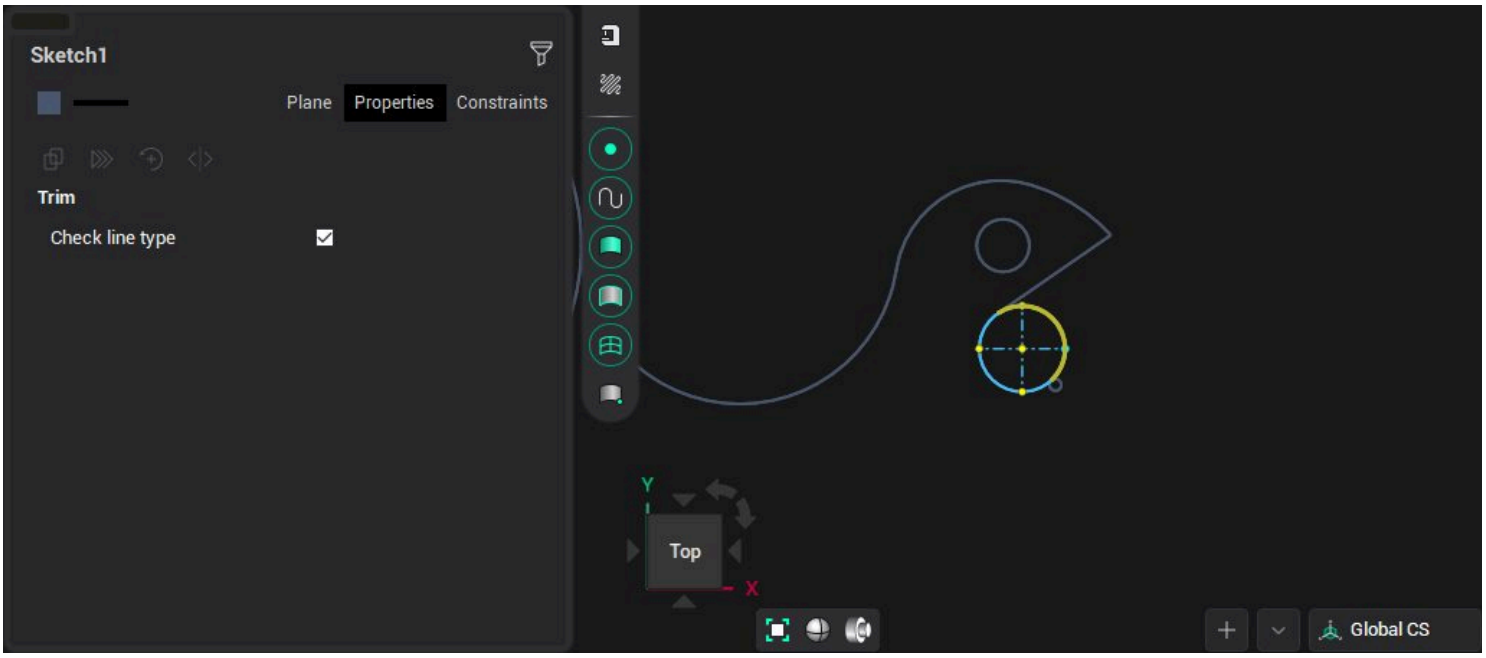
We construct another equidistant line:



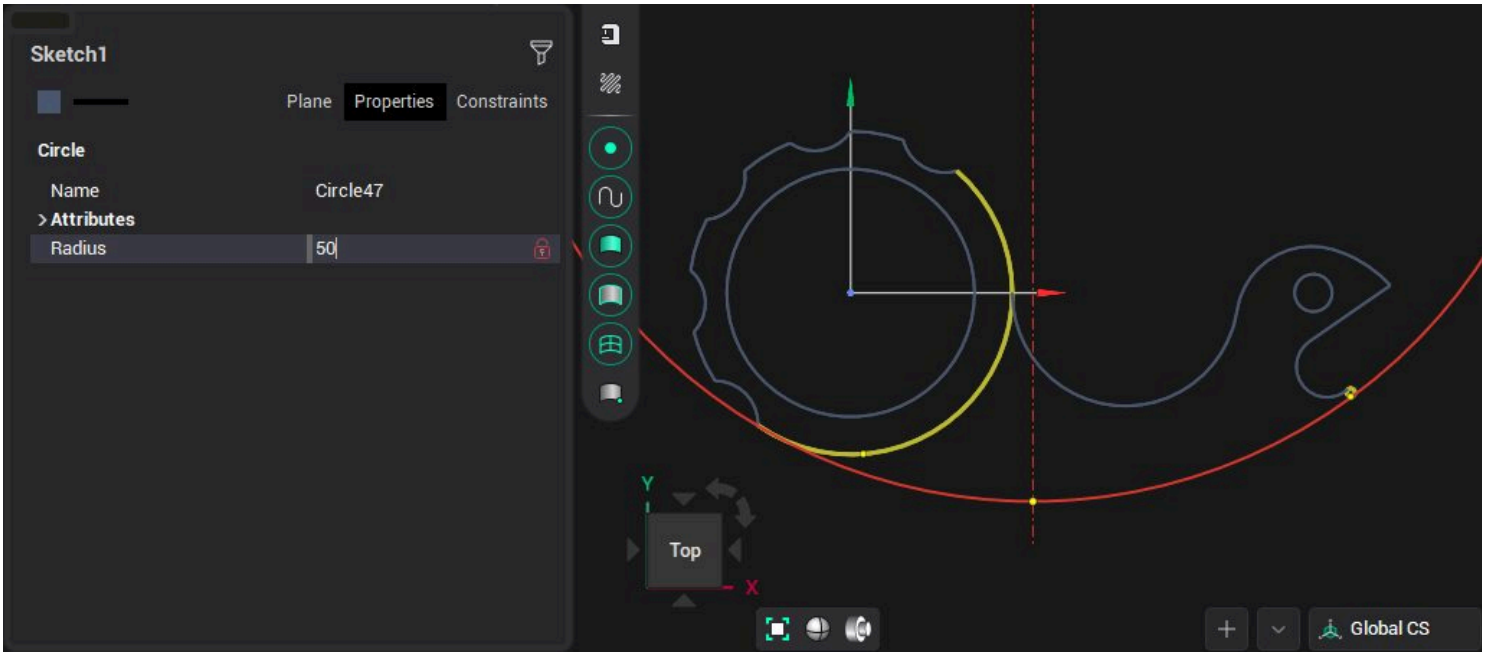
And a circle tangent to a circle and a line with a radius of 0.4:



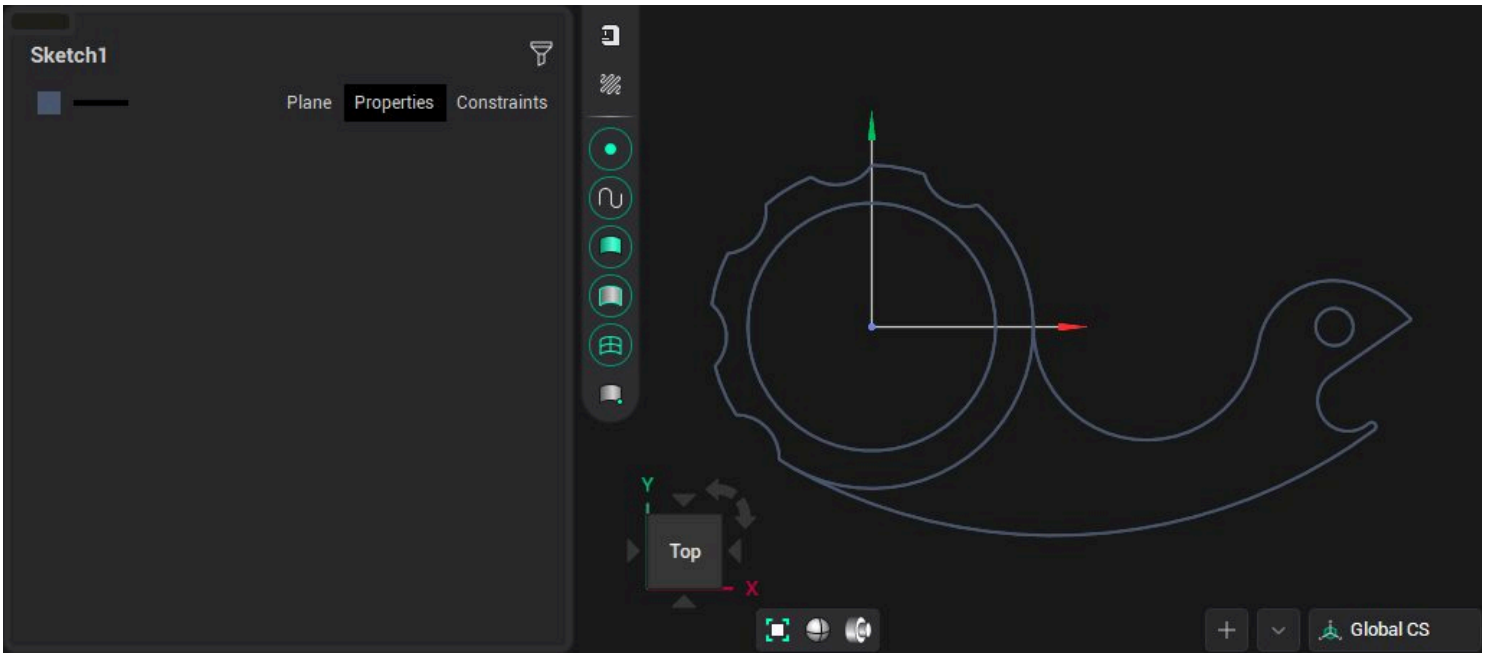
Unnecessary more auxiliary elements are deleted, the excess is cut off:



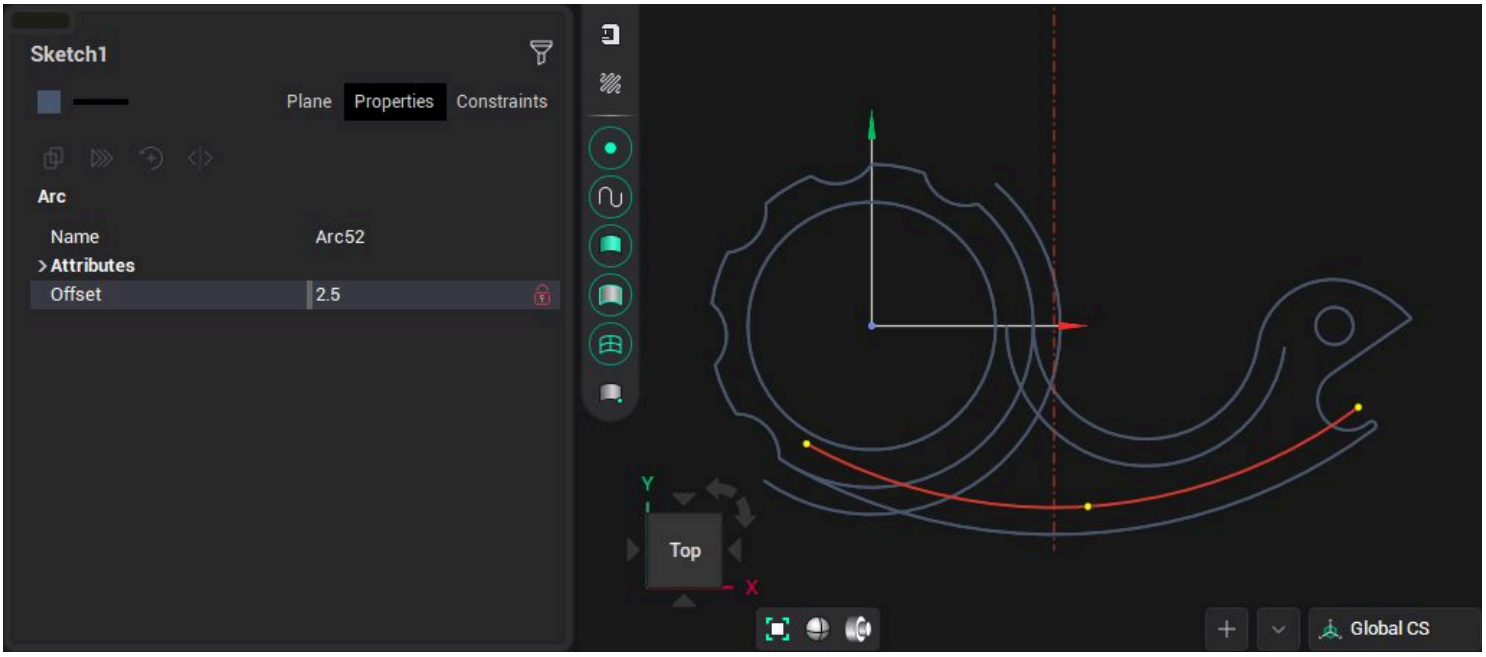
We build another circle, tangent to the previously constructed ones, with a radius of 50:



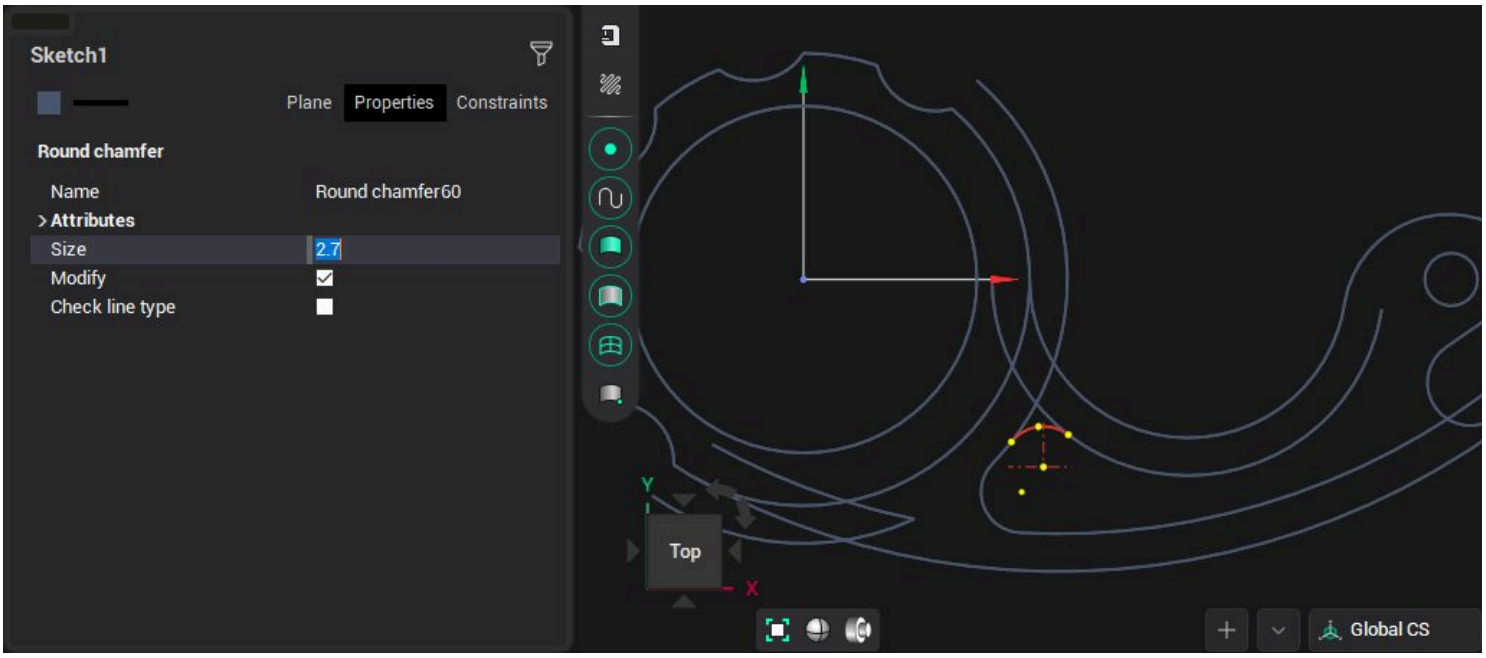
We cut off the extra sections of the circles and get the required figure:



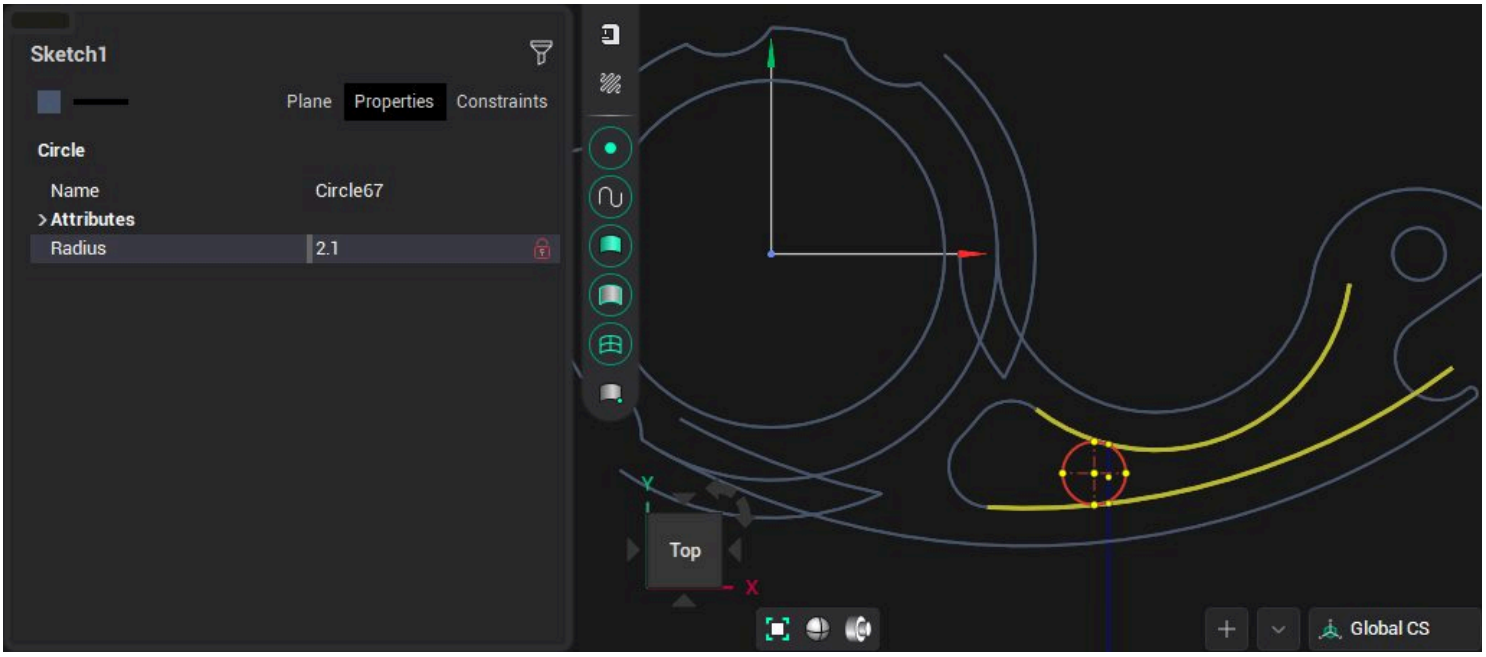
To build the inner figure, we build three equidistant arcs with a distance of 2.5 from the original ones:



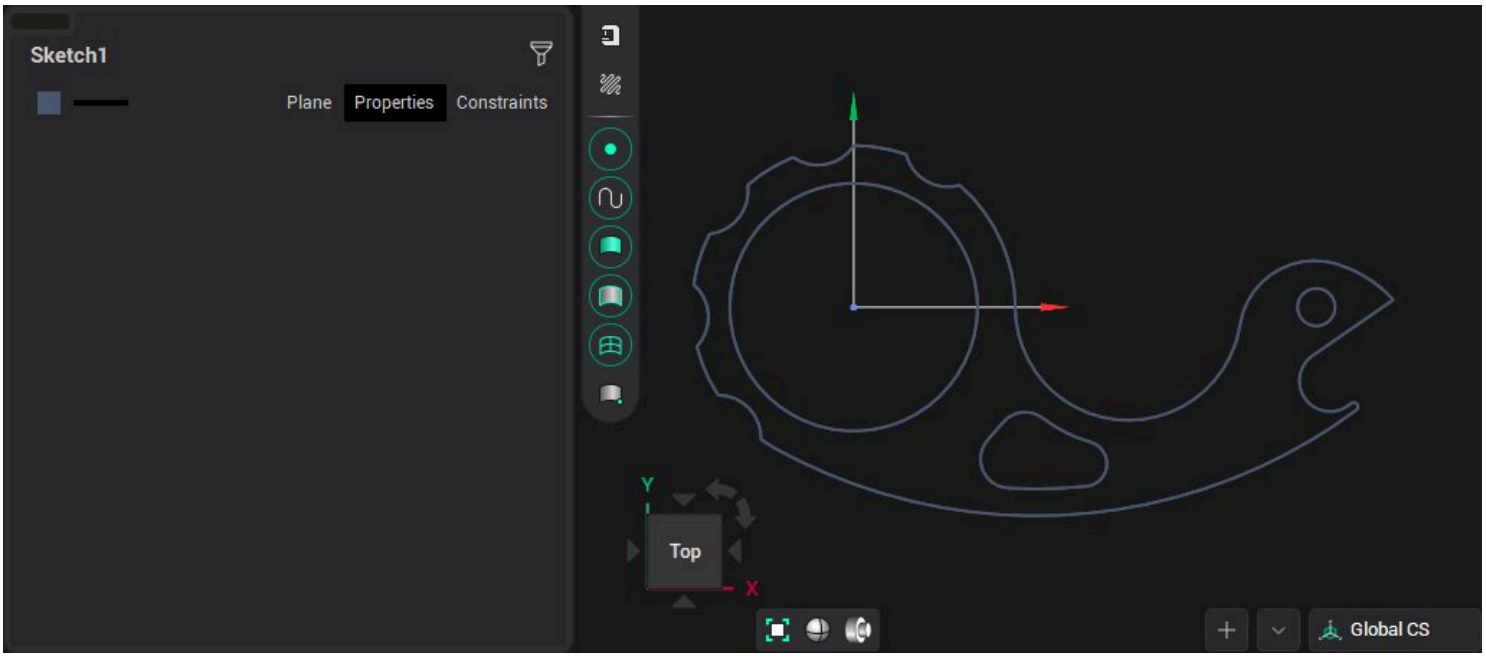
Turn on the construction of the rounding  and specify two elements with a size of 2.7:



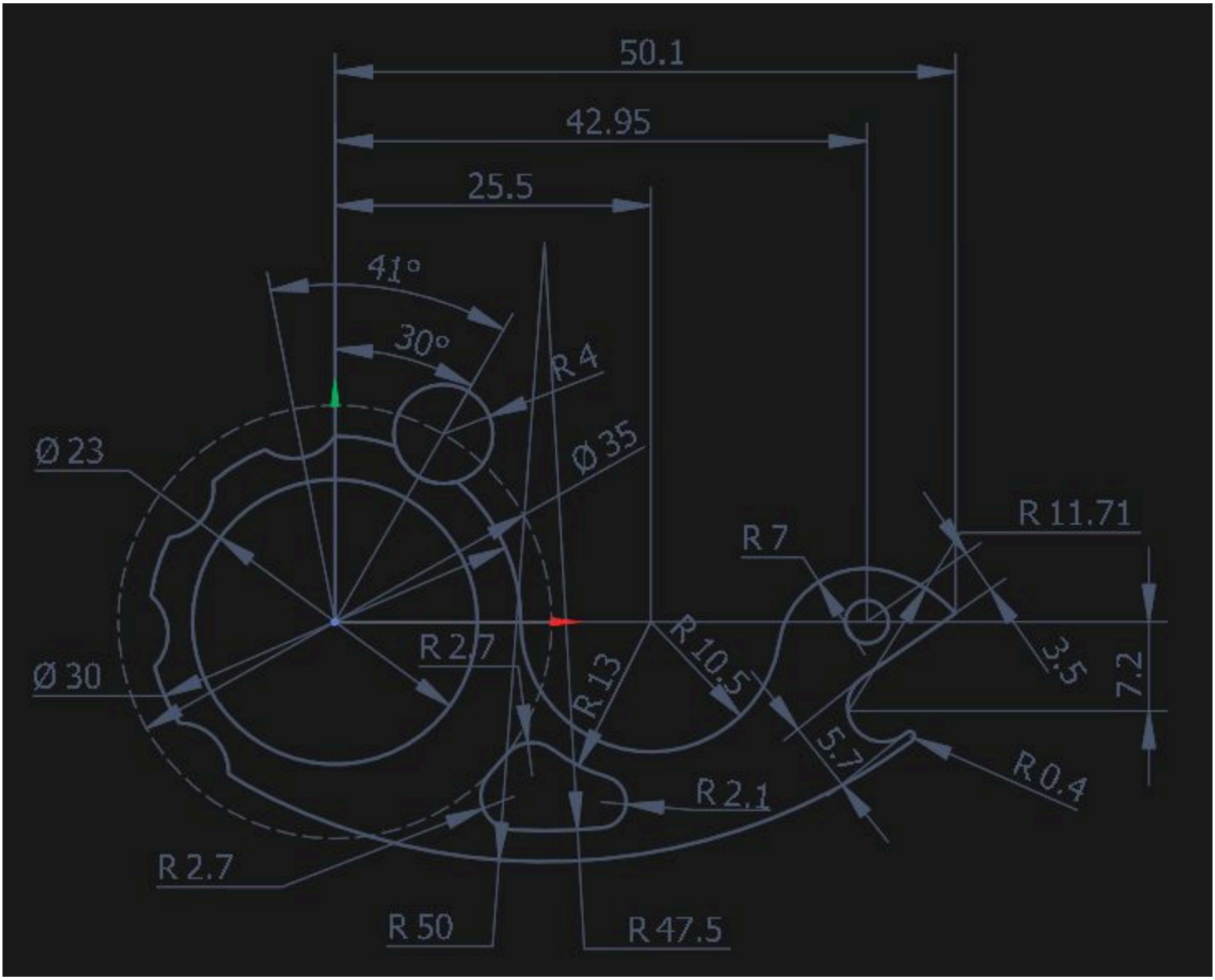
We construct the last element as a circle tangent to two arcs and with a radius of 2.1:



Delete trim elements and cut off unnecessary parts of the arcs, this completes the construction:

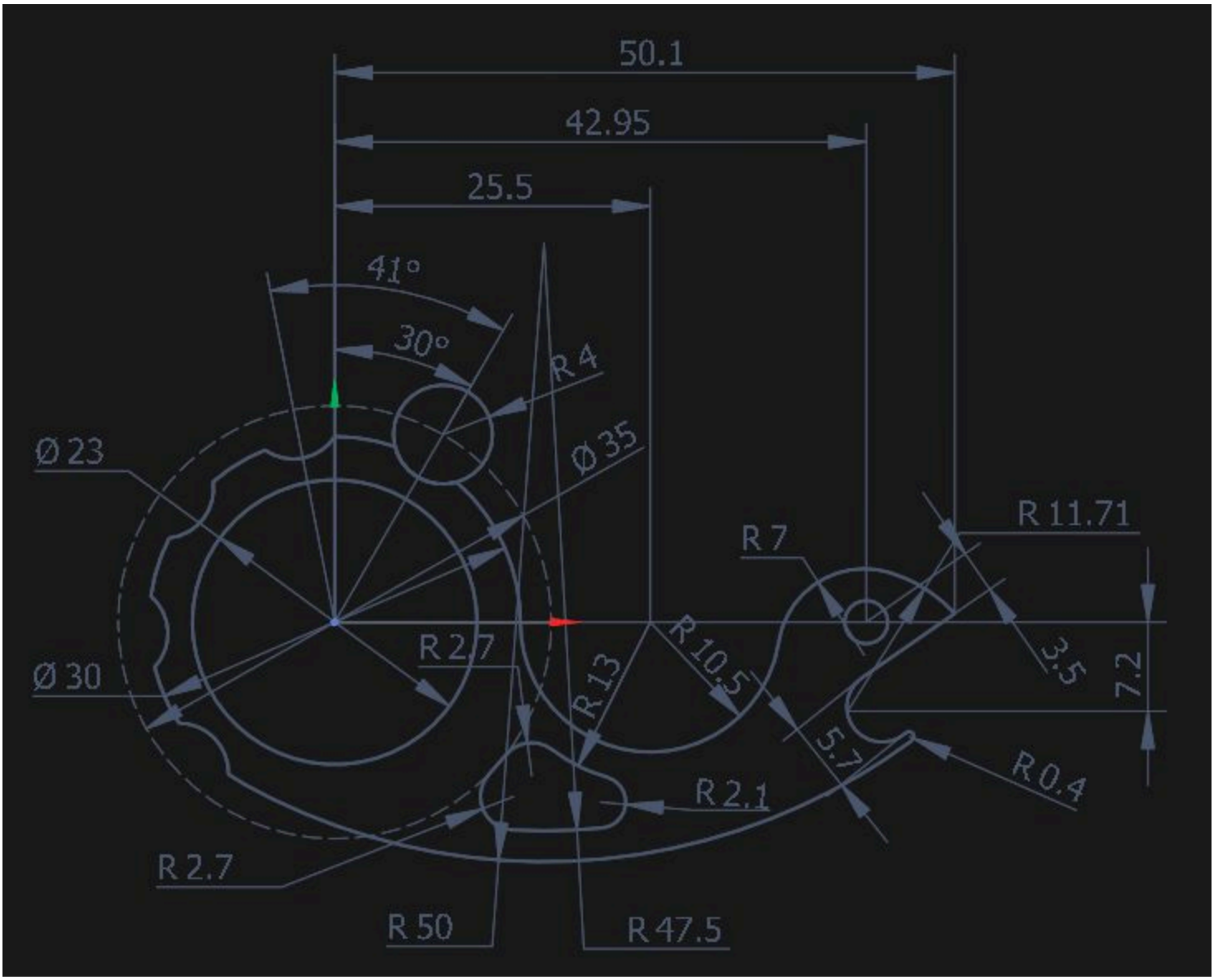


And our drawing after dimensioning:

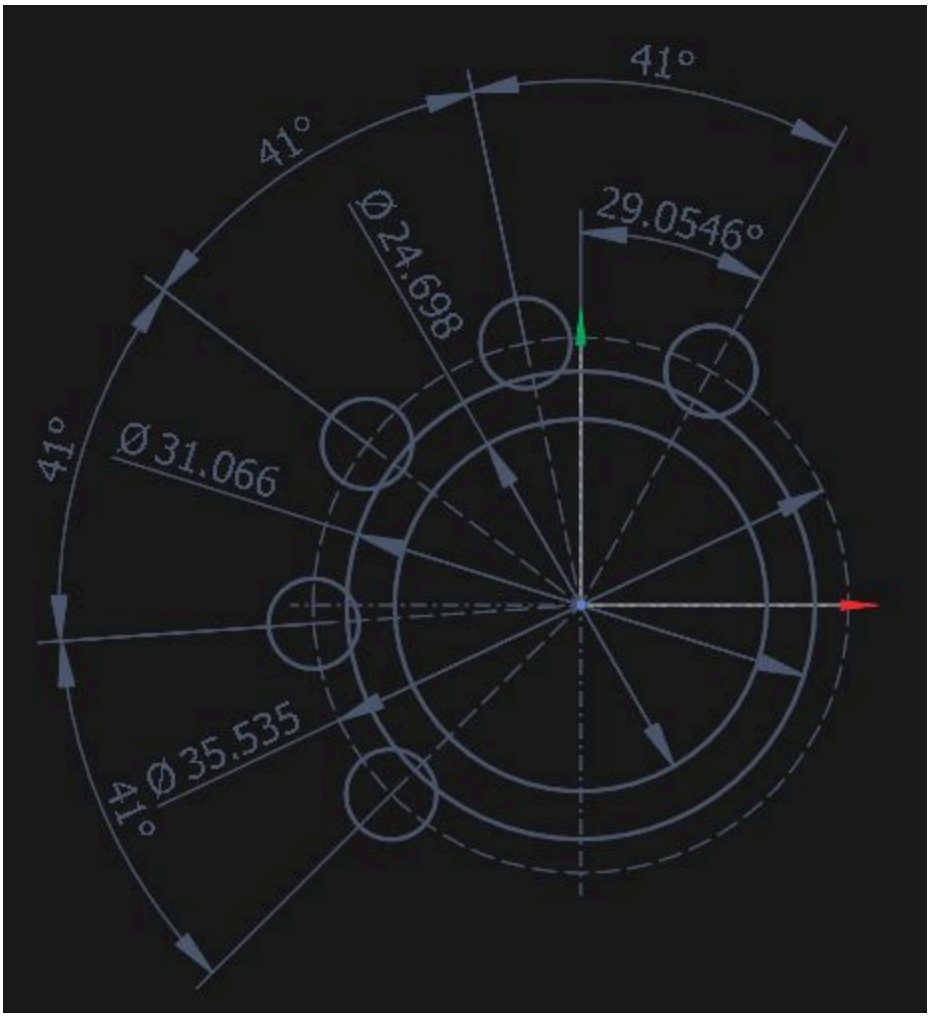


Example of building a drawing using constrains

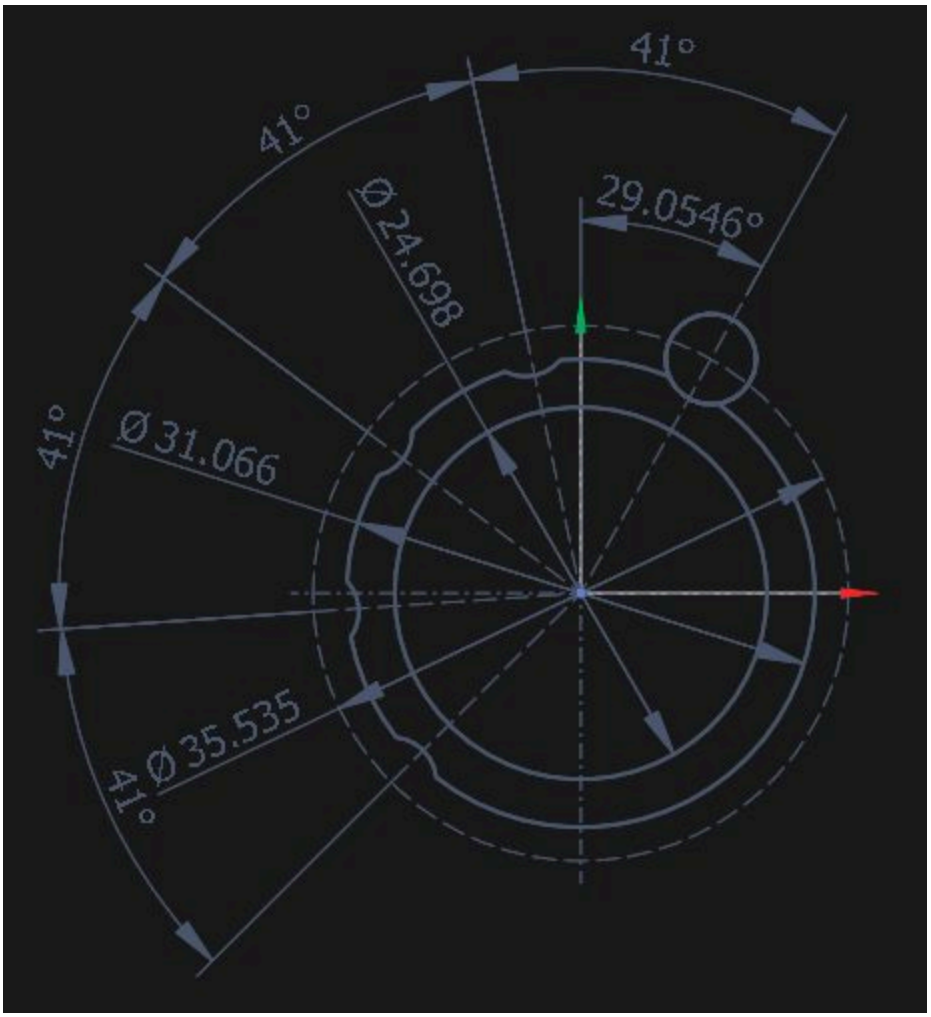
As example, let's take the same drawing as previously:



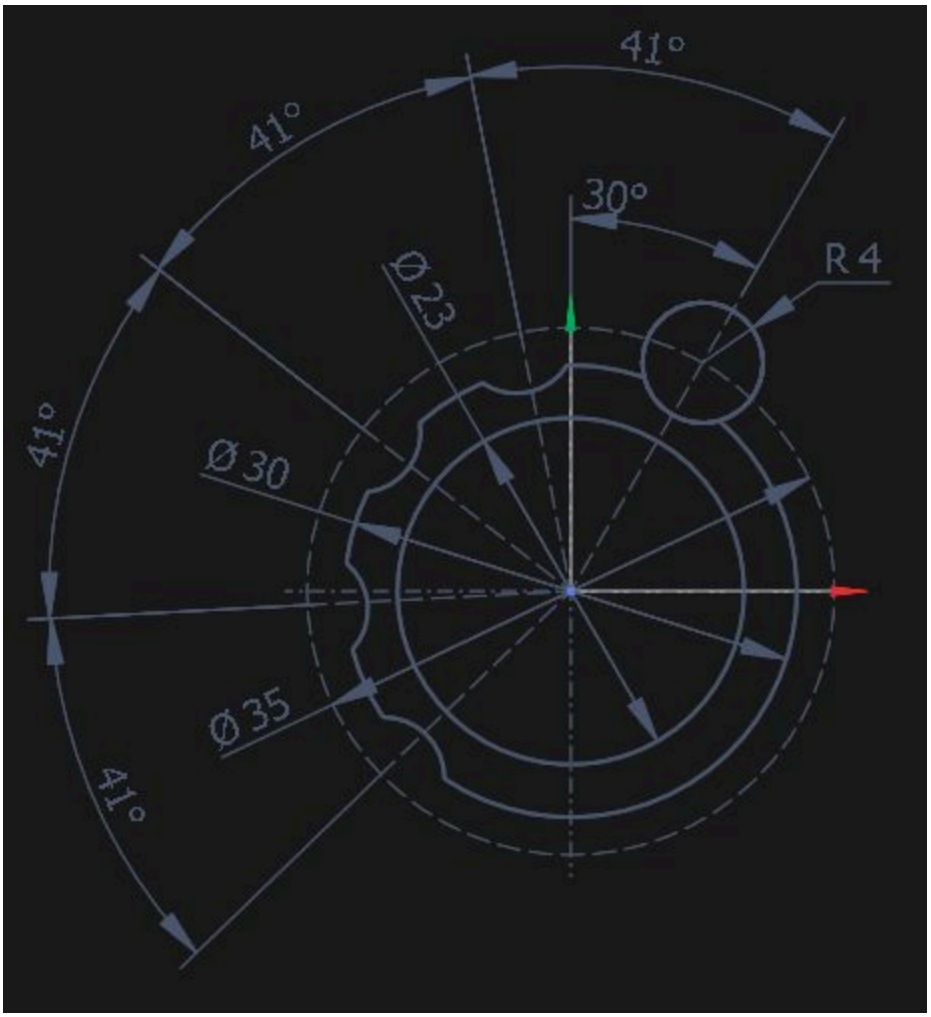
To begin with, from the point 0,0 we build three circles with an arbitrary radius, but more or less respecting the scale. From the center we draw a line at an arbitrary angle for the circle R4. We do not comply with the dimensions - then we will bring them to those indicated on the drawing. We build a circle and rotate with copying at an angle of 41 °, having previously indicated the center of rotation at the point 0.0. Next, we build axes to the resulting circles and arrange the angular and radial dimensions:



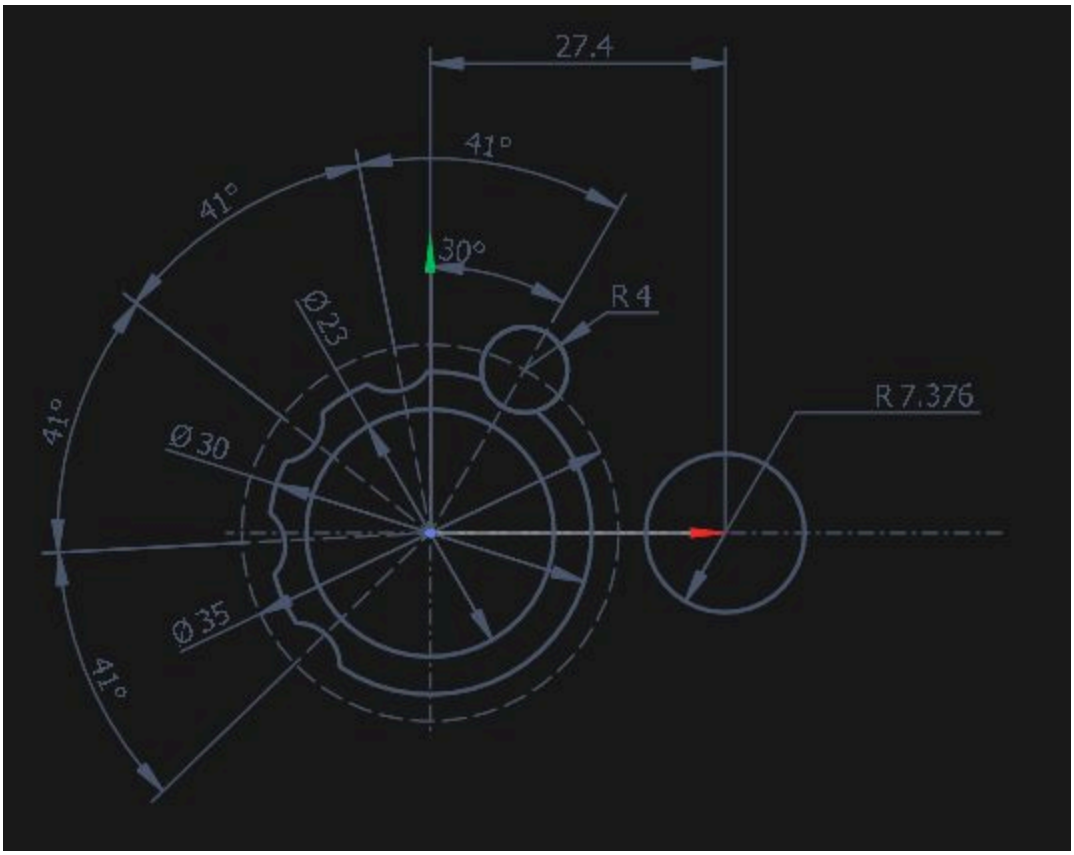
Trimming off the excess:



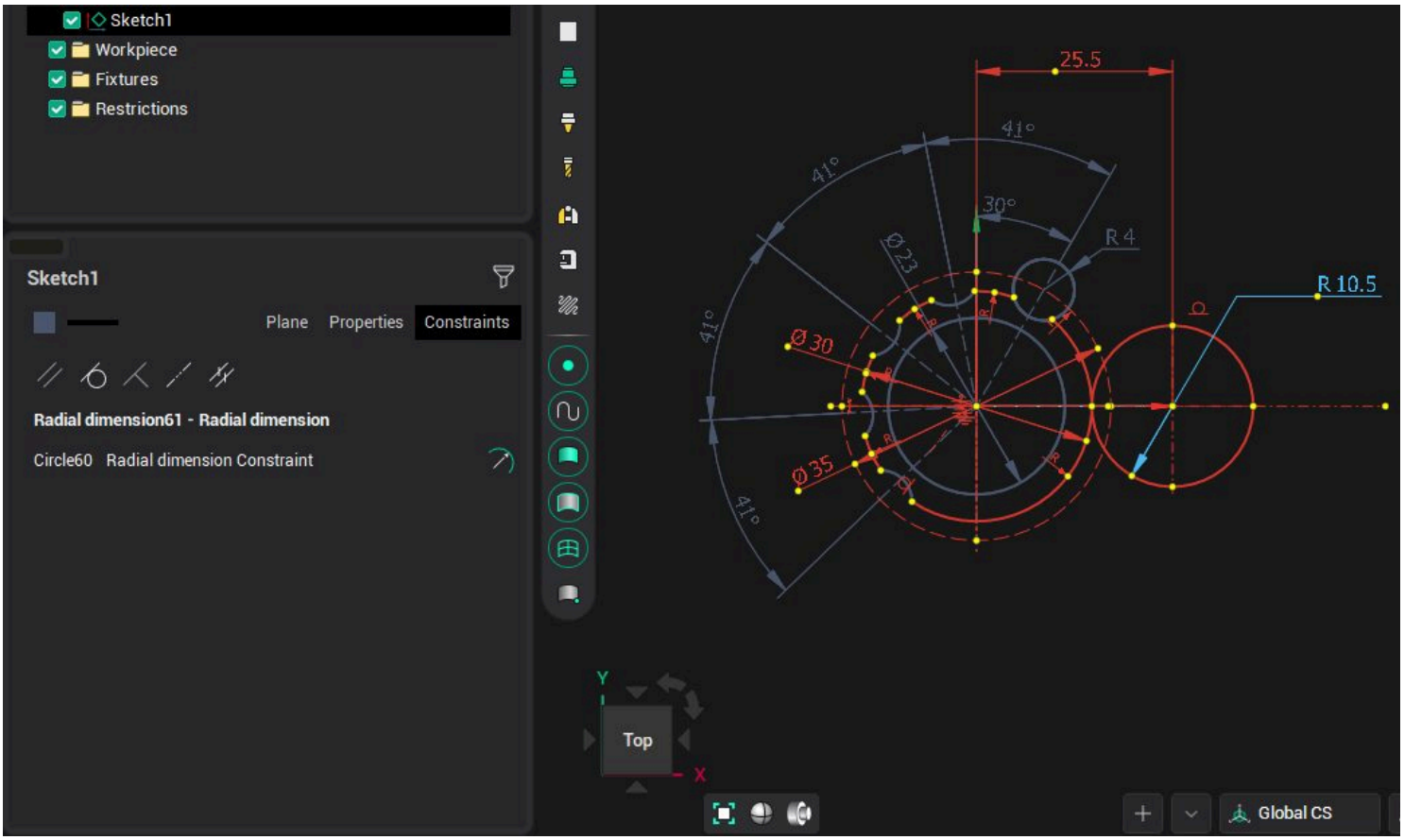
Now select the entire drawing with the key combination [Ctrl-A] and change the radius of the circle to 4, then change the angle to 30. After that, fix the angles of the segments and angular dimensions of 41°, after which we again select all the elements in the block and set the values of the diameters according to the drawing:



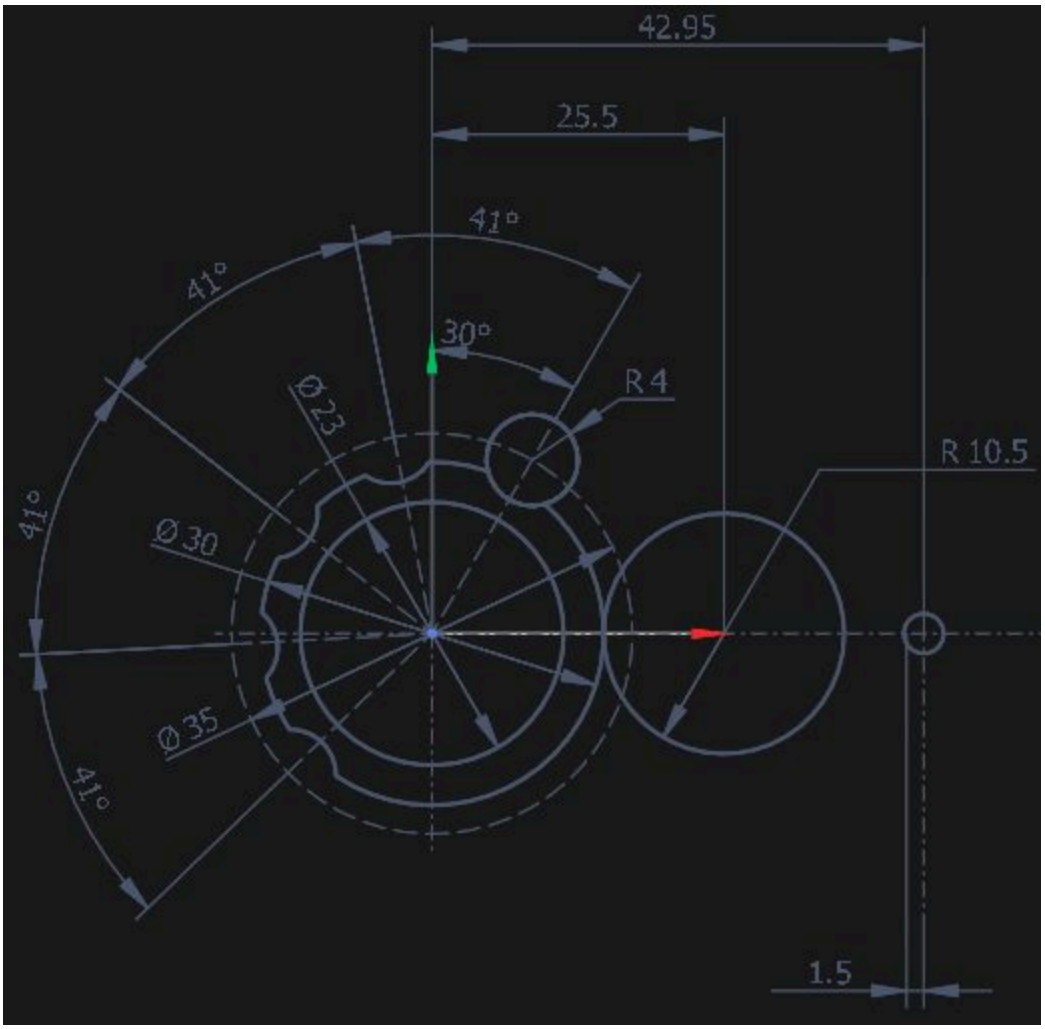
We fix the radii of the previously constructed circles and the resulting arcs. Next, we extend the circle axis to the right with a margin for further constructions. We build a circle with a center on the resulting line with a margin, then we cut off the excess and measure it:



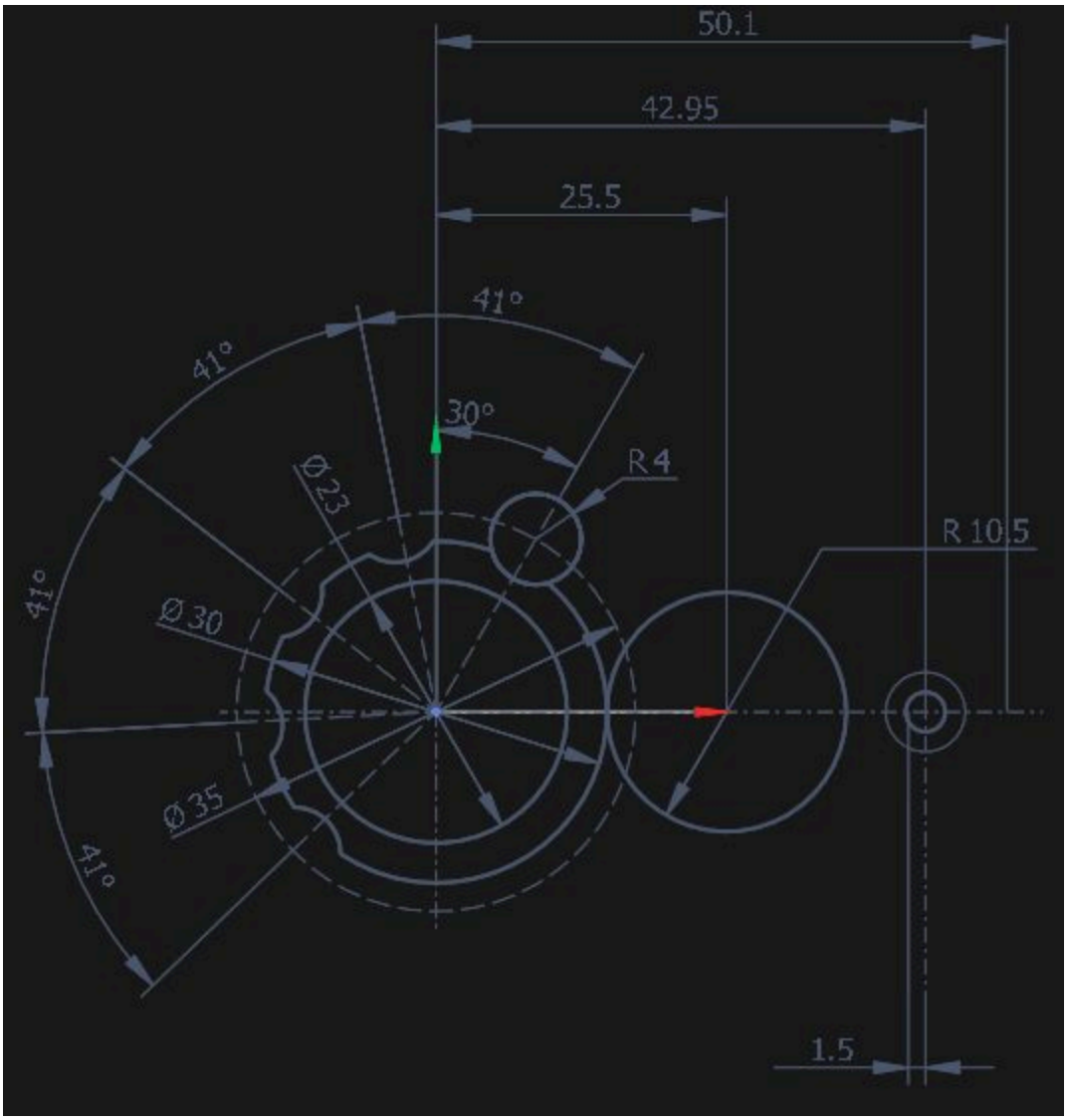
We fix the center of the arc Ø30, select a group of elements into a block, activate a new circle in the block. Next, in the inspector, in the Constraints tab, add the constraint Tangential to the circle and specify the part of the circle Ø30, set the radius according to the drawing and fix it:



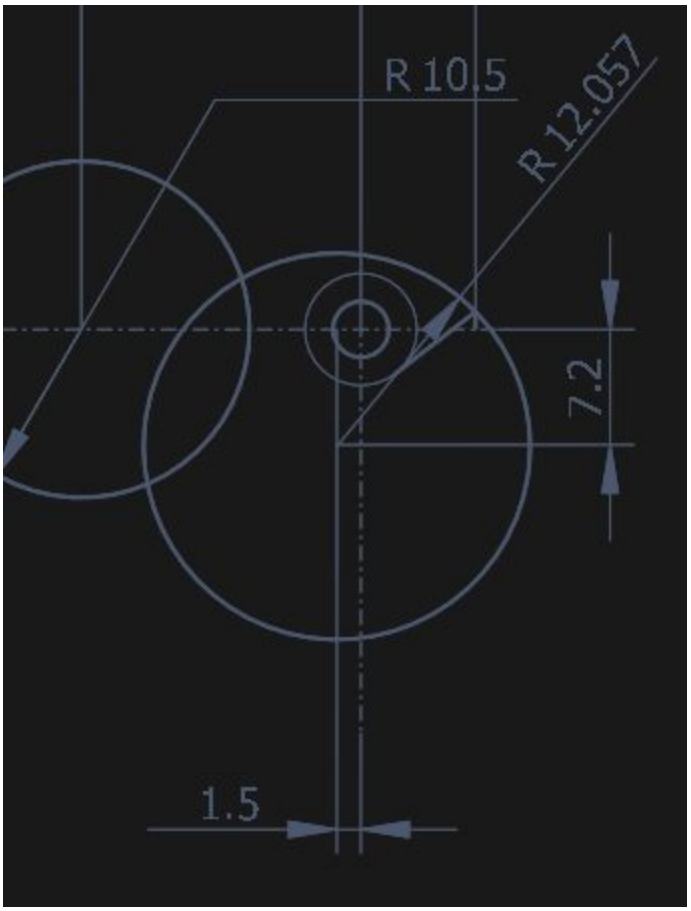
At the resulting circle, you can fix its center. Next, on the axis we build a small circle with a diameter of 3.5 and build a linear dimension from the zero point to its center. In the block, we expose according to the drawing and fix the center. We build a vertical axis for a circle of 3.5 and equidistant from this axis at a distance of 1.5 to build a point inside an arc with a diameter of 5.7 and fix its angle:



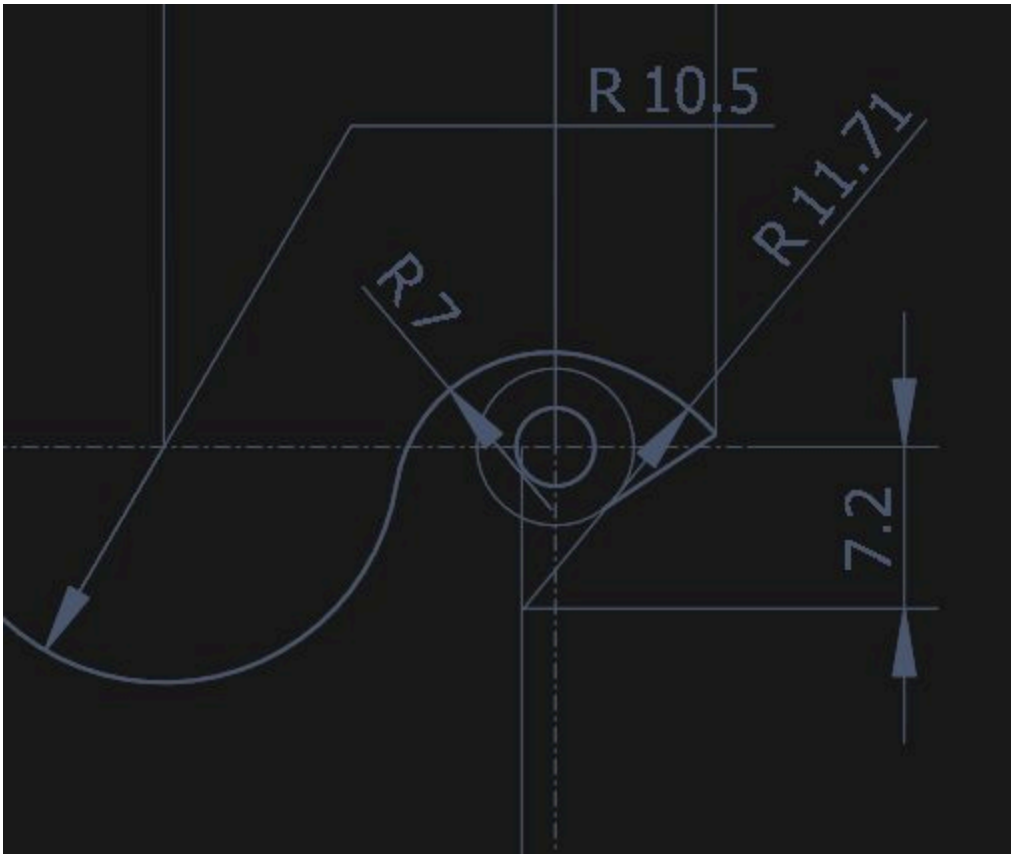
Now we build an auxiliary line for the size 50.1 from the zero point, adjust the size and build an auxiliary circle with a radius of 3.5:



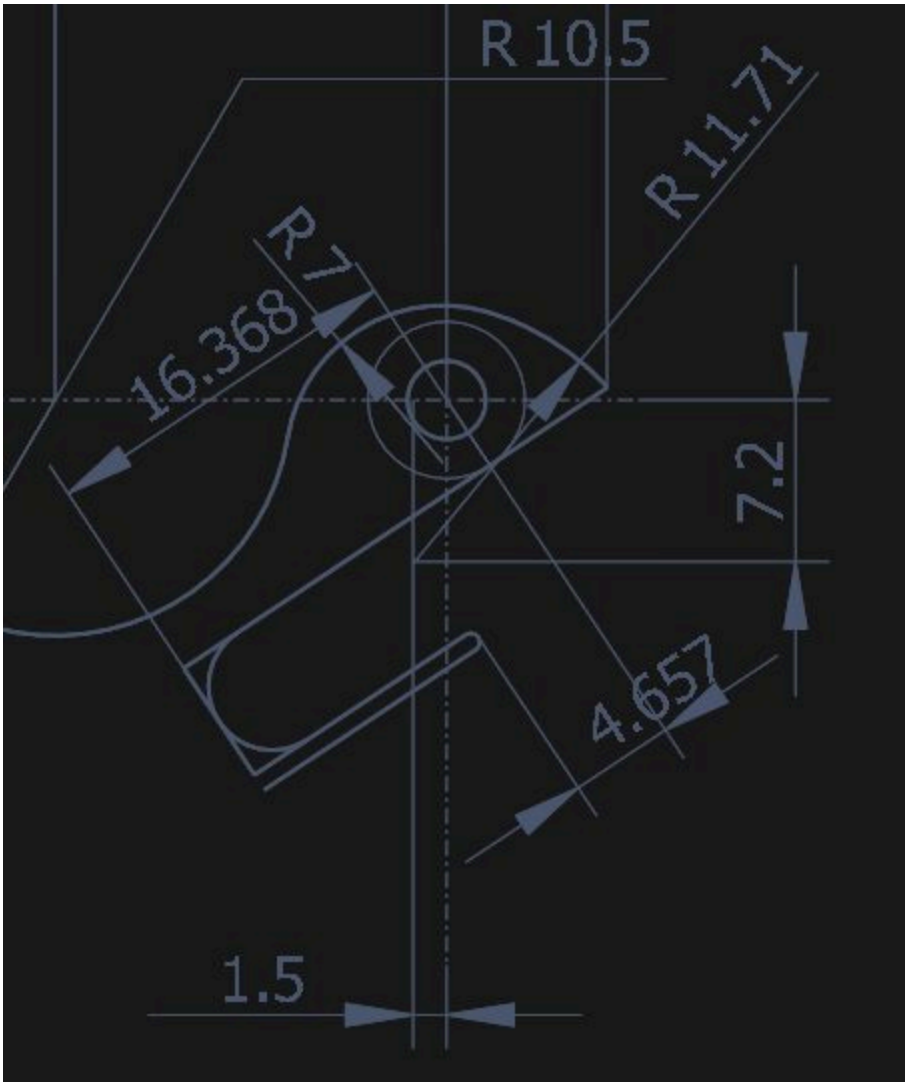
We build a vertical size and set it to 7.2 in the block, after fixing the top point. We build a line tangent to the auxiliary circle with a radius of 3.5, which would intersect the auxiliary line just above the axis. We cut off the auxiliary line and build a circle with the center at the resulting point and passing through the intersection point of the tangent and the vertical at 65.4. We measure the resulting circle:



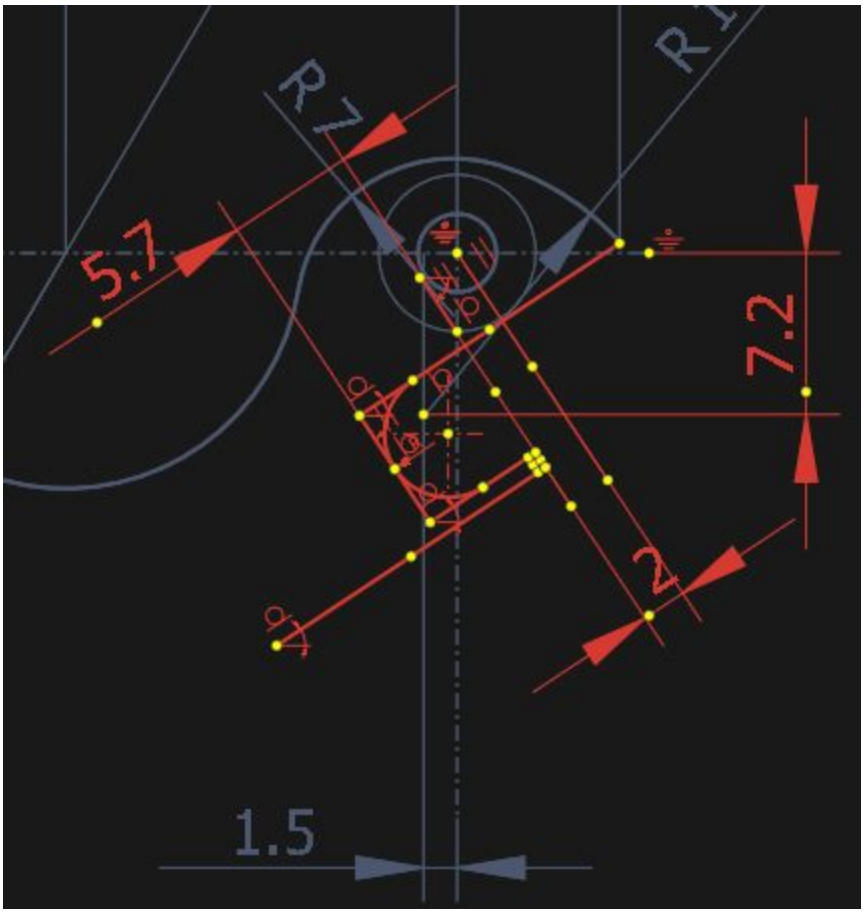
In the block, set the radius to 11.71, then build an arc with a radius of 7 tangent to the circles R10.5 and R11.71, cut off the excess.



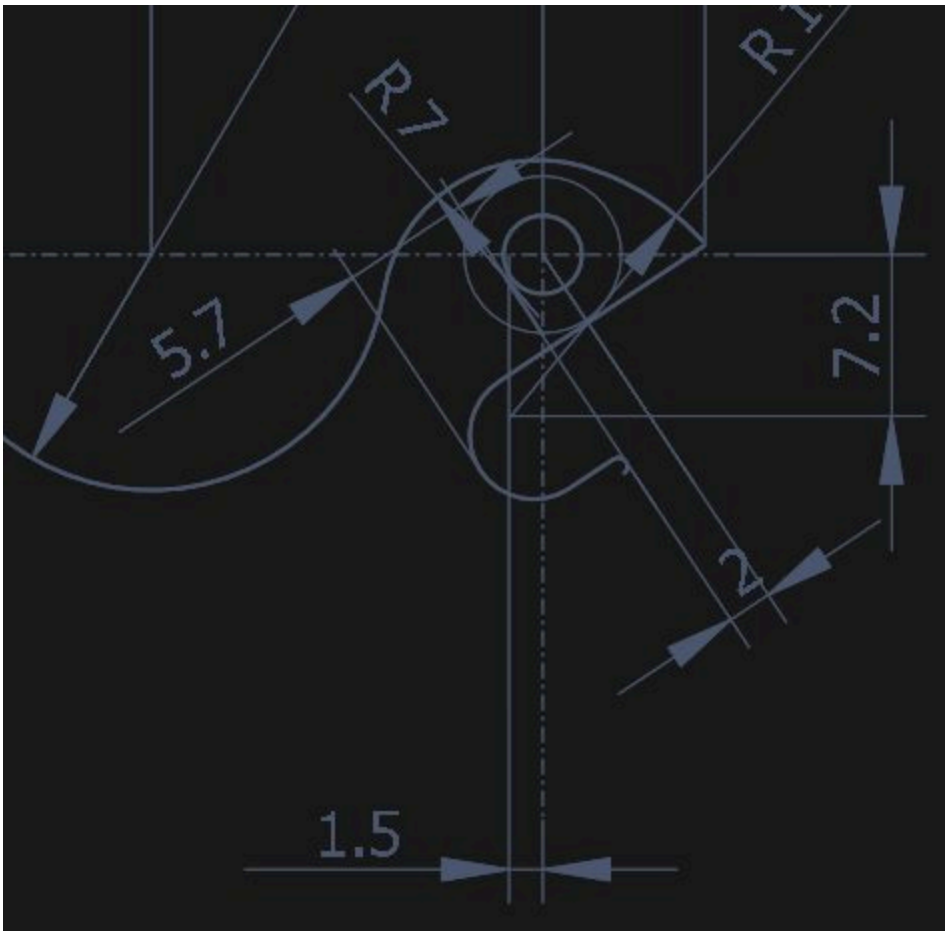
To build the arc R5.7 and R0.4, we extend the inclined line and build an equidistant line to it at a distance of 5.7, and from it another one at a distance of 0.8, connect the first two with a line and enter an arc between them. We build a time line passing through the center of a small circle and parallel to the third line, and from it equidistant and also enter a small arc. We measure the received elements, delete and cut off the excess.



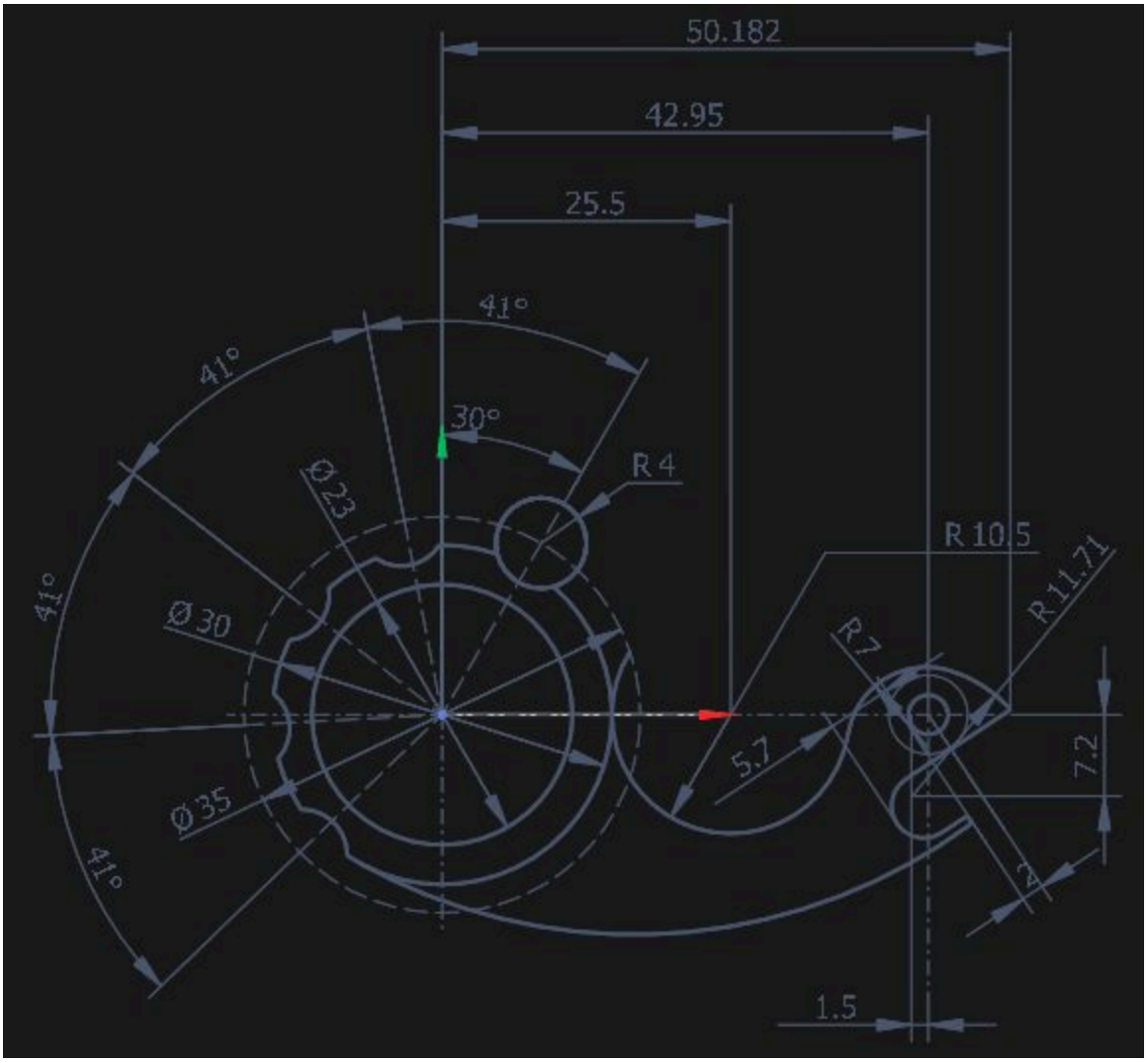
We fix the angles of the lines, and the points of the dimensions lying in the center of the circle and in the block are changed to the values from the drawing.



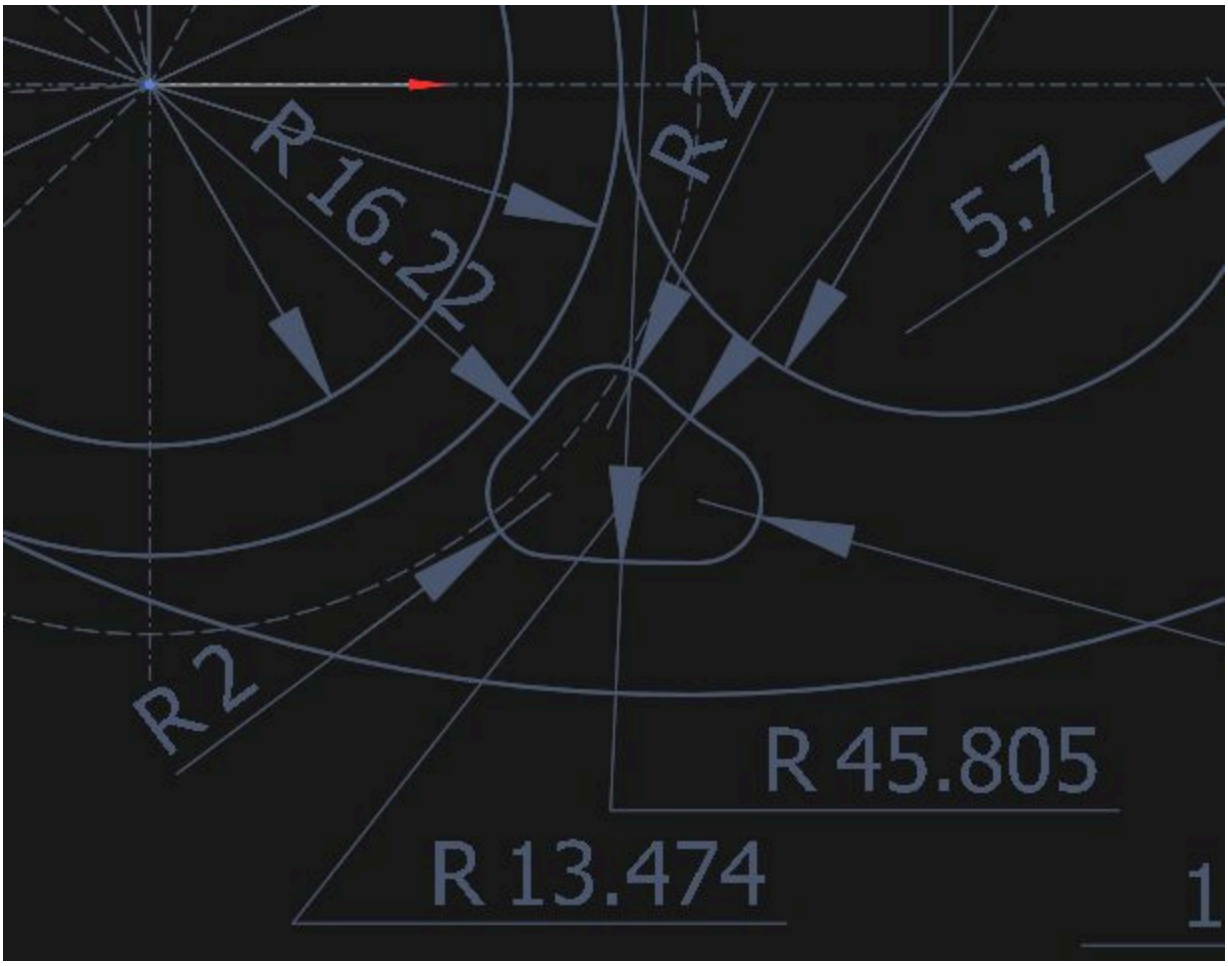
We cut off the excess or change the line type.



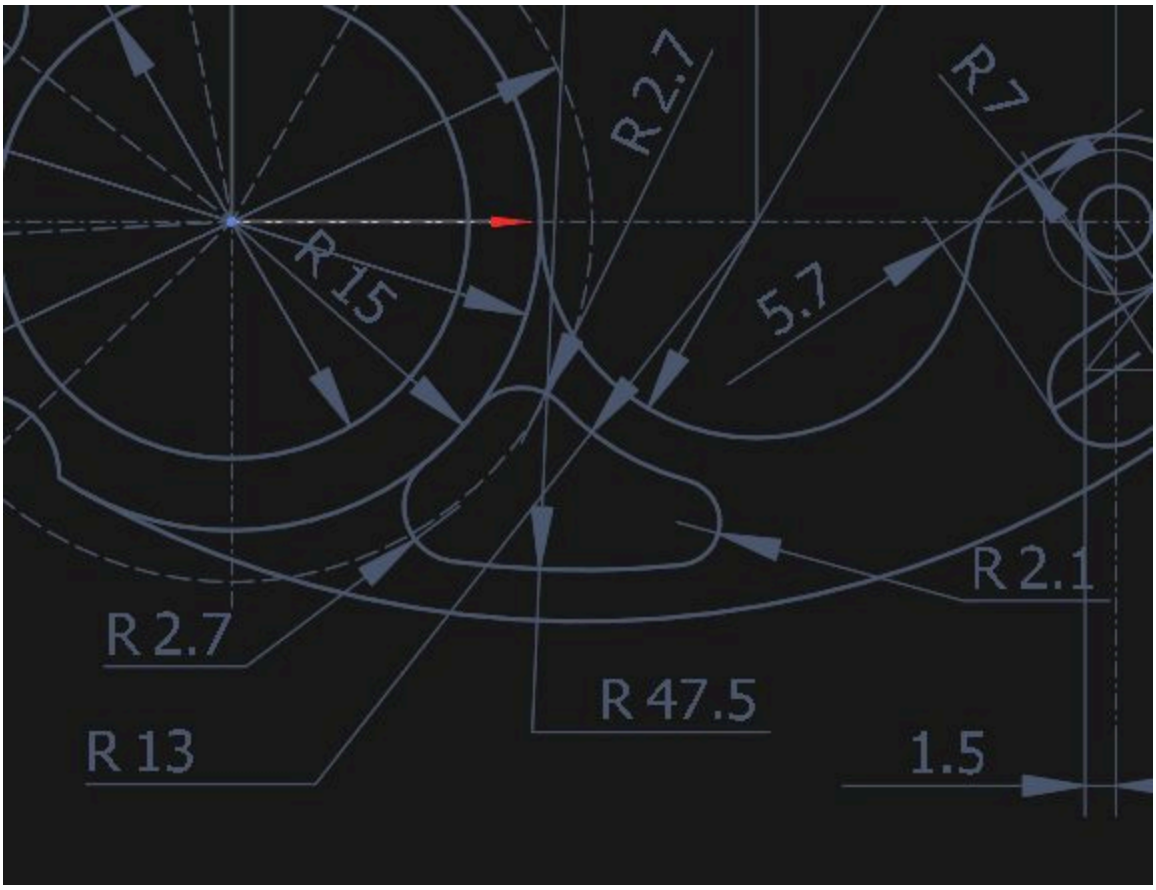
Conjugate arc R0.4 and left circle with R30 arc R50. We cut off the excess from the arc R0.4



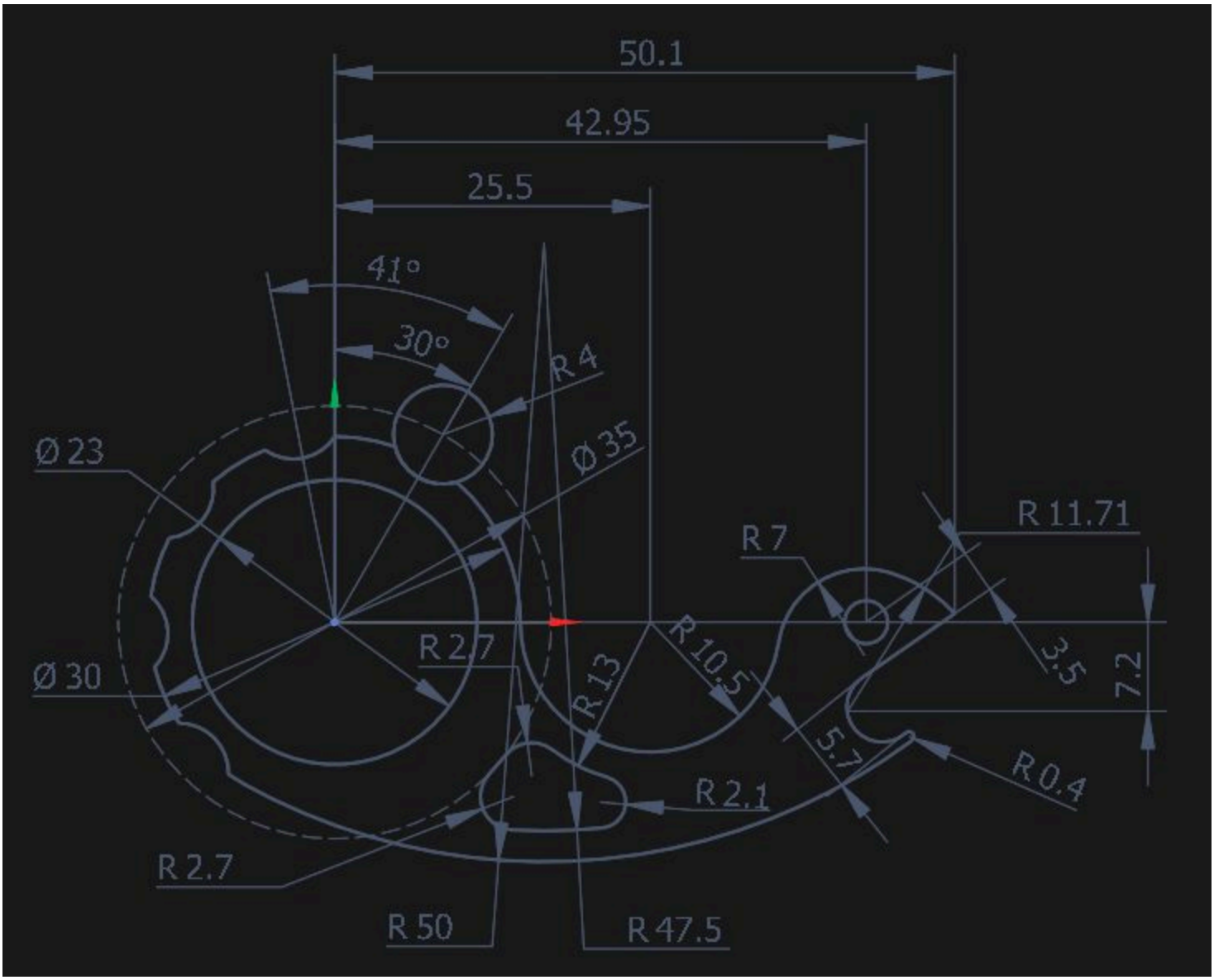
Now we build 3 arcs parallel to R50, R10.5 and R30, fix the centers and match them with arcs, set the dimensions and cut off the excess.



In the block, we set the dimensions according to the drawing. First, we set the radii for large arcs and fix the radius after each adjustment. Next, set the radius of rounding:

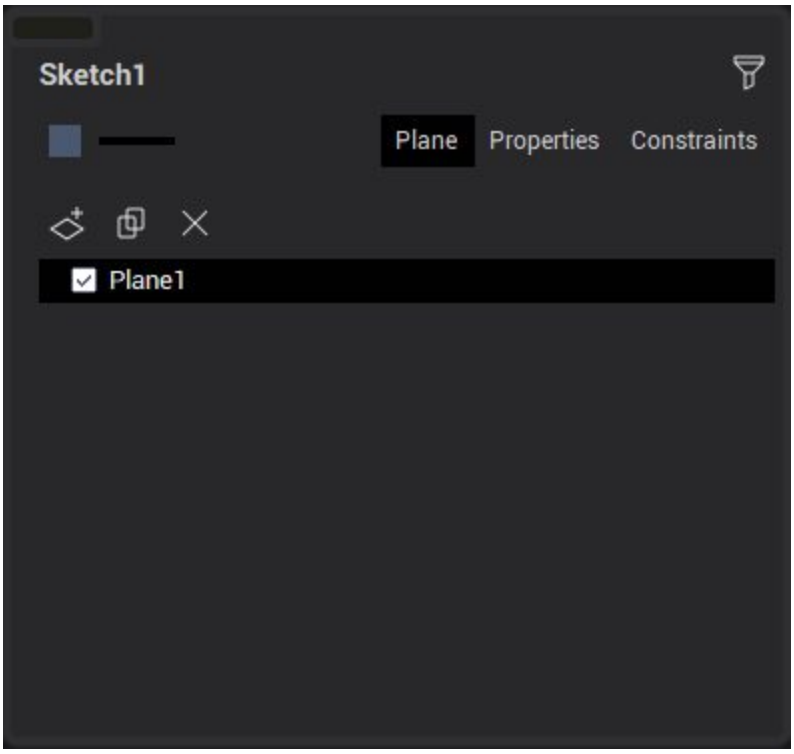


Thus, we got the required drawing.


50.1
42.95
25.5
41°
30°
R 4
Ø 23
Ø 35
R 11.71
R 7
R 27
R 13
R 10.5
3.5
7.2
Ø 30
5.7
R 0.4
R 2.1
R 2.7
R 50
R 47.5

Planes

Planes are controlled on the corresponding tab:

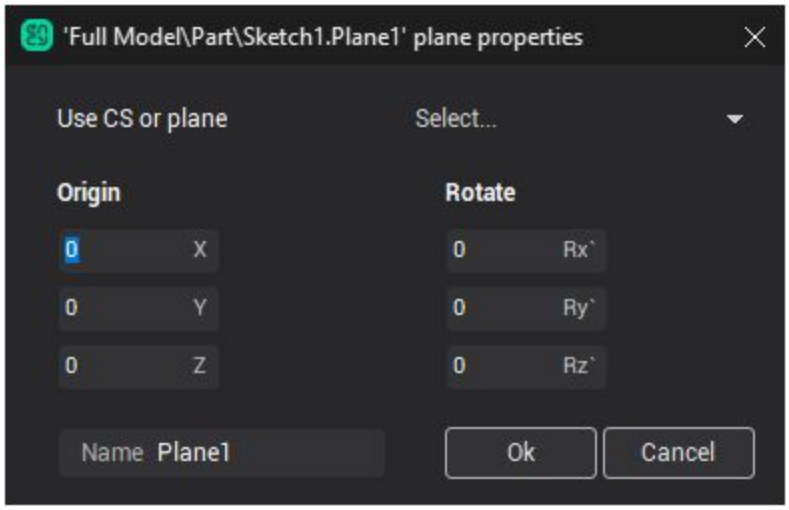


Here:

-  - making new plane,
-  - copying selected plane,
-  - deleting current plane.

To edit the coordinate system of the current plane, double-click on it or select an item in the context menu.

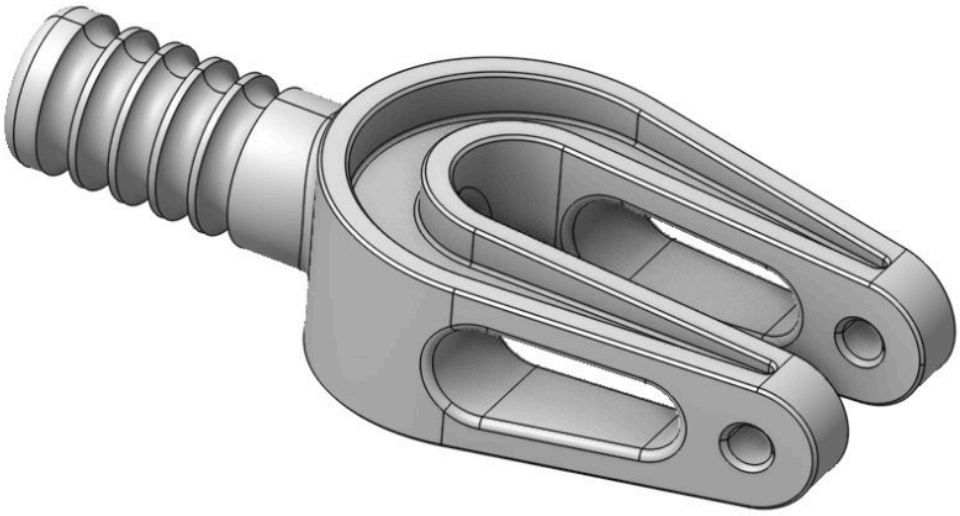
Plane making/editing window:



The plane is defined by specifying the origin of coordinates and rotating about the coordinate axes.

Further, all new objects will be created in the current plane. Snapping to objects in another plane is not performed.

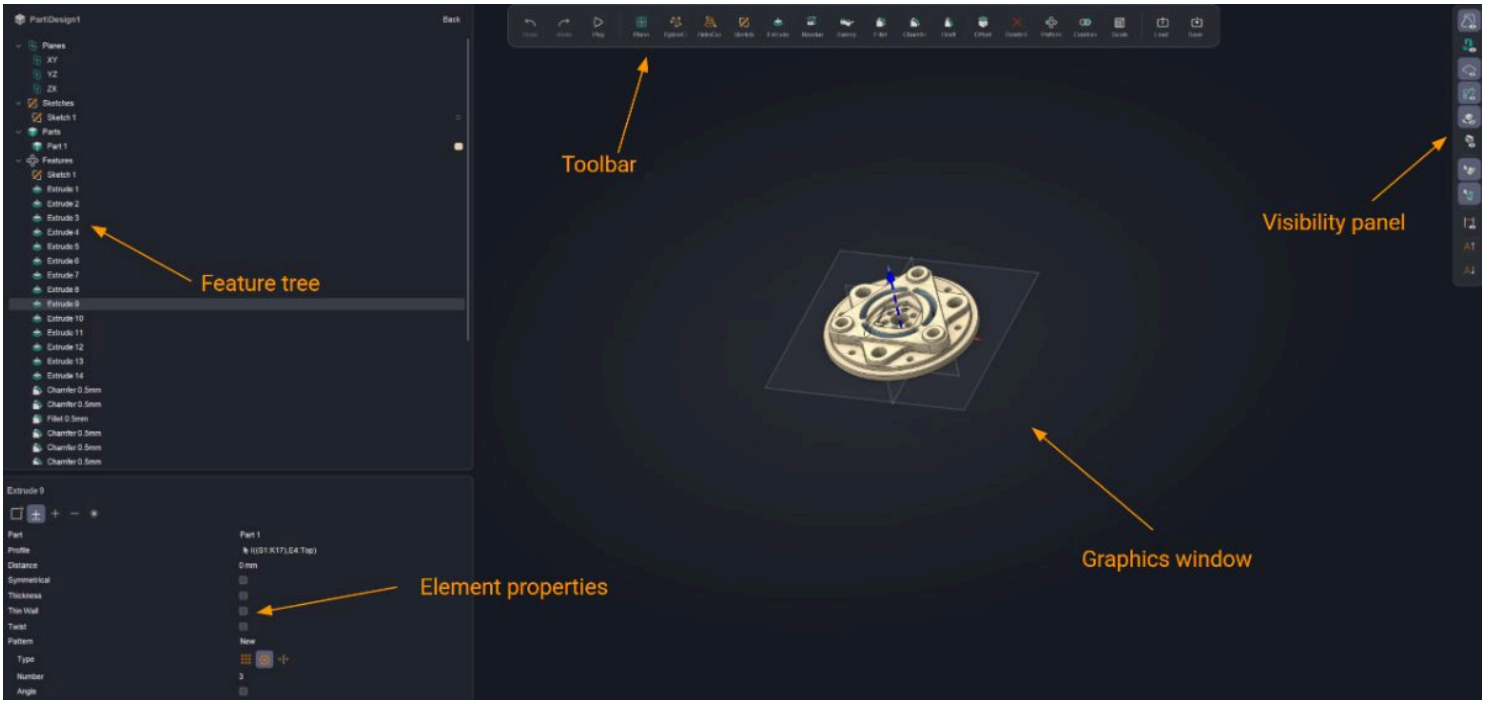
Design (3D Geometry)



Application area:

The integrated modeling module allows for the creation of both 2D drawings and 3D models based on them. Within the CAM system, this module is used to develop simple models intended for use as fixtures, blanks, or machined parts.

Main window



Application area:

The content of the main window's elements depends on the 2D or 3D working mode.

The main window consists of a feature tree, a toolbar, element properties, and a visibility panel.

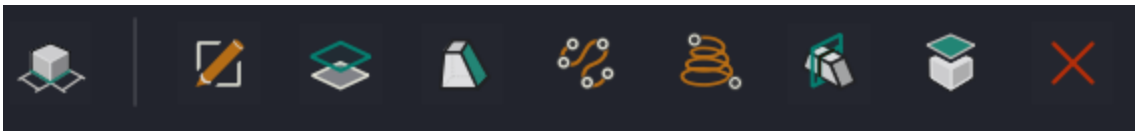
Feature tree.

The feature tree displays the chronological sequence of operations performed to create the 3D model. It is important to note that the feature tree is strictly sequential. Therefore, it is not possible to create two non-intersecting extrusions within a single body. The system will require the creation of a new body.

The tree consists of 3 base planes, sketches, parts and features.

Sketches. Defines a drawing in the construction tree

▼ Click here to expand..
Planes are created using a pop-up menu (by clicking the right or left mouse button on a surface or plane). Its content changes depending on the selected element.



Then you need to click on the drawing button.

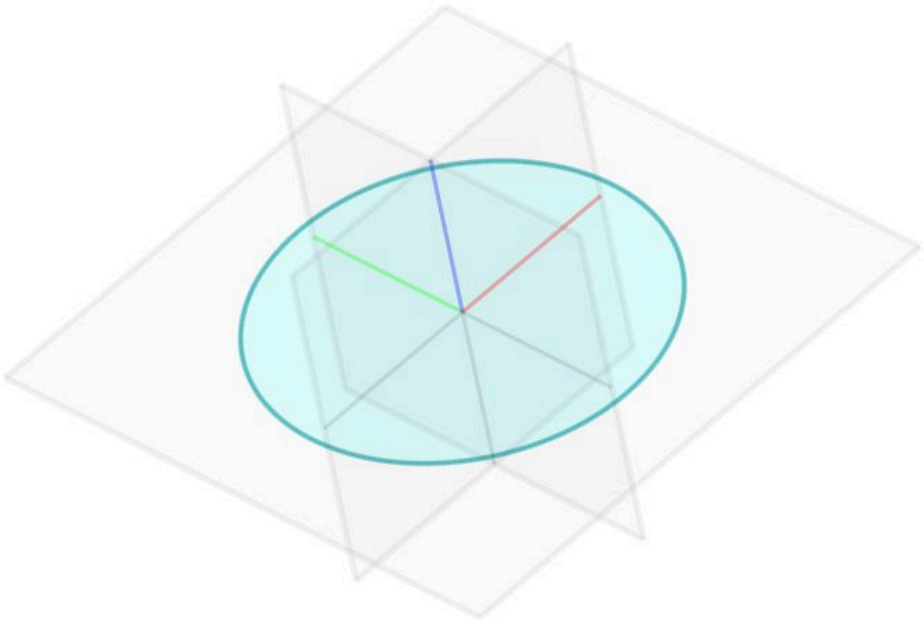


The methods for defining a plane are described in another chapter. Details

Next to each sketch in the tree, there is a visibility toggle that allows you to show or hide it in 3D mode.



If the contour is closed, the system highlights it, indicating that a boolean operation can be performed.

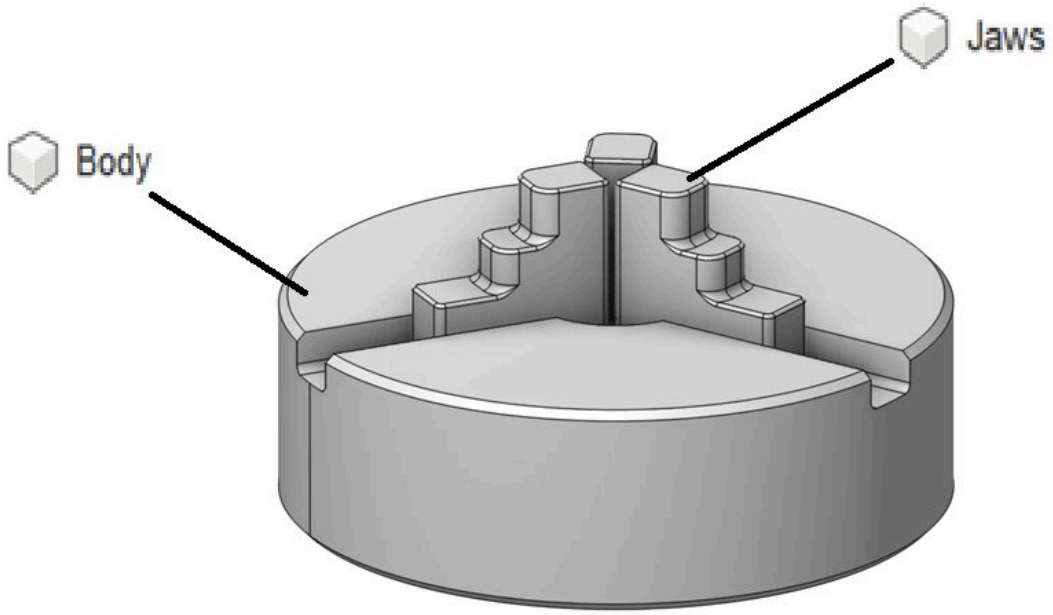


If the contour is not highlighted for extrusion, check the sketch visibility and ensure that the contour is closed.
Curves. They represent spatial curves (Helix, Spline). Details

Parts. The Design can consist of multiple parts (one by default). This allows for the creation of assemblies from individual parts. This is necessary for creating assemblies (for example, a jaw chuck

consists of jaws and a base). Details

▼ Click here to expand..



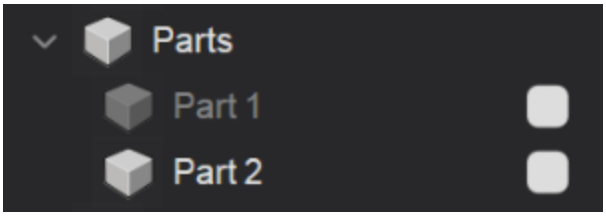
The Create Part button is located in the Element Properties panel when performing the Extrude command.



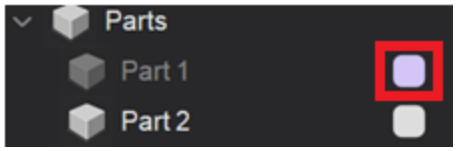
To have two parts appear in the tree, you need to create at least two sketches and perform the Extrude operation on each.



The visibility of a part is controlled by clicking its icon.



You can also change the color of the model.





Features. It contains various elements (sketches, extrudes, etc.).

Toolbar.

This is the toolbar for 2D (Details) or 3D (Details) mode. The toolbar's contents change depending on the selected mode.

Element properties.

This panel allows you to change the properties of Features.

Visibility panel.

This panel allows you to control the visibility of various elements (such as the machine, planes, sketches, etc.).

▼ Click here to expand..

Show 3D model. Determines the visibility of 3D models not involved in the model design.

Show machine. Determines the visibility of the equipment model

Planes. Determines the visibility of all planes

Sketches. Determines the visibility of all sketches

Ground plane. Determines the visibility of the floor

Section plane. Enables (or disables) a special mode for displaying the internal structure of an object. Details

Edges. Allows selecting model edges

Features. Allows selecting model features

Dimensions. Determines the visibility of all dimensions.

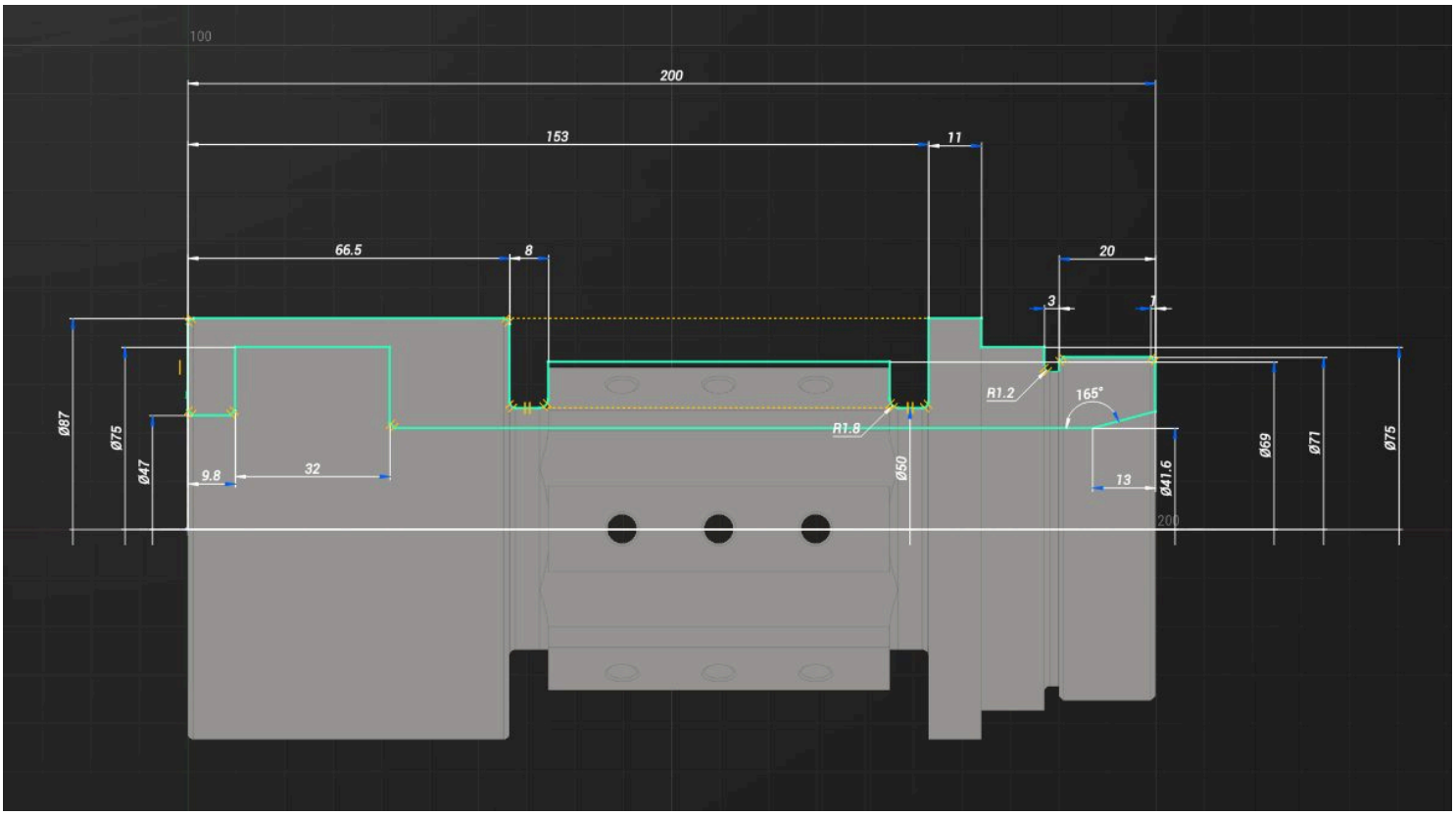
Increase PMI Font. Increases the dimension size

Decrease PMI Font. Decrease the dimension size

Graphics window.

This is the main window where the drawing, model, or assembly is displayed.

Design in 2D mode



Application area:

This mode allows you to create drawings.

Defining planes for drawing creation was covered in the next chapter. Details

Toolbar/Element properties.

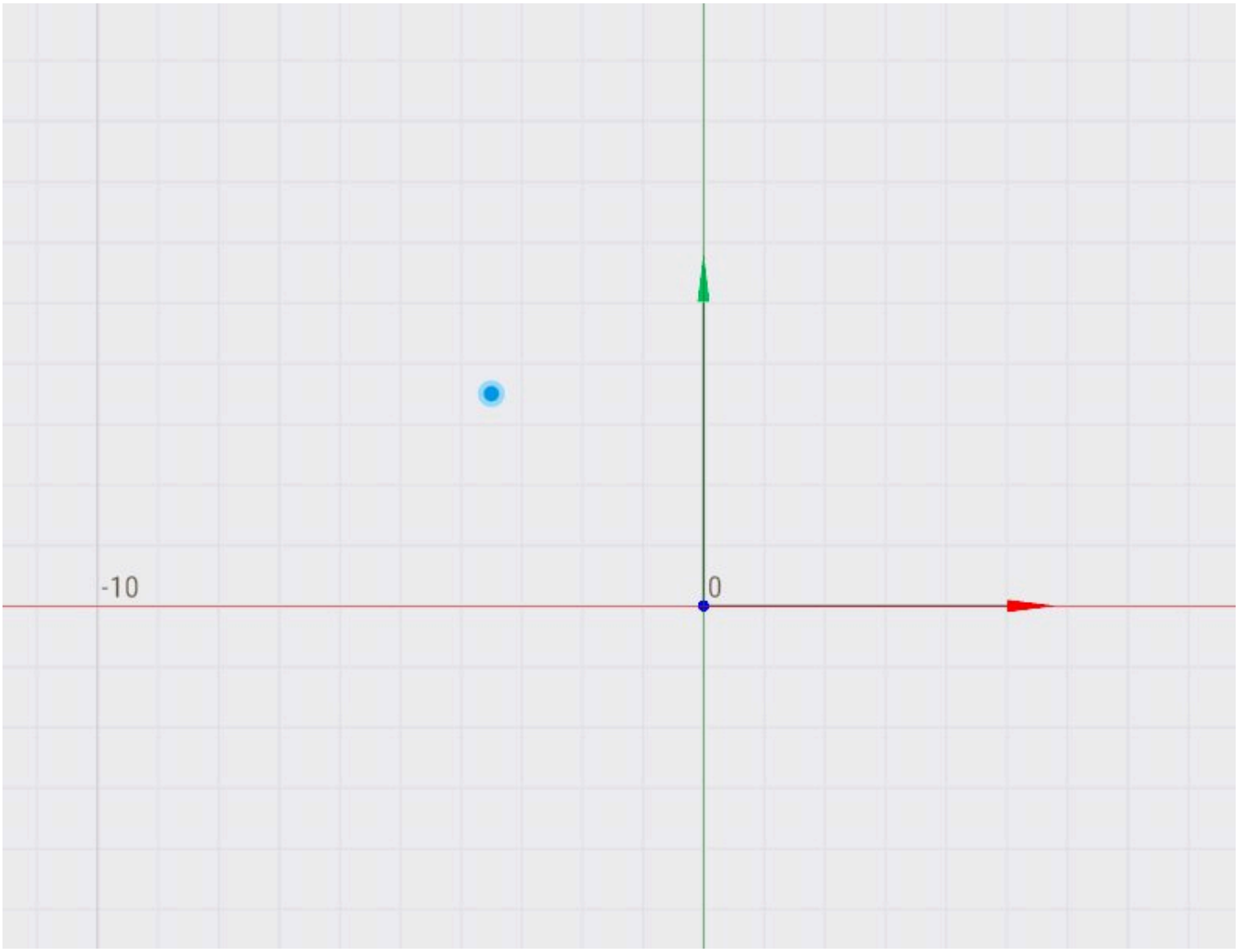
▼ Click here to expand..

Undo. Reverses the last action.

Redo. Reapplies the last undone action.

Sketch. Allows you to exit a drawing or create a drawing

Point. Add points to the sketch



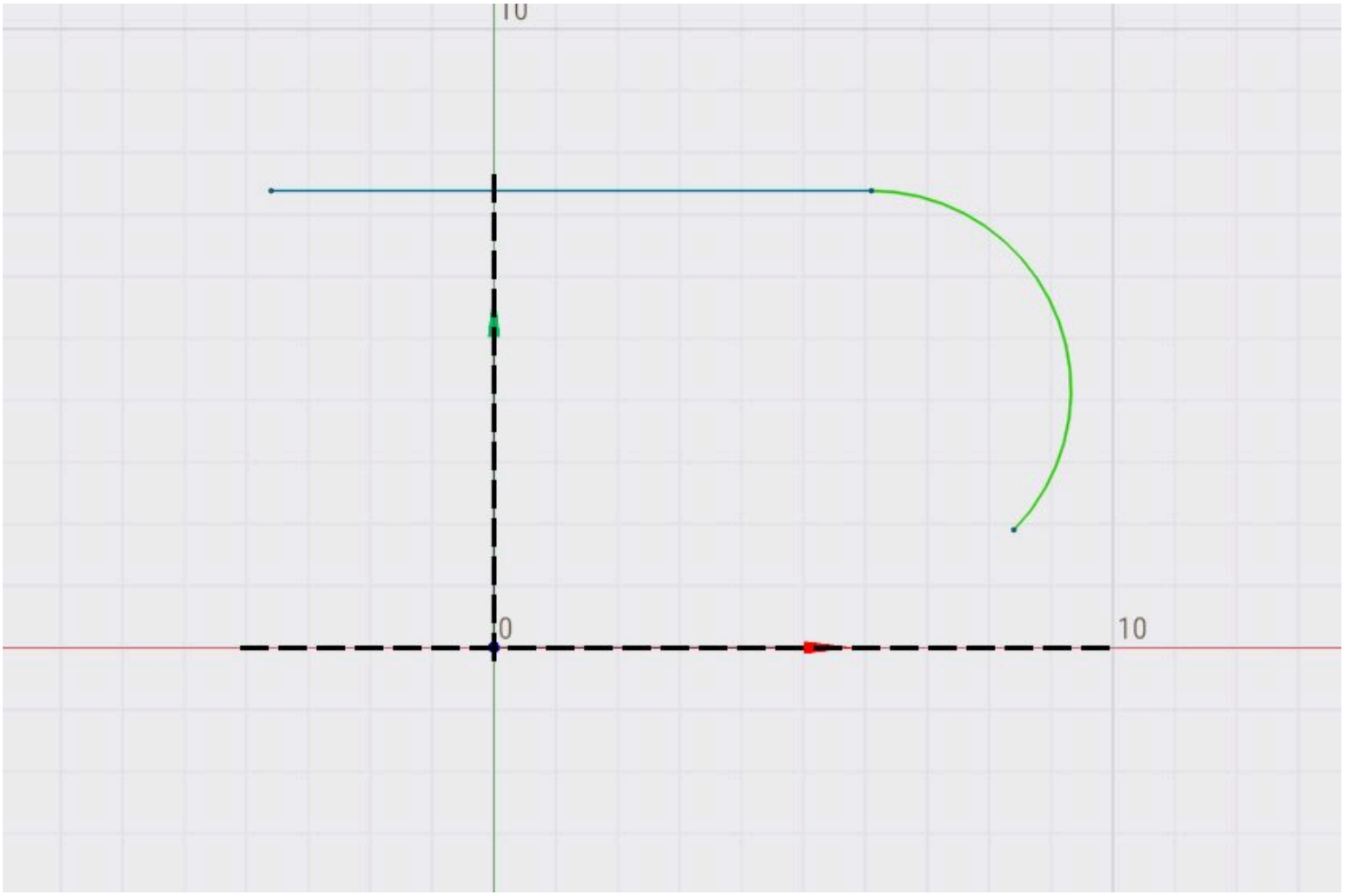
Coordinates: Enter the coordinates of the point separated with [space].

Split Contour. Allows you to split a curve into two

Hot keys:

RMB: Exit.

Contour. Draw a contour with lines and arcs Click to add the next line or tangent arc.



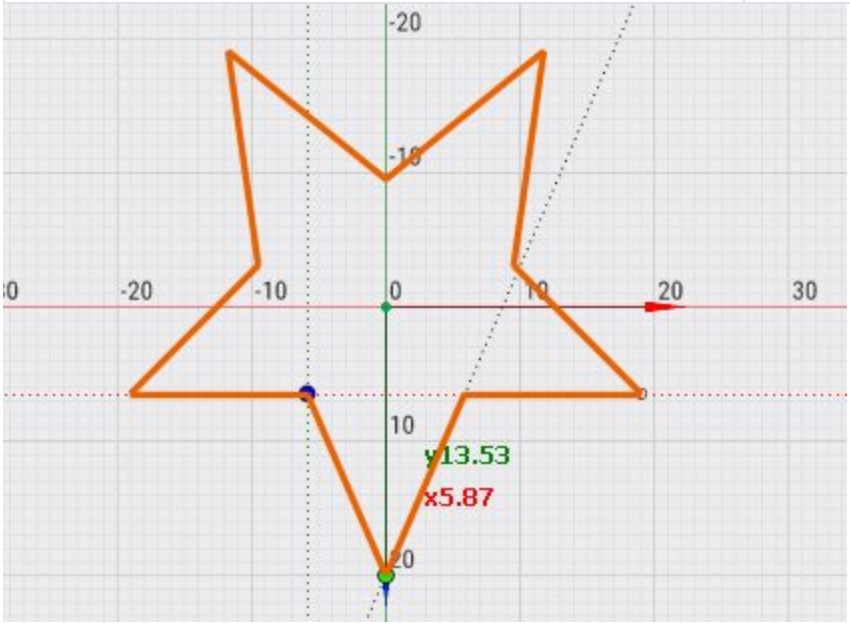
Coordinates: Enter the coordinates of the point separated with [space].

▼ Click here to expand..
You can use a star as an example.

1.You need to copy this table to the clipboard 2. Enter the data in the appropriate field 3.Press enter

X	Y
0	20
5,87	6,47
19,02	6,47
9,51	-3,09
11,75	-19,02
0	-9,51
-11,75	-19,02

X	Y
-9,51	-3,09
-19,02	6,47
-5,87	6,47



Hot keys:

Space: Hit Space to switch between drawing lines and arcs.

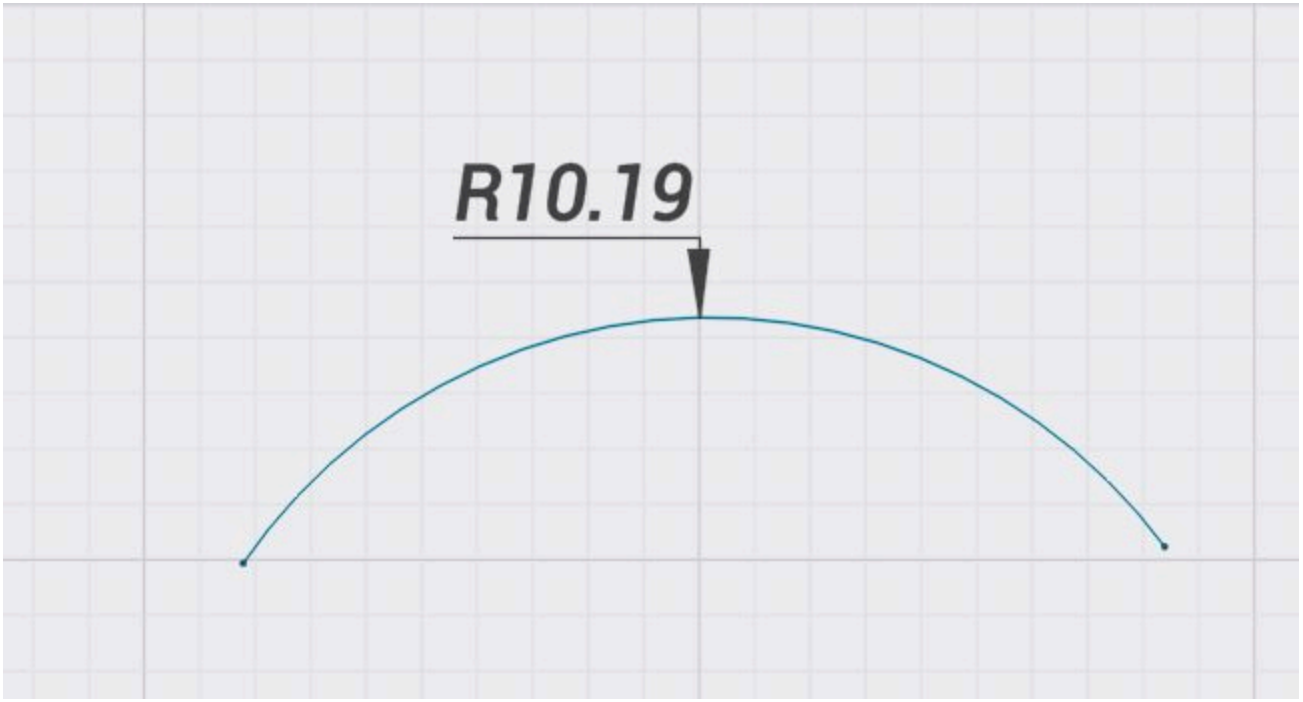
Shift: Hold Shift and click on an arc or a circle to make the line tangent to it.

Trim a contour Hold and drag the left mouse button to trim out a piece of a contour.

RMB: Exit.

Arc by two points. Draw an arc connecting two points.

1. Click the first point of the arc.
2. Click the second point of the arc.
3. Either click the third point on the arc or enter the radius of the arc in the Radius field.



Radius. Predict the radius between two points.

Start point. Allows you to specify point by coordinates.

End point. Allows you to specify point by coordinates.

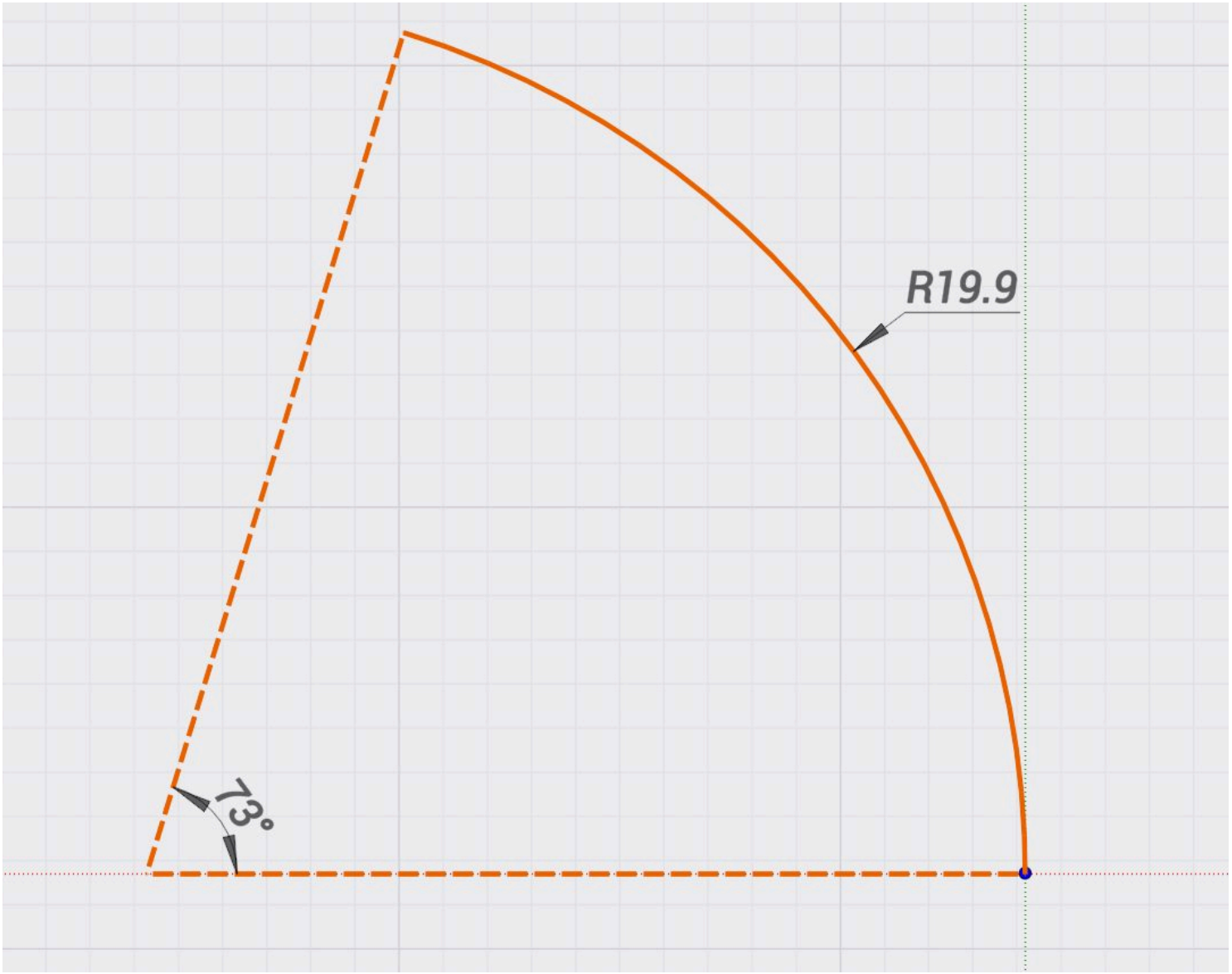
Hot keys:

Shift: Hold Shift and click on an arc or a line to make the arc tangent to it.

RMB: Exit.

Arc from center. Draw an arc starting with the center point

1. Click the center point of the arc. 2. Click the first point of the arc. 3. Click the second point of the arc.

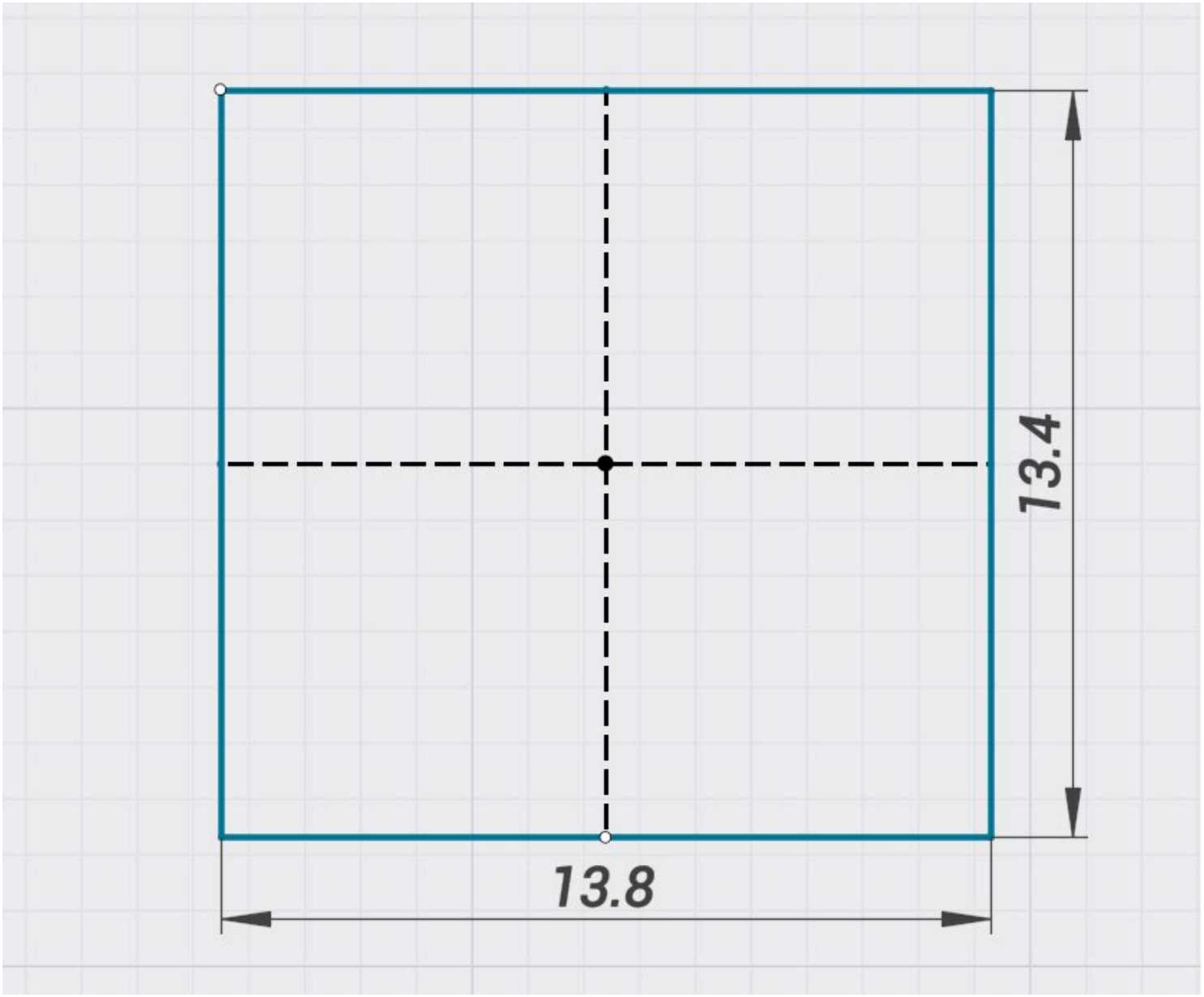


Hot keys:

RMB: Exit.

Rectangle by 2 points. Draw a rectangle between two diagonal points

1. Click the first corner point of the rectangle. 2. Either Click the second corner point or enter the dimensions of the rectangle in the field above.



Dimension. Enter only one dimension to make it a square. (Example 67.5 67)

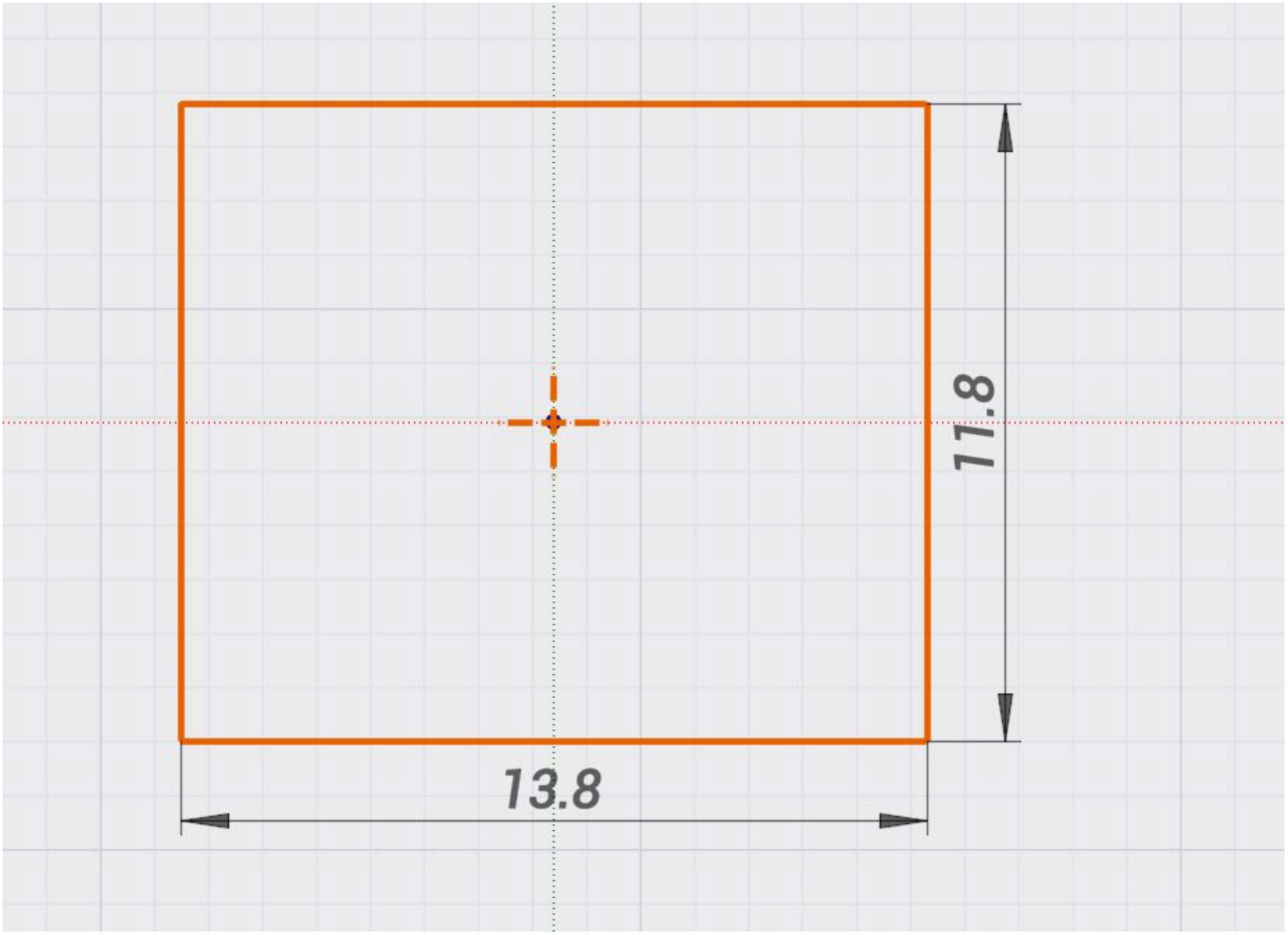
Start point. Allows you to specify point by coordinates.

Hot keys:

RMB: Exit.

Rectangle from center. Draw a rectangle from the center point.

1. Click the center point of the rectangle. 2. Either click the corner point or enter the dimensions of the rectangle in the field above.



Dimension. Enter only one dimension to make it a square. (Example 67.5 67)

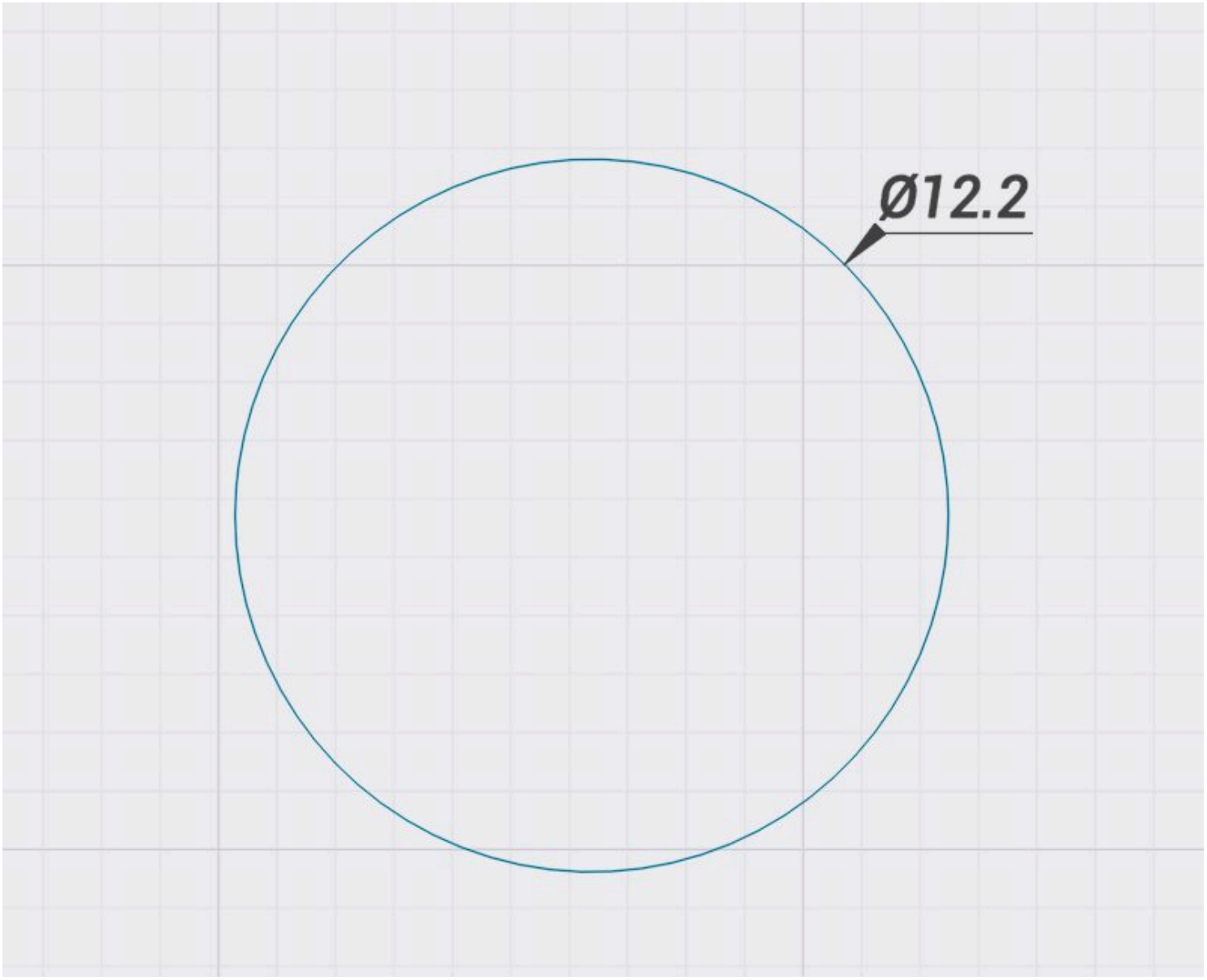
Center point. Allows you to specify point by coordinates.

Hot keys:

RMB: Exit.

Circle. Draw a circle starting from the center

1. Click the center point of the circle.
2. Click a point on the circle, or enter the size of the circle in the field above.



Diameter. Determines the diameter of a circle.

Center point. Allows you to specify point by coordinates.

Hot keys:

Hold Shift and click on a segment to make the circle tangent to the segment.

Hold Ctrl to create multiple circles with the same diameter.

Draw a circle tangent to another geometry

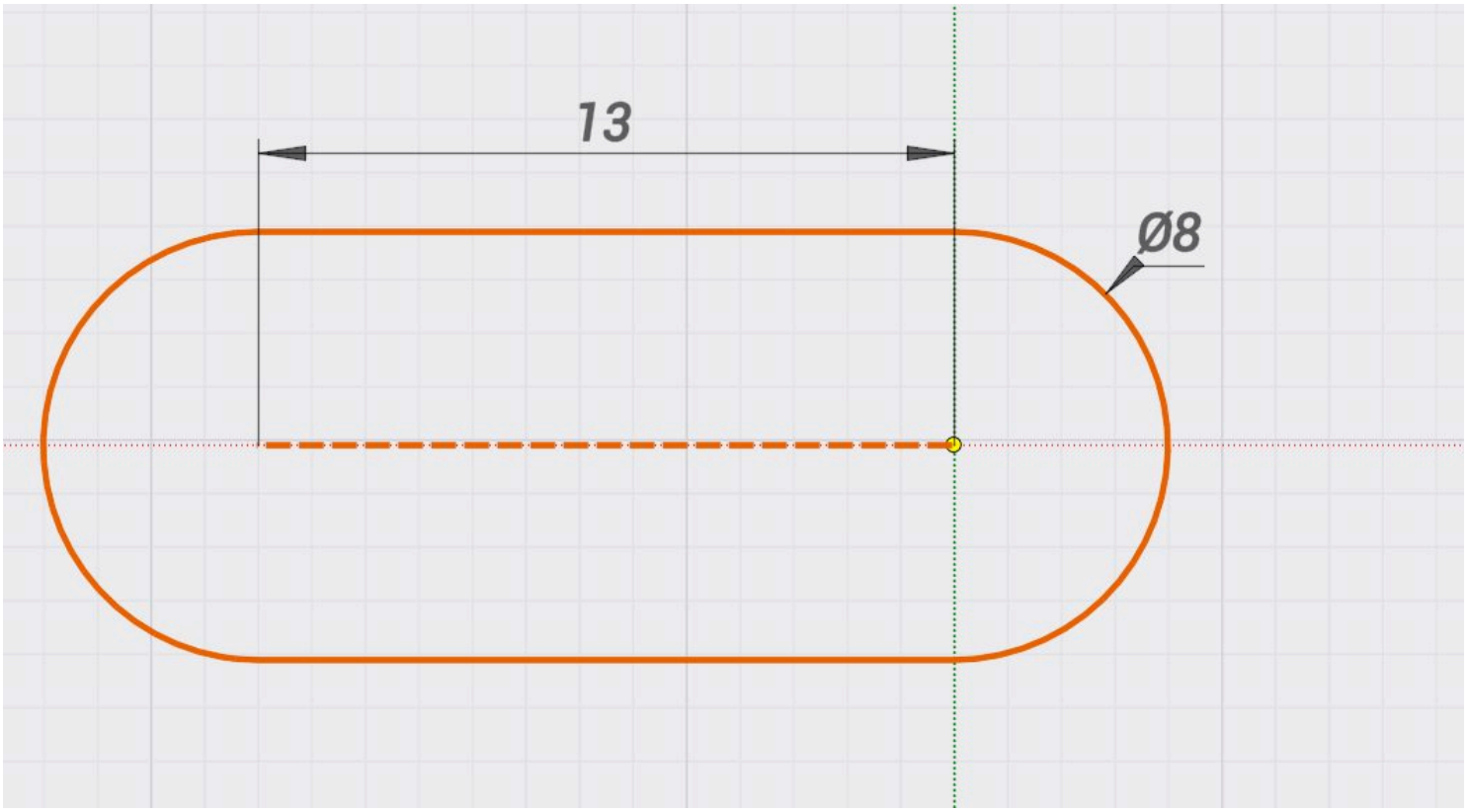
Hold Shift and click on lines, arcs, circles or points to draw a circle tangent to those entities.

Enter the size of the circle in the field above when needed.

RMB: Exit.

Slot linear. Draw a straight slot between two points

1. Click the first point of the slot.
2. Click the second point of the slot.
3. Click the periferal point of the slot to define the slot width.

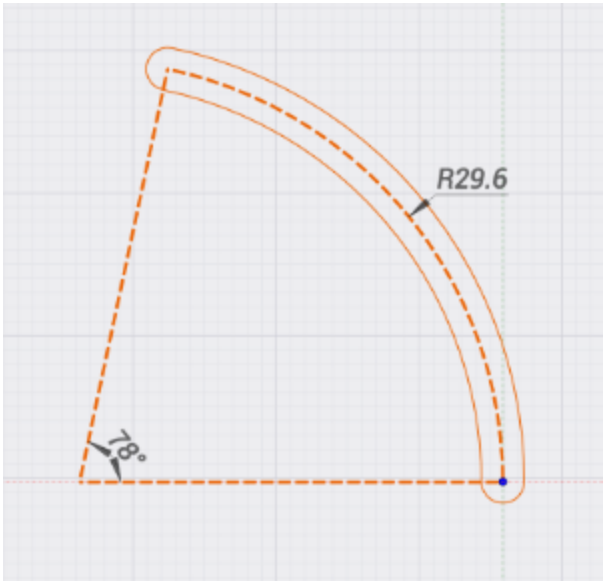


Hot keys:

RMB: Exit.

Slot circular. Draw a circular slot from the center point

1. Click the center point of the slot arc.
2. Click the first point of the slot arc.
3. Click the second point of the slot arc.



Hot keys:

RMB: Exit.

Offset. Allows you to set the offset of the selected element.

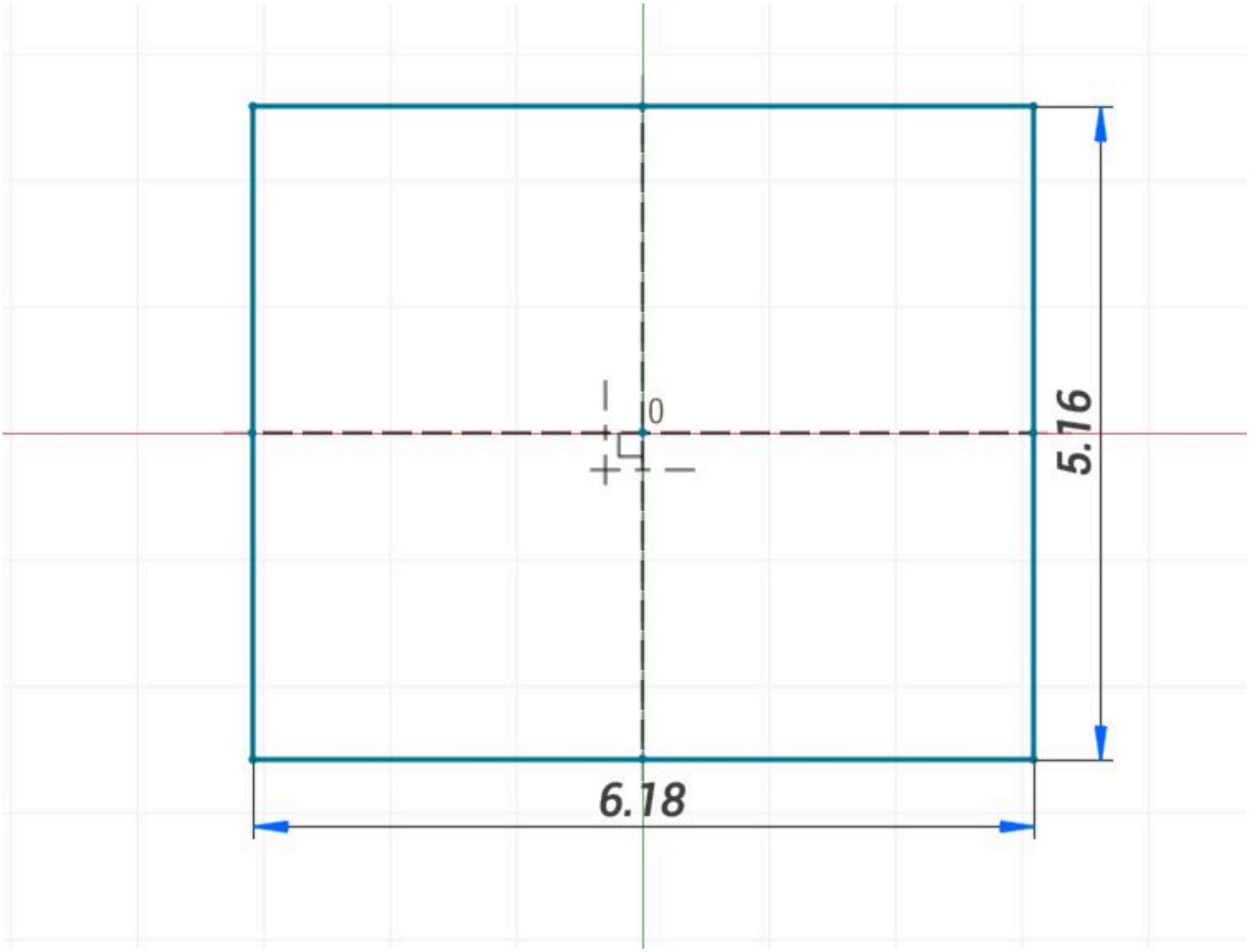
Left. Allows you to set an offset to the left of the contour

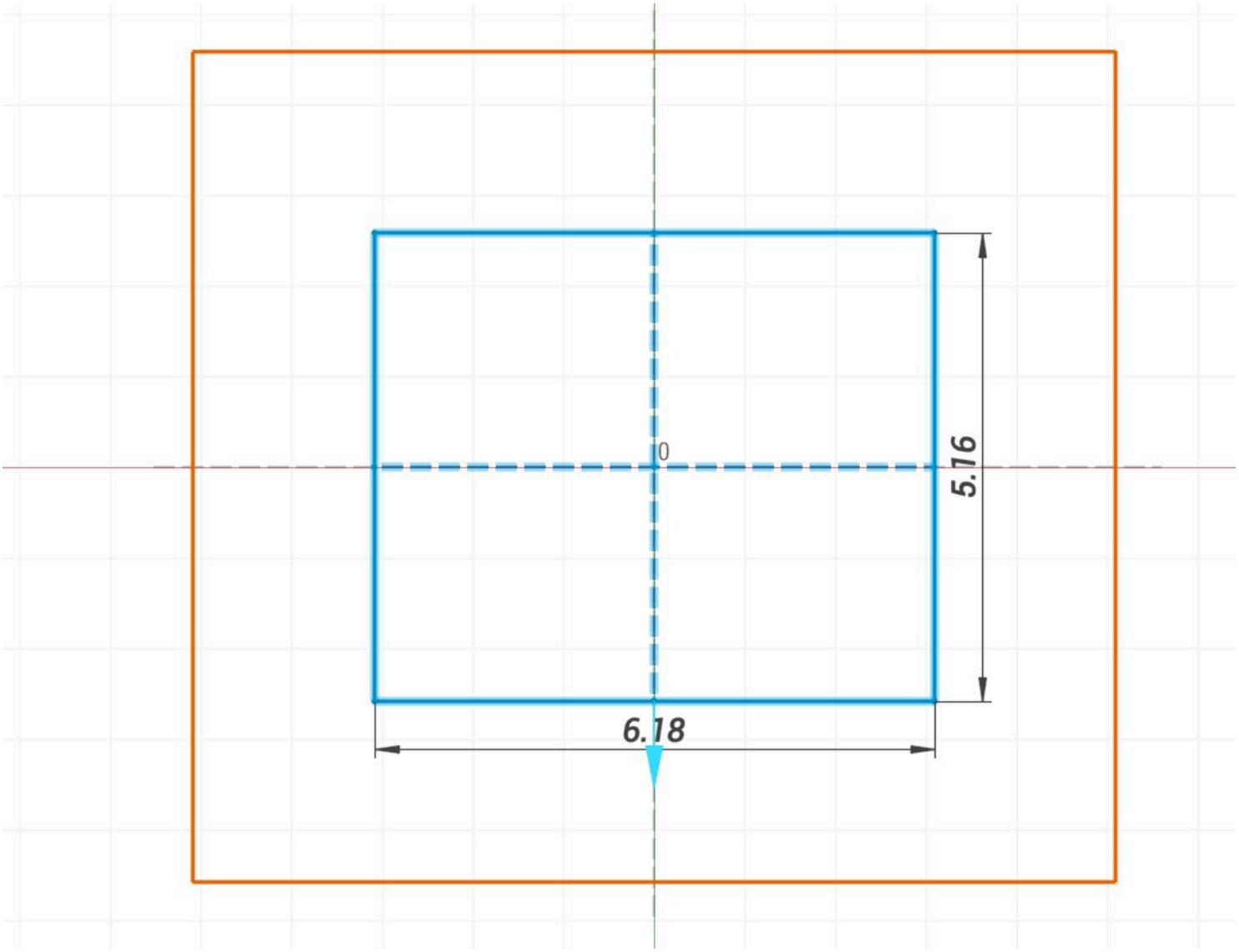
Right. Allows you to set an offset to the right of the contour

Both sides. Allows you to set an offset to the left and right of the contour

Distance. The distance by which the contour must be shifted

Roll Corners. Allows you to round off the corners of an offset contour


0
5.16
6.18



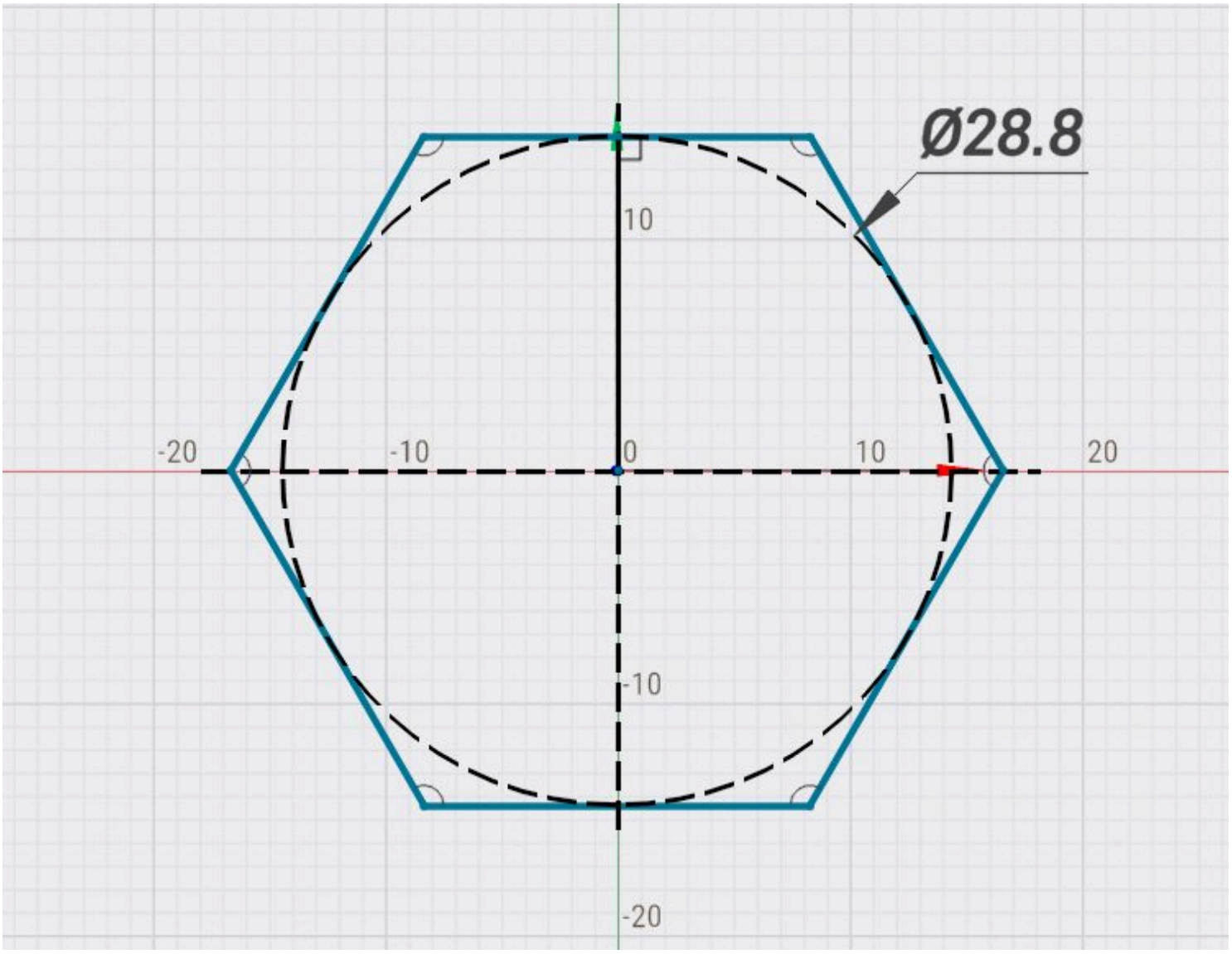
Hot keys:

Select the contour and press Ctrl to drag in the desired direction.

RMB: Exit.

Polygon. Draw a polygon starting with the center point

1. Specify the number of sides of the polygon in the field above.
2. Click the center point of the polygon.
3. Click the peripheral point of the polygon.



Second point on the side. The second point is located on the side

Second point in the corner. The second point is located in the corner

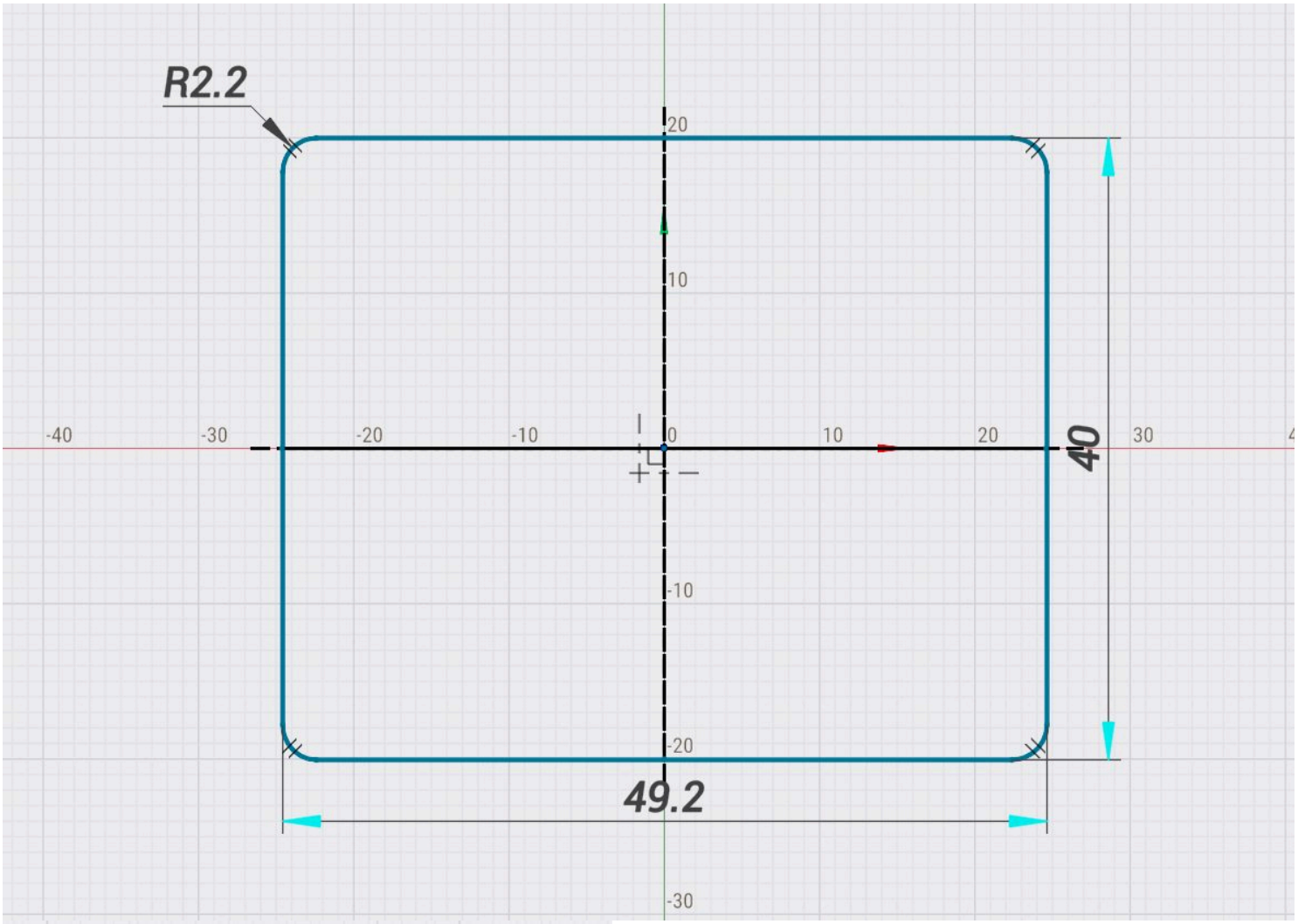
Number of sides. Defines the number of sides

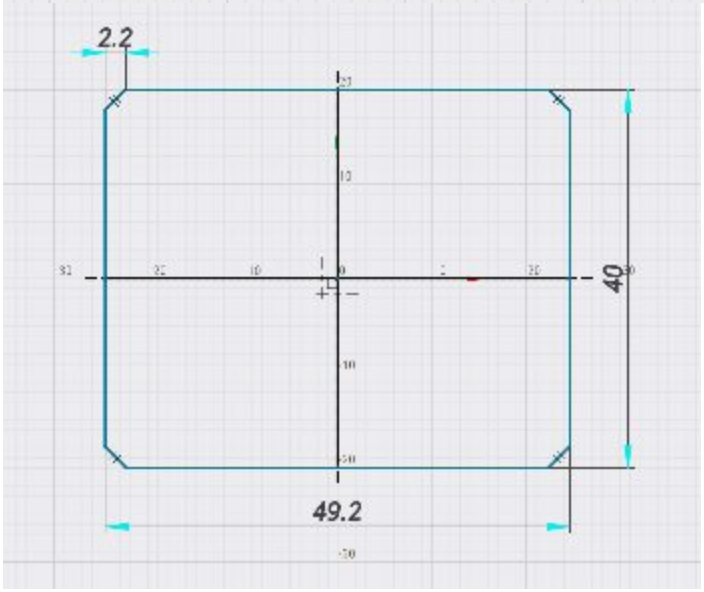
Hot keys:

RMB: Exit.

Rounding/Chamfer. Create sketch roundings and chamfers.

Hover the mouse over the sketch entities where you want to insert the feature and click.





Radius/Size. Defines the size of an element

Hot keys:

Press Space to toggle between rounding and chamfering.

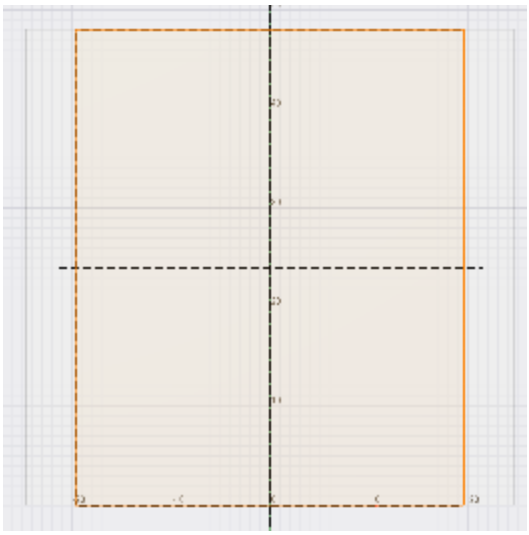
Rotate the mouse wheel while holding [Ctrl] to increase/decrease the feature size.

RMB: Exit.

Import object. Import reference geometry into the sketch

Click on a face, curve or an edge to project it into the sketch. The imported entities are marked with color.

Remove reference geometry from the sketch Click on the imported entity to remove it from the sketch.





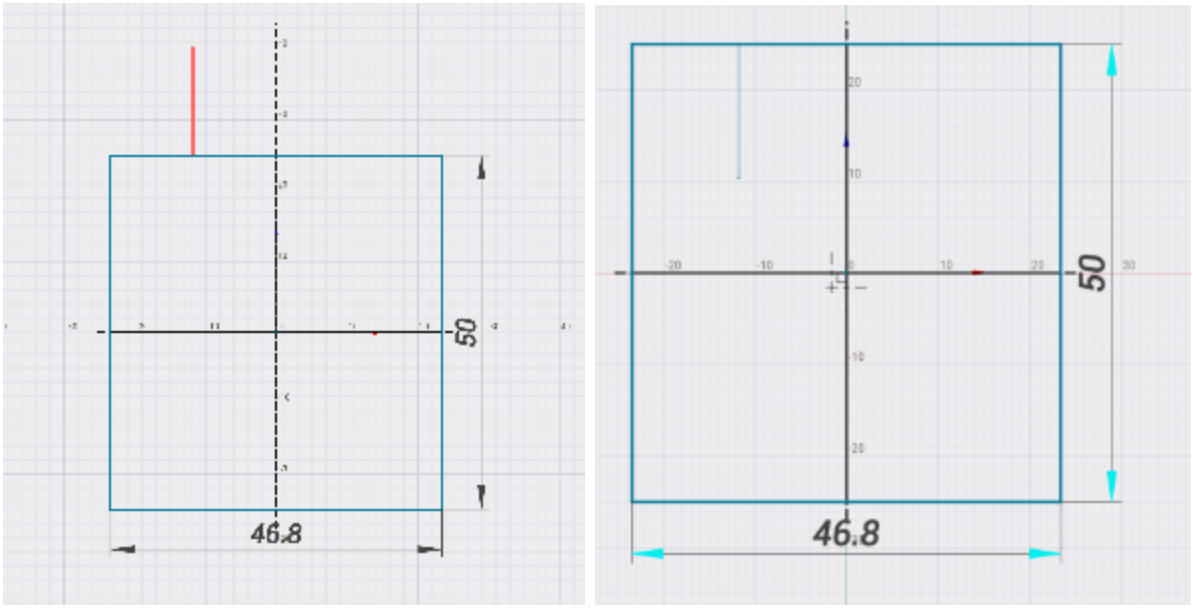
Hot keys:

RMB: Exit.

Trim. Trim out a piece of a contour.

Hover the mouse over the entities you want to trim and click. The entities to be removed will be highlighted.

Note: The tool is used to trim entities that intersect with other entities. If you want to delete noninteresecting entities use [Del] instead.



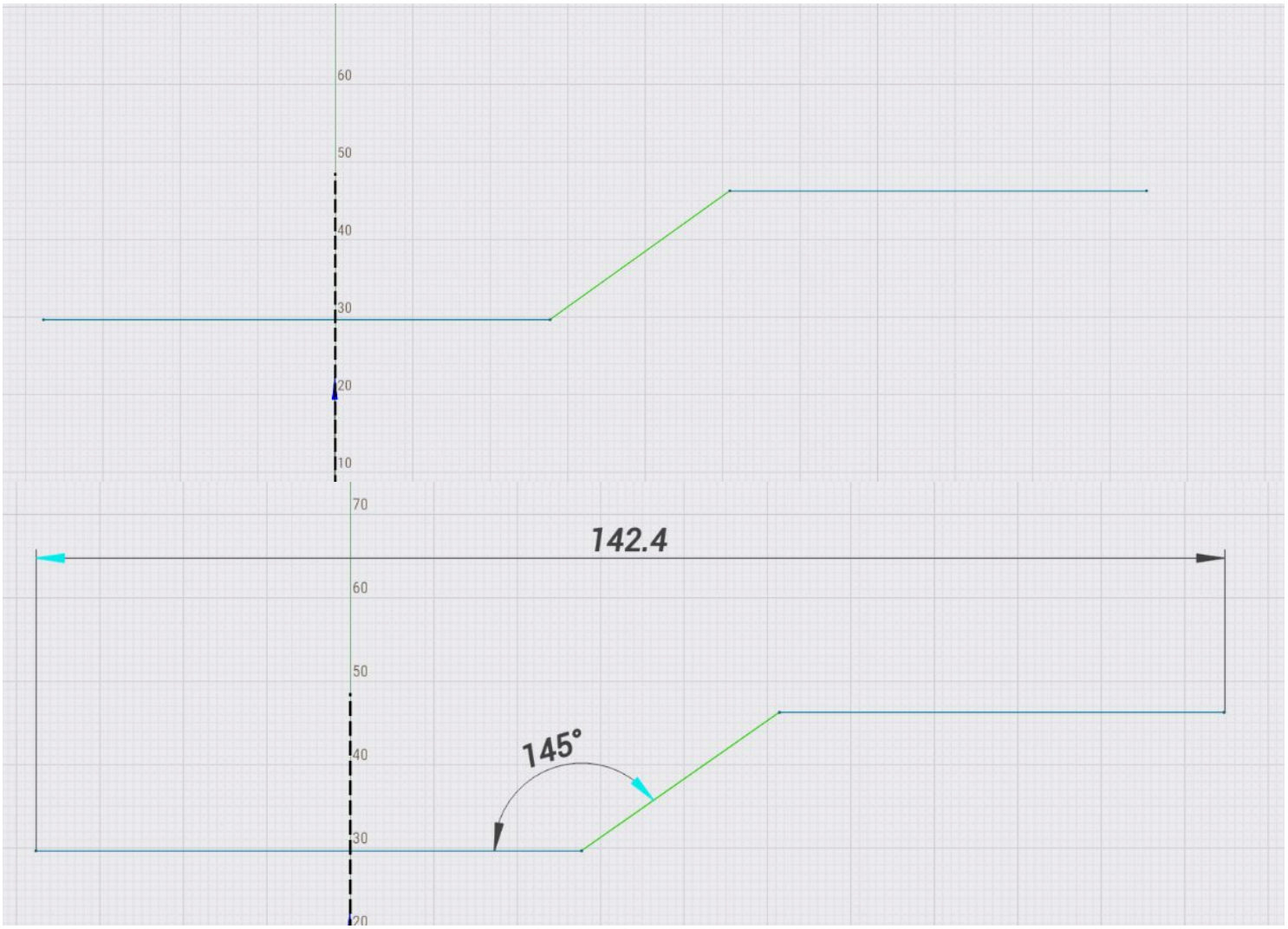
Hot keys:

RMB: Exit.

Dimension. Add dimensions to the sketch.

Select one or two geometry entities with the mouse to add a dimension. The type of the dimension will be determined automatically based on the type of the selected entities.

You can add linear, radial, diametral or angular dimensions this way.

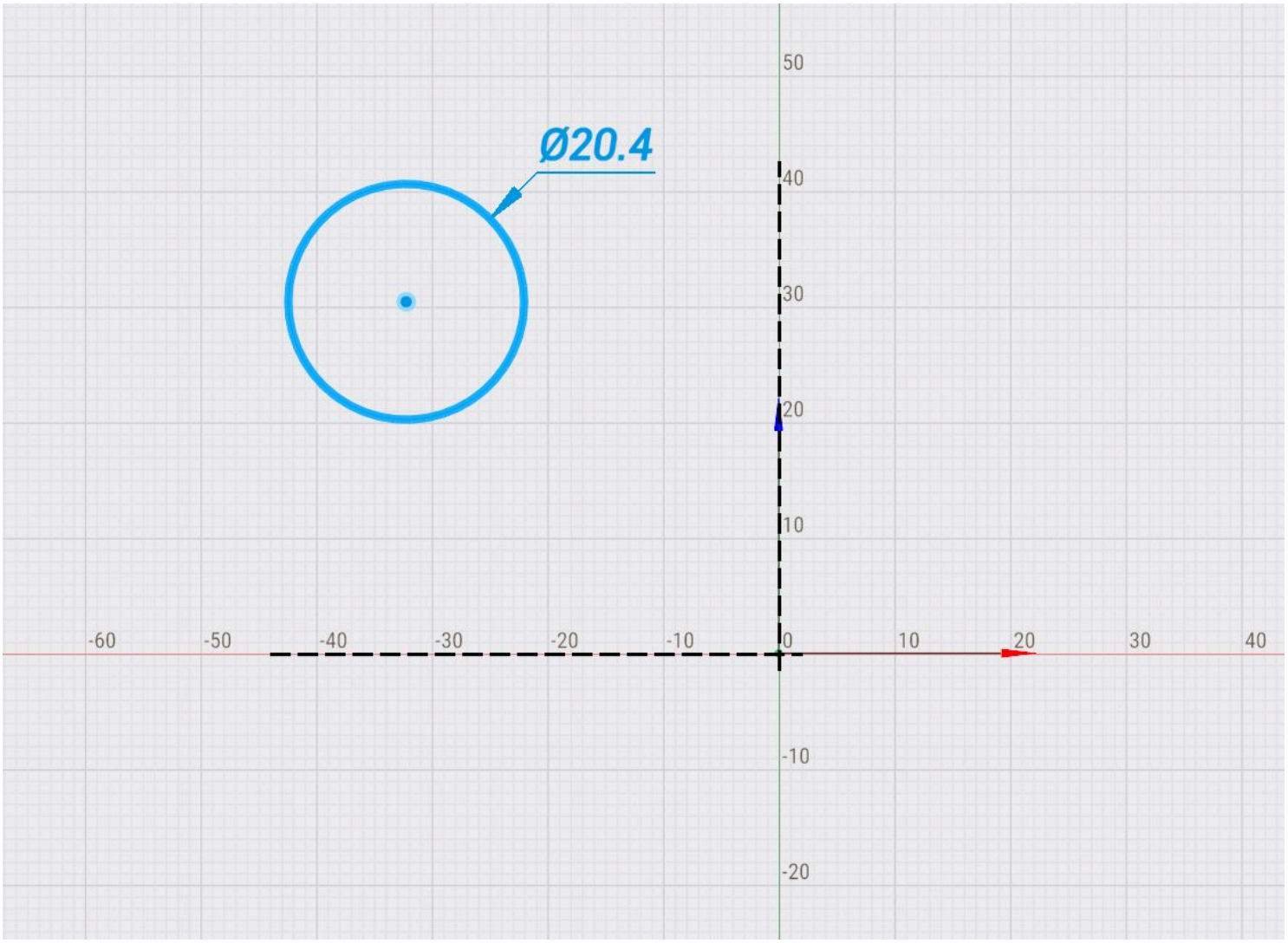


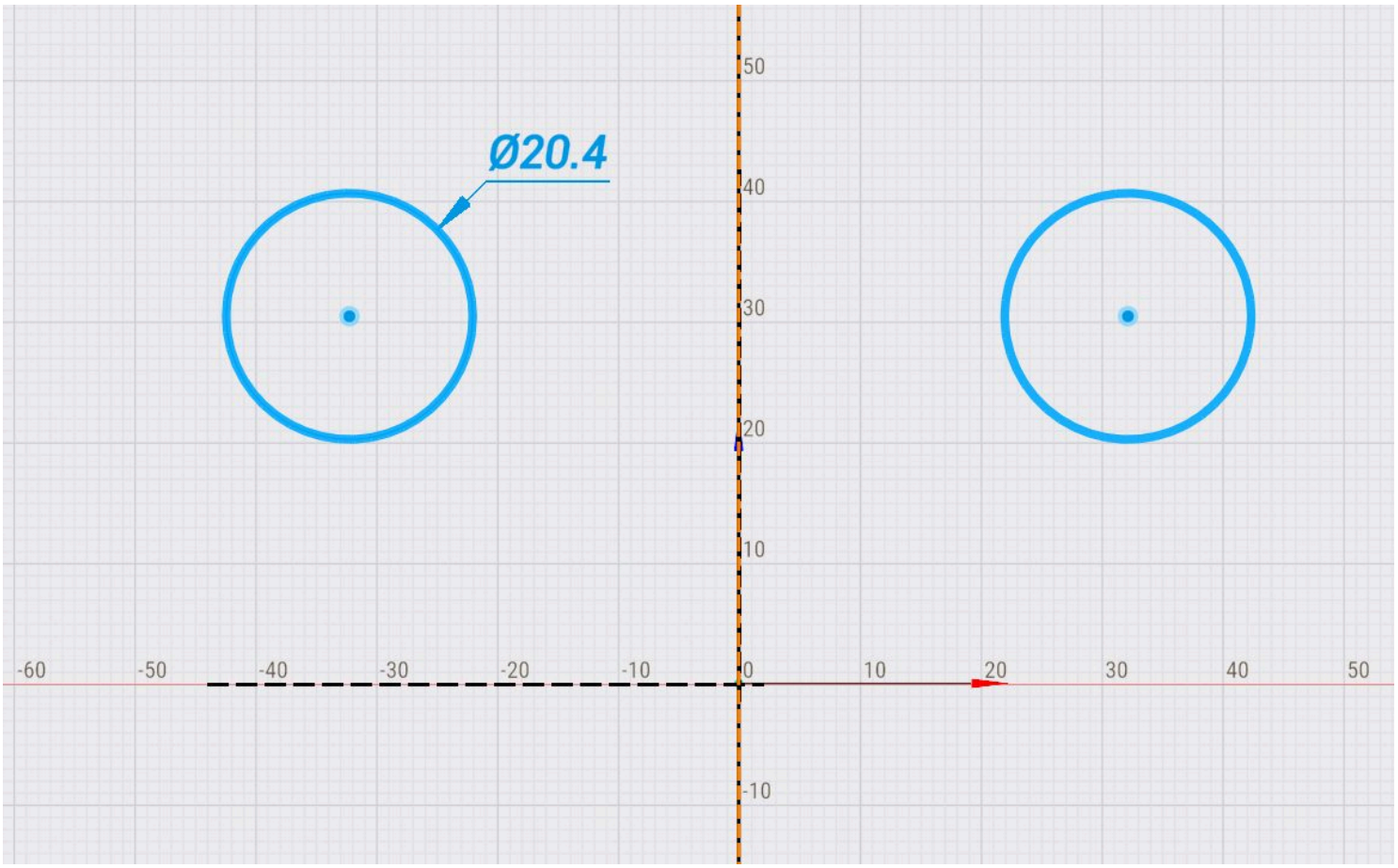
Hot keys:

RMB: Exit.

Mirror. Mirror selected sketch entities.

Select a line of the sketch with mouse as the symmetry axis.


Ø20.4
50
40
30
20
10
-10
-20
-60
-50
-40
-30
-20
-10
0
10
20
30
40

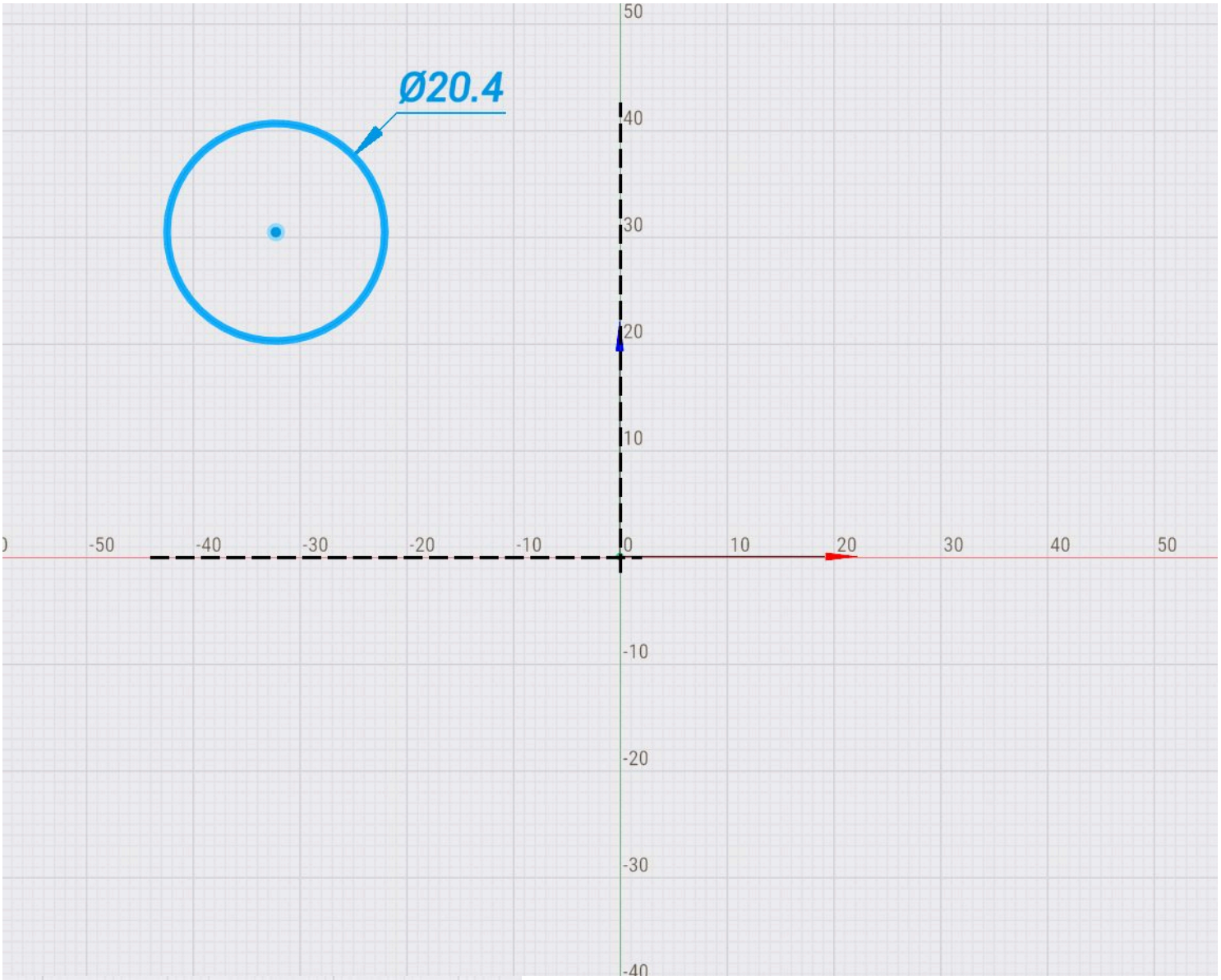


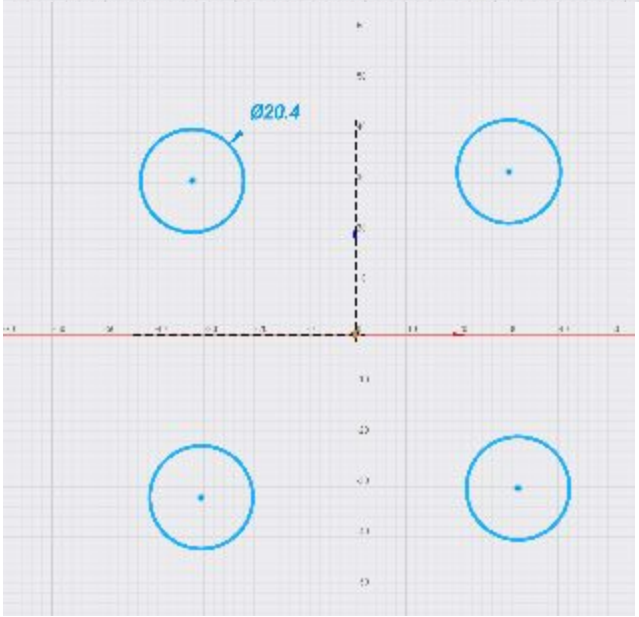
Hot keys:

RMB: Exit.

Circular pattern. Create a circular pattern of the selected entities

1. Enter the number of copies in the field above.
2. Select a point of the sketch to define the center point of the pattern.





Number of copies. Specifies the number of elements to copy.

Hot keys:

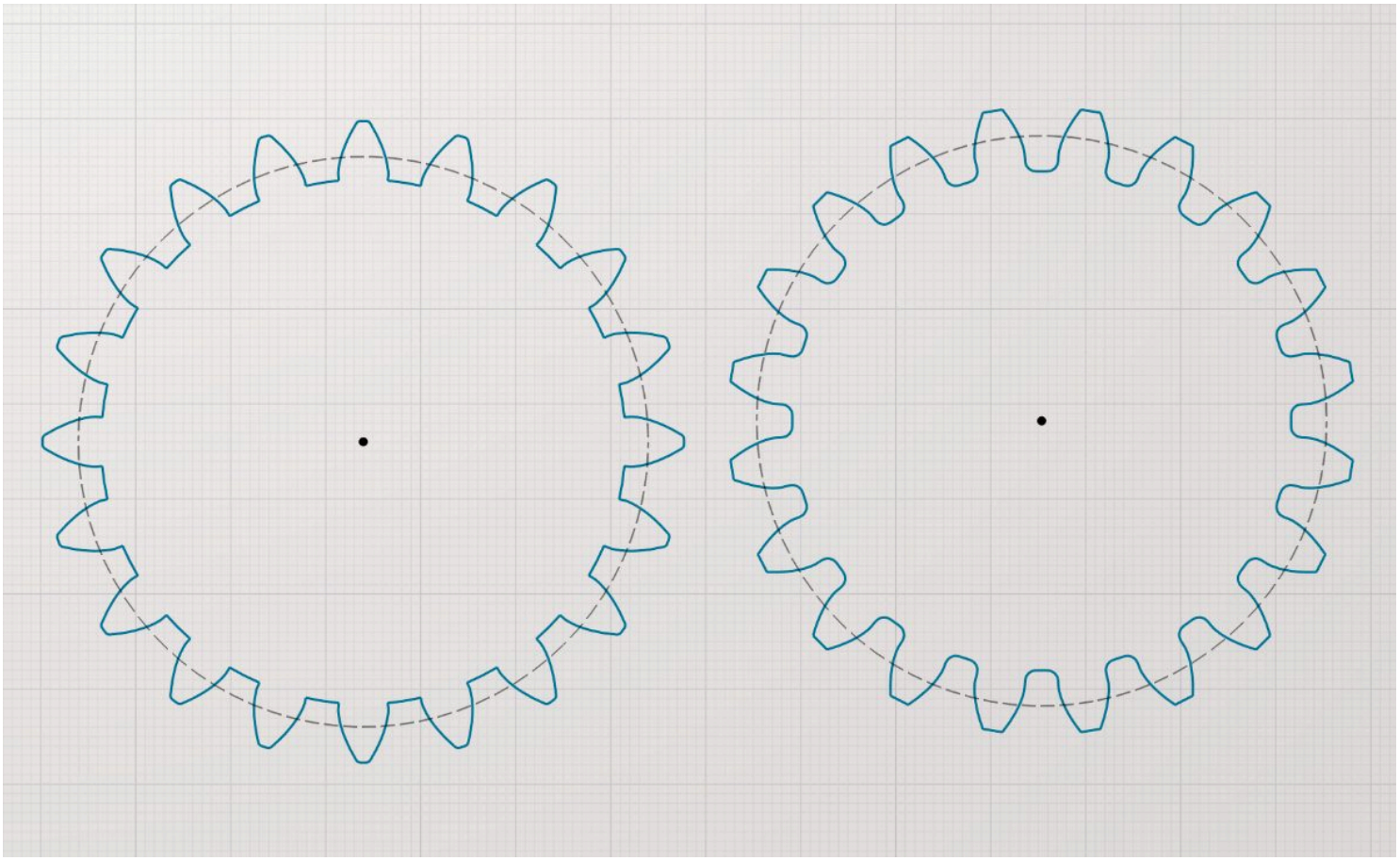
RMB: Exit.

Text. Allows you to create custom text.

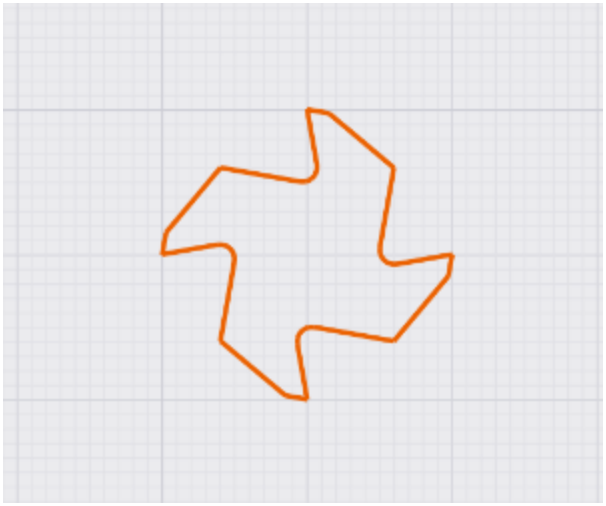


Helixal gear. Allows you to build a gear profile

Helic angle. Angle setting for choosing between a spur or helical gear in the 3D model.



Mill profile. Allows you to build a cutter profile



Insert decal image. Allows you to insert an image to outline it

Load. Allows you to download a drawing in *.gsf format

Save. Allows you to save the drawing in *.gsf format

Features of working with drawings.

▼ Click here to expand..

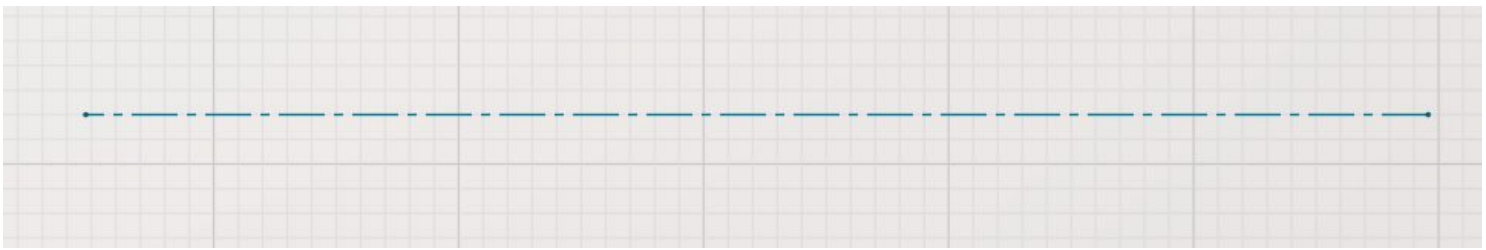
When working in the drawing mode and selecting curves (Left or right mouse button), various settings are possible (changing the line type, restrictions, etc.)

In drawing mode, you can select 3 main lines:

- Main - Used for drawing a part and its visible sides.

- Construction - Image of invisible outlines of an object

- Center line - Designation of axial lines and center lines of circles



Lock position. Allows you to enter restrictions (angle, size, etc.) The object does not move in space

Lock agle. Allows you to limit the angle of an element

Make horizontal. Allows you to make the curve horizontal. The curve can be part of the contour

Center lines. Allows you to set the centerlines of a circle/radius

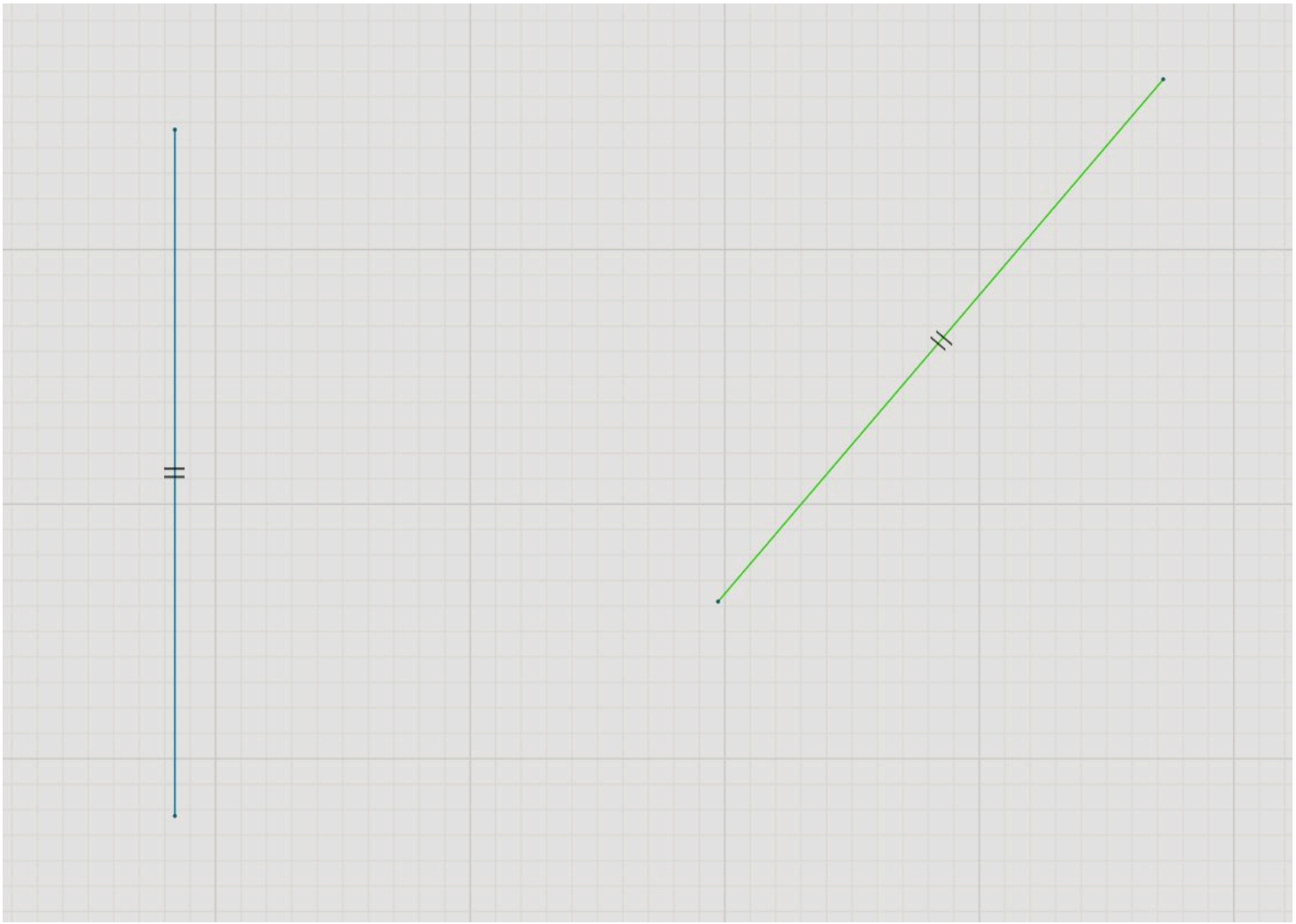
Dimension. Allows you to set the dimensions. As a rule, the size is set from the 0 point of the drawing to the element.

Defeature. Allows you to exclude an element (for compound figures, contours, etc.)

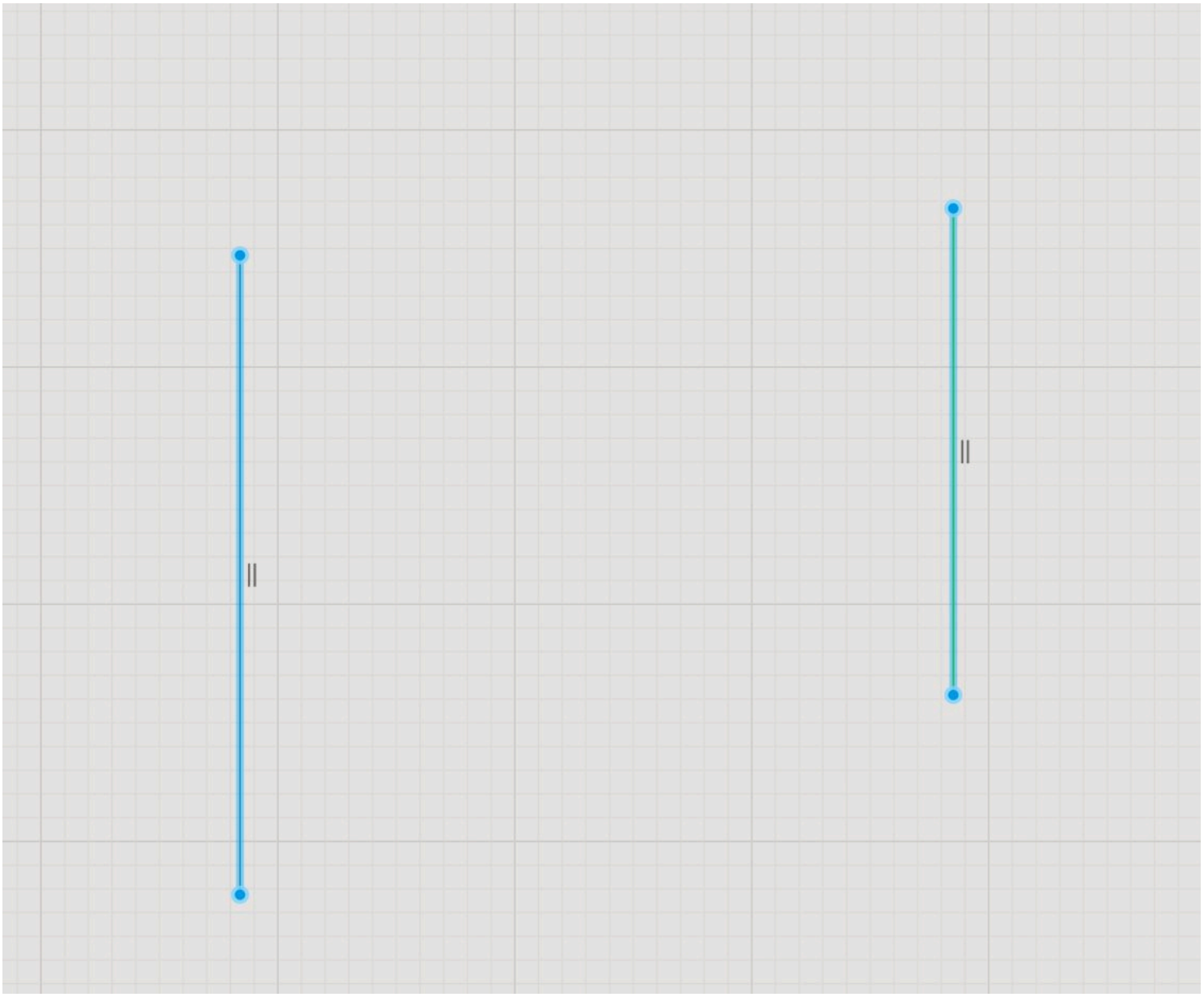
When selecting multiple elements (for example 2 segments), the following restrictions are available:

The restriction can be removed by selecting the restriction sign and pressing the DEL key.

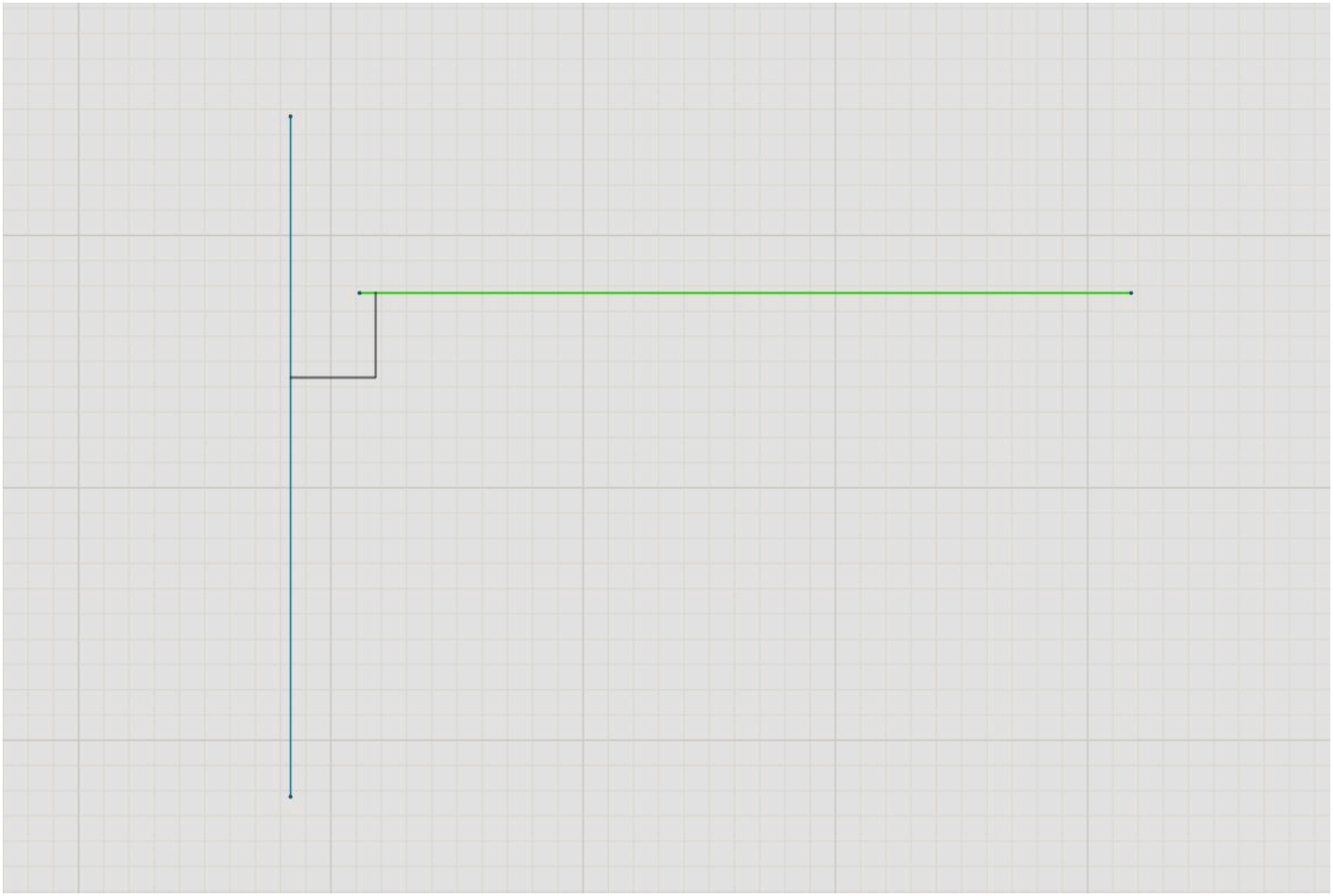
Equal. Introduces a length limit (two segments become equal). They are denoted by the equal sign. T



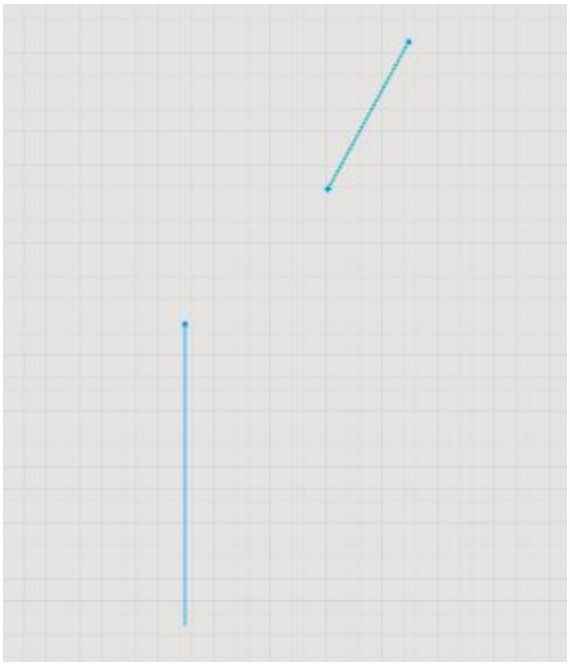
Parallel. Makes two elements parallel to each other. The limit sign is visible when one of the elements is selected.



Perpendicular. Makes two elements perpendicular to each other.

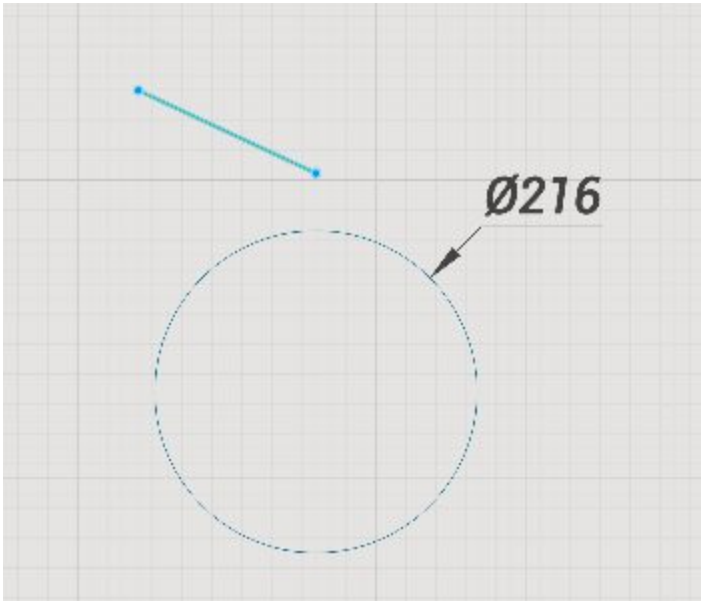


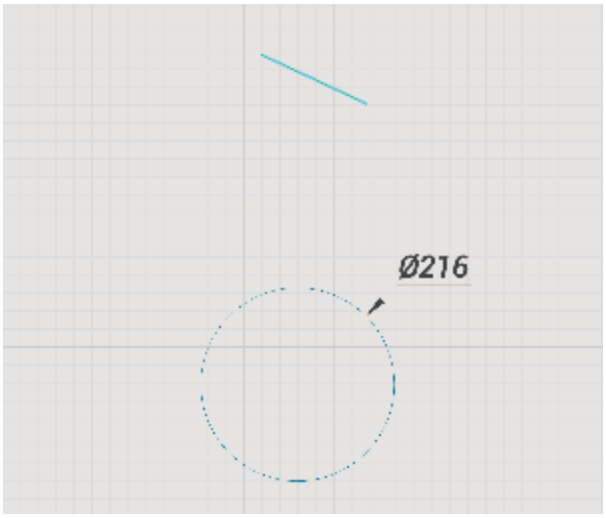
Colinear. Allows you to set 2 curves on 1 line.



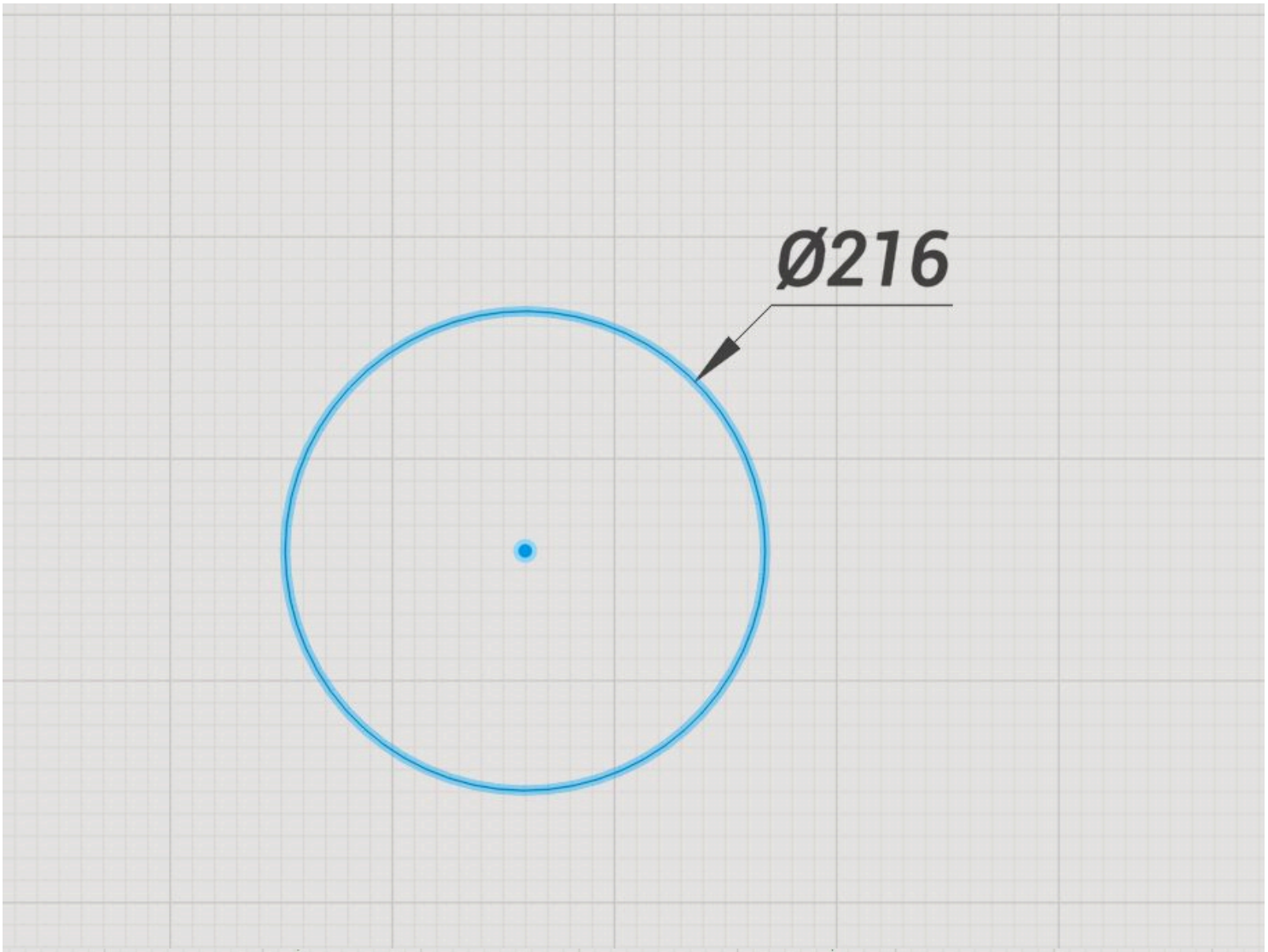
The drawing also actively uses the different keys.

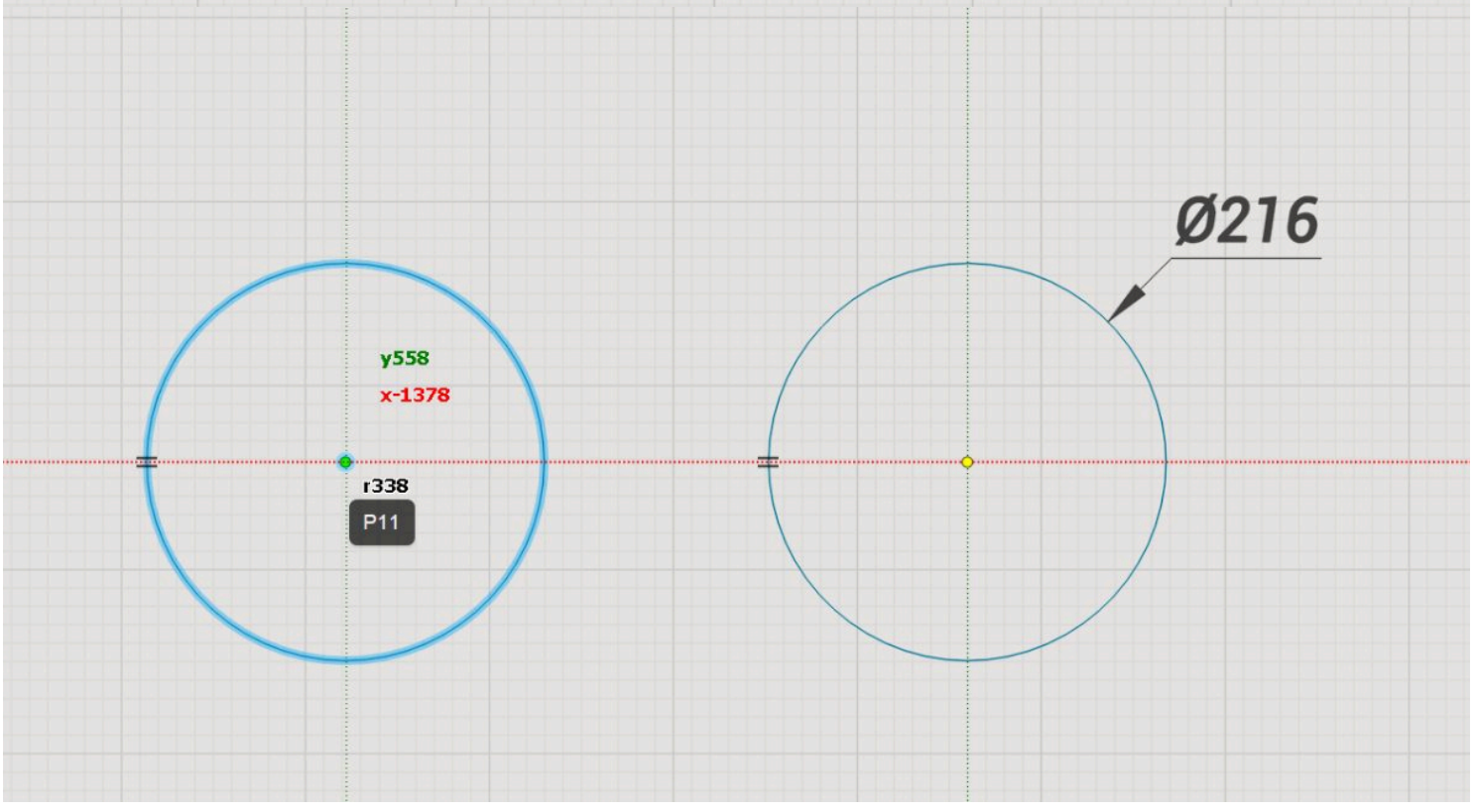
- To move, select an element, hold down Shift and drag to the side



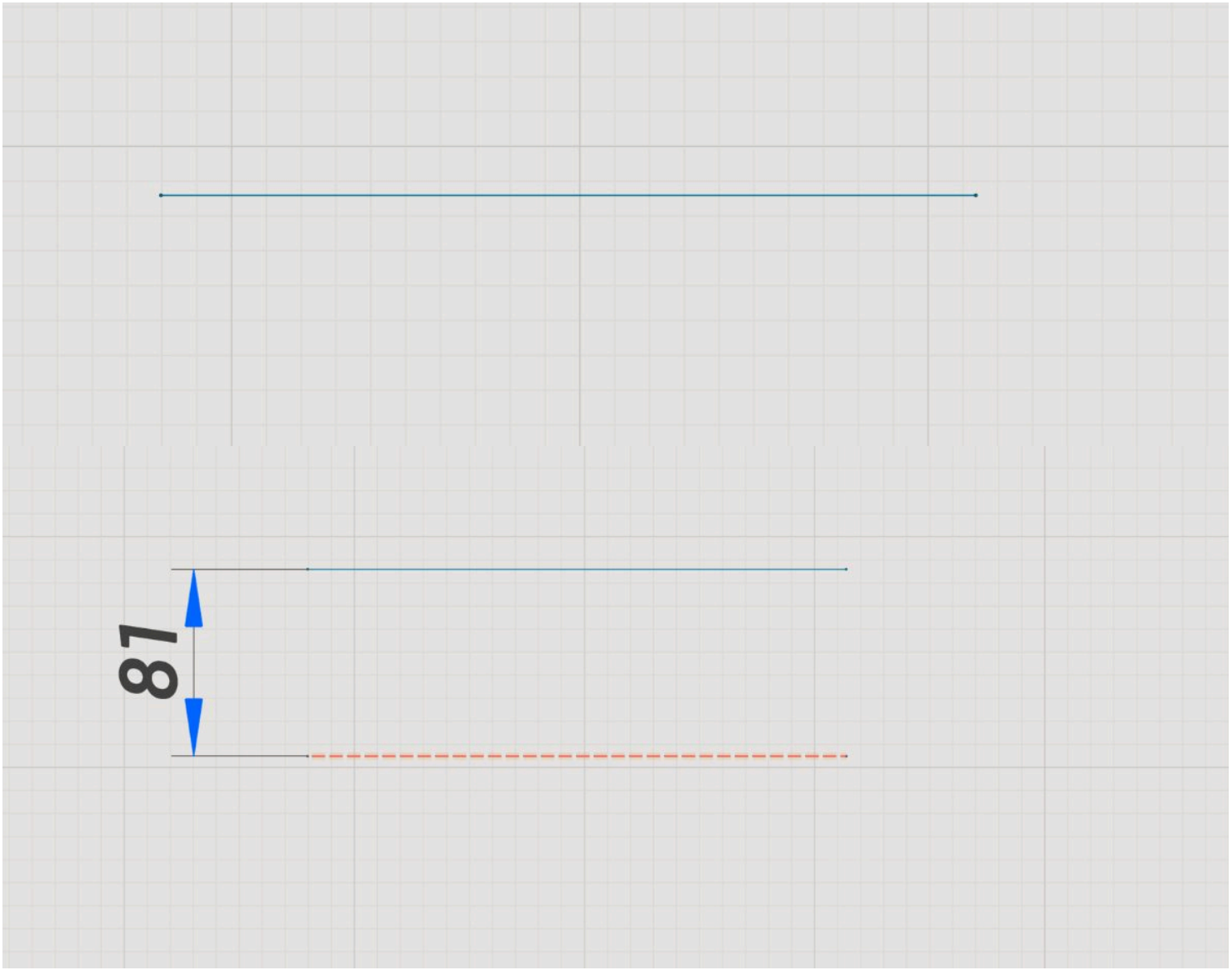


To create a copy, select the element. Grab the point and use the Ctrl key

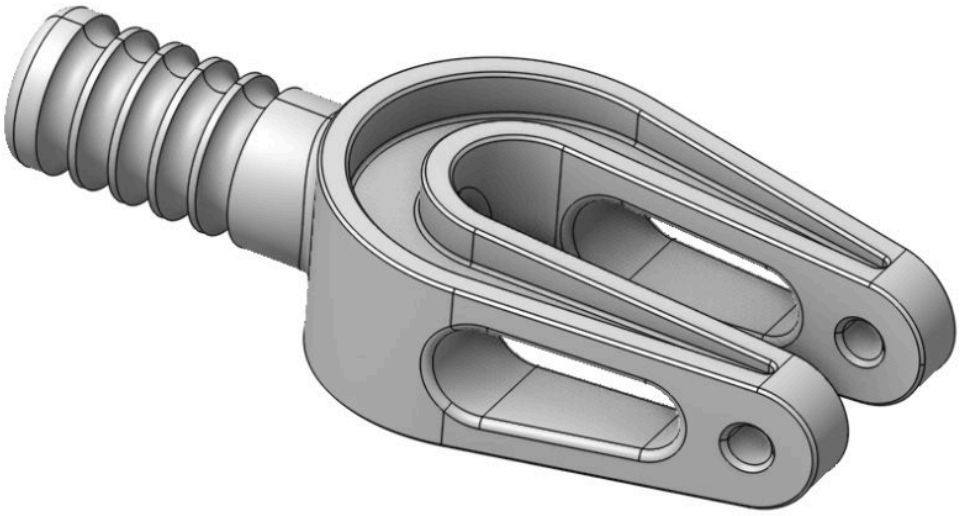

Ø216


Ø216
y558
x-1378
r338
P11

To move, grab the element and press the Ctrl key.



Design in 3D mode



Application area:

This mode allows you to create 3D model. Sketches must be used to build the model. Details

Toolbar/Element properties.

Undo. Reverses the last action.

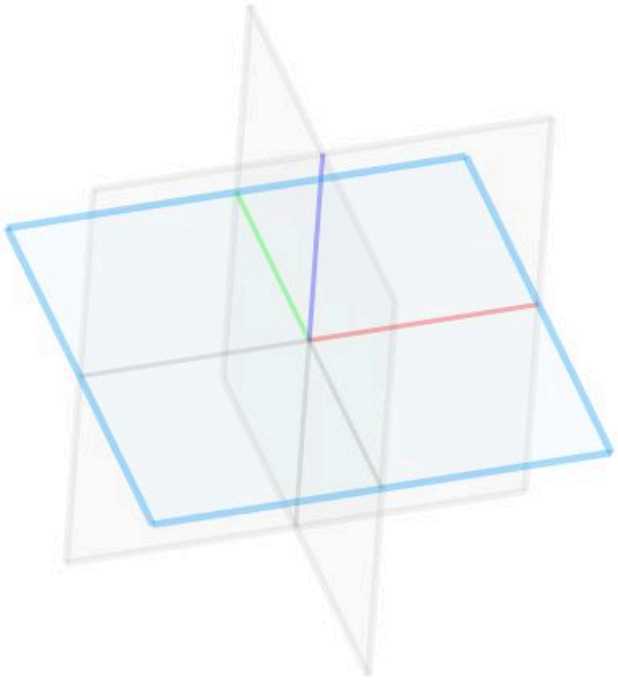
Redo. Reapplies the last undone action.

Play. Animate the model history step by step to get a better understanding of the design intent (The model must contain several elements in the build tree.)

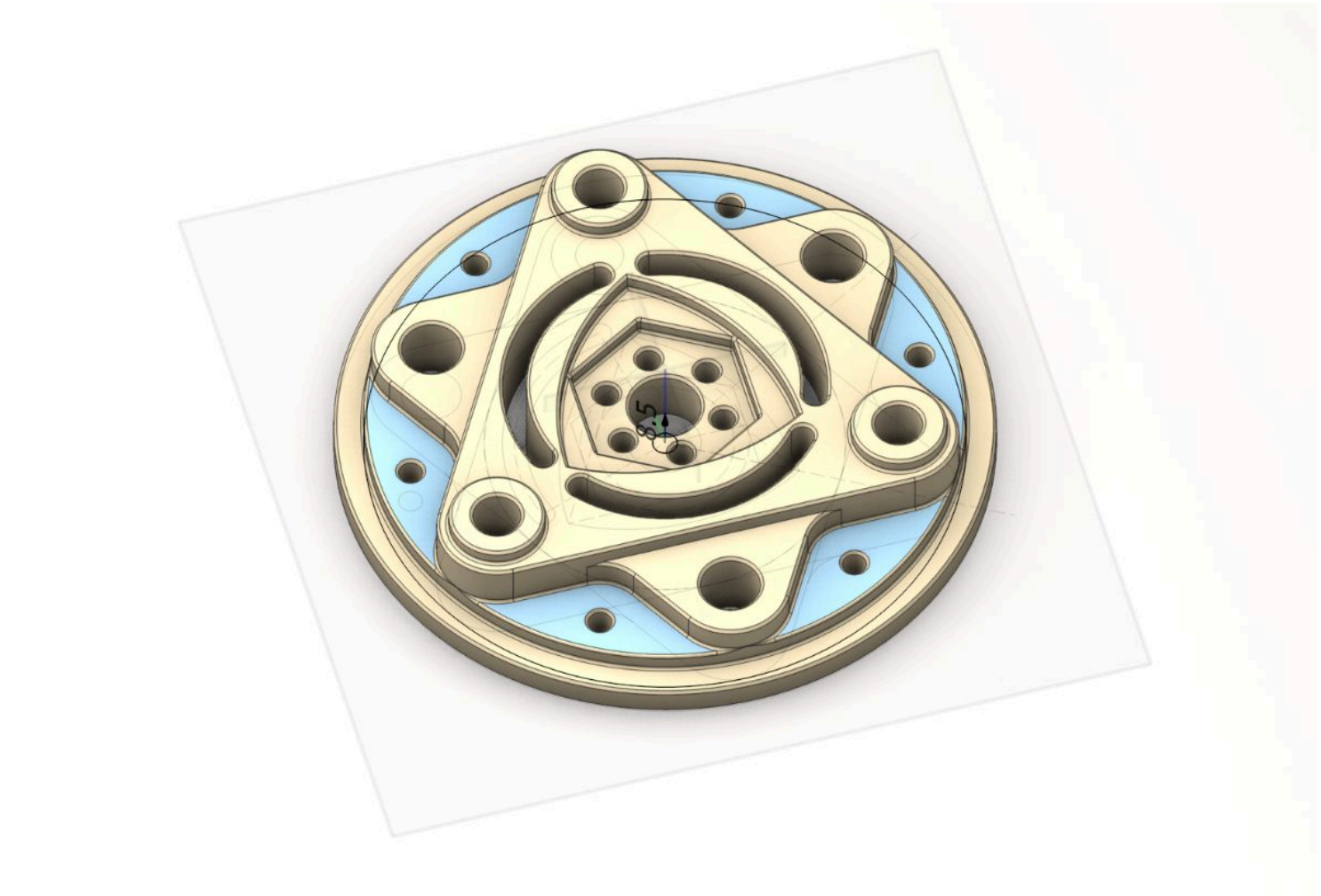
Plane. Allows you to set a plane for a drawing.

▼ Click here to expand..
You can create sketches on various surfaces, with offset, and more:

- Relative to the base planes



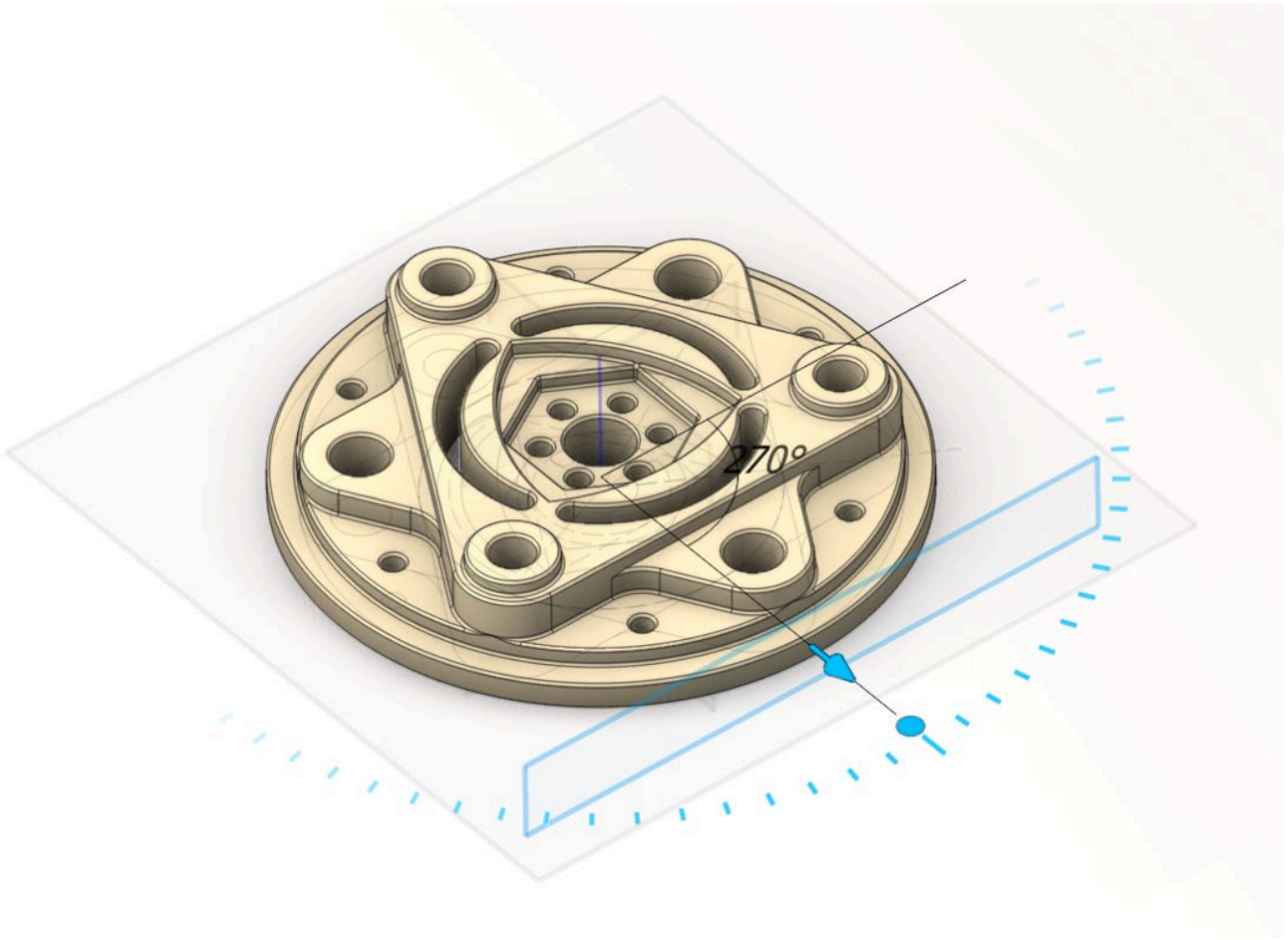
- On the surface of the model



Tangent plane. Create a tangent plane

Click on a plane or a face to create a plane.

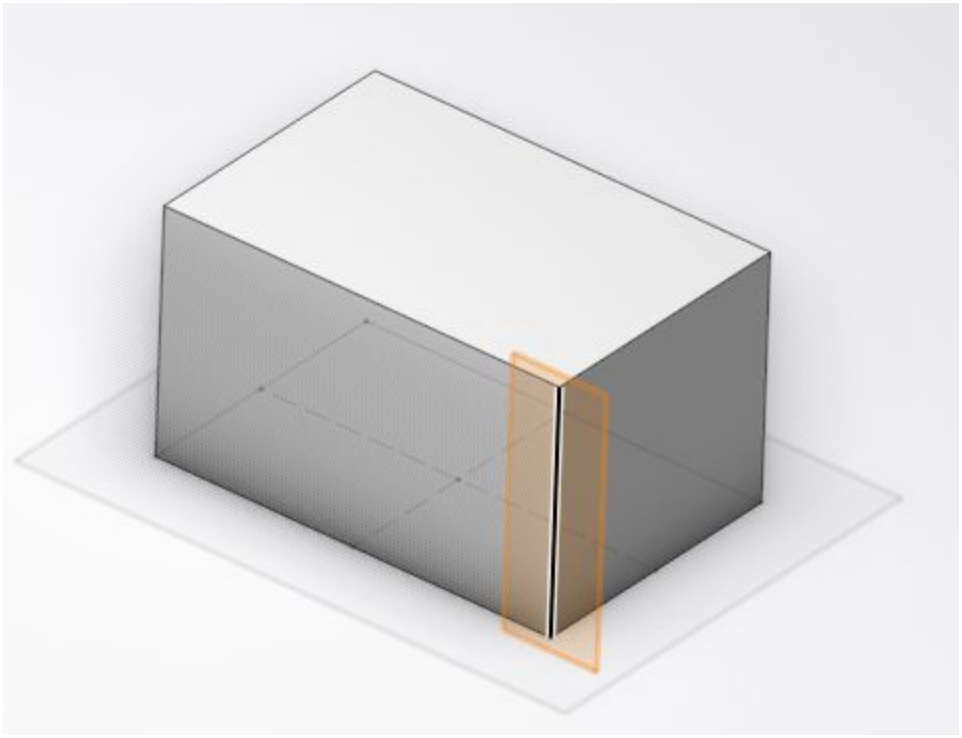
RMB: Exit.



Axial plane. Create a plane through an axis

Select a straight edge, a sketch line or a surface of revolution to create a plane.

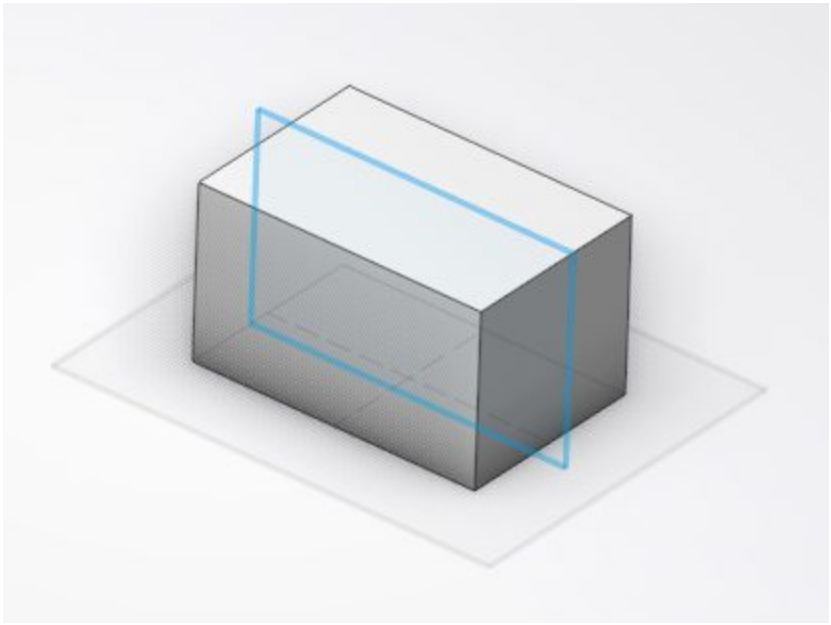
RMB: Exit.



Middle plane. Create a middle plane.

Select two flat faces to create the middle plane.

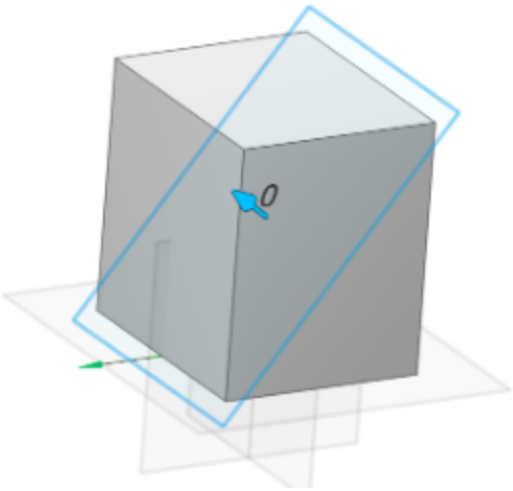
RMB: Exit.



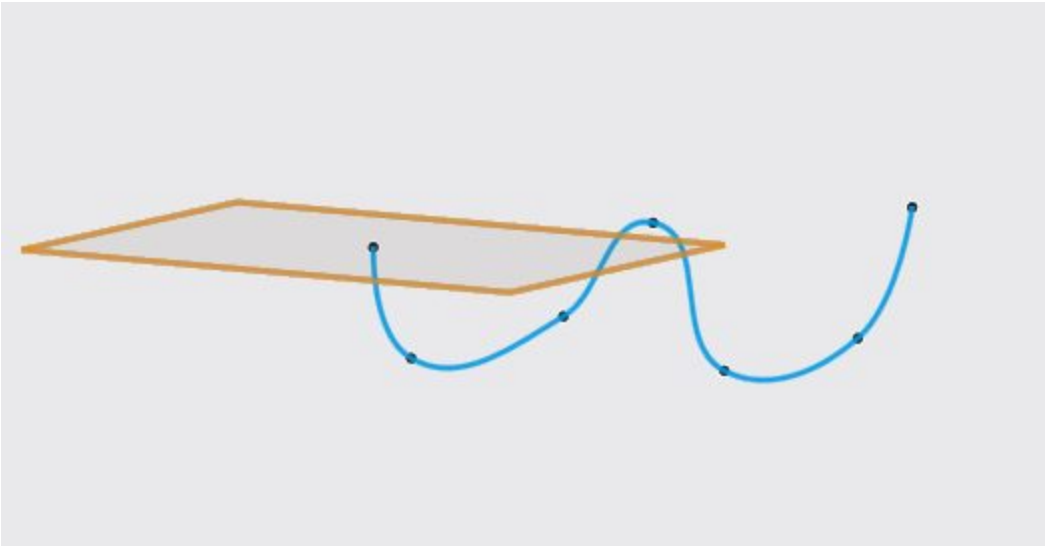
Plane throw two lines. Create a plane trhough two lines

Select two straight edges to create a plane.

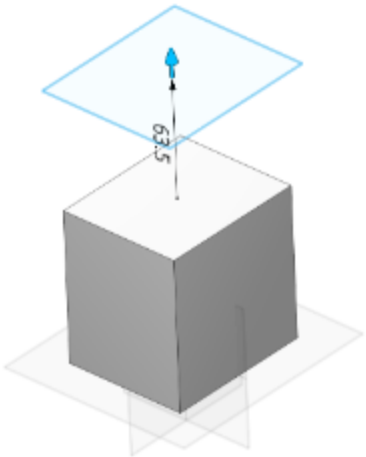
RMB: Exit.



Create normal plane. Creates a plane normal to a curve. Used with function Spline.



Offset plane. Allows you to set a plane at a distance. Called by right clicking on a surface or plane.



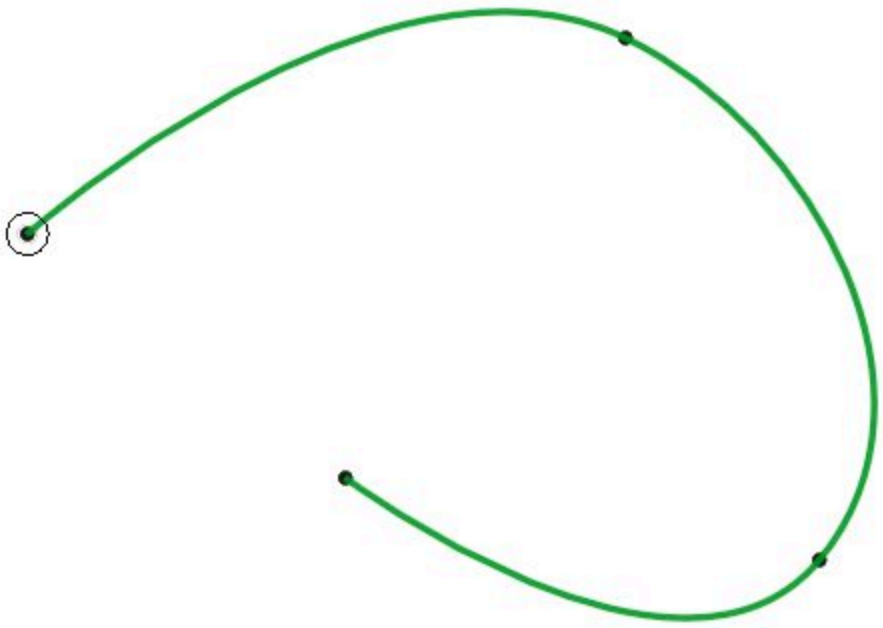
Spline. A spatial curve, represented by a spline, a polyline, and a helical line.

▼ Click here to expand..

1. Call the Spline command on the toolbar Choose a curve option Spline or Polyline

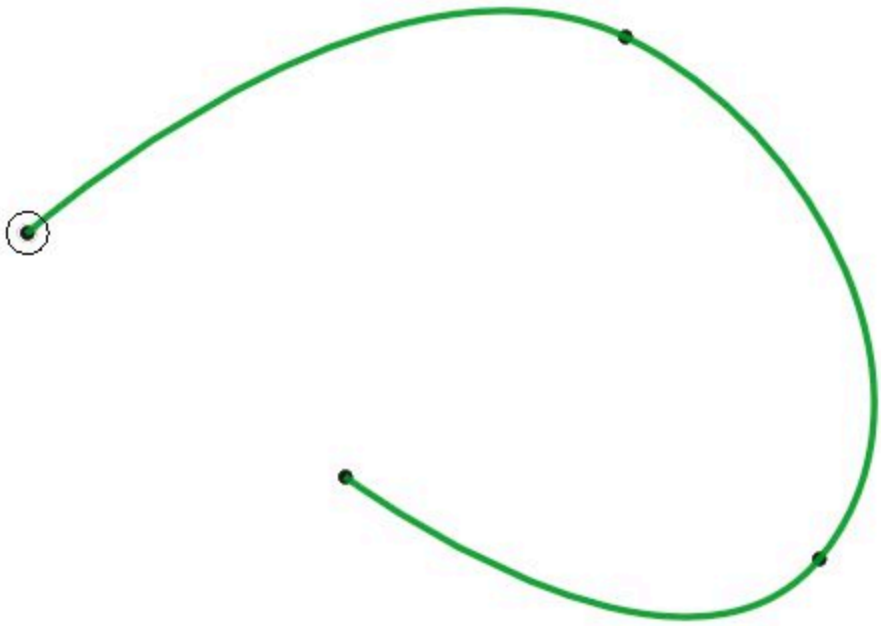
2. Set points with the mouse (snapping works) or set numeric coordinates in the appropriate field.

3.Use the right mouse button to complete the point assignment.

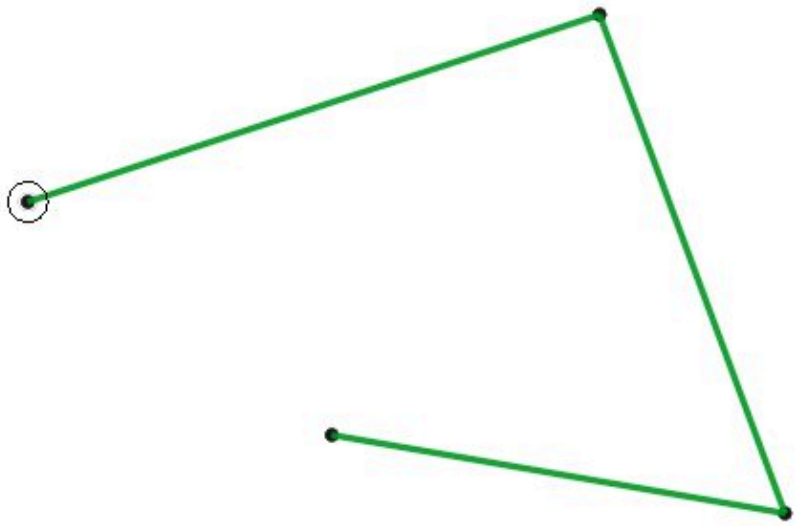


Curve type. Defines the curve type: Spline or Polyline.

Spline.



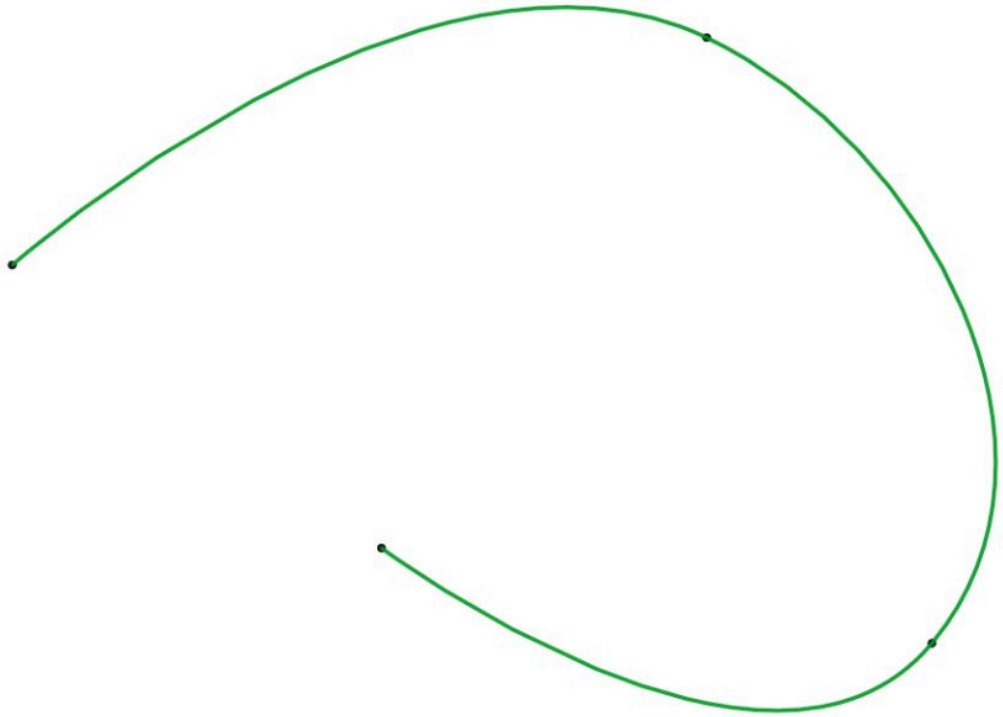
Polyline.

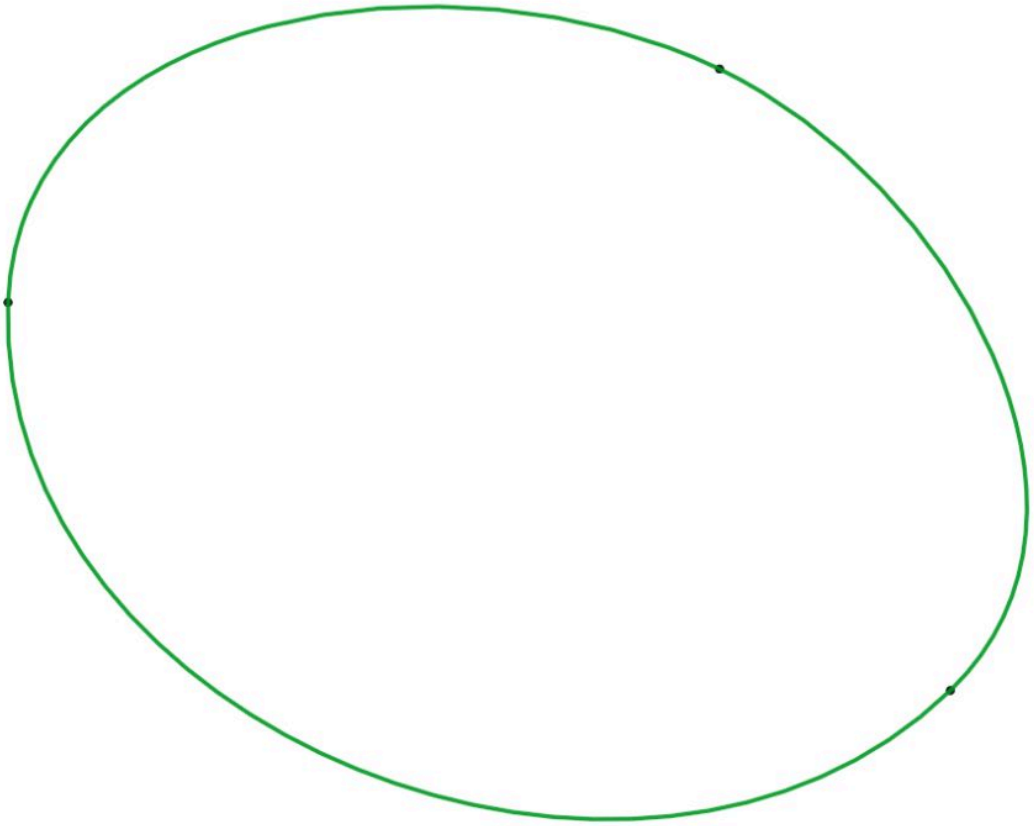


Draw. Switching the mode for entering new points or editing existing ones is done by pressing/releasing the Draw button.

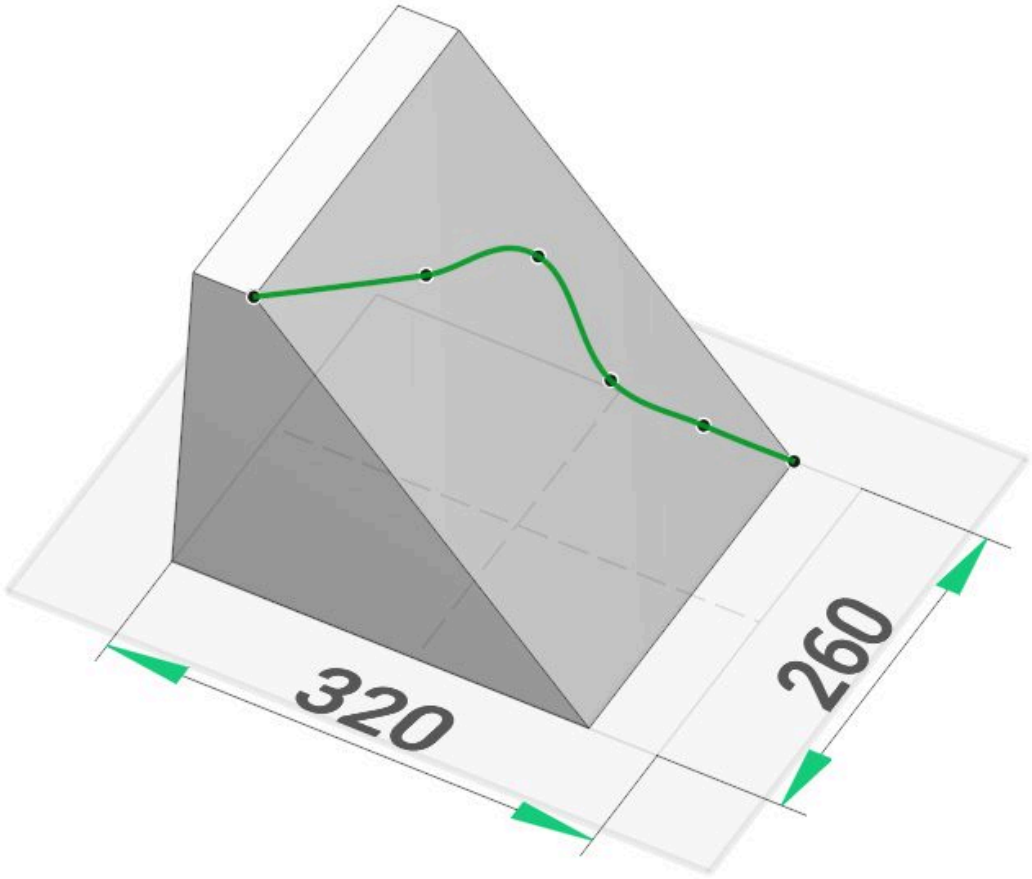
In the editing mode, the selected point is circled, and its coordinates are displayed in separate input fields (X, Y, Z). You can change the coordinates of the selected point either by entering the coordinates or by dragging the mouse. The coordinate input fields are inactive if no point is selected.

Closed. Allows closing the spline.





On Plane. You can place a spline on a plane by selecting a plane or a flat edge in the OnPlane drop-down list (highlighted when selected). The curve points will be projected onto the selected plane. This function is not supported on cylindrical surfaces or circular geometries



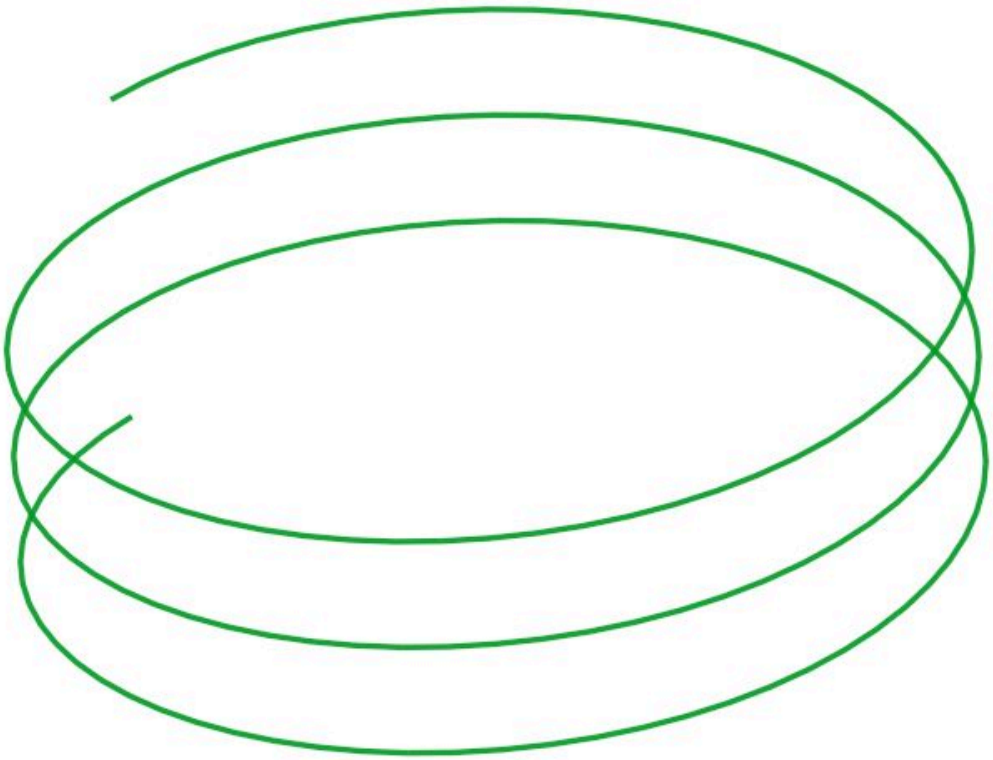
Point. Defines a point for the spline

Coordinates. Determines the coordinates of a given point. To set a series of points, you can copy a set of coordinates from MS Excel or a text file and paste them into the Coordinates input field.

An alternative way to create a spline is to select a plane or planar face in model space and select Spline from the pop-up toolbar.
Helix. A spatial curve.

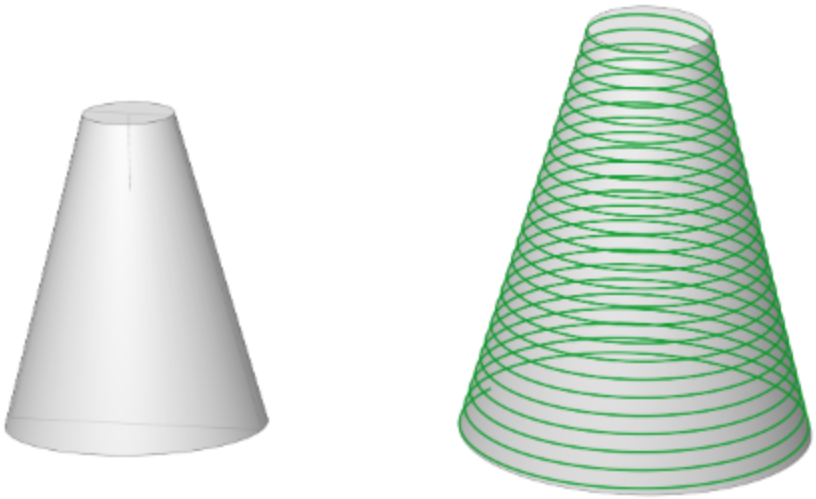
▼ Click here to expand..

1. Call the Helix command in the toolbar and set the coordinate of the helical line center (point with the mouse or enter the coordinates in the input field)

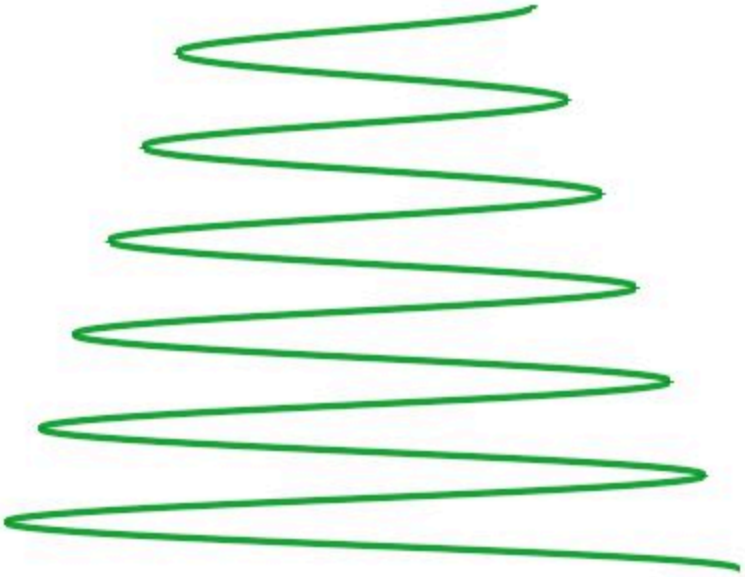


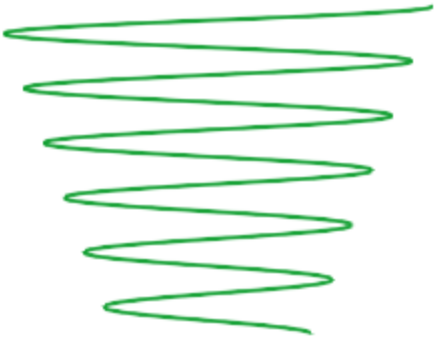
The geometry of the surface on which the helical line is located is determined by the dimensions: initial and final radii, height, cone angle. You can change the dimensions by entering values in the input fields, adjusting the corresponding visual objects, or dragging the control points with the mouse.

Base element. To construct a helical line, it is possible to use previously constructed geometry. To do this, in the curve creation/editing mode, click the mouse on the face or surface while the Base element selection field is pressed.



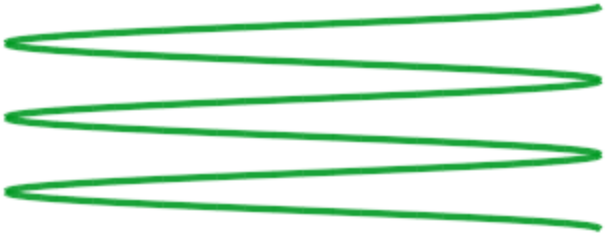
Change the helix direction by clicking the arrow button. →



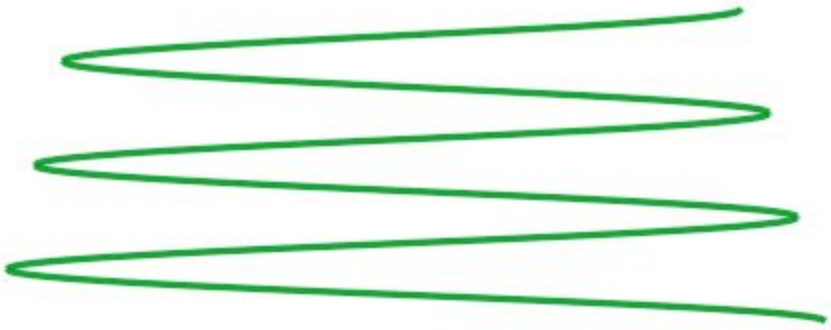


Helix type. There are three possible variants of the helical line.

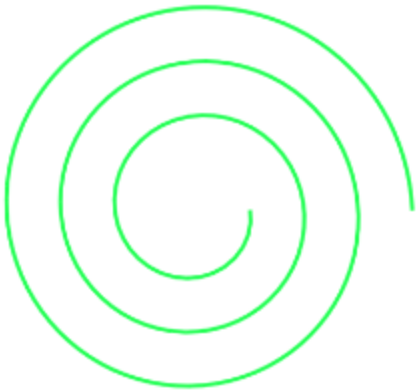
Cylindrical.



Conical.



Flat (Archimedes).



Input method. The helical parameter of the curve can be set in four ways: By Pitch, By Angle, By Count.

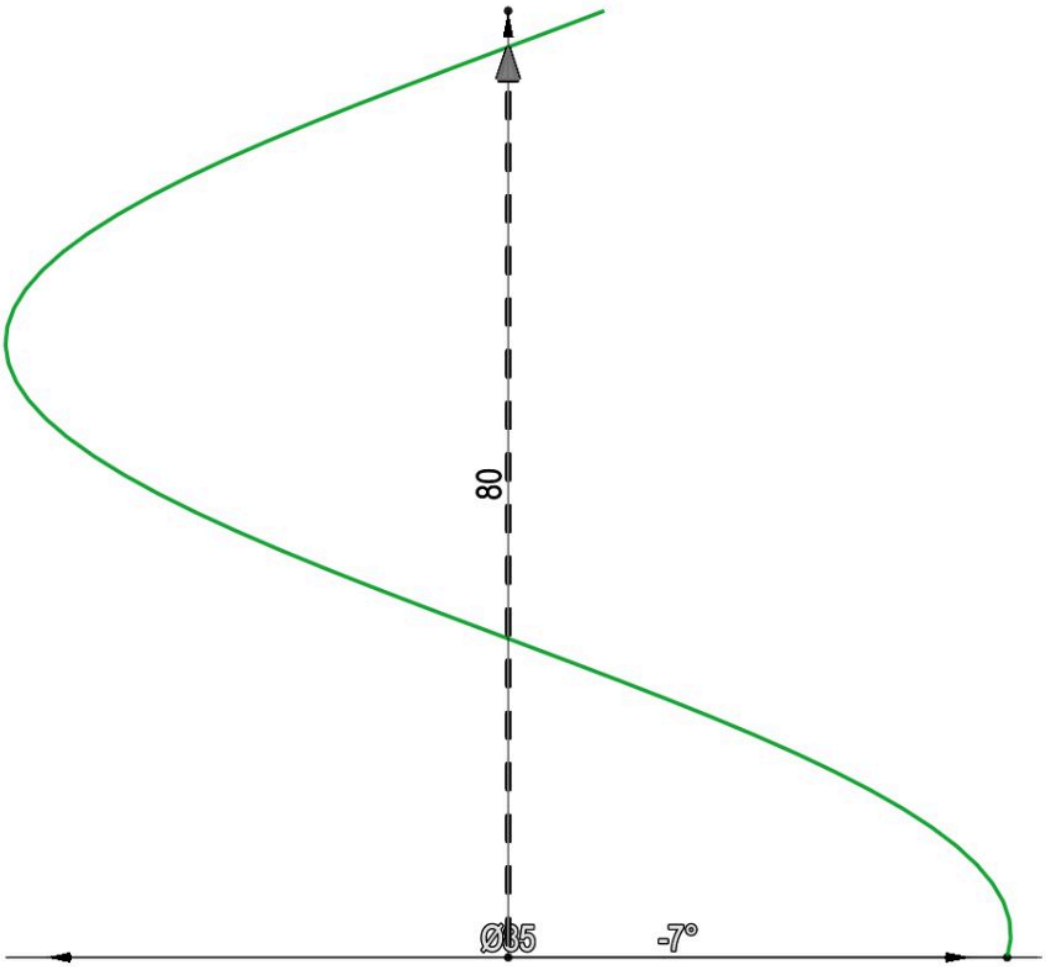
When using the By Pitch option, the step is controlled by the Pitch parameter.

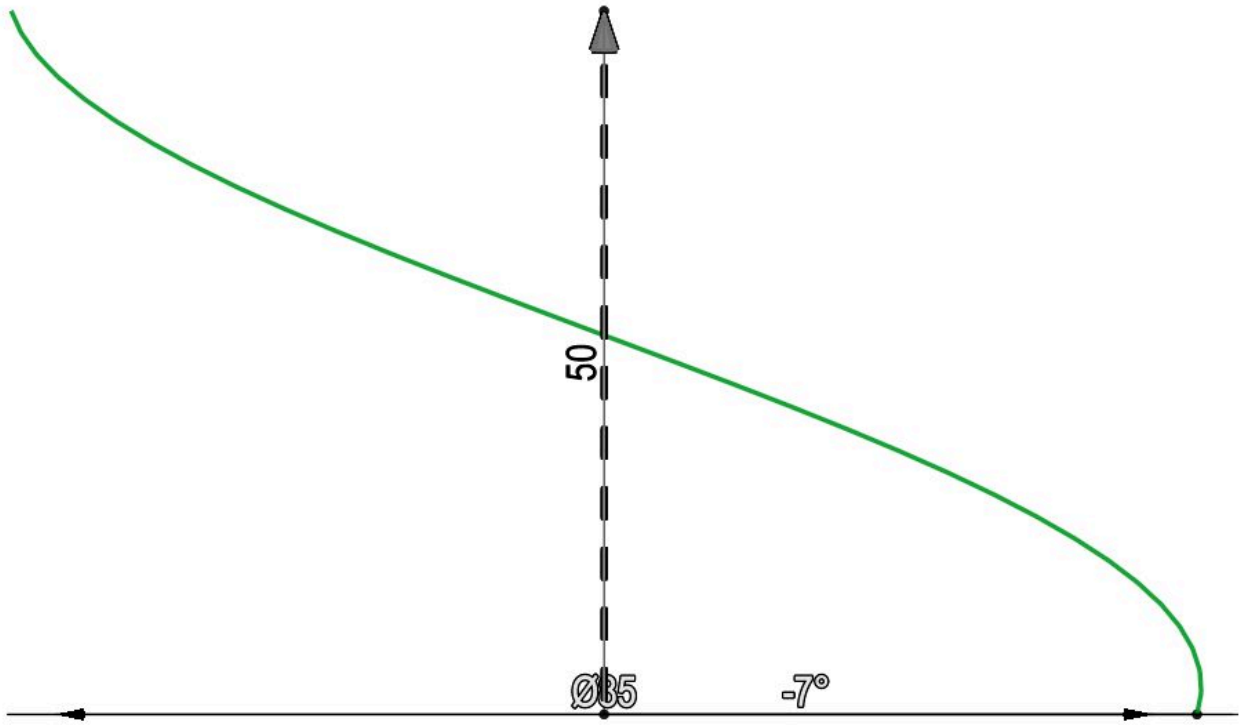
With the By Angle option, the length is controlled by the Total Angle parameter.

For the By Count option, the step is controlled by the Number of Threads (NB of Threads) parameter.

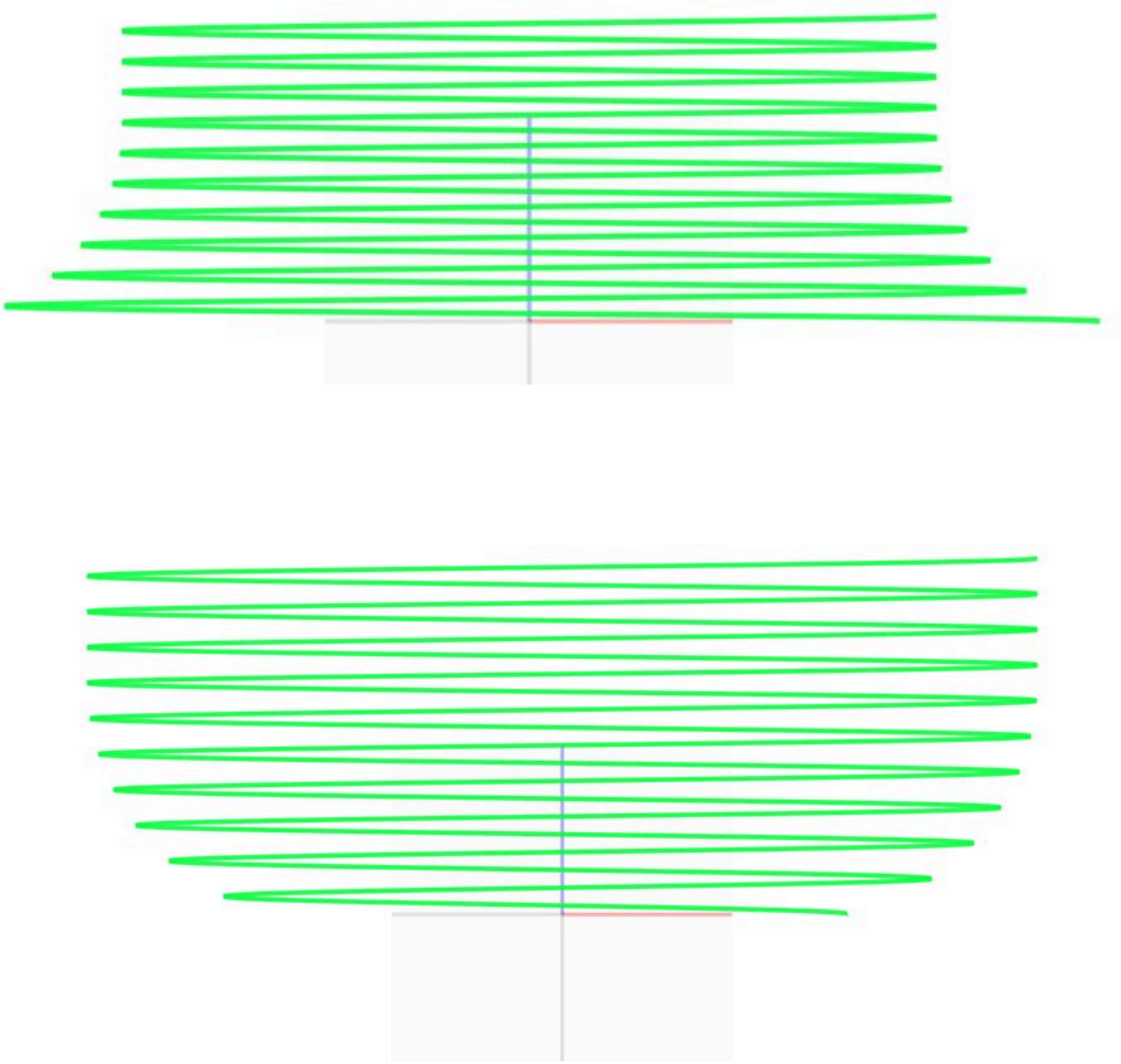
Below are all remaining parameters available depending on the Helix type.

Height. Determines the height of the spiral.

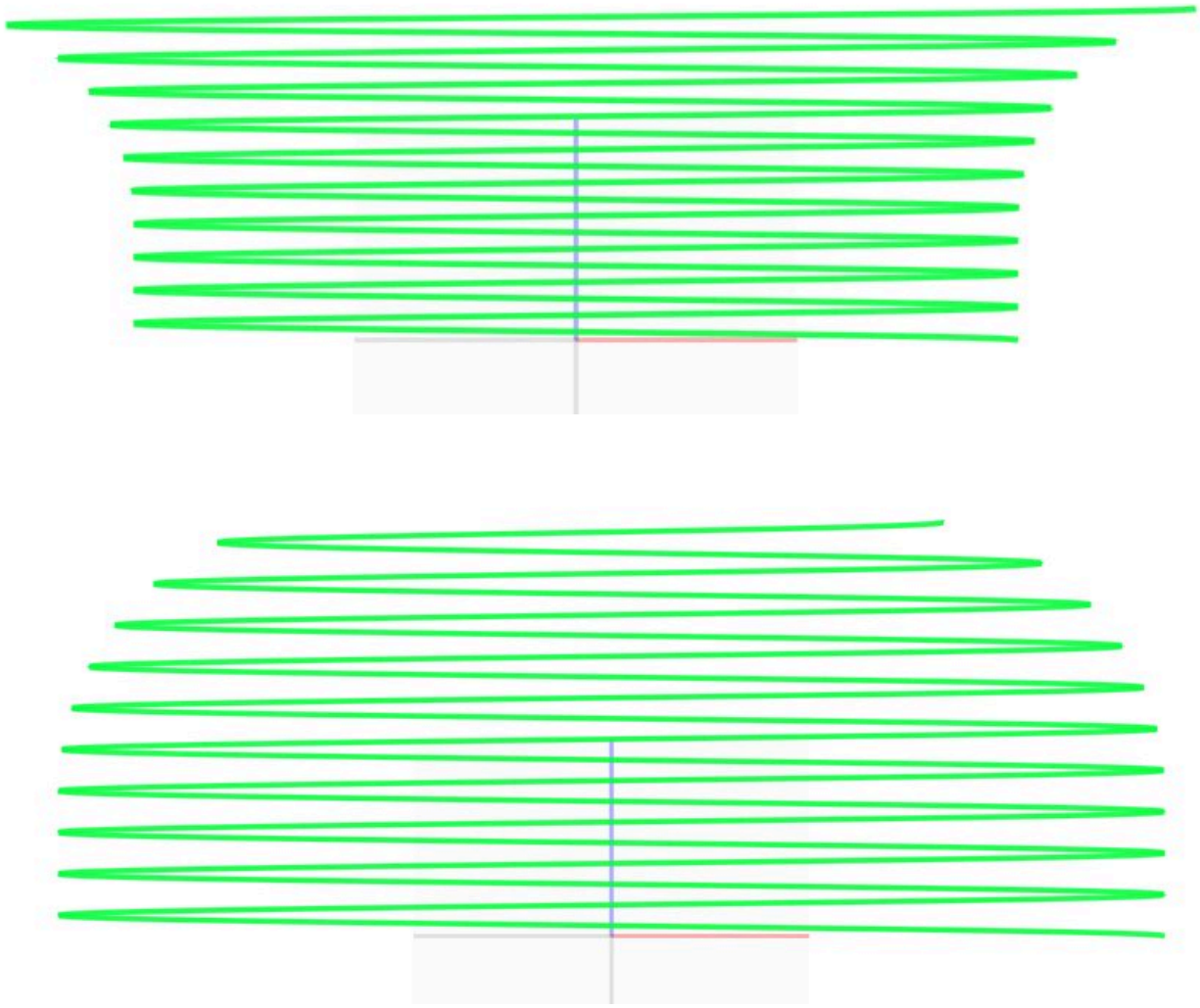

80
Ø35
-7°



Start Runout. Radius Exit for the Start diameter. A positive value increases the diameter outward from the base diameter, while a negative value subtracts twice the exit radius value from the base diameter.

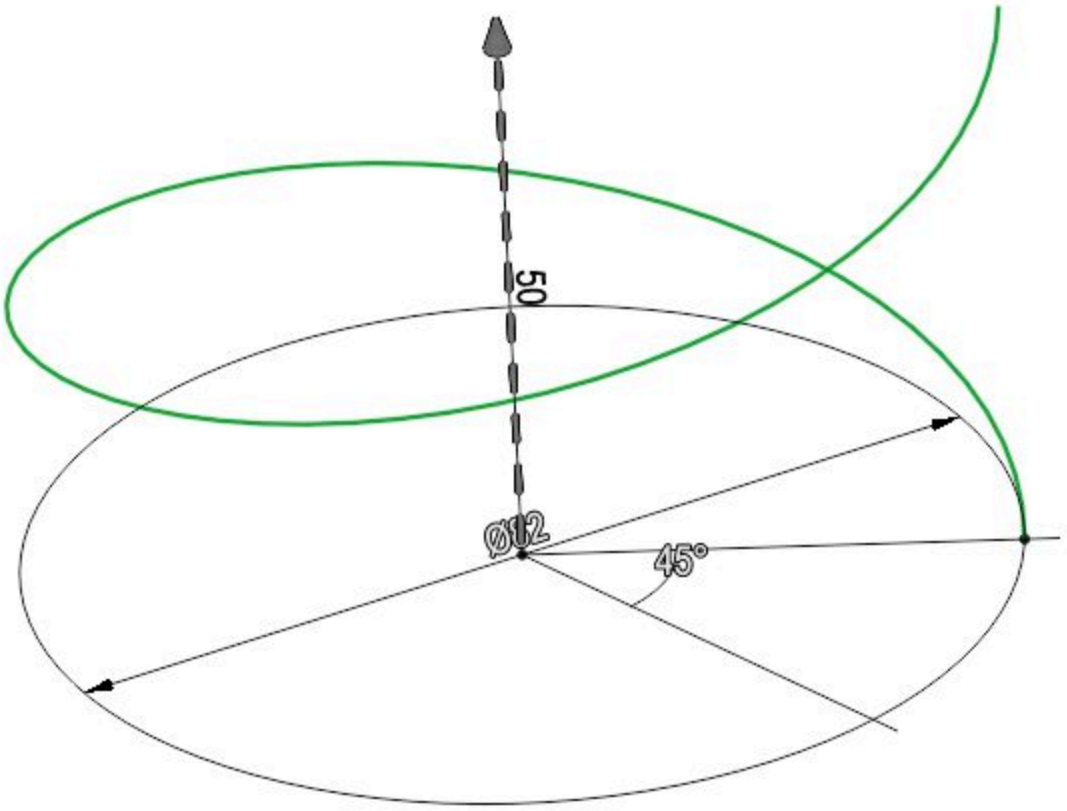


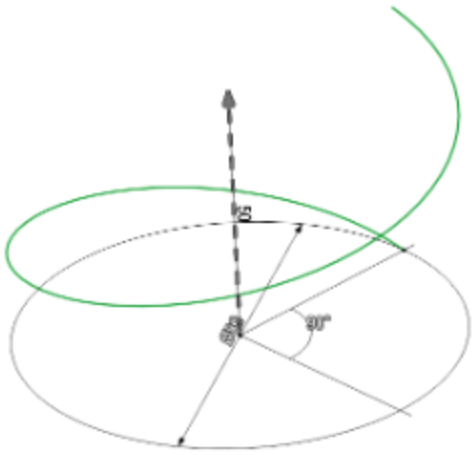
End Runout. Radius Exit for the End diameters. A positive value increases the diameter outward from the base diameter, while a negative value subtracts twice the exit radius value from the base diameter.



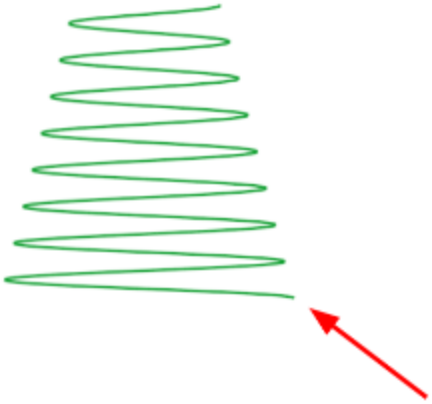
Diameter. This option allows you to change the diameter of the helix.

Start Angle. This option allows you to change the angle of the starting point relative to the center of the helix.

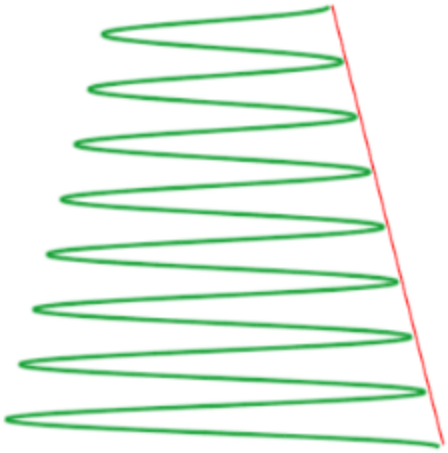




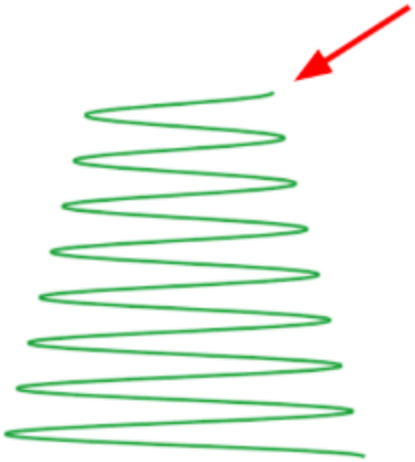
Start diameter. This option allows you to change the lower diameter of the helix.



Cone agle. Determines the angle of the spiral.



End diameters. This option allows you to change the upper diameter of the helix.



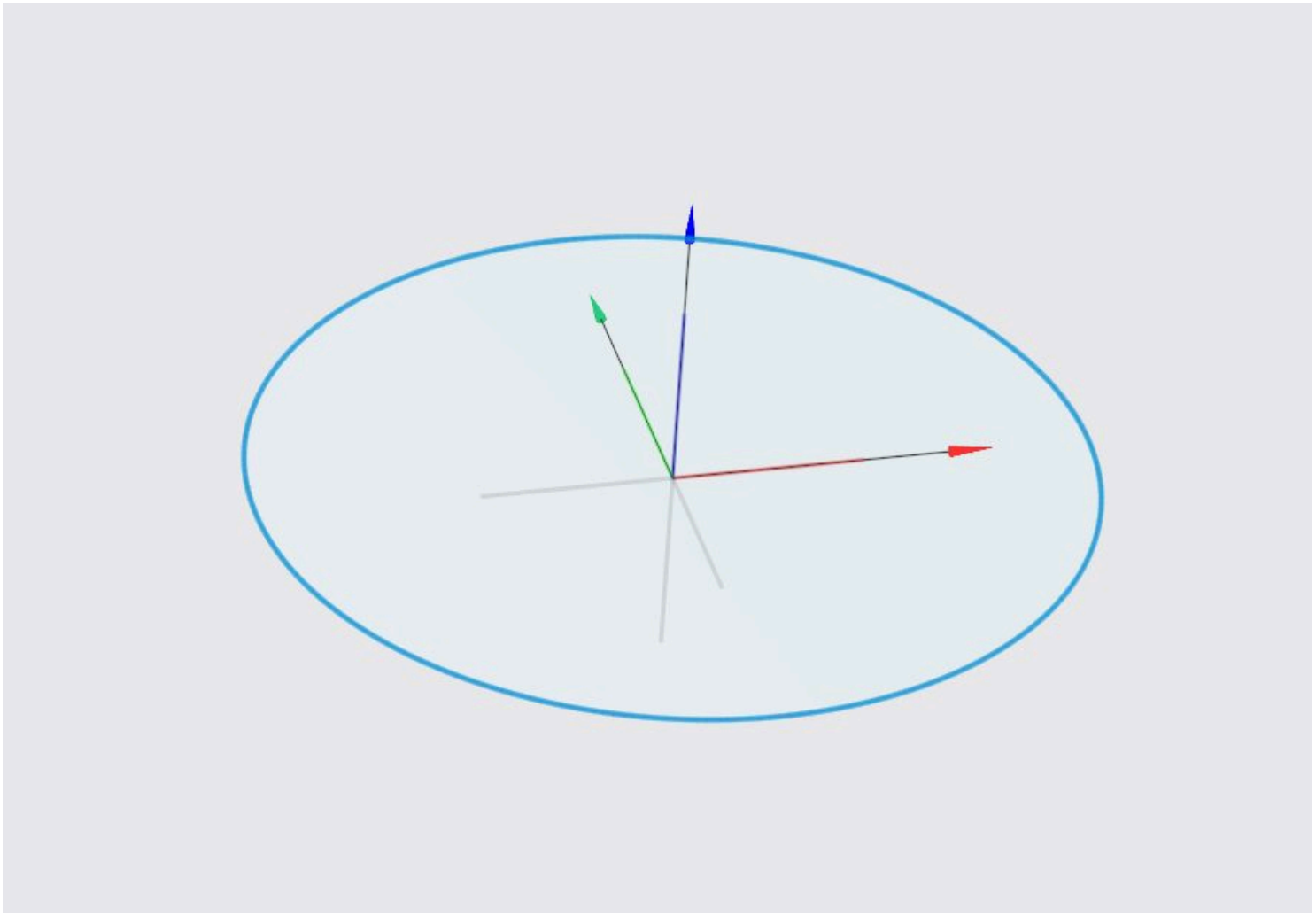
Sketch. Allows you to create a sketch in the selected plane

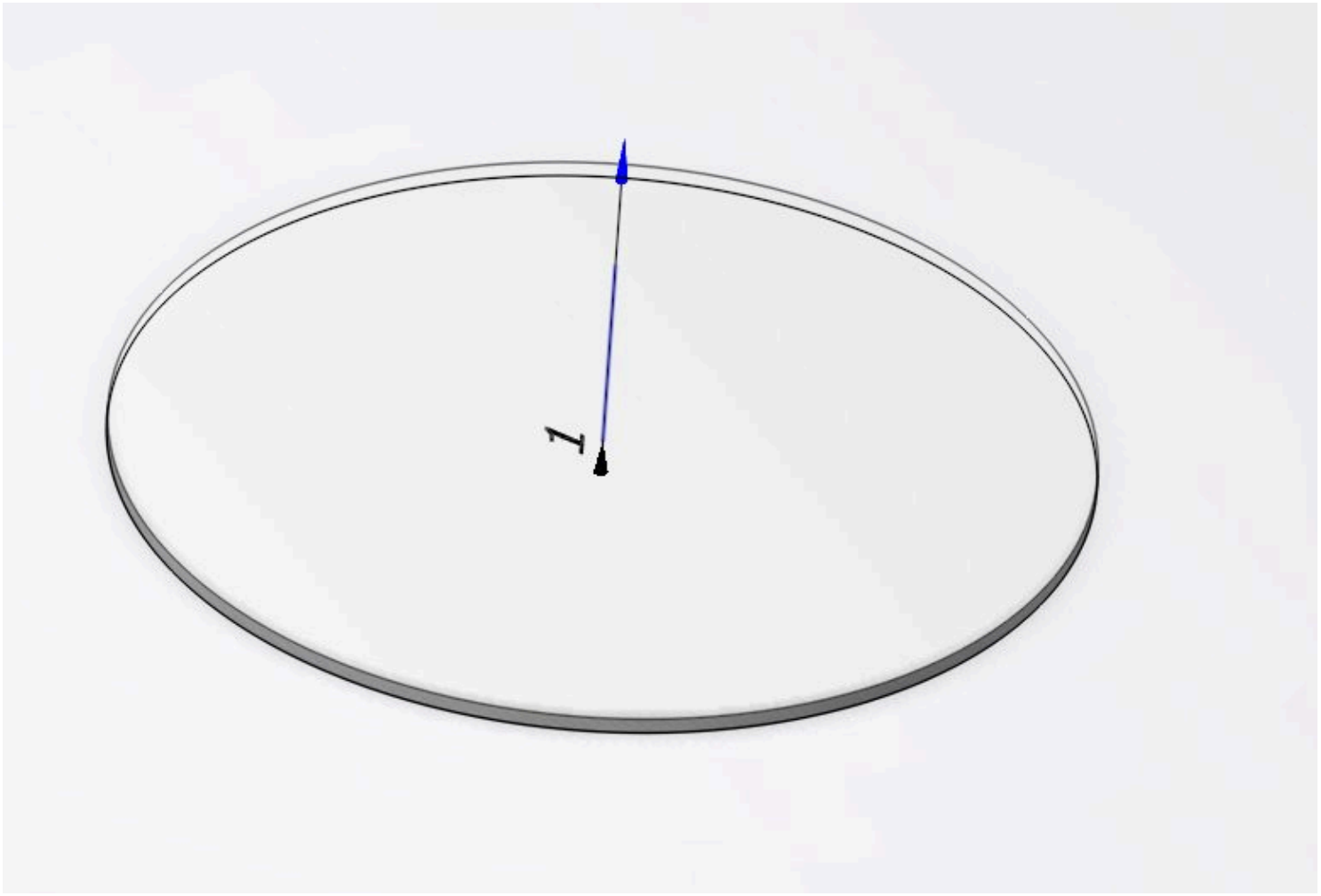
Extrude. Extrude areas of the sketch to create a prismatic shape.

▼ Click here to expand..

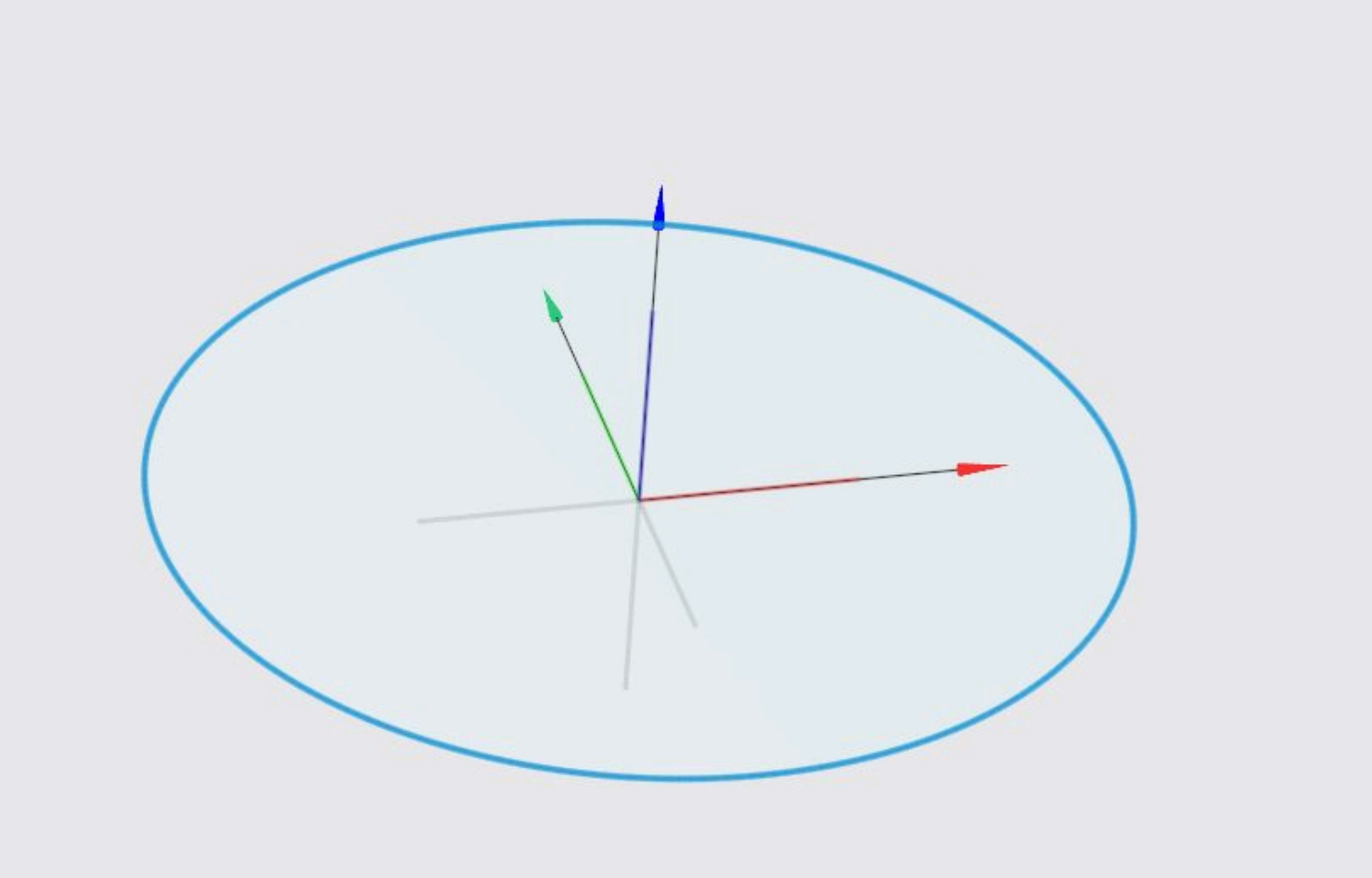
1. It is necessary to select a closed contour of the drawing

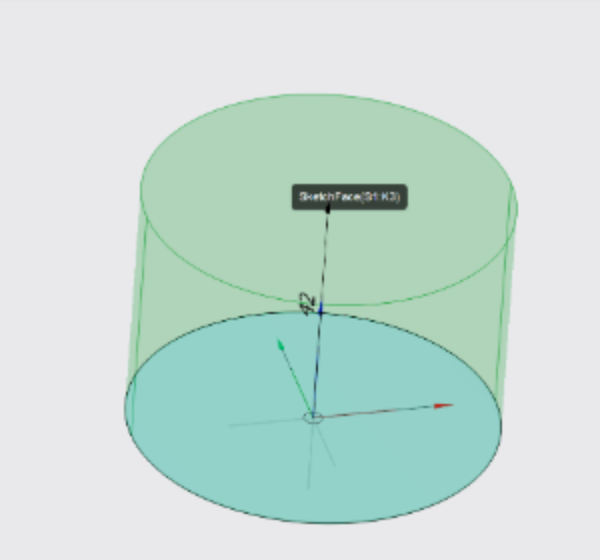
2. On the toolbar, click the button Extrude



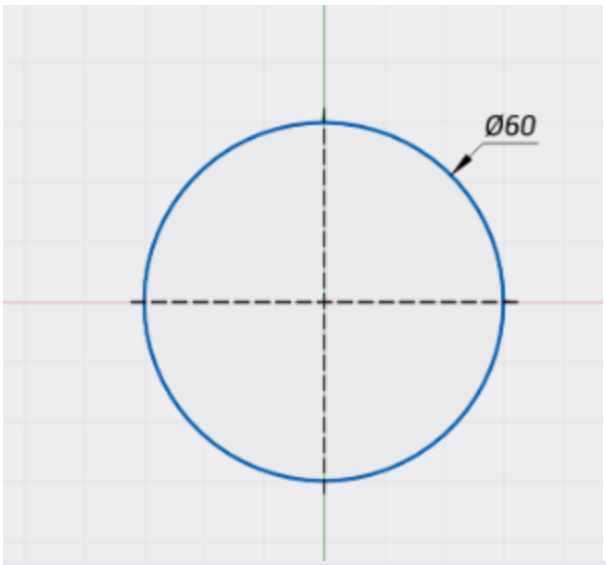


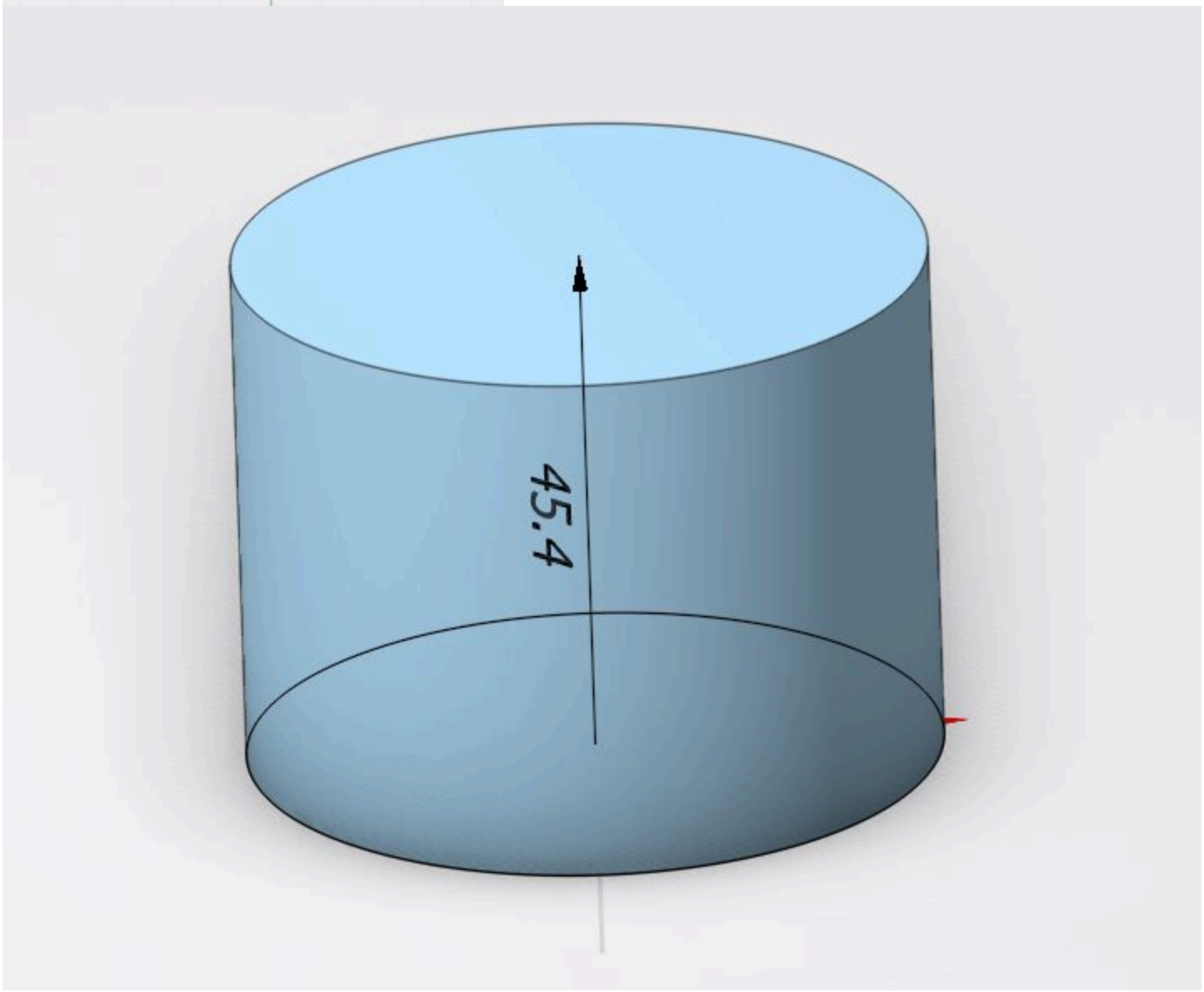
For quick model building, it is better to select closed contours and pull them out interactively (by pulling in the desired direction)

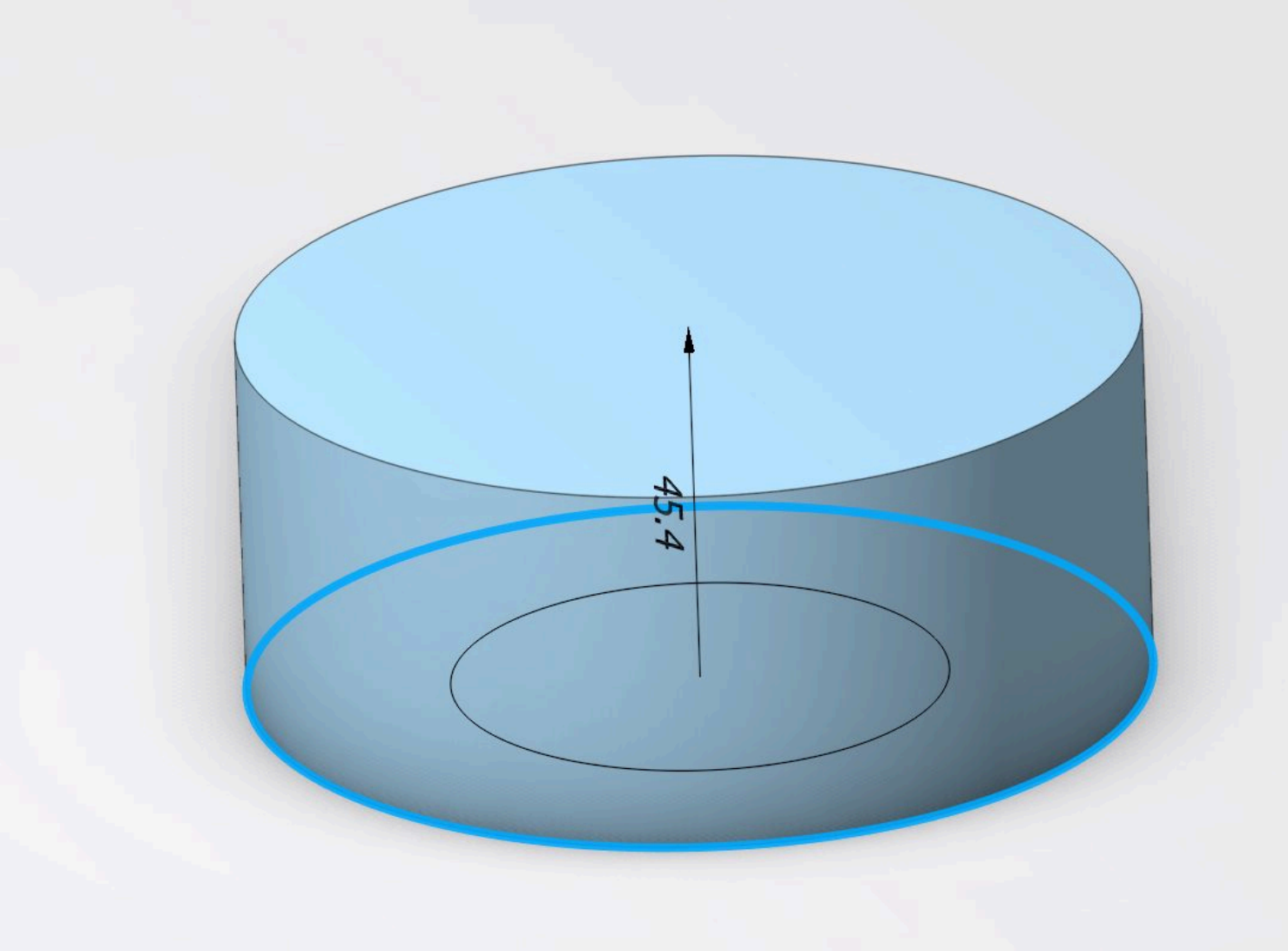


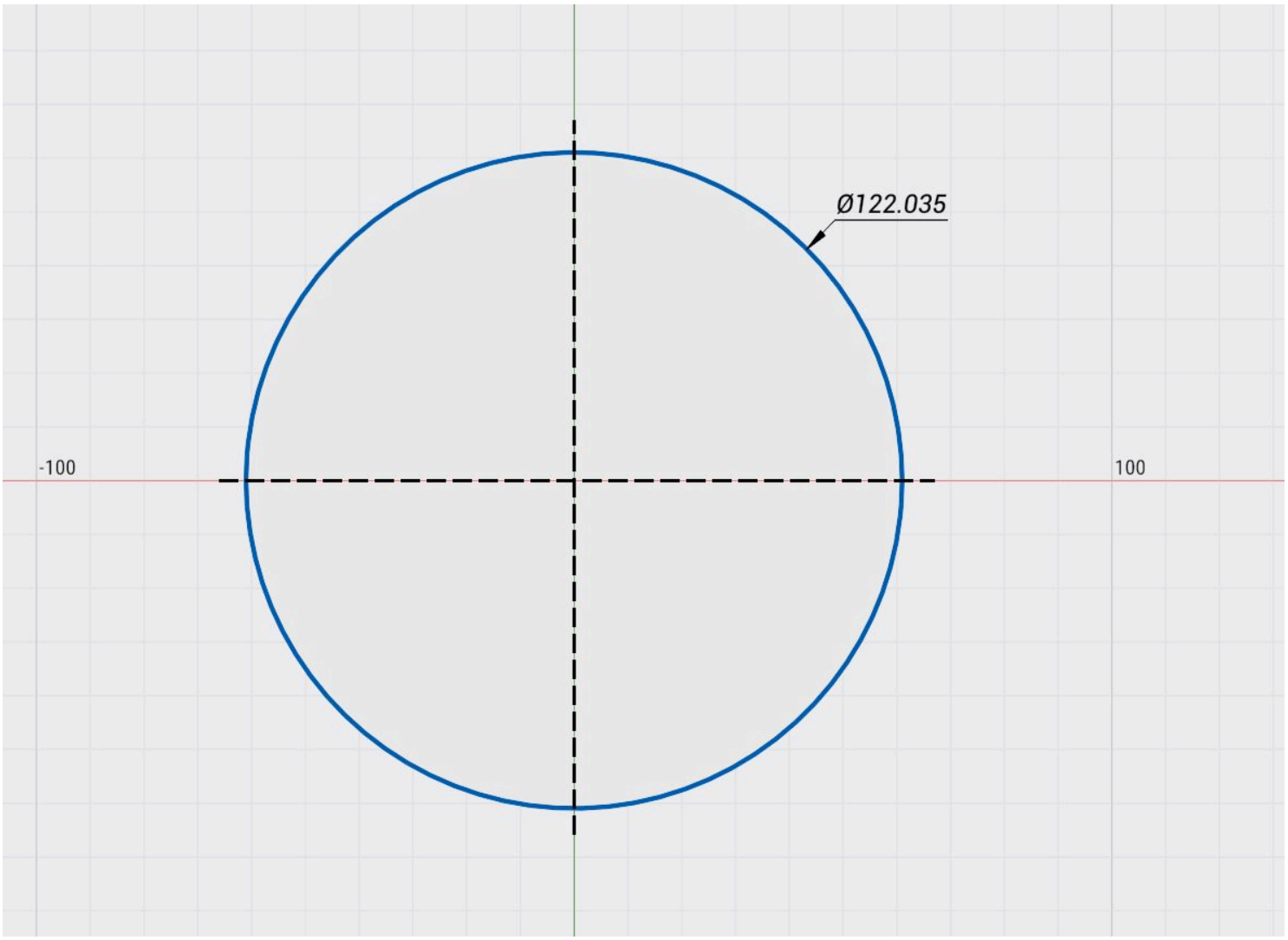


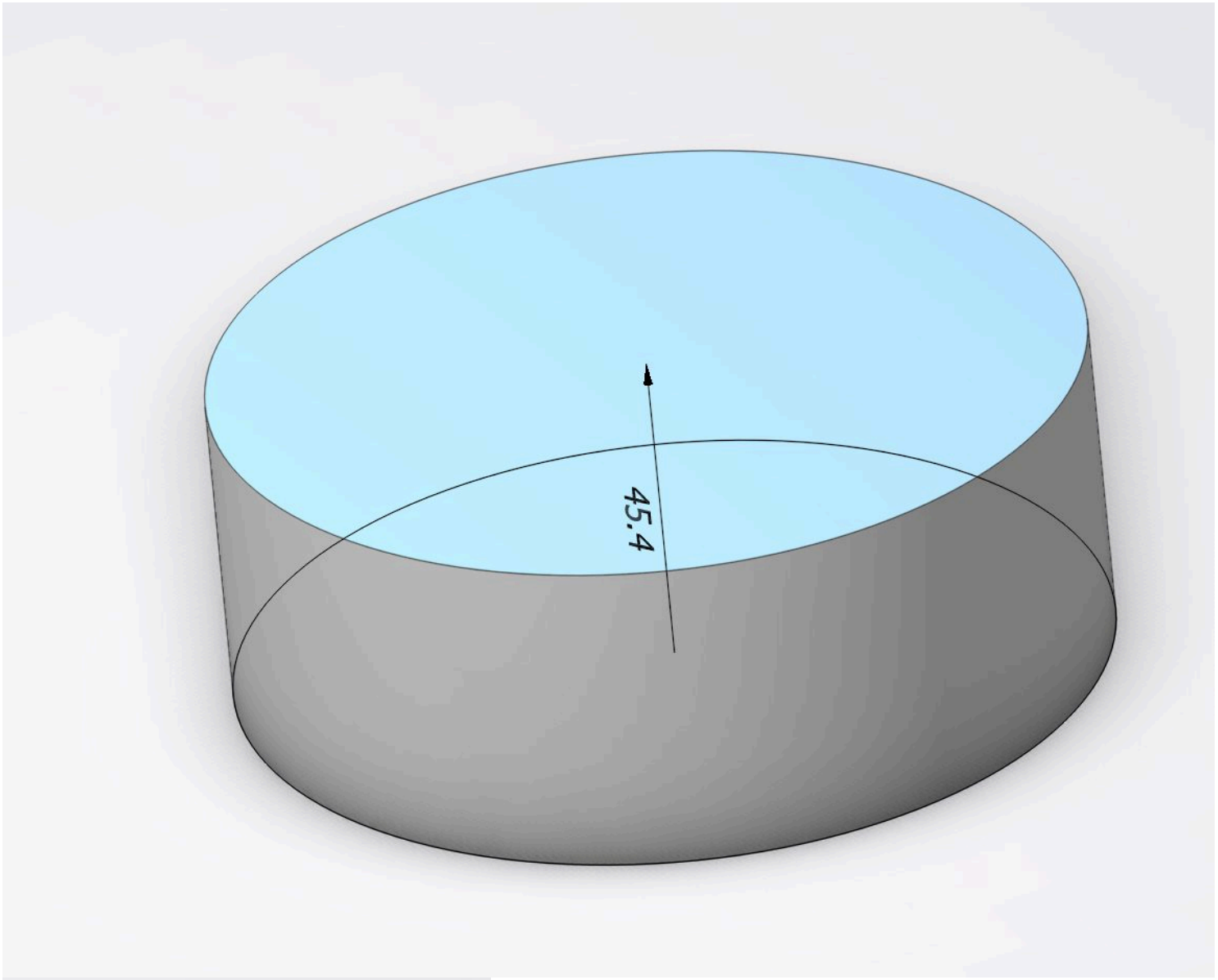
You can also change the dimensions of the model at any time without going over to the drawing. To do this, you can simply grab the edge and move it interactively. This will change the dimensions in the drawing.

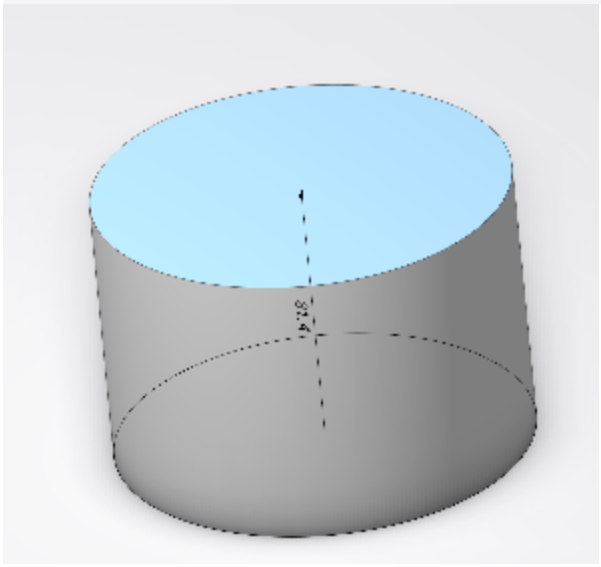

Ø60


45.4


45.4


Ø122.035
-100
100



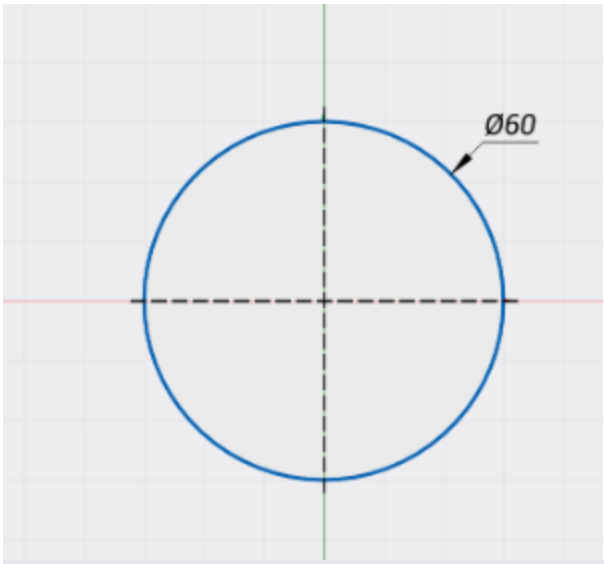


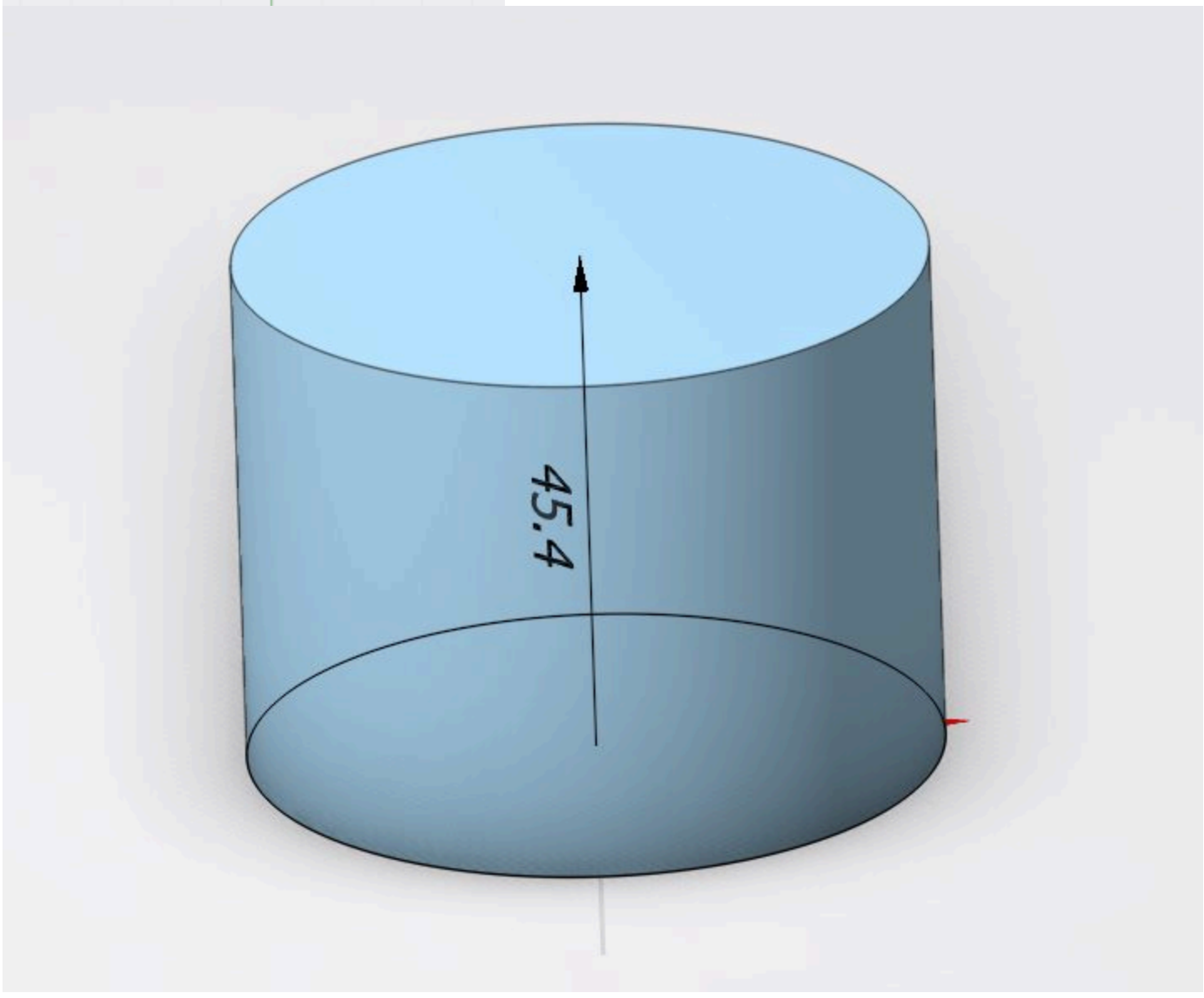
Boolean operations.

New part. Allows you to create new parts. Details

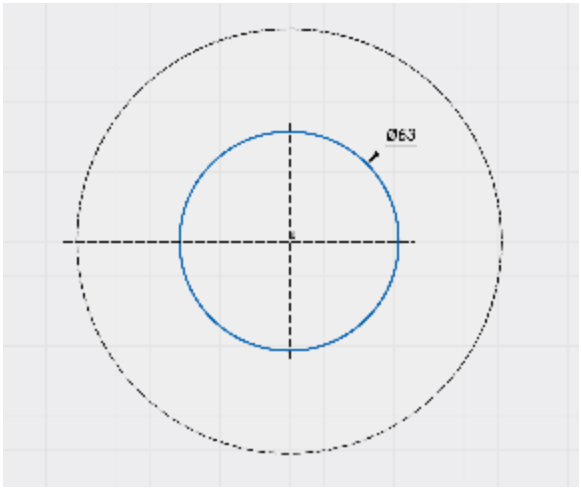
Auto. Automatically selects what needs to be done (subtract or add).

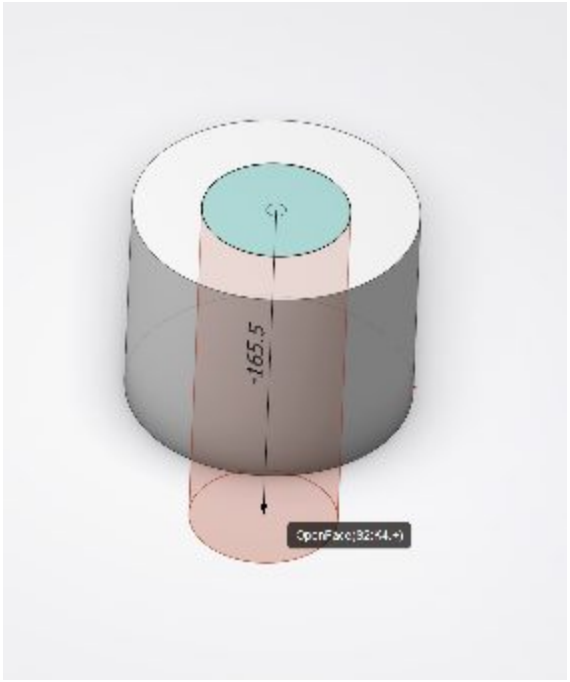
Add. Allows you to pull out drawing elements.

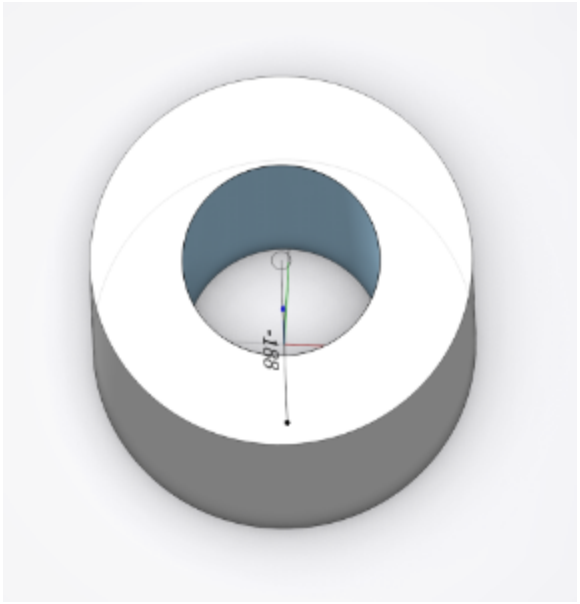




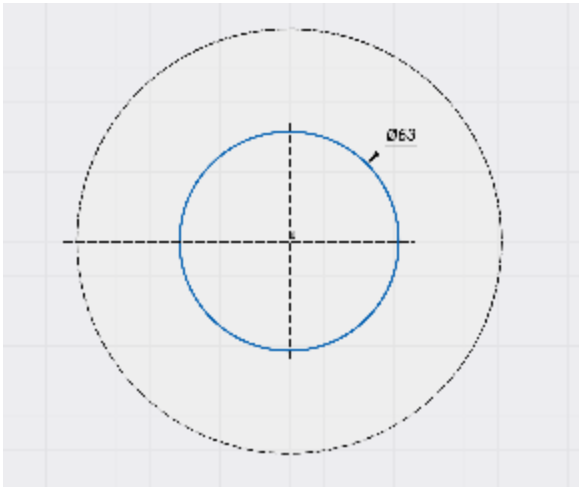
Subtract. Allows you to subtract drawing elements from an existing body.

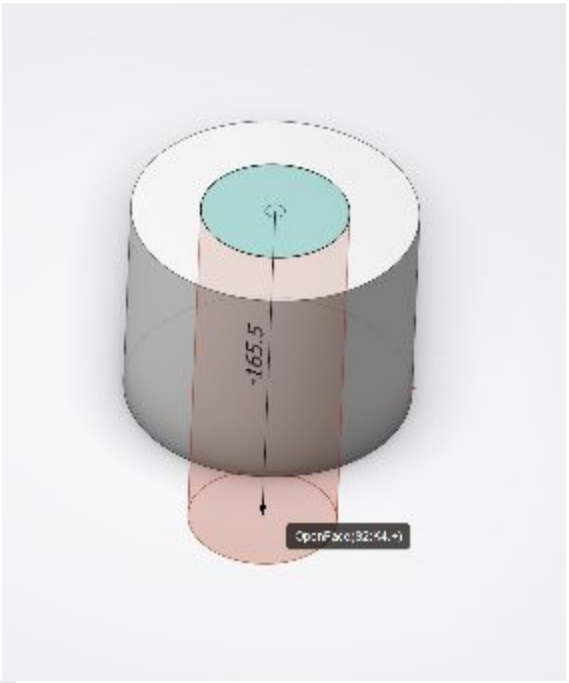


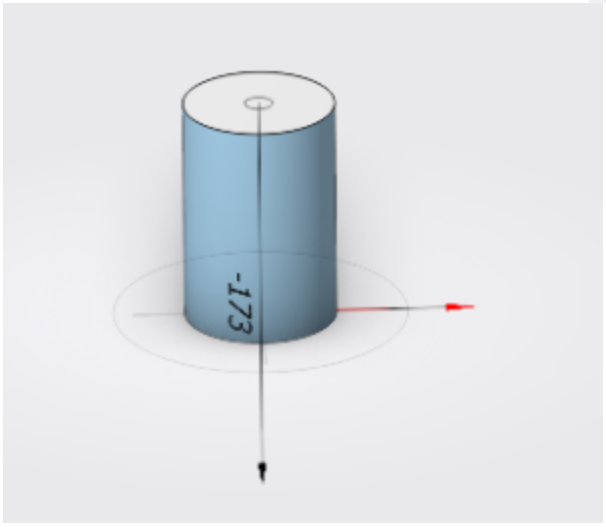




Intersect. Allows you to create new geometric objects based on the intersecting areas of existing objects.





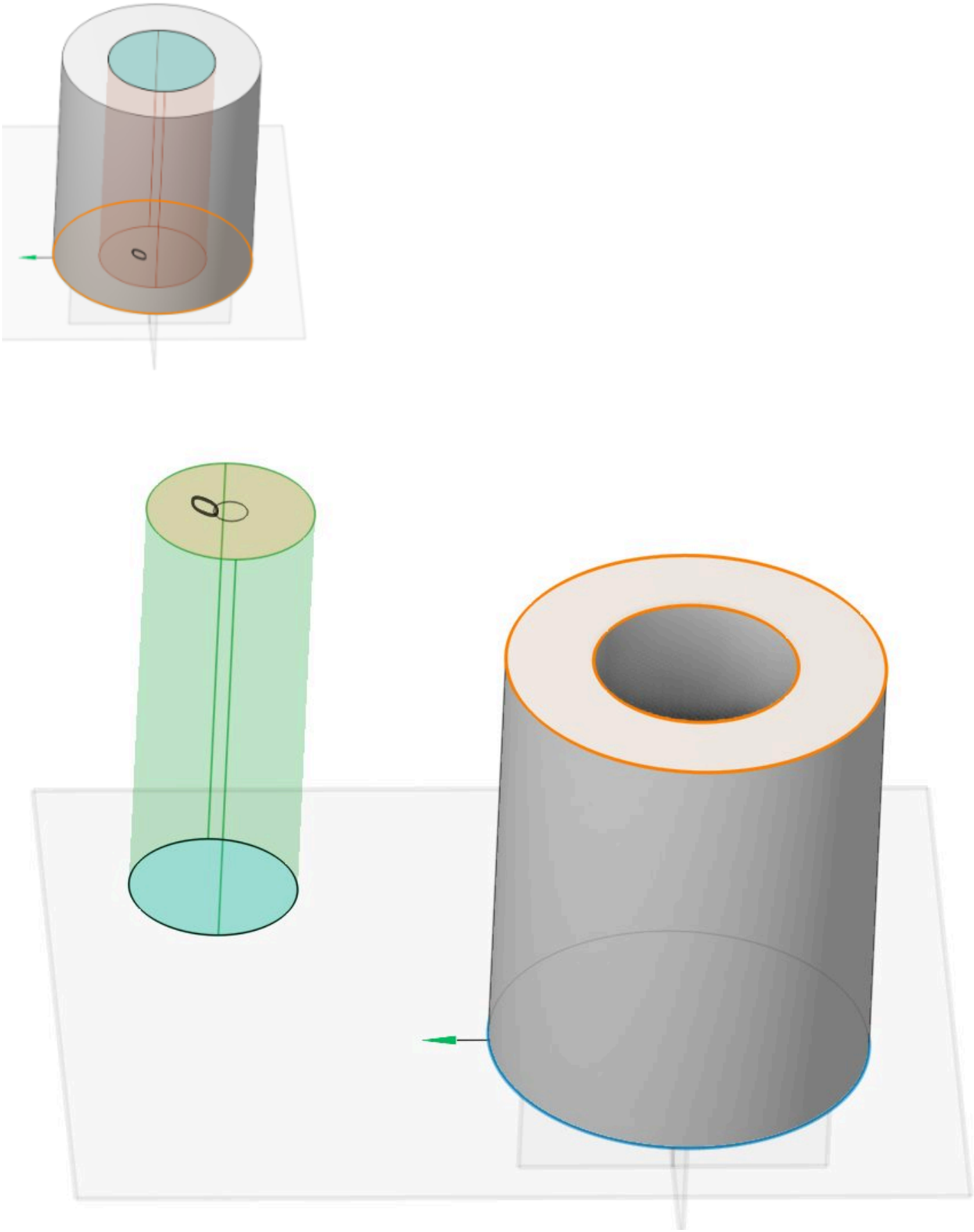


Part. The name of the part to which this operation applies.

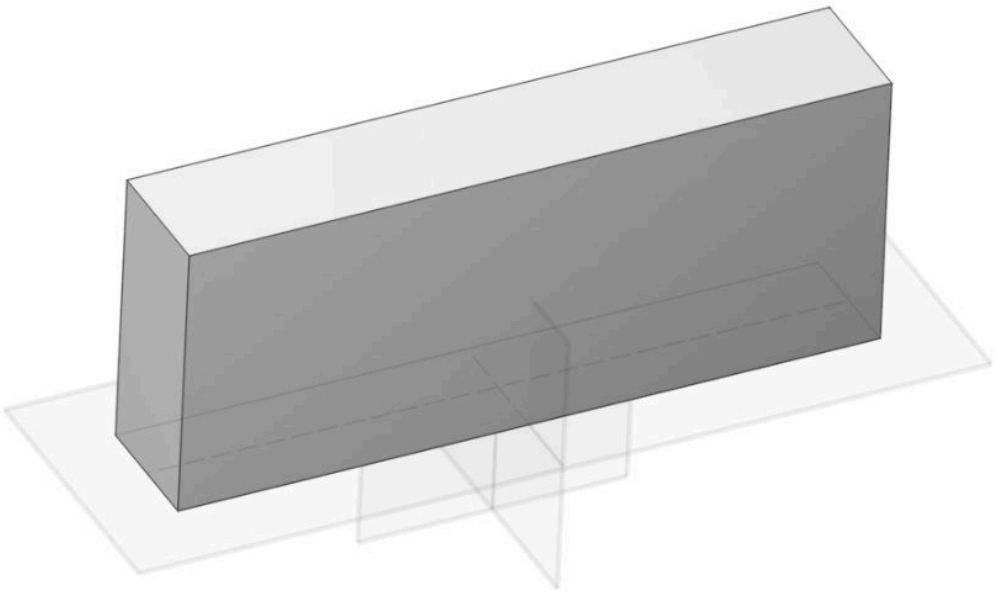
Profile. Defines the profile of the drawing

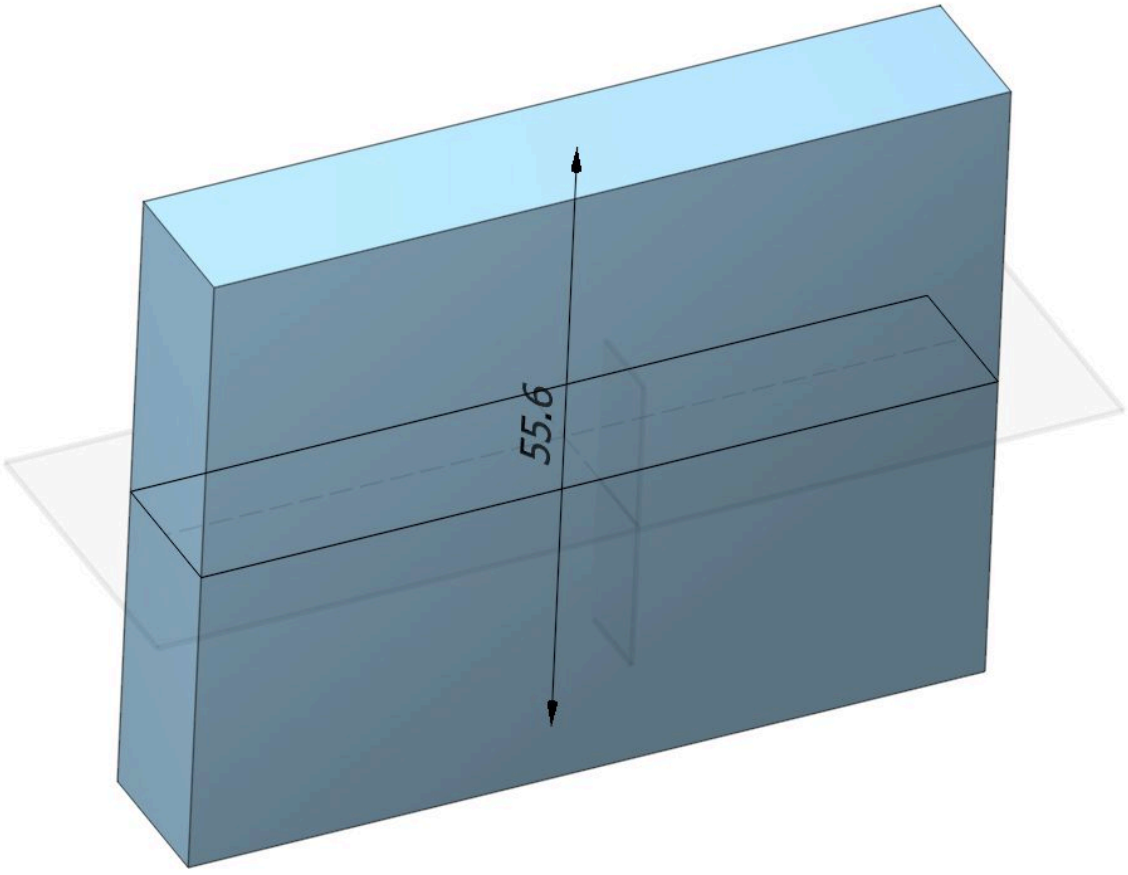
Distance. Determines the distance of extrusion and subtraction. Possible to change interactively.

Also, when subtracting or adding, it is possible to snap to different surfaces of already extruded elements. In this case, the Distance value will be equal to 0.

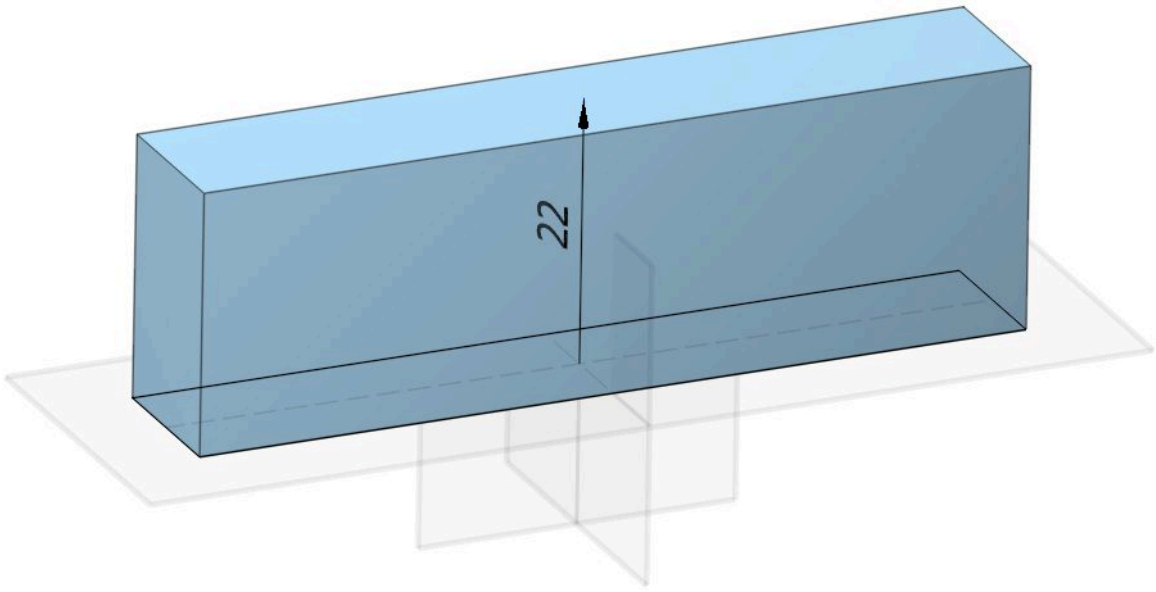


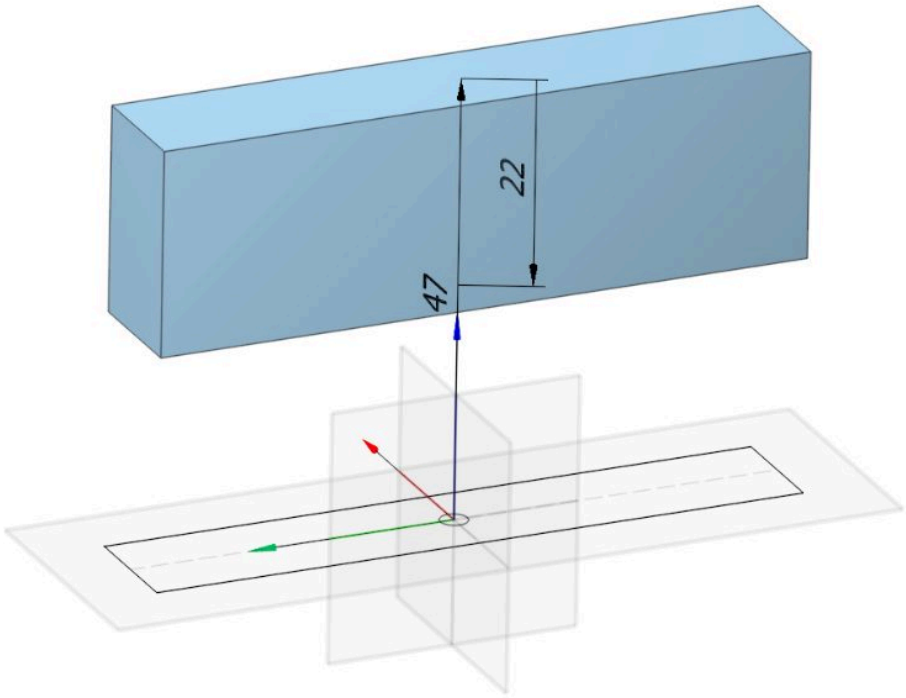
Symmetrical. Extrude symmetrically both sides from the sketch plane.



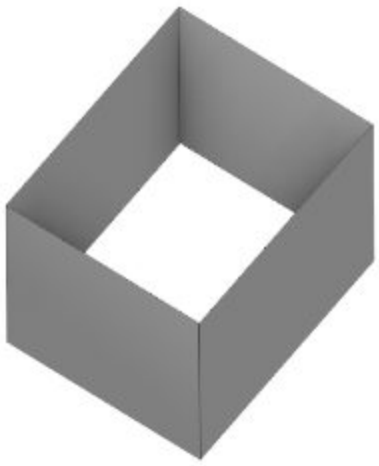


Thickness. Thickness of the extrusion relative to the top level.



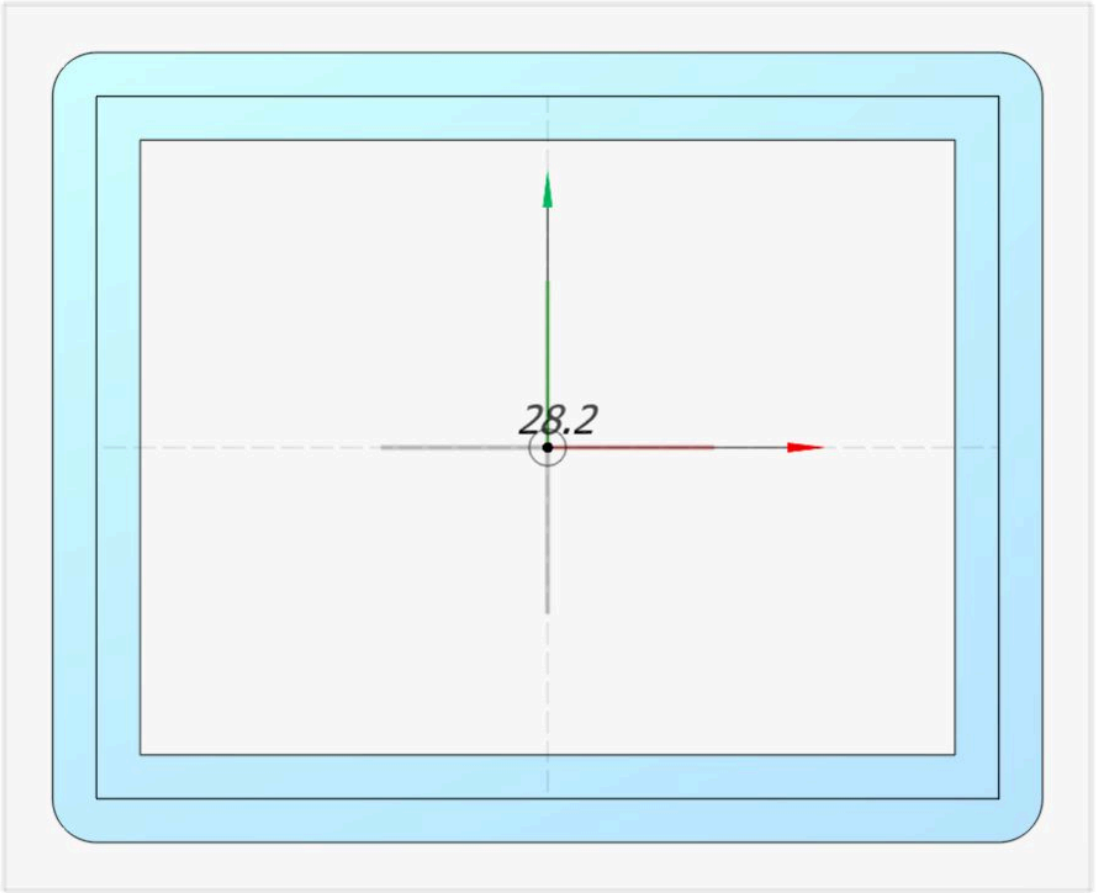


Thin Wall. Allows you to create thin-walled elements.



The direction of adding thickness contains the following options (in the form of pictograms):

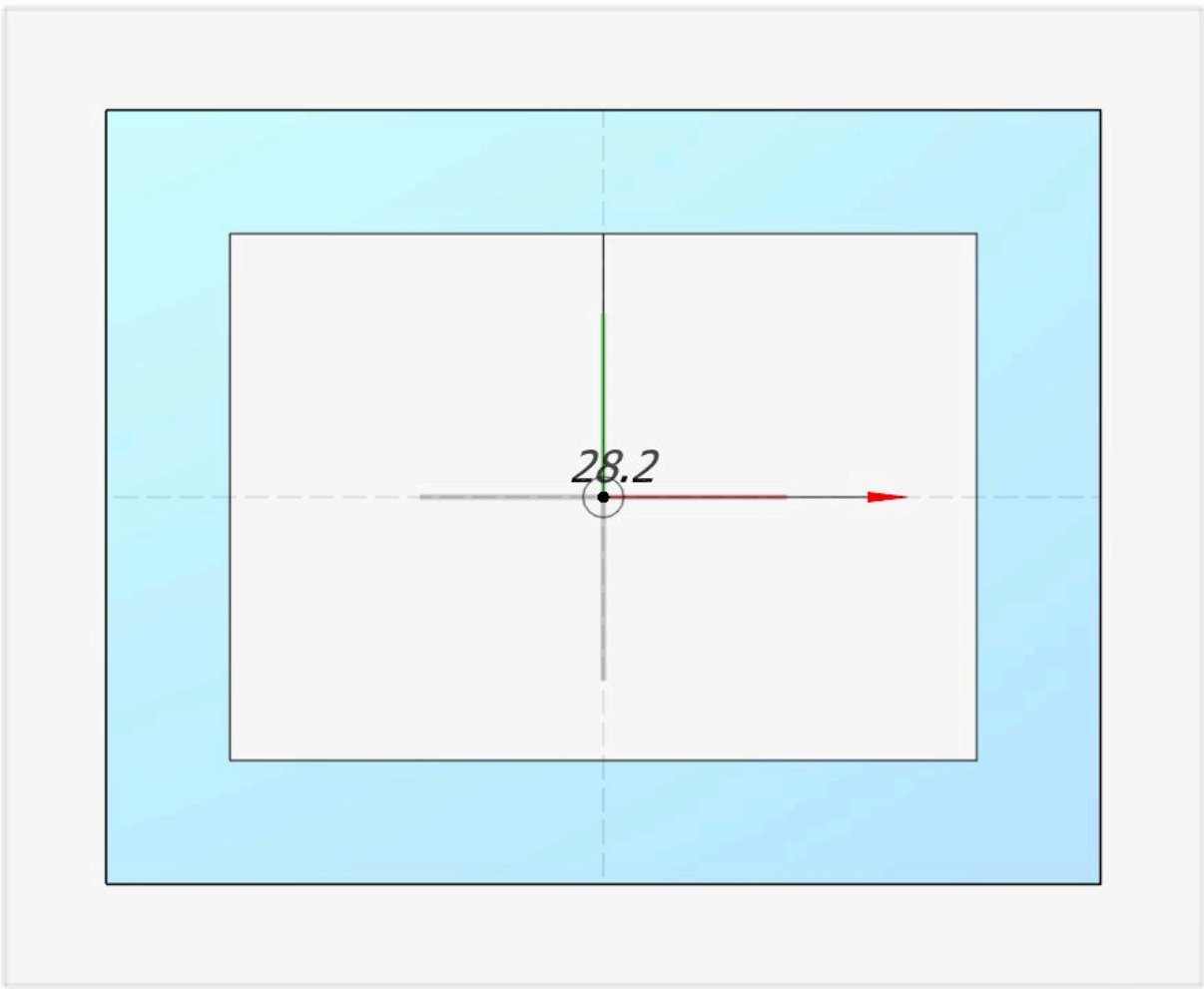
<> - in both directions,



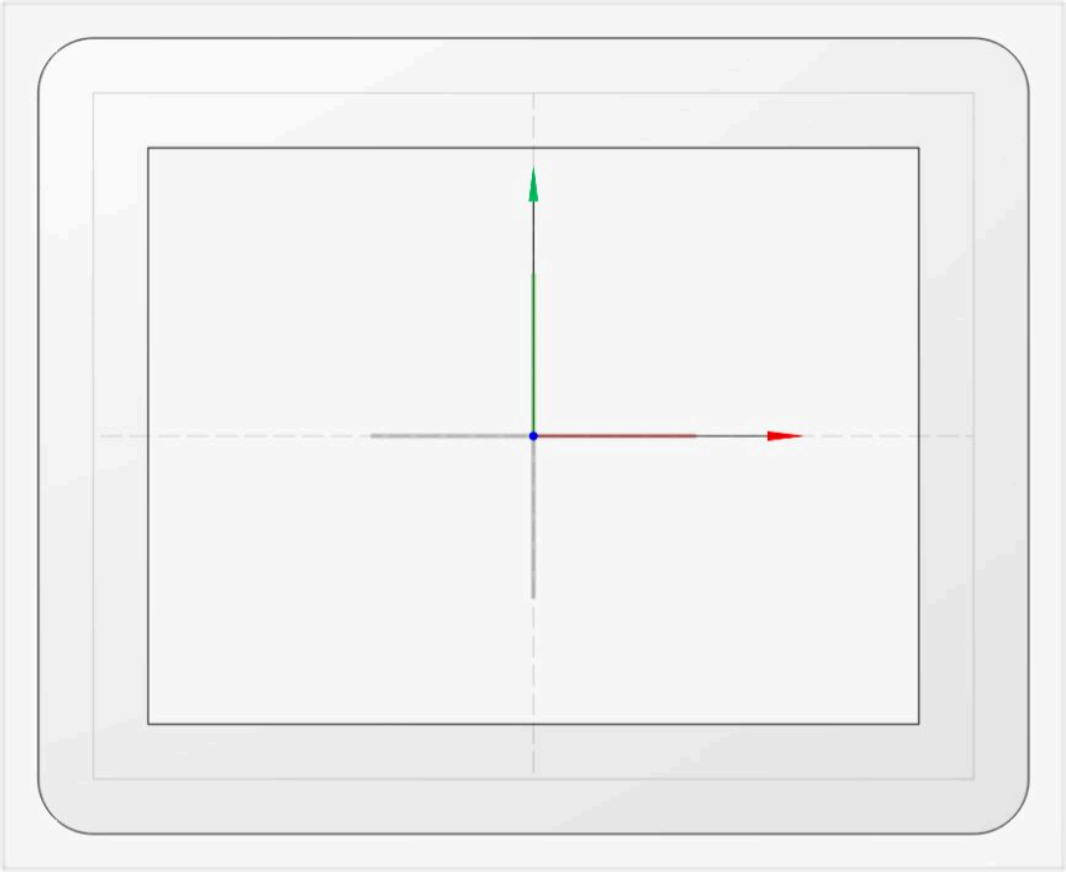
-> outward,

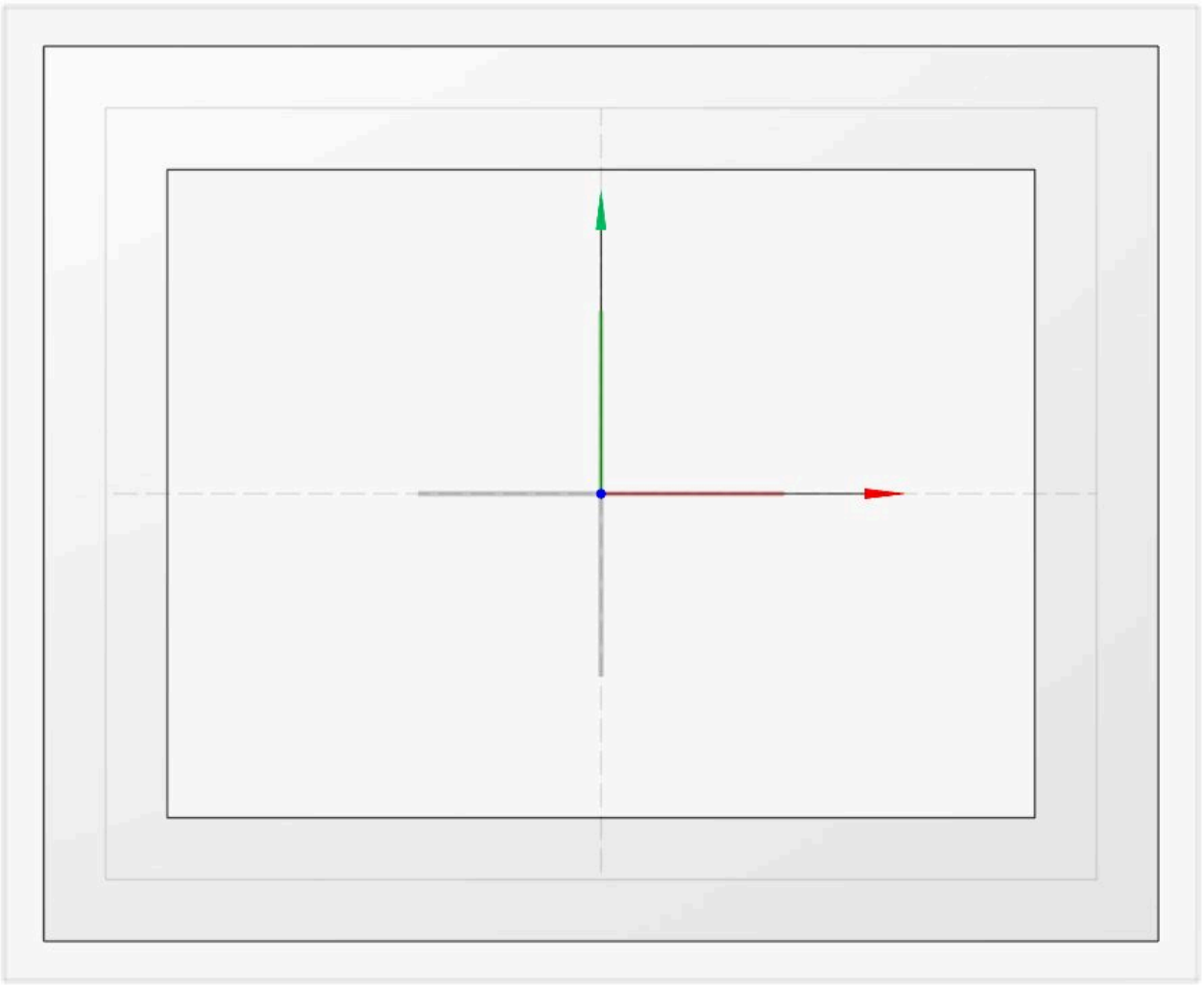


<- inward.



Corner rounding provides sharp or rounded corners for the edges of the outer contour.





Wall thickness input field (can be zero). If the thickness is non-zero, a closed 3D body is formed to which all Boolean operations are applicable.

Twist. Twisting along the normal to the sketch. The parameters are the same as in Helix. For an extrude operation, the twist is performed around the Z axis of the sketch.

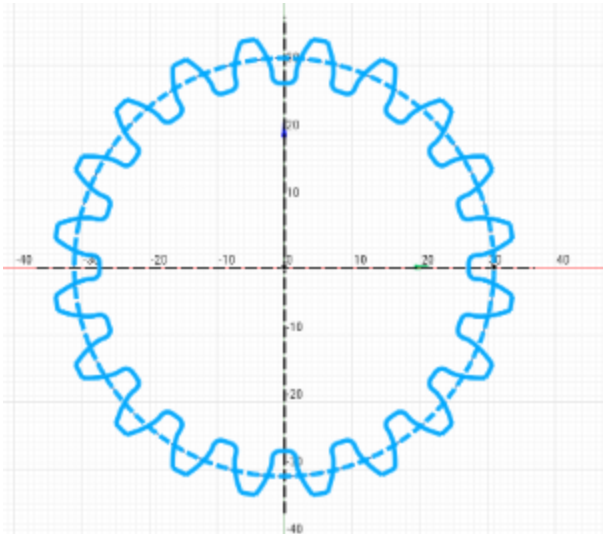
When using the By Pitch option, the step is controlled by the Pitch parameter.

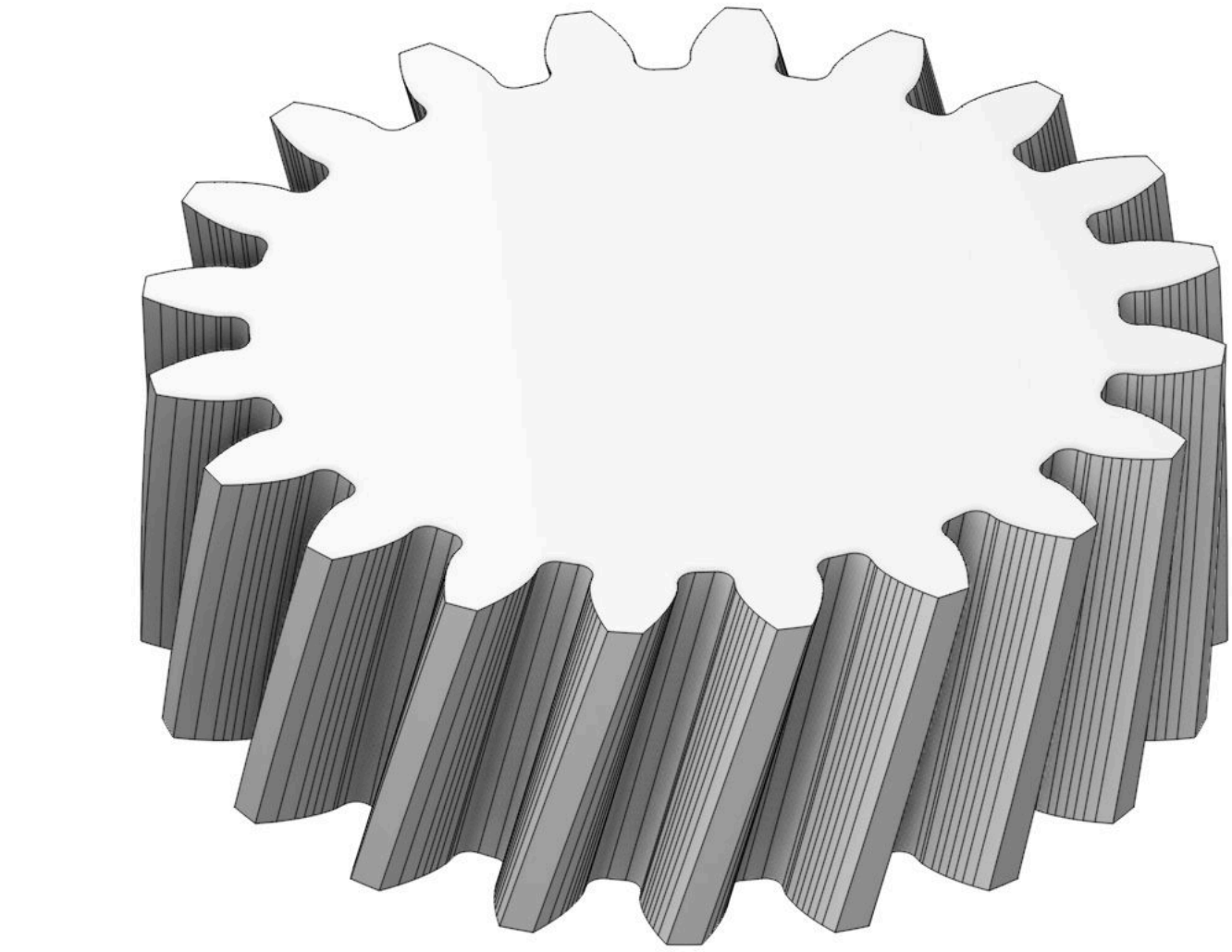
With the By Angle option, the length is controlled by the Total Angle parameter.

For the By Count option, the step is controlled by the Number of Threads (NB of Threads) parameter.



When pulling out a gear profile constructed in a sketch (Details) as a block with a given value of the helical angle of the teeth, the twisting is applied automatically, and its parameters are taken from the sketch - Helic angle.



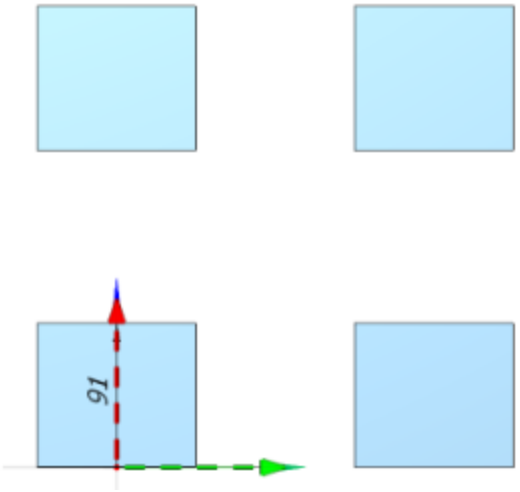


There is no option to disable the twisting. If this is necessary, the corresponding block should be destroyed in the sketch and then the operation repeated.

Pattern. Allows you to create a template for copying elements.

Type. Defines the template type.

Linear. They represent a two-dimensional array.



Defined by. Distance definition method: Extend or Spacing.



Number. Nurmber of copies along the first axis.

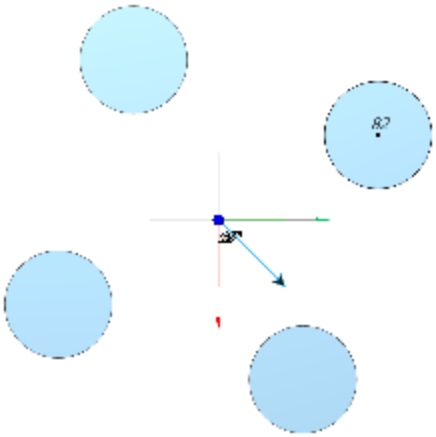
Distance. Distance along the first/second axis

Direction. Copy direction.

Second direction. Number of copies along the second axis

Inherited. Allows you to use another breeding template

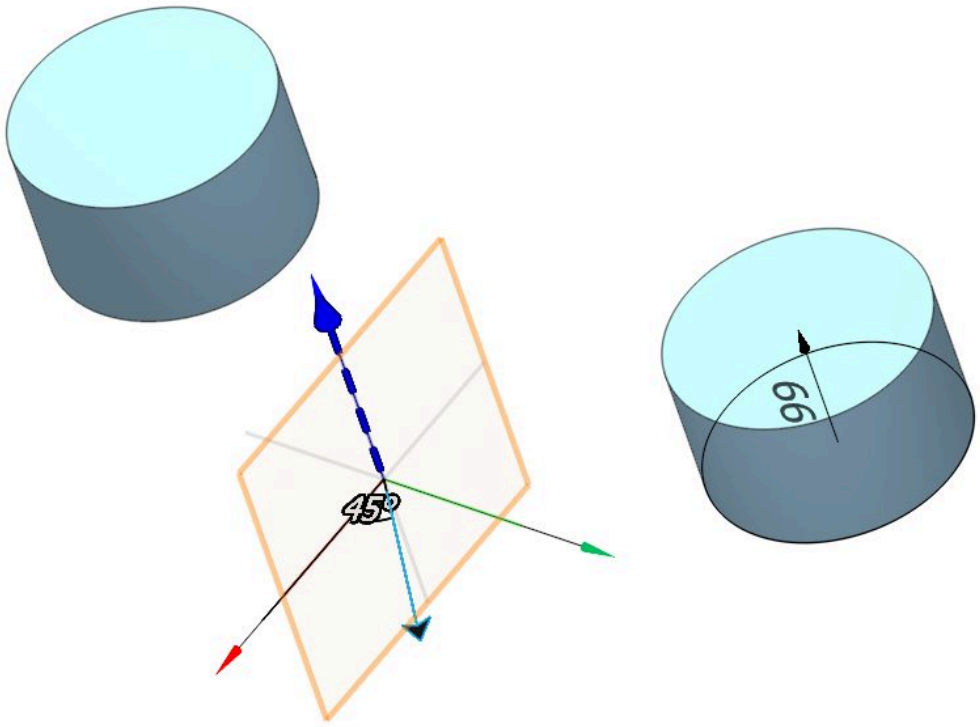
Circular. Allows you to create a circular array



Angle. Angular span of the pattern

Along axis. Copies along the Z axis

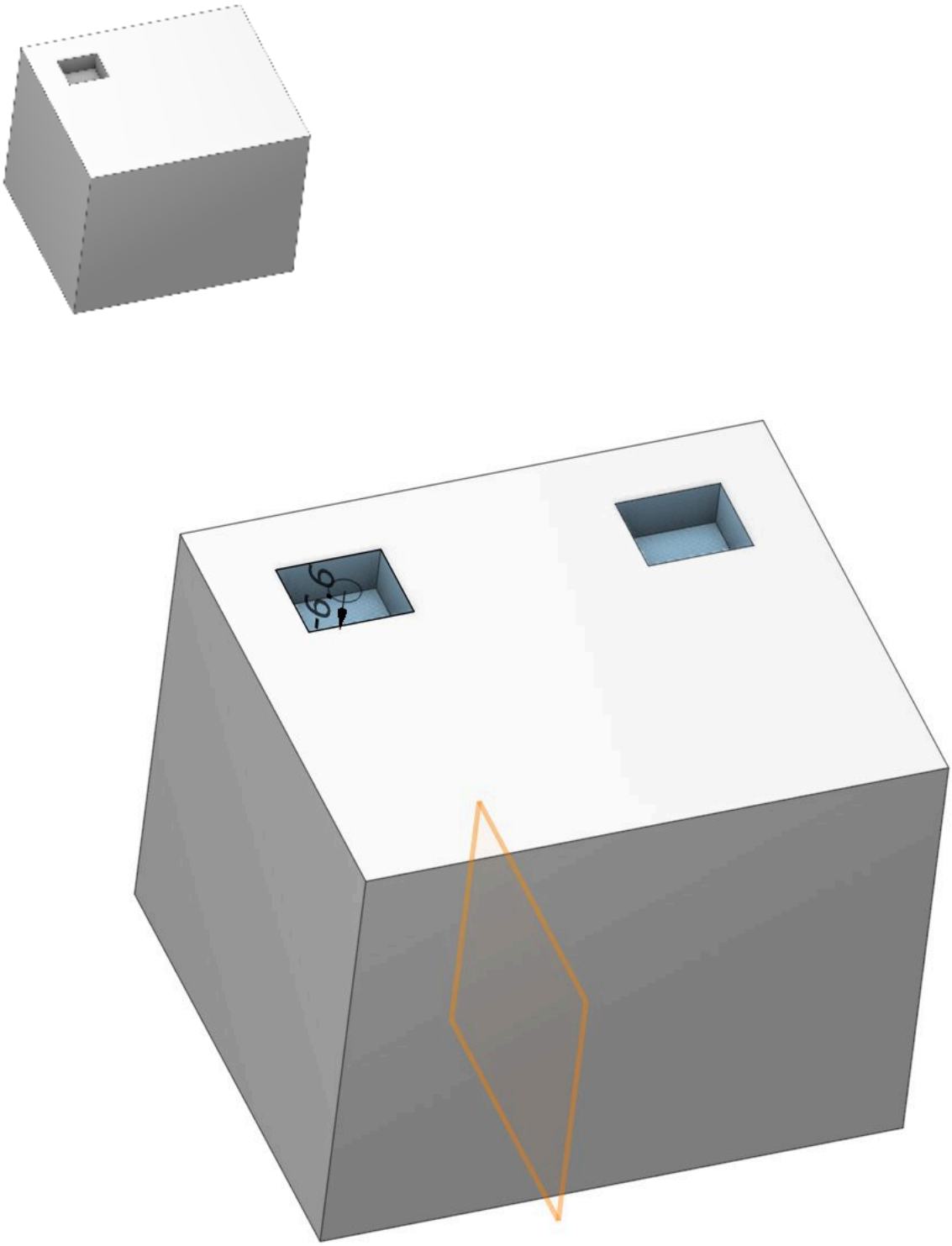
Mirror. Allows you to create an element relative to the selected plane.



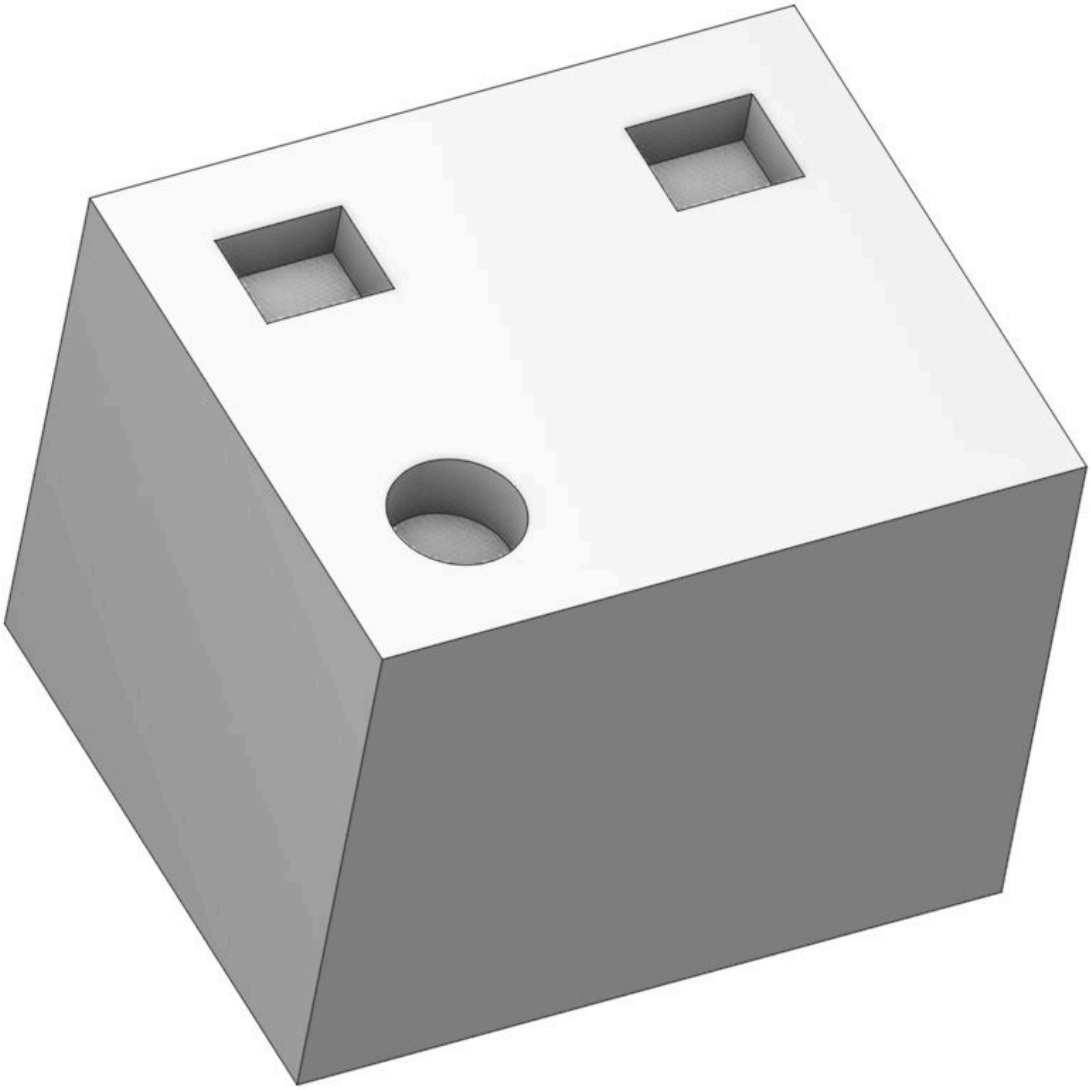
Plane. By default, 3 base planes (XY,YZ,ZX) are available. You can also add your own plane before extrusion and use it in this functionality.

The new Boolean operation can use patterns that were used in previous Boolean operations.

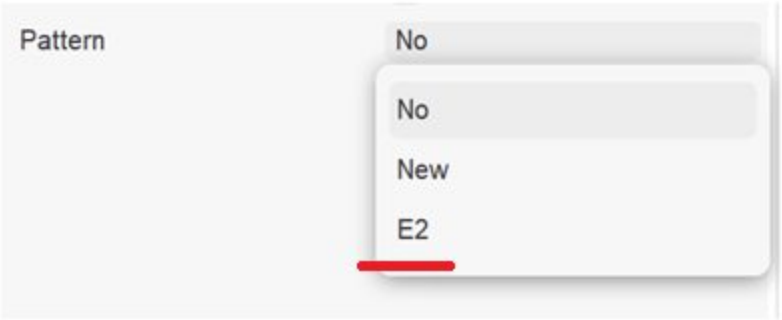
1.For example, we will extrude a prism and use the symmetry reproduction scheme



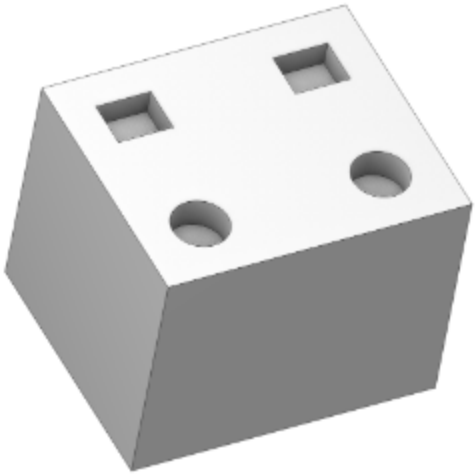
2.Extrude a circle



3.Select the template of the previous extrusion

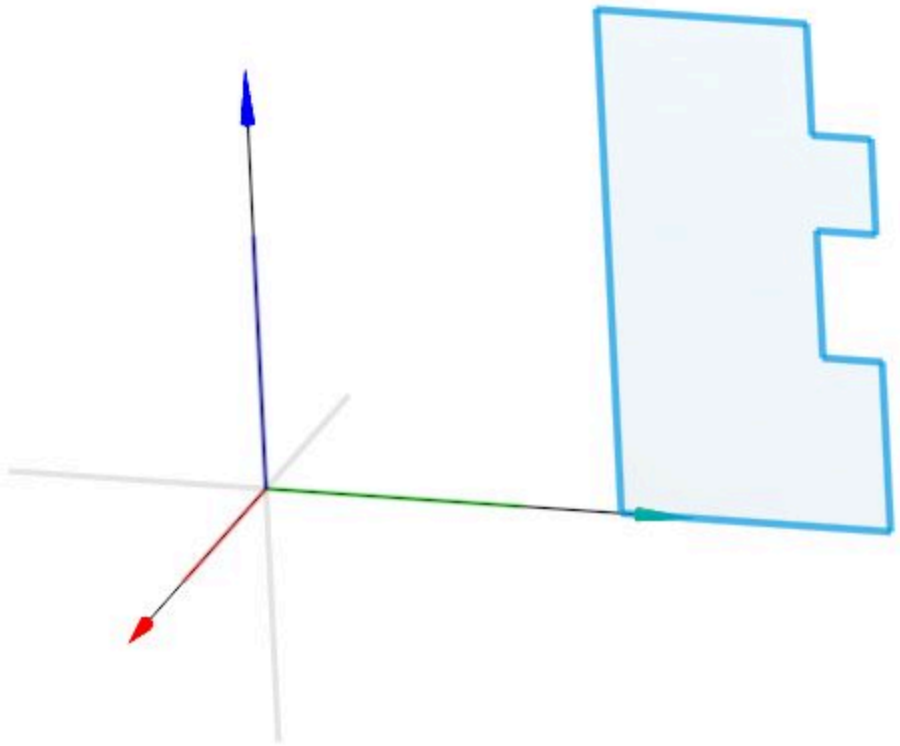


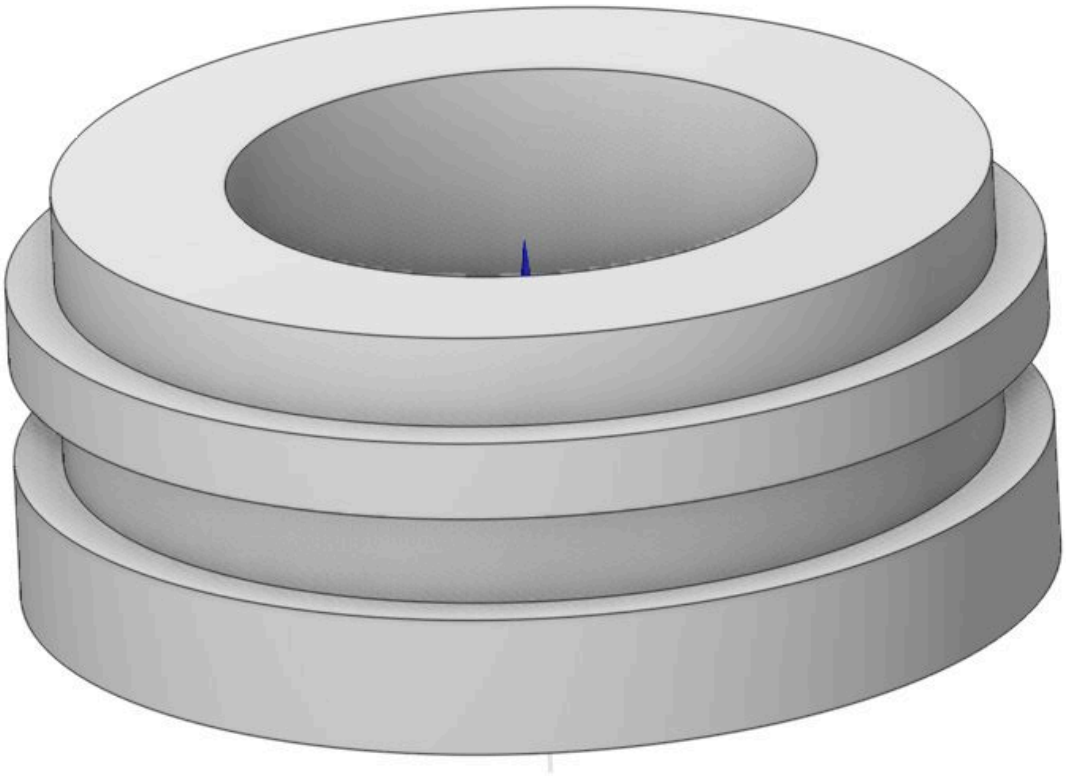
We get a similar result



Revolve. Revolve areas of the sketch around an axis.

▼ Click here to expand..

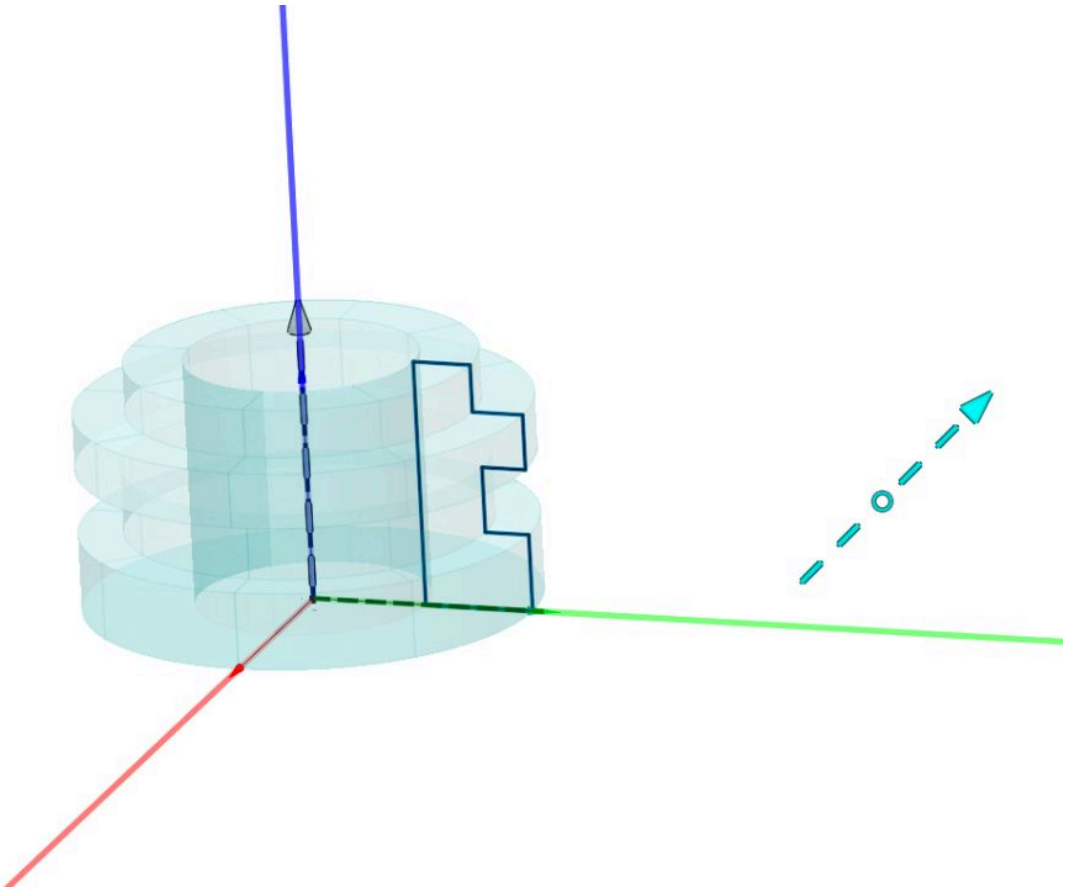




Part. The name of the part to which this operation applies.

Profile. Defines the profile of the drawing

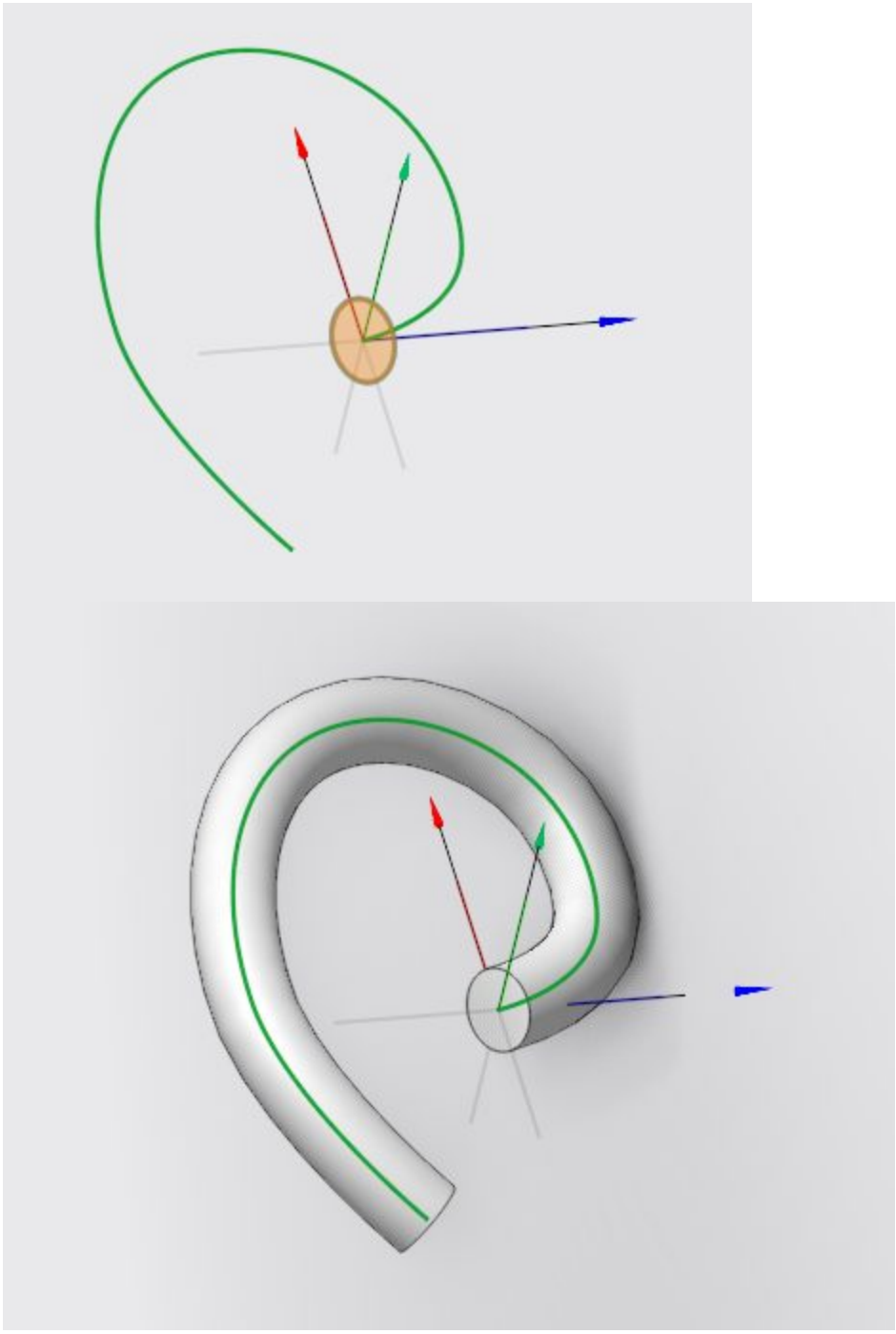
Axis. Rotation axis. Can be set interactively by binding to a curve or base axes of the SC



Thin Wall. Allows you to create thin-walled elements. The functionality is described in Extrude

Pattern. Allows you to create a template for copying elements. The functionality is described in Extrude
Sweep. Stretching the profile along the trajectory. The profile can be a closed or open outline sketch (Profile). The contours of a sketch, spatial curves, and edges of bodies can act as a Path

▼ Click here to expand..
1.Call the Sweep command from the toolbar, then sequentially select the sweep profile with the mouse - the sketch outline (the Profile selection field is pressed) and one or more path segments (the Path selection field is pressed). As you add segments, the Path field displays the number of segments, and a preview of the 3D element is generated in the main window.



If you need to exclude a selected trajectory segment, click on it again. To complete the construction, you need to click the mouse in the empty space of the model.

An alternative way to create an operation is to select the sketch outline and choose the Sweep command from the pop-up toolbar, then select the path segments.

When you select an open contour as a profile, an open body - a surface - will be constructed.

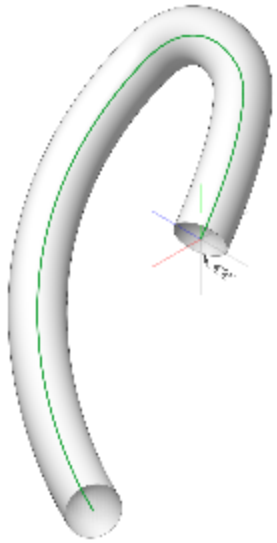
Part. The name of the part to which this operation applies.

SweepType. Defines the profile of the drawing

Profile. Defines the profile of the drawing

Path. It's a curve along which a 2D profile is moved to create a 3D object

Thin Wall. Similar to the Extrude function.



Twist. Similar to the Helix function.

Pattern. Similar to the Extrude function.
Coil. Generation of a 3D solid by extruding a sketch along a helical trajectory (unlike the Sweep operation, explicit creation of a helical guide curve is not required).

▼ Click here to expand..

1. Call the Coil command from the toolbar, then select the profile (sketch outline) to extrude with the mouse.



By default, the coil axis is aligned with the sketch's X-axis; its position can be later adjusted (restriction: the axis must lie within the sketch plane).

Part. The name of the part to which this operation applies.

Profile. Defines the profile of the drawing.

Change the Coil direction by clicking the arrow button. 

Base Surface. To construct a helical line, it is possible to use previously constructed geometry. To do this, in the curve creation/editing mode, click the mouse on the face or surface while the Base Surface selection field is pressed.

Input method. Similar to the Helix function.

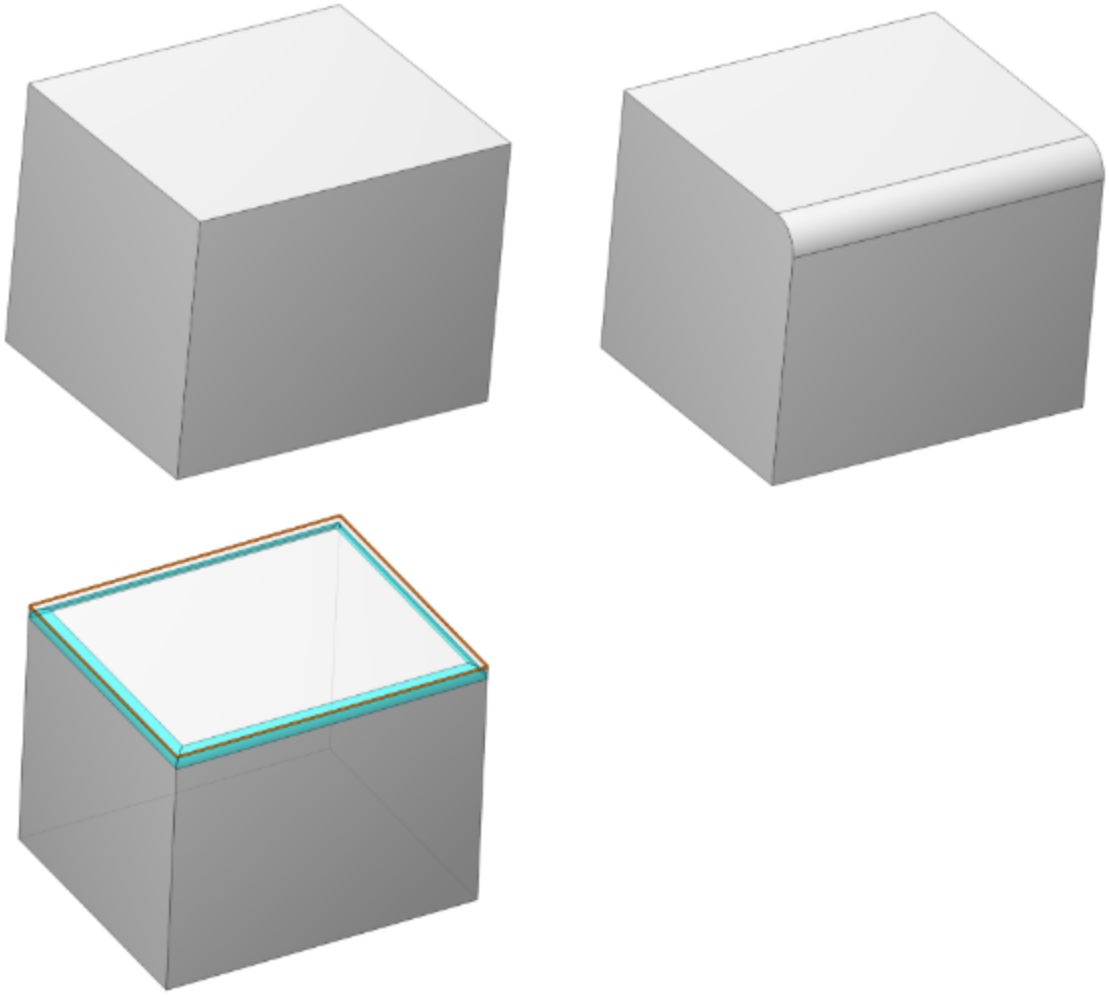
Height. Allows you to change the element height.

End Runout. Similar to the Helix function.

Pattern. Similar to the Extrude function.
Filet. Round sharp corners of the model.

▼ Click here to expand..

1. Click on an edge of the part to add it to the list. The fillet will be propagated to the tangent edges. Double click on the edge to select similar edges.



Remove fillets: Click on an added edge to remove it from the list.

DEL: Press DEL on the keyboard to clear the edges list.

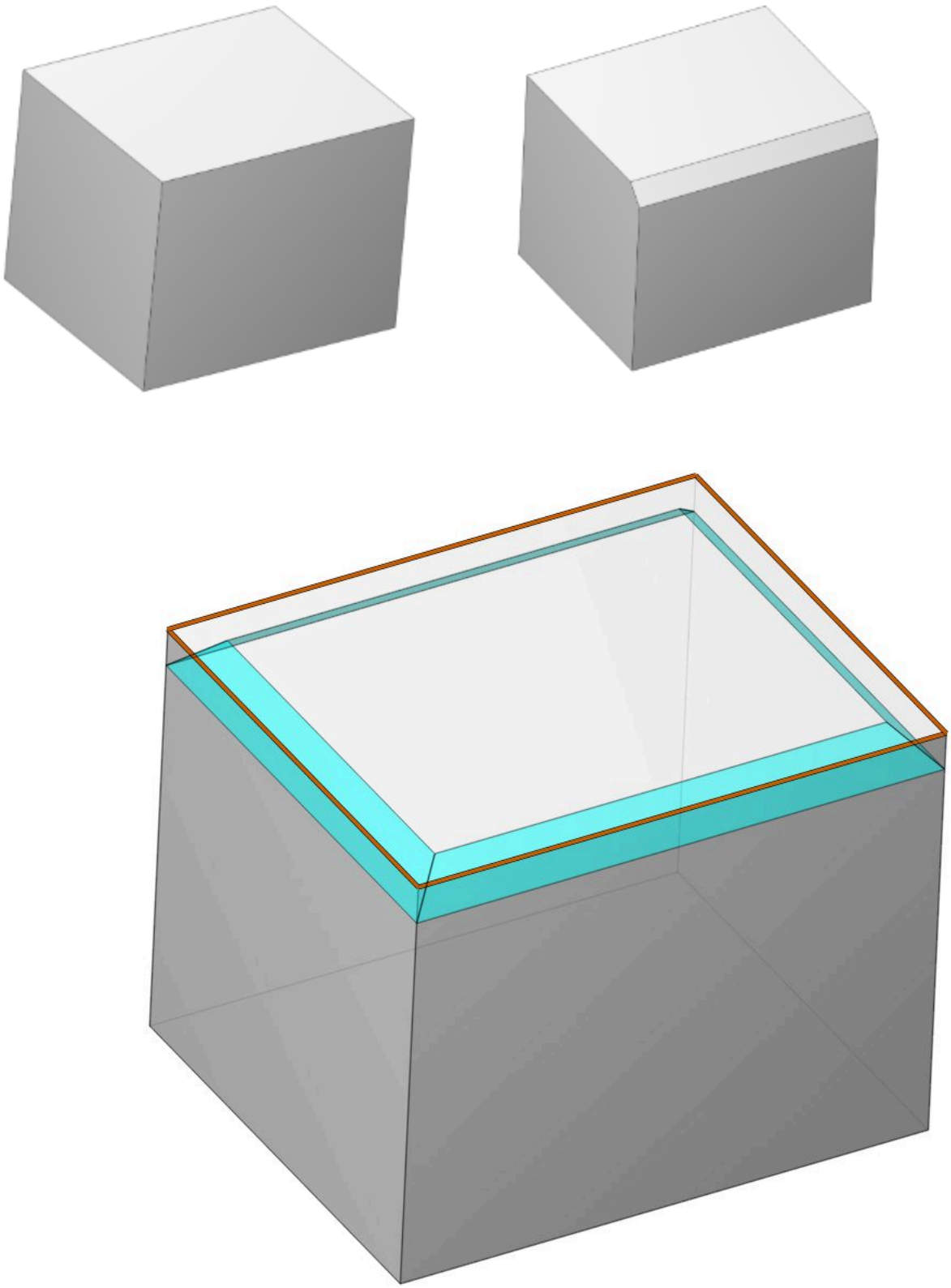
RMB, LMB: Exit.

Edges. Edge for rounding

Radius. Rounding radius
Chamfer. Chamfer sharp corners of the model.

▼ Click here to expand..

1. Click on an edge of the part to add it to the list. The chamfer will be propagated to the tangent edges. Double click on the edge to select similar edges.



Remove chamfer: Click on an added edge to remove it from the list.

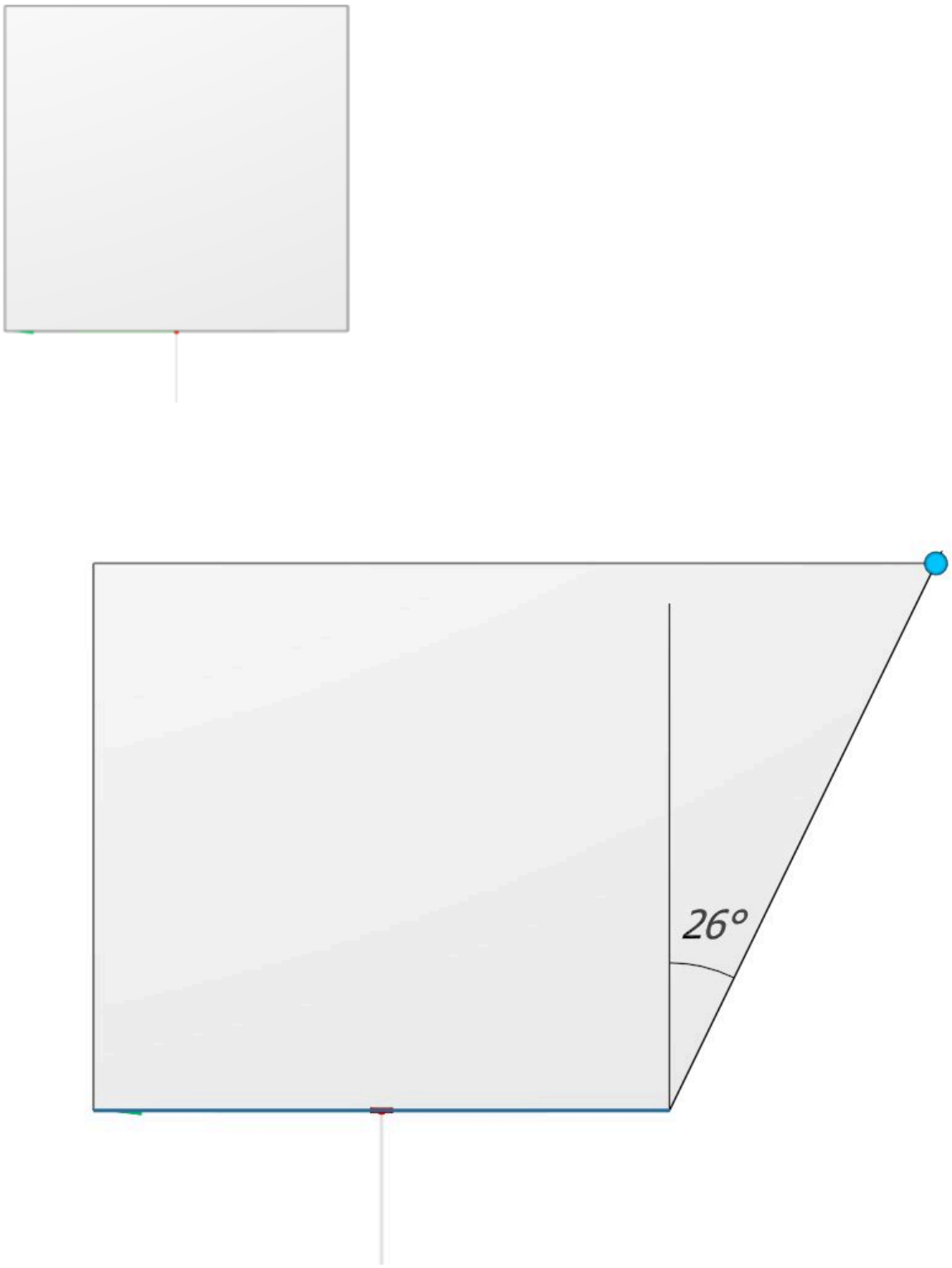
DEL: Press DEL on the keyboard to clear the edges list.

RMB, LMB: Exit.

Edges. Edge for rounding

Size. Determines the size of the chamfer
Draft faces. Tilt the selected surfaces relative to a given plane.

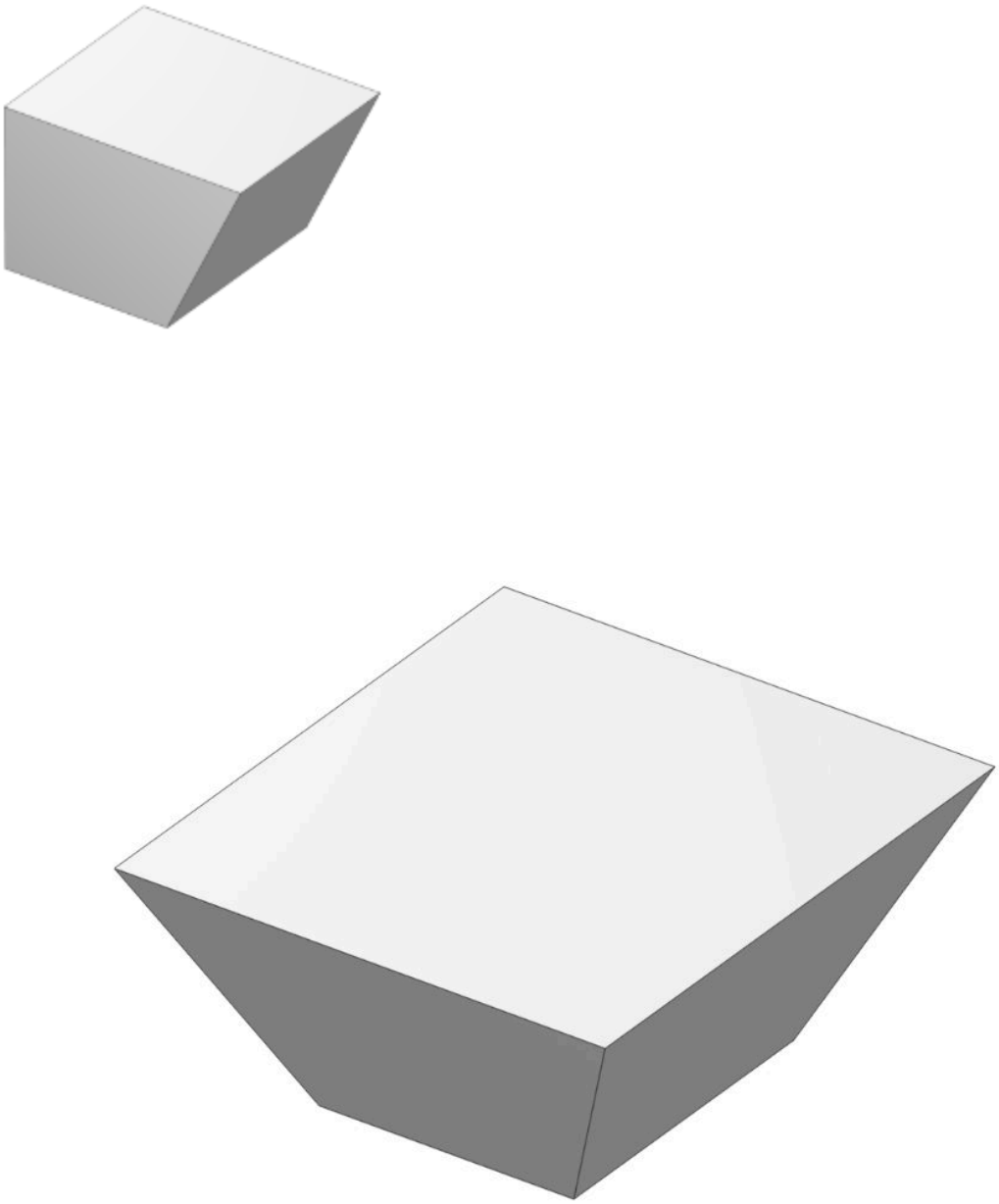
▼ Click here to expand..
1.Select draft faces Click on a part face to add or remove it to the draft faces list.



RMB, LMB: Exit.

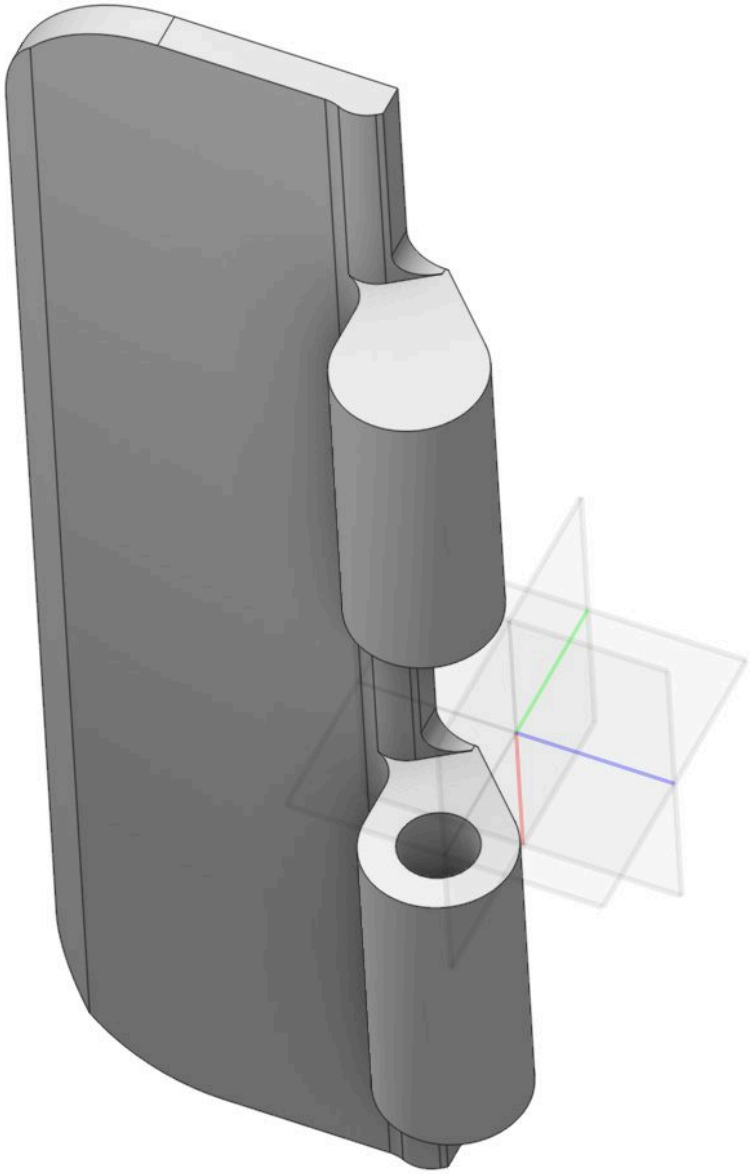
Faces. Click to select the faces to draft

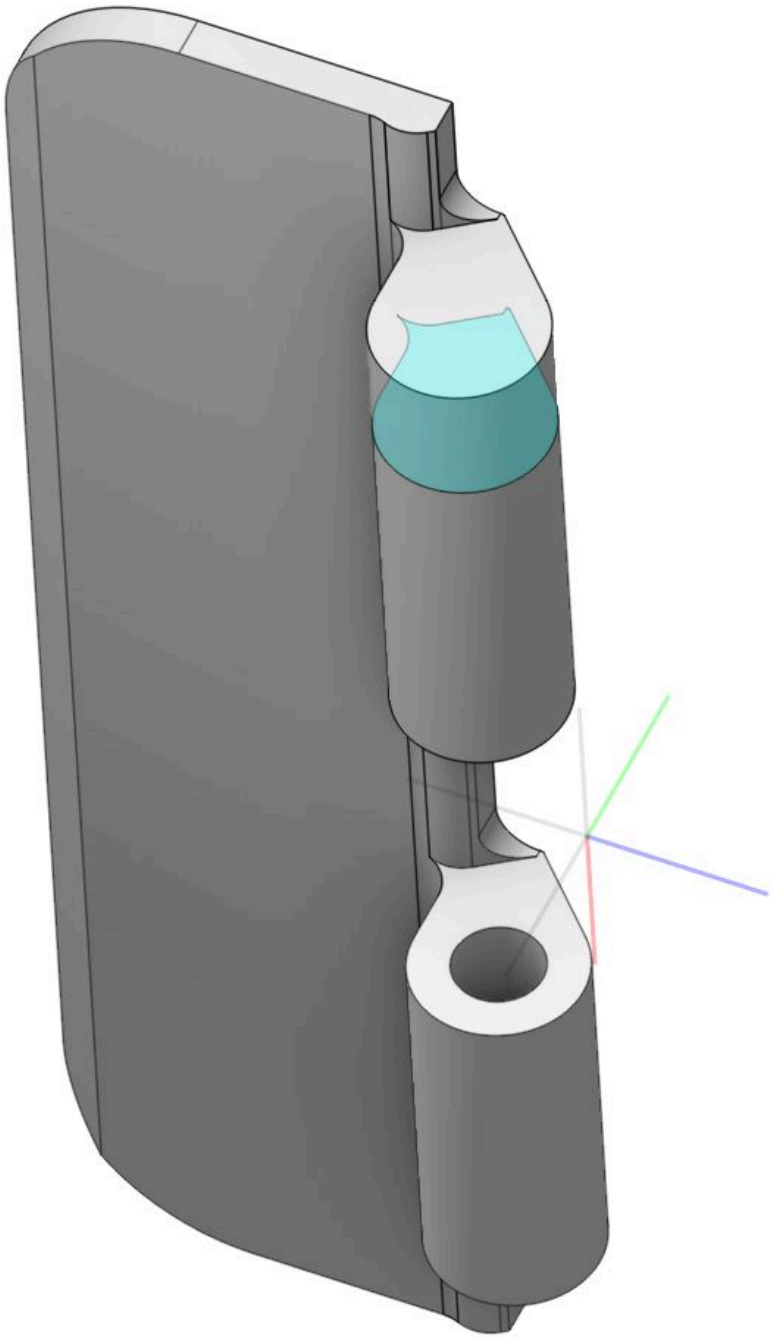
Feature Propagation. Propagate selection to the faces of the same feature.

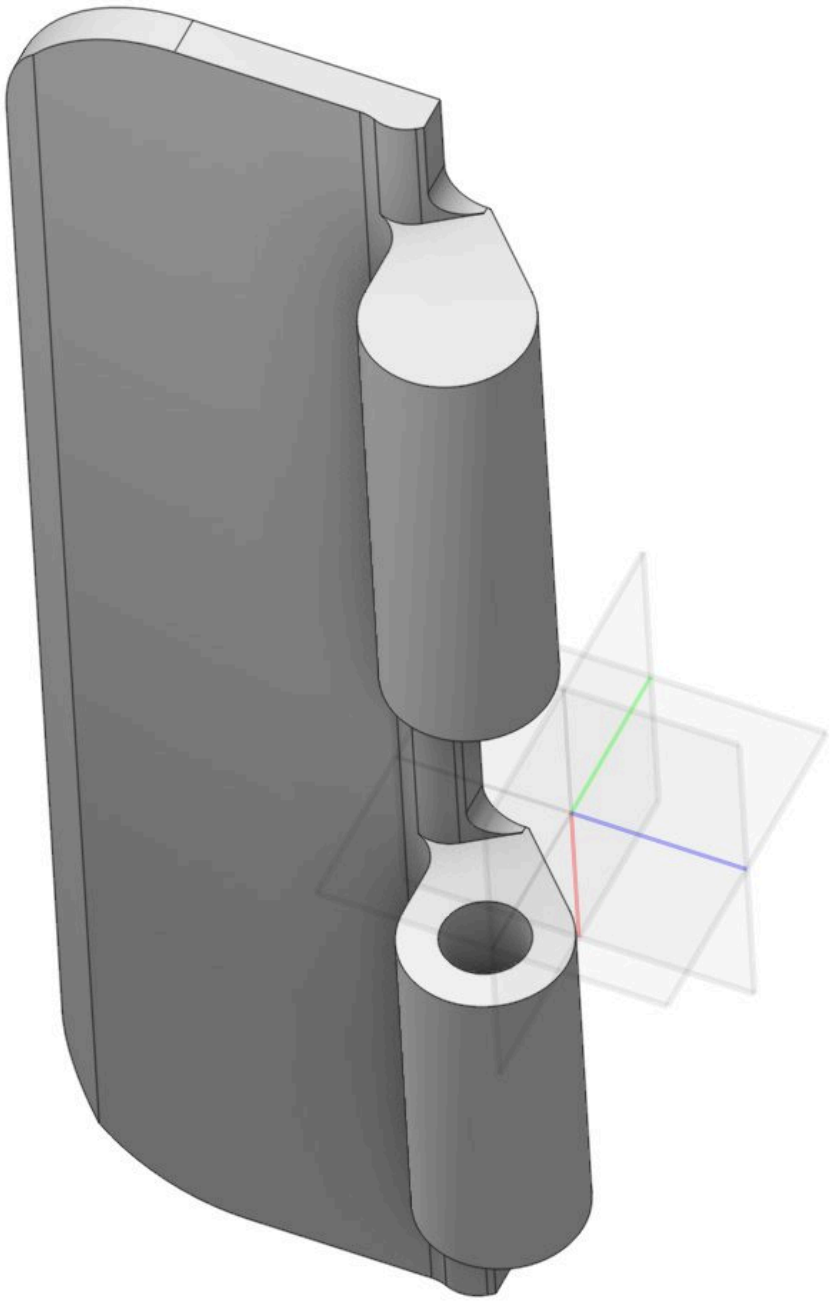


Taper Angle. Enter the angle of inclination of the faces
Offset. This functionality allows you to offset a selected surface by a specified distance. It can be used for creating a blank. Details

▼ Click here to expand..

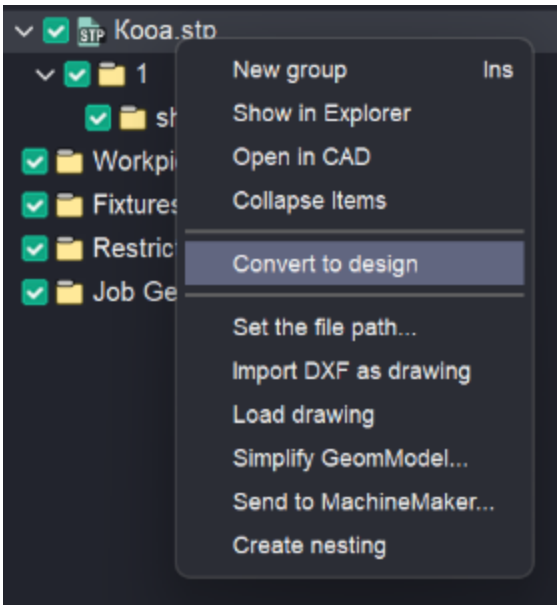


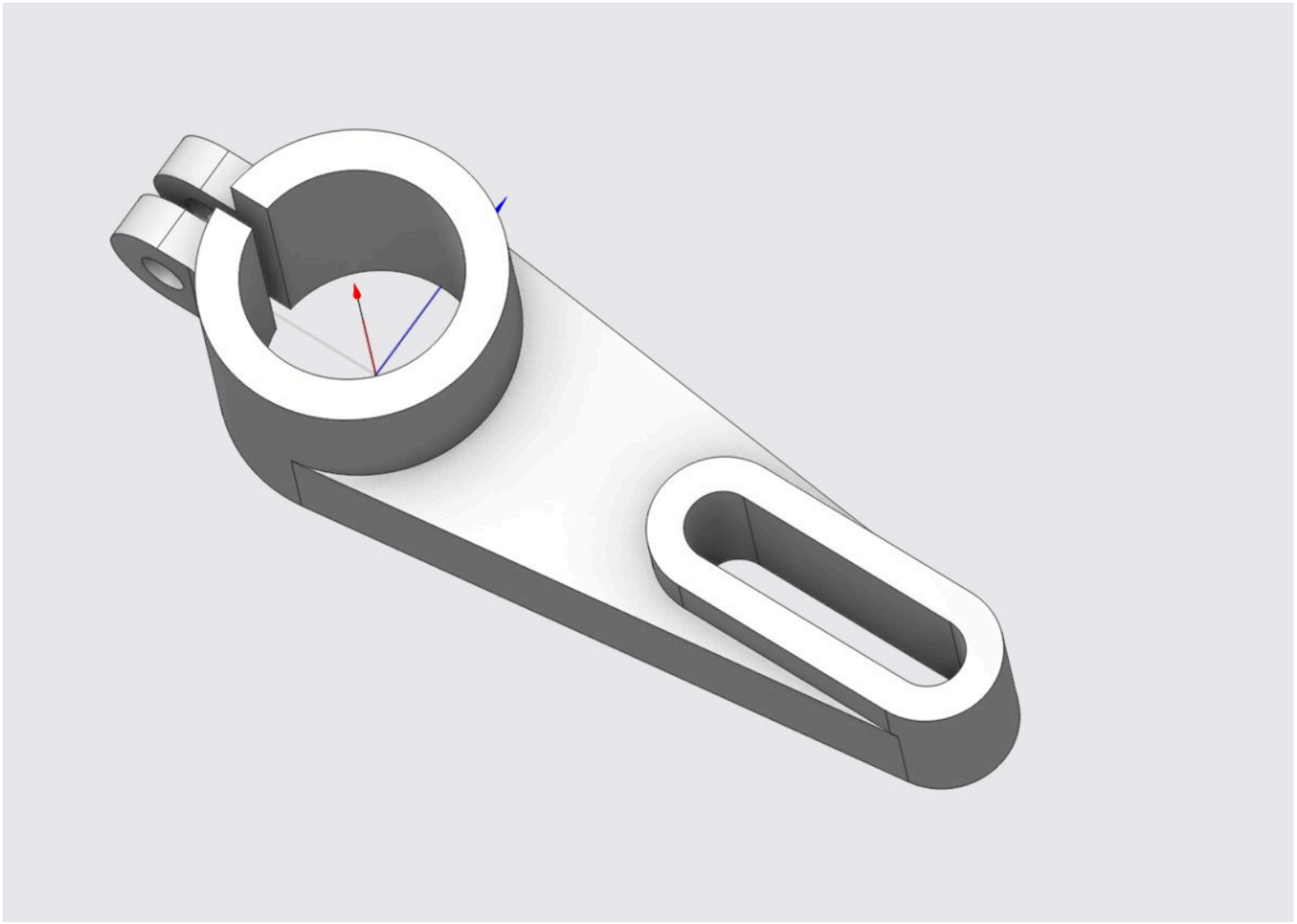


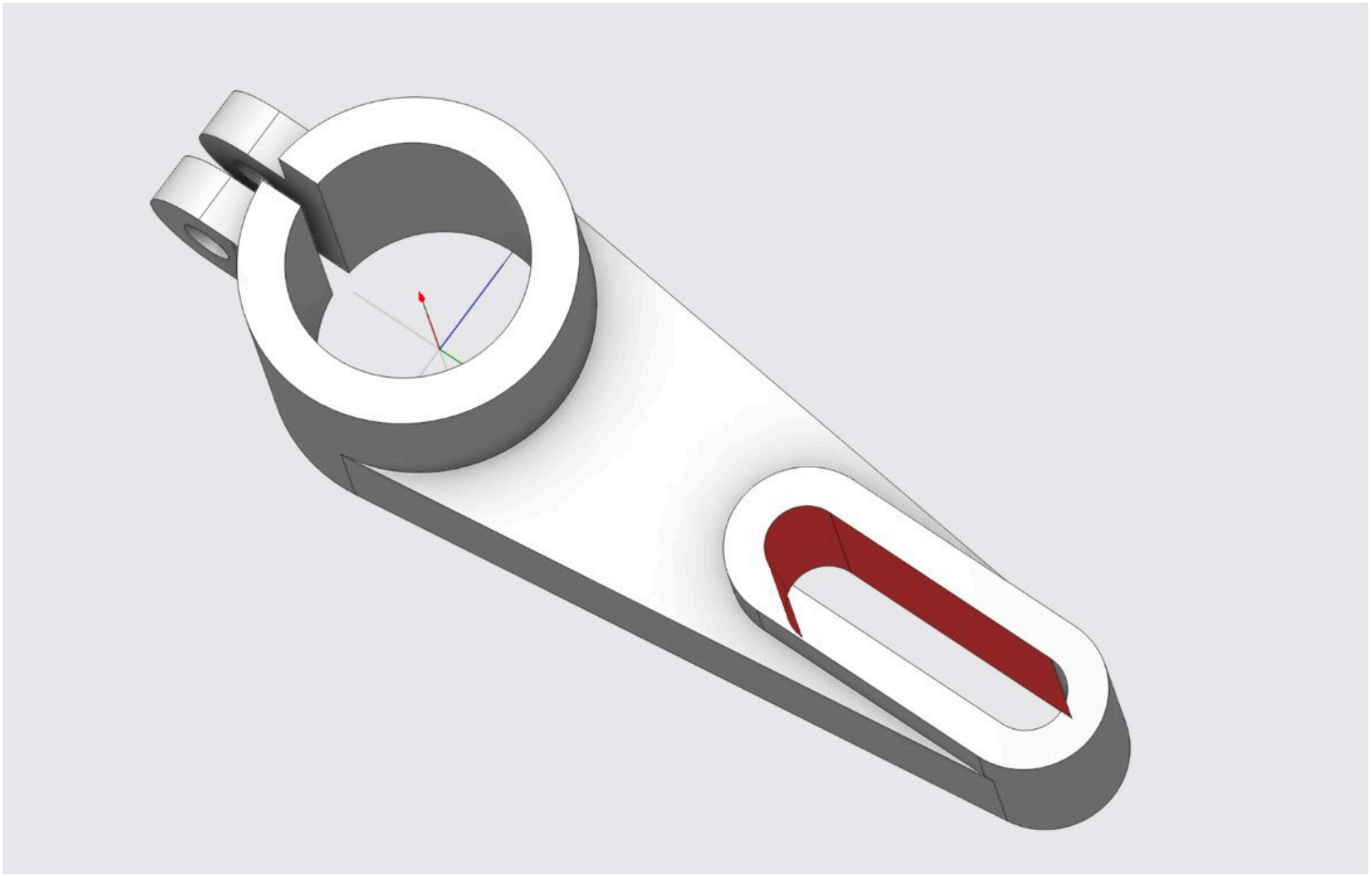


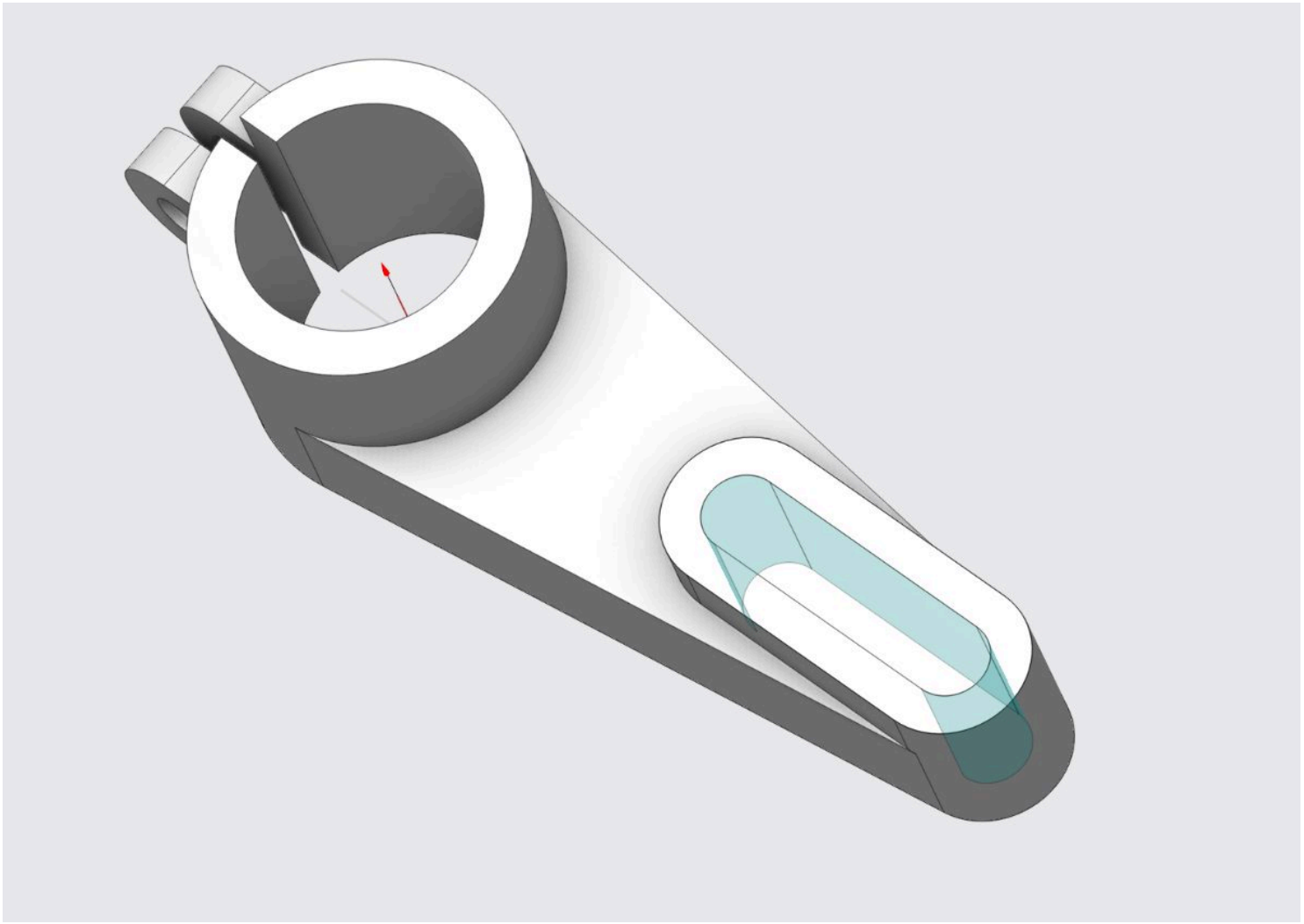
Delete face. R emove elements of a part (holes, grooves, chamfers, etc.) It can be used for creating a blank. Details

▼ Click here to expand..
Use it with the option Convert to design. (The option allows you to transform the model and open it in Design mode)


Kooa.stp
1
sh
Workpi
Fixtures
Restric
Job Ge
New group
Ins
Show in Explorer
Open in CAD
Collapse Items
Convert to design
Set the file path...
Import DXF as drawing
Load drawing
Simplify GeomModel...
Send to MachineMaker...
Create nesting







Faces. Defines the selected surfaces

Self-Intersect.
Pattern a part. Mirror or create a linear or circular pattern of the given part. In the window, select the model and specify the type of reproduction. The reproduction patterns are described in the Extrude.

Combine parts. Combine two parts together with boolean addition, subtraction or intersection. The result is one body out of two. Works similar to the Extrude function, only with details.

▼ Click here to expand..
Workpiece. Select the part of the model used as the workpiece body

Tool. Select the part of the model used as the tool body

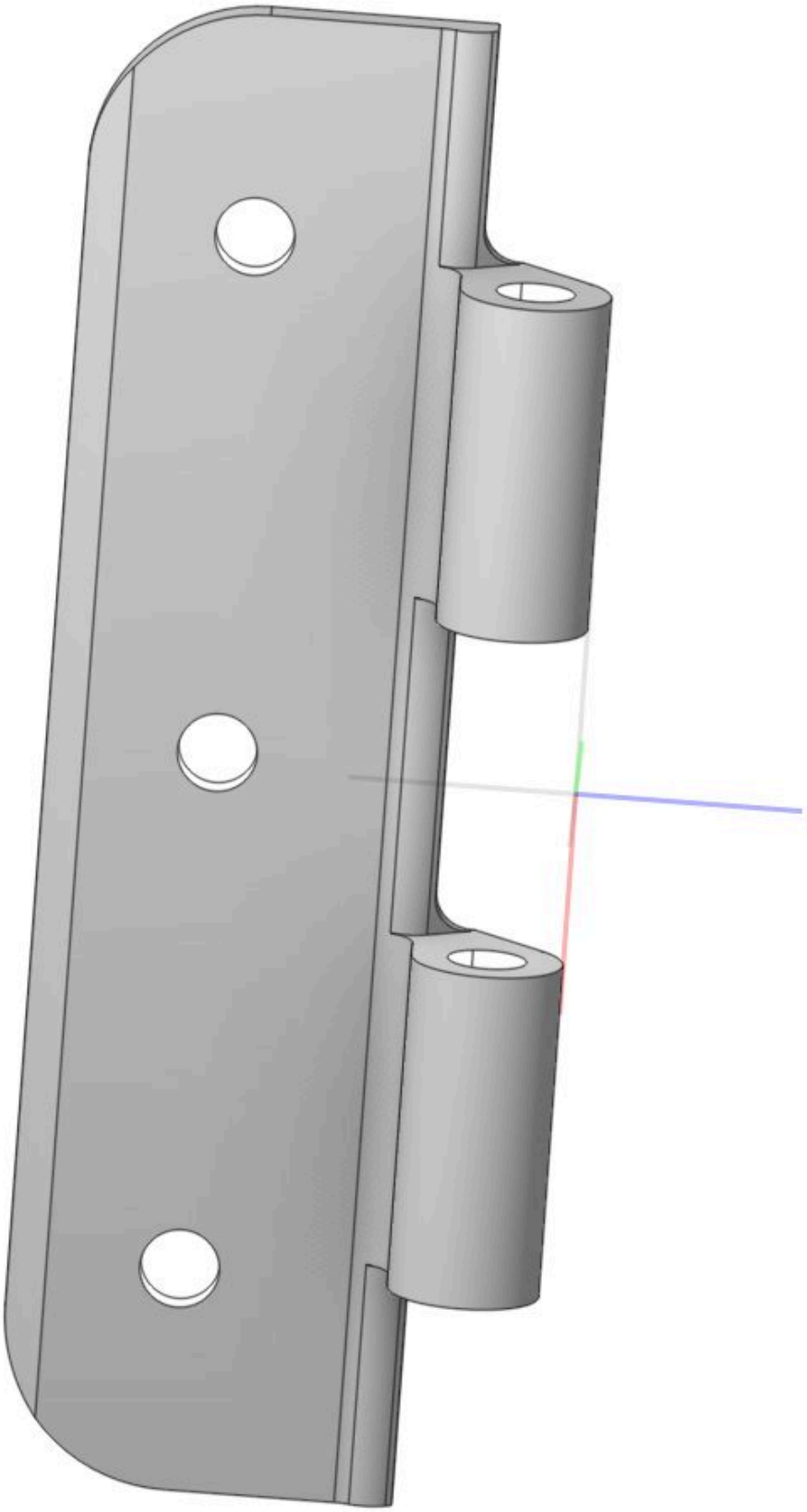
Keep tool. Keep the tool body in the model after merging it with the workpiece. Allows you to save the second part in the tree.

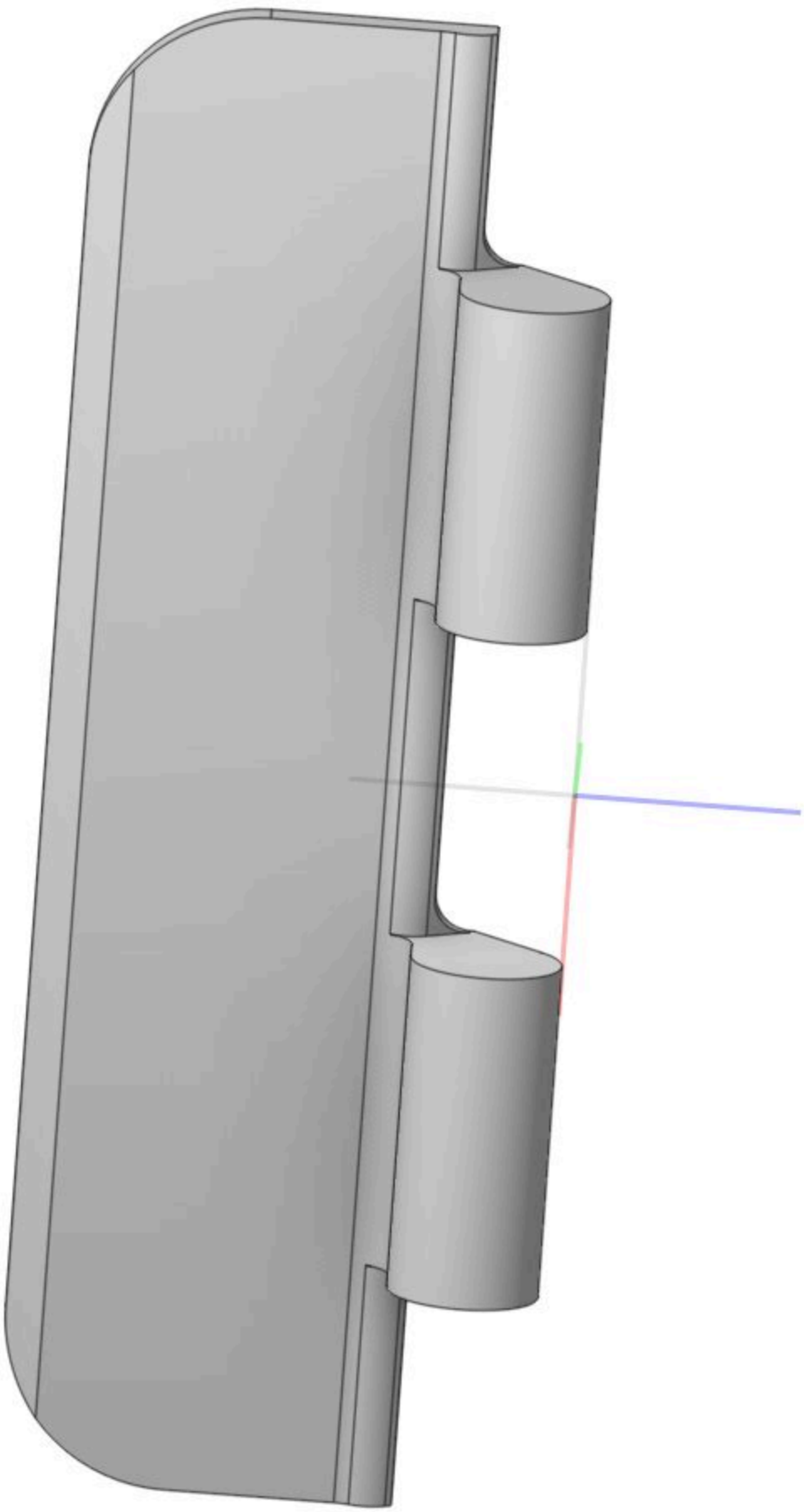
Scale the whole model. Scale the whole model by specifying the scale factor or entering a new dimension of the model.

Load. Allows you to download internal format (*.gsm) and sample projects. There is a New Model button that allows you to clear the 3D mode.

Save. Allows you to save in internal format (*.gsm) and export to format (*.stp).

Direct Modeling



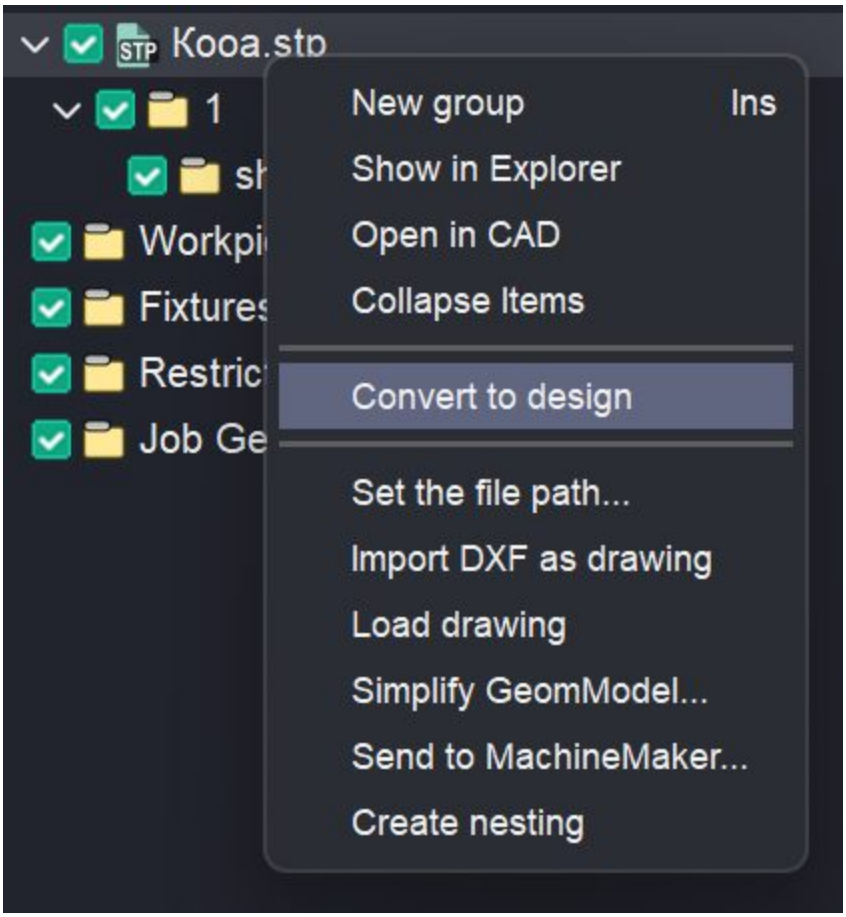


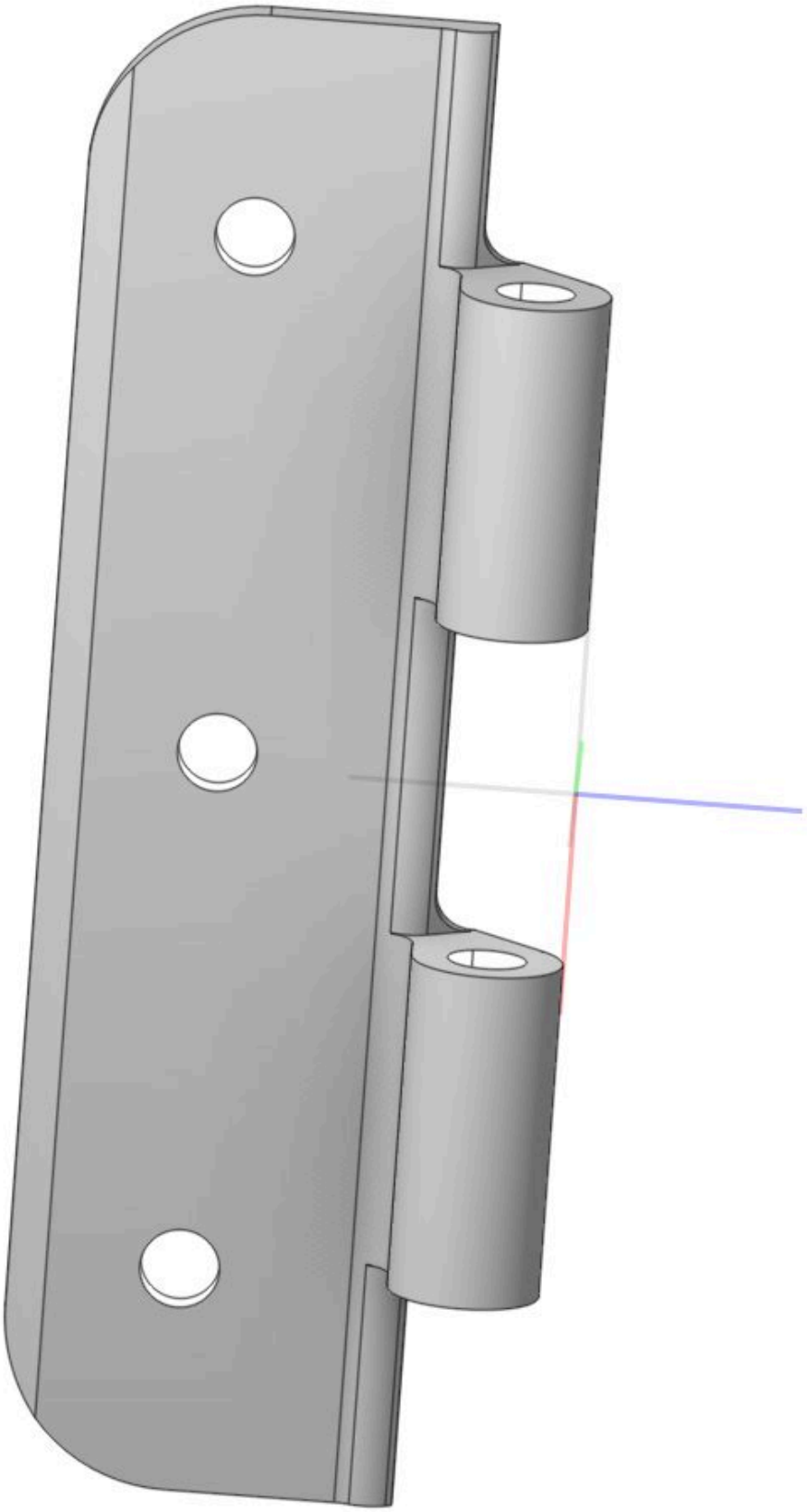
Application area:

Editing a CAD model directly (moving, deleting faces) without considering the design history or parametric dependencies.

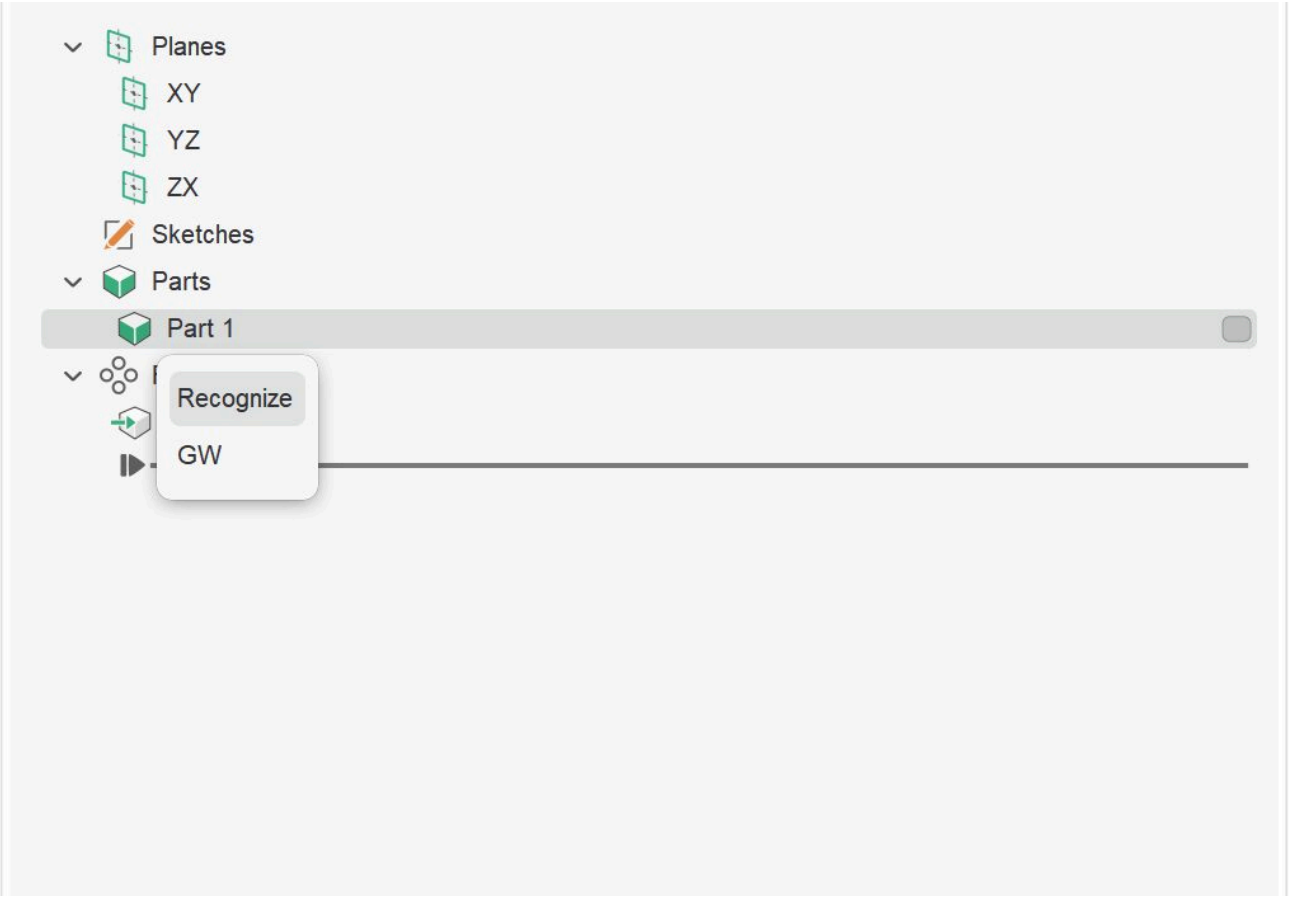
Example: creating a stock blank from an external file for subsequent machining

1. Converting the 3D Model to a CAM System Format. This requires right-clicking the model in the model tree and choosing Convert to design.


Kooa.stp
1
sh
Workpi
Fixtures
Restric
Job Ge
New group
Ins
Show in Explorer
Open in CAD
Collapse Items
Convert to design
Set the file path...
Import DXF as drawing
Load drawing
Simplify GeomModel...
Send to MachineMaker...
Create nesting



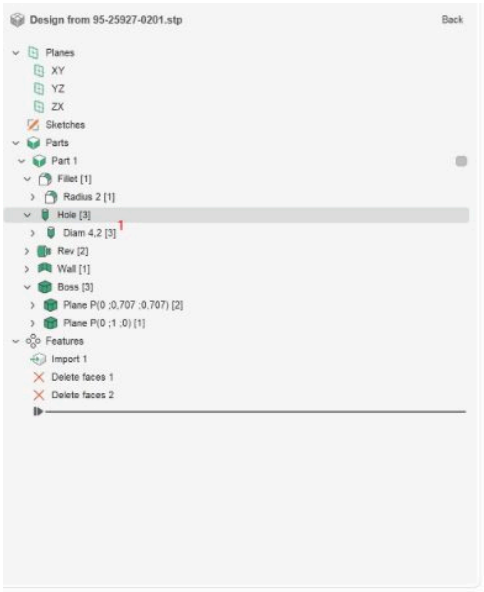
2. After the model opens in Design mode, right-click on the model in the construction tree and select Recognize.

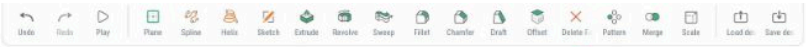


3. After this, typical features such as holes, pockets, and others will be highlighted in the construction tree.

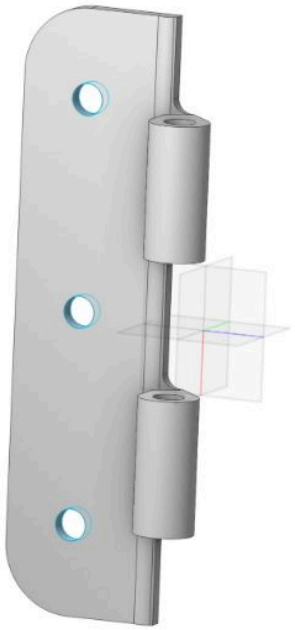


4.Using the delete or offset tools, you can create a blank according to your criteria.



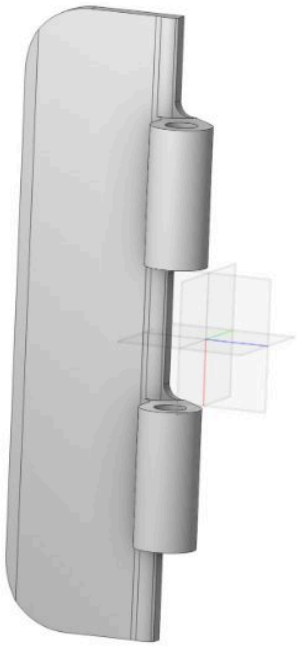


2

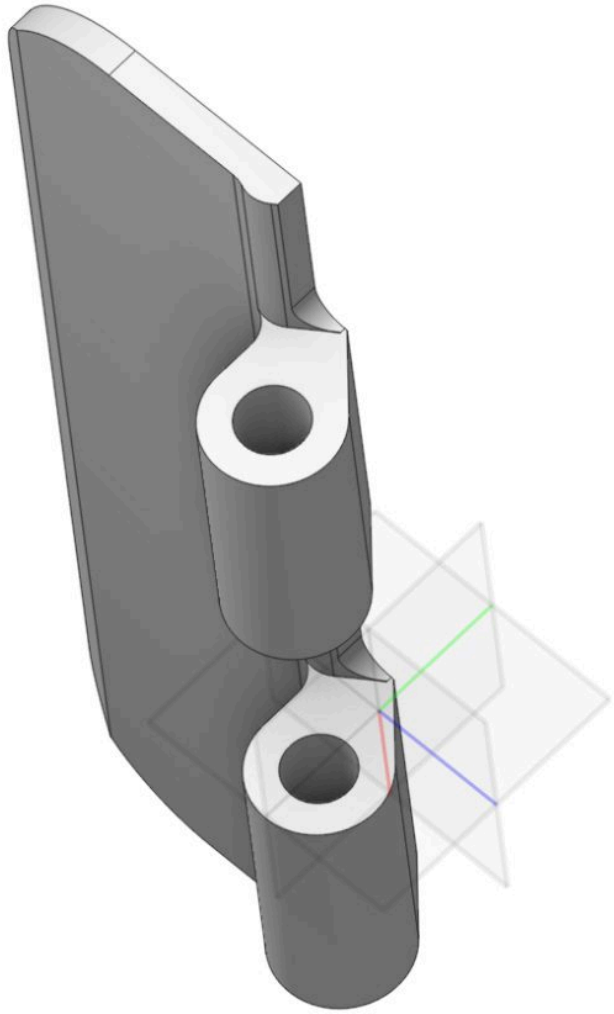


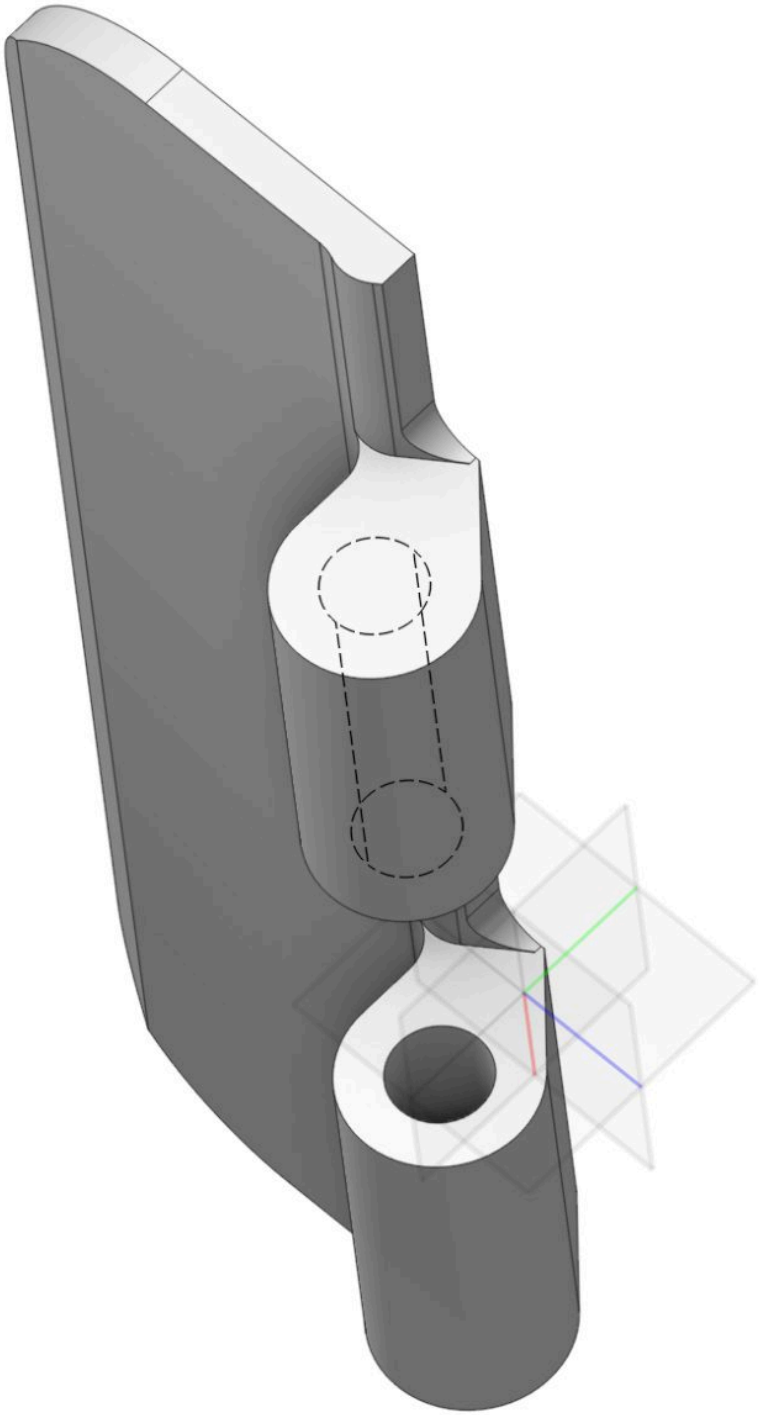




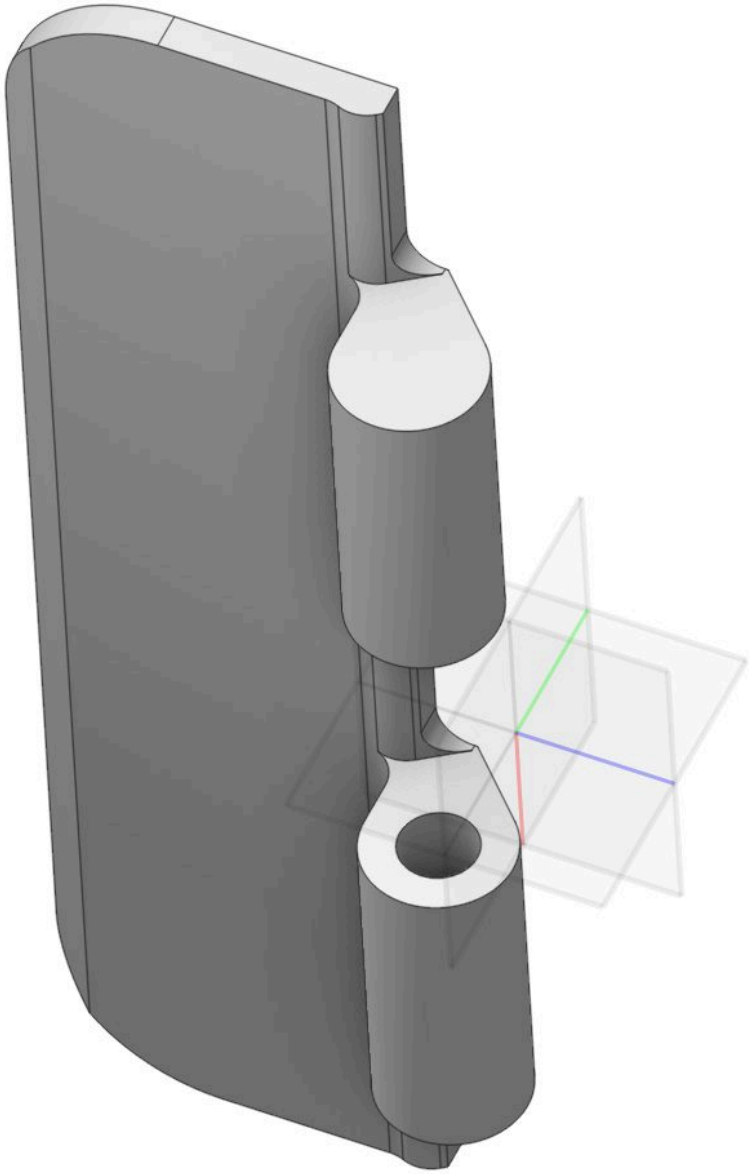


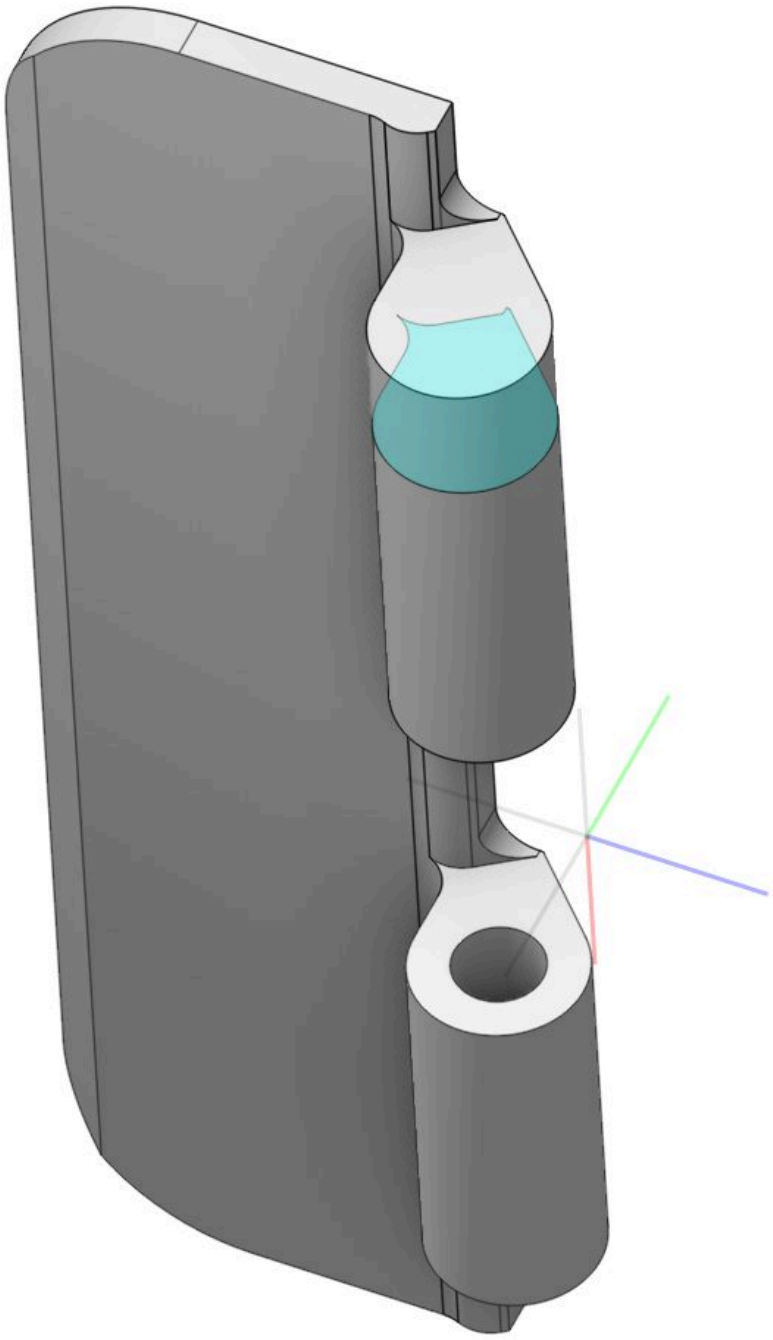
You can also select a separate surface to simplify it. Use the remove surface functionality.

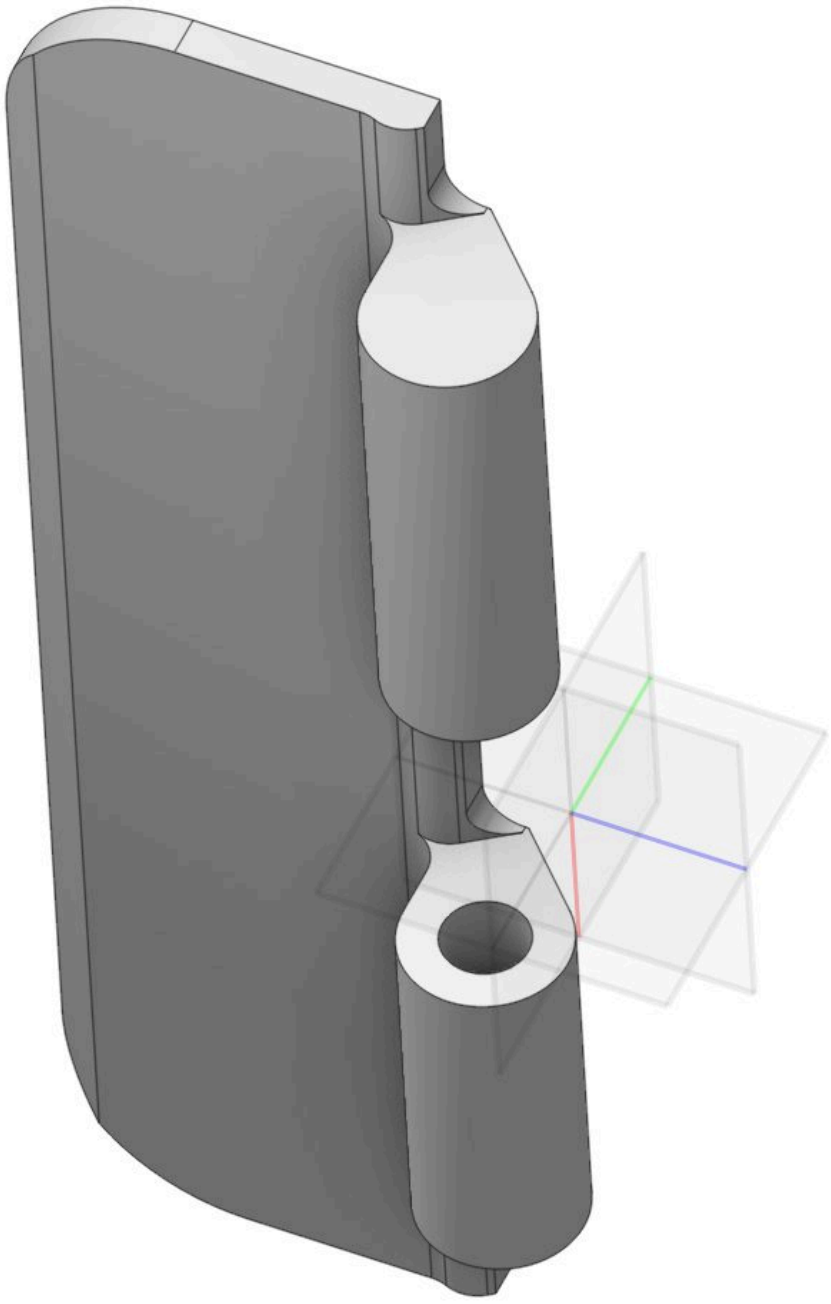




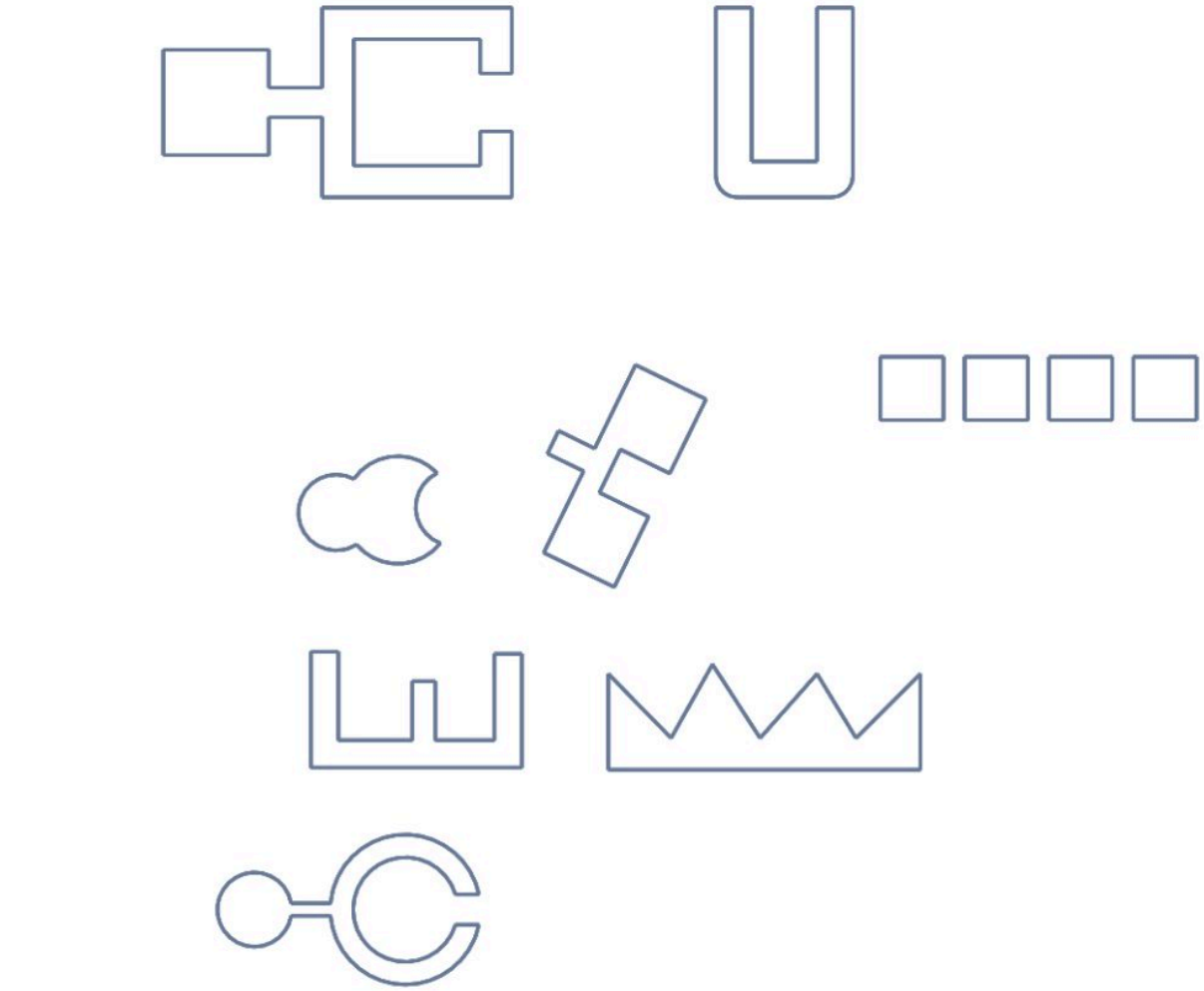
If you also need to offset the surface, use the offset function.

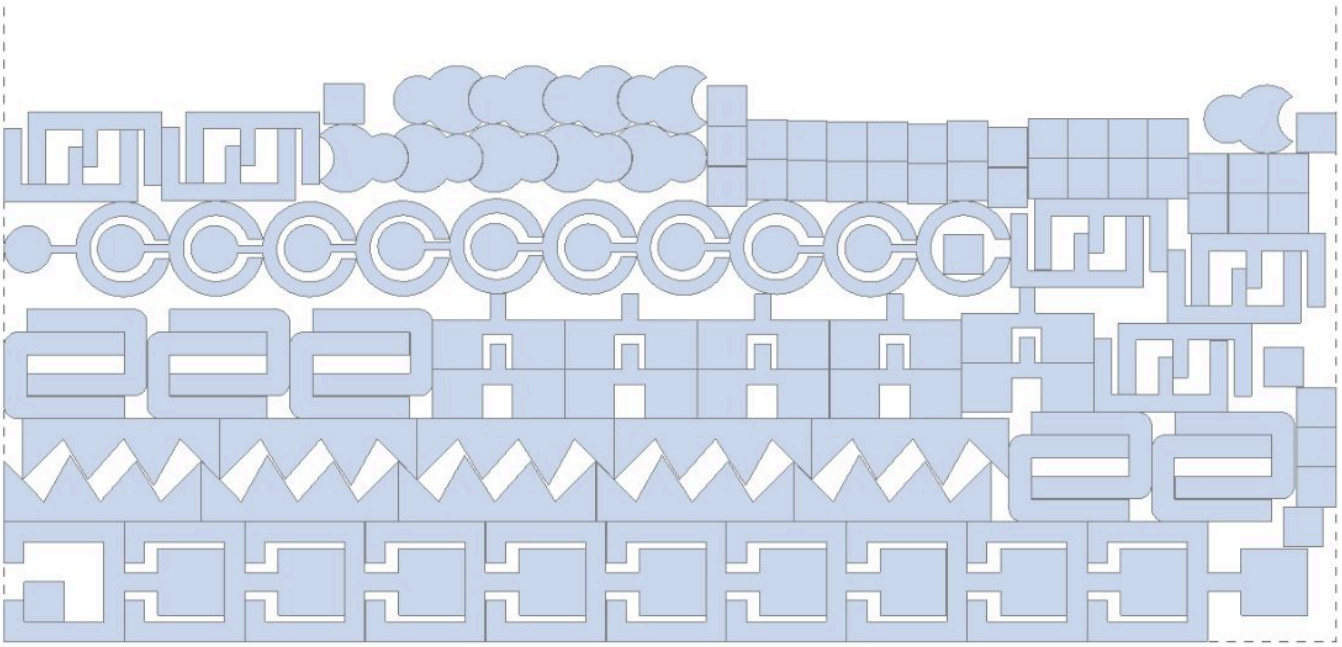






Nesting





Application area:

The Nesting tool is available from the Model tab. Right-click a sheet and select Create nesting to automatically arrange selected parts and optimize material usage.

The Nesting tool automatically arranges selected parts on a sheet to optimize material usage and reduce waste. Configure sheet dimensions, spacing, margins, part copies, rotation, mirroring, grain direction, and placement rules before generating the layout.

Window layout.

▼ Click here to expand..

1. Options tab. Contains settings that define the nesting calculation.

2. Parts tab. Displays nestable parts, calculated sheets, unnested parts, and ignored parts. It is used to define the quantity and individual placement rules for each part.

3. Run button. Starts the nesting calculation using the current settings. The arrow next to the button opens additional calculation commands.

4. Sheet navigation. Switches between sheets included in the nesting layout. The Add sheet command adds another sheet.

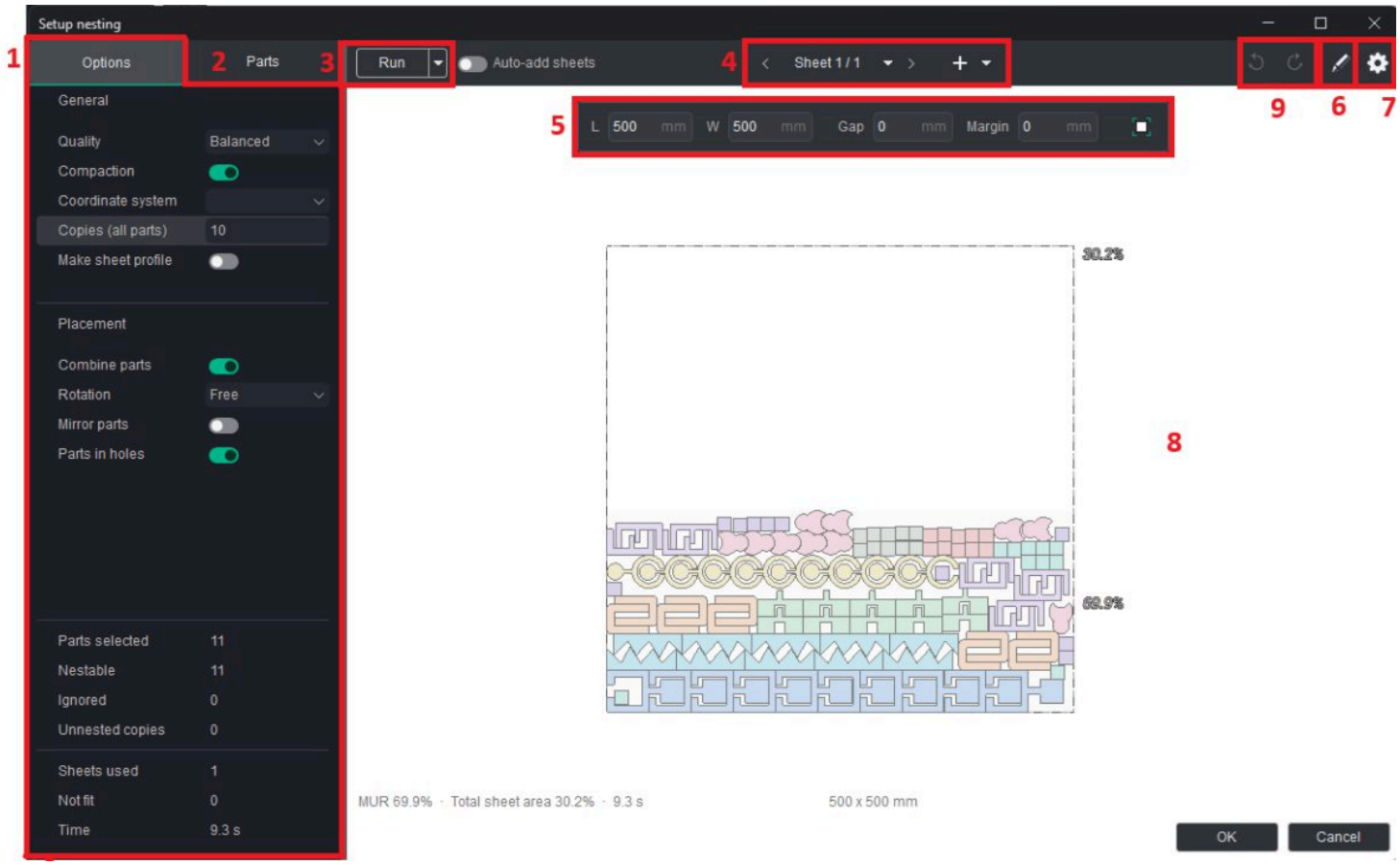
5. Sheet parameters. Define the length, width, gap, and margins of the current sheet.

6. Manual editing. Allows you to manually edit the nesting layout.

7. Settings button. Opens nesting settings.

8. Nesting layout. Displays the calculated arrangement of parts on the current sheet. The layout is generated according to the selected sheet parameters, material requirements, and placement rules.

9. Undo/Redo. Undo (Ctrl+Z) - reverses the last change. Redo (Ctrl+Y) - restores an undone change. After a new change is made, previously undone steps can no longer be redone.

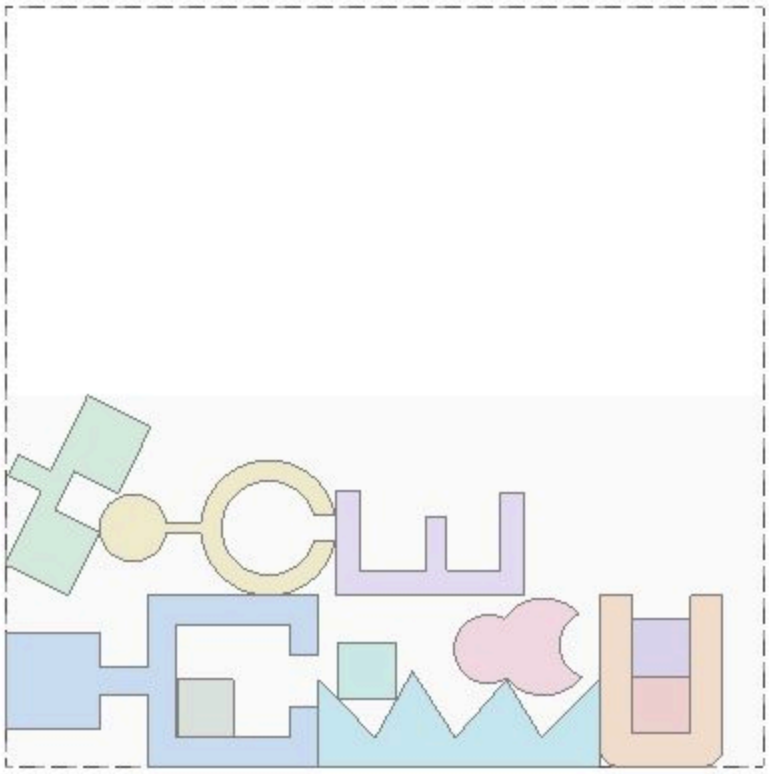


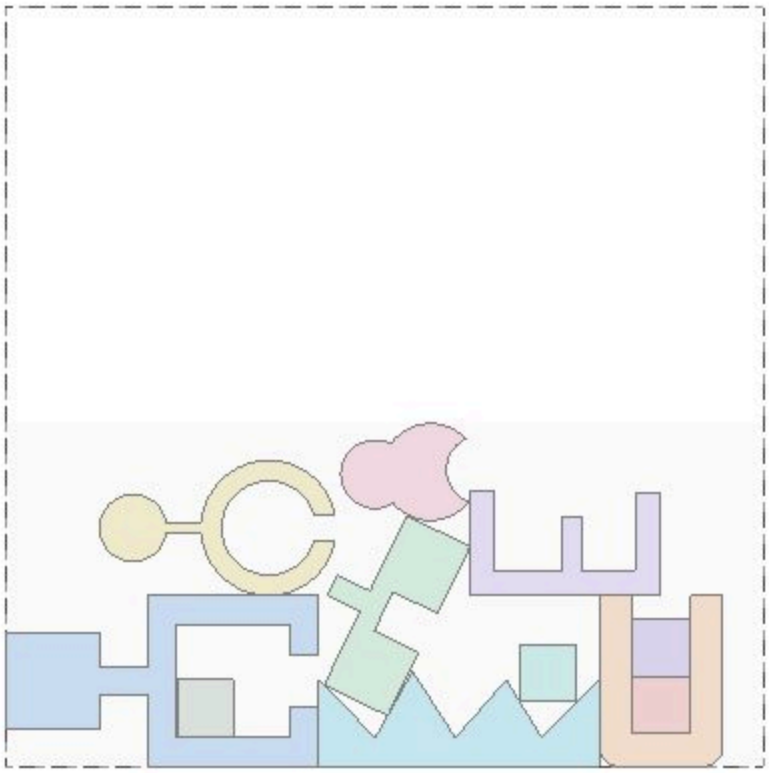
Options tab.

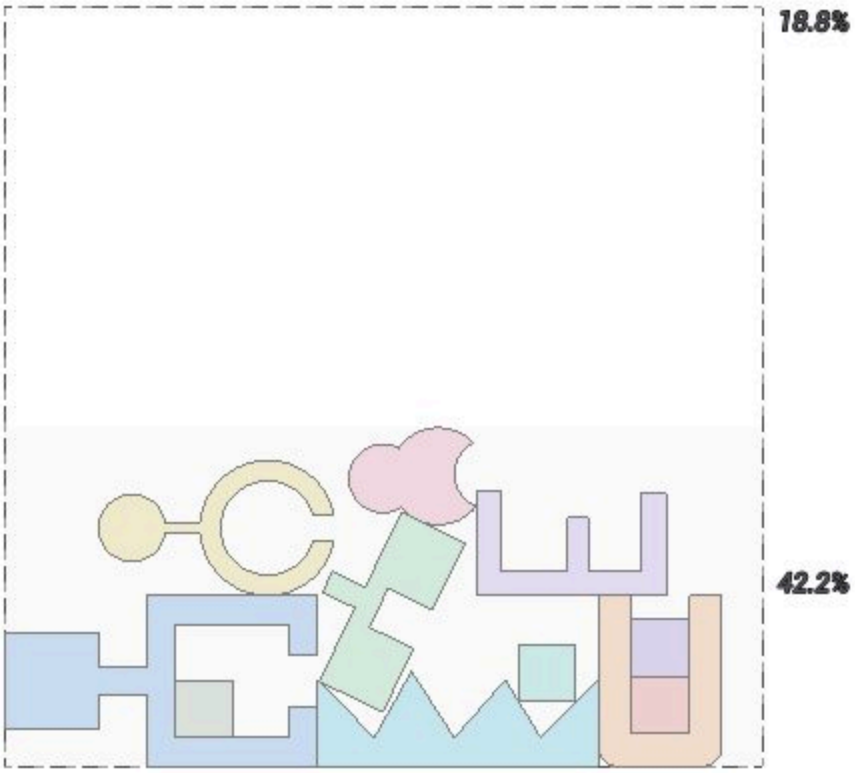
▼ Click here to expand..

General

Quality. Selects the balance between calculation speed and nesting quality.


18.8%
38.5%

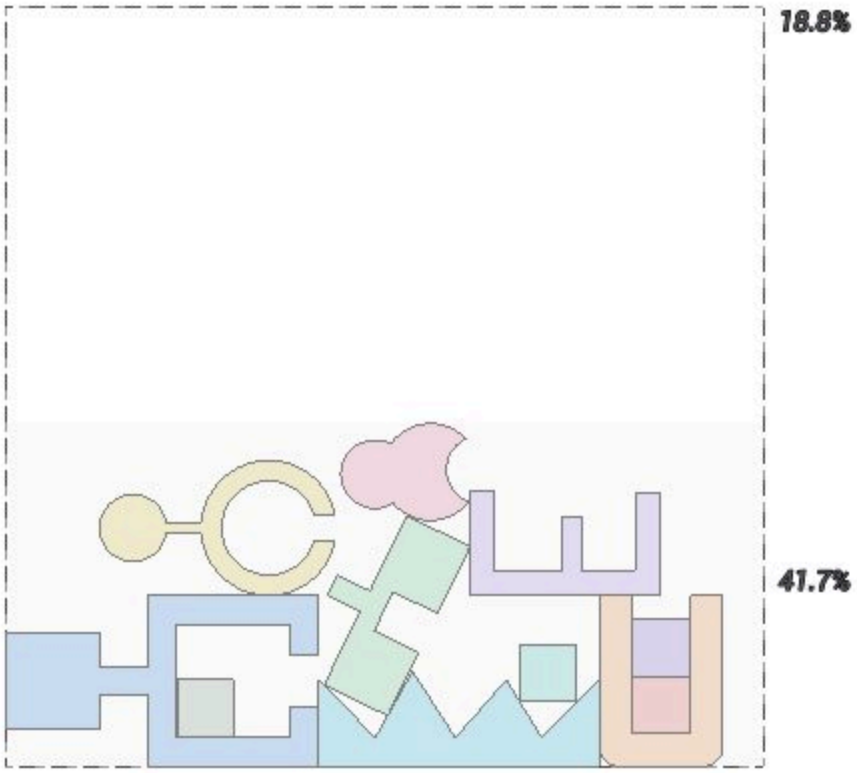

18.8%
41.7%

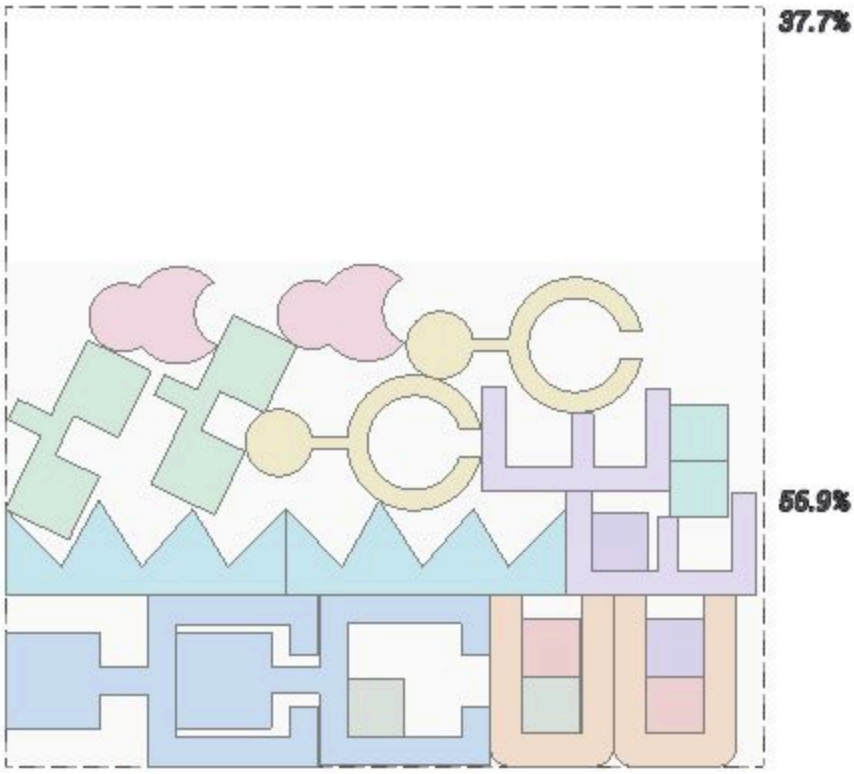


Compaction. When ON, reduces unused space between parts where possible.

Coordinate system. Selects the coordinate system used for the sheet layout.

Copies (all parts). Specifies a common number of copies for all selected parts.

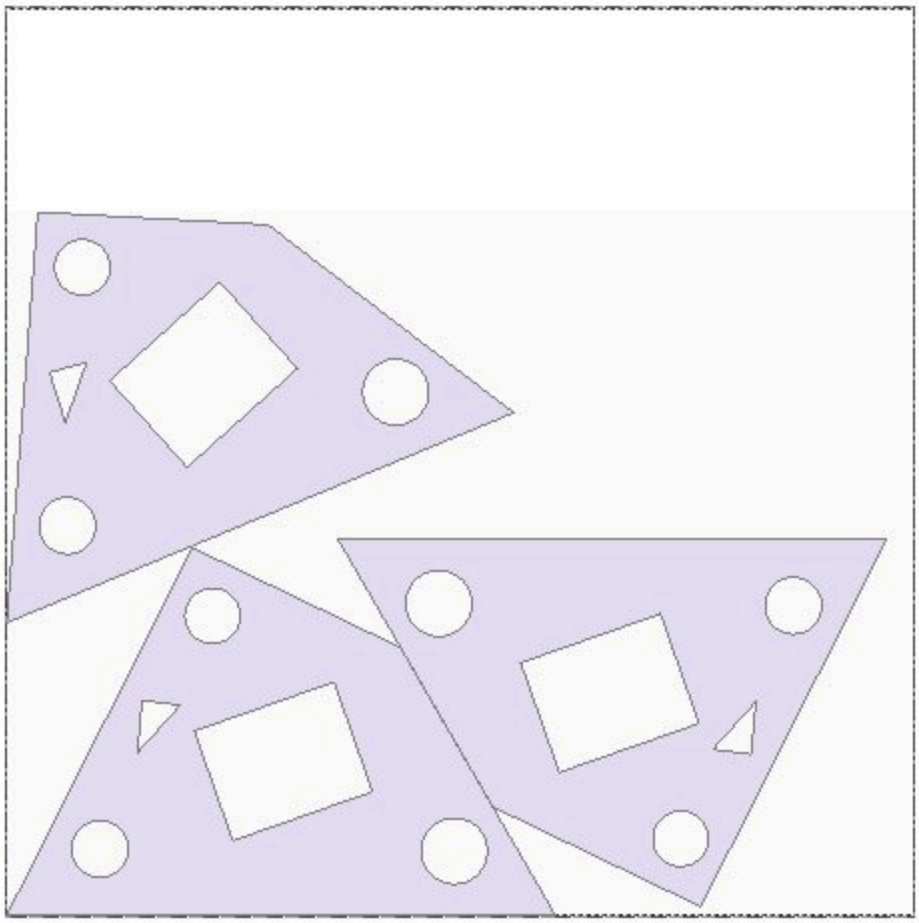

18.8%
41.7%

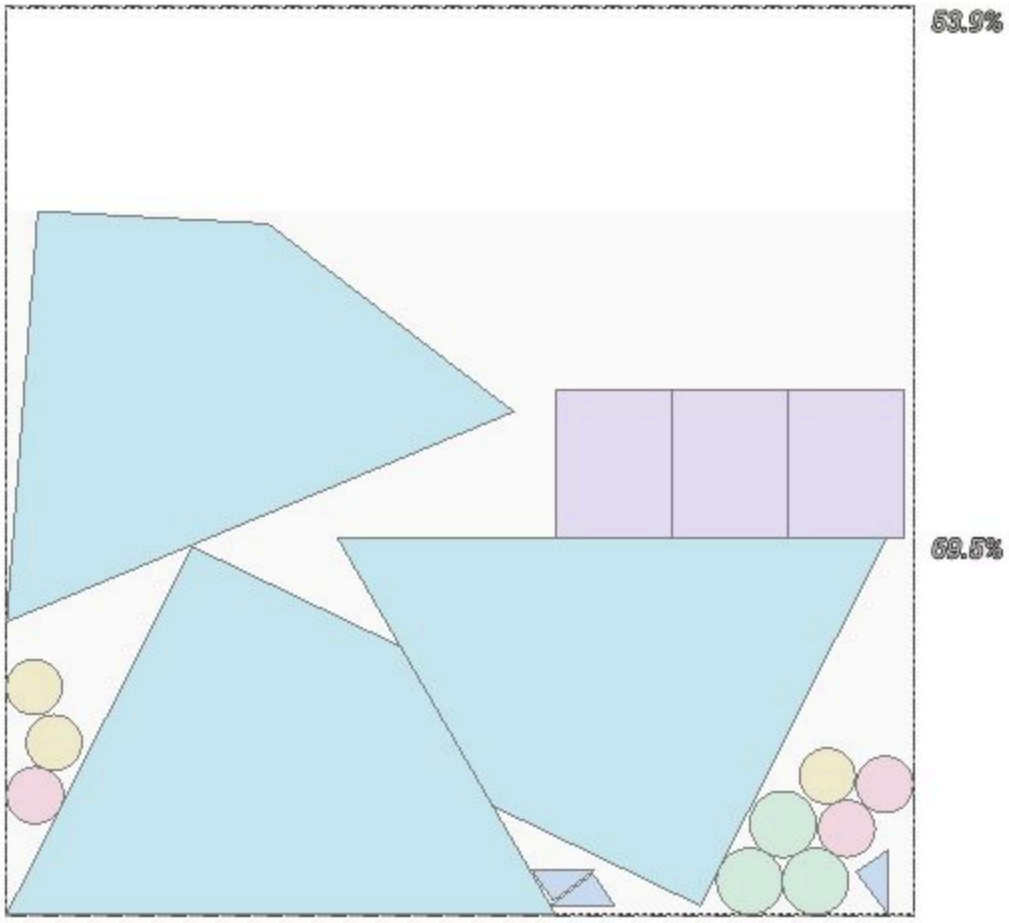

37.7%
56.9%

Make sheet profile. Keeps the sheet in the project after the nesting is created.

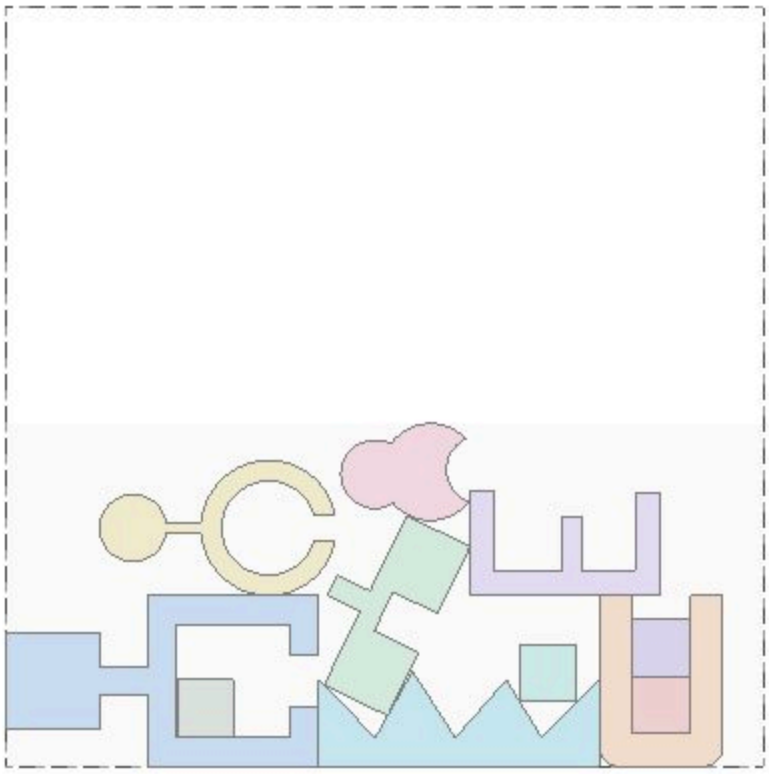
Placement.

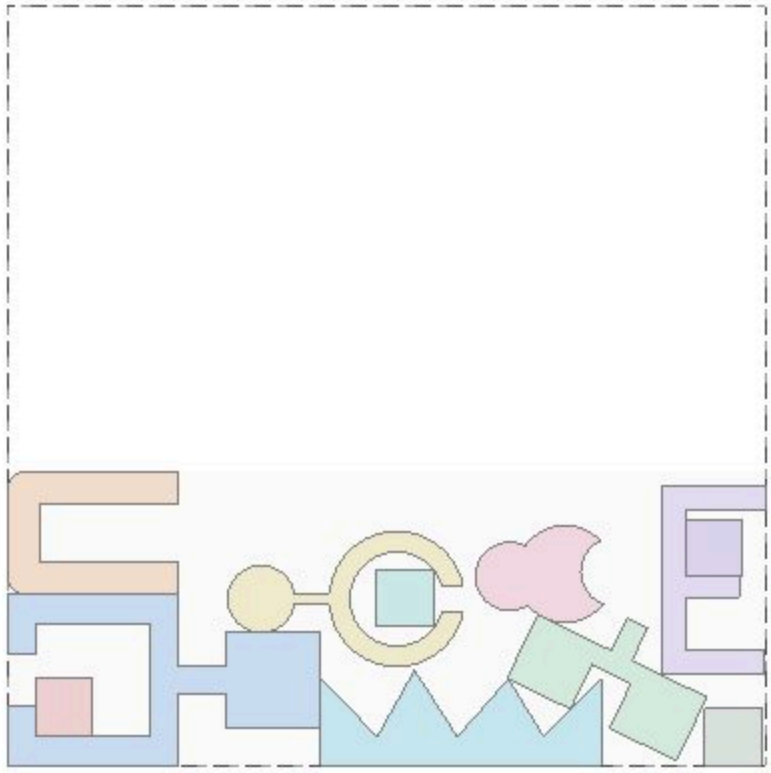
Combine parts. When ON, allows compatible parts to be arranged together on the same sheet.

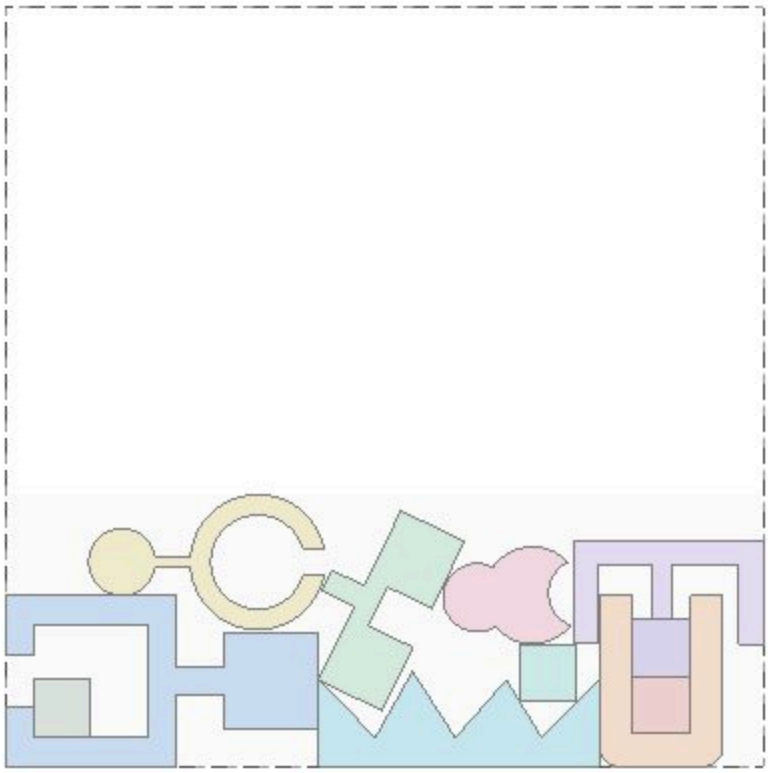

34.9%
44.9%

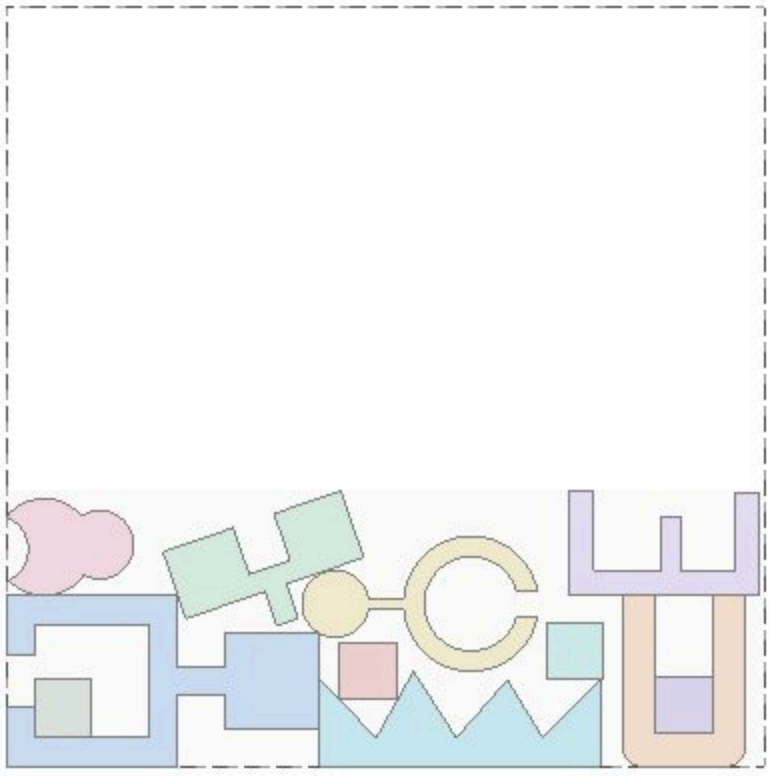


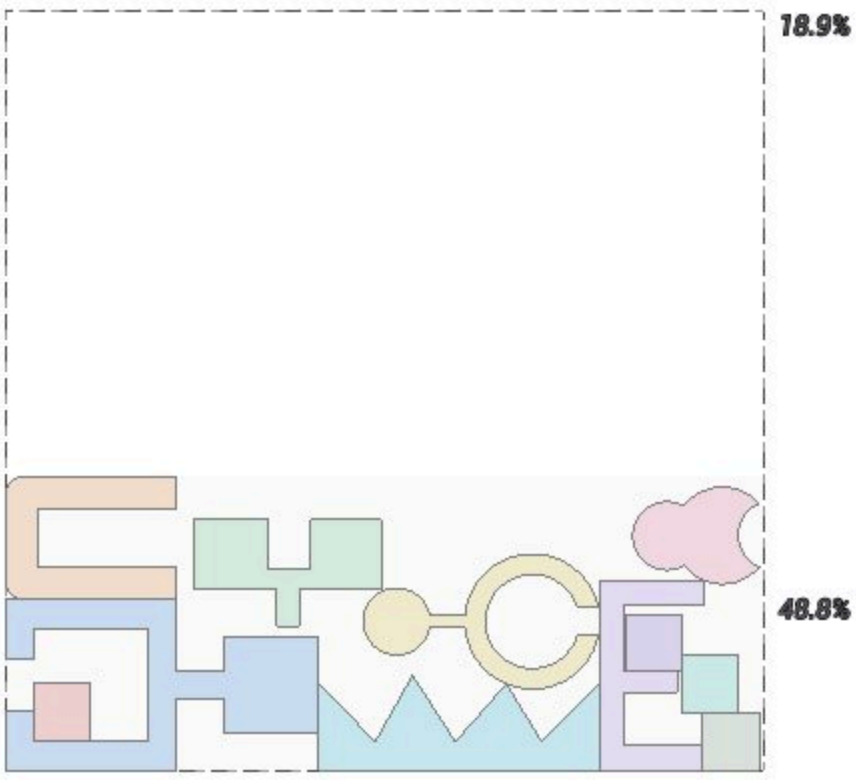
Rotation. Defines the permitted rotation of parts during nesting.


18.8%
41.7%

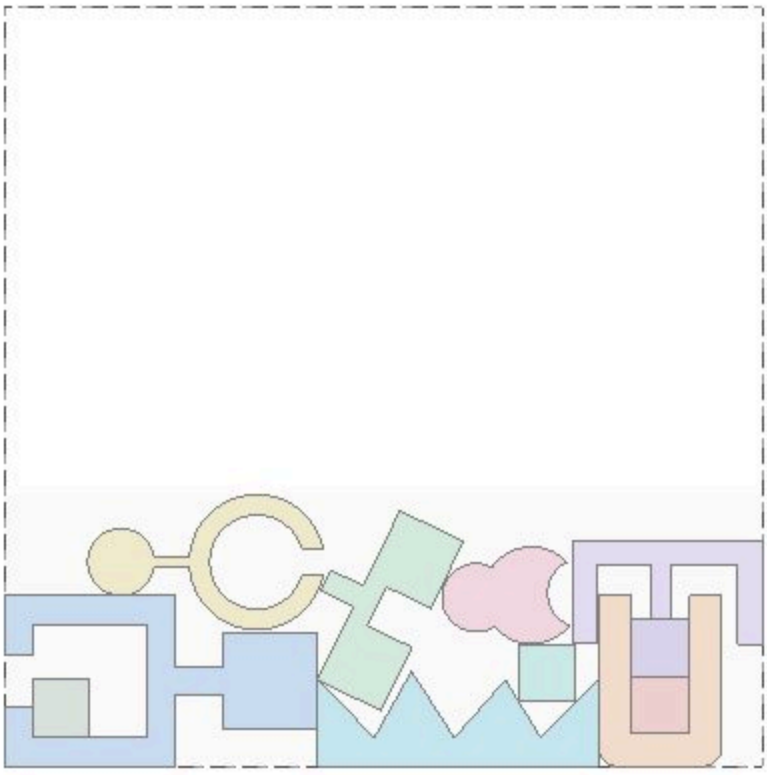

18.9%
48.7%

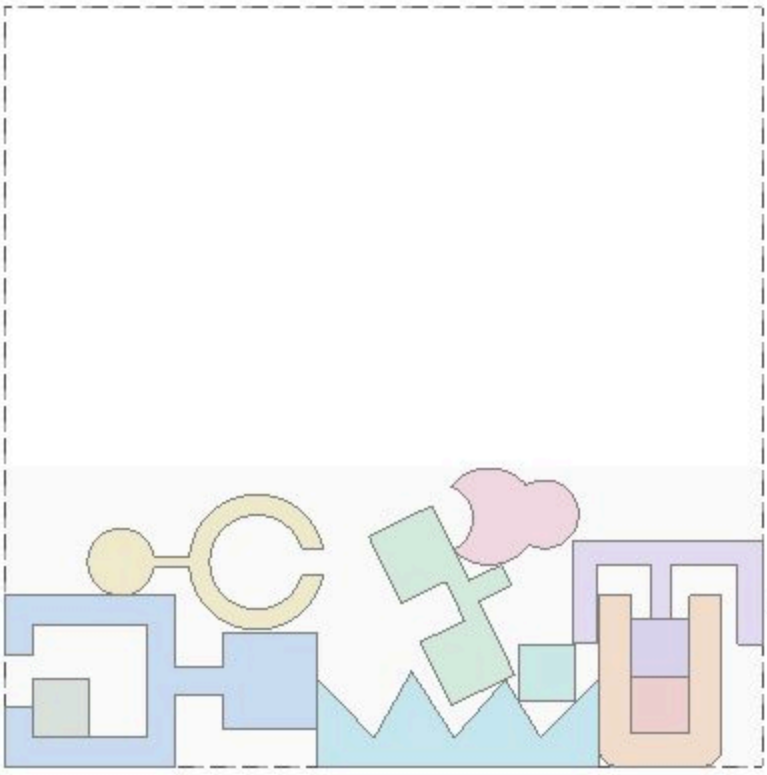

18.8%
52.6%


18.9%
52.0%

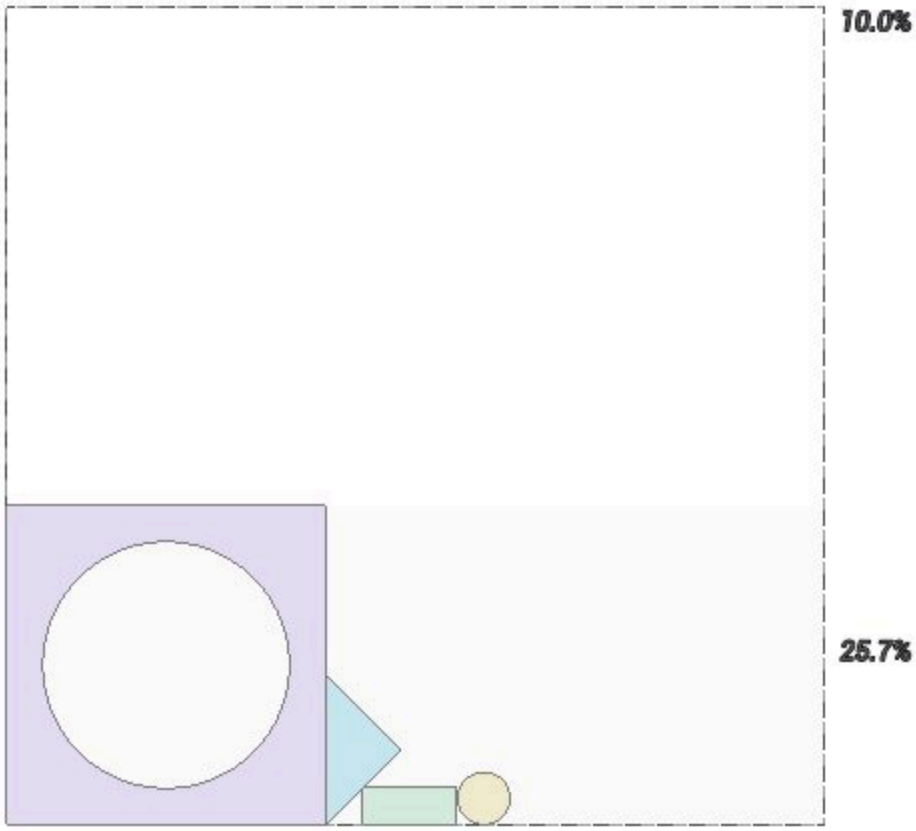


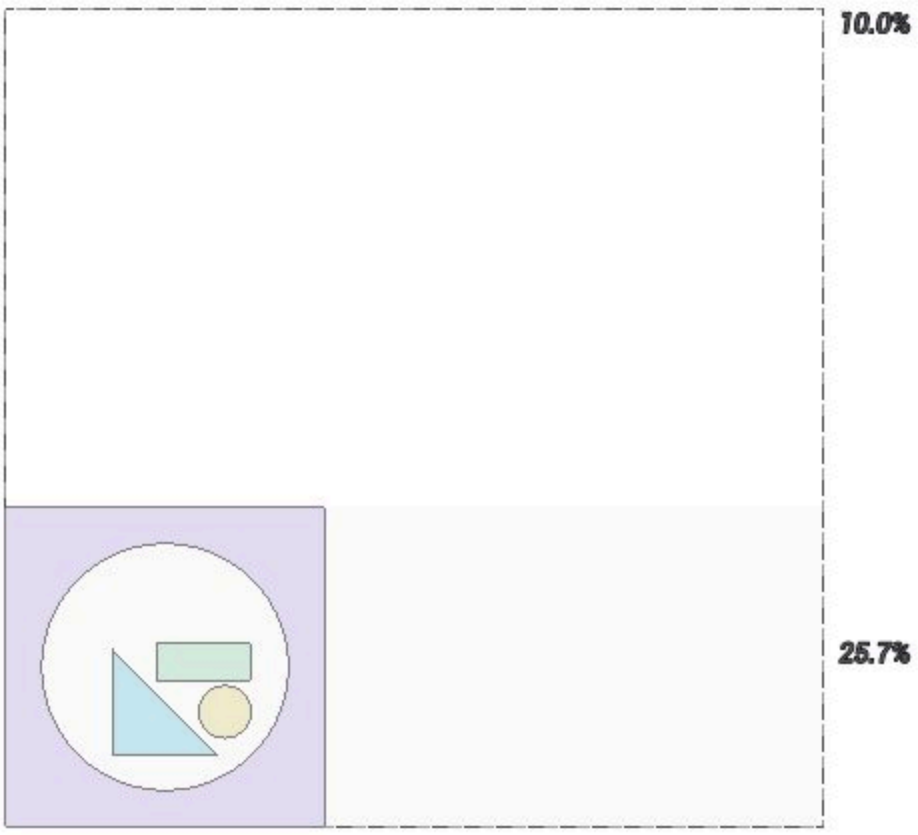
Mirror parts. When ON, allows mirrored copies of parts to be placed.


18.8%
52.6%


18.8%
47.9%

Parts in holes. When ON, allows parts to be placed inside suitable internal contours of other parts.


10.0%
25.7%



Nesting result.

Parts selected. Shows the number of parts selected for nesting.

Nestable. Shows the number of selected parts that meet the current nesting conditions.

Ignored. Shows the number of parts excluded from the calculation.

Unnested copies. Shows the number of requested part copies that could not be placed.

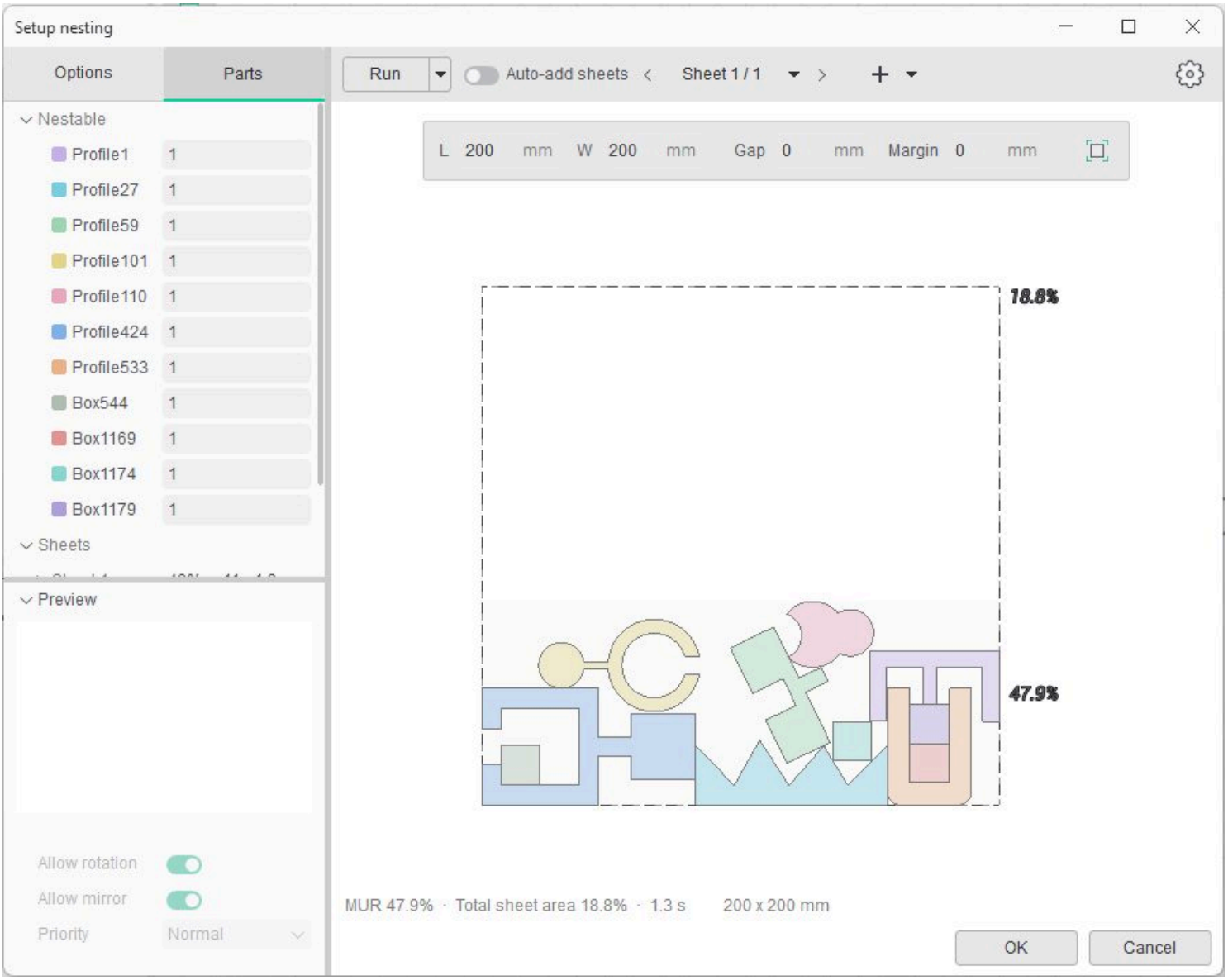
Sheets used. Shows the number of sheets required by the calculated layout.

Not fit. Shows the number of parts that do not fit on the available sheet format.

Time. Shows the time required to calculate the nesting result.

Parts tab.

▼ Click here to expand..



Nestable. Lists the parts that can be included in the nesting. The field next to each part defines its required quantity.

Sheets. Displays sheets produced by the calculation. The entry shows the material utilization, number of placed parts, and calculation time.

Unnested. Contains parts that could not be placed on the available sheets under the current conditions.

Ignored. Contains parts excluded from the calculation.

Preview. Displays a preview of the currently selected part.

Allow rotation. When ON, permits rotation of the selected part during nesting.

Allow mirror. When ON, permits a mirrored placement of the selected part.

Priority. Defines the placement priority of the selected part.

Sheet parameters.

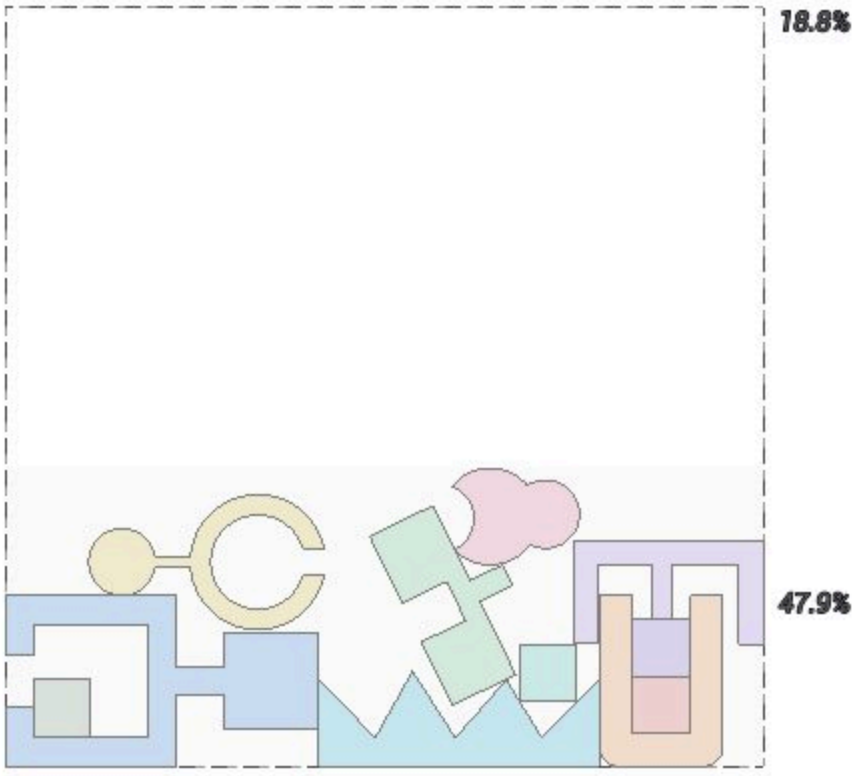
The controls above the preview define the dimensions and placement restrictions of the current sheet.

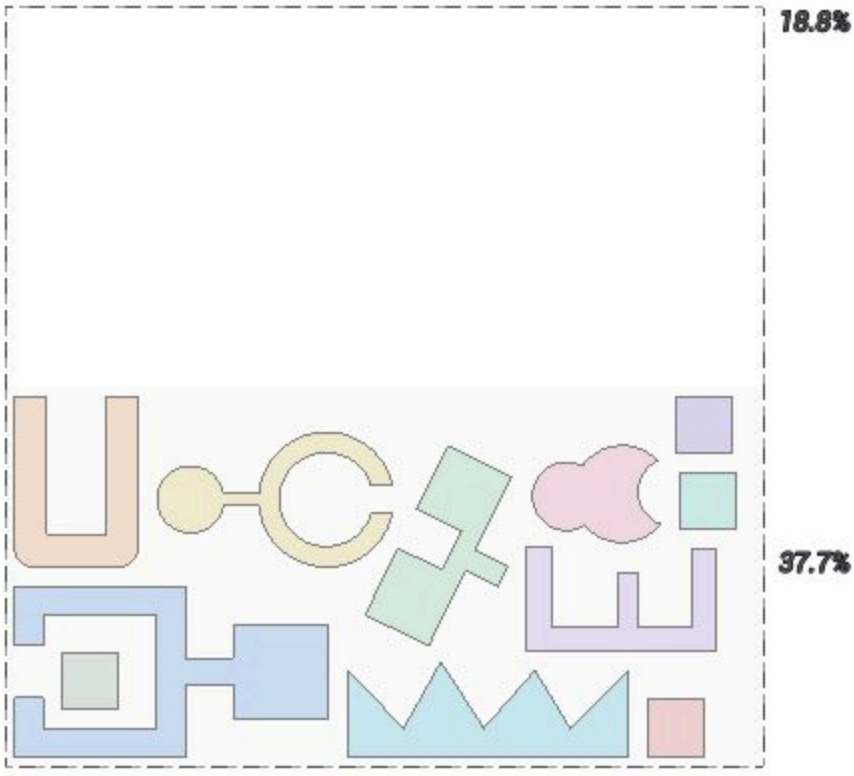
▼ Click here to expand..

Length. Defines the length of the current sheet.

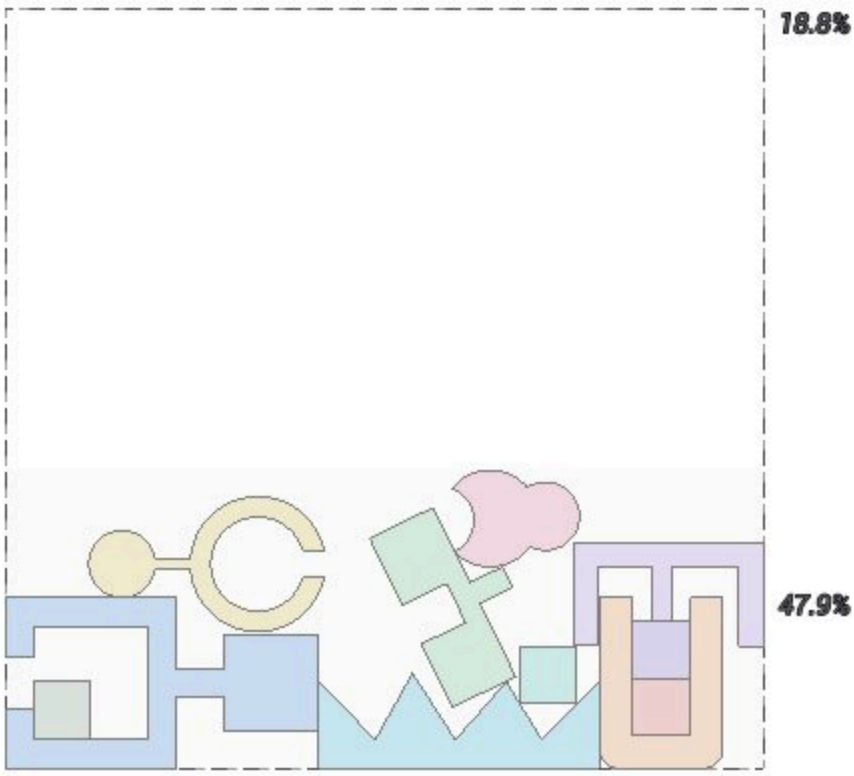
Width. Defines the width of the current sheet.

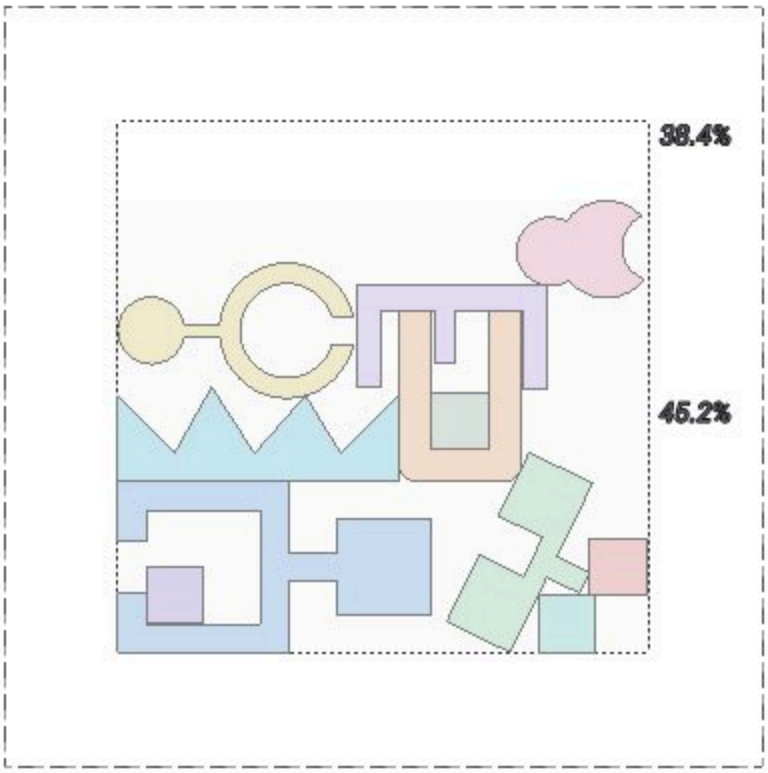
Gap. Defines the minimum distance between placed parts.


18.8%
47.9%


18.8%
37.7%

Margin. Defines the unused border area along the sheet edges.

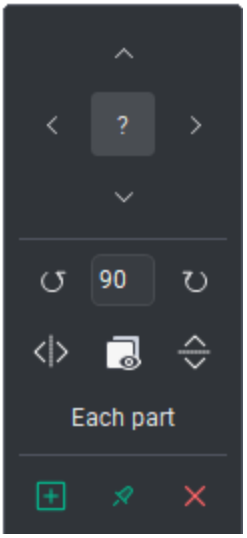




Manual editing.

Use Manual editing to adjust the arrangement of parts on the current sheet.

▼ Click here to expand..



Arrow buttons (←, →, ↑, ↓). Move the selected part until it touches an adjacent contour or the edge of the sheet.

? Hides tooltips.

Rotate. Rotates the selected contour around its center by the specified angle increment.

Hold R and scroll the mouse wheel to rotate the selected contour around the cursor position. If multiple contours are selected, hold E and scroll the mouse wheel to rotate each selected contour around its own center. Each mouse wheel step rotates the contour by 5° by default. Hold Shift for 1° increments or Alt for 90° increments.

Angle like a sample. Copies the rotation angle and mirroring from another contour. Select the contour you want to modify, press A, and then click the contour whose properties you want to copy.

Mirror. Mirrors the selected contours horizontally or vertically. If multiple contours are selected, enable Each part to mirror each part relative to its own center.

Remove. Removes the selected contours from the nesting layout and adds them to the Unnested Copies list.

Add. Adds the contours selected in the left panel from the Unnested or Nestable list to the current sheet.

Placed by Hand. Locks the selected contour in its current position on the sheet.

Editing behavior:

- Hold Ctrl to perform these actions without collision or sheet boundary checks.

- A red outline indicates a copy that violates the required clearance from an adjacent part or the sheet edge.
- Ctrl+Z and Ctrl+Y work throughout the entire window. They can undo or redo manual edits, Run, Remove Sheet, and changes made to sheets.
- Removing a sheet or changing its size clears the Placed by Hand status.

Settings button.

▼ Click here to expand..

The New sheet defaults popup is opened from the gear icon in the top-right corner. It defines default parameters for sheets added to the nesting.

Stock format. Selects a predefined stock format or Custom for manually specified dimensions.

Length and width. Define the dimensions of every newly added sheet.

Tech gap. Defines the technical gap used when placing parts on a new sheet.

Margin. Define the unused top, bottom, left, and right areas of every newly added sheet.

Creating the layout.

▼ Click here to expand..

Open the Model Tab

In Ency, go to the Model tab.

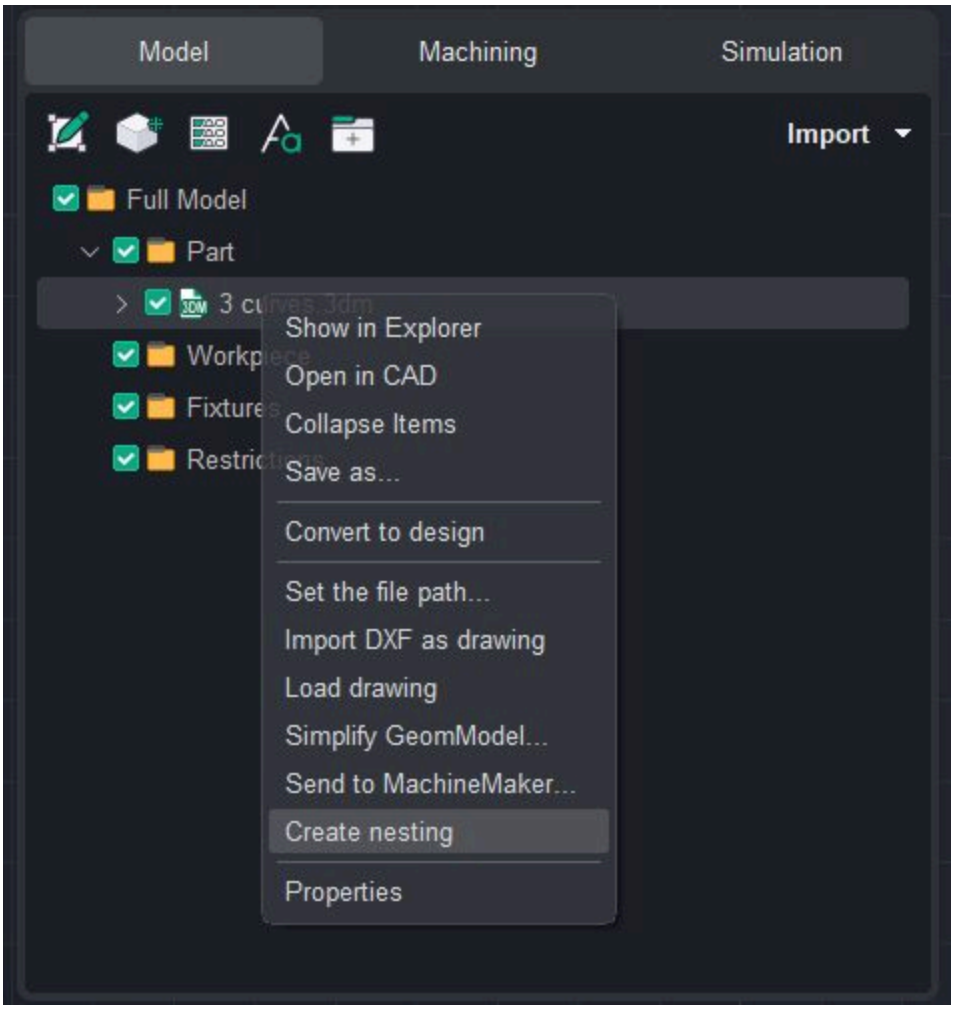
Import the Model

Import the following model from the standard Ency library:

Milling_25D\3 curves.3dm

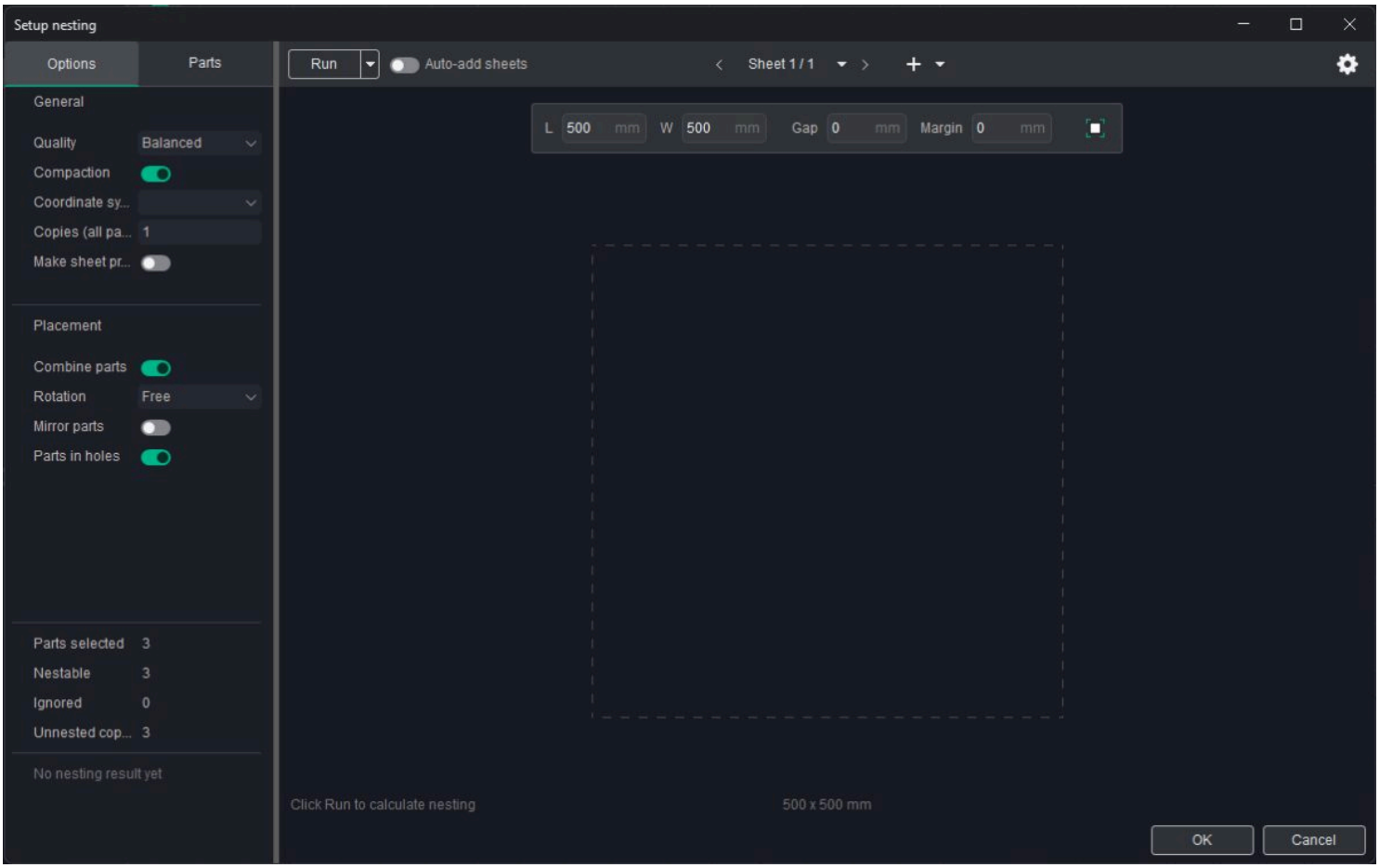
Create a Nesting Layout

Right-click on the contour and select Create Nesting from the context menu.



Open the Nesting Window

The Setup nesting window will open. This window contains the parameters used to configure the nesting layout.

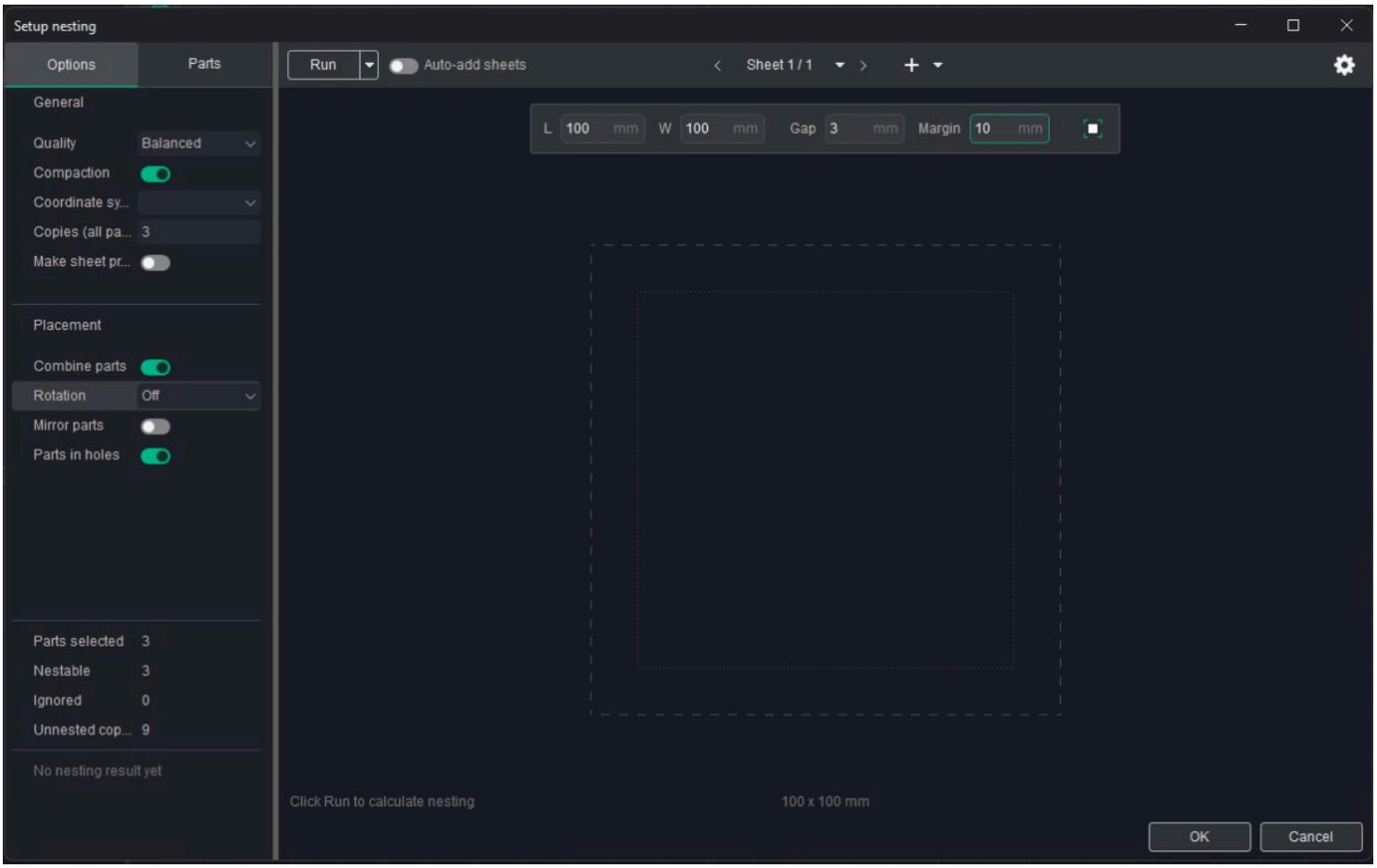


Set the Nesting Parameters

Set the following parameters:

- Copies — 3
- Rotation — Off
- Width — 100 mm
- Length — 100 mm
- Gap — 3 mm
- Frame — 10 mm

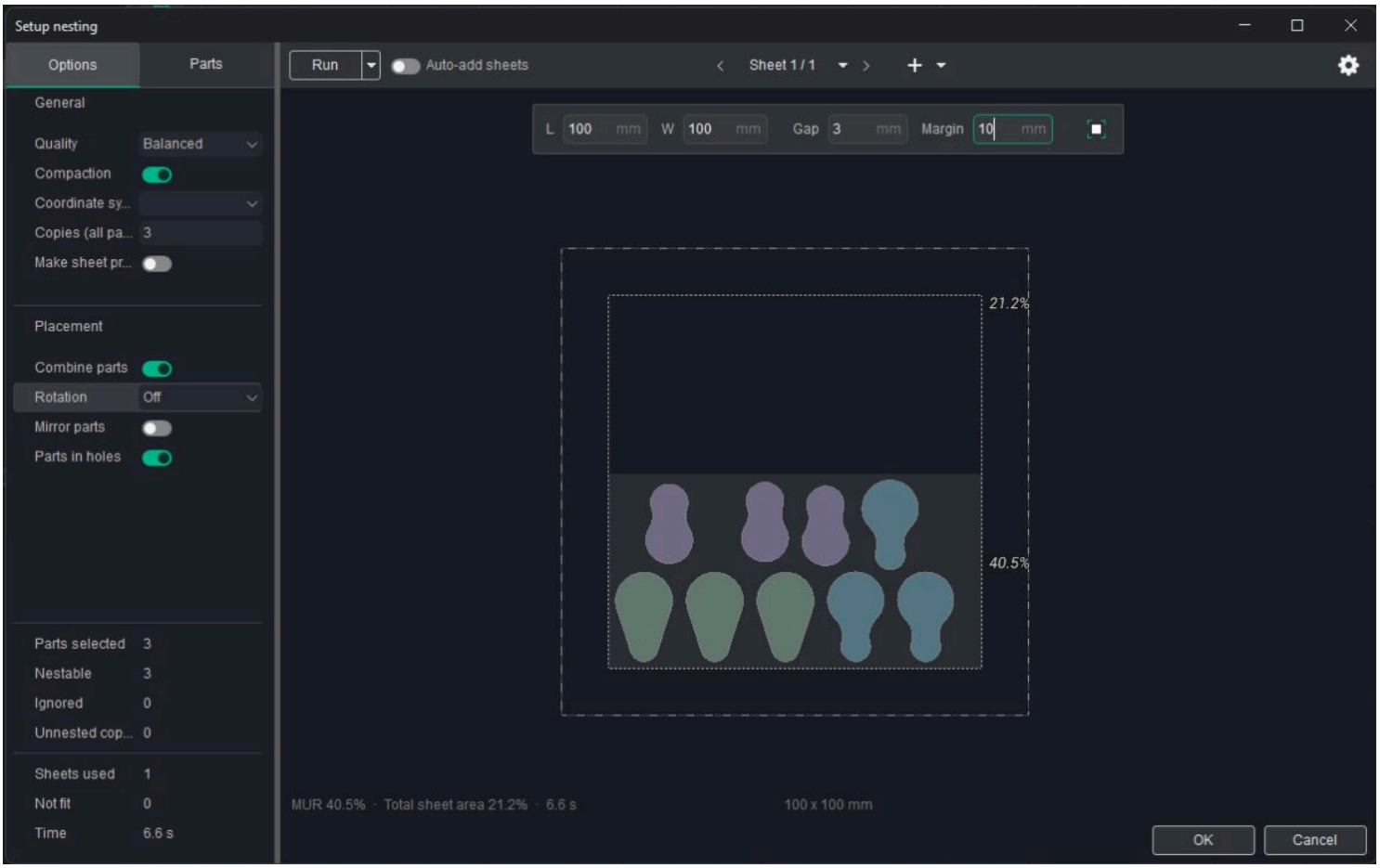
Make sure all parameters are entered correctly before proceeding.



Run the Nesting Process

Click Run and wait for the nesting process to complete.

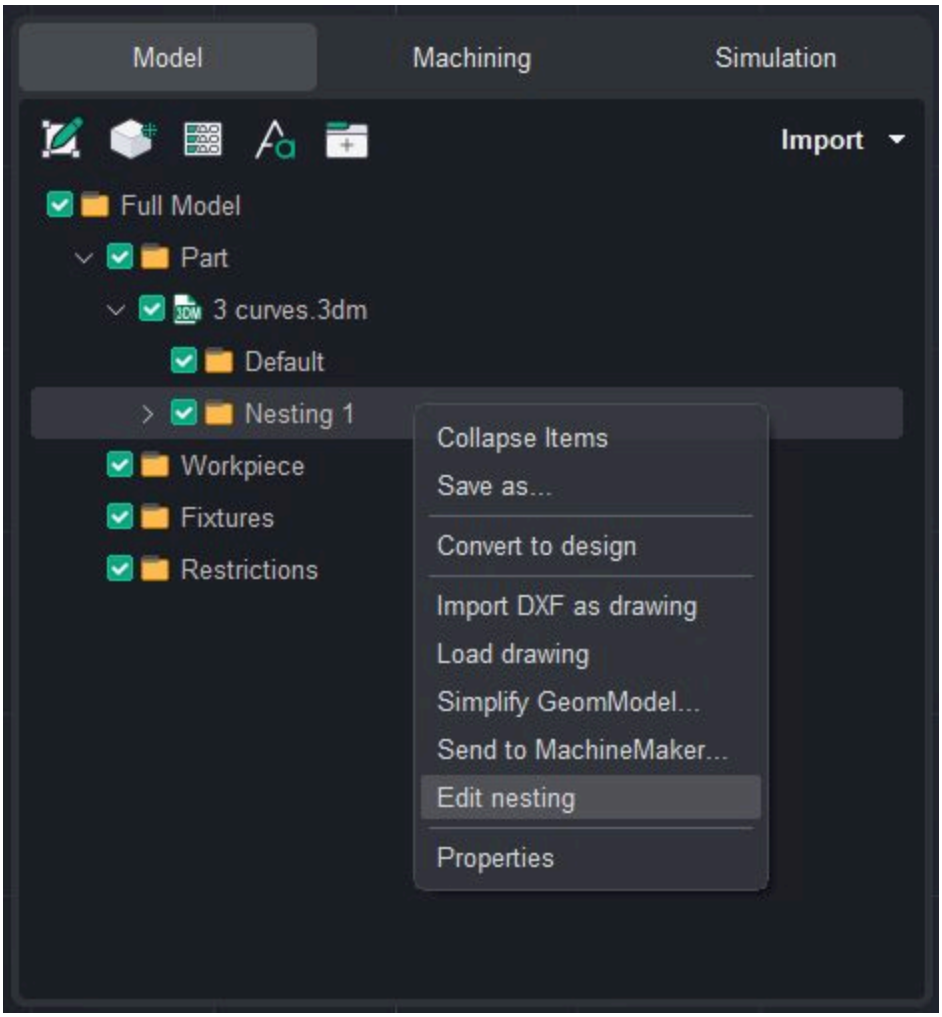
Once the layout has been generated, review the result. If you are satisfied with the arrangement of the parts, click OK to confirm it.



Review and Edit the Result

After confirming the nesting layout, a new folder containing the generated layout will be automatically created in the Parts folder.

If necessary, the generated nesting layout can be edited manually.


Model
Machining
Simulation
Import
Full Model
Part
3 curves.3dm
Default
Nesting 1
Workpiece
Fixtures
Restrictions
Collapse Items
Save as...
Convert to design
Import DXF as drawing
Load drawing
Simplify GeomModel...
Send to MachineMaker...
Edit nesting
Properties

Creating machining technology

This section describes machining technology creation: the structure of the machining mode and operation groups, sequencing, operation types, and milling, turning, EDM, robotic, additive, and other machining processes.

Common principles of technology creation

The process of creating a machining technology in Cam system in general consists of the following consecutive steps:

1. Selecting or creating the machine. The machine schema is created in CAM system format if needed.

- During the opening, CAM system loads the last used machine. If you need another machine for your purposes, you can select it in the machine selection window (you might need to specify the directory where the machine representation files are located).
- If there is no item for your particular machine in the library, you would need to create it using the Machine Maker or manually editing the.xml file.

2. Importing the part(s) and setting up their positions. System supports Part (group), also you can define different positions for each part using the Setup Stages (below there is more information about setup stages and project structure). Additional mechanisms and devices can be added to the project as fixtures.

3. Creating the operations describing the technological process. For each operation you need to select the tool and define the job assignment, the strategy and other parameters. After that you can calculate the toolpath, verify the trajectory using simulation and generate the CNC code using the postprocessor.

Machine schema

The machine kinematics is represented as the hierarchical set of nodes in the.xml file. Each node can represent a linear or rotary machine axis, a tool or a workpiece holder. A node can reference an image file with the 3d model.

Machining sequence

The operation is the basic unit of the technological process in Cam system which defines the particular way of manufacturing/production. The main parameters of the majority of the operations are:

1. The cutting tool
2. Job assignment consisting of the geometrical objects such as splines, faces, edges etc. These objects define the machining toolpath
3. The strategy and other parameters

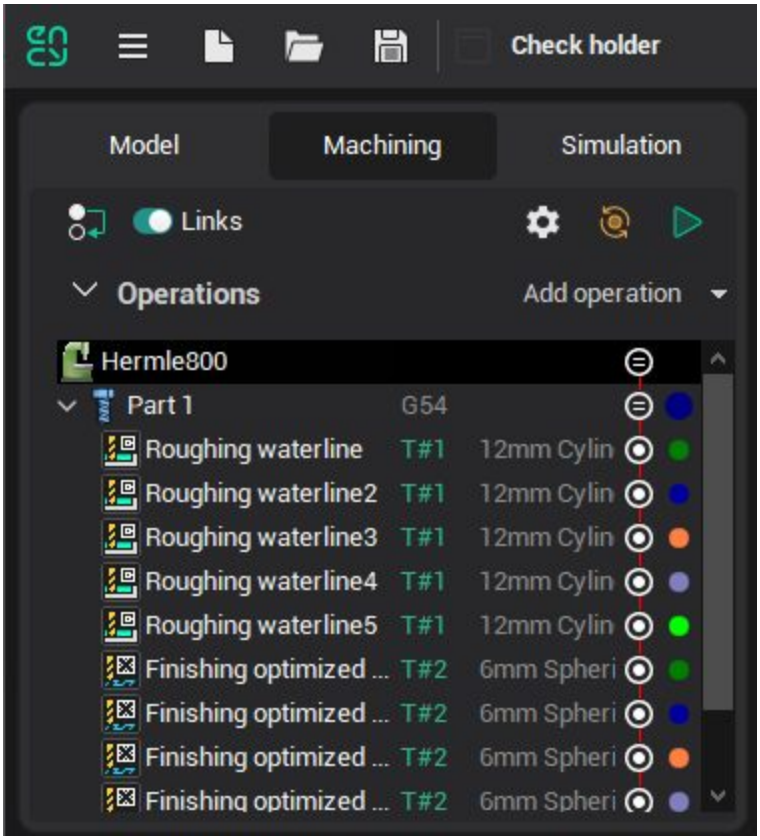
See the Defining machining sequence article for more info about the operations' common user interface.

Operations tree

The machining sequence in Cam system is organized into a hierarchical tree with operations as nodes. A machining tree may have an arbitrary complexity to meet any certain requirements. An example of such tree is shown below.

Basic node of the tree is an Operation. In particular the operation is determined strategy of parts machining containing the set of parameters, that is individual for every type of machining.

To bring an order into the machining sequence the operations groups, such as parts and setup stages, are introduced.



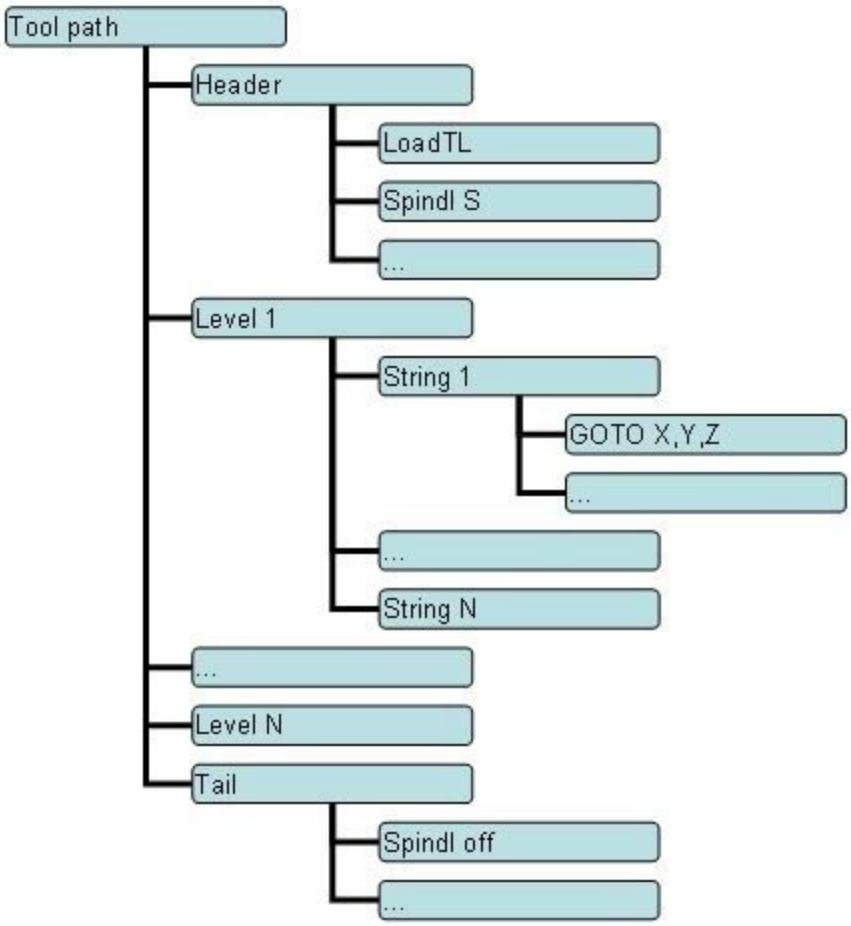
Operations tree example.

The operation tool path

The tool path at the bottom level is a sequence of the machine commands in the CLDATA format that include both the tool motion commands and the technological commands such as the feed rate switch, spindle and coolant switch on/off, etc. A tool path has a hierarchical structure like the machining tree: the elementary commands are united into the logical groups. The types and the contents of groups depend on the operation type.

The work on creating a machining sequence as well as defining its input data and specifying the parameters of operations is performed on the Machining tab.

For example, the structure of the tool path of the Roughing plane is shown below:

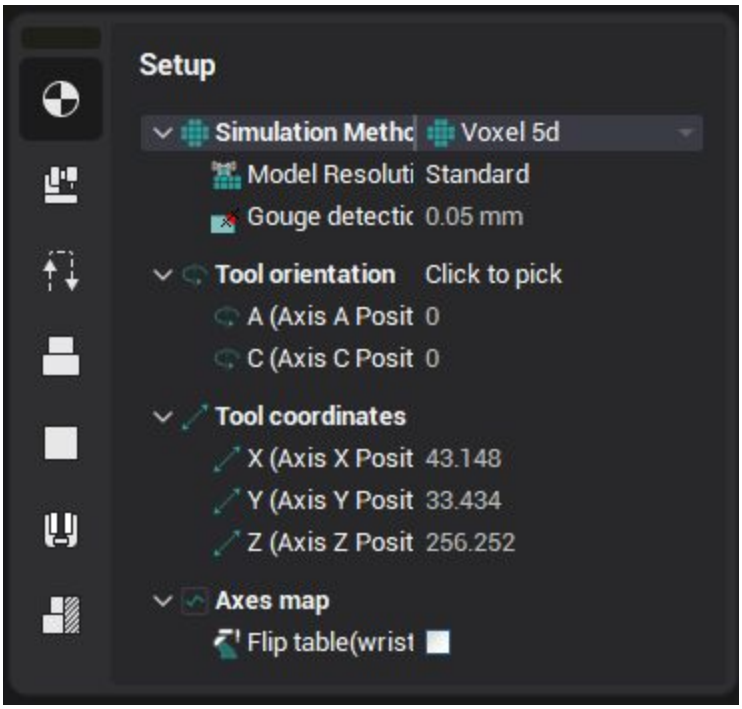


Setup stages

Setup stage is the special group of operation to machine the once placed part(s). If you need the manual part fixturing you need to create another setup stage. A new setup stage contains all the same parts that the previous stage contains. All the parts can be overturned or fixed in another place, but the initial workpiece for these parts are taken as the machining result of the previous stage. If create/destroy/rename a part in a setup stage, it will be created/destroyed/renamed in all the next stages.

The workpiece position is defined by the Workpiece connector and Workpiece setup parameters. If the setup stage contains one or more parts, these parameters are hidden, rather they must be defined for each part separately. The operations which are inside the setup stage or part group don't have this parameters, meaning that they work with the fixed workpiece (only special Move part operations can also alter the workpiece setup). If these parameters were changed for an ordinary operation in the project of the older version, a special compatibility mode is activated after opening such project, which enabled the workpiece setup parameters for such operations.

The machine (the very first operation of the tree) also has the workpiece setup parameters for the usual mode of working with the single part.

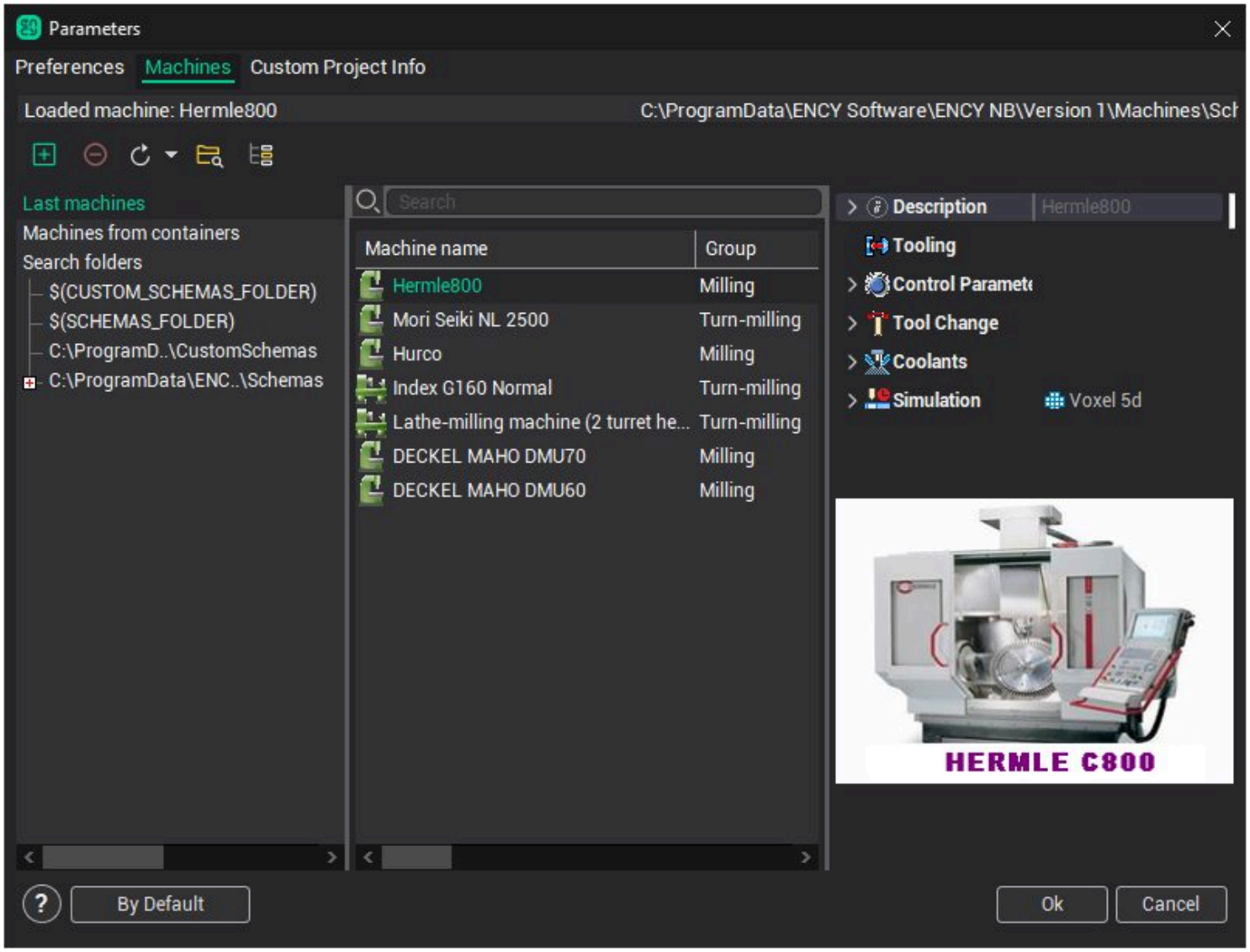


Workpiece setup parameters in the Setup stage operation.

Selection of a machine and its parameters definition

The first step of the working process after new project creation is selection of an appropriate machine. The type of the machine and its parameters define the list of available operations, the operations capabilities, the default parameters, as well as the subsequent behavior of the system. For example, if a lathe machine is selected, then only turn operations are available, for a mill machine there are only mill operations available, while for a mill-turn center are both turn and mill operations available. Selecting a five-axis mill machine makes it possible to indicate the position of the rotary head in a number of 3D operations, and so on.

To change a machine, you need to select the root node in the machining tree, then press the Parameters button.



The above shown dialog will appear. On the left side of this window is the list of folders where system will search machine files (they are united in Search folders group). And three additional groups are show also

at the top:

- Last 30 machines. The list of 30 machines recently used on the computer, sorted by date of last use (after the first installation on a new PC it contains machines which defined by the supplier of the program).
- Top 30 machines. List of the most frequently used machines on the computer, sorted by frequency of use (after the first installation on a new PC it contains machines which defined by the supplier of the program).
- Recent projects. The list of several last opened projects. They can contain machines, and you can select any of them.

When you select one of the groups or folders in the left list the middle panel of window will contain the machines that are in this group. To set the desired machine simply select one of the machines in the list and click "Ok". When you select a particular machine on the right side of the window shows its properties and image.

At the top is a toolbar with buttons to manage the folders and lists.

-  – It allows you to add a folder to the list of search paths. After pressing the button the standard folder selection dialog will appear. You can add a folder where you store all your machines.
-  – removes the selected folder from the search paths.
-  – search for machines' files in the specified folders and update the list. The same procedure is performed automatically when you open this window.
-  – opens the folder of selected machine or selected folder in Windows explorer.
-  – expands a folder in the left pane, in which is machine selected in the right panel. Allows you to quickly determine where the file of the machine.

By clicking on the Cancel button the window closes, and all the actions made by a list of machines and the current machine will not be saved.

Most parameters of a selected machine are shown in the properties' inspector in "read only" mode. Some basic properties are listed below.

The Machine name, NC system name, Developer and Commentary fields are filled by the machine scheme developer.

The Postprocessor file field specifies the name of the postprocessor, which will be used to generate CNC code by default.

The Interpreter file field specifies the name of the interpreter, which will be used to Simulation of machining by NC program.

In the Tool library field one enters the name of a tool library which the tools for operations will be selected from. This library is used both by automatic tool selection and by defining the tool manually via the operations parameters window.

The group called Control parameters contains settings for a used control. Here Tolerance (Digits) determines the number of digits after decimal point to output in CLDATA commands. Furthermore, a generated toolpath will not contain tool movements shorter than the specified tolerance. If the Use arcs box is unchecked then a generated tool path will contain only linear movements. If the box is checked, then arcs in selected planes will be generated, but only those, whose length is greater than the specified Minimal arc length, and whose radius is less than the specified Maximal arc radius. All other arcs will be approximated with lines.

The fields from Tool Change group contain coordinates of the appropriate points that will be used by tool path calculation.

The Scheme section defines the configuration of the used machine as well as availability of machine components such as index tables, rotary heads and other.

A machine is described as a tree of components, moving relative to each other. The root node Scheme corresponds to the machine base. The machine elements mounted direct on the base are listed under this node. Each node can contain sub nodes – the components, mounted on and moving relative to the parent. The leaf nodes of the tree have to be either a workpiece or a cutting tool. The way and direction the component moves is described in the fields Axis Type and Direction. The axis type can be either linear or rotary. For rotary axes the Direction defines orientation of the revolution axis.

The Address field specifies a prefix, by which the component is addressed in an NC program. The Min and Max fields bound ranges of available element movements. The Point field defines coordinates of the point in the machine coordinate system which the node axis passes through.

The distributive of system has an extensive range of machines and robots of the various groups. If none of the available machines doesn't suit your needs, turn to your dealer or to the support team for a help.

The machine configuration is stored in a specific XML files. They can refer to 3D models of machine components, which are usually located in separate files *.osd or *.stl, as well as auxiliary *.xml and *.supplement files in the same folder as the main machine file. Sometimes files of the machine can be placed inside the encrypted zip-archive with the extension *.stfc or inside project *.stcp files.

Defining machining sequence

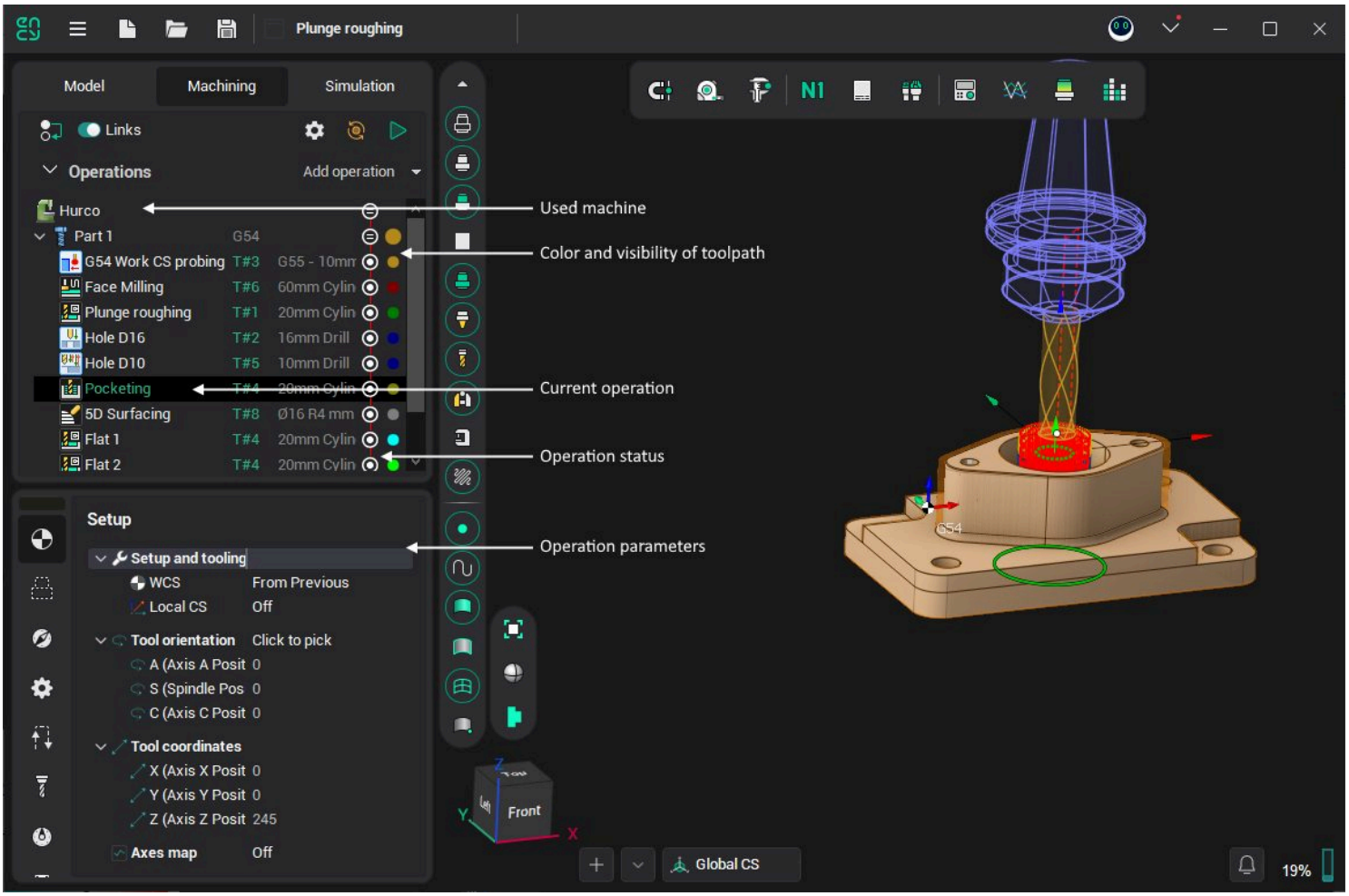
Before defining a machining sequence one should specify the produced part, the stock and the fixtures for the sequence. That is, one should define what is produced at first (the part), than what from it is produced (the workpiece) and only than how it is to be produced (the machining sequence).

The specified Part will be checked by all subsequently created operations irrespective to an operation machines the whole part or only some piece of it.

The Workpiece model should be necessary specified except of cases when the part is machined only with finishing operations without taking the initial and intermediate work pieces into account. The workpiece model needs to be a solid, i.e. represent a bounded space. The faces composing a workpiece are either to be sewed using prescribed tolerance or to close to the specified level. The workpiece also can be defined using solids generated on curves as well as swept around the part (e.g. box, cylinder around part). The workpiece defines the initial stock being modified by machining operations. That is, the workpiece of an operation is the machining result of the preceding operation. As result, editing some operation changes input data for all subsequent operations and involves its resetting. The workpiece of the machining sequence is also used in the Simulation mode as initial stock.

The Fixtures define the initial restrictions to be not machined. If no operation overrides the fixtures, these affect all the operations.

The Machining Result is the stuff leaved after machining of the workpiece by series of operations. The node is introduced to offer a visual check for the rest material as well as for transparence of the workpiece transmission over operations. The item is calculated automatically and its contents can not be edited. If the toolpath of operation is not yet generated then the machining result of such an operation is the same as its workpiece.



In the technology mode the top part of the window contains the structured sequence of machining operations as well as the nodes for easy access to the operations' main parameters. The bottom part contains the parameters of the selected node.

- New – opens the dialog to create a new operation. If the current operation is a group then the new operation will be created inside the current group. Else, the new operation is inserted after the selected ones.
- Parameters – opens the dialog to edit the parameters values of the current operation: tool, federate, approach-retract, strategy.
- Run – runs the tool path calculation process for the current operation. If the current operation is, a group then all operations inside will be calculated.
- Reset – resets the tool path of the current operation. If the current operation is a group then the tool path of all operations in the group will be deleted.

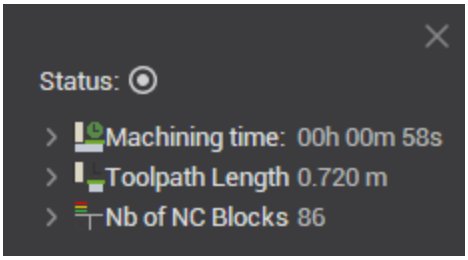
Current operation can be deleted, renamed, copied or cut to the clipboard by the standard keys or from the pop-up menu:

- [Del] – deletes the current operation;
- [Ctrl+R] – renames the current operation;
- [Ctrl+X] – cuts the current operation to the clipboard;

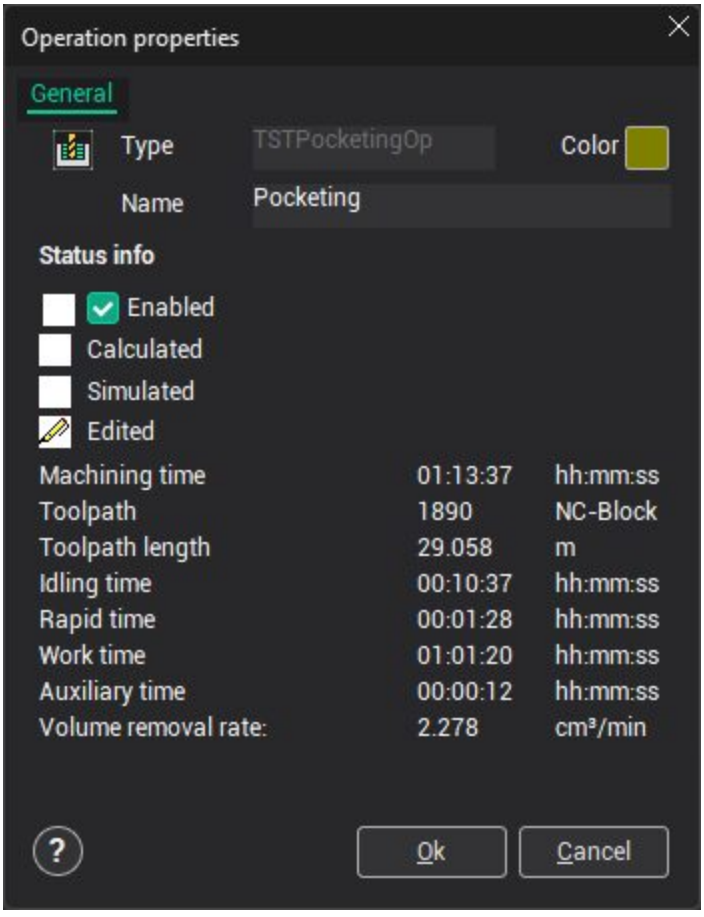
- [Ctrl+C] – copies the current operation to the clipboard;
- [Ctrl+V] – inserts the operation from the clipboard. Operation is added to the end of the current group.

Mouse can change the structure of the job tree. To do it press and hold the left mouse button on the required operation and move it to the required place.

There is the icon near the every operation. This icon shows the operation status. Details



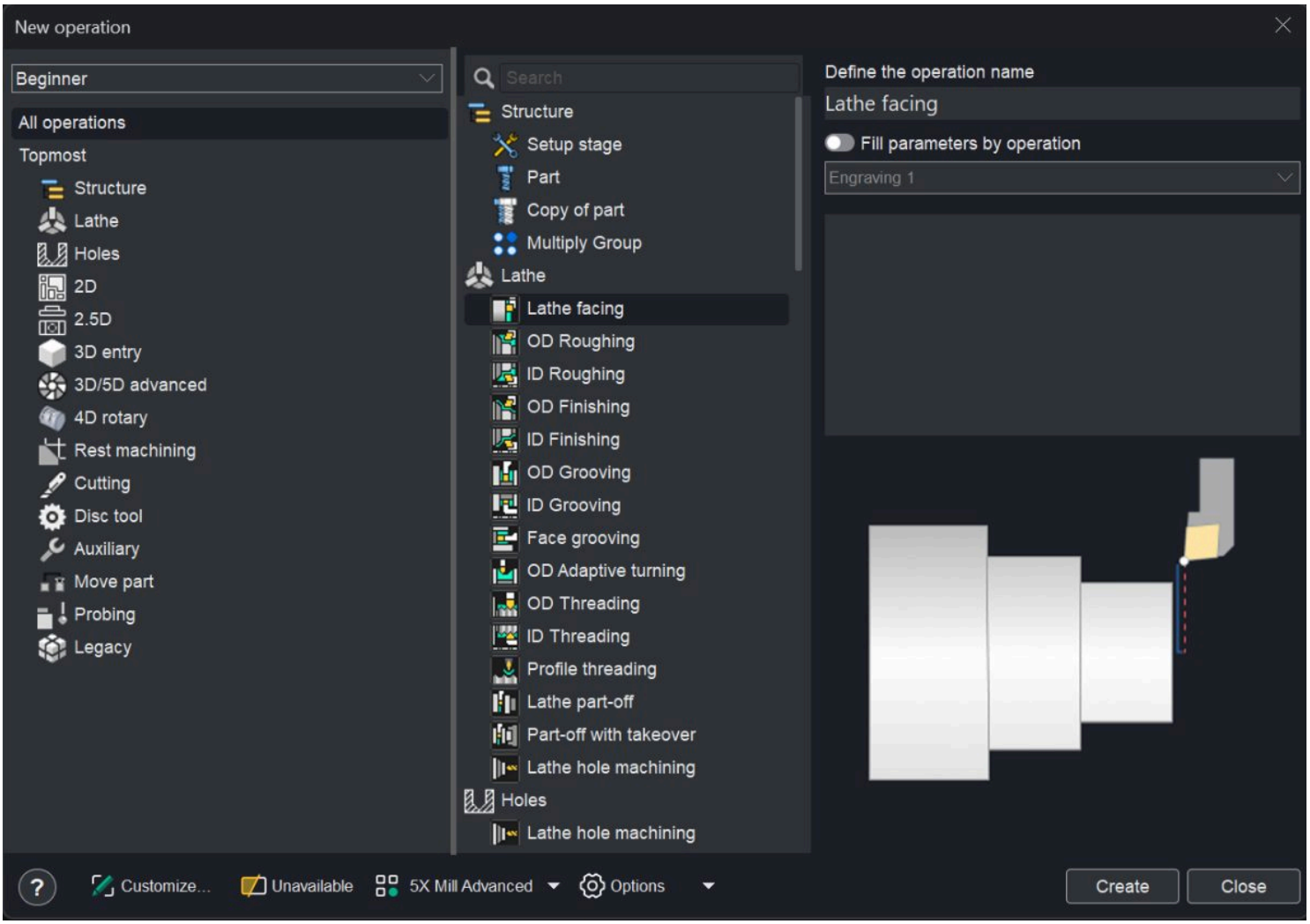
Detailed information about operation with the operation status is displayed in the property window. To open it click the right mouse button on the operation and select the "Properties:" item from the context menu.



The window shows the operation type and the icon corresponding to the type, operation name, tool path color. If the operation is calculated then the number of the commands in tool path and the machining

time is shown. If the operation is simulated then idle, work, rapid and auxiliary time is shown. If the errors were detected, while the simulation then the information about error is shown.

Creating new operation



Application area:

To add a new machining operation to the technical process, CAM system provides a dedicated creation window where you can browse, search, and configure operations before adding them to your project.

There are two ways to work when creating an operation:

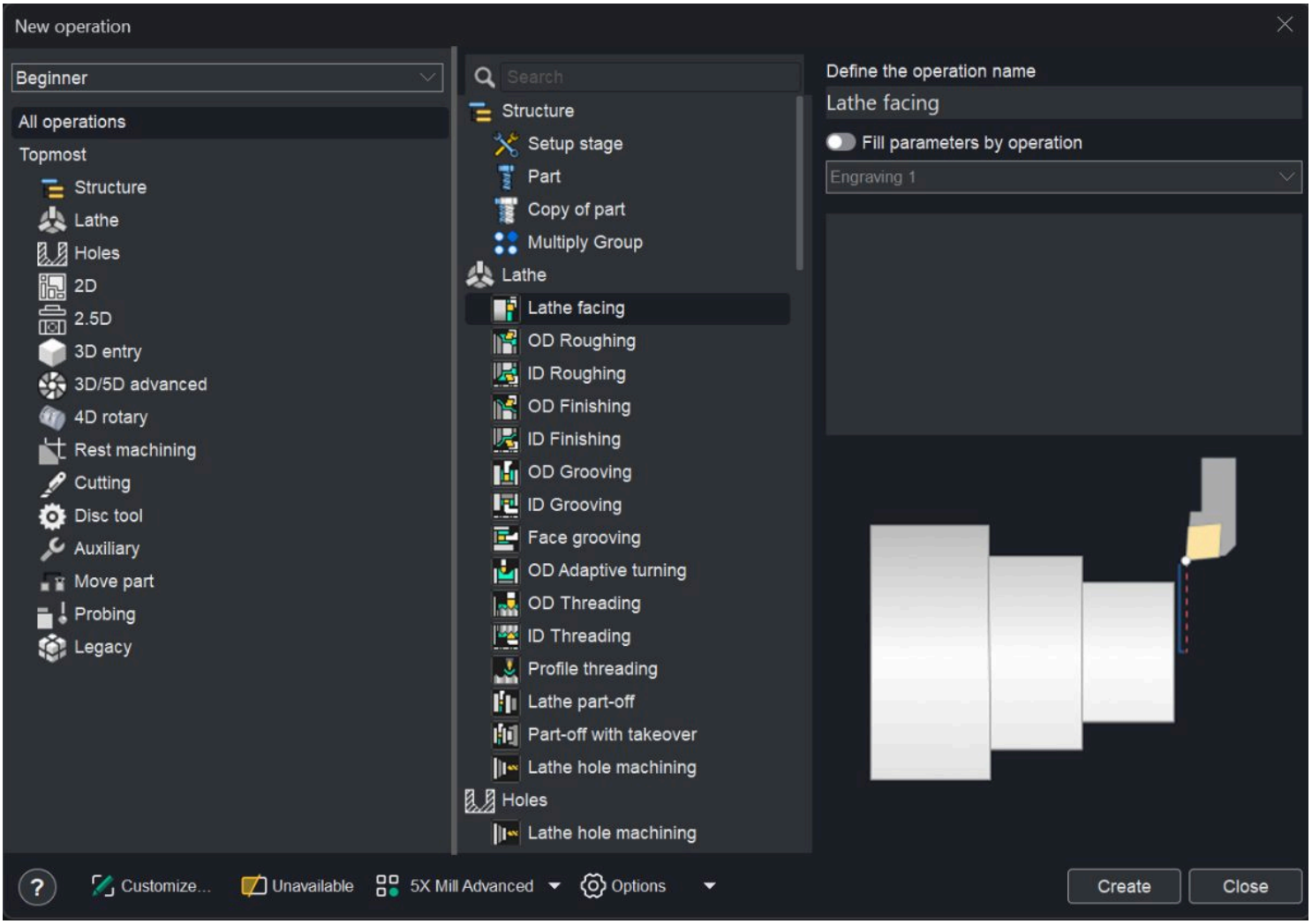
- The first one is more suitable for beginners who are getting familiar with the system.

▼ Click here to expand..
To open the window for creation a new machining operation just press the Add operation button.

After that, the operation creation window will appear. This mode is preferable for new users, as it allows them to view the operation details.

There are two ways to create an operation in the window.

- By double click of the left mouse button on the operation in the technical process, an operation of the appropriate type is adding. Thus creation operation window remains open.
- In order to create an operation select it in the list and click on the "Create" button, the operation will be created and the window will be closed.

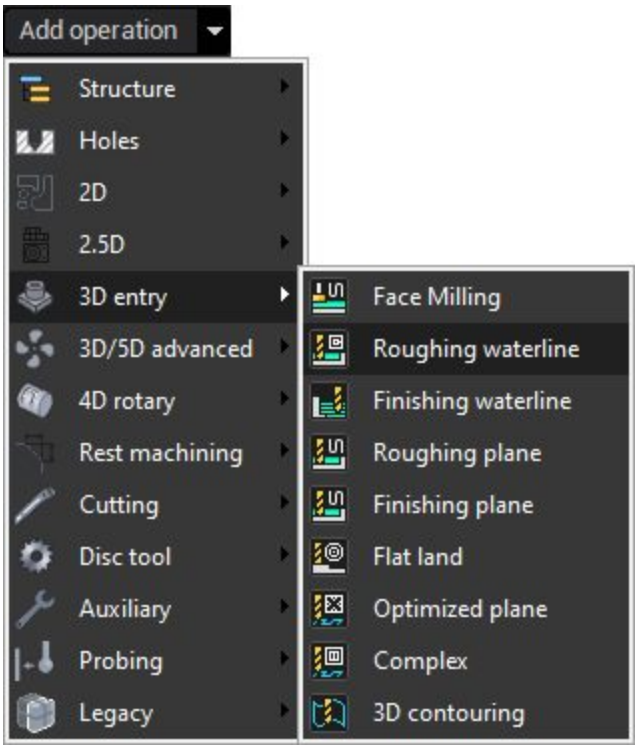


- The second one is intended for more experienced users.

▼ Click here to expand..

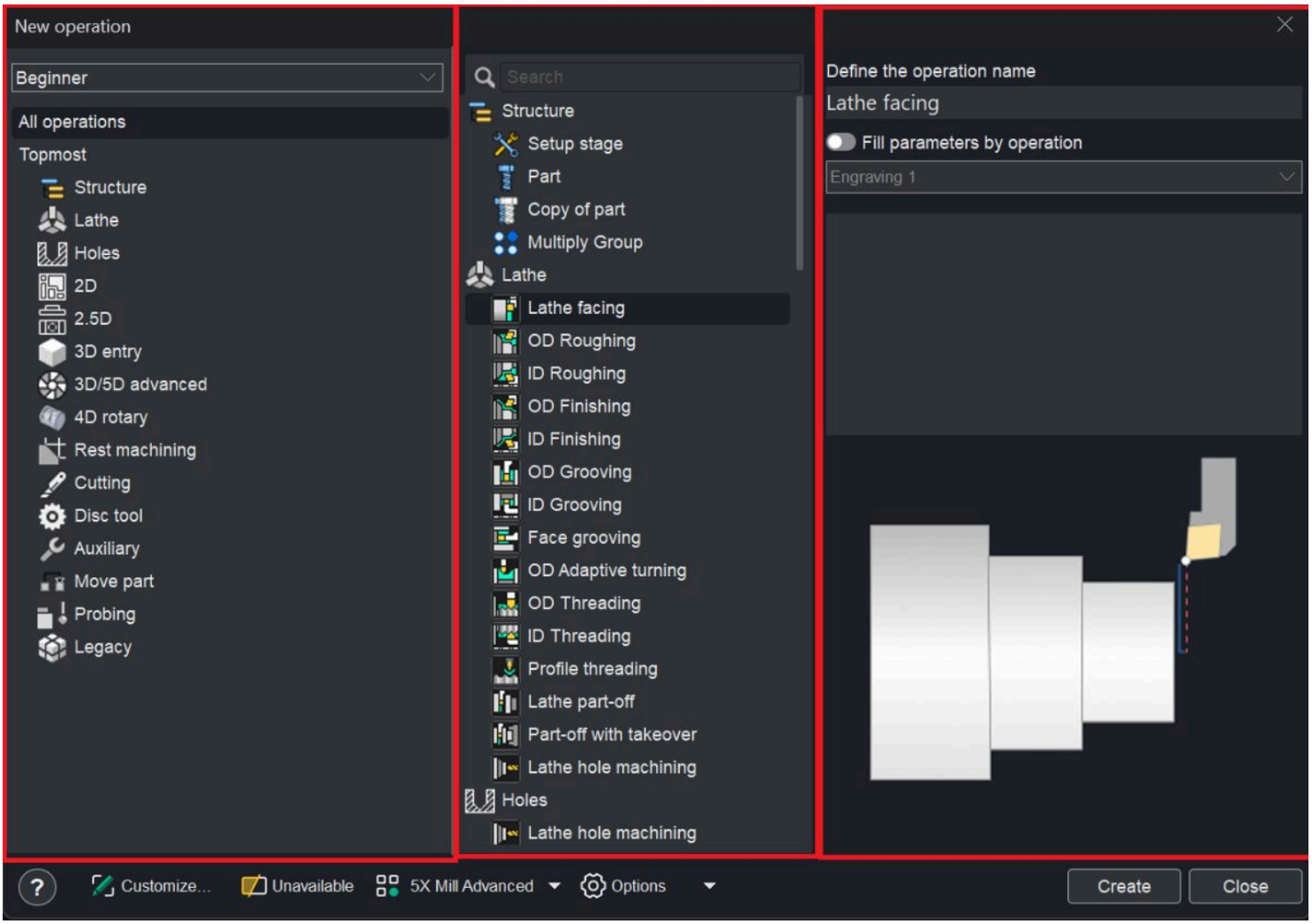
There is also one more quick and easy way to create a new operation. To create an operation, you can click the arrow next to Add operation and choose from the menu the desired type of operation.

This method of creating operations significantly accelerates the work, as it eliminates the need to constantly open and close the windows.



The creation window is divided into three panes:

- Left pane — lists operation groups for category selection.
- Middle pane — displays the operations belonging to the selected group.
- Right pane — shows details about the currently highlighted operation.



Right pane.

This panel presents details for the selected operation and supports the following capabilities:

Operation name specification. The name can be defined using the Define the operation name field.

Settings copying from an existing operation. Enabled via the Fill parameters by operation checkbox; the source operation is selected from the dropdown list.

Explanatory image with video playback. located at the bottom of the panel; a single click toggles video playback.

Toolbar.

The bottom toolbar provides functionality for managing the content displayed within the window. It comprises the following elements:

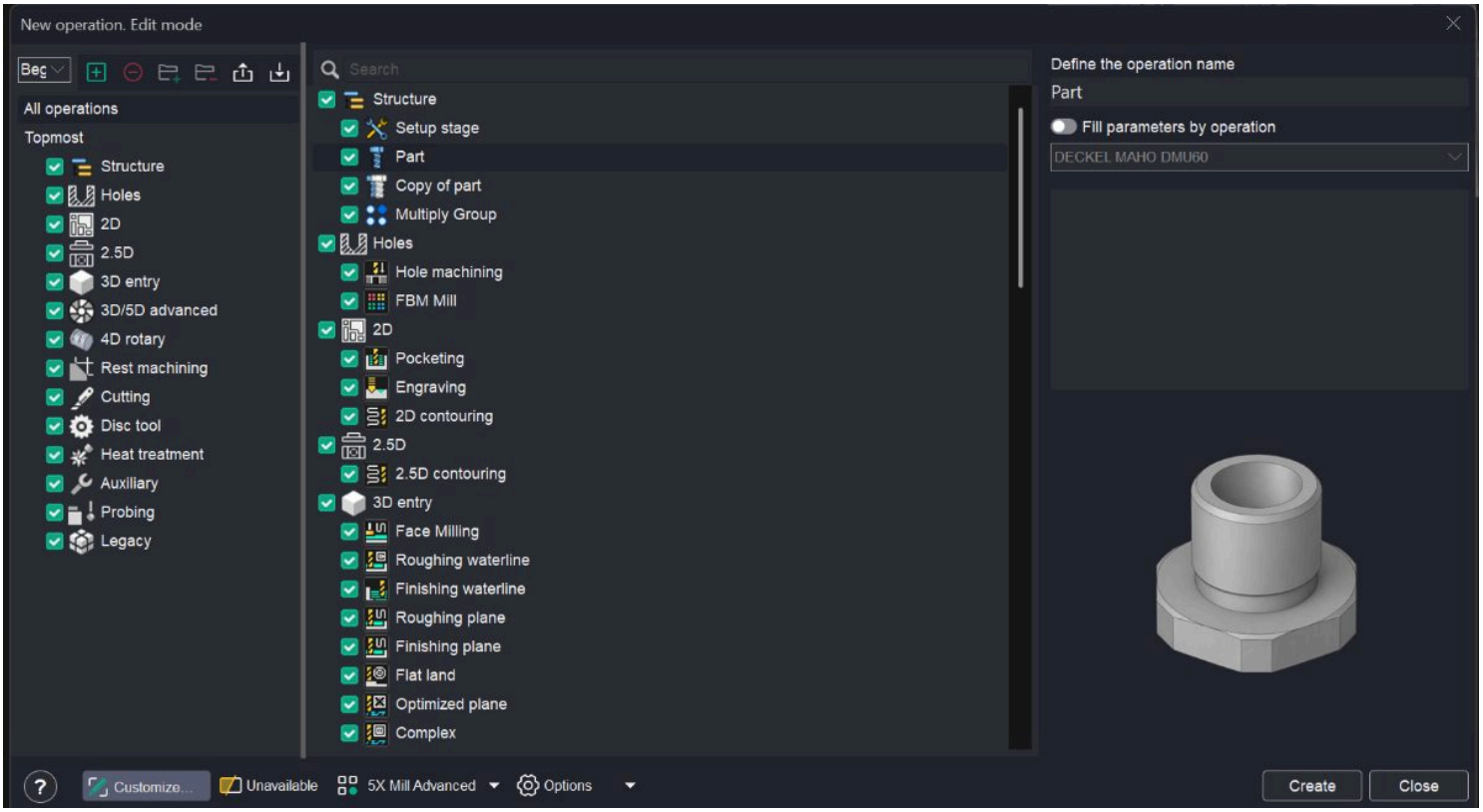
Question button. It allows you to open the help section for this topic.

Customize... Customize which operations and groups should be visible in this window and in Create operation dropdown menu.

▼ Click here to expand..

The updated window layout introduces enhanced controls for managing operations and groups, as detailed below:

- Additional controls adjacent to operations and groups — new buttons and checkboxes are displayed next to the names of operations and groups. The checkboxes govern the visibility of the corresponding operations and groups within the interface.
- Drag-and-drop functionality — operations can be repositioned between groups via drag-and-drop interaction.
- Group creation and deletion — new groups can be created, and existing groups can be deleted by using the dedicated buttons located in the upper section of the window.
- Group name modification — the name of any group can be edited by clicking on it and entering a new label.
- Group repositioning and nesting — groups can be moved to alternative locations within the hierarchy or nested inside other groups using drag-and-drop operations.



Create configuration. Adds the new configuration, all properties are copied from the current configuration.

Remove configuration. Removes current configuration. Last configuration in the list can not be removed.

Add group. Add new group of operations. Use it to create a sub-menu in "Create operation" dropdown menu. Click to a group to change its name. You can drag operations to any group.

Remove group. Remove selected group of operations.

Export current configuration. Export current configuration to external file. A ZIP archive will be created. The User defaults in the Inspector will also be exported. Details

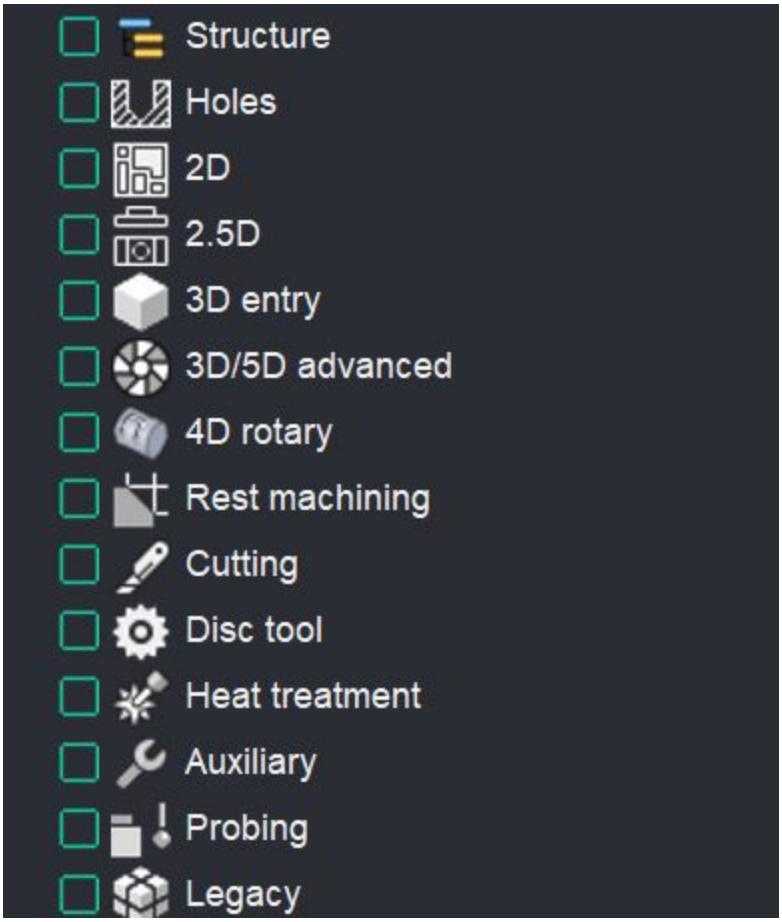
Import current configuration. Import current configuration to system.

An example of creating your own configuration.

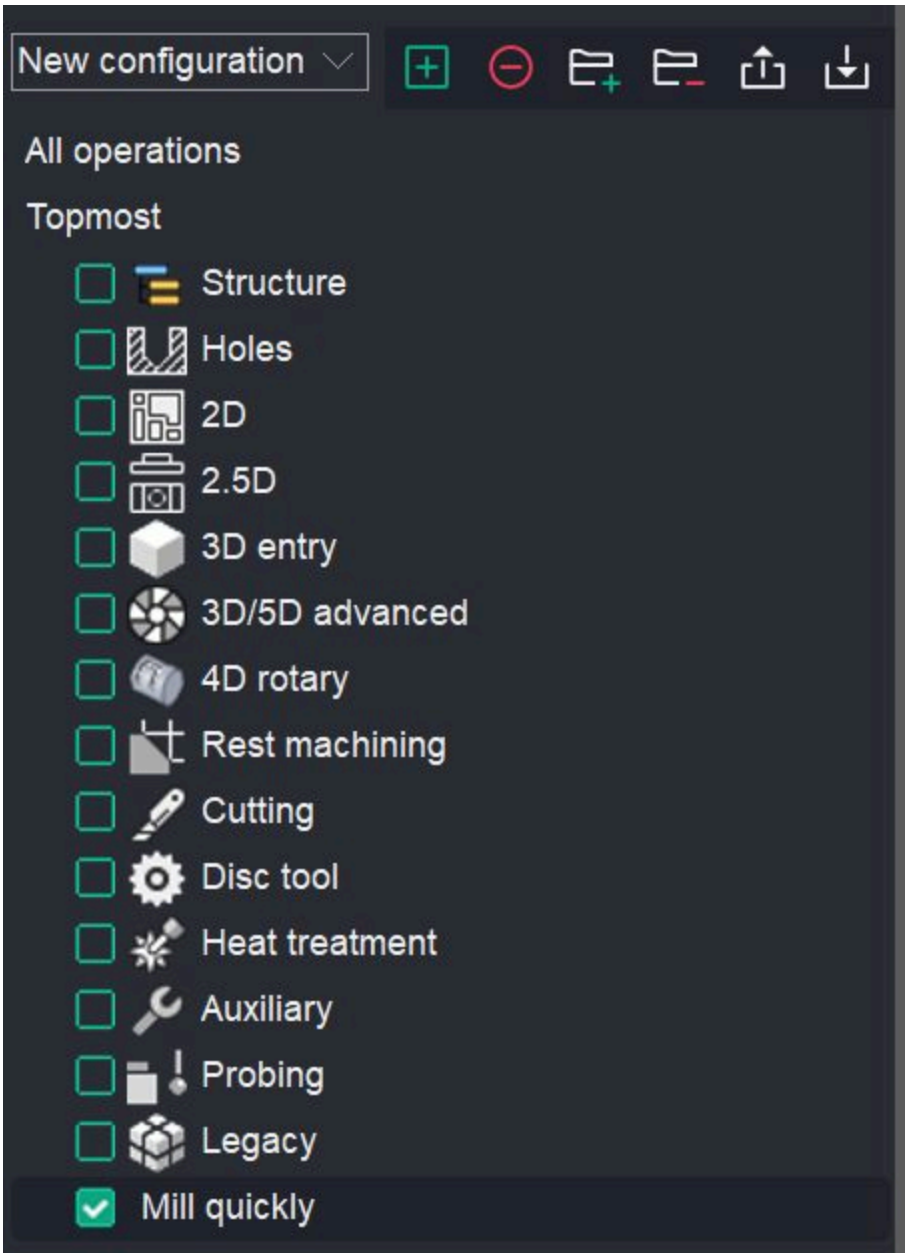
First, you need to create a new configuration. The name "New Configuration" will appear in the configurations window.



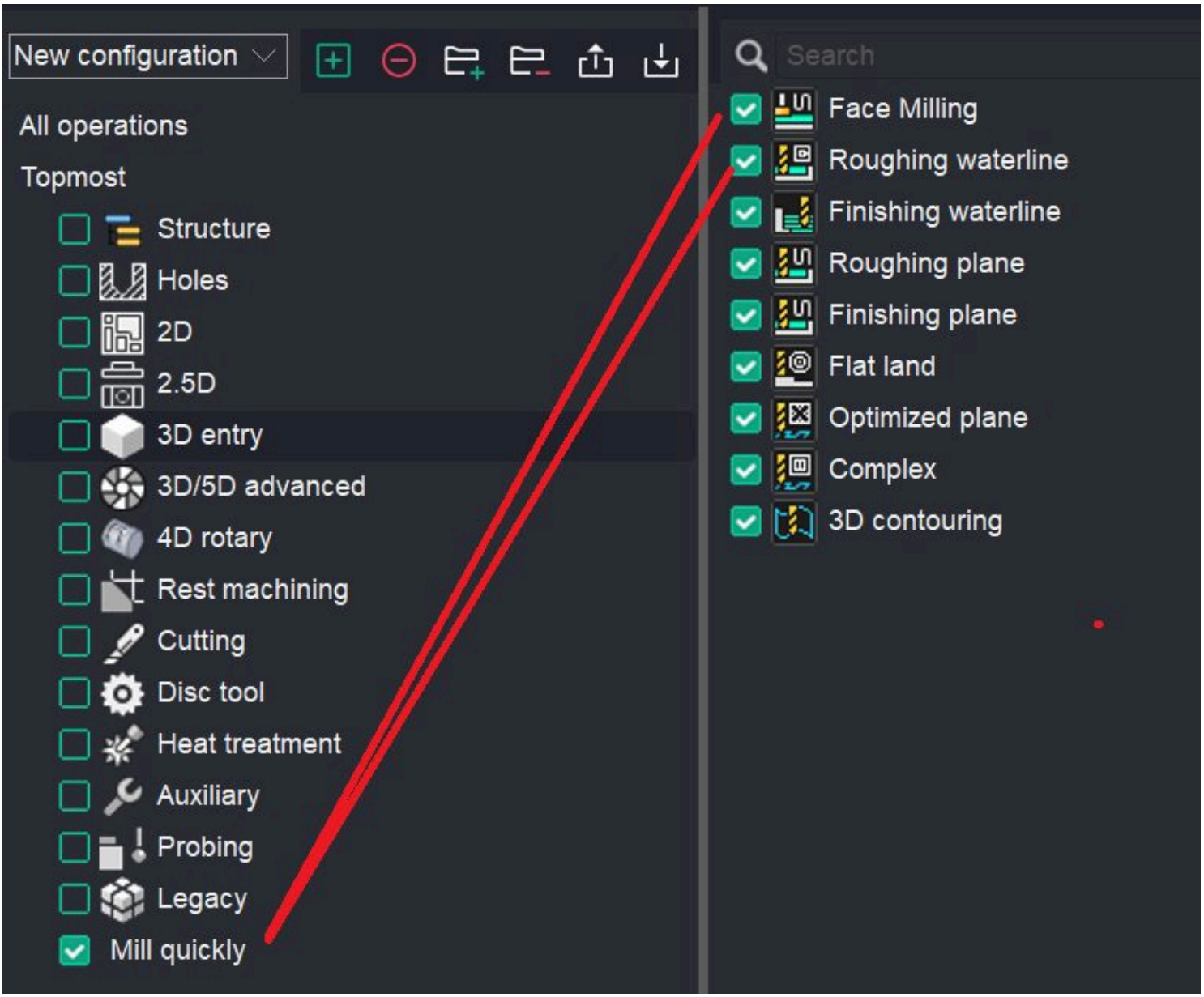
Disable all groups.



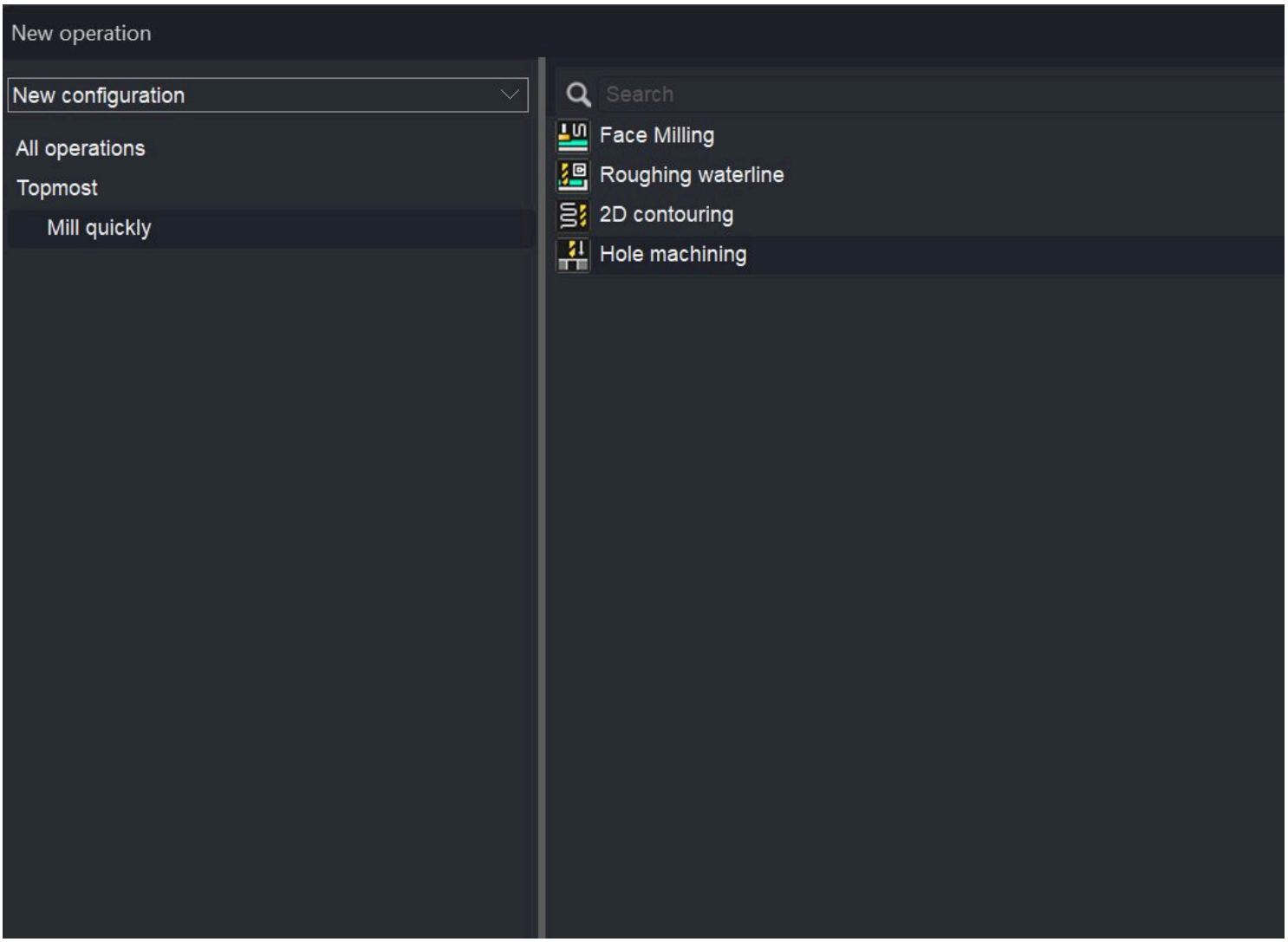
Use the Add Group function on the toolbar to create a custom group. Name it "Mill Quickly".



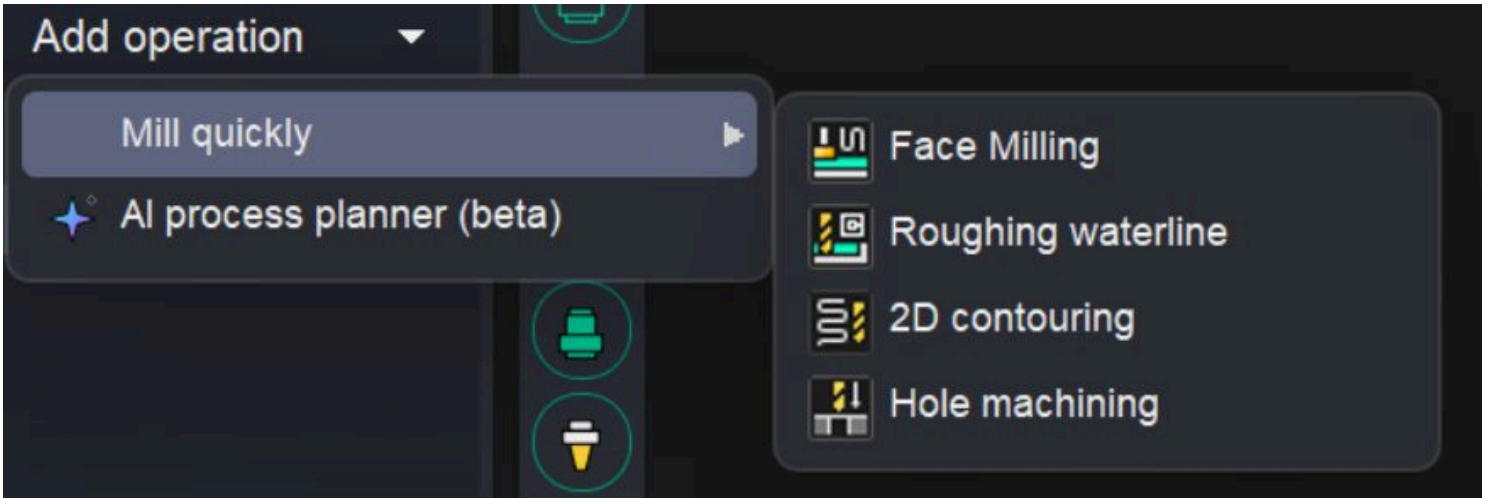
Drag and drop a set of operations into this group.



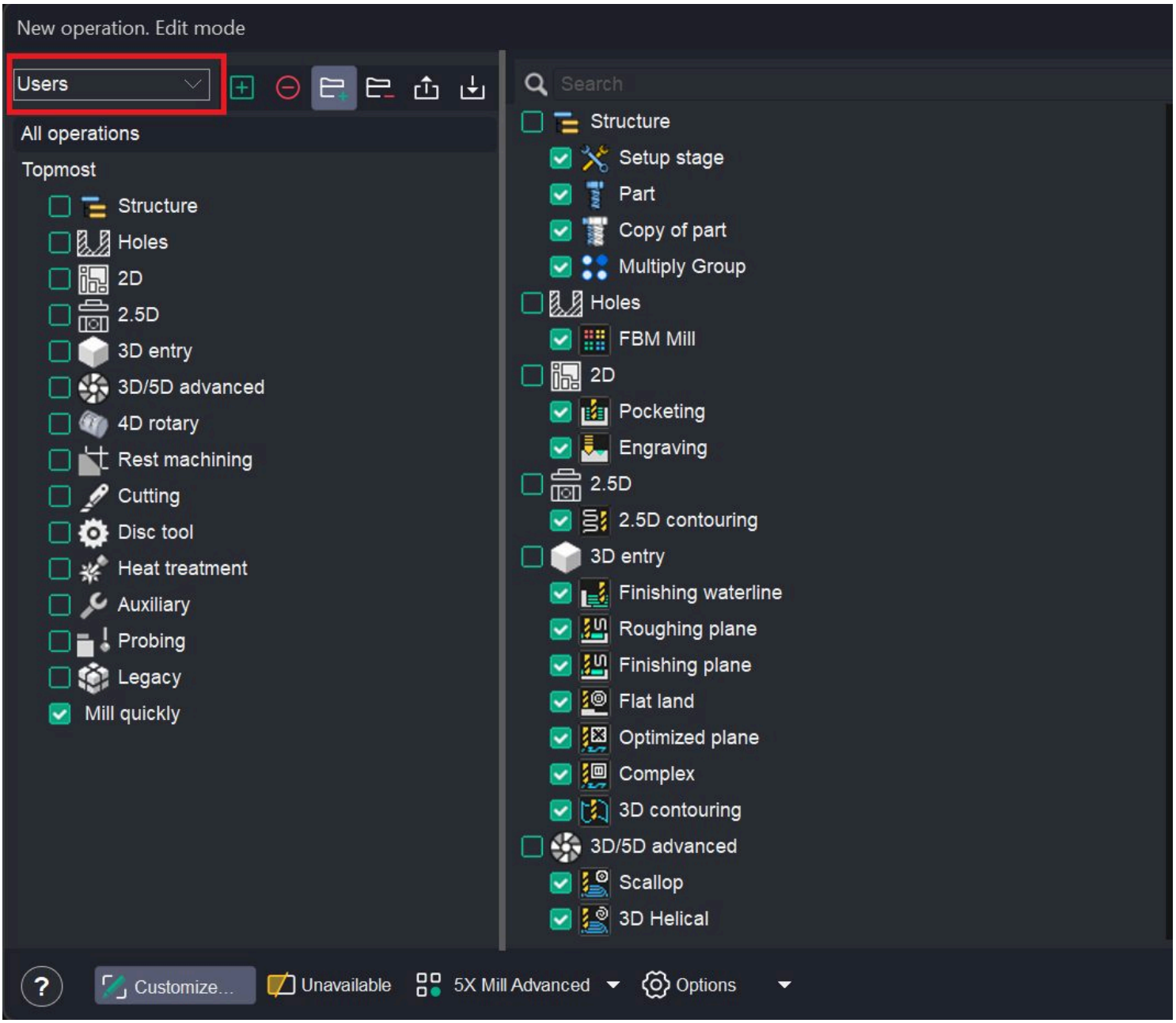
You can click the Customize··· button. Your configuration with the operation group will be displayed in the operation creation window.



Also in the dropdown list.



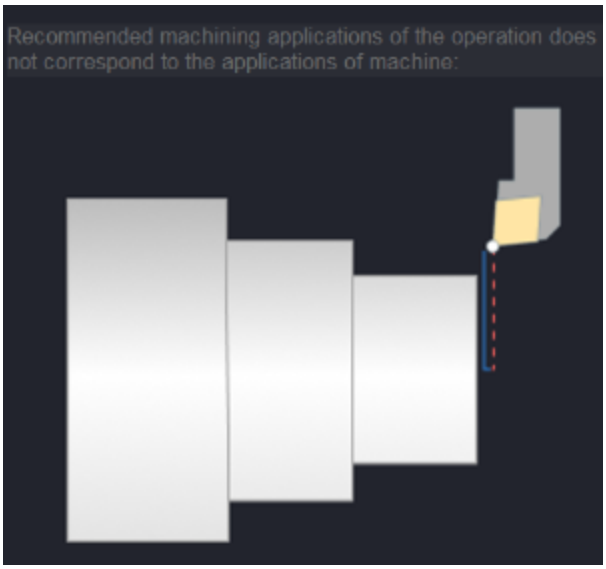
You can rename the configuration.



Unavailable. Show operations which are currently unavailable. Operations may be unavailable due to the machine type or license availability.

▼ Click here to expand..

The reason why an operation is unavailable is displayed in the right-hand panel.



Operation preset. Show operations preset for machining. (Express, Cutting, Cutting 5D, 2.5X Mill, 3X Mill, 3X Mill Advanced, Rotary, 5X Mill, 5X Mill Advanced). Depending on the selected preset, the groups and set of operations will change.

Options. It allows you to activate the options.

▼ Click here to expand..
Among the available options, there is a 3+2 mode option.

If the user has a license (e.g., 3X Mill Advanced) and this option is enabled (ON), the operation will be created with the Local CS parameter visible on the Setup tab. Details This enables machining in 3+2 mode.

If the option is disabled (OFF), the Local CS parameter will not be displayed.

This option also affects the following parameters:

- Approximate safe motions;
- Safe surface (Cylinder, Sphere);
- Rotary Transformations.

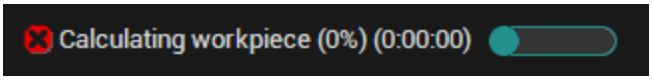
Executing operation

Having created an operation the system automatically assigns the values of all its parameters with regard to the machining method; model dimensions, selected tool etc., or it copies the parameter values from an earlier created operation. Thus, the operation is ready to be executed as soon as it has been created.

It is very easy for the user to alter, if necessary, any of the parameter values for the current operation. To edit the parameters for the current operation (highlighted) the Parameters button is used. One should note that altering the operation parameters might cause alteration of the toolpath and the order of machining commands. As a result, after modification of the parameter values for a calculated operation, the computation results will be reset and to obtain the new toolpath, the operation needs to be recalculated.

To perform the operation calculation, press the run  button. Calculation of the toolpath for complex models that have a large number of complex surfaces may take considerable time.

The process indicator at the bottom of the main window



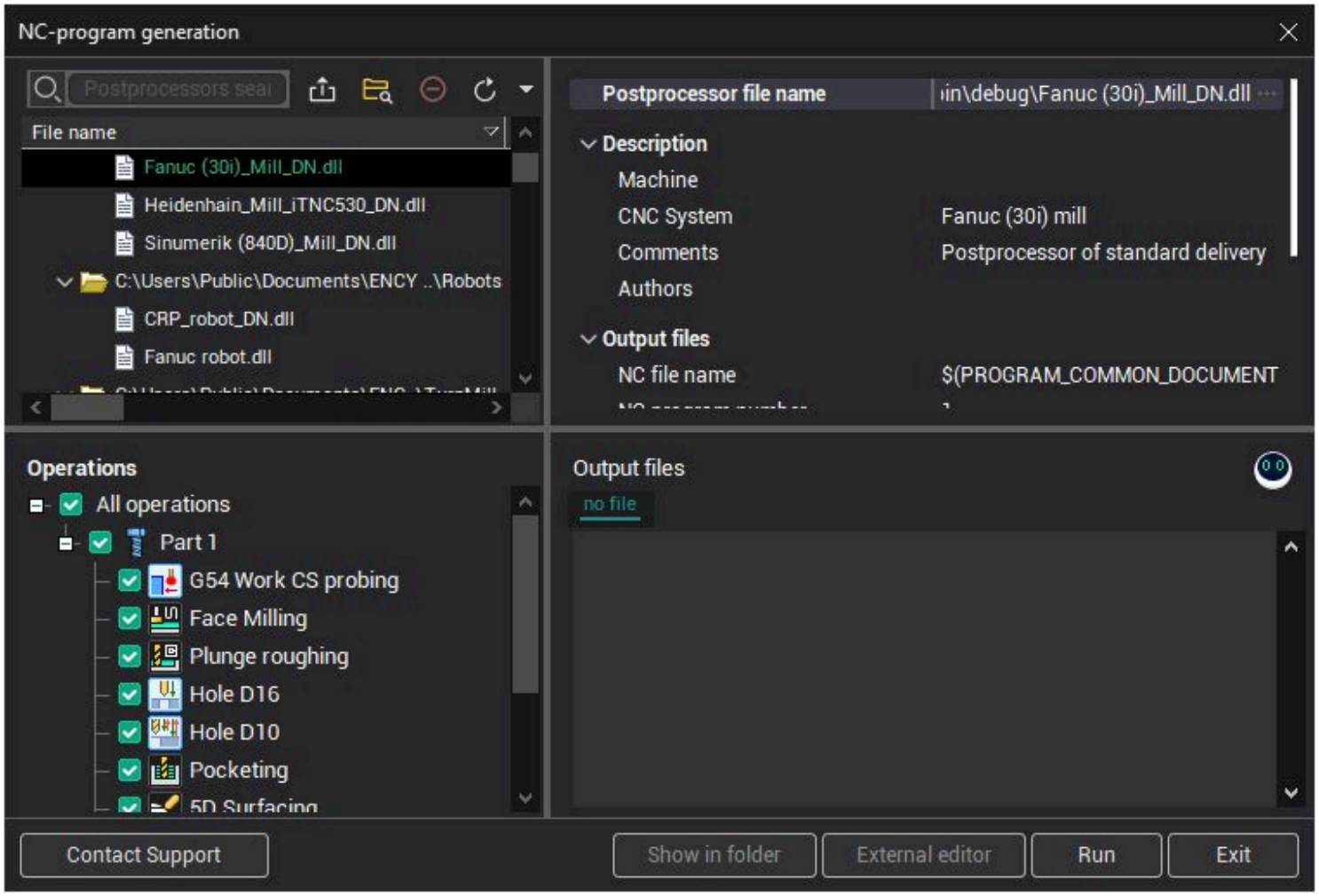
displays how far the current operation calculation has progressed. Clicking within the process indicator area will interrupt execution of the operation. The system will request confirmation of process interruption. If Yes is selected, the calculation will be cancelled, if No, the calculation will continue.

During the calculation process all visualization control buttons are enabled.

Generating NC code

Generation of NC programs is performed by a postprocessor that transforms the CLDATA commands of a calculated toolpath into the format of the selected CNC system. The postprocessor files have the *.sppx extension. The output of CNC code is performed on a standard text file, so it can be easily transmitted from the computer, where the distribution package is installed to the CNC control using any standard methods of copying files and folders.

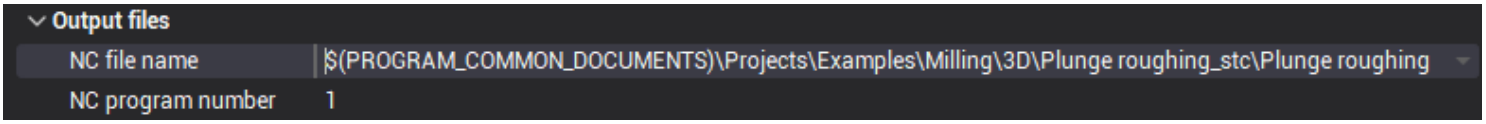
To generate the CNC code, press the N1 button. The NC-program generation panel will appear.



Here you can select the required postprocessor and run it to get an NC program. If the list doesn't contain the postprocessor you want, try to select another Folder with postprocessor files using the appropriate controls at the top of the panel. General information about the selected postprocessor is displayed in the left-hand bottom corner of the panel.

In the Output file field, one enters the name and the path to the file, which the NC program should be output to.

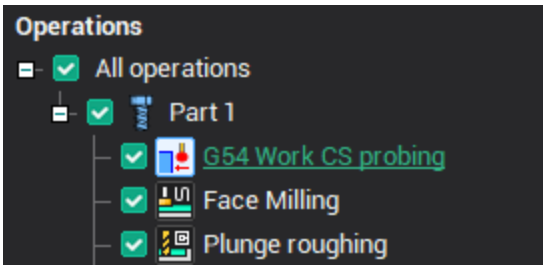
Use drop down button to see list of previous NC program files, or select output NC-program to the same folder as project or default folder. Default folder is set in System settings window.



To start generating NC program for the selected CNC system, press the Run button. In the full version of the program, NC code is written to the output file, and, if selected, into the postprocessor window. In the demo version, output into a file will not be performed, and only the last few strings of the NC program will be displayed in the postprocessor window.

Please note, that the postprocessor only creates NC code for all calculated machining operations that are 'enabled' (, or status icon in the machining process list window) and also have Generate NC code property set at the time the postprocessor is run (the operations tree below postprocessor list).

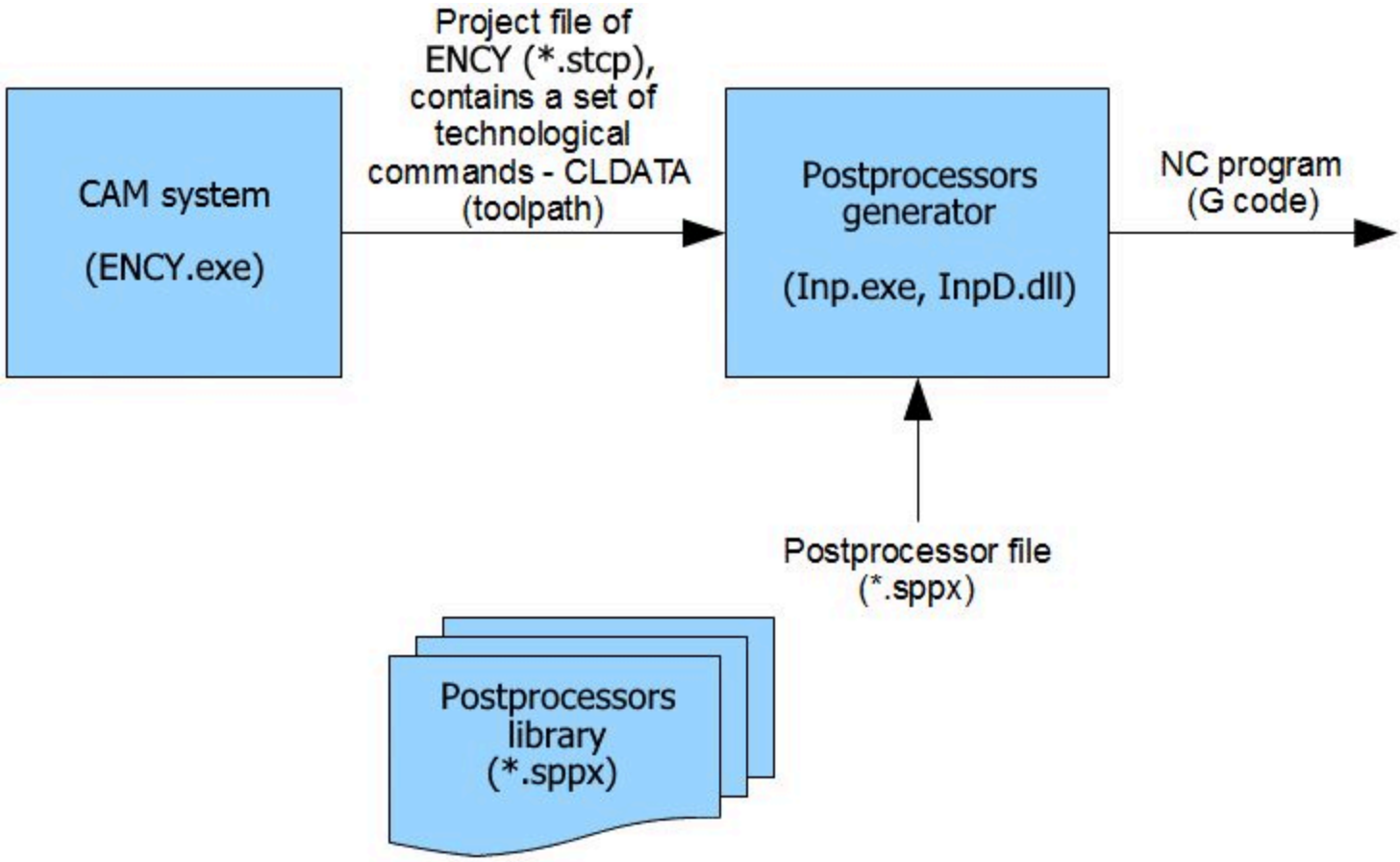
> Note:
>
> To generate several NC programs for different operations, use the operations tree to enable or disable NC code generation for operations, and then run the postprocessor to create the separate NC programs. Please note that 'Generate NC code' property is a property of operation. It will be saved in project and User operations default settings.



If some operations have Generate NC code property disabled when hinting will be visible. The root All operations node checks or unchecks all enabled operations in the technology tree.

The postprocessor system is a separate standard module, and can, if required, be run separately from CAM system.

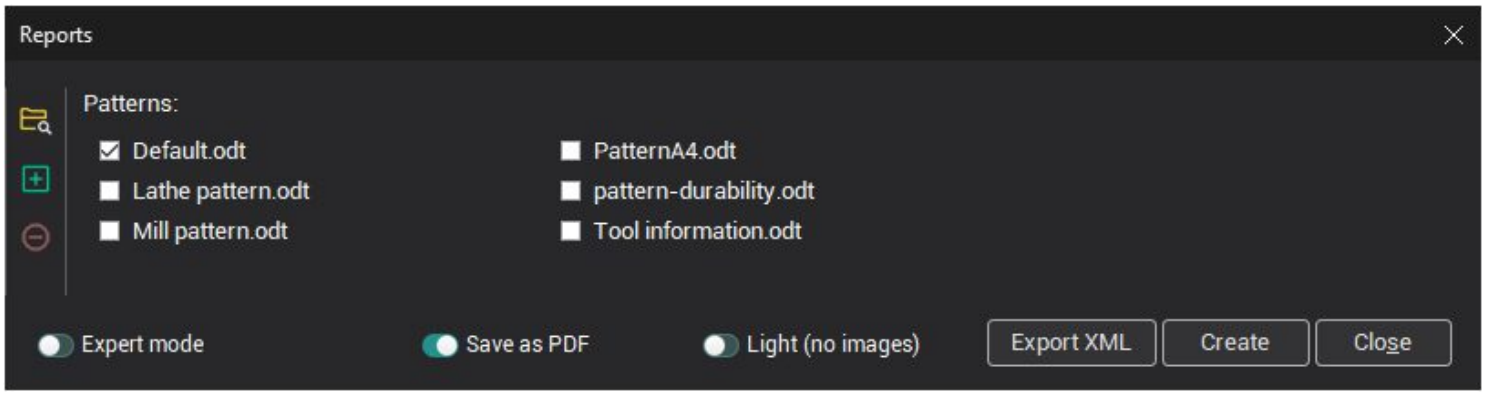
. The name of the executable file is – Inp.exe. Postprocessor tuning files for CNC systems are created and edited using the Postprocessors generator (INP.exe).


CAM system
(ENCY.exe)
Project file of ENCY (*.stcp), contains a set of technological commands - CLDATA (toolpath)
Postprocessors generator
(Inp.exe, InpD.dll)
NC program (G code)
Postprocessor file (*.sppx)
Postprocessors library (*.sppx)

Machining report

System is contain build in the machining report feature. The result machine report can be edited by the any text editor supported.odt format. The result report document can be saved in the ODT or PDF format (PDF need the internet connection). The application to open the result machining report is defined in the setup window.

Press the  button from the Machining bookmark to open the Machining report generation window.

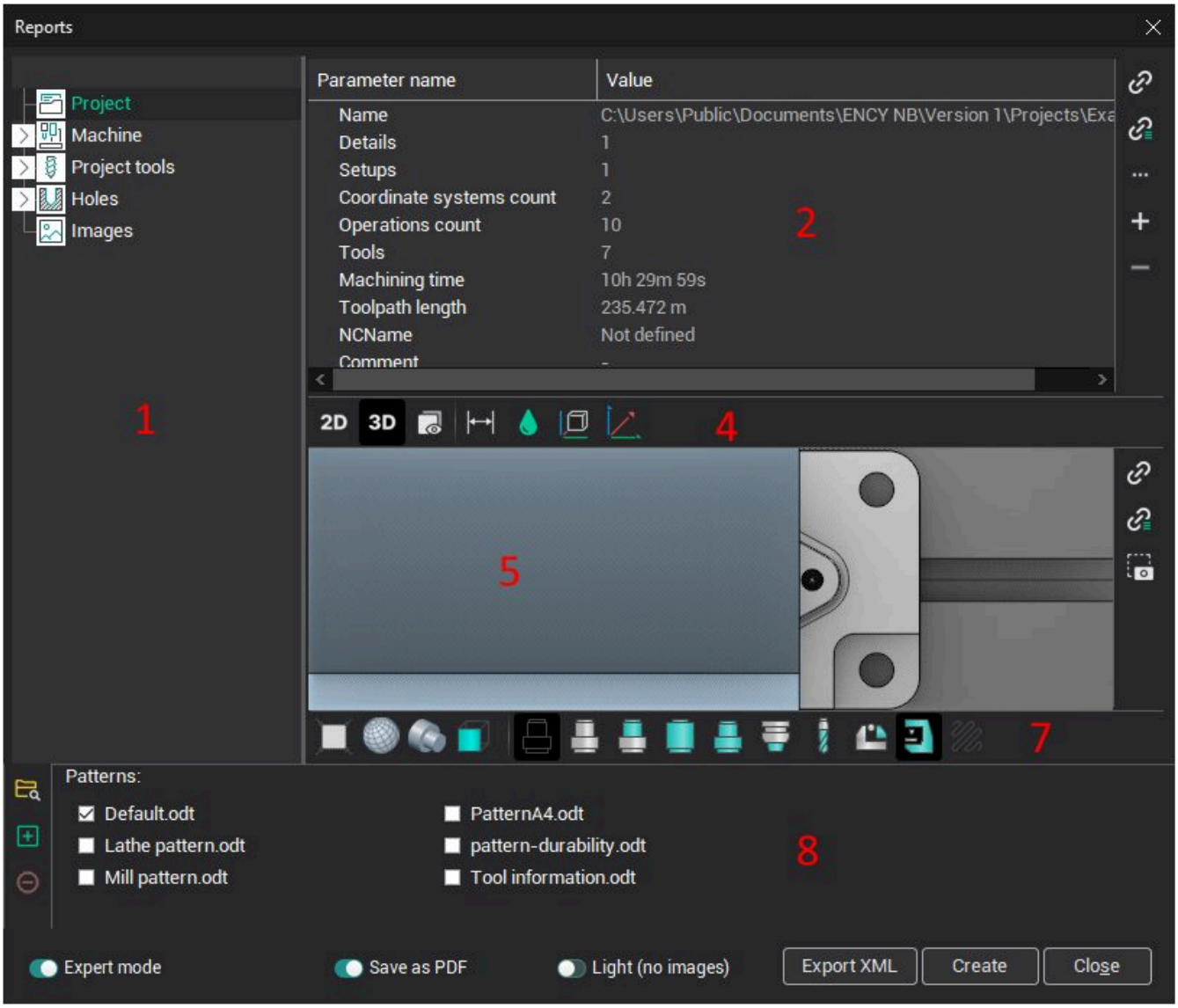


As default the biggest part or the machining report window is occupied by list of patterns from the default folder. It is possible to select several patterns and generate reports by the one click.

At the right part of windows is placed the control buttons for add/remove additional patterns to list and change default pattern folder.

Save as PDF - Change output report format to PDF (Internet connection neccessary);

Expert mode - Expand window to the full mode used for report patterns creation/edit or tune some report params.



The full machining report window contain the next elements:

1 - Tree with the information from the project that can be used for reports;

2 - Selected item parameters list. This panel show parameters for the selected item which can be used for output in the report. (Used for pattern creation);

3 - The control buttons for a report pattern creation/modify;

- - Make command to insert to the pattern simple parameter;
- - Make command to insert to the pattern parameter to output as array;
- ... - menu with additional command;
- + - add custom parameter;
- — - remove custom parameter;

4 - Control buttons for tune the output images for the report.

System automatically chooses the view and creates the dimensions. The dimensions can be shown in the top, bottom, left, right, front and back views only. Three distances to the origin and three overall dimensions are shown.

5 - The main graphic window showing the image for the selected item in the tree;

6 - The control buttons for report pattern creation/modify;

7 - Panel defines the drafts options. The draft can contain the workpiece, source model, tool path, origin, axes. These objects appear if press the corresponding buttons in panel;

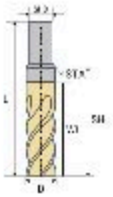
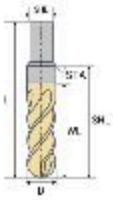
8 - Pattern list.

Select patterns from the list and click Create to create the machining reports. The application to open the result document is defined in the setup window (for ODT format).

Note: *If document will be used later it must be saved manually.*

Example of result document show below:

Tools information report

ID	Name	Type	Length	Draft
1	8mm Cylindrical mill	Cylindrical mill	40	
2	6mm Spherical mill L30mm	Spherical mill	30	
3	6mm Spherical mill L80mm	Spherical mill	80	

Machining report creation/modifing.

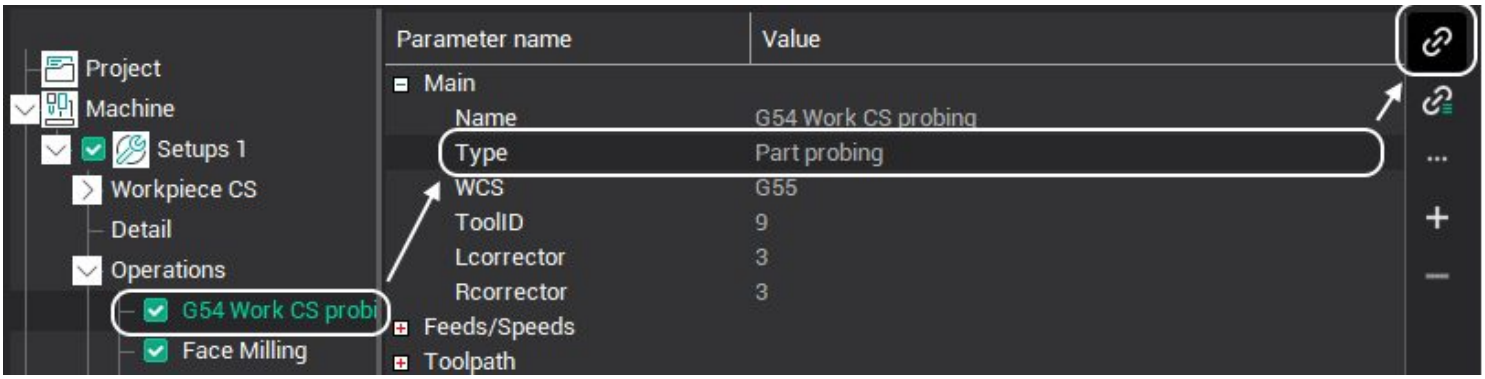
Reports can be created using a variety of templates, offering flexibility in formatting and content.

Formation of the template is performed directly in editor supporting the ODT format.

Setting variables for substitution is done only in the notes object.

To insert a variable, specify the full path to it in the structure of the stcx file. You need to use for it the control buttons (part 3 and part 6 on the full window image).

For example, to display the values of an operation type in a report, create a note in the template (Ctrl + Shift + C) and make next action in the report window:



As a result of the command, the value "Roughing waterline" will be displayed in the report.

Similarly, any other values are output.

The reporting system allows you to add custom variables to reports using this syntax.

VAR (Variable Name, Title, Default Value).

Example:

VAR (Detail, Detail, Case)

When you select a template, a list of available variables is displayed in the RTK window.

A variable can be described in any part of the template. The format of the variable call:

${Variable name}

Example:

$ {Detail}

All commands and variables should be found only in the notes objects.

Separate commands have also been added to work with images.

Using example pictures

In this case, an arbitrary image whose properties and format can be fine-tuned is inserted into the document. When substituting, the image source is replaced, previously configured properties remain

unchanged.

The command for displaying the image is set (using OpenOffice as an example) in the Name field on the Settings tab.

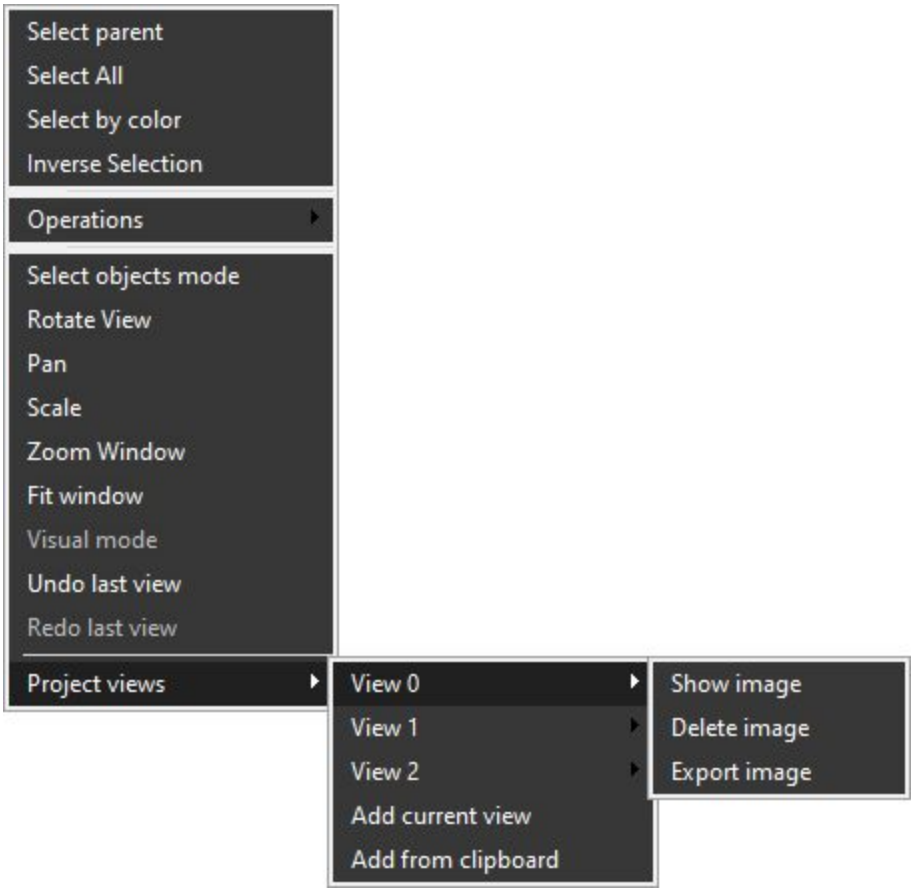
You can view images added to the project by right-clicking in the graphic window and choosing Project Views from the menu.

Add current view - Forms the image according to the current view and adds it to the list of project images

Add from clipboard - pastes the image from the clipboard and adds it to the list of project images.

Already added images can be viewed and deleted.

When deleting images, index shift does not occur.



For a better understanding of the principles of working with this type of template, it's recommended to review the provided examples within the software distribution and explore the included lessons on creating machining reports.

Standard machining sequences

In practice to manufacture similar parts often the same approaches are used. For example, the machining sequence for a typical die can consist of three operations:

- Roughing waterline;
- Complex finishing;
- Complex rest milling.

If a large amount of operations parameters, such as used tools, stocks, steps, etc are the same for several parts then creating a standard machining sequence makes some sense. For that CAM system offers capabilities to import and export a whole machining sequence as well as a distinct operation. The access to these functions is provided via the operations shortcut menu:

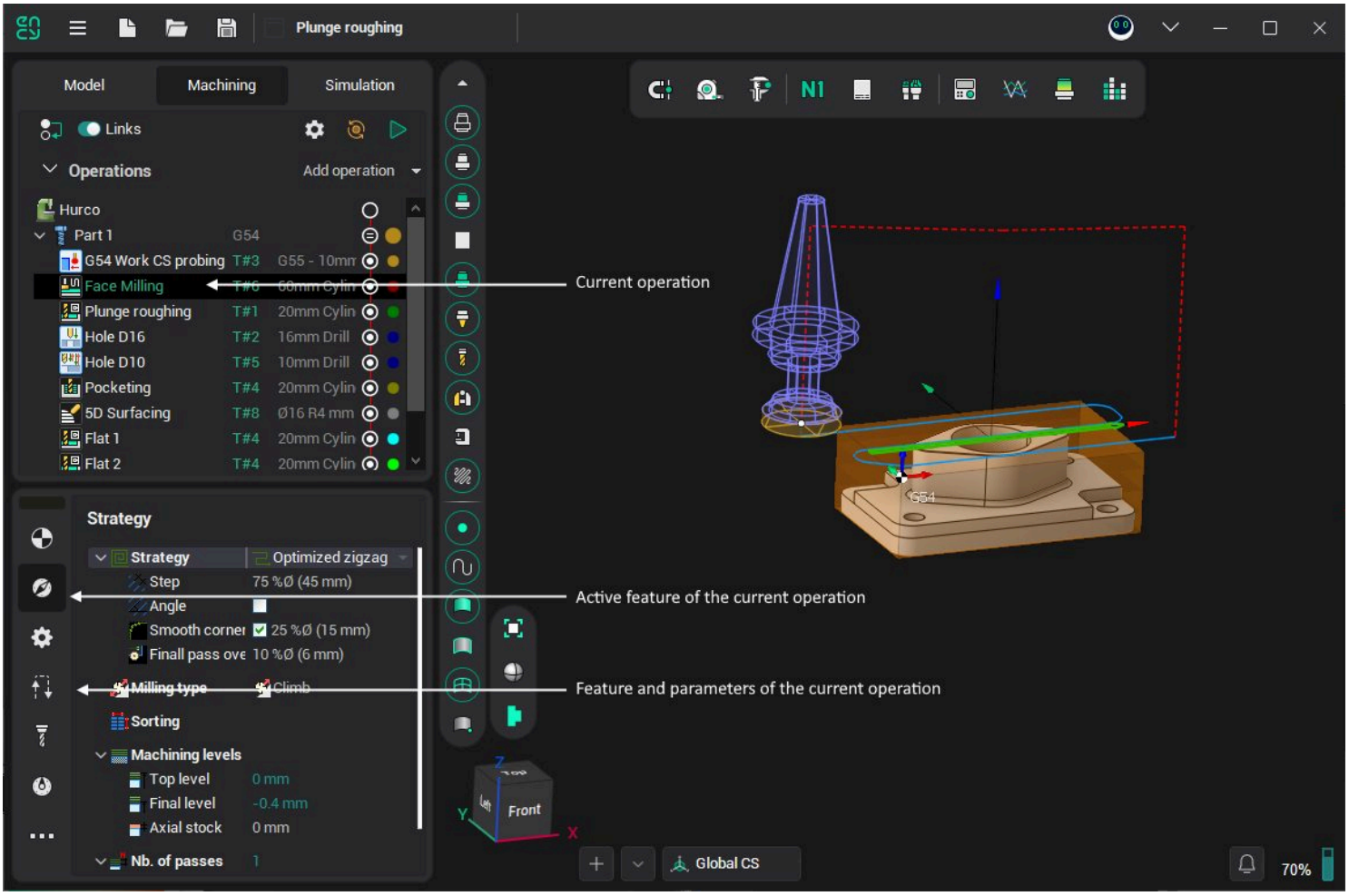
The Export operation command saves all the parameters of the selected operation into the specified file with the *.sto extension.

The Import parameters command fills the parameters of the selected operation with the values previously saved in the *.sto file.

The Import operation command adds to the machining sequence one or more operations either from the CAM system operation file (*.sto) or from the any given CAM system project file (*.stcp).

Operations setup

The parameters of an operation define what is to be machined and the way it is to be machined. Selecting a parameter node inside the operation tree changes the bottom side of the tab to display the tools used to define and edit the parameter properties.



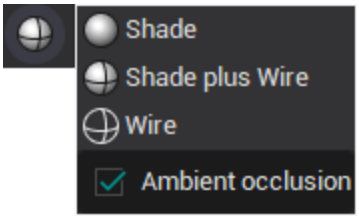
Geometrical parameters of an operation

The geometrical parameters of an operation are displayed in the view. Those include the part, the workpiece, the fixtures and the job assignment. The visual properties of the parameter can be edited using the light icon next to the parameter's node. Left-click on that icon switches the object on/off. However, a selected parameter is always visible irrespective to its state.

In pop-up menu you can specify the object color, the transparence as well as select its display mode.

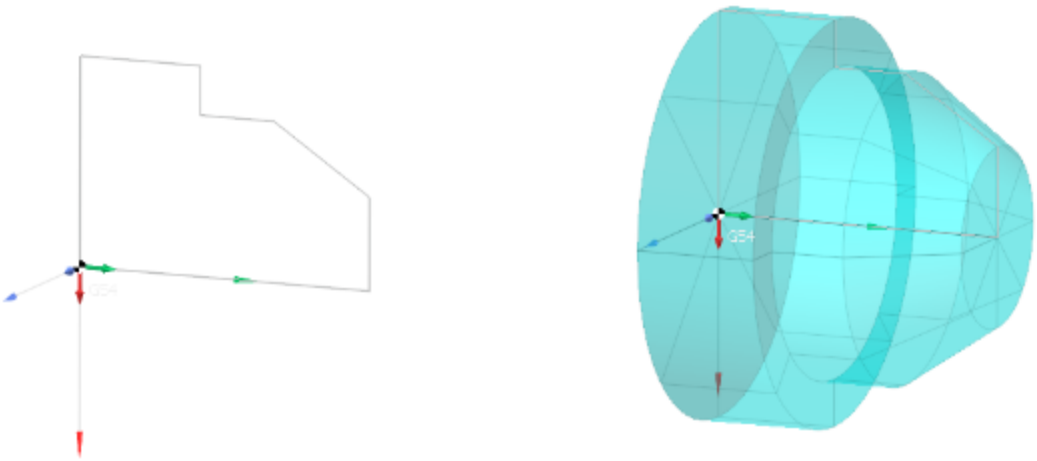


The defaults display mode is used to specify the object should be displayed using the global settings set with the button on the main toolbar.



The rest of the modes force an object to be displayed irrespective to the global settings.

Defining part, workpiece and fixtures



Application area:

These elements allow you to specify a part, workpiece, restrictions and fixture.

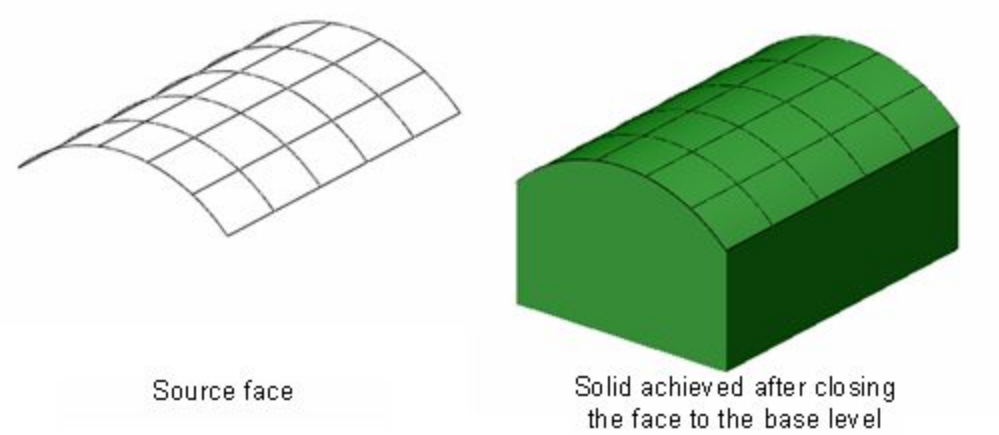
Each item of the list defines a surface, a solid etc to take into account while calculating toolpath. The item caption displays the full path to the item in the geometry model tree. If the source object of an item is removed or renamed the item is marked with the question sign. The icon left to the item defines either the item type or the way the item was achieved from the source object. To add an item of some type to the list the appropriate buttons are used.

Face. The button adds selected faces to the list. If no faces are selected then the dialog with the geometry model tree appears.

▼ Click here to expand..
In this dialog one should find and select the faces, the meshes and the groups to be added to the list. If a group is added then only 3D-objects of the group will be taken into account. When defining a workpiece the Add Faces button is used to make a solid from the selected faces. The solid item represents a bounded space. It is marked with the ✔ icon.

There are two possible ways to make a solid:

- By sewing faces. The added faces need to form one or more solids. The tolerance that the faces are connected with to each other is specified in the dialog that appears.
- By closing faces to the given Z level.



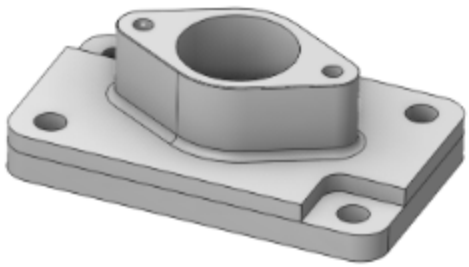
Primitive. Allows you to use different templates to define a workpiece in the form of primitives.

▼ Click here to expand..
Workpiece of thr previous operation. The special link items are introduced. Those are used to provide relations between operations as well as to define simple parametric features. At that, a link item is the reference to the source object, that in turn can be a reference to another object. Changing a source object forces all link items to be reset and/or recalculated.

Machining result of the previous operation. The special link items are introduced. Those are used to provide relations between operations as well as to define simple parametric features. At that, a link item is the reference to the source object, that in turn can be a reference to another object. Changing a source object forces all link items to be reset and/or recalculated.

Restrictions of the previous operation. Allows you to set a primitive from the previous element (operations, setup stages, parts).

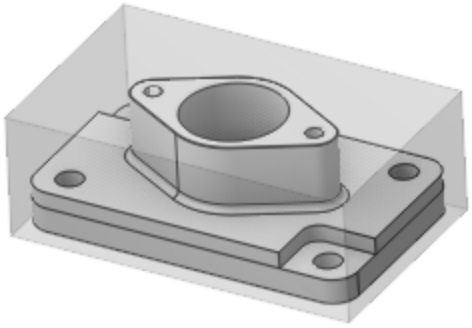
Empty workpiece. Allows you to specify a blank. Often used in painting.



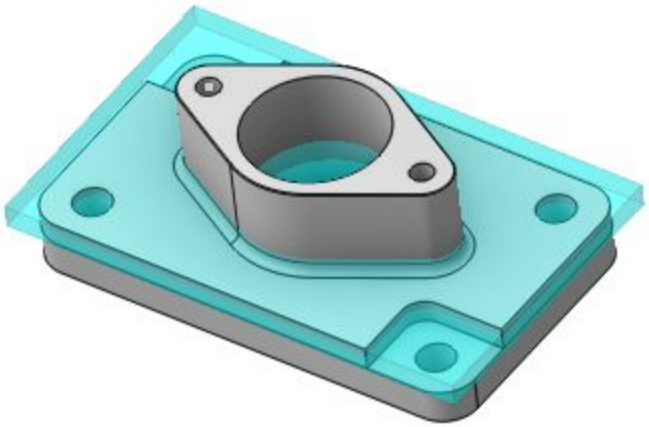
Box. Allows you to set a simple prism around a part.

▼ Click here to expand..
Definition method.

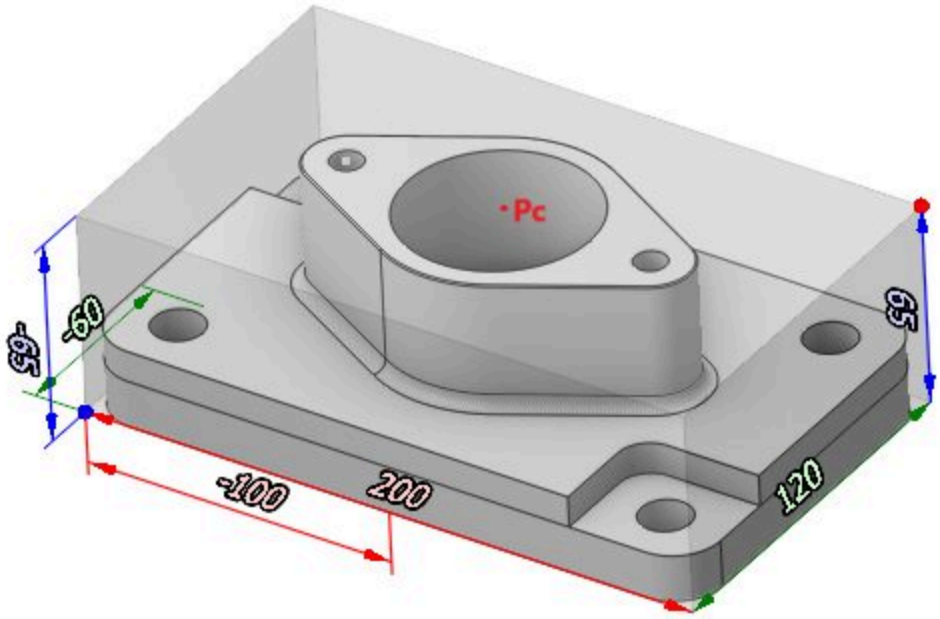
Around part. Allows you to set a box around the part with the required stocks. The allowance is measured from the dimensions of the part.



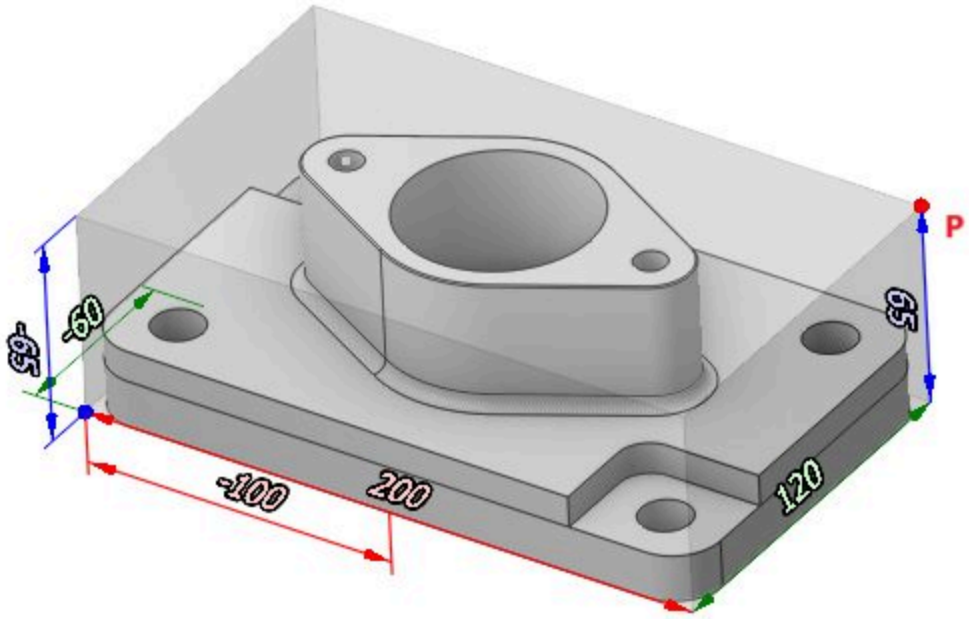
Around geometrical model. The allowance is added from the selected geometry (parts, surfaces).



From center point. Allows you to specify a workpiece by points.The distance of the workpiece is determined from the central point (Pc) to the overall dimensions of the part.

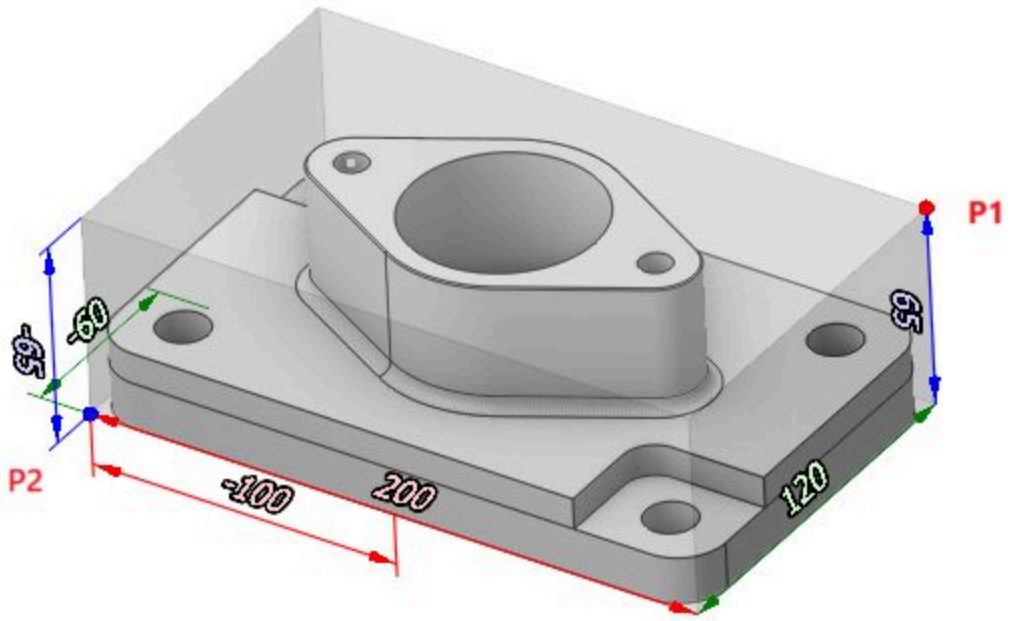


From bottom southwest point. Allows you to specify a workpiece by points. The distance of the workpiece is determined from the bottom point (P) to the overall dimensions of the part.



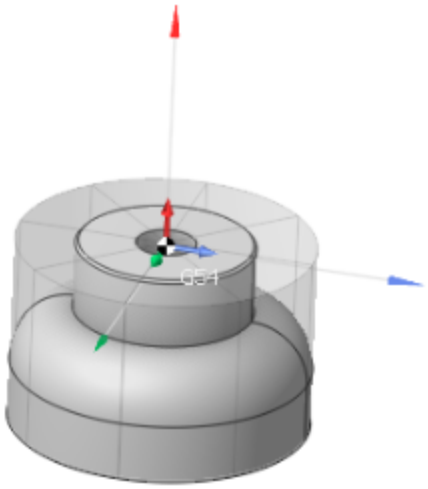
From two points. Allows you to specify a workpiece by points. The distance is measured between point P1 and P2. In this case, the points in the window are specified along three axes (X,Y,Z) relative to the global

coordinate system.

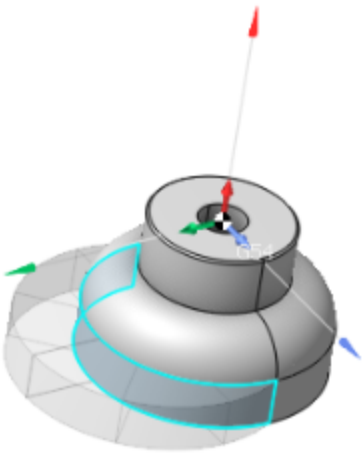


Cylinder. Allows you to define a cylinder around a part as a workpiece.

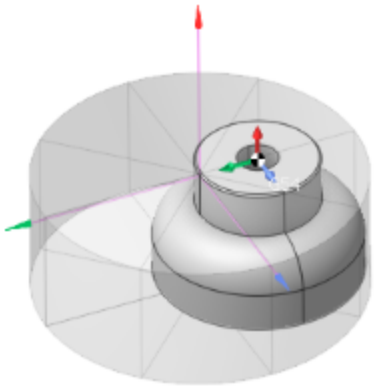
▼ Click here to expand..
Around part. Builds a cylinder based on the dimensions of the part around the global axis of the coordinate system (if selected Center of Box).



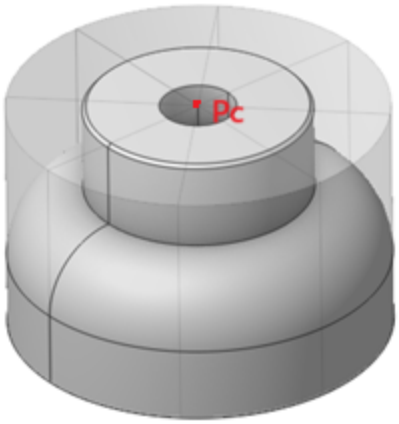
Around geometrical model. Builds a cylinder based on the selected element (face,part) around the global axis of the coordinate system (if selected Center of Box).



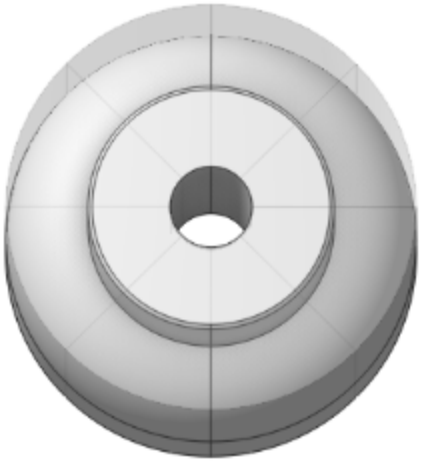
if selected Center of CS then the workpiece is built in the same way, only around the selected SC.



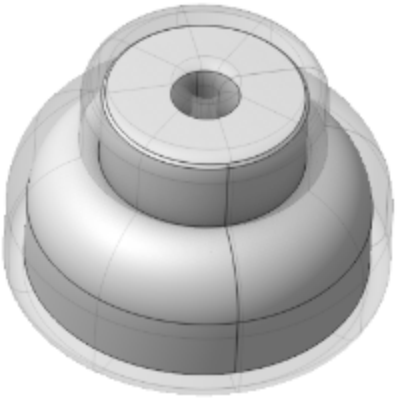
From center point. Allows you to specify a workpiece by points.The distance of the workpiece is determined from the central point (Pc) to the overall dimensions of the part.



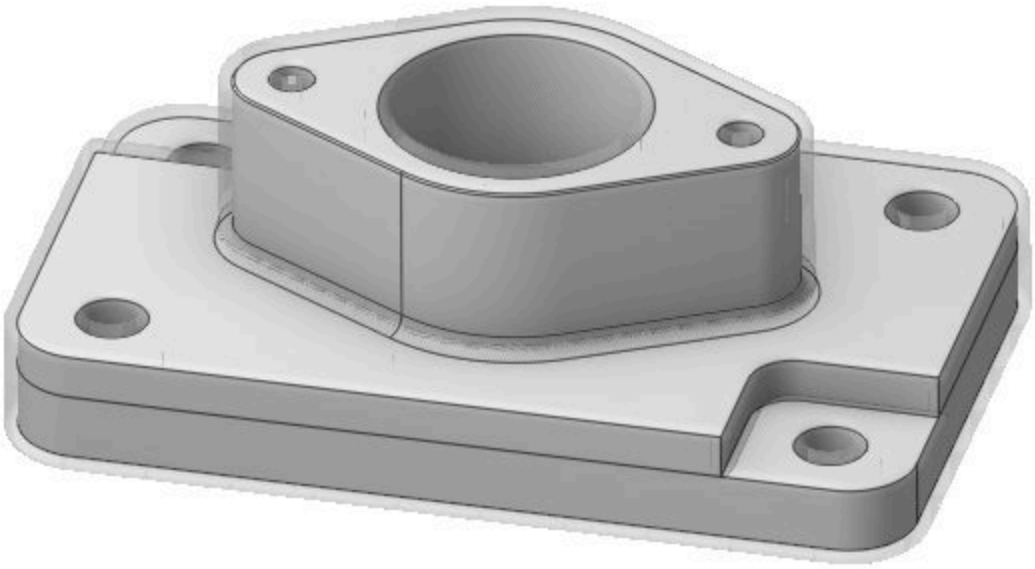
Tube. Allows you to define a cylinder around a part with a hole in the center as a workpiece. Works similar to a cylinder.



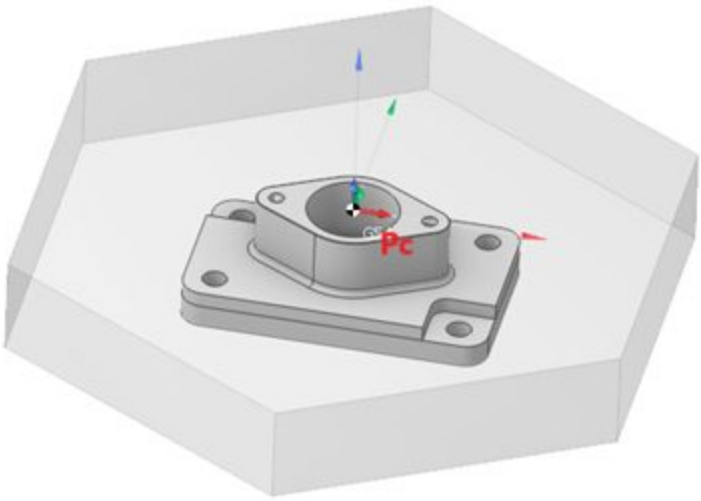
Turn Envelope. Allows you to rotate the workpiece around the part.



Castling. Allows you to set the casting with an equal offset of the workpiece from the surface of the part

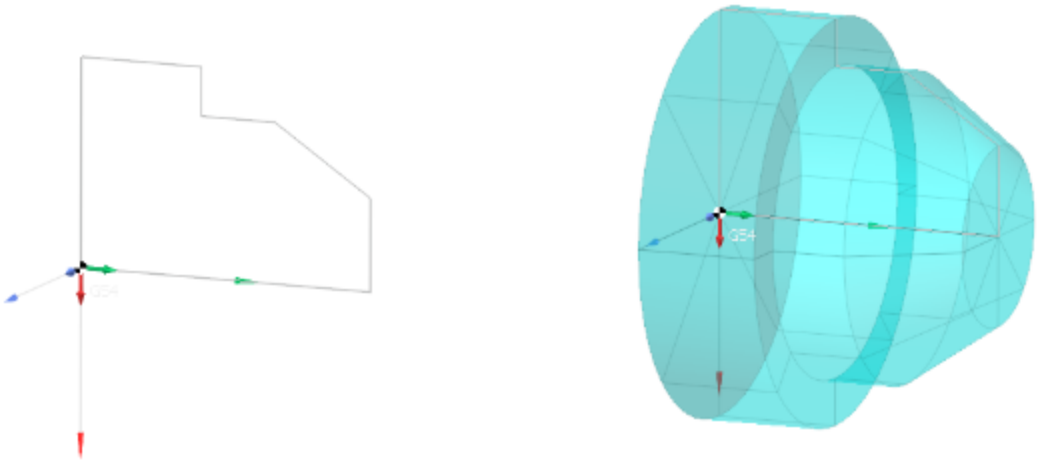


Polygonal prism. Allows you to set the workpiece as a polygon from the center of the workpiece (Pc).



Turn. The turn button makes a revolution solid generated from the specified curves. If no curve is selected the dialog that helps find and select the generating curves for the solid appears.

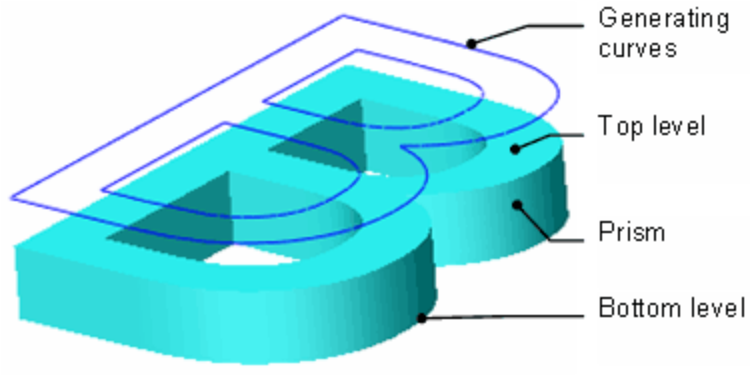
▼ Click here to expand..



Extrude. The extrude is used to make a prism.

▼ Click here to expand..
Closed curves as well as group of curves forming closed profiles can be used as the source objects. The specified curves are used as the base of the prism. In addition, the top and the bottom levels of the prism should be specified.

If one wants define a prism with holes and isles he needs place the source curves in one folder and specify that folder as the prism base. As a rule the outer curve defines the body and the inner curves define the holes. It is possible to set the outer curve as a hole for the fixtures. To do it is necessary to check the Throw hole box.



User operations

Every technological operation of CAM system have the rich parameters' kit, which allow you to build flexible machining strategy for the complex technological processes for the exact parts. After creation the operation already has some predefined parameters values – it is so called "system default parameters". So the operation is ready for toolpath calculation.

Developing of technology process generally includes the following steps:

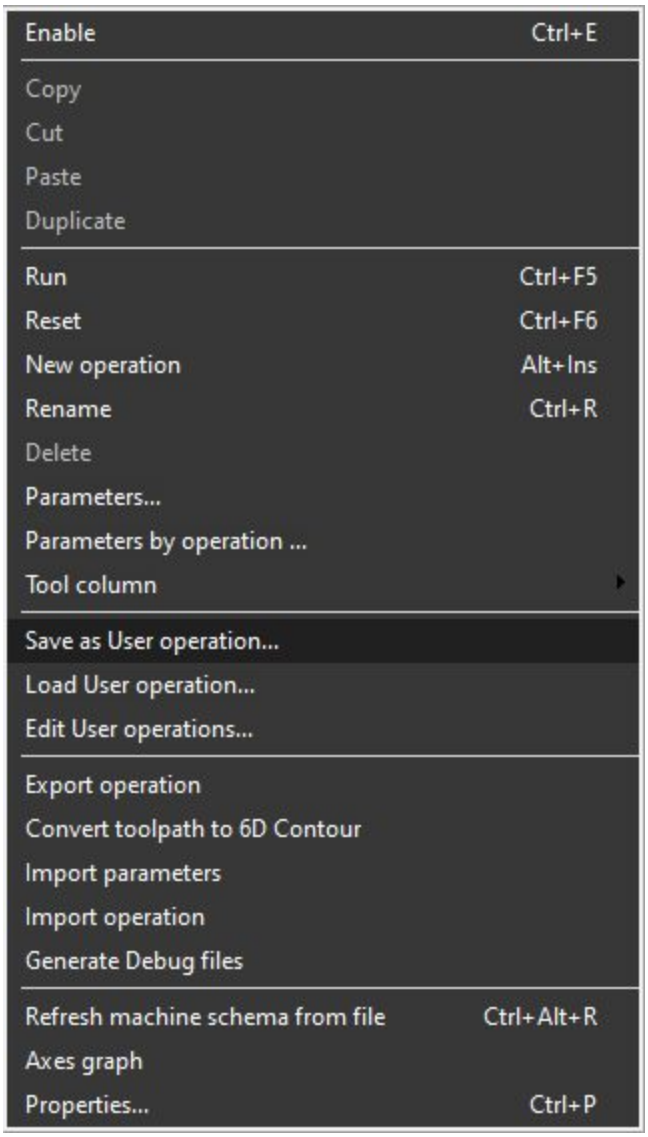
- Determining machining strategy for a specific element of a part.
- Determining operation which is able to implement the proper strategy.
- Creating operation and specifying the machined part's element as it's job assignment.
- Calculating toolpath.
- Analyzing toolpath.
- Modifying technological parameters of the operation to achieve better results.
- The latter three steps should be repeated until the desired result will reached.

To provide universal and versatile approach CAM system operations have a lot of parameters. Therefore the process of the operation customizing may take a long time and significant effort. CAM system provide the User operations feature to reduce the required time to design operations. It allows to specialize the universal CAM system's operations

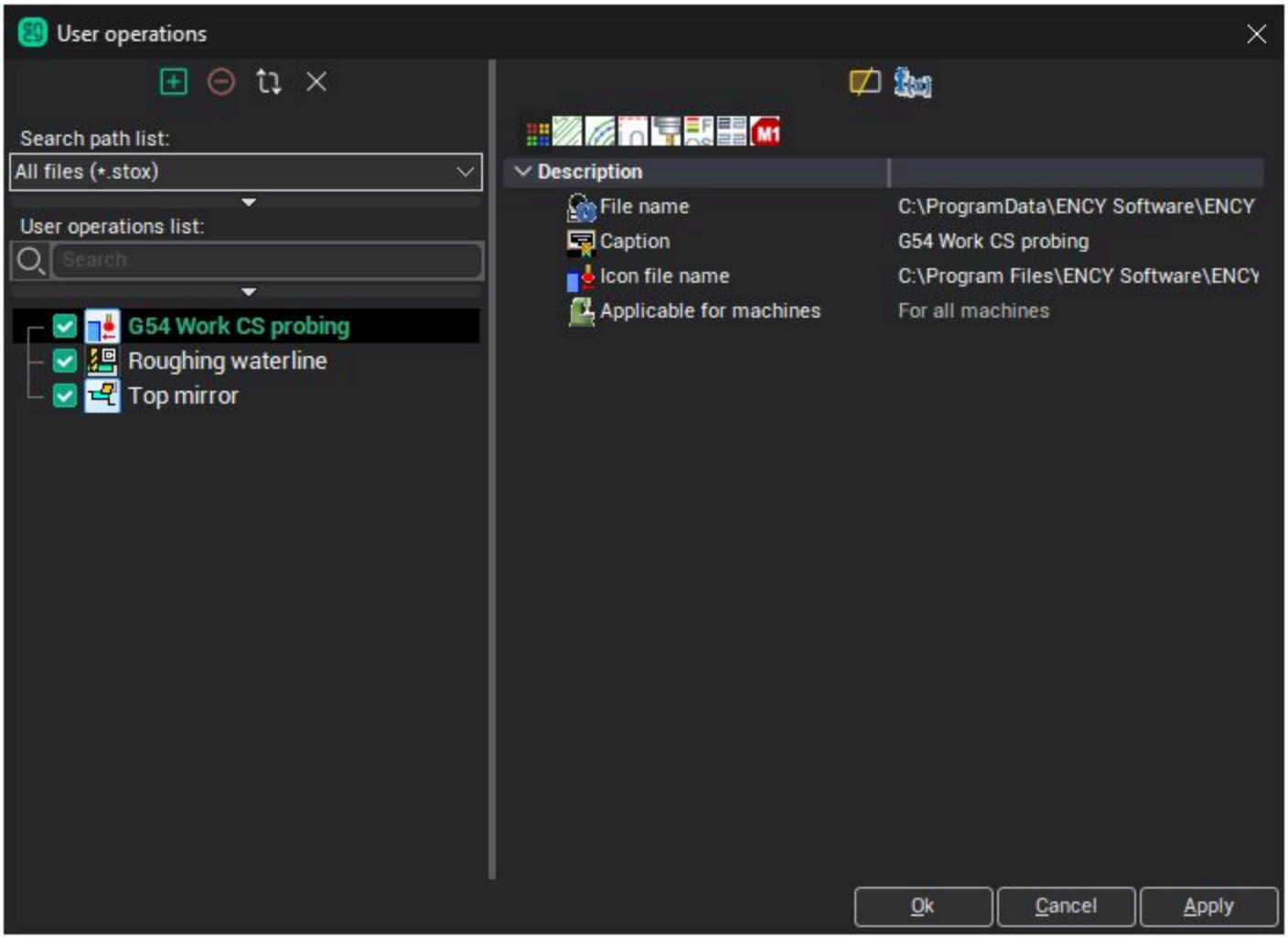
After setting up the operation for specific purposes it is possible to save it's parameters set under the certain name to the external *.stox file. When processing the next similar element possible to download this set of parameters and, thereby, greatly reduce the time for setting up the operation. As a user operation can be saved not only the individual operations but also several operations. For this purpose these operations are placed in the group and this group is saved as the user operation.

User operations can easily be imported to another computer by simply transferring *.stox files of the corresponding operations.

To save and load default parameters use the pop-up-menu of operations in the List of technological operations on the Technology tab.



After choosing the `Save as User operation...` item the current operation's parameters will be saved in special *.stox file independently from the current project and, so they can be used in other projects. Before saving the User operations window will open in which you can set some properties.



In the left part of the window there are the Search path list to search for files of user operations, the list of available user operations and the filter panel. Filter allows you to quickly find the operation by the location of the file, by the operation type, by the machine or simply by name.

To control the lists should use the buttons above them at the top left of the window.

- Opens the folder selection dialog and allows you to add the selected folder to the list of paths intended to the search for the user operations' files (*.stox).
- Removes the selected folder from the search path list. Physically the folder is not deleted.
- Allows you to force restart the searching the user operation *.stox files in folders that are specified in the list of search paths.
- Physically deletes the selected user operation file from the hard drive and exclude it from the list. The check box next to the name of the operation in the list allows you to turn off the visibility and accessibility of user operations without deleting the file.

From the right side of the window there is the properties inspector that displays parameters of the selected user operation grouped by tabs. The first tab has the following basic parameters.

- File name - the full name of the *.stox file. Depending on where you save the file, the operation can be accessed by any user of this computer, or only the current user. By default, files are saved in a

special folder for user operations available for all users. However, you can choose a completely arbitrary name and location to save the file, including the network location.
- Caption - name of the user operation, which will be displayed in a list and assigned by default when adding operation to technological process.
- Icon file name - image file name of any of the common image formats that will appear next to the name of operation.
- Applicable for machines - option allows you to limit the scope of operation and allows to make it inaccessible to add to the technological process, depending on the currently active machine. This option can be useful when working with multiple machines on single computer. When selecting For all machines item the operation will be available regardless of the selected machine. If the For selected machines item specified then a list of machines becomes available in which you can mark the machines for which the operation should be available.

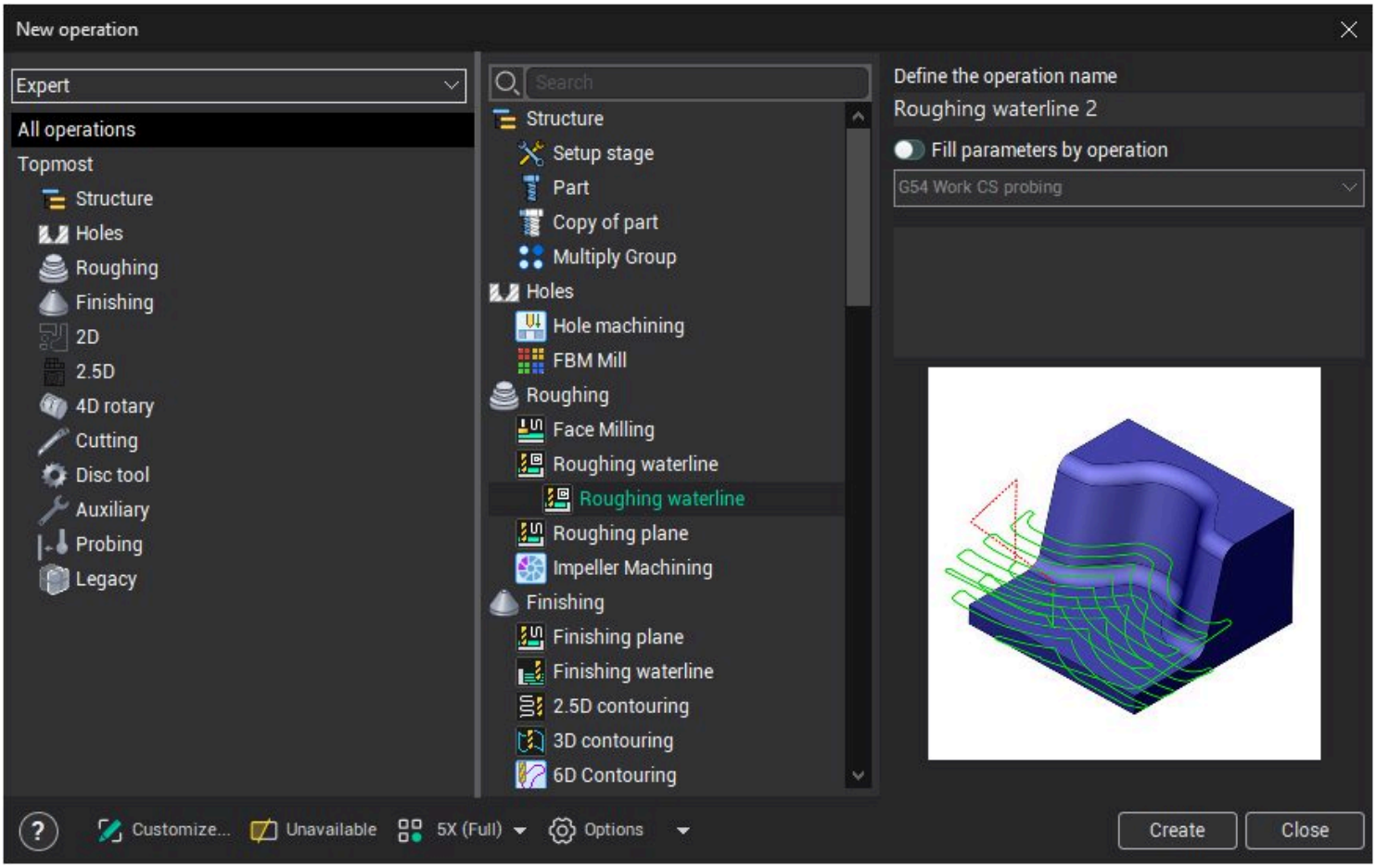
The parameters are located on the other tabs may vary and depend on the particular type of operation. All of them are also available to allow you to easily edit values without changing the parameters of the source CAM system's operation from which this user operation was created.

Above the list of parameters are located a few buttons to control the display of these parameters.

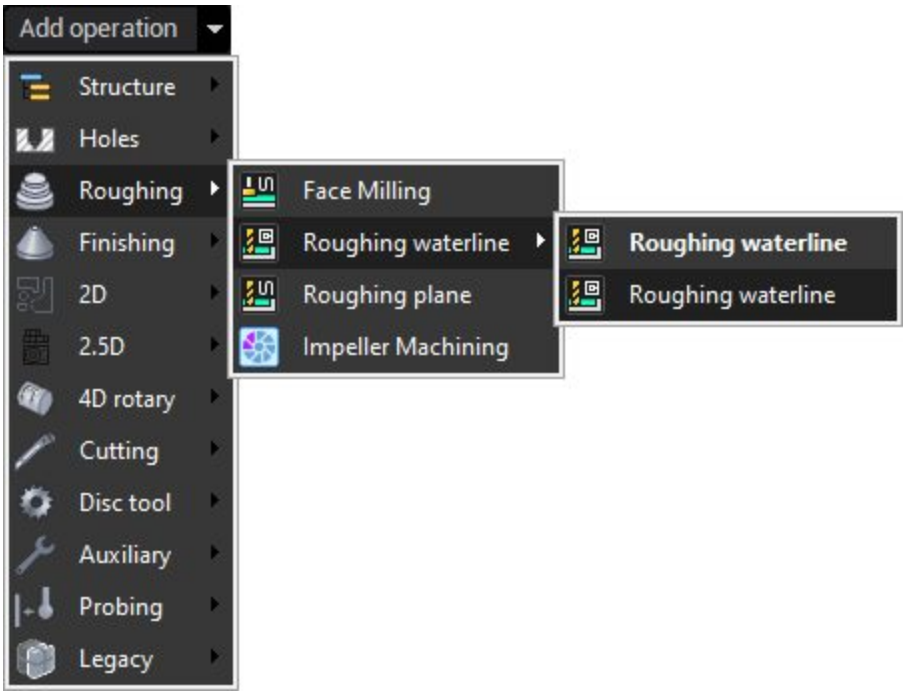
-  allows you to hide / display the properties whose values coincide with the system default values for this type of operation. This makes it easier to distinguish between the important parameters that have been changed with respect to the source. In inspector values coinciding with the default values are displayed with gray color. In order to return the edited value of any field in the default state, you need to select the contents of the field and press the Delete key on your keyboard.
-  allows you to view and edit the formulas for the calculated parameters. Values for the parameters of operations can be defined not only with constants numbers and strings, but also with formulaic expressions. This allows you to make them dependent on other parameters and more flexibility to manage their behavior. Calculated fields are displayed in the inspector with green color. Expressions can use mathematical operations "+, -, *, /, ^, (,), ,", trigonometric functions, as well as links to other parameters. To add a link to another parameter into an expression you need to specify the parameter name in square brackets like this: [Length].

There are several ways to add an instance of a previously created user operation into technological process of the exact project.

In the New operation window, select a previously created user operation.



The creation of user operation also available from the new operation pop-up next to the New button at the Technology tab.

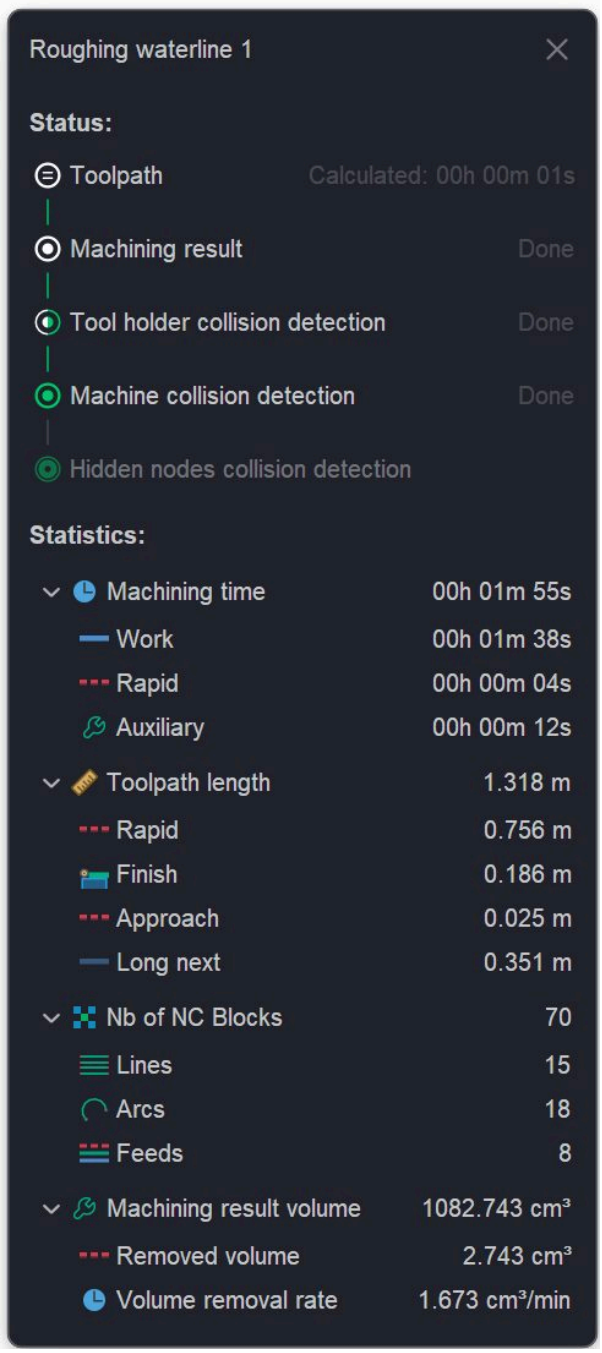


Parameters from the user operation may be applied to the operation not only during its creation but also to the existing operation. Select the `Load from User operation...` menu item in the operation popup

for this. The list of user operations will appear that is available for this type of cycle. When you select an item parameters of the selected user operation will be loaded into the current operation of CAM system.



Operation status panel



Application area:

When clicking on the operation status icon inside technology tree next to the operation name then the status panel appears.

The Status panel is a mandatory element of the interface, designed to visualize the current status of technological operations execution.

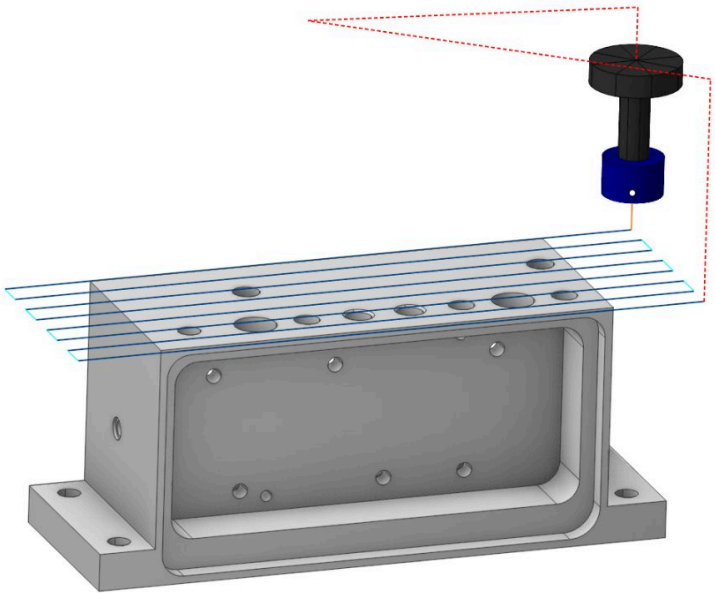
It performs the following functions:

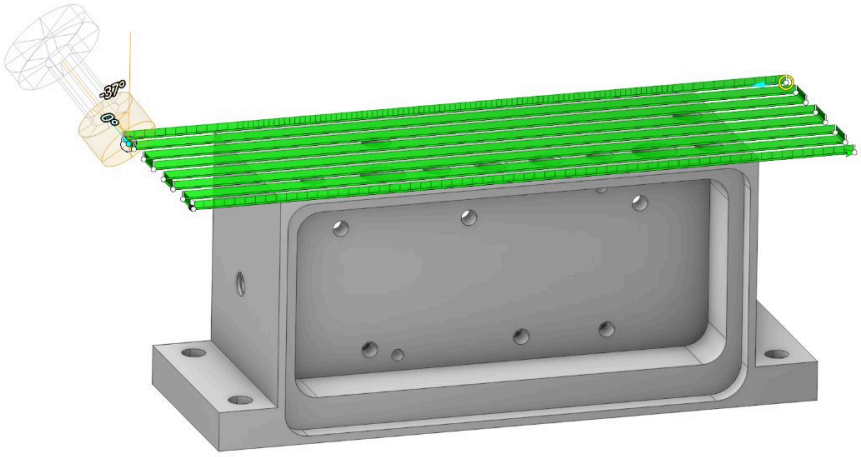
- displays the processing stages (toolpath calculation, machining result, collision detection);

- provides statistical data on tool movement time at different feed rates;
- allows viewing key metrics of the control program (Number of NC blocks for each node type).

You can refer to the following section for a more detailed description of operation statuses. Details

Conversion of arbitrary toolpath to 6D Contour operation



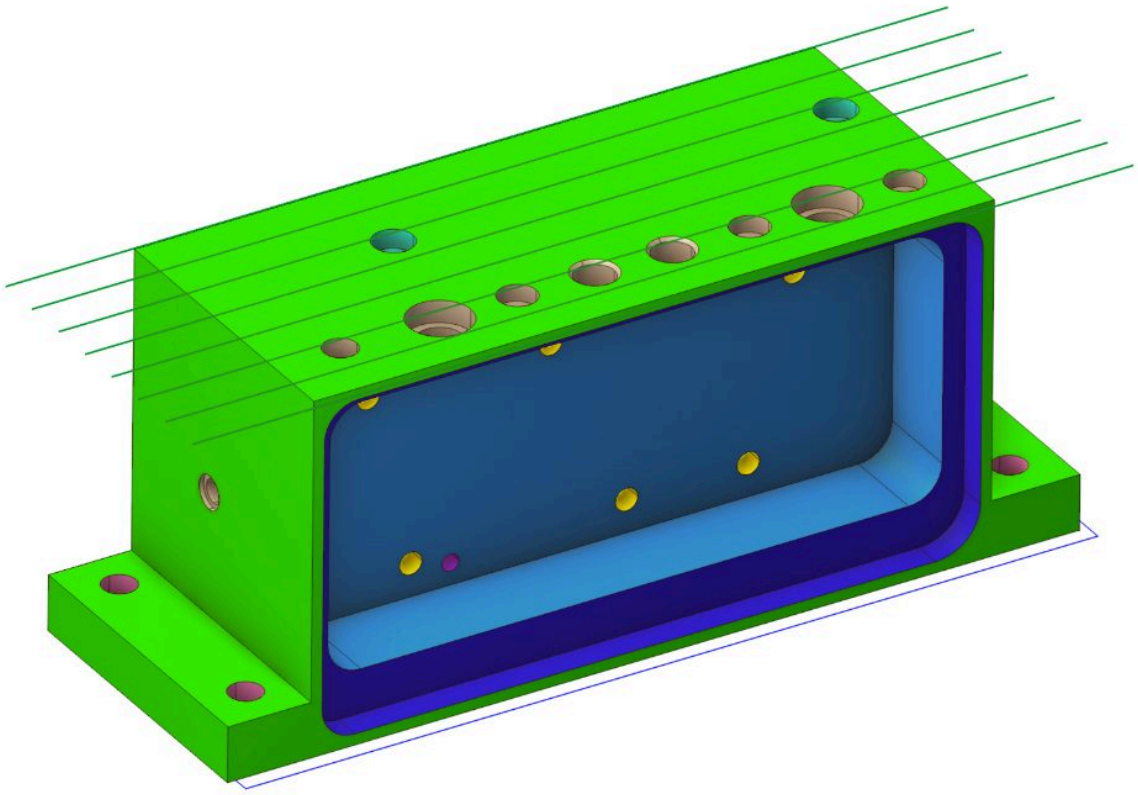


Application area:

it's possible to take (almost) any operation, and convert its toolpath to "6D Contour" operation, which has this toolpath as job assignment. This allows you to make slight modifications to the original toolpath, such as altering tool normals, before post-processing.

To activate, you need to right-click on a specific operation, select in the dialog box that appears Convert toolpath to 6D Contour operation.

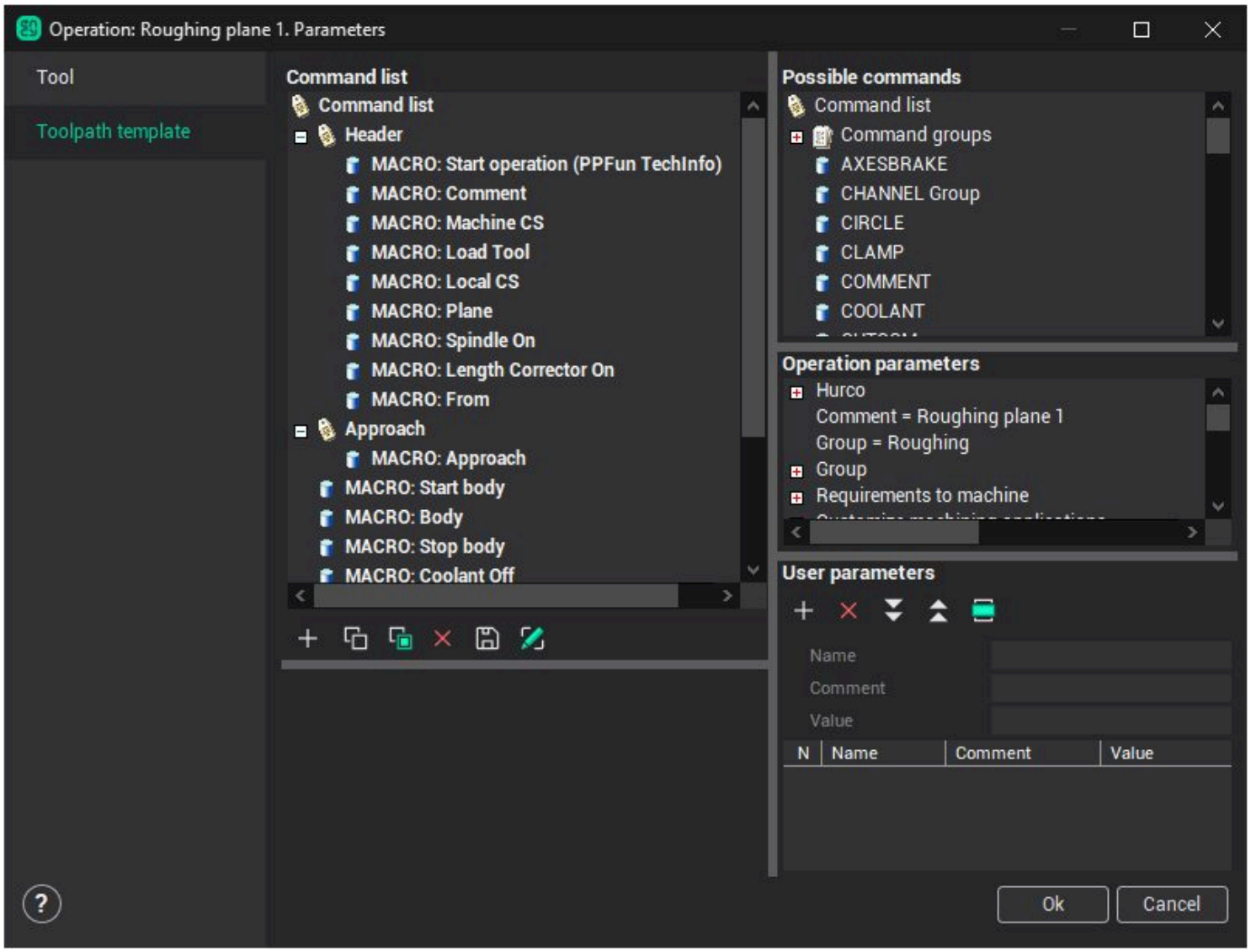
In this case, a group of splines is created on the Model tab.



Tool path template

The toolpath template defines the control data output format. Later this template can be saved and used many times.

Editing of auxiliary operation properties are made in window, which is opening by pressing Parameters button in technological window.



In parameter's editing window five primary objects exists:

- command list;
- list of possible commands;
- property inspector;
- operation parameters;
- user parameters.

Command list includes list of technological commands CLData (command of turning on the spindle shaft, cooldown, feed transmission, movement by circle, directly and so on), which will be contained in

operation after it's calculation. The list may be formed by user from the beginning or may contain a list of defaults commands, which was made by developer of that operation.

List of Possible commands includes all CLData commands which exists in CAM system and macro command (MACRO), and which can be used by user for forming a template. The macro command is a procedure, that do output of standard sequence of operations to the control data.

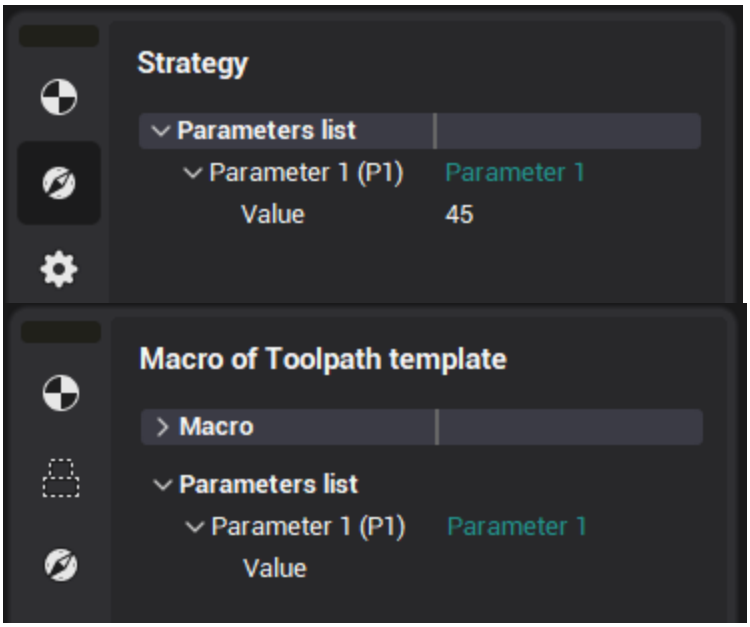
For adding one of commands needs to select a row with a corresponding name and drag it into wishful place in command list. The root element of the list of available commands MCDTree also may be inserted into command list, and inside of MCDTree unit other CLData command may be placed. Thus, commands in the list may be grouped by any other concept. For editing of the list the tools panel may be used.

Any CLData command has special set of parameters (feed magnitude, coordinate position, comment string and so on). Destination of property inspector is on representation and editing pointed parameter, it display choosing command in list. To properties of technological operation may be assigned either simple values (number, string, elements from the given kit), or expressions using common arithmetic operations and mathematical functions (`+`, `-`, `*`, `/`, sin, cos). As an argument of operators and functions may be not only numerical or symbol constants, but also named parameters from the Parameter list, created and edited in the same window. For example, a node DELAY have property Pause value, that can be defined in inspector by this way:

[2*Parameters List(Pause)]

The expression is framed by square brackets specially to show the system that the expression contains references to the list of parameters and that before the computation it is necessary to replace parameters by their concrete values. In indicated example in the parameters' list the parameter Pause must exist. The string ParametersList is reserved key word for access to the parameter's list.

Each parameter from the list has two fields: Name – unique descriptor of parameter in the list and Value – in general, expression that contains numeric or string constants, and also references to other parameters from the list. The editing of parameter's list is carried out with the help of the buttons on the tool's panel. To create the reference of property of technological command to the parameter from the list in addition to manual editing of the text field it is possible to use drag transfer of corresponding parameter from the list to the required block in the list of commands. The values from parameter's list are available for editing not only from parameter's window but also in global inspector of operation's properties and in the window technical process.

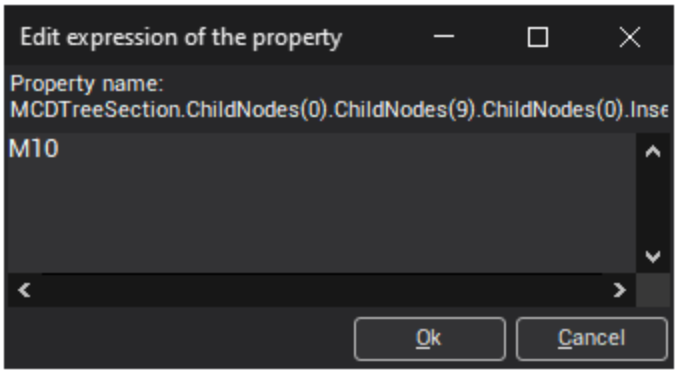


So, toolpath template represents parametrized list of CLData technological commands.

The buttons of control board of parameter's window have the following meaning.

The buttons of the left board:

- Add new command. It adds the command, indicated from the list of Possible commands to the place which is indicated in the Command list.
- Copy command to clipboard. Copies command to clipboard.
- Paste command from clipboard. Pastes command from clipboard. The command will be inserted immediately after the end of selected group or after selected command.
- Delete command. It deletes the detailed command from the Command list.
- Save group. It saves selected group of commands.
- Edit expression. Opens next window:

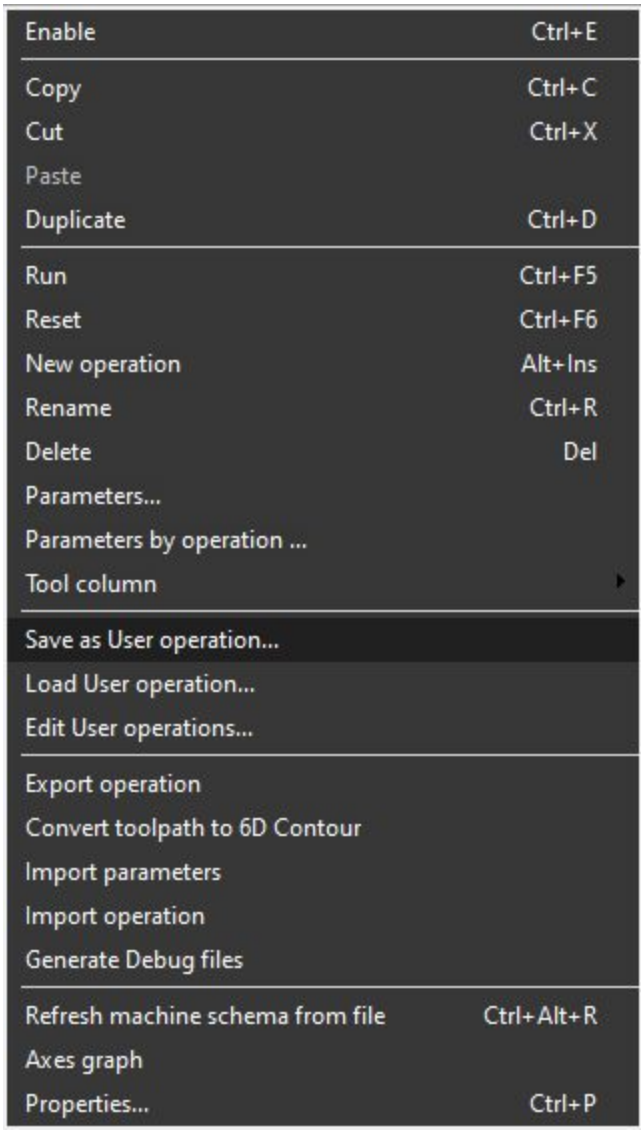


The button of the right board:

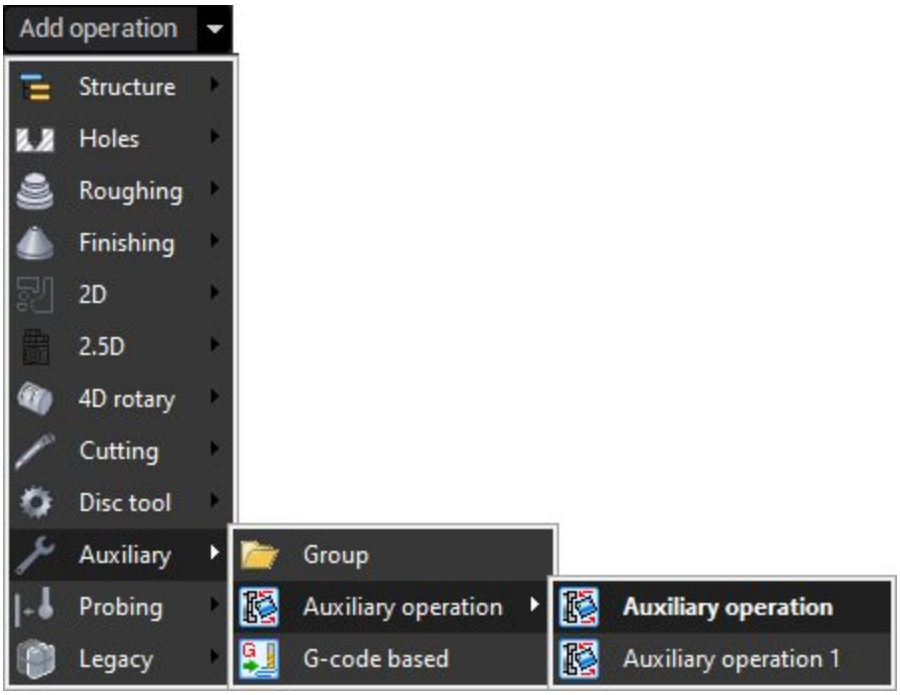
- Add new parameter. It adds new parameter to the detailed place in the Parameter list.
- Delete parameter. It delete parameter from the Parameter list.

-  Move parameter down. It lows down the parameter in the Parameter list.
-  Move parameter up. It raises up the parameter in Parameter list.
-  Insert parameter. It substitutes as a value of a property in the Command list, the parameter, what is selected in Parameter list.

Once created the template may be saved for the next use during the creation of other CAM system projects. For that it should be used mechanism of operations' named default parameters. The functions of saving and loading of the default parameters executing from the context menu which opens at the click of mouse right button on the name of concrete operation in the list of technological operations in regime Technology.

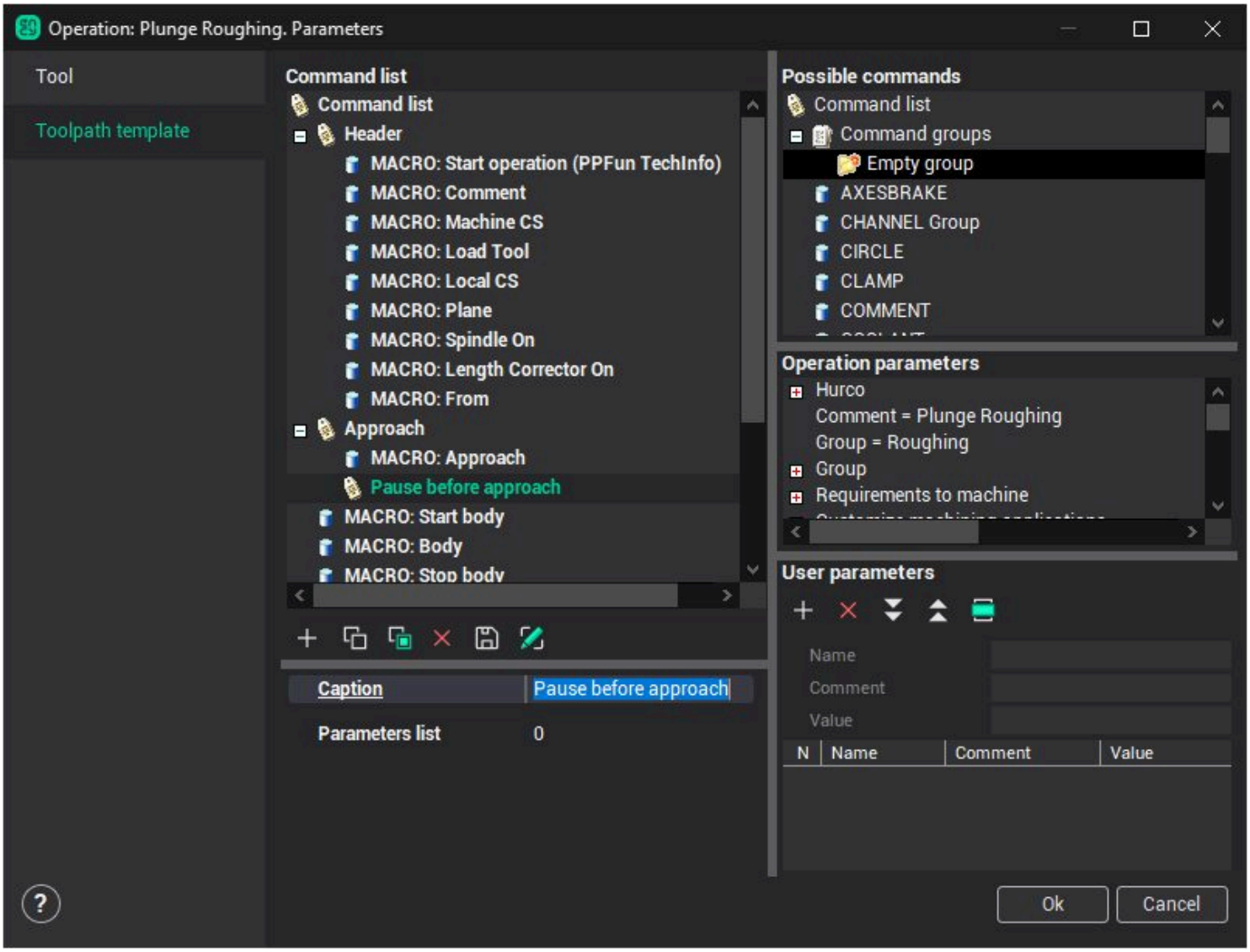


On choosing the `Save as User operation...` the current operation parameter's kit organize into the separate object which is kept safe in a special file independently from the current project and, so, may be used in other projects. After saving the default parameters become available during the creation of new operations (in the window of creation the new operation and drop down menu of new operation creation).

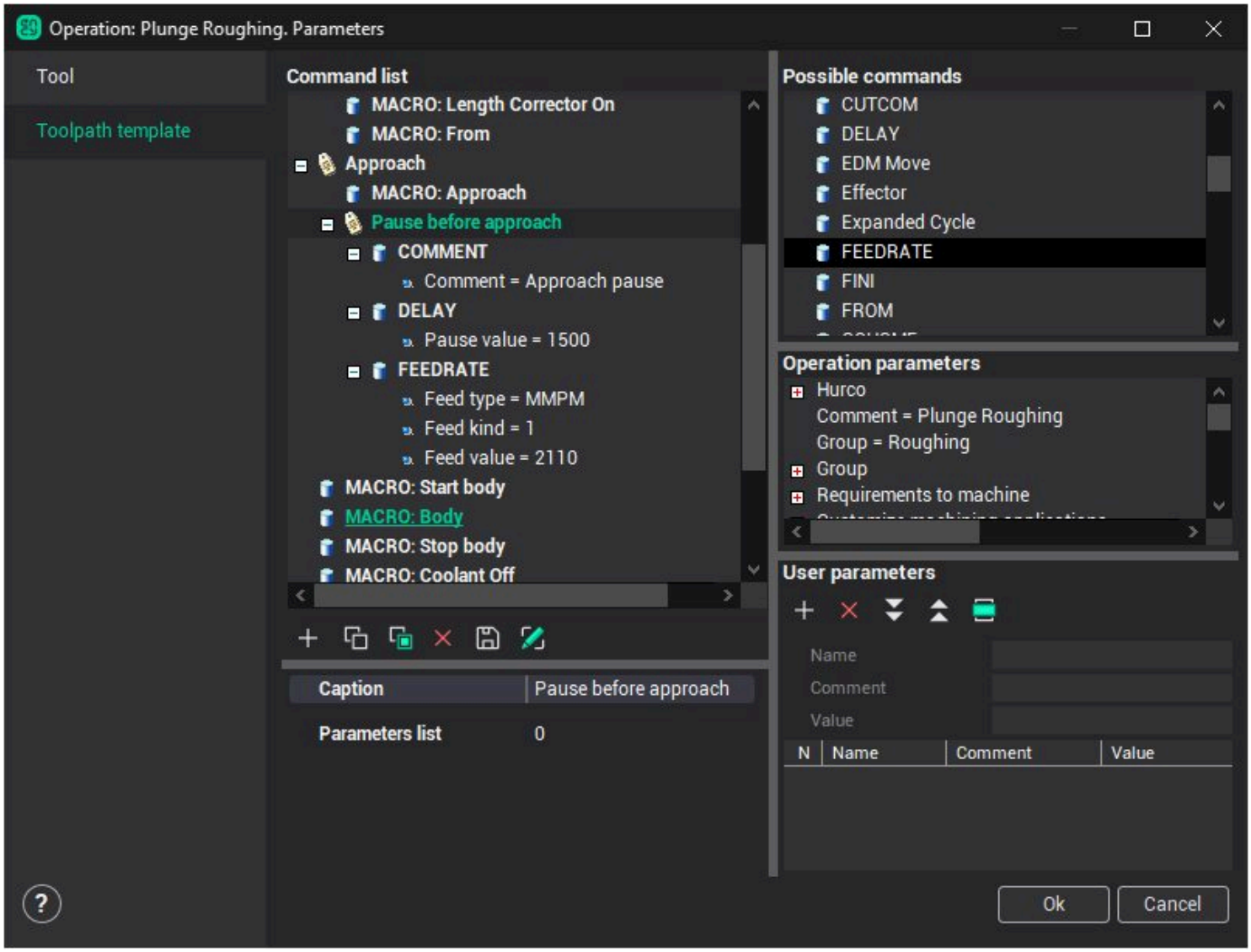


The default parameters may be applied to the operation not only during its creation but also to already existing operation. For that in the context menu of operation should be chosen `Load User operation...`. On the screen it will appear available for this type of operation list of defaults.

It is possible to add new group to the Commands list by choosing Empty group item in Possible commands list. You can set the group name an parameters count.

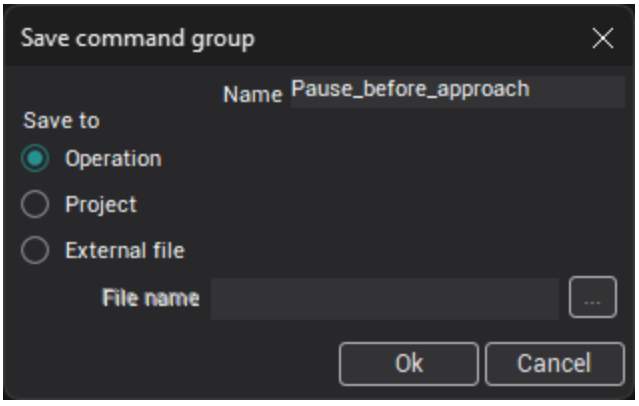


It is possible to add commands from Possible commands list to the group. So, you can form the set of commands for a specific task.



It is possible to save formed set of commands .

Saving parameters window will be shown:



Set the name for group. Specify the name space parameter to provide a unique name.

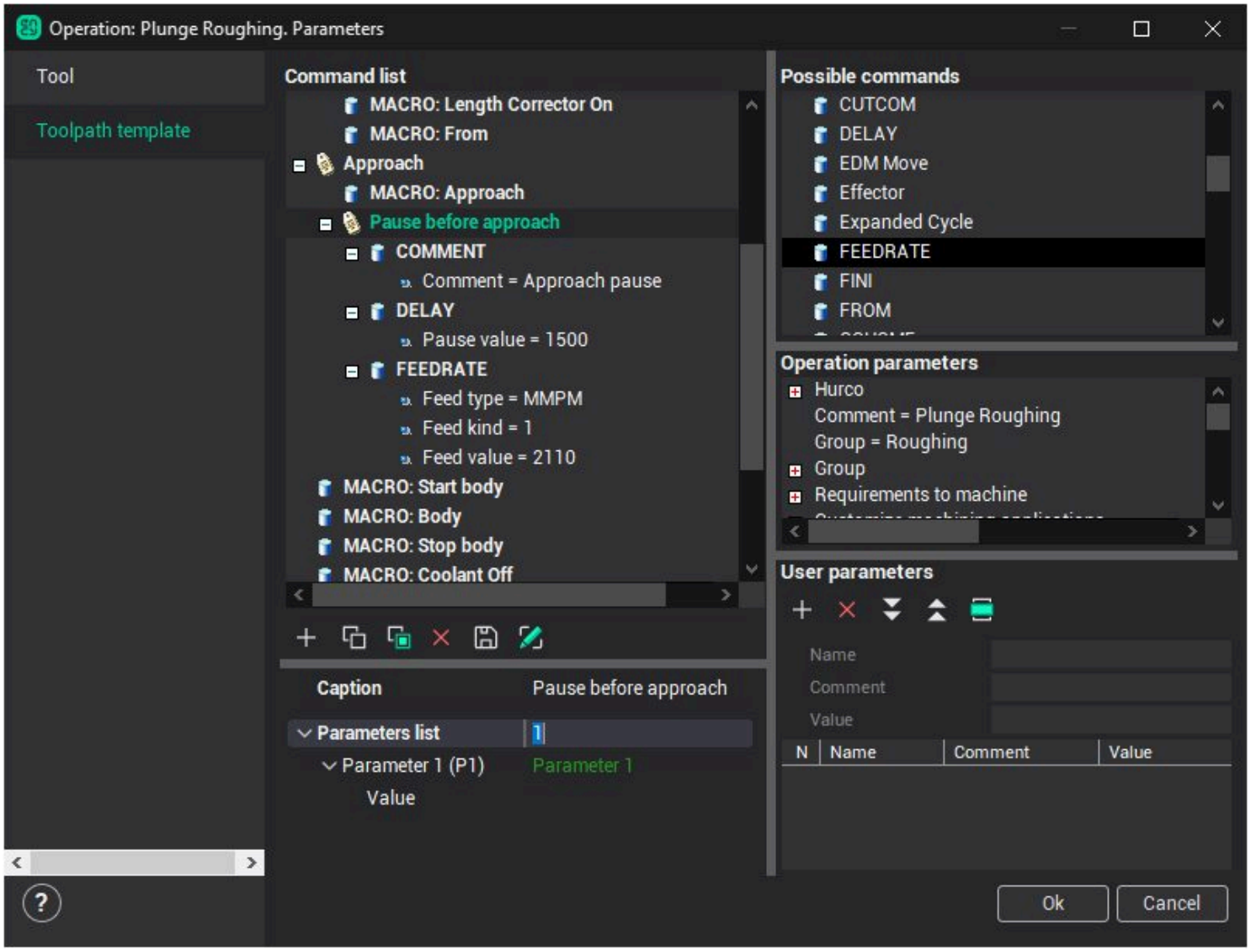
There are 3 group saving modes:

Operation. Saves group inside the operation. Saved group becomes visible in the Possible commands list. Saving the template will save the group too. But this group becomes unavailable in the other operations.

Project. Saves group inside the project. At the next loading of the project, all operations and this group will be loaded too.

External file. Saves group in the external file. This allows you to use the group in all operations and projects. But it is necessary to remember that parameters specified on the ParametersList are unavailable in the other operations. So, it is impossible to use that parameters.

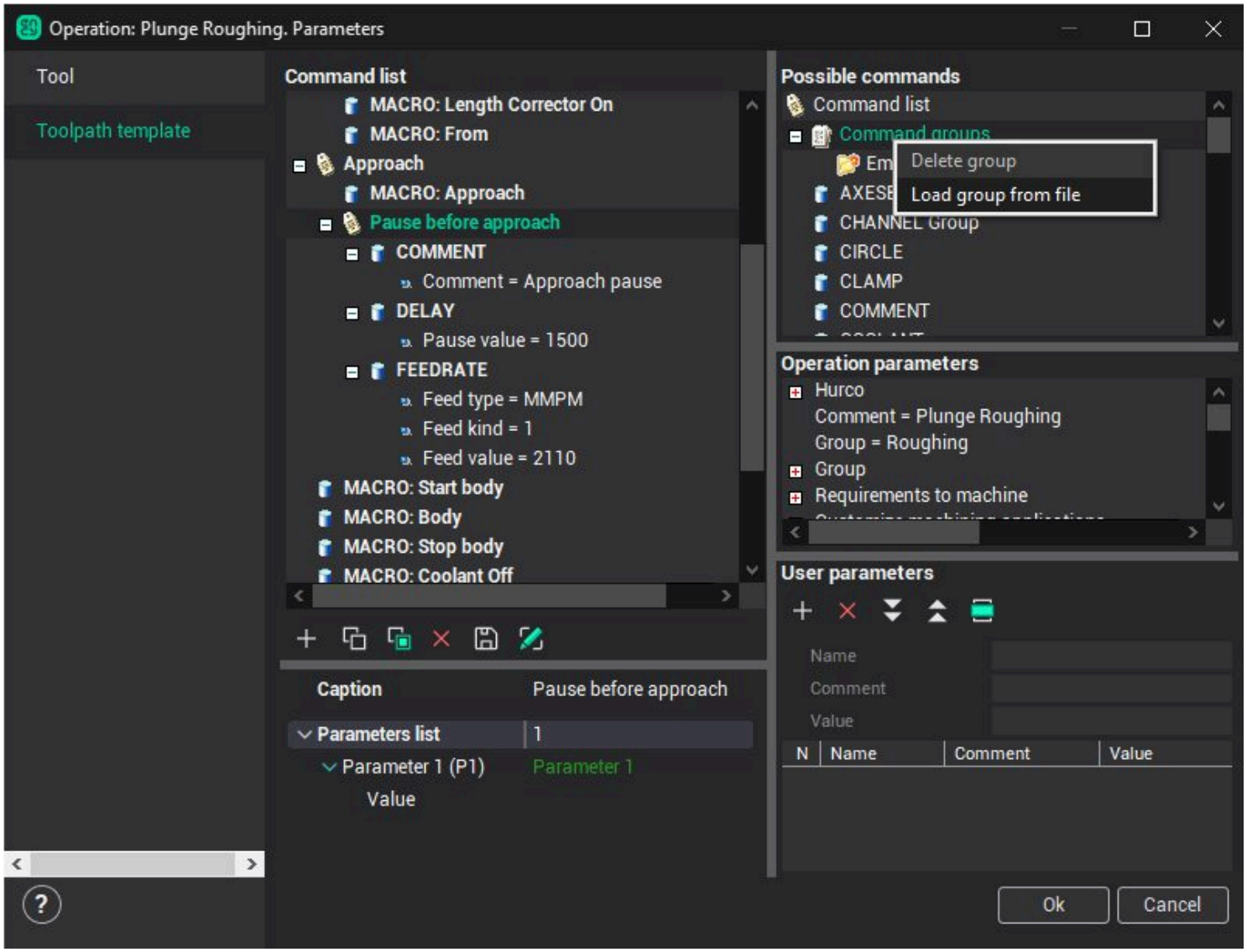
To use the parameters, create them directly inside the group. These parameters will be saved with group. To create parameters inside the group, set the parameters count and specify them all.



These parameters will be available inside the group.

So it is possible to save all typical groups of commands to external files and even use them on the another computer. To load the group of commands from an external file, open the context menu by right-

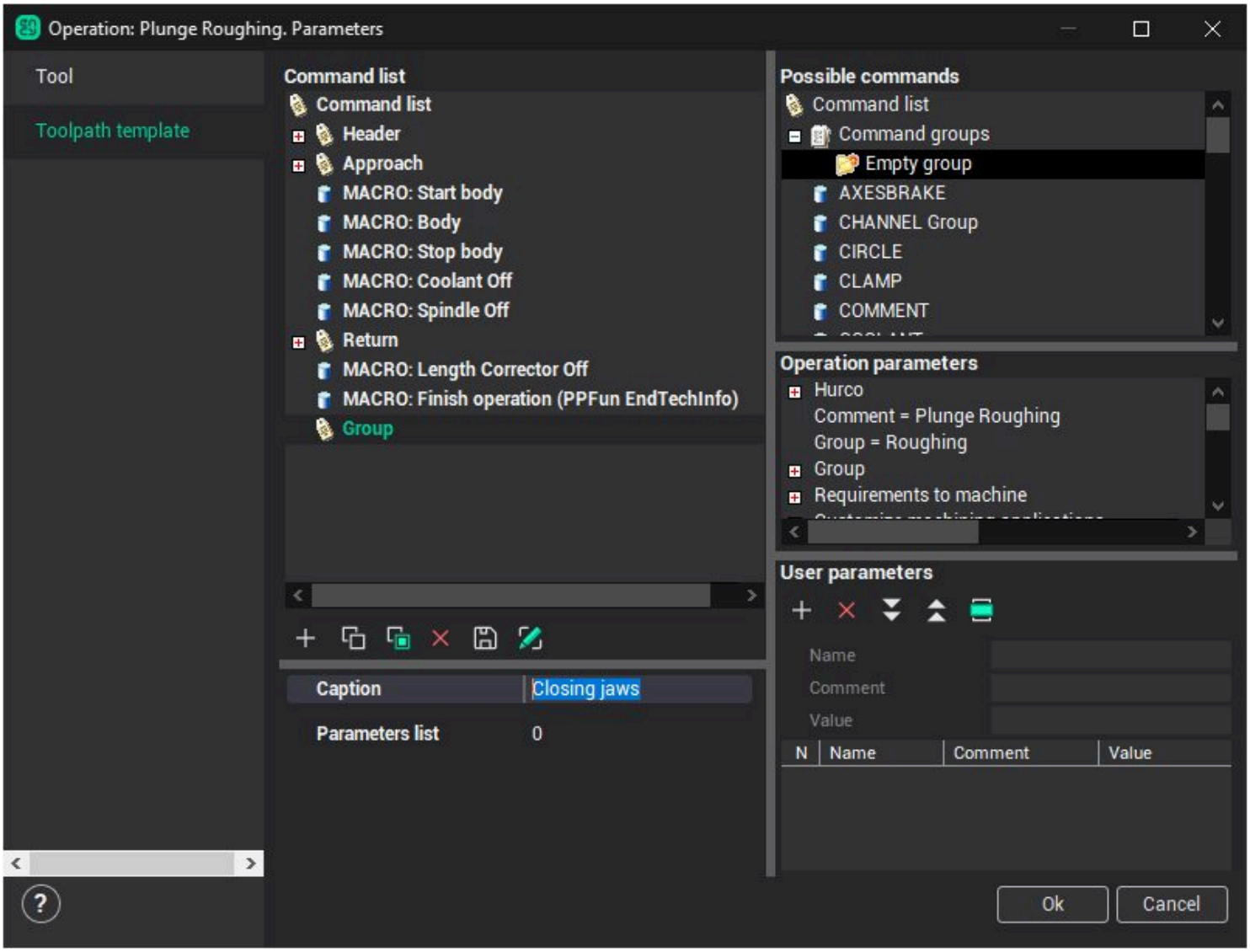
clicking on the list of possible commands and choose Load group from file item.



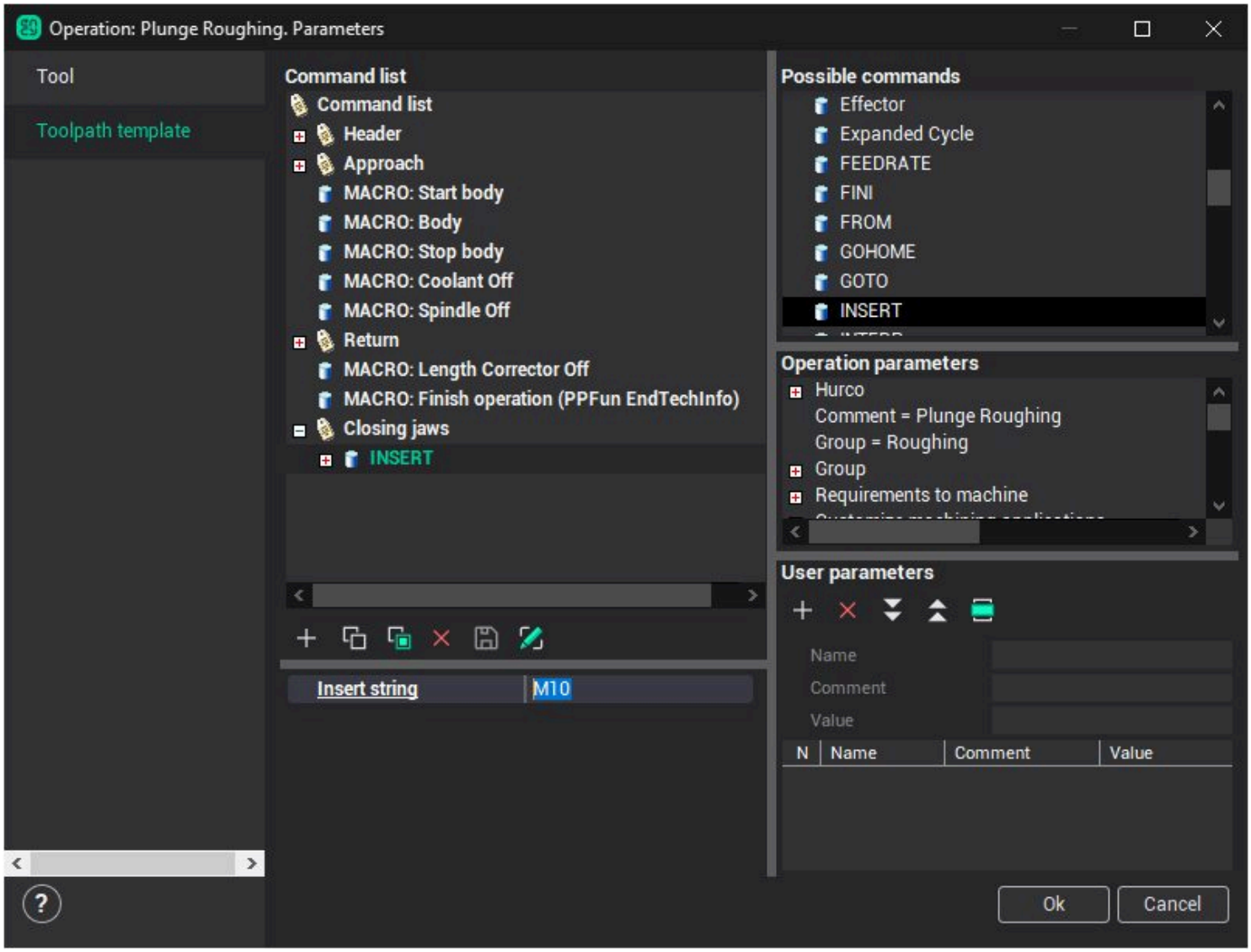
There is the example of using groups.

Suppose you want to close workpiece holder's jaws before the machining. It is necessary to send M10 command to the NC program to do this.

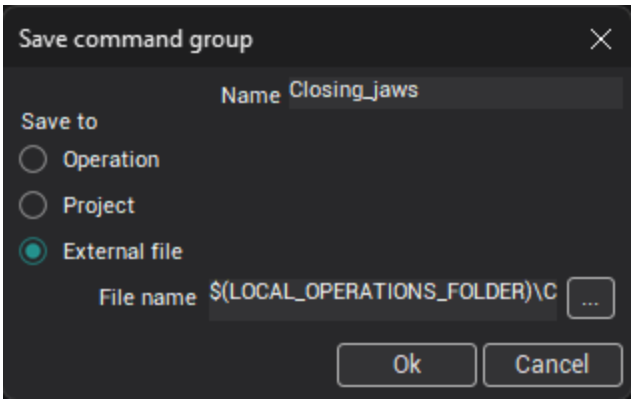
Step 1. Create the empty group in the command list and set the name Closing jaws for it.



Step 2. Provide the sending M10 command to the NC program. Add INSERT command to the group and set value = "M10".

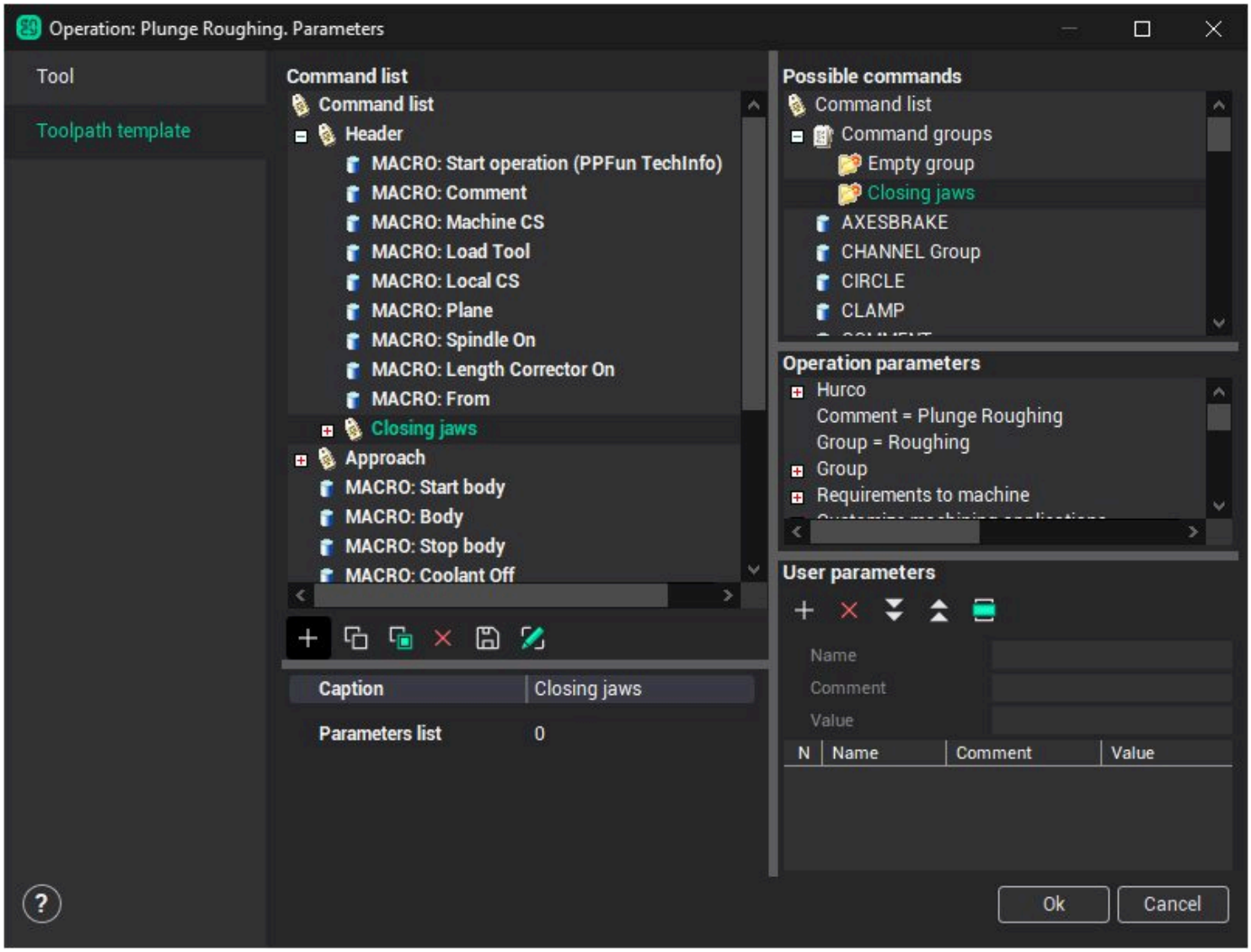


Step 3. Save formed group to the external file.

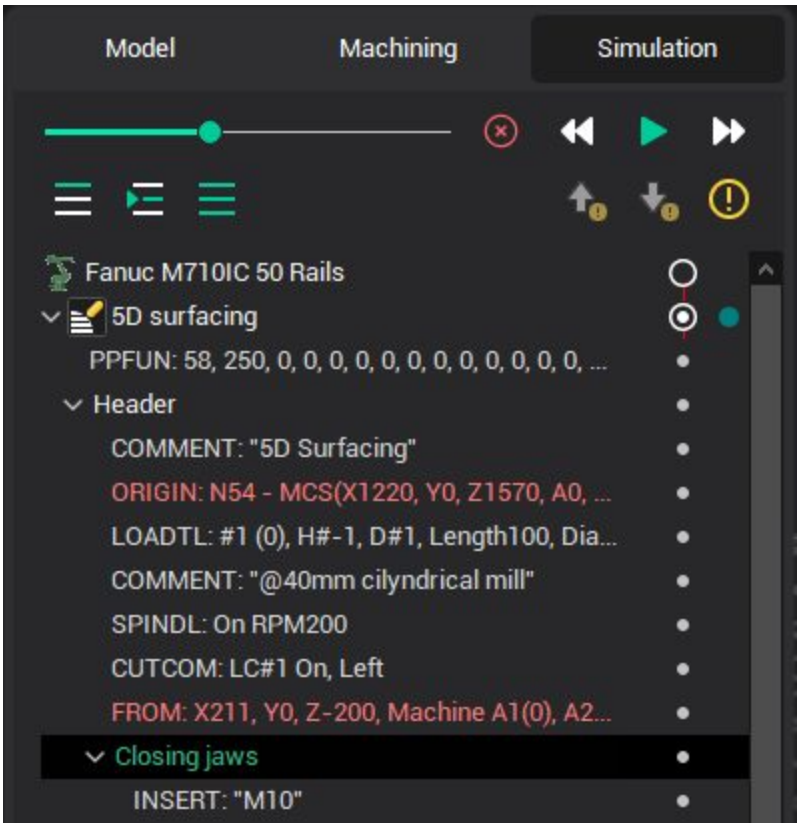


Formed group is available in the Possible commands list.

Step 4. Add closing workpiece holder's jaws before the machining operation. Select the Header operation group and add saved Closing jaws group



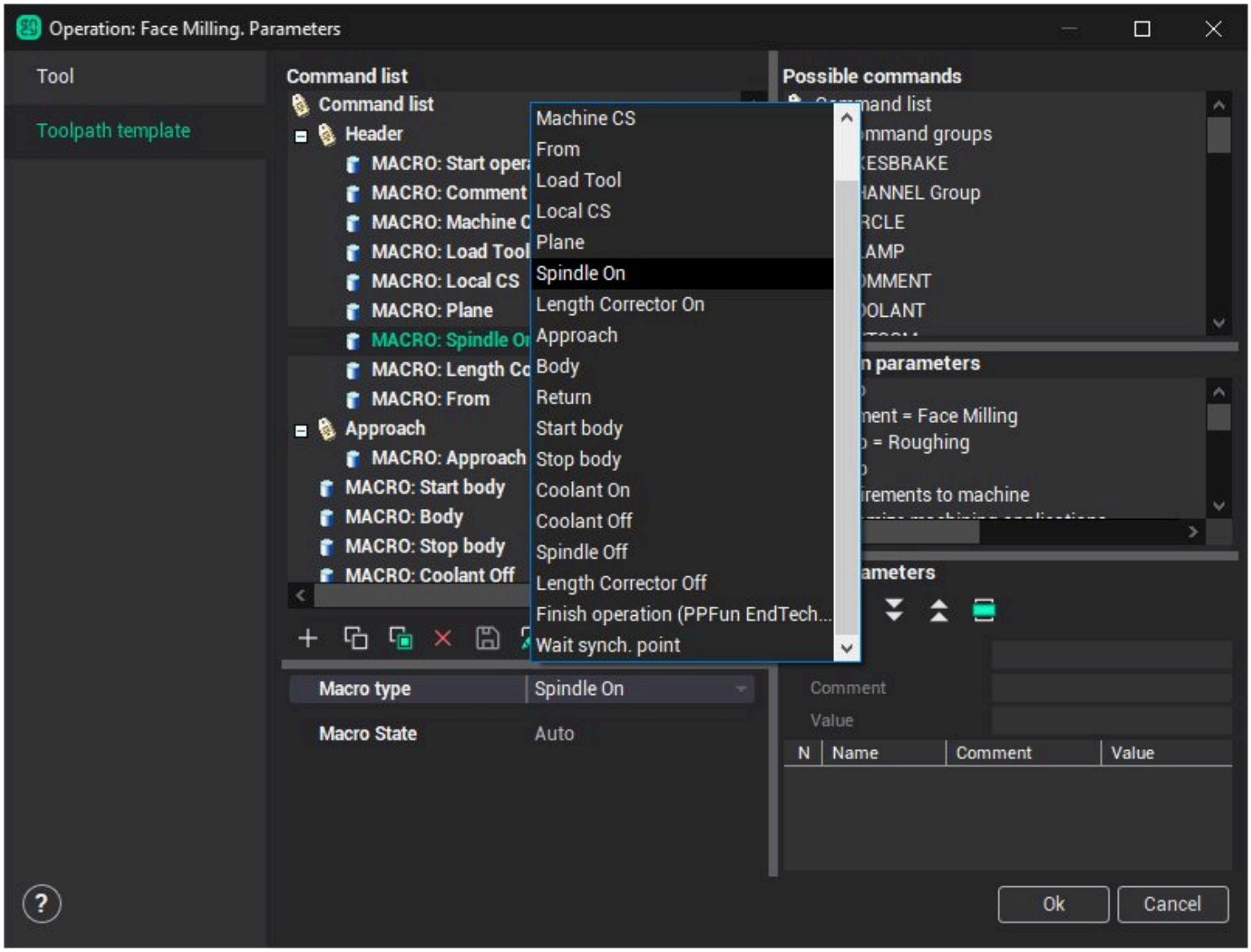
Ready! The closing workpiece holder's jaws command (M10) will be generated before the machining.



In this realization the system does not visualize closing jaws process. Closing jaws command must move jaws to the some position. To do this, it is necessary to send MULTIGOTO command to the NC program. It is possible to specify the movement value in the ParametersList. But if the group will be saved in the external file, it is necessary to add parameter to the group, not to the ParametersList.

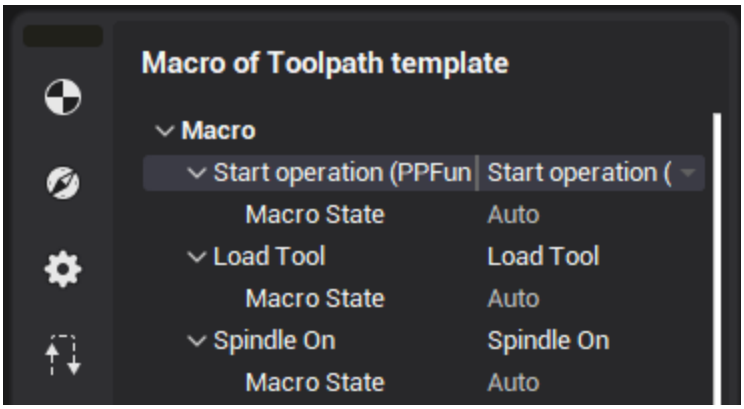
Macro commands in the tool path template

Simple command in the tool path template directly outputs the corresponding command to the CLData. Unlike, the macro command runs the subroutine that is realized inside CAM system. Some commands can be output in CLData as the result of this routine running. Type and the sequence if these commands depends on the Macro type. The list of the macro subroutines is opened while selecting the macro type on the page of the template.

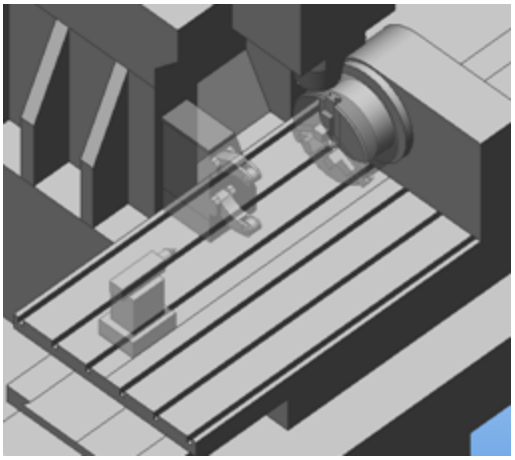


Besides the macro command has the State. State defines the running way of the subroutine. Auto state is the default state. In this mode the CLdata command are output if it is necessary.

If the state is disabled then the CLData commands is not output. It allows to exclude the unnecessary commands without the appreciably template editing. For example, it is possible to output coolant switch off command even if it was not disabled in the previous operation. For the comfortable editing of the macro commands states it is located on the additional page of the operations properties inspector.


Macro of Toolpath template
Macro
Start operation (PPFun
Start operation (
Macro State
Auto
Load Tool
Load Tool
Macro State
Auto
Spindle On
Spindle On
Macro State
Auto

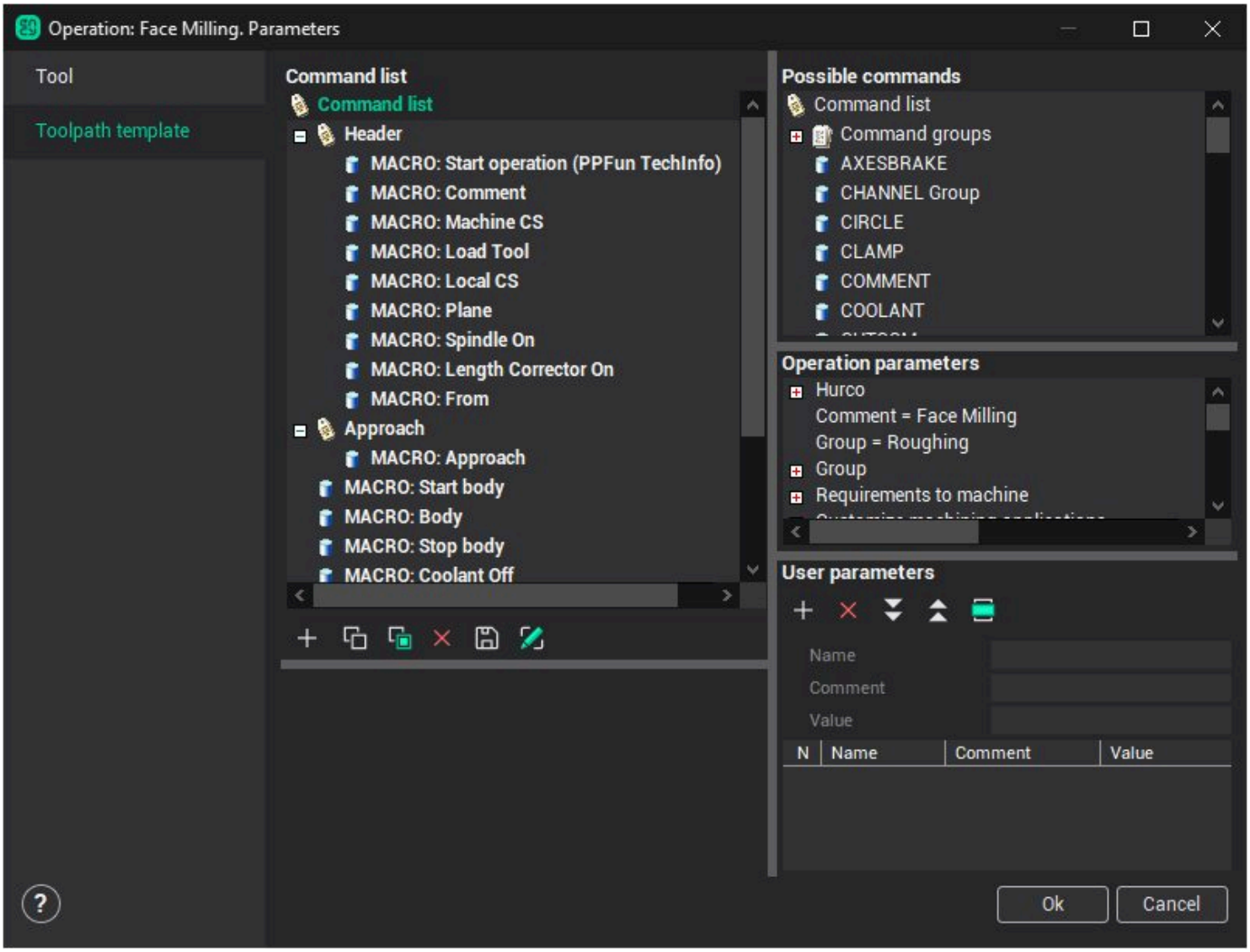
Auxiliary operation



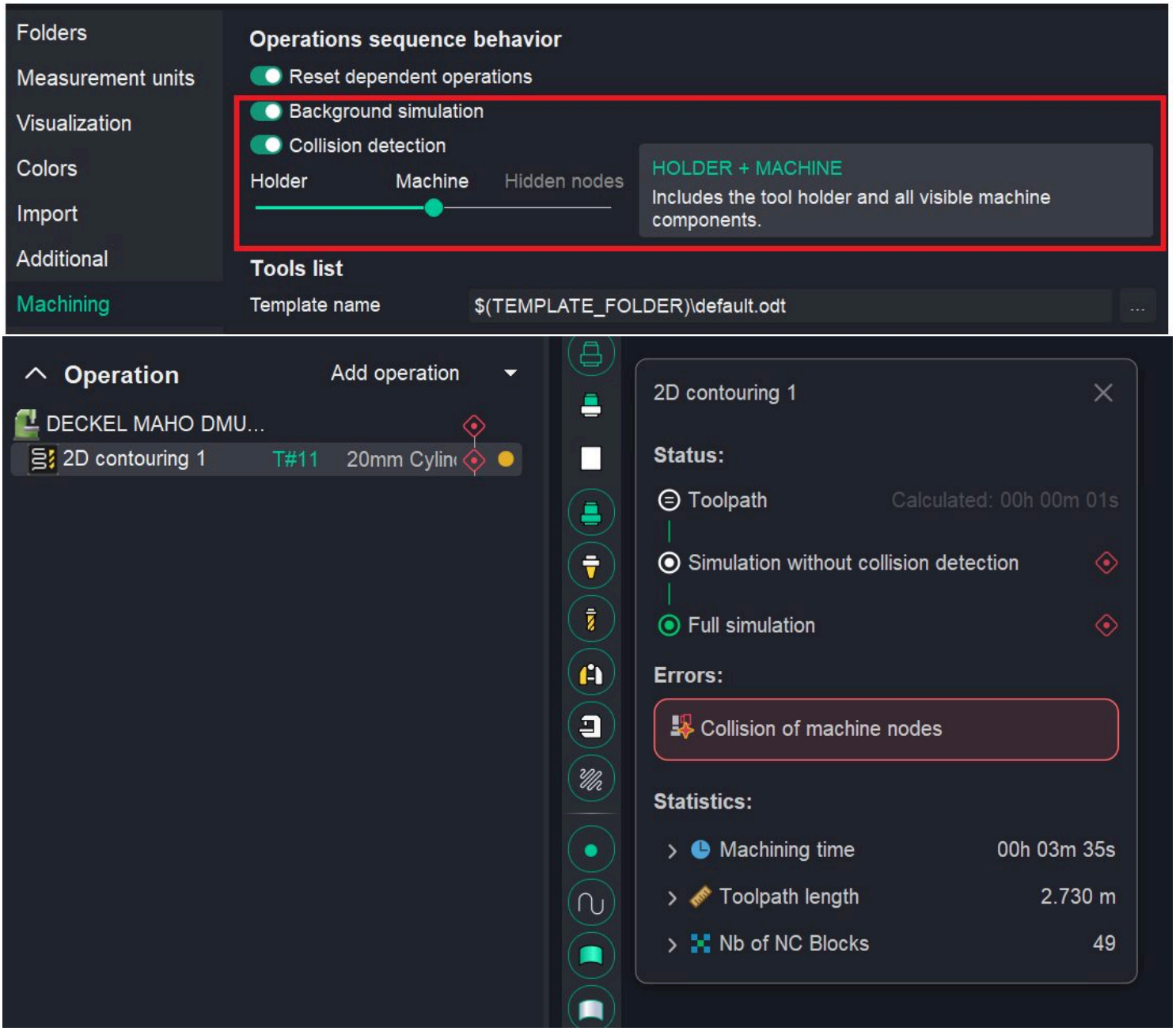
Auxiliary operations of CAM system designed to store the specific sequence of CLData commands (for specific types of machines, for particular company) into the named list, which can be saved and used many times in a process of work with the system. This, for example, may be such types of operations as clamping a chuck, tool interchange, approaching tail stock, part overturn, set of the active workpiece coordinate system G54-G59 and so on.

Instance of CAM system auxiliary operation is created as any other operation, on Create button click in the window of process, or with the help of contextual menu this button. In groups of operations equally with group Roughing, Finishing and Rest milling the Auxiliary become available. In this group enumerated the list of available operations for specific machine. By defaults for all types of machines only Abstract auxiliary operation is active, which presents by itself a semimanufactured article with empty set of CLData and list of parameters.

Editing of auxiliary operation properties are made in window, which is opening by pressing Parameters button in technological window on the Toolpath template tab.


Operation: Face Milling. Parameters
Tool
Toolpath template
Command list
Command list
Header
MACRO: Start operation (PPFun TechInfo)
MACRO: Comment
MACRO: Machine CS
MACRO: Load Tool
MACRO: Local CS
MACRO: Plane
MACRO: Spindle On
MACRO: Length Corrector On
MACRO: From
Approach
MACRO: Approach
MACRO: Start body
MACRO: Body
MACRO: Stop body
MACRO: Coolant Off
Possible commands
Command list
Command groups
AXESBRAKE
CHANNEL Group
CIRCLE
CLAMP
COMMENT
COOLANT
Operation parameters
Hurco
Comment = Face Milling
Group = Roughing
Group
Requirements to machine
User parameters
Name
Comment
Value
N
Name
Comment
Value
Ok
Cancel

Collision Detection in Background Simulation



Application area:

The Collision Detection feature in Background Simulation automatically verifies toolpaths for physical interferences with the machine and workholding equipment without interrupting the user's workflow. After the trajectory calculation is complete, the Technology tab displays not only the calculated toolpath but also the workpiece model after machining. Additionally, the system generates alerts for any detected gouges or issues with machine components.

This functionality is particularly useful for simple projects or scenarios where fine finishing operations with small stepovers are not required. As a result, the user can continue designing the manufacturing

process directly in the Technology tab without launching a separate machining simulation, enabling immediate assessment of the final result.

However, calculating the results of Background Simulation requires significant computational resources and time, consuming a substantial portion of the central processing unit (CPU) capacity and random-access memory (RAM) volume. This leads to a noticeable increase in the time required for simulating machining results and trajectory monitoring when a large number of dependent operations need to be recalculated, multiple workpiece re-setups are required, or fine finishing with a very small stepover is performed.

Firstly, you have the option to disable all Background Simulation calculations. If you do so, the CAM system will generate the toolpath based solely on the 3D model. You can then verify the processing results at a later stage in the Simulation tab. However, the CAM system includes certain automated

operations that require the results of the previous processing step. These operations include:

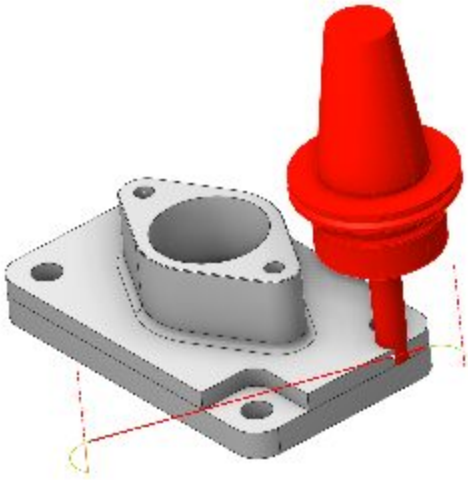
1. Roughing waterline;
2. Roughing plane;
3. Pocketing;
4. Hole machining;
5. All turning operations.

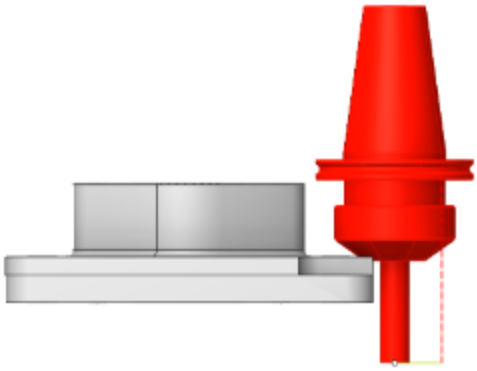
For these operations, the CAM system will automatically enable the calculation of the previous processing results when generating toolpaths — even if background calculations are otherwise disabled.

There are two switches in the settings on the Machining tab that control background simulation:

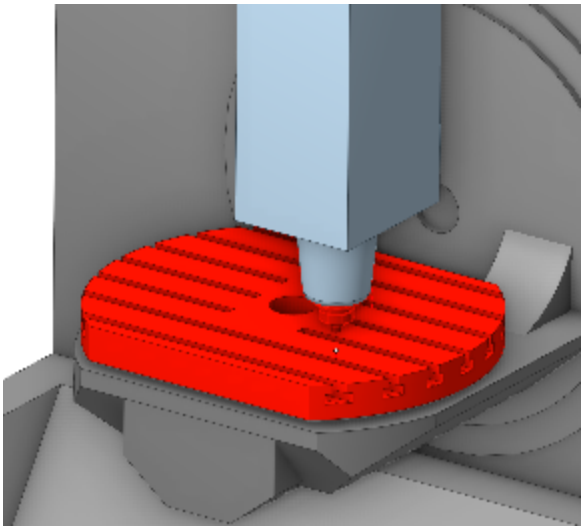
1. Background simulation is needed to calculate the intermediate workpiece. It will also indicate if the workpiece is being cut at non-cut feeds; however, it will not detect collisions with the arbor, fixtures, or machine tool. Error detection requires the configuration of appropriate filters.
2. To enable background collision detection, the Collision Detection option must be activated in the system settings. Once this option is enabled, a list of verification criteria becomes available. The user can select one of the following collision-checking modes:

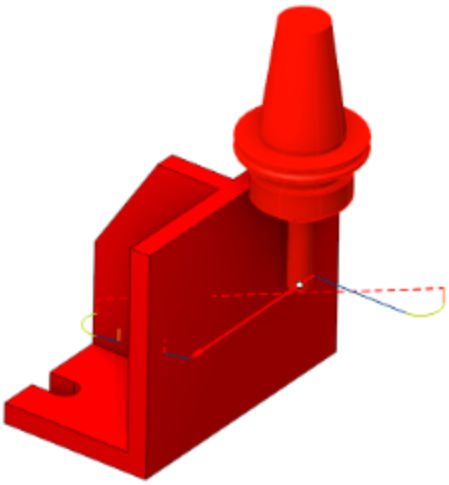
- Holder - Check collisions with the tool holder only. Fastest computation.



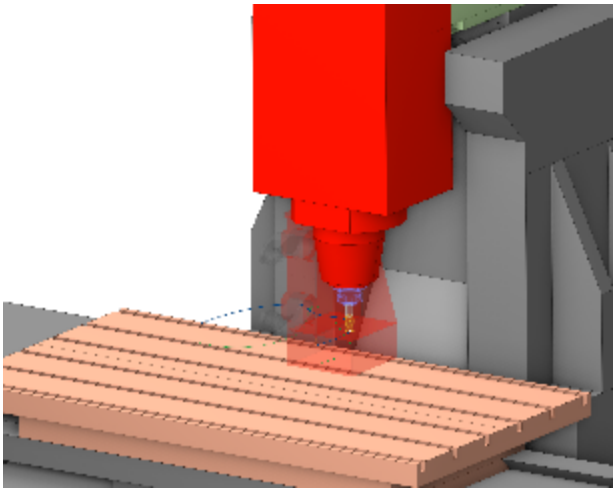


- Machine (Holder+Machine) - Includes the tool holder and all visible machine components





- Hidden nodes (Holder+Machine + Hidden nodes) - Full check: includes holder, machine, and hidden machine nodes default by the user.



The more elements included for verification, the longer the operation calculation will take. Upon completion of the check, the user is presented with the status of each operation. Details

In addition to Background Simulation, you can run step-by-step simulation on the Simulation tab. The key aspects of the interaction between Background Simulation and step-by-step simulation are covered in another chapter of the documentation. Details

List of Operation Statuses

The Operation Status List is needed to monitor operation stages, quickly diagnose issues

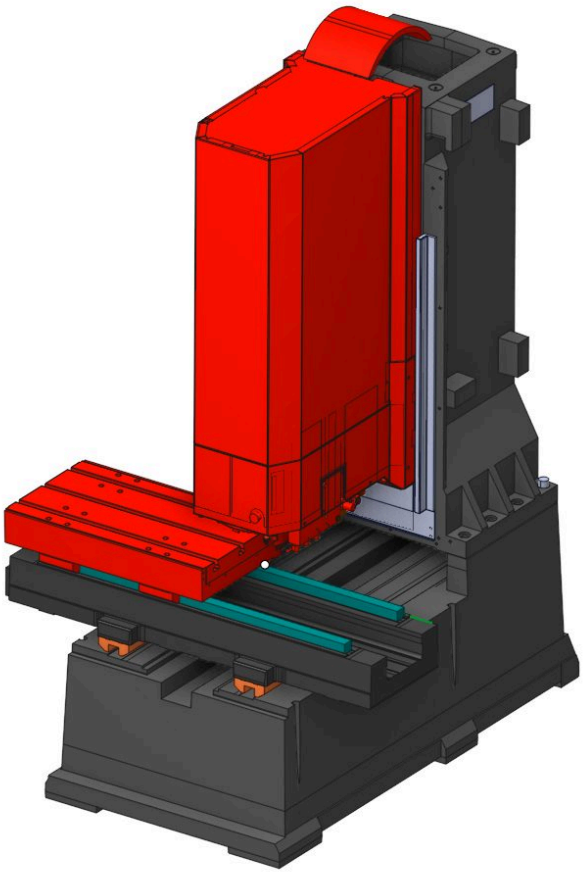
List of Operation Statuses:

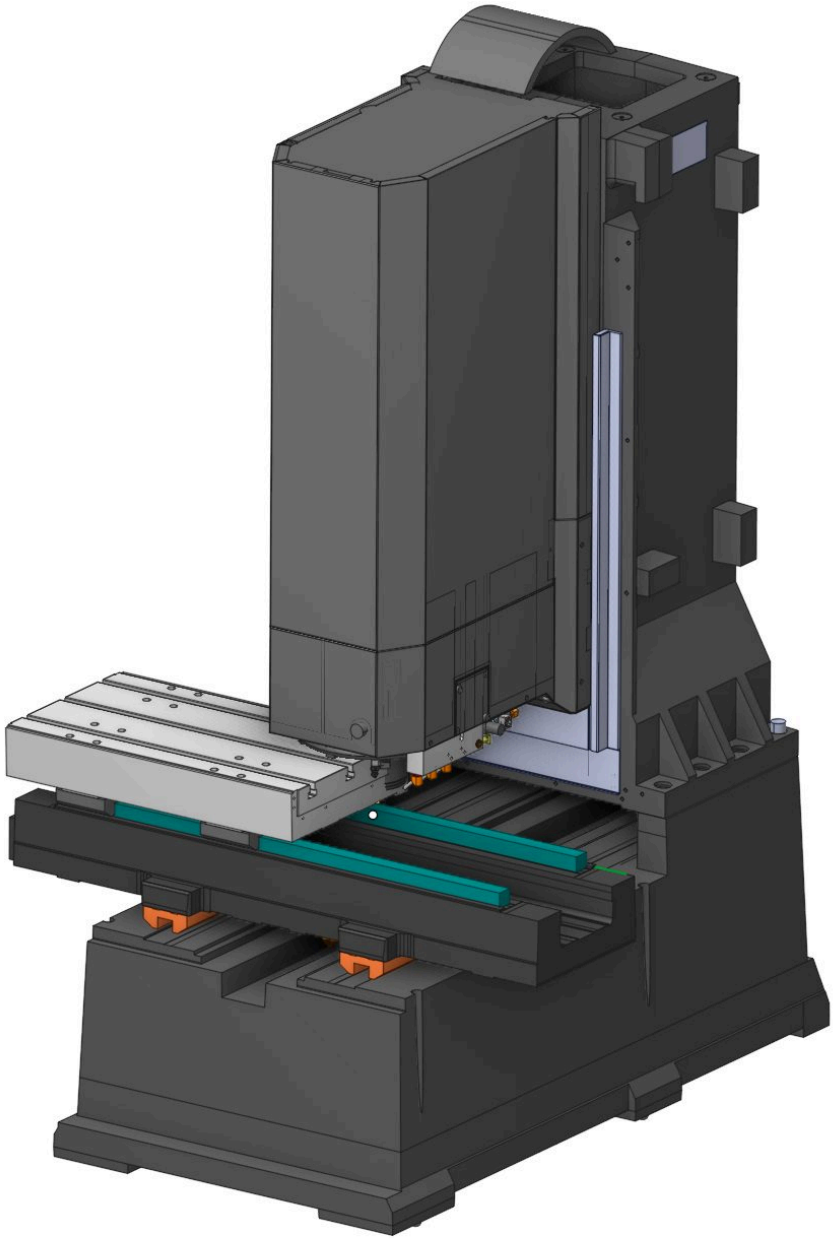
-  - Status: The operation is not calculated.
-  - Status: The operation is calculated (Background Simulation Disabled / Not Performed and step-by-step simulation Not Executed).
-  - Status: The operation is calculated and the result is machine-calculated (Background Simulation Performed - Calculated only: verification with the workpiece. step-by-step simulation Not Executed).
-  - Status: The operation is calculated and the result is machine-calculated. Collision check with the toolholder is performed. (Background Simulation Performed — Toolholder Only (Idle-run workpiece entry); step-by-step simulation Not Executed).
-  - Status: The operation is calculated and the result is machine-calculated. Full background simulation has been performed. There are hidden machine components that were not included in the background check. (Background Simulation and step-by-step simulation Completed).
-  - Status: The operation is calculated and the result is machine-calculated. Full background simulation has been performed. (Background Simulation and step-by-step simulation n Completed).
-  - Status: The operation disabled.
-  - Status: The operation has a collision.

Background Simulation is discussed in Chapter. Details

Step-by-step simulation is discussed in Chapter. Details

Editing of the Collisions Ignore List





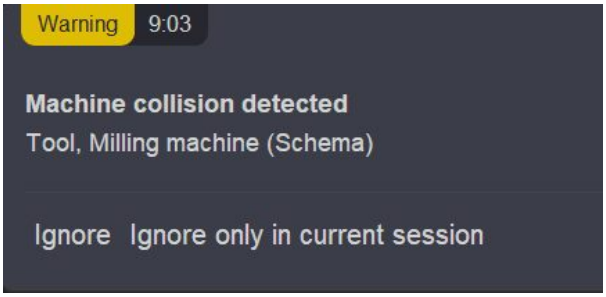
Application area:

Implemented direct editing of the CDIgnoreList section in the machine scheme directly from CAM system. Previously, editing required manual data entry in Notepad or the use of MachineMaker.

When a collision occurs during simulation, a notification appears indicating that a collision between nodes has occurred. A Ignore or Ignore only in curent session button has been added to this notification.

Functionality:

1. When a collision is detected during a simulation, a notification alerts the user, specifying the colliding nodes (e.g., node X and node Y). This notification includes an Ignore or Ignore only in curent session buttons. Pressing the "Ignore" button adds the identified node pair to the CDIgnoreList, effectively ignoring future collisions between those specific nodes.

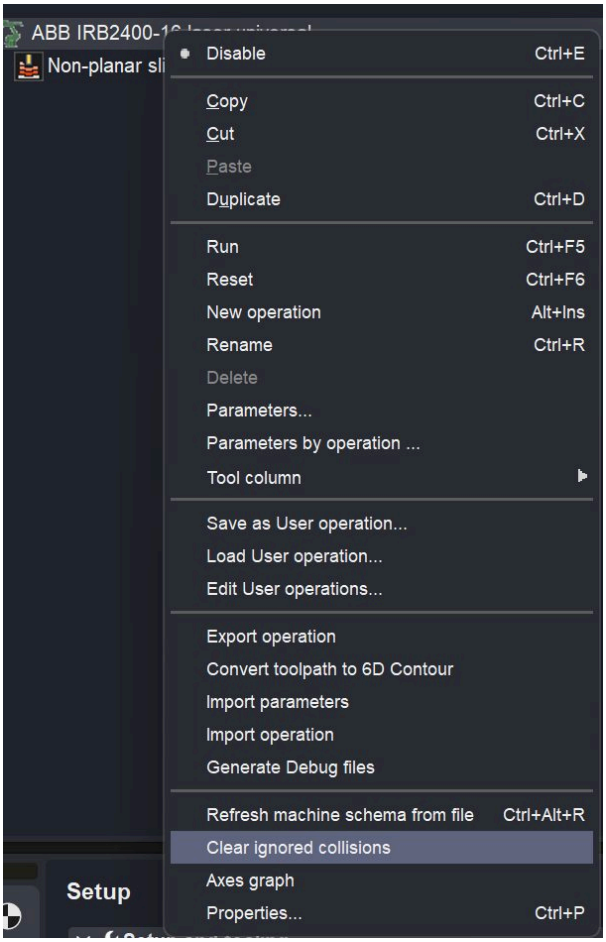


In the first case, a file with the extension ".cdil" is created next to the machine kinematics file. If the file doesn't exist, it is created.

When creating a new CAM project, the system references a machine. If an ignore list file exists next to the machine file, it is loaded from there; otherwise, the data is loaded directly from the machine file.

In the second case, a file is not created. Everything only works within the current session.

2.The file can be cleared through the machine's popup menu (on the technology tab).



Toolpath recalculation control

Application area:

Toolpath recalculation control is used to identify differences between the current operation parameters and the values used to calculate the existing toolpath.

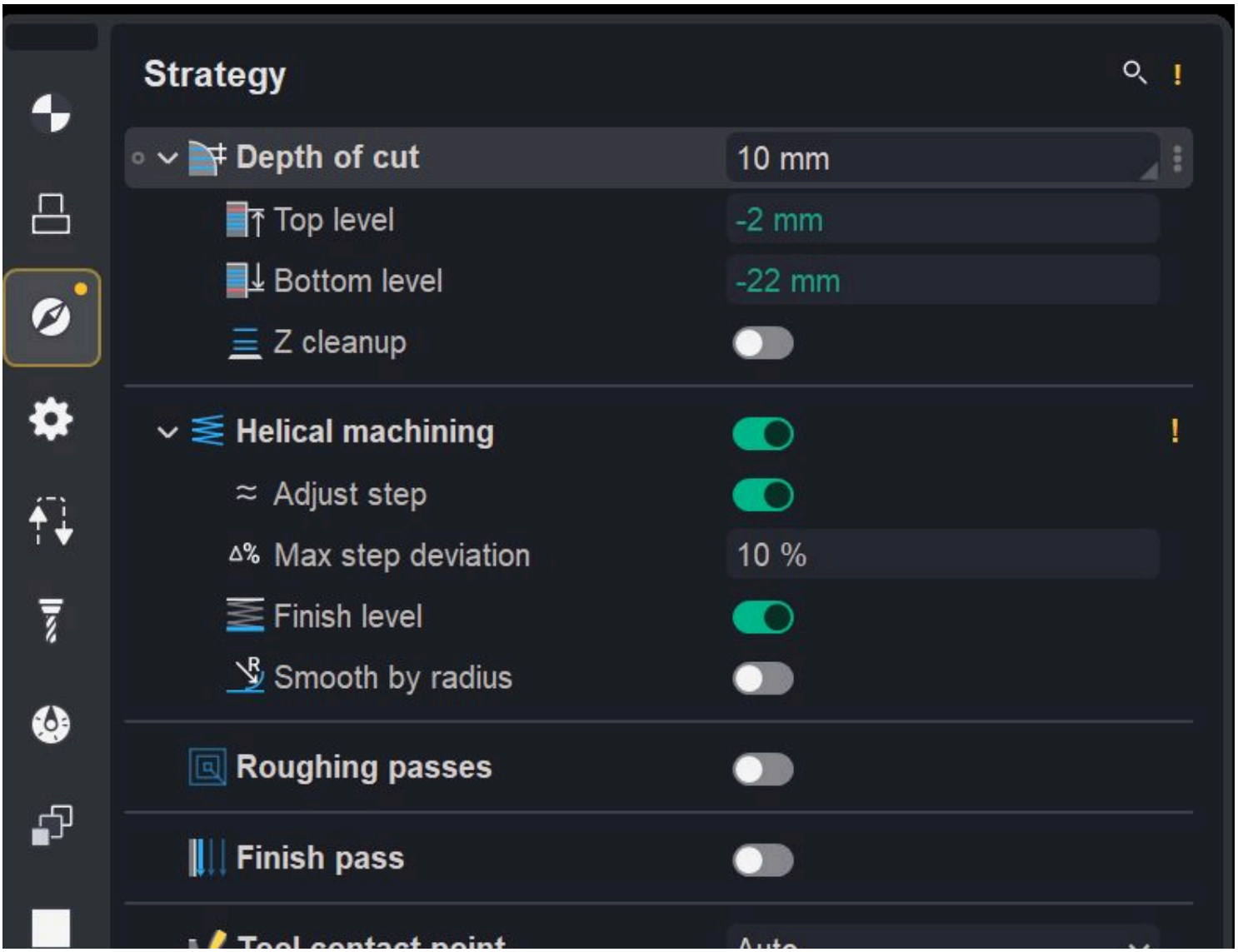
The function helps locate edited parameters, restore the previously calculated parameter state, and update the affected parts of the toolpath when a full recalculation is not required.

Changed parameter indicators:

Calculated parameter state. The CAM system saves the operation's parameter values when the toolpath is calculated. These values are used to identify subsequent changes.

Operation status. A yellow warning status indicates that the current operation parameters differ from those used to calculate its toolpath.

Inspector indicators. Changed parameters, their groups, and the corresponding inspector pages are marked to help locate settings that require recalculation.



Changed parameter indicators on the Strategy page.

Recalculation and parameter restoration:

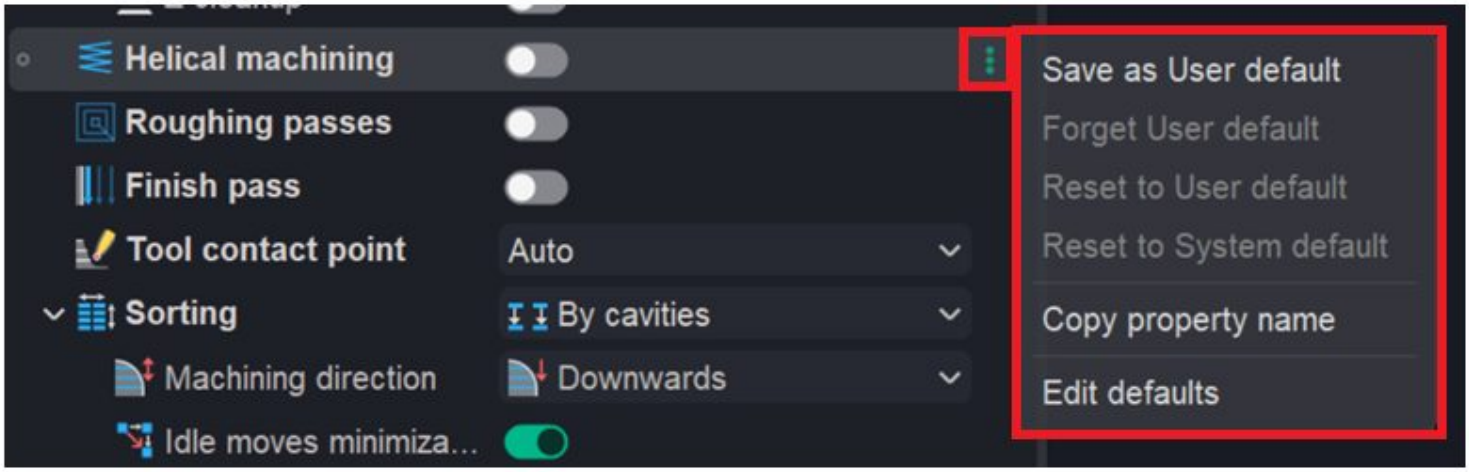
Partial recalculation. If the parameter changes do not require a full recalculation, the calculated toolpath is preserved. Only the affected parts, such as links, approaches, and retracts, are updated.

Parameter restoration. Before recalculation, the parameter values used to calculate the current toolpath can be restored.

Settings:

Auto-reset toolpath. The setting for hiding the parameter change hints is located in System setup > Additional > Auto-reset toolpath.

User defaults in inspector



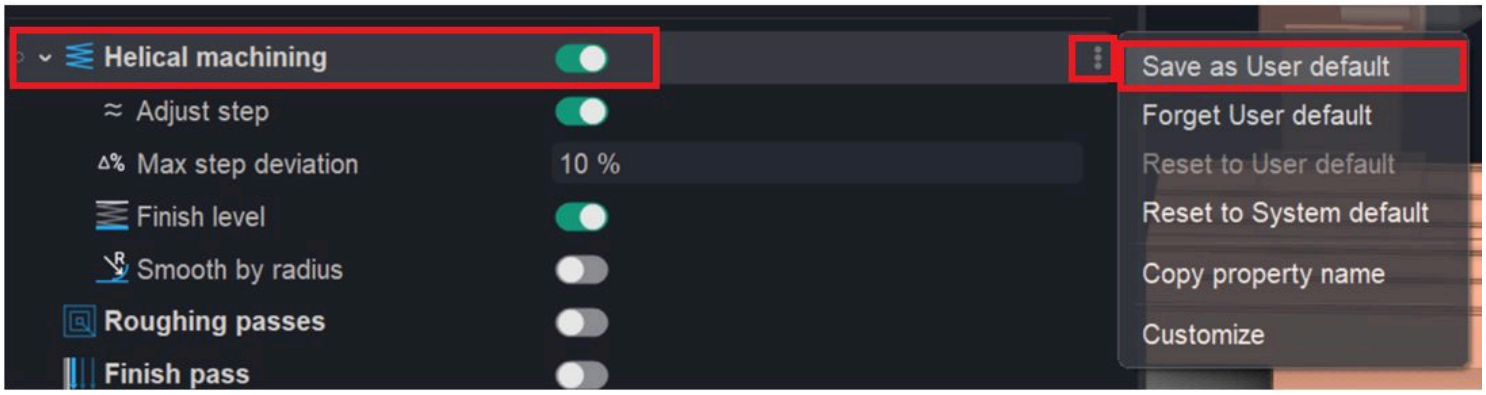
Application Area:

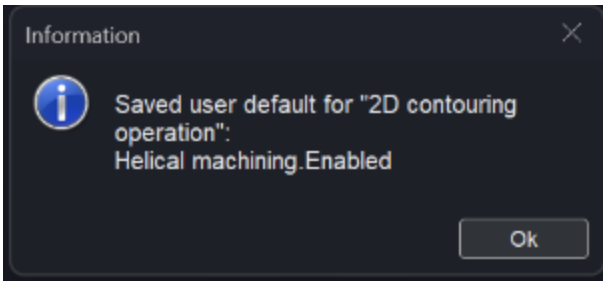
These menu options let users save, reset, or customize property settings and copy property names for use elsewhere.

Save as User default - A separate file with the *.usrdef extension is created in the folder (%userprofile%\Documents\ENCY NB\Version 3\Operations\Defaults). This file contains the new default values for all parameters displayed in the inspector row.

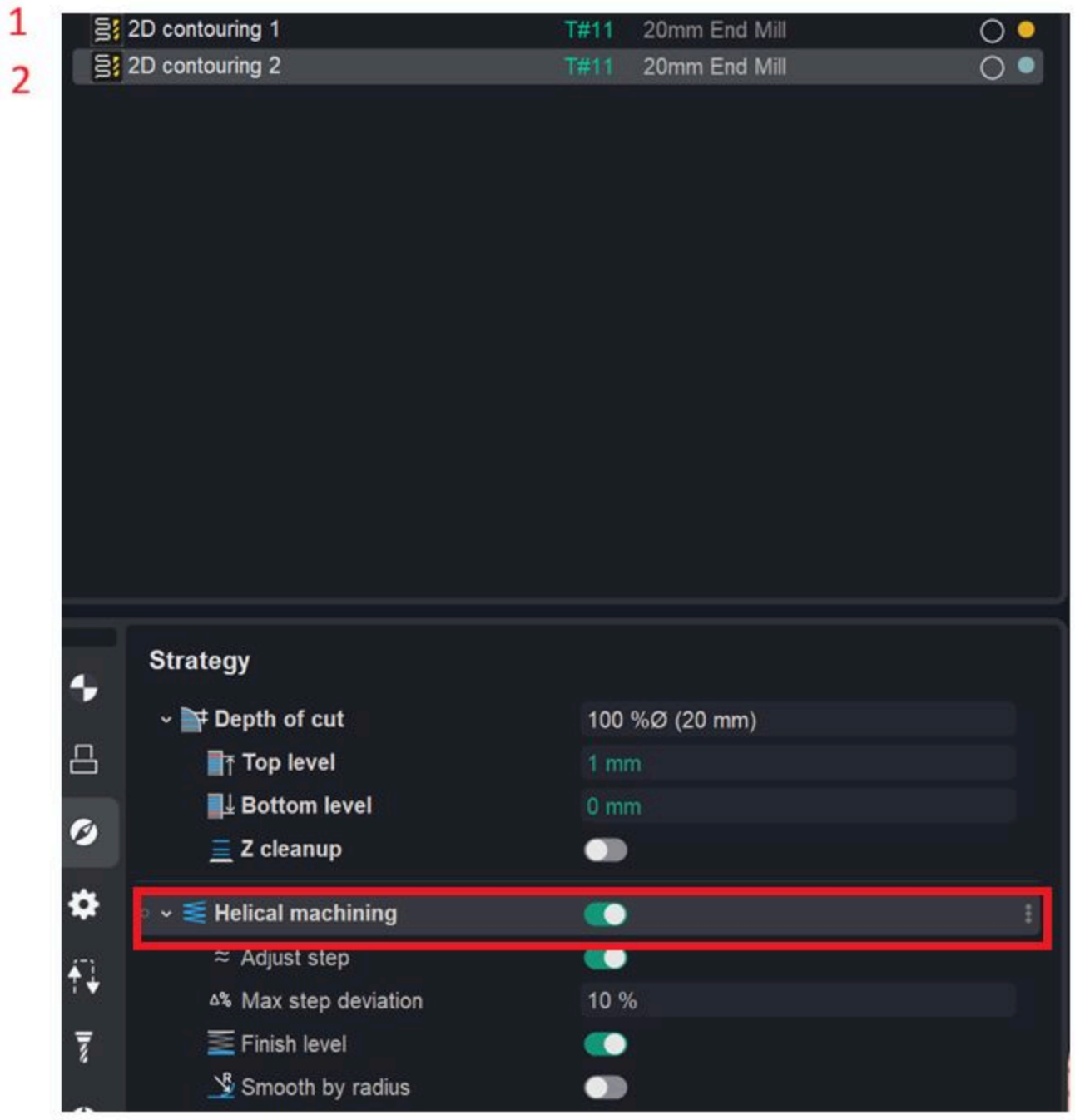
▼ Click here to expand..

For example, if you set Helical machining to true in a 2D Contouring.





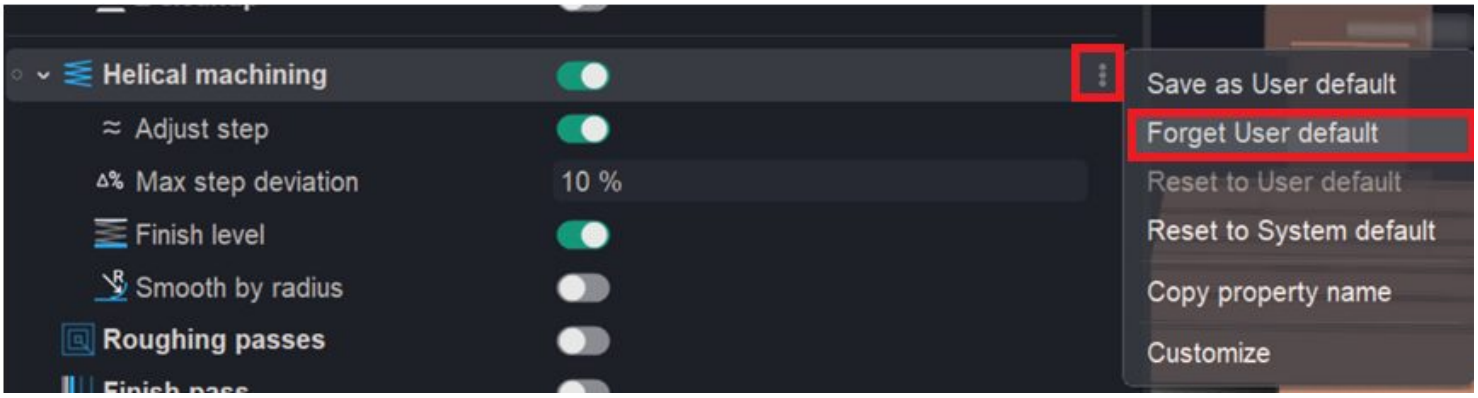
The next 2D contouring operation will have Helical machining already set to true.

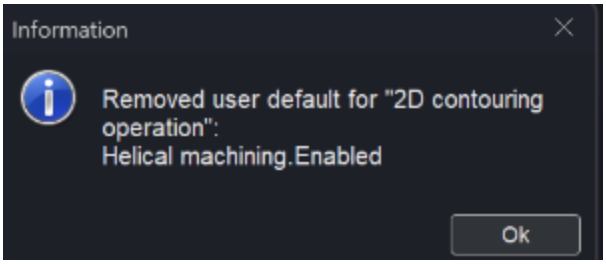


Forget user default — forget (delete) the user default. The value for all parameters of the current inspector row is removed from the *.usrdef file. If no defaults remain in the file, the entire file is deleted. The current value in the inspector remains unchanged.

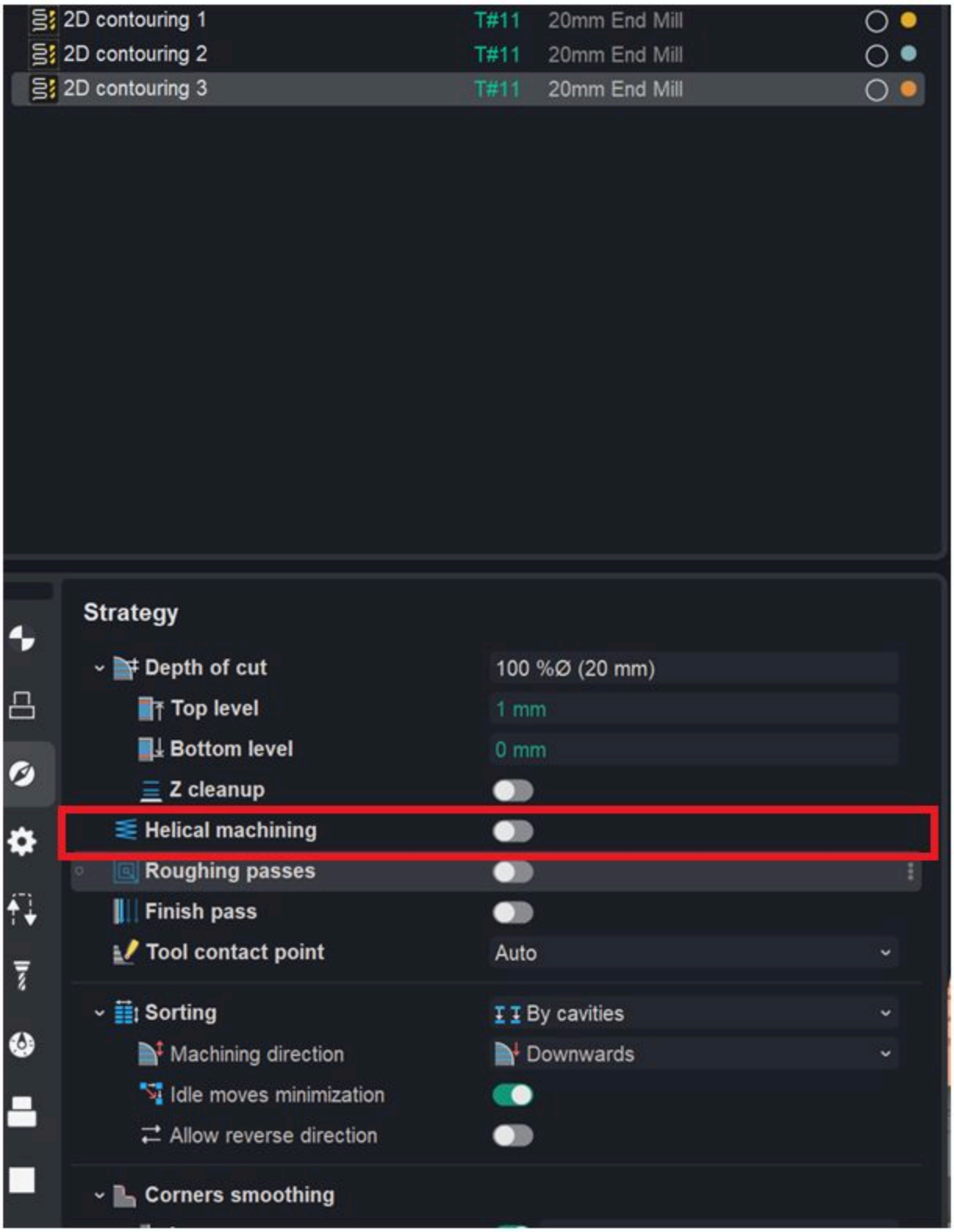
▼ Click here to expand..

For example, you can also choose to forget this rule.





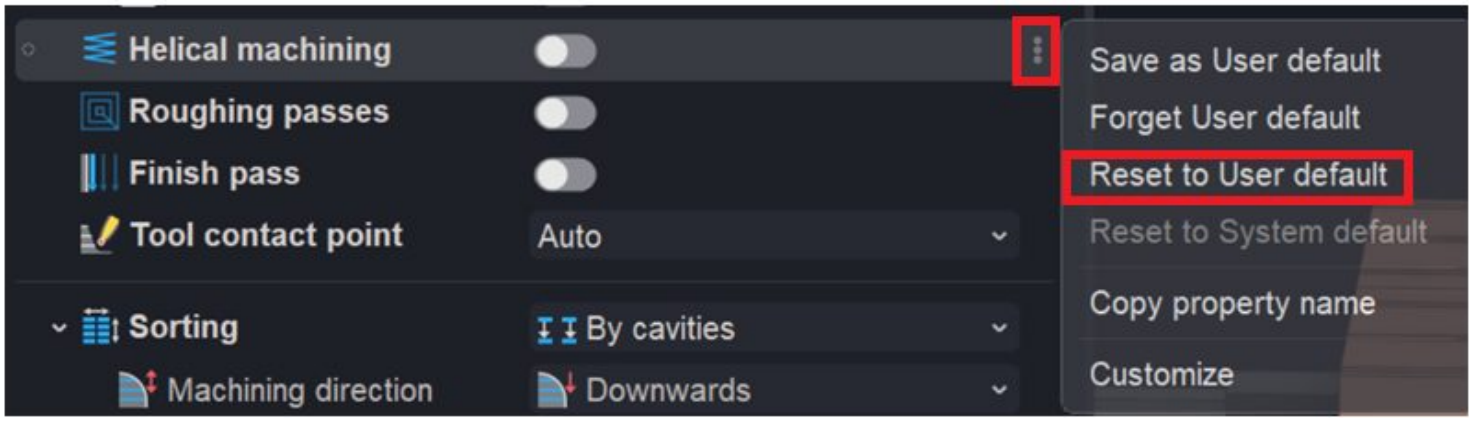
In this case, the next operation will be created without applying this setting.

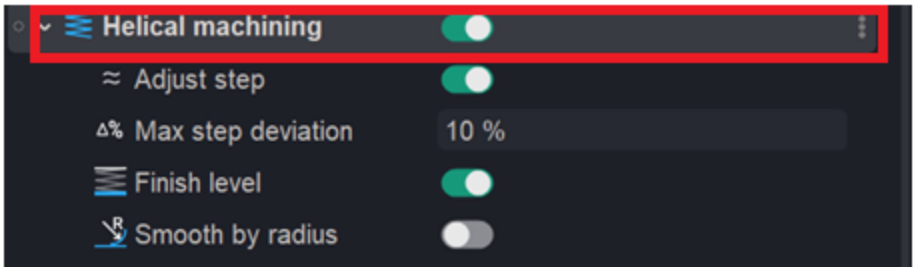


Reset to User default — reset to the user default. The current property value is replaced with the value saved in the user default settings.

▼ Click here to expand..

For example, if you need to apply your custom rules to an existing operation, use Reset to User default — the operation will adopt your saved defaults. Important: this applies to only one specific parameter.

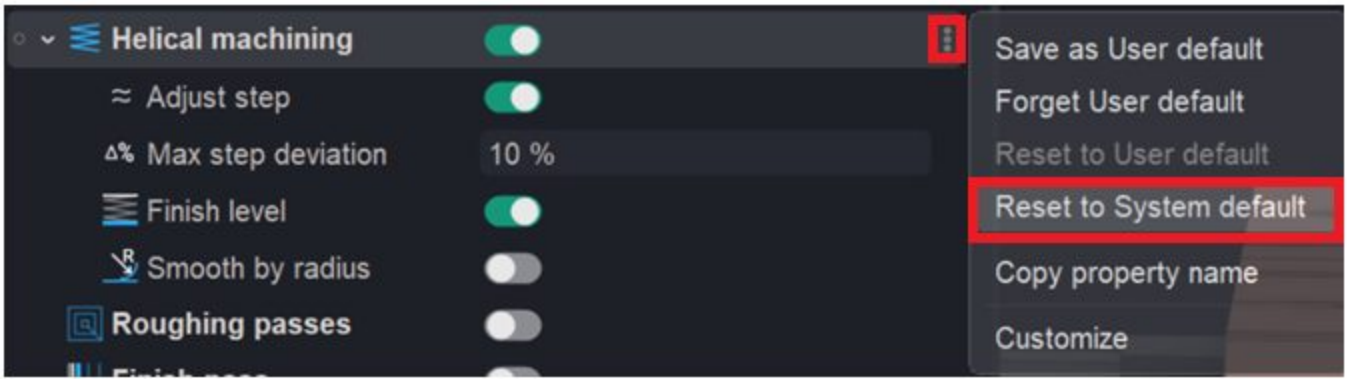


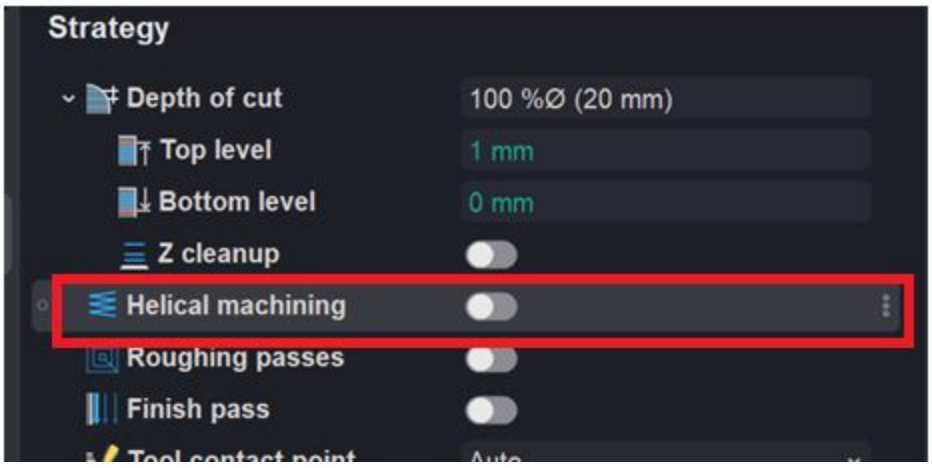


Reset to System default — reset to the system default. The current property value is replaced with the value saved in the system default settings.

▼ Click here to expand..

For example, if you need to restore the system default settings for an existing operation, use Reset to System default — the operation's parameter will be reset to their original factory values. Important: this applies to only one specific parameter.

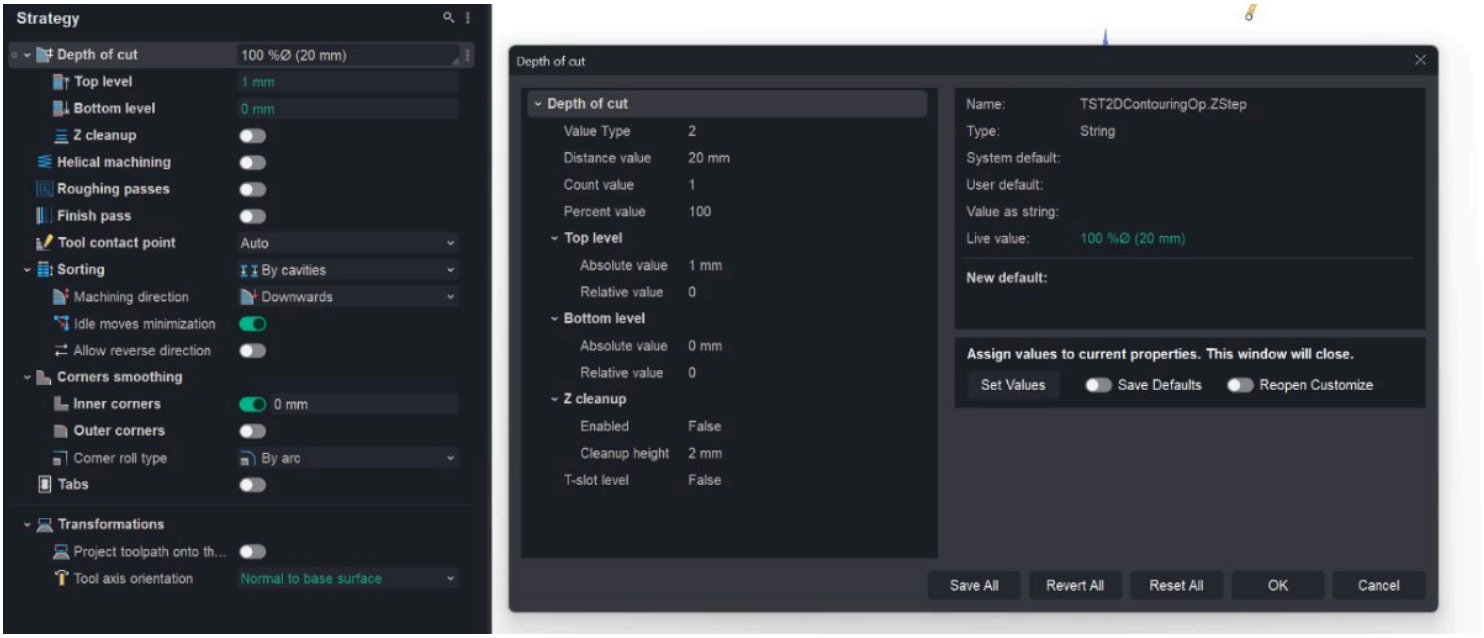




Copy property name — copy the property name. The unique property name is copied to the clipboard. You can use this name to refer to the property in expressions of other properties, in post-processors, and in extensions via the API. If multiple properties are displayed in the row, their names are copied, separated by commas.

Edit defaults. A window will open where you can view and edit the expression for the selected property.

▼ Click here to expand..



Set Values. When enabled, the assigned values are also written as the new default (the "New default" field above), so they persist for future use — not just for this instance.

Save Defaults. When enabled, the assigned values are also written as the new default (the "New default" field above), so they persist for future use — not just for this instance.

Reopen Customize. When enabled, after setting the values the Customize window reopens instead of staying closed, so you can keep editing.

Save All. Saves the current values of all properties as their defaults.

Revert All. Discards your unsaved edits and restores all properties back to their last-saved values.

Reset All. Resets all properties to the system default values

OK. Confirms and applies the changes, then closes the dialog.

Cancel. Closes the dialog without applying any changes.

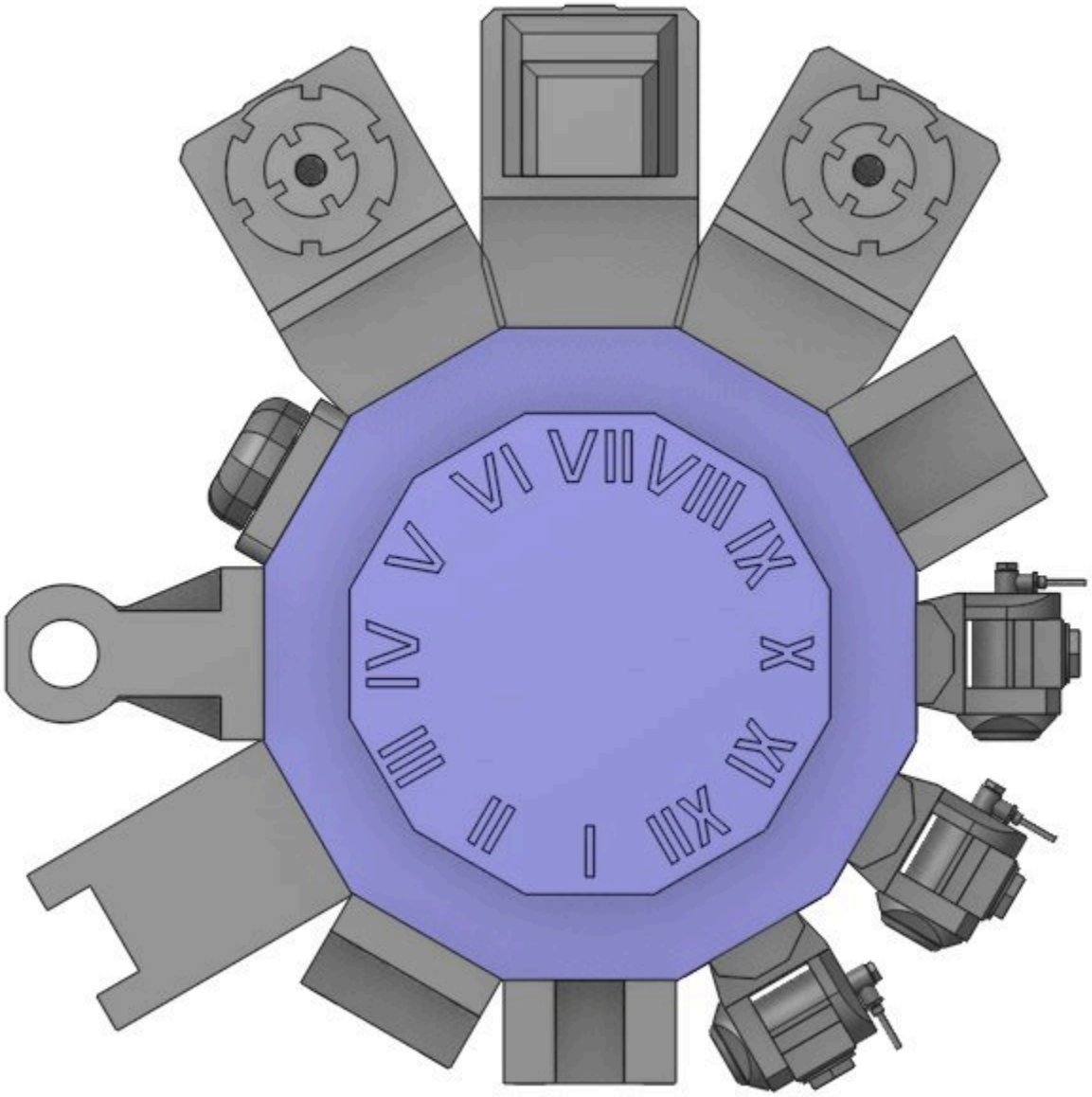
The export and import functionality for settings (files in the *.usrdef format) is implemented similarly to the mechanism for working with user operation files. It can be accessed via the window for creating a new operation in Customize mode. Details

Structure of the machining mode

Application area:

Here are the key sections you will discover when using the 'Technology' tab.

Machine setup panel



Application area:

This tab is necessary for configuring the fundamental project parameters, primarily related to the kinematic scheme of the equipment.

Description.

Machine name. Must be unique.

▼ Click here to expand..

Comment. Description of the equipment used

Group. Machine group is used only for sorting in a machine library. Seven values are available: - Jet cutting - Wire EDM - Milling - Lathe - Turn-milling - Robotics - Other

Developer. Developer of this machine. It can be xml schema author or cnc machine manufacturer.

NC System name. Name of NC-system.

Postprocessor file. Path to postprocessor file.

Interpreter file. Path to interpreter file (*.snci)

Measurements. Parameters of measurements

Tooling.

Tooling parameters

▼ Click here to expand..
Position for tool block. Configures the contents and parameters of tool blocks in the tool positions of the machine. You can put a tool block wherever you want, or hide it by making it empty.

Visible. Configures the visualization parameters of the tool block. When the flag is active, the tool block is visible, and its color can be altered in this state. The parameters of this group are the same as in the Frame group.

Control Parameters.

Machine control parameters.

▼ Click here to expand..
Arcs. The parameters related to arcs.

Use arcs. The parameter determines whether arcs will be displayed in CLData (e.g., circular interpolation). When the option is disabled, arcs are not displayed, and the trajectory is represented by line segments.

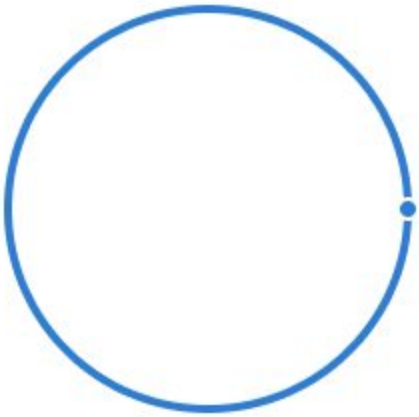
Use arcs in XY plane. The parameter determines whether arcs will be displayed in a specific plane.

Use arcs in YZ plane. The parameter determines whether arcs will be displayed in a specific plane.

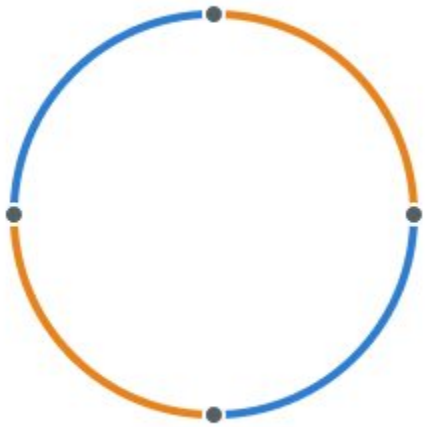
Use arcs in ZX plane. The parameter determines whether arcs will be displayed in a specific plane.

Circles division. The parameter allows you to divide circles into segments (Do not divide, by quarters, by halves). It is necessary for generating certain interpolations (e.g., generating circular interpolation using R).

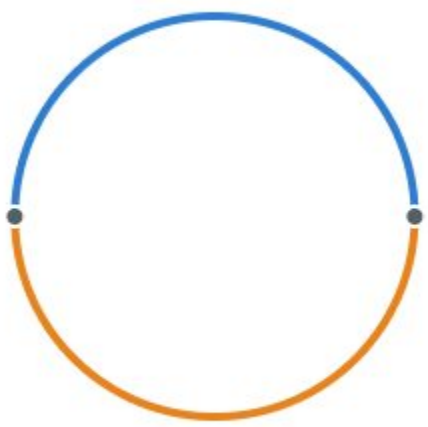
Do not break. The circles do not split into arcs.



Into quadrants. The circles are divided into quarters



Into halves. The circles are divided into halves



Minimal Arc Length. If the length of the arc is less than the value set in this parameter, the arc will be displayed as line segments.

Maximal Arc Radius. If the radius of the arc is greater than the value specified in this parameter, the arc will be displayed as line segments.

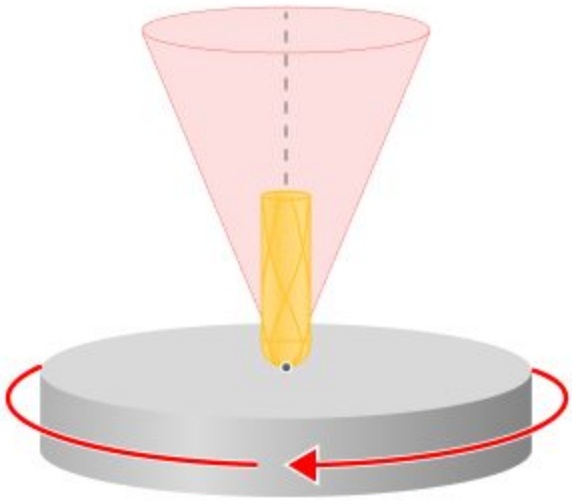
Spatial arcs. The arcs do not lie in planes. The parameter usually is used for robots to generate MULTIARC outputs. MULTIARC movement command defines a circular arc passing through three points with a predetermined orientation of the tool in space for each of these points.

Minimal distance between start and end points. If the distance between the start and end points of a spatial arc is less than a specified value, line segments are generated between these two points.

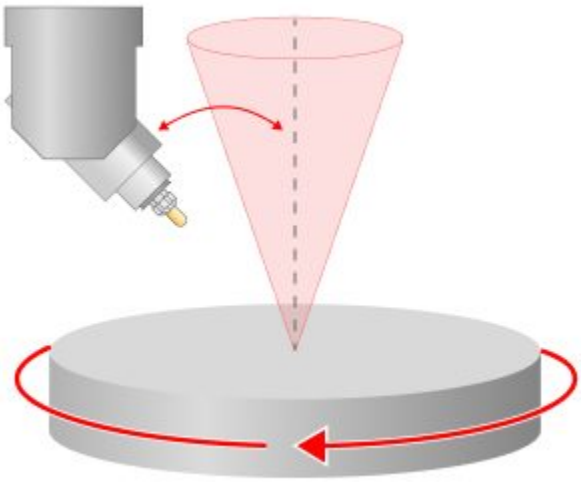
Reorientation mode. The law of interpolation at an intermediate point can refer to either relative to path or relative to base CS.

Consider middle point orientation. The parameter controls whether orientation is considered at the midpoint.

Singularities. This parameter group defines the system's ability to automatically prevent singularities. Red — singularity zone. It occurs when the rotary table rotates and the tool axis stays in line with the rotation axis.



Allowed axis deviation. This parameter specifies the angle by which machine's or robot's nodes can deviate from the nominal calculated position to bypass the singularity zone.



Robots operate with additional singularity parameters specifying permissible deviations for axes in their components: Wrist, Elbow, Base. Each component supports individual singularity avoidance angle configuration.

Default machine(robot) configuration. The state of flip status equipment elements as indicated on the kinematic equipment shema.

Origin list parameters. The parameters related to displaying the local coordinate system.

Origin prefix. The parameter determines the prefix used for displaying the local coordinate system (LCS).

Initial number. The parameter sets the initial value for displaying the local coordinate system (LCS).

Increment value. The parameter determines the step size for calculating the number of the next local coordinate system (LCS).

Create new origin for part group. The parameter determines whether automatic creation of a new Local Coordinate System (LCS) is required for the next group of parts of a certain type.

Radius Compensation Parameters. Defines the toolpath construction features considering the tool radius compensation.

Sharp Corner. When considering the tool radius compensation at contour sections containing sharp angles, the system applies special algorithms to construct the tool path. This parameter defines the maximum angle that the system will consider as sharp.

Rotary Transformations. Defines the availability of rotational transformation parameters in the Transformation tab.

Polar interpolation is available. When the flag is active. one can change a linear axis to the rotary one in the simple 3-axes milling process, using the Rotary Transformation parameter on the Transformations tab. Usually it is necessary on the lathes that has the drive mill tool. or for the rotary operations at the four- or five-axis machines.

CNC supports polar interpolation. When the flag is active, then CNC interpolation tick on the Transformations tab in the Rotary Transformation group is available. If this parameter is on, then one can generate the G-code with the commands to switch on/off the polar interpolation.

Cylindrical interpolation is available. When the flag is active, one can utilize the cylindical interpolation function, using the Rotary Transformation parameter on the Transformations tab. It gives the possibility to mill the side surface of cylindrical parts by programming the unrolled curves. The system programms the unrolled curves in the [X,Y,Z] coordinates, but most of the turn-milling machines can process only in [X,C,Z] coordinates. So the cylindrical interpolation makes the transformation [X,Y,Z] => [X,C,Z].

CNC supports cylindrical interpolation. When the flag is active, then CNC interpolation tick on the Transformations tab in the Rotary Transformation group is available. If this parameter is on, then one can generate the G-code with the commands to switch on/off the cylinrical interpolation.

Start from C=0. When the flag is set, then the parameter Start from C=0 on the Transformations tab in the Rotary Transformation group is active by default. It sets rotary axis to zero position before switch on the polar or cylindrical interpolation.

Tool frame output. Defines the format for outputting the tool orientation parameters in CLDATA commands containing five-axis movements.

Additional transformation. `*Base coordinate system*` defines the coordinate system with respect to which the transformation will be performed. It can be: Tool CS, Workpiece CS, Global CS or any other geometrical CS you have created before. Also additional translation and rotation of this CS can be defined inside Additional transformation group of parameters.

Set machine state flags. When programming robots, their physical coordinates relative to Cartesian coordinates are also determined by the flip position of their joints. This parameter specifies whether to output flags in CLData indicating the flip position of the joints.

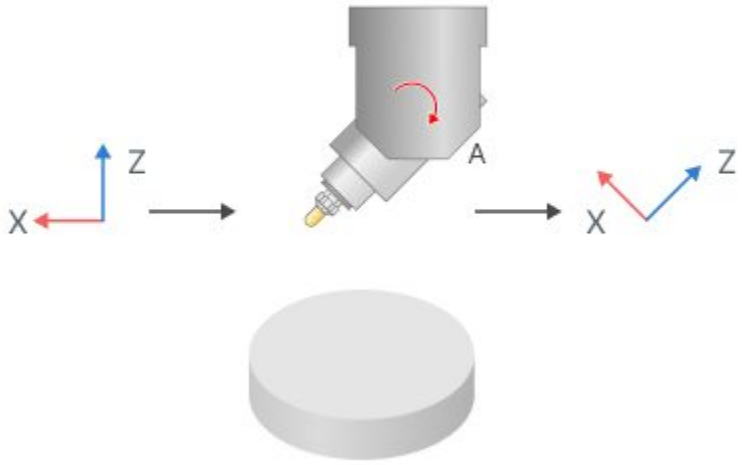
Swap Tool and Workpiece frames. In some situations, the robot's tool block holds the workpiece instead of the tool. In such cases, swap the tool and workpiece coordinate systems. Set the flag if the robot controller does not support this transformation.

Set tool contact surface normal vectors. For five-axis tool radius compensation, the tool must offset along the normal to the surface from their contact point. When setting the flag, CLData outputs an additional TLContact command before each movement, specifying the parameters of the normal to the surface at the tool-workpiece contact point. Details

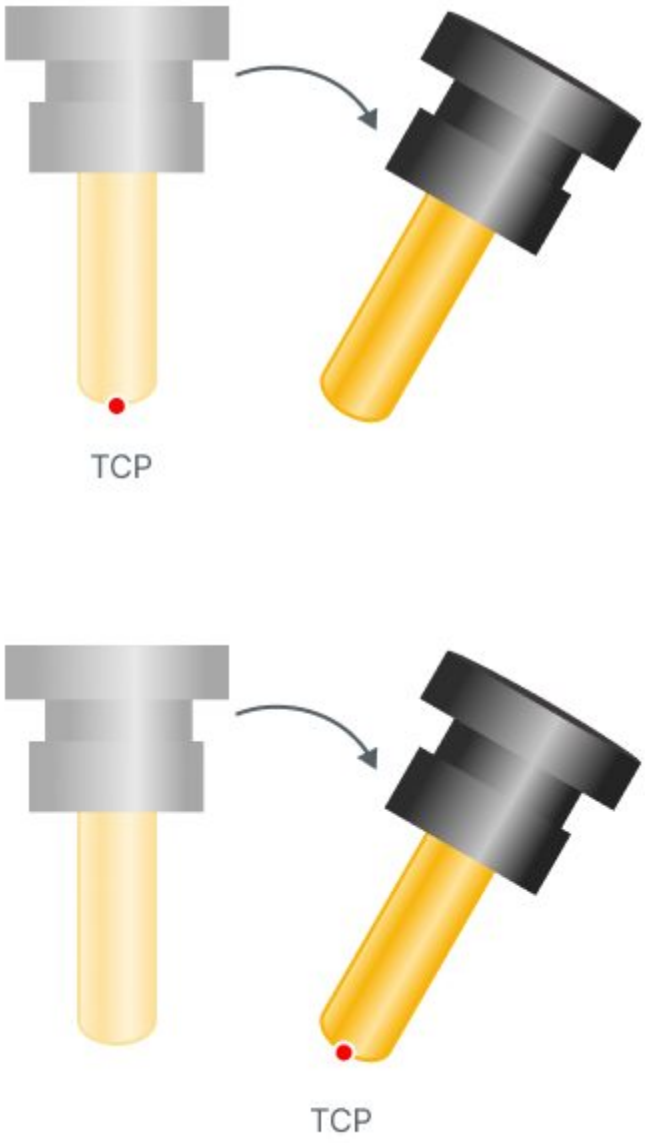
Prefer GOTO for linear movements. Prefer to form a GOTO command in CLData for movements instead of MULTIGOTO, if it contains only movements along three geometric linear coordinates X, Y, Z.

Indexed 5-axis machining. Specifies the accessible parameters and coordinate transformation modes for indexed five-axis machining.

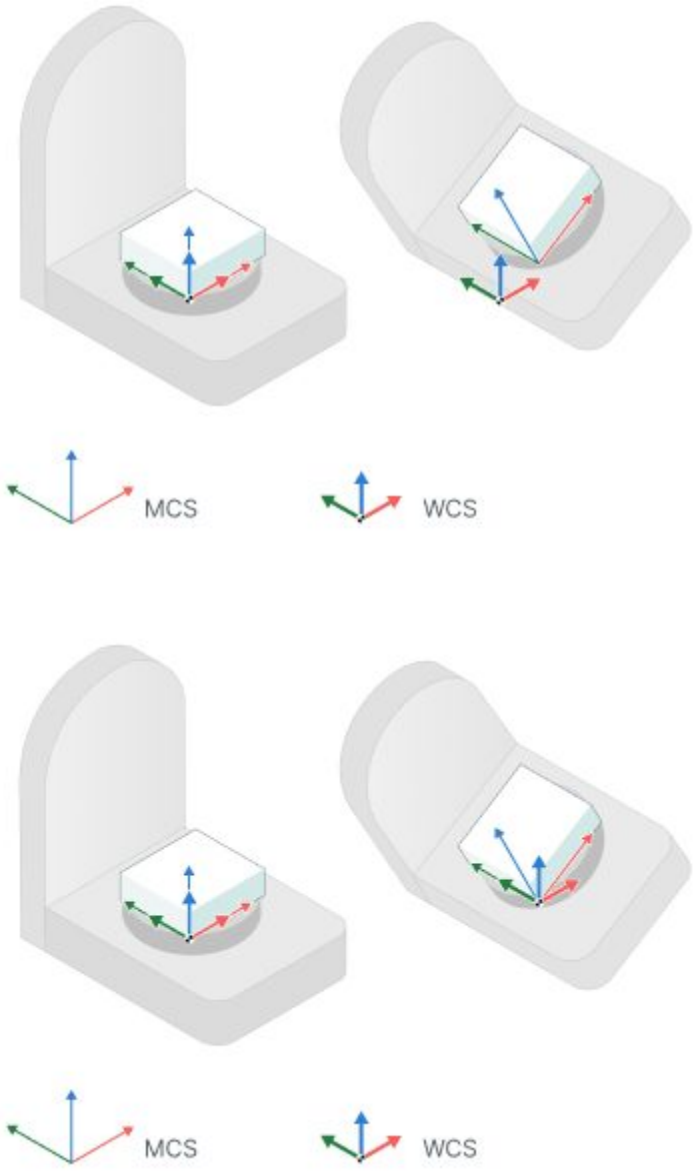
Indexed 5Axis compensation mode. Enables coordinate rotary transformation in the indexed five-axis processing mode (for inclined plane functions). When these options are activated, the coordinate system is automatically rotated (recalculated) to follow any rotary-axis movement of the machine, so that programming stays aligned with the inclined working plane. These compensation modes are used when the machine control system does not support built-in tilted-working-plane functions (such as Siemens CYCLE800, Heidenhain PLANE/Cycle 19, or Fanuc G68.2), nor rotational tool-center-point control (TCPM/RTCP). In such cases the coordinate transformation cannot be performed by the CNC and must be calculated by the CAM system and embedded into the output program by the postprocessor. If the control already provides these functions, the transformation is handled by the machine and these options are not required.



5 Axis tooling point compensation. Option is responsible for rotation of the tool center point together with the rotary head.



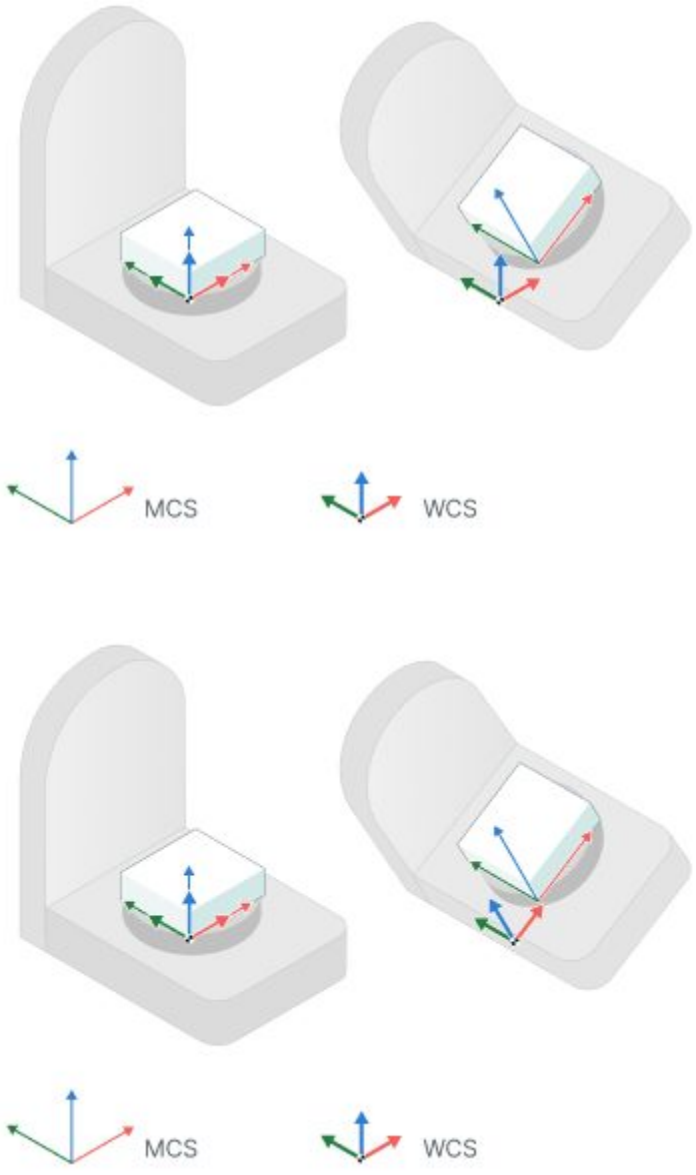
5 Axis workpiece zero point compensation. Workpiece zero point is rotated together with rotary table. Only origin position follows table, while axes orientation stays aligned with machine (MCS).



MCS - Machine Coordinate System

WCS - Work Coordinate System

5 Axis coordinate system compensation. Coordinate system orientation is rotated together with rotary table. Origin stays in place, only axes orientation follows table rotation.

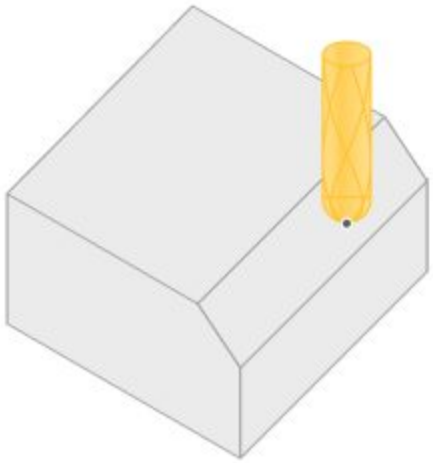


MCS - Machine Coordinate System

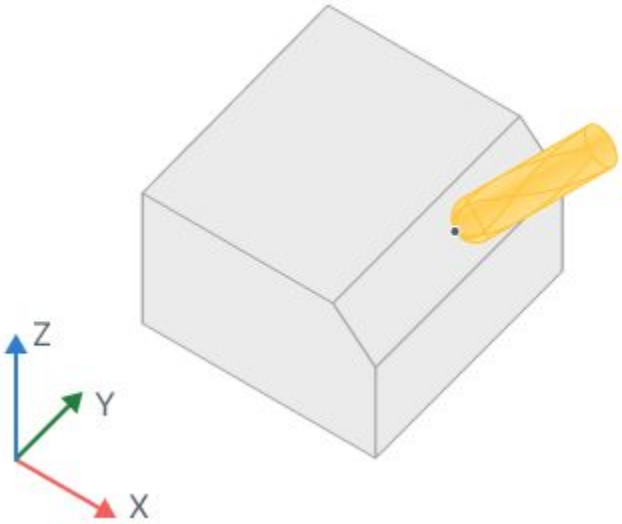
WCS - Work Coordinate System

Local coordinate system (ORIGIN). Defines the display mode of the Local CS parameter in each operation's Setup tab.

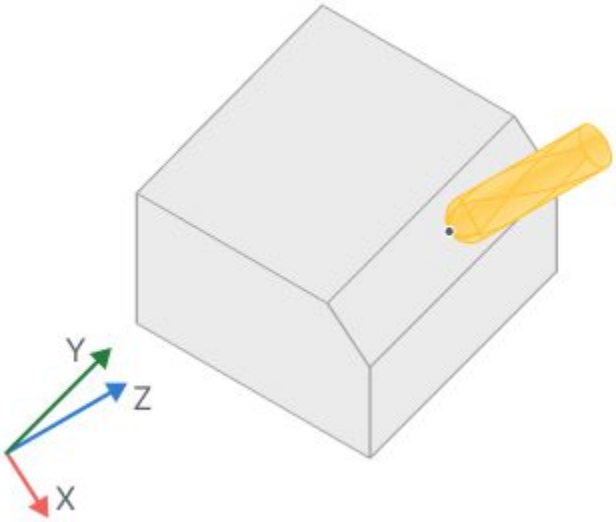
Unavailable. The Local CS parameter does not display.



Off. The Local CS parameter displays but defaults to Off.

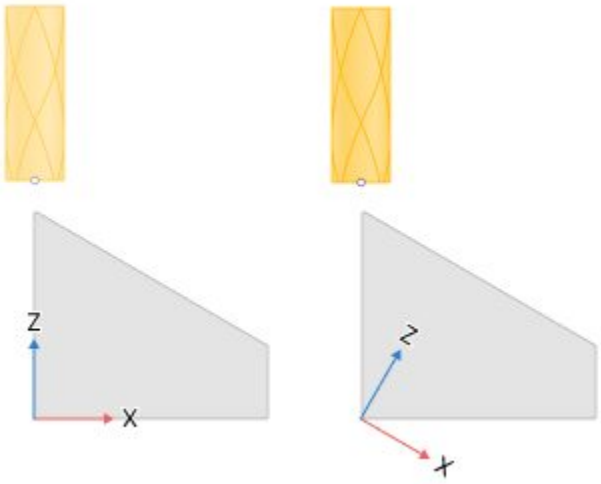


Auto: The Local CS parameter displays and defaults to Auto. In this mode, rotational transformations align the WCS so that the Z-axis matches the tool axis, and its origin remains fixed to the user-selected part point.

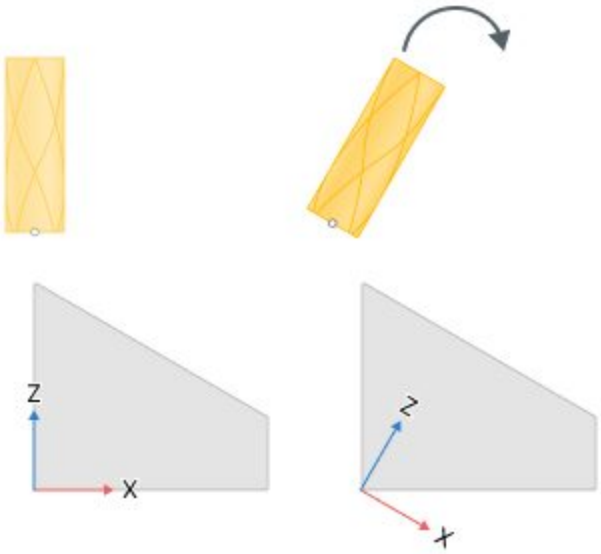


Local CS positioning mode. Defines the behavior of the machine's physical axes during rotational transformations.

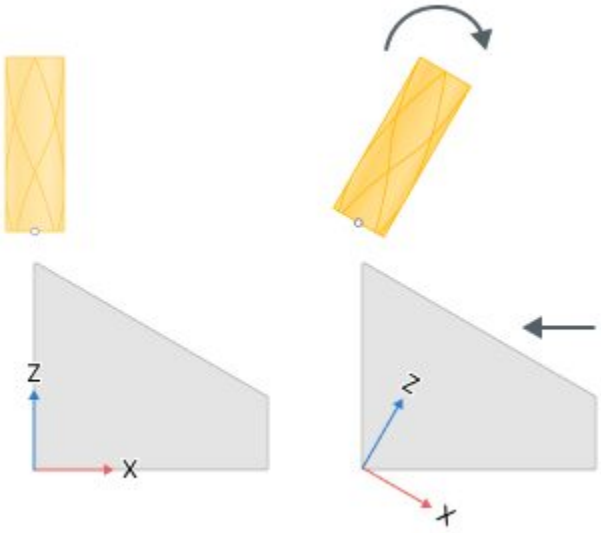
Stay. Rotates the coordinate system without moving the machine's physical axes.



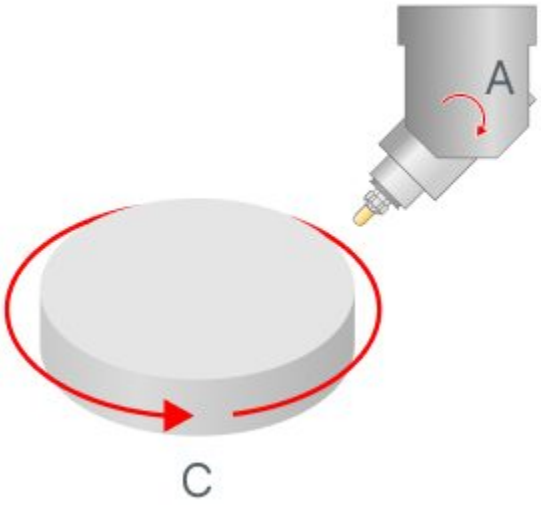
Turn. Rotates the coordinate system and the physical axes of the machine accordingly, without compensating for the misalignment of the tool's set point and the coordinate system origin with linear axes movement.

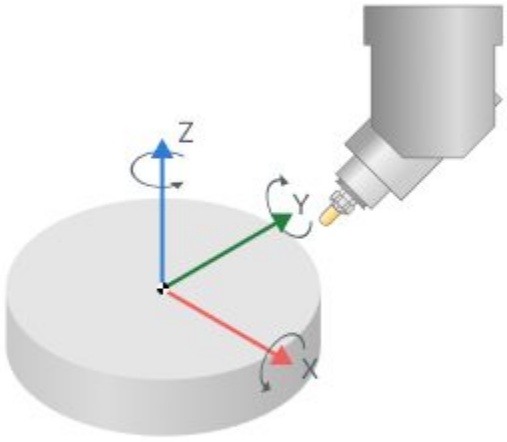


Move. Rotates the coordinate system and the physical axes of the machine accordingly, and compensates for the misalignment of the tool's set point and the coordinate system origin with linear axes movement.



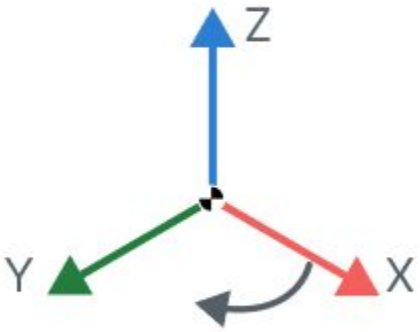
Is spatial. Defines whether to perform coordinate system transformations using spatial angles or the machine's rotational axes. When the flag is set, the transformation uses spatial angles.

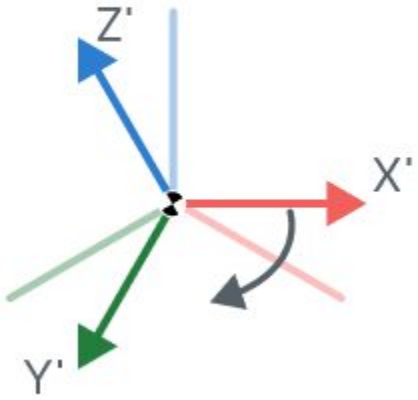




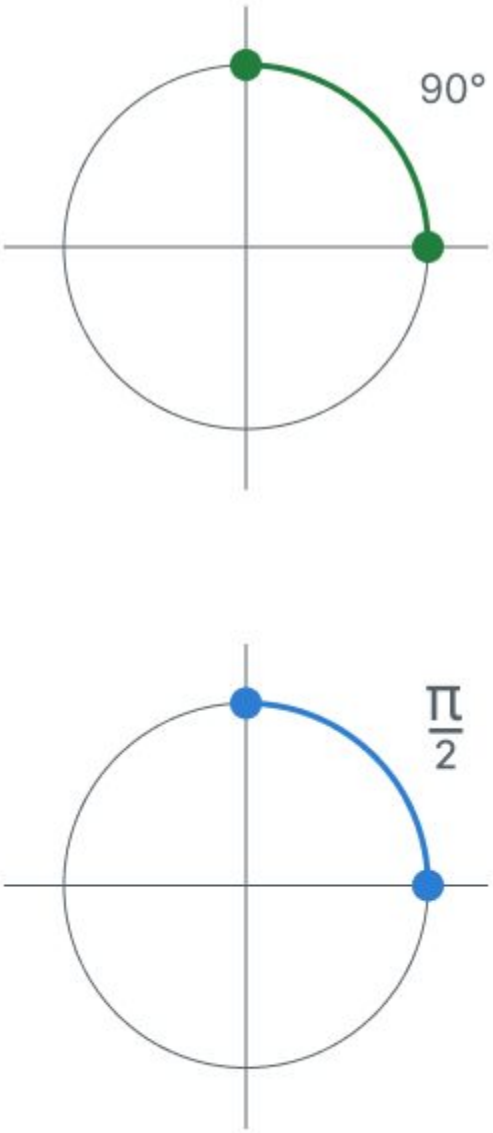
Euler angles type (rotation sequence). Specifies the sequence of rotations when performing rotational transformations using spatial angles.

Rotations around movable axes. Determines whether to perform rotations around fixed axes or previously rotated axes. When the flag is set, each subsequent rotation uses the previously rotated coordinate system axes.

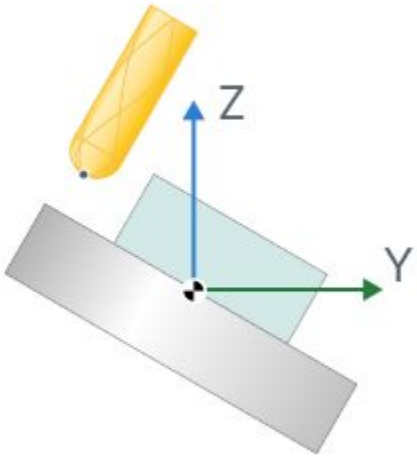


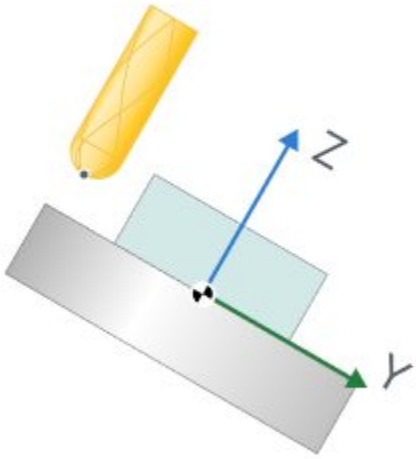


Angles in degrees. Specifies the unit of measurement for rotation angles. When the flag is set, the system measures angles in degrees; otherwise, it measures in radians.



Move auto LCS with table. Determines the position of the WCS during rotational transformations in Auto Local CS mode. When the flag is set, the WCS attaches to the part during rotational transformations, aligning the Z-axis of the WCS with the tool axis. When the flag is disabled, the WCS remains in its original position and does not move with the workpiece.





Auto LCS rotation law. Defines the rule how to locate the Local CS when Local CS option has the value Auto for mill operations.

Snap to Tool CS. Use it in machines where rotational axes are realized through tool rotations. In this case, the axes rotate in conjunction with the tool..

Snap to Machine CS. The axes direction connects to the fixed global coordinate system of the machine.

Snap to Workpiece CS. Use it in machines where rotational axes are realized through workpiece rotations. In this case the axes rotate in conjunction with the workpiece.

Exclude Spatial Angle A. In rotational transformations, the work coordinate system's X-axis remains parallel to the machine coordinate system's X-axis.

Exclude Spatial Angle B. In rotational transformations, the work coordinate system's Y-axis remains parallel to the machine coordinate system's Y-axis.

Exclude Spatial Angle C. In rotational transformations, the work coordinate system's Z-axis remains parallel to the machine coordinate system's Z-axis.

Turn Auto LCS rotation law. Defines the rule how to locate the Local CS when Local CS option has the value Auto for turning operations.

Z along turn axis, X along tool. The Z axis of the local coordinate system is located along the turning axis of rotation. The X axis is directed toward the tool..

Z along turn axis, -X along tool. The Z axis of the local coordinate system is located along the turning axis of rotation. The minus X axis is directed toward the tool.

Z along tool, X by snap rule. The Z axis of the local coordinate system is located along the main direction of tool (along its axis). The X and Y axes are located according to the Auto LCS rotation law rule.

Rotatable Workpiece CS. When the flag is set, you can both shift and rotate the workpice coordinate system in the WCS parameter in the Setup tab.

Continuous 5-axis machining. Specifies the accessible parameters and coordinate transformation modes for continuous five-axis machining.

TCPM mode is available. Enables the Tool center point management parameter in the Setup tab when the flag is active. Use it only when the CNC system supports this function. When the flag is disabled, the system uses coordinate transformation parameters from the Indexed 5Axis compensation mode.

TCPM mode default state. Activates the Tool center point management parameter by default in the Setup tab when the flag is enabled. When the flag is disabled, this parameter is inactive by default.

Disable TCPM when rotate back. When the flag is set, you can access the option to disable TCPM mode during links while rotating the workpiece back to its original position.

TCPM 5Axis compensation mode. Defines coordinate transformation parameters for each block of the NC program in continuous 5-axis machining mode.

5 Axis tooling point compensation. Option is responsible for rotation of the tool center point together with the rotary head.

5 Axis workpiece zero point compensation. Option is responsible for rotation of the workpiece zero point rotation tothether with the rotary table.

5 Axis coordinate system compensation. Option is responsible for rotation the workpiece coordinate system together with the rotary table.

Tool Change. Tool change system behavior configuration parameters.

▼ Click here to expand..
Go to tool change position. The setting defines the necessity of retracting to the tool change location.

Only if tool change is needed. The system retracts to the tool change location only if the next operation requires a different tool. No retraction occurs at operation end if the same tool is used.

Always in the end of operation. Retraction to the tool change station occurs irrespective of the tool used in the subsequent operation — whether it's a new tool or the same one.

Never. Tool change position retractions will be omitted

Output mode. The address configuration parameter specifies how tool change retraction will be output in the control program.

Tool tip coordinates. The defined tool tip coordinates in relation to the workpiece coordinate system will be included in the NC program as the tool change point.

Reference point (ISO G28). The tool change retraction is performed by specifying a movement to the reference point (G28 command).

Machine coordinates (ISO G53). G53 function is used to perform retraction movements to the tool change position in the machine coordinate system.

Tool change position. Tool change position coordinates are defined here.

Heavy axes. Addresses of axes to be avoided during tool change approach/retraction are defined. The parameter is utilized for automatic approach/retraction calculation to the tool change position under "Collision Avoidance" rule. When enabled, the CAM system optimizes movements to prevent collisions using different machine axes. Heavy axes (those undesirable to move) are listed in this section.

Tool change time calculation. This setting defines the method for calculating tool change duration in the CAM system:

Default law. Project's default calculation methodology rule.

Undefined. The CAM system's tool change time is preset in the algorithm and not subject to modification.

Constant time. The tool change duration is set manually via the Constant Time parameter value.

Advanced laws. Multiple calculation rules can be activated via this parameter, appearing as a dropdown menu. The specified number determines how many rules will be enabled, and each list entry can utilize any of the previously described rules.

Rapid movements time calculation method. Method for calculating the time of rapid (non-cutting) tool movements.

Maximum axis movement time (default). The system calculates the travel time for each machine axis (X, Y, Z, and rotary axes if applicable). The rapid move time is determined by the axis that takes the longest time to complete its motion.

Time of tool tip movement. Calculates the rapid movement time based on the actual path of the tool tip, considering the vector sum of speeds across all axes (X, Y, Z).

Coolants.

Set of parameters managing the usage of coolants in the project

▼ Click here to expand..
The following parameters are available for all types of coolants:

ID. Symbolic representation of coolant delivery mode.

Tube name. Title of cutting fluid delivery system.

Icon file. Link to the pipeline model file.

Switch time. Time necessary to activate the coolant delivery system.

Simulation.

Set of parameters managing system performance in project simulation.

▼ Click here to expand..
Some parameter group works similarly to the Setup panel.

Simulation Method. Determines visualization mode in simulation environment.

Solid. Solid modeling technique utilizing mesh representation for 3D model visualization. Characterized by higher computational demands but provides enhanced quality in rendering machined surfaces.

Voxel5d. Point voxels are used, improving rendering of complex volumes but requiring significant resources. They are applicable for both 3-axis operations and turning operations, as well as operations with a large number of axes.

Voxel3d. Column voxels are used. They accelerate the modeling process but can only be applied to 3-axis operations.

Model resolution. Rendering resolution for model simulation. It defines the number of voxels used to divide the workpiece model in simulation. Higher resolution results in improved model rendering quality.

Gouge detection tolerance. Minimum penetration value for a gouge classification. Penetrations less than this threshold are disregarded and no overcut warnings are generated.

Revolution bodies simulation. This parameter is used in turning projects and is responsible for rendering, for example, lathe chuck jaws as a single rotating body. When enabled, the chuck jaws form a single rotating body during simulation. Collisions with other machine components are checked against this body. Note that using this parameter slows down the simulation process.

Check inappropriate tool and spindle direction. This setting controls the validation of spindle rotation direction. It ensures that the spindle's rotational direction is compatible with the cutting action of the milling tool (such as matching a right-hand cutter with clockwise spindle rotation) and verifies proper alignment in turning processes regarding the tool placement..

Collisions to ignore. The setting specifies machine component pairs that will be excluded from collision detection during simulation modeling. Adding these components to the ignore list helps avoid false collision alerts from the system. How to add through Machine Maker you can read, for example, for Robot Details.

Show active part only. By enabling this option, you will only see the current part. In a project with multiple parts, the current part is determined by the focus in the tree, i.e., on a specific part or operation intended for processing that part.This allows reducing the load on your PC when working in different modes (such as Technology or Modeling).

System settings.

▼ Click here to expand..
Rotation angles (rotation sequence). This parameter defines the method for executing coordinate axis rotation angles when applying local coordinate systems.

Rotations sequence. The setting determines the order of executing rotations about the coordinate axes.

Rotations around movable axes. This parameter specifies whether rotations through spatial angles will occur around fixed axes or pre-rotated axes. With the flag activated, every successive rotation will take place around the axis of the previously rotated coordinate system.

Angles in degrees. This setting configures the angular measurement unit to degrees.

Machine state parameters.

This section contains groups that define the values of individual axis parameters. The checkbox in the axis designation determines whether it will be displayed in the list of axes on the operations Setup tab.

▼ Click here to expand..
Address. The axis address that is output to the NC program.

Axis Control. Specifies the motion control mode for this axis.

Countinuous. This axis operates in continuous mode.

Indexed. This axis performs movement through incremental steps.

Manual. Modification of this axis's coordinate is allowed only via manual operation.

Force control with map. Path determination for this axis movement may be processed through the Axes map. Details.

Group. Axis affiliation to the motion type group. There are Rotary axes, Linear axes, and Others.

Increment. The least movable increment along this axis which the machine drive system can handle.

Min. Lowest coordinate limit for this axis.

Max. Highest coordinate limit for this axis.

Initial value. The default coordinate location for the component.

Design time value. The shift amount that adjusts the Min-Max travel range of this axis.

Axis has brake. This parameter, when activated, triggers the inclusion of brake control commands in CLDATA.

Priority. This number affects the ordering of the axis in the axes list under Machine State Parameters. The smaller the number, the higher the axis appears in the list.

Order. This is the ordinal number that identifies the axis position within the Machine State Parameters list.

Support shortest path rotation. This parameter defines whether the rotary axis supports rotation using the shortest path between initial and final positions in 3D space.

Frame.

This subsection describes the machine components tree, detailing the order in which machine components are connected to each other. Each component includes the name and axis identifier for the movable component's movement.

▼ Click here to expand..
Visible. This controls the visualization parameters of the node. When the flag is enabled, the node becomes visible, allowing its color to be changed in this state.

Metallic. Setting the flag gives the surfaces of the tool block a metallic gloss.

Transparent. With the flag set, the tool block becomes transparent. You can adjust the transparency level.

Display mode. The parameter manages the rendering of the tool block's 3D model. The default mode uses the display mode set for the system's graphical window. You can also set a unique display mode for this specific tool block. Details.

Channel. The channel within the NC system that controls the axis connected to this particular movable component.

Workpiece index. This axis is assigned to work only with the workpiece indicated by the designated number.

Rapid feed. This parameter defines the rapid traverse speed for this axis: mm/min for linear axes and degrees/min for rotary axes.

Scale. This option sets the relative size or scale for visualizing the component.

Inverse rotation direction. Reverse spindle rotation direction to the opposite.

ID of rotary axis that is used as spindle. This designation marks the rotary axis configured to operate as the spindle in workpiece or tool mounting units.

Default clamp ID. This unit reference marks the specific mounting station used for workpiece or tool clamping.

Supported applications. Available tool categories for this specific mounting unit.

Sub machines.

Submachine is a list of parameters for the `tool holder, workpiece holder` pair. This is especially topical for complex machines with multiple spindles, multiple places for the workpiece, etc. Details.

Default reordering mode.

The parameters of this group define the capability to reorder the operation list sequence.Details.

▼ Click here to expand..
No reordering. Reordering functionality for the operation list is disabled.

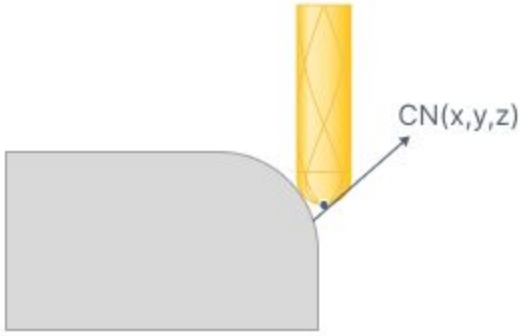
Minimize tool changes. If the project includes more than one part, the actual machining sequence can be redefined to minimize tool changes.

Swiss lathe (takeover last). If this mode is activated then the operations of the second part are placed in the beginning of the reordered list.

Machine dimensions.

This set of parameters establishes dimensional links in machine nodes and between components of the robotic manufacturing system (offsets, etc.). Initial values are set in MachineMaker, with redefinition allowed in the kinematic configuration file.

3D Tool Compensation

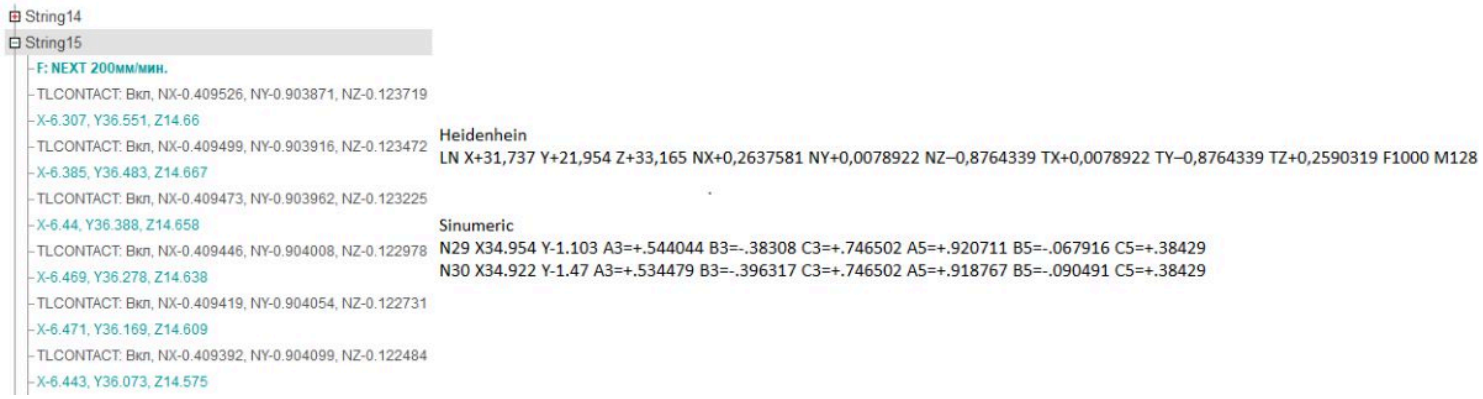


Application area:

For surface finishing operations, the G-code output may include the surface normal vector at the tool contact point. This enables the implementation of 3D tool compensation — a controlled displacement of the cutting tool relative to the workpiece surface at each point along the toolpath.

To include surface normal vector data in the G-code output, enable the Set tool contact surface normal vectors (on Machine setup panel Details) flag in the machine schema settings. This generates a TLCONTACT (NX, NY, NZ) command in the CLData for each toolpath point. The postprocessor must then either:

- output the contact normal directly to the G-code frame, or
- manually shift each path point along this vector by the specified correction amount, depending on the 3D compensation support of the target CNC system.



Setup panel

Application area:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more.

Part properties (available for the Part group). See more

▼ Click here to expand..

Part number. Unique identifier assigned to a machined part (workpiece) in a CAM system.

Workpiece material. Material of the part being machined, selected from a reference catalog in the CAM system. Each entry defines the material's composition, condition (annealed, hardened, etc.), ISO machinability group (P, M, K, N, S, H, O), and hardness (e.g. HB value). The CAM system uses these properties to calculate cutting parameters — speeds, feeds, and tool engagement — for the machining operations. Details



Setup and tooling.

▼ Click here to expand..

Workpiece connector. List of workpiece fixturing points on the machine (fixed table, rotary table, left chuck, etc.) The arrangement of the fixturing systems and their positioning is determined during the construction of the machine's kinematics.

Workpiece setup. Determines the position of the part on the equipment. Details

Base setup. The same as Workpiece setup but for robots.

WCS. Workpiece coordinate system (WCS, G54 – G59) defines the NC-program "zero" on the workpiece. Details

Base. The same as WCS but for robots.

Tool center point management. Involves continuous rotation mode of the workpiece coordinate system along with the workpiece itself. It is utilized for continuous 5-axis machining (provided that the CNC controller supports such a mode).

Local CS. Allows generating a trajectory within a coordinate system additionally offset and rotated concerning the workpiece coordinate system. For instance, a program for machining an inclined pocket concerning the workpiece coordinate system will be much simpler if derived in a local coordinate system aligned with the pocket's inclination. This method is used for positional 5-axis (3+2) machining. When selecting the 'Auto' value, the direction of the Z-axis in the local coordinate system is automatically set along the tool rotation axis, which is defined by parameters in the Tool Orientation group. Details

Tool connector. Determines the interchangeable node for installing the tool.

Tool orientation.

It determines the orientation direction of the tool. Details

This group specifies the primary rotational axes.

Machine limits.

The reduced limits are used when the toolpath and maps are calculated. They define the range within which the axis positions are considered valid, so the resulting motion is constrained to the specified envelope rather than to the full machine travel.

The limits are treated as operation limits: they describe the range acceptable for the current operation and are applied in addition to the global machine limits imported from the machine.

Tool coordinates in WCS

This tab displays geometric coordinate systems. In parentheses, showed the name of the physical axis corresponding to movement along a specific axis.

Linear physical axes

Coordinates values of the linear axes of the machine concerning the physical zero point of that axis (the end sensor of that axis). Here, display all the primary physical coordinates of axes not involved in the Tool coordinates in WCS.

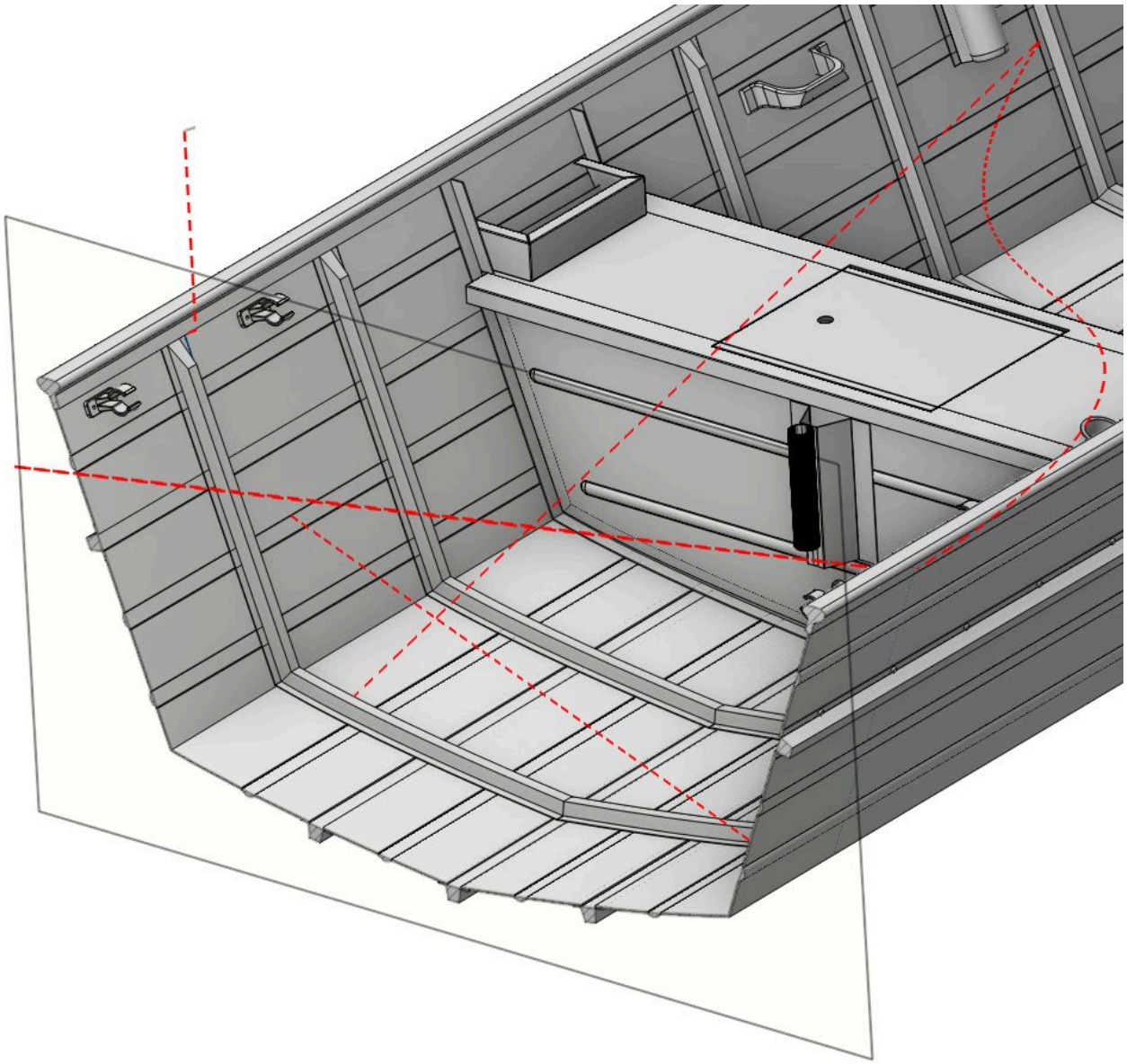
Other axes

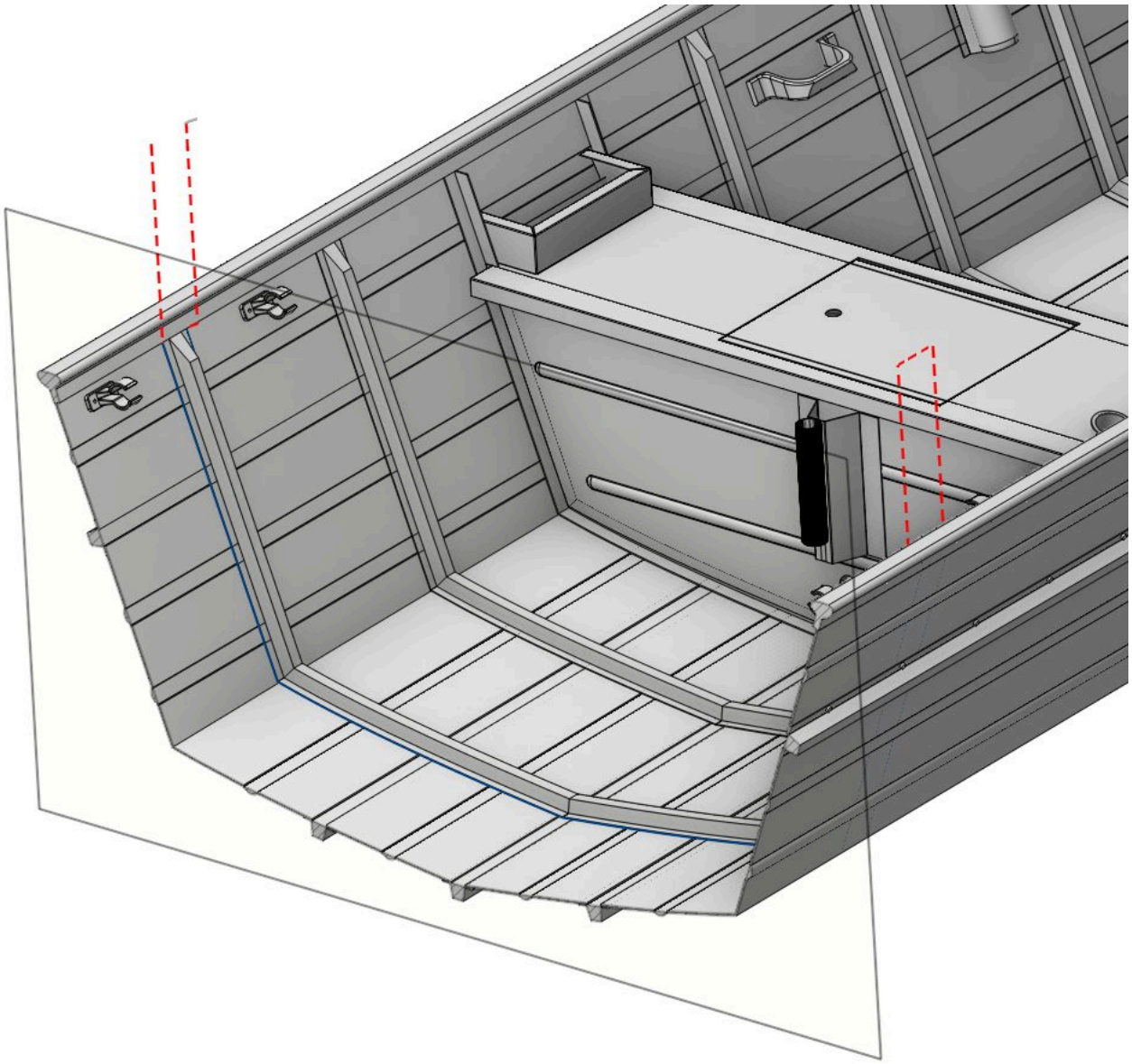
This group specifies the auxiliary axes, such as the axes related to the movement of chuck jaws.

Kinematics.

The Kinematics group defines the kinematic behavior of the equipment while the toolpath is executed — control of excessive degrees of freedom, motion optimization, and selection of the axis configuration. It is used to set up how a multi-axis machine or robot physically travels along the computed path.

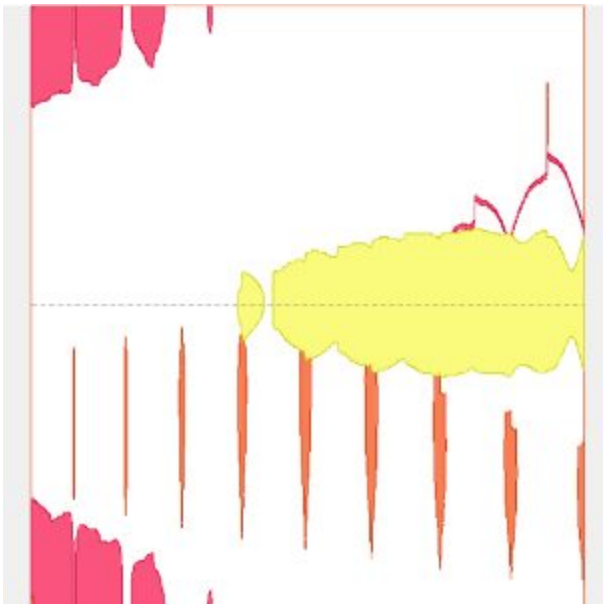
▼ Click here to expand..
Smart Motion. Smart Motion enables robot trajectory generation for 5D and 6D points without requiring a robot axes map. The resulting trajectory stays within axis limits and includes singularity and collision avoidance. Additionally, the trajectory can be fine-tuned using the robot axes map.





Axes map. The Axes Map allows to define manual and fine-tune automatic control laws for the excessive degrees of freedom of a robot (the 6th axis, the rails axes, the rotary table axes). The feature is available by pressing the ellipsis button next to the Axes map parameter. To optimize processing, it's possible to switch between different available axes of the equipment.

Mapping is applicable for robots(Details) and 5-axis machines (Details).



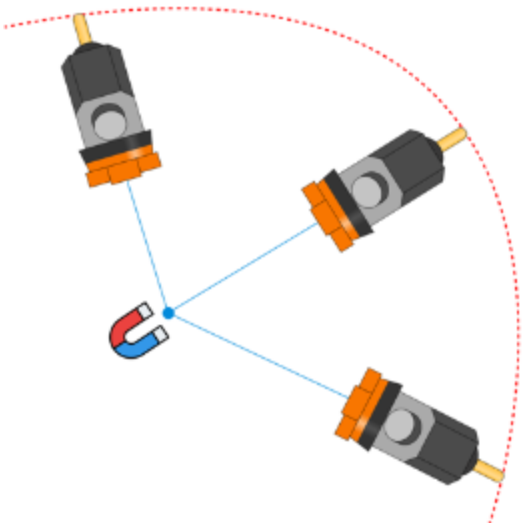
Flips. Joint configuration switching. The J1, J3, and J5 buttons toggle the configuration (posture) of the corresponding joint, selecting an alternative inverse-kinematics solution. They make it possible to reach a different axis arrangement while preserving the tool position and orientation — for example, to avoid a travel limit or a collision.

6th axis control

To position an axial tool (mill, beam, jet), five degrees of freedom are required. A standard industrial robot has six axes. If the tool is not aligned with the rotation axis of the robot's last joint, there are infinite possibilities to position the robot precisely at a particular point by rotating the tool around its own axis. This provides additional flexibility in accessing hard-to-reach areas, helps circumvent various kinematic singularities, and avoids mechanical collisions between equipment components Details

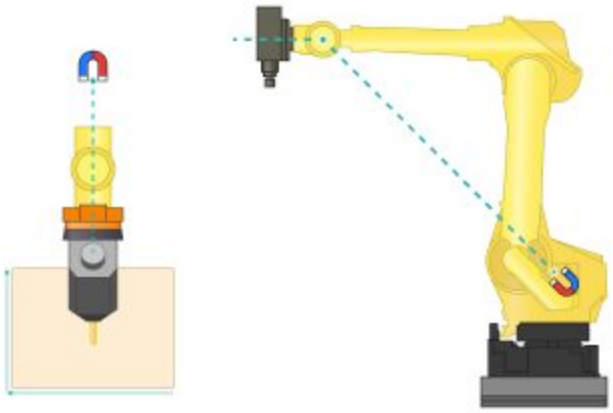
▼ Click here to expand..
Direct to point.

In this mode you specify a 3d point which attracts the point of intersection of the 5th and 6th axes to the minimum distance during the machining.

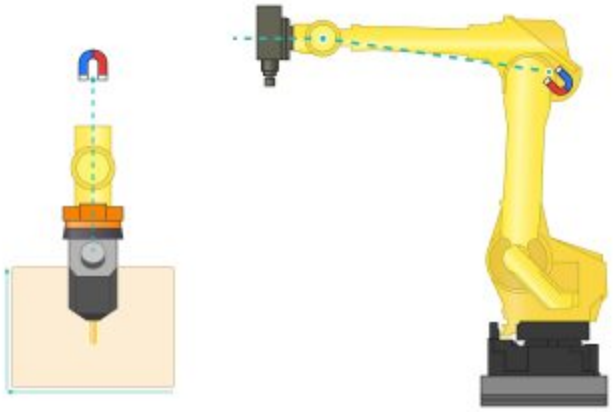


Point.

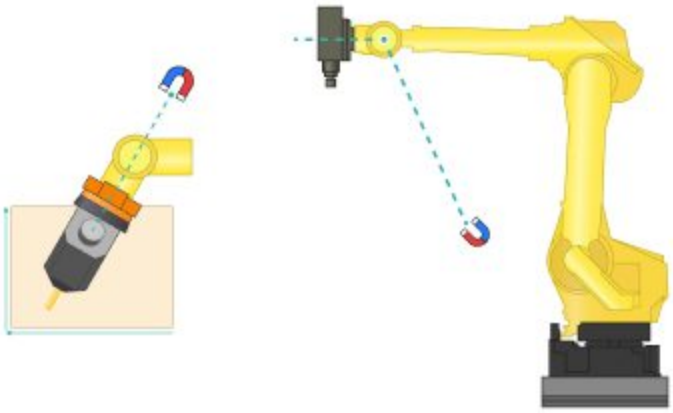
Robot base point. In this mode, the point will be positioned at the robot's home/base point



Robot elbow point. In this mode, the point will be positioned at the robot's elbow point.

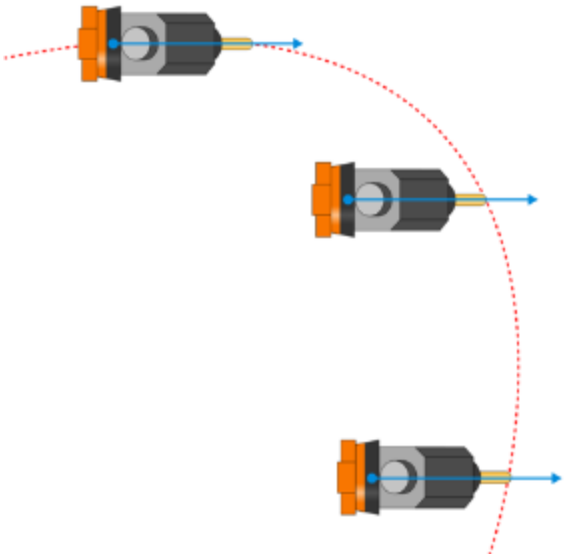


Custom. In this mode, the location of the point will be determined according to the user's instructions.

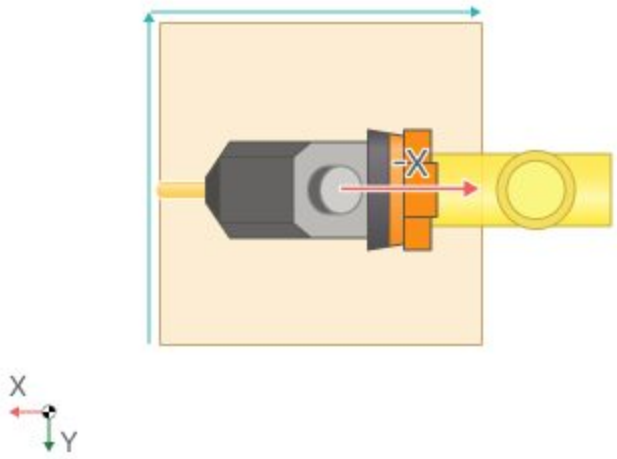


Fixed vector.

In this mode you define the desired direction of the robot flange (6th axis) in the coordinate system of the robot base.

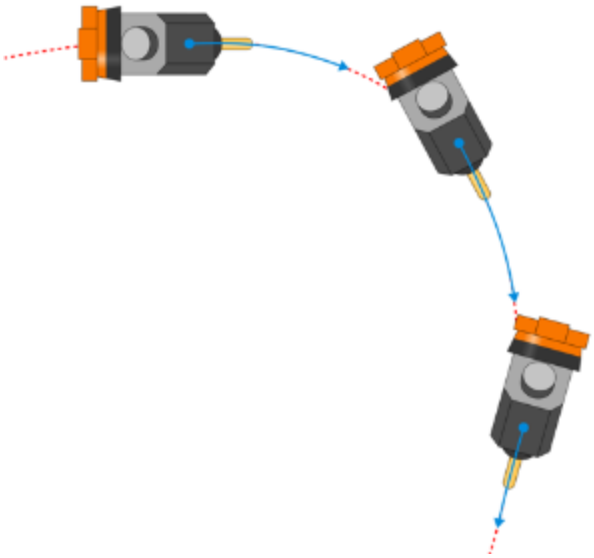


Vector. Vector for the 6th axis of the robot.

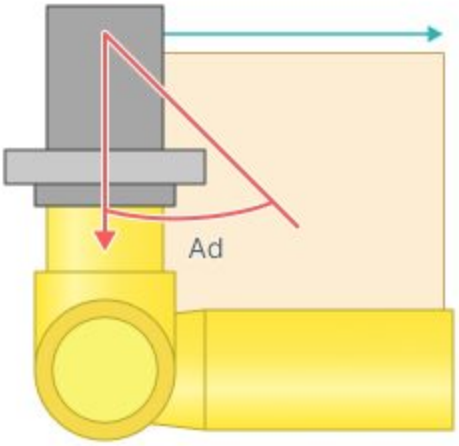


Tool path

In this mode one axis of TCP is aligned with the tool path direction.

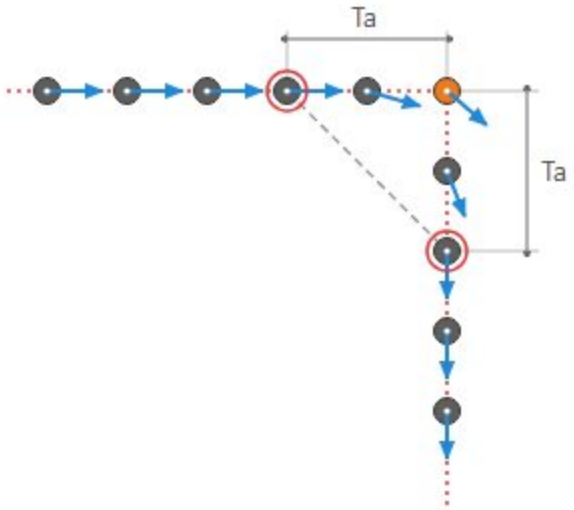


Angular deviation. Defines an additional angle for the 6th axis along the tool.

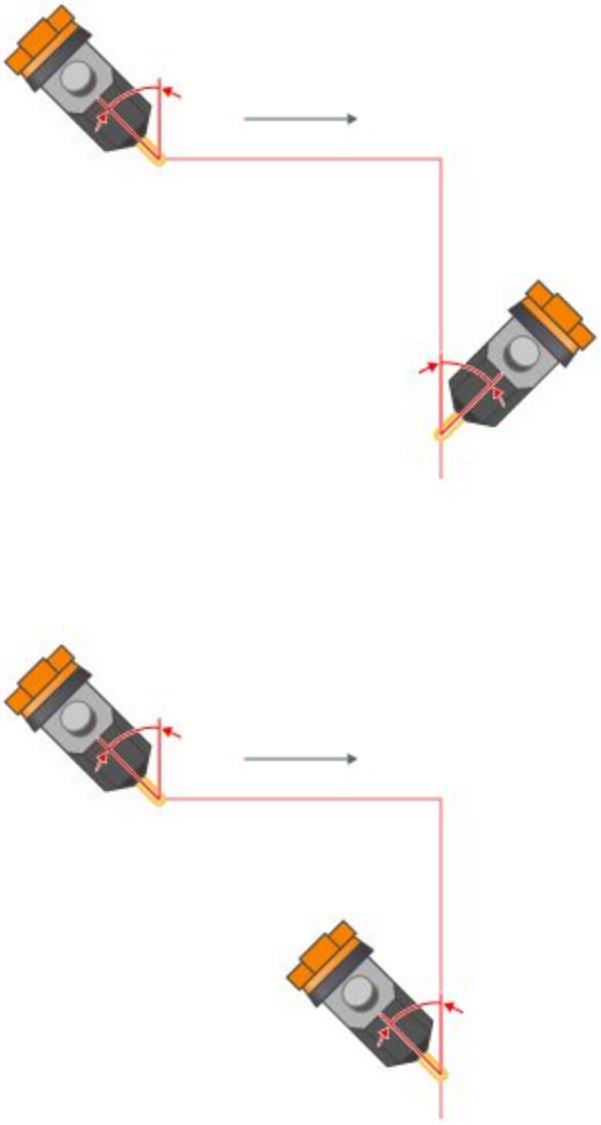


Ad - Angular deviation

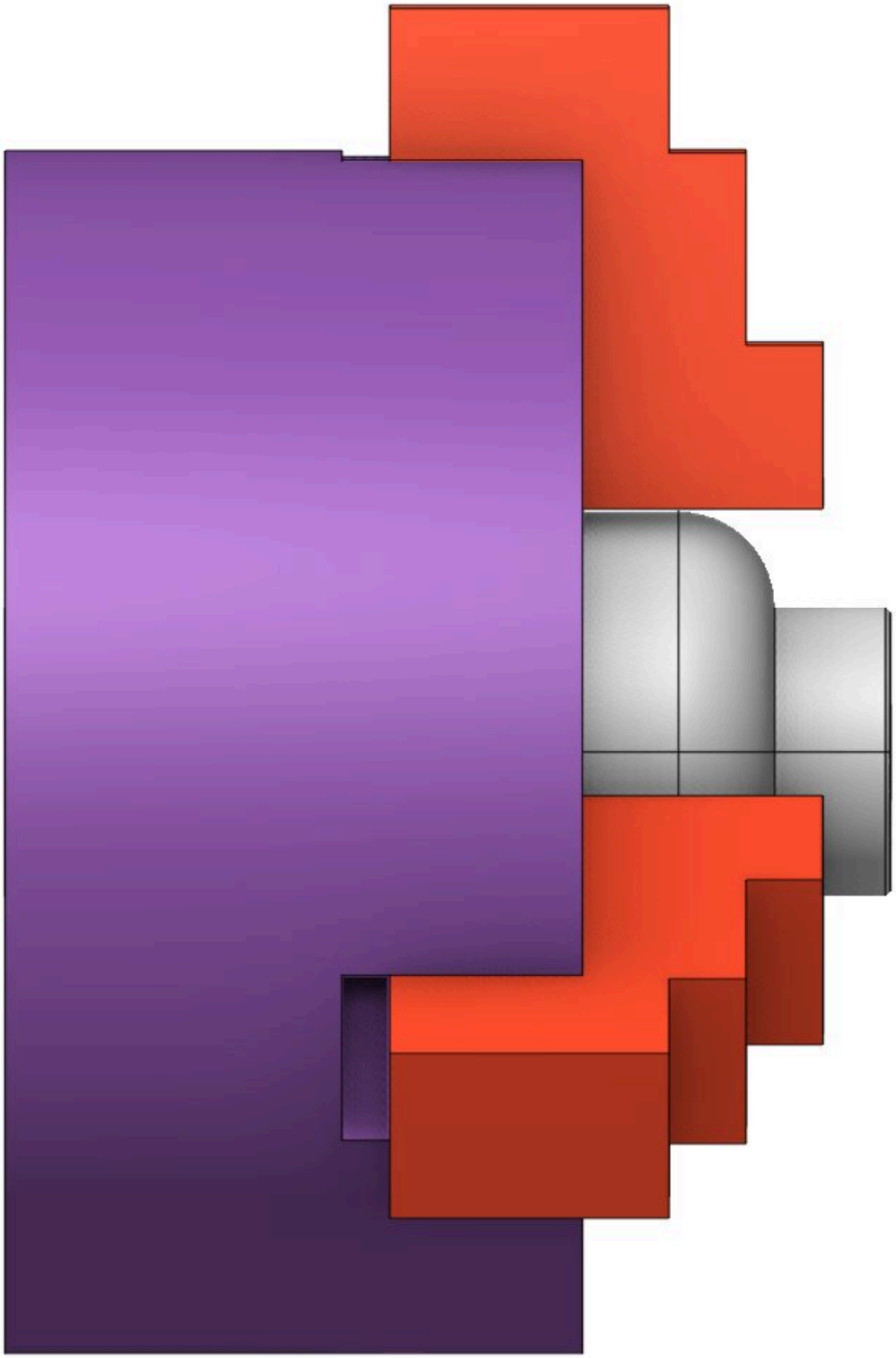
Tangent approximation. Smooths the rotation of the 6th axis in the Tool path mode. The TCP X axis direction is taken not from the tangent at the point itself but from the line between the toolpath points before and after it (Ta in the picture). At sharp corners the 6th axis starts turning before the corner and finishes after it instead of flipping at the vertex. The larger the value, the smoother the rotation, but the more the X axis deviates from the direction of motion on curved sections. The value is set as a distance or as a percentage of the tool diameter; the default is 50% of the diameter. 0 means the exact tangent. On vertical plunges and retracts the X axis keeps the direction of the adjacent cutting section.

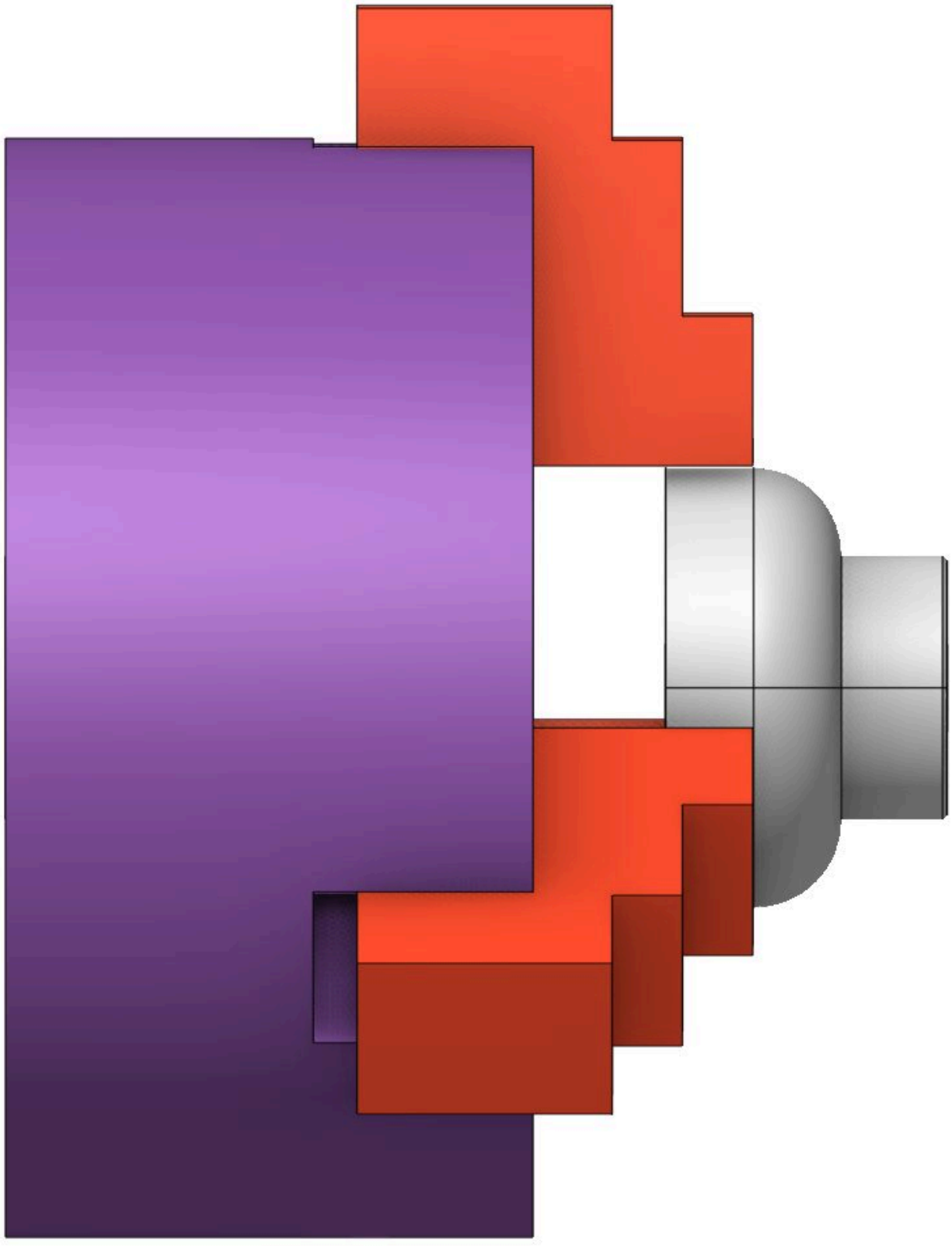


Fixed lead direction. Fixes the movement of the tool at the selected angle in map (Lean angle, Lead agle) relative to the selected global coordinate system.



Positioning of part at machine (Workpiece setup)





Application area:

The position of the part at the machine is determined by the Workpiece Setup parameter in the Setup panel (Details).

Principles of working with a window.

To change the workpiece setup just click on the ellipses button next to the parameter. The workpiece setup dialog should appear.

In this dialog you can set the Geometry CS of the workpiece and the additional workpiece transformations composed of the translation along the machine X, Y, Z axes and the rotation around the same machine axes.

The Geometry CS determines the initial setup: the workpiece is attached to the workpiece connector in such a way the Geometry CS of the workpiece coincides with the joint LCS of the workpiece connector determined by the Machine CS parameter.

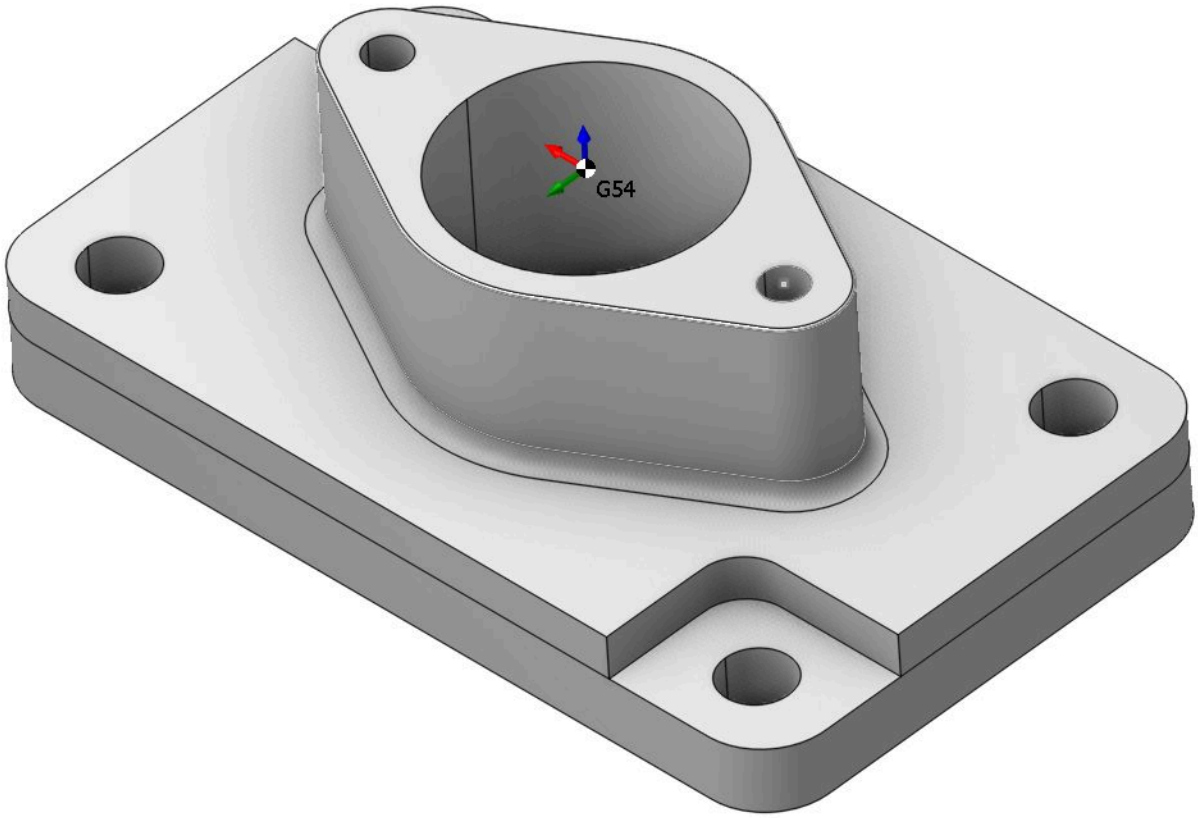
The Offset option is used to shift the workpiece along the XYZ axes.

The Rotation option is used to rotate the workpiece around the XYZ axes. The way of rotations depend on machine schema settings.

The most easiest way to change the workpiece location is to use the mouse wheel to change the values of the Translate and Rotate boxes. The workpiece position is immediately updated in the graphic view after each parameter change.

Another convenient way is to interactively drag and drop a workpiece in the graphics window. Grab the highlighted anchor points and drag or rotate in the desired direction until you reach the desired position. Or click on the square on the desired side to put the workpiece with this side on the table (machine connector's up side).

Workpiece coordinate system (G54 - G59)



Application area:

Workpiece coordinate system (WCS, G54 – G59) defines the NC-program "zero" on the workpiece. You can always see the Workpiece CS on the screen while working in the Technology and the Simulation modes.

Principles of working with a window.

There are two ways you can define the Workpiece CS. You can select the Workpiece CS from the drop down list and you can set the Workpiece CS in the interactive mode by clicking on the ellipses button.

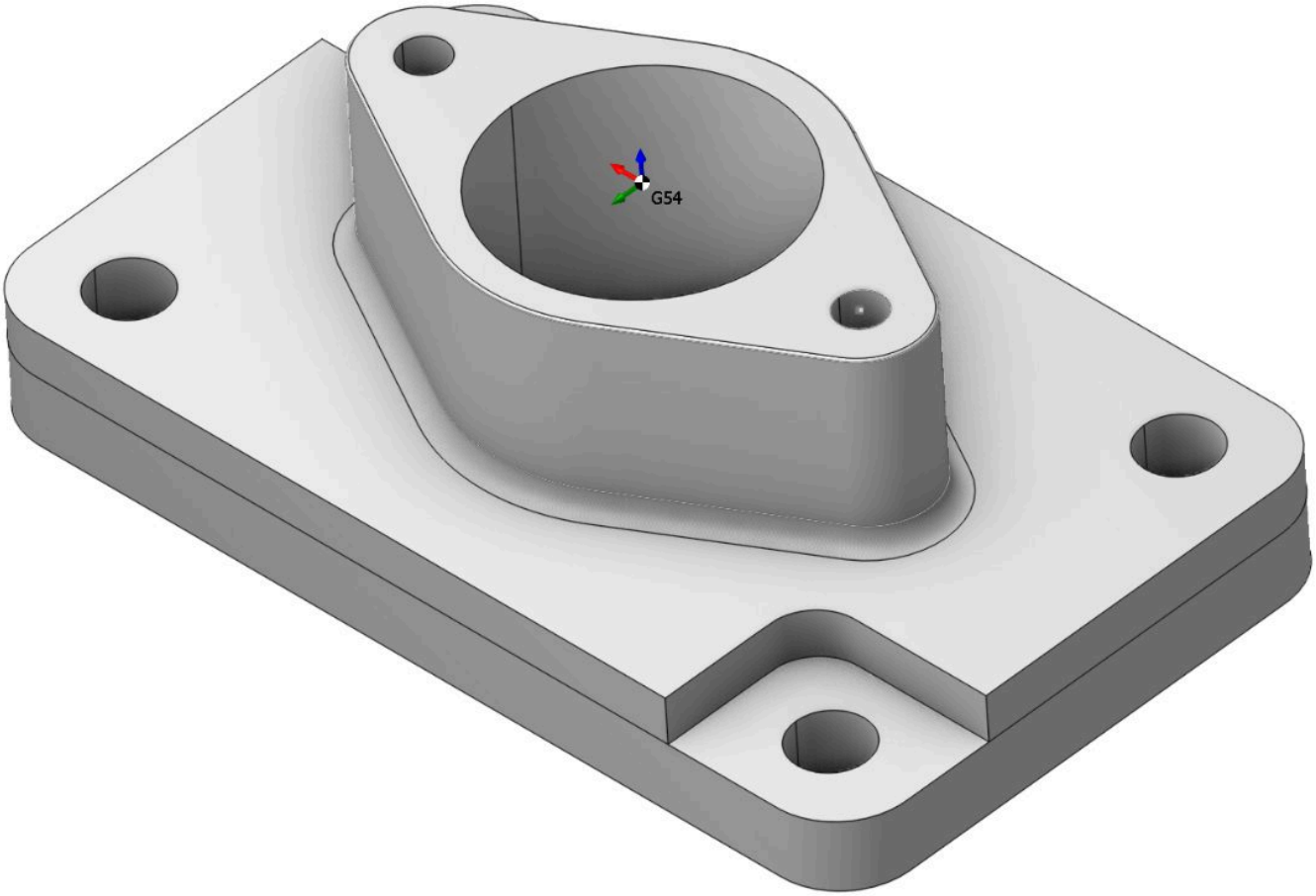
The drop down list contains the following options.

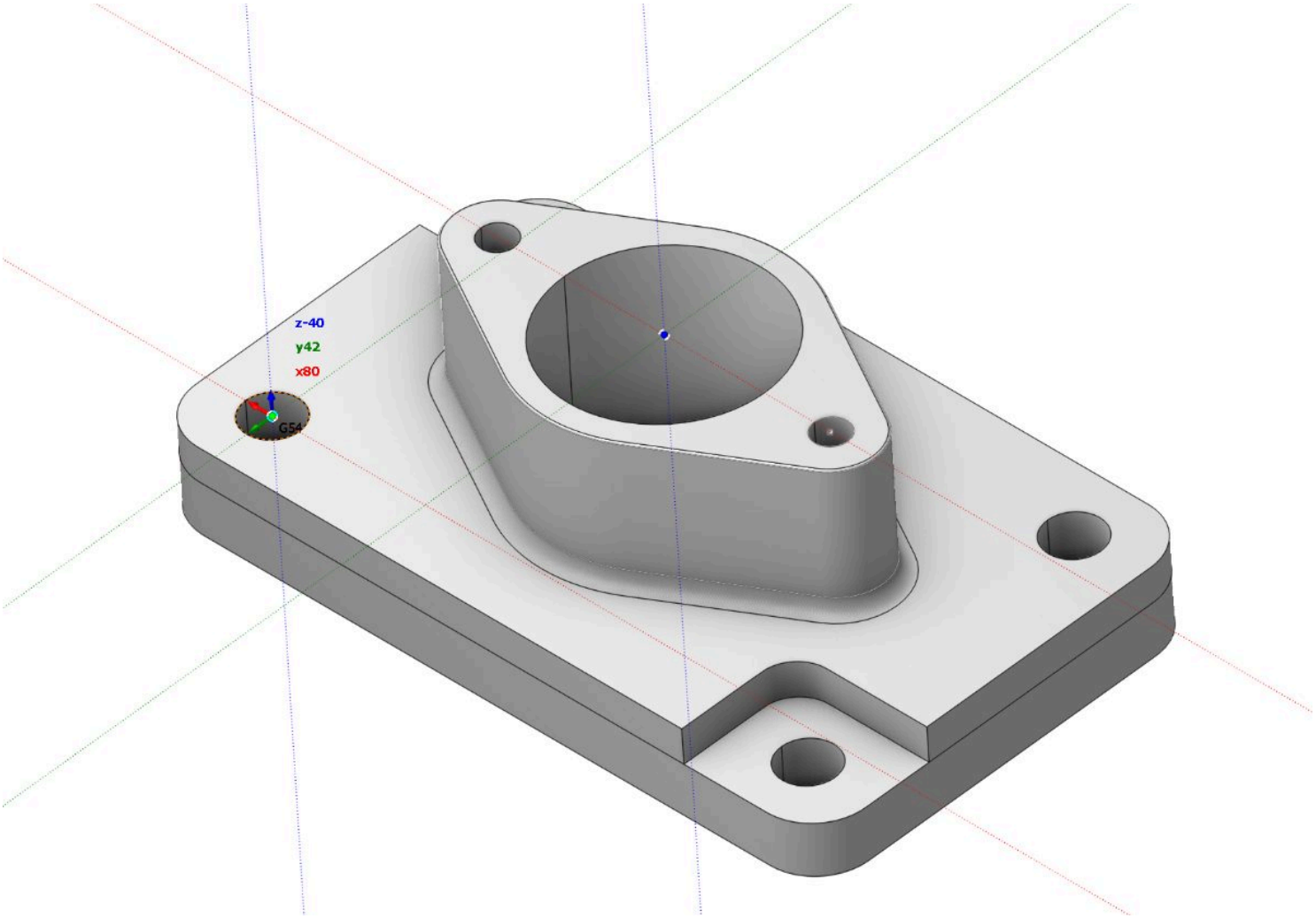
1. Off - the workpiece coordinate system is disabled. It means the workpiece zero coincides with the machine zero point G53.
2. From previous - the workpiece coordinate system is taken from the previous operation.

3. The list of geometrical coordinate systems of the project. You can use a geometrical coordinate system as the workpiece coordinate system. In this case only the origin of the geometrical coordinate system is used, the axes orientation of a geometrical coordinate system play no role for the Workpiece CS definition.

You can specify the WCS in the interactive mode by clicking on the ellipses button next to the WCS parameter.

In the interactive mode you can change the position of the WCS origin in the graphic view using the standard drag&drop technique. Just hover the mouse pointer over the WCS on the screen. It should become highlighted. Than click on it with the left button to start dragging. After that you can change the location of the coordinate system by snapping either to the part or to the workpiece geometry. In the interactive mode you can also change the WCS number (54-59 etc.).

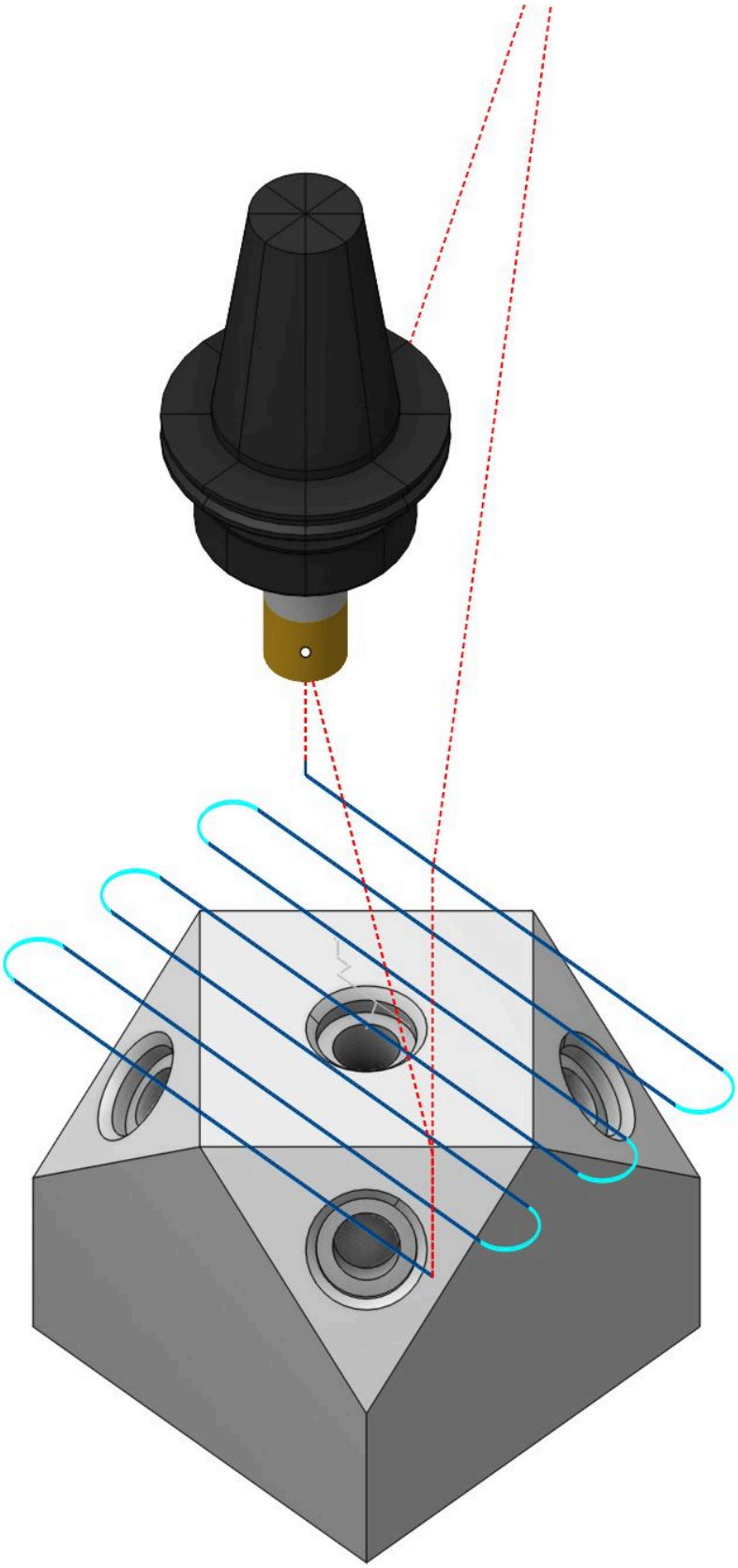

G54

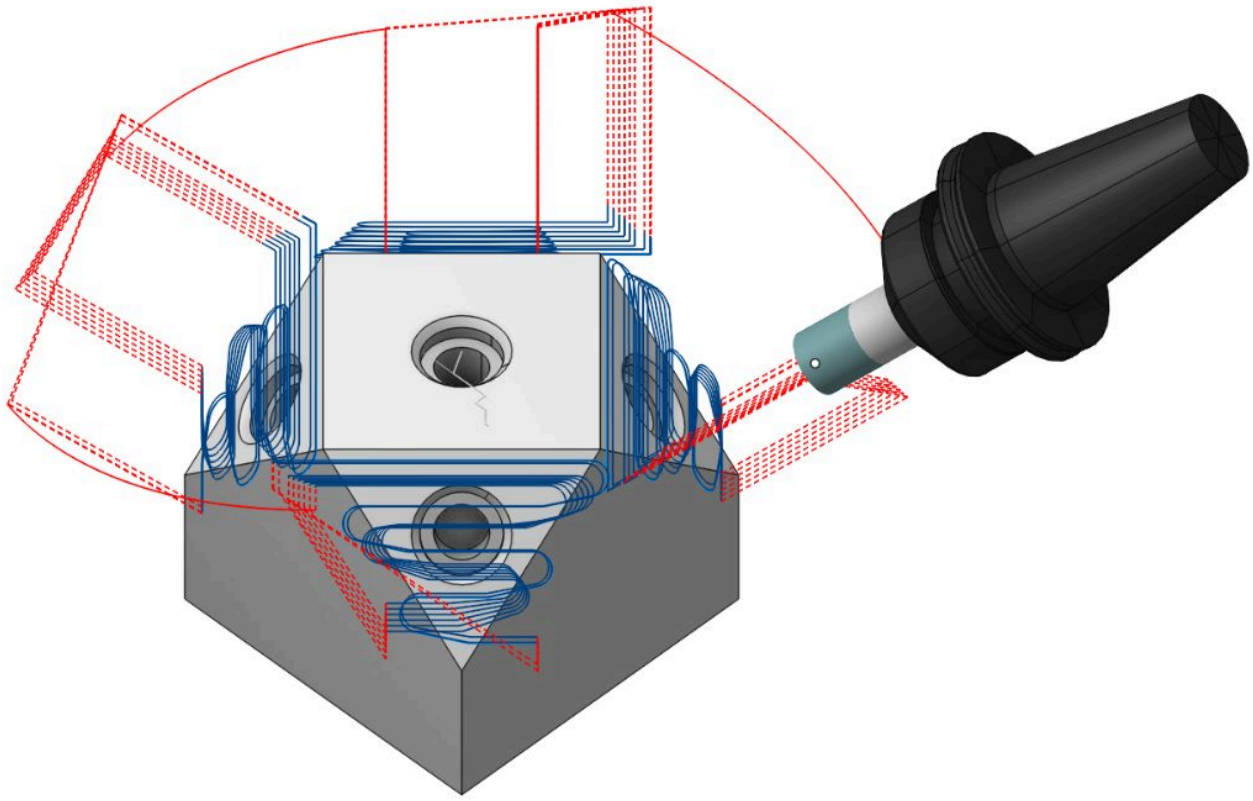


Note that CAM modifies operation header CLData according to your changes. The `ORIGIN G54` command is added to the header in the example:

```
ORIGIN: G54 - MCS(X735.5, Y-286.5, Z-655.66, A0, B0, C0), WCS(X-14.5, Y43.5, Z203, A0, B...
```

Operation local coordinate system





Application area:

Allows generating a trajectory within a coordinate system additionally offset and rotated concerning the workpiece coordinate system. For instance, a program for machining an inclined pocket concerning the workpiece coordinate system will be much simpler if derived in a local coordinate system aligned with the pocket's inclination. This method is used for positional 5-axis (3+2) machining. When selecting the 'Auto' value, the direction of the Z-axis in the local coordinate system is automatically set along the tool rotation axis, which is defined by parameters in the Tool Orientation group.

Principles of working

An operation generates the toolpath in a local coordinate system. By default an operation generates the toolpath in the workpiece coordinate system (G54). However you can change this behavior by specifying the Local CS parameter in the Setup panel.

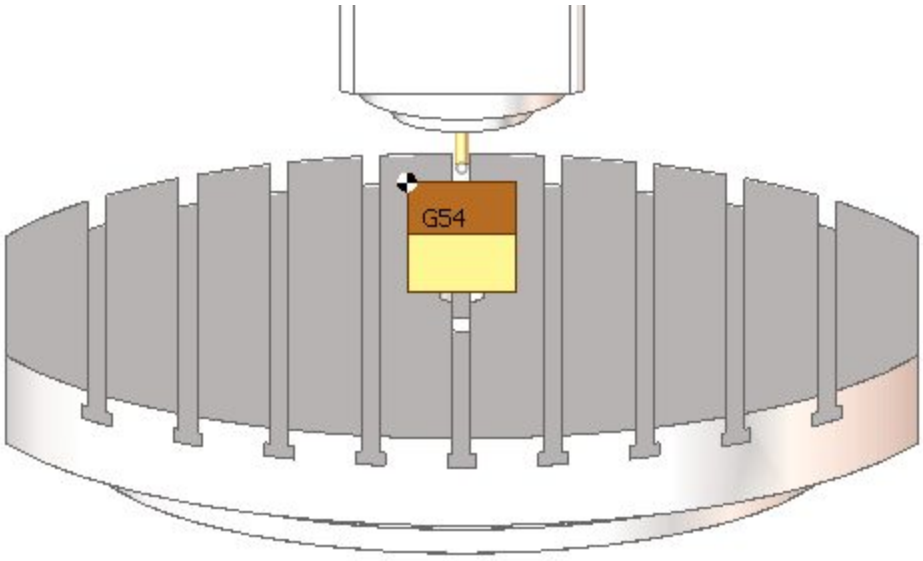
The parameter can be set by two ways. You can select the local coordinate system from the drop down list, and you can set the local coordinate system in the interactive mode by clicking on the ellipses

button.

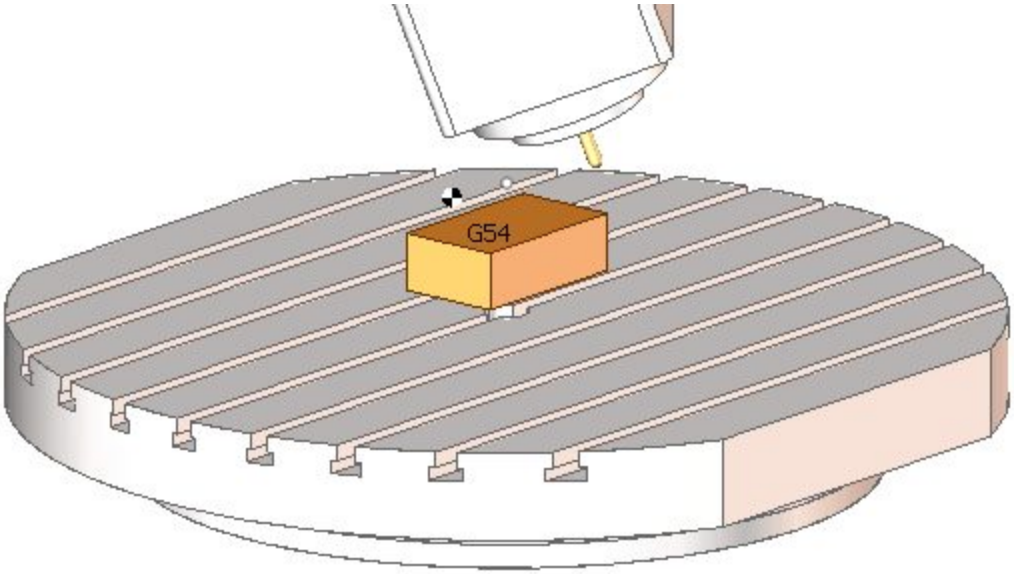
The drop down list contains the following options:

1. Off - the local coordinate system is disabled - the toolpath is generated in the workpiece coordinate system.

Initial machine configuration (A0 C0). G54 is at the top left corner of the workpiece, the tooling point is at the tool tip.



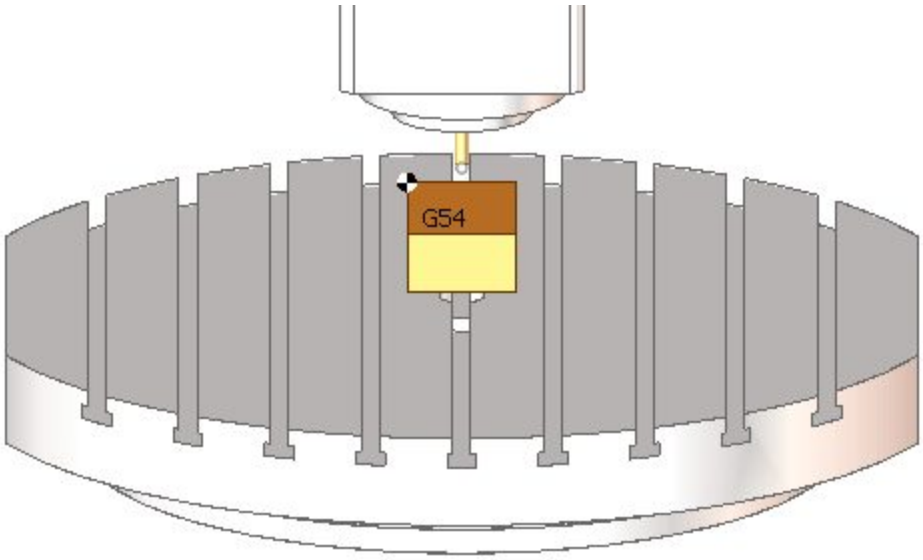
Machine configuration after 5 axis positioning without Local CS enabled (A20 C-40). The workpiece and the tool are moved, but the G54 and the tooling point still stay the same. The generated toolpath will be depend on the workpiece setup and the tool length.



2. Auto - the position of the local coordinate system is determined by the current position of the machine rotary axes. The Z axes of the LCS is aligned to the tool axis direction. The origin of the LCS

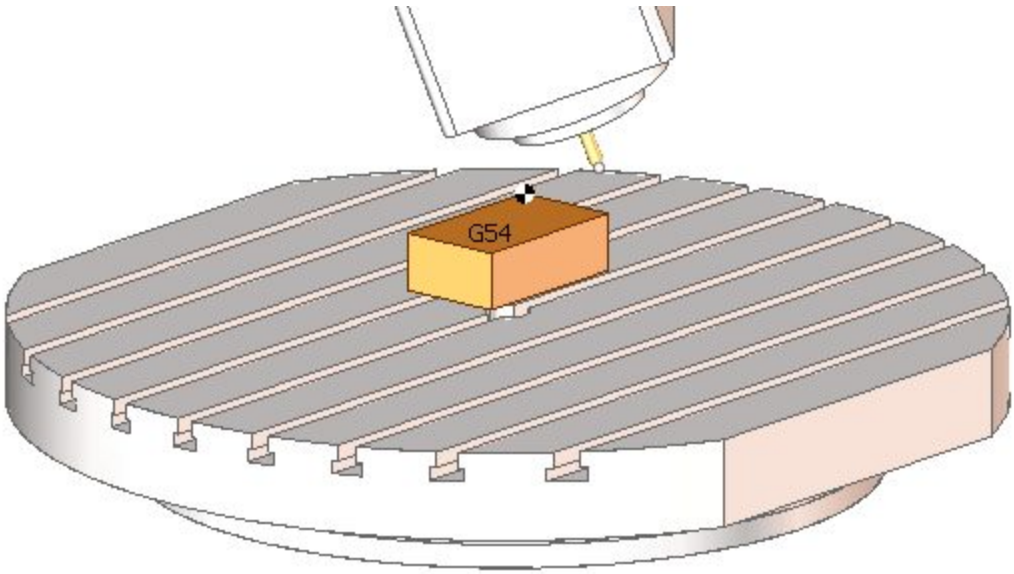
coincides with the position of the workpiece zero point (G54), but you can easily relocate it in the interactive mode (the ellipses button).

Initial machine configuration (A0 C0). G54 is at the top left corner of the workpiece, the tooling point is at the tool tip.



Machine configuration after 5 axis positioning with Local CS set to AUTO (A20 C-40). The G54 is again at the workpiece top left corner, the tooling point is again at the tool tip. The generated toolpath will be independent on the workpiece setup / tool length

So generally you should use the Auto option of the Local CS parameter. For more information refer to the Operation local coordinate system topic.



3. The list of geometrical coordinate systems. To define the operation local coordinate system exactly the same as you want you can create an appropriate geometrical coordinate system and select it

from the Local CS drop down list.

When you enable a local coordinate system CAM system generates the Origin command in the header section of the CLData. The origin command contains all the data required for postprocessing toolpath. These are:

1. MCS - the position of the local coordinate system relative to the machine coordinate system (G54). This matrix is used in old controls without the tool center point management function (TCPM).
2. WCS - the position of the local coordinate system relative to the workpiece coordinate system (G54 rotated together with the workpiece). This matrix is used in the controls with the TCPM function.
3. Coordinates of the rotary axes positioning the tool axis along the z axis of the local coordinate system.

By clicking on the ellipses button next to the Local CS parameter you enter the interactive mode of Local CS definition. After positioning the local coordinate system just click OK to apply the changes. Details

In addition some controls do not support the definition of a local coordinate system by spatial angles. Those controls require the local CS is defined by the actual machine rotary angles, aligning the tool axis direction along the Z axis of the local coordinate system. CAM system supports such controls. The corresponding option is available under the Control parameters section in the machine properties.

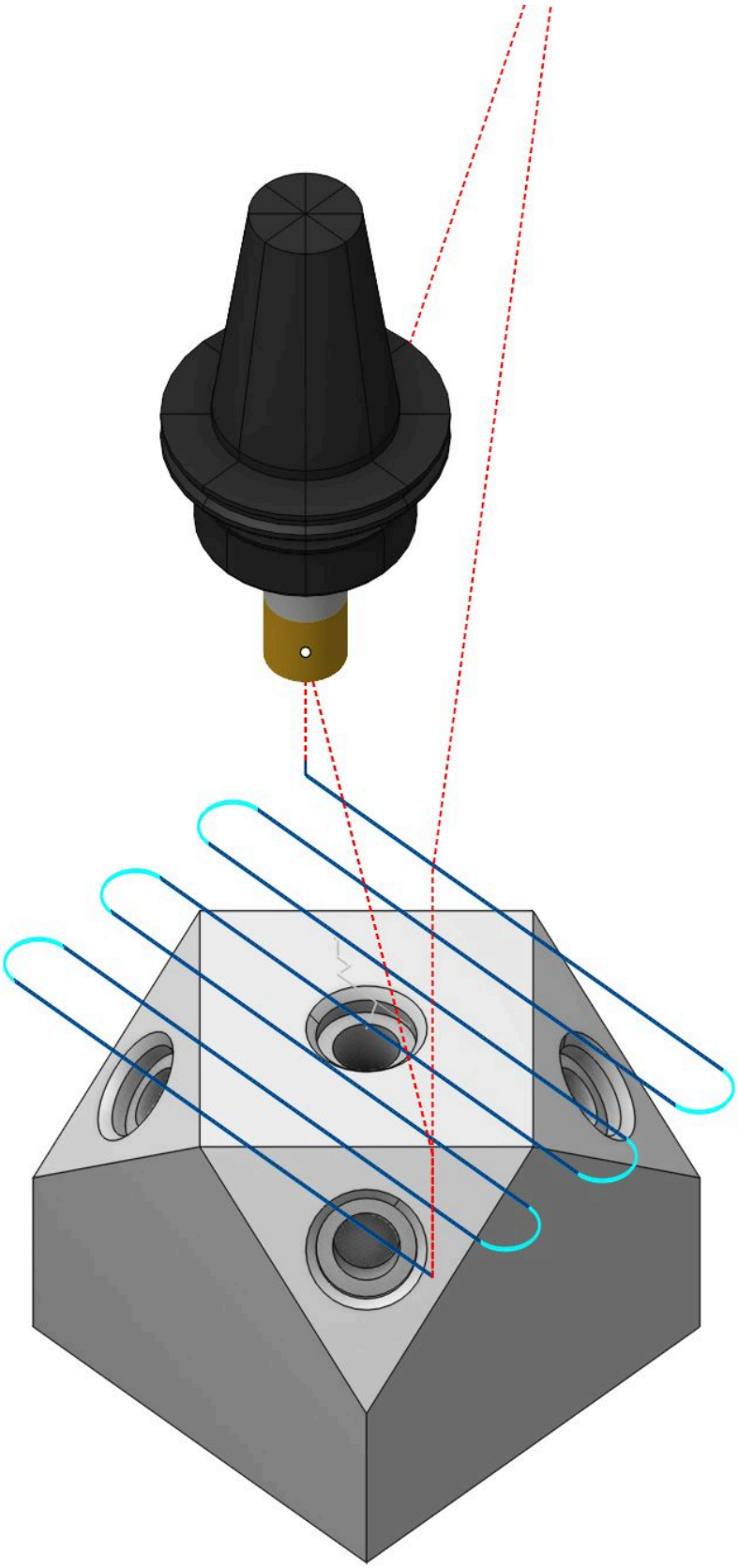
Another control feature is the Local CS positioning mode. This option determines the behavior of the control when it treats the Origin command. CAM system supports the following modes of the Local CS positioning. (Machine Setup - Control parametrs - Indexed 5-sxis machining - Local coordinate system (ORIGIN))

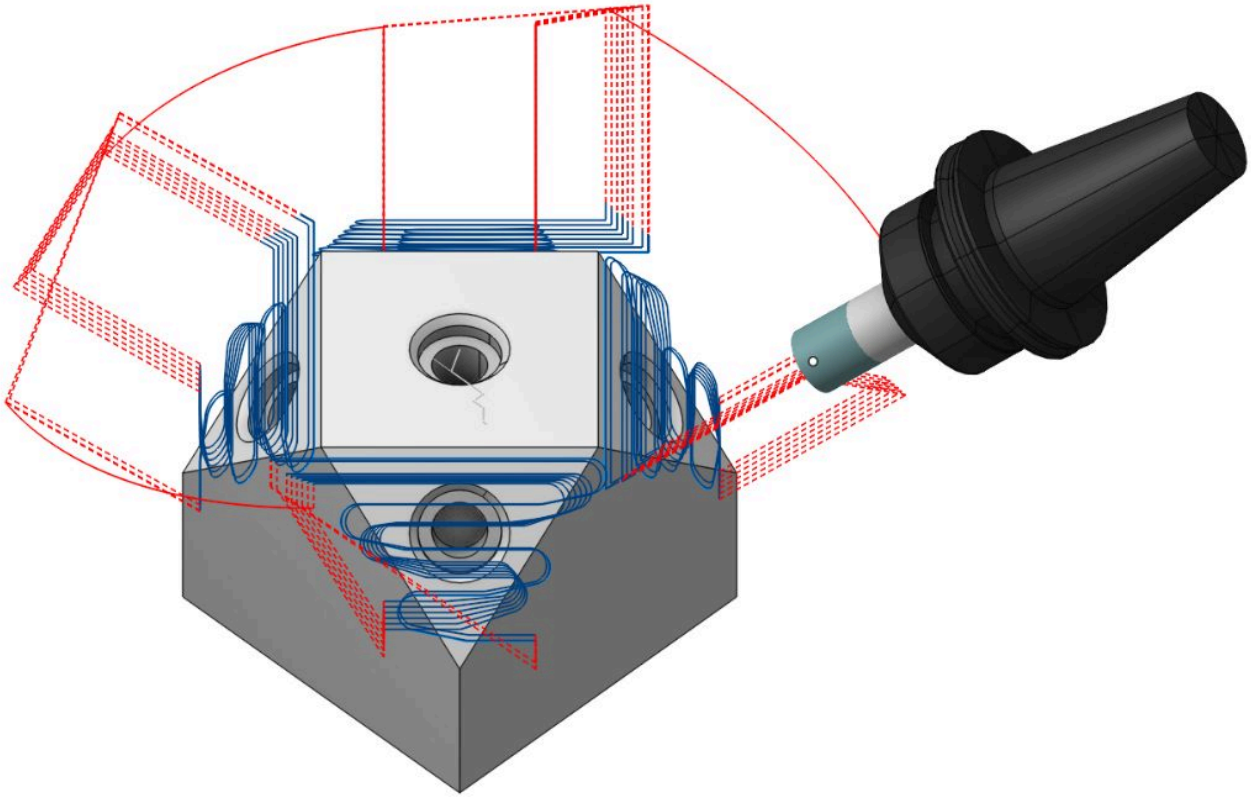
1. Stay - the origin command does not move the machine axes.
2. Turn - the origin command rotates the machine axes in such a way the tool axis direction becomes aligned with the Z axis of the local coordinate system.
3. Move - the origin command rotates the machine axes to align the tool axis with the Z axis of the local coordinate system and moves the linear axes in such a way the tool tip position stays the same relative to the workpiece.

In the next table there are commands for the most commonly used CNC controls which can be generated by the postprocessor instead of the ORIGIN command.

Origin	G92 G68	TRANS ROT	Cycle 7 Cycle 19

Tool orientation





Application area:

5-axis positioning provides a convenient method of manufacturing parts that require milling on multiple faces by minimizing setups. 5-axis positioning is performed by setting the appropriate values to the Tool orientation parameter section in the operation Setup panel. All the milling operations may be used for part's processing from different sides. In the case the equipment doesn't allow to rotate the part or the axis of the tool, it is necessary to change the scheme of part's workholding. In CAM system you can easily set the rotary axes position by clicking on the part face which the tool axis has to be aligned to

Principles of working with a window.

For a particular operation you can set the actual values for the rotary axes position as you need. After generating toolpath the Approach section of the CLData will contain the commands of rotary axes positioning. The entire program is then generated with regard to the new rotary axes position.

Tool orientation window.

Up, Down, Front, Back, Left, Right. If you want to specify one of the standard orthogonal orientations, you can use the buttons with names corresponding to the direction you need

View vector. You can also orient the machine axis along the current view vector used in the graphics window

X,Y, Z. Determine the rotation vector

Flank. Option will help you orient the tool sideways in relation to the selected direction instead of along it.

Rotation angle. Additional turning angle.

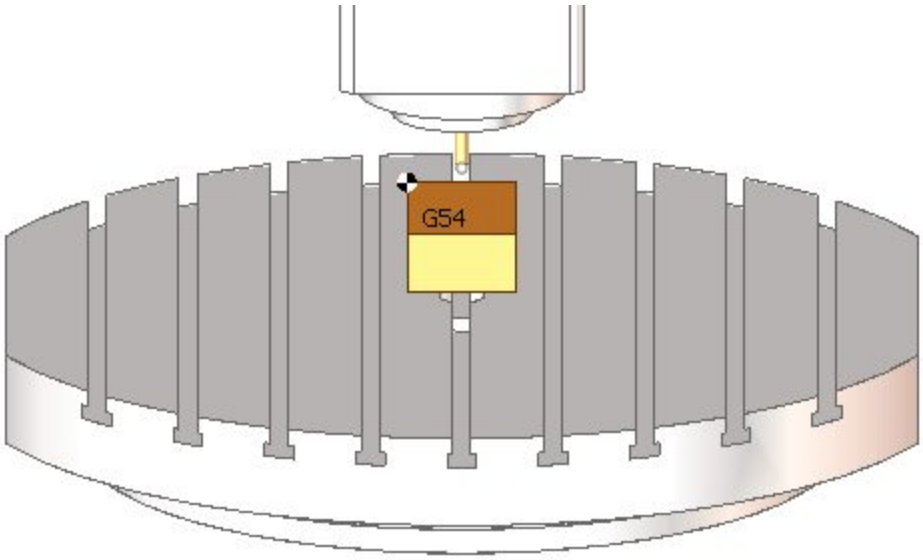
Alternate solution. Option if the machine allows you to provide the same relative position of the tool relative to the workpiece in several ways.

You can easily set the rotary axes position by clicking on the part face.

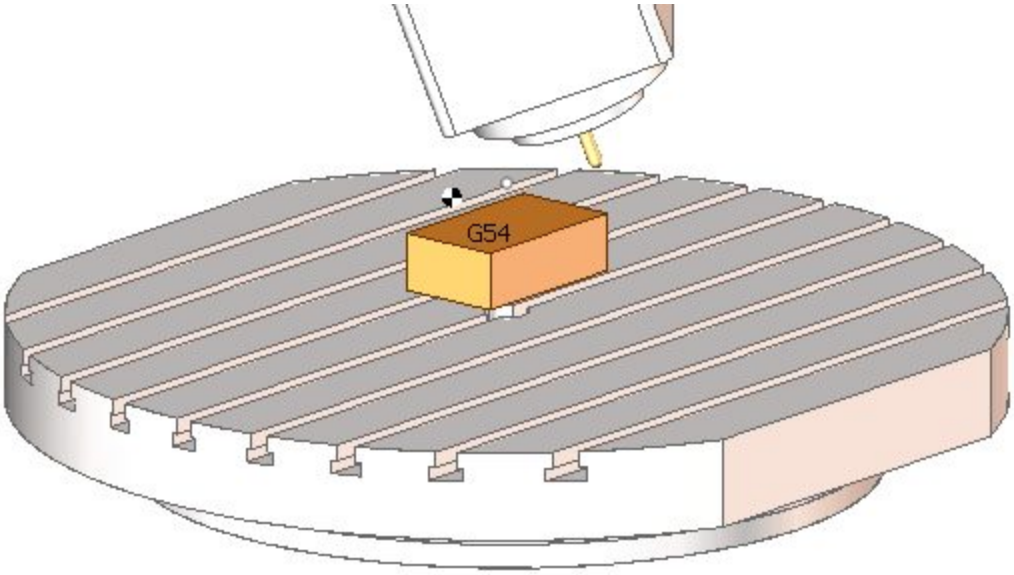
By default an operation generates the toolpath in the Workpiece coordinate system (G54 - G59), but you can change this behavior by specifying the Operation local coordinate system parameter in the Setup panel.

Many today controls require definition of the local coordinate system for 5 axis positioning. Without this definition the position of the workpiece zero point (G54) and the position of the tooling point is not updated after changing the position of rotary axes - the control behaves as it knows nothing about the machine kinematics - as a regular 3 axis control. But when you specify the local coordinate system (the PLANE function at HDH, the ROT function at Sinumeric), the control updates the position of the workpiece zero and the position of the tool tip regarding to the actual workpiece-tool orientation. It looks like at the following figures.

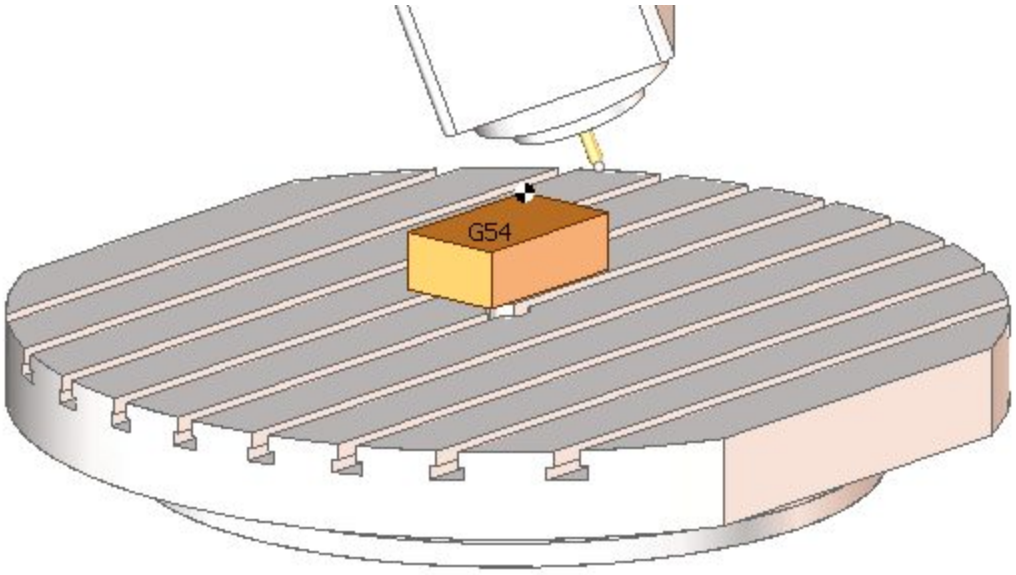
When changing the orientation of a tool, it is most often necessary to set a local coordinate system so that the machine can compensate for the position of the workpiece coordinate system and the tool's setup point. Details



Starting position of the machine tool (A0 C0). G54 is located at the upper left corner of the workpiece, the tool setting point is aligned with the tip.



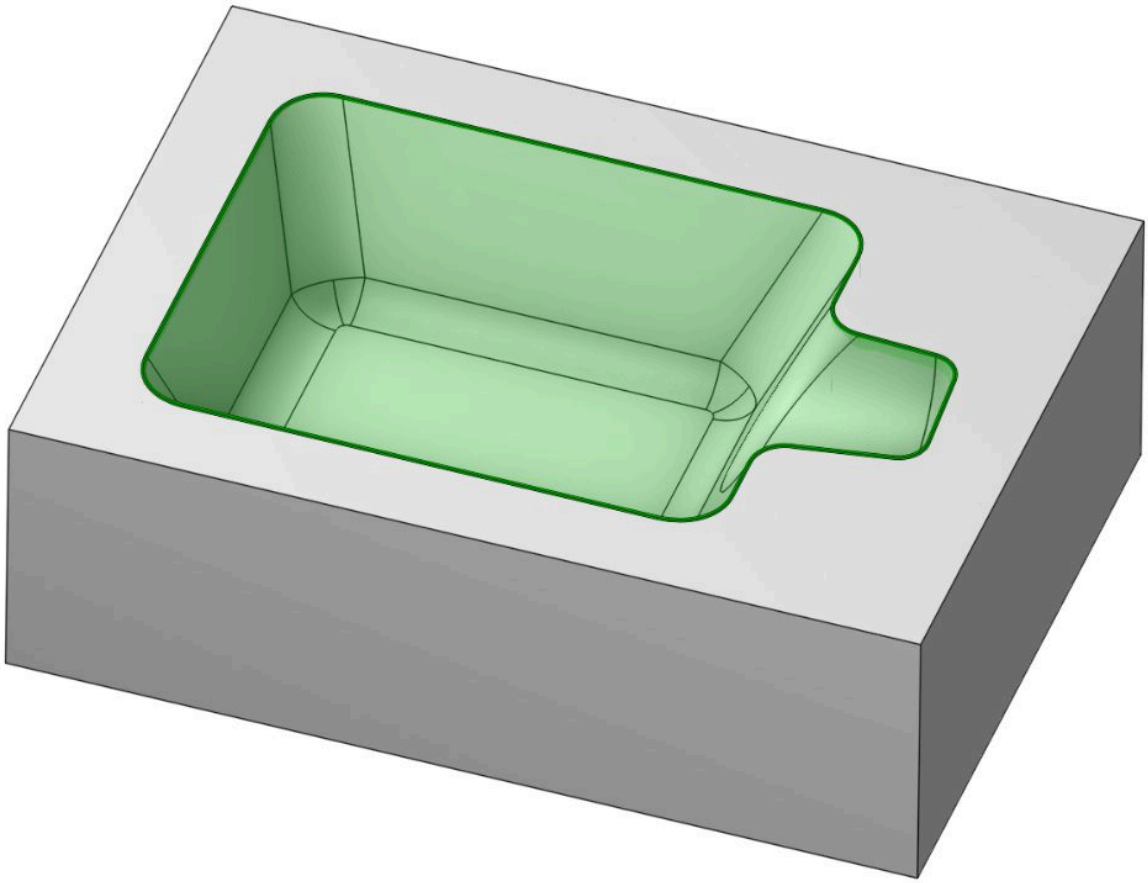
Machine position after five-axis positioning without specifying a coordinate system (A20 C-40). The workpiece zero and the tool set point have not changed their position in space, while the workpiece and tool have. The calculated trajectory will depend on the position of the workpiece on the machine and the tool reach.



Machine position after five-axis positioning with coordinate system reference (A20 C-40). G54 is again at the upper left corner of the workpiece and the tool set point is again aligned with the tool tip. The calculated control program will be independent of the workpiece position on the machine and the tool reach.

For this reason, in most cases, it is recommended to set the Local SC parameter to Auto for racks equipped with the coordinate system rotation function. For more information, refer to the Local coordinate system section of this manual.

Job assignment



Application Area:

The job assignment specifies the surfaces and areas to be machined, depending on the operation type. Different methods can define the job assignment.

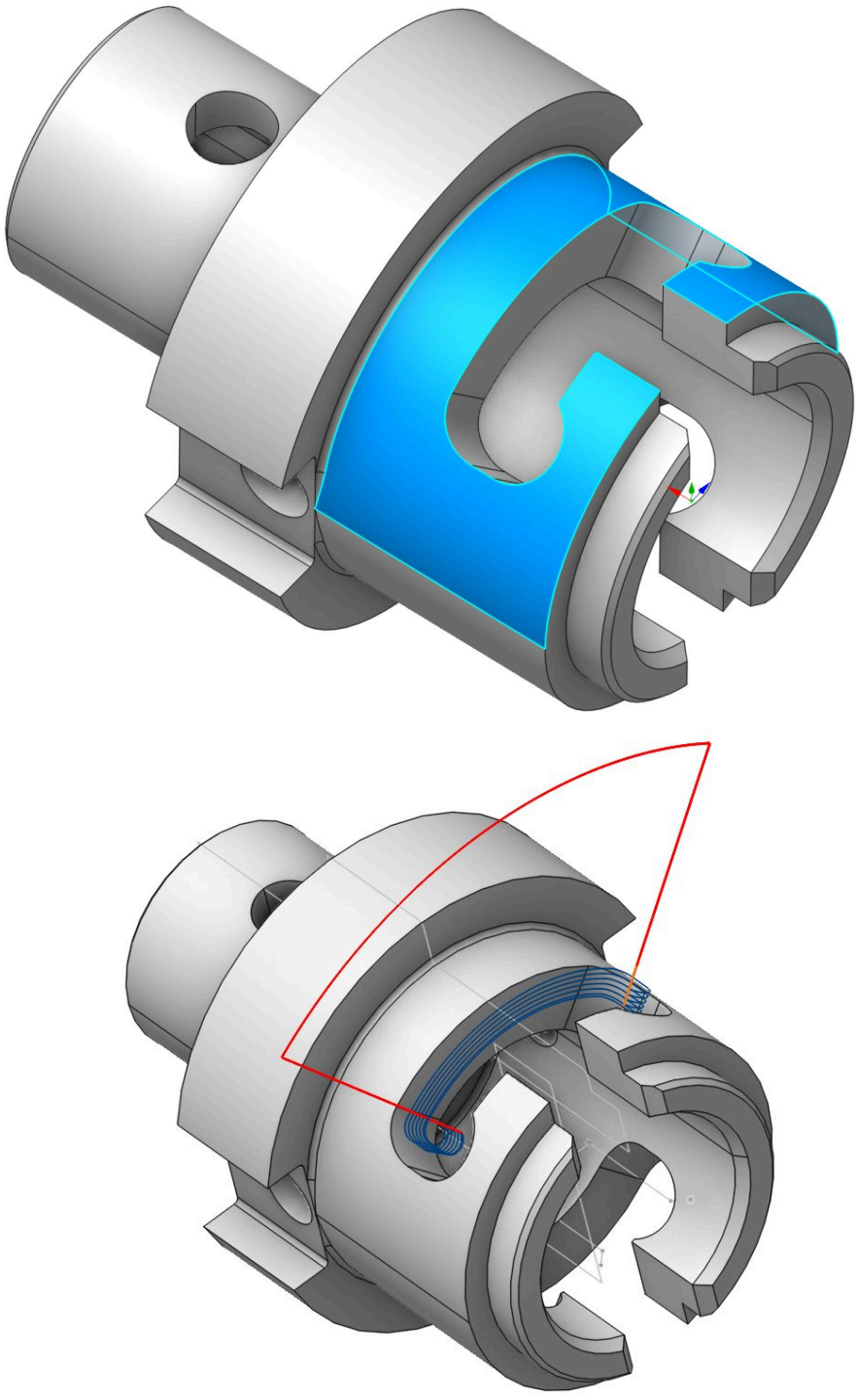
Defining the job assignment

- For curve machining operations, such as 2D contouring and 3D curve milling, the assignment can be defined by curves, groups of curves, and points for circle centers. The toolpath considers user-defined options like tool offset and additional stock for curves.
- In engraving and pocketing operations, the assignment is based on top machining areas created from curve projections. The toolpath removes material outside the model, considering the stock and additional features like ridges and cavities defined by curves.
- Volume machining operations use solids, surfaces, and meshes for the job assignment. Additional stock can be specified for objects, influencing the toolpath to remove material outside the

assignment without cutting the assigned objects.

- Turn operations offer various ways to specify fragments of the part's turn generatrix to be machined.

Base surface



Application Area:

It is possible to create 2D Contouring, 2D Pocketing, Engraving, Area Cladding operations that generate toolpath on a base surface. As a base surface you can use planes, cylinders and revolution surfaces. The base surface is specified in the Job assignment of a pocketing/engraving operation. You can use existing

3d model surfaces as a base surface. If you do so you also can add surfaces connected to the base surface as job assignment items pockets and bosses. This facility is very useful for letter engraving.

Base Surface properties dialog box:

Configure the following base surface parameters in the Properties dialog box:

Item type:

None. Leave base surface parameters unconfigured.

Plane. Set the base surface as a plane passing through the Center point, and is perpendicular to the coordinate axis defined in the Axis Directions fields. You can designate a plane from the 3D model as the base plane by selecting it in the graphic field. The system will automatically calculate its parameters.

Cylinder, radius. Set the base surface as a cylinder with its axis determined by the Axis Directions parameters, passing through the Center point, and with a radius specified in the dialog box. You can designate a cylindrical surface from the 3D model as the base cylinder by selecting it in the graphic field. The system will automatically calculate its parameters.

Revolution. Select a user-defined surface of revolution as the base surface.

Position.

Axis. Select the axis that defines the position of the base surface (the orthogonal vector to the Plane or the axis of the Cylinder).

Axis Direction. If using a Custom axis, input the unit vector components along the coordinate axes that define the axis.

Center (Pc). The point defining the origin of the base surface.

Parameters.

Alternate Front Side. This feature changes the machined side of surface to the back one.

Curves Project Method. Specifies the approach for mapping the curve onto the base surface.

Nearest. The curve maps directly to the nearest surface according to its position and extent.

Map to UV domain. Curve projection is determined by the surface UV map.

Curves Transformation. Modify the UV coordinates to change the projected curve.

Xmin. Adjust the initial U coordinate of the map for the base Plane.

Xmax. Adjust the final U coordinate of the map for the base Plane.

Ymin. Adjust the initial V coordinate of the map for the base Plane.

Ymax. Adjust the final V coordinate of the map for the base Plane.

Hmin. Adjust the base Cylinder's starting position.

Hmax. Adjust the end position of the base Cylinder.

Amin. Adjust the initial angle of the base Cylinder sector.

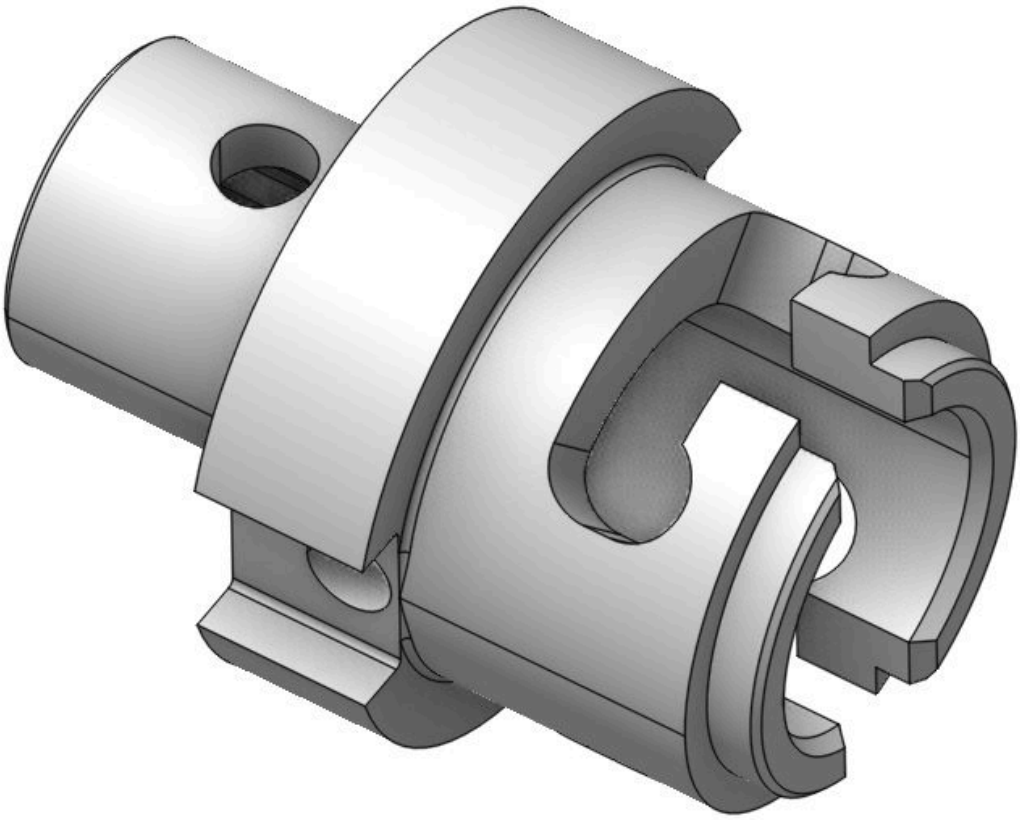
Amax. Adjust the final angle of the base Cylinder sector.

Rotation Angle. Rotate the curve around the map center.

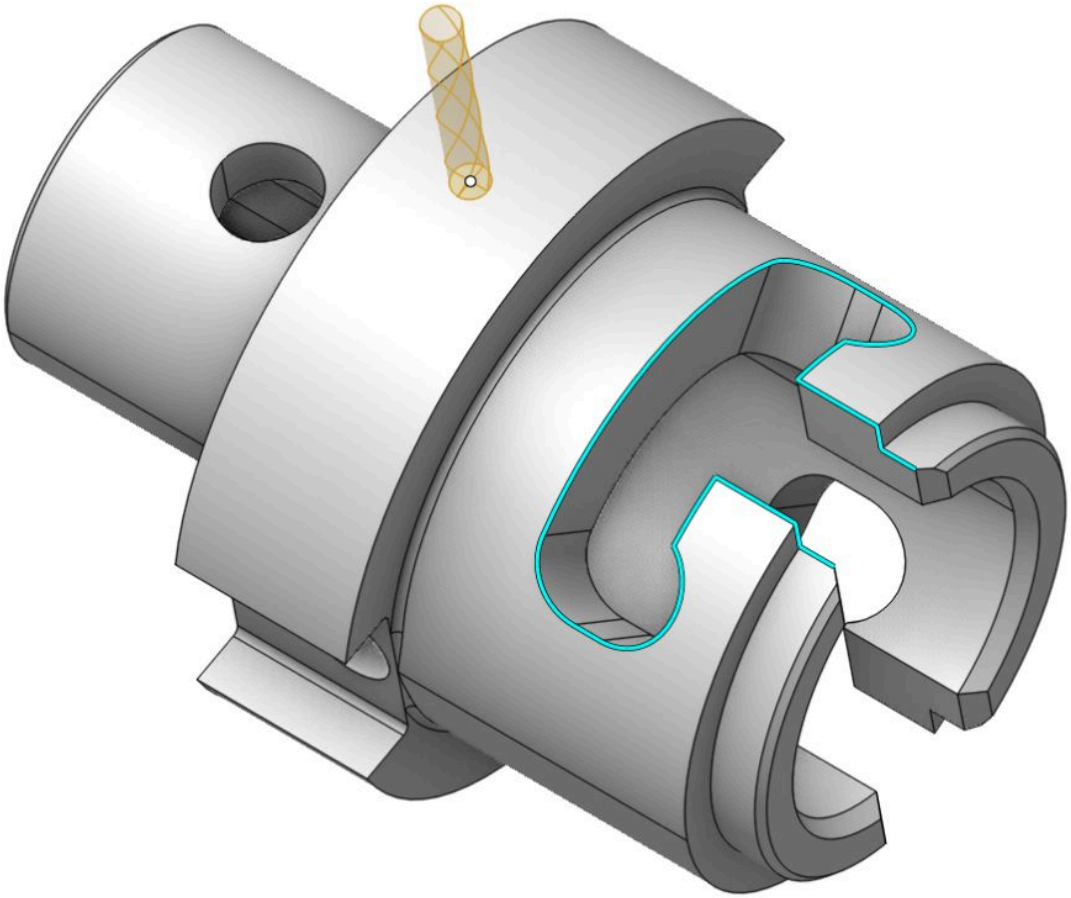
Preserve ratio. Maintain the proportions between curve elements during transformations.

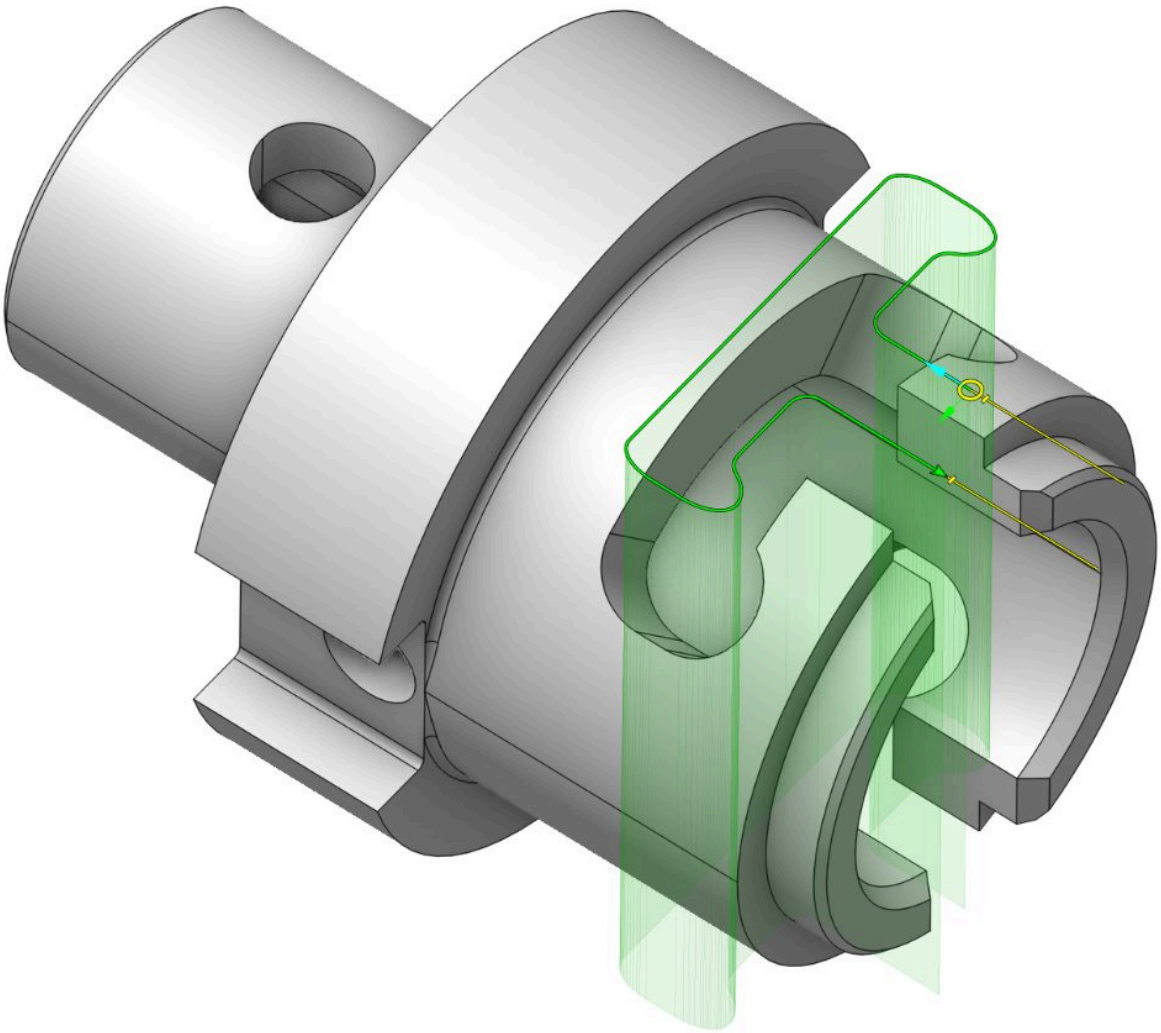
Working with Base surface.

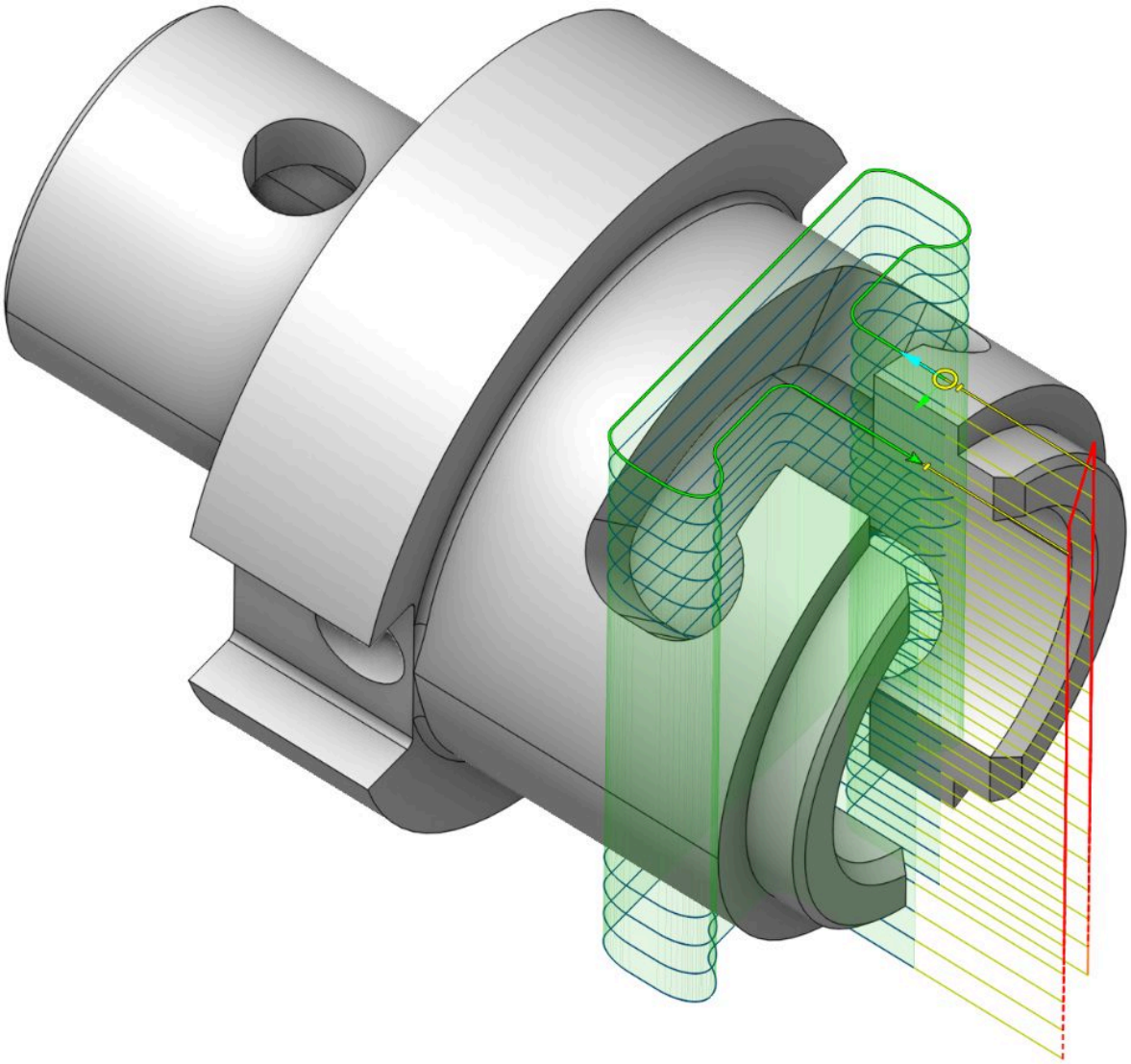
As an illustration, we will attempt to machine this pocket on a mill-turn machine utilizing the C-axis and cylindrical interpolation.



We will be using the 2D contour operation. We will select the edges on the model and calculate the toolpath. We can see that this toolpath isn't satisfactory.

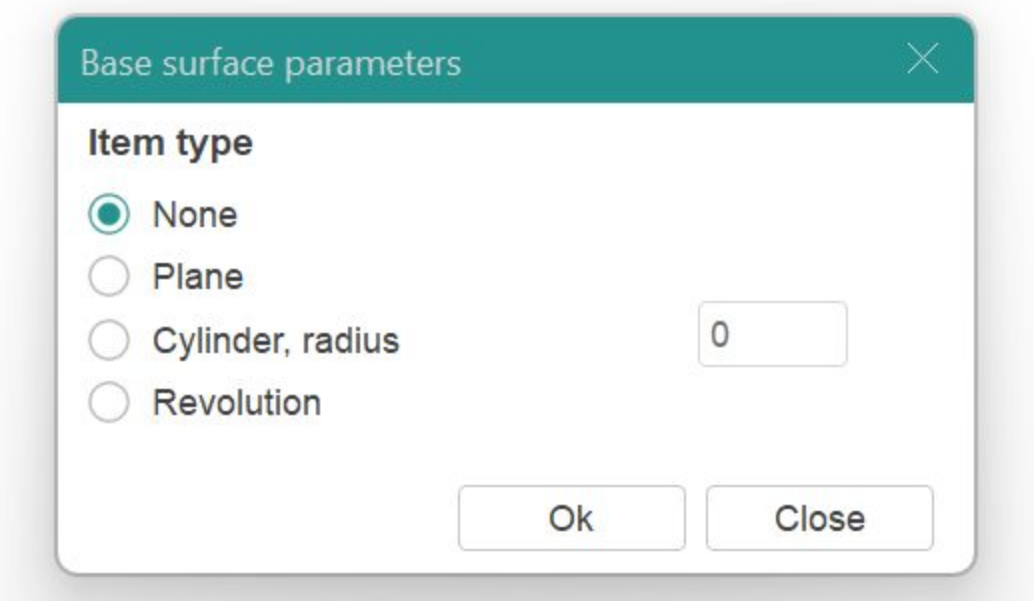


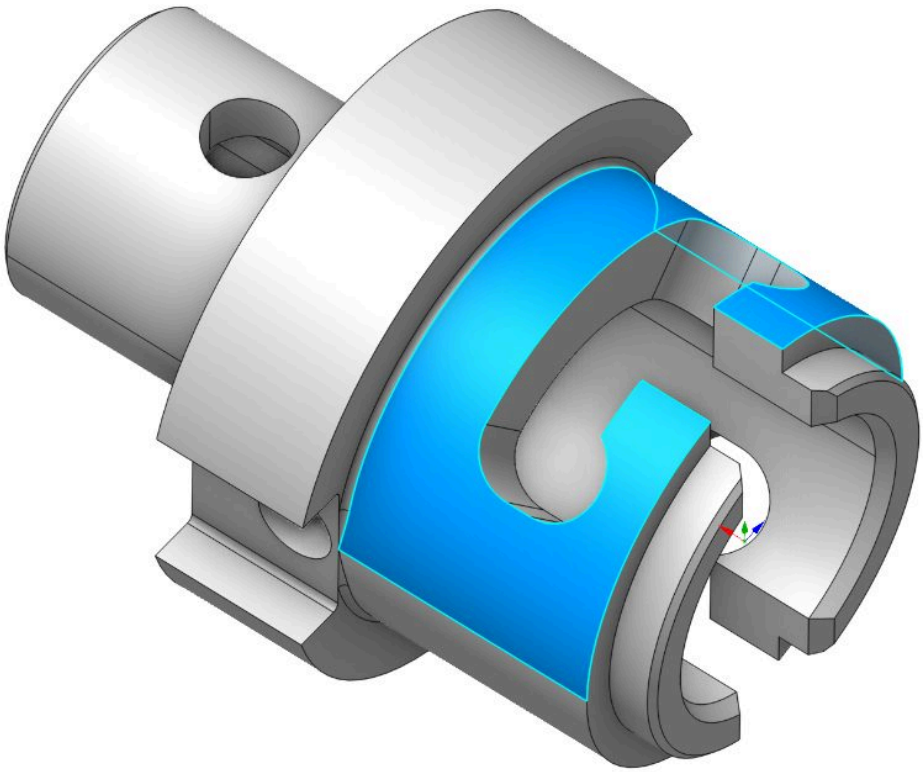


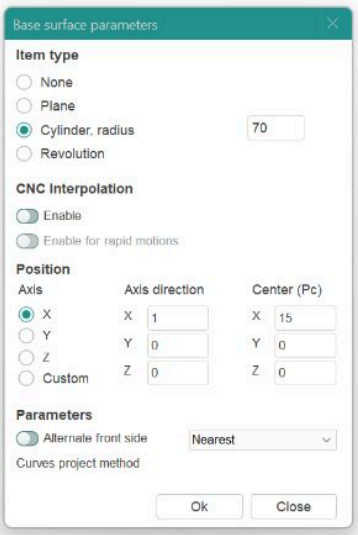


In the job assignment, we will call up the base surface window.

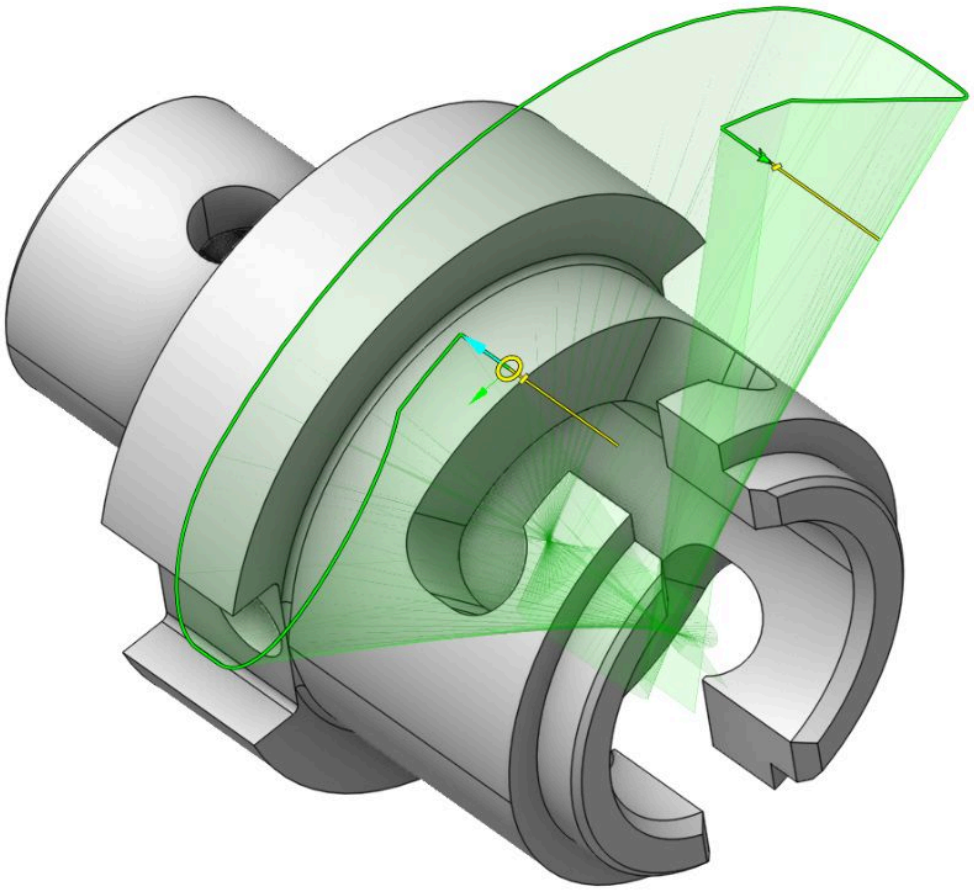
Double-click on the cylindrical surface of the part. The radius of the base surface will be automatically determined in the window.



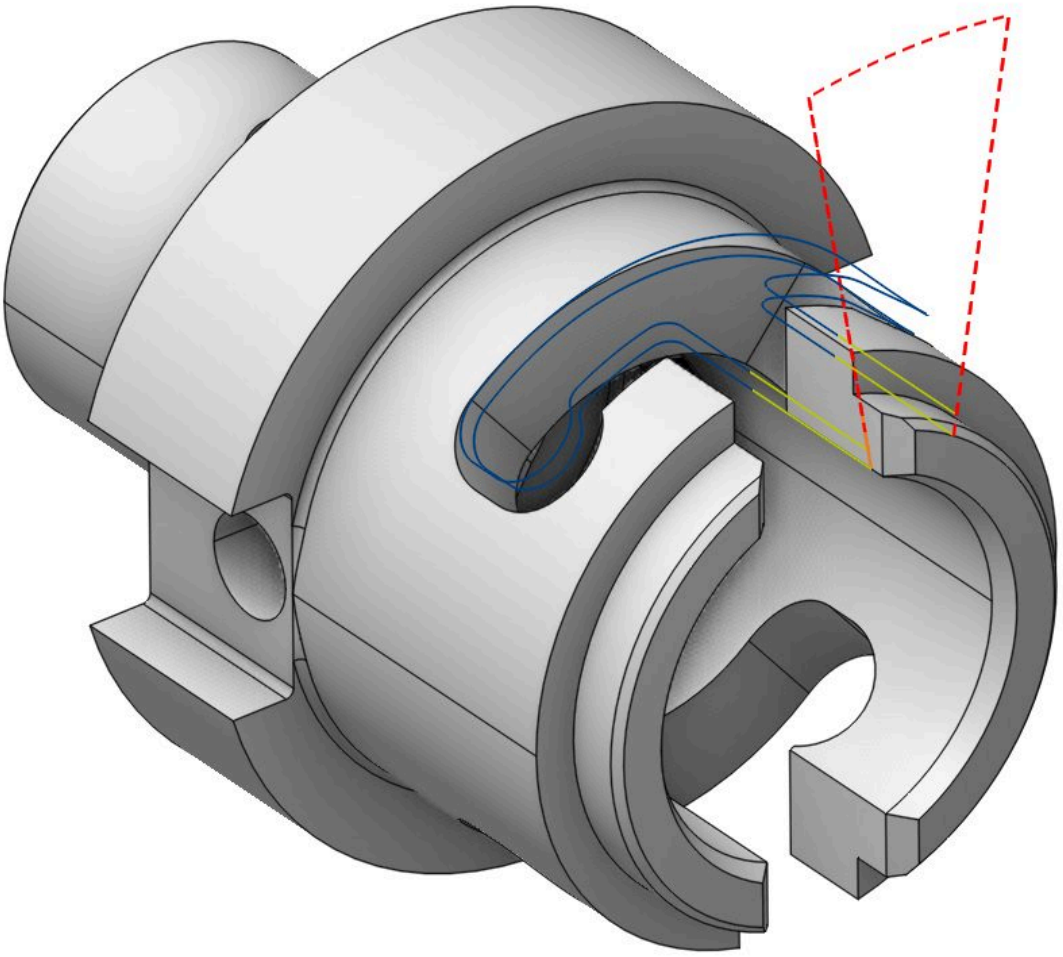




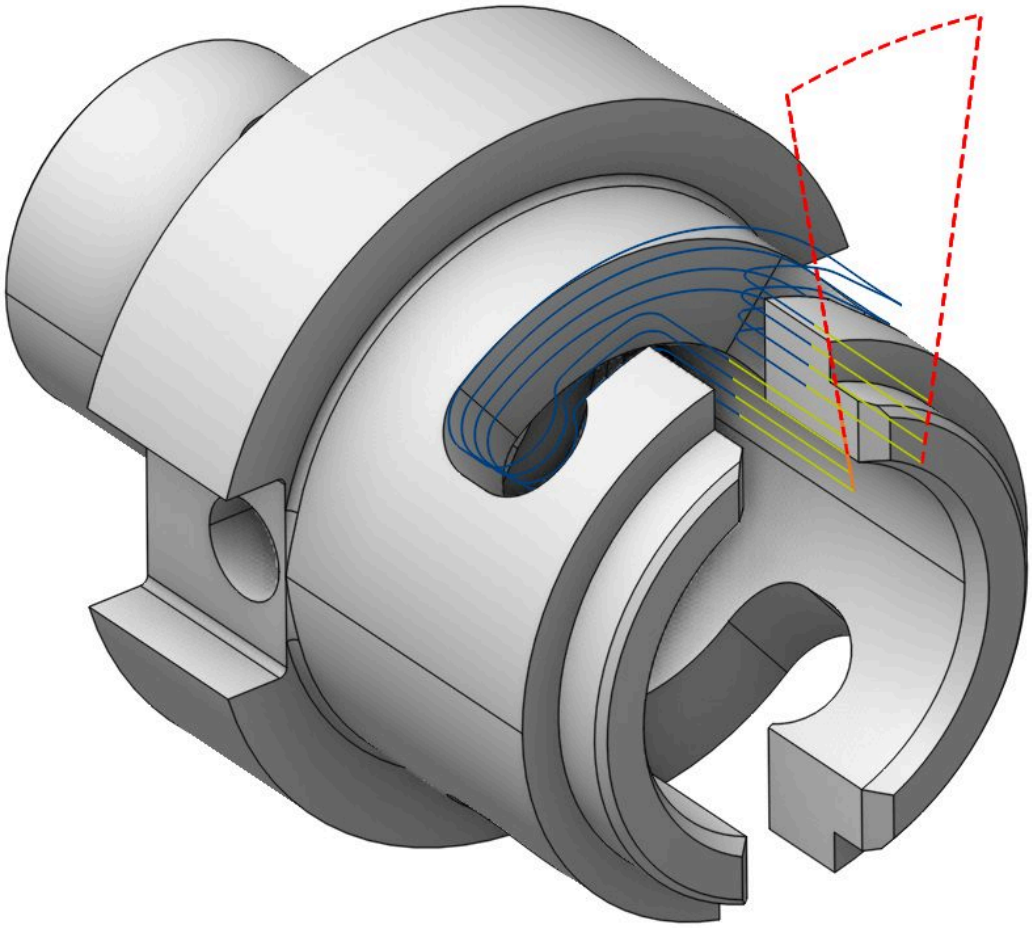
We will delete the contour and assign it again. We will get the following result. Now we need to set up the machining levels.



In the Strategy tab, when selecting this base surface, specify the bottom level as 0 and set the top level to 10. Set the Depth of cut to 5 mm. Calculate the toolpath.



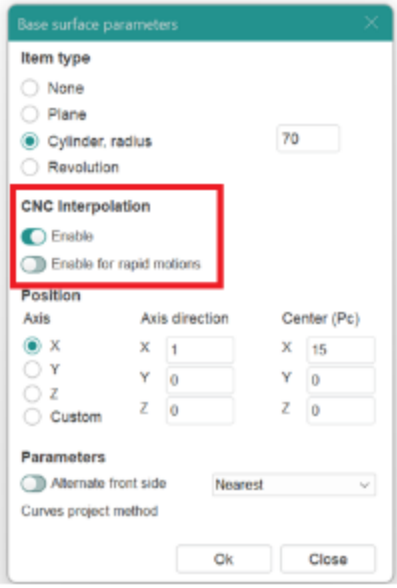
We can see that the bottom level is now on the base surface. To cut the pocket, we can change the bottom level to -10 mm.



However, the CLData output is missing commands for cylindrical interpolation.

RAPID: 10000
MultiGOTO: X0, Y170, Z-26.5, C9.12
X0, Y85, Z-26.5
COOLNT: On, #1
F: APPROACH 200mm/min.
X0, Y75, Z-26.5
F: ENGAGE 200mm/min.
X0, Y75, Z15
F: WORK 200mm/min.
X0, Y75, Z38.623
MultiGOTO: Z38.868, C9.13

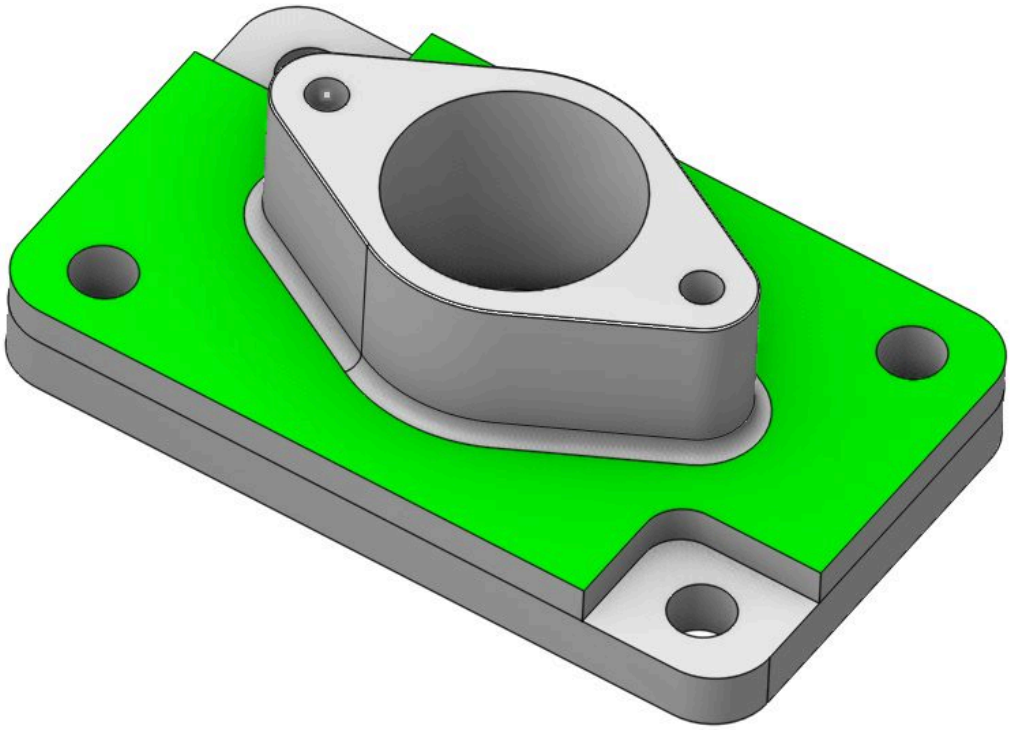
To enable it, you need to go to the base surface tab and enable the corresponding options. (It is important that the cylindrical interpolation support checkbox is also enabled in the machine configuration Control Parameters - Rotary Transformations. Details)

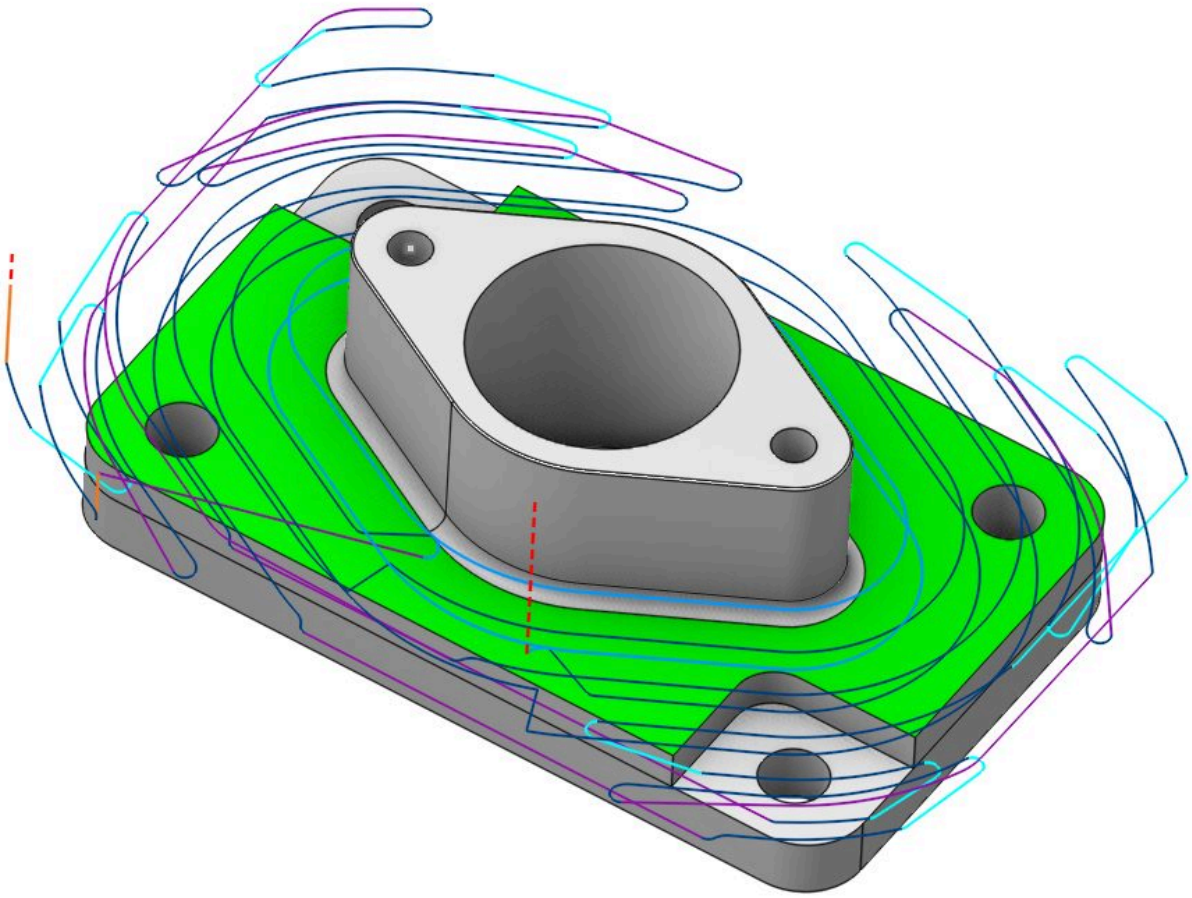


Level: 5
RAPID: 10000
MultiGOTO: X0, Y170, Z-26.5, C9.12
MultiGOTO: Y85
COOLNT: On, #1
INTERP Cyl On, R70
F: APPROACH 200mm/min.
X11.143, Y85, Z-26.5
X11.143, Y75, Z-26.5
F: ENGAGE 200mm/min.

After that, the corresponding commands will appear in the CLData, which can then be processed in the post-processor. You can learn more about cylindrical interpolation in the following section Details.

Faces





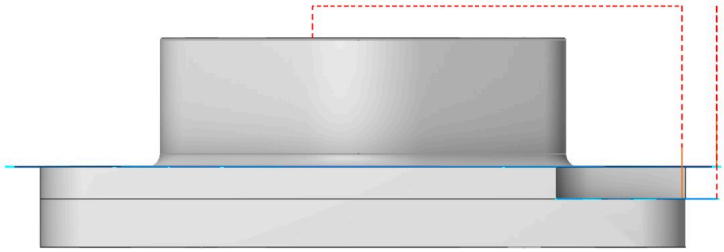
Application Area:

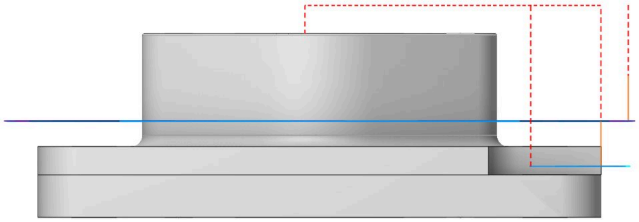
The button adds selected faces to the list. If no faces are selected then the dialog with the geometry model tree appears. In this dialog one should find and select the faces, the meshes and the groups to be added to the list. If a group is added then only 3D-objects of the group will be taken into account.

You can study the features of choosing faces in the corresponding section. Details

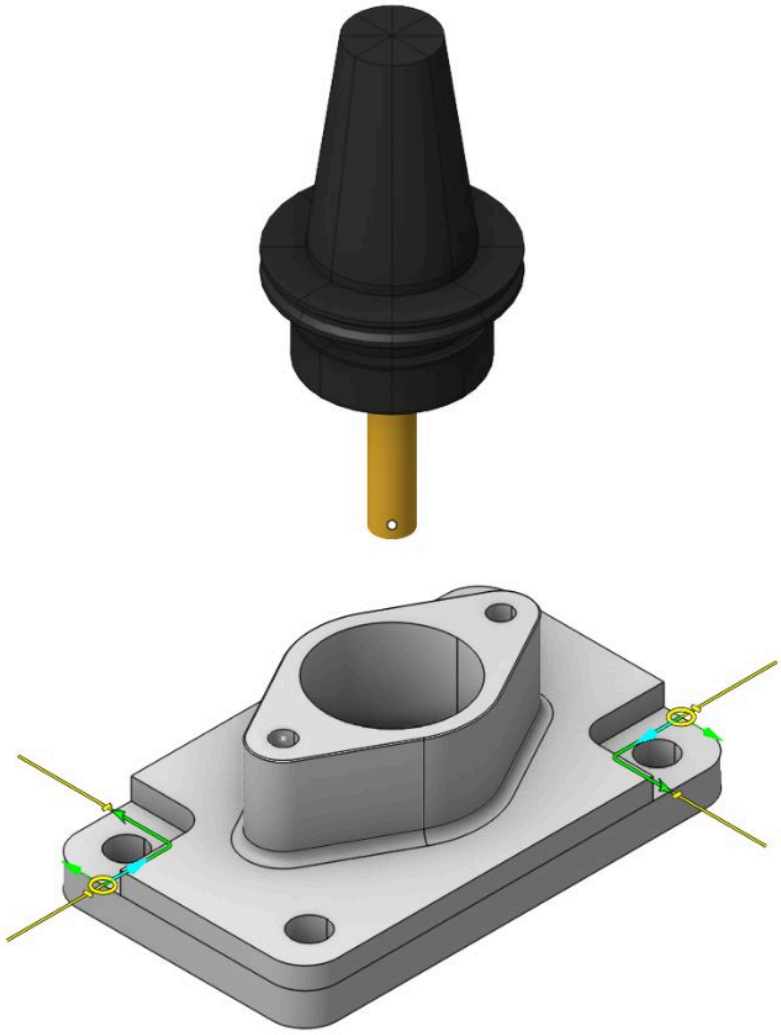
Surface stock

If you select different surfaces as a Job assignment, then in some operations you can use the surface property (double-click on the surface element in the Job assignment) to specify different stocks for different surfaces.





Curve



Application Area:

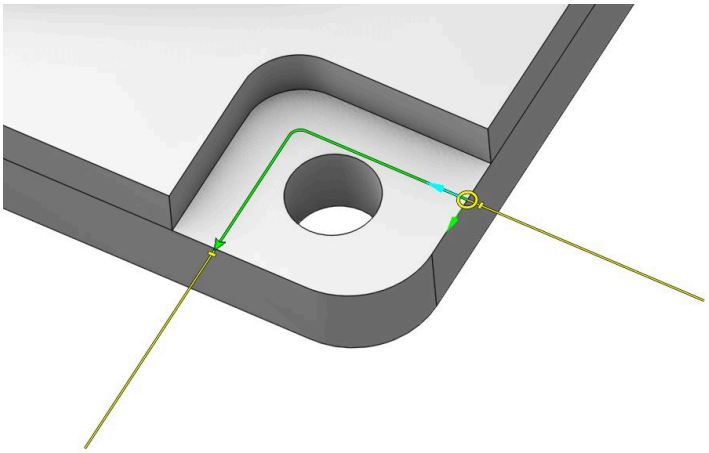
The 2D and the 3D contouring operations are simple, at the same time the powerful cycle for creating planar tool paths based on specified curves, edges, faces or pockets. By defining a base surface you can also generate tool paths on cylindrical and revolution surfaces. Using the Project on part feature you can project the generated tool path onto the part, and by specifying the Tool axis orientation mode you can generate four and five axis tool paths by normal to the part or to the base surface.

You can study the features of choosing curves in the corresponding section. Details

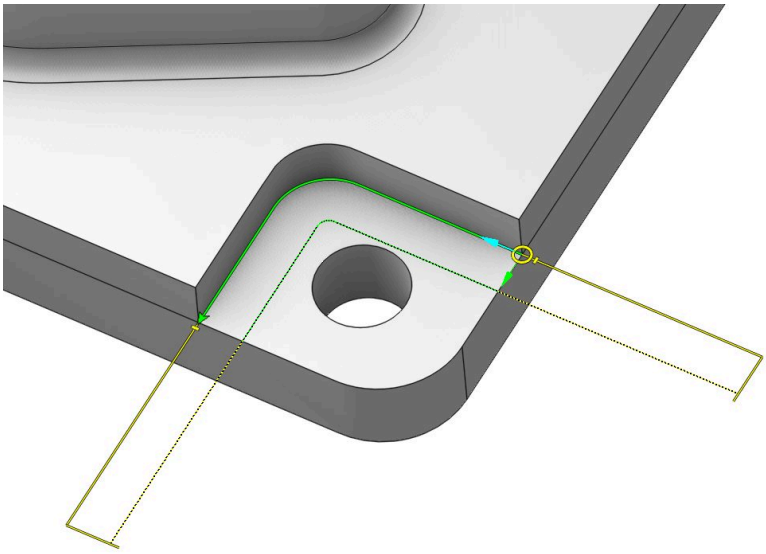
Working with curves

1. To add a new contour into the job assignment just select curves, edges or vertical faces you want to machine and press the Curve button. The connected entities will be joined into contours.

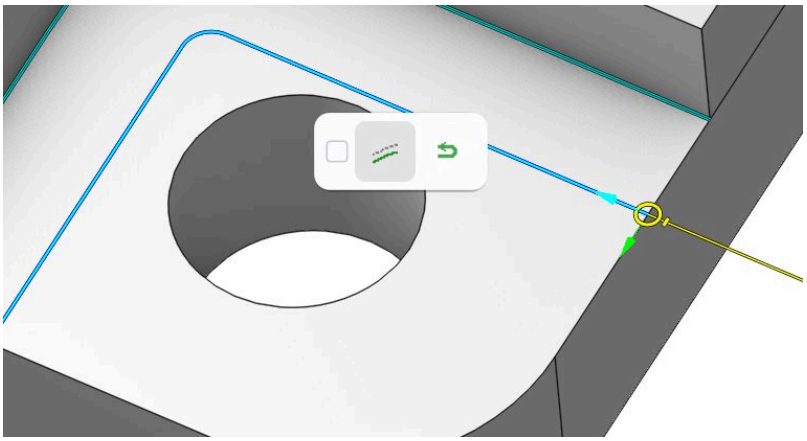
Contour itself. The green line represents the curve which will be output into the CLData. Approaches are marked in yellow (You can change them interactively or on the Links/Leads page.).



If a contour is compensated and the Radius compensation is set to Control than an additional dashed line is displayed, which represents the tool center curve (the curve along which the tool will be moved).



To access contour parameters just left click on a contour in the graphic view. You will see the action bar. Here you can switch the Compensated flag of a contour and its Machining with return flag. For closed contours you can also switch the Fix start point flag.

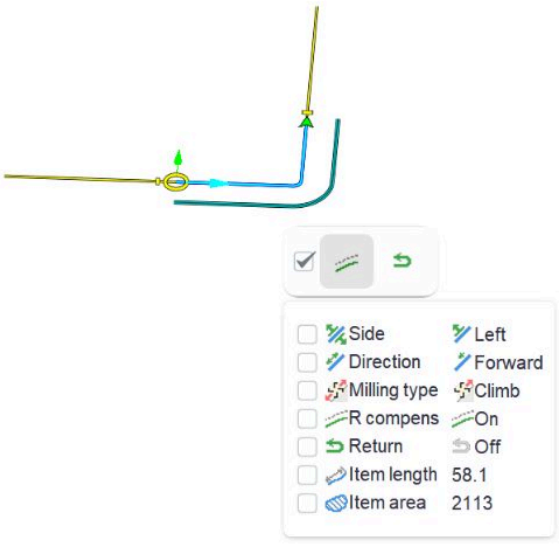


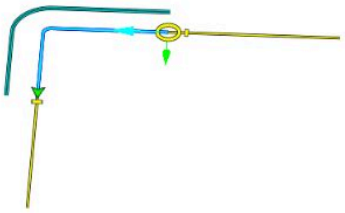
Changing parameters of several contours together

To do this you should select several contours and edit parameters of one of them. When you make a change to a parameter of one contour this change applies to all the selected contours. In such a way you can change the Compensated flag, the Machining with return flag, the Fix start point flag, the contour direction, the contour side.

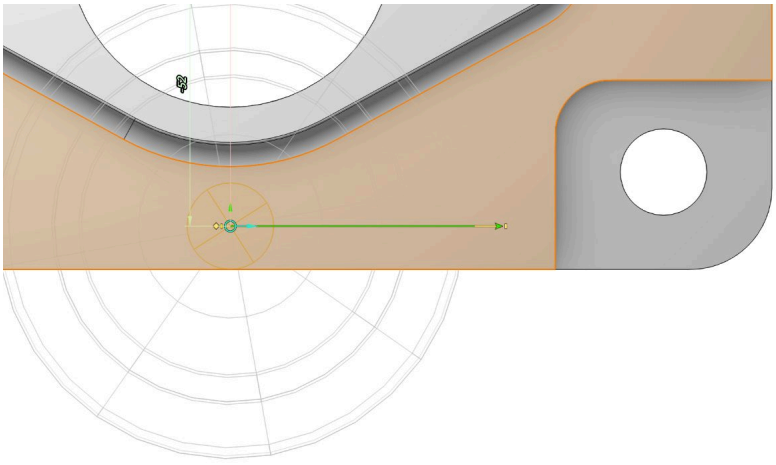
Selecting multiple contours

- To select several contours you can hold the [Ctrl] key on the keyboard and select them with the mouse.
- To select all contours in the job assignment just double click on any of them in the graphic view.
- To select contours by some criteria click on a contour, than on the check box in the action bar that appears and check the criteria in the properties tree that appears.

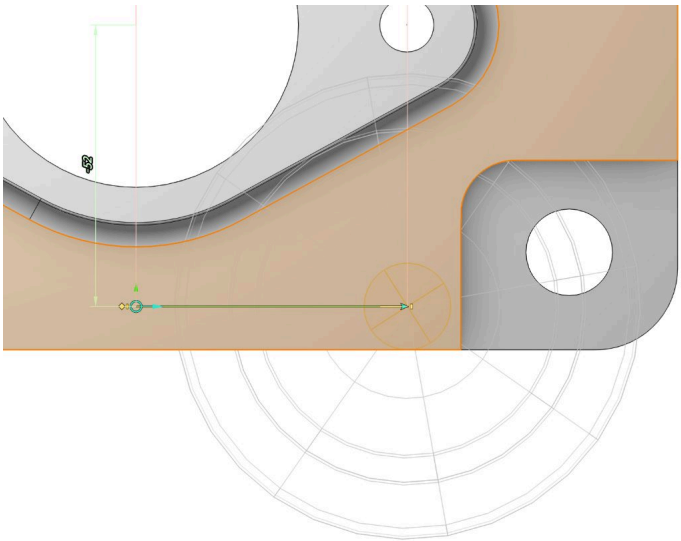




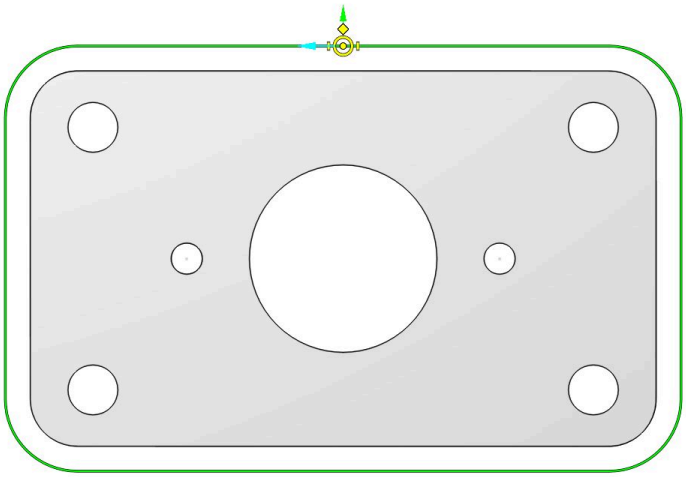
2. Contour start point (yellow circle). To change start point position just drag it with the mouse. You can also change start point position by X and Y dimensions. Just left click on the point and edit the dimensions that appear.

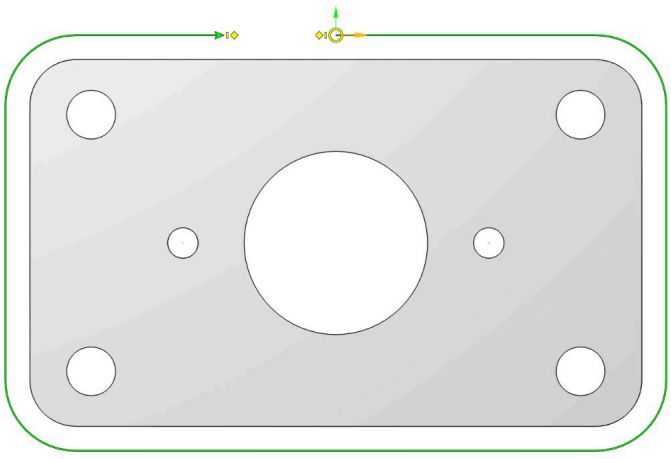


3. Contour terminate point (green arrow for an open contour and yellow dot in the middle of the yellow circle for a closed contour). You can drag the terminate point with the mouse, and you can change its position by X, Y dimensions.



The default start point position for closed contours is Auto. It means the position of a point is determined by the operation during toolpath generation. You can distinguish auto start points from regular ones by their visual representation. Auto start points are depicted without yellow dot inside, regular ones points are displayed with a yellow dot inside.





4. Contour direction. This is an aqua colored arrow attached to the start point at the above images. To change the contour direction just click on this arrow. The direction arrow can be of two colors: the aqua and the orange. The color of the direction arrow indicates milling mode of a contour, It's aqua for climb milling contours and orange for conventional milling contours.



5. Contour side of machining. This is a lime arrow attached to the contour start point at the images above. To change the side of machining just click on this arrow.

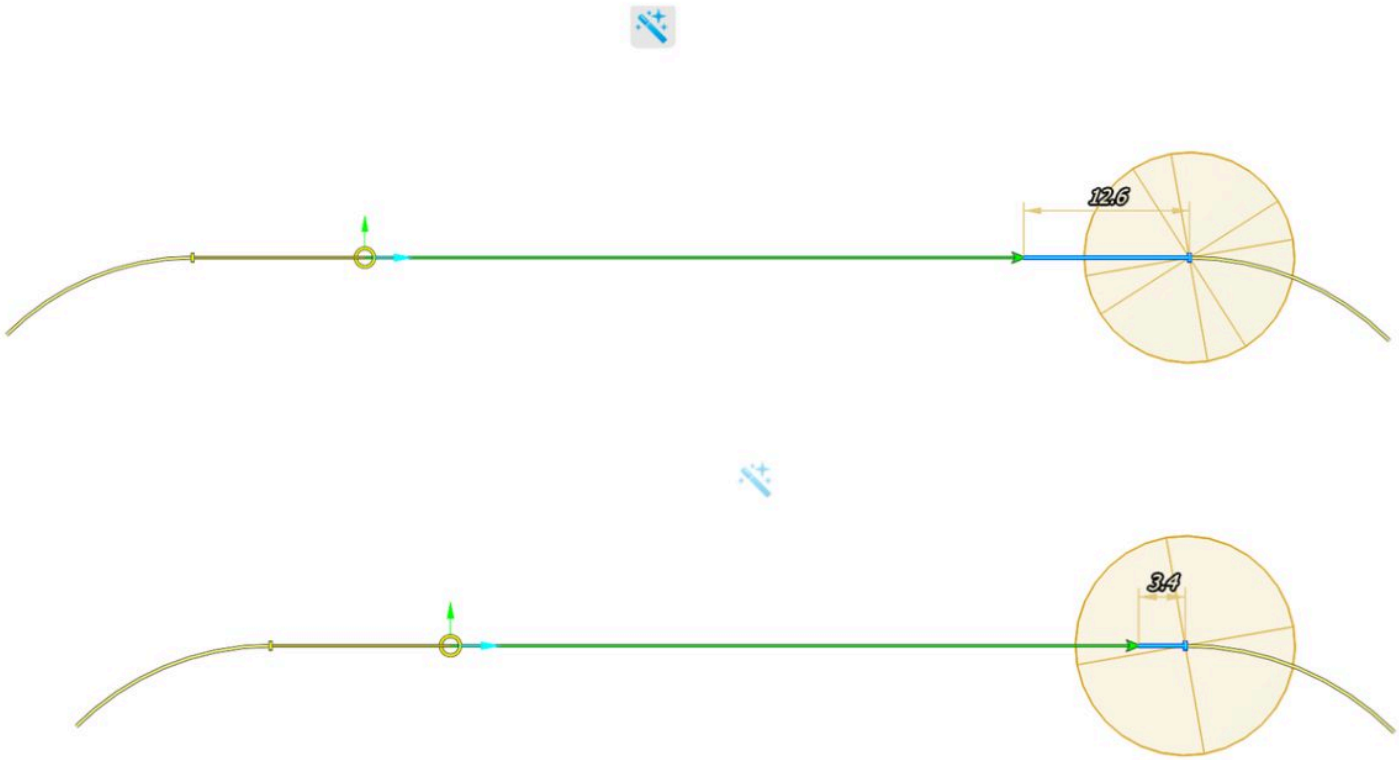


6. Contour overlaps. These are yellow dashes next to contour start and terminate points at the images above. An overlap is used to extend a toolpath beyond the contour itself. To change the overlap value you can either drag the yellow dash with the mouse or click on it and edit the dimension value which appears.

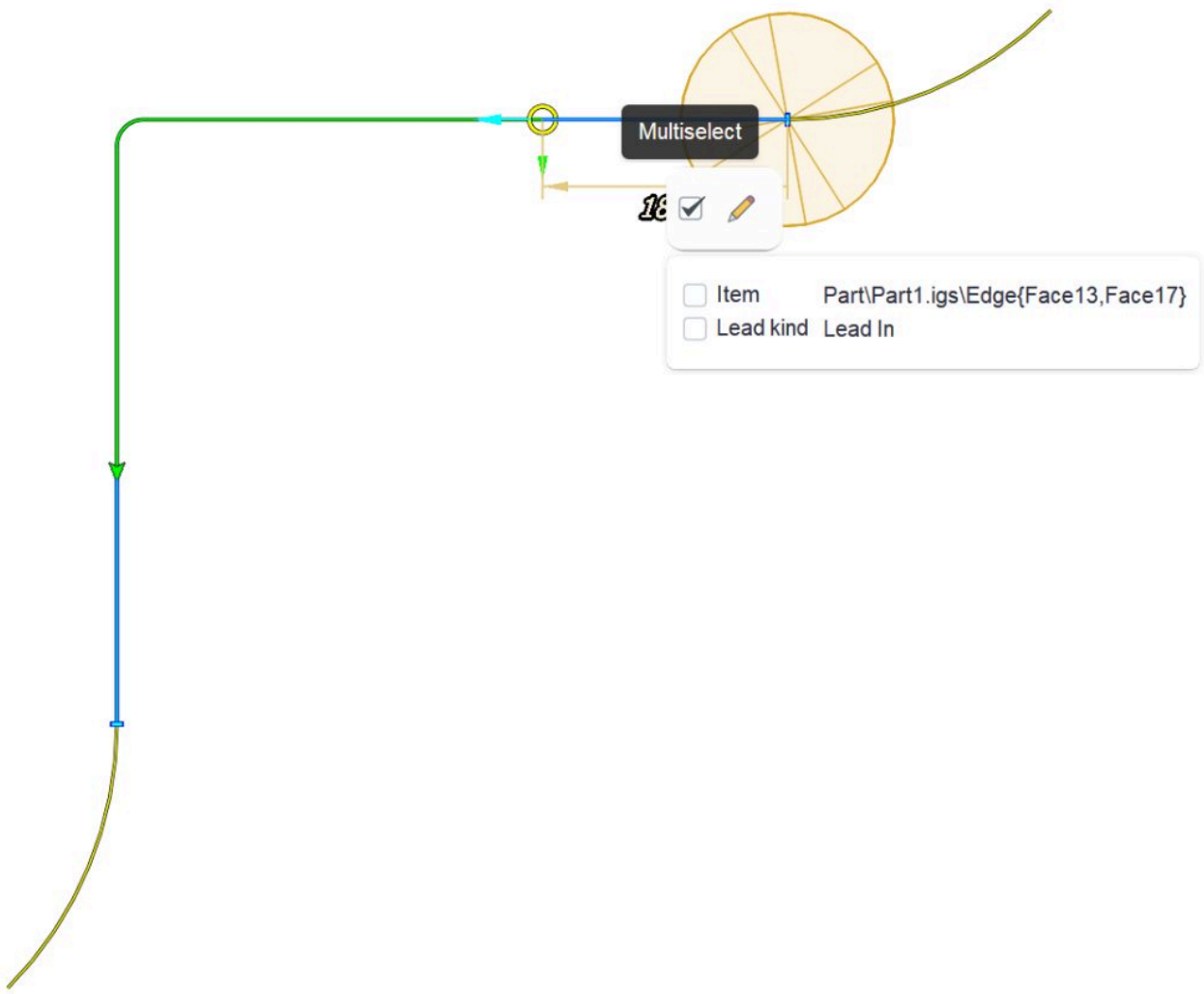


Editing multiple overlaps in one click

You should know that when you edit an overlap parameter this change applies to all the selected overlaps. To make the process of editing multiple overlaps quick and simple CAM system automatically highlights and selects all identical overlaps. However you can always disable this feature. Just click on the magic wand button at the top of the graphic view.

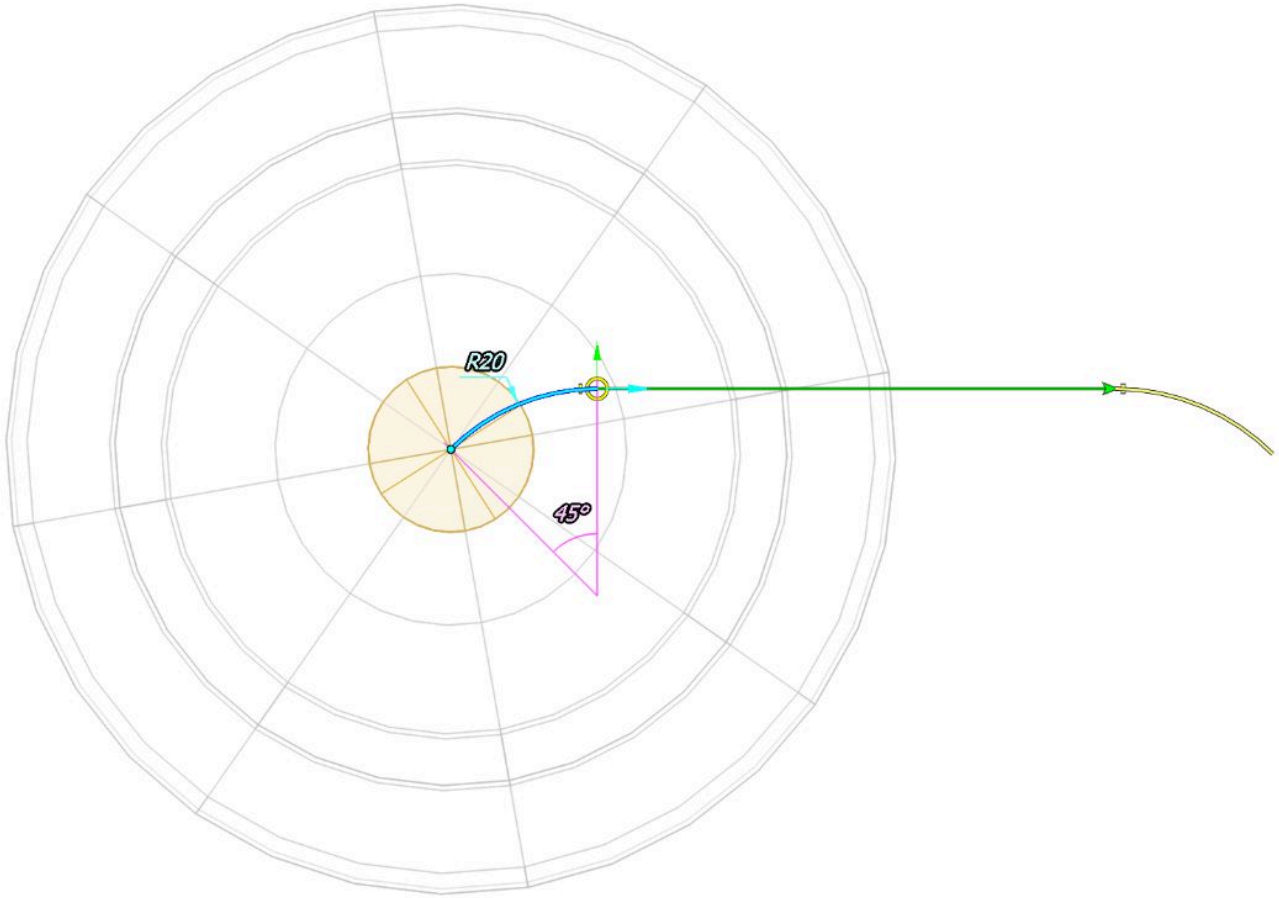


To select overlaps by a criteria you should click on one overlap, than on the check box in the action bar which appears and check the criteria on which you want select overlaps.

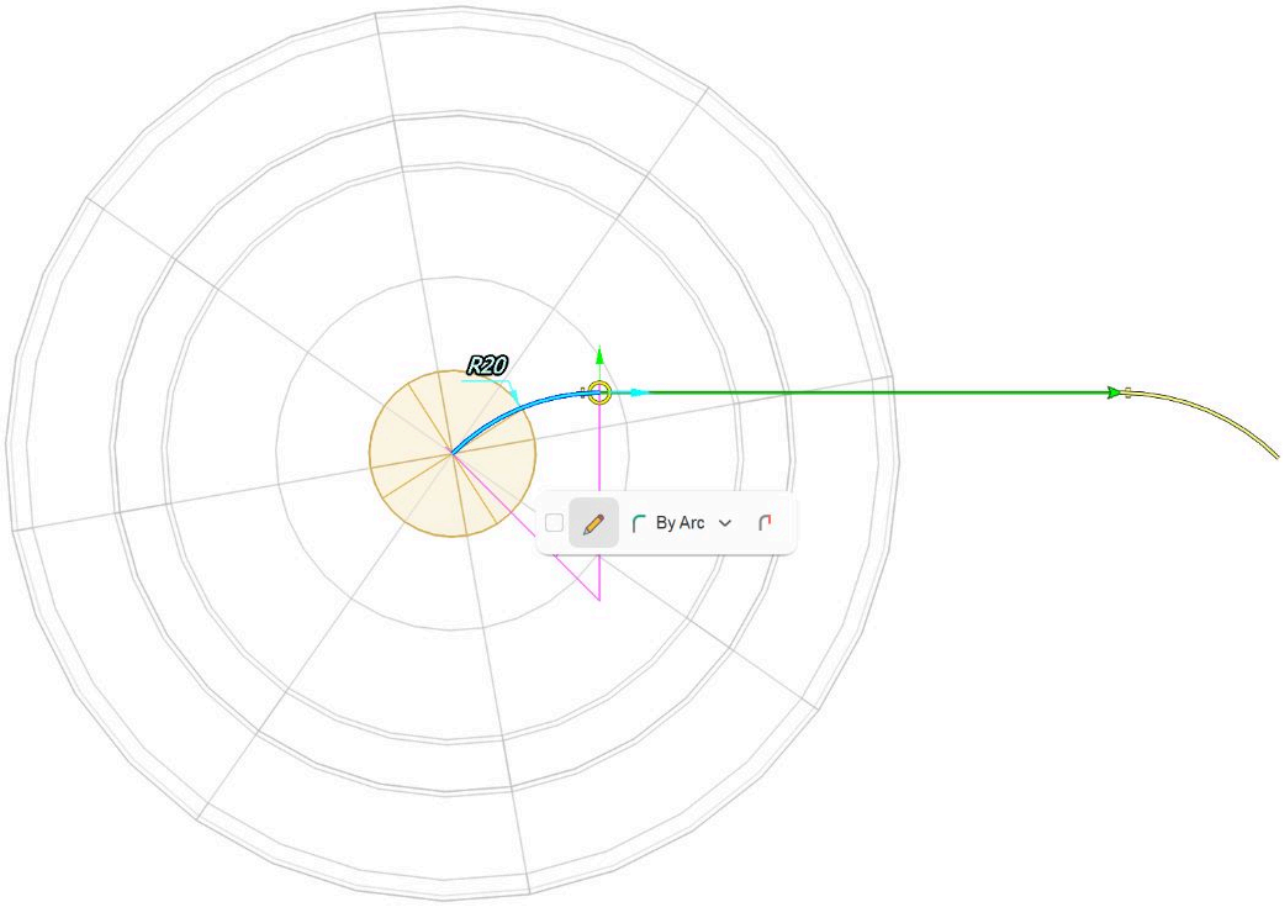


7. Contour leads. These are yellow diamonds at the above pictures. To change a lead just drag the corresponding diamond with the mouse. After that you can edit lead dimensions that appear.

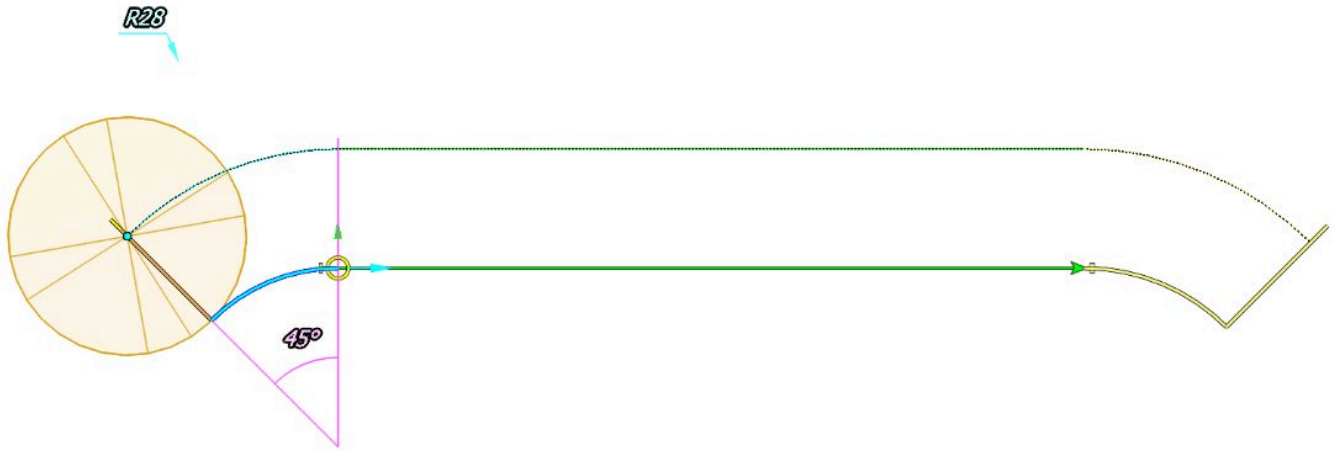




There are three lead types available: none, line and arc. To change a lead type, left click on a lead and select the lead type from the action bar that appears.

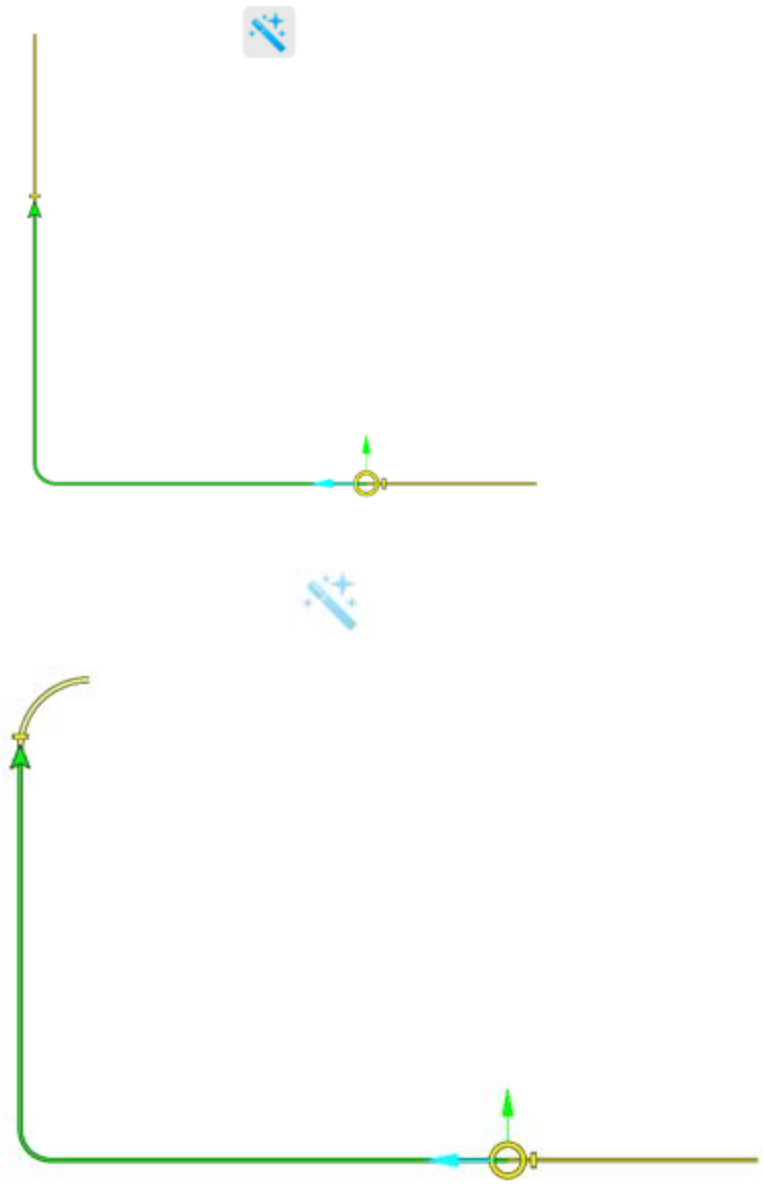


For compensated contours and Control radius compensation an additional dashed line is displayed. This line represents the tool center curve. Lead dimensions are specified for the tool center curve. This makes lead parameters independent of the current compensation type. If you want to change the size of a compensated lead using drag and drop action you should click on the yellow dot where the tool center curve is ending, and not where the uncompensated lead curve is ending.

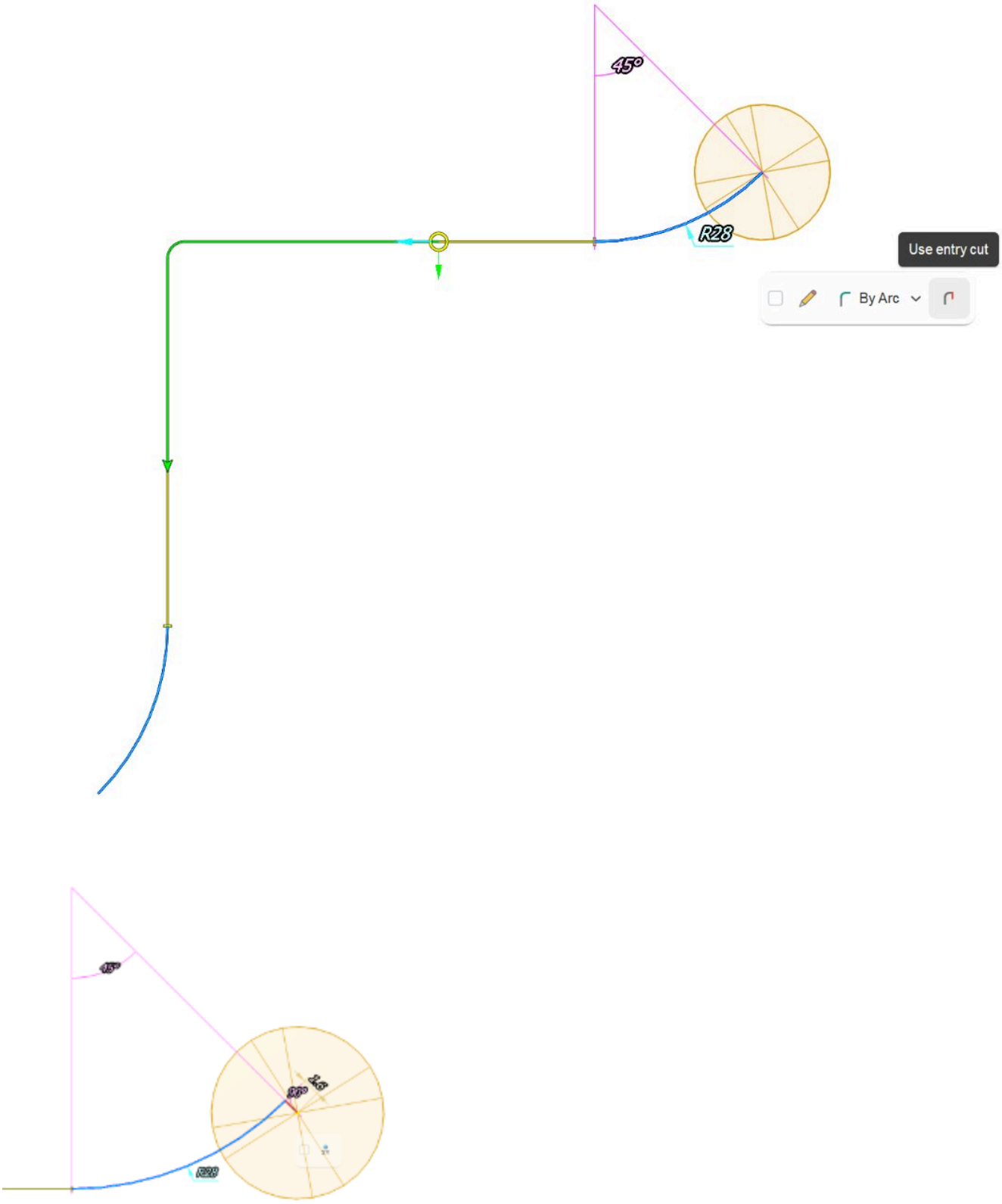


Editing multiple leads in one click

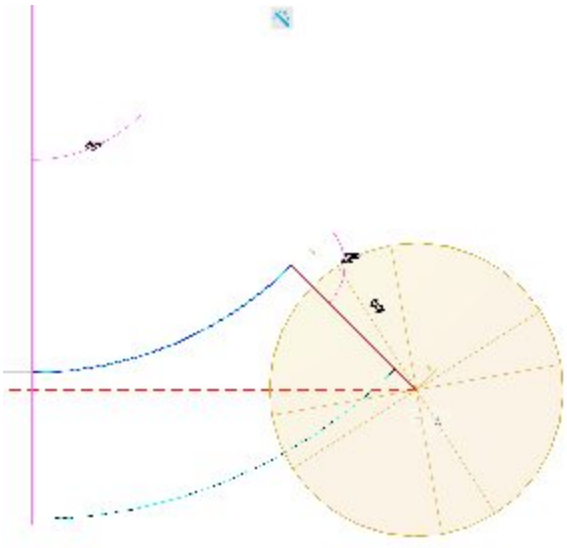
You should know that when you edit a lead parameter this change applies to all selected leads. To make the process of editing multiple leads quick and simple CAM system automatically highlights and selects all identical leads. You can disable this behavior by clicking on the magic wand button at the top of the graphic view. To select only one lead you should click on it twice. To select leads by a criteria you should click on one lead, than on the check box in the action bar which appears and check the criteria on which you want select leads.



8. Contour entry lines. Entry lines are automatically added to the leads of the compensated contours when the Compensation type is other than Computer. They play the role of compensation switch lines. Entry lines are not added to leads by line, as that makes no sense. But you can also switch entry lines explicitly by clicking on a lead and checking the Use entry line button in the action bar that appears.

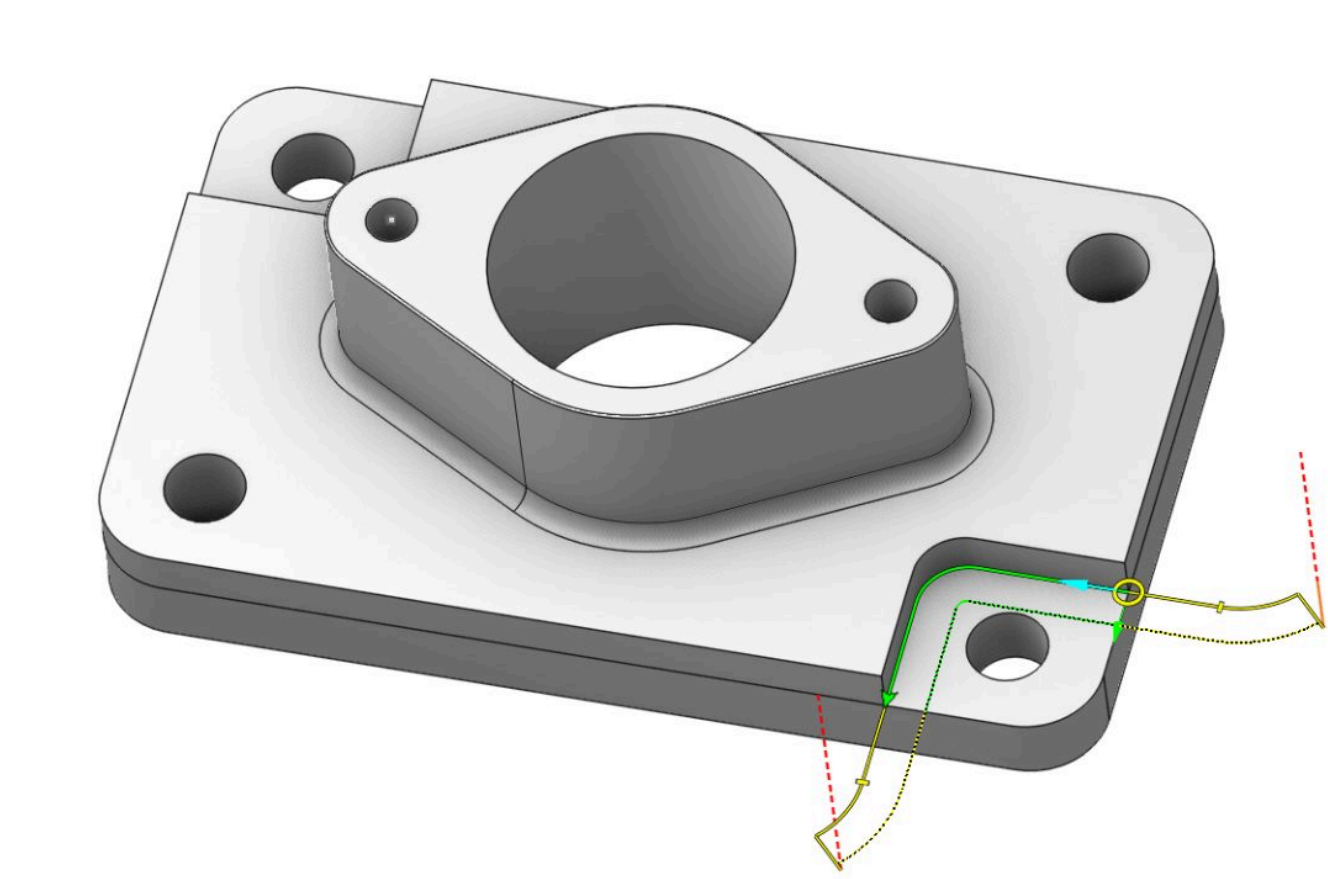


Here is an example of entry lines with Control radius compensation.



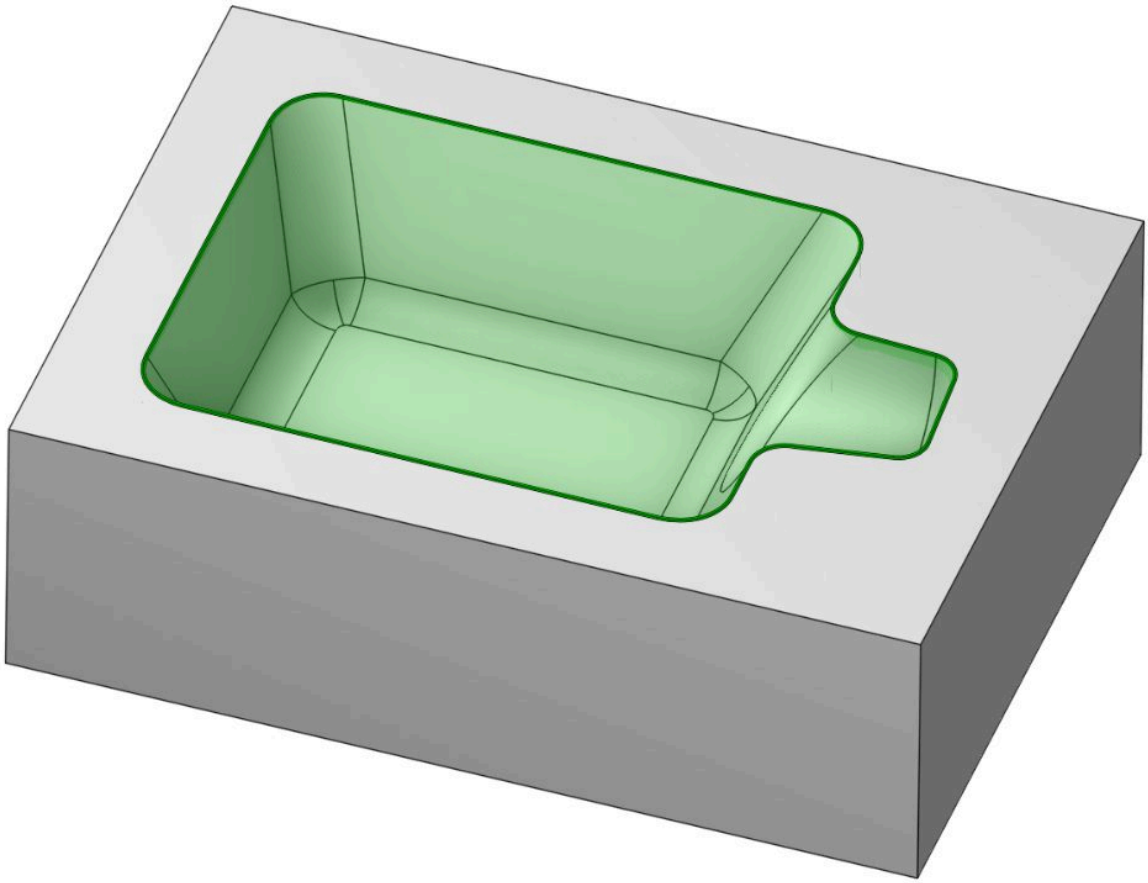
9. Contour offsetting/radius compensation

Contour radius compensation is generally used when you have a part line curve (e.g. an edge or a face of a vertical wall) and want to machine material from a side of this part line. To turn the contour radius compensation on just select a contour in the graphic view and click the Compensated button in the action bar that appears. The tool maximal radius is used by default as the contour offset radius. But you also can specify the tool contact point explicitly in the Tool parameters. In that case the contour offset distance will be taken from the tool contact point.



The method of radius compensation is determined by the operation parameter Radius compensation. You can select it either in the parameters inspector or in the operation parameters windows. For example for 2D contouring - Output. Details

Job zones



Application Area:

Job zones are used to define the part and workpiece areas that have to be machined by roughing and finishing milling operations. Job zones can be closed and open. Open job zones first introduced in legacy CAM system offer an easy and intuitive way to define open machining areas using only source 3d model entities.

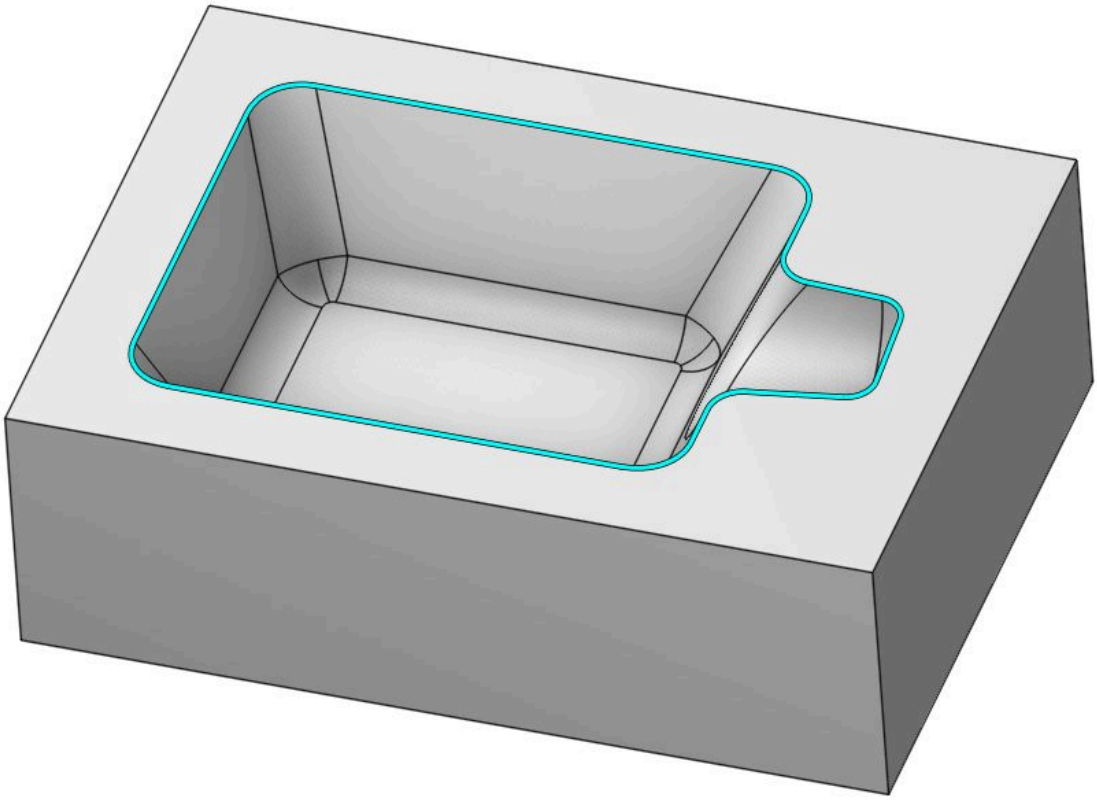
Working with Job zones.

Using closed job zones

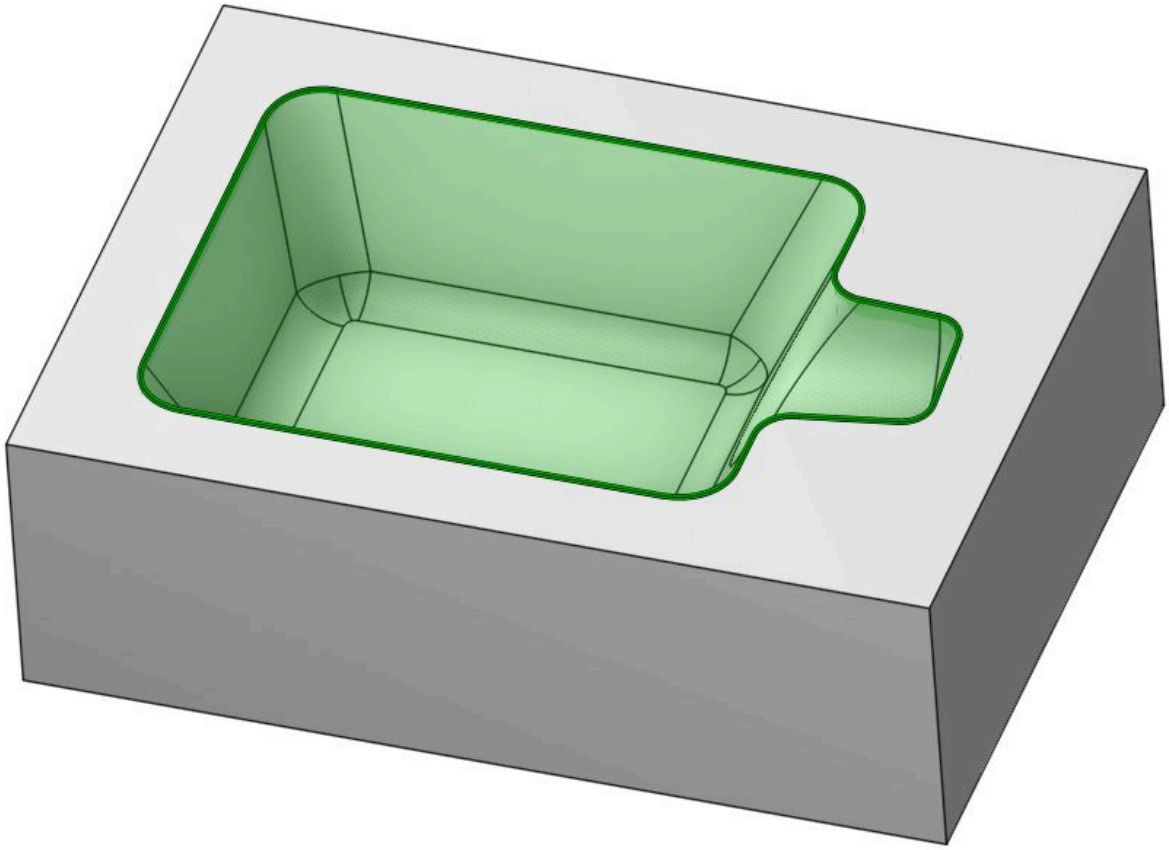
As a source geometry for closed job zones can be used closed chains of curves, edges and vertical walls. The best practice of using closed job zones includes the following steps.

- In the graphic view select geometry entities defining the job zone. The easiest way is to use 3d Model edges. You can easily select the whole chain of tangent edges by simple double click on an

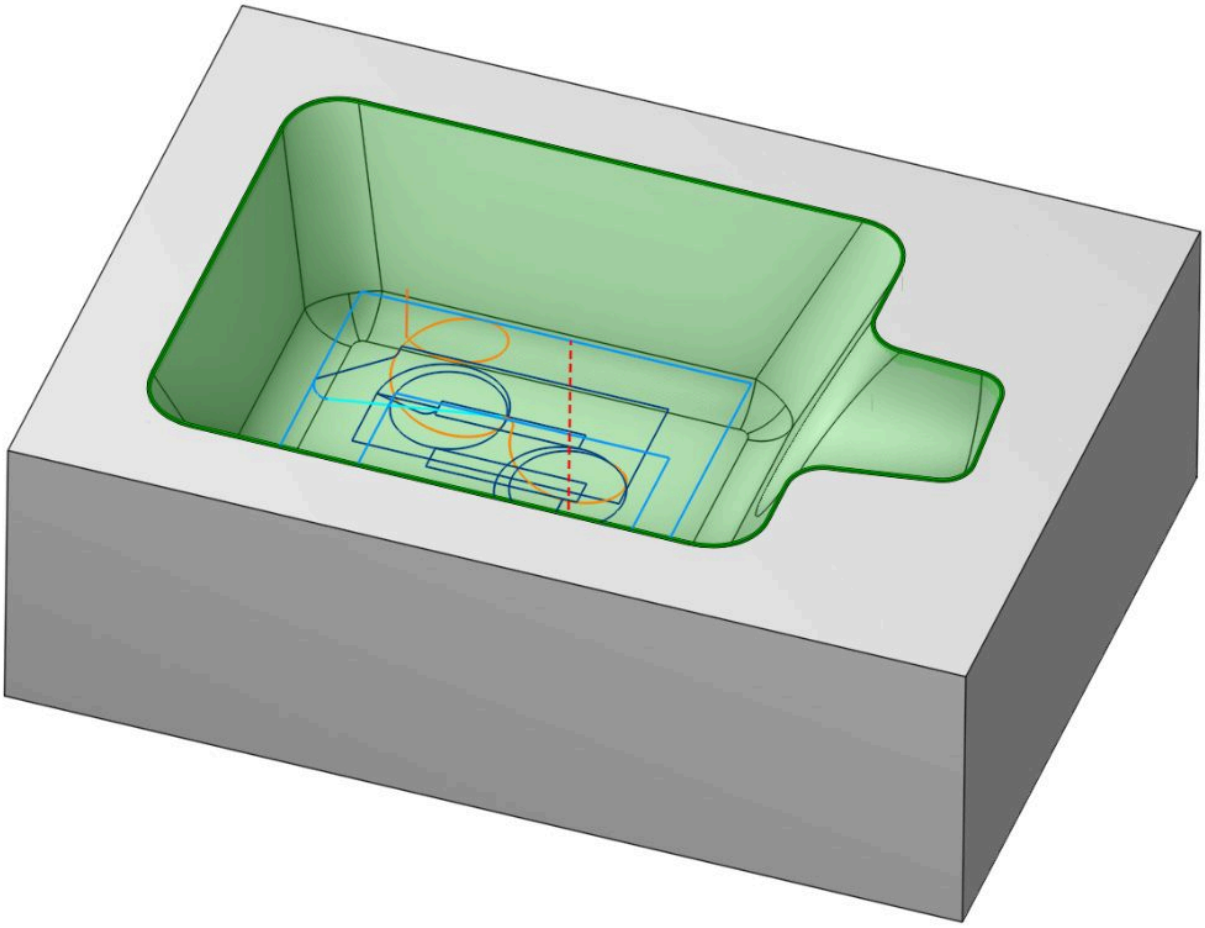
edge of a chain.



- At the Job Assignment panel press the Job Zone button.



- Generate toolpath. The trajectory should be contained inside the Job Zone limits.



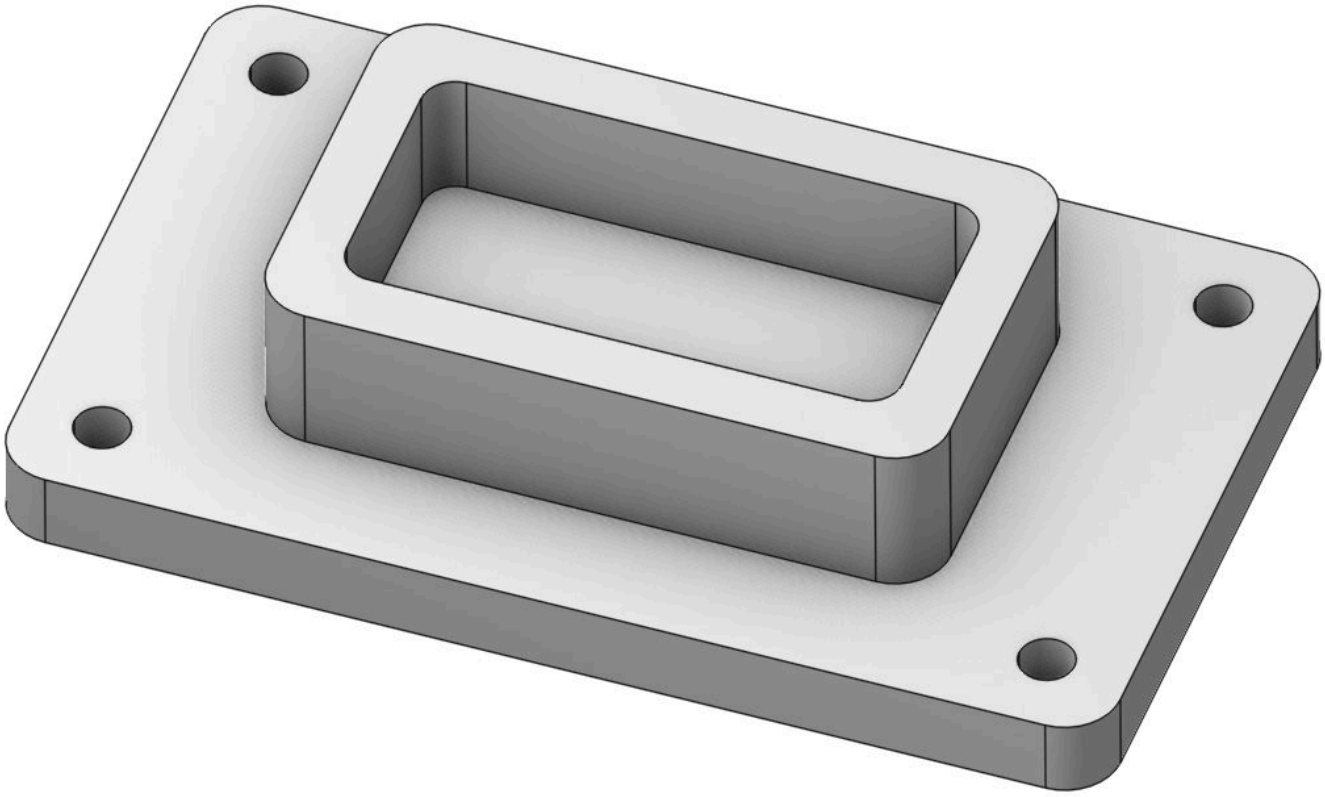
Using open job zones

An open job zone is a half-space constructed by one or more entities. As source geometry for an open job zone you can use distinct curves, 3d edges and vertical walls as well as connected chains.

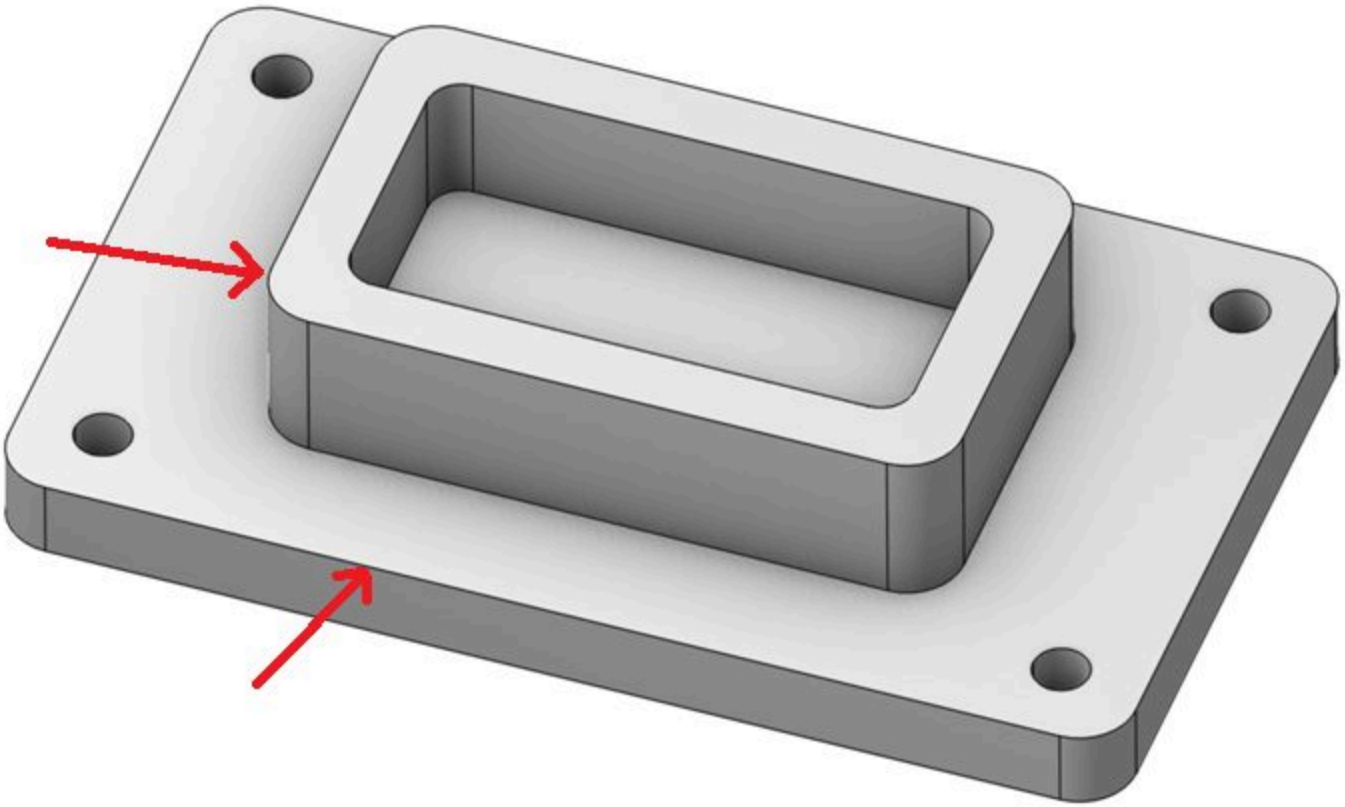
The best way to understand how and when to use open job zones is to look at practical examples.

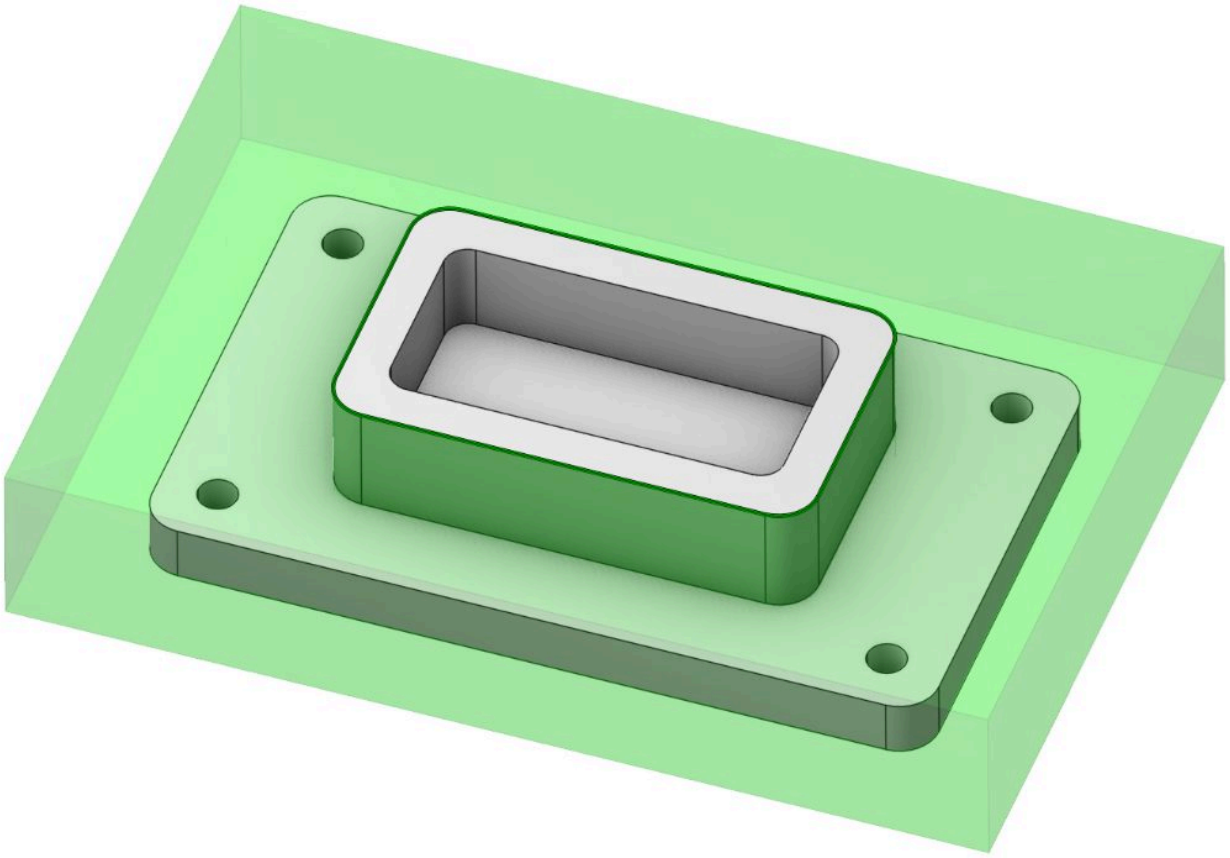
Using open job zones in a roughing operation

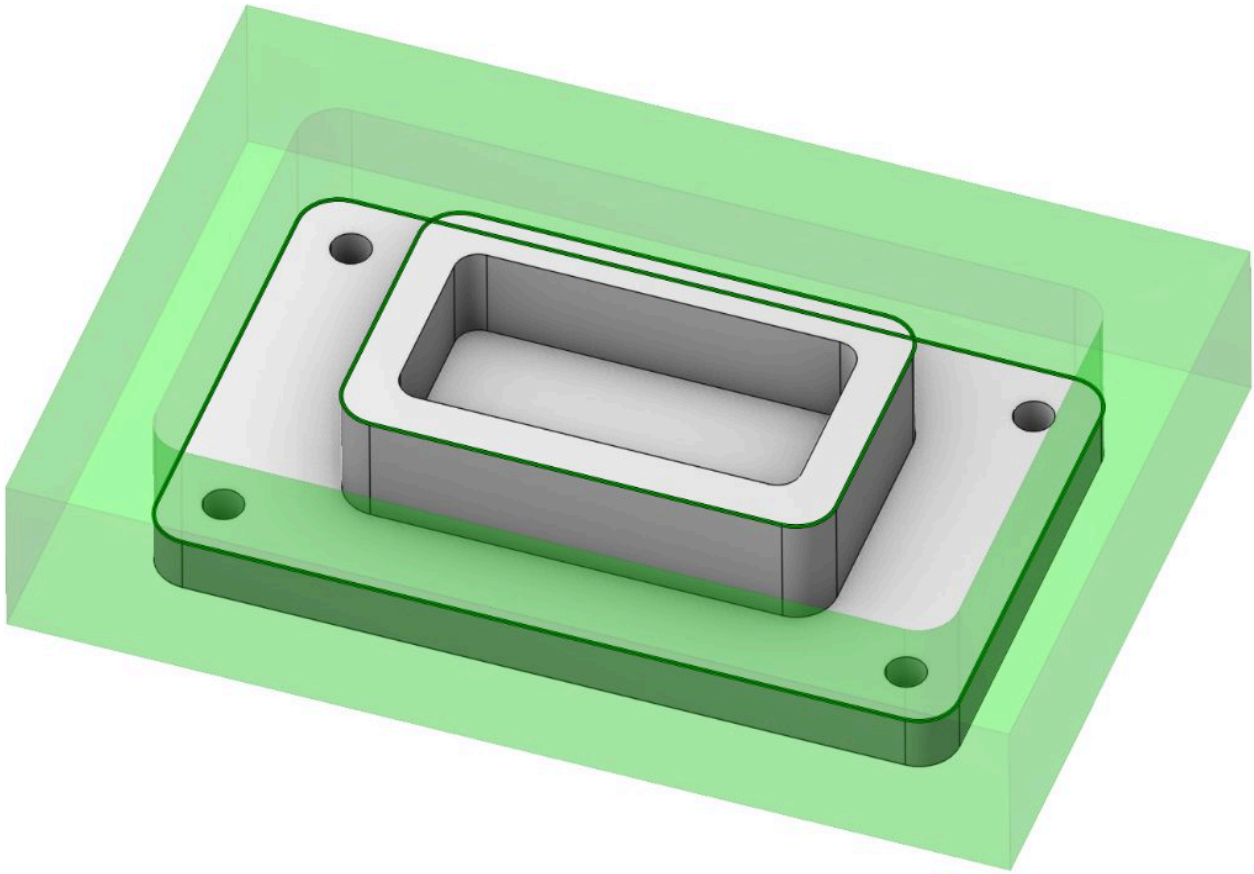
As a demonstration, we will attempt to constrain the work area with curves to prevent the machining of the central pocket and to keep the machining zone within the part's profile.



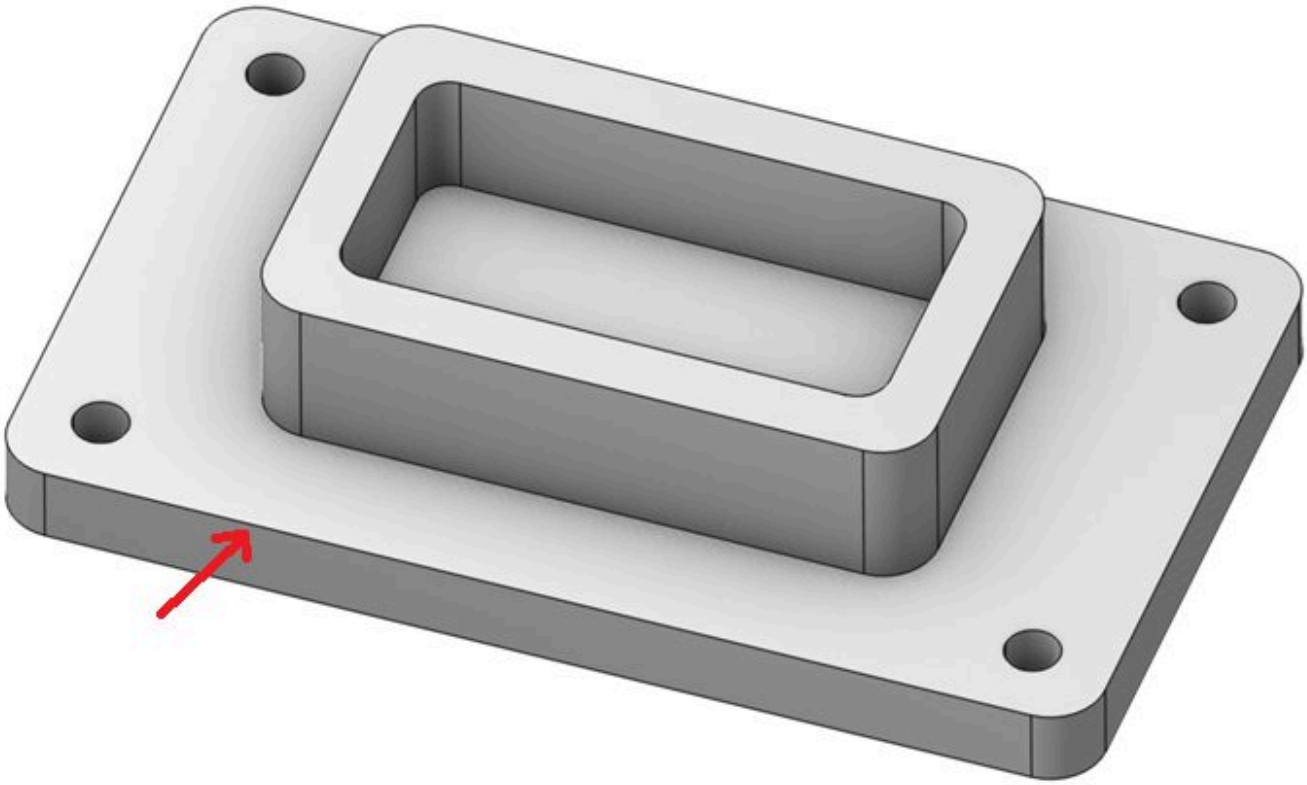
For this, we will select two edges as Job zones.

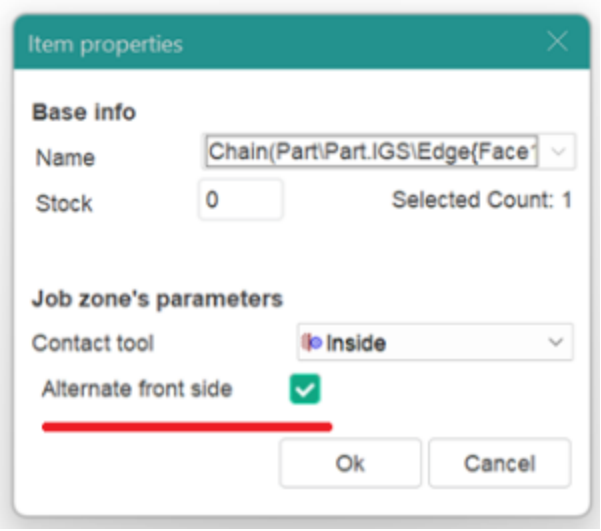




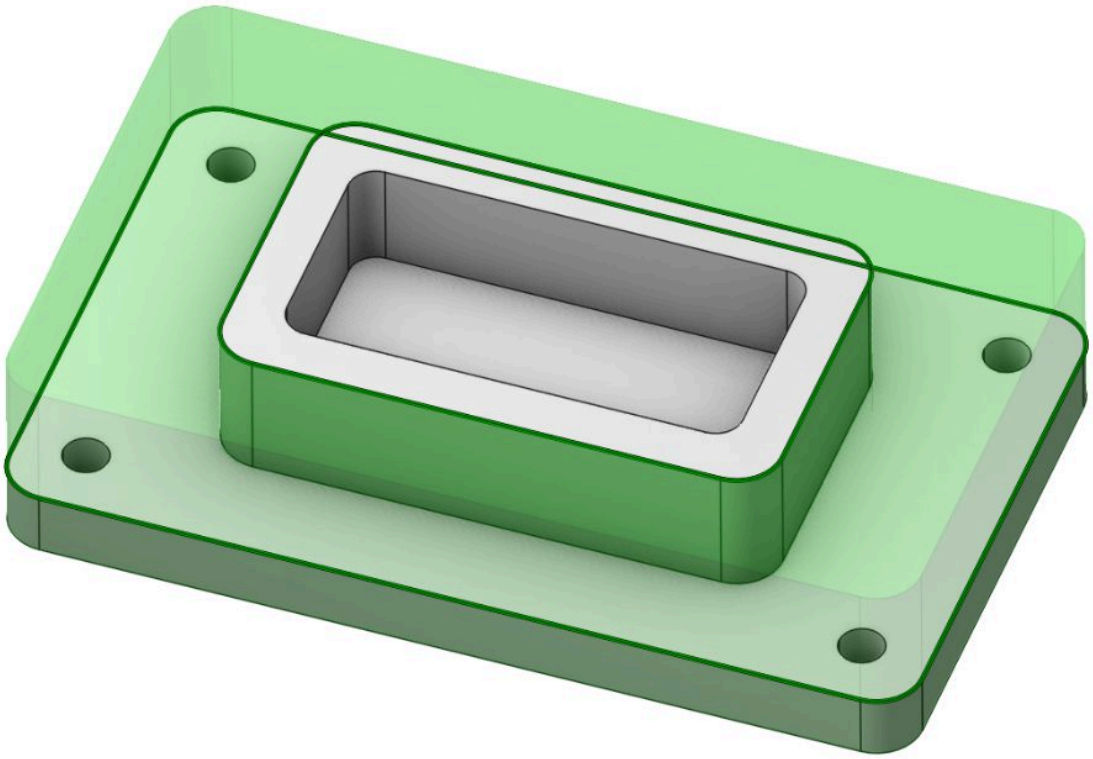


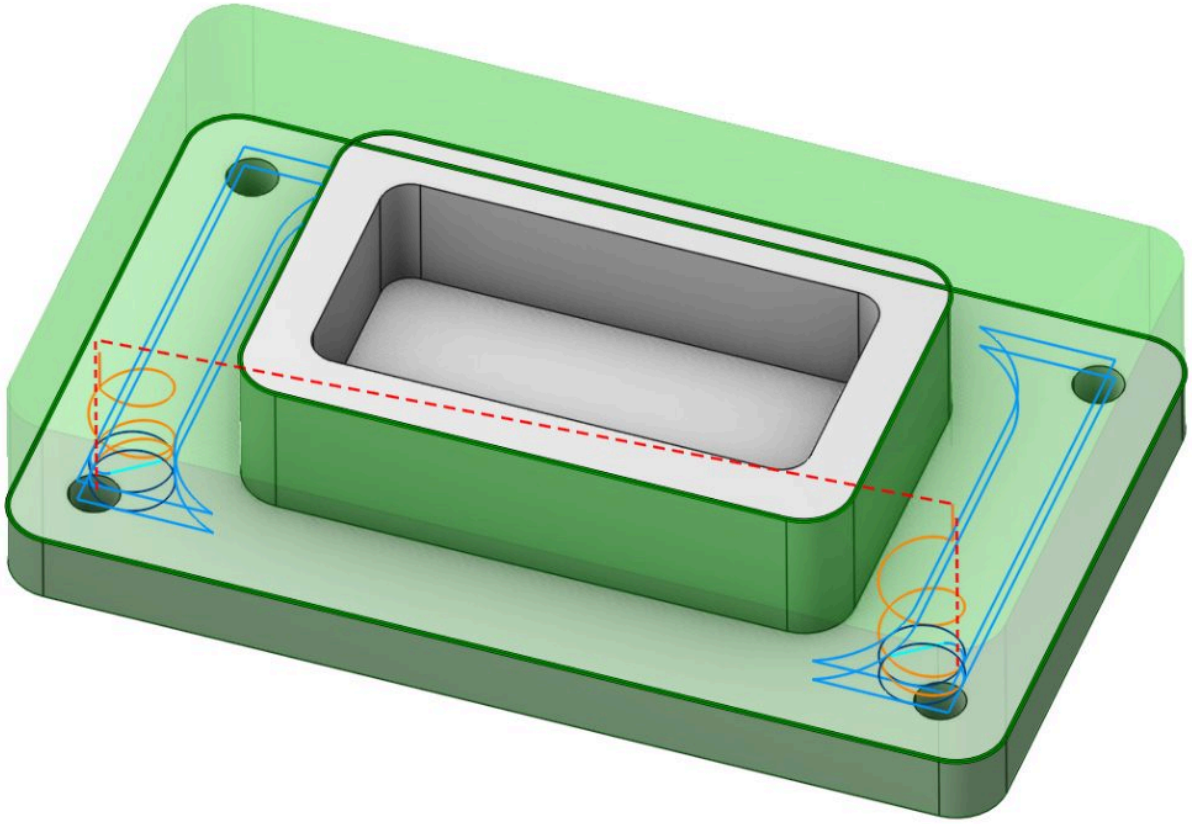
We can see that one of the zones is facing the opposite direction. It needs to be inverted. In the work assignment, we find this zone. We double-click on it and change its direction.





Now the result obtained corresponds to the task. You can apply these skills when choosing a work assignment for your parts.

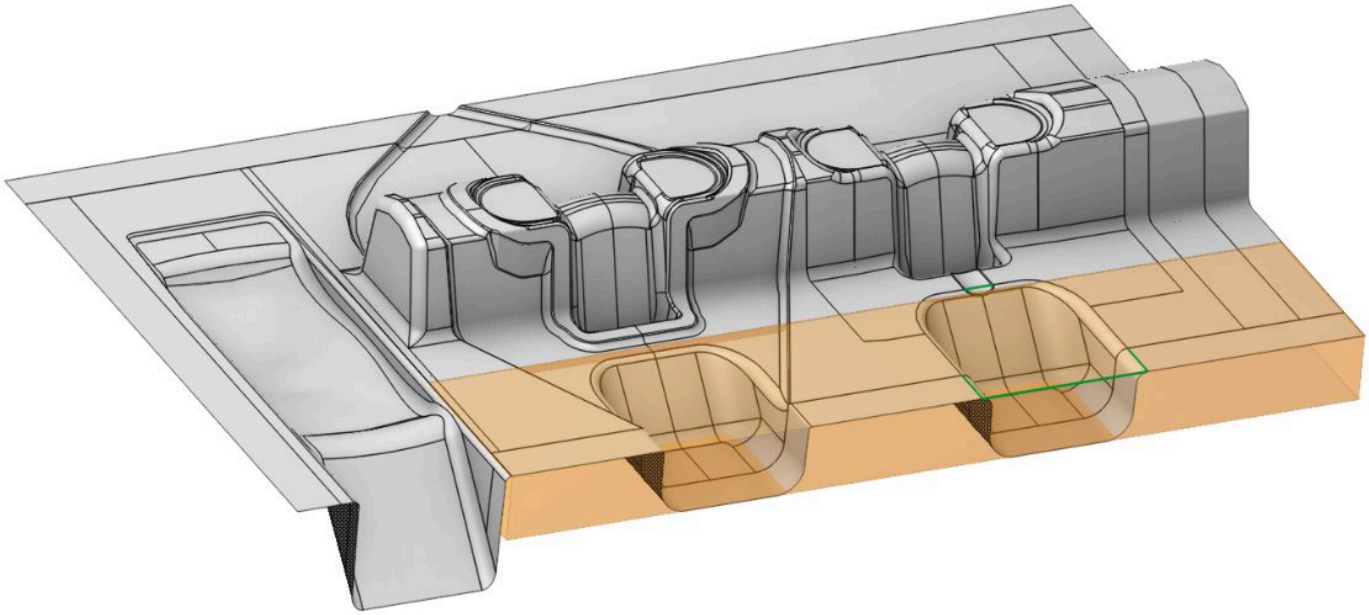




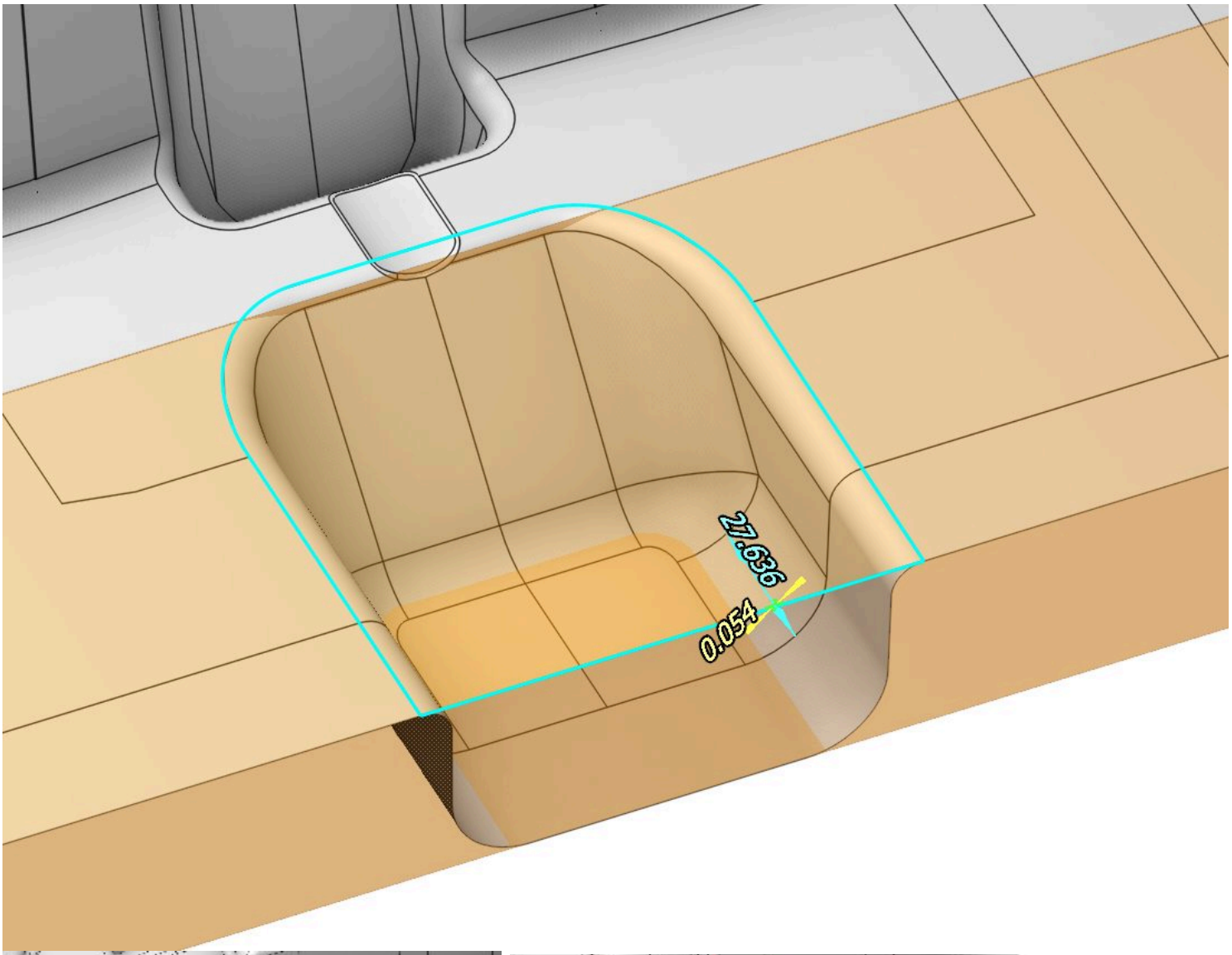
Using open job zones in a finishing operation

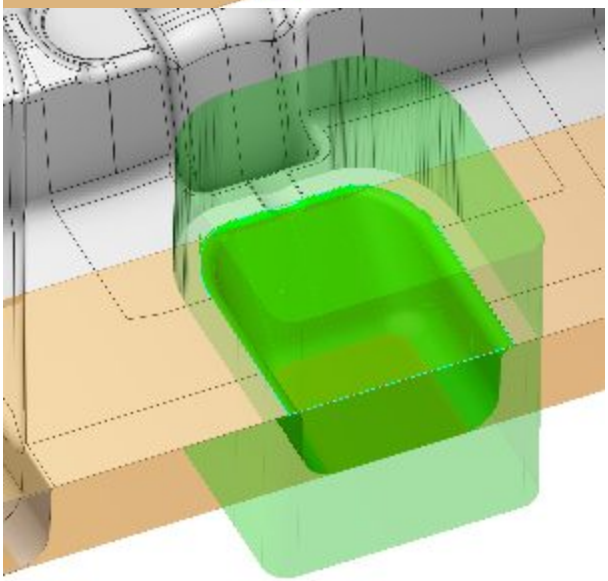
At the picture below you can see a die with three open freeform pockets. We will machine one small pocket using Finishing plane operation.

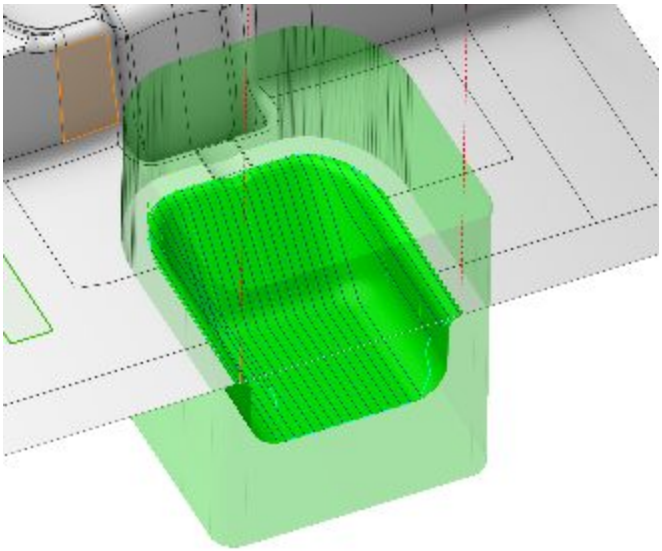
To accomplish this task we just need select the edge chain around the pocket and add it into the job assignment as a Job Zone.



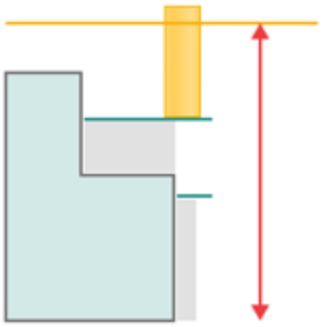
After that we generate toolpath. The result should look as follows.

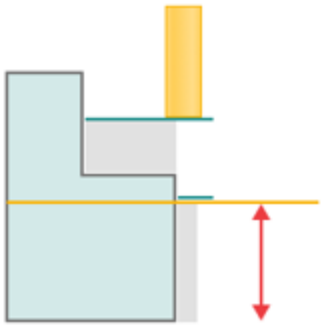

27.636
0.054





Top and bottom levels





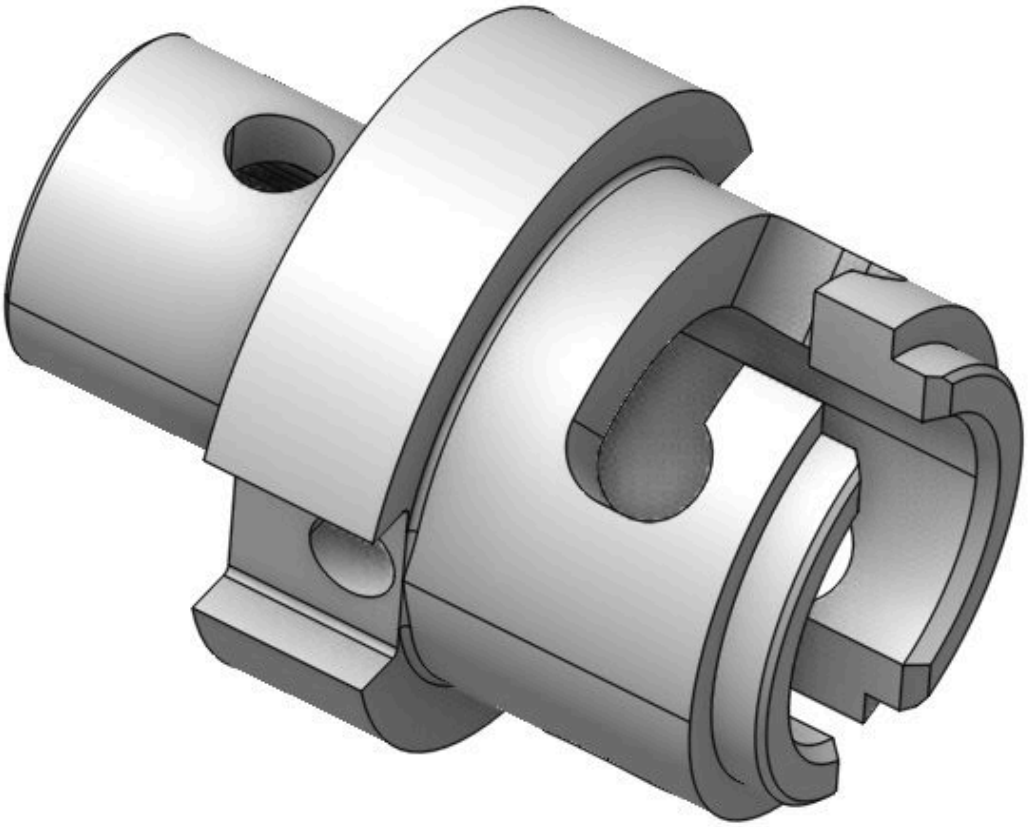
Application Area:

The common task while machining prismatic and turn-milling parts is to define the Top and Bottom level parameters of an operation based on the geometrical position of the machined feature. The most straightforward way to accomplish this task is to use the corresponding items of the Job Assignment.

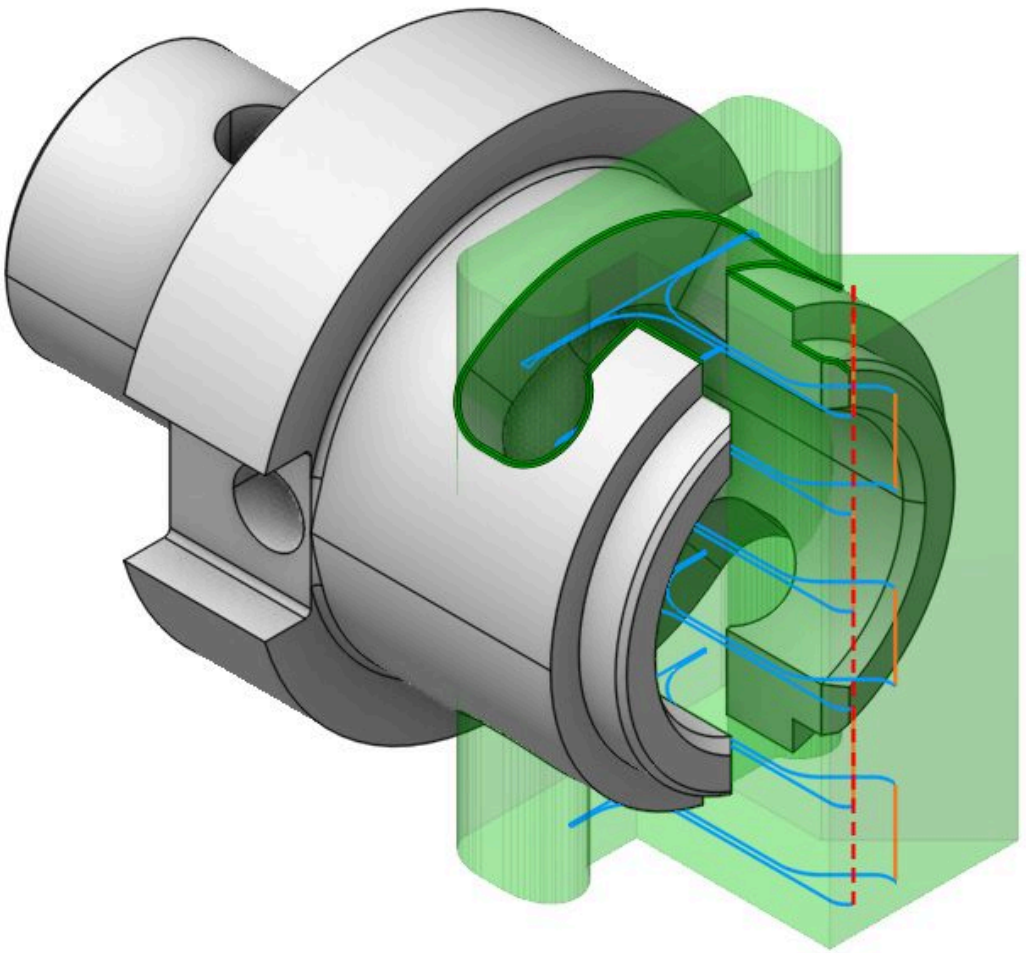
As source geometry for the Top and Bottom Level, items of the Job Assignment can be used on any 3d Model entities, including groups. The actual values of the levels are computed from the bounding box of the geometry entity. In addition, you can set a positive or negative stock in the item property dialog.

Working with Job zones.

In the following picture you can see a turn-milling part. We will machine the front open slot.

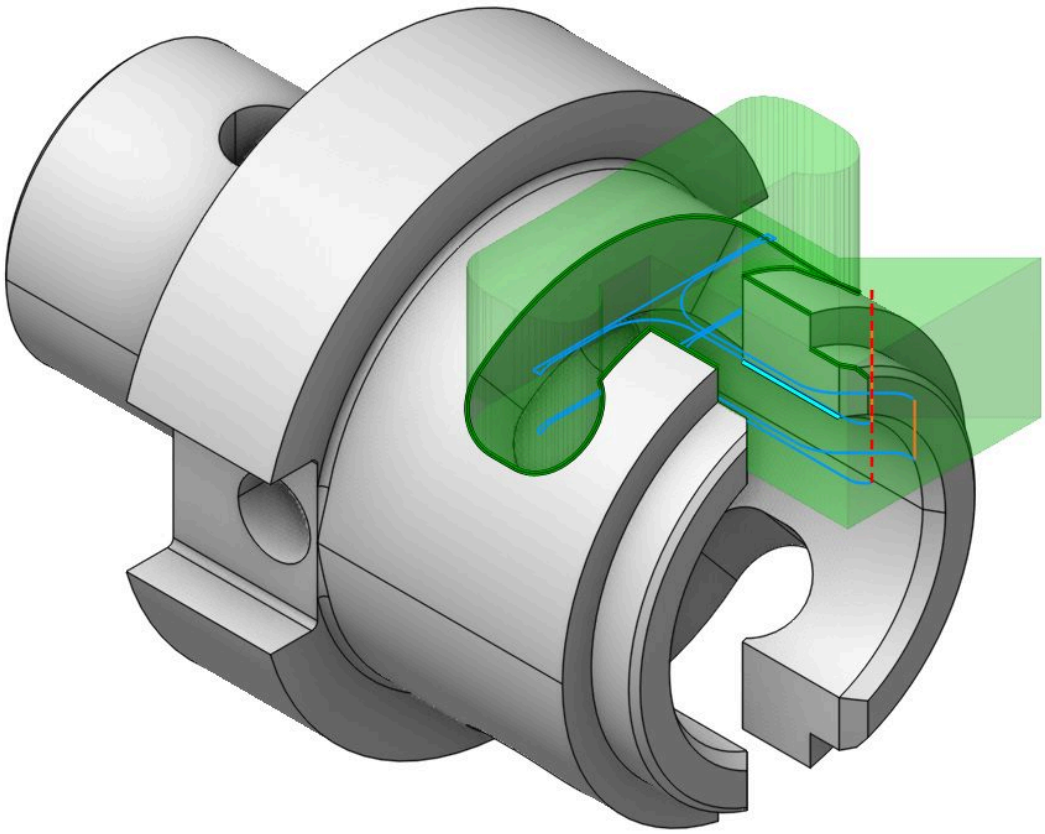


Using open job zones, we construct the machining zone in the XY plane. As you can see in the picture below, we have to limit the toolpath by Z. Both top and bottom levels have to be adjusted.

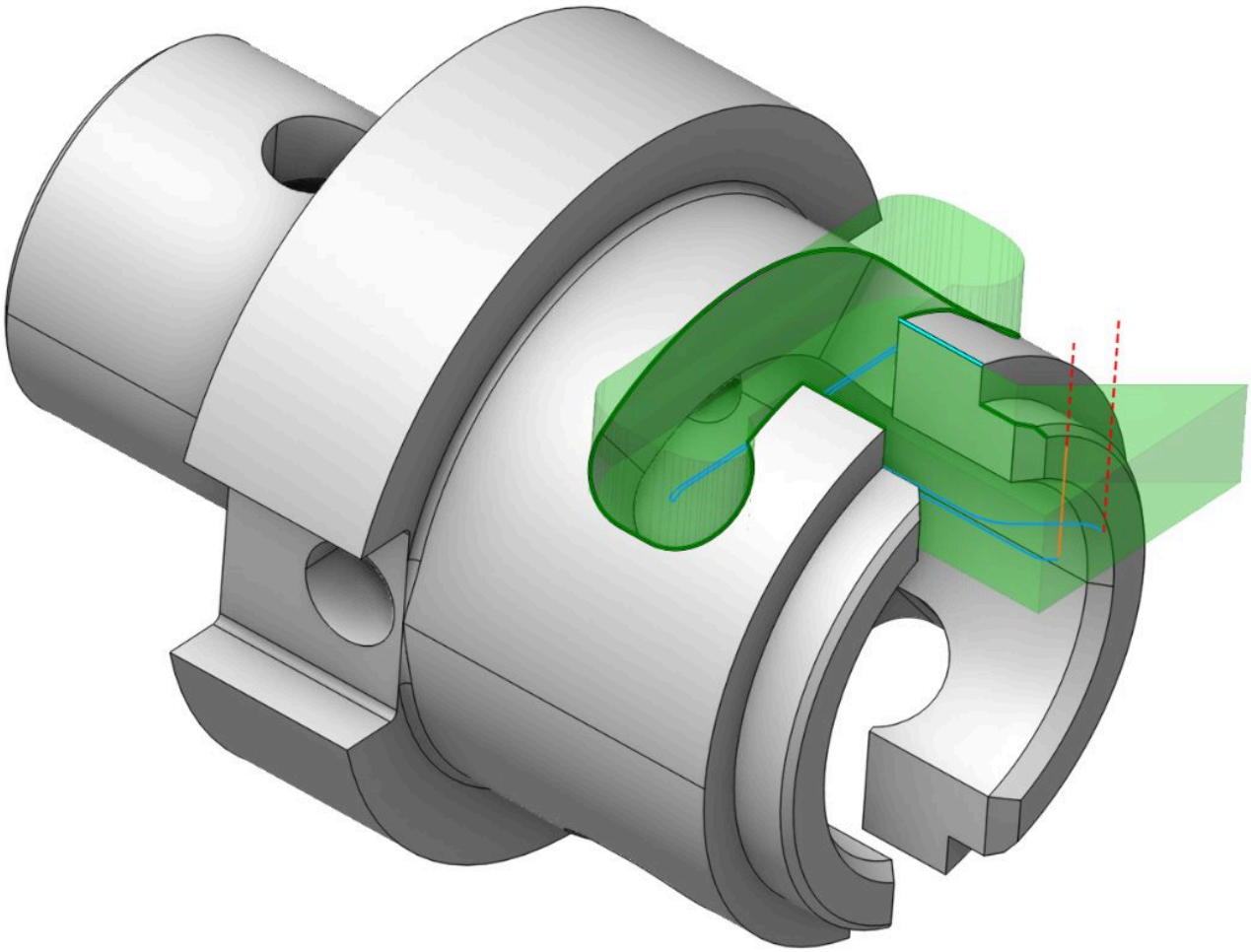


To accomplish this task we will add the side face of the slot as the Bottom Level item into the Job Assignment.

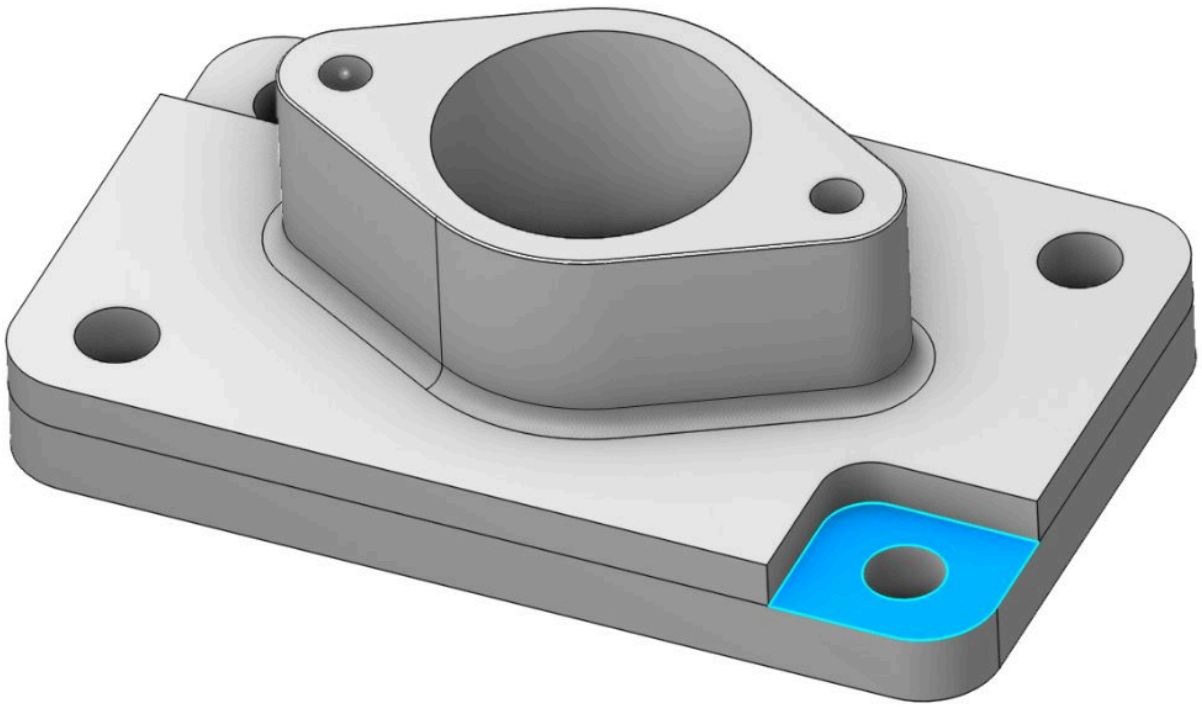
The generated toolpath will look like depicted below.

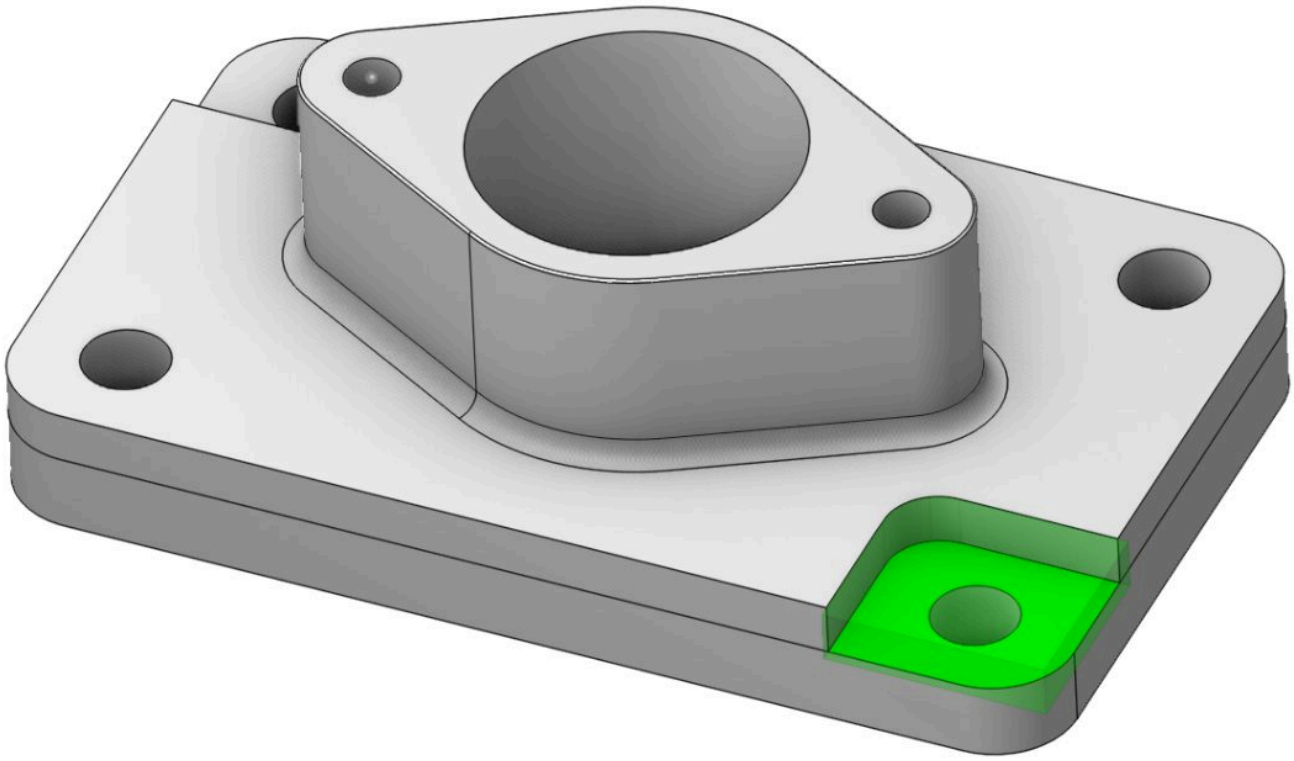


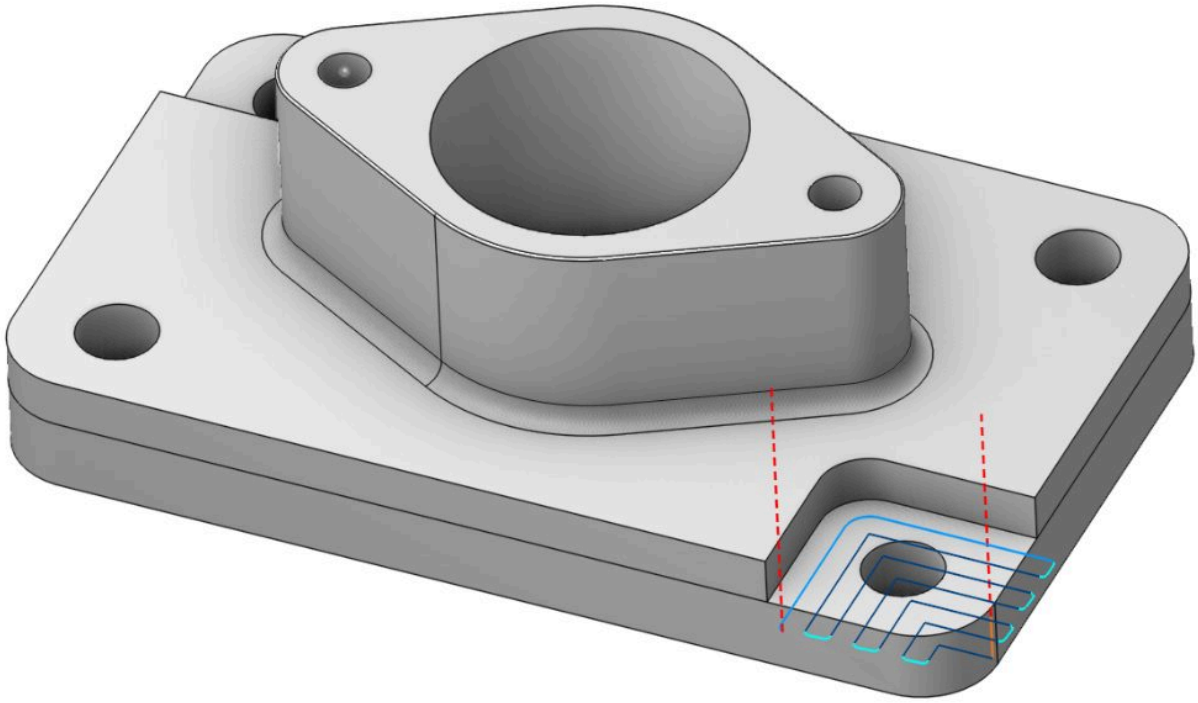
To limit toolpath from top we add the outer diameter of the slot as the Top Level item into the Job Assignment.



Pocket feature







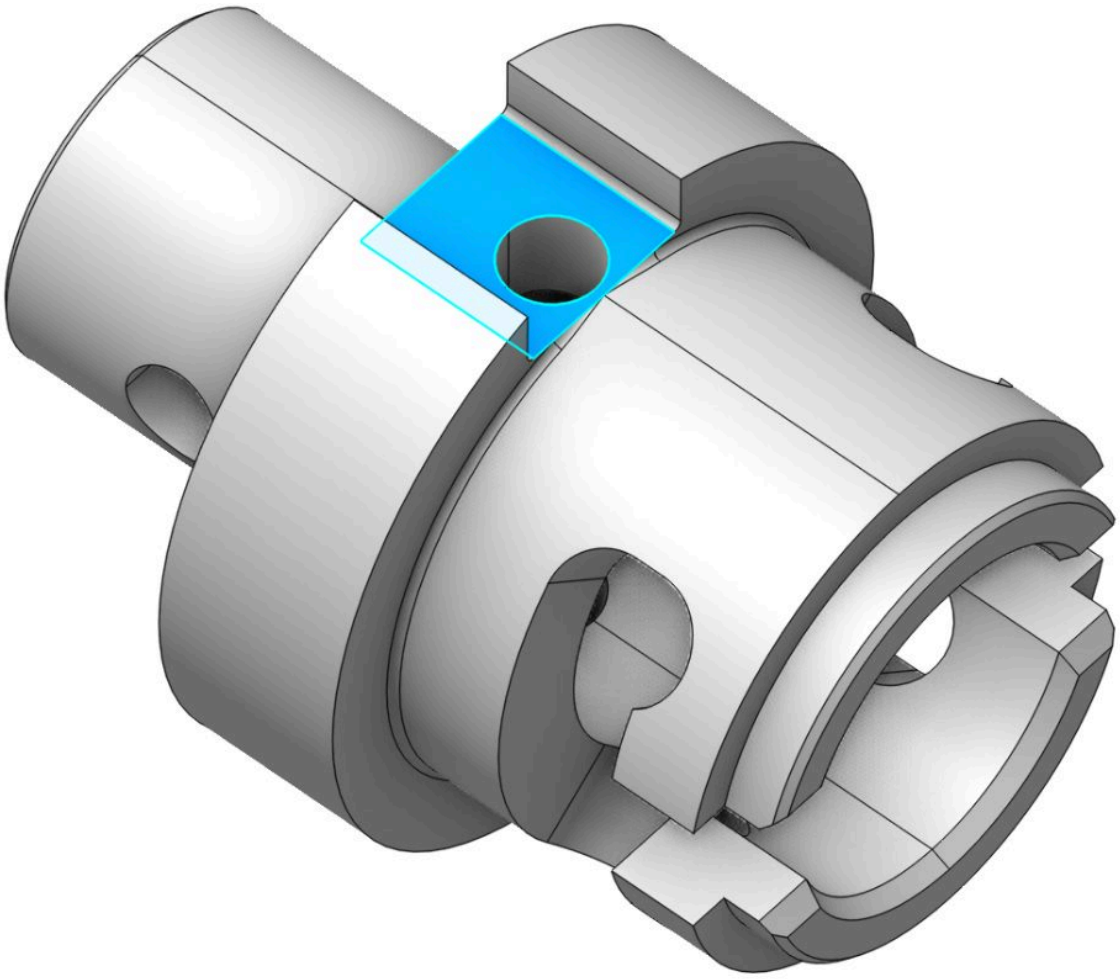
Application Area:

A typical prismatic or turn-milling part consists of many simple shapes. Creating technology for such parts using only job zones, restrict zones and machining levels is very time consuming process. To simplify this task, CAM system can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels.

The Pocket is a 2d machining feature that consists of 2d Job Zone, Top level and Bottom level, local coordinate system and associated attributes such as corner and fillet radii. The pocket feature is recognized from a base surface that can be either a flat bottom face of a pocket or a side face of a through pocket. You can add a Pocket feature item into the Job Assignment of any 3d milling operation and 2d contouring operation.

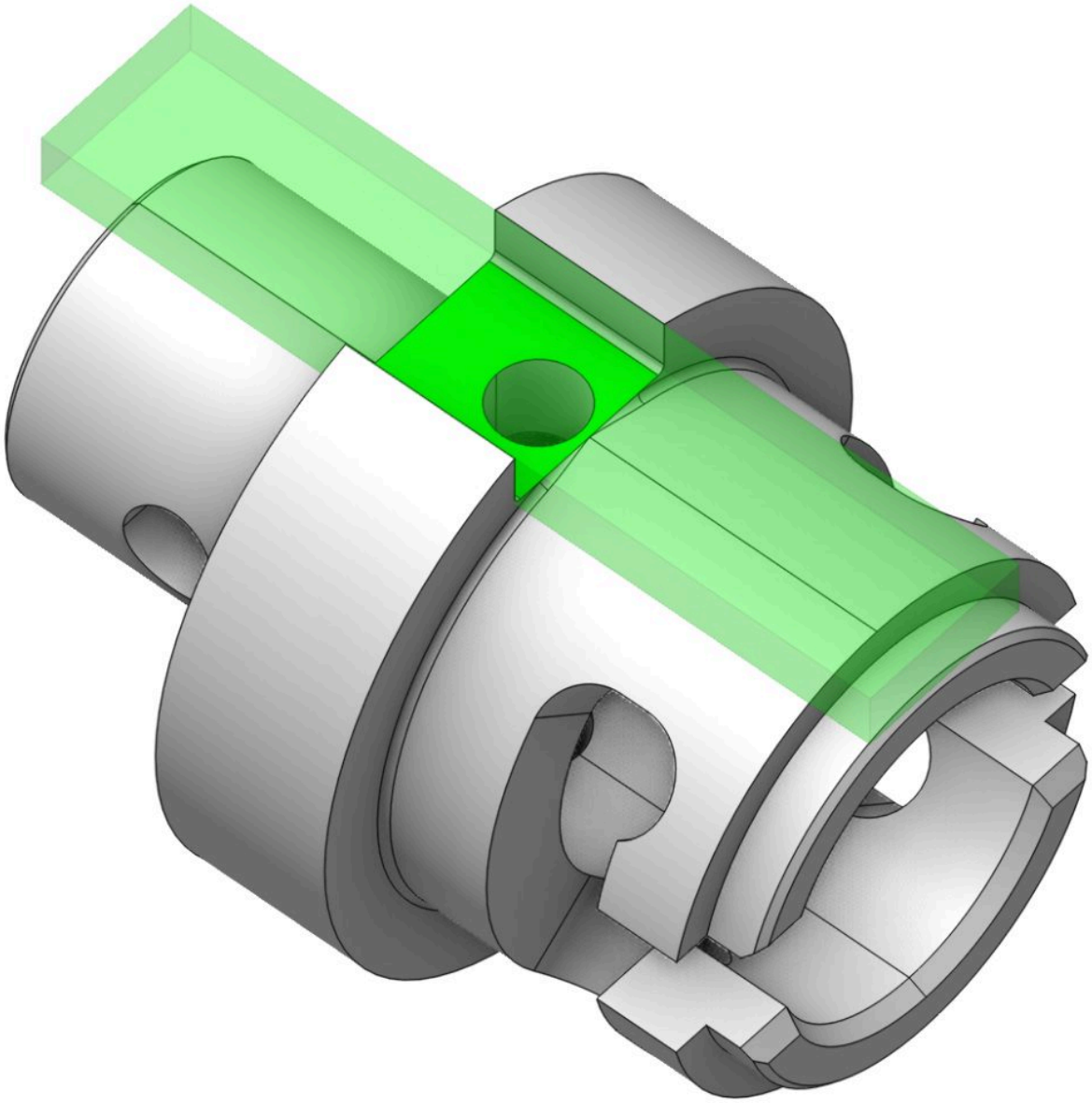
Working with Pocket.

1. The first step is to select the base surface of a pocket. At the picture below you can see a turn-milling part. We will machine the straight open slot at the top of the part.

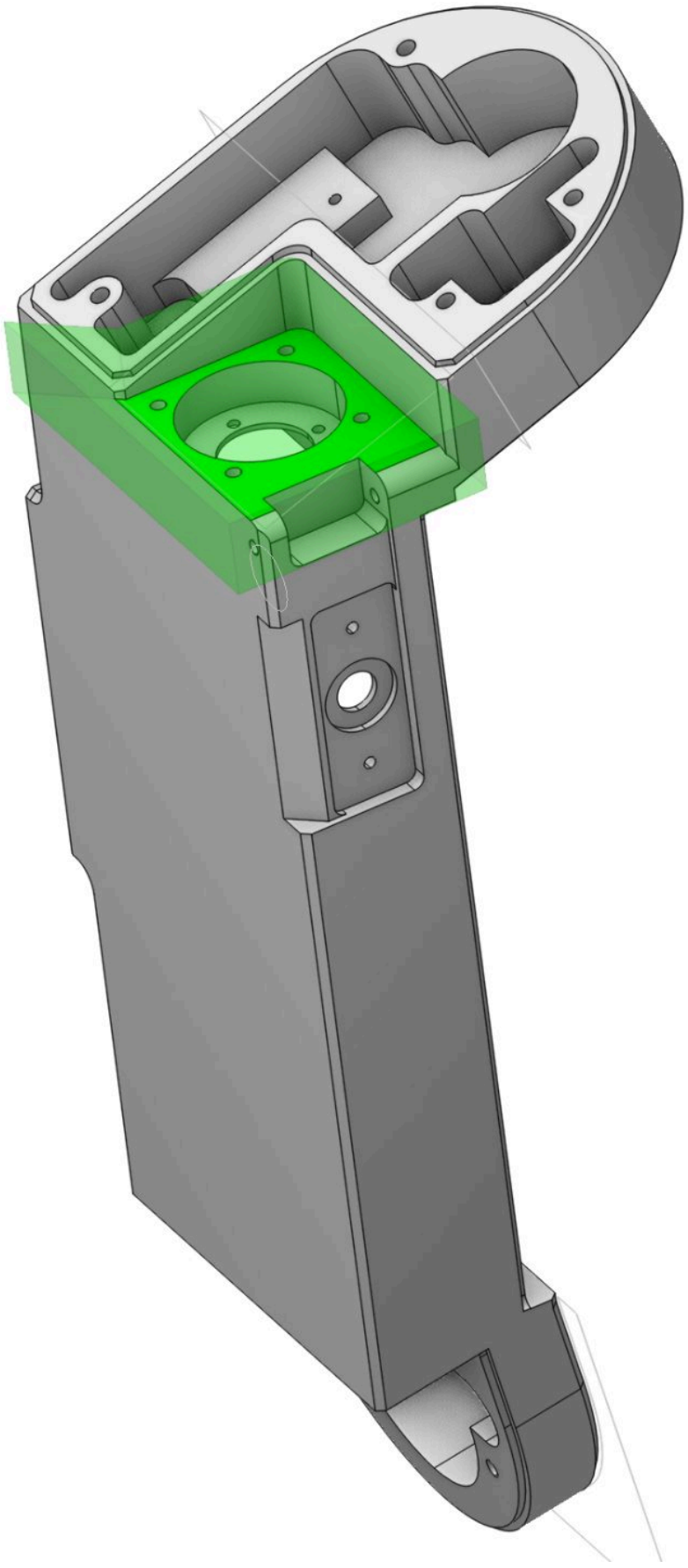


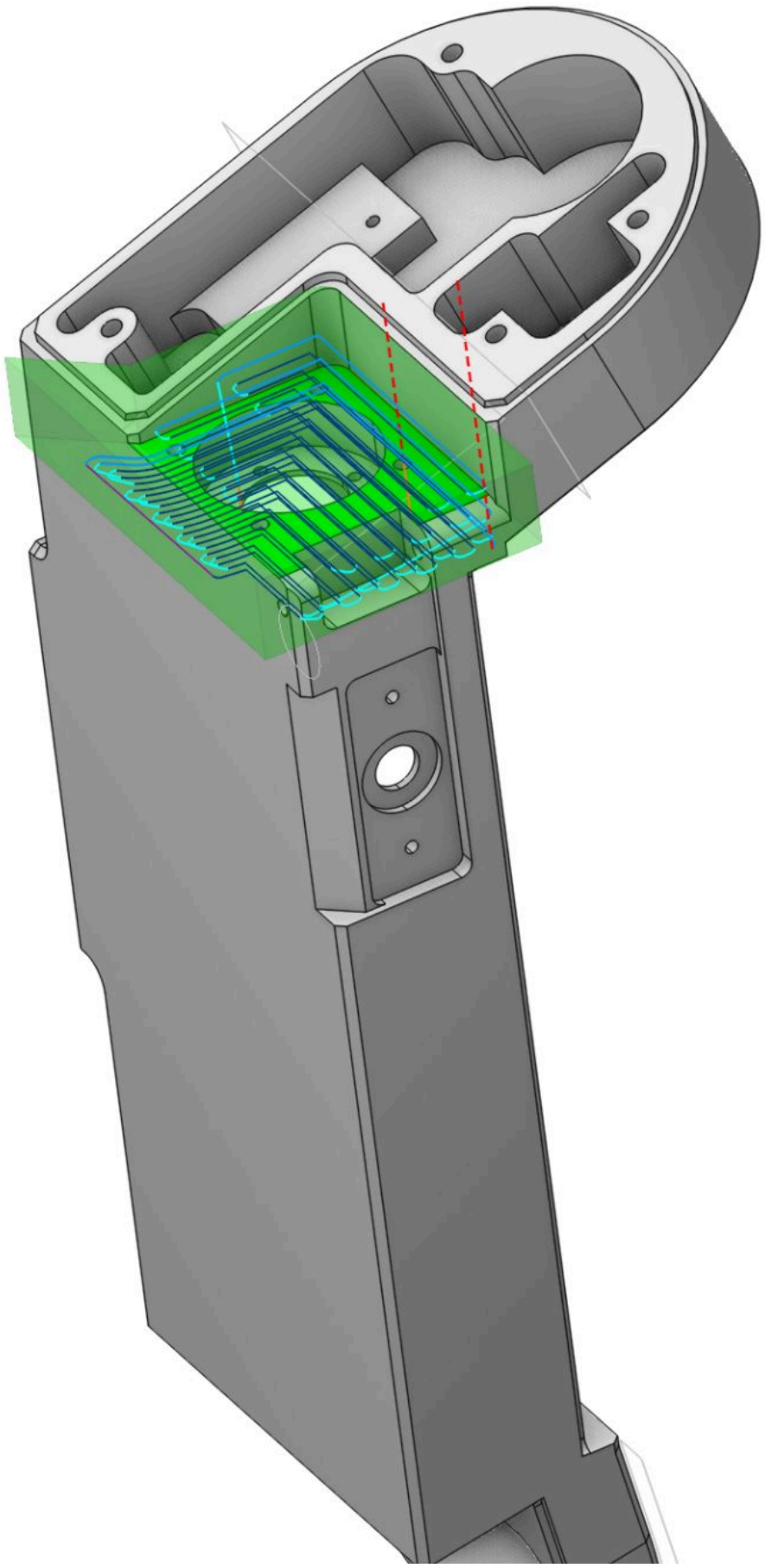
2. We select the bottom face of the slot and press the Pocket button at the Job Assignment panel.

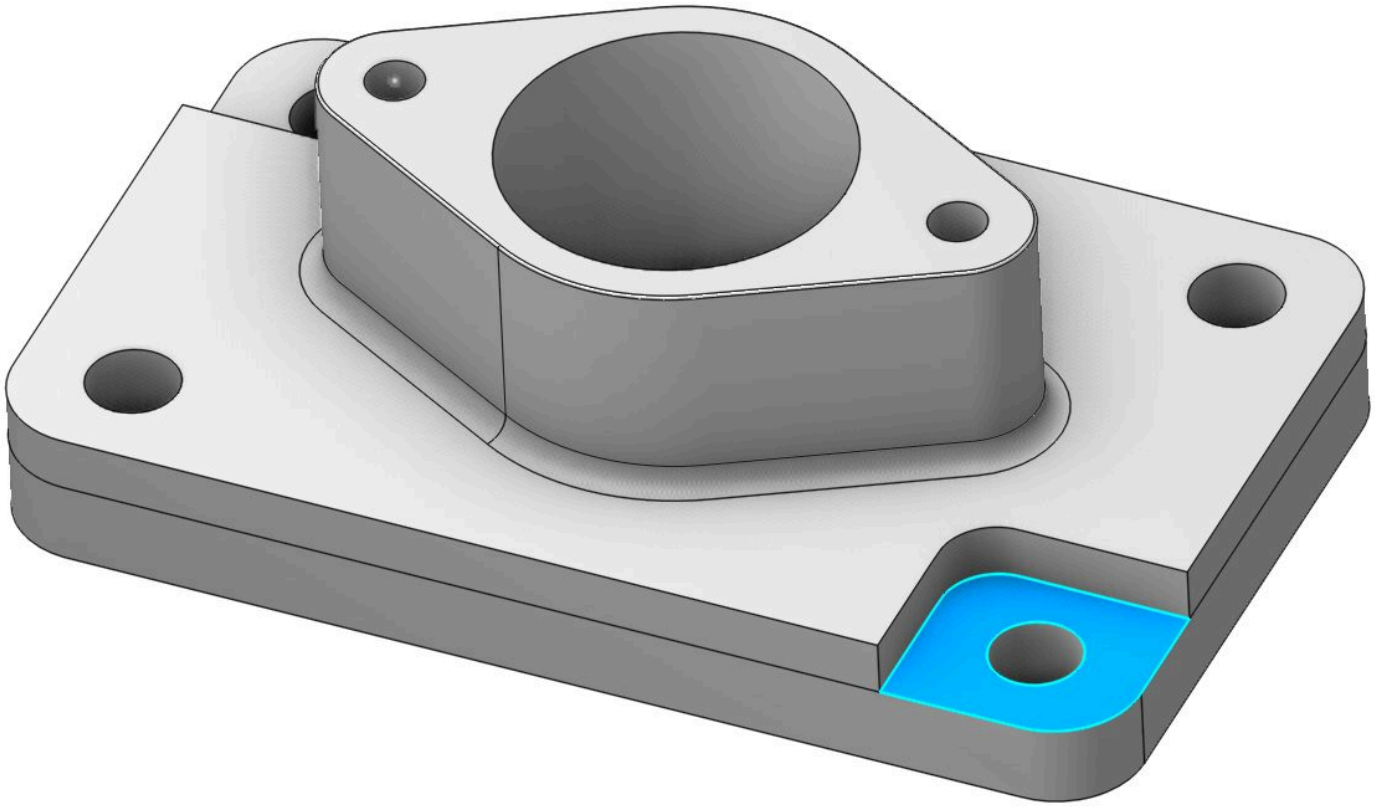
3. The pocket was converted to an open Job zone and Top level and Bottom level.

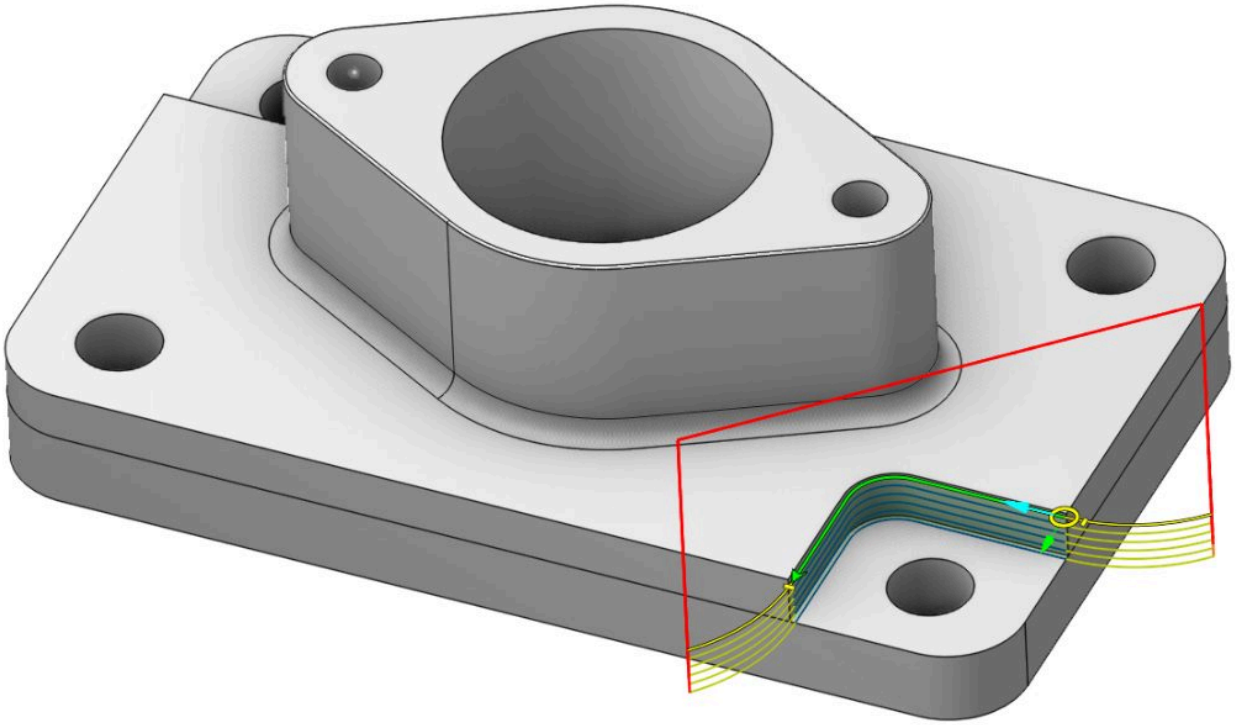


Typical uses of a pocket feature

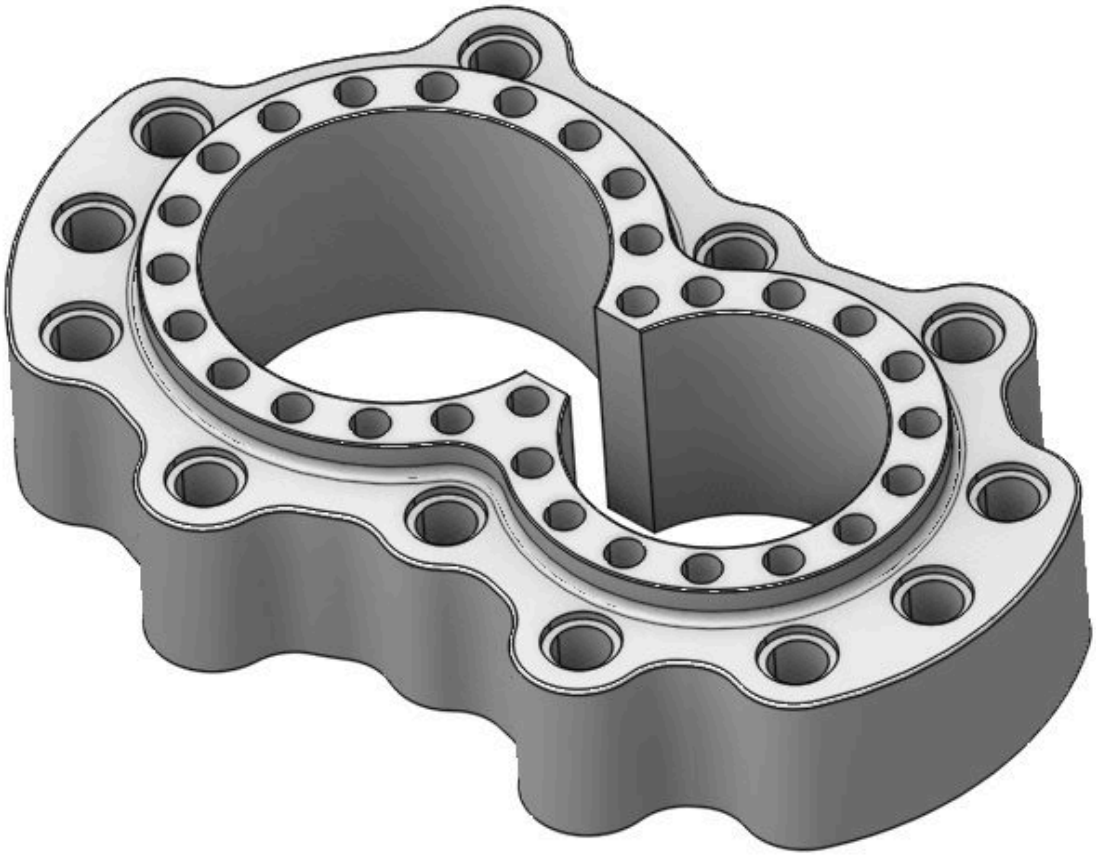


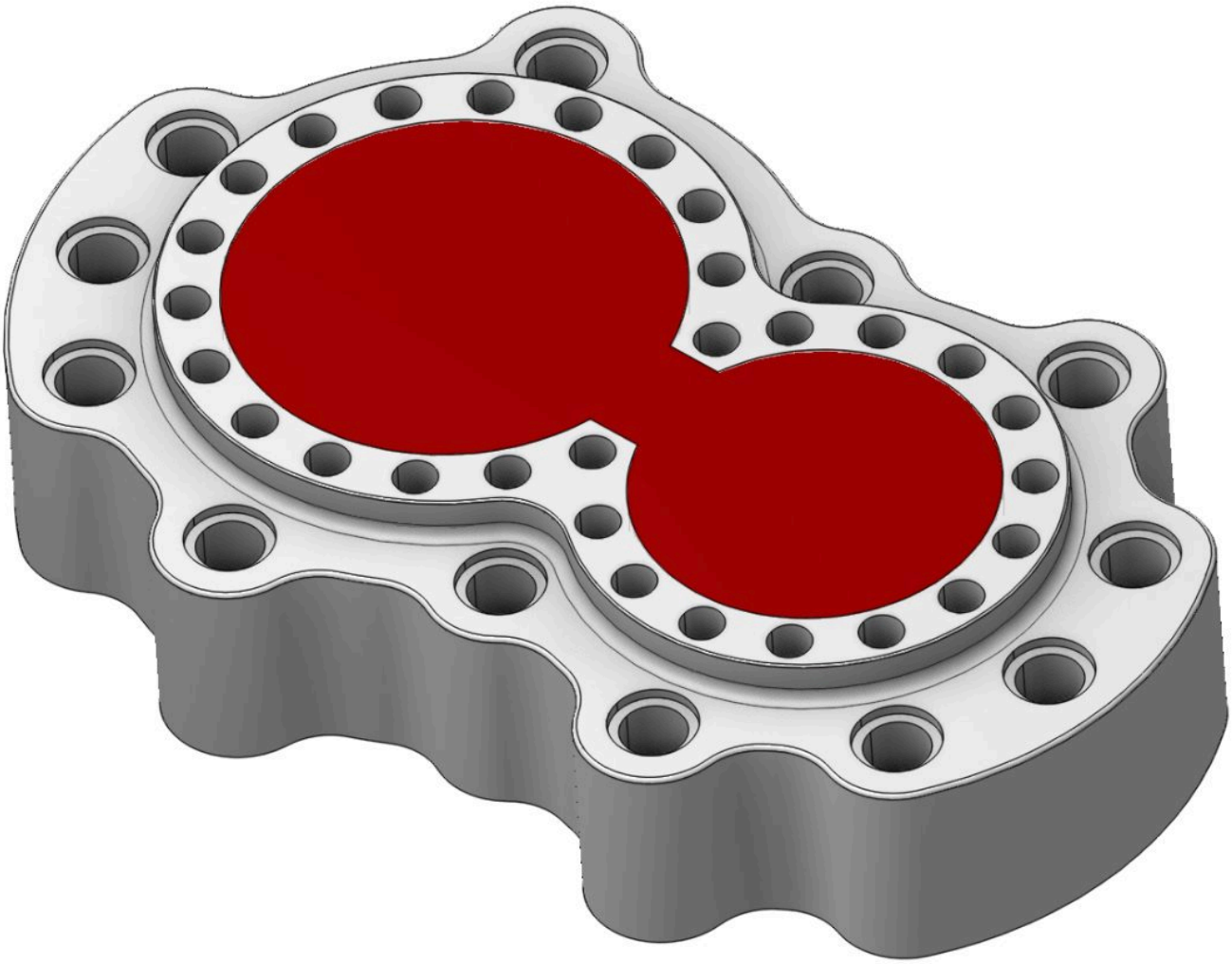






Restrict zones



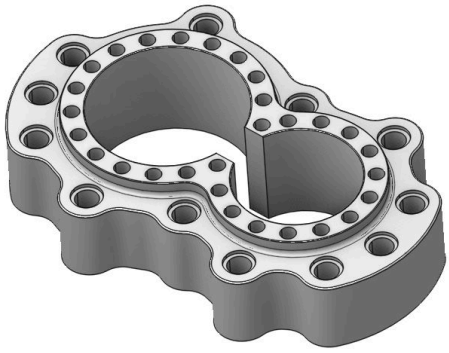


Application Area:

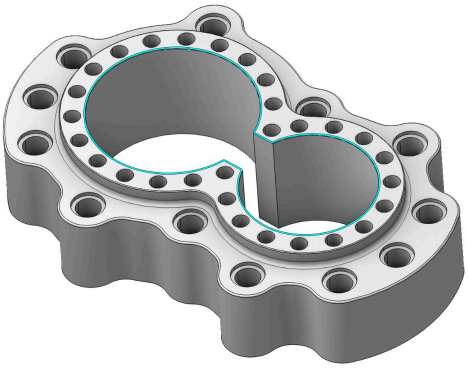
In addition to Job Zones in CAM system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Opposed to a job zone a restrict zone is always closed. As source geometry for restrict zones you can use any curves, 3d edges as well as vertical walls and connected chains.

Working with Restrict zones.

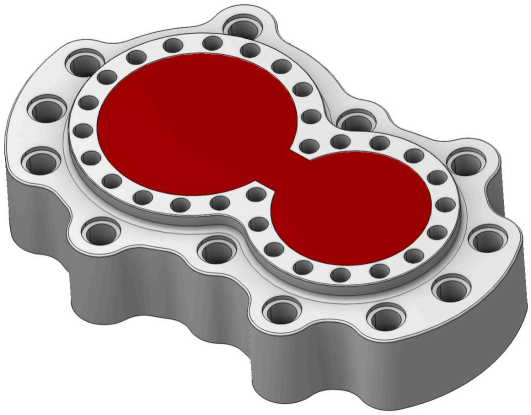
At the picture below you can see a part with a window. We will rough the outside of this part. To accomplish this task we have to add a restrict zone covering the inner window.



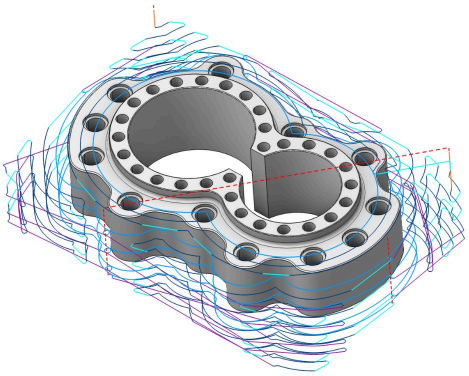
At first we select the edges chain around the window.



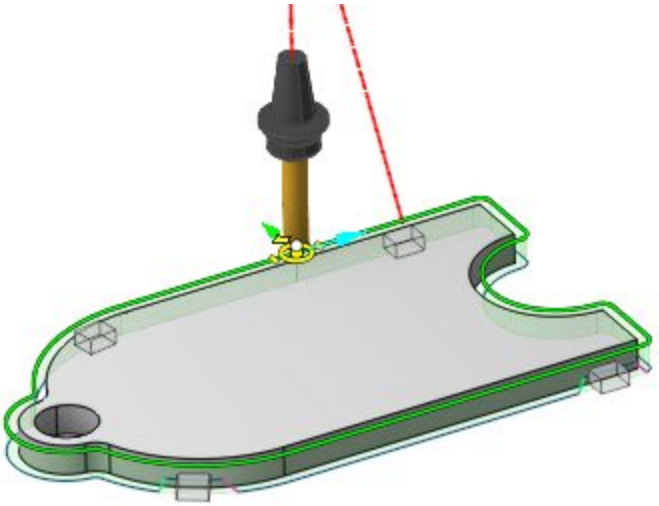
After that we add a new Restrict Zone into the Job Assignment of the Roughing operation.

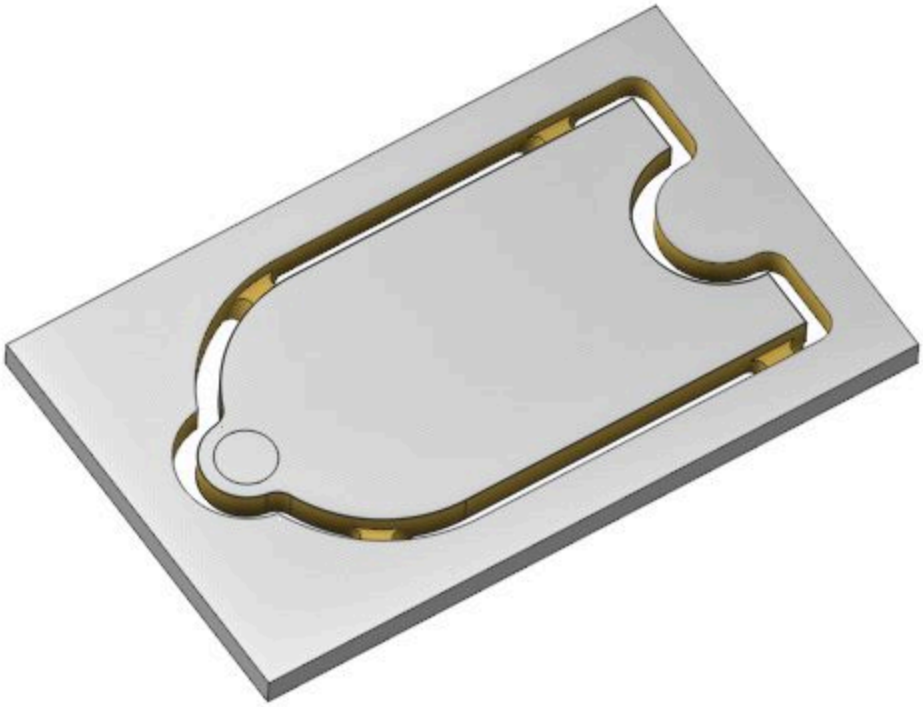


After generating toolpath we will see the following picture. As you can see the inner window has been leaved uncut.



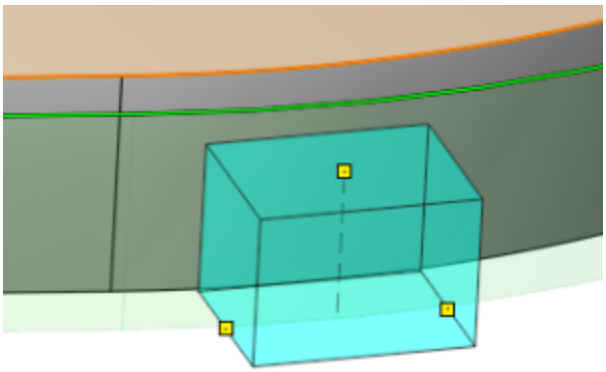
Tabs

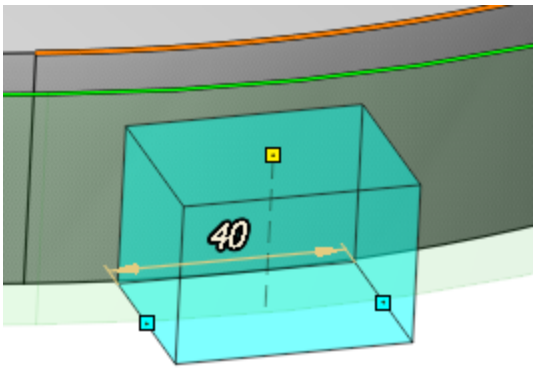


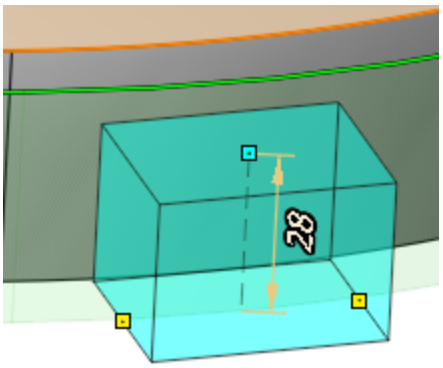


Application Area:

Tabs are needed to hold the part or fragments of the workpiece from falling out at the final stage of processing, which can lead to tool breakage or defective parts. The tabs window in the Job Assigment only describes its geometry and methods of reproduction along the contours. In this window, you can perform the initial addition of tabs with automatic placement according to a certain rule. Then you can edit the position and size of each tab directly in the graphic window.







Add tabs.

Necessary for the formation of the geometry of the tabs and their relative positioning.

▼ Click here to expand..
Scope

Necessary to determine the types of contour to which the tabs are applied.

Closed and unclosed. Tabs will be created in both open and closed contours.

Closed only. Tabs will be created in closed contours.

Unclosed only. Tabs will be created in open contours.

Selected countours only. We define the rule for which contours to add tabs to - all or only the selected ones. By default, if no contours were selected in the work order - all contours, if at least one was selected - selected contours only.

Replace existing tabs. Whether to recreate tabs for those contours where tabs were created earlier. Defaults to true.

Dimensions

Determines the dimensions of thetabs.

Tab size. Defines the length of the tabs. "By default", in mm or in % of the tool diameter.

Tab height. Defines the height of the tabs. "By default", in mm or in % of the tool diameter, FullHeight (the full height of an element or operation).

Default dimensions

By default is defined in Default dimensions.

Default tab size. Specifies the values for the defaulttab sizes.

Default tab height. Specifies the values for the defaulttab sizes.

Placement strategy

Defines the tabs placement strategy.

Quantity per countour. A specified number of tabs is automatically created for each contour. Thetabs are distributed evenly along the length of the contour.

Desired tab count. Number of tabs.

Linear density. One tab is automatically created for each section of the specified length Linear density on each curve.

Linear density. Density by length in mm, or as a % of the tool diameter. Default is 500%.

Minimal tab count. Minimum number oftabs per circuit. Default is 0.

Area density. One tab is automatically created for each Planar density elementary area on each closed curve.

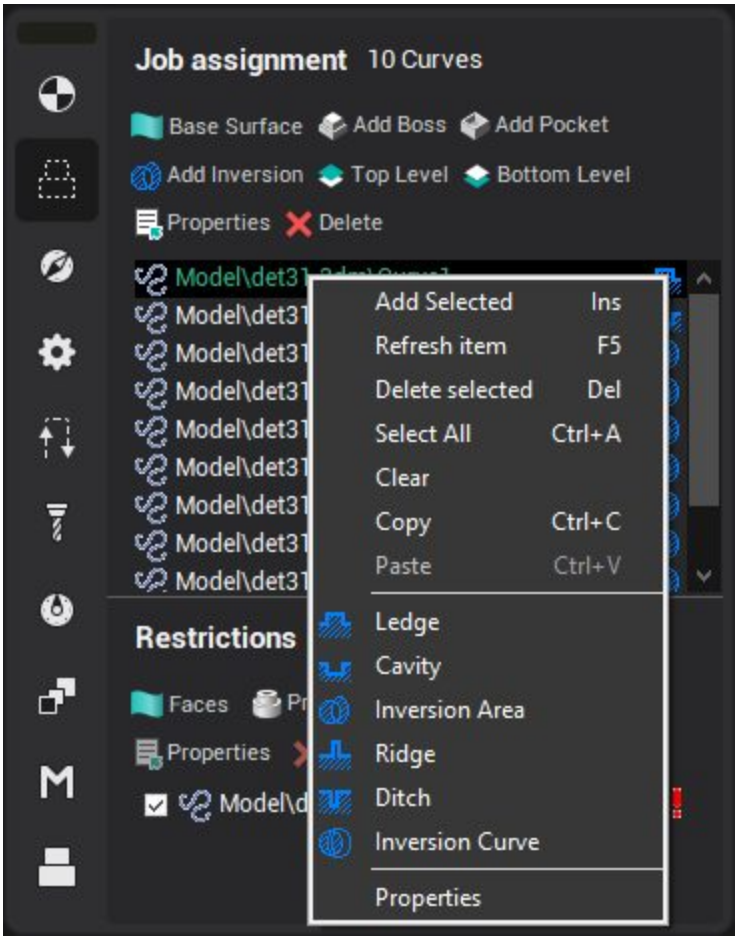
Area density. Area density in mm^2, or in % of the tool diameter. Default is 1000%.

Minimal tab count. Minimum number oftabs per circuit. Default is 0.

Avoid corners. When the Avoid corners option is enabled, the algorithm starts checking whether the given tab falls into a corner. If it does, it tries to shift it a small amount to one side, then to the other, until it finds a place outside the corner. If it fails, it leaves the tabs in its original place.

Merge tabs of neighbor curves. This parameter removes tabs from adjacent contours that affect this contour.

Job assignment for engraving and pocketing operations



Application Area:

For the engraving and pocketing operations, the model is defined by curves as for the 2D and 3D curve machining operations. However, unlike the curve machining operations, the system forms a model for machining from the defined curves. A model for machining represents a flat area, which only exists where there is the curve to be machined. The task of the user is to create areas from the available curves. Every curve is a 'border' of the model. Its selected type defines how each 'border' is machined:

Boundary types

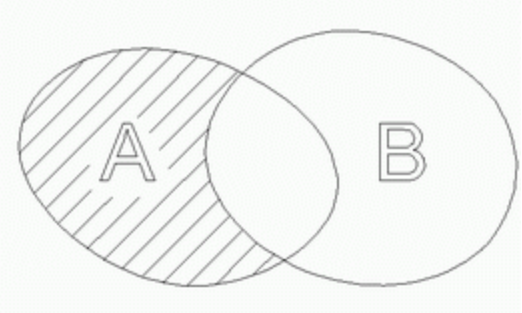
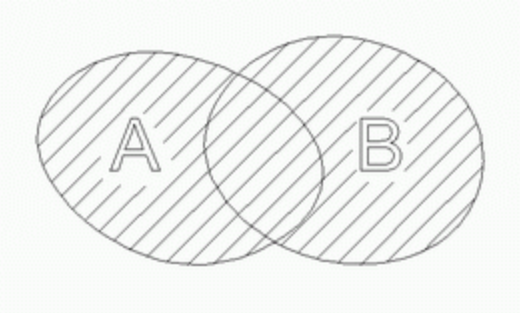
- Ledge – indicates a closed area, which will not be machined;
- Cavity – indicates a closed area, which will be machined;
- Inversion area – indicates a closed area, inside which the machining rules will be reversed, i.e. machined areas will become unmachined and vice versa;
- Ridge – indicates a curve along which material will not be removed;
- Ditch – indicates a curve along which material will be removed;
- Inversion curve – indicates a curve along which machining will not be performed, if it goes into machining area or vice versa.

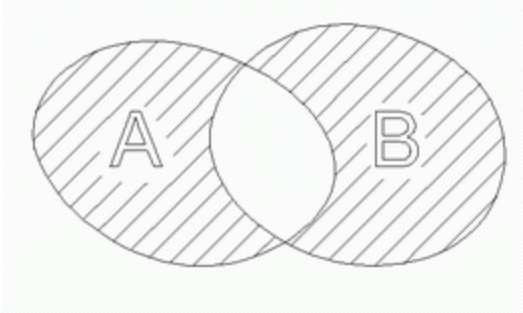
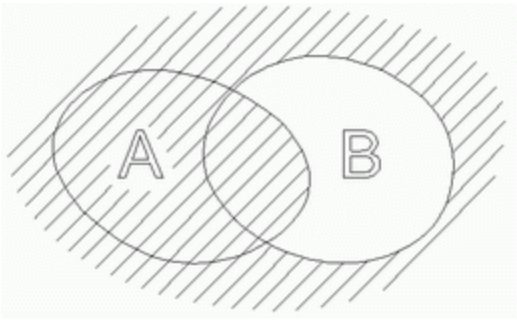
A Ledge, Cavity or Inversion area can only be defined by closed curves. When an open curve is selected, then these types will be unavailable. If a closed contour consists of several fragments, then it must be joined into a single curve (see Curve joining). A Ridge, Ditch or Inversion curve can be defined by any curve, either closed or not. To define their thickness, use additional stock.

Forming the machining area

The formation of an area is performed by successive execution of Boolean operations on the selected curves. The order that the curves appear in the list is important. The first object of the list dictates the status of any unbounded area, i.e. workpiece area not enclosed by any curves. Should the first object be a Cavity or a Ditch, then the model is considered to occupy the entire unbounded area, otherwise machining is possible in this area. All subsequent objects modify the area of the model by the method with which they are defined. If an object is a Ledge or Ridge it will be added to the area occupied by the model. If an object is a Cavity or Ditch it is subtracted from the area occupied by the model. If an object is an inversion area or curve, then it will reverse the status of the area it overlays. Should a group consisting of several curves be an element of the list, then an area will be formed from that group by inverse addition of each curve; the obtained area modifies the result according to its defined type. Surfaces, faces and points that enter the group, are ignored.

The results of area formation by two curves are shown in the pictures below. Model (solid) areas are shaded.

List contents	Resulting area
A – ledge B – cavity	
A – ledge B – ledge	

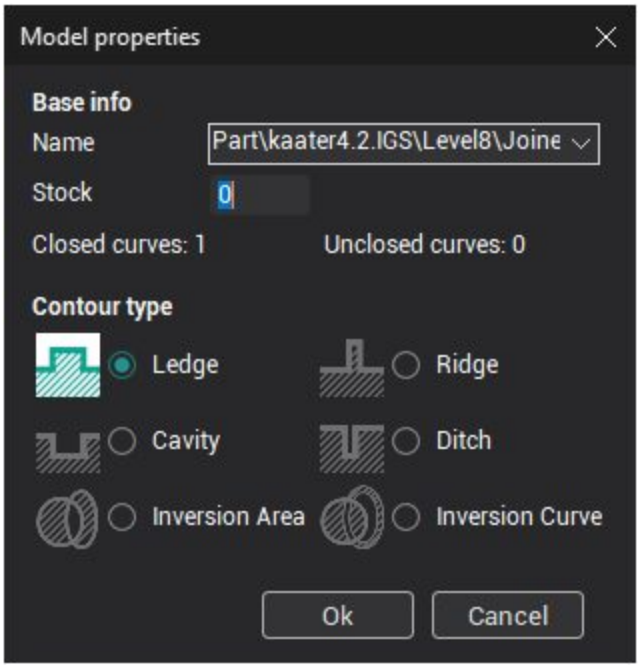
List contents	Resulting area
A – ledge B – inversion area	
B – cavity A – ledge	

Take a look at the first and the last examples. In both cases curve A is a Ledge and curve B is a Cavity. In the first case the size of the model is defined by the curve A, with further subtraction from its area limited by the curve B. In the last case, curve B defines the area where machining can be performed, curve A further limits machining in area B.

The order of the geometrical objects in the list can be changed by mouse dragging. The model being formed can be dynamically displayed in the graphical window.

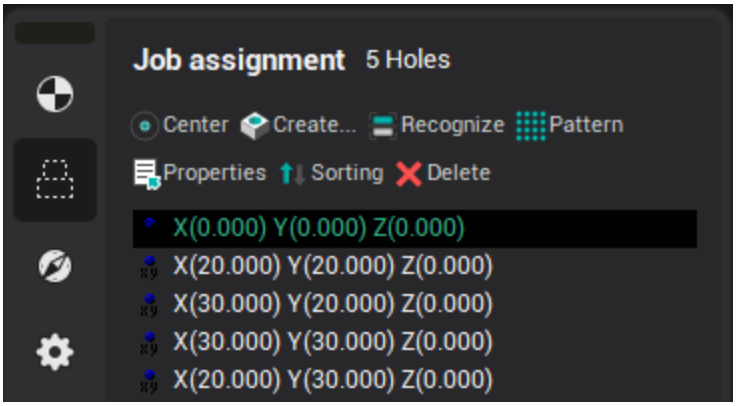
Item properties

There is the properties window to set the selected item parameters. The window can be opened by the double click on the item or from the pop-up menu. The window is shown below.



Note: Dynamic showing of parameters possible only in a Shade mode. For turn on it press the  *button on the main panel.*

Job assignment for hole machining operation



Application Area:

When defining the parameters for the hole machining operation it is possible to define the data for holes to be drilled. In the hole machining operation, the holes list defines the number, sequence and parameters of the holes to be machined. The order can be altered by mouse dragging.

Job assignment controls

- Center — Create hole by center point
- Create... — Create hole by coordinates input
- Recognize — Automatically recognize holes in the part
- Pattern — Create holes array by pattern
- Properties — Properties of the selected items
- Delete — Delete selected items

Hole parameters

Each hole is defined by the coordinates for it's center, the diameter and also the value of the upper and bottom levels. There are two methods to define the center coordinates of holes: by coordinates or by a geometrical "point" object.

Machining levels

Regardless of the center definition method used, the depth of the hole is defined directly on the Model page. The holes specified by coordinates are marked with the sign while the holes defined by center point are marked with the icon. To define the top and the bottom levels, it is necessary to select the desired points from the list on the right and enter the Zmax and Zmin values.

- Zmax – defines the Z coordinate of the top of the hole. The coordinate can be defined either directly or calculated automatically. When calculating automatically, transition to work feed is performed using the safe distance from the workpiece.

- Zmin – defines the Z coordinate of the bottom of the hole. The coordinate can be defined either directly or calculated automatically. When calculating automatically, the coordinate is taken from the model being machined.

Drill tip compensation

In addition to the ZMin parameter operation can have the Drill tip compensation parameter specified. This parameter can be one of the following:

- Off – the drill tip descends to the ZMin level.
- Drill tip – drill descends below the ZMin level to the value of the tapered part of the drill, thus providing cylindrical drilling area to the ZMin depth.
- Length – drill top descends below the ZMin level to the specified value.

Drilling direction

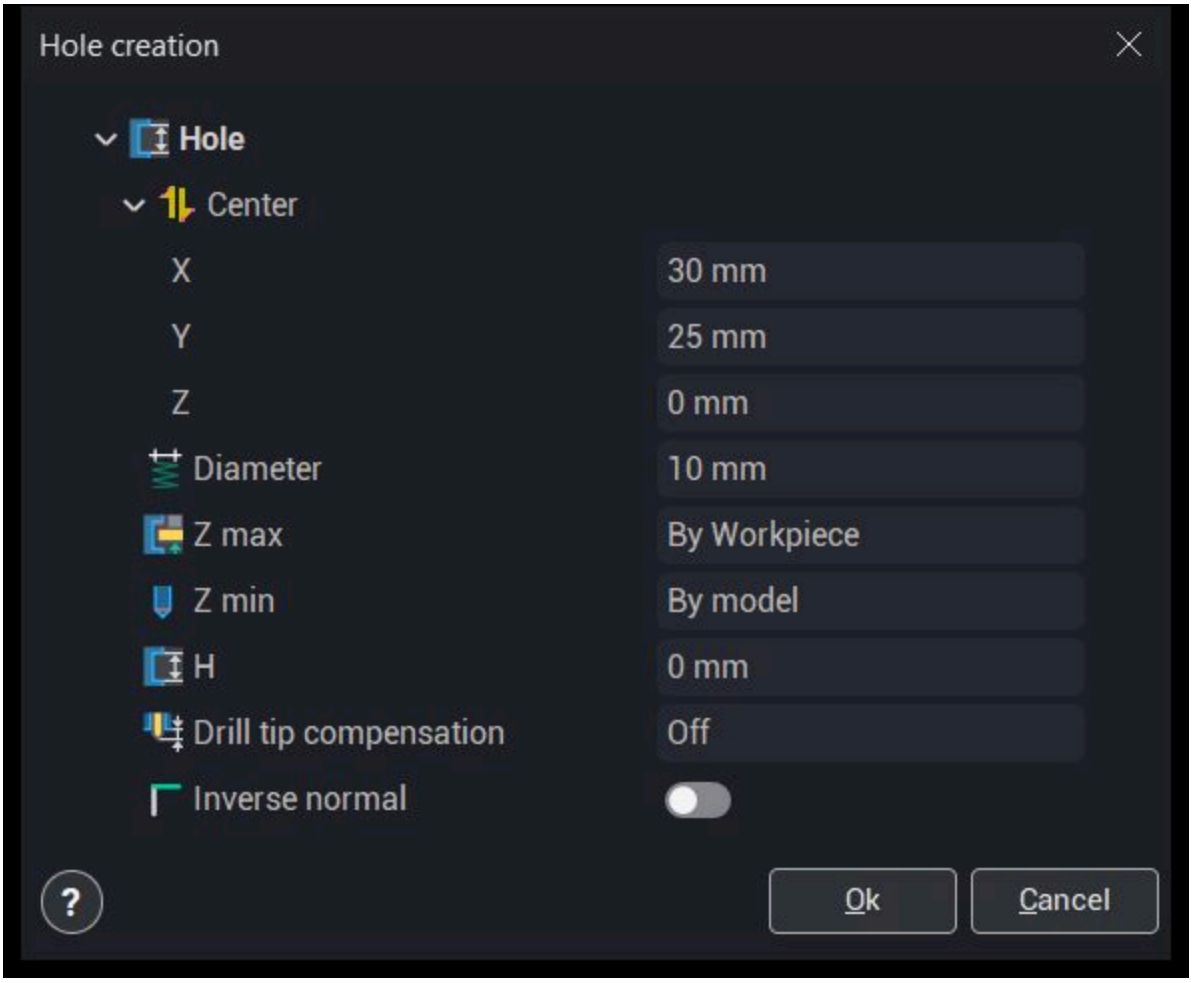
Hole Machining operation supports two ways to specify drilling direction for each hole center. Use the Inverse normal field of the Hole editing window or specify the normal in the graphical window.

Exporting and sorting holes

Select holes in the holes list and use the context menu Export selected in DXF item to export the list into the DXF-file.

To sort holes with different parameter values use the Sorting.

Defining holes by coordinates



Application Area:

The window that opens is designed for parameter assignment of new or existing holes. The Create command opens the Hole creation window. To edit the parameters of an existing hole, double click the required hole in the list or use the Properties command: the Hole editing window opens. Details

Creating and editing holes

To create a hole the user should define the values of its parameters (position, diameter, depth) and press the Ok button. The created hole will be automatically added to the list.

When hole's parameters are being altered, the changes are displayed in the graphic window.

Hole parameters

Center. Coordinates X, Y and Z of the hole center.

Diameter. Diameter of the hole.

Z max - top level mode:

- By workpiece - the top level is defined as the highest coordinate of the workpiece edge along the tool travel path;
- By feature -the top level is defined as the highest coordinate of the geometric element selected as the Job Assignment along the tool travel path;
- Default – the default mode is used; parentheses indicate this mode. See Default settings below;
- Manual – user set value manually.

Z min - bottom level mode:

- By feature -the bottom level is defined as the most remote coordinate of the geometric element selected as the Job Assignment along the tool travel path;
- By model - the bottom level is determined by the most remote coordinate of the hole position on the part in the tool travel direction;
- Default – the default mode is used; parentheses indicate this mode. See Default settings below;
- Manual – user set value manually.

H. Depth of the hole: the distance between Z max and Z min. It is set as a length or in % of the hole diameter (% D).

Drill tip compensation - choose the way the hole depth is specified:

- Off – last tool path point matches the drill tooling point;
- Drill tip – last tool path point matches the drill tip point;
- Length – same as Off but the drill travels the specified value down from the drill tooling point.

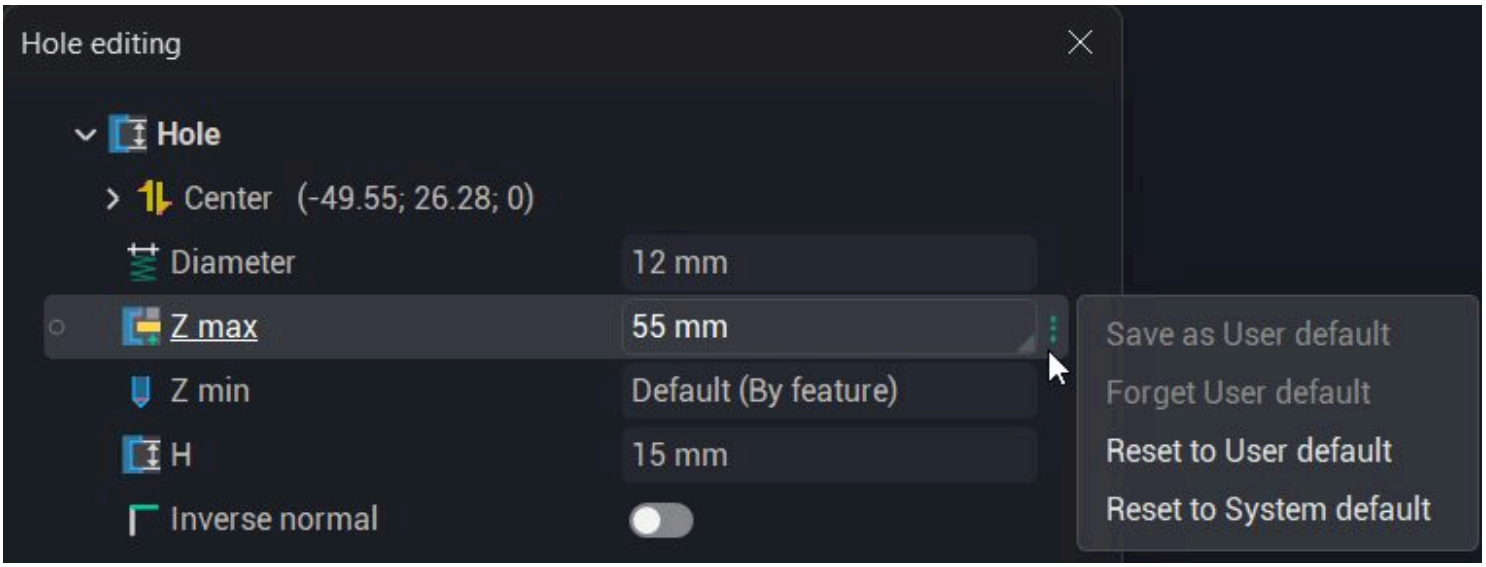
Inverse normal. Reverses the drilling direction of the hole. The direction of an existing hole is changed in the Hole editing window.

Default settings

To change the default modes of Z max and Z min, hover over the Z max or Z min line and click the ... button on the right. The menu contains the following commands:

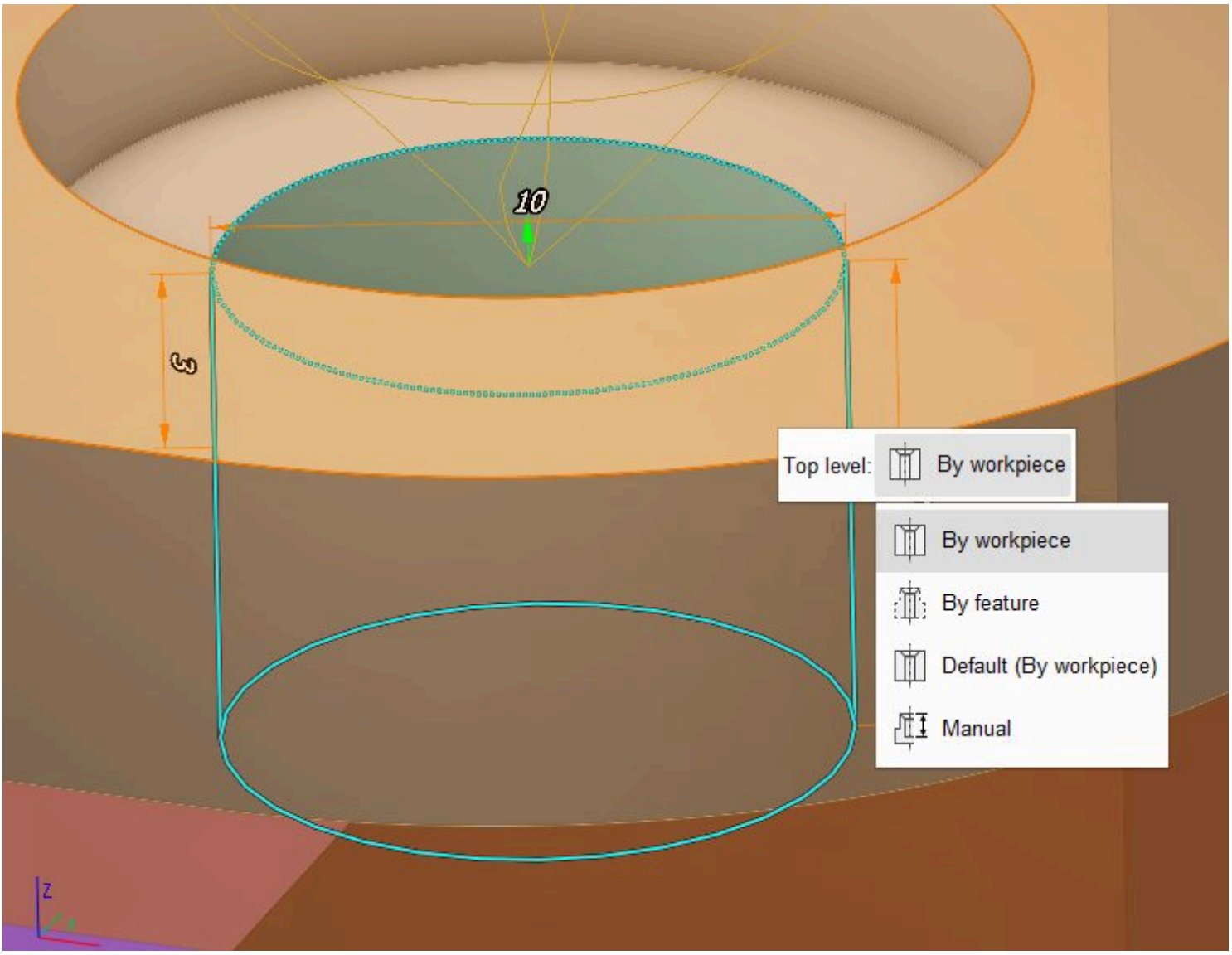
- Save as User default – the current mode of the level becomes the default mode. Available when the level uses a mode that can be the default: By Workpiece or By feature for Z max, By feature or By model for Z min;
- Forget User default – the default mode returns to the system default;
- Reset to User default – sets the Default mode for the level;
- Reset to System default – sets the system default mode for the level.

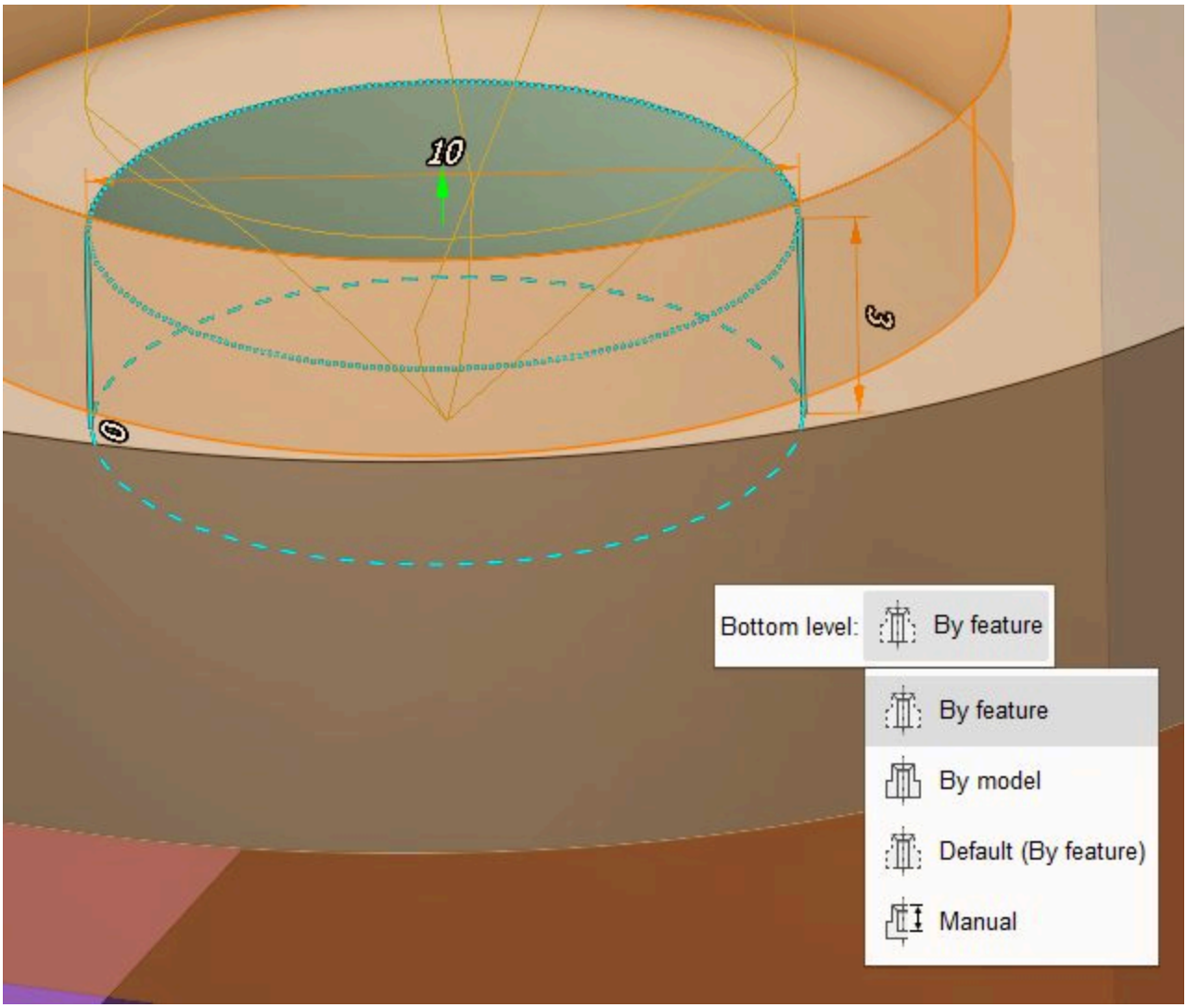
Commands that do not change anything are unavailable. These settings will be applied for the whole system, not just for a current project.



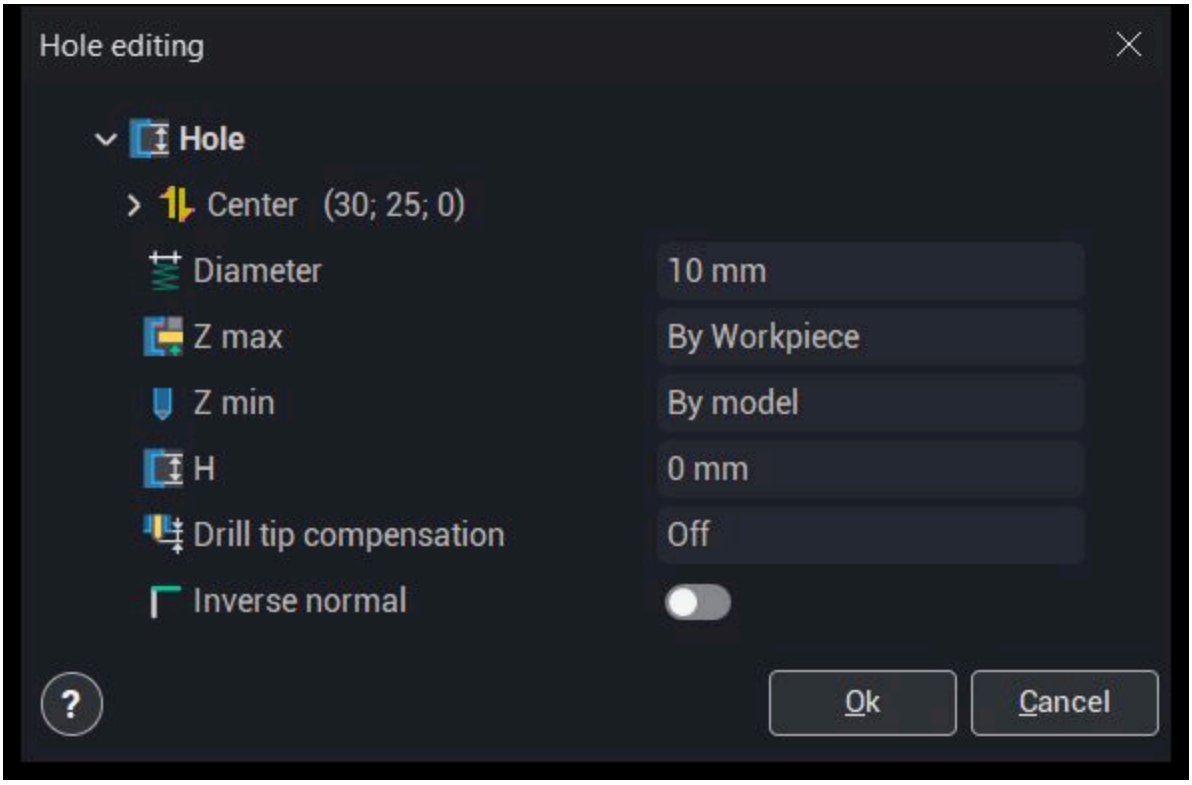
Editing levels in the graphic window

Also you can set Z max and Z min mode in graphics window. Click on top or bottom level and you see action menu. Each mode is displayed differently:


10
6
Top level:
By workpiece
By workpiece
By feature
Default (By workpiece)
Manual
Z


10
3
0
Bottom level:
By feature
By feature
By model
Default (By feature)
Manual

Defining holes by using a geometrical point object



Application Area:

The window that opens is designed for parameter assignment of new or existing holes. To edit the parameters of an existing hole, double click the required hole in the list or use the Properties command: the Hole editing window opens.

Creating and editing holes

To create a hole the user should define the values of its parameters (position, diameter, depth) and press the Ok button. The created hole will be automatically added to the list.

When hole's parameters are being altered, the changes are displayed in the graphic window.

Hole parameters

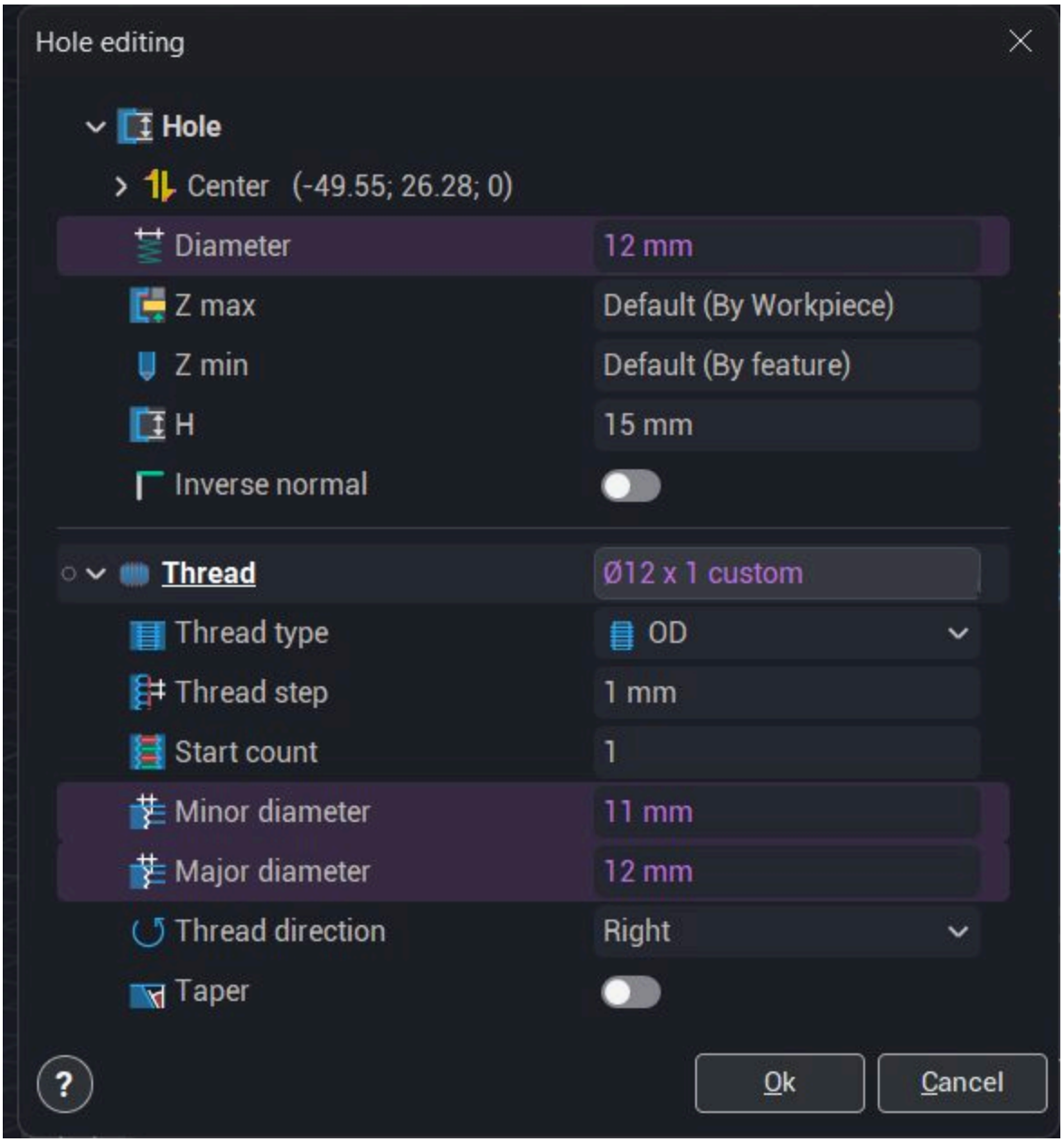
The Hole editing window contains the same parameters as the Hole creation window. Details

Center. The coordinates of the hole center are shown in one line as *(X; Y; Z)*. Expand the line to edit them.

Inverse normal. Reverses the drilling direction of the hole.

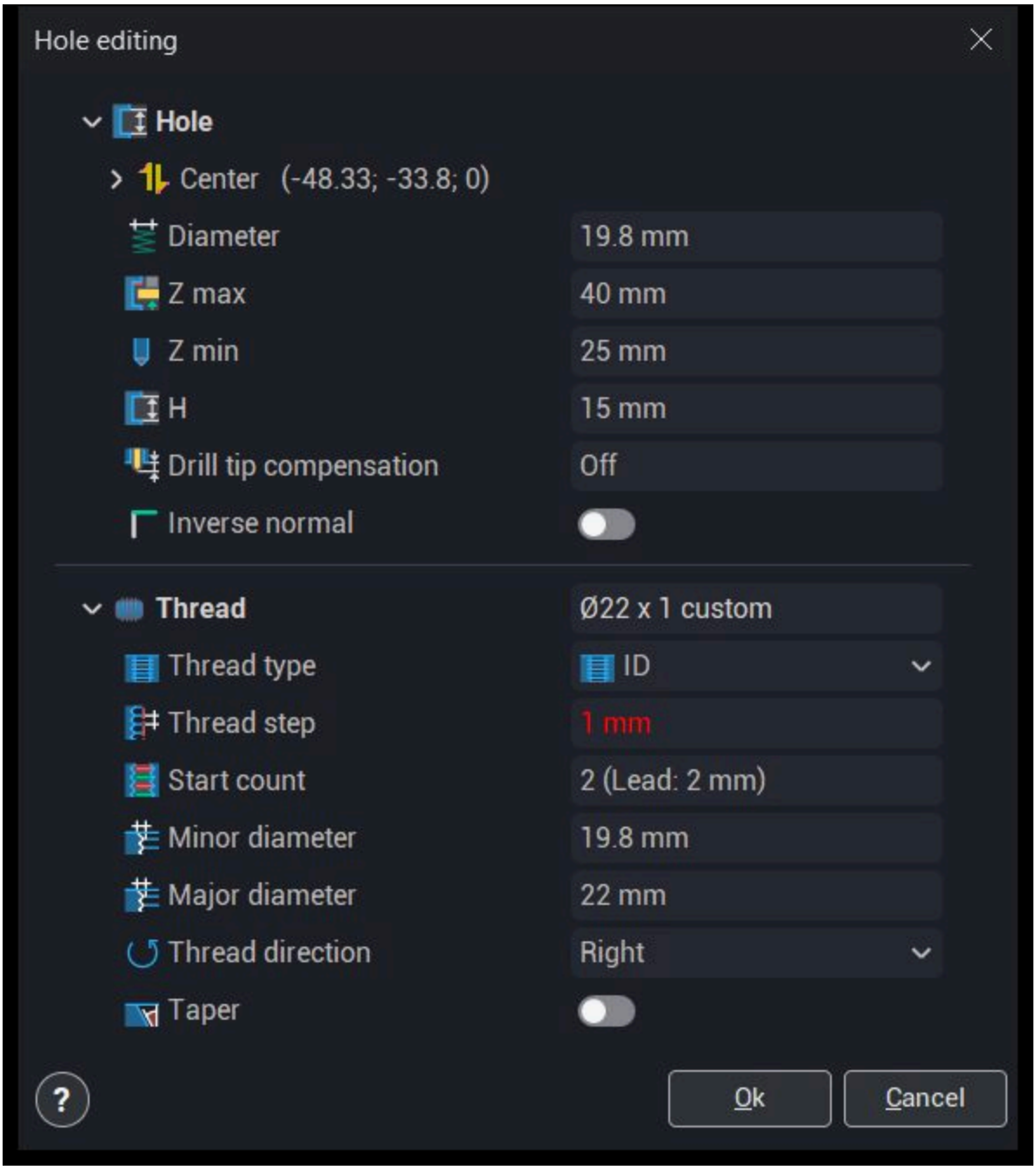
Editing several holes

To edit several holes at once, select them in the list and use the Properties command. The window shows the parameters of all selected holes, including the Thread group. If the selected holes have different values of a parameter, the parameter is highlighted in purple. A value entered in the window is applied to all selected holes.



Thread

The Thread group is shown when the operation uses the Thread milling drilling type. It defines the thread of the hole. The machining strategy is common for all holes of the operation and is set on the Strategy tab. Details



Thread. The line shows the thread designation and the source of the thread data:

- custom – the thread data is entered manually;
- from CAD model – the thread is taken from the CAD model, the thread data is read-only. Selecting it again repeats the search in the model. Available when the model holds a thread for this hole;
- from library – the thread is selected in the thread library. If the designation is changed after that, the source becomes custom.

Thread type. Specifies whether the thread is internal (ID) or external (OD).

Thread step. Thread pitch: the distance between adjacent thread turns. It is set as a length, as a number of threads per unit, or taken from the tool (From tool). The value is highlighted in red when the thread mill of the operation has several rows of teeth and its tooth step differs from the thread step: such a mill cuts

only its own step. The number of thread starts does not matter. The calculation of the operation reports the same error.

Start count. Number of thread starts. The lead is shown in parentheses: for a multistart thread the lead equals the pitch multiplied by the start count.

Minor diameter. Inner (minimal) diameter of the thread.

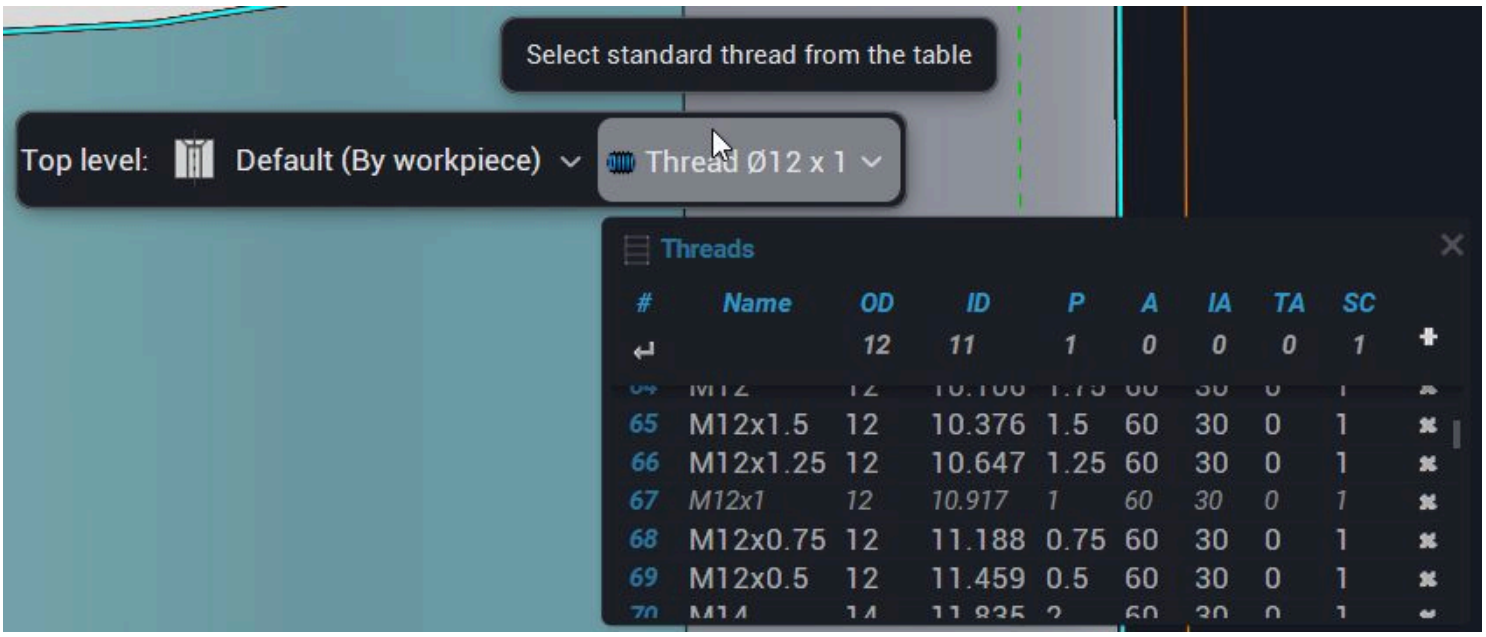
Major diameter. Outer (maximal) diameter of the thread. The thread depth is half the difference between the major and minor diameters.

Thread direction. Thread kind: Right or Left.

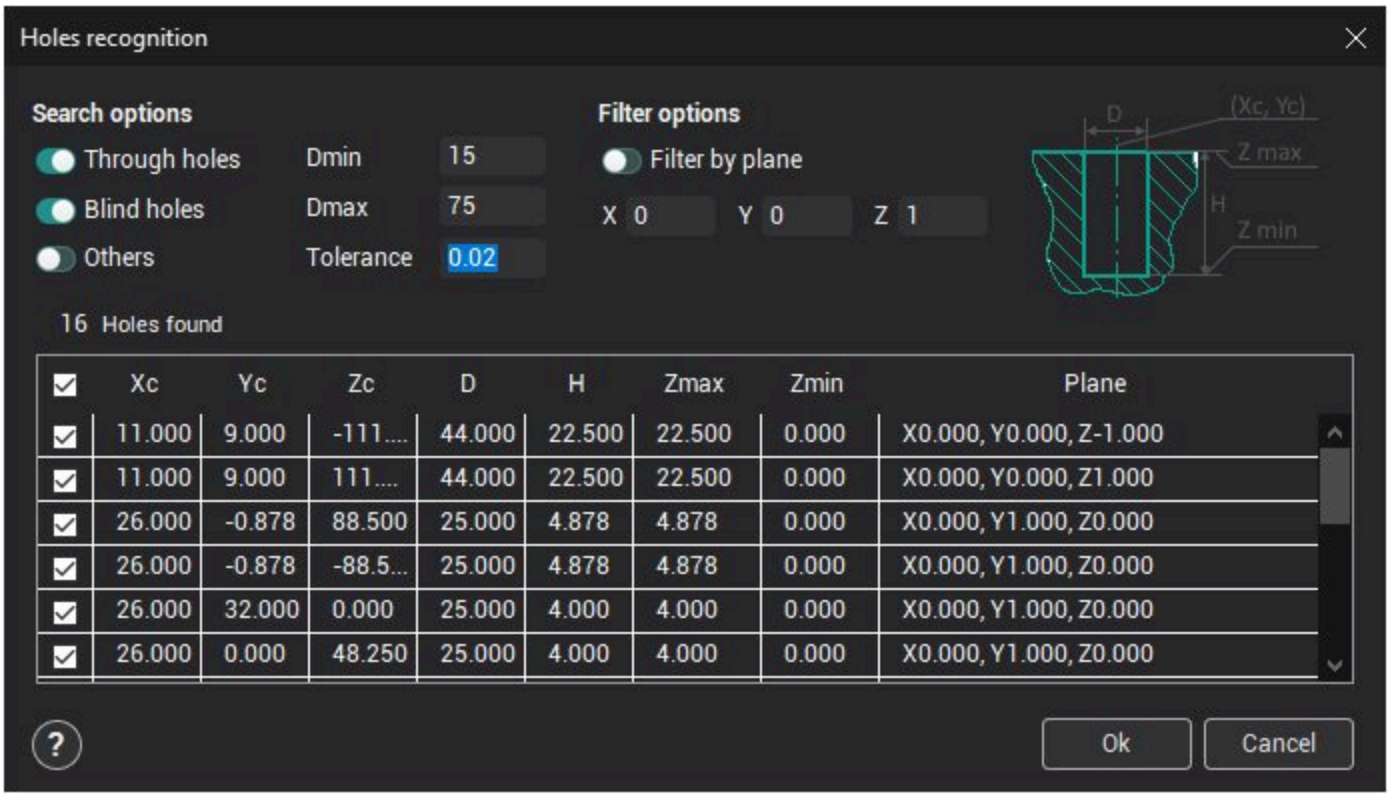
Taper. Enables a tapered thread and sets the taper angle.

Selecting a thread in the graphic window

A standard thread can also be selected in the graphic window. Click the top level of a hole: the action panel shows the Top level mode and the Thread button with the current thread of the hole. Click the Thread button to open the Threads table and select a thread in it. The selected thread fills the designation, step, start count, minor and major diameters and taper of the hole thread. The button is available when the operation uses the Thread milling drilling type.



Automatic hole recognition



Application Area:

The holes are found in the part. When a hole is found, it will be automatically added to the holes list.

Search options

Hole recognition is performed according to the selected search options. Only those holes that lie within the defined range will be added to the list. All holes are divided into three types:

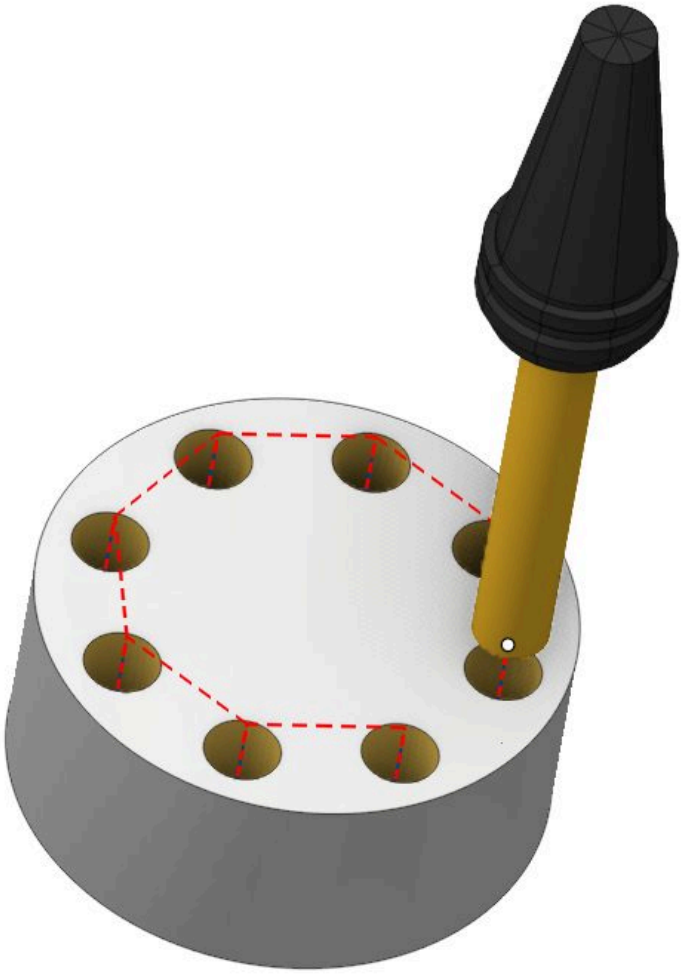
- Through – holes, which go through the model, or with a bottom level that is lower than the bottom machining level of the operation.
- Blind – holes, the end of which lie in the model between the top and the bottom levels for the operation.
- Others – holes, for which only the center coordinates, can be defined but not the diameter and/or the depth of the hole. Such holes might have a variable diameter e.g. with facets, or just be curves.

Editing and adding recognized holes

Parameters for the holes found in a search operation can be edited. When the parameters for a hole are being edited, the hole is highlighted in the graphic area. The parameters for holes can be altered by left clicking on it in the search window and typing the new values.

When the Ok button is pressed, all holes selected with a tick will be added to the holes list. Left clicking on the heading of the first column will activate or deactivate all holes.

Creating hole pattern



Application Area:

In the CAM system, it is possible to set various types of hole templates. It is not necessary for a template to have a geometric construction of the hole.

Pattern types

Using certain rules, you can define the following templates:

- Linear
- Circular

- Angular
- Concentric
- Parallelogram

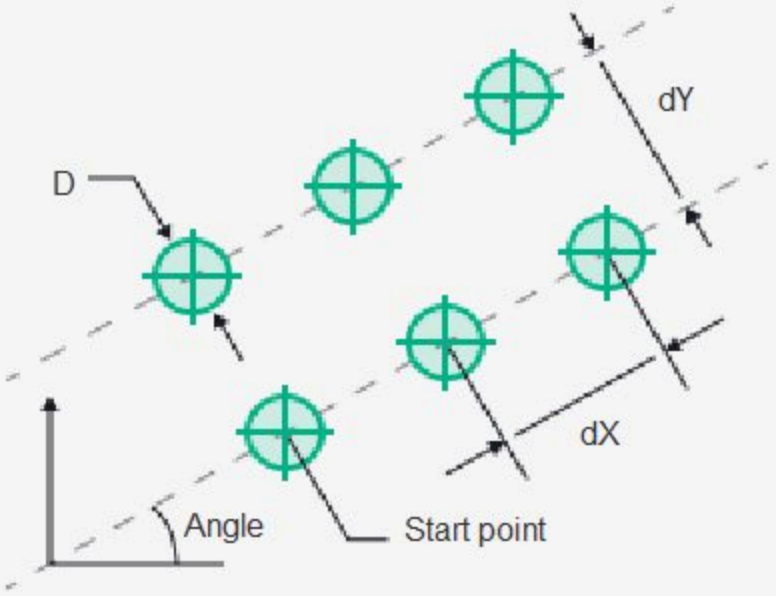
The system uses five types of pattern: Linear, Circular, Angular, Concentric and Parallelogram.

Linear

On the Linear page user can create linear holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

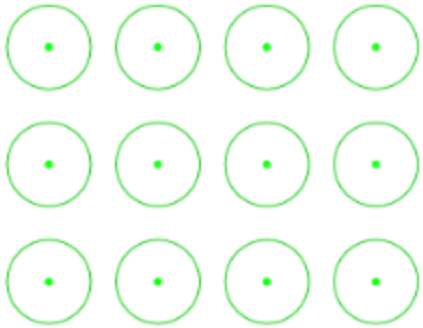


Columns	2	D	20
Rows	2	dY	10
Angle	0	dX	10

Start point

X	0	Zmin	Auto		
Y	0	Zmax	Auto	H	0

Ok | Cancel | Apply

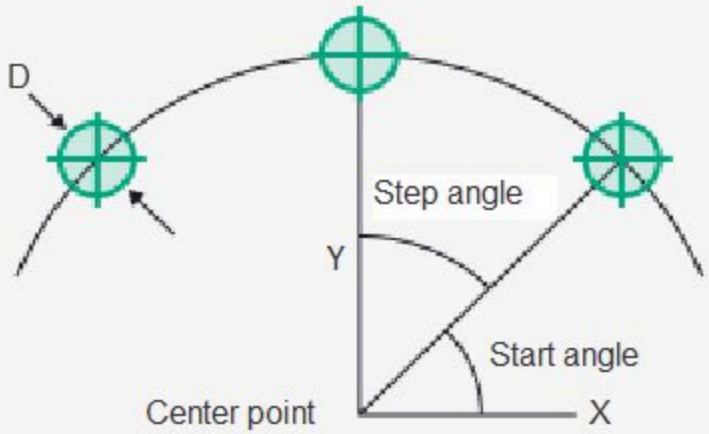


Circular

On the Circular page user can create circular holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

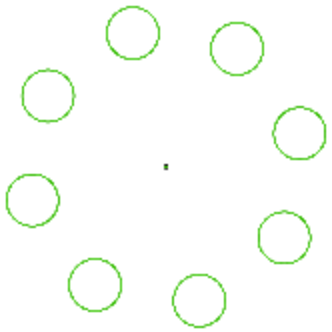


Radius	53	D	20
Number	8	Start angle	14
Step angle	45		

Start point

X	-14	Zmin	0		
Y	0	Zmax	0	H	0

Ok | Cancel | Apply

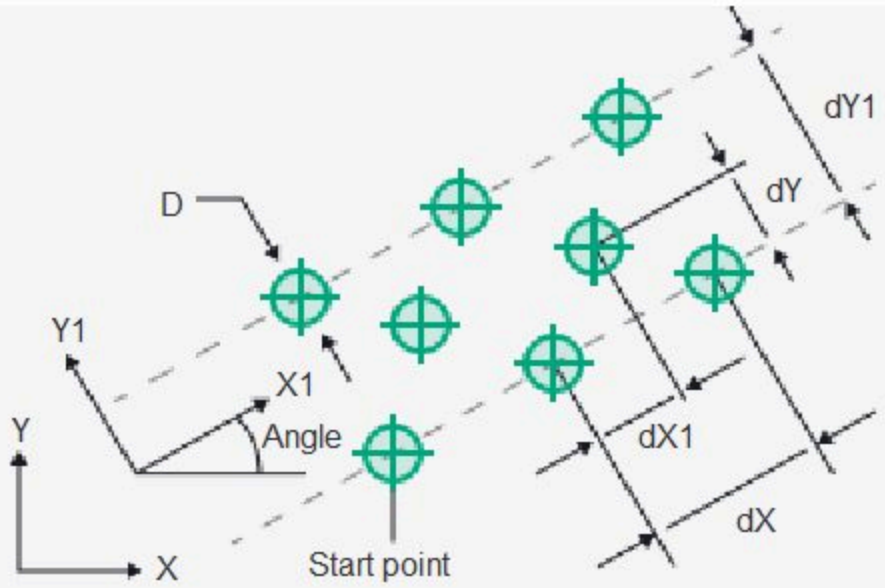


Angular

On the Angular page user can create linear holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

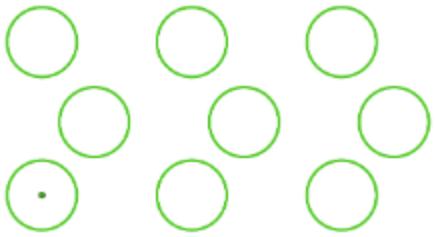


Columns	3	dX	20
Rows	3	dY	20
Angle	0	dX1	10
D	20	dY1	10

Start point

X	-14	Zmin	0		
Y	0	Zmax	0	H	0

Ok | Cancel | Apply

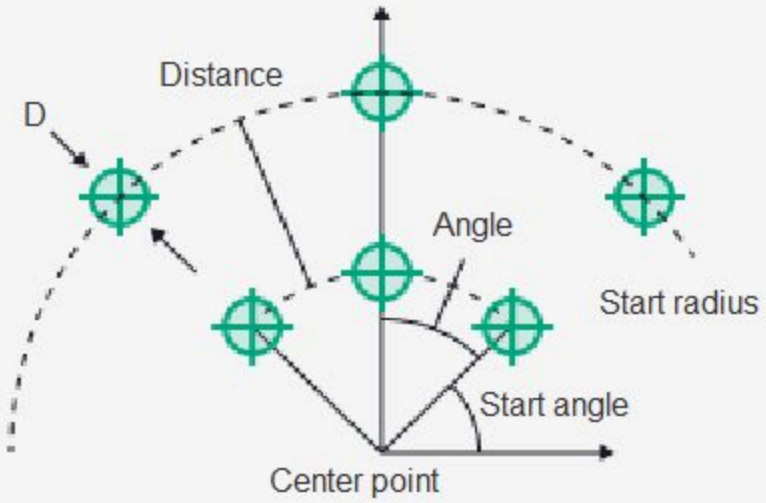


Concentric

On the Concentric page user can create concentric holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

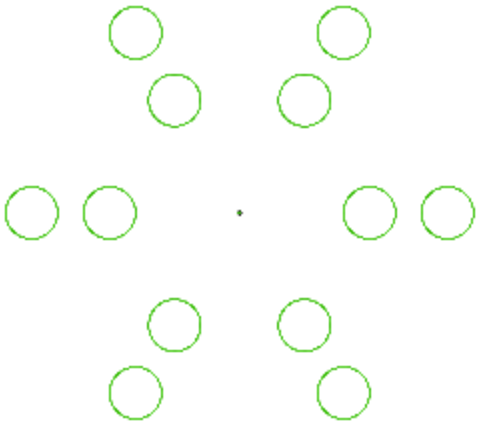


Start radius	50	D	20
Circles num.	2	Start angle	0
Holes num.	6	Distance	30
Angle	60		

Start point

X	-14	Zmin	0		
Y	0	Zmax	0	H	0

Ok | Cancel | Apply

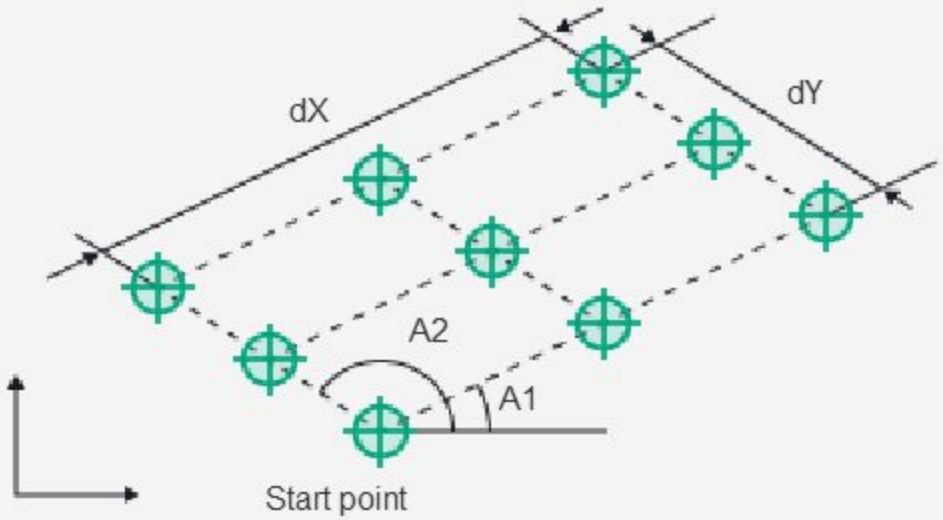


Parallelogram

On the Parallelogram page user can create parallelogram pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram



Columns	5	D	20
Rows	3	Steps	Absolute
Angle1	2	dX	153
Angle2	130	dY	100

Start point

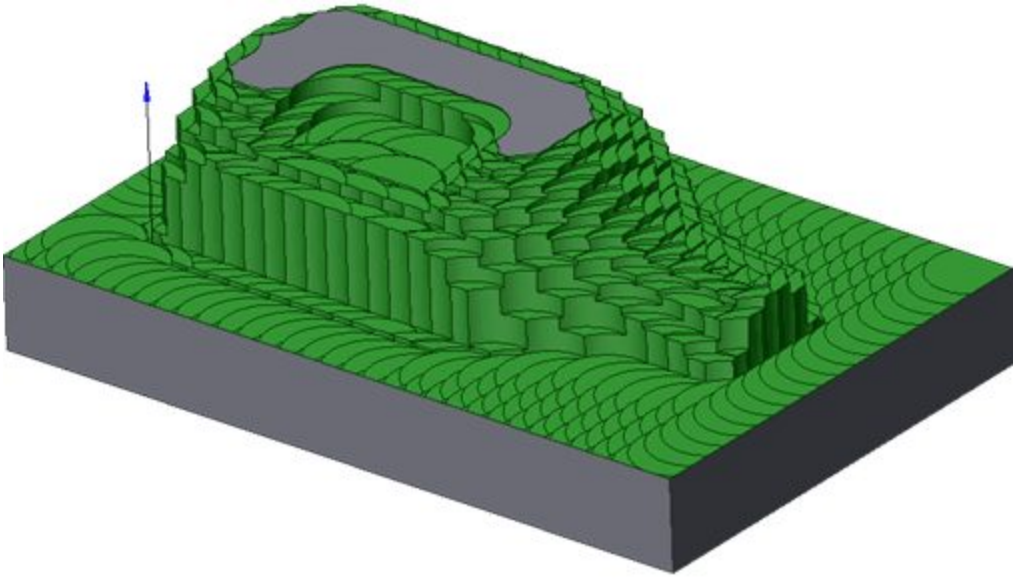
X	-14		Zmin	0	
Y	0		Zmax	0	H 0

Ok | Cancel | Apply

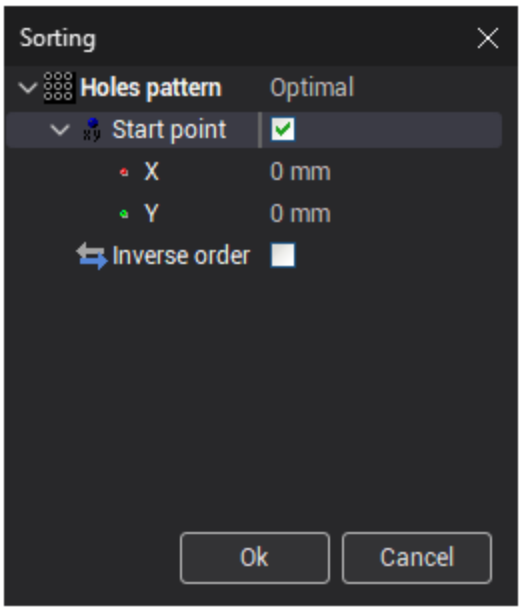


Using patterns for axial plunging

Using the hole patterns together with automatic determination of hole levels allows someone perform roughing machining of the part by the axial plunging strategy.



Holes sorting



Application Area:

The holes sorting window allows you select one of the type of sorting using some parameters to sort holes list.

Sorting patterns

The system has four pattern of holes sorting: Rectangular, Circular (Rings), Circular (Sectors), Optimal.

Rectangular

The pattern Rectangular allows you to sort the holes along XY rows. This type of sorting has the following parameters:

- Start point — starting position of calculation rows
- Rows angle — angle to rotate rows
- Rows step — distance between rows
- Direction — direction to calculate drilling trajectories (Available values: Top-down, Bottom-up)
- Order — order of moving between rows (Available values: One way, ZigZag, Optimal)
- Inverse order — change order in the opposite direction

If Start point is unchecked then the system will select first hole of drilling trajectories.

Circular (Rings)

The pattern Circular (Rings) allows you to sort the holes along the rings located from specified center. T his type of sorting has the following parameters:

- **Center point** — center of calculation rings

- Start angle — angle to set start position
- Start radius — distance between center and first ring
- Step — distance between rings
- Direction — direction to calculate drilling trajectories (Available values: Clockwise, Counter-clockwise)
- Radial order — direction to calculate drilling trajectories (Available values: Outside to inside, Inside to outside)
- Order — order of moving between rings (Available values: One way, ZigZag, Optimal)

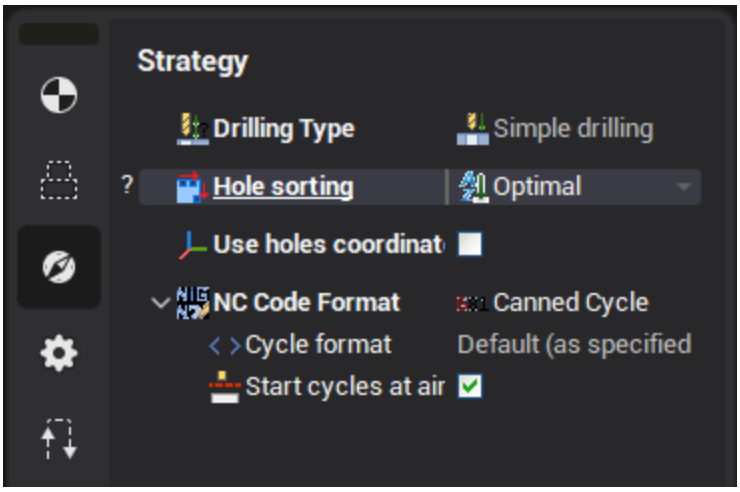
Circular (Sectors)

The pattern Circular (Sectors) allows you to sort the holes using sectors of a circle. T his type of sorting has the following parameters:

- Center point — center of calculation sectors
- Start angle — angle to set start position
- Step — distance between sectors (There are two input mode: Count and Degrees)
- Direction — direction to calculate drilling trajectories (Available values: Clockwise, Counter-clockwise)
- Radial order — direction to calculate drilling trajectories (Available values: Outside to inside, Inside to outside)
- Order — order of moving between sectors (Available values: One way, ZigZag, Optimal)

Optimal

The pattern Optimal is similar to Optimal in Strategy except for configurable settings.

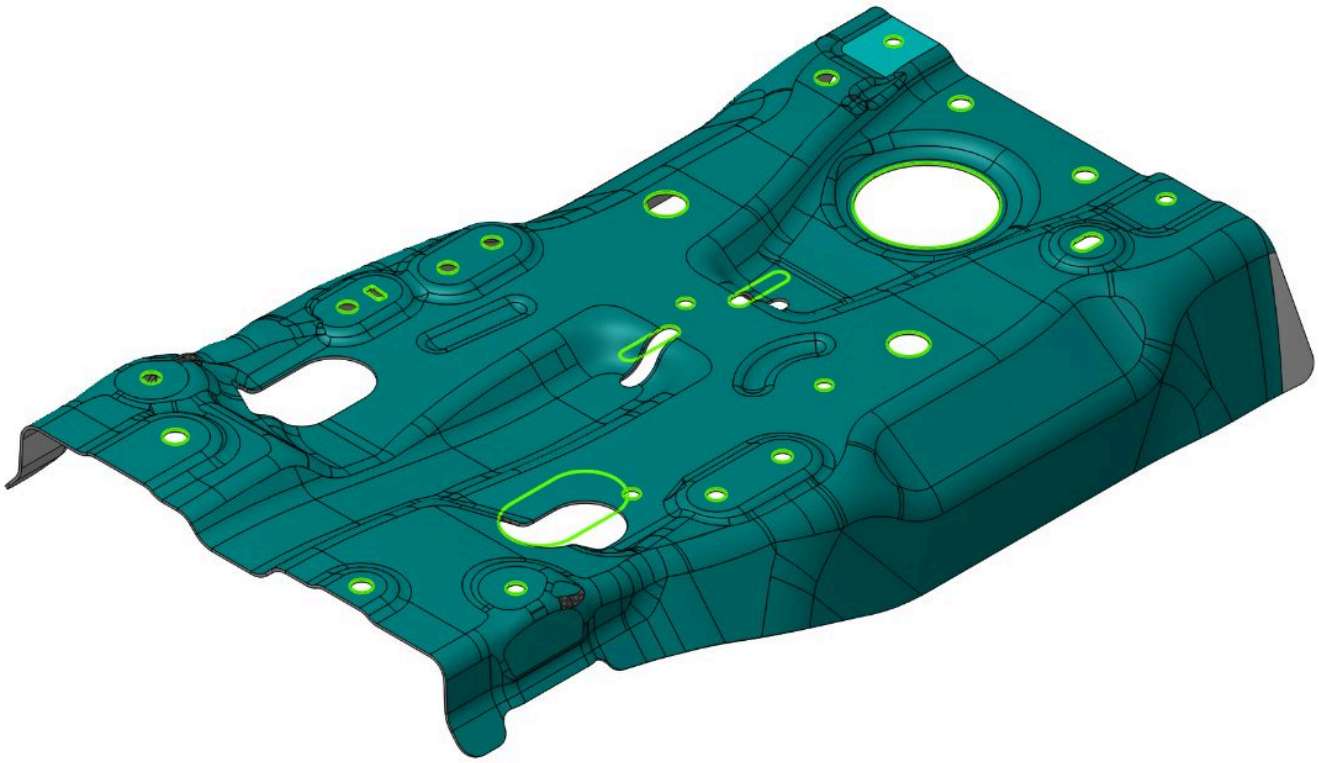


This type of sorting has the following parameters:

- Start point — starting hole to calculation optimal drilling trajectories
- Inverse order — change drilling trajectories in the opposite direction

If Start point is unchecked then the system will select first hole of drilling trajectories.

Recognize features



Application Area:

The Recognize features option has been added for the 6D Contouring operation (Details). It allows recognizing the following constructive elements: Circle, Rectangle, Rounded Rectangle, Slot, Hexagon, and Keyhole. This feature replaces primitive motion commands (line segments, arcs) in NC programs with high-level constructive elements to improve code readability, compactness, and enable advanced post-processing features

Recognize features

Feature Filter

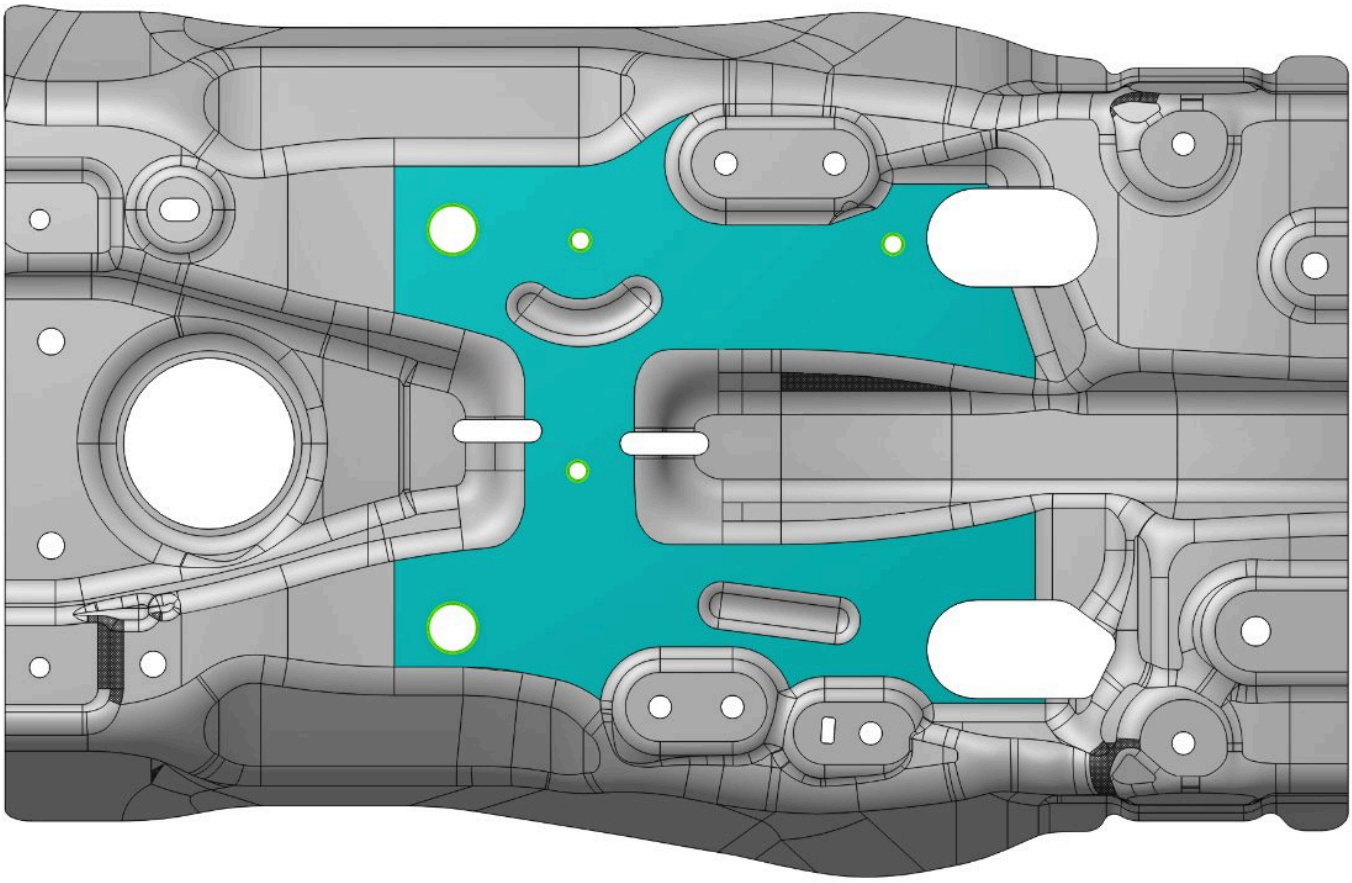
Allows you to select which types of recognize elements will be recognized. The default value is All.

Face Selection

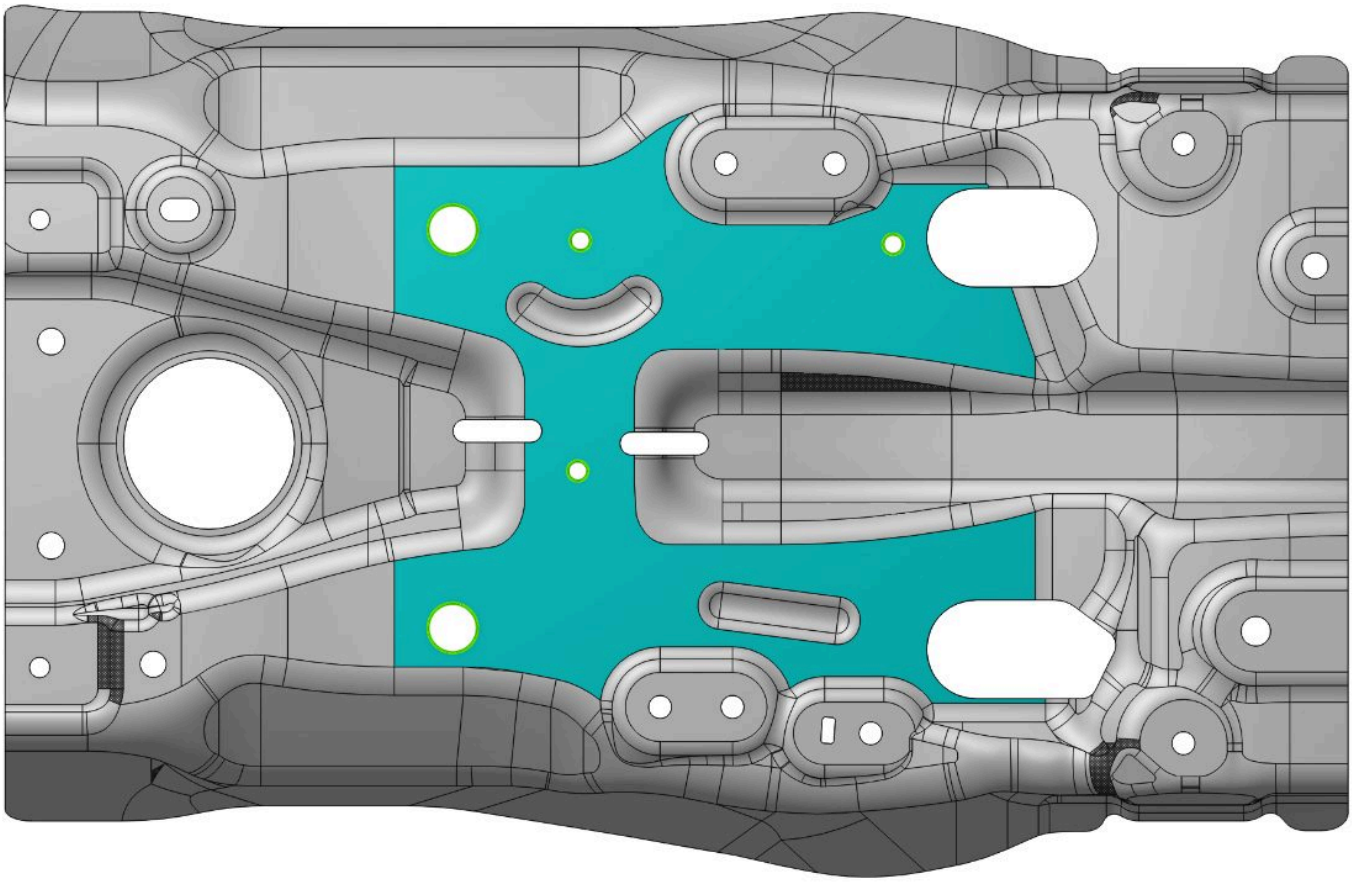
Allows you to select the surface on which the feature will be recognized. (*To select specific surfaces, you need to disable Tangency Propagation.*)

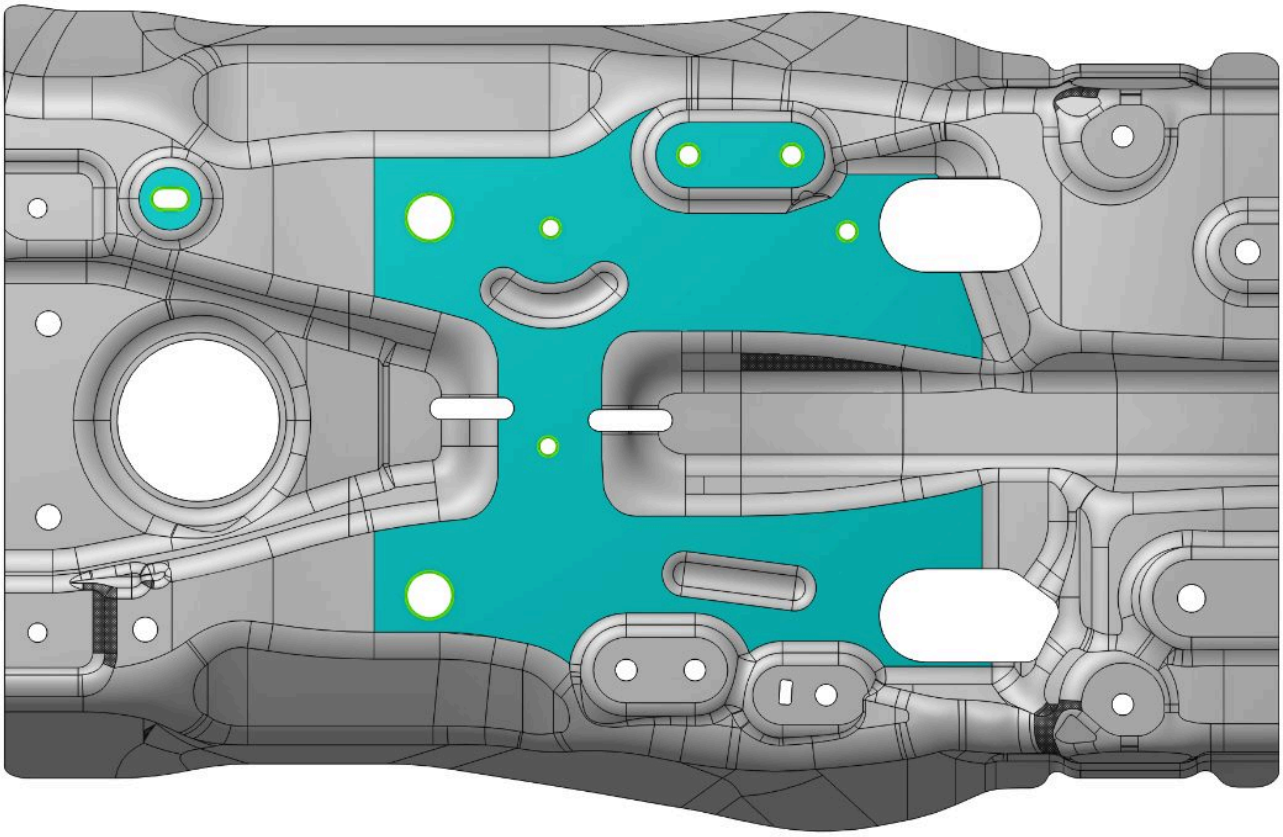
▼ Selecting faces

The selected surface is highlighted in the corresponding color. Additionally, any found elements are also highlighted.

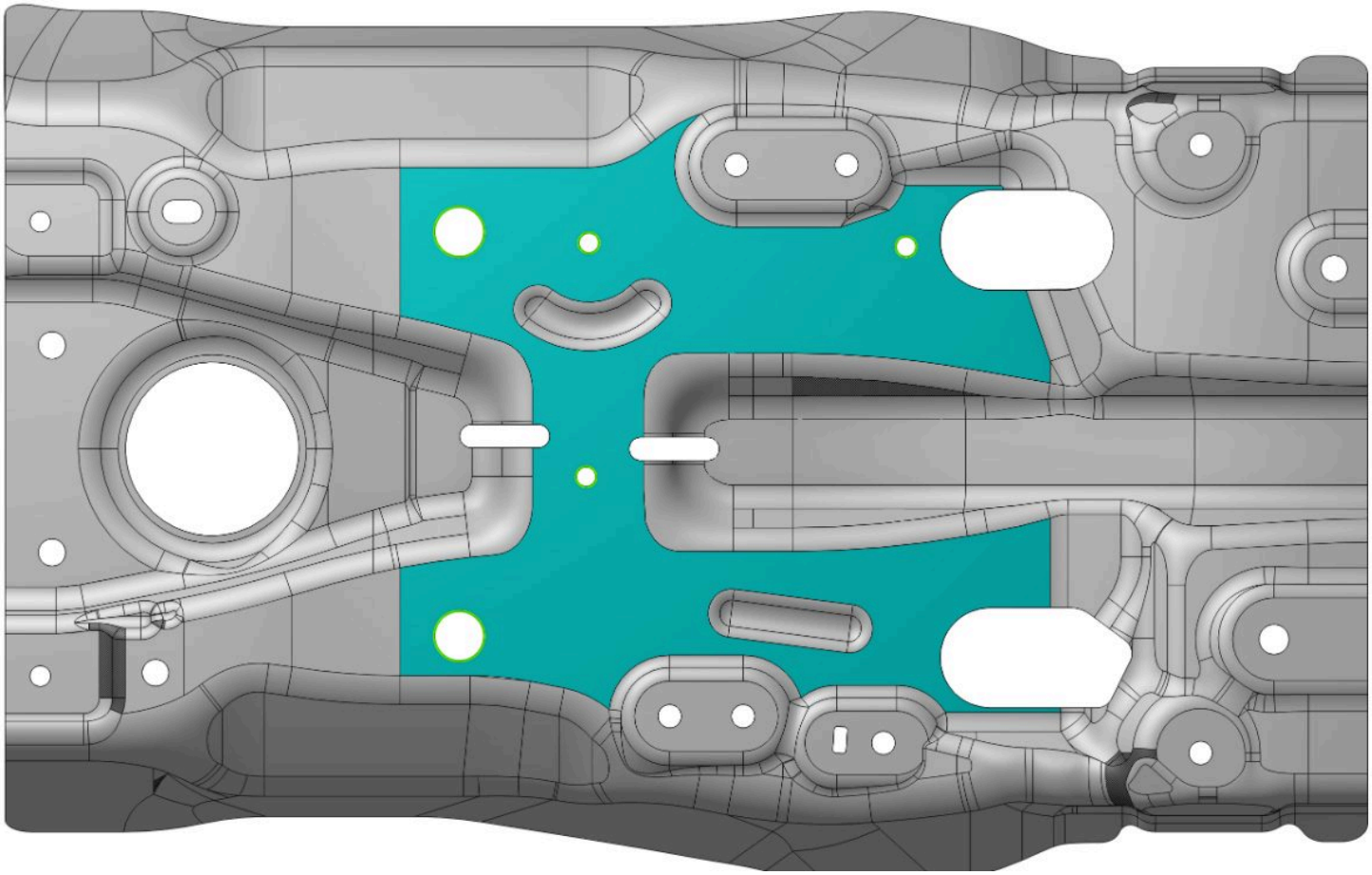


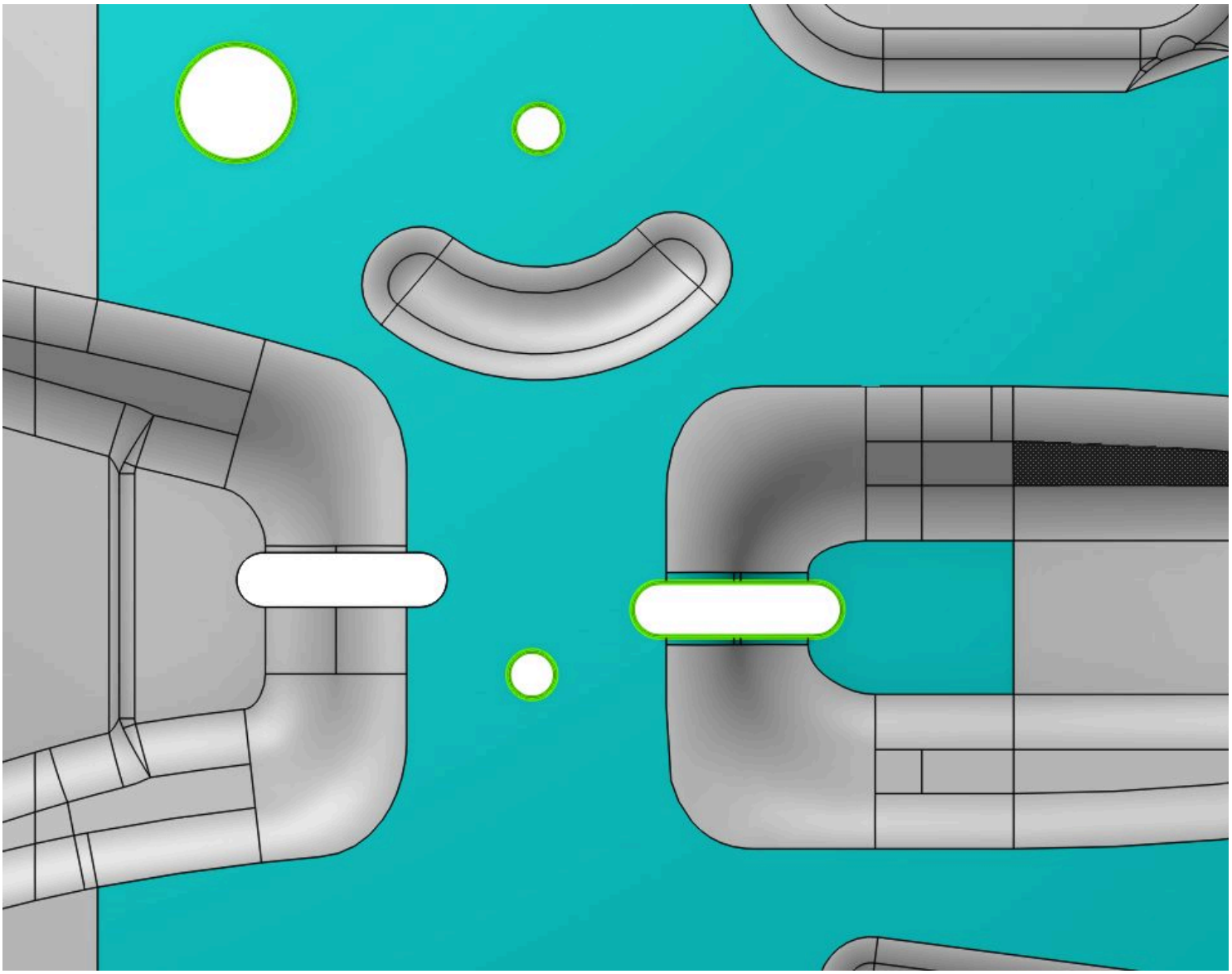
Click on Selection and choose the required surface. By selecting surfaces one by one, you build a list of surfaces. To remove a surface from the list, click on the surface that has already been selected.





If an element is located on multiple surfaces, all relevant surfaces must be selected.

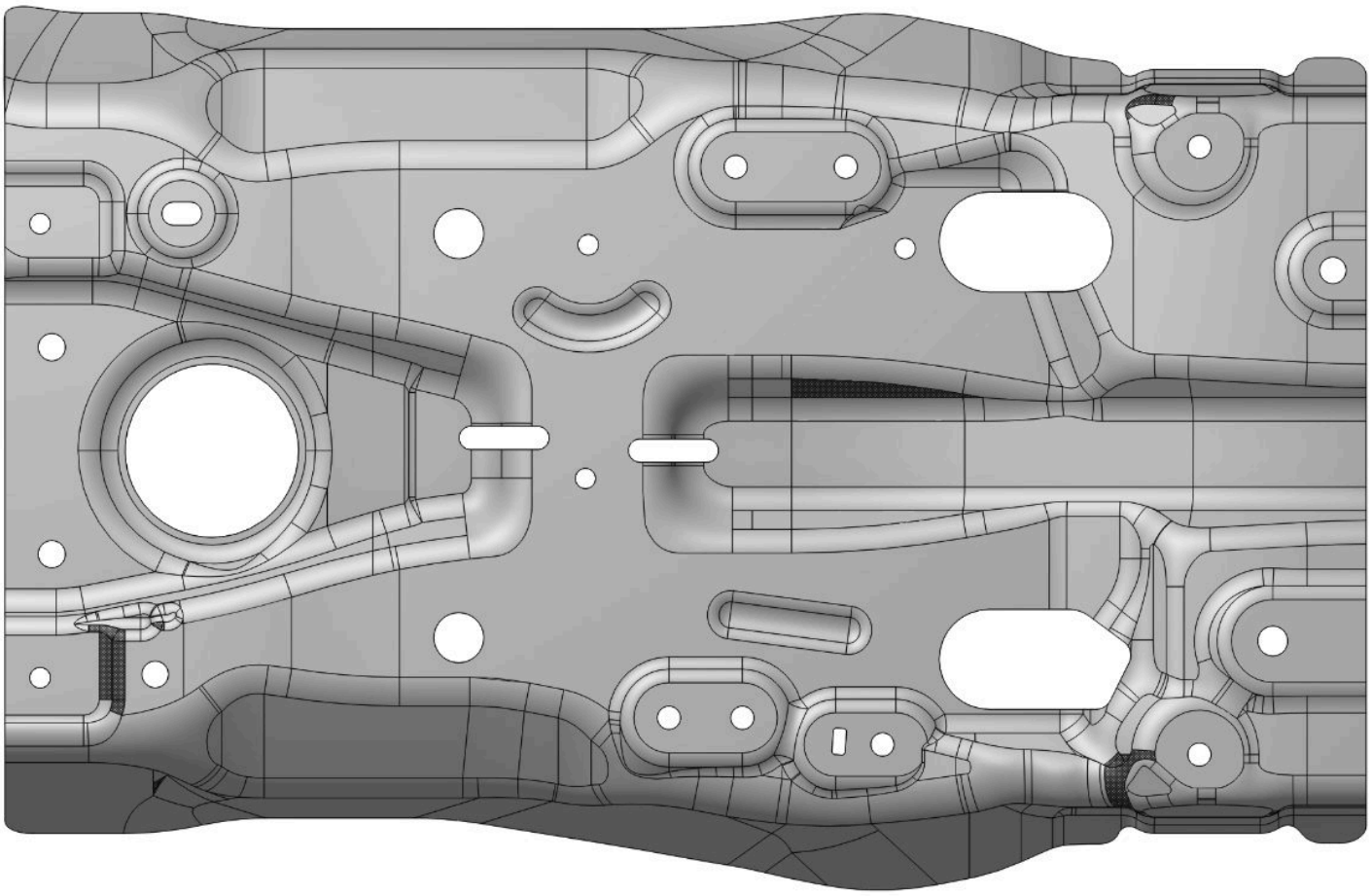


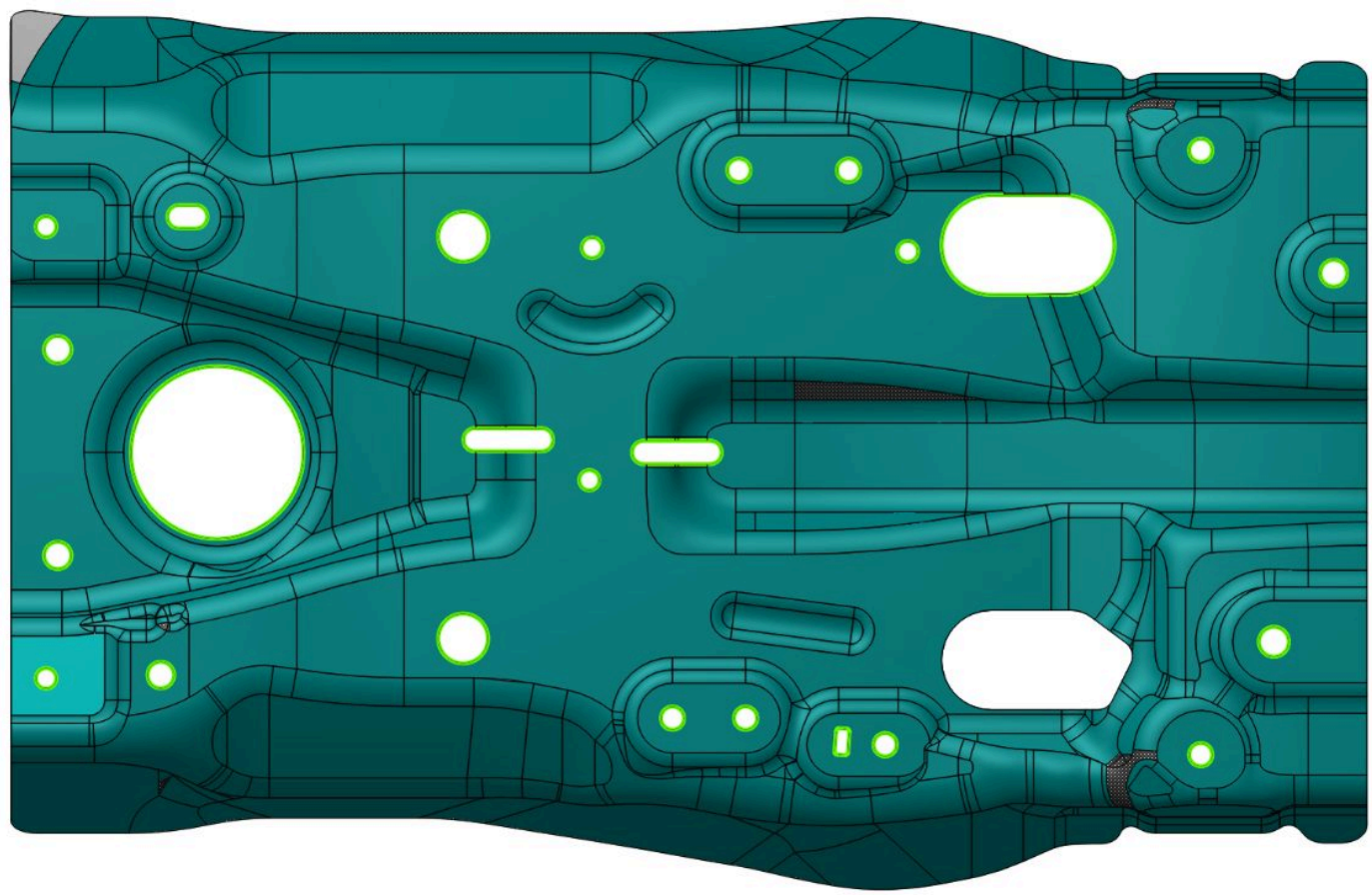


To reduce the labor intensity of surface selection, the Tangency Propagation option is enabled by default.
Tangency Propagation

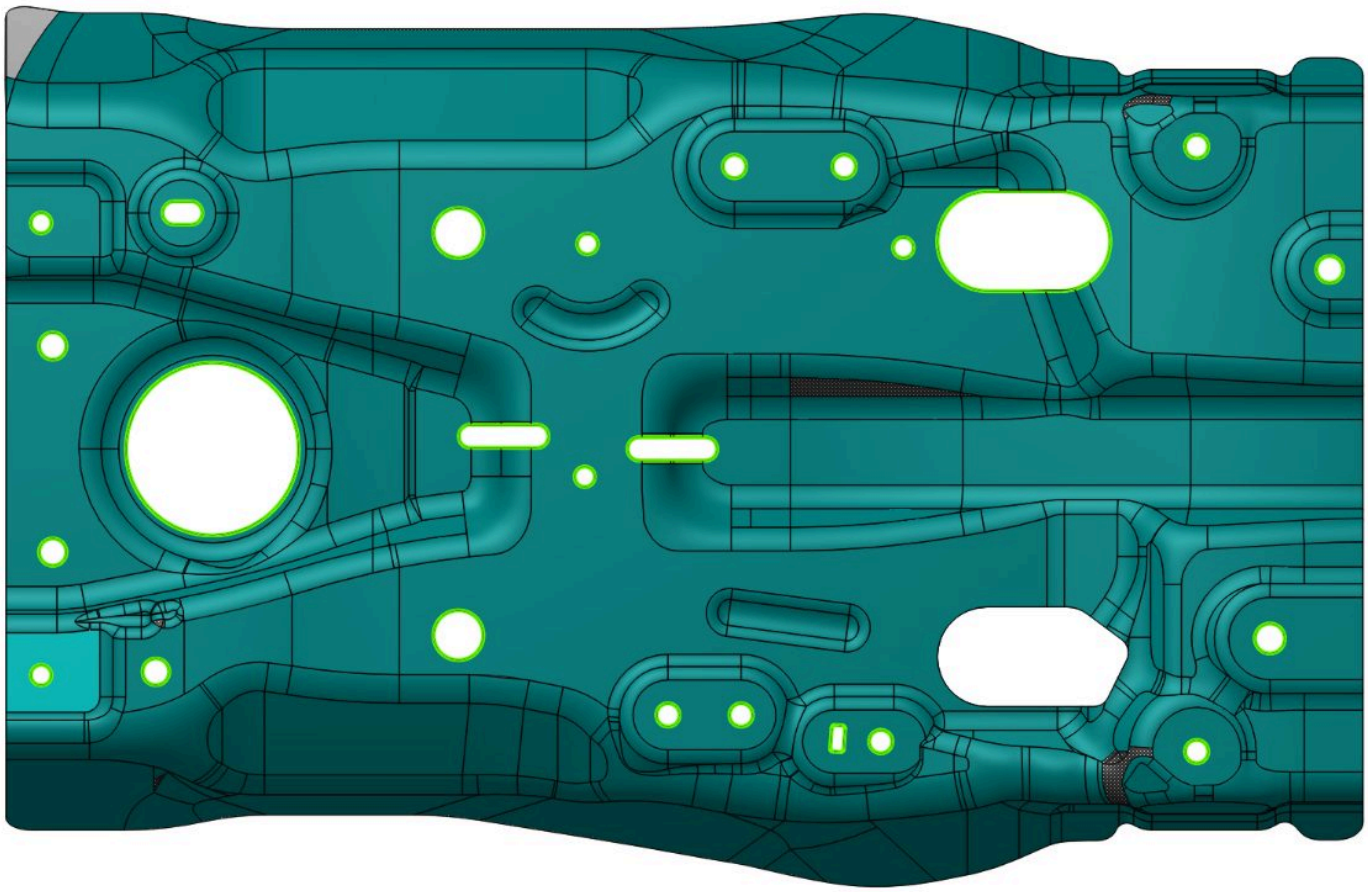
This feature allows selecting smoothed surfaces.

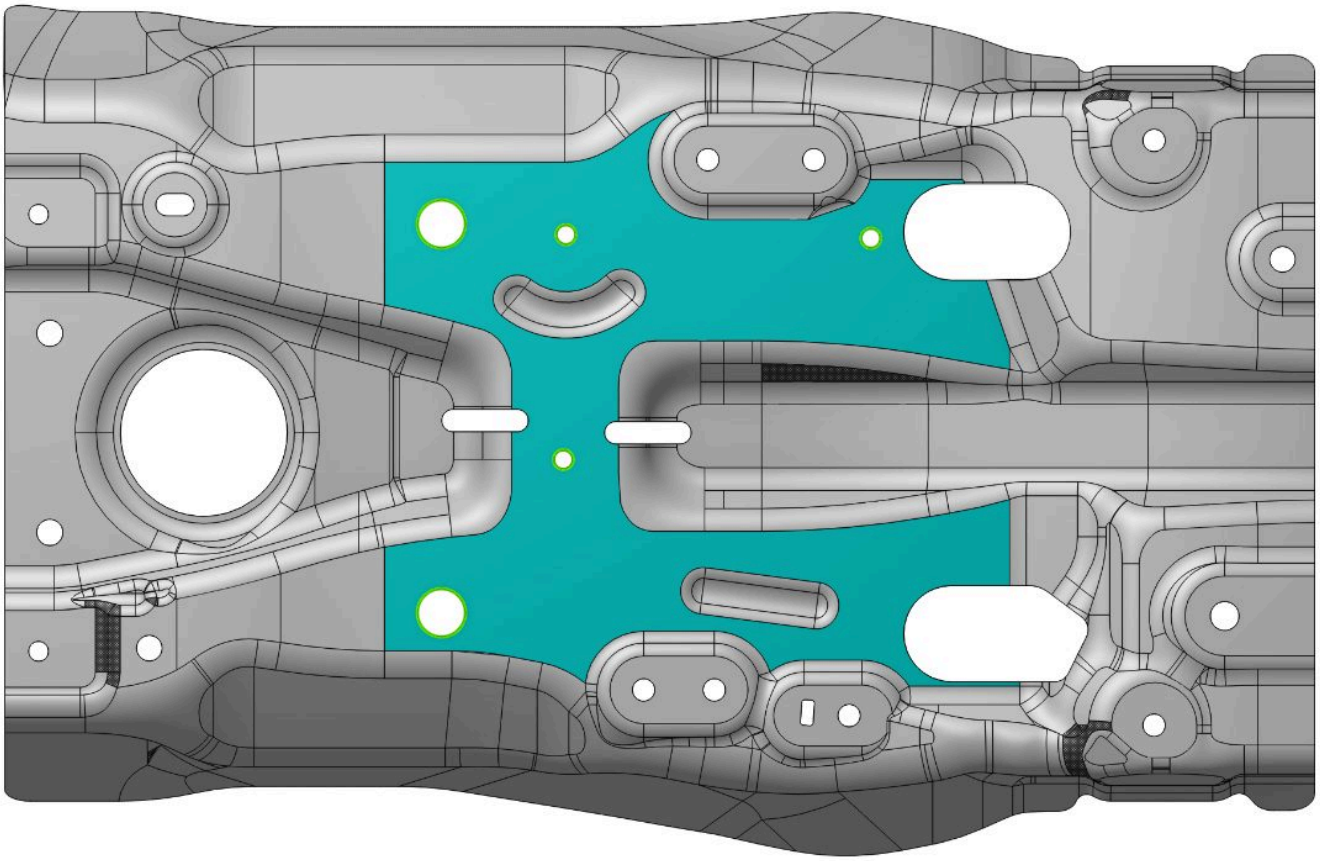
▼ Tangency propagation examples
By default, the angle is set to 0. In this case, the function operates similarly to the Smooth surface selection option. Details





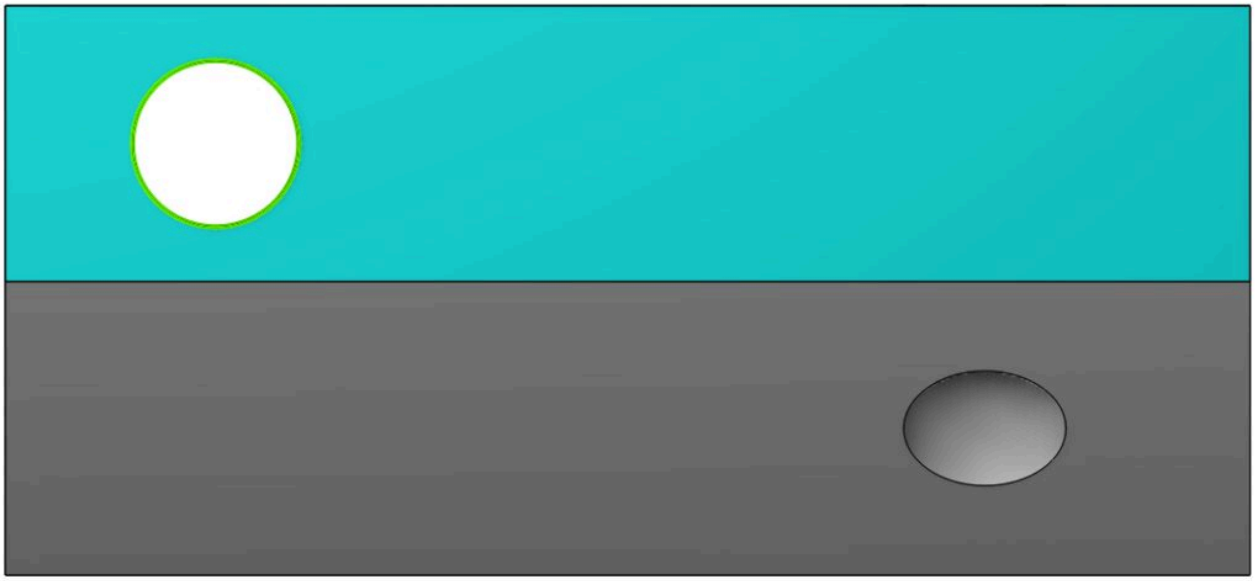
If this option is disabled, the system will recognize elements only on the surfaces that have been explicitly selected.



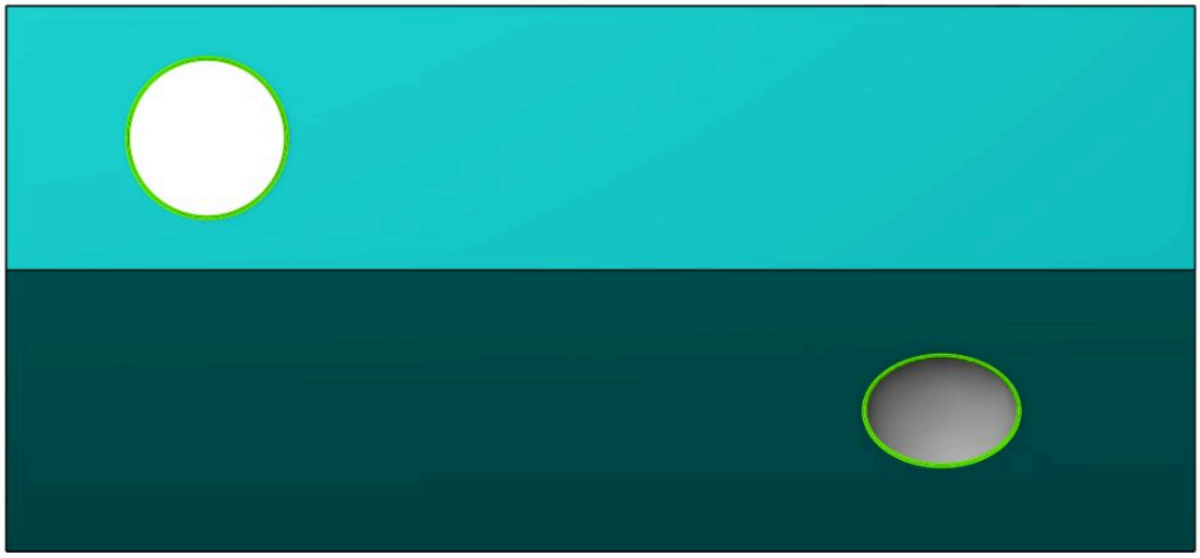


Increasing the angle allows selecting non-smoothed surfaces as well.

Angle = 0



Angle = 45

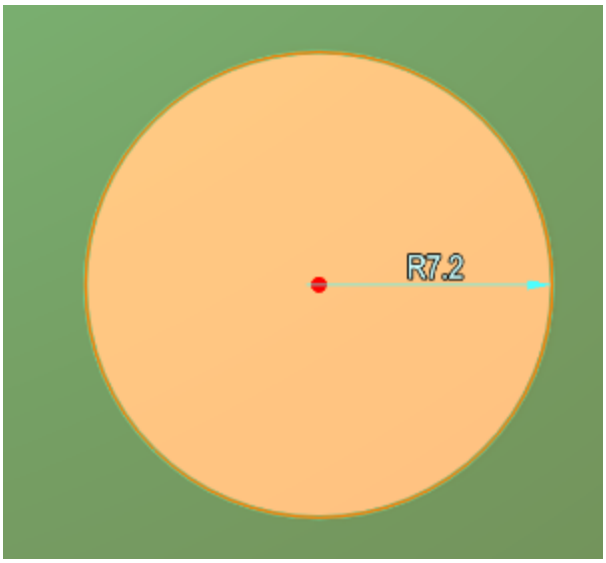


Recognized Features

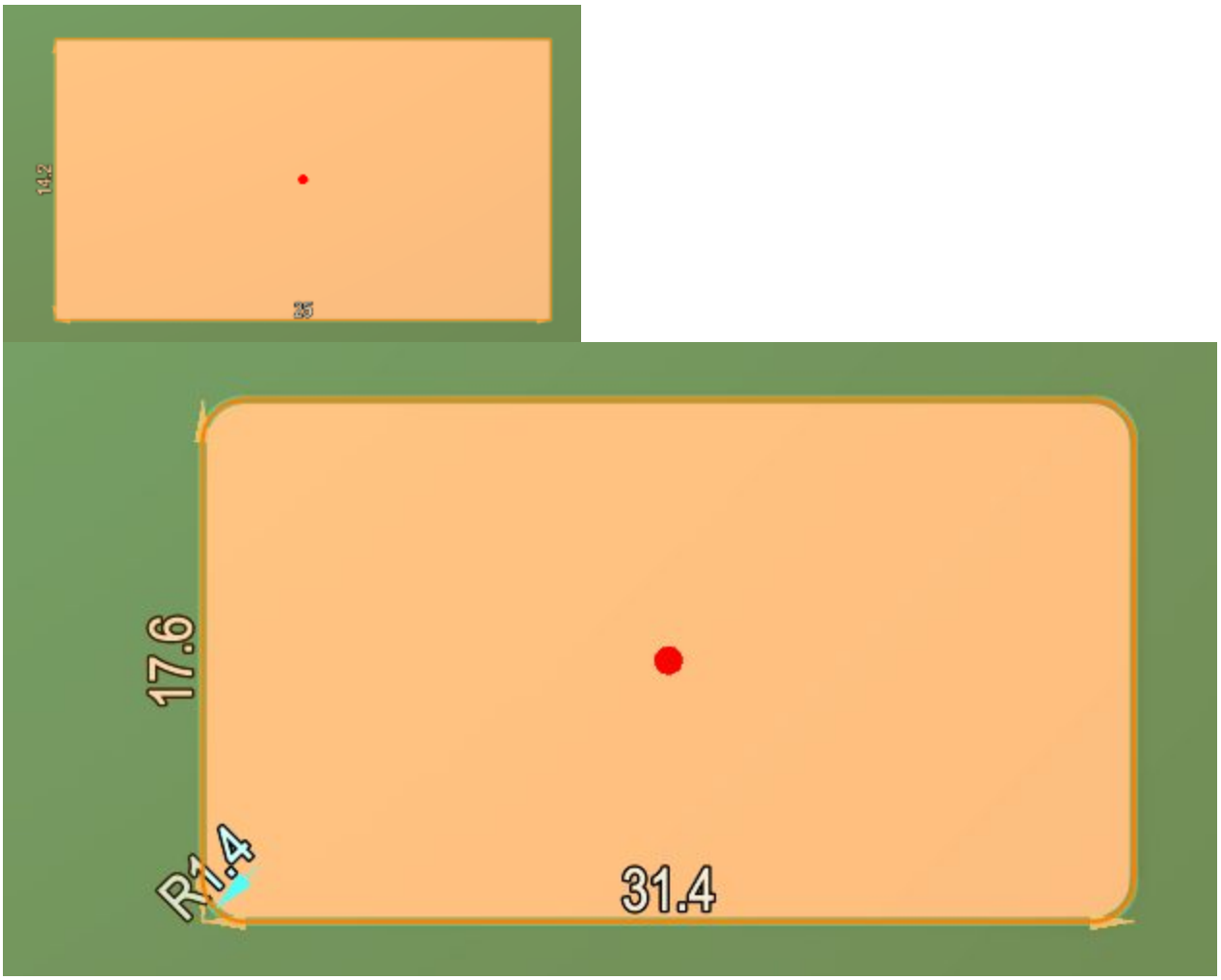
List the types of recognize elements.

▼ Recognized feature types

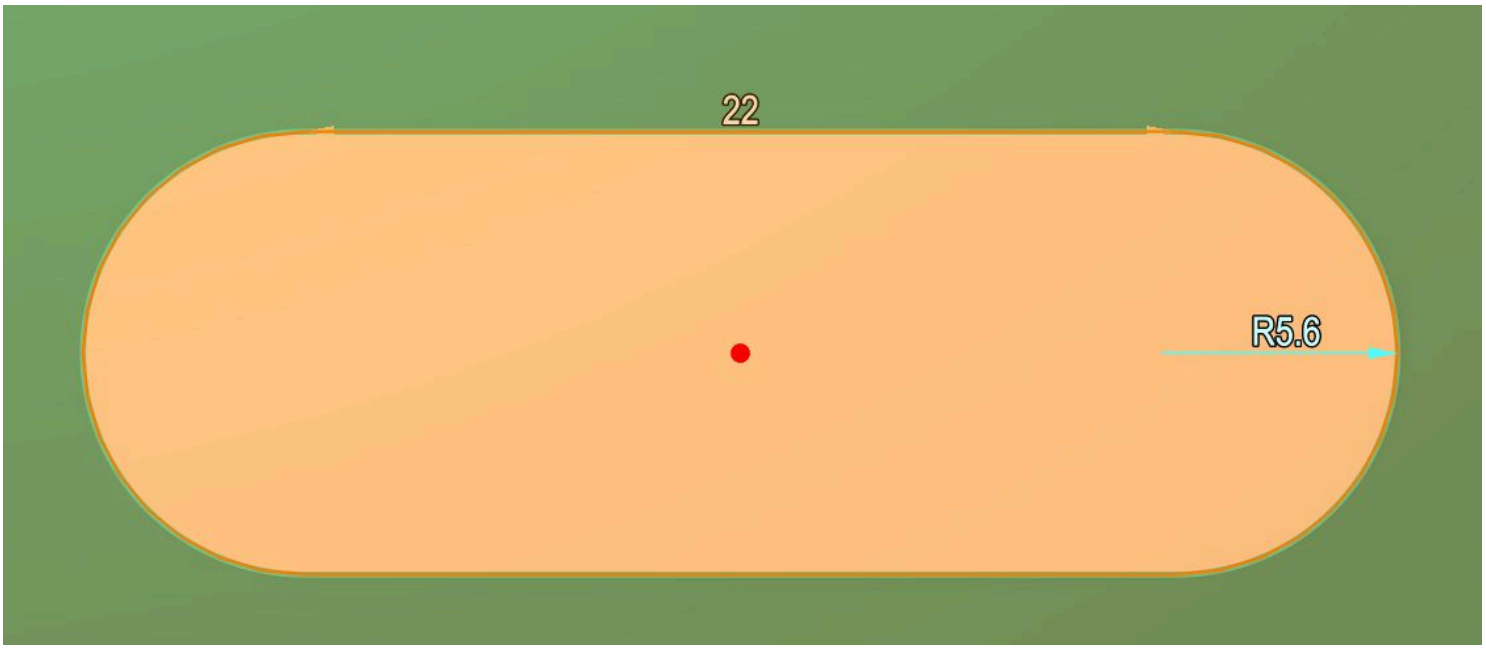
Circle. Defines a circle (it is possible to change the circle's radius).



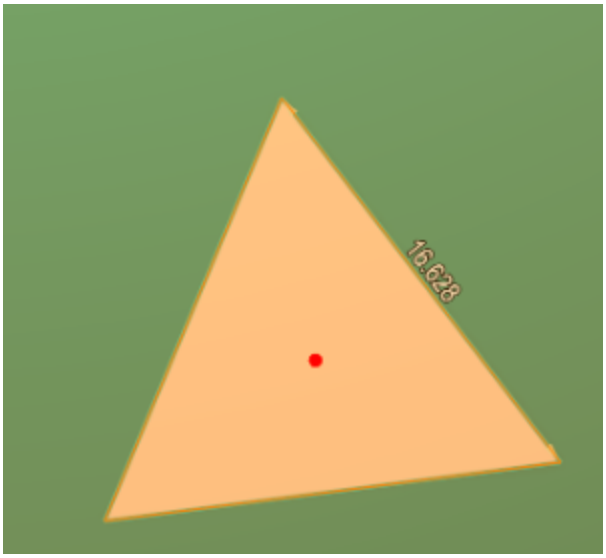
Rectangle. Allows defining a rectangle with or without corner rounding (It is possible to modify the width, length and corner radius.)

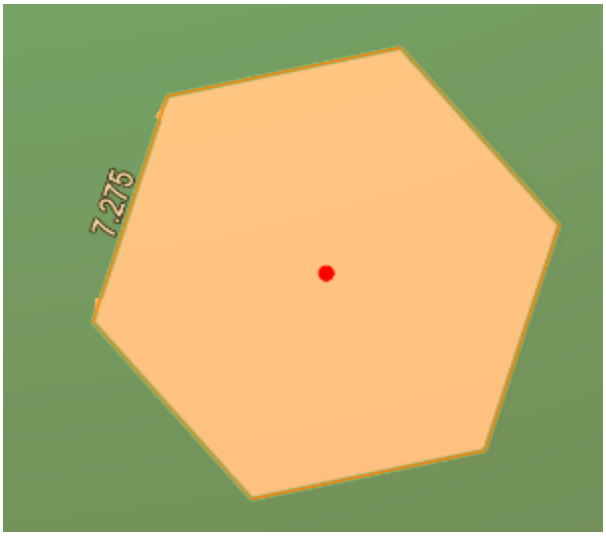


Slot. Defines a slot (it is possible to change the slot's length and radius, as well as the radius of the semicircular ends).

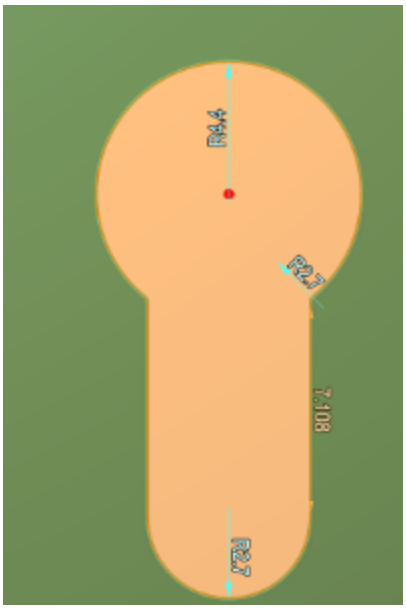


Polygon. The Polygon tool identifies all polygonal shapes having between 3 and 6 sides inclusive (edge length can be changed.).

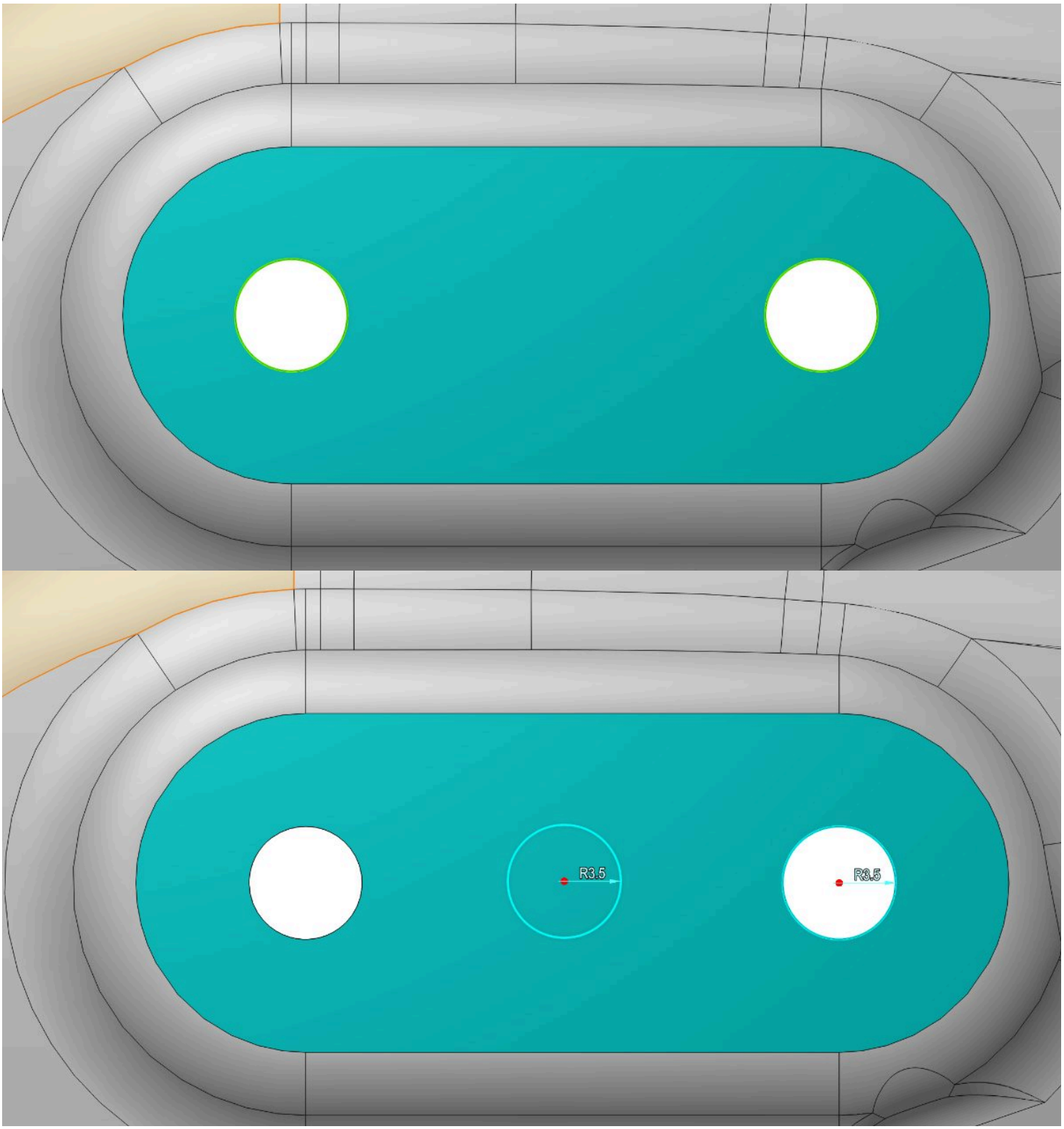




KeyHole. Defines a keyhole shape (it is possible to adjust the dimensions of both the circular part and the slot-like extension of the keyhole).



In addition to modifying the recognized elements, all elements can be moved within the selected plane by grabbing the red reference point at the element's centre.

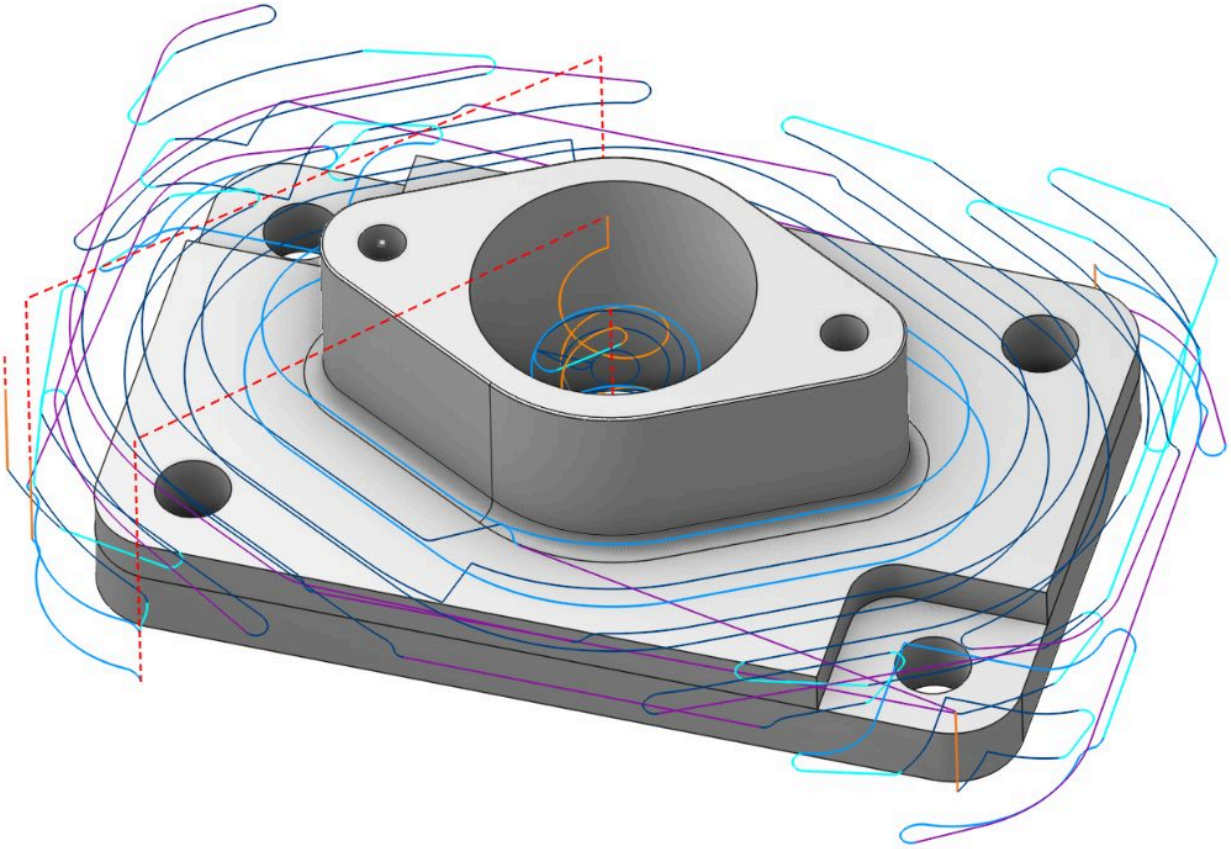


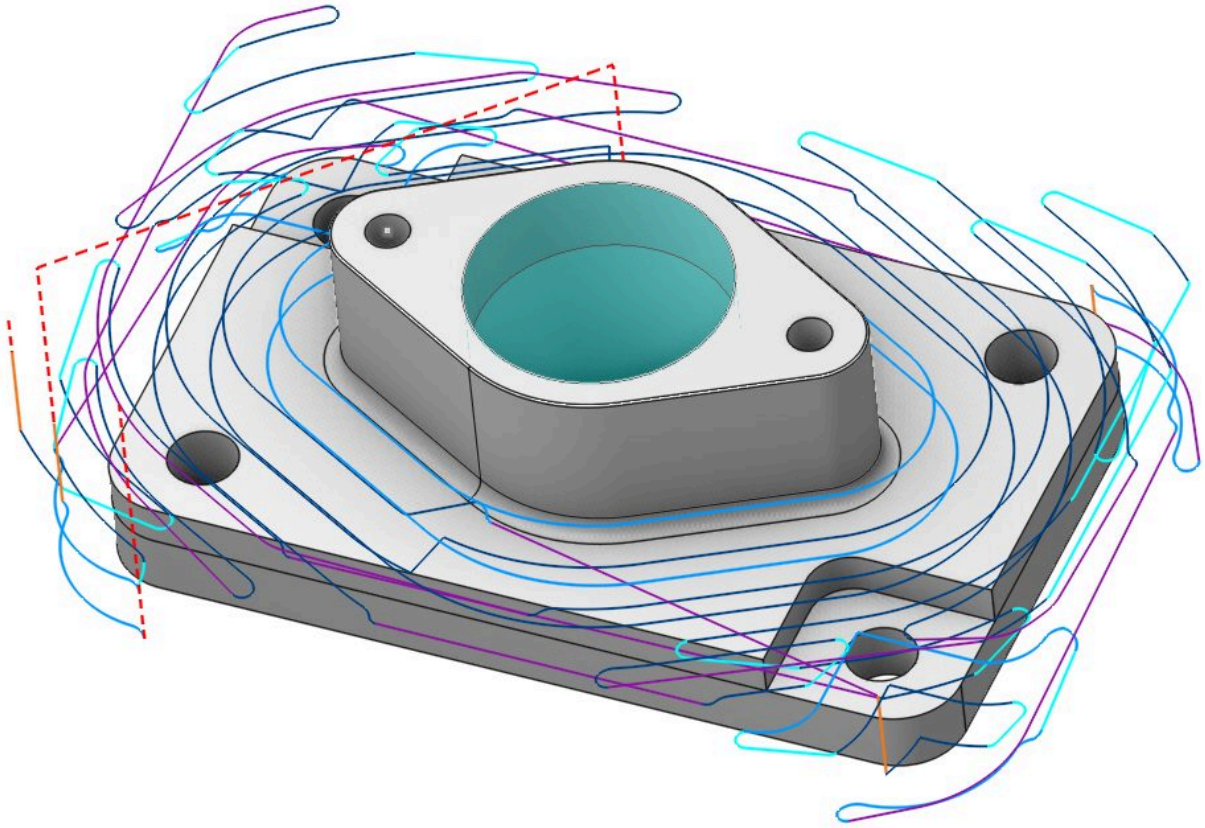
After the toolpath calculation, the CLDATA will contain commands corresponding to the constructive elements.

- Circle(Radius(7.2))
- KeyHole(Width(7.11), Head Radius(4.4), Neck Radius(2.7))
- Polygon(Width(7.27))
- Polygon(Width(8.43))
- Polygon(Width(16.63))
- Rect(Width(25), Height(14.2), Radius(0))
- Rect(Width(31.4), Height(17.6), Radius(1.4))
- Slot(Width(22), Height(5.6))

The trajectory output method can be controlled on the Parameters tab in the Features group. Details

Restrictions



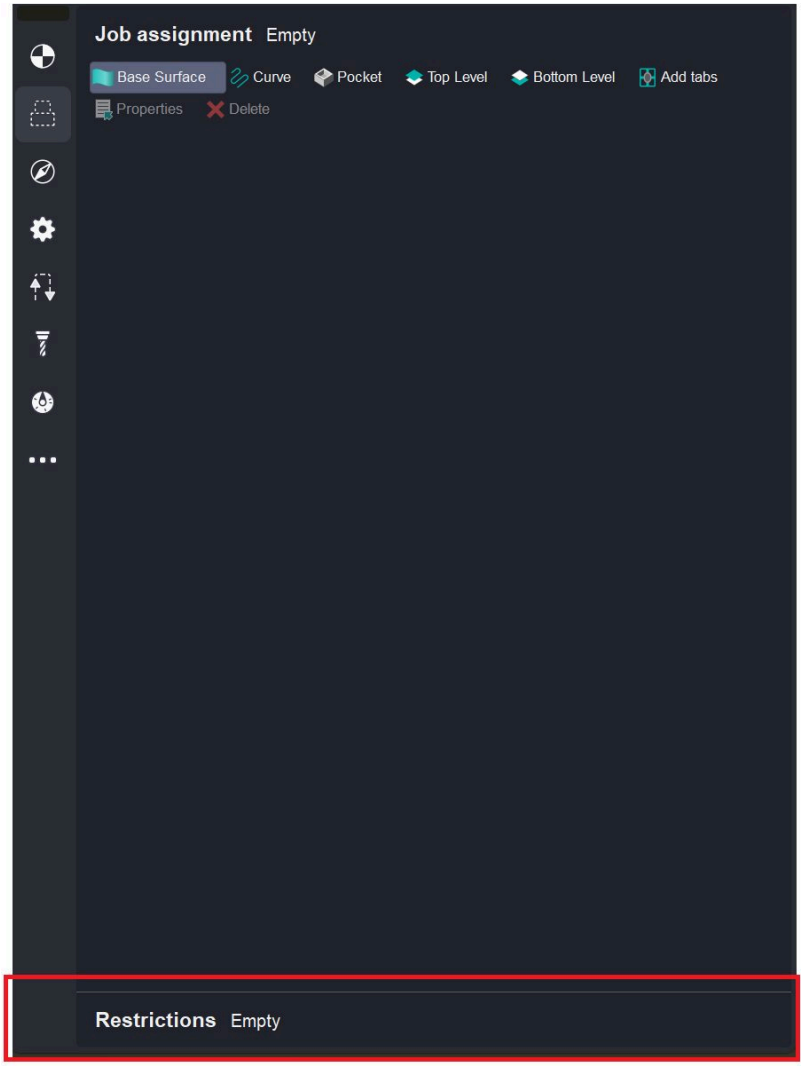


Application Area:

It allows you to restrict areas that should not be machined. A distinguishing feature compared to fixtures (Details) is that these constraints are applied within the scope of a single operation, whereas fixtures are utilized across the entire Setup stage (Details).

Defining restrictions

It is located at the bottom of the Job assignment. Double-clicking it expands this menu for easier geometry definition. Double-clicking it again collapses the section.



Methods for determining the restrictions are indicated here. Details

Visibility

Visibility is controlled by turning on and off the visibility of the Job assignment visibility.

Operation parameters

Application area:

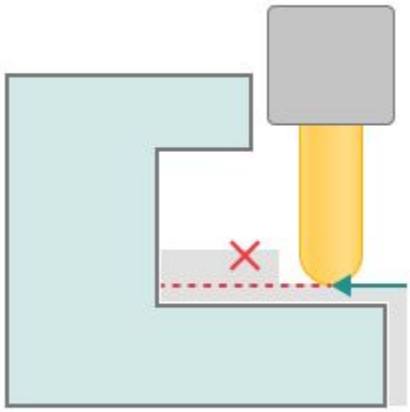
This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory.

Below are all the parameters. Depending on the specific operation being used, certain parameters will be available.

Check part.

The part geometry is taken into account while calculating the toolpath. A gouge-free toolpath is guaranteed, however the calculation time can be long when the part geomety is complex.

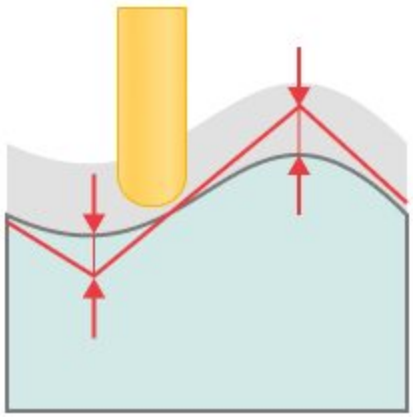
▼ Click here to expand..



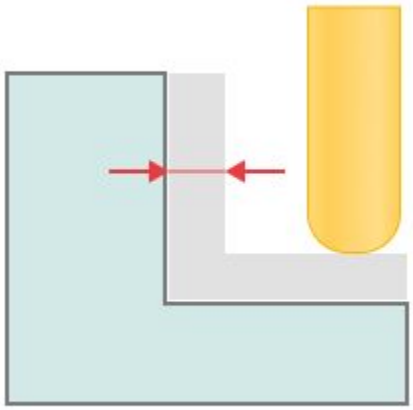
Tolerance. This feature defines the 'maximum deviation' between the 'ideal' and 'approximated' toolpath. A larger tolerance amount reduces the number of moves in the toolpath. The toolpath calculation aims to ensure that deviations from the part geometry do not exceed the specified tolerance.

One should note that increasing the tolerance would increase the calculation time and the size of the NC program.

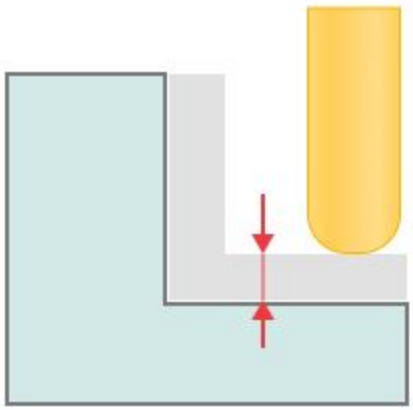
The tolerance must be more than zero, otherwise it will be impossible to construct an approximated toolpath.



Radial stock. It defines the stock in radial direction of the tool for the future machining.



Axial stock. It defines the stock in axial direction of the tool for the future machining.



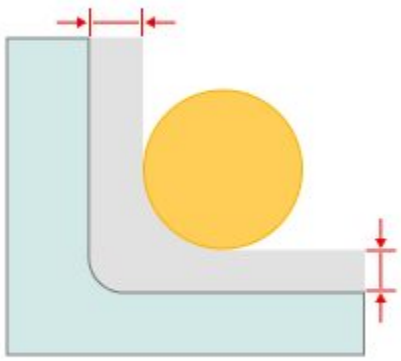
Use fast calculation method. It uses more fast algorithms for the calculation. Try to disable this option if toolpath tolerance doesn't suits you. If Use fast calculation method is used, toolpath calculation time reduces dramatically. A speedup on complex parts may reach up to 10–20 times.

By now, fast toolpath calculation methods use triangulated surfaces as input data. This fact involves some limitations that should be taken into consideration while generating toolpath. First of all, the quality of the generated toolpath may be worse than when using traditional exact methods. The maximum deviation of the toolpath depends heavily on the value specified in the Tolerance panel of the operation parameters dialog. This value determines the tolerance of model triangulation. Secondly, triangulated surfaces require a lot of operation memory.

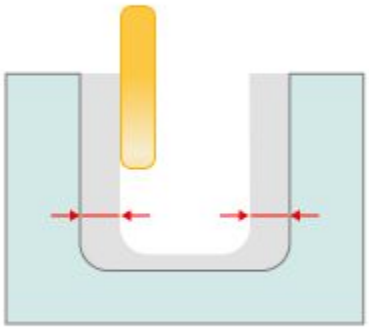
Described limitations are today a common place for all CAM systems, and only our CAM systems gives you the alternative between speed of approximate calculation methods and the quality of exact calculation methods.

Use new calculation method. The new algorithm enables fast multi-core computation and ensures a smooth tool path, taking into account surface normals and the side angle. Details

Disc diameter stock. This design allows for a stock to be left at the walls of the parts, ensuring sufficient material for subsequent processing or fitting



Disc thickness stock. Disc thickness stock allows for a margin to be left adjacent to the part wall on the side of the disc.



Work passes interpolation.

▼ Click here to expand..

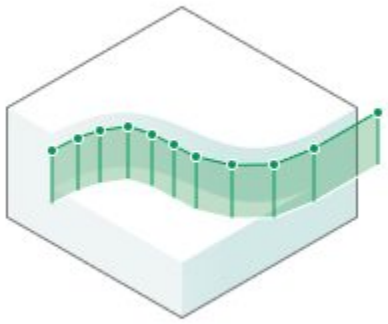
Auto. The system automatically selects the interpolation mode based on machine capabilities and path type.

- If spatial arcs are supported and the path is 5-axis or 6-axis, "Spatial arcs" will be used.
- If only planar arcs (XY, YZ, ZX) are supported and the path is 3D, "Planar arcs and helices" will be used.
- If arcs are not supported, "Deviation-based" segmentation will be used.

Off. Toolpath is used as-is. No interpolation.

Deviation-based. Interpolates the path using linear segments within the specified linear and angular deviation limits.

▼ Click here to expand..

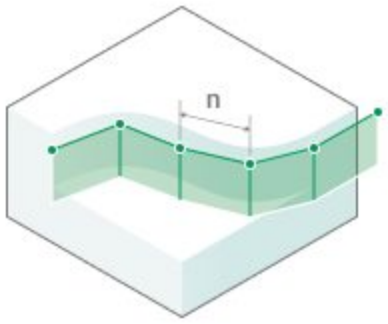


Max linear deviation. Sets the maximum distance the resulting line segment can deviate from the original spatial trajectory. Decreasing the value results in a more accurate approximation; increasing it results in a coarser approximation, fewer segments, and faster calculations, but may result in noticeable errors in the trajectory's shape.

Max angular deviation. Sets the maximum allowable rotation between adjacent line segments in the approximation. A low value smooths out the turns, creating more segments at the points of curvature; a high value allows sharper angles between segments.

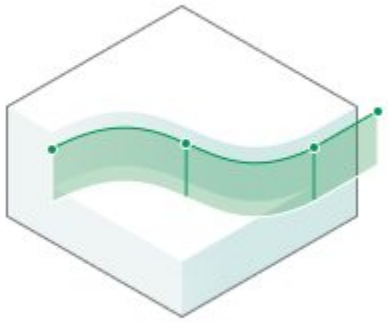
Fixed segment length. Divides the path into equal-length segments. Works best for smooth trajectories. May clip sharp corners.

▼ Click here to expand..



Segment length. Specifies a fixed length of linear segments into which the original toolpath is divided during interpolation.
Planar arcs/helices. Arcs interpolation in the XY / YZ / ZX planes with the Circle command (depending on the operation).

▼ Click here to expand..



Max linear deviation. Sets the maximum distance the resulting line segment can deviate from the original spatial trajectory. Decreasing the value results in a more accurate approximation; increasing it results in a coarser approximation, fewer segments, and faster calculations, but may result in noticeable errors in the trajectory's shape.

Max angular deviation. Sets the maximum allowable rotation between adjacent line segments in the approximation. A low value smooths out the turns, creating more segments at the points of curvature; a

high value allows sharper angles between segments.
Spatial arcs. The path is approximated with arcs in arbitrary planes and line segments within the specified deviations. The mode is available when the machine supports spatial arcs.

▼ Click here to expand..
Max linear deviation. The maximum distance by which the arcs and segments can deviate from the original path. A smaller value gives a more accurate approximation and more elements.

Max angular deviation. The maximum deviation of the tool axis from its original orientation within one segment or arc. The parameter is shown when the operation uses rotary axes.

Max angle. The maximum central angle of one arc: Default (140°), 180° or 90°. A longer arc is split into several arcs.

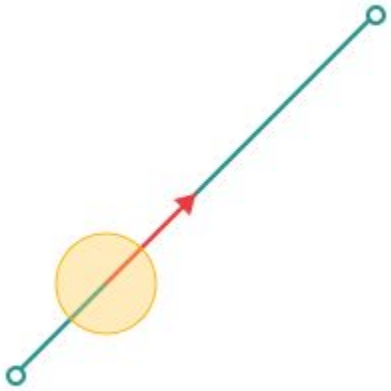
Fixed length spatial arcs. The path is divided into sections of the specified length. Each section is output as an arc through its start, middle and end points; straight sections are output as line segments. The mode is available when the machine supports spatial arcs.

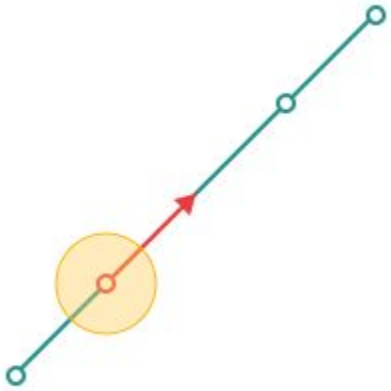
▼ Click here to expand..
Segment length. The length of a path section replaced by one arc.

Max motion length.

▼ Click here to expand..
If enabled: The tool path motions will be divided to not exceed the given length. Smaller trajectory segments enhance contour-following accuracy, particularly on high-precision machines.

If disabled:. Motion moves will not be divided.



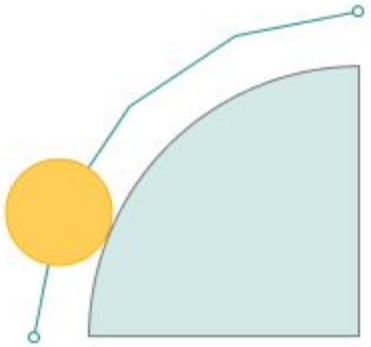


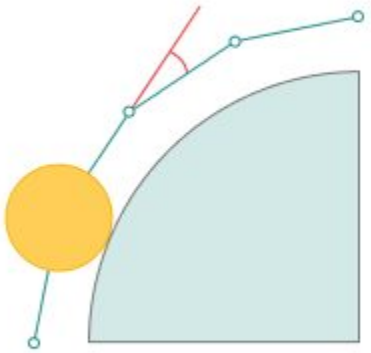
Max motion rotation.

▼ Click here to expand..

If enabled: The tool motion trajectory is divided into segments so that the rotation angle on each segment does not exceed the specified value. The CAM system automatically inserts intermediate rotation points if the calculated rotation angle on a segment exceeds the Max motion rotation limit.

If disabled:. Motion moves will not be divided.





Links interpolation.

These parameters define the interpolation of links, the moves between work passes: the retract from the machined surface, the transition to the next work pass and the engage. Links can be interpolated differently from work passes. For example, work passes can be output as spatial arcs and links as line segments.

The group is available in the same operations as Work passes interpolation. It is hidden when Planar arcs/helices is selected for work passes.

▼ Click here to expand..
Same as work passes. Links are interpolated with the same mode and parameters as work passes (the Work passes interpolation group). This mode is the default.

Off. Links are output as calculated, without interpolation.

Deviation-based. Links are approximated with line segments within the specified deviations.

▼ Click here to expand..
Max linear deviation. The maximum distance by which the segments can deviate from the calculated link path. A smaller value gives a more accurate approximation and more segments.

Max angular deviation. The maximum deviation of the tool axis from its calculated orientation within one segment or arc. A smaller value gives more segments where the tool axis rotates. The parameter is shown when the operation uses rotary axes.

Fixed segment length. Links are divided into segments of equal length; the last segment can be shorter. Sharp corners can be cut off.

▼ Click here to expand..
Segment length. The length of the segments into which links are divided.

Spatial arcs. Links are approximated with arcs in arbitrary planes and line segments within the specified deviations. The mode is available when the machine supports spatial arcs.

▼ Click here to expand..
Max linear deviation. The maximum distance by which the arcs and segments can deviate from the calculated link path. A smaller value gives a more accurate approximation and more elements.

Max angular deviation. The maximum deviation of the tool axis from its calculated orientation within one segment or arc. The parameter is shown when the operation uses rotary axes.

Max angle. The maximum central angle of one arc: Default (140°), 180° or 90°. A longer arc is split into several arcs.

Fixed length spatial arcs. Links are divided into sections of the specified length. Each section is output as an arc through its start, middle and end points; straight sections are output as line segments. The mode is available when the machine supports spatial arcs.

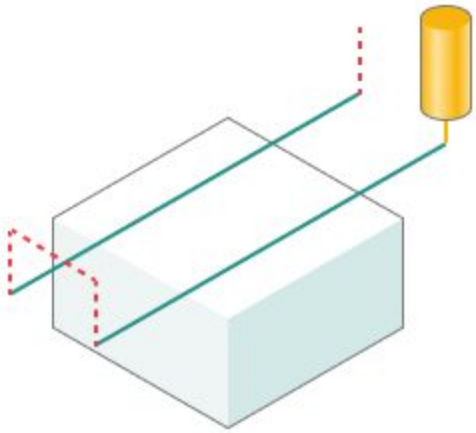
▼ Click here to expand..
Segment length. The length of a link section replaced by one arc.

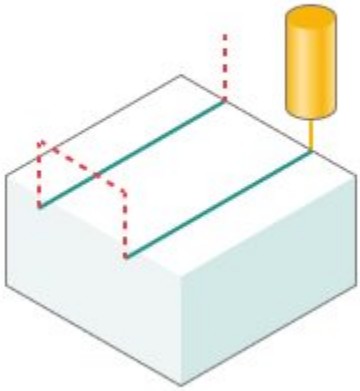
Max motion length. If enabled, link moves longer than the specified value are divided into parts no longer than it. If disabled, link moves are not divided. The parameter is shown in the Off, Deviation-based and Spatial arcs modes. In the Same as work passes mode, the Max motion length of work passes applies; in the fixed length modes, moves are divided by the Segment length.

Check workpiece.

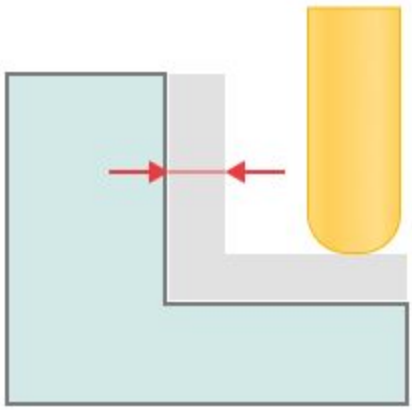
Group of parameters, that define the way of workpiece calculation for the current operation.

▼ Click here to expand..

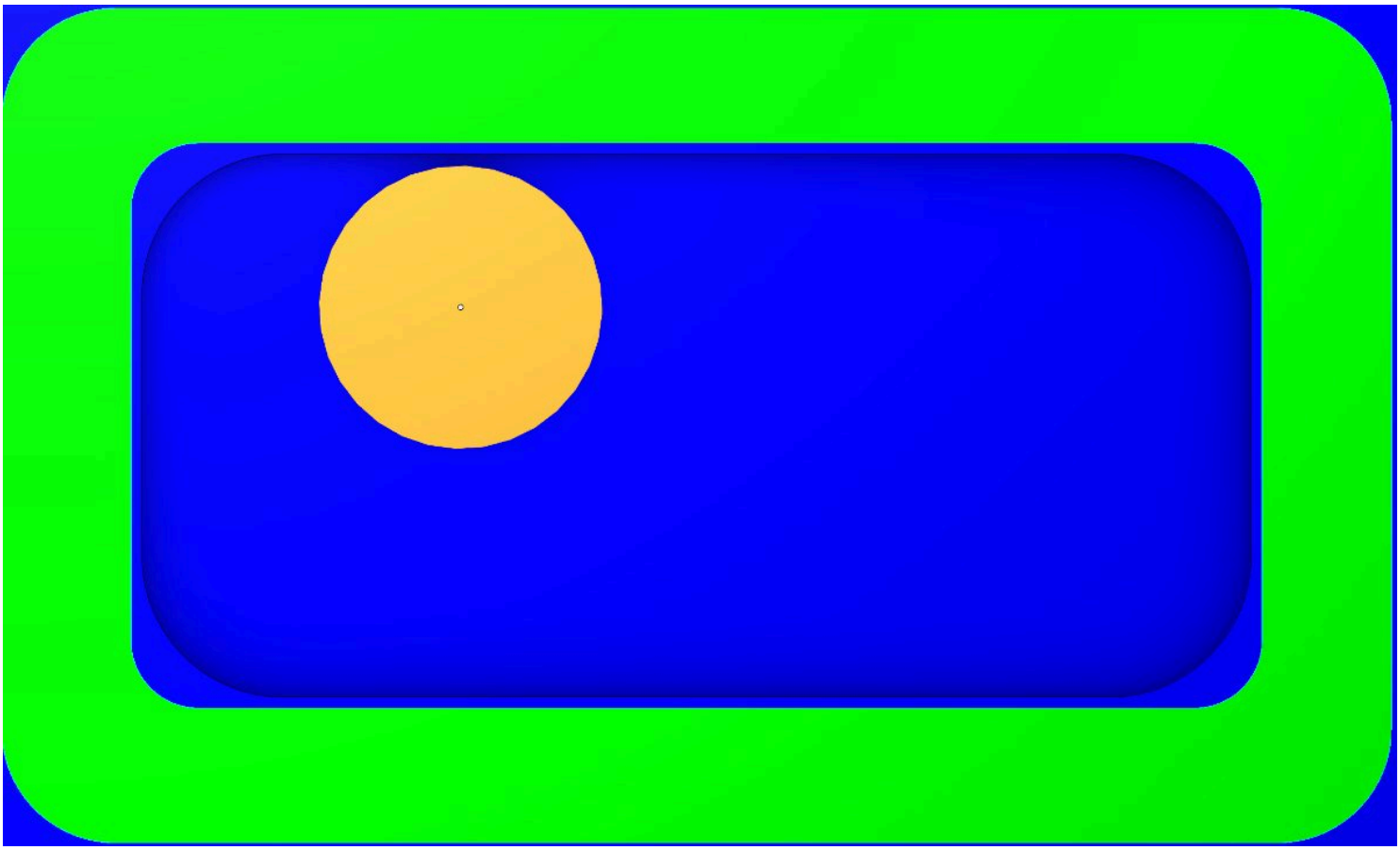




Radial ignore thickness. Unlike the stock in the Check part group, the algorithm relies on analyzing the workpiece after the previous operation: the toolpath is not generated in areas where the radial stock is less than the specified value. The parameter value must be less than the Depth step and Safe distance.

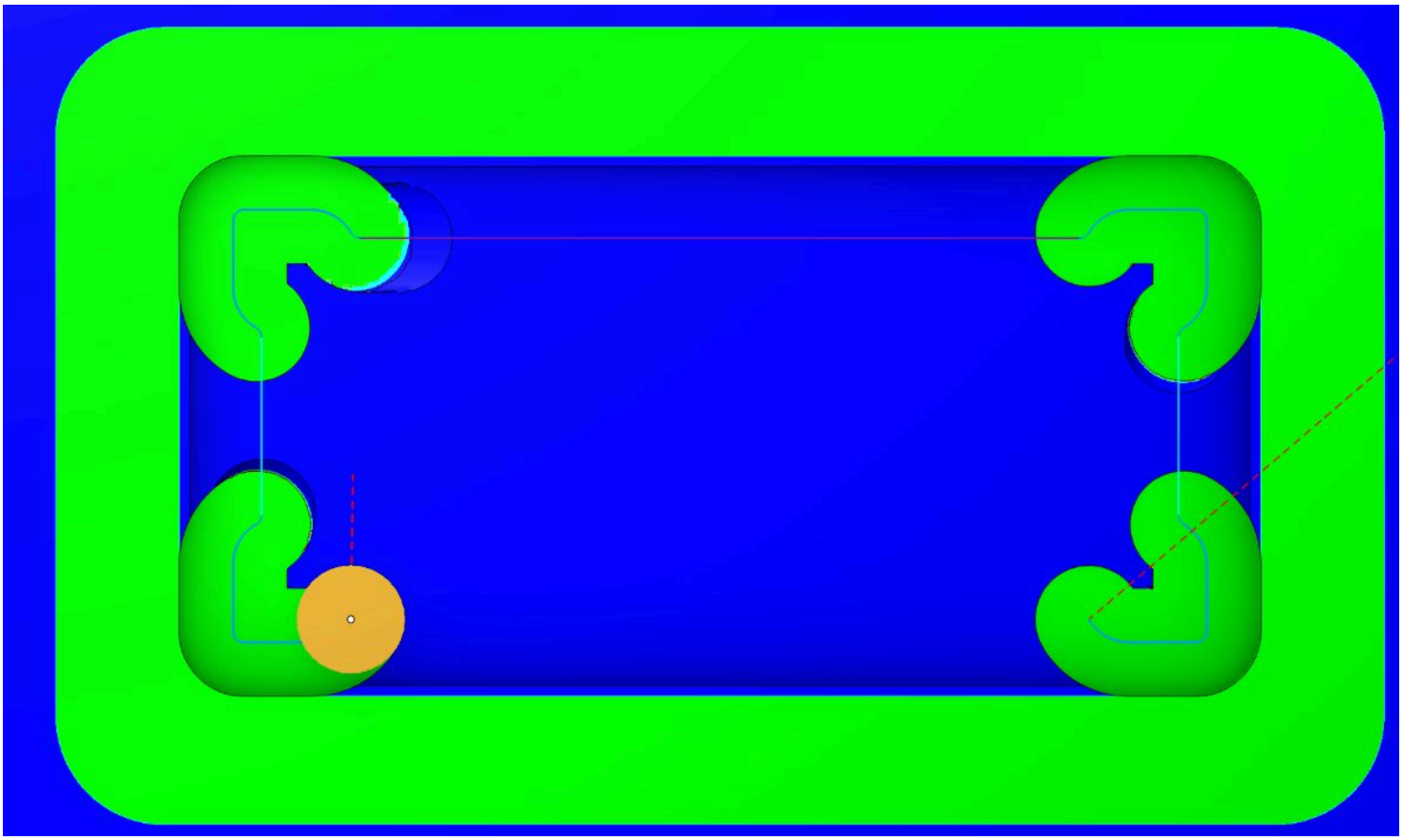


If, during the first pocket machining operation, the Radial stock and Axial stock values are set to 1 mm and the tool diameter exceeds the pocket's fillet radii, the stock allowance will be uneven. On straight wall sections, it will be 1 мм, whereas in corner areas it will exceed 1 mm.

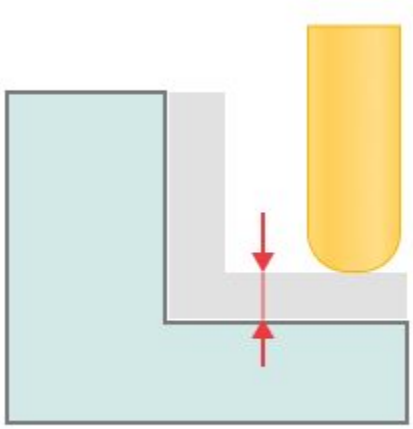


In the second machining operation, you can set the Radial ignore thickness and Axial ignore thickness values to 1,1 mm. When using a tool with a diameter smaller than the pocket's fillet radii, machining will

be performed exclusively in the corner areas of the part — specifically where the stock allowance exceeds 1 mm.



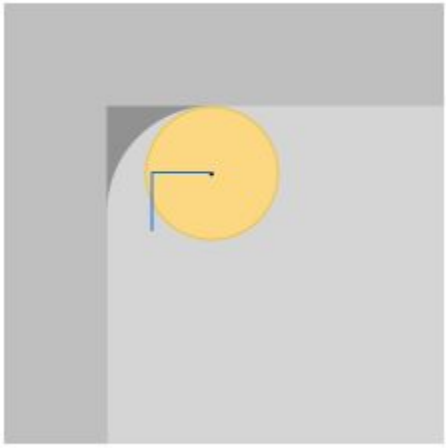
Axial ignore thickness. Unlike the stock in the Check part group, the algorithm relies on analyzing the workpiece after the previous operation: the toolpath is not generated in areas where the axial stock is less than the specified value. The parameter value must be less than the Depth step and Safe distance.

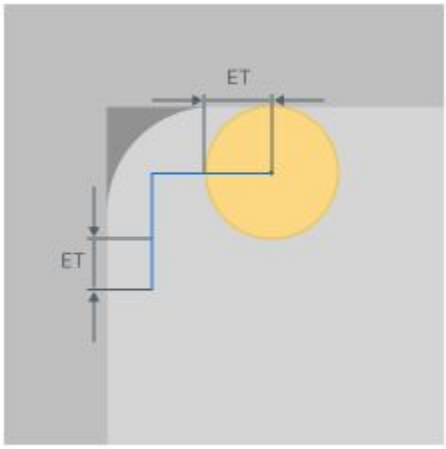


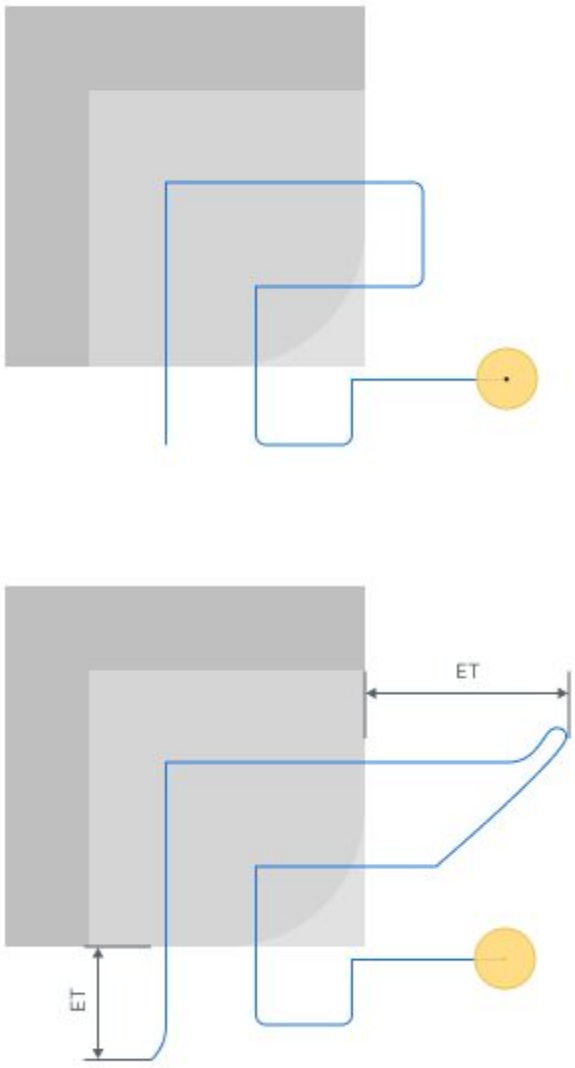
Extend toolpath. Extend the toolpath so that cutting starts before the workpiece edge. This is primarily used in finishing operations: Finishing waterline and Finishing plane, when the Check workpiece option is enabled and the toolpath is being constrained (a ccording to the workpiece). This option allows extending the approach to the entery point.

The parameter is similar to the Safe distance setting on the Link/Leads tab. Details

The difference is that the extended approach for Safe distance follows a tangent from the cutting point, while Extend toolpath contours around the workpiece.



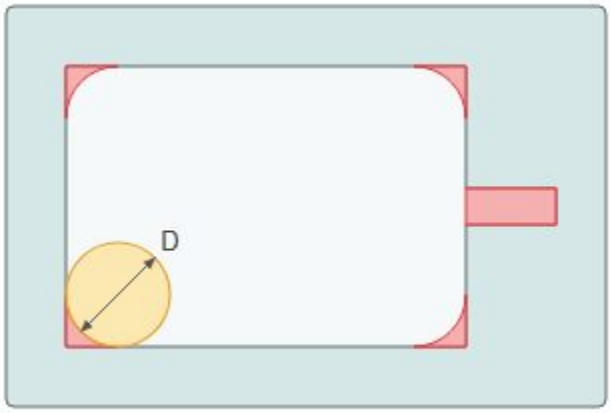




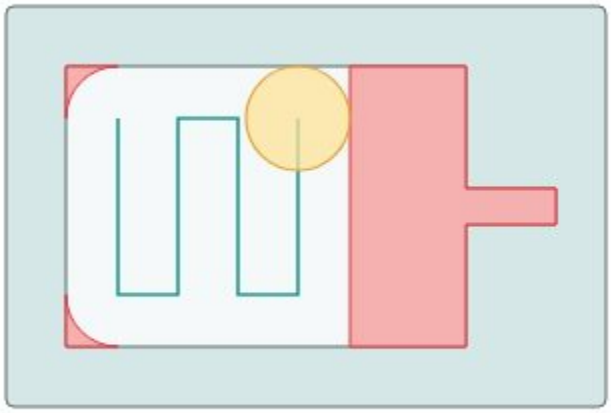
ET - Extend toolpath

Theoretical rest material. Defines how the workpiece for the current operation is calculated: from the zones inaccessible to a tool or from the simulated toolpaths of previous operations.

▼ Click here to expand..
ON. The workpiece is calculated as theoretical rest material. Only zones that the tool could not reach, such as inner corners and slots narrower than the tool, are included. The real toolpaths of previous operations are not taken into account; only the tool is used.

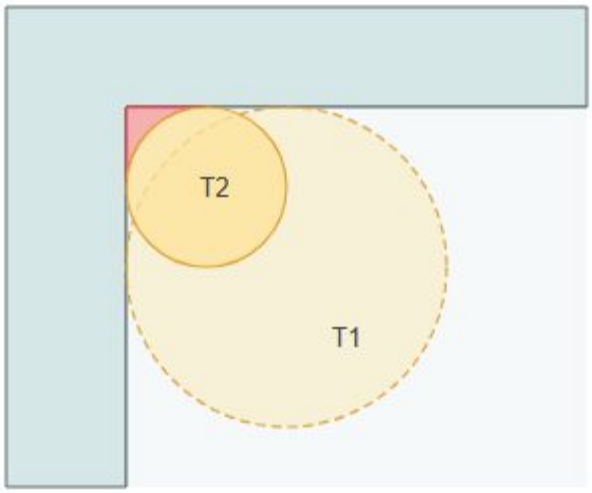


OFF. The workpiece is calculated by simulating all previous operations. The material left by their real toolpaths is used as the workpiece for the current operation.

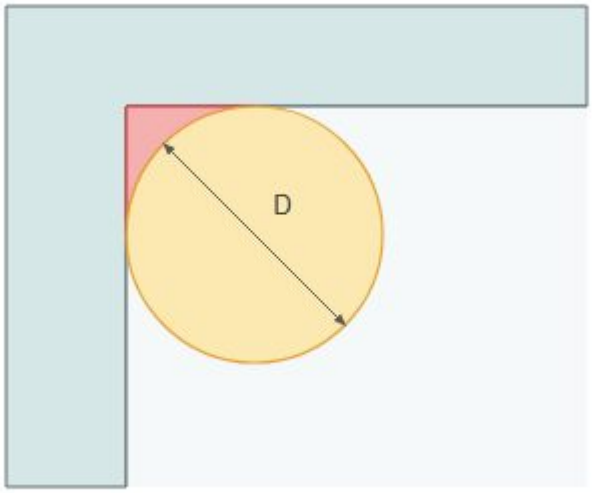


After tool diameter.

OFF. The rest material is calculated after the tools used in all previous operations.



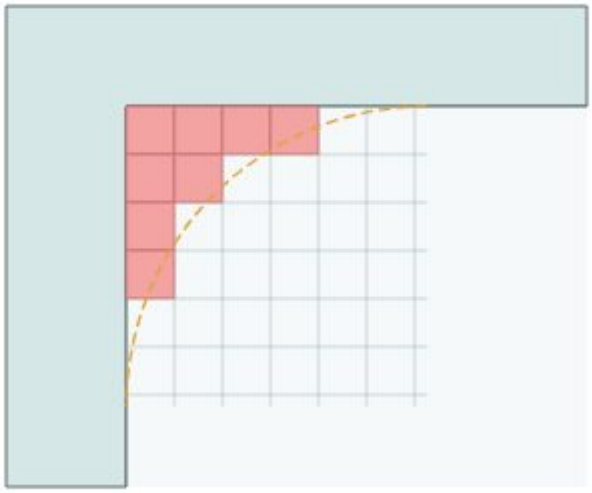
ON. The rest material is calculated after a tool with the specified diameter D.



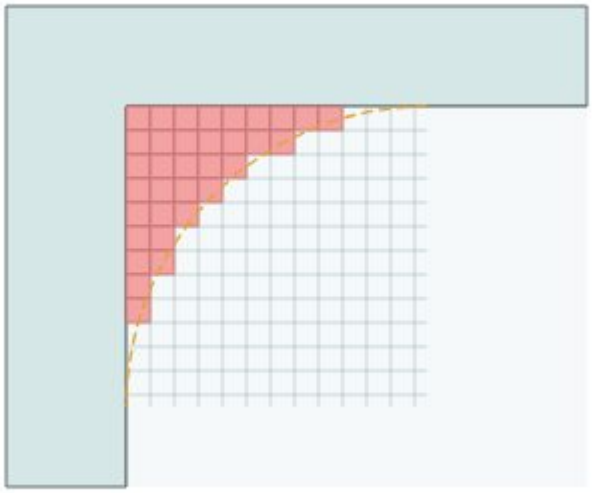
T1, T2 - T1 — previous tool (larger diameter); T2 — current tool

Model resolution. Defines the grid step used to calculate the rest material. A higher resolution finds the rest material more accurately in corners, but increases calculation time and memory usage.

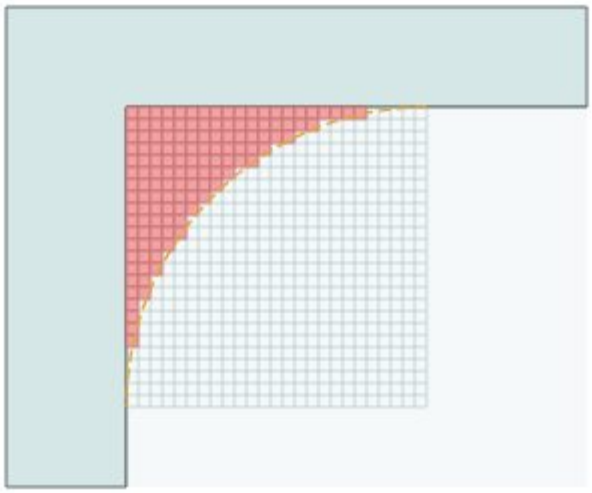
Standard. Uses the basic grid step of the rest material model.



High. Uses a grid step two times smaller than Standard.



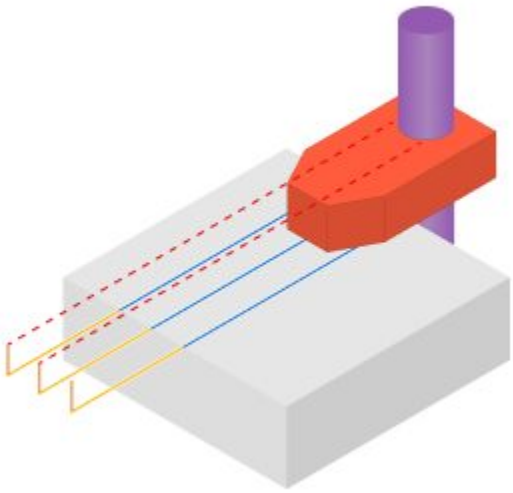
Ultra High. Uses a grid step four times smaller than Standard.

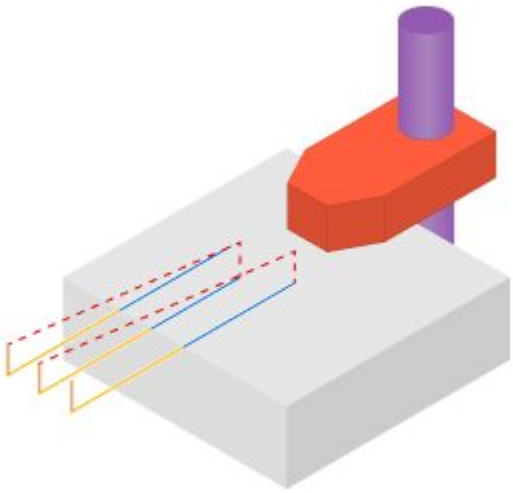


Check fixtures.

When generating the toolpath, fixtures are treated as no-go zones to prevent collisions.

▼ Click here to expand..





Weaving.

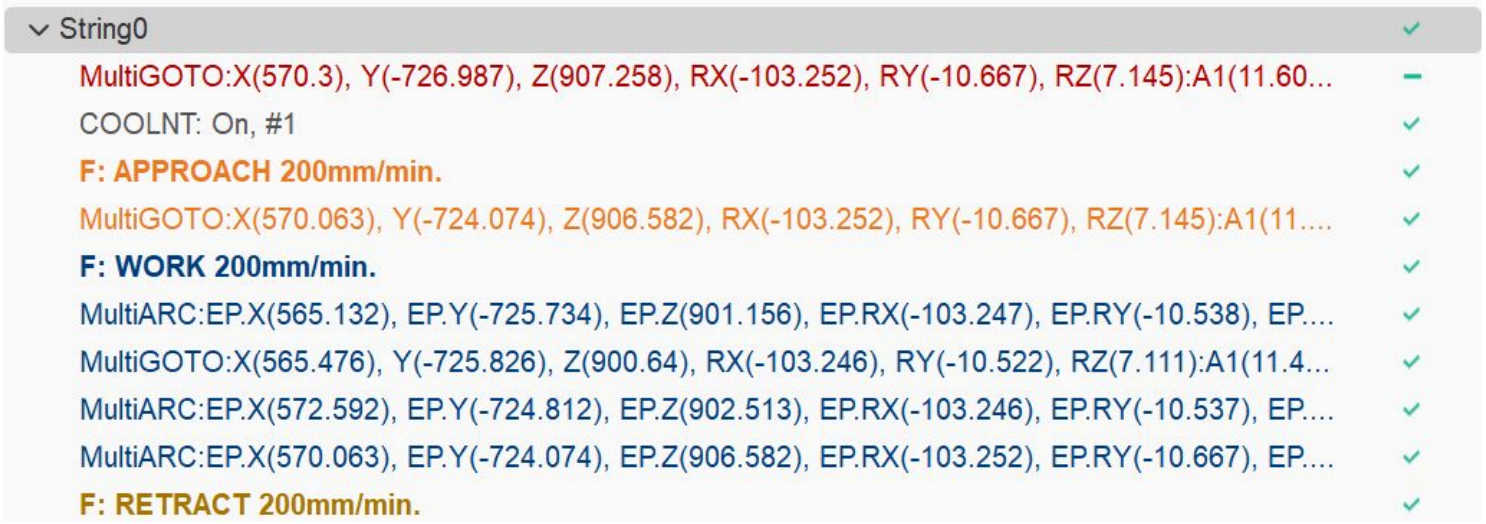
Mechanical weaving is used, for example, to compensate for tolerances or to bridge gaps in a seam. The torch moves across the seam in this instance and the weave oscillation is thus superposed on the seam motion. It is also possible to rotate the torch in relation to the plane of the weld (direction of welding). Can also be used for polishing. Details

Features.

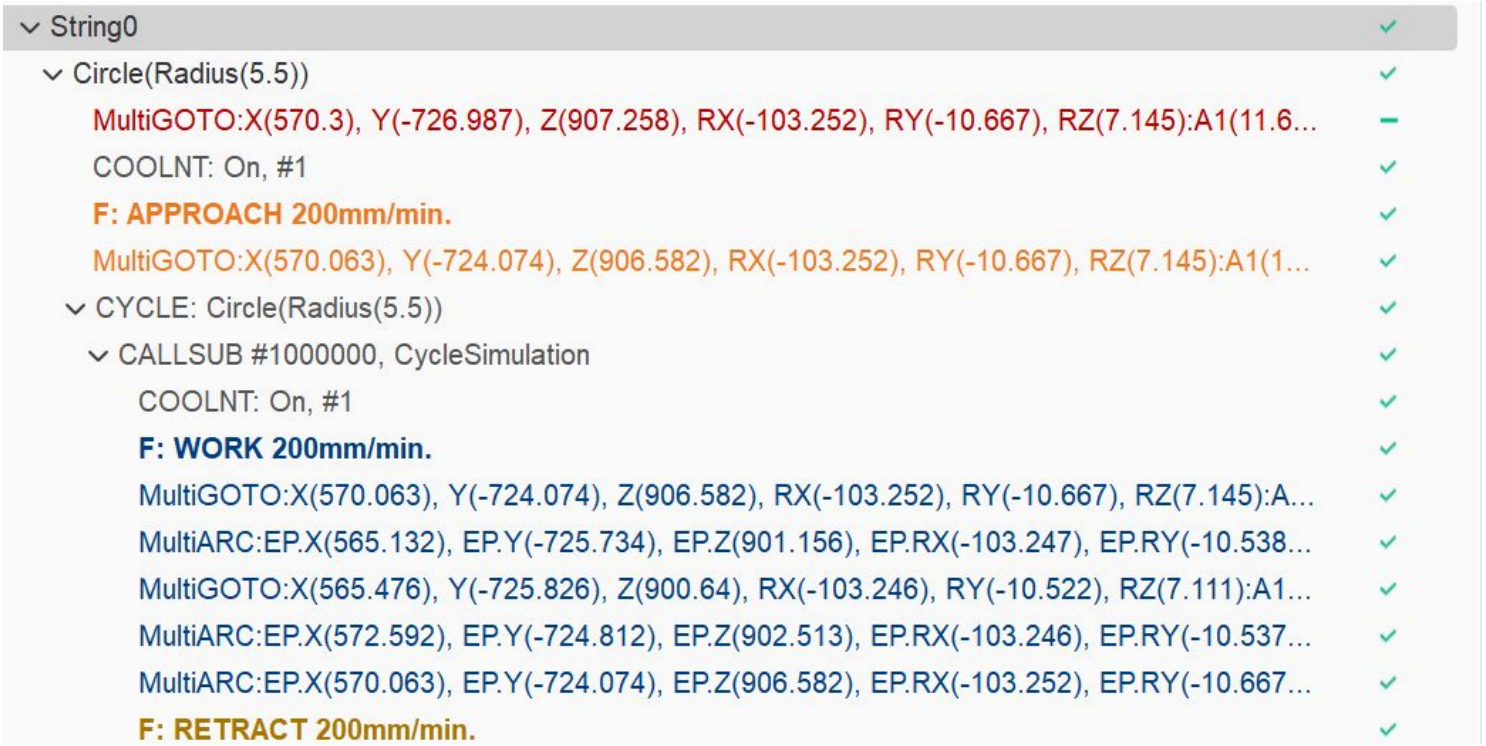
The trajectory output method is used for Recognize features. Details

▼ Click here to expand..
Cycle format.

None. Only simple NC commands (Without indicating cycles or recognized features).



Without internal movements (EXTCYCLE only). Canned cycles are used, with identification of recognized machining features



With internal movements. Only simple NC commands with indication of recognized features.



Start and end at center. The toolpath starts and ends at the center of the feature. When the option is disabled, cutting starts from the wall edge.





Check Holder.

The "Check Holder" feature detects segments of the toolpath where the tool holder collides with the part and modifies those segments according with the specified strategy. Details

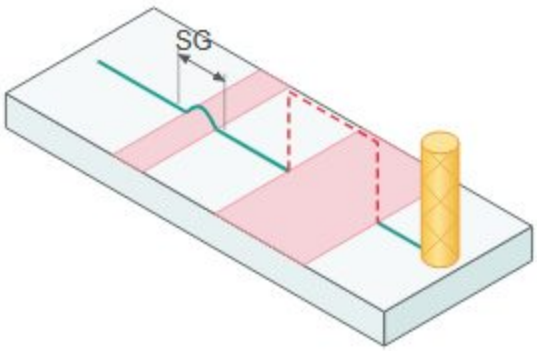
Plunge roughing.

Makes a plunge passes based on the current trajectory. Details

Advanced.

▼ Click here to expand..

Small gap size. After the undercut check a pass can break into pieces. If the gap between the pieces is less than the specified Small gap size, they are joined with a smooth curve and the tool does not retract. Larger gaps are linked with a retract.



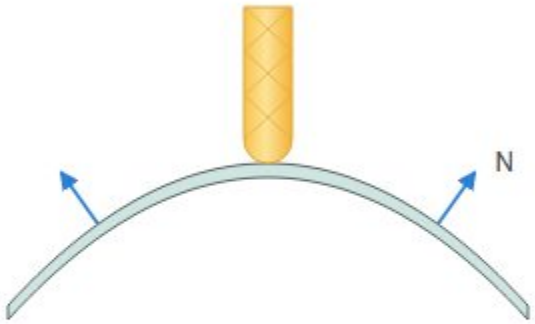
SG - Small gap size

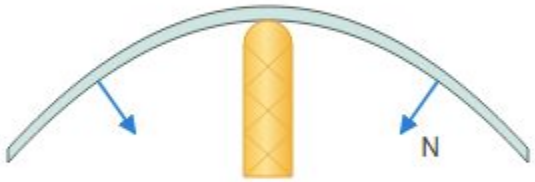
Small gap size: 0 joins no gaps. By default, the value is half of the tool diameter.

Reverse side. Controls the side from which the surface is machined by reversing its normal.

Reverse side, disabled. The surface is machined from the side its normal N points to.

Reverse side, enabled. The surface normal N is reversed, and the surface is machined from the other side.





Simulation.

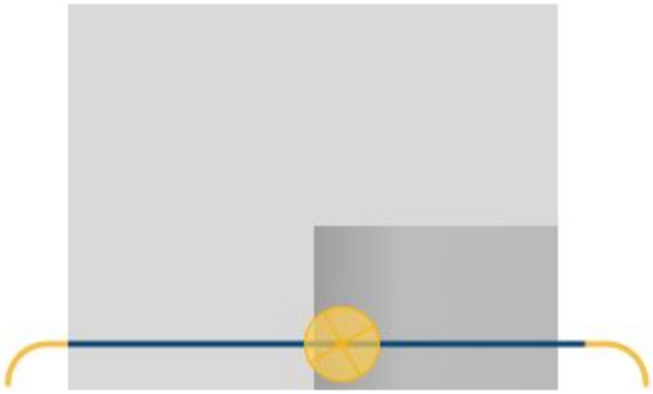
Group of parameters to define the simulation options for the current operation.

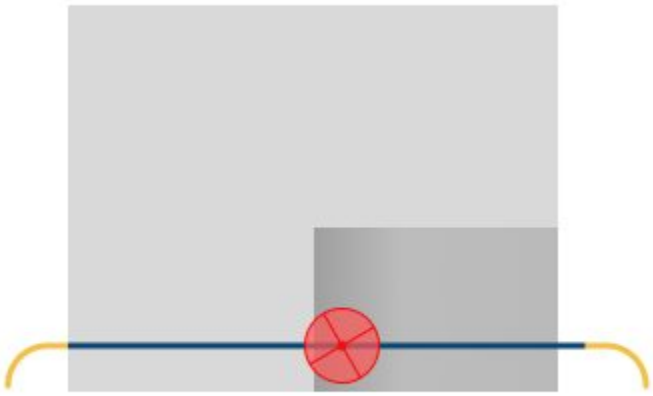
▼ Click here to expand..

Check for gouges.

If enabled: If the toolpath of this operation gouges the part, the gouging motions will be highlighted in red.

If disabled: If the toolpath of this operation gouges the part, the gouging motions will not be highlighted.





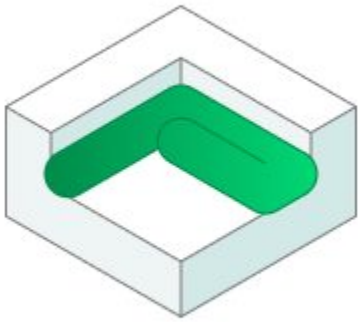
Simulation type.

Auto. The simulation type is automatically assigned based on the selected operation type.

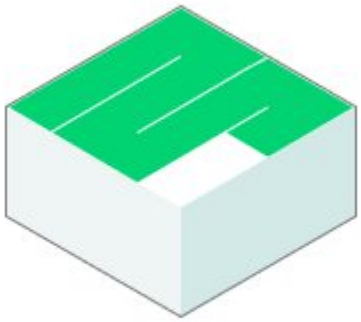
Subtractive. Modeling is carried out by subtracting material.



Additive. Modeling is done by adding material to the tool level.

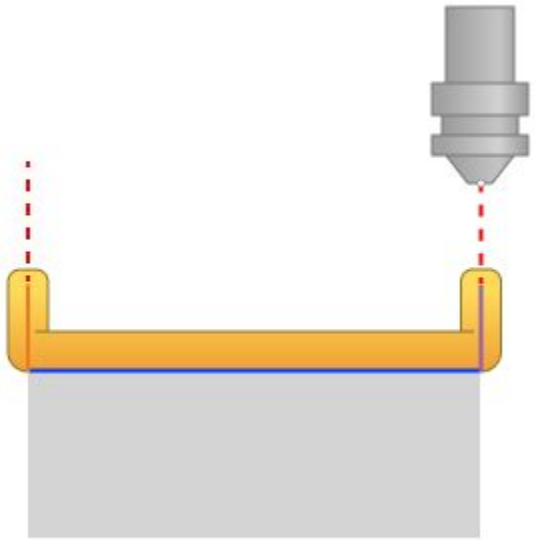


Painting. Modeling is done by adding material in layers. Details

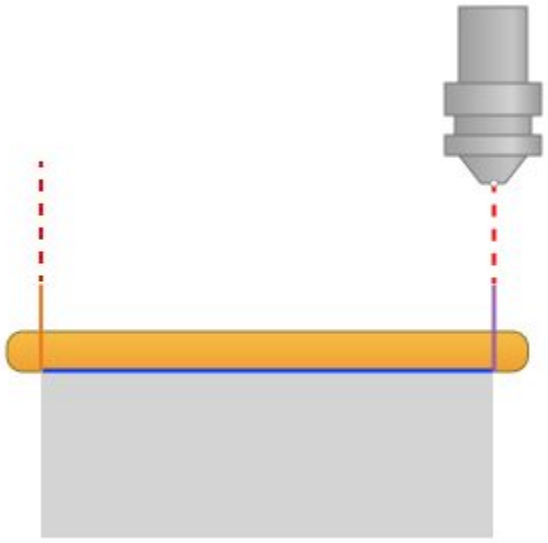


Effective feeds. Allows modeling the welding deposition process according to feed rules.

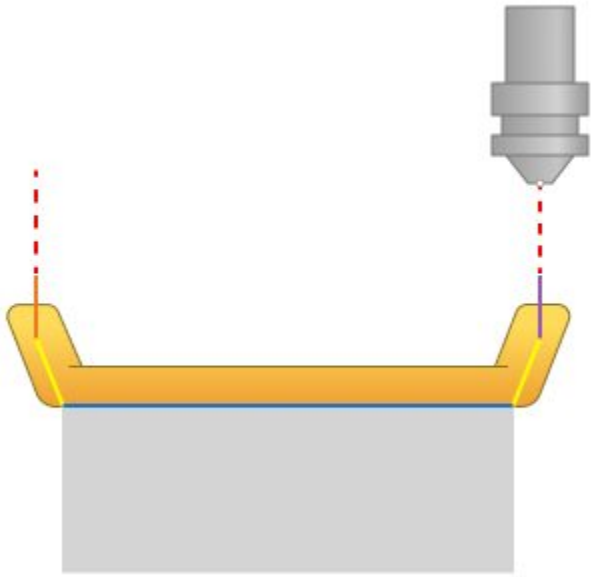
All non rapid. The material is added, if current feed type is NOT 'Rapid'.



Working feed only. The material is added if the current feed type is 'Working', 'Finish pass' or 'First pass'.



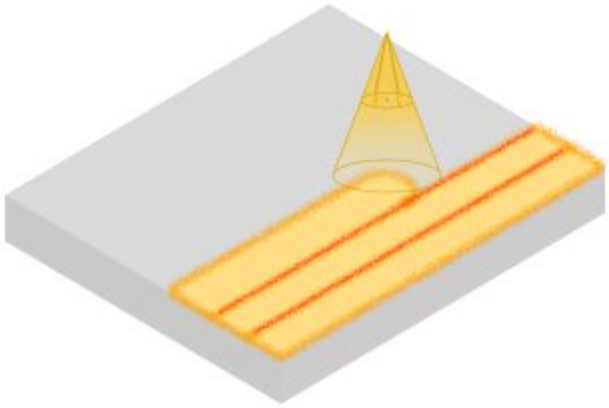
Working feed, retracts and engages. The material is added if the current feed type is 'Working', 'Retract' or 'Engage'.



Custom. The material has been added to the chosen feed. (Work feed, Engage feed, Retract feed, Short link feed, Long link feed, Finish feed, First feed, Approach feed, Return feed, Plunge feed.)

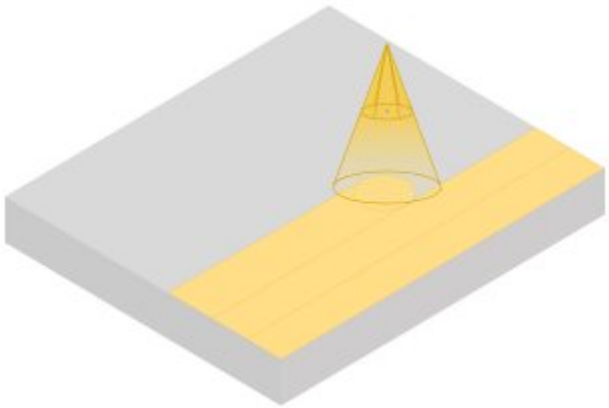
Painting type. Specifies the type of paint application modeling. Details

Thickness measurement. This type of painting simulation shows not only the areas to be painted, but also displays the approximate thickness of the future paintwork. To view the thickness result of the applied paint, you need to activate Verify Compare in the top toolbar within the simulation.



Paint flow rate. Paint flow rate is needed to ensure consistent and high-quality application of paint.

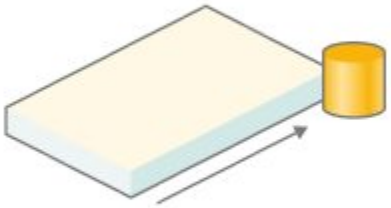
Simple. This type of painting simulation shows which areas of the model will be painted.

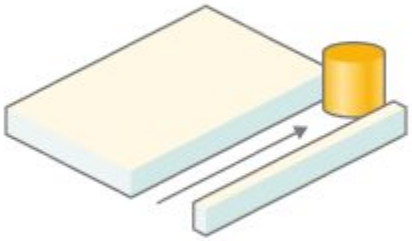


Delete chips.

The deleting chips will happen automatically at the end of the operation simulation process.

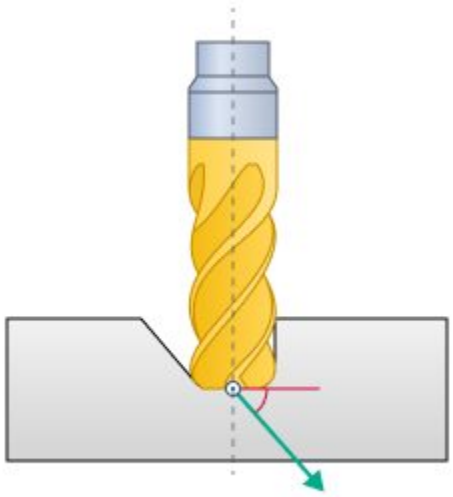
Disabled. The deleting chips is disabled.



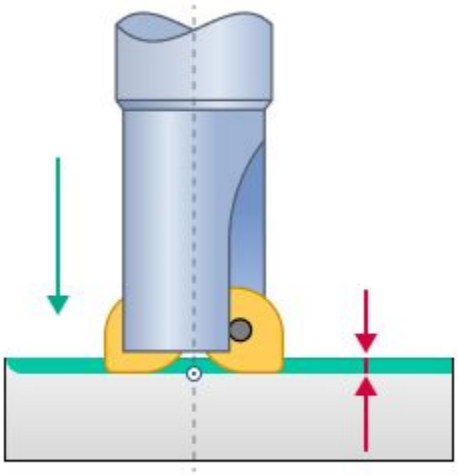


Check for plunges. Simulation checks for the incorrect tool lowering.

Allowed ramp angle. If the ramp angle of tool path is more than this value then simulation marks these lines by the error sign.



Allowed axial plunge. If the vertical lowering of the tool into the material is greater than this distance, then the tool path is marked with an error.



Tolerance.

This feature defines the 'maximum deviation' between the 'ideal' and 'approximated' toolpath. A larger tolerance amount reduces the number of moves in the toolpath.

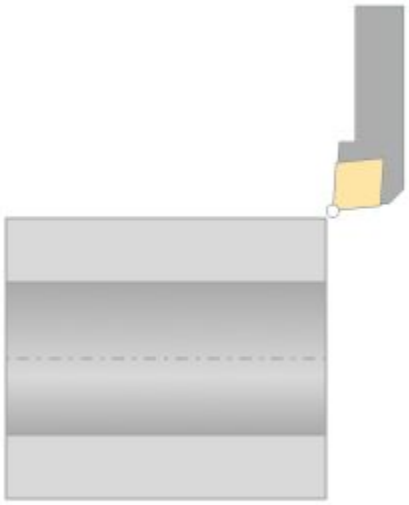
Preferred tool orientation.

In turning operations, it defines the basic position of the cutting tool relative to the workpiece rotation axis.

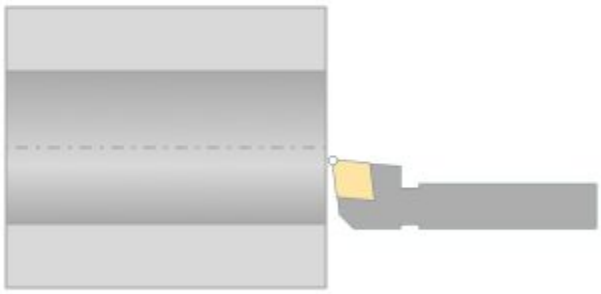
▼ Click here to expand..

Undefined. Indicates that the system imposes no rigid constraints on tool orientation relative to the workpiece rotation axis. The orientation is either user-defined or automatically determined based on the workpiece geometry and the type of operation performed.

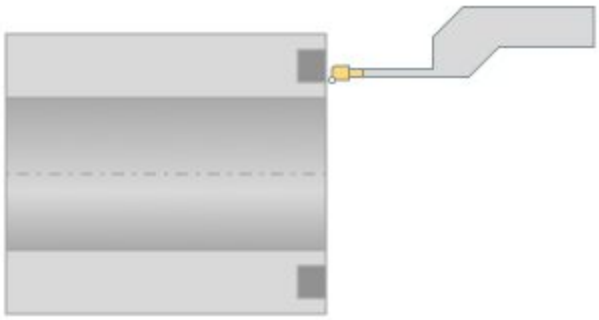
Outer radial. The tool is oriented radially from the outside — the cutter approaches the workpiece from the side, perpendicular to the rotation axis, for machining external diameters.



Inner radial. The tool is oriented radially inside — the cutter enters the bore and works on internal surfaces.



Axial. The tool is oriented along the rotation axis (Z-axis), with feed parallel to the workpiece axis.



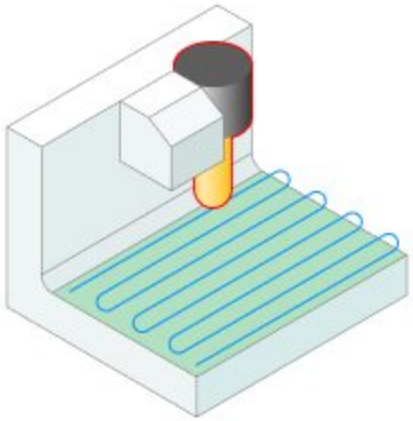
Minimum layer time.

This group of parameters allows you to set the minimum allowed time in seconds for printing a single layer. Details

Power.

This option adjusts the extruder speed based on layer height, modifying the Power (material feed) parameter. Details

Check Holder (Avoid collisions)

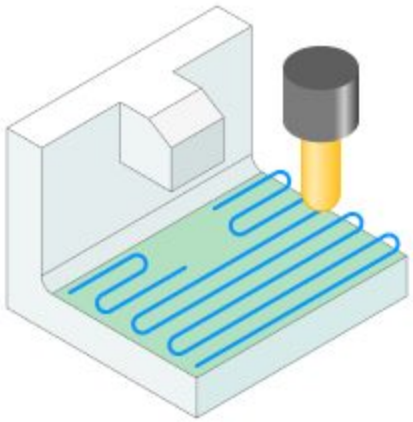


Application area:

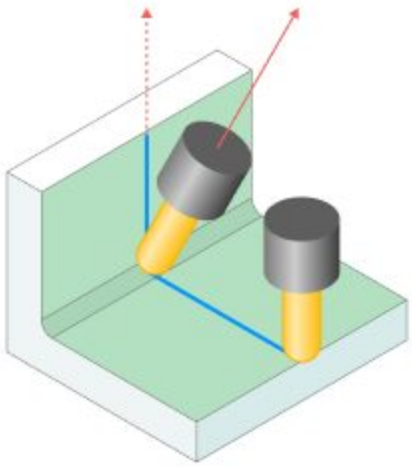
With the Check Holder function, this feature detects segments of the toolpath where the tool holder collides with the part and modifies those segments according with the specified strategy.

Methods of Avoiding Collisions:

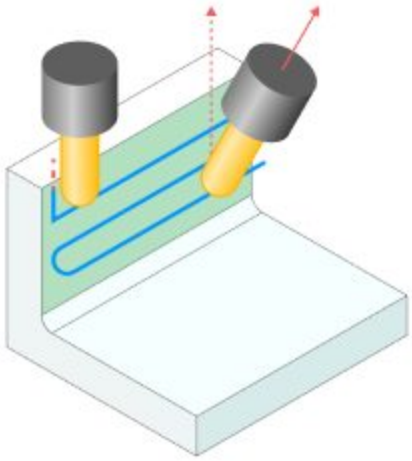
Trim Toolpath. The collided segments (red) are excluded from the toolpath.



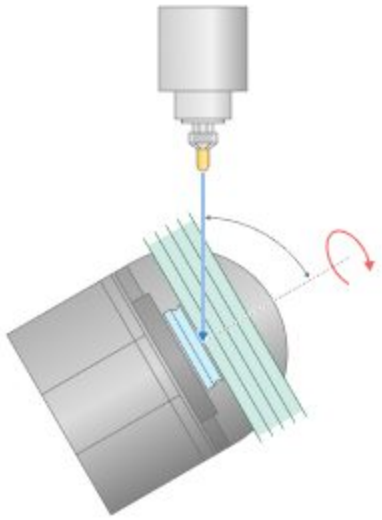
Frontal Tilting. The tool tilts away from the collision in the frontal plane (in a plane of tool pass).



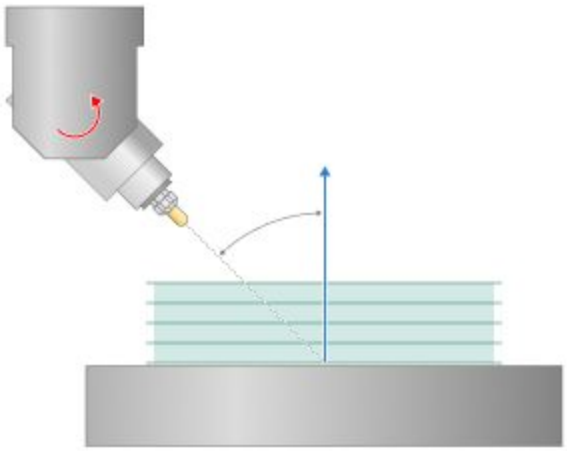
Side Tilting. The tool tilts away from the collision sideways (in a plane perpendicular to the toolpath).



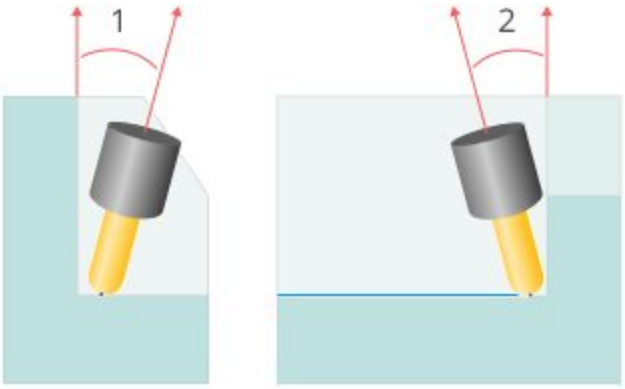
Tilt to Rotary Axis. T he tool deviates from the initial orientation by specified Tilt Angles, in the plane formed by the Rotary Axis and the tool contact point. Works only with 5-axis machines that have a rotary axis around the axis of rotation. For example, the rotary axis on table C.



Tilt Around Rotary Axis. T he tool deviates from the initial orientation by specified Tilt Angles, in the plane perpendicular to Rotary Axis. Works only with machines that do not have a rotary axis around the axis of rotation.



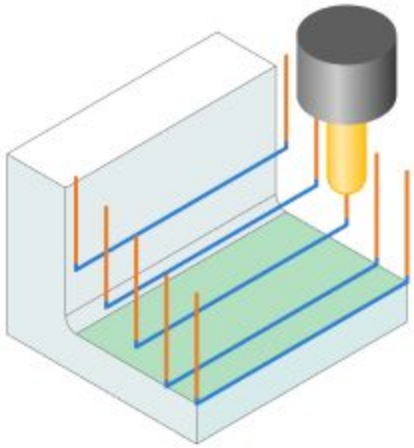
Adaptive tilting. This feature enables simultaneous angular deviation along two axes.



1 - First corner from the wall.

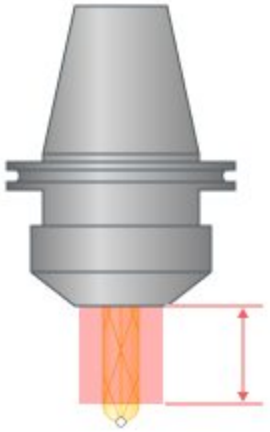
2 - Second corner from the second wall.

Offset Along Tool. Th e toolpath shifts along the tool axis within a specified distance to avoid tool holder and workpiece collision. The toolpath remains continuous in this case.

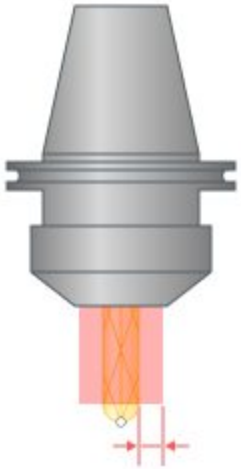


Parameters of Avoiding Collisions.

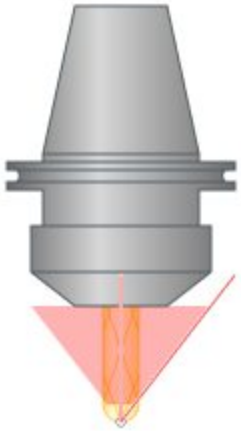
Tool Working Length. Specify the value of the tool's cutting length along its axis (Usable length LU by ISO 13399).



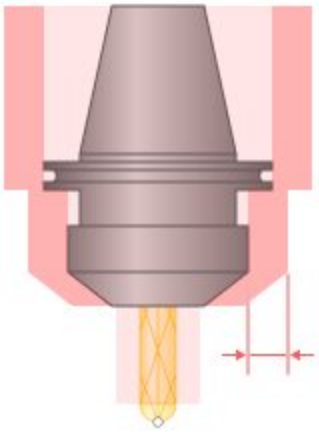
Tool Radial Clearance. Specify the minimum safe distance between the tool's non-working part and the workpiece in the direction perpendicular to the tool axis.



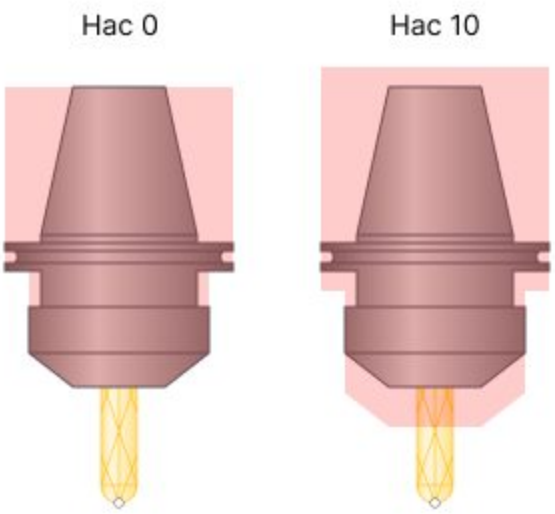
Tool Angular Clearance. The safety zone around the non-working part of the tool is a truncated cone with a radius at the apex equal to the tool radis plus Tool Radial Clearance and with an angle equal to the Tool Angular Clearance.



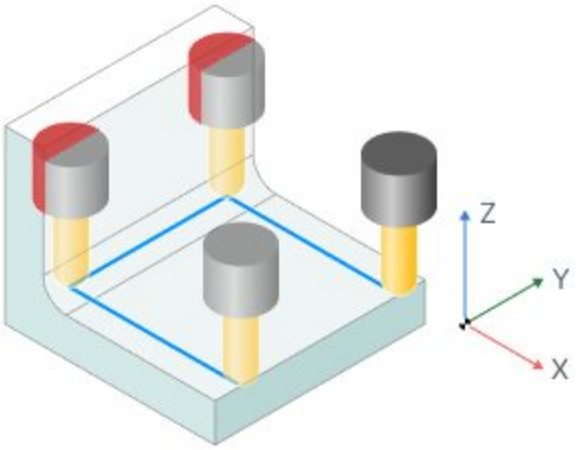
Holder Radial Clearance. Specify the minimum safe distance between the holder and the workpiece in the direction perpendicular to the tool axis.

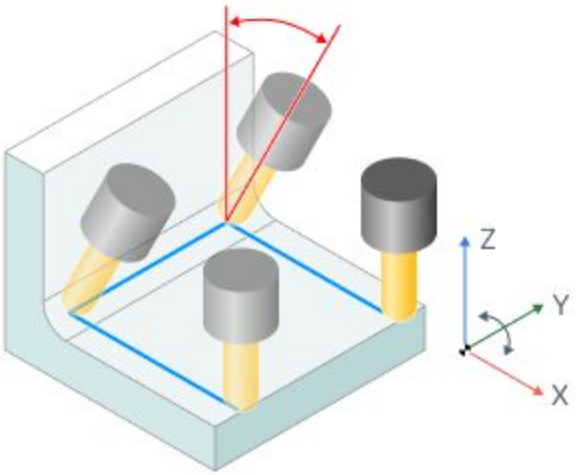


Holder Axial Clearance. Specify the minimum safe distance between the holder and the workpiece along the tool axis. Hac - Holder Axial Clearance.

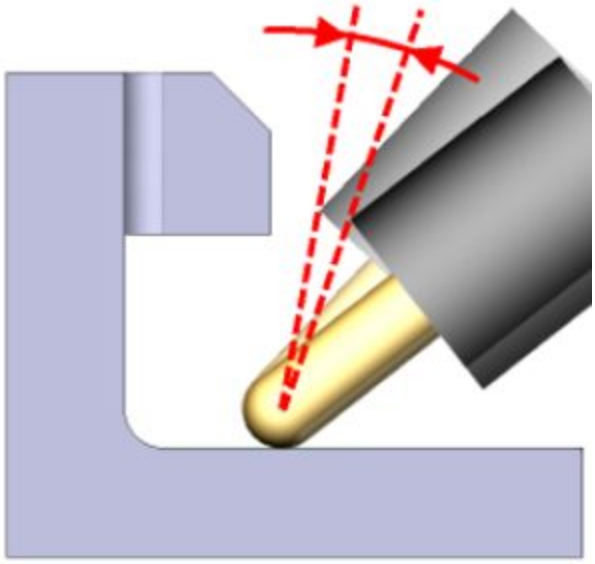


Fixed Lead Direction. Fixes the movement of the tool at the selected angle (taking into account the specified gaps in the Check Holder) when approaching an obstacle relative to the selected global coordinate system.

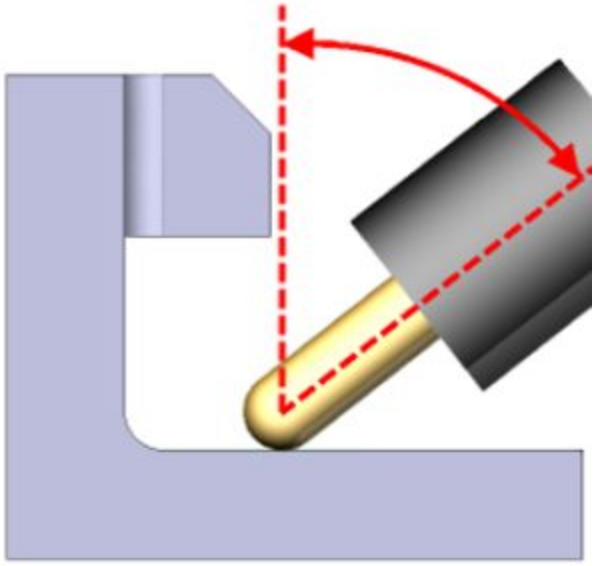




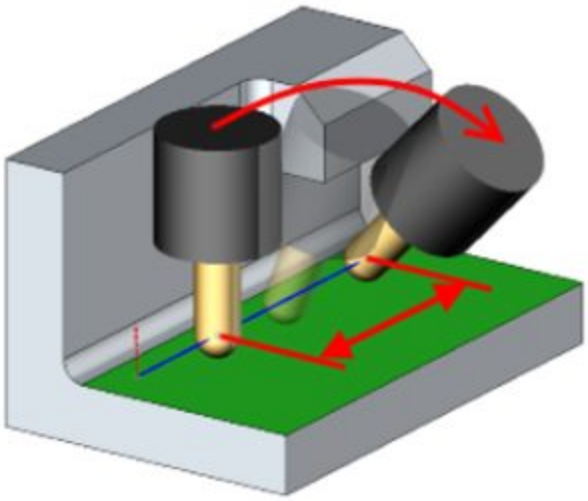
Additional Tilt Angle. Additional clearance angle in Avoid Collisions tilting strategies to guarantee a safe collision-free toolpath.



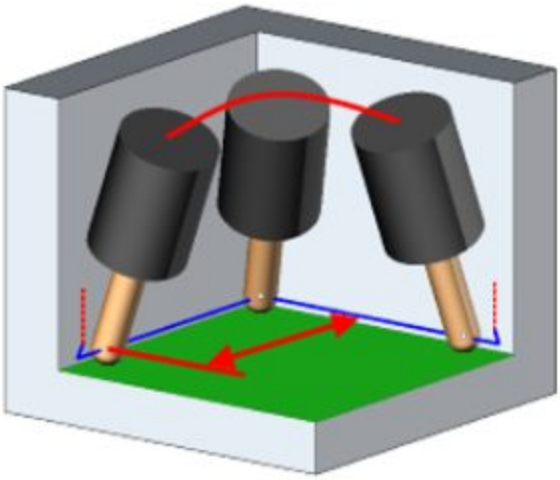
Max. tilt angle. The maximum allowed tilting angle.



Inertia. The greater the value, the smoother the tool axis tilting is. 0 means no smoothing.



Normal blend distance. The distance between two points of the toolpath used for the calculation of the current tool tilting direction. Use greater values when machining irregular shapes to eliminate unnecessary tool axis oscillations. Use values greater than the holder radius when machining inner corners.



Additional Distance. The displacement of the toolpath along the tool axis to avoid tool holder and workpiece collision.

Max. Distance. The maximal allowed Additional Distance.

Check Spindle. Parameters controlling collision avoidance between the workpiece and the machine's spindle unit.

Auto. This mode uses 3D models of the machine's scheme to perform collision calculations.

Manual. In this mode, the system monitors collisions between the workpiece and a safety zone comprising a truncated cone and a cylinder. The top Radius, the Angle, and the Max Radius define the cone. The Max Radius define the cylinder.

Spindle Radial Clearance. Specify the minimum safe distance between the spindle node and the workpiece in the direction perpendicular to the tool axis.

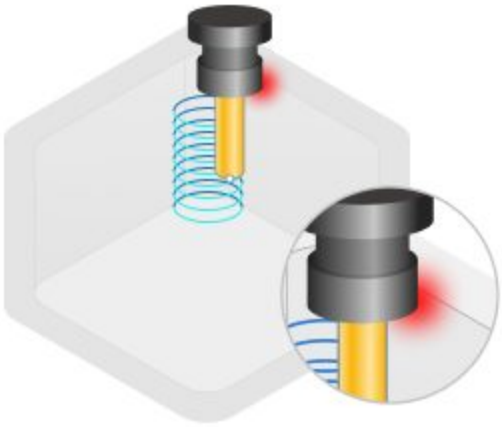
Spindle Axial Clearance. Specify the minimum safe distance between the spindle node and the workpiece along the tool axis.

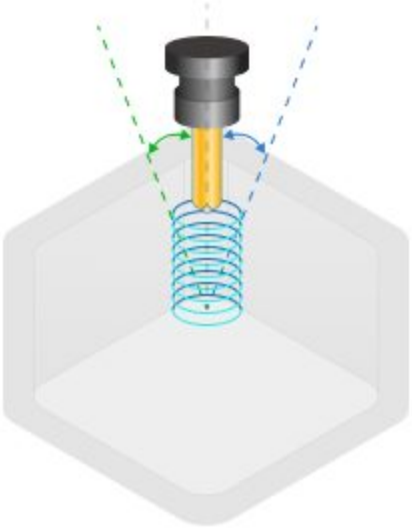
Max Radius. In the Auto mode it is the radius of a cylinder encompassing spindle-adjacent components (such as the spindle head) that might collide with the workpiece during machining. In the Manual mode it is the maximal radius of the safe zone.

Radius. It is the envelope radius of the spindle node.

Angle. The safety zone around the spindle node is a truncated cone with a radius at the apex equal to the Radius plus Spindle Radial Clearance and with an angle equal to the Angle.

Check links. Function that enables controlled angular deviation of the tool in two directions during transitions between machining passes to prevent collisions with the workpiece.



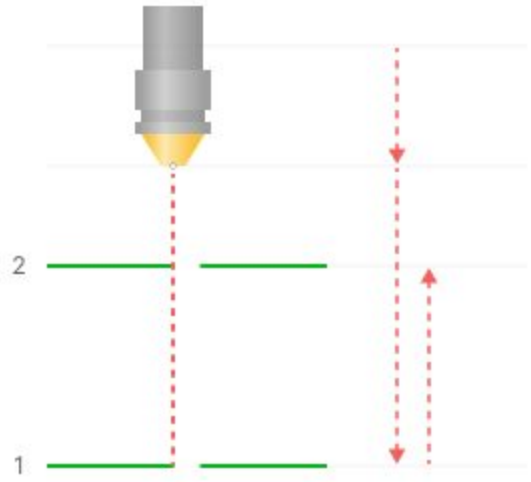


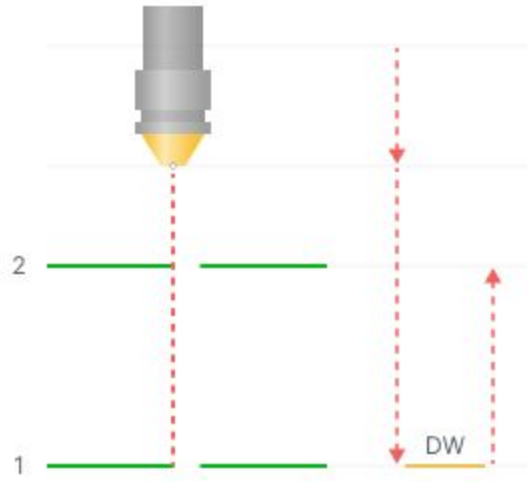
Layer time

This page describes parameters that control how long a single layer is being printed in additive operations. Use "Minimum layer time" to make sure a fast layer has enough time to cool down before the next one, and "Maximum layer time" to avoid an overly long layer that would dry out before the next one starts (cold joint). Both groups can be used independently or together.

> Note
>
> You can find these parameters on a "Parameters" page in our "Additive" operations, inside the "Layer time" group.

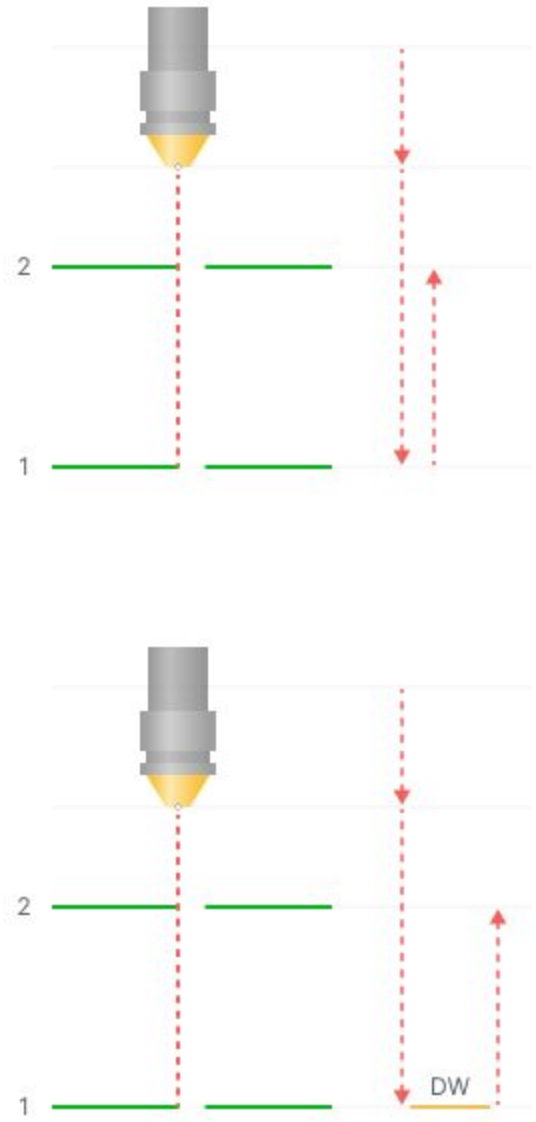
Minimum layer time





Description:

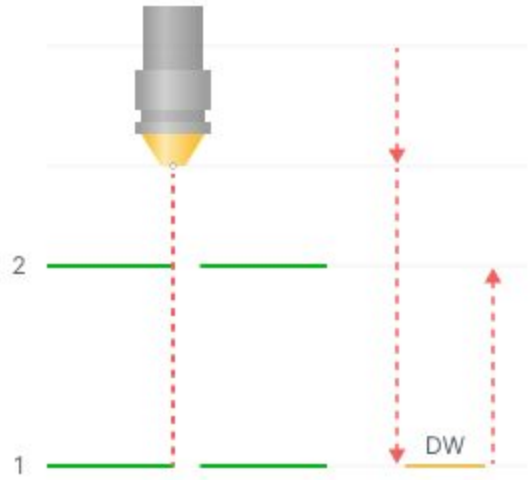
This group of parameters allows you to set the minimum allowed time in seconds for printing a single layer. If the layer is printed faster, the operation will be paused until the specified time is reached. Also it's available to use additional parameters such as "Lift head" and "Minimum feed".

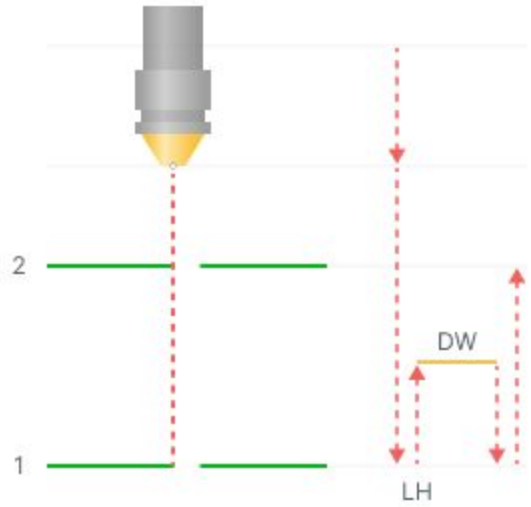


DW - Dwell

Lift head:

This parameter allows you to additionally retract the extruder during the delay to the specified distance. The retract is performed strictly according to the normal to the last printing point.



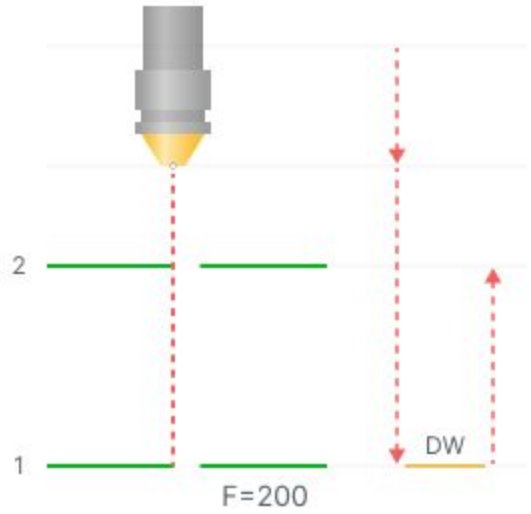


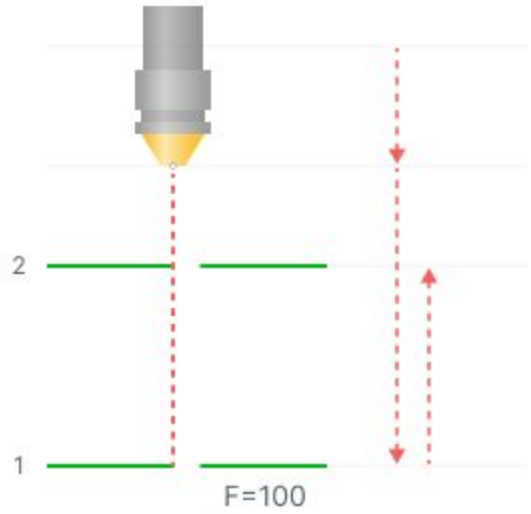
DW - Dwell

LH - Lift head

Minimum feed:

This parameter allows you to reduce the feed rate in order to increase the layer time and avoid inserting the "delay" command. The "Minimum feed" value defines the lowest limit for the feed rate. If all feeds have been reduced to this minimum but the layer time is still less than the minimum layer time, the "delay" command will be inserted.



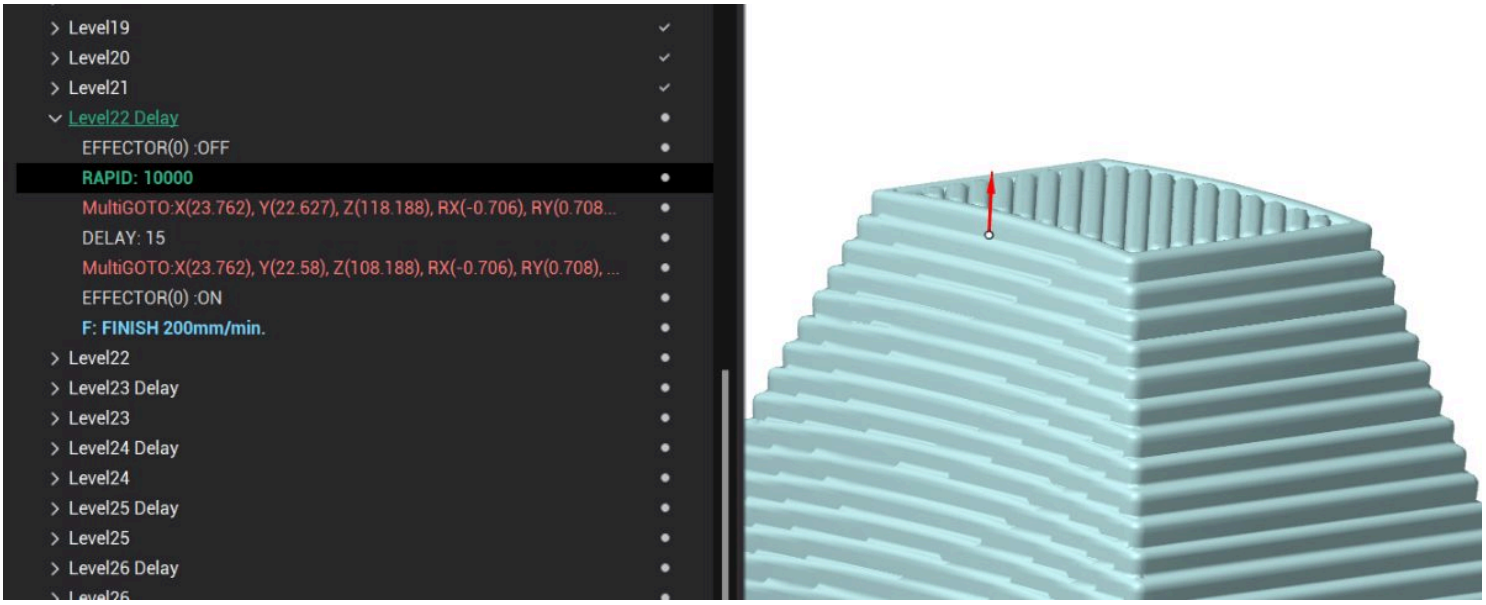


> Example
>
> Let's say our minimum layer time is 100 seconds and actual layer time is 75 seconds with a feed rate of 180 mm/min (3 mm/sec). If this parameter is turned off, a 25-second delay will be inserted. However, if this parameter is set to 100 mm/min, the feed rate will be reduced to 135 mm/min (2.25 mm/sec). As a result, the layer time will become 100 seconds, and no delay will be inserted.

> Note
>
> The system will not reduce "Rapid" feed rate. Additionally, it will not reduce "Approach from safe surface feed", "Return to safe surface feed" and "Transition on safe feed" if they are set as "Rapid".

Result:

The result can be seen on the "Simulation" page. Delay command with additional commands will be added to a separate group in CLDATA and this group will be inserted before printing the next layer.



Inside this group with the added name "... Delay" we can see "Effector off" command, retract command, then DELAY command, approach command and "Effector on" command.

The DELAY command will be inserted with a time value in seconds, which equals the difference between the minimum layer time and the current layer time. So, the delay time is the time we need to wait before the minimum layer time is reached.

> Note
>
> If the "Lift head" parameter is off, then retract and approach commands will not be inserted.
>
> If the "Minimum feed" parameter is used, you will see reduced values of the feed rate in this tree, and in some cases delay will not be necessary.

Maximum layer time

Description:

This group of parameters allows you to set the maximum allowed time in seconds for printing a single layer. If the layer takes longer, the working feed is scaled up to the specified "Maximum feed" to fit within the limit. If even the "Maximum feed" is not enough, the operation continues and a warning is added to the affected layer in the CLData tree.

Maximum feed:

This parameter defines the upper limit of the feed rate used to fit the layer within the maximum allowed time. Working feeds in the layer are scaled up but never exceed this value. Unlike "Minimum feed", this

parameter has no on/off switch — it is always active when "Maximum layer time" is enabled.

> Example
>
> Let's say our maximum layer time is 10 seconds and actual layer time is 20 seconds with a feed rate of 180 mm/min (3 mm/sec). If we set "Maximum feed" to 360 mm/min (6 mm/sec), all working feeds in the layer will be scaled up by a factor of 2, so the layer time becomes 10 seconds and fits within the limit.

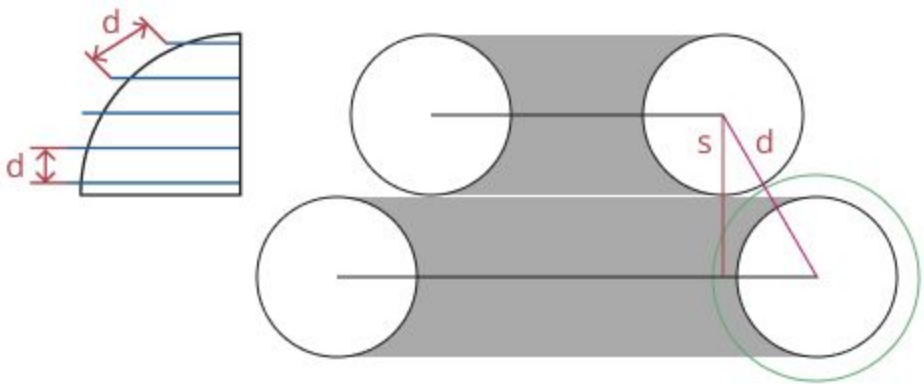
> Note
>
> The system will not increase "Rapid" feed rate. Additionally, it will not increase "Approach from safe surface feed", "Return to safe surface feed" and "Transition on safe feed" if they are set as "Rapid".

Result:

The result can be seen on the "Simulation" page. In the CLDATA tree the working feed for the affected layer will be replaced with a higher percentage value, calculated to keep the layer within the maximum allowed time.

If even the "Maximum feed" is not enough to fit within the limit, an additional item will appear in the tree near the affected layer, showing the actual elapsed time and the limit — for example, MaxTimeExceeded: 117>100sec means the layer interval reached 117 seconds while the configured limit was 100 seconds. The operation log will also show a warning. Printing is not stopped — the layer is still printed at the maximum possible feed.

Power



Green circle — real extruded material, black circle — theoretical extruded material.

S - Horizontal step

d - Step on the plane

Application area:

This option adjusts the extruder speed based on layer height, modifying the Power (material feed) parameter.Using the link, you can learn how to work with the POWER command in the postprocessor. Details

This is calculated as follows:

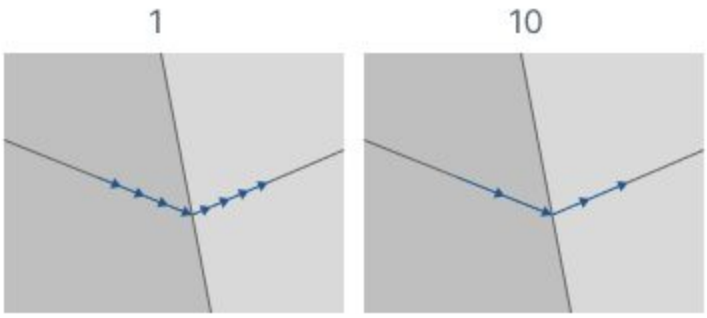
Power = d/S*100%

Thus, the Power function allows increasing the material feed depending on the varying step. That is, the larger the step, the more material needs to be extruded.

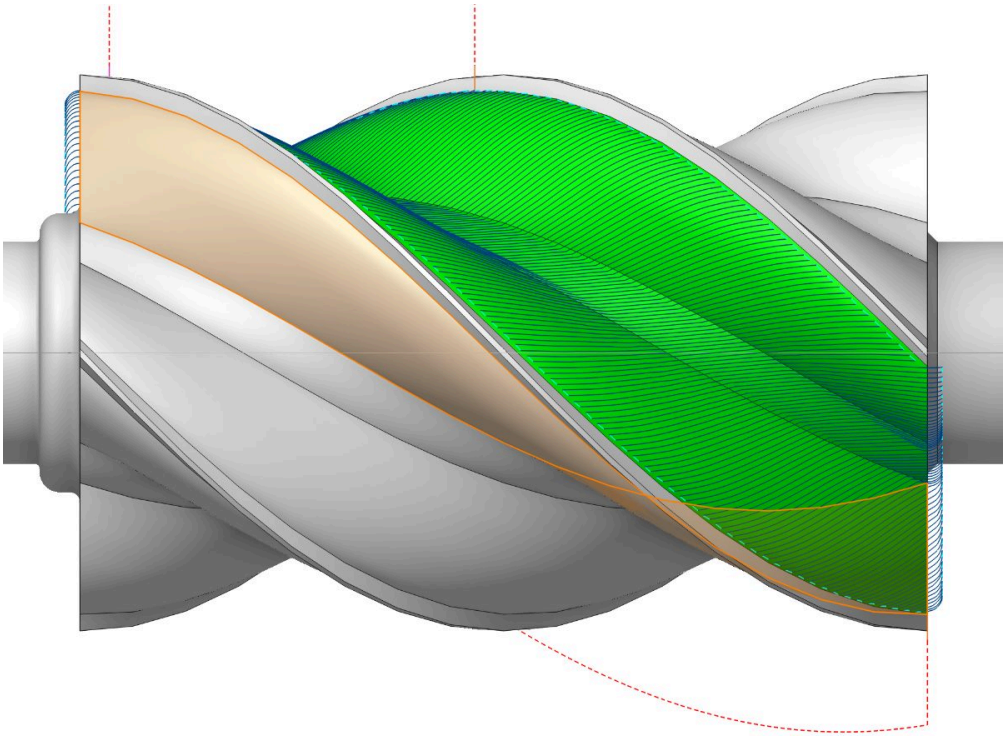
In CLDATA, you will see it as follows:

```
EFFECTOR(0) :OFF
F: SHORT LINK 200mm/min.
X0, Y108.874, Z18
POWER(0): 112.25%
EFFECTOR(0) :ON
F: FINISH 200mm/min.
X0, Y108, Z18
POWER(0): 116.67%
X55, Y108, Z18
POWER(0): 130.17%
X56, Y108, Z18
POWER(0): 111.8%
X56, Y109, Z18
```

Max dist deviation. Interpolation parameter.



Use new calculation method for Rotary roughing/ finishing



Application area:

To resolve issues with unsatisfactory toolpath quality in the normal direction relative to the surface, the calculation algorithm for the rotary finishing operation has been rewritten. Details

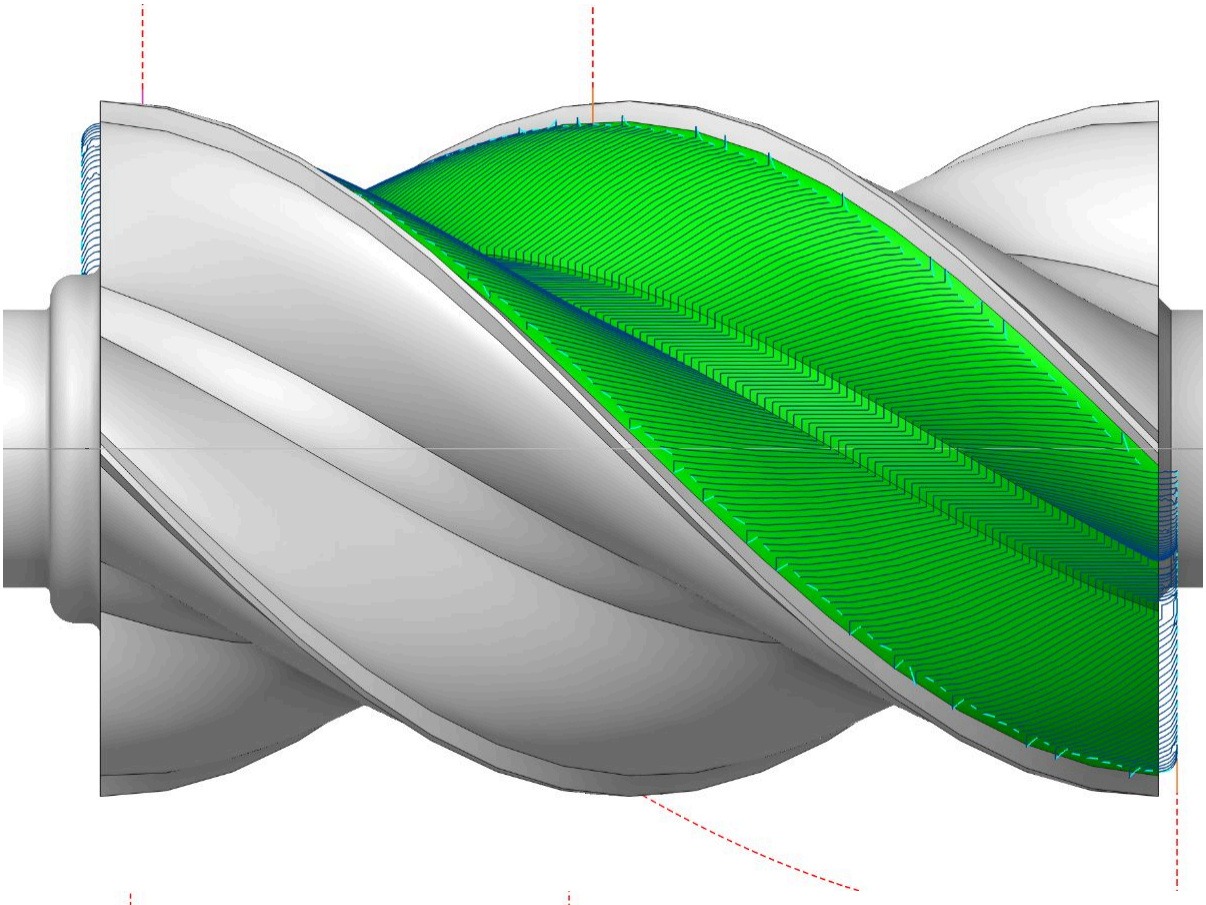
This parameter is located on the Parameters tab and allows running the old calculation algorithm when disabled, or the new one when enabled.

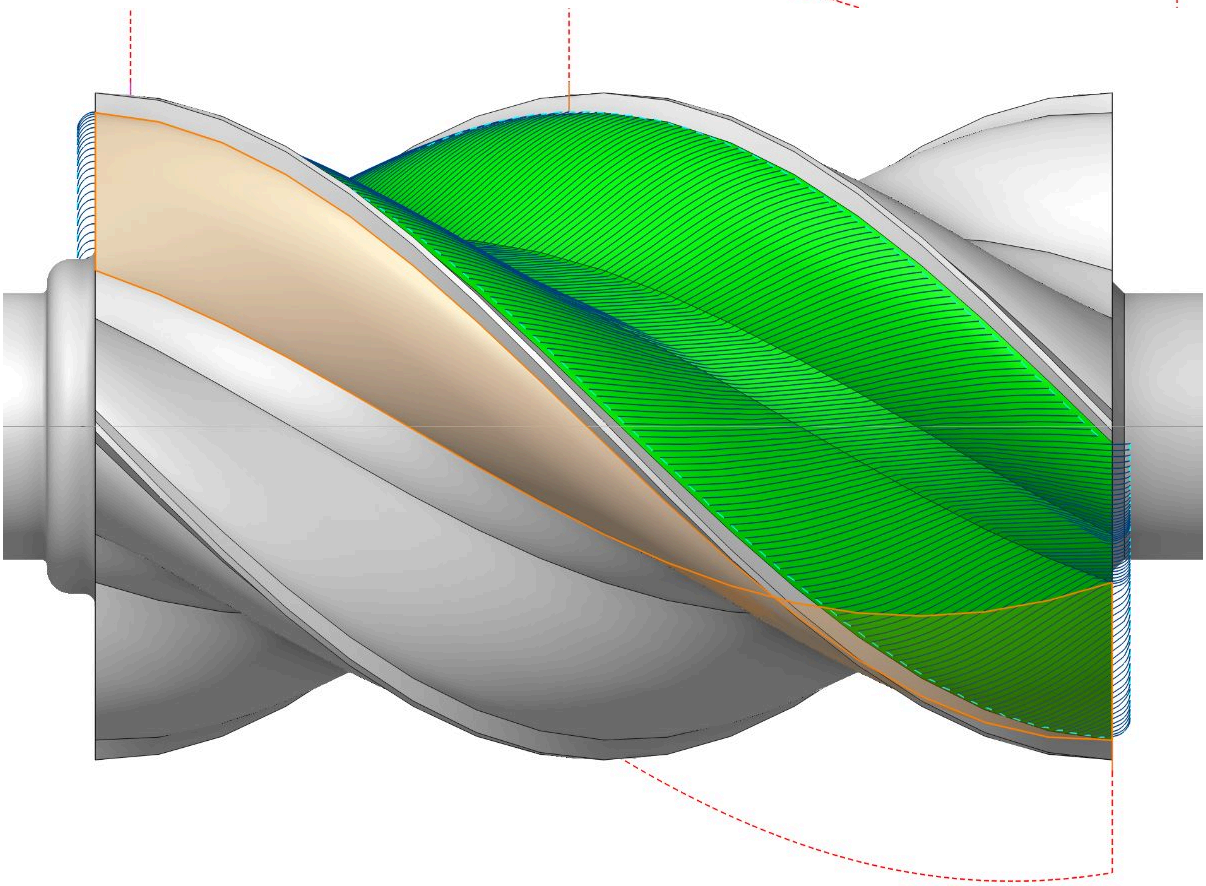
Benefits of the Updated Calculation Algorithm.

Enhanced Performance. The calculation process is significantly accelerated by distributing the computational load across all available CPU cores, maximizing hardware utilization.

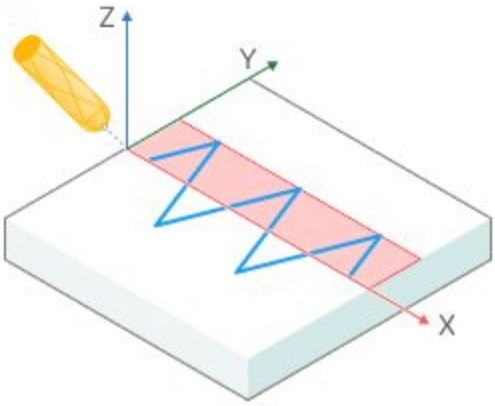
Optimized Toolpath Smoothness. When the tool axis orientation is set to follow the surface normal, the resulting toolpath is notably smoother. Additionally, users can fine-tune the smoothing effect by adjusting the Normal smoothing distance parameter. Details

Superior Toolpath Geometry. The algorithm demonstrates a marked improvement in toolpath quality when the Side Angle strategy is employed, leading to more efficient and precise machining.





Weaving



Application area:

Mechanical weaving is used, for example, to compensate for tolerances or to bridge gaps in a seam. The torch moves across the seam in this instance and the weave oscillation is thus superposed on the seam motion. It is also possible to rotate the torch in relation to the plane of the weld (direction of welding). Can also be used for polishing.

Weaving.

NC Code Format.

Defines how the trajectory is output.

▼ Click here to expand..
Long Hang. The trajectory is output with transformation according to the parameters.

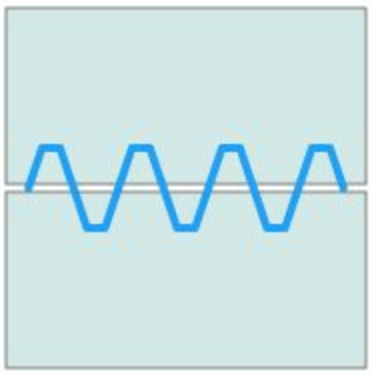
Canned Cycle. The trajectory is output as lines from the work order. At the beginning of each section, the cycle start command is output, and at the end, the cycle end command. In order for this to work, it is necessary to register the corresponding commands in the postprocessor.

Pattern. Pattern defining the type of weaving.

▼ Click here to expand..
Triangle. Deflection of the welding torch in 1 direction as Triangle.

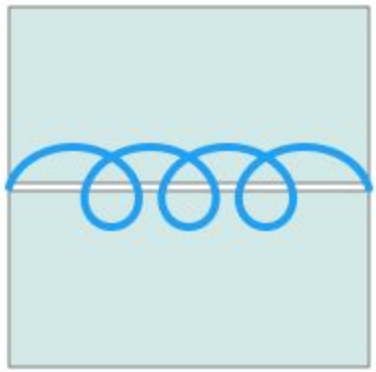


Trapezoid. Deflection of the welding torch in 1 direction with dual frequency.

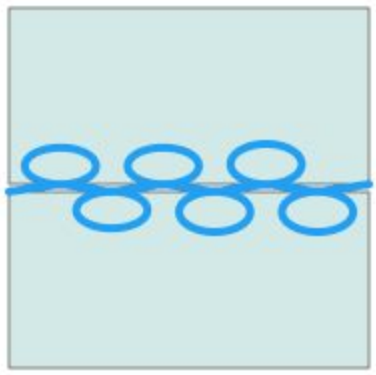


Unsymetric Trapecoid. Work with robot cycles.

Spiral. Deflection of the welding torch in 2 directions (amplitude = weave length/2).



Double 8. Figure of eight weaving.

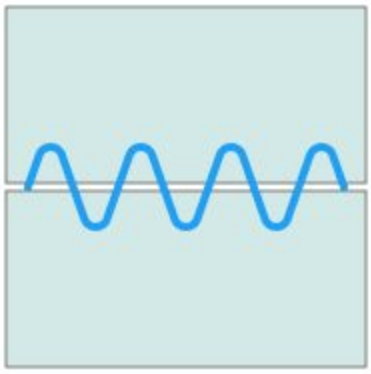


On Seam. Work with robot cycles

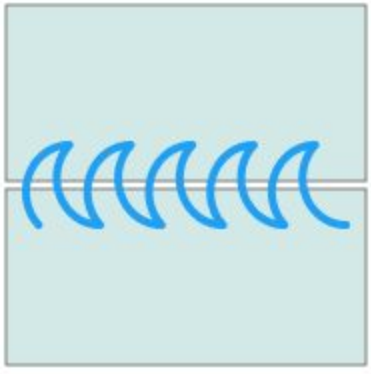
Edge Bottom. Work with robot cycles

Edge Top. Work with robot cycles

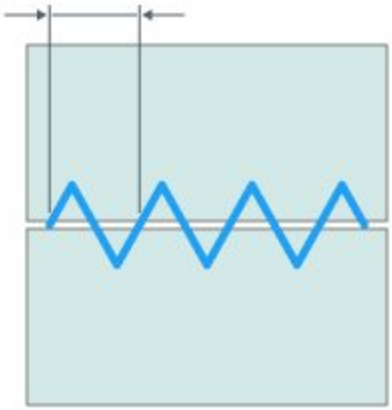
Curvy. Creates a curved zigzag toolpath (with arcs) on the piece.



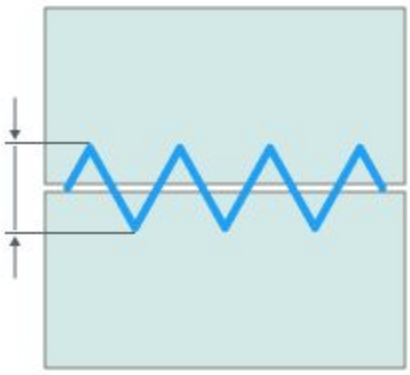
Crescent. Generates a template in the form crescent.



Weave length. Oscillation step



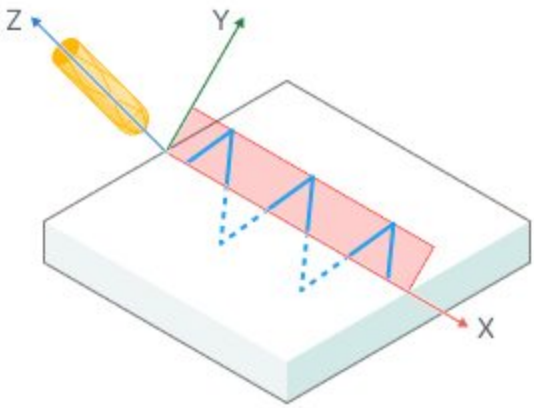
Amplitude. The maximum displacement of a point on wave from its equilibrium position.



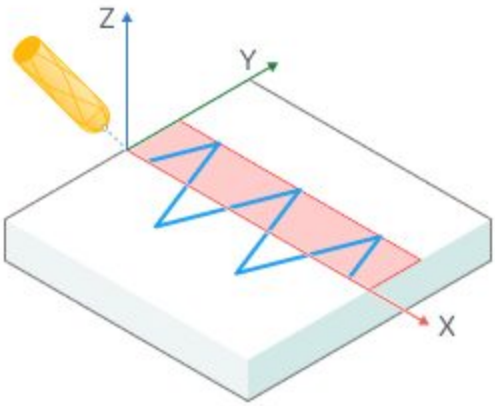
Weave plane. The plane in which it is determined weave.

▼ Click here to expand..

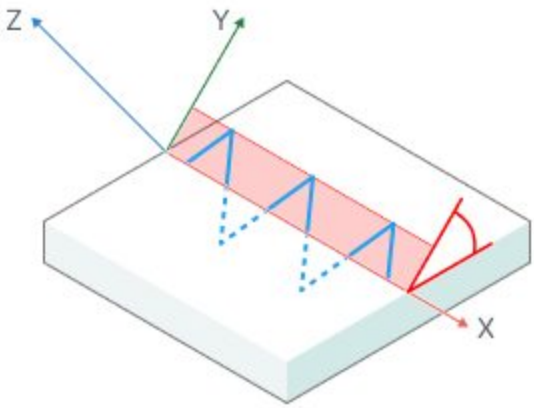
Tool axis. The plane normal vector corresponds to the tool vector.



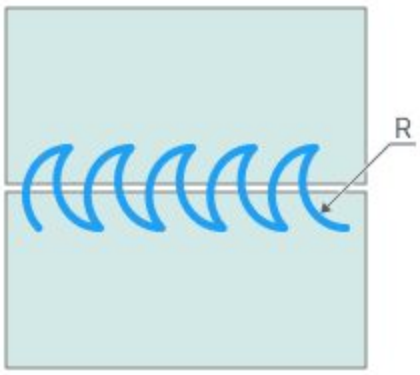
Normal to surface. The normal vector of a plane corresponds to the vector of the surfaces. Does not work with 5d Curve



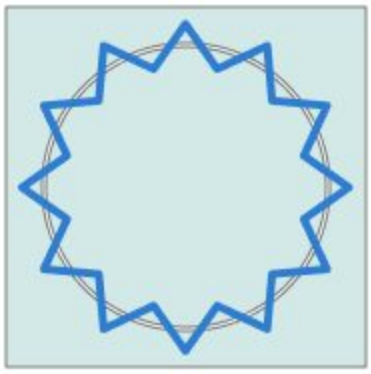
Plane angle. Defines an additional angle of rotation of the plane



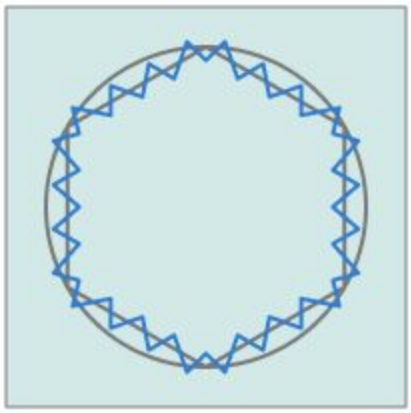
Radius. Radius for crescent.



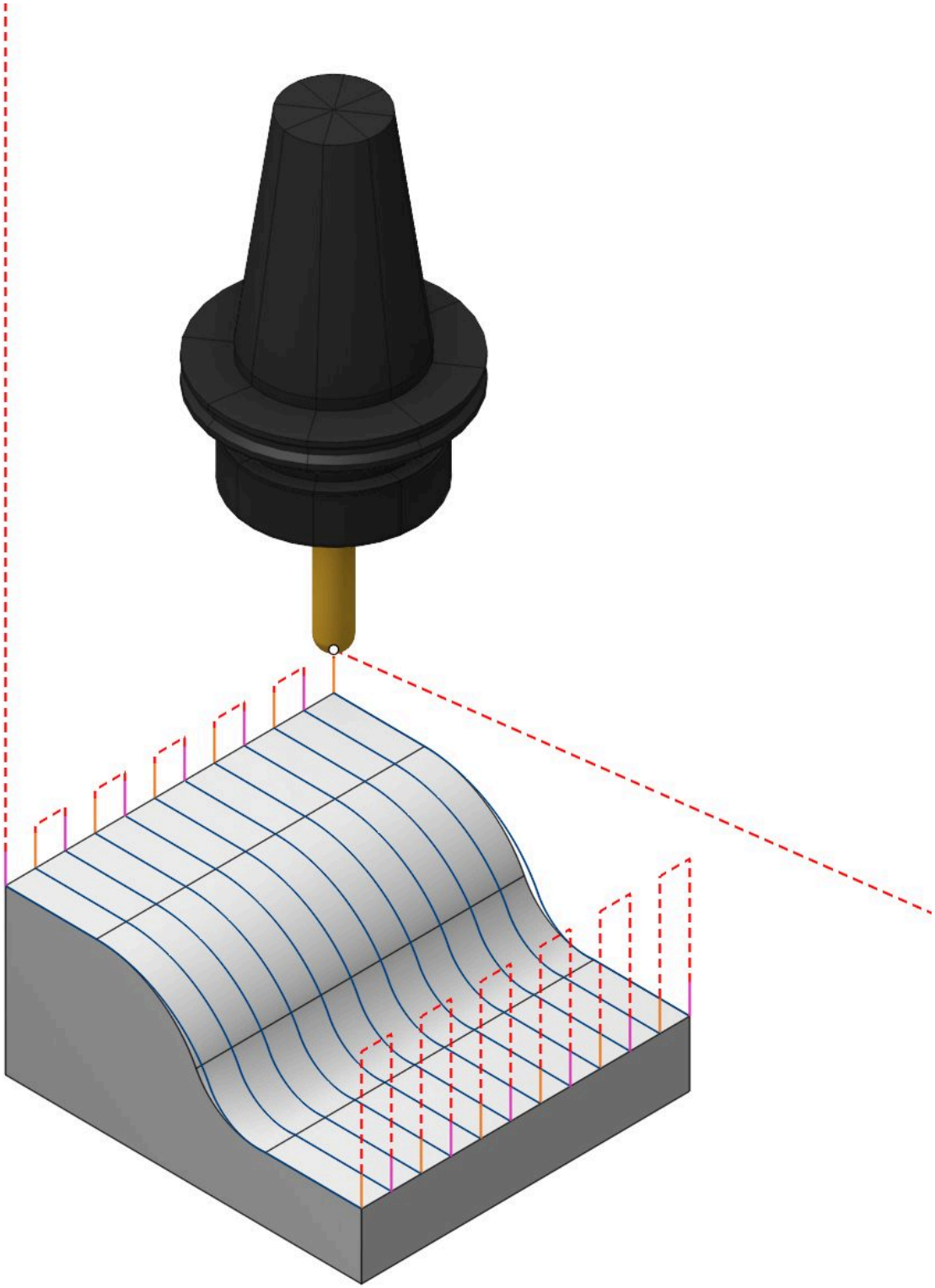
Project Onto Curve. If enabled: Follows all the curves of the trajectory



If disabled: Oscillations are built on straight segments equal to the length of the oscillation.



Links/Leads



Application Area:

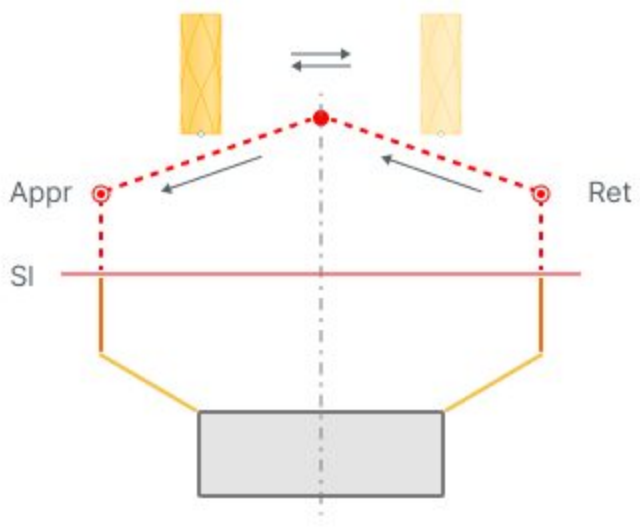
In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can

configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

Typically, the paths for non-cutting movements (idle strokes) consist of:

- Approach means the path the tool takes to arrive at the first point of an engage or plunge move either from a tool change position or from the last position of the previous operation.
- Lead In (Engage) involves a series of tool movements that facilitate a smooth transition to the initial point of the working stroke.
- Transitions are the toolpath sections between work passes.
- Lead Out (Retract) describes a sequence of tool passes, which allows its smooth departure from the last point of a working toolpath to the safe surface or to the safe distance.
- Return involves a series of tool movements that transition the tool from the exit point of the last operation to the tool change point via a safe plane.()

Approach/Return



Appr - Approach

Ret - Return

Sl - Save level

Application Area:

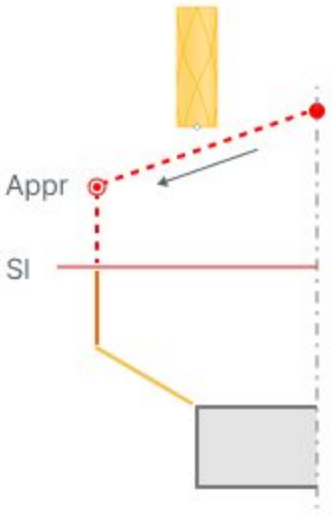
This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back.

Approach:

The approach describes the section of the toolpath that facilitates movement to the engage or to the lead in point for the initial working stroke from the tool change position or from the last position of the prior operation.

Typically, the approach divides into two movements:

1. Idle movement to a safe surface (Save level), safe distance, or designated level. Specify the safety surface and its location in the Safe Motions group, and for five-axis operations, use the Links group. You can create multiple trajectory segments during the approach. Methods to form the trajectory include: Short, Avoid Collisions, `Custom,` and `Rule from Previous/Root/Submachines`. Details.
2. Move at an accelerated feed rate to the working feed distance for the first working stroke along the Z-axis of the machine (for 3D milling operations) or along the tool axis (for 5D milling operations). Specify the final approach target and the corresponding distance values in the Links and Distances groups.



Appr - Approach

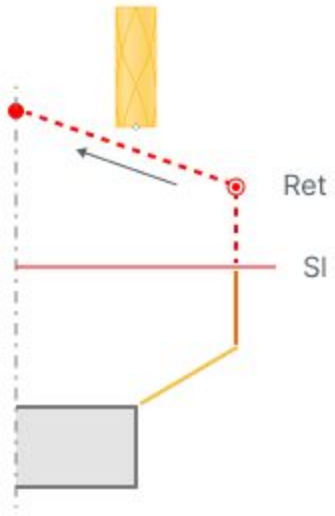
Sl - Save level

Return:

The return describes a series of tool movements that facilitate the tool's transition from the departure point of the final working stroke to the tool change point, passing through a safe surface or a safe distance.

Typically, the return divides into two movements:

1. Move at an accelerated feed rate from the retraction point of the last working stroke along the machine's Z-axis to a safe surface, safe distance, feed switch level, or a designated level. The principle for forming the withdrawal remains the same as for the approach.
2. Idle movement to the tool change position.

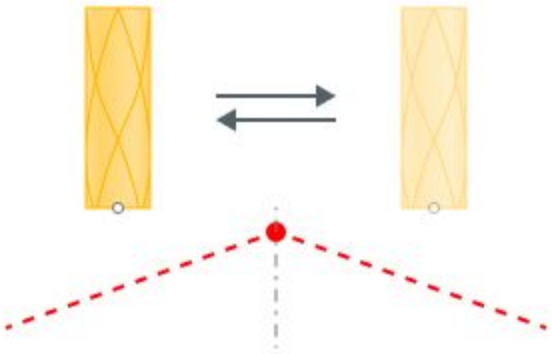


Ret - Return

Sl - Save level

Tool Change Position:

The tool change point is the machine position where the tool change occurs. In the CAM system, you can define the tool change position for the overall tech process or for separate operations. Details.



Approach and return rules

The Approach and return rules define an additional tool approach/return path to/from the first/last machining point. When used together with the tool change position approach rule determines a tool path from a tool change position to the first machining point, while the return rule determines a tool path from the last machining point to a tool change position.

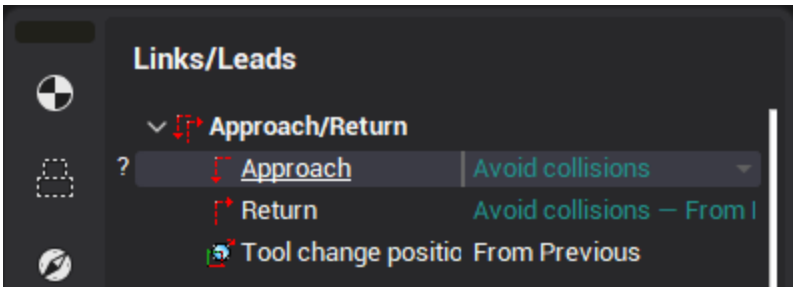
Defining approach/return rule of an operation

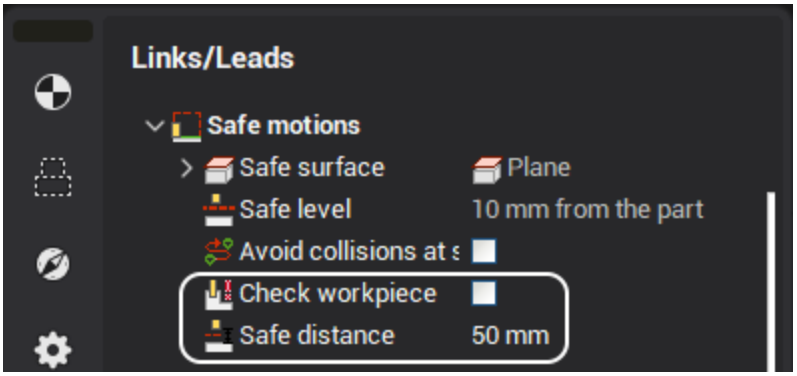
The approach/return rule is specified under the Approach/Return section in the operation Links/Leads inspector panel. The check box indicates whether or not the approach/return is automatically generated to avoid collisions.*Collision avoidance* - generate approach/return trajectory automatically with special algorithm, while avoiding machine collisions. Disable the checkbox to select this approach/return type. If this type is chosen you can additionally define "*Safe distance*" and "*Check workpiece*" parameters for the approach/return.

If the approach/return check box is enabled, this means that the rule is defined explicitly or has one of the listed below special types. The edit field shows the actual rule, computed according to the approach/return (hidden) type. To adjust the approach/return used by the operation you can just edit the command directly.

Approach/return definition types

The approach/return can have one of the following definition type. This type is hidden, and is shown only in the Approach return rules editing form.





Default rule - the approach/return for a newly created operation has this type. To reset the rule back to the default one clear the edit field and press Enter. The following factors which are considered during the

actual rule computation of the default type (in decreasing priority): if the submachine is defined for the operation's workpiece holder/tool holder pair, then use the submachine approach/return. See Submachine definition in the machine schemas for more info about the submachines. if the operation has the same workpiece holder and tool connector as the previous operation, then the approach/return is taken from the previous operation. otherwise use the rule from operation's setup stage or the rule defined in the first operation of operations tree (the machine), in case there are no setup stages or the operation is a setup stage itself.

2. *Rule from the previous operation.*
3. *Rule from root operation* - use the rule, specified in the root of operations tree on 'Technology' tab (not the single rule defined in the machine, as in previous legacy CAM system).
4. *Custom* - the rule is defined explicitly as the list of commands.
5. *Short* - no additional points are added into toolpath. A tool moves from a tool change position to the first point of a machining toolpath directly by the shortest distance. If the machine is robot, then this movement is done using physic axes (PhysicGOTO).
6. *Rule(s) from the machine approach/return list* - operation can reference one of the items in the machine's approach/return list. Several approaches/returns can be specified in the machine under meaningful names. First, the name of the rule is displayed, then, in the round brackets, the rule's sequence of commands. More information about the list is available in the separate article.

Approach/return rules syntaxis

A Custom approach/return rule defines explicitly the outputted CLData in the "Approach" or "return" section of the toolpath. A single approach or return rule consists of several *commands*, which are separated by the semicolons. A *command* is defined by the *keyword*, which may be followed by the list of machine axis names and values (coordinates).

Axes value list is defined as a whitespace separated list of machine axis ids (or addresses) with optional exact positions of machine axes in the state which corresponds to the given *command*. The axis value can be specified in brackets or directly after the axis id. Examples of the axis value definitions:X100 Y(200) Z A1 A2(253.2) A3(100.4) A4 A5 A6

The following command types (*keywords*) are supported:MultiGOTO (or no command) + *Axes value list* - defines multi coordinate movement ("Multigoto" CLData node). PhysicGOTO (or G53) + *Axes value list* - defines movement in physical axes coordinates ("Physicgoto" CLData node). GOTO + *Axes value list* (must be "X", "Y" or "Z") - defines simple linear movement ("Goto" CLData node). GOHOME + *Axes value list* - movement to the "Tool change" position ("GOHOME" CLData node). Intended for use in the return rules only. LCS - multi purpose command. The common logic is that some machining mode is enabled in the particular place in the approach, and then disabled using the same command in the return. The outputted CLData depends on the operation parameters: If the Operation local coordinate system is

enabled, then the LCS enable command ("Origin LCS: On") is outputted inside the approach, and the LCS disable (Origin LCS: Off) in the return. If the "Tool center point management" is enabled, then the TCPM mode enable command ("Interp 5axis: On") is outputted inside the approach, and the TCPM disable (Interp 5axis: Off) in the return. Note: "TCPM" mode can't be used simultaneously with the Local coordinate system. Polar interpolation enable/disable U-axis turning mode enable/disable SLCS(...) - used for temporary enabling of the "Local coordinate system" inside the approach or return. See the "Approach for the TCPM enabled operations" section below for more info about this feature.

Approach/return for the TCPM enabled operations using Local CS

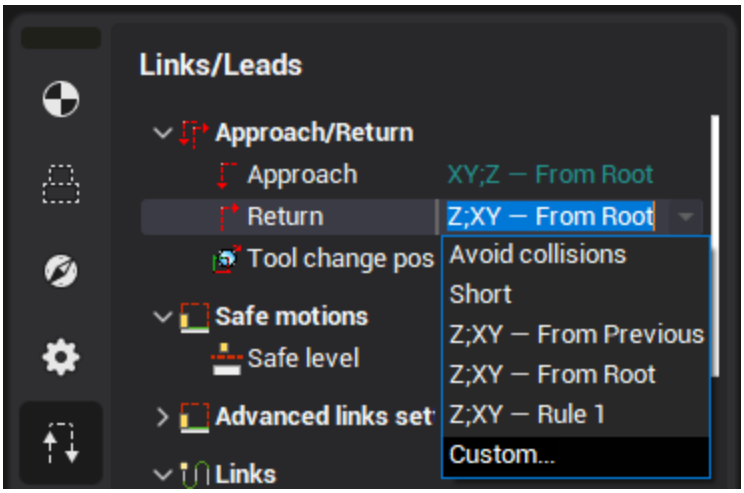
The "Tool change point management" mode is commonly used in the 5 axis machining, but some problems arise when the machine kinematics and workpiece mounting do not match completely the real machine kinematics and part mounting. This will most likely cause a collision. The solution is to perform some machine movements, which are inside the SLCS(...) block, in the specific "Local coordinate system" *before* the "TCPM" mode is enabled in the approach. If the SLCS() command is used the "TCPM" mode will be activated in the end of the approach; it will be deactivated in the beginning of the return, if the given command sequence is in the return rule.

The LCS used by this command is not defined by the operations' parameters, instead, it corresponds to the first point of the toolpath, if the SLCS command is inside the approach section and to the last point of the toolpath, if the command is inside the return section. The rotary axes movements should be done before enabling this LCS.

Inside the brackets one or more movement commands can be specified, e.g. SLCS(G53 X100; YZ). These movements will be performed in the given Local coordinate system. Example of the full approach rule using SLCS(): `G53 Z(-0.5); G53 X(0.5) Y(-0.5); AC; XY; SLCS(XYZ)`

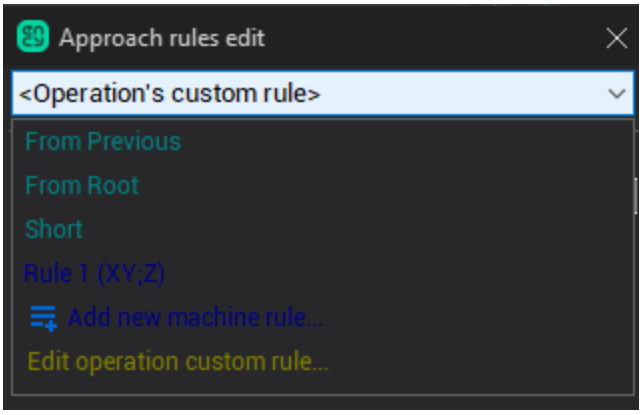
Advanced approach/return rules editing

The edit field allows quick adjustment of the approach/return of the operation. If you need, for example, to link the operation's approach with the previous operation or specify a long sequence of commands, use the approach/return edit form. To open it use the ellipsis button of the edit field. It also allows to edit the machine approach/return list.


Links/Leads
Approach/Return
Approach
XY;Z — From Root
Return
Z;XY — From Root
Tool change pos
Avoid collisions
Short
Z;XY — From Previous
Z;XY — From Root
Z;XY — Rule 1
Custom...
Safe motions
Safe level
Advanced links set
Links

Approach/return rules edit window

Selecting the approach/return type



Use the above combo box to select the definition type of the approach. For example, '*From Previous*' item means the approach will be the same as in the previous operation. See the main article for more info about the approach/return types.

Interactive editing of approach/return

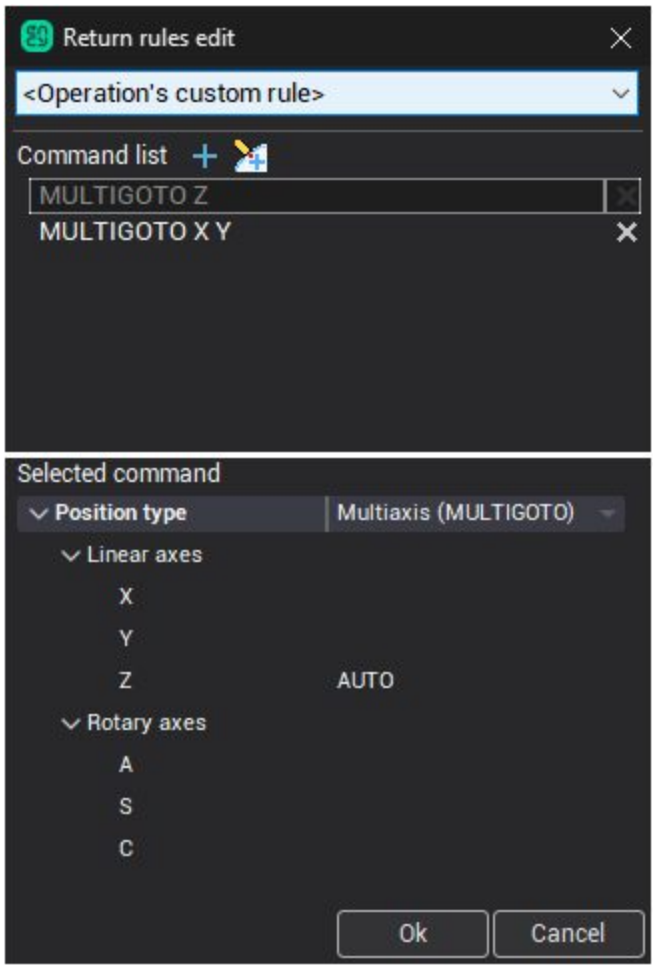
In the approach/return edit dialogue you can define the operation's approach/return tool path or edit the machine's rule list. The dialogue window is split into three areas.

1. In the top most combobox you can select to edit operation's own approach/return or one of the items in the rule list.

2. Below, in the center of the form, there is the list commands which define current selected rule. Each line (command) in that list represents a distinct point of the lead path. A point has the following format.If G53 is present in the beginning of a line the subsequent coordinates are the coordinates in the machine coordinate system, otherwise the coordinates are the coordinates of the tool tip in the current workpiece coordinate system. The subsequent text has the following format: Moved Axis 1 name, than optional moved axes 1 value in round brackets (e.g. X1 or X1 (30)), then space, than Moved axis 2 name, then optional moved axes 2 value, then space, and so on.

If no coordinate is specified after an axis name, it means the corresponding coordinate is calculated by CAM system automatically. For an approach path the coordinate is taken from the first point of the machining toolpath, for a return path the coordinate is taken from the tool change position. So, for example, if you set an approach path to "X Y; A;C; Z", the tool will move from a tool change position to the first point of the machining toolpath first by the X and Y axes simultaneously, than by A axis, than by C axis, the final motion will be a vertical plunging along the Z axis.To add a new empty command press the + button, and to add current machine state as a command press the button next to previous. To delete command from rule click on 'X' icon in the right column. Use drag'n'drop to arrange commands in the desired order. Double click on command to set the tool position according to the defined coordinates in selected command Using popup menu (right click on the command) or corresponding hotkey you can

cut, copy, paste or delete selected commands. You can edit the command's text directly using the 'Edit' popup item (or press Ctrl+E).

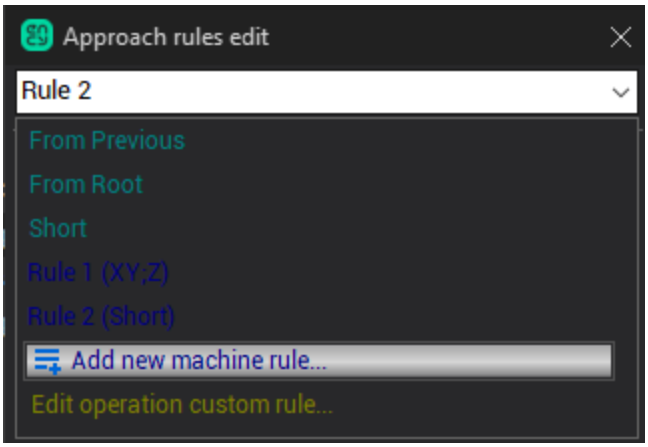


3. In the bottom area of the form there is a list of machine axes. To add a new reference point to the path you should either fill the coordinates of the appropriate axes with desired values, and/or set some of them to AUTO by clicking on the ellipses button next to an axis. To edit a reference point just click on it in the top list, the bottom list will be updated accordingly. After that you can change the reference point parameters.

The 'Position type' option in the middle of the inspector is used to set the mode of a reference point. When checked it turns a reference point to a G53 reference point. For a G53 reference point coordinates are specified in the machine coordinate system. Those points are output into a CLDATA using PhysicGOTO commands, while regular reference points are specified as coordinates of a tool tip in the current workpiece coordinate system. Those points are output into a CLDATA using GOTO and MULTIGOTO commands.

As approach and return reference points can be specified in terms of machine axes without explicit coordinates values, it is possible to define for an actual machine standard approach and return sequences. E.g. for a five axis milling machine a default approach sequence can look like XY; A C; Z, a return sequence can look like Z; XY; A C.

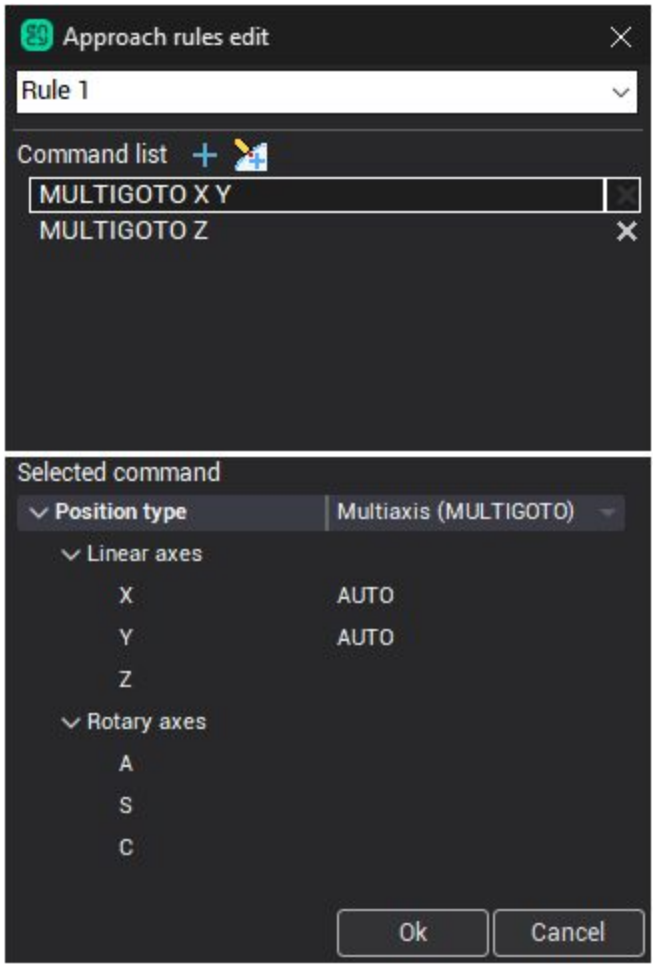
Editing the machine's rule list To interactively edit one of the approach/return rules in the machine first select it by it's name using the topmost combobox. The command tree will be refreshed according to the selected rule. Then the rule can be edited the same way as the operation's own rule. The combobox can be used as an edit to change the current rule's name. To delete rule from the list click on the 'X' icon which appears when hovering mouse over an item in the combobox. To add new empty rule to the list use Add new machine rule button in the dropped down combobox. To switch back to operation's rule click on Edit operation custom rule..



Machine's approach/return list

Approach/return list overview

In the "Leads" section of machine's xml file several rules for approach/return can be specified for convenient use. Each item of the list has a name to distinguish it from other rules. If the rule in the list was edited, it will affect all the operations which reference this rule. To assign a rule to operation, select the necessary item in the Approach/return edit form:



Approach/return list alternative

There is an alternative way to have several rules for approach or return - create submachines for the workpiece holder/tool holder pairs used in the project and define appropriate approach and return rule for each submachine. See Submachine definition in the machine schemas for more info about submachines in CAM system.

Editing approach/return list

There are 2 ways of editing the machine's approach/return list.

1. Direct editing of machine's xml file (shown in this article)

2. Using the approach/return edit form. To display the form click the ellipsis button in the approach/return edit field.

Editing machine's xml file

The machine's approach/return list is located in the Leads section (previously here could be specified just one approach/return rule). ApproachCommands and ReturnCommands are the names of the respective subsections. Each rule has 3 fields:

1. RuleID - unique GUID of the rule. It's used as a method for operations to reference a particular rule. See the operation's GUID as an example.
2. Name - name of the rule.
3. Command - the rule itself, the sequence of intermediate points of approach/return. To specify the approach/return with collision avoidance use keyword Auto.
4. Type - type of the rule which defines possible use case for the approach/return. Currently there are 4 available types:

- LCS - approach/return can be used only if the local coordinate system is enabled for the operation.
- TCPM - can be used only if the tool center point management (TCPM) is enabled (both LCS and TCPM cant be enabled at the same time).
- General - the rule can be used only if both LCS and TCPM are off.
- Undefined - the rule doesn't depend on LCS or TCPM state. If the type wasn't explicitly defined it is assumed to be 'Undefined'.

Below is the partial example of xml file:

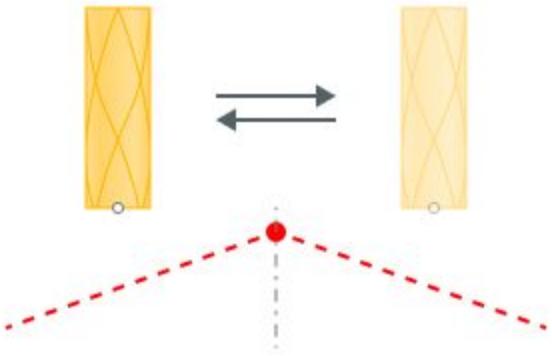
```
                    MaxTurn65WithCounterSpindle
<SCType ID="MaxTurn65WithCounterSpindle" Caption="MaxTurn65 with Counter Spindle"
type="MaxTurn65" Enabled="true">
        <GUID DefaultValue="{8E0CEF0A-8045-436D-89FD-BBE70D387AB1}"/>
        <Priority DefaultValue="172"/>
        <Name DefaultValue="MaxTurn 65 with Counter Spindle"/>
        <Comment DefaultValue="MaxTurn 65 with Counter Spindle"/>
        <Leads>
        <ApproachCommands>
                <SCArray>
                        <Rule>
                                <RuleID>{41D3BB1C-2F23-47AC-B5F9-5DAF7030A015}
</RuleID>
                                <Command>C;Z10;X;Z</Command>
                                <Name>Left spindle approach</Name>
                        </Rule>
                        <Rule>
                                <RuleID>{54FC19E5-8ACB-491A-8E94-FC9990FC8680}
```

```
</RuleID>
                                <Command>C2;Z-10;X;Z</Command>
                                <Name>Right spindle approach</Name>
                        </Rule>
                </SCArray>
        </ApproachCommands>
        <ReturnCommands>
                <SCArray>
                        <Rule>
                                <RuleID>{380D355A-6708-4C86-BA69-7521A0198A8E}
</RuleID>
                                <Command>Z10;X;Z;C</Command>
                                <Name>Left spindle return</Name>
                        </Rule>
                        <Rule>
                                <RuleID>{18B61D64-EEC9-4403-8922-CCFCC017E53E}
</RuleID>
                                <Command>Z-10;X;Z;C2</Command>
                                <Name>Right spindle return</Name>
                        </Rule>
                </SCArray>
        </ReturnCommands>
</Leads>
```

Conversion of the older version machines/projects

When the older version machine is opened for the first time, the list consisting the machine's previous approach/return rule is created. Also the root operation references this rule (if no custom rule is assigned to it). As a result, operations having 'From root operation' rule type use the same approach/return as before.

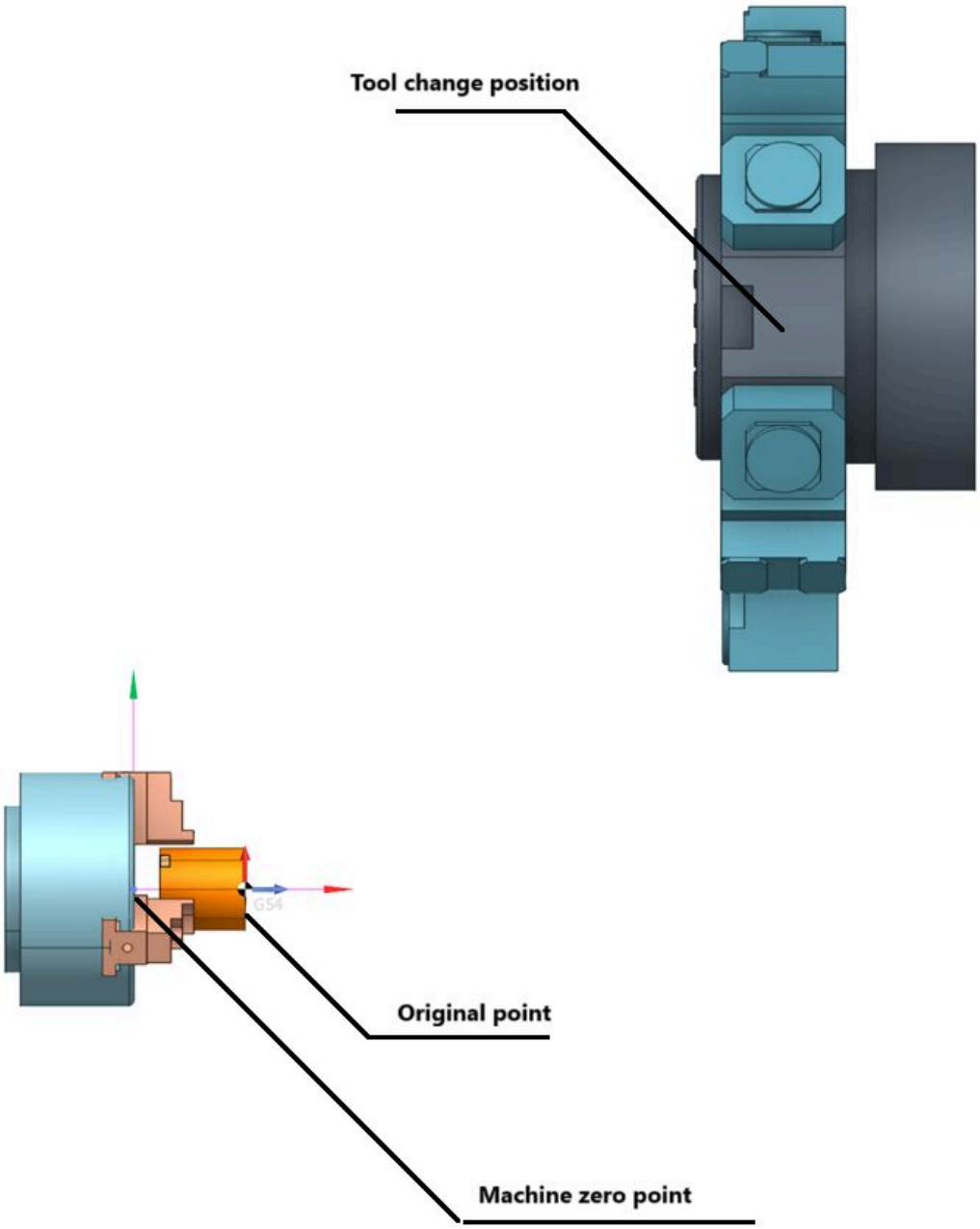
Tool change position



Application Area:

Tool change position is the machine position in which tool changing is performed. In CAM system you can specify a default tool change position for a whole job list and override this setting in individual operations as you need. The tool change position is specified under the Approach/Return section in the operation Setup panel. Defining a tool change position is not obligatory. If you not define a tool change position, then in the Simulation mode tool changing is not simulated. A tool is just appearing in the beginning of an actual operation and disappearing in the end of an operation. This setting is usually used for conventional 3 axis milling when there is no risk to damage a machine or a part while tool interchanging. However for complex machines like five axis milling centers and mill-turn machines specifying a valid tool change position is crucial for safe and time-efficient machining.

A tool change position is defined as a set of machine coordinates (e.g. at the above picture X1(0) Y1(0) Z1(0)...) in the machine coordinate system (G53). Those coordinates are the physical coordinates, not the coordinates of the tool tip in the workpiece coordinate system. So for a mill-turn machine those coordinates are the coordinates of the turret head center point.



For an actual operation you can select a tool change position from the list of the following options

1. From the previous operation - the tool change position is inherited from the previous operation.
2. From machine - the tool change position is inherited from the position specified under the machine node.
3. Custom. If you select this item, a tool change position dialog appears.

The tool change position dialog is used to define a tool change position as well as the tool change behavior for an actual operation in an interactive mode. Here you can set:

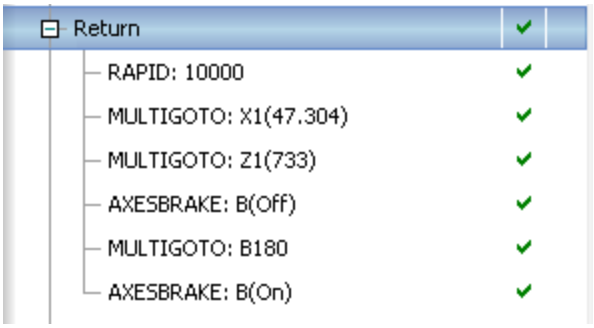
1. Mode (way of assignment) (Avoid colision, Short, From root, From the previous operation and Custom).

2. Using of tool change position.

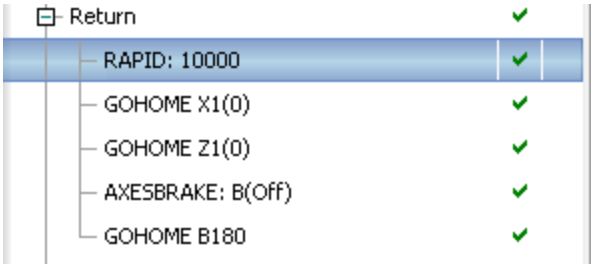
- Auto (go to position for the tool change). If this option is selected the tool change position is used only when two neighboring operations use different tools with different tool numbers. If two neighboring operations use the same tool then no NC code is generated for the tool change, the second operation just starts machining in the last point of the preceding operation. This is the default behavior.
- Use anyway. When the option is selected, a tool unconditionally goes to the tool change position in the end of the operation.
- Do not use. The tool does not go to the tool change position in the end of machining.

3. Tool change output mode.

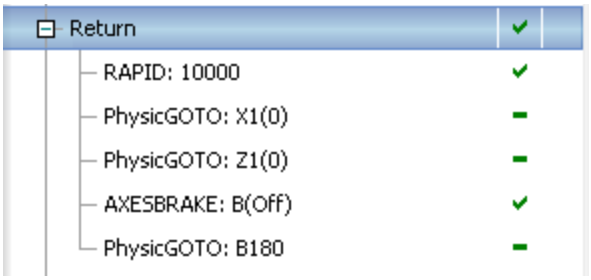
- Tool Tip coordinates. CAM system generates the tool change movements using regular GOTO and MULTIGOTO commands, representing the coordinates of the tool tip in the workpiece coordinate system.



- Reference point (ISO G28). When selected the option forces CAM system to generate the GOHOME commands for the tool change positioning. The GOHOME commands represent coordinates in the machine coordinate system and can be used in the postprocessor to generate "Go to reference point" commands (ISO G28).



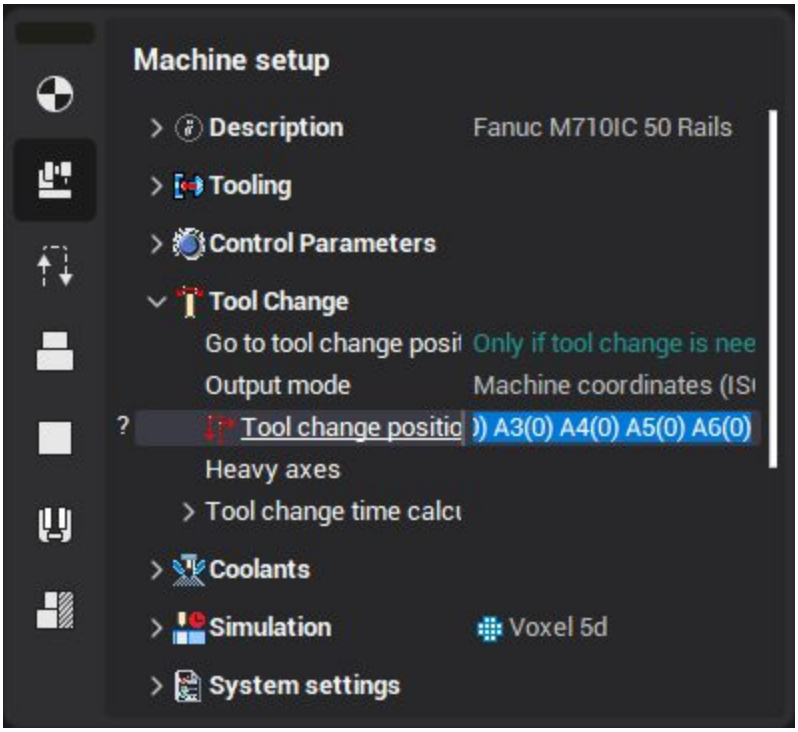
- Machine coordinates (ISO G53). When selected the option forces CAM system to generate the PhysicGOTO commands for the tool change positioning. The PhysicGOTO commands represent coordinates in the machine coordinate system and can be used in the postprocessor to generate "Go to machine coordinate" commands (ISO G53).



4. Coordinates of the tool change position.

The coordinates are specified in the list. To define a tool change position you should check the boxes near the appropriate axes and set the values for the axes. You can set the axes coordinates both in the machine (G53) and the workpiece (G54) coordinate systems. The result is displayed on the screen. A convenient way to define a tool change position is to use the mouse wheel to change axes coordinates while looking at the result in the graphic view.

The default tool change position can be specified in the machine parameters panel



and in a machine configuration file under the ToolChangeMachineState tag.

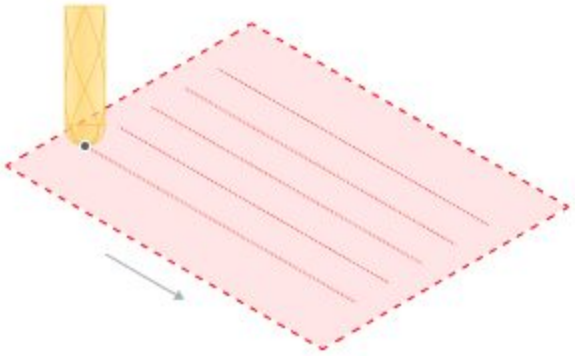
```
  <SCType ID="TMyMachine" type="AbstractMachine" Enabled="true">

SimulateToolChange DefaultValue="true"/

ToolChangeMachineState DefaultValue="X1(0) Y1(0) Z1(0) B180 A0 X2(0) Z2(0)"/

/SCType
```

Safe Motions



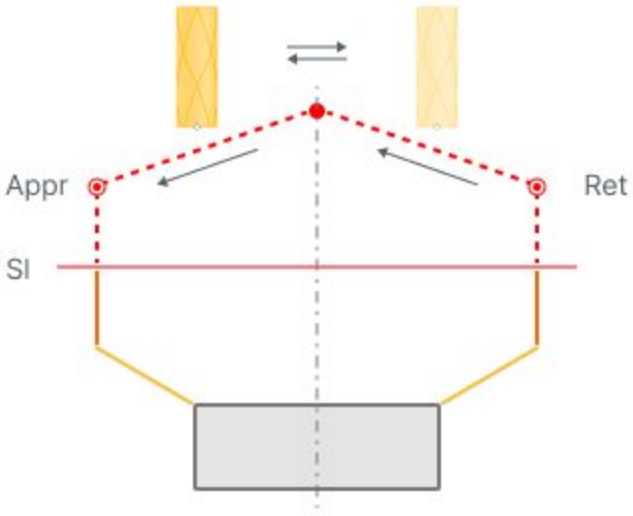
Application Area:

Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it.

Safe Level:

By default, sets the distance from the most protruding part of the workpiece to the Safe surface.

▼ Click here to expand..



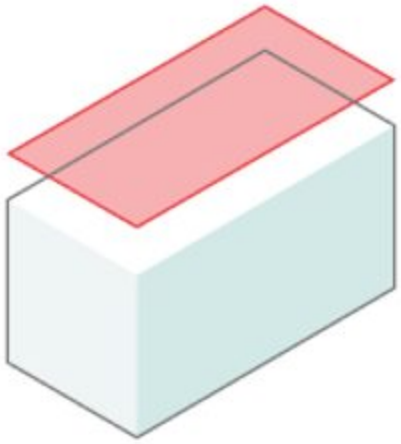
Appr - Approach Ret - Return Sl - Safe level

Safe Surface:

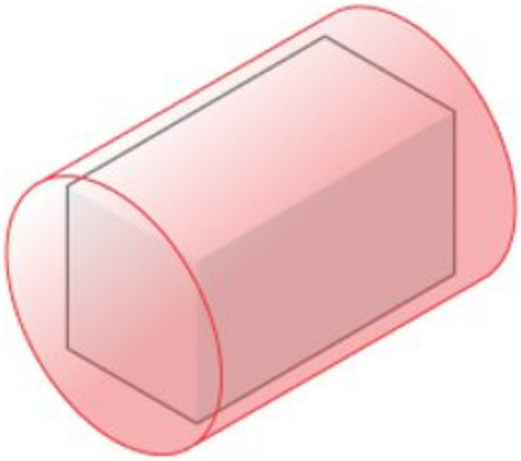
Defines the surface shape that determines the transition path between the working passes of the tool if the links type is specified as On Safe Surface.

▼ Click here to expand..

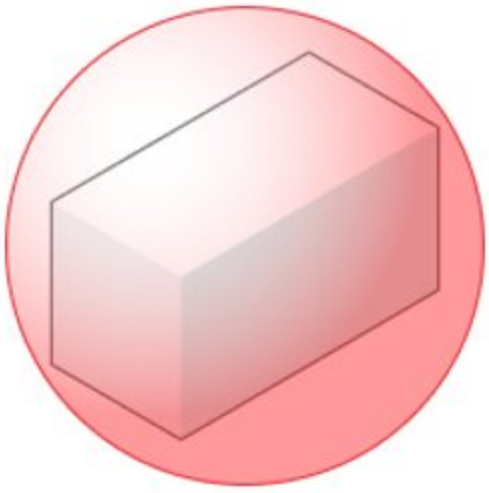
Plane. The path for transitions is set on a plane located at the Safe Level above the workpiece. This type of surface is the only option for three-axis operations.



Cylinder. The transition path is set on the cylindrical surface located at a Safe Level above the most protruding part of the workpiece. This surface option is applicable for four- and five-axis operations.



Sphere. The transition path is set on the surface of a sphere located at a Safe Level above the most prominent part of the workpiece. This surface option is applicable for five-axis operations.



The following options can be used to define surfaces:

Origin.

For a Plane, it establishes the base point from which the Direction vector begins, aligned with the normal to the plane. For a Cylinder, it sets the starting point of the axis. For a Sphere, it defines the sphere's center.

Center of the Part.

The Origin point is located at the geometric center of the part.

Custom Point.

The Origin point is defined by its X, Y, Z coordinates.

Direction.

For a Plane, it sets the direction of the normal vector to the plane. For a Cylinder, it sets the axis vector.

Auto.

The system automatically sets the direction vector: for a Plane, it runs along the tool axis; for a Cylinder, it follows the part axis or the machine axis with rotational coordinates.

Custom Vector.

The Direction results from projecting the unit vector onto the X, Y, and Z axes.

Max Retraction Distance:

It enables the restriction of the maximum retraction height. This option applies only to certain operations, including Hole Machining and Probing.

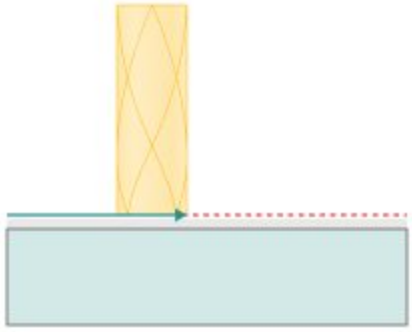
Avoid Collisions at Safe Plane:

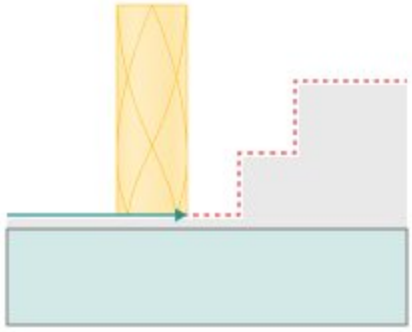
When the flag is set, the system automatically generates a trajectory for transitions within the operation while avoiding collisions. This option applies only to certain operations, including Chamfering, 5D Cladding, Point Welding, 6D Welding, and Spraying, mainly in robotic applications.

▼ Click here to expand..
Check Workpiece.

If the option is active, the system considers collisions with the workpiece when generating transitions that avoid collisions.

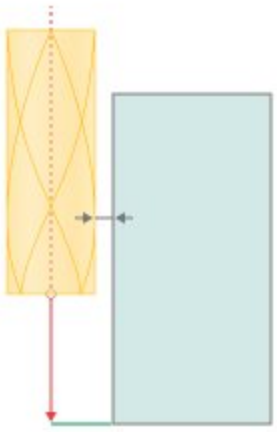
If the option is inactive, the transition paths consider only collisions with the part during collision avoidance. Use this option if the workpiece and the part of the operation differ slightly (or differ within a safe distance). This parameter also affects the approach and return with collision avoidance.





Safe distance.

If the collision avoidance for links is enabled, the parameter defines the minimum allowed distance between the tool, part (workpiece) and the machine parts. This parameter also influences the approach/return with collision avoidance.



Advanced Links Settings

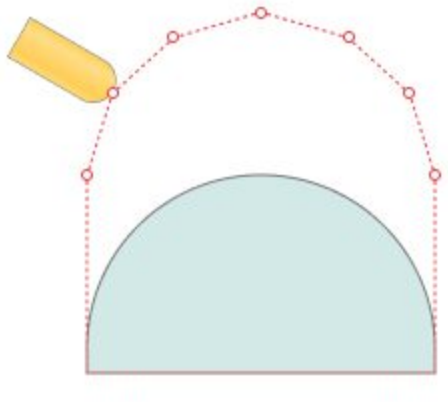
Application Area:

This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes.

Approximate Safe Motions:

This parameter manages the trajectory composition for safe movements on surfaces like cylinders and spheres, in five-coordinate operations, where safe motions utilize rotational coordinates.

▼ Click here to expand..



Off. Safe motions on the safety surface execute in a single block.

Always. The trajectory on the safety surface divides into basic movements, so the angular coordinate shift does not exceed the specified value.

When Needed.

Advanced Axes Limits Control:

Using this option prevents two specific problems in the five-axis trajectory:

1. The system excludes overturn during the working pass whenever feasible. Overturn take place not during working passes but in segments of the trajectory with rapid movements.

2. The system creates a smooth path in singularity zones. A singularity zone refers to a position of the machine where the coordinate of one axis cannot be calculated. For example, if axis A equals zero, then axis C can take any value.

▼ Click here to expand..
Avoid Singularity on Safe Surface.

Allowed Axis Deviation in Singularity.

The parameter defines the angle by which the machine or robot joints can deviate from the nominal calculated position to bypass the singularity zone.

Machine Optimized Transitions:

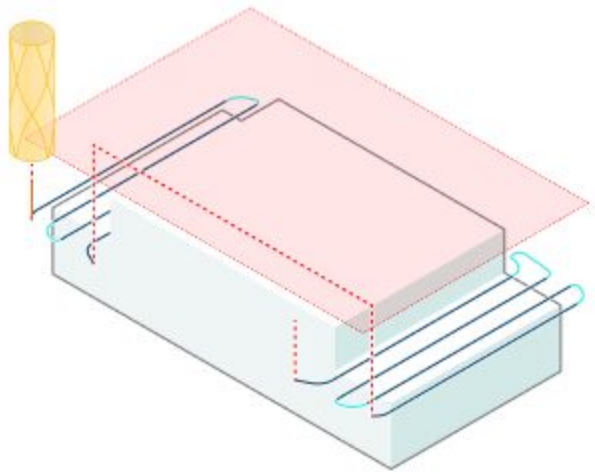
When the option is disabled, the transitions at the level defined by the Rapid Distance are made by the euclidian space shortest path. Let's say you want to move from point1(X1,Y1,Z1,Nx1,Ny1,Nz1) to point2(X2,Y2,Z2,Nx2,Ny2,Nz2), where X, Y, Z are coordinates of the tool center, and Nx, Ny, Nz are coordinates of the tool axis direction vector. The euclidian shortest path is a linear interpolation of these geometrical vectors. It means that the shape of the connecting curve is a straight line with linearly interpolated tool axis direction vector along it.

When the option is enabled, the transitions at the level defined by the Rapid Distance are made by the machine space shortest path. Let's say you want to move from point1(A1,C1,X1,Y1,Z1) to point2(A2,C2,X2,Y2,Z2), where X, Y, Z, A, C are machine axes. The machine coordinates shortest path is a linear interpolation of these axes. It means that the shape of the connecting curve depends on an actual machine schema configuration. Machine optimized transitions look longer, but in fact they are more optimal in many cases than the geometrical ones.

Links

The Link constitutes an elementary section of the toolpath existing between working moves. The parameters of this tab establish the transition execution method, which is dependent on the operation type.

Links based on the Roughing waterline

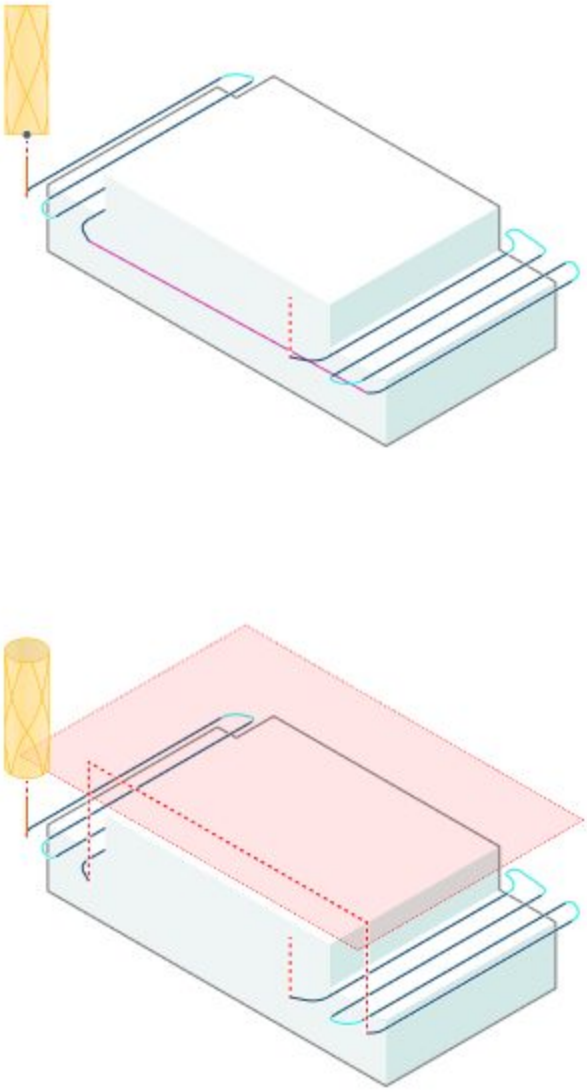


Application Area:

This group defines the settings for cutting in or transitions between areas.

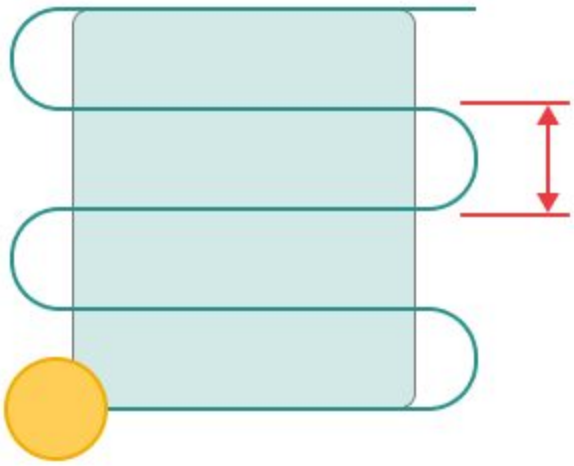
Go up if farther.

▼ Click here to expand..
Enables checking the length of links, and if the length of the link is greater than defined value, then the link will be made via the safe level.

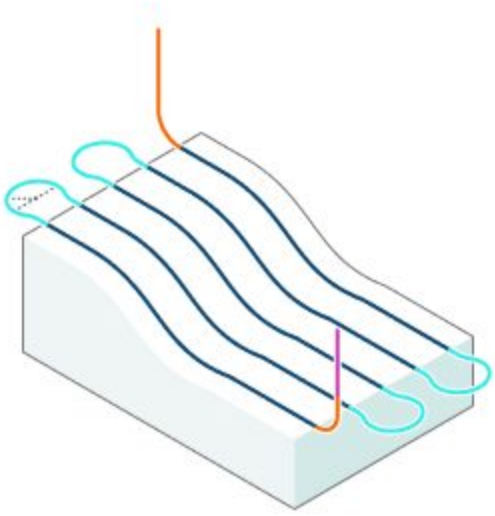


GF - Go up if farther
Short link max distance.

▼ Click here to expand..
Sets the maximum distance at which links are made by the short link type.



Links for 3D/5D advanced operations



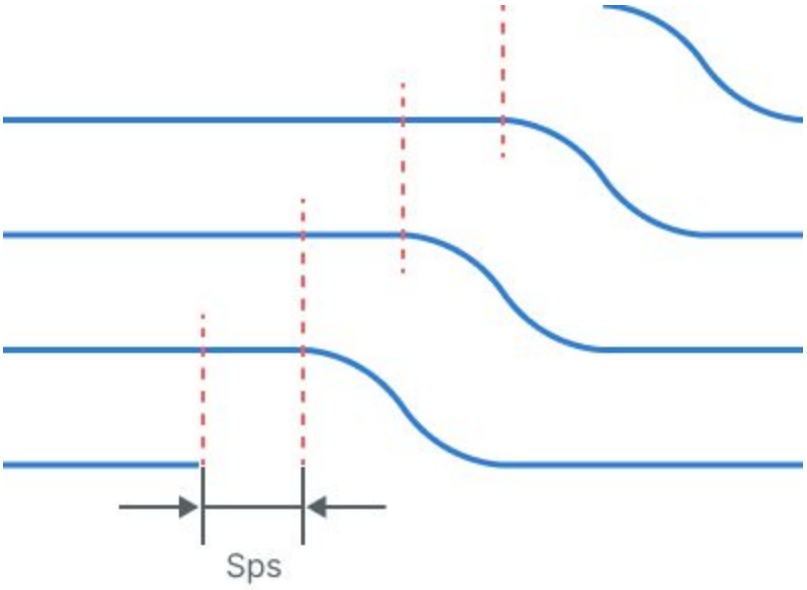
Application Area:

Defines the rules governing Entry/Exit links and links between work moves.

Start point shift (For additive operations).

▼ Click here to expand..
The start point shift relative to the previous layer allows to get a more even distribution of zones where the feed of material starts/stops and where the quality of the formed surface is usually lower.

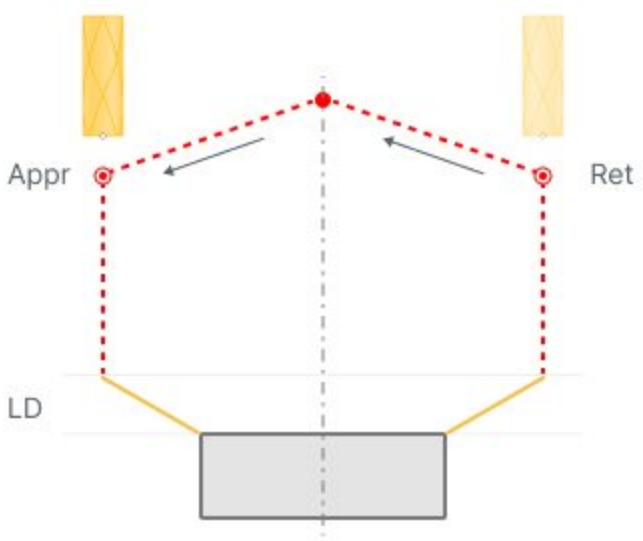
Note: For "Non-planar slicing" operation this option is available only if "Start points" parameter on the "Strategy" page is set to "Auto" or "Custom for first layers" mode.



Sps - Start point shift
Entry/Exit links. Defines the general rule of approach/return.

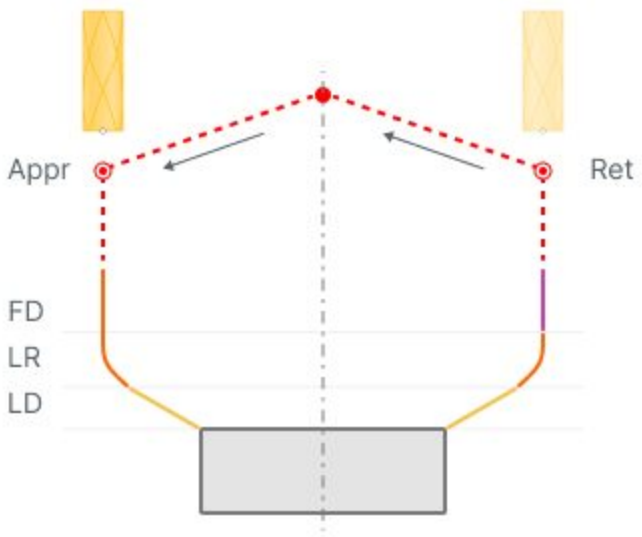
▼ Click here to expand..
Straight.

With this rule, the approach to the part is carried out: 1. Tool change position 2.Appr- Approach 3. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2.Ret - Return 3. Tool change position



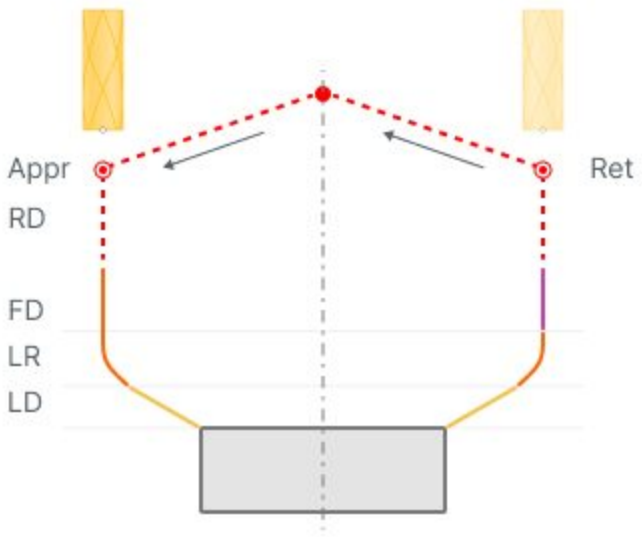
By Feed Distance.

With this rule, the approach to the part is carried out: 1. Tool change position 2.Appr- Approach 3. FD - Feed distance 4. LR - Linking Radius (depends on the type of Leads. If the Leads is located vertically, then as a rule, the arc is not built) 5. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2. LR - Linking Radius (depends on the type of Leads. If the Leads is located vertically, then as a rule, the arc is not built) 3.FD - Feed distance 4.Ret - Return 5. Tool change position



By Rapid Distance.

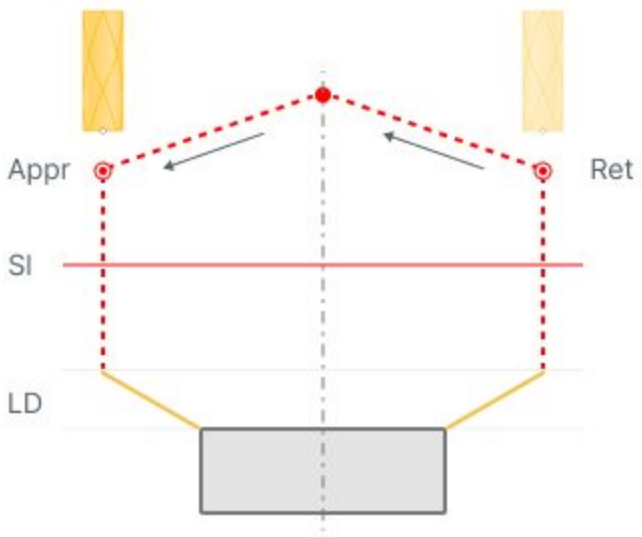
With this rule, the approach to the part is carried out: 1. Tool change position 2. Appr- Approach 3. RD - Rapid distance 4. FD - Feed distance 5. LR - Linking Radius (depends on the type of Leads. If the Leads is located vertically, then as a rule, the arc is not built) 6. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2. LR - Linking Radius (depends on the type of Leads. If the Leads is located vertically, then as a rule, the arc is not built) 3. FD - Feed distance 4. RD - Rapid distance 5. Ret - Return 6. Tool change position



On Safe Surface.

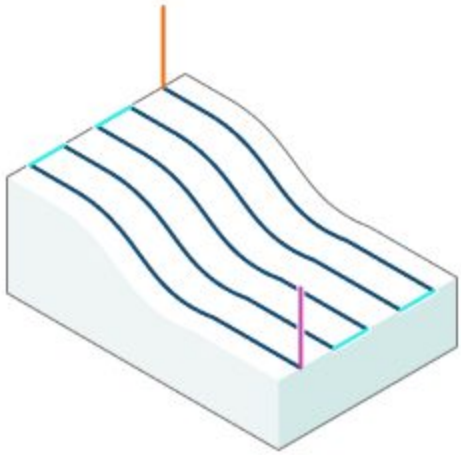
With this rule, the approach to the part is carried out: 1. Tool change position 2.Appr- Approach 3.SL - Safe level 4. FD - Feed distance 5.LR - Linking Radius (depends on the type of Leads. If the Leads is

located vertically, then as a rule, the arc is not built) 6. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2. LR - Linking Radius (depends on the type of Leads. If the Leads is located vertically, then as a rule, the arc is not built) 3. FD - Feed distance 4. SL - Safe level 5. Ret - Return 6. Tool change position

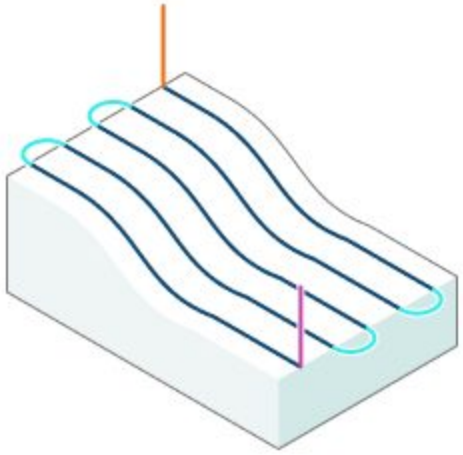


Short links/Long links. Defines the rule for transition between work moves.

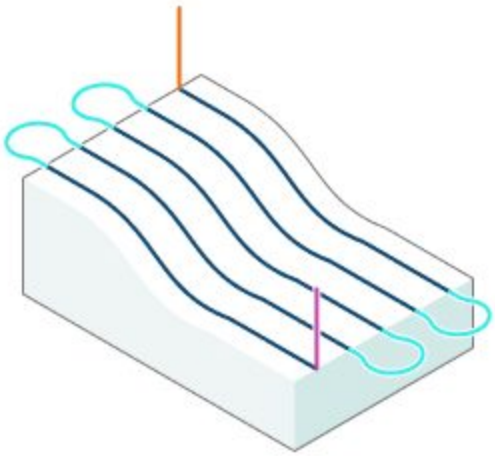
▼ Click here to expand..
Straight. Links are made by the shortest distance.



Smooth. Links are made by a smooth spline curve.

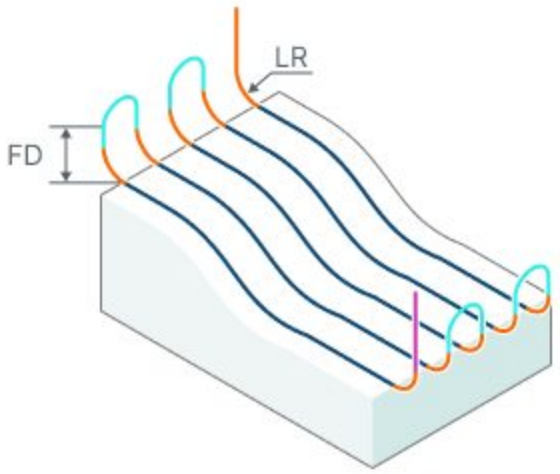


High Speed. Links are made by a smooth curve in such a way that the Link radius is nowhere exceeded.



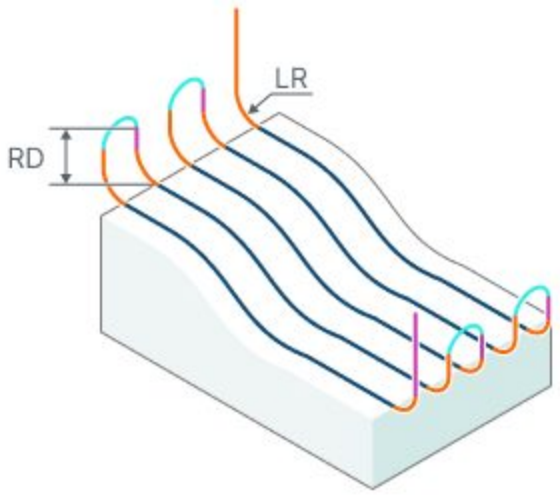
By Feed Distance.

The links made from from the Feed distance. Additionally transition curves may be smoothed by the Link radius. FD - Feed distance LR - Link radius



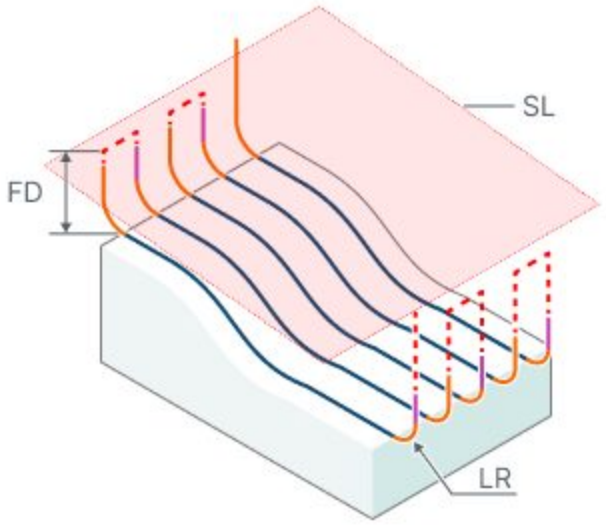
By Rapid Distance.

The links made from from the Linking Radius and Rapid distance. RD - Rapid distance LR - Linking Radius



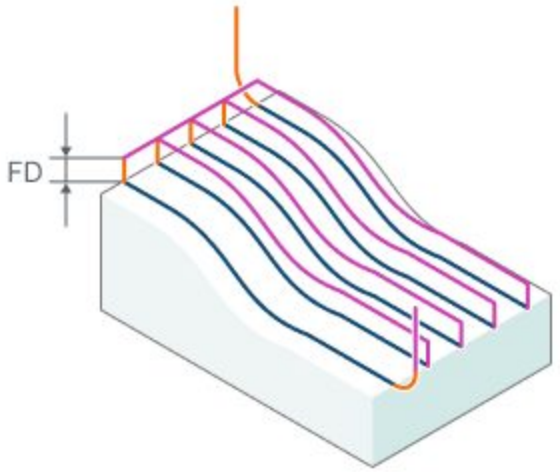
On Safe Surface.

The links made from from the Linking Radius + Feed distance to the Safe level. LR - Linking Radius FD - Feed distance SL - Safe level

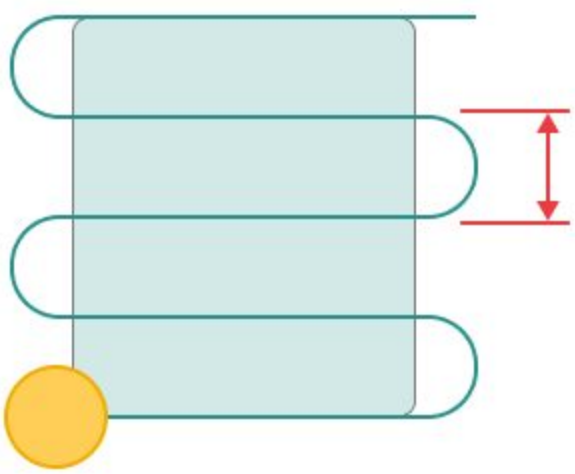


On Offset Surface.

Works as a displacement of the treated surface by a distance Feed distance. Works with milling type: Climb, Conventional. FD - Feed distance



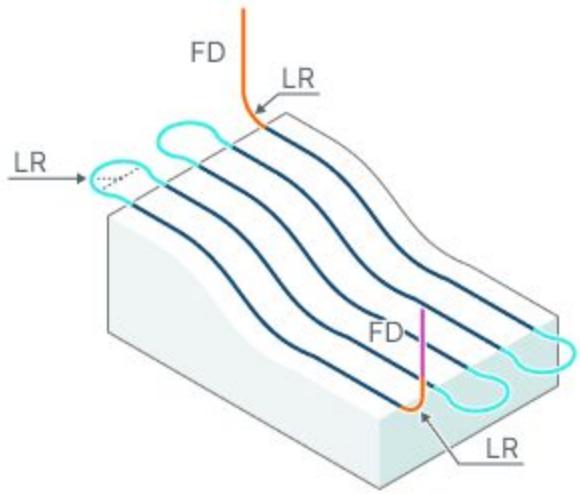
Short link max distance. Sets the maximum distance at which links are made by the short link type.



Linking Radius.

▼ Click here to expand..
Link radius sets the radius of smoothing for

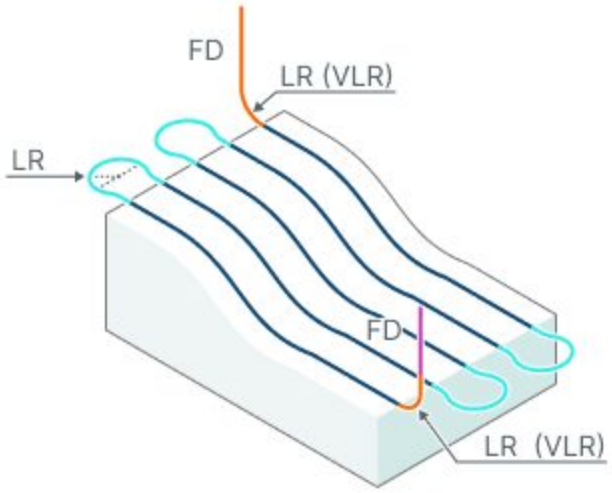
- Highspeed links - Feed distance links (depends on the type of Leads. If the Leads is located vertically, then as a rule, the arc is not built)



FD - Feed distance LR - Link radius

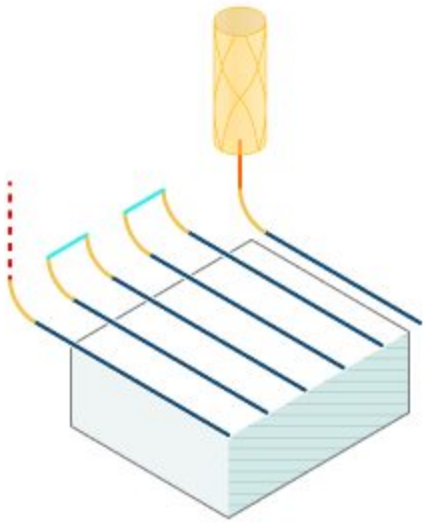
Vertical Lead Radius.

▼ Click here to expand..
The Vertical Lead Radius parameter, in combination with the Linking Radius, expands the connection zone to include a vertical displacement, enabling "Frog Jump" transitions.



FD - Feed distance LR - Link radius VLR - Vertical Lead Radius

Links for 3D operations



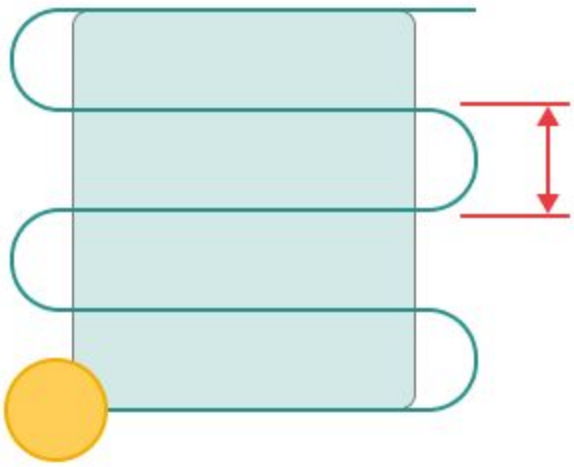
Application Area:

This group defines the settings for cutting in or transitions between areas.

Short link max distance.

▼ Click here to expand..

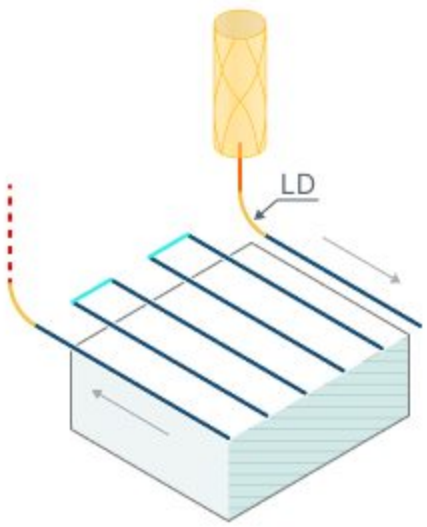
Sets the maximum distance at which links are made by the short link type.



Short link type.

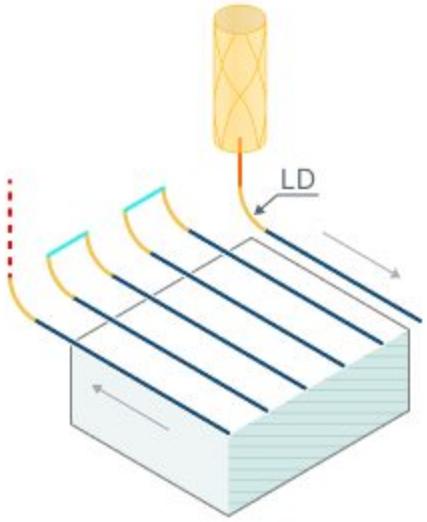
▼ Click here to expand..

Direct. Links are made by the shortest distance.



LD - Leads

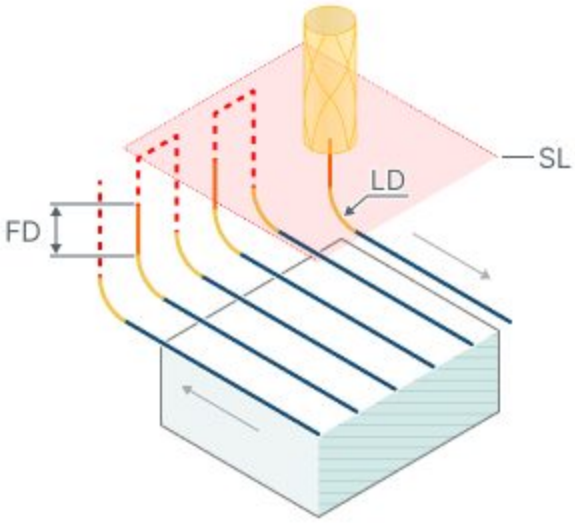
Retract-engage. Links are set by Leads and shortest distance



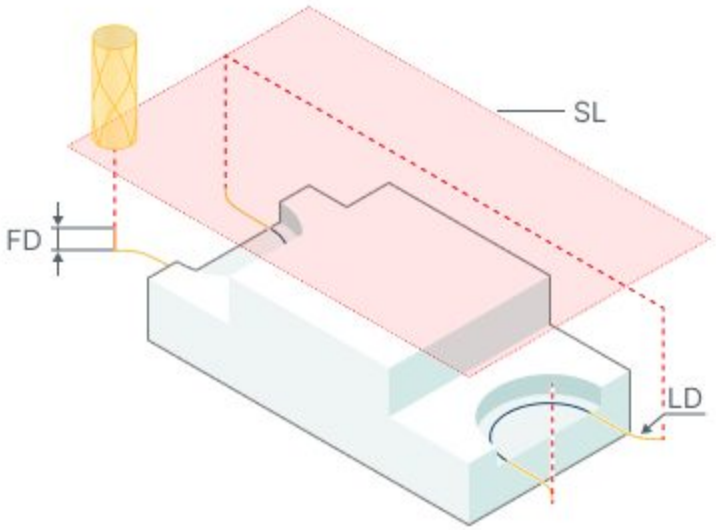
LD - Leads

Via safe plane.

Links on the return: 1.Leads (LD) 2.Safe level (SL) Movement on the plane of safety Links on the approach: 1.Safe level (SL) 2. Feed distance (level) (FD) 3. Leads (LD)


SL
LD
FD

Links for countouring operations

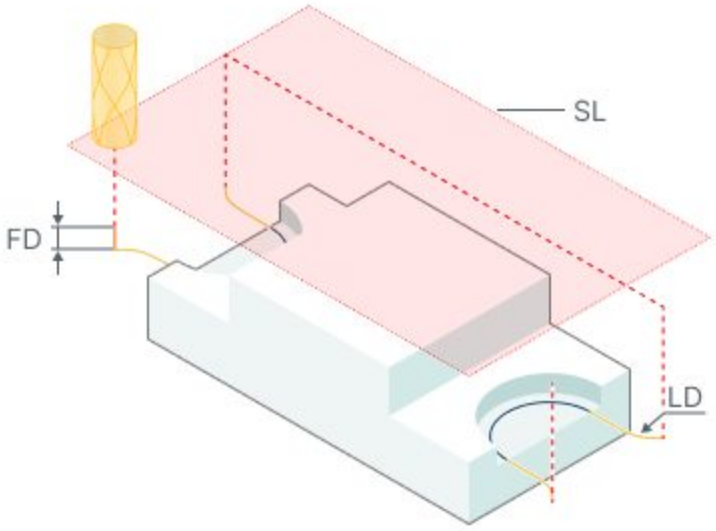


Application Area:

This group defines the settings for cutting in or transitions between areas.

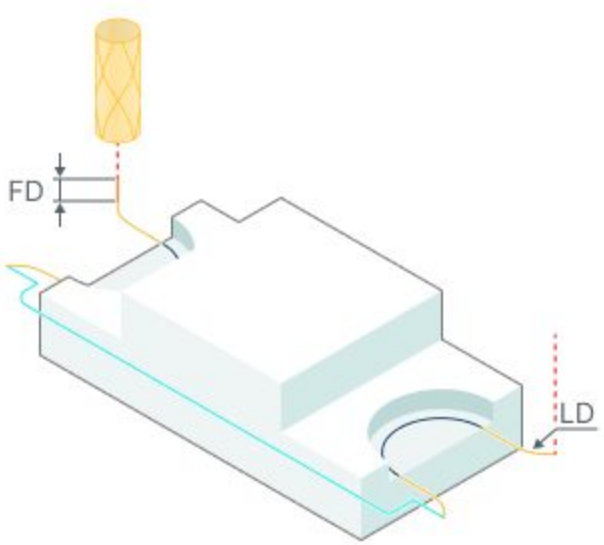
Transitions.

▼ Click here to expand..
Via safe plane. The transition is carried out through a safe level



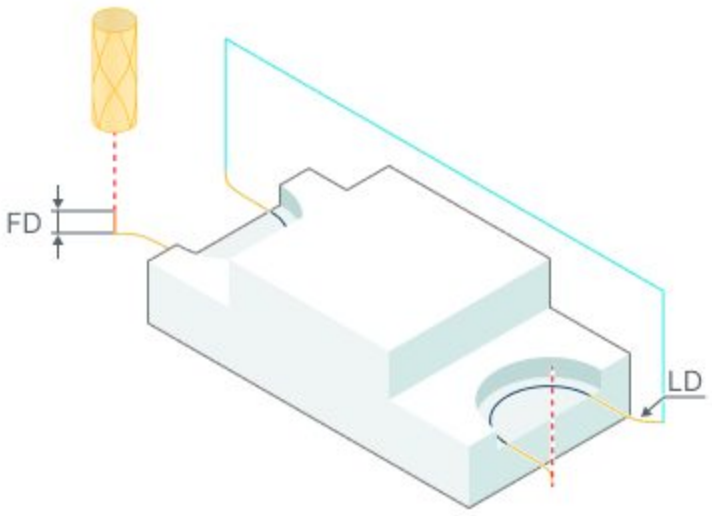
FD - Feed distance SL - Safe level LD - Leads

Around workpiece. The transition is carried out at the level of the working path. For its operation, it is necessary to enable Check workpiece in the Parameters tab. The algorithm will build the shortest tool path without crashing into the workpiece.



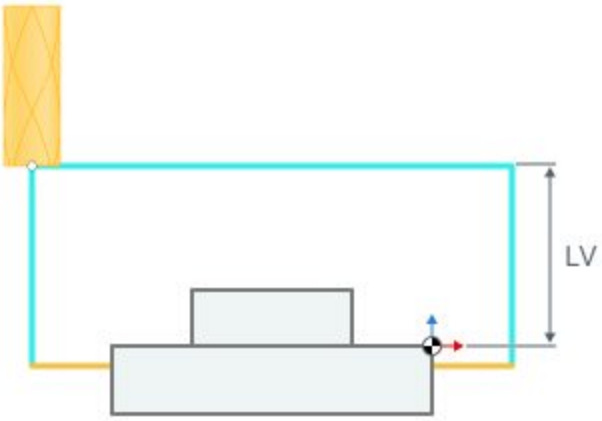
FD - Feed distance LD - Leads

By level. The transition is carried out through a level that is measured from the Workpiece coordinate system.



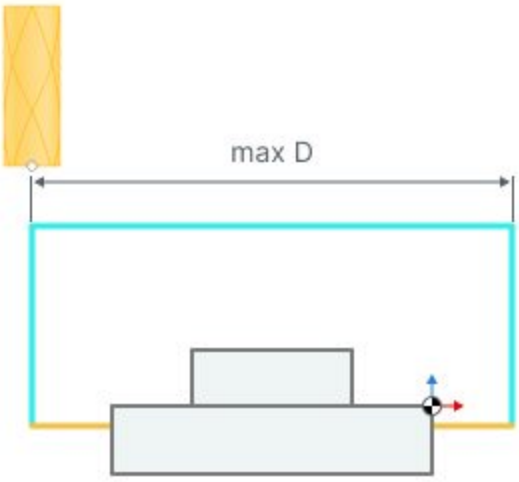
FD - Feed distance LD - Leads

Level. Level that is measured from the Workpiece coordinate system



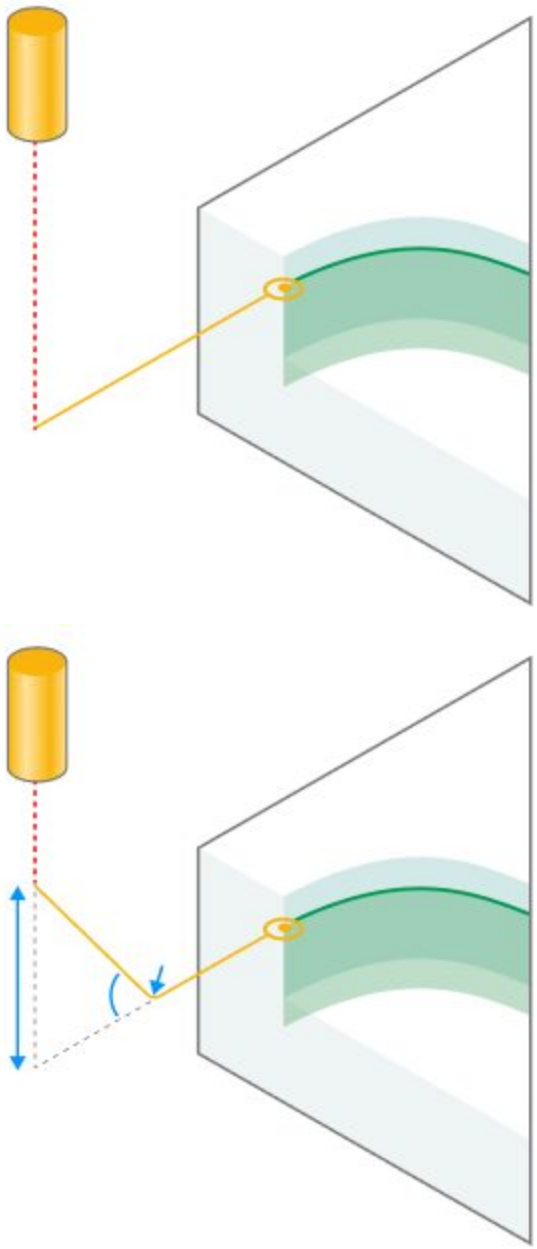
LV - Level

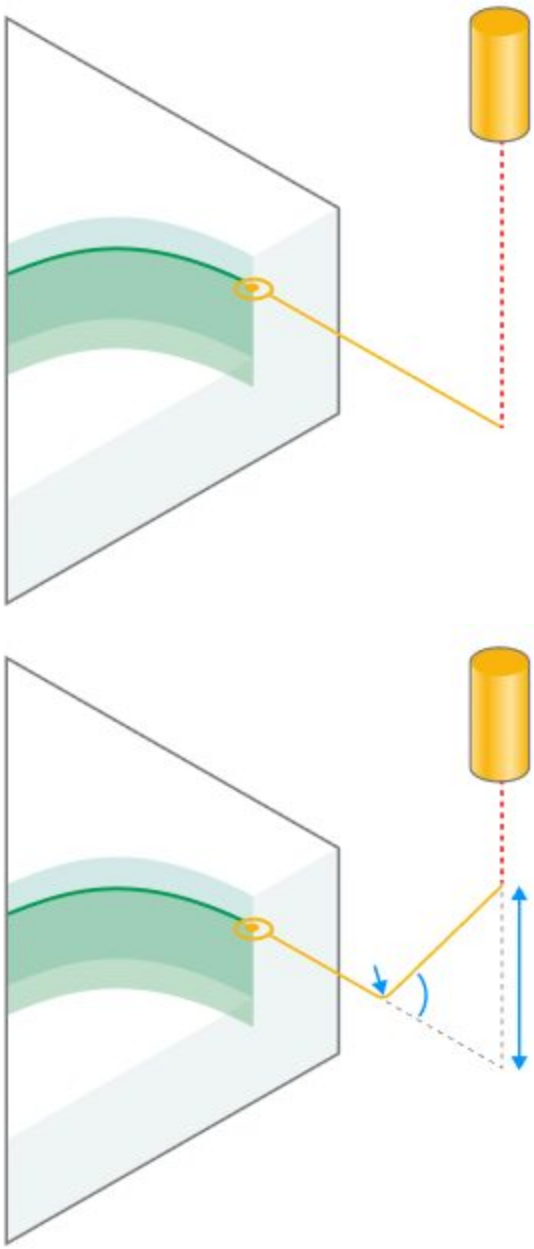
Maximal distance. Defines the distance at which the transition through the level is performed. If the distance is greater than the value in the parameter, then the transition is performed through the safety level.



max D - Maximum distance
Plunge on Lead In/ Rise on Lead Out.

▼ Click here to expand..
Allows you to set up the plunge/rise method.





Height. Determines the height for plunge/rise at an angle.

Ramp angle. Determines the angle of plunge/rise.

Smooth arc radius. Allows you to smooth the trajectory through the radius when plunge/rise.

Links in cladding operations

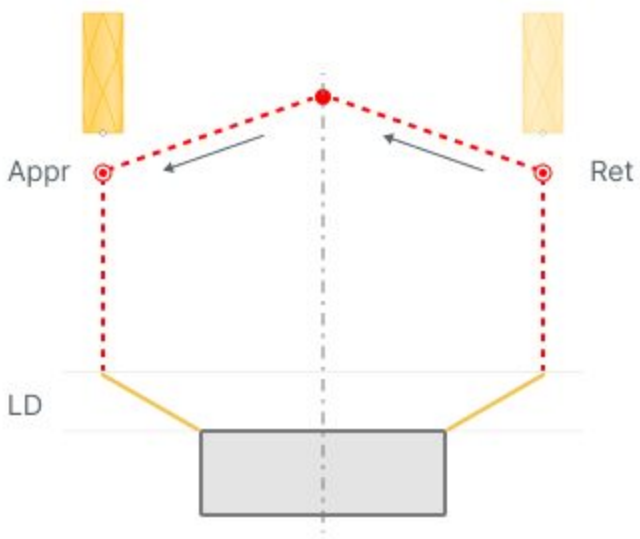
Application Area:

This group defines the settings for cutting in or transitions between areas.

First links / Last links. Defines the general rule of approach/return.

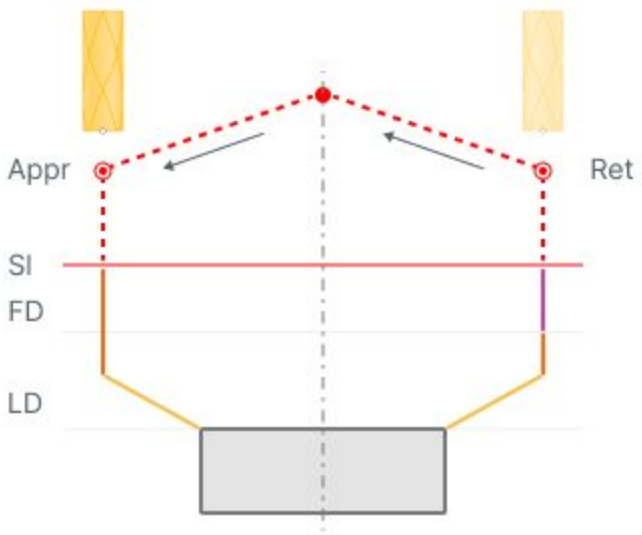
▼ Click here to expand..
Straight.

With this rule, the approach to the part is carried out: 1. Tool change position 2.Appr- Approach 3. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2.Ret - Return 3. Tool change position



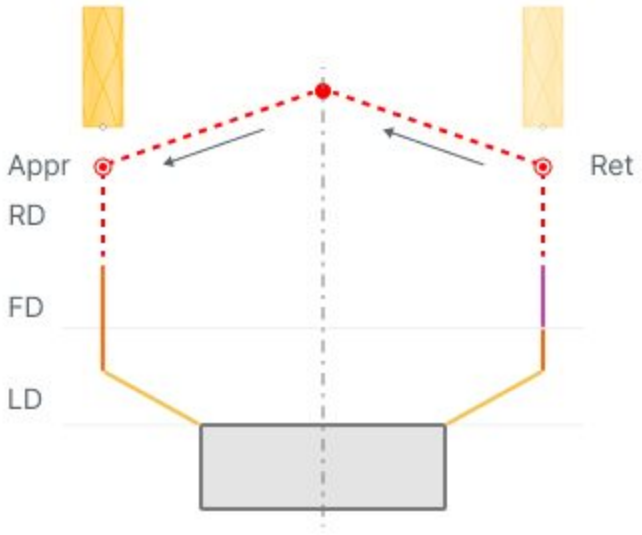
On Safe Surface.

With this rule, the approach to the part is carried out: 1. Tool change position 2.Appr- Approach 3.SL - Safe level 4. FD - Feed distance 5. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2. FD - Feed distance 3. SL - Safe level 4. Ret - Return 5. Tool change position



By Rapid Distance.

With this rule, the approach to the part is carried out: 1. Tool change position 2. Appr- Approach 3. RD - Rapid distance 4. FD - Feed distance 5. LD - Leads (If they are enabled) The returnl is carried out in the reverse order: 1. LD - Leads (If they are enabled) 2. FD - Feed distance 3. RD - Rapid distance 4. Ret - Return 5. Tool change position



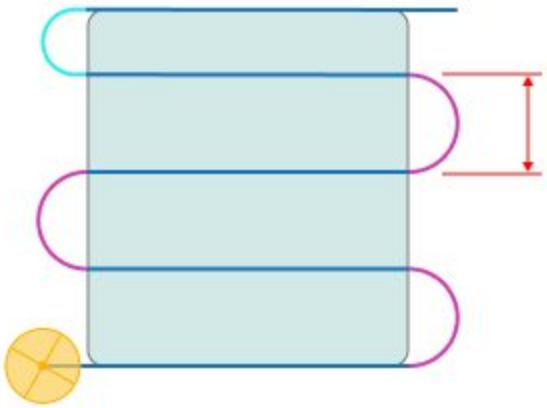
Neighbor links.

▼ Click here to expand..
Continuous.

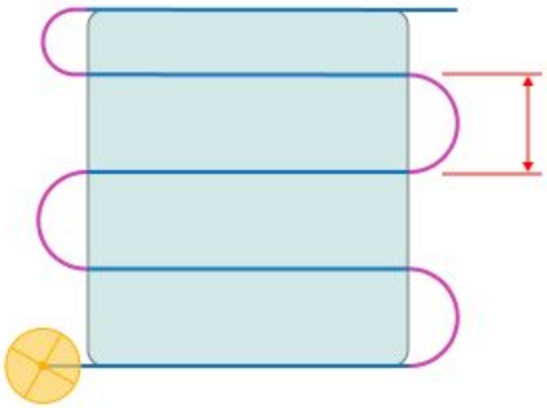
Transitions between neighboring passes are made continuously along the part at the Short link feed, without retracting the tool. Combined with Effective feeds set to all non-rapid, this produces a continuous

cladding bead. Applied only to transitions shorter than Neighbor link max distance.

> Note
>
> For Area cladding and Cladding 3D operations in the Parallel strategy the transition follows the equidistant of the outer contour; in the Offset strategy it is built as the shortest direct move.



As short link. Transitions between neighboring passes are executed as regular short links, according to the Short links setting.

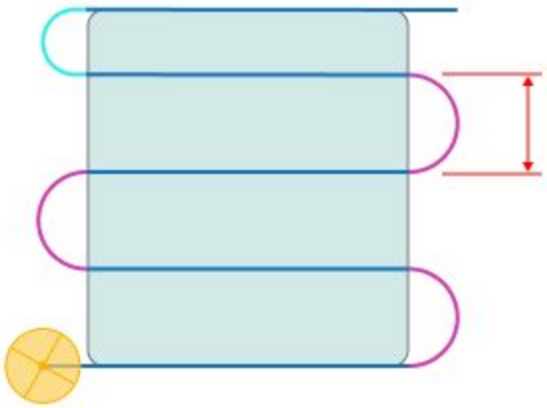


Neighbor link max distance.

The maximum distance between the end points of two neighboring passes at which the transition is treated as a continuous neighbor link. If the distance is greater, the transition is built as a Short link or

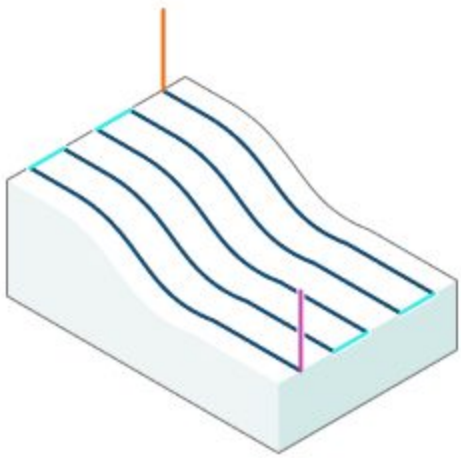
Long link instead.

> Note
>
> Short links operate in the gap between this value and the Short link max distance. If this value reaches the Short link max distance, that gap closes and every short transition becomes a neighbor link.

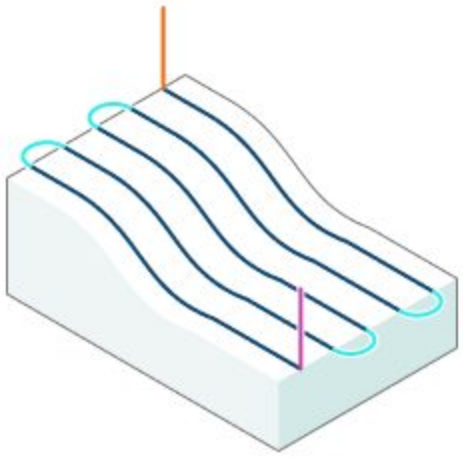


Short links / Long links.

▼ Click here to expand..
Straight. Links are made by the shortest distance.

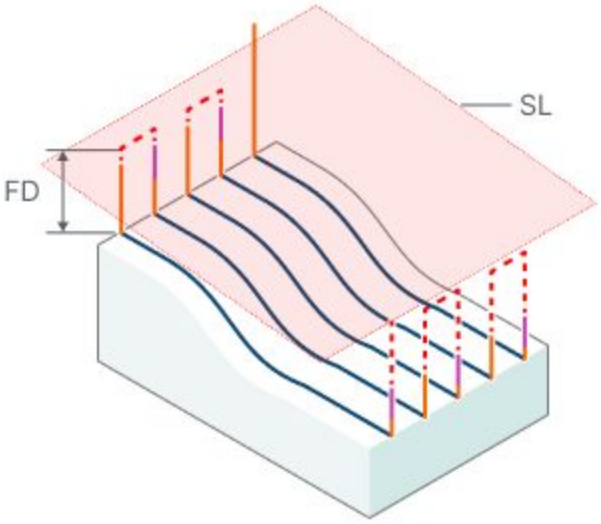


Smooth. Smooth Bezier spline transition between passes.



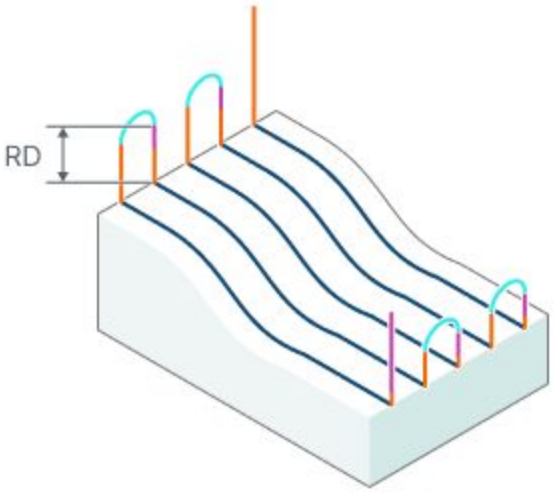
On Safe Surface.

The links made from from the Feed distance to the Safe level. FD - Feed distance SL - Safe level

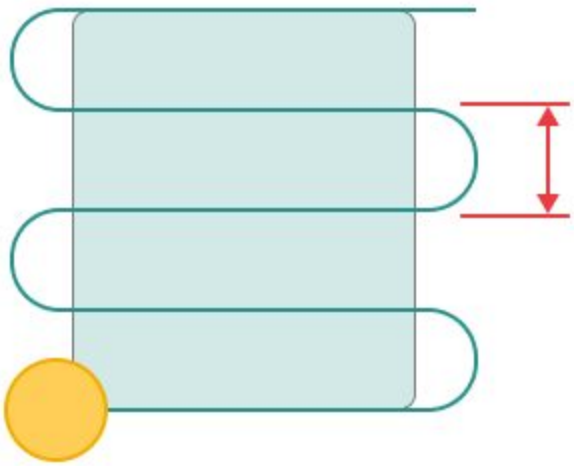


By Rapid Distance.

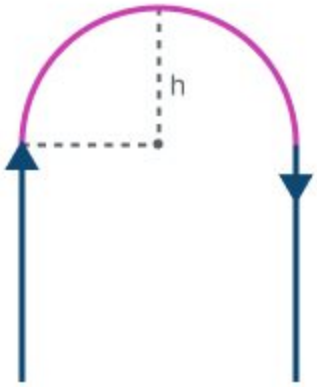
The links made from from the Rapid distance. RD - Rapid distance



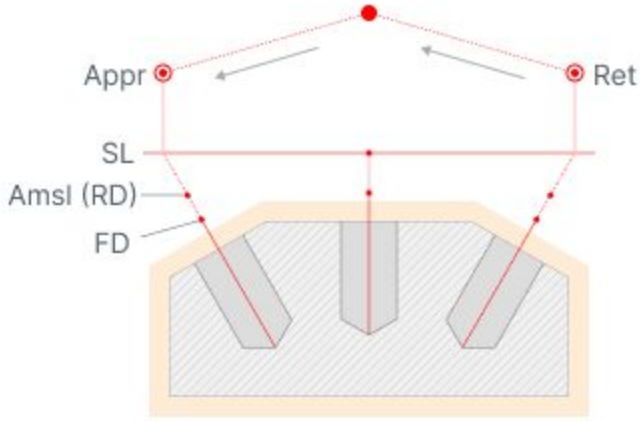
Short link max distance. Sets the maximum distance at which links are made by the short link type.



Smooth link height. Height of the link arc above the straight chord connecting two neighboring passes. The specified value is exact for the reference case — the ideal zigzag. In other configurations (Parallel forward/backward, Offset, etc.) the actual arc height may differ.


h

Links in drilling operations

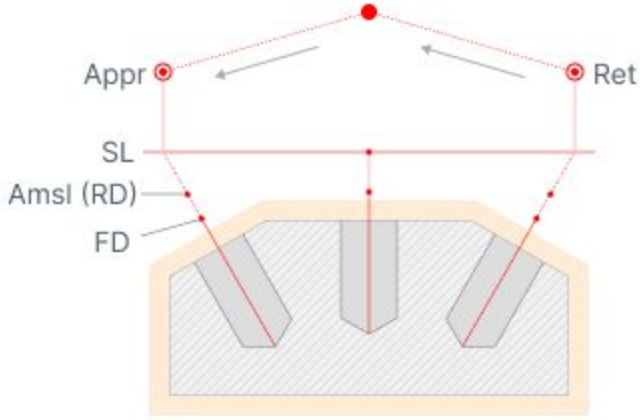


Application Area:

The parameters of this group detail the final point for the approach/return movements and specify the technique for auxiliary movements when transitioning between holes.

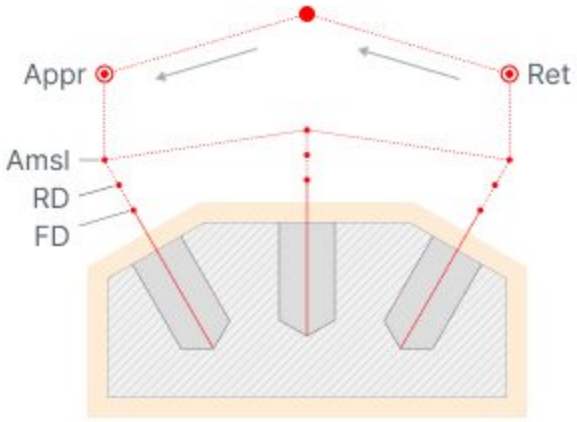
First Link Type. Defines the method for bringing the tool to the first processing hole.

▼ Click here to expand..
Via Safe Surface. The toolpath continues until the hole axis intersects with the specified safe surface (plane, cylinder or sphere).



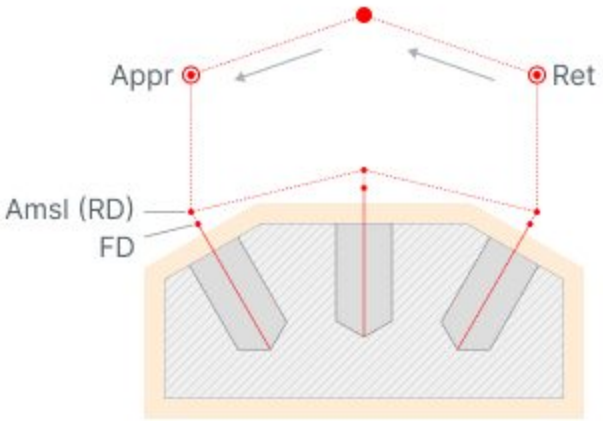
With this rule, the approach to the first hole is carried out: 1. Tool change position 2.Appr- Approach 3.SL - Safe level 4. Amsl - Air move safety level. If enable Start cycles at air move safety level, else RD - Rapid distance 5.FD - Feed distance Retraction to the tool change point is carried out through Ret -Return

Via Air Move Safety Level. The toolpath continues along the hole axis up to the air move safety level.



With this rule, the approach to the first hole is carried out: 1. Tool change position 2.Appr- Approach 3.Amsl - Air move safety level 4.RD - Rapid distance (If disable Start cycles at air move safety level) 5.FD - Feed distance Retraction to the tool change point is carried out through Ret -Return

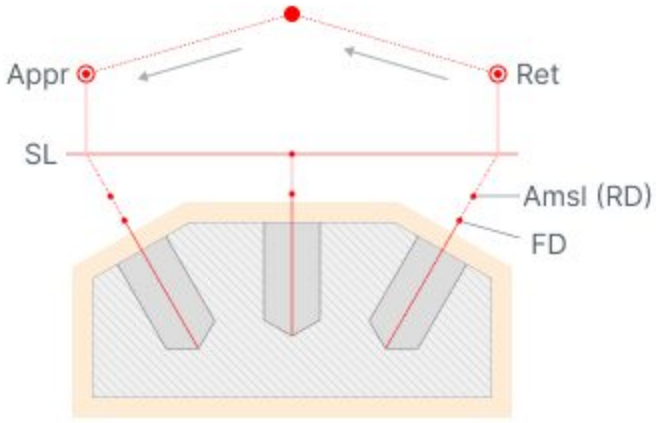
Straight. Straight transition between holes' first cycle points



With this rule, the approach to the first hole is carried out: 1. Tool change position 2.Appr- Approach 3.Amsl- Air move safety level. If enable Start cycles at air move safety level, else RD - Rapid distance

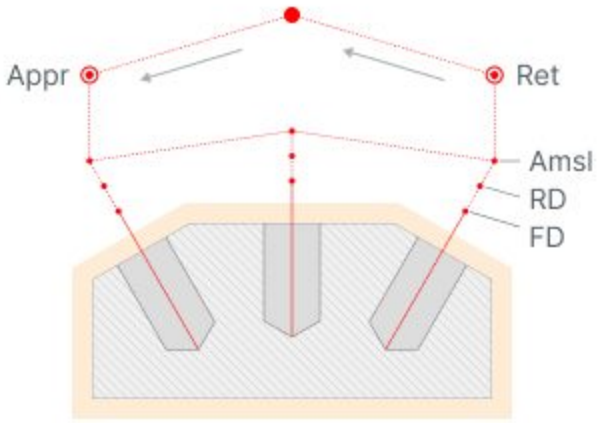
4.FD - Feed distance Retraction to the tool change point is carried out through Ret -Return
Last Link Type. Specifies how the tool is retracted from the last processed hole.

▼ Click here to expand..
Via Safe Surface. The toolpath continues until the hole axis intersects with the specified safe surface (plane, cylinder or sphere).



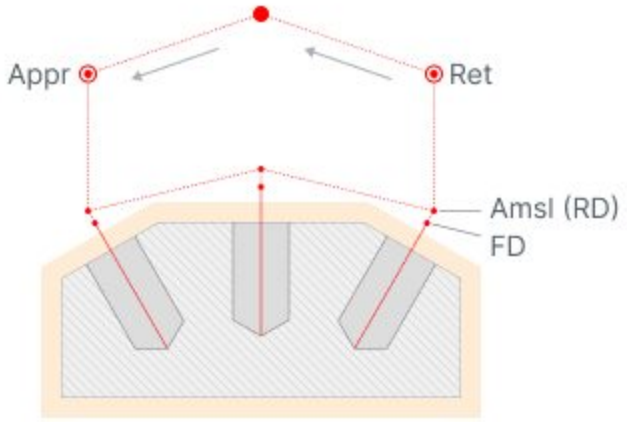
With this rule, the approach to the last hole is carried out: 1.SL - Safe level 2. Amsl - Air move safety level. If enable Start cycles at air move safety level, else RD - Rapid distance 3.FD - Feed distance Retraction to the tool change point is carried out through Ret -Return

Via Air Move Safety Level. The toolpath continues along the hole axis up to the air move safety level.



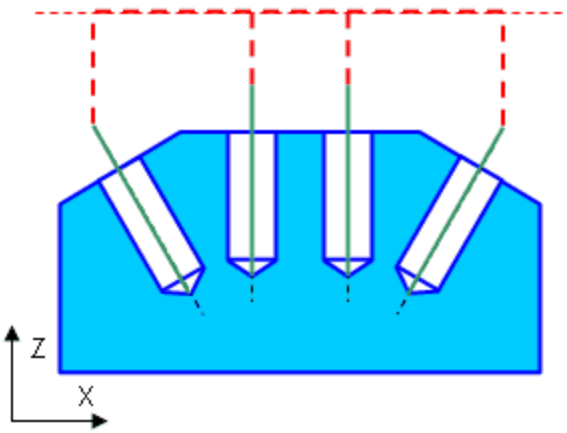
With this rule, the approach to the last hole is carried out: 1.Amsl - Air move safety level 2.RD - Rapid distance (If disable Start cycles at air move safety level) 3.FD - Feed distance Retraction to the tool change point is carried out through Ret -Return

Straight. Straight transition between holes' first cycle points



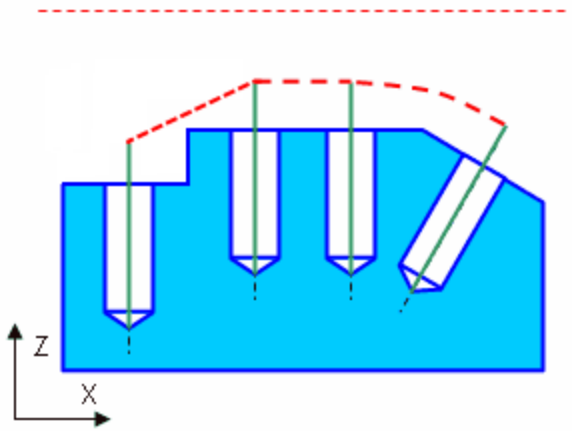
With this rule, the approach to the last hole is carried out: 1.Amsl- Air move safety level. If enable Start cycles at air move safety level, else RD - Rapid distance 2.FD - Feed distance Retraction to the tool change point is carried out through Ret -Return
Long Link Type. Defines the method for transitioning the tool between consecutive processing holes when the transition length exceeds the value set in the Short Link Max Distance parameter.

▼ Click here to expand..
Via Air Move Safety Level. The tool follows the shortest path to the next hole, arriving at the coordinate specified by the Air Move Safety Level parameter, and then moves towards the Feed Distance along the tool's normal.

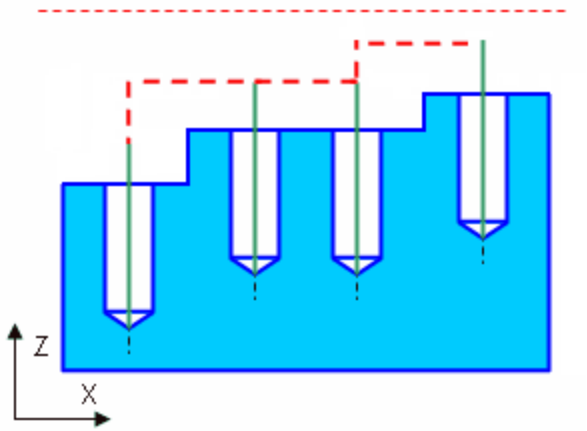


Via Safe Surface. The tool moves along the Safe Surface to the next hole, arriving at the coordinate set by the Safe Surface Level parameter, then proceeds toward the Feed Distance in the direction of its normal.

Straight. The tool follows the shortest route to the next hole, arriving at the coordinate specified by the Rapid Distance parameter, then moves toward the Feed Distance along the tool's normal direction.



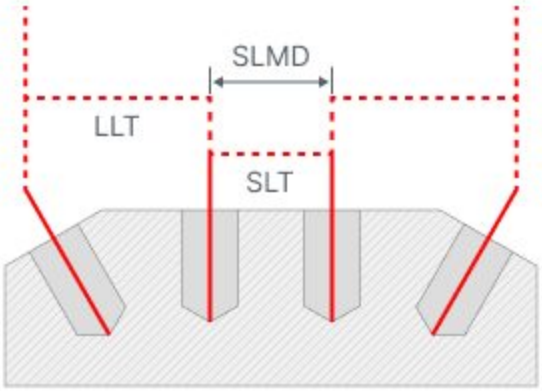
Orthogonal. If the Rapid Distance at the next hole exceeds that of the current hole, the tool retracts along its normal to the next hole's Rapid Distance level, then follows the shortest path to the next hole, and then lowers along its normal to the Feed Distance.

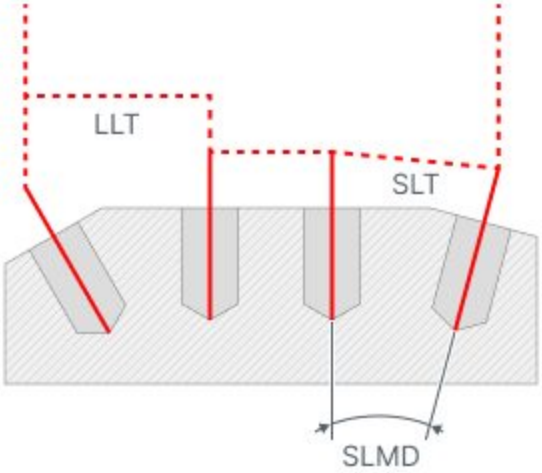


Short Link Type. Defines the method for transitioning the tool between consecutive processing holes when the transition length is smaller than the value set in the Short Link Max Distance parameter. This option uses the same parameters as the Long Link Type.

Short link max angle / Short link max distance.

▼ Click here to expand..
Using the Short link max distance and the Short link max angle parameters you can divide the long and short transition cases to be possible to make diferent type of links. If Short link max distance and Short link max angle less than the distance between the holes, then the transition is performed as Long link type. Otherwise it will be executed on Short link type





SLMD - Short link max distance

SLMA - Short link max angle

SLT - Short link type

LLT - Long link type

Leads

Leads define the engage and retract motions at the beginning and end of working passes. These settings control how the tool enters and leaves the machining path, including the shape, direction, and dimensions of the additional motions.

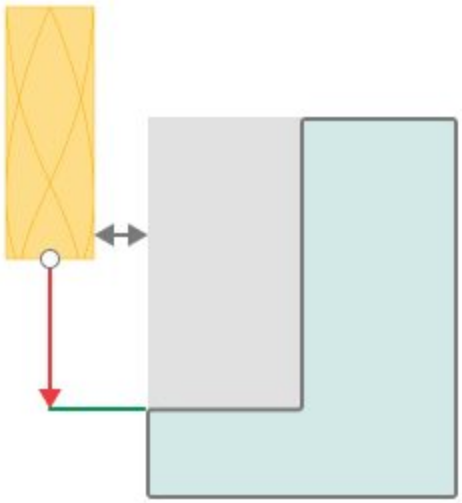
This section describes the lead options available for different operation types, including roughing waterline, face milling, 3D and 5D machining, contouring, and cladding. Refer to the topic for the required operation type when choosing how to connect working passes to approach, retract, and linking motions.

Leads based on the Roughing waterline

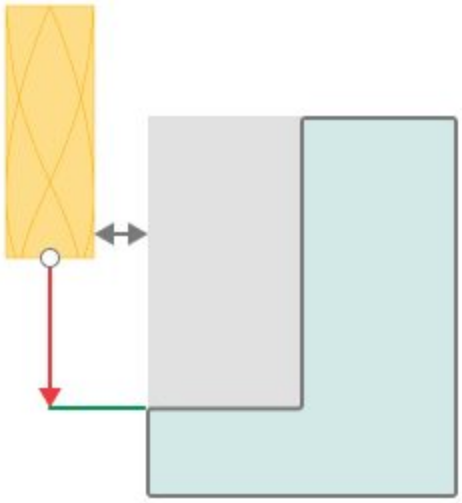
Application Area:

Allows you to add additional engage and retract to the working moves using their feed.

Safe distance. The distance from the workpiece when switch from/to rapid feed or long link feed.



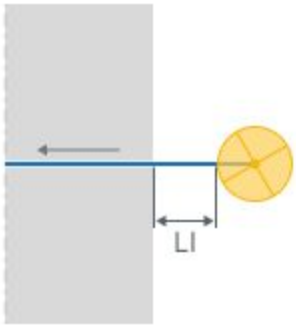
Leads based on the Face Milling



Application Area:

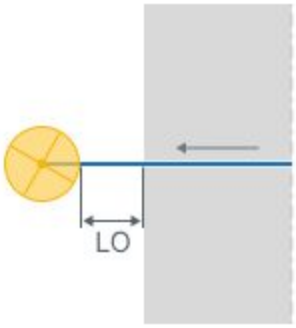
Allows you to add additional engage and retract to the working moves.

Lead in. The distance from the workpiece at which a work pass starts. Used to avoid vertical plunging of the tool into material.



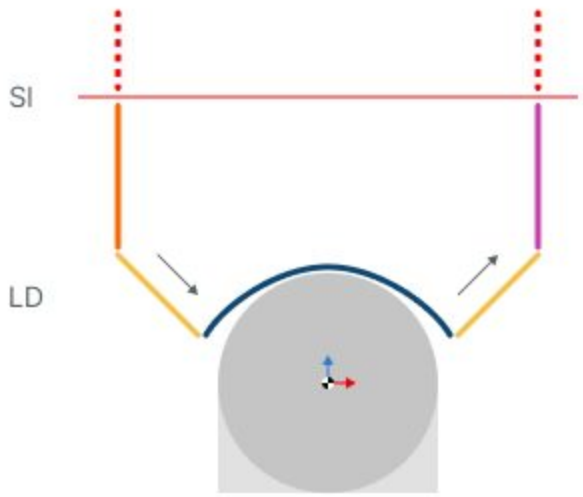
LI - Lead in

Lead out. The distance from the workpiece at which a work pass starts. Used to avoid vertical plunging of the tool into material.



LO - Lead out

Leads for 3D operations



SI - Safe level

LD - Leads

Application Area:

Allows you to add additional engage and retract to the working moves using their feed.

Engage.

▼ Click here to expand..

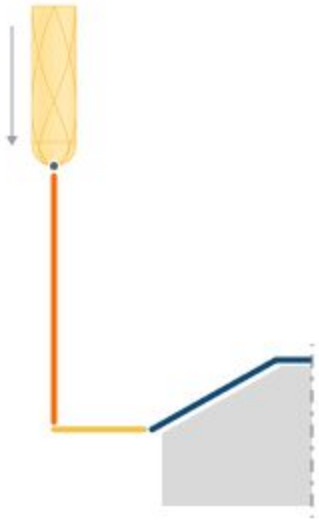
Off. The Engage and Retract toolpath parts is not generated.



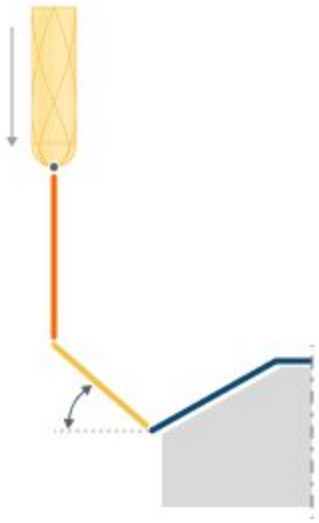
Vertical. The engage is performed vertical to the first point of the work pass.



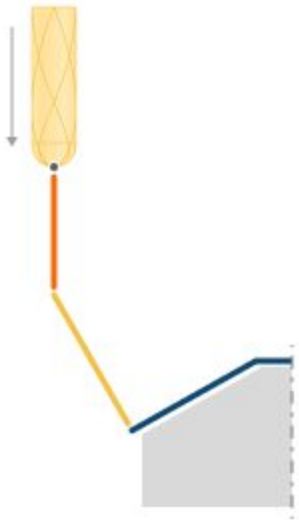
Horizontal. The engage is performed perpendicular to the tool axis to the first point of the work pass



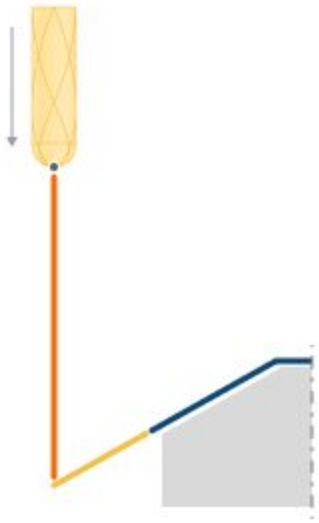
Angle to Z Axis. The engage is performed by angle to vertical to the first machining point



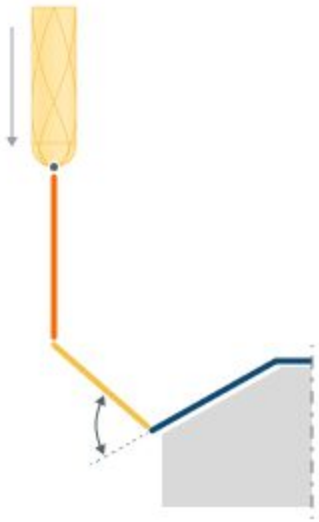
By Normal. The engage is performed along the normal to surface at the first point of the work pass



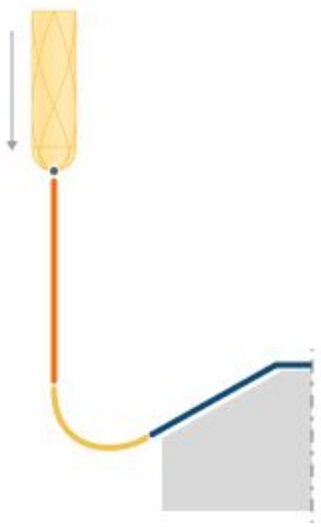
By Tangent. The engage is performed tangentially to the first machining point



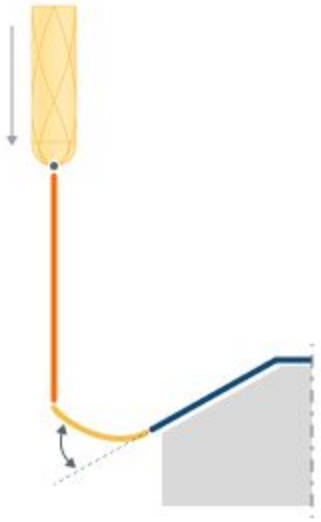
Angle to Tangent. The engage is performed by angle to tangent to the first machining point



By Arc. The system adds an arc to the first point on the curve using the defined radius.

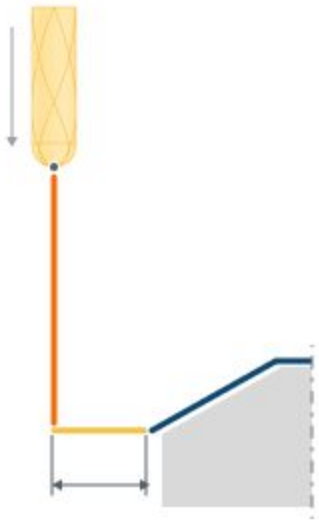


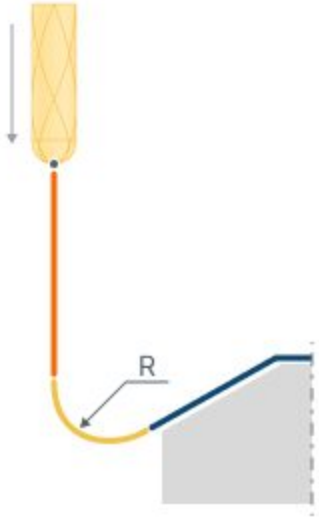
By Arc (Angle). The system adds an arc to the first point on the curve using the defined radius. The arc angle is constructed from the tangent of the work pass.



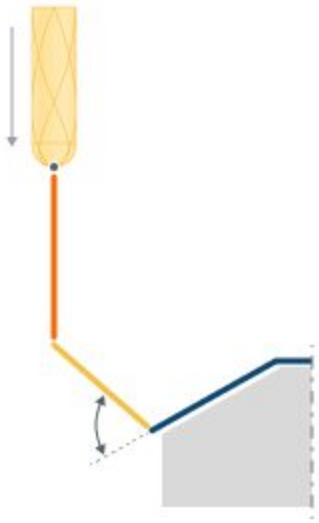
Depending on the engage, it becomes possible to control additional parameters:

Length (Radius). Changes the length





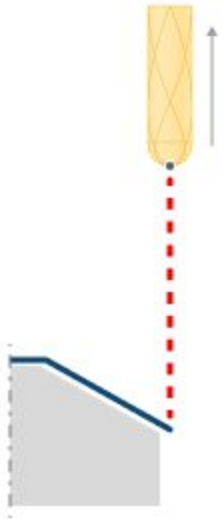
Angle. Changes the vertical angle



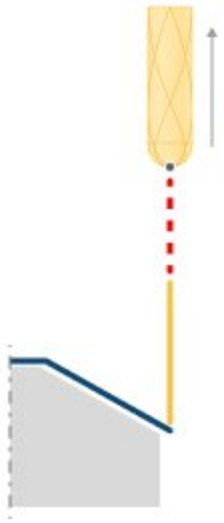
Retract.

▼ Click here to expand..

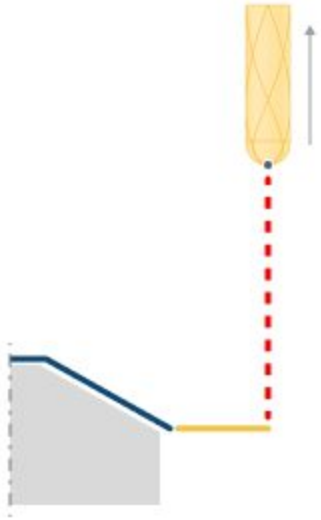
Off. The Engage and Retract toolpath parts is not generated.



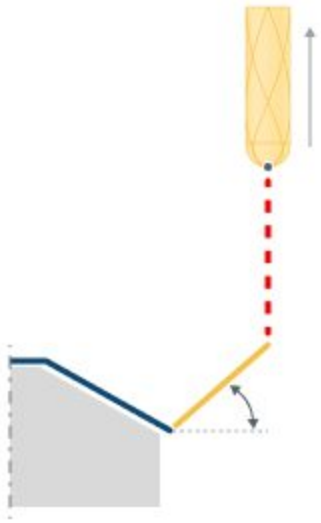
Vertical. The retract is performed vertical to the last point of the work pass.



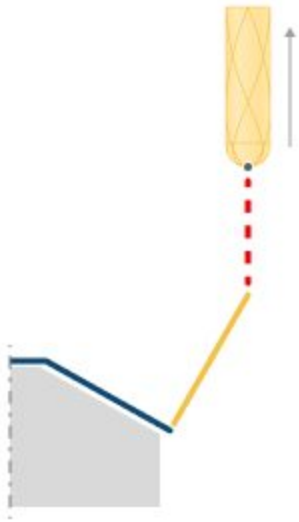
Horizontal. The retract is performed perpendicular to the tool axis to the last point of the work pass



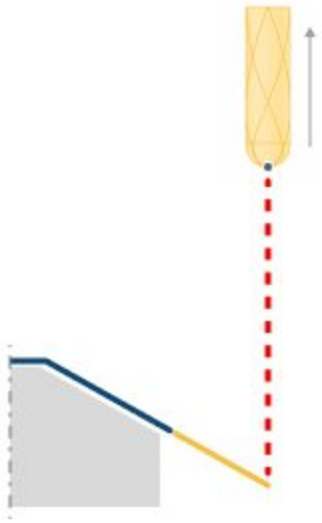
Angle to Z Axis. The retract is performed by angle to vertical to the last machining point



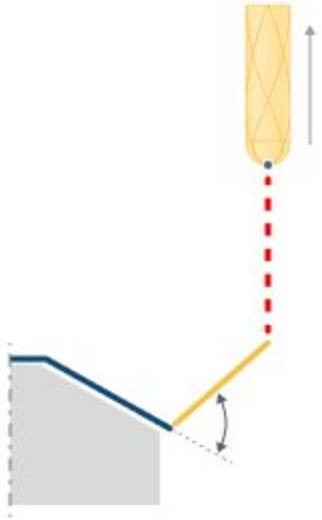
By Normal. The retract is performed along the normal to surface at the last point of the work pass



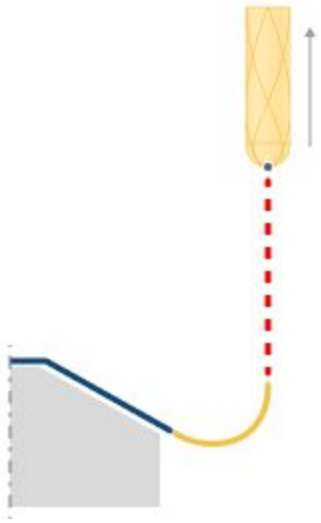
By Tangent. The retract is performed tangentially to the last machining point



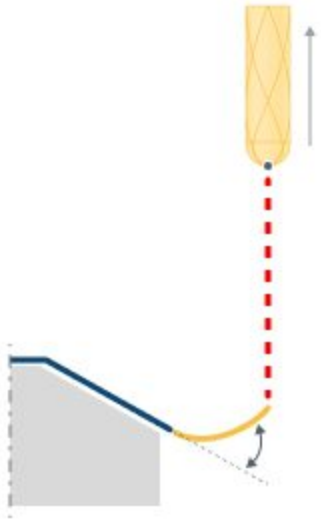
Angle to Tangent. The retract is performed by angle to tangent to the last machining point



By Arc. The system adds an arc to the last point on the curve using the defined radius.

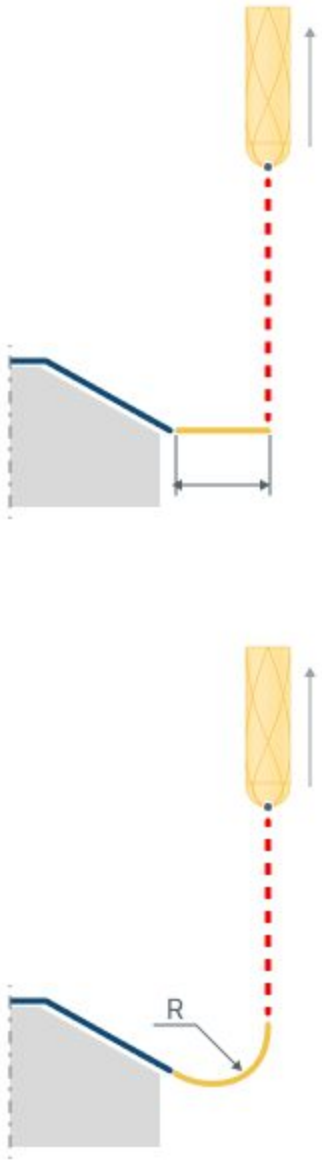


By Arc (Angle). The system adds an arc to the last point on the curve using the defined radius. The arc angle is constructed from the tangent of the work pass.

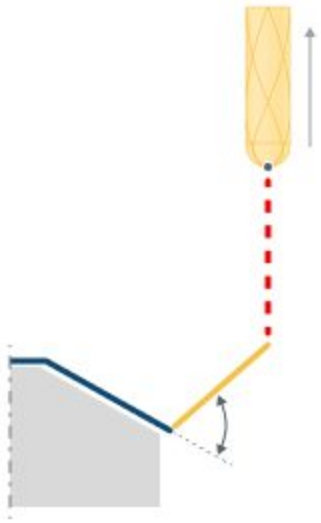


Depending on the engage, it becomes possible to control additional parameters:

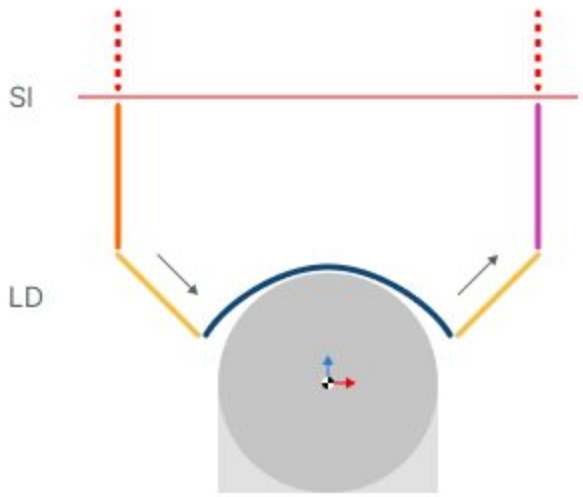
Length (Radius). Changes the length



Angle. Changes the vertical angle



Leads for 3D/5D advanced operations



SI - Safe level

LD - Leads

Application Area:

Allows you to add additional engage and retract to the working moves using their feed.

Lead from hole center (available for the chamfer operation).

▼ Click here to expand..
Enable: The lead is formed from the center of the hole.

Disable: The lead is formed according to the Lead in/out for closed curves or Lead in/out for open curves rule.

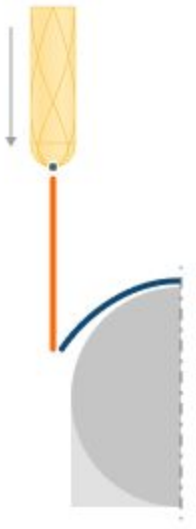




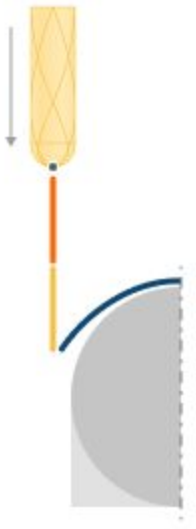
Engage.

▼ Click here to expand..

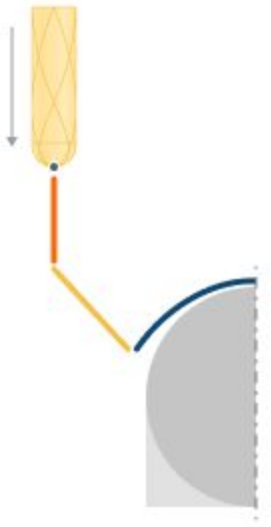
Off. The Engage and Retract toolpath parts is not generated.



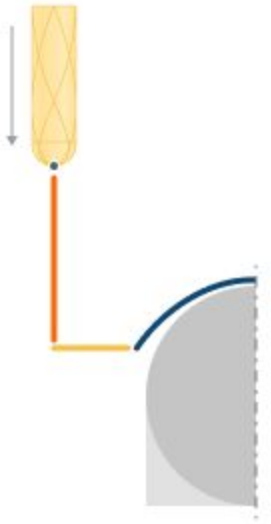
By Vertical Normal. The engage is performed along the tool axis to the first point of the work pass.



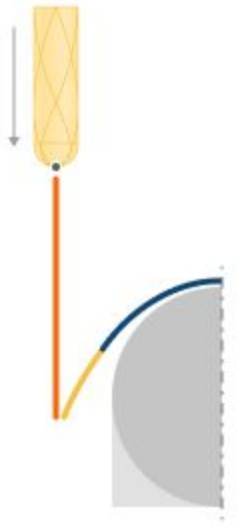
By Surface Normal. The engage is performed along the normal to surface at the first point of the work pass



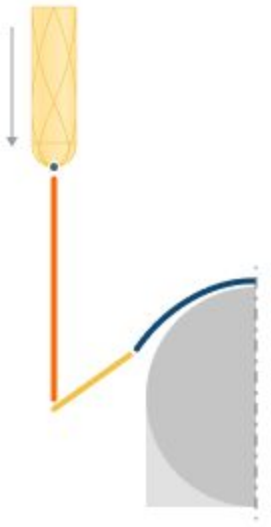
By Horizontal Normal. The engage is performed perpendicular to the tool axis to the first point of the work pass



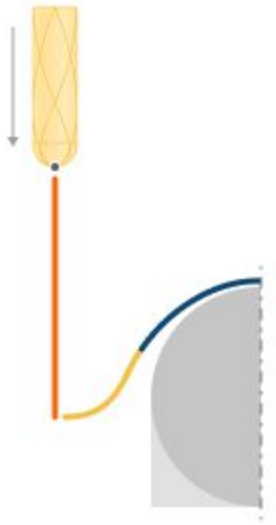
By Tangent. The engage is performed tangentially to the first machining point.



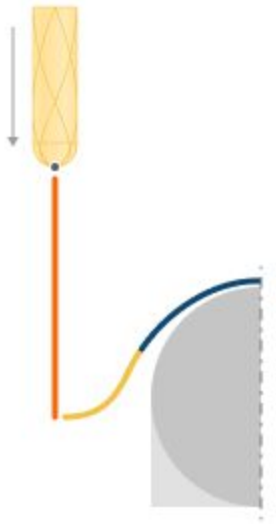
By Line. The engage is performed by angle to tangent to the first machining point.



By Vertical Arc. The system adds an arc to the first point on the curve using the defined radius. The arc lies in the vertical plane and is tangent with the first toolpath applied to the contour. The angle of the curve is defined by user. The tool plunge is performed at the vertical end of the arc, and then moves along the arc and then onto the work pass.



By Normal Arc. The system adds an arc to the first point on the curve using the defined radius. The arc lies in the vertical plane and is tangent with the first toolpath applied to the contour. The angle of the curve is calculated so that the tangent on the other side of the arc is vertical. The tool plunge is performed at the vertical end of the arc, and then moves along the arc and then onto the work pass.

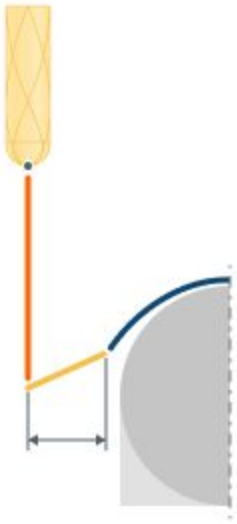


By Horizontal Arc. Used when constructing a trajectory in the horizontal direction. The system adds a tangent with an angle in the horizontal direction.

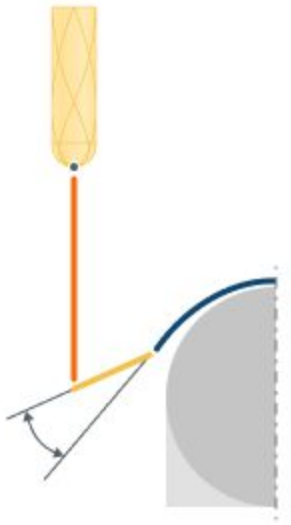


Depending on the engage, it becomes possible to control additional parameters:

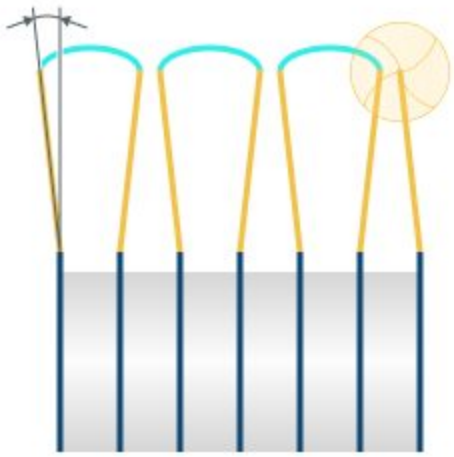
Length (Radius). Changes the length



Angle. Changes the vertical angle



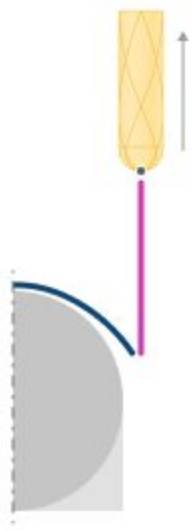
Plane angle. Changes the horizontal angle



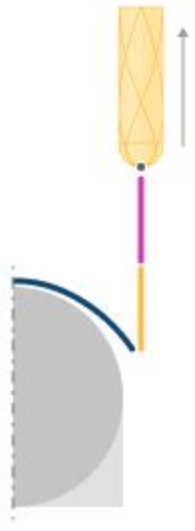
Retract.

▼ Click here to expand..
Same as engage. Builds the retract in the same way as the engage; separate retract size settings are not used.

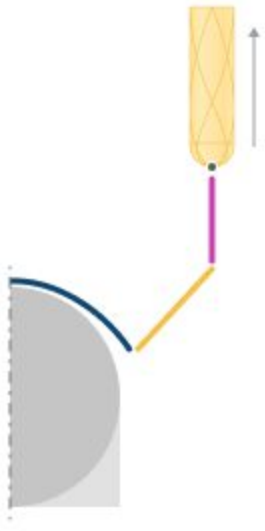
Off. The Engage and Retract toolpath parts is not generated.



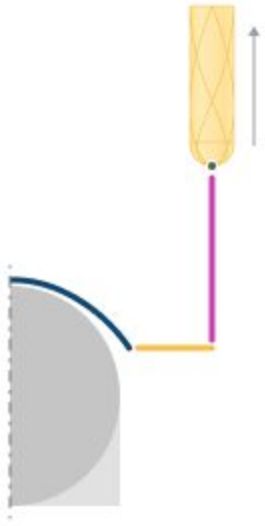
By Vertical Normal. The retraction – along the tool axis from the last point of the work pass.



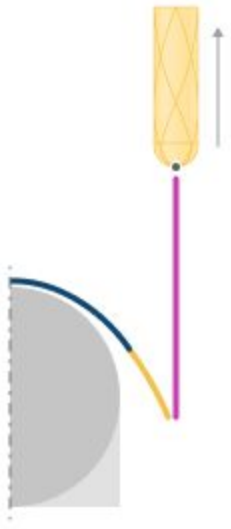
By Surface Normal. The approach is performed along the normal to surface at the first point of the work pass



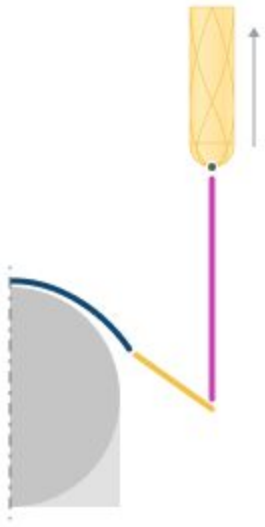
By Horizontal Normal. The retraction is performed perpendicular to the tool axis from the last point of the work pass.



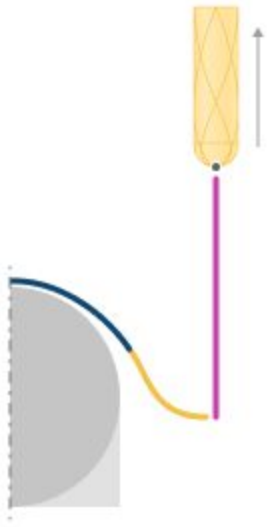
By Tangent. The approach is performed tangentially to the first machining point.



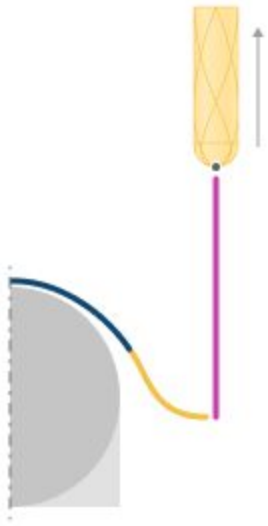
By Line. The approach is performed by angle to tangent to the first machining point.



By Vertical Arc. The system adds an arc to the first point on the curve using the defined radius. The arc lies in the vertical plane and is tangent with the first toolpath applied to the contour. The angle of the curve is defined by user. The tool plunge is performed at the vertical end of the arc, and then moves along the arc and then onto the work pass.



By Normal Arc. The system adds an arc to the first point on the curve using the defined radius. The arc lies in the vertical plane and is tangent with the first toolpath applied to the contour. The angle of the curve is calculated so that the tangent on the other side of the arc is vertical. The tool plunge is performed at the vertical end of the arc, and then moves along the arc and then onto the work pass.

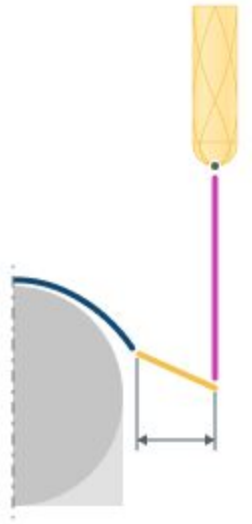


By Horizontal Arc. Used when constructing a trajectory in the horizontal direction. The system adds a tangent with an angle in the horizontal direction.

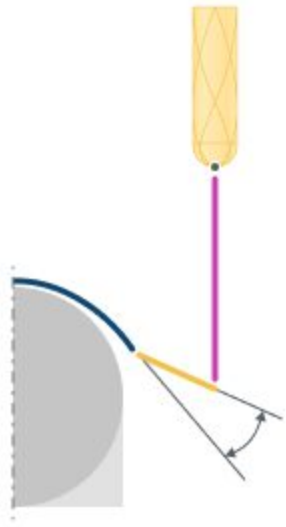


Depending on the engage, it becomes possible to control additional parameters:

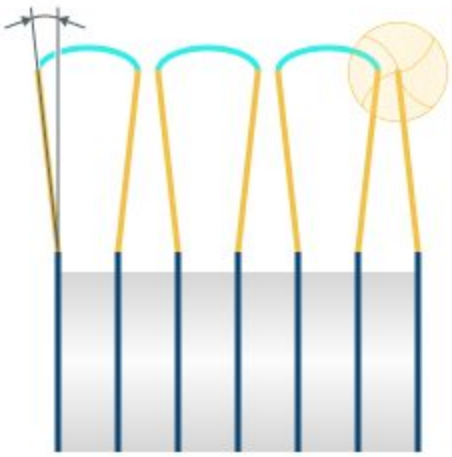
Length (Radius). Changes the length



Angle. Changes the vertical angle

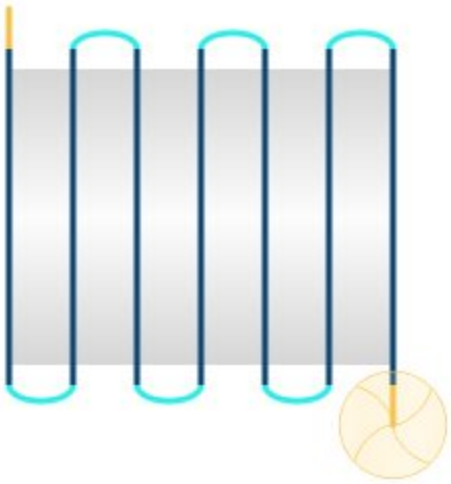


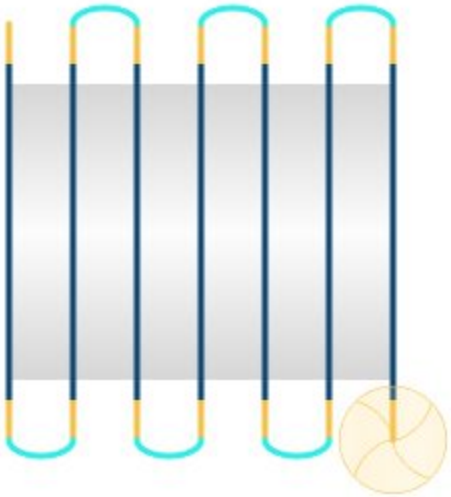
Plane angle. Changes the horizontal angle



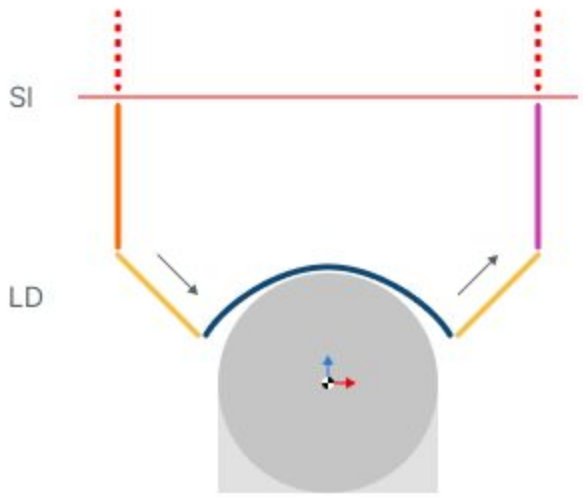
Use leads for short links. Enable: When enabled leads are added to all links, including the short ones.

Disable: When disabled leads are added only to long links and entry/exit inks.





Leads for countouring operations



SI - Safe level

LD - Leads

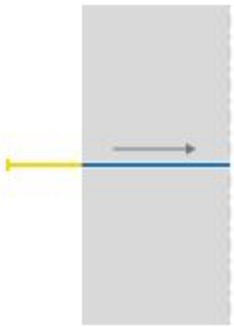
Application Area:

Allows you to add additional engage and retract to the working moves using their feed.

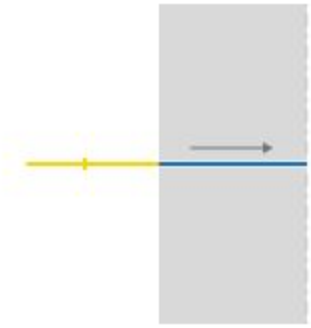
Engage.

▼ Click here to expand..

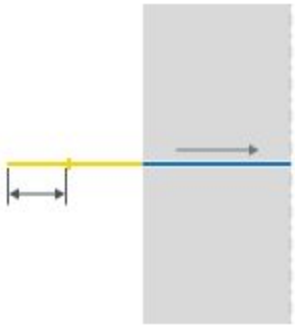
OFF. The Engage and Retract toolpath parts is not generated.



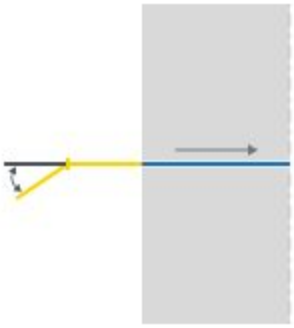
By Line.



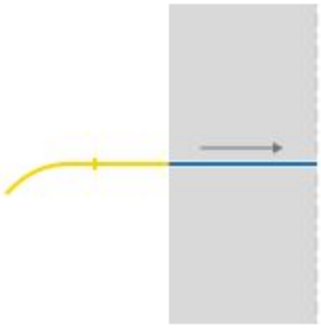
Length.



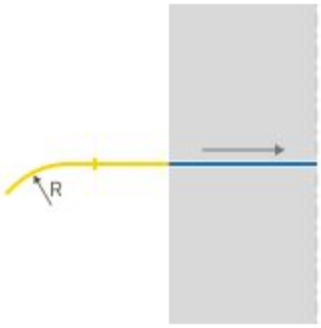
Angle.



By Arc.

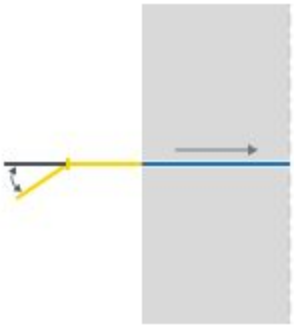


Radius.



R - Engage radius

Angle.

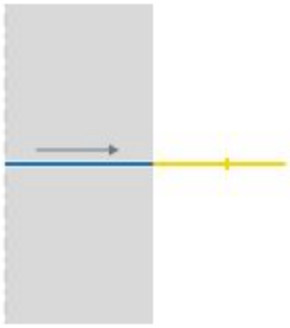


Retract.

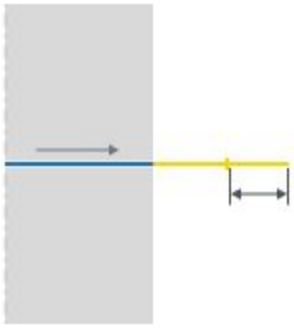
▼ Click here to expand..
OFF. The Engage and Retract toolpath parts is not generated.



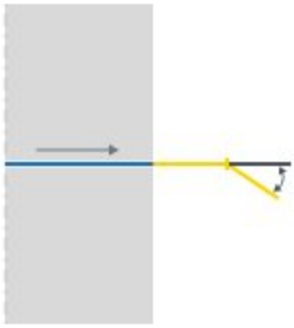
By Line



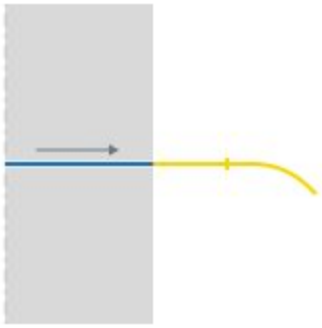
Length.



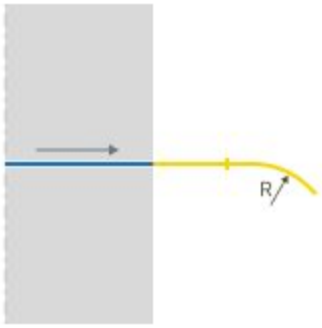
Angle.



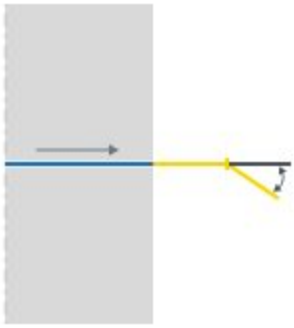
By Arc.



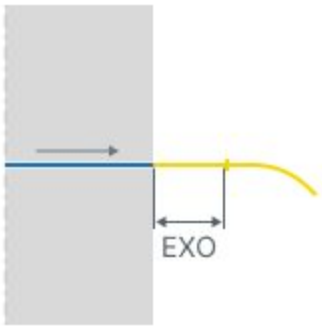
Radius.



Angle.

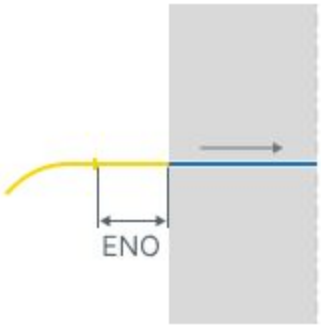


Exit overlap. An overlap is used to extend a toolpath beyond the contour itself. To change the overlap value you can either drag the yellow marker with the mouse or click on it and edit the dimension value which appears.



EXO - Exit overlap

Entry overlap. An overlap is used to extend a toolpath beyond the contour itself. To change the overlap value you can either drag the yellow marker with the mouse or click on it and edit the dimension value which appears.

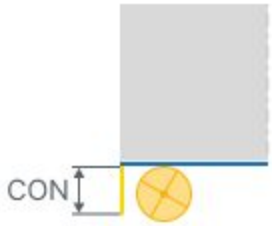


ENO - Entry overlap

Compensation on cut. Inlet segments are automatically added to the main approach paths of the compensated contour when the Compensation type is not set to Computer. By default, input/output segments are not added directly to the straight feed paths, as this would be technically impractical.

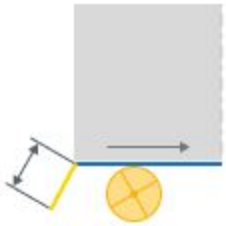
▼ Click here to expand..



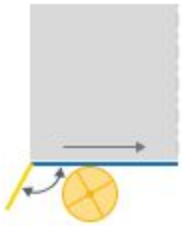


CON - Compensation on cut.

Length. Allows increasing inlet segment length.



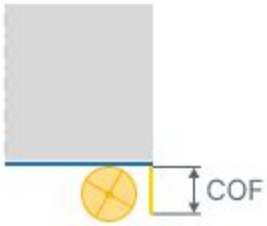
Angle. Allows increasing inlet segment angle.



Compensation off cut. Outlet segments are automatically added to the main approach paths of the compensated contour when the Compensation type is not set to Computer. By default, input/output segments are not added directly to the straight feed paths, as this would be technically impractical.

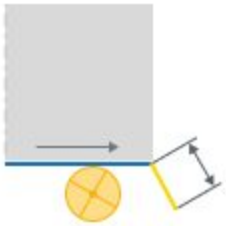
▼ Click here to expand..



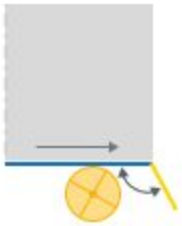


COF - Compensation on cut.

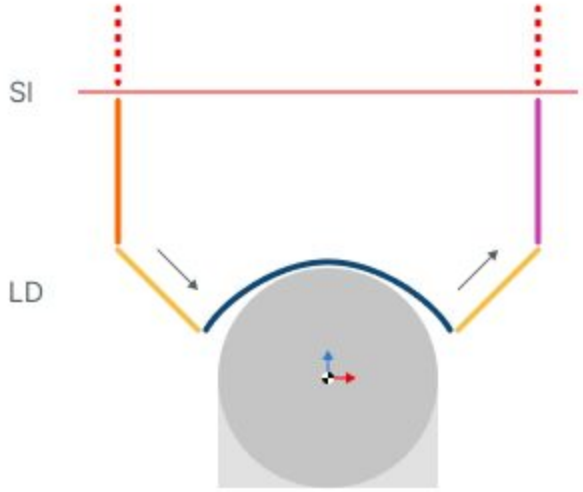
Length. Allows increasing outlet segment length.



Angle. Allows increasing outlet segment angle.



Leads for cladding operations



SI - Safe level

LD - Leads

Application Area:

Allows you to add additional engage and retract to the working moves using their feed.

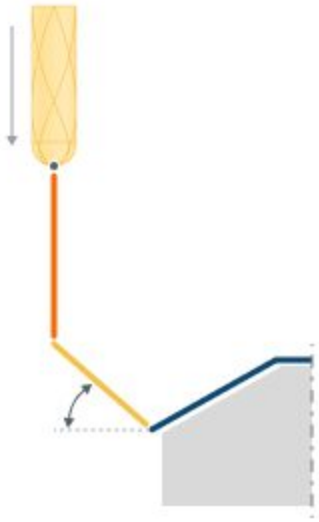
Engage.

▼ Click here to expand..

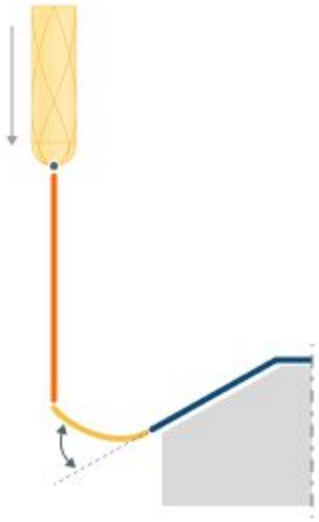
Off. The Engage and Retract toolpath parts is not generated.



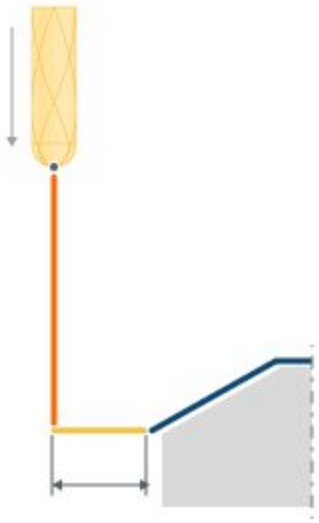
By Line. The engage is performed by angle to vertical to the first machining point

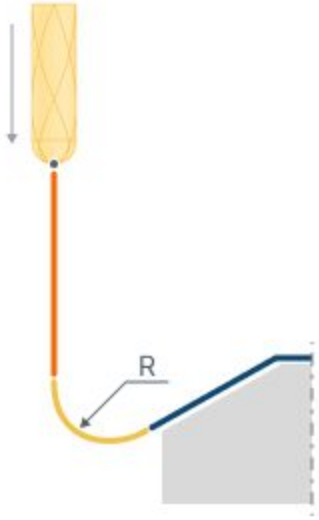


By Arc. The system adds an arc to the first point on the curve using the defined radius. The arc angle is constructed from the tangent of the work pass.

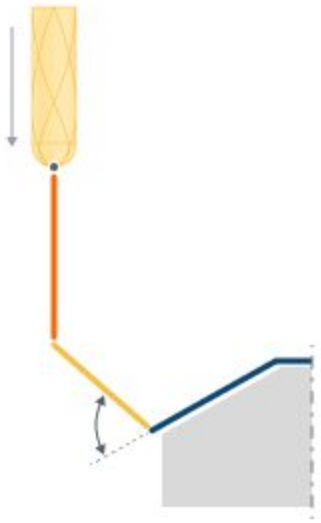


Length (Radius). Changes the length





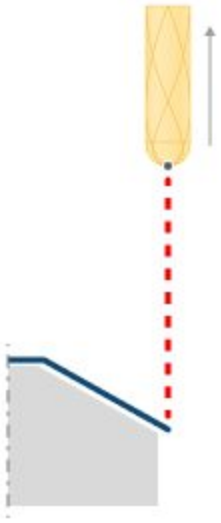
Angle. Changes the vertical angle



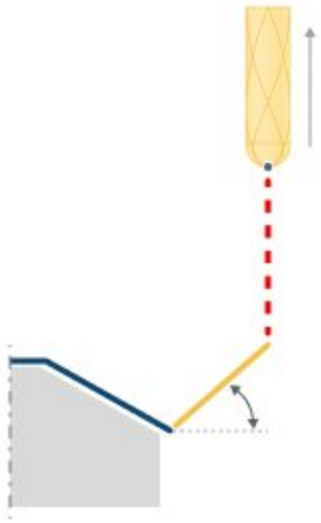
Retract.

▼ Click here to expand..

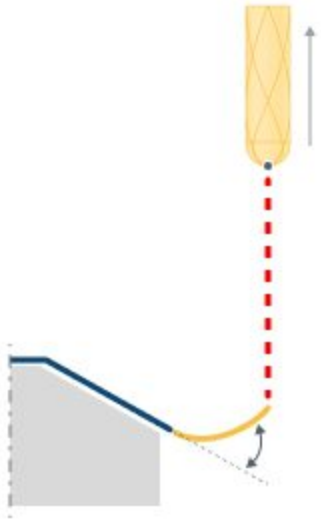
Off. The Engage and Retract toolpath parts is not generated.



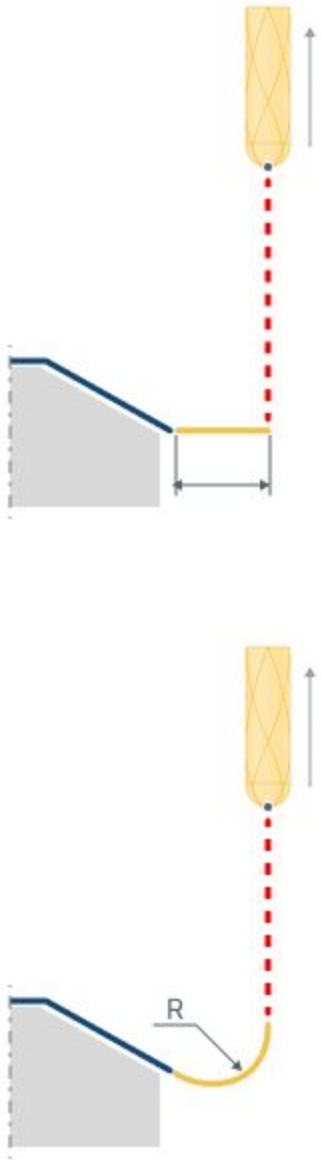
By Line. The retract is performed by angle to vertical to the last machining point



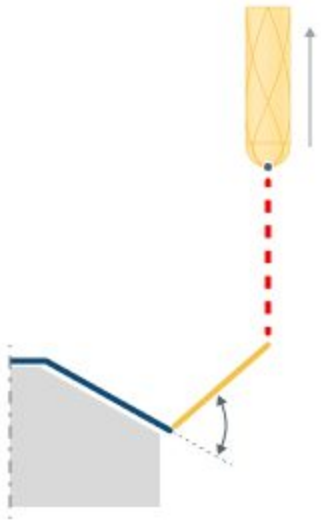
By Arc. The system adds an arc to the last point on the curve using the defined radius. The arc angle is constructed from the tangent of the work pass.



Length (Radius). Changes the length

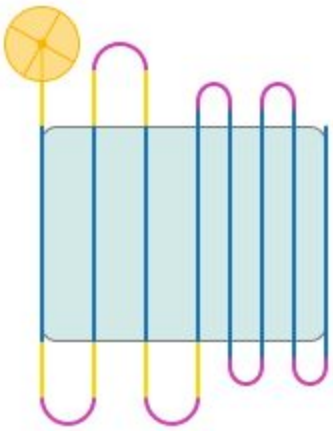


Angle. Changes the vertical angle

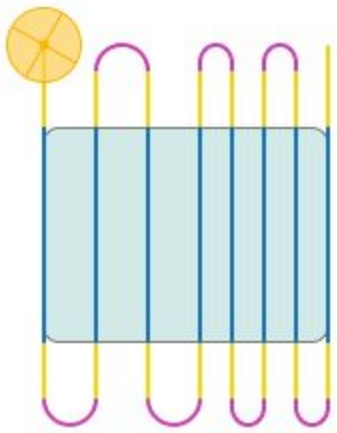


Use leads for.

▼ Click here to expand..
Long links. Leads is used only on the Long links.



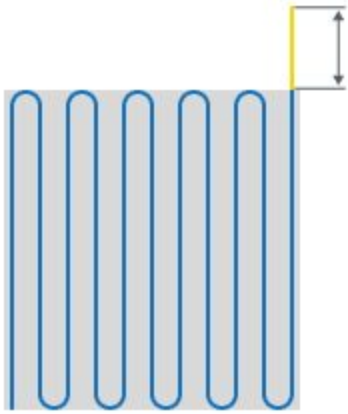
Long and Short links. Leads is used on both Long and Short links.



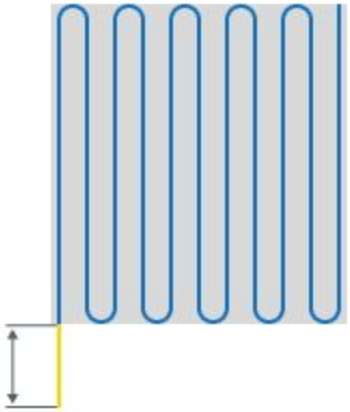
Overlaps.

▼ Click here to expand..

Entry overlap. Allows you to set an overlap at the beginning.

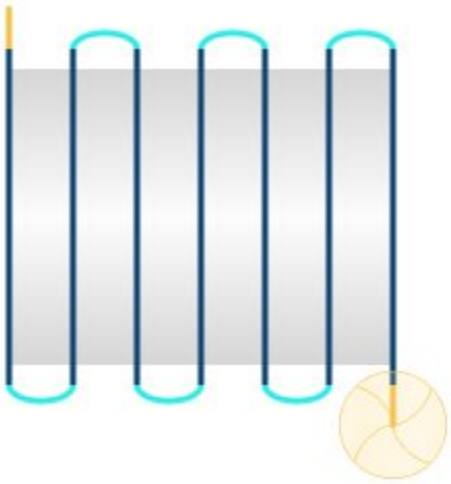


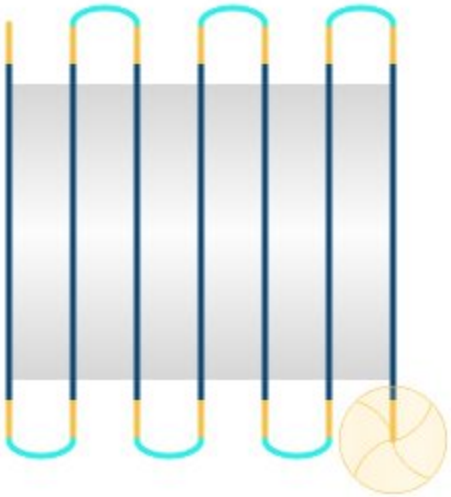
Exit overlap. Allows you to set an overlap at the end.



Use leads for short links. Enable: When enabled leads are added to all links, including the short ones.

Disable: When disabled leads are added only to long links and entry/exit inks.



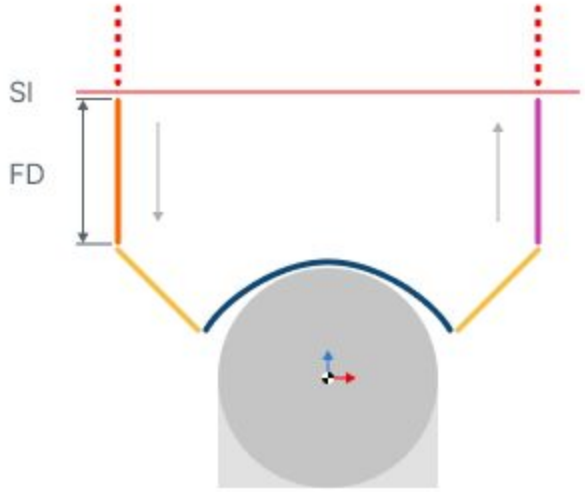


Distances

Distances define the clearances and levels used during tool approaches, retracts, and links between working passes. Depending on the operation type, these parameters determine where the tool switches between rapid motion and feed motion, how far it retracts, and how linking moves are positioned relative to the workpiece.

This section explains the distance settings available for 3D machining, face milling, roughing waterline, advanced 3D/5D operations, and drilling. Refer to the corresponding topic to check how each distance is measured and how it affects the toolpath.

Distance for 3D operations

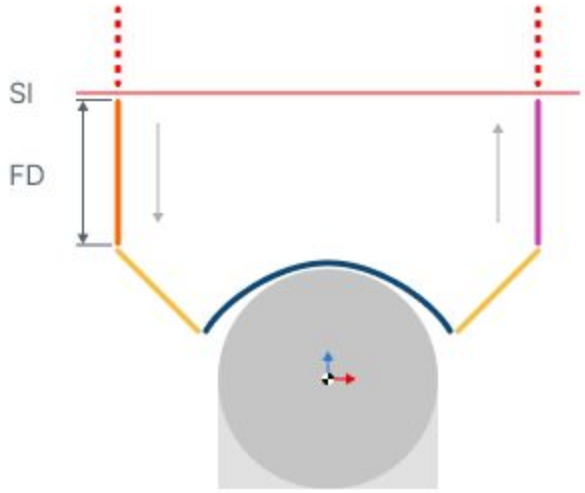


Application Area:

Distances required to switch feeds.

Feed distance (level). The distance measured along the tool axis from the activation of approach feed to the point where it switches to the work feed or engage feed.

▼ Click here to expand..



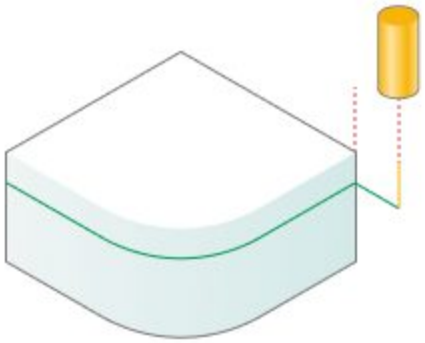
SL - Safe Surface

FD - Feed Distance

The approach feed starts during movement toward the workpiece from the Safe Surface or from the Rapid Distance. Within this distance, the approach feed is active. The switch to the work feed occurs at the level of the working passes or when the engage motion with engage feed begins. This value can be set:

- in length units from the engage start point;
- in length units relative to the workpiece level;
- as a percentage of the tool diameter;
- as a percentage of the distance between passes.

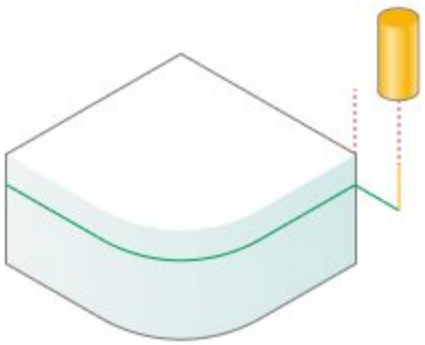
Distances based on the Face Milling



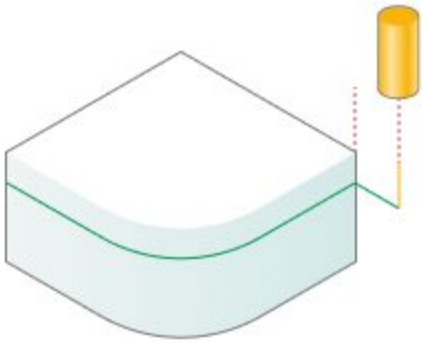
Application Area:

Distances required to switch feeds

Feed distance. Distance where vertical motion switch to work feed.



Distances based on the Roughing waterline

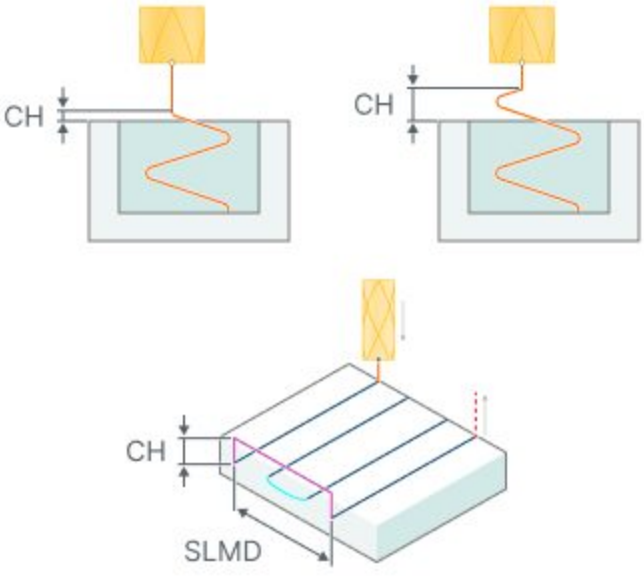


Application Area:

Distances required to switch feeds

Clearance height.

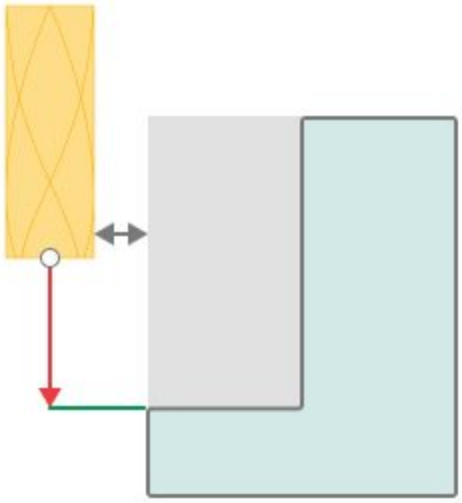
▼ Click here to expand..
This distance is used to back-off the tool, when it goes over the already machined zone (If the transition value is greater than the value `*Short link max distance*`). It is also used as a clearance value when cutting into closed areas.



CH - Clearance height SLMD - Short link max distance

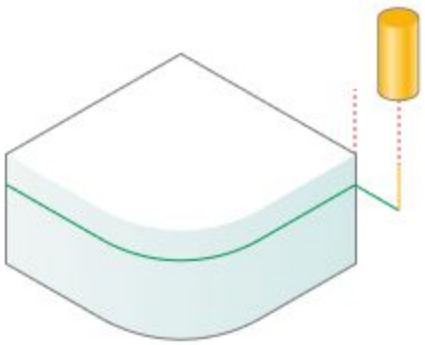
Safe distance.

▼ Click here to expand..
The distance from the workpiece when switch from/to rapid feed or long link feed.



Feed distance (level). The distance measured along the tool axis from the activation of approach feed to the point where it switches to the work feed or engage feed.

▼ Click here to expand..

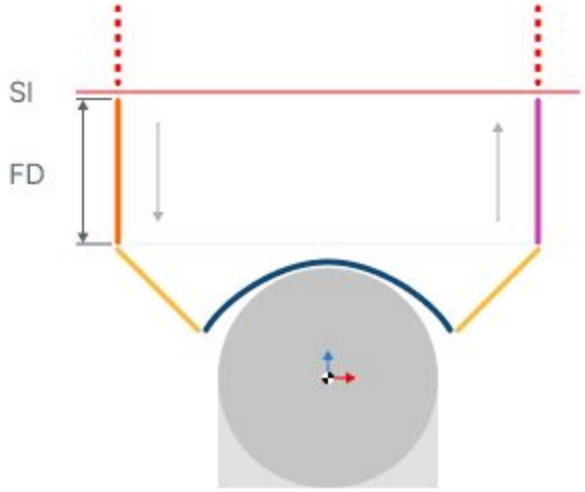


The approach feed starts during movement toward the workpiece from the Safe Surface or from the Rapid Distance. Within this distance, the approach feed is active. The switch to the work feed occurs at the level of the working passes or when the engage motion with engage feed begins. This value can be set:

- in length units from the engage start point;

- in length units relative to the workpiece level;
- as a percentage of the tool diameter;
- as a percentage of the distance between passes.

Distances for 3D/5D advanced operations

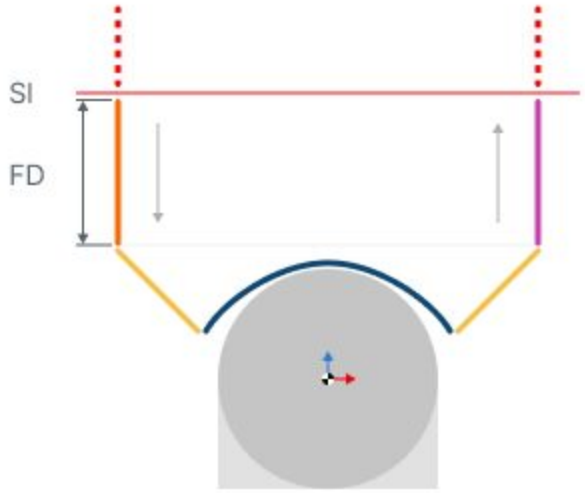


Application Area:

Distances used in Links rules

Feed Distance.

▼ Click here to expand..
The distance measured along the tool axis from the activation of approach feed to the point where it switches to the work feed or engage feed.



SL - Safe Surface

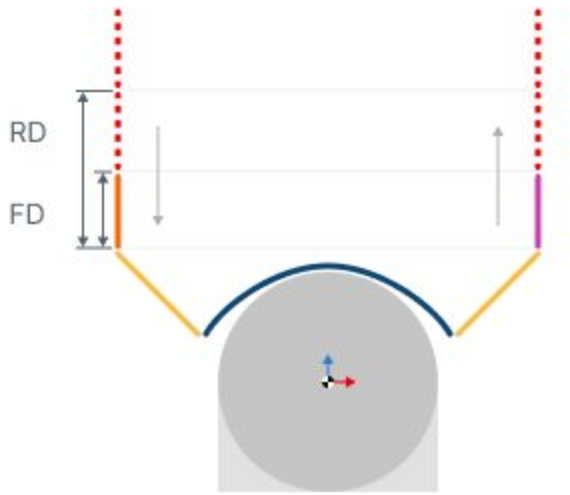
FD - Feed Distance

The approach feed starts during movement toward the workpiece from the Safe Surface or from the Rapid Distance. Within this distance, the approach feed is active. The switch to the work feed occurs at the level of the working passes or when the engage motion with engage feed begins. This value can be set:

- in length units from the engage start point;
- as a percentage of the tool diameter;

Rapid Distance.

▼ Click here to expand..
It is the distance from the workpiece where entry movements take place at machining start or exit movements after its completion.



FD - Feed distance

RD - Rapid distance

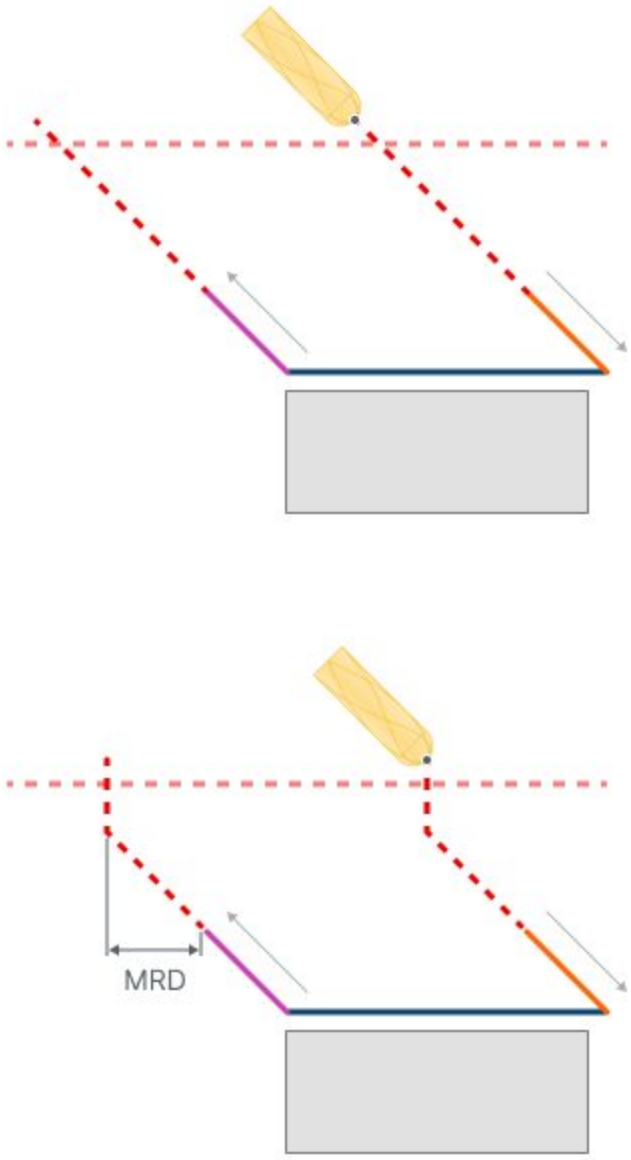
This serves as an alternative method to specifying the entry/exit or links levels for the tool instead of the Safety Surface. The Rapid Distance must exceed the Feed Distance.

Max Retraction Distance.

▼ Click here to expand..
The longest distance the tool can travel along its axis while retracting at rapid speed.

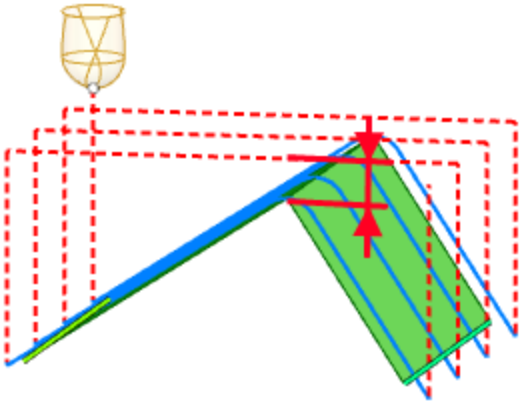
Enable: If the option is turned on, the tool first moves this distance along its axis during Links and Retracts, and the remaining travel to the Safety Surface occurs perpendicular to it.

Disable: When the option is disabled, the tool travels to the Safety Surface along its axis during Links and Retracts.

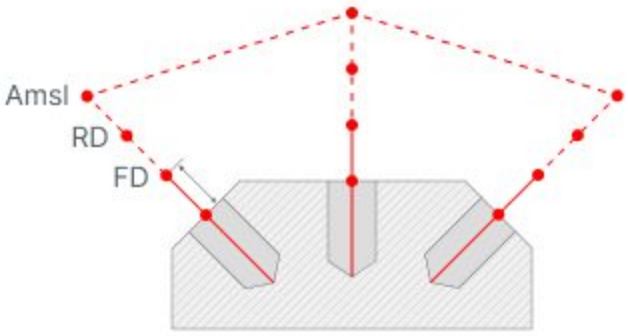


MRD - Max retraction distance
Rapid move safe distance

▼ Click here to expand..
When making a rapid distance link the tool is checked against the workpiece. At positions where the distance between the tool and the workpiece along the tool axis direction is less thatn the Air move safety distance, the tool is raised by the rapid distance. So, for this type of link, Air move safety distance is the minimum distance, while the rapid distance is the maximum distance between the tool and the workpiece along the tool axis direction.



Distances in drilling operations

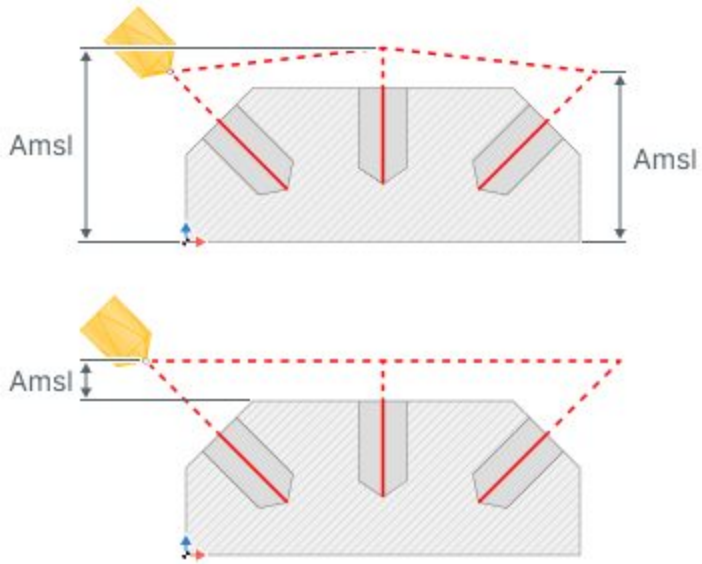


Application Area:

Distances used in Links rules

Air move safety level.

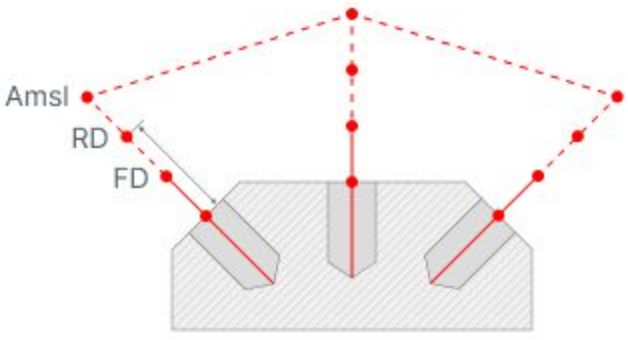
▼ Click here to expand..
The toolpath continues along the hole axis up to the air move safety level. The air move safety level can be definied as absolute distance from the origin of CS or incrementally from the part (plus workpiece and fixtures) top level.



AMSL - Air move safety level

Rapid distance.

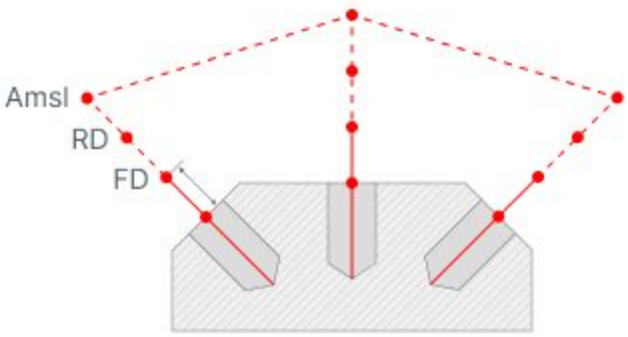
▼ Click here to expand..
The Rapid distance defines the point on the holes axis in which the cycle should start. The tool will go to this point at rapid feed. If the Start cycle at air move safety level option is enabled then this point is extended to the air move safety level.



AMSL - Air move safety level RD - Rapid distance FD - Feed distance

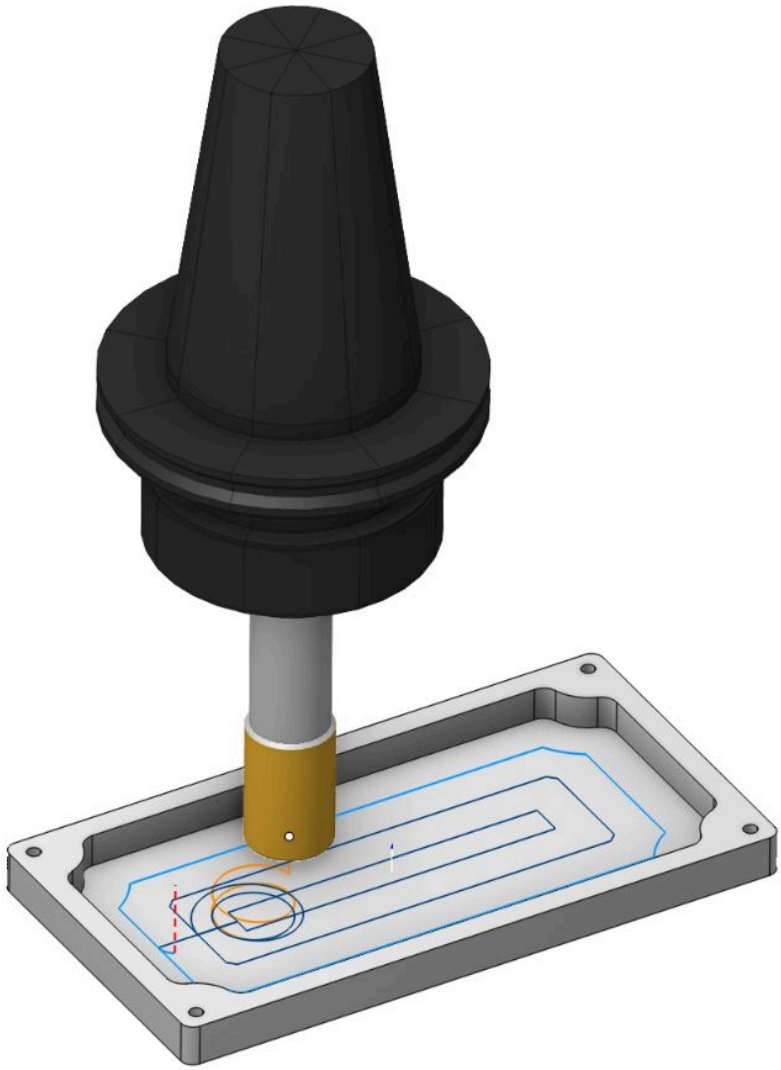
Feed distance.

▼ Click here to expand..
The Feed distance dimension defines the point on the hole's axis from the upper level of the hole on which the feedrate is switched from the rapid to the work one. So this distance is used to avoid the collision on the rapid feed.



AMSL - Air move safety level RD - Rapid distance FD - Feed distance

Allow plunges



Application Area:

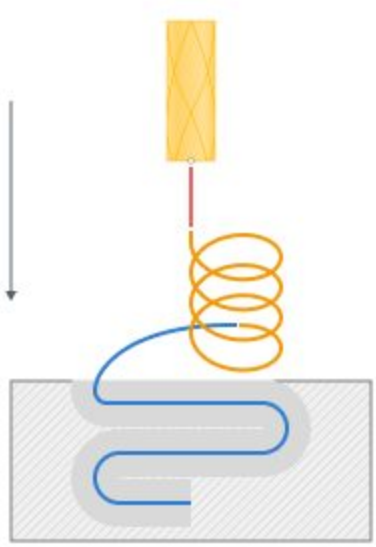
When it is impossible to approach the machining area from the outside, the system automatically generates a plunge movement to the first point of the work pass. A plunge is a toolpath section along the Z axis within the workpiece body. The Plunge is performed as follows:

- Tool approach to the XY plunge point at the safe plane of the operation.
- Traverse vertically at rapid feed to the safe level or the safe distance before cutting begins (depending on settings). The safe level is the Z top level of the current coordinate system. The safe distance is the distance to the start of the defined plunge.
- Vertical descent at the approach feeds into the beginning of the defined plunge.
- The selected plunge method is applied at the approach feed to the approach point.
- Approach starts.

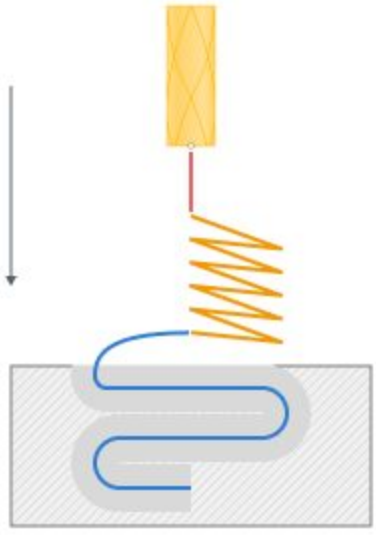
The following plunge methods are available in the system:

None. Set it if you don't want to try plunge.

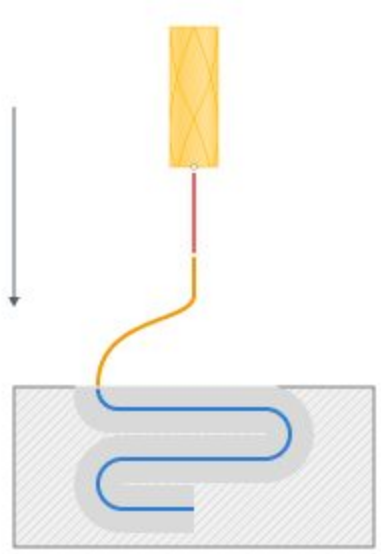
Spiral. The tool performs a helical motion along a circle, connected at its last point with the first approach point. The radius of the circle is a user-defined option.



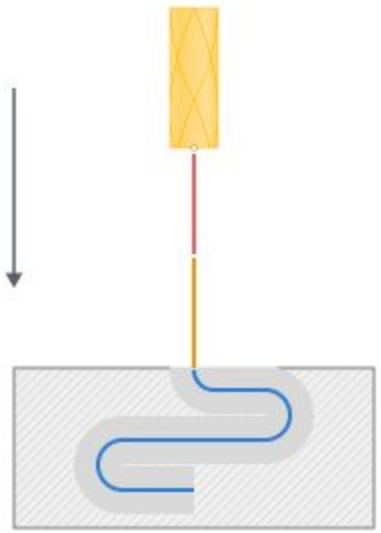
Zigzag. The tool performs reciprocal movements along a straight section, connected to the first approach point. The length of the section is a user-defined option.



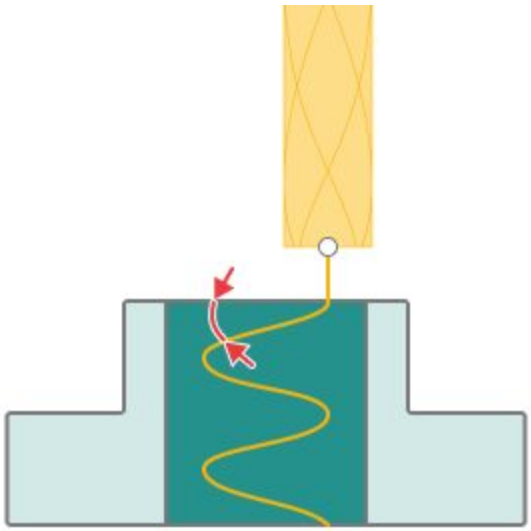
Along curve. The tool plunges along the approach curve. The plunge move is completed at the end of the approach move.



Axial. Performed along the vertical straight to the first point of approach.



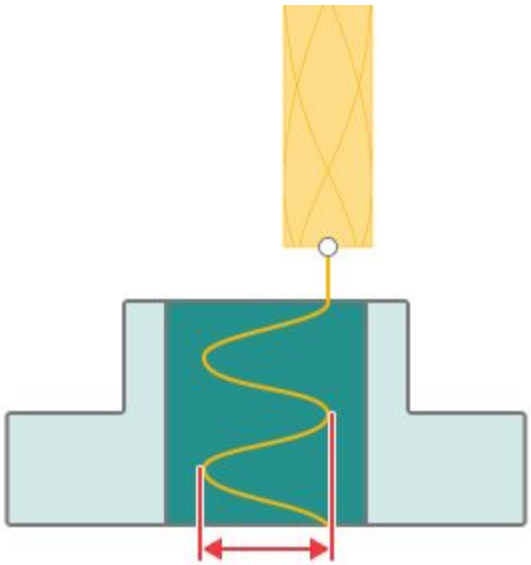
Ramp angle. Plunge Angle determines the speed of plunging. The greater the angle the faster the tool plunges into material.



Min size. The Plunge Min size defines:

- the minimum allowed diameter of a spiral in Spiral Plunges,
- the minimum allowed amplitude of zigzag moves in Zigzag and Along Curve Plunges.

If CAM software can not form a gouge-free plunge of the selected type with the given minimal size, it tries to use the next available Plunge Type. If no more plunge types are available the pocket is left unmachined.

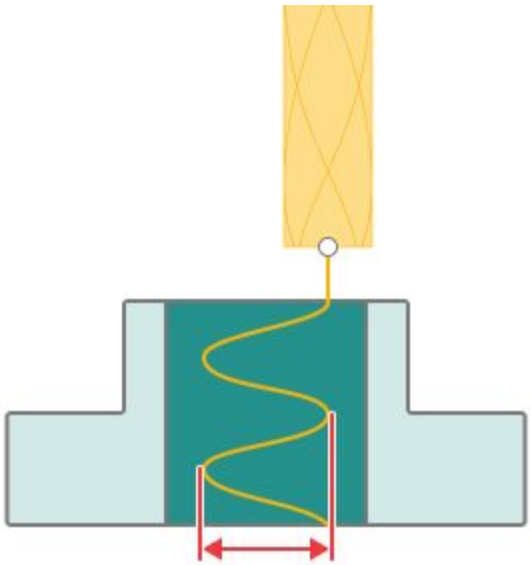


Max size. The Plunge Max size defines:

- the optimal diameter of a spiral in Spiral Plunges,

- the optimal amplitude of zigzag moves in Zigzag and Along Curve Plunges.

When CAM software can not form a gouge-free plunge of the selected type with the given optimal size, it tries to decrease the size by the amount specified in the Degression parameter until either the plunge becomes possible or the size reaches the minimum allowed value, then CAM software tries to apply another Plunge type from the list.



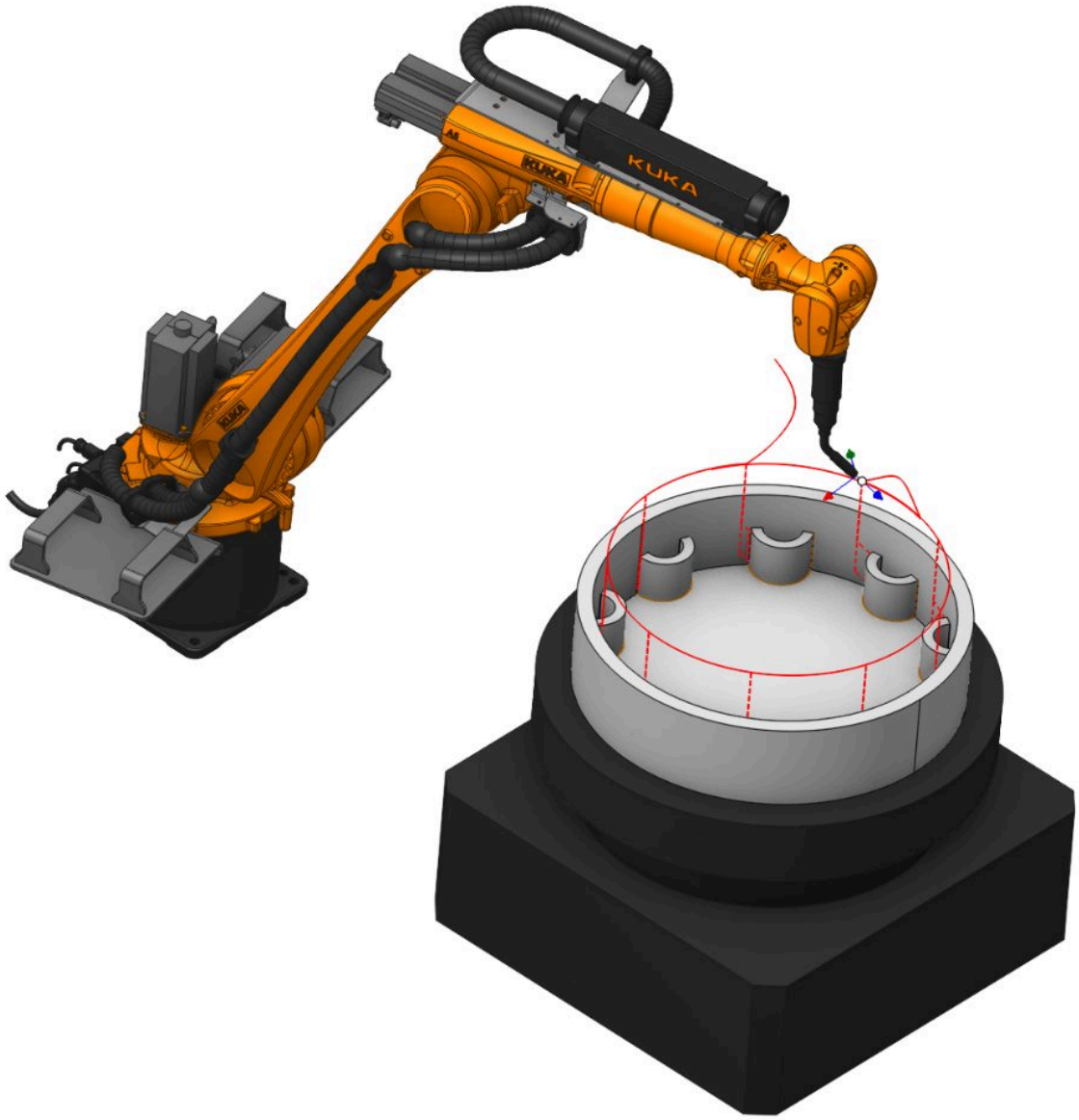
Degression. Degression defines the value by which CAM software decreases the plunge size when it can not form a gouge-free plunge. When CAM software can not form a gouge-free plunge of the selected type with the given optimal size, it tries to decrease the size by the amount specified in the Degression parameter until either the plunge becomes possible or the size reaches the minimum allowed value, then CAM software tries to apply another Plunge type from the list.

Smooth radius.

If enabled: The connection of plunge motion and horizontal motion is rounded by smooth radius.

If disabled: The plunge toolpath is connected with the first horizontal motion under plunge angle.

Transformation



Application area:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool.

Main settings:

Part copying.

The way of output of the prototype part.

▼ Click here to expand..
Copying method. The method of outputting processing code for a copy of the part.

Using real copies. Utilizing expanded code.

Using subtouutines. Utilizing subroutines.

Default. Using real copies

Multiply toolpath by axis.

The Multiply toolpath feature can help with machining of repeating model fragments. You can define machining for only one of repeating parts and then just determine the amount and orientation of the same elements. Works with physical axes of equipment

▼ Click here to expand..
Machining order.

Consistently. Repeating copies of a toolpath place along a circle, with the count and the angle step given in the appropriate field

Multiply Step. Step axle-direction in units within the given axle.

Multiply Count. Quantity of block's repetitions.

Most distant. Repeating copies of a toolpath place along a circle, with the count and the angle step given in the appropriate field. Elements are sorted more distantly from each other.

Manually. Repeating copies of a toolpath place along a circle. The count and the angle of each element are set by user.

Steps. Quantity of block's repetitions with own angle

Angle1..N. Angle of specific block's repetitions

Formalize as subroutine. The repeated block design as subprogram. Additionally you can set its name in the field. If Formalize as subroutine is disabled, then the repeated block over and over again insert in CLData with re-calculated coordinates.

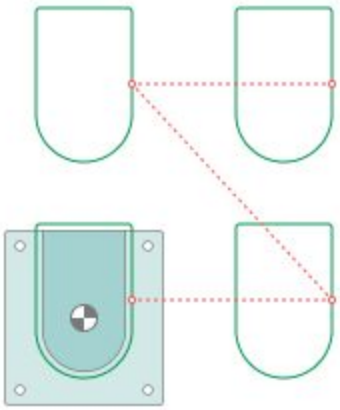
Multiply scheme.

The Multiply toolpath feature can help with machining of repeating model fragments. You can define machining for only one of repeating parts and then just determine the amount and orientation of the same elements. Works with Multiply schemes.

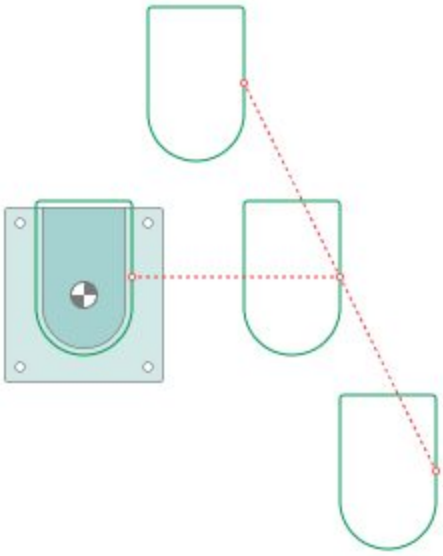
▼ Click here to expand..

None. Don't use Multiply scheme.

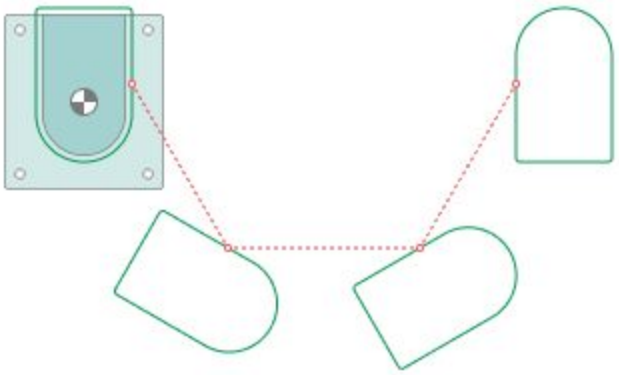
Two dimensioanal array. Forms a two-dimensional array.



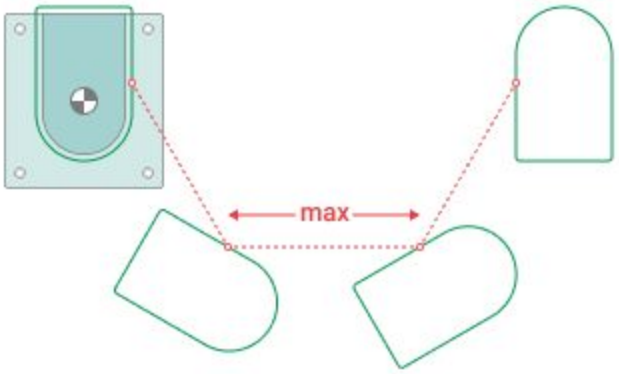
Two dimensional array (manually). Enables the formation of an array from elements with arbitrary placement.



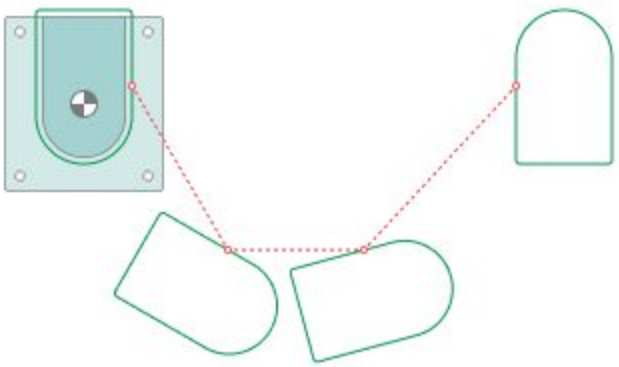
Round array. Array arranged in a circular pattern.Sequential processing.



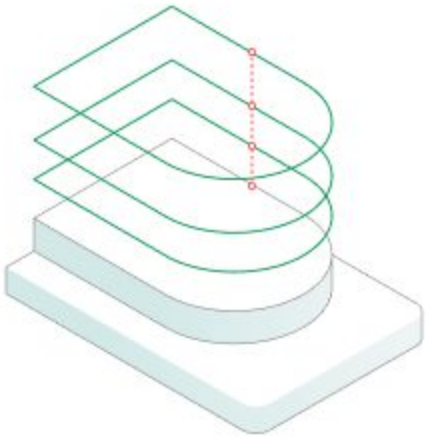
Round array (most distant). Array arranged in a circular pattern. The algorithm aims to find the next processing element located at the maximum distance from the current position



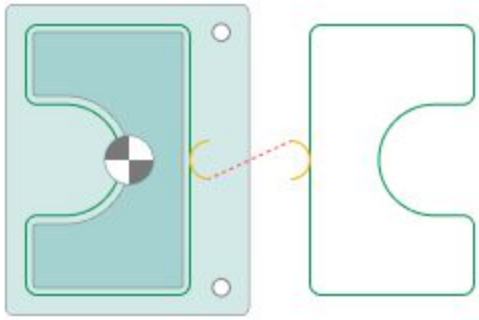
Round array (manually). Array arranged in a circular pattern. Allows setting a specific angle for each element.



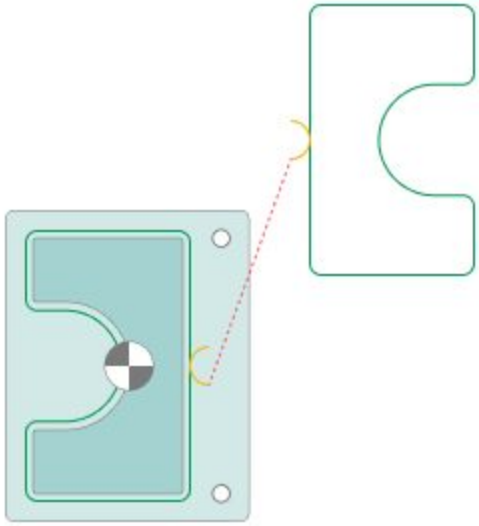
Level array. The trajectory is copied along the tool axis.

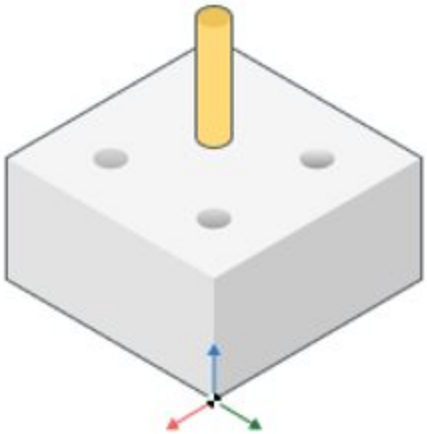


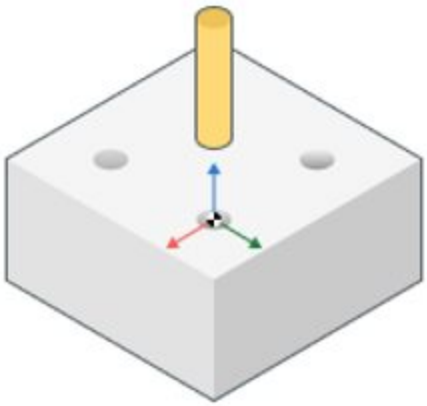
Axis symmetry. Enables creating a mirrored copy relative to the axis.



Point symmetry. Enables creating a mirrored copy relative to the point





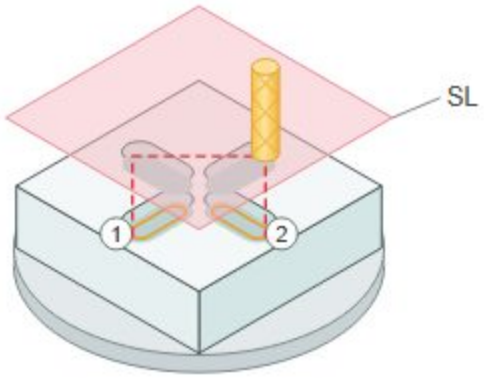


Base coordinate system. Defines the coordinate system with respect to which the transformation will be performed. It can be: Tool CS, Workpiece CS, Global CS or any other geometrical CS you have created before. Also additional translation and rotation of this CS can be defined inside Additional transformation group of parameters.

Additional transformation. Allows you to enter additional offsets along the axes

Custom transition between copies.

By default, the tool moves from one copy of the toolpath to the next one through the safe surface of the operation. The Custom transition between copies parameter lets you define this transition with a rule, the same way the operation approach and return are defined.



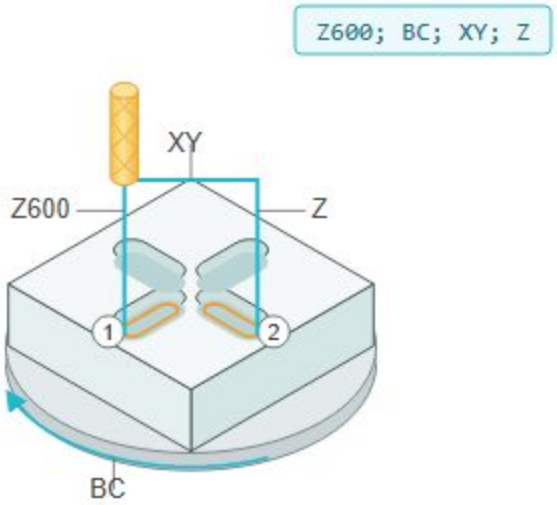
SL - Safe level

The parameter is on the Transformations tab under Multiply scheme. It is shown only when a multiply scheme mode other than None is selected.

To define the transition:

1. Turn on Custom transition between copies.
2. In the field next to it, select a rule from the list or type it in:
    - Same as approach rule — the approach rule of the operation is used (default);
    - Avoid collisions, Short, From Previous, From Root, the submachine rule and the rules from the machine approach/return list — the same as for the operation approach;
    - Custom... — open the editor window and define your own sequence of movements.

The rule is written the same way as an approach rule. For example, the rule `Z600; BC; XY; Z` means: move up along Z to the level of 600, rotate the B and C axes, move along X and Y, then move down along Z to the start of the next copy. Details



The transition movements are output to CLData in the section of each copy (Copy: 2, Copy: 3, ⋯) before the toolpath of the copy or the subroutine call.

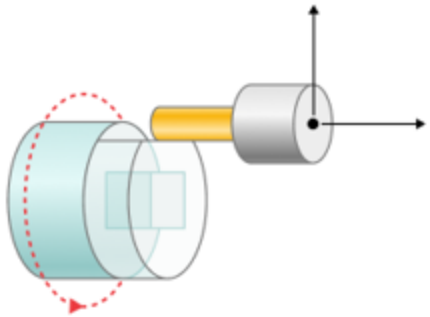
Limitations:

- the transition is defined only for Multiply scheme; Multiply toolpath by axis does not have this parameter;
- for operations inside a Multiply group the parameter has no effect — the group controls the transitions itself;
- it works both with and without Advanced axes limits control, including when the copies are formalized as a subroutine.

Rotary transformations.

Allow at 3-coordinate milling processing to change the displacement of the tool in direction of one of the linear axle to the rotary motion of the billet, if it is possible to execute on the machine.

▼ Click here to expand..
Polar. Polar interpolation changes a linear axis to the rotary one in the simple 3-axes milling process. Usually it is necessary on the lathes that has the drive mill tool. Sometimes the polar interpolation is used with another king of machines. Ordinary lathe has two linear axes, usually its are X and Z, and the spindle rotation – usually axis C. Details



Rotary axis. A rotary axis, which replaces a linear axis (usually a spindle, rotary table).

Radial Coordinate. A linear axis perpendicular to the part axis.

Axial Coordinate. A linear axis coinciding with the axis of rotation of the part.

Negative Radial Coordinate. If enabled: Radial coordinate outputs with POSITIVE sign.

If disabled: Radial coordinate outputs with NEGATIVE sign.

CNC interpolation. If enabled: Toolpath is interpolated by CAM system with usage of the Radial and Rotary axes.

If disabled: Special CNC commands are used to activate/deactivate interpolation on CNC-control

Rapid Motions. Way of output (G0,G1) for rapid motions

Fast feed. It outputs linear motion (G1) for rapids inside interpolation block

Rapid feed. It outputs positioning (G0) for rapids inside interpolation block

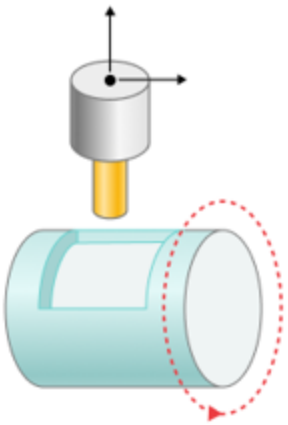
Fast feed. Feed value that will be output if "Rapid Motions=Fast Feed"

Start from C=0. If enabled: It sets rotary axis to ZERO POSITION before switch on the interpolation.

If disabled: it sets rotary axis to the BEST POSITION before switch on the interpolation. P.S. Some postprocessors implemented till 2023 does not support this option. Contact your PP provider or use "Zero position".

Tolerance. Characterize the deviation of transformed trajectory from ideal in millimeters (inches).

Cylindrical. The cylindrical interpolation gives the possibility to mill the side surface of cylinder by programming the unrolled curves. The unrolled curves are programmed in the [X,Y,Z] coordinates, but the cylinder milling is performed in [X,C,Z] coordinates. So the cylindrical interpolation makes the transformation [X,Y,Z] => [X,C,Z]. Details

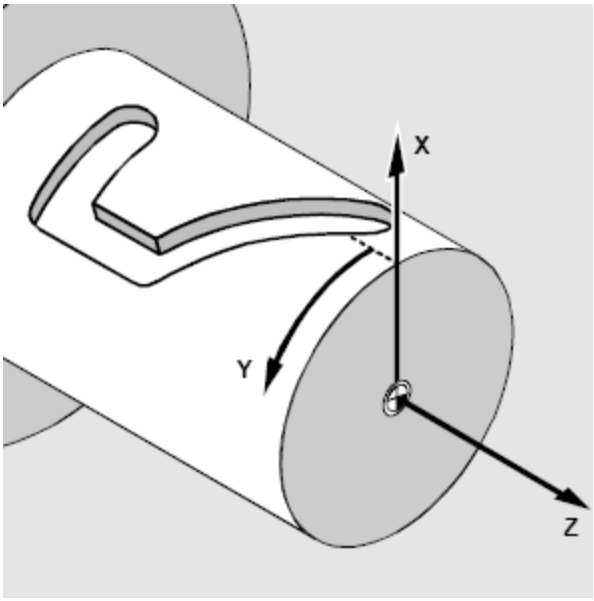


Some parameters are similar to polar interpolation.

For cylindrical interpolation, it is necessary to specify on the Job Zone tab as the Base surface Cylinder and enable interpolation.

Off. Does not output Rotary transformations.

Cylindrical interpolation



The cylindrical interpolation is available in operations: 2D contouring, pocketing, 2.5D pocketing, 2.5D wall machining.

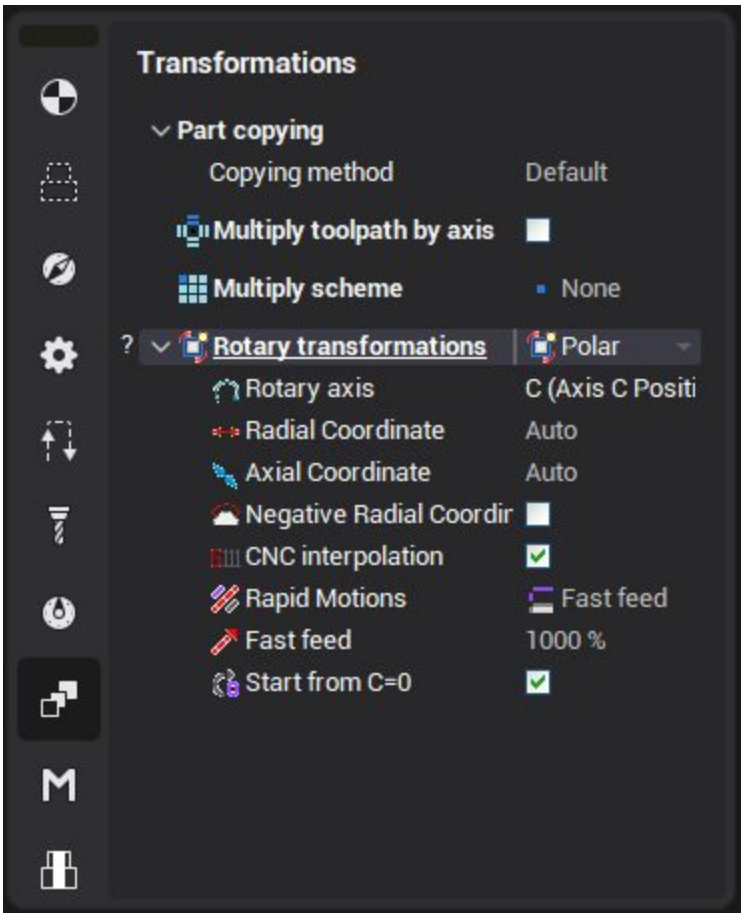
The cylindrical interpolation gives the possibility to mill the side surface of cylinder by programming the unrolled curves. The unrolled curves are programmed in the [X,Y,Z] coordinates, but the cylinder milling is performed in [X,C,Z] coordinates. So the cylindrical interpolation makes the transformation [X,Y,Z] => [X,C,Z].



The possibility to perform the milling of the side surface of cylinder depends on the machine construction.

1. It must exist the Rotary axis (rotary table, lathe spindle) that rotates the workpiece.
2. The rotary axis must be located perpendicularly to the tool rotation axis.
3. The tool rotation axis have to intersect the workpiece rotation axis.
4. It must exist the Rotary axis that moves the tool in the plane that is perpendicular to the rotary axis.
5. It must exist the Axial axis that moves the tool along the rotary axis.

If all listed condition are performed and the machine variable Machine –> Control parameters –> Rotary transformations –> Cylindrical interpolation is available is set then the rotary transformation panel will be available on the transformations page.



The Mode field defines the rotary transformation mode: none, polar or cylindrical transformation. The tolerance defines the deviation of the transformed tool path from the ideal one. It is measured in millimeters (inches).

The cylindrical transformation performs the next calculation

$$A = \left(\frac{Y}{R} \cdot \frac{180}{\pi} \right),$$

where:

A – the position of the rotary axis in degrees,

Y – the position of the virtual unrolled axis that corresponds to the rotary axis,

R – the radius of the cylinder.

The corresponding fields defines the machine axes that are taken as the rotary axis, radial axis and etc. The default values for these parameters are defined in the machine schema.

The modern numerical controls have the possibility to perform the cylindrical transformation. So the described transformation is performed inside the control, not inside the CAM software. For such machines it is better to mark the CNC interpolation tick. In this case the G-code is generated in the

[X,Y,Z] coordinates, and control makes the [X,Y,Z] => [X,C,Z] transformation. The G-code in the most cases looks like the next sequence:

1. The positioning to the start point that is programmed in the real machine axes.
2. Switch on the cylindrical interpolation mode with the specifying of the cylinder radius.
3. The motion along the profile that is programmed in the coordinates [X,Y,Z]
4. Switch off the cylindrical interpolation mode

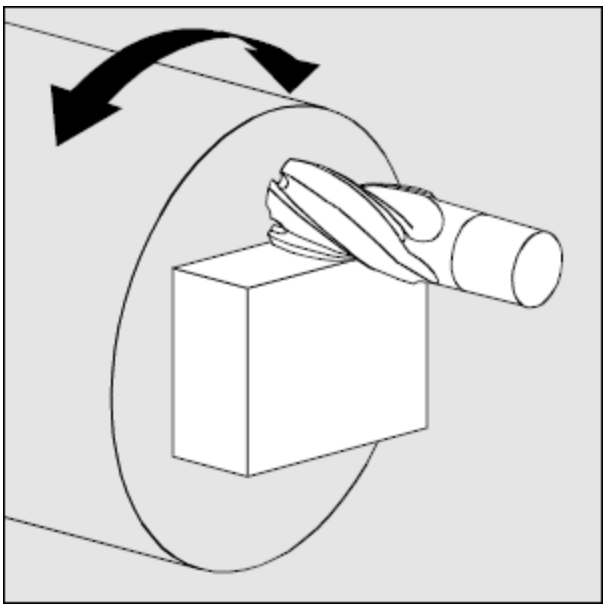
The corresponding commands for the well known controls are shown in the table below.

Numerical control	Command to switch on the cylindrical interpolation	Command to switch off the cylindrical interpolation
FANUC, Mori Seiki, HAAS etc.	G07.1 (G107)	G07.1 (G107)
Sinumeric	TRACYL	TRAOFF
Heidenhain	Cycle 27	-

If the machine variable Machine –> Control parameters –> Rotary transformations –> CNC support cylindrical interpolation is set then CNC interpolation tick is available. If this parameter is on then the G-code generated with the commands to switch on/off the cylindrical interpolation. Else the G-code is generated in the [X,C,Z] coordinates.

If Allow rapid motions inside interpolation block is checked then the interpolation switch on in the beginning of the tool path and switch off in the end of tool path. Else CNC interpolation is started before the work feed and closed before the rapid feed motion.

Polar interpolation



The cylindrical interpolation is available in all milling operations if machine allows to use it.

Polar interpolation changes a linear axis to the rotary one in the simple 3-axes milling process. Usually it is necessary on the lathes that has the drive mill tool. Sometimes the polar interpolation is used with another king of machines. Ordinary lathe has two linear axes, usually its are X and Z, and the spindle rotation – usually axis C.

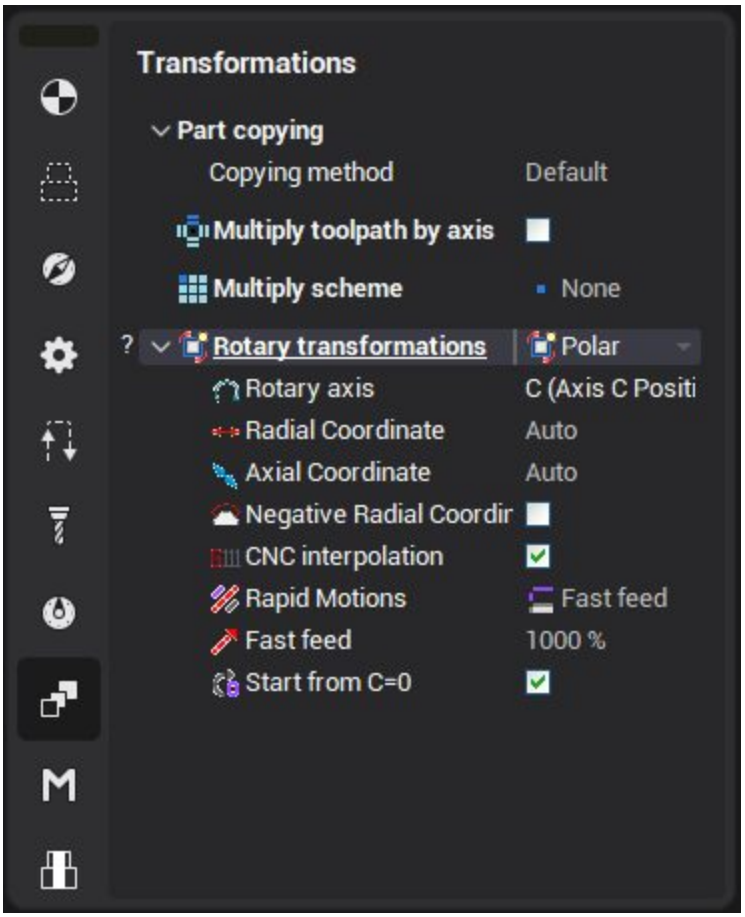


In simple mode CAM system generates the G-code in [X,Y,Z] coordinates. If polar transformation is active then the same g-code is generated in the [X,C,Z] coordinates. So polar interpolation transforms [X,Y,Z] => [X,C,Z]

The possibility to use the polar interpolation depends on the machine construction:

1. It must exist the Rotary axis (rotary table, lathe spindle) that rotates the workpiece.
2. The rotary axis must be located parallel to the tool rotation axis.
3. It must exist the Radial axis that moves the tool in the plane that is perpendicular to the rotary axis.
4. It must exist the Axial axis that moves the tool along the rotary axis.

If all listed condition are performed and the machine variable Machine –> Control parameters –> Rotary transformations –> Polar interpolation is available is set then the rotary transformation panel will be available on the transformations page.



The Mode field defines the rotary transformation mode: none, polar or cylindrical transformation. The tolerance defines the deviation of the transformed tool path from the ideal one. It is measured in millimeters (inches).

The polar transformation performs the next calculation:

$$R = \sqrt{X^2 + Y^2}$$

$$A = arctg\left(\frac{Y}{X}\right),$$

where:

R – radial axis position,

A – rotary axis position,

X – position of the first linear axis,

Y – position of the second linear axis.

The corresponding fields defines the machine axes that are taken as the rotary axis, radial axis and etc. The default values for these parameters are defined in the machine schema.

The modern numerical controls have the possibility to perform the polar transformation. So the described transformation is performed inside the control, not inside the CAM software. In this case the G-code is generated in the [X,Y,Z] coordinates, and control makes the [X,Y,Z] => [X,C,Z] transformation. The G-code in the most cases looks like the next sequence:

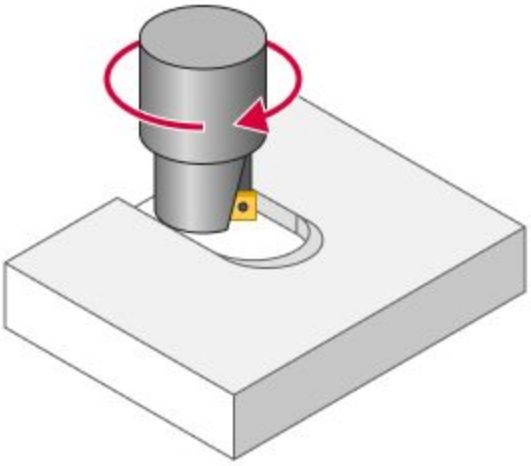
1. The positioning to the start point that is programmed in the real machine axes.
2. Switch on the polar interpolation mode with the specifying of the cylinder radius.
3. The motion along the profile that is programmed in the coordinates [X,Y,Z].
4. Switch off the polar interpolation mode.

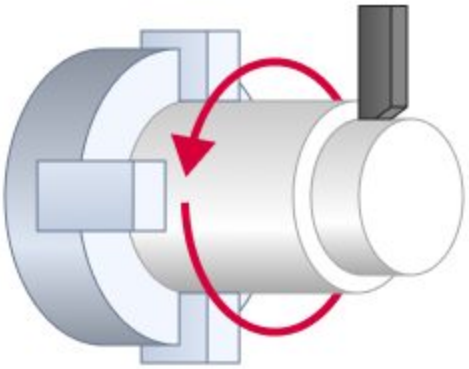
The corresponding commands for the well known controls are shown in the table below.

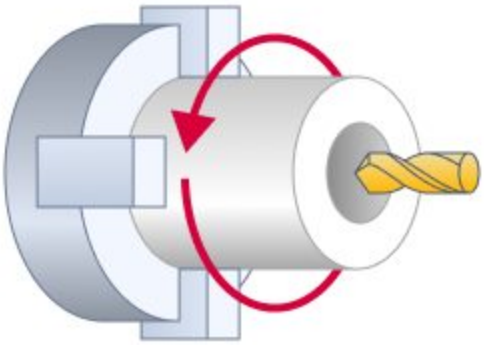
Numerical control	Command to switch on the polar interpolation	Command to switch off the polar interpolation
FANUC, Mori Seiki, HAAS etc.	G112	G113
Sinumeric	TRANSMIT	TRAOFF
Heidenhain	-	-

If the machine variable Machine –> Control parameters –> Rotary transformations –> CNC support polar interpolation is set then CNC interpolation tick is available. If this parameter is on then the G-code generated with the commands to switch on/off the polar interpolation. Else the G-code is generated in the [X,C,Z] coordinates.

Feeds/Speeds







Application Area:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter.

Feeds/Speeds (Calculate).

The calculator enables users to set and automatically calculate spindle speeds and feed rates. Details

Spindle.

This parameter group configures the spindle rotation properties.

▼ Click here to expand..
Spindle Speed.

Specify the direct spindle speed in revolutions per minute (RPM) or cutting speed in meters per minute (CSS). In the second case, the CNC system continuously calculates and adjusts the spindle's revolutions per minute based on the current workpiece diameter. Instead of a fixed revolutions per minute, only the maximum allowable value is specified.

Max RPM.

If the spindle speed is set via cutting speed, it is necessary to specify the maximum rotational speed to restrict its increase when the diameter changes.

RPM before CSS.

In turning operations, this parameter enables RPM mode at the tool change position and activates CSS mode following the approach moves.

If the tool is positioned at the tool change point and the spindle is activated in CSS mode, its rotation will be slow since the tool is far from the workpiece's rotational axis. During the first rapid movement when the tool approaches the workpiece, the spindle accelerates significantly, causing extreme loads on the machine. To prevent this, some machine manufacturers recommend initially running the spindle in constant RPM mode and switching to CSS after the tool approaches the workpiece. When the "RPM before CSS" option is enabled, the spindle first runs in RPM mode at the speed corresponding to the first cutting feed point, then the approach movement is executed, and finally, the mode switches to CSS.

Range.

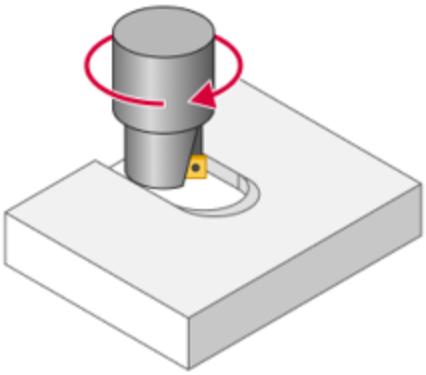
Specify the range number required to obtain the desired spindle speed on a machine with a mechanical gearbox.

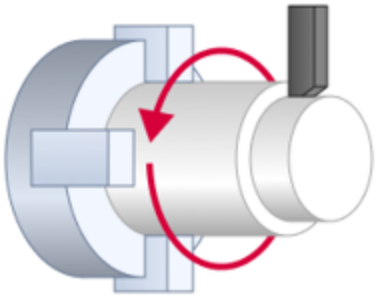
Rotation direction.

Configure the spindle rotation direction.

Forward.

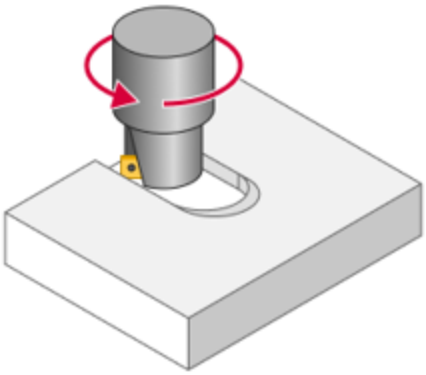
This is a clockwise spindle rotation when viewed from the motor side, represented in the NC program by M3. If a tool is installed, this direction tightens a right-hand screw. In turning operations, the spindle rotates towards the operator.

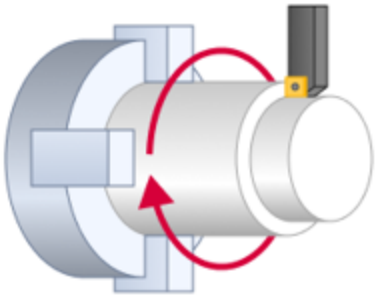




Reverse.

This is a counterclockwise spindle rotation when viewed from the motor side, represented in the NC program by M4. If a tool is installed in the spindle, this direction corresponds to tightening a left-hand screw. In turning operations, the spindle rotates away from the operator.





Grip RPM. The subspindle performs the part capture at this RPM.

Plunge speed.

In a Roughing Waterline operation, a specific spindle speed can be defined for the cutter's plunge movement into the material when milling enclosed surfaces like pockets. Details.

Coolant:

This parameter group controls the coolant supply to the cutting area. The postprocessor uses these parameters to generate the appropriate commands for enabling and disabling coolant flow in the NC program. Users can define coolant supply methods and configure their parameters in the Coolants section under the Equipment tab. Details.

▼ Click here to expand..
Flood.

Activating this parameter initiates coolant supply through the nozzle.

Mist.

Compressed air sprays the liquid to produce a mist and transports the air-liquid mixture to the cutting zone.

Tool.

Activating the parameter enables coolant delivery under pressure through the tool channels.

Coolant 4.

Activating the parameter enables coolant delivery to the tool using an alternative method, such as an air blast.

Cut Feeds:

This parameters group determines the feed rates for cutting, tool engagement, approach, and retraction. Each parameter can be specified in speed units such as mm/rev, mm/min, and for milling operations, also in mm/tooth. All speeds, except for the working feed rate, can be defined as a percentage of the working feed rate.

▼ Click here to expand..
Work feed (in lathe operations Cut Feed).

The primary feed rate value maintained for stable cutting during the process. In roughing operations material removal will be performed at this speed, in finish – detail surface machining.

Calculable.

In the milling operations the feed value can be either permanent or calculated, depending on the slope angle of every elementary toolpath section. When assigning a calculated feed, defining will be feed values and coefficients when moving down, horizontally and up. The real feed values when moving down, horizontally or up will be equal to multiplication of the corresponding correction coefficient to the feed value. With intermediate values of slope angle of an elementary toolpath section, the real feed value will be calculated proportionally to the defined border values. For example, with a feed values equal to up 300, horizontally – 200, down – 100, the real feed value on a section with tool movement up under angle of 45 degrees will be equal to 250. Use of the calculated feed, allows the user to decrease the machining time due to more flexible control over cutting modes.

Up feed value.

Feed rate during vertical upward motion of the tool.

Down feed value.

Feed rate during vertical downward motion of the tool.

Engage feed.

The feed rate value used for engage movement. Details.

Retract feed.

The feed rate value used for retract movement. Details.

First feed (in lathe operations First Cut Feed).

The feed rate applied during working passes at the initial cutting depth. First feed is useful when machining a workpiece with an initial tough exterior (skin).

Finish feed (in lathe operations Finish Cut Feed).

The feed rate value used for finishing passes in operations with separate roughing and finishing stages. It is advised to use when need to obtain a surface of high quality after a roughing operation.

Plunge feed.

Feed applied to tool plunge moves, for example, it ia applied for plunging to a lower machining layer in waterline roughing and pocketing operations. Details.

Chip Breaking Feed.

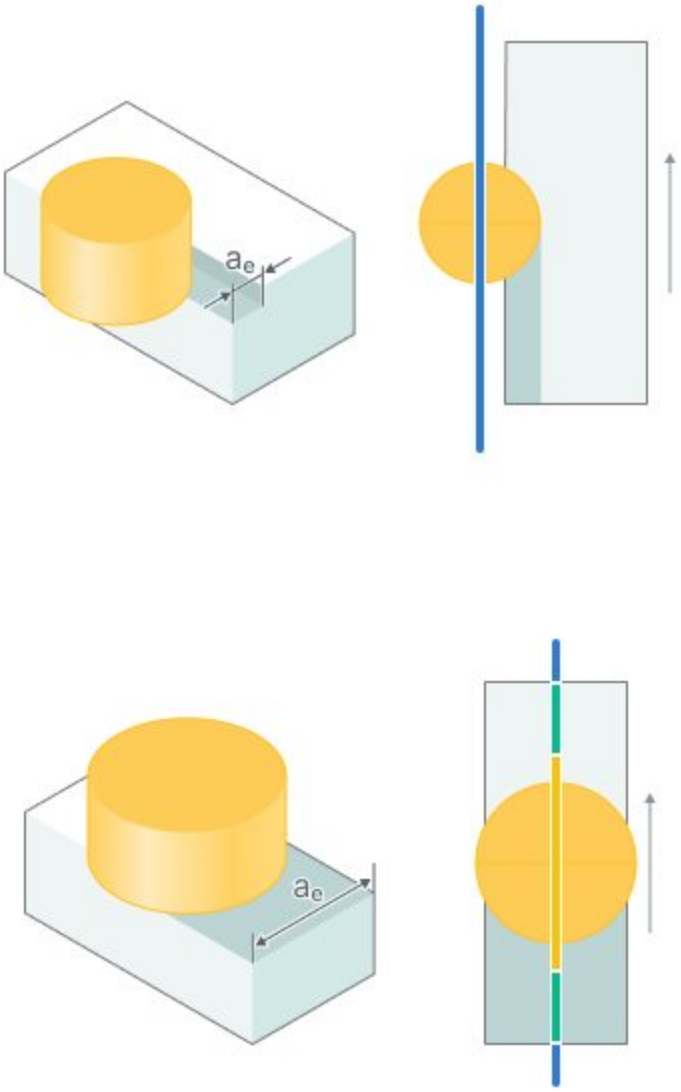
In lathe operations, this value specifies the speed for retracting from the workpiece and repositioning during chip breaking.Details.

Adaptive feedrate.

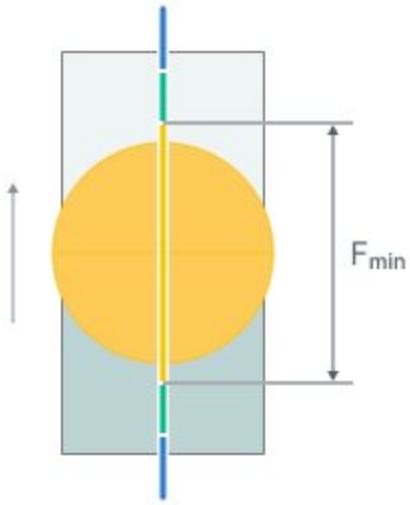
The feature automatically adjusts the toolpath feedrate based on the actual tool engagement. So it reduces feedrate in tight corners where the tool engagement approaches 100% and increases feedrate as the tool engagment approaches 0% (the air cutting).

To activate this parameter, the main condition is the width of the overlap.

The tool engagement approaches 100% (Ae~100%). Option is active. The zones are marked in black in CAM system.

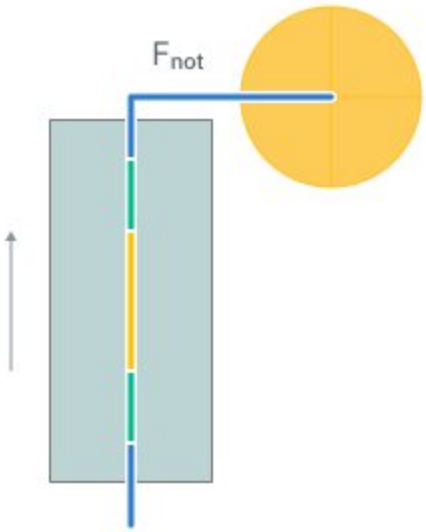


Full Cut Feed. Minimum final feed, in percents of the Work Feed.



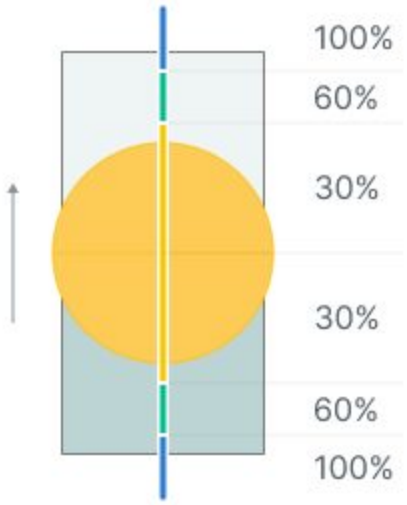
Fmin - Full Cut Feed

No Cut Feed. Feedrate on return from the zone (without cutting), in percents of the Work Feed.



Fnot - No Cut Feed

Feed Increment. Step to change feed.



Mark overloads. Allows you to mark the segments where the tool engagement is maximal in the Simulation tab. In the parameter you can specify the value from the work feed that needs to be marked. Details

Override Feeds.

This parameter becomes available in lathe operations, and its activation allows defining separate feed rate settings—Cutting, Engage, Retract, Finish Cutting, and Traversal feeds—for designated cycles in the Job Assignment.

Non-cut Feeds:

This parameter group defines the speeds of auxiliary movements, including approach, return, and transitions between working passes. Each parameter can be set in speed units such as mm/rev, mm/min, and for milling operations, also in mm/tooth. Additionally, speeds can be specified as a percentage of the working feed rate or as rapid feed.

▼ Click here to expand..
Rapid feed.

The feed at which all rapid tool transitions (positioning) are performed. Its value is the rapid motion velocity for the current machine. Movements at this speed are output to the NC program under the G00 address. This figure is used to calculate accurate machining process time. Toolpath sections, performed at rapid feed are displayed in red. When switched to rapid feed, the system creates the RAPID command in the CLDATA program.

Short link feed.

The feed rate applied to transition movements executed according to the Short Links rule. Details.

Long link feed.

The feed rate applied to transition movements executed according to the Long Links rule. Details.

Approach feed.

The feed rate applied for the approach movement, covering the distance from Feed Distance to the start of the Plunge or Engage. Details.

Approach from safe surface feed.

The movement speed value from the Safe Surface to the Feed Distance. Details.

Return to safe surface feed.

The movement speed value from the Feed Distance to the Safe Surface level. Details.

Transition on safe surface feed.

The movement speed value during the links over the Safe Surface. Details.

Traversal Feed.

In lathe operations, this speed defines the transition rate from one work pass to the next.

Return feed.

In milling operations, this is the movement speed from the end of the Retract motion to the Feed Distance level. In hole machining operations, this is the retraction speed in some drilling cycles. e.g. deep drilling cycles with chip removal or breaking. Details.

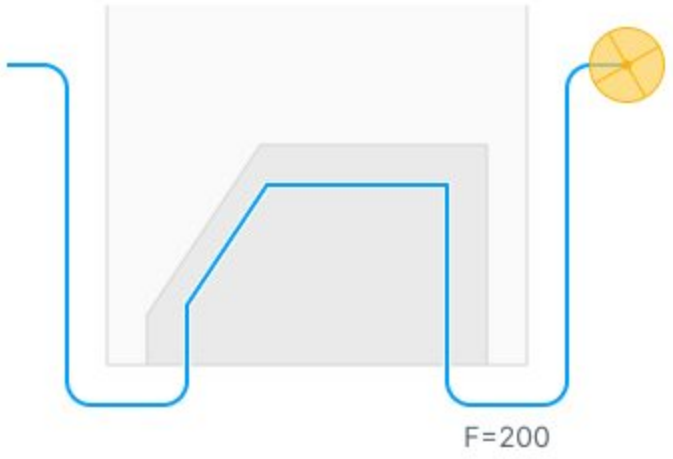
Rapid idle moves (Rapid non-work moves).

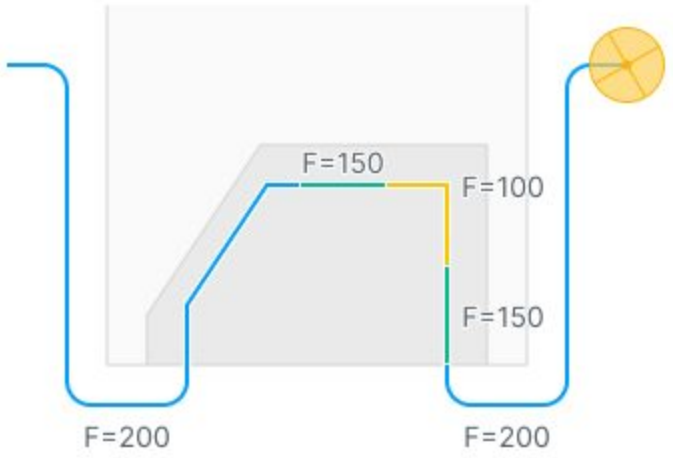
When this parameter is activated, all the tool retraction movements from the machining areaб intended to be performed at Return feed rate are instead executed at Rapid feed.

Corner control:

This group of parameters allows you to to add some specific behaviour to the toolpath corners. It is located on the bottom of "Feeds/Speeds" page. Details

Corner control





Application Area:

This group of parameters allows you to to add some specific behaviour to the toolpath corners. It is located on the bottom of "Feeds/Speeds" page.

> Note
>
> The corner control is only applied to the toolpath segments with an effective feed, which you can set through the Effective feeds parameter on the "Parameters" page.
>
> For the subtractive operations, the default set of the "Effective feeds" includes:

> - Work feed
> - Engage/Retract feeds
> - First feed
> - Finish feed

You can find the full list of operations that support "Corner control" option below:

▼ Click here to expand..

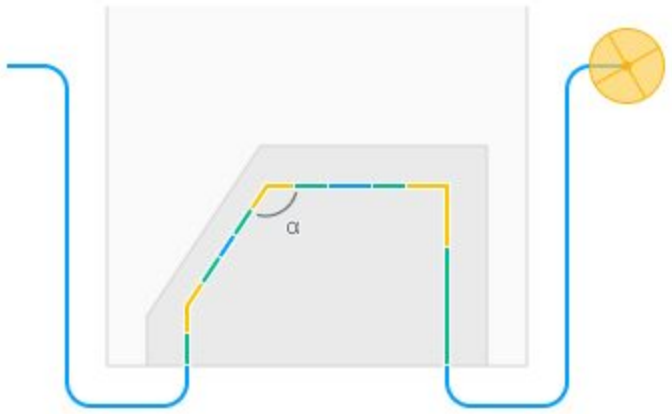
- 2D
  - 2D contouring
- 2.5D
  - 2.5D contouring
- 3D
  - 3D contouring
- 3D/5D advanced
  - 6D contouring
- 4D Rotary
  - 4D contouring
- Rest machining
  - Chamfering
- Cutting
  - Jet cutting
  - Jet cutting 5D
- Additive
  - Area cladding
  - Curve cladding
  - Cladding 3D
  - Cladding 5D
  - Non-planar slicing
- Welding
  - Welding 6D
- Spraying
  - Contour spraying
  - Surface spraying
  - Morph spraying
  - Rotary spraying
- Legacy
  - Welding 5D
  - 5D contouring

There are two main groups of parameters that allow you to control corners: Detection method and Action type.

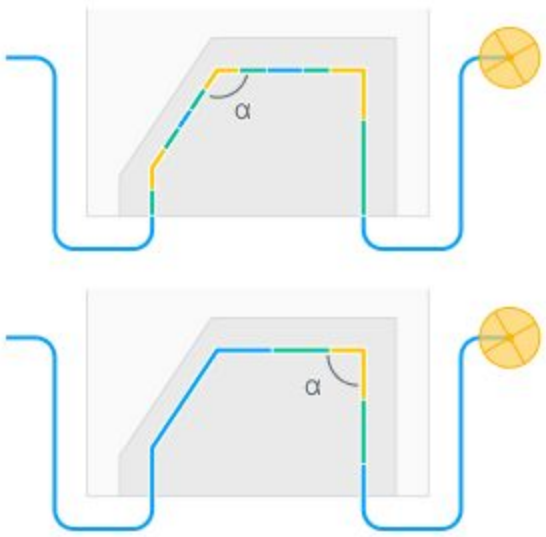
Detection Method:

"Detection method" group of parameters shows the system how to detect corners. Now, only one parameter is available - By angle.

By Angle.



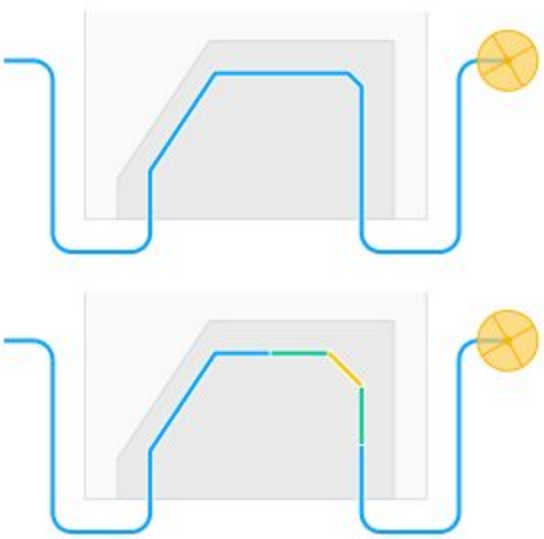
By angle. parameter defines the angle value in degrees. Two consecutive segments of the toolpath will be detected as a "corner", if the angle value between them is less than or equal to the defined value.



For example, the first angle with a value of 102 degrees will be marked as a corner in the system if the "Angle" value is set to 110 degrees.

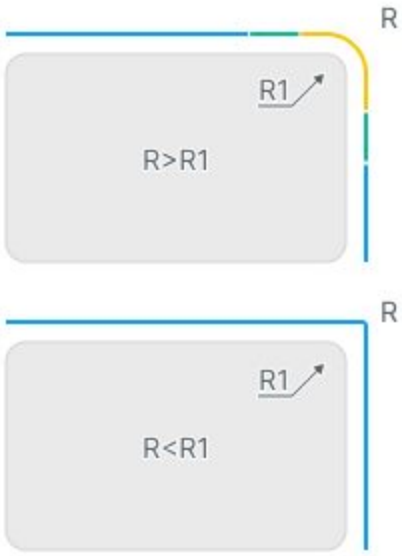
Also there are two additional parameters for more precise adjustment: Minimum line length and Minimum arc radius.

Minimum line length parameter allows you to smooth the toolpath before running the corner detection algorithm using the desired tolerance (or we can say the minimum line length) value. So, the detection method is applied to a smoothed toolpath, which ensures there are no false corner detections or missed corners. *Note: this parameter doesn't affect the final toolpath. It is only used for internal detection algorithms.*



Minimum arc radius. Sets the radius threshold R used to detect rounded corners. In the illustration, R1 is the radius of the fillet on the model.

- R > R1: the fillet is treated as a corner, and the selected Action type is applied.
- R < R1: the fillet is not treated as a corner by this radius criterion.



Action Type:

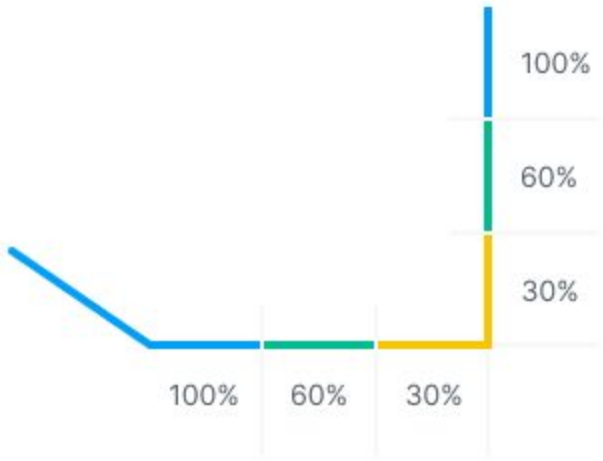
"Action type" group of parameters controls the system's behavior at toolpath corners. Three modes are available: Adjust feed, Adjust power and Adjust feed & power.

> Note
>
> The last two modes ("Adjust power" and "Adjust feed & power") are only available for Cladding 5D and Non-planar slicing operations and only when the Power function is enabled on the Parameters page.

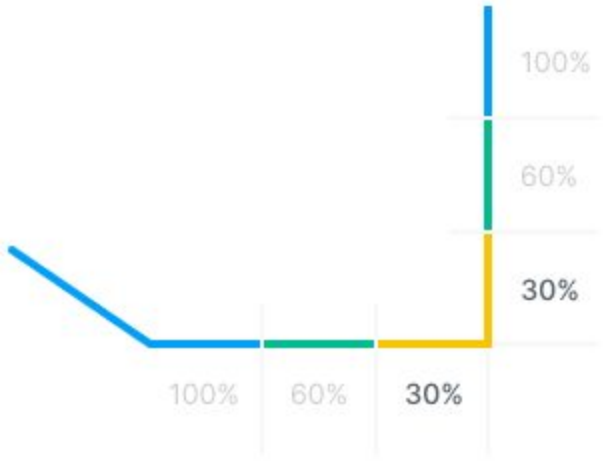
This group consists of 3 available options: Desired value, Number of steps and Step length.

Adjust feed, Adjust power, Adjust feed & power

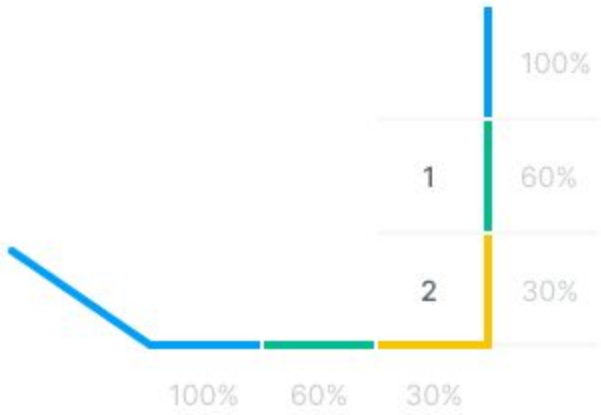
These modes allow you to adjust the feedrate and/or power at detected toolpath corners separately or simultaneously using a step-wise approach.



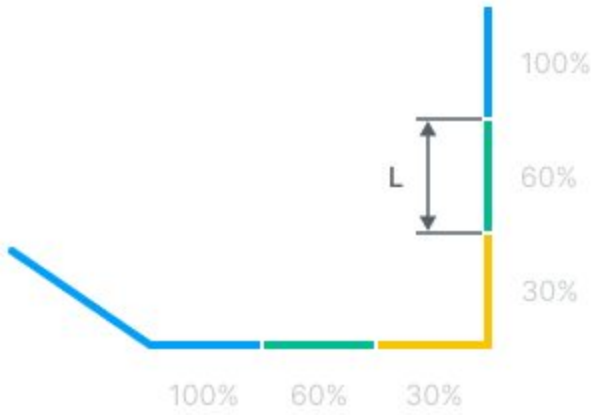
Desired value defines the target value as a percentage of the base parameter in the detected corner. Values below 100% will reduce the parameter, while values above 100% will increase it.



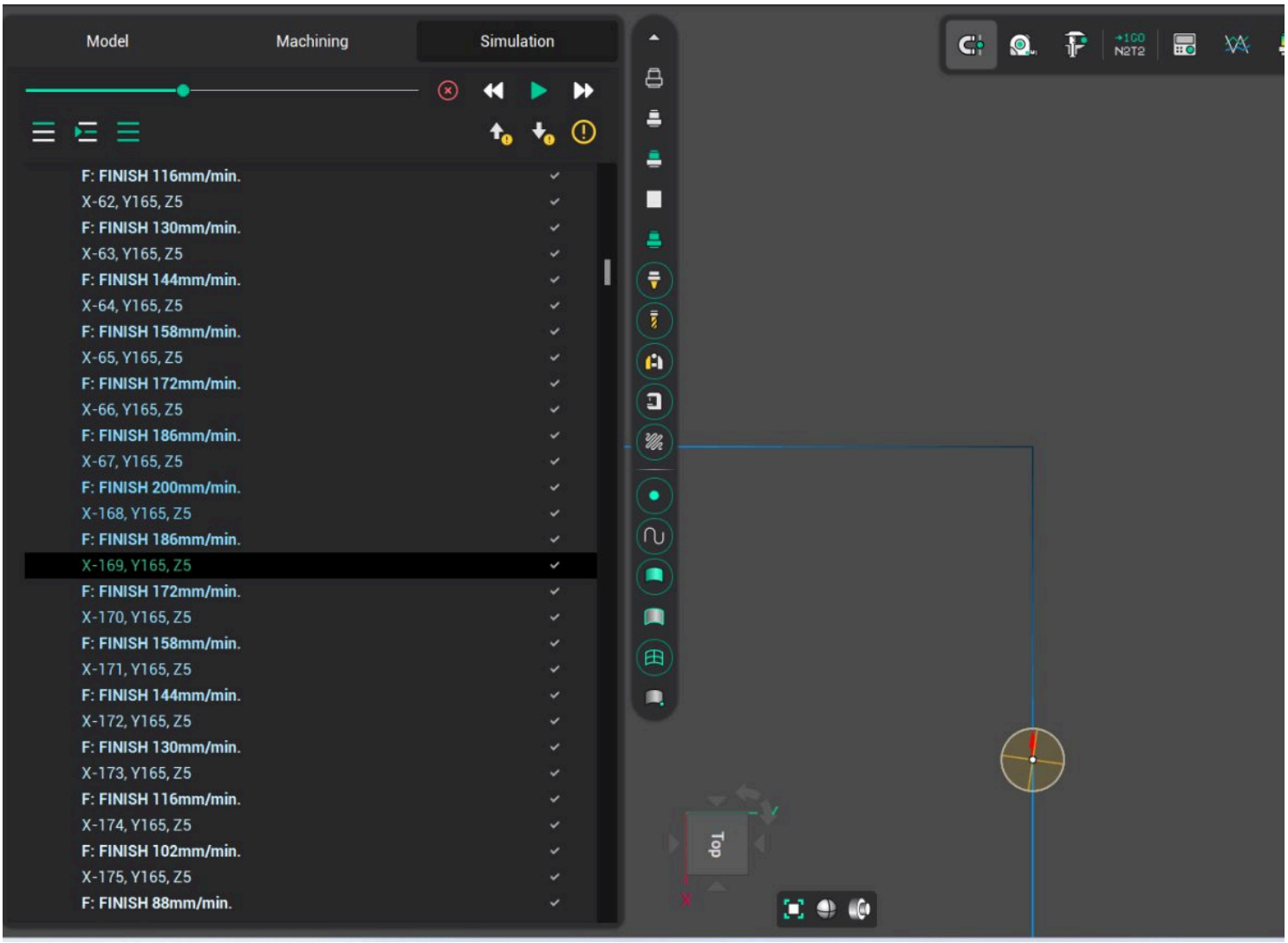
Number of steps defines the number of steps required to reach the desired value.



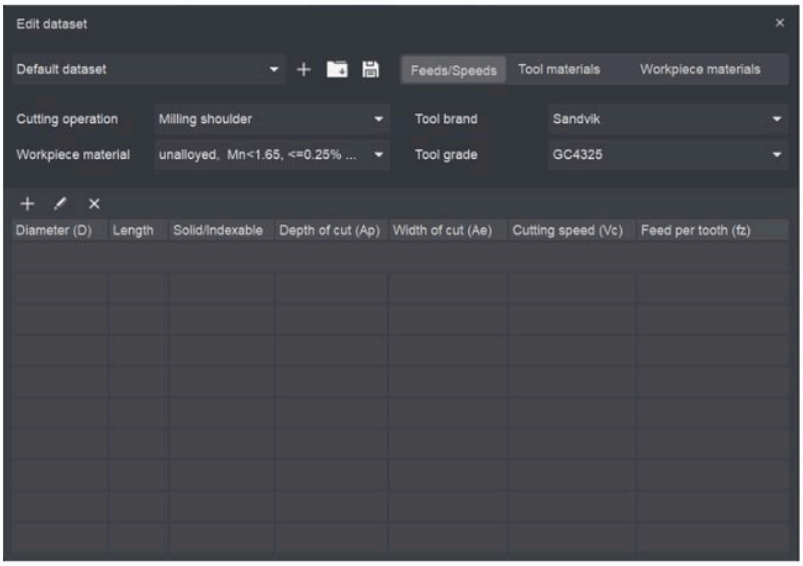
Step length defines the length of one step.

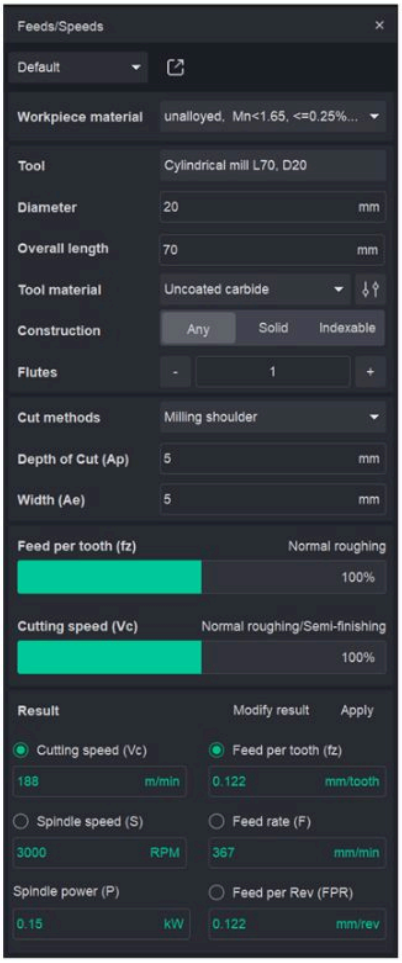


The effect of these parameters can be viewed on the Simulation page. Toolpath segments within corners will be displayed with a color gradient. Additionally, new commands with reduced feedrates will be added to the CLDATA and similarly highlighted with a gradient.


Model
Machining
Simulation
F: FINISH 116mm/min.
X-62, Y165, Z5
F: FINISH 130mm/min.
X-63, Y165, Z5
F: FINISH 144mm/min.
X-64, Y165, Z5
F: FINISH 158mm/min.
X-65, Y165, Z5
F: FINISH 172mm/min.
X-66, Y165, Z5
F: FINISH 186mm/min.
X-67, Y165, Z5
F: FINISH 200mm/min.
X-168, Y165, Z5
F: FINISH 186mm/min.
X-169, Y165, Z5
F: FINISH 172mm/min.
X-170, Y165, Z5
F: FINISH 158mm/min.
X-171, Y165, Z5
F: FINISH 144mm/min.
X-172, Y165, Z5
F: FINISH 130mm/min.
X-173, Y165, Z5
F: FINISH 116mm/min.
X-174, Y165, Z5
F: FINISH 102mm/min.
X-175, Y165, Z5
F: FINISH 88mm/min.
Top

Feeds/Speeds (Calculate)



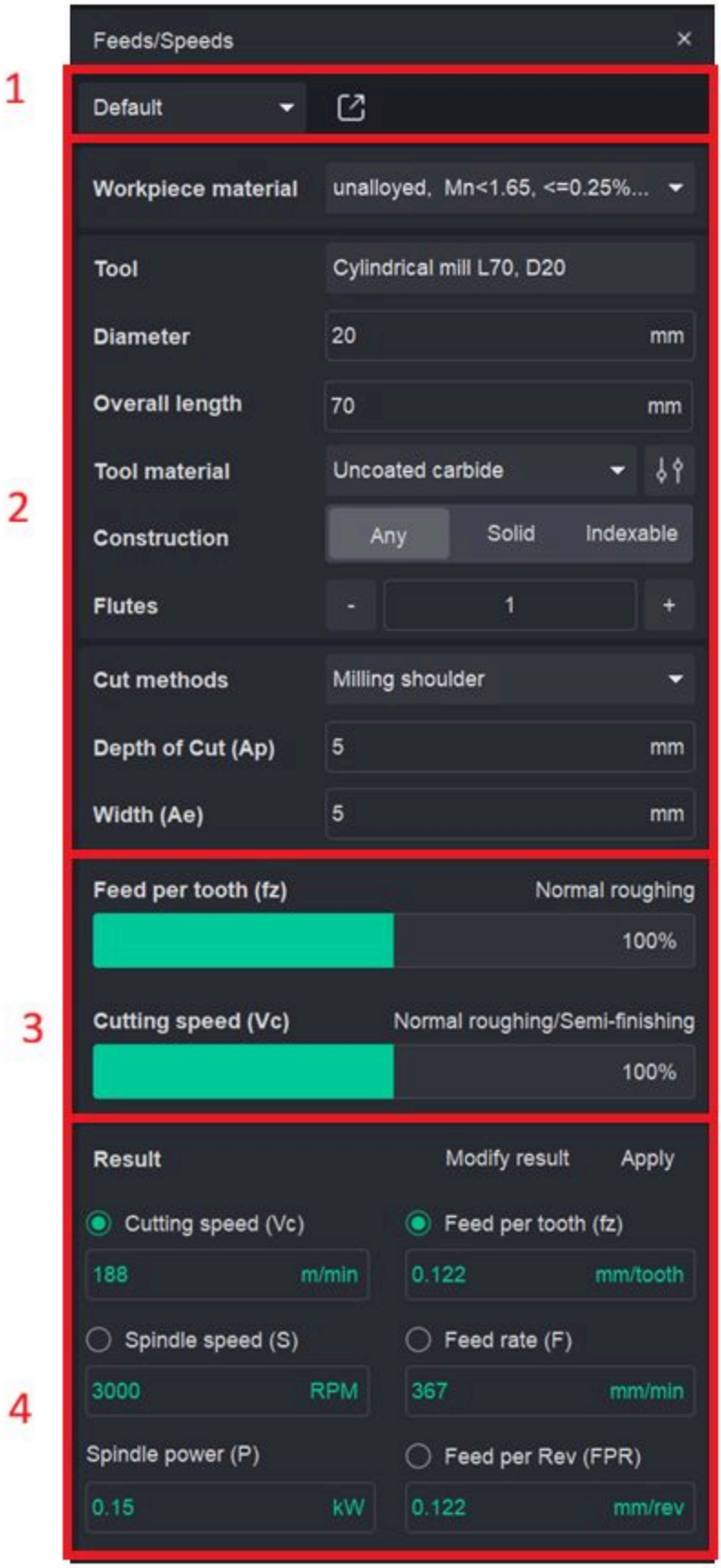


Application Area:

The Feeds/Speeds calculator computes cutting parameters — cutting speed, spindle speed, feed per tooth and feed rate — from the selected tool, workpiece material and cut method. Computed values are written into the machining operation and can be fine-tuned manually before they are applied. The calculation of cutting parameters is performed automatically in real-time upon modification of any parameter, without the need to press a dedicated Calculate button.

The calculator panel is divided into four areas, top to bottom:

1. Toolbar area. It allows the user to manage custom tables for populating data used by the calculator.
2. Input area. Defines the tool, the workpiece material and the cutting conditions.
3. Load sliders. Set the cutting load relative to the recommended range.
4. Result. Shows the computed cutting parameters.

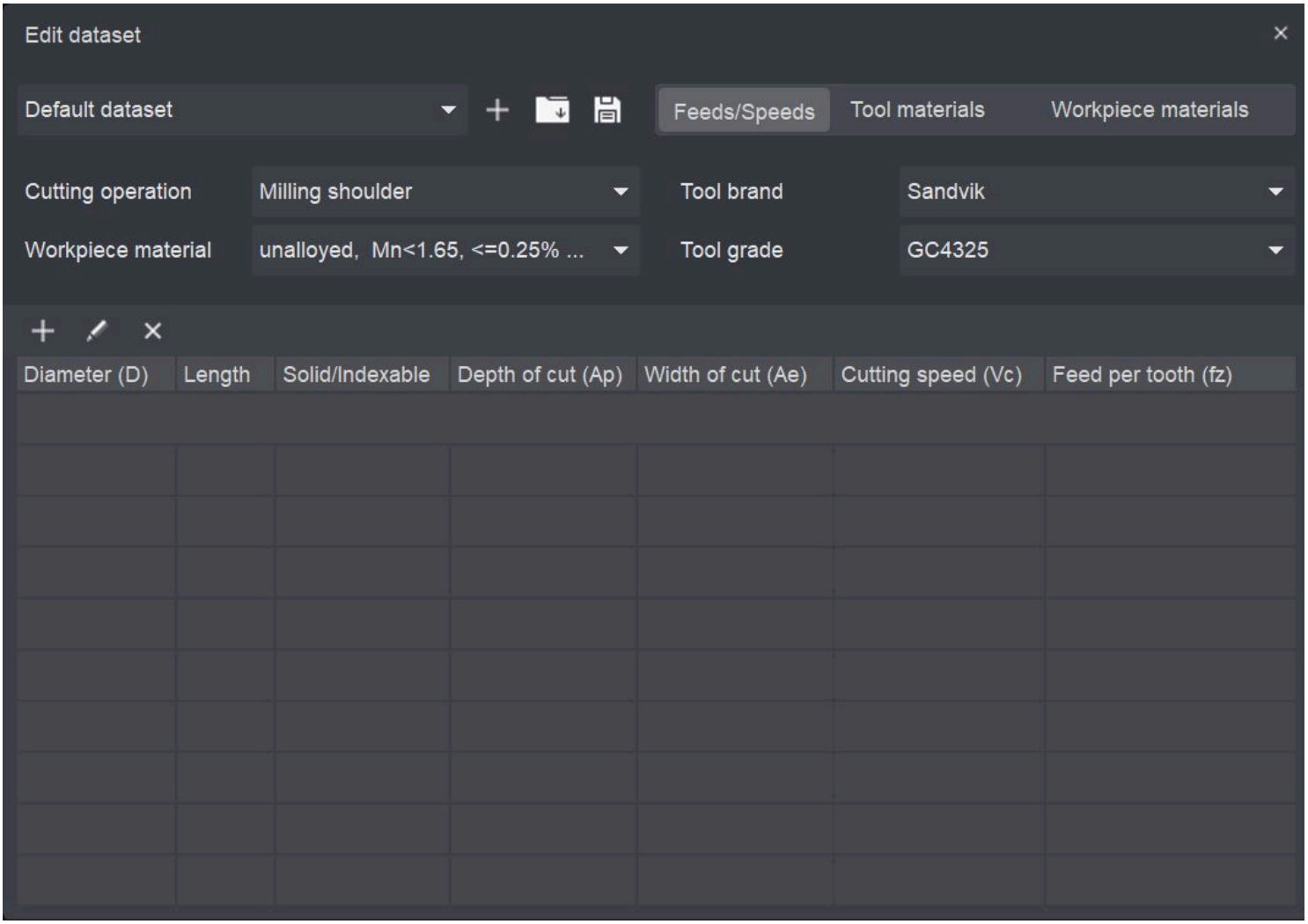


Toolbar.

Dataset selector. Selects the active cutting data dataset. Defaults to Default dataset. These tables allow the user to define their own cutting parameters, tool materials, and workpiece materials.

Open editor. Opens the Edit dataset window for managing the tool and workpiece material databases.

▼ Click here to expand..



The filters and displayed list dynamically adapt according to the type of the selected table. (Feeds/Speeds, Tool materials, Workpiece materials)

Toolbar.

Dataset selector. Selects the dataset to edit.

Add. Creates a new dataset.

Open. Loads an existing dataset.

Save. Saves the current dataset.

Tabs. Feeds/Speeds, Tool materials, Workpiece materials — switch between the edited sections.

Feeds/Speeds tabs.

Filters

Define which records are shown and edited:

Cutting operation. The operation type the data applies to.

Workpiece material. The material being filtered.

Tool brand. Manufacturer (for example, Sandvik).

Tool grade. Manufacturer grade (for example, GC4325).

ISO group. Material group — P M K N S H O. Select the group the material belongs to.

Subgroup. Refines the ISO group (for example, Unalloyed).

List. It displays a data list depending on the selected table.

Input parameters.

Workpiece material. The material being machined, selected from the dataset (for example, unalloyed, Mn<1.65, <=0.25%C).

Tool. The tool the calculation applies to (for example, End Mill L70, D20). Taken from the active operation.

Diameter. Tool cutting diameter, mm.

Overall length. Total tool length, mm.

Tool material. Cutting material of the tool (for example, Uncoated carbide). The icon to the right switches the material preset.

Construction. Tool body type: Any, Solid or Indexable.

Flutes. Number of cutting edges. Adjust with the – / + buttons.

Cut methods. The milling strategy (for example, Milling shoulder).

Depth of Cut (Ap). Axial depth of cut, mm.

Width (Ae). Radial width of cut, mm.

Load sliders.

Two sliders set the working load as a percentage of the recommended range. The caption above each slider shows the corresponding machining condition.

Feed per tooth (fz). Sets the feed load. The caption shows the regime, for example Normal roughing. Expressed as 0–100%.

Cutting speed (Vc). Sets the speed load. The caption shows the regime, for example Normal roughing/Semi-finishing. Expressed as 0–100%.

Moving a slider re-evaluates the dependent result fields when you press Calculate.

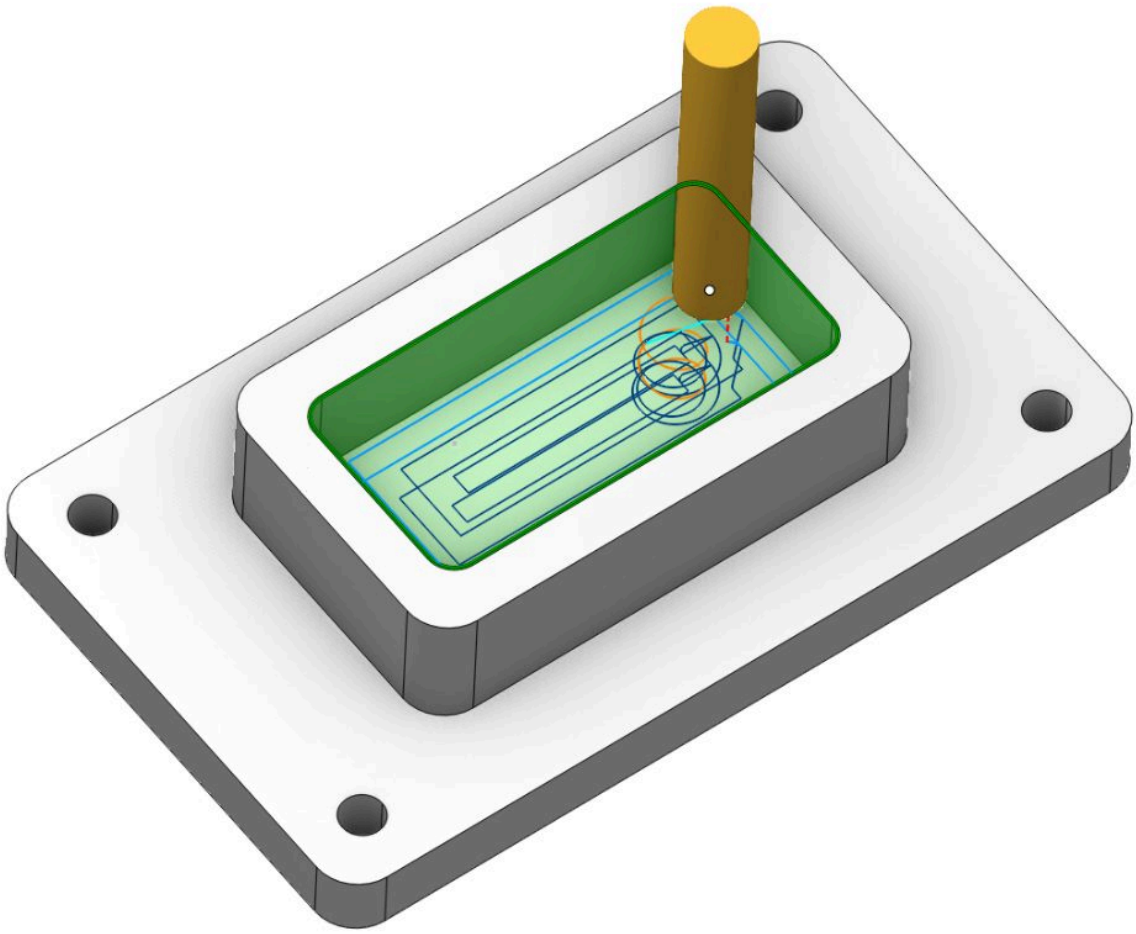
Result.

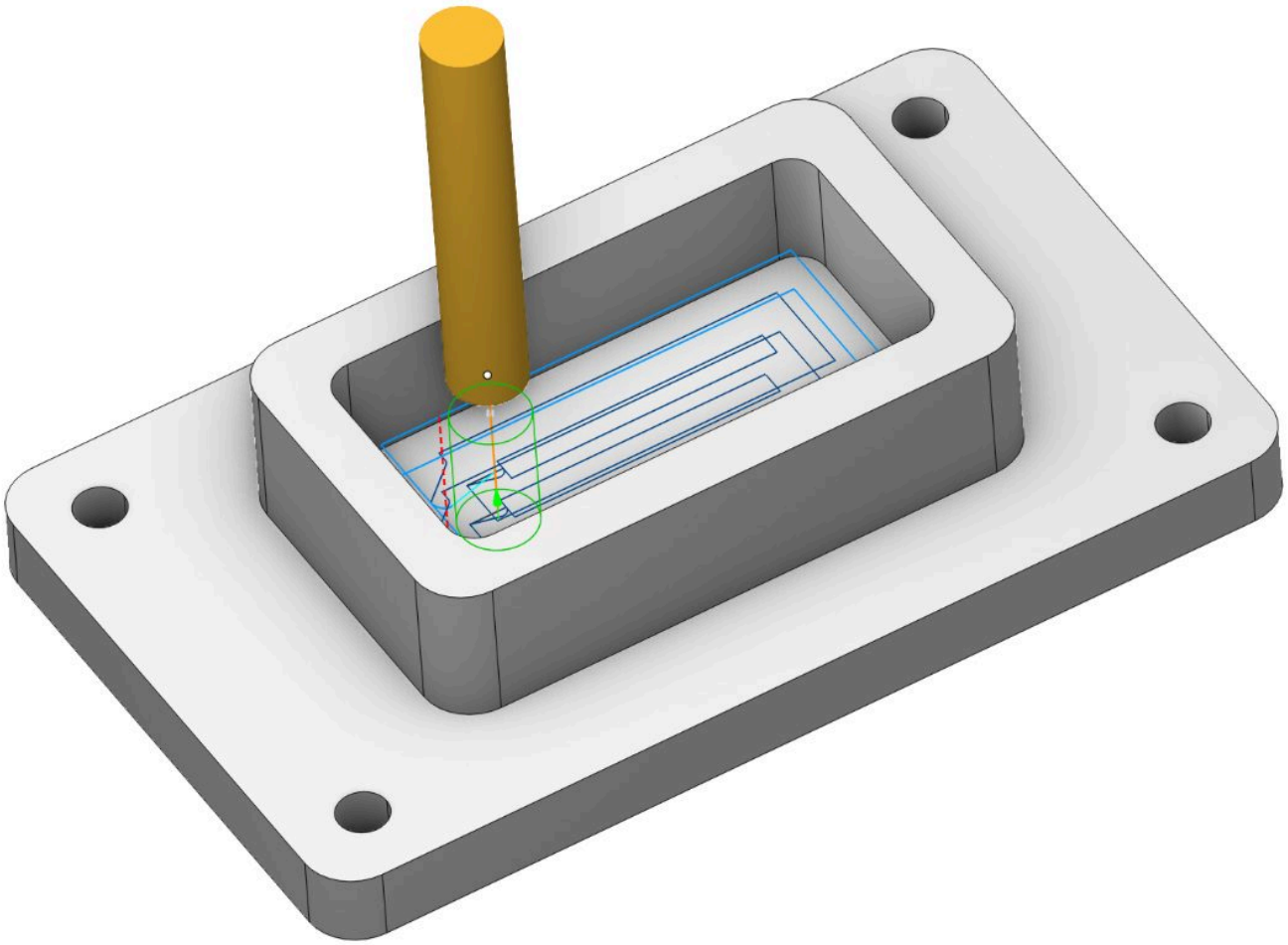
Parameter	Symbol	Unit	Example
Cutting speed	Vc	m/min	153
Feed per tooth	fz	mm/tooth	0.115
Spindle speed	S	RPM	2435
Feed rate	F	mm/min	280
Spindle power	P	kW	0.15
Feed per Rev	FPR	rev/min	0.115

The computed values. Click Modify result to edit a value manually.

The Apply function enables the transfer of computed cutting parameters to a designated machining operation. Within this interface window, the user specifies the format for data output to the operation, including options to define parameters as cutting speed and feed rate in mm/min or as spindle speed and feed rate in mm/rev.

Holes (Drill points)





Application Area:

In the Roughing waterline (Details) and Pocketing (Details) operations, if it is impossible to approach the area being machined from outside, then the approach can be performed through a drill point with appropriate parameters. All drill points for a machining process are registered in a unified list. This list contains data on the point coordinates, hole depth and tool diameter. The user can add and remove points in the list.

The method of defining the hole:

Center. Create hole by center point.

Create. Create hole by coordinates input. Details

Recognize. Automatically recognize holes in the part. Details

Pattern. Create holes array by pattern. Details

Properties. Displays the properties of an element. You can also call this menu by double clicking on an item in the list. Details

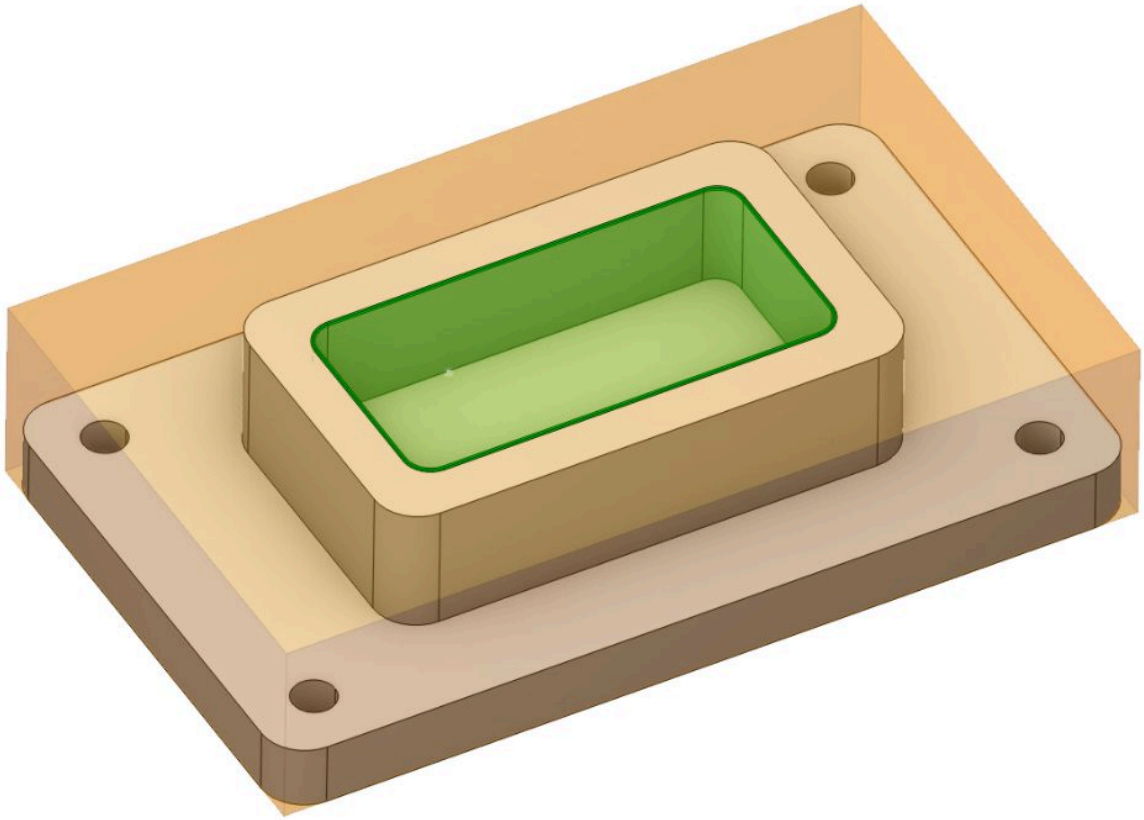
Sorting. Sorting using some parameters to sort holes list. Details

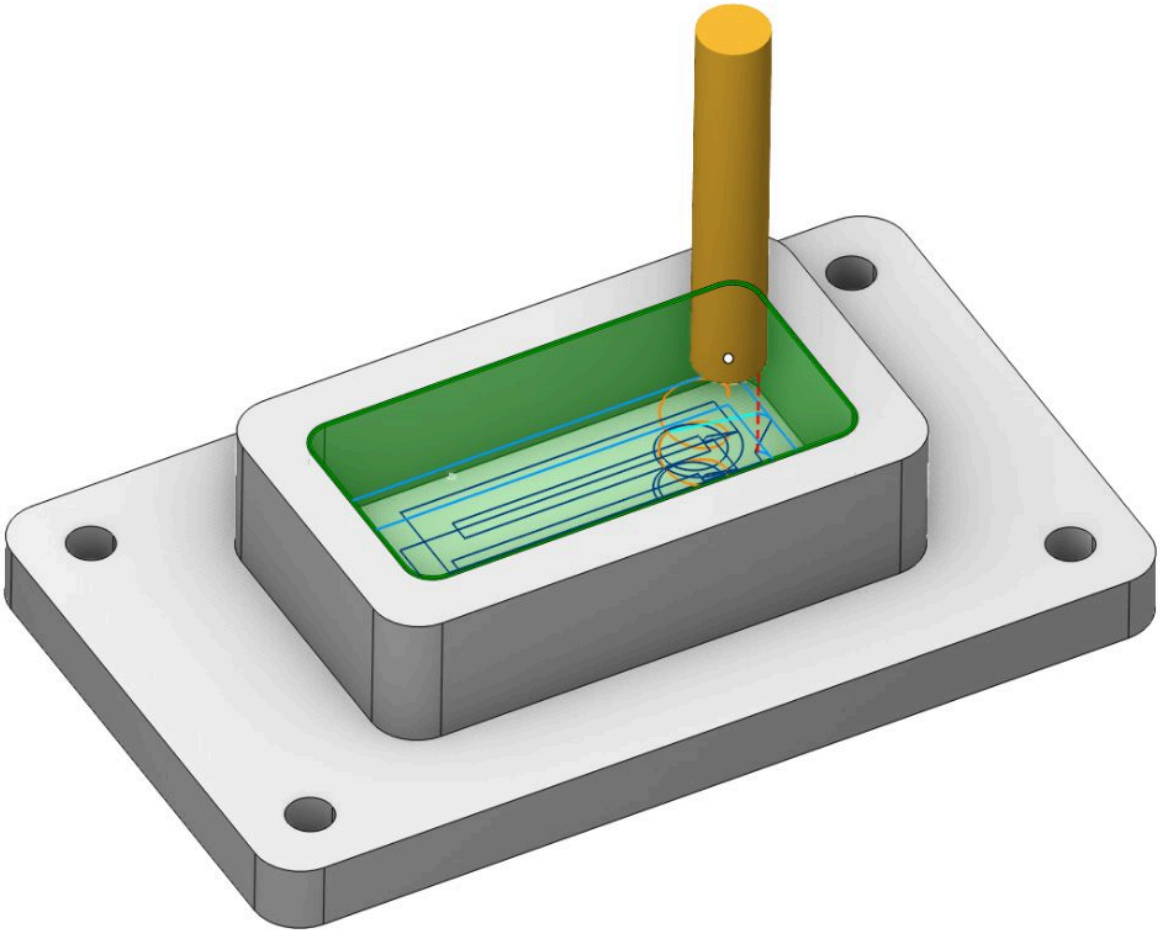
Delete. Removes an item from the list.

When removing stock material from an isolated area (e.g., cavity, which cannot be approached from outside on the current level), the system first tries to select an appropriate point by coordinates, depth and diameter from the list. If there is no suitable point in the list, a new point will be automatically calculated and added to the current list.

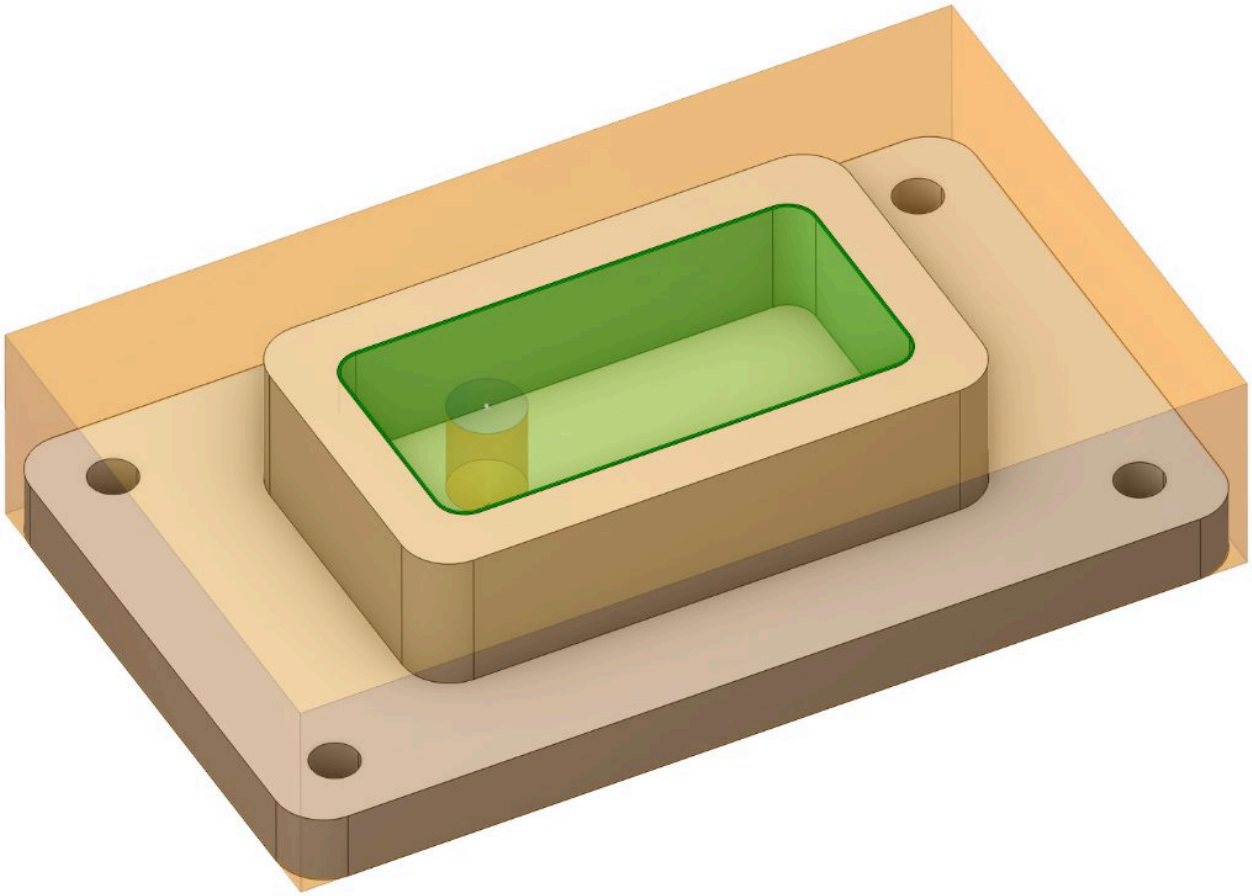
For example:

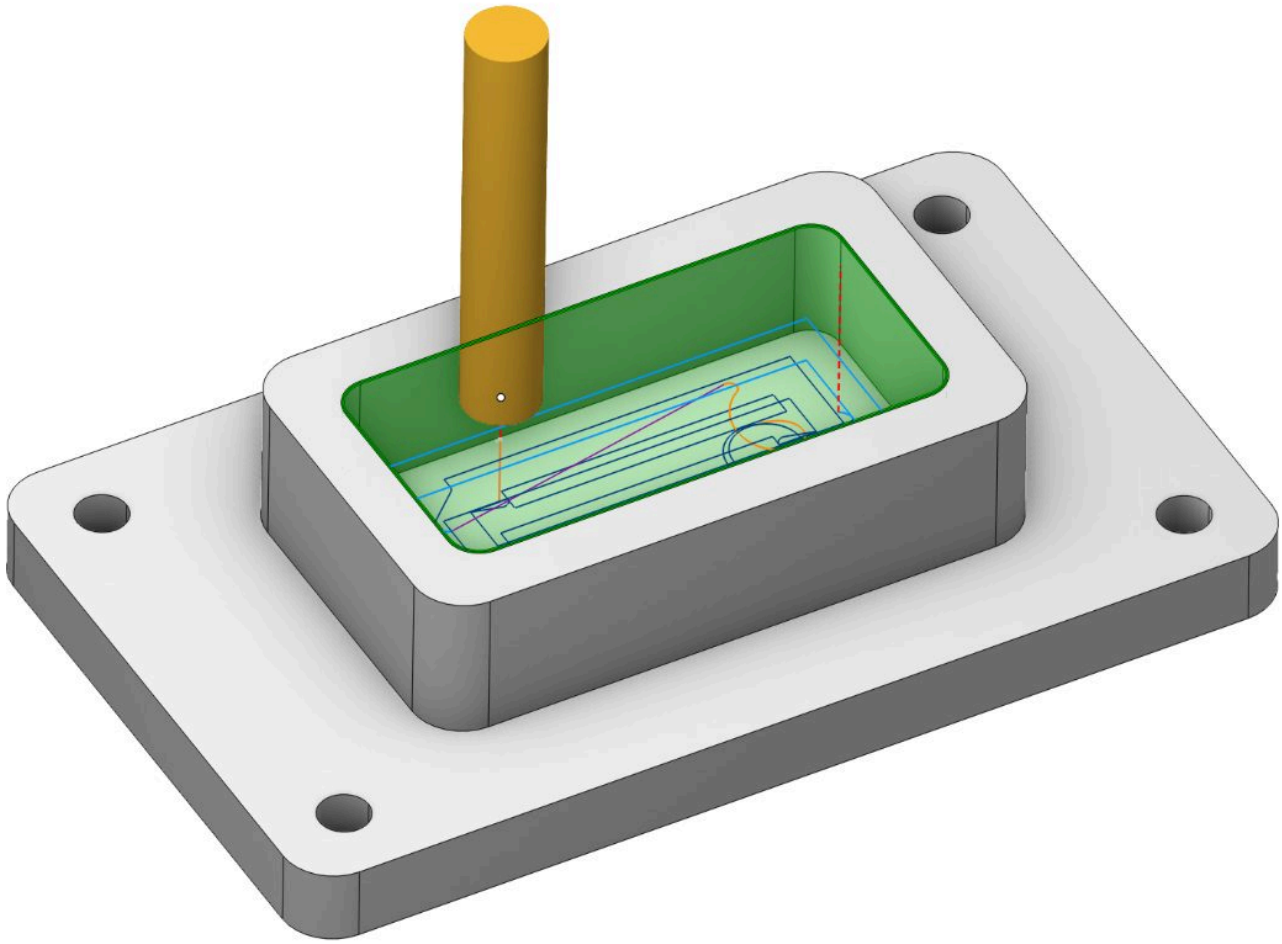
The project uses one operation, Roughing waterline.





The project uses the following operations: Hole Machining, followed by Waterline Roughing.

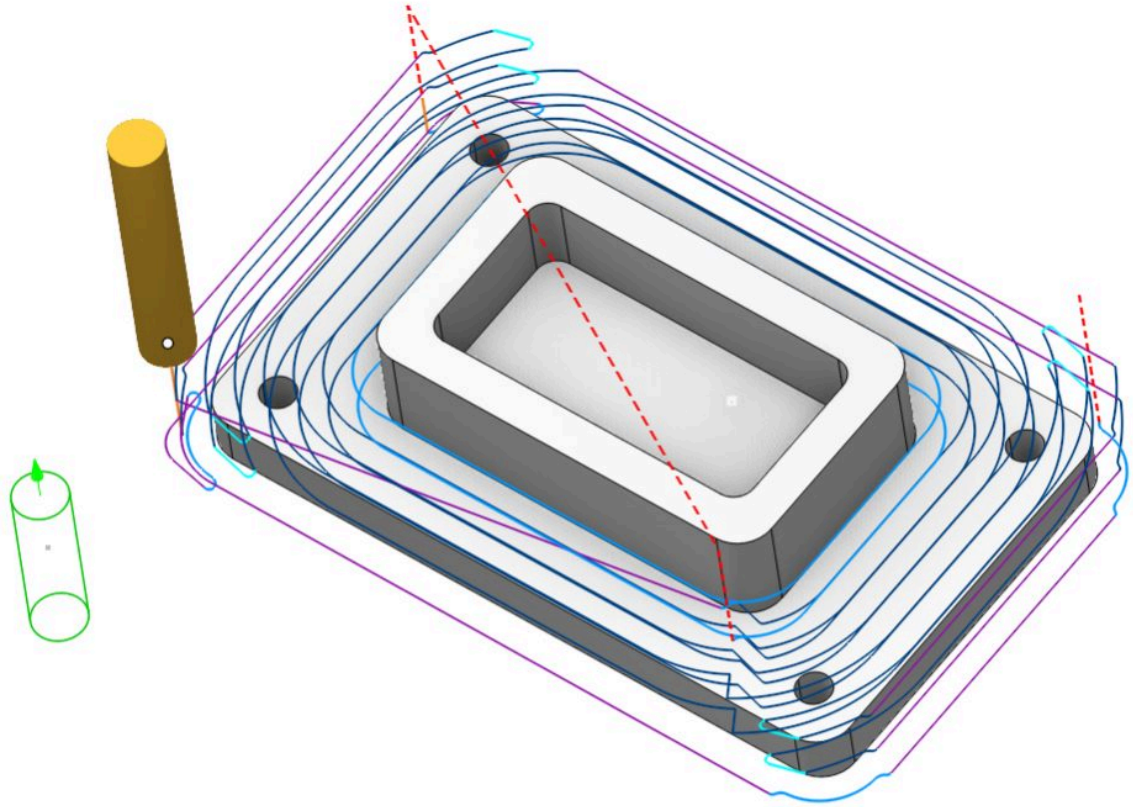


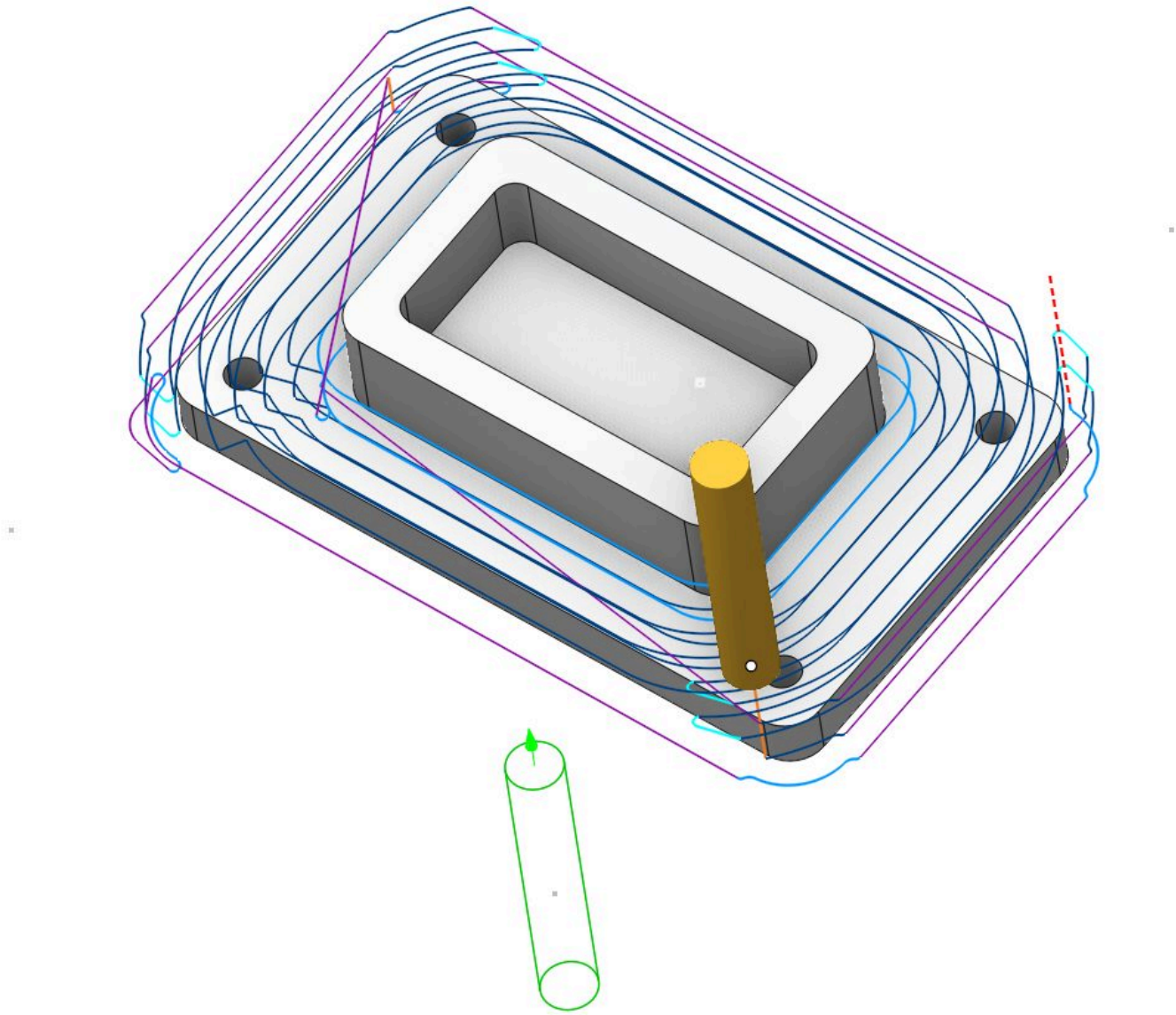


Note: If accurately defined drill (plunge) points are required in the operation, then the parameters for the point(s) should be added to the list, before calculating the operation.

Points from the unified list can be defined as parameters for the hole machining operation.

If in the list of drill points, there are points that lie outside the workpiece borders, then in the waterline roughing and pocketing operations they will be perceived as the recommended tool lowering points outside the workpiece. If the area being machined can be approached from outside, then the system searches for the 'plunge' point in the list (coordinates, depth to which it is possible to sink, maximum distance to the area being machined, which corresponds to the diameter). If there are several such points, then the closest will be selected. If no suitable points are found, then the tool plunge will be performed at an arbitrary point.

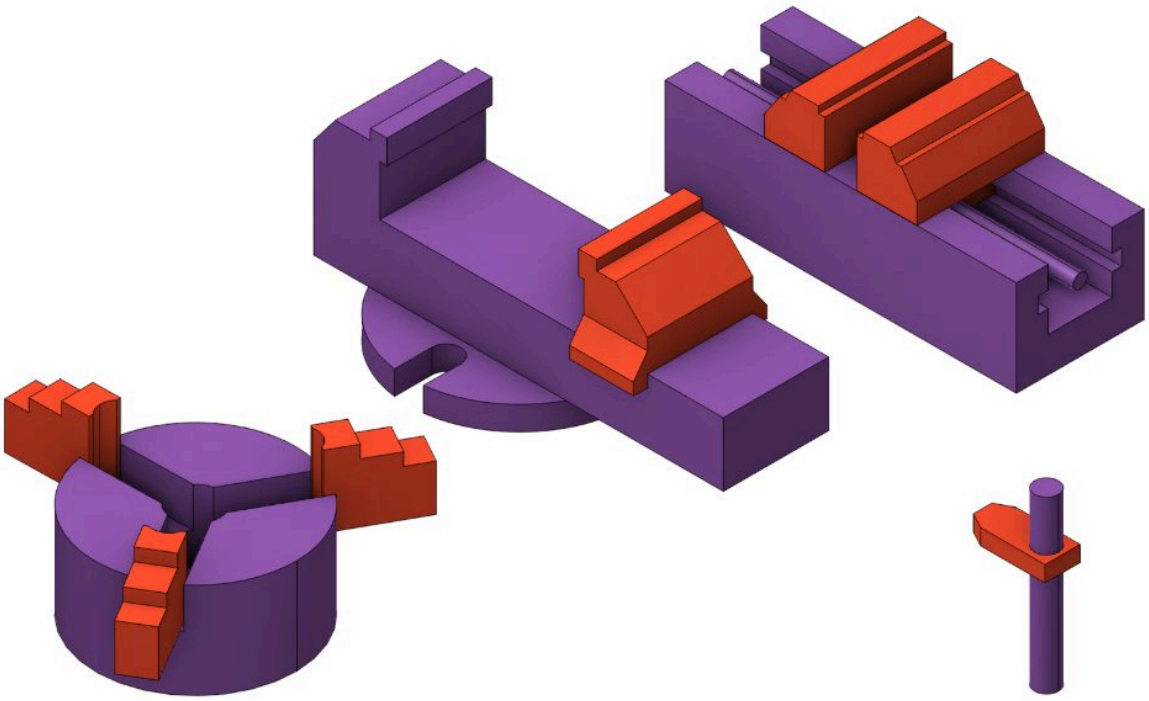




Export selected drill points to DXF:

Select the required points in the holes list and choose Export selected in DXF from the context menu to save the selected points to a DXF file.

Fixtures



Application area:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. While calculating a toolpath the fixtures are used as the models the tool should not collide with while machining the part. The fixtures can be constructed from solids, surfaces and features generated from curves.

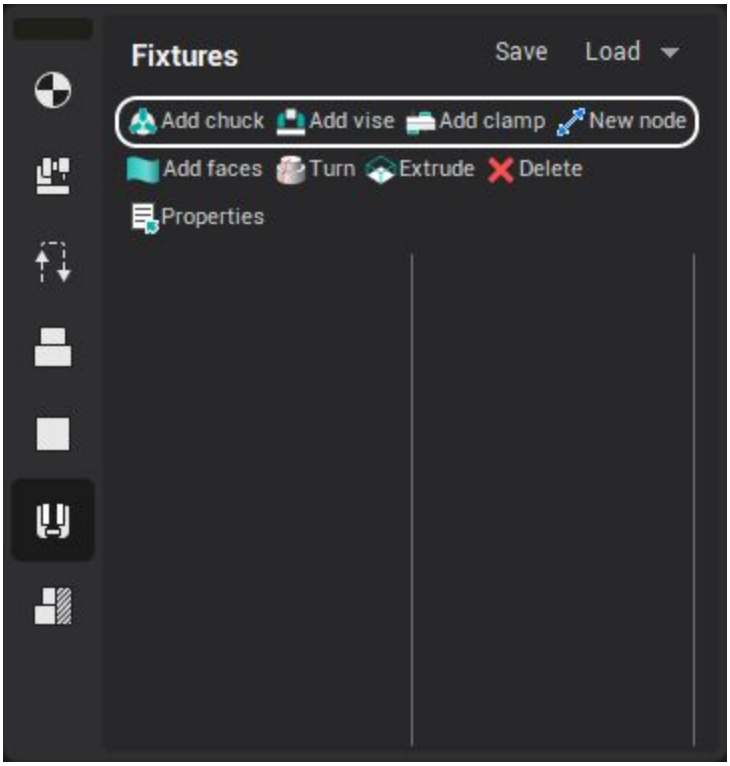
The algorithm works as follows: When the working pass intersects the fixtures, that section is cut away, extending to a safe plane. This algorithm works only with work passes.

The difference between a Restriction (Details) and a Fixture is that a Fixture is intended to be used throughout the entire Setup stage (Details).

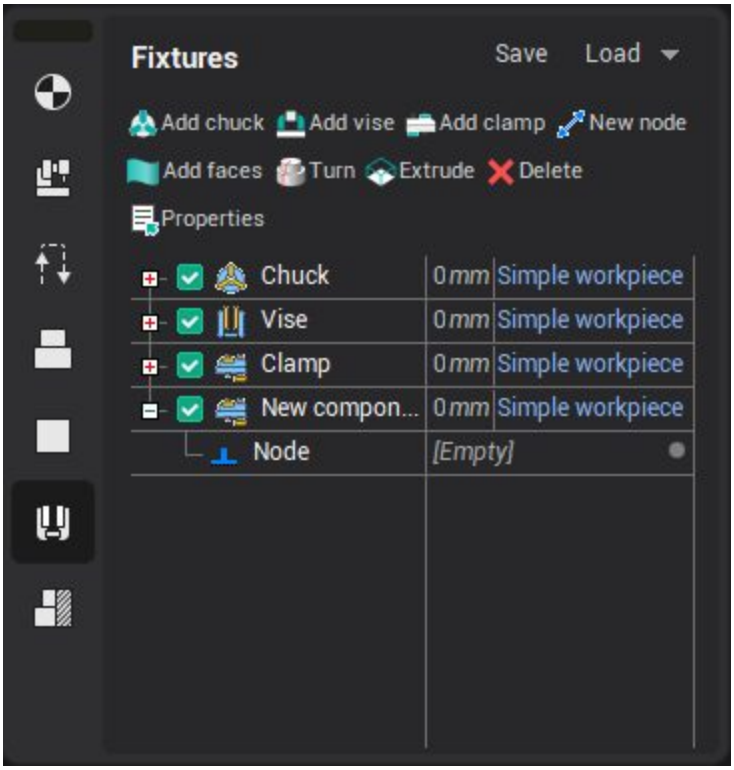
The Fixtures can be set in several ways:

1. Through tools that help define the part, workpiece and r estriction. (Details)
2. Creating a new fixtures

To create a fixture, you should select an existing template or create your own version of fixture using the "New node" button.

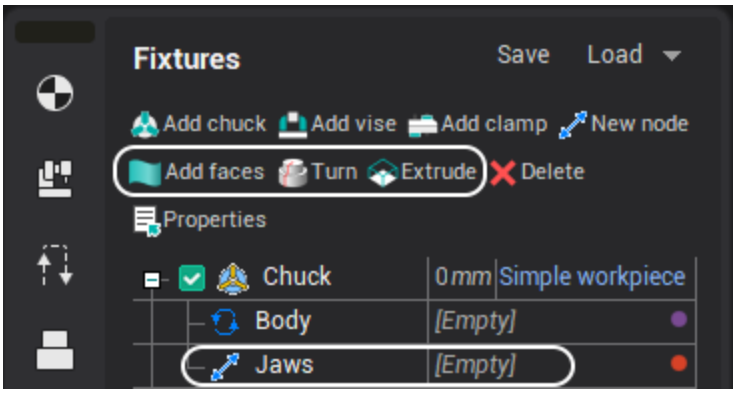


The newly created template and nodes are seen in the fixture tree.



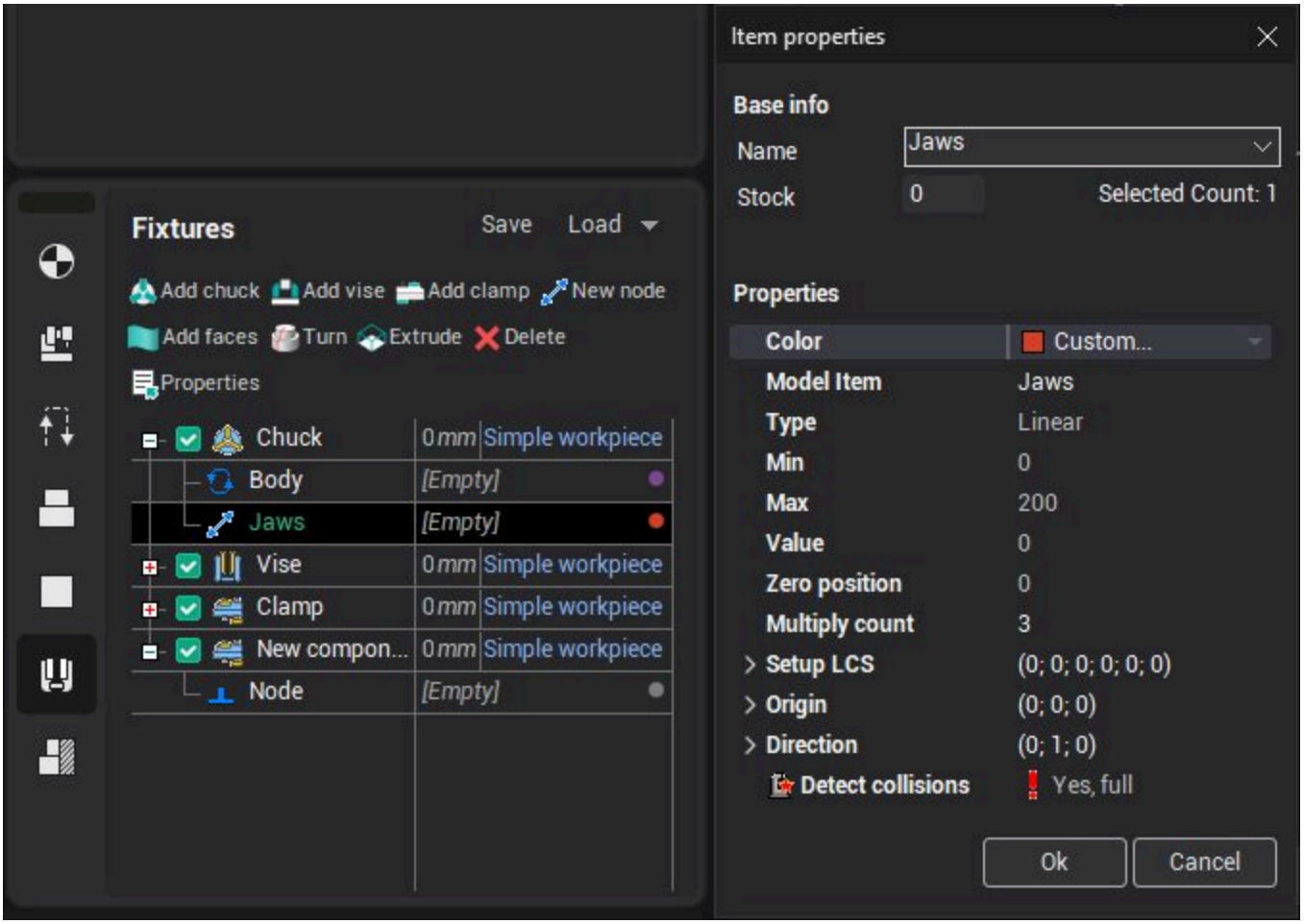
Geometry

To add geometry, select a fixture node and click one of the geometry buttons.

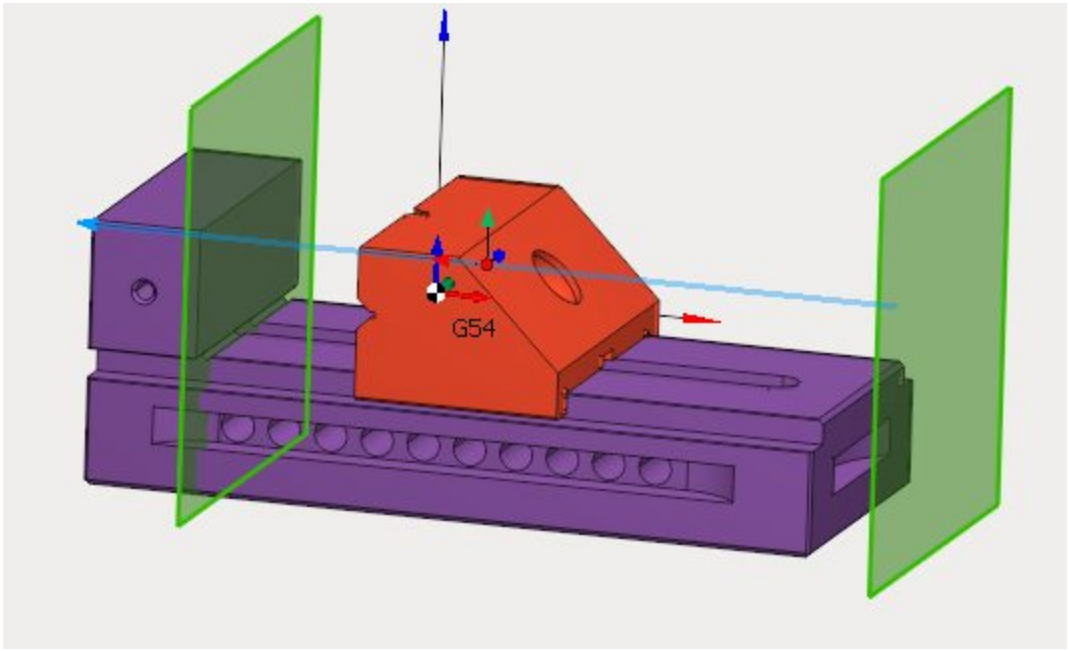


Configuring node parameters

Configuring node parameters are set in the Properties window that can be opened by double-clicking on a tree node or clicking the properties button on the main panel of the page.



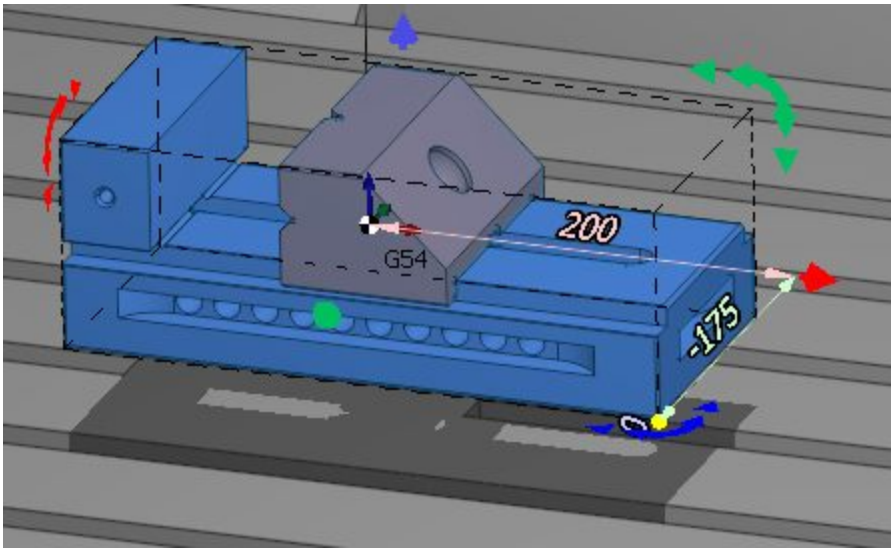
The main parameters of the node, such as the maximum (minimum) position, direction and rotation axis, can also be specified using visual objects.



Working with Fixtures.

Component setting

The position of the component is adjusted using a visual object.

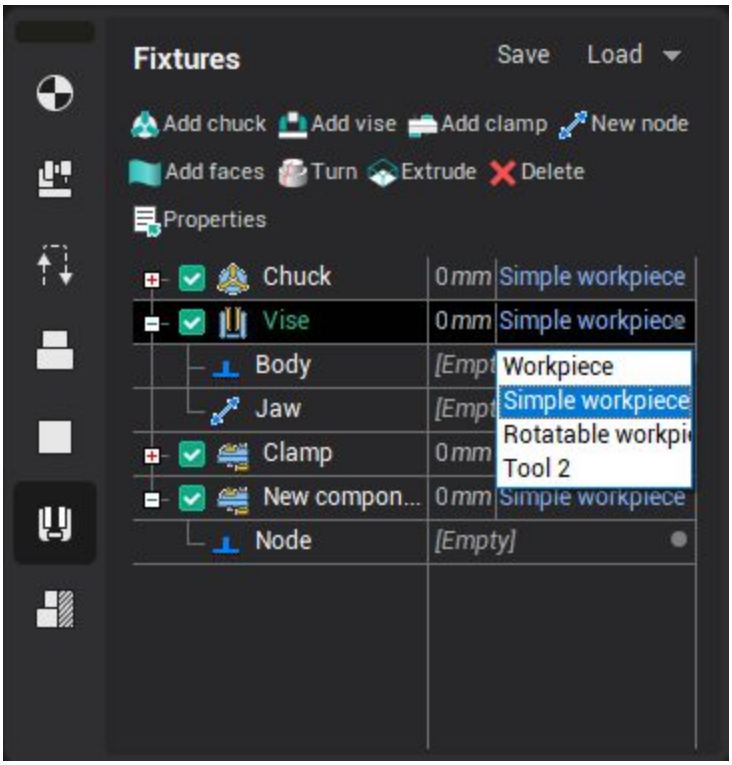


It can be invoked by double-clicking on the node or by one-click in component edit mode. .

When you right-click on a visual object, panel is showed up. It allows you to copy and delete components.

Snapping a coordinate systems

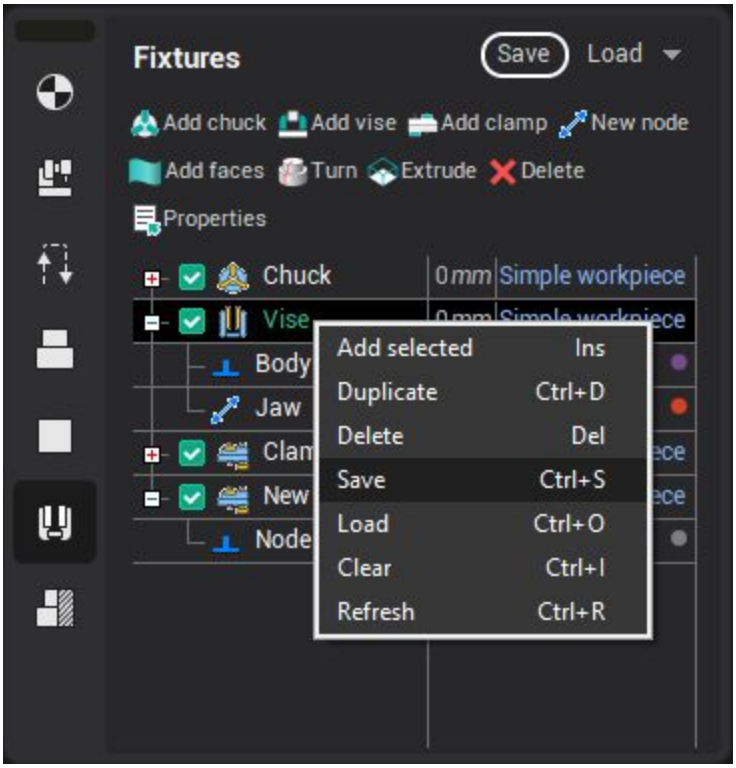
If your fixture is defined in a machine unit, then it is linked to a specific setup and moves with the machine.



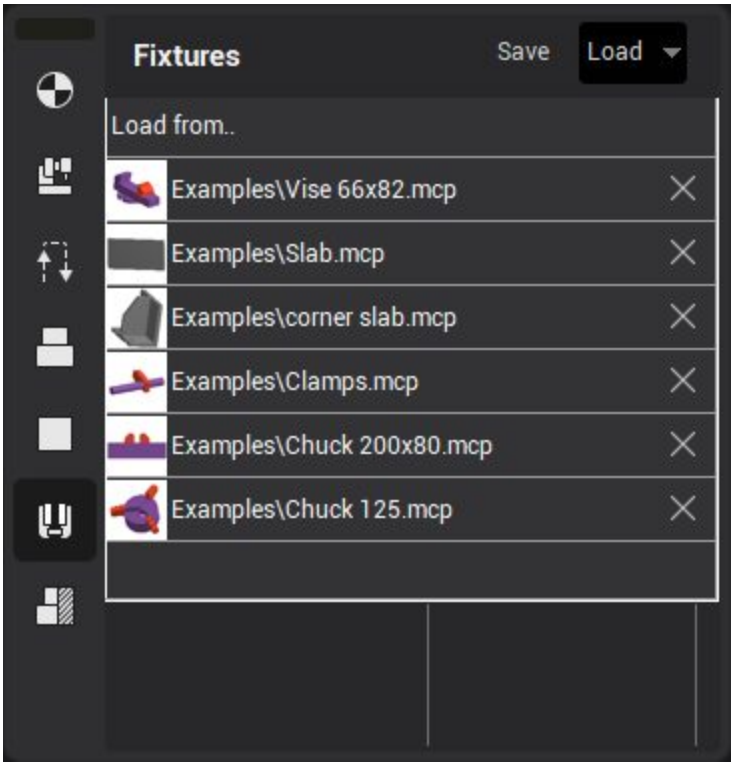
If it is in the operation node, then it is linked to the current setup and can be moved both with the machine tool and with the part.

Saving and loading

A fixture can be saved by right-clicking in the Component Tree or by clicking the Save button.



A fixture can be loaded by right-clicking in the Component Tree or in the drop-down panel of the Load button.



Structure group of operations

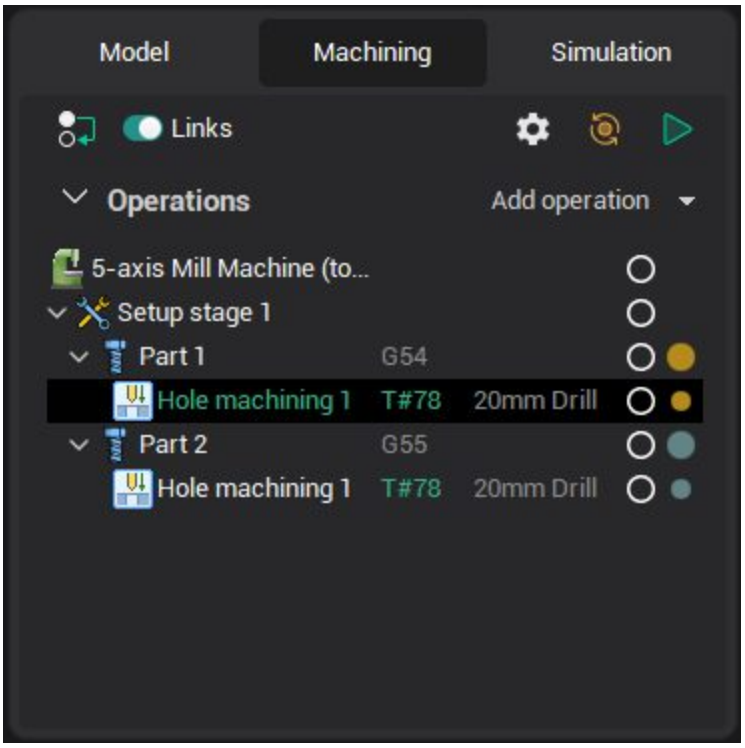
Application area:

The group required to build a technological process.

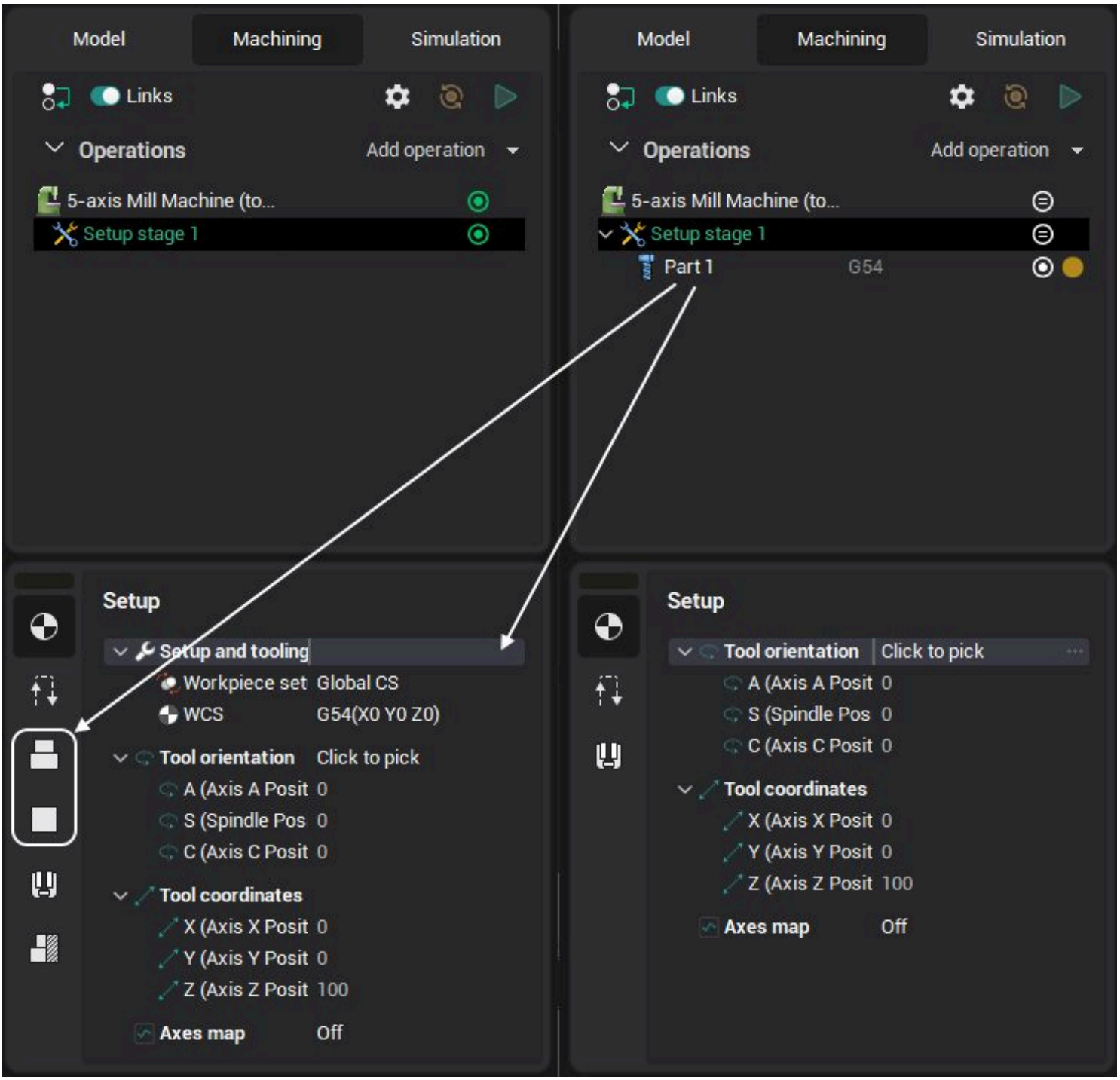
The peculiarities of working:

The components included in the operation tree have a hierarchical structure similar to the sequence of technological processes. Nodes of the Part or Part Copy type are nested in the Structures group node. In turn, the Multiply Group node is nested within the Part and Part Copy nodes.

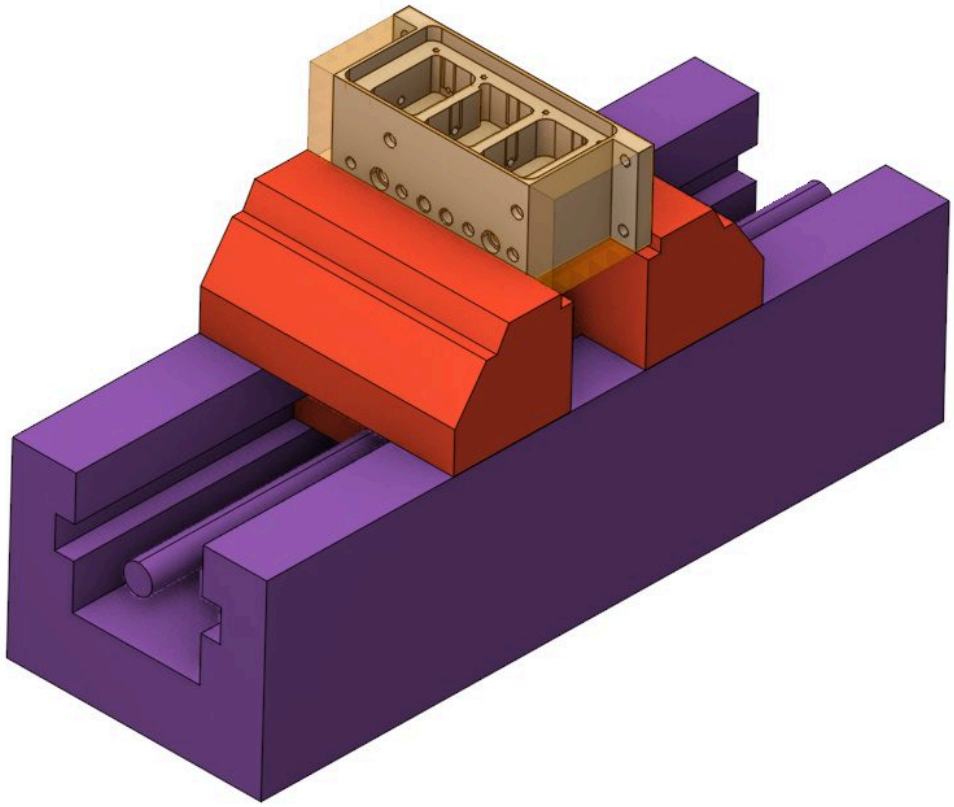
Accordingly, operations are located within these nodes.

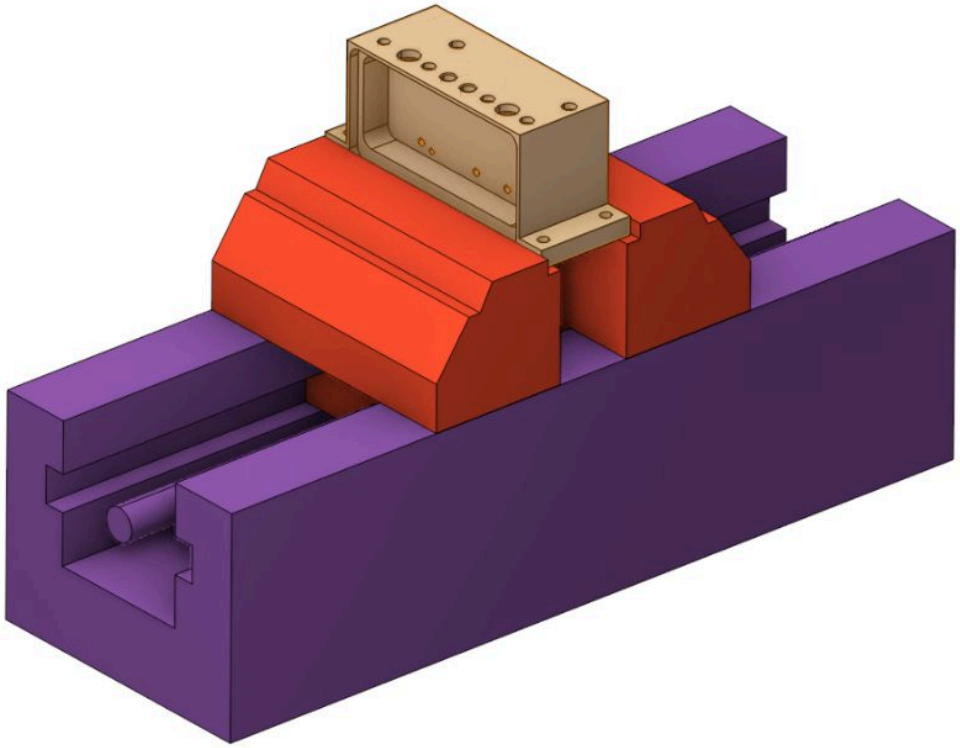


The tabs located in the groups are similar to the element in the Equipment tree. Adding a new subgroup in the structure reduces the number of tabs in the current group, and the nested group will inherit some of the same parameters as the main group. However, these parameters are hidden in the main group.


Model
Machining
Simulation
Links
Operations
Add operation
5-axis Mill Machine (to...
Setup stage 1
Part 1
G54
Setup
Setup and tooling
Workpiece set Global CS
WCS G54(X0 Y0 Z0)
Tool orientation Click to pick
A (Axis A Posit 0
S (Spindle Pos 0
C (Axis C Posit 0
Tool coordinates
X (Axis X Posit 0
Y (Axis Y Posit 0
Z (Axis Z Posit 100
Axes map Off

Setup stage





Application area:

This group is necessary to define the Setup. This group allows the part to be processed from different sides, and the previous setup does not interfere with the subsequent ones due to the hiding of the part, workpiece, and other previous setups.

In this group, you can add tooling that will be taken into account in the subgroups (Part, Copy of part).

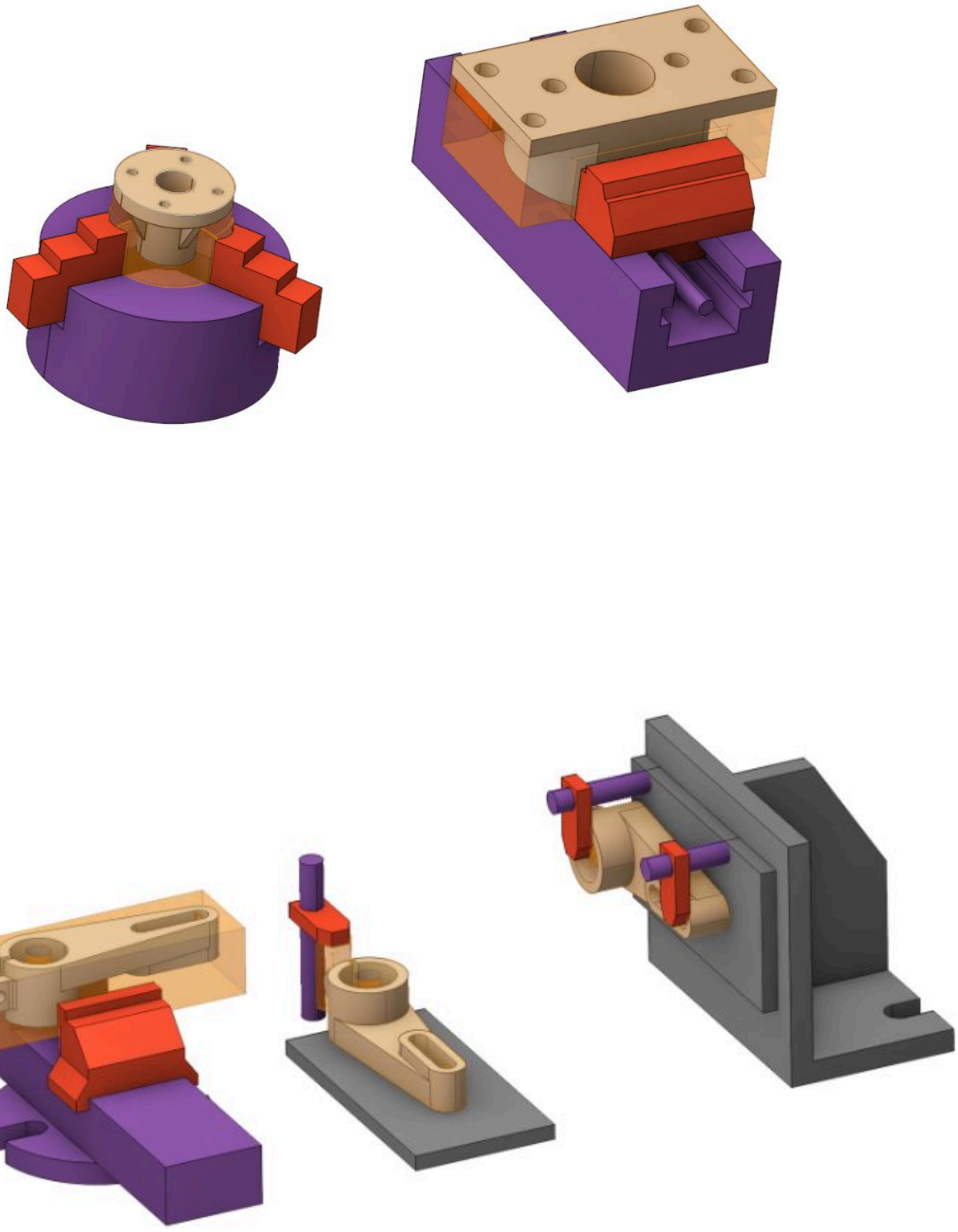
Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part (group)



Application area:

This group is necessary for creating multi-part projects. These can be several different (design) parts or standard parts divided in the technology by operation, similar to a conveyor. To sort operations, use Sequencing mode. Details

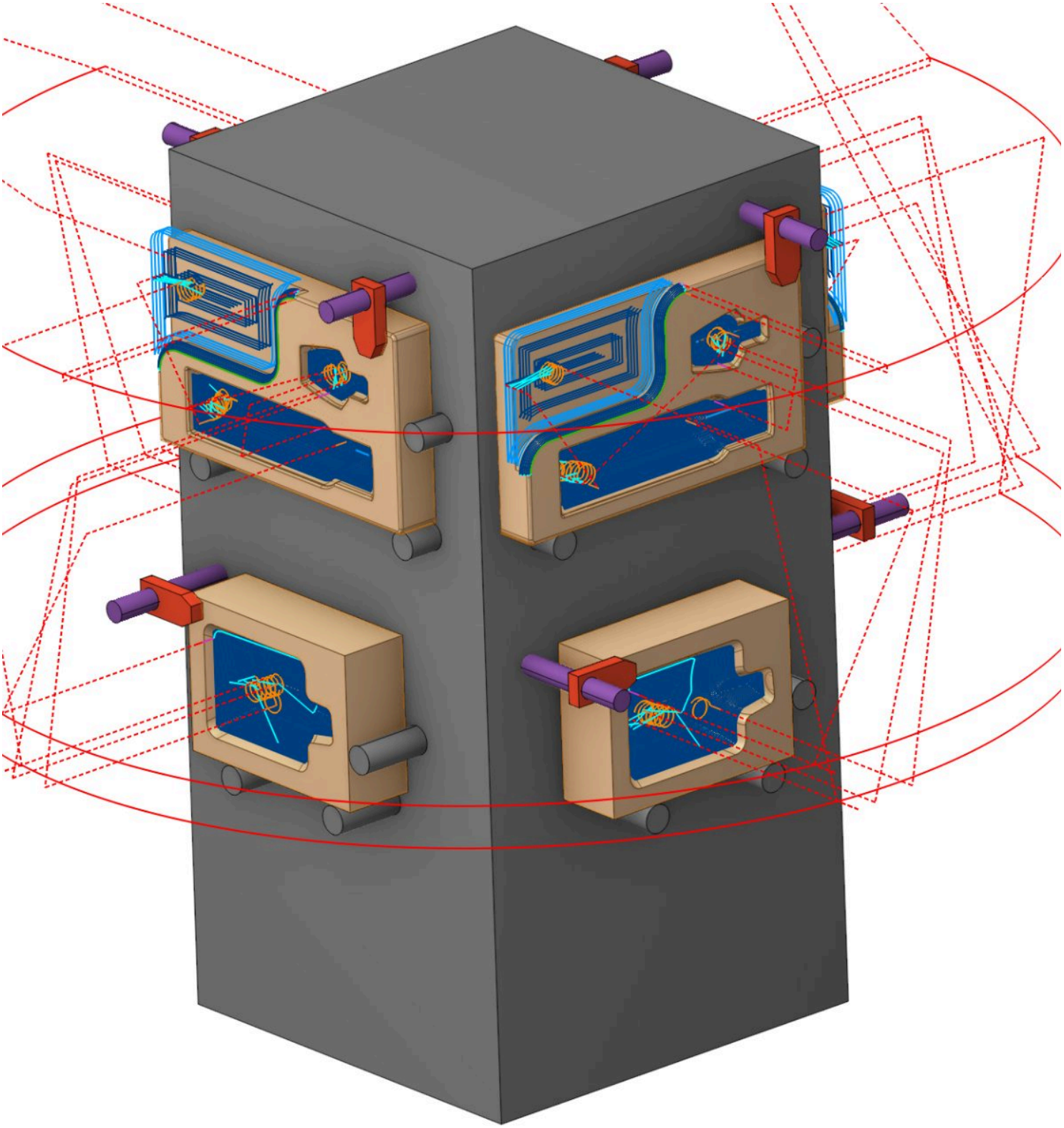
Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Copy of part



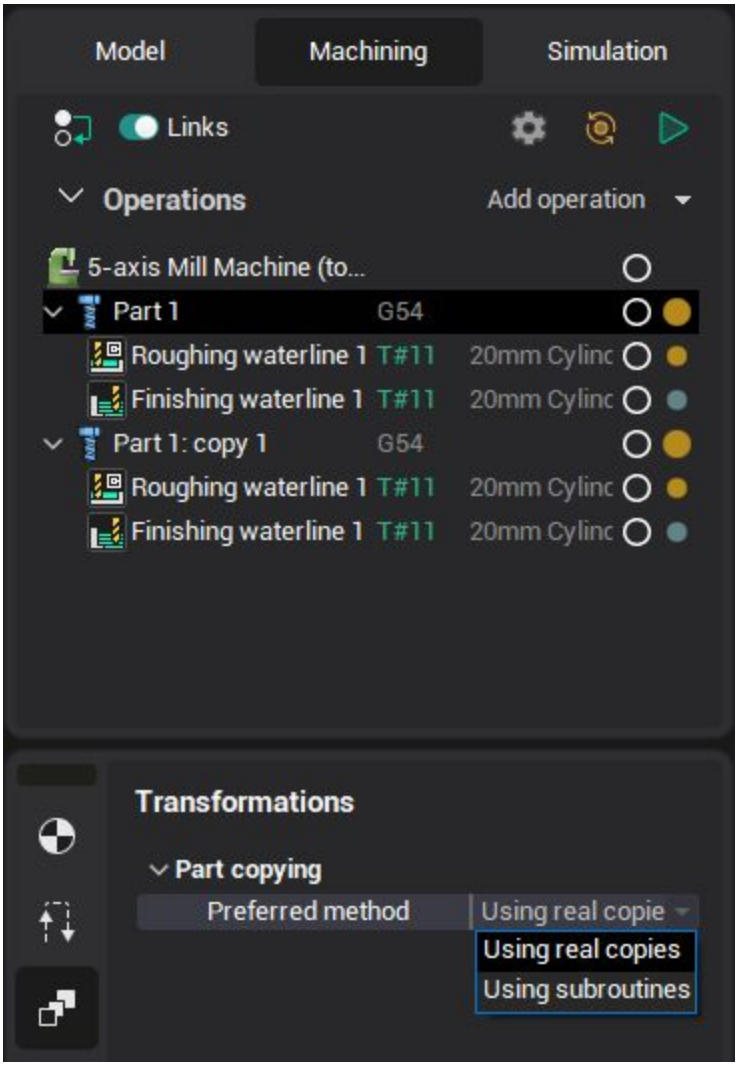
Application area:

Copy of part is the special kind of simple part. It's designed to prepare the project with the several identical parts. Usually it can be done with the next steps:

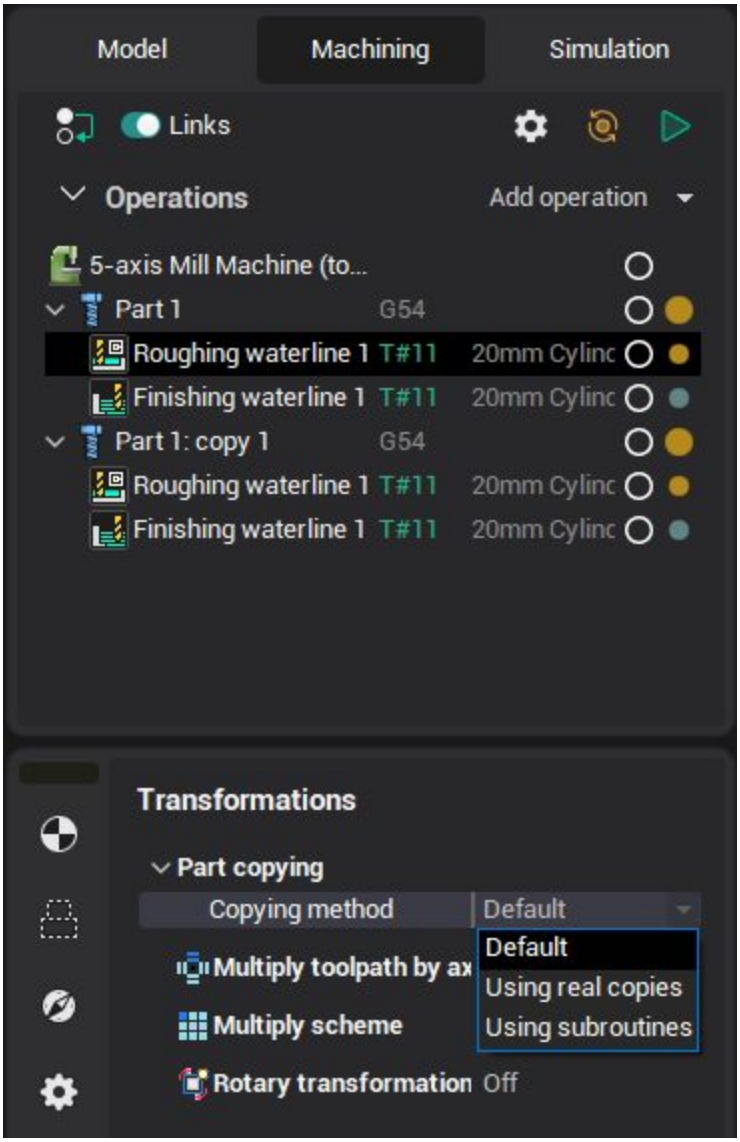
- Create the prototype part. Add operations, that are neccesary for the machining.
- Create copy of part. If the project contains more than one part then the copy of the current selected part will be used as prototype.
- Assign the workpiece setup and workpiece CS for the copy.

Copy of part is a group of operations, that is syncronized with the prototype group (part). It's not possible to add or delete the operation inside the copy. If an operation is created/deleted in the prototype part, then the copy of this operation will be created/deleted in all part copies. Copy of part contains inside the special operation copies. Operation copy doesn't calculate the toolpath itself. It just apply the toolpath of the prototype operation to the place of the part copy. So if reset the prototype operation then all copies of this operation will be reset. Recalculation of copies must be started manually.

There is the possibility to output the copies machining as the subroutine calls. The way of output is defined in the parameters of the prototype part. Before usage the subroutines be sure that your postprocessor supports it.



You can set individual copy parameters for each operation in the group.



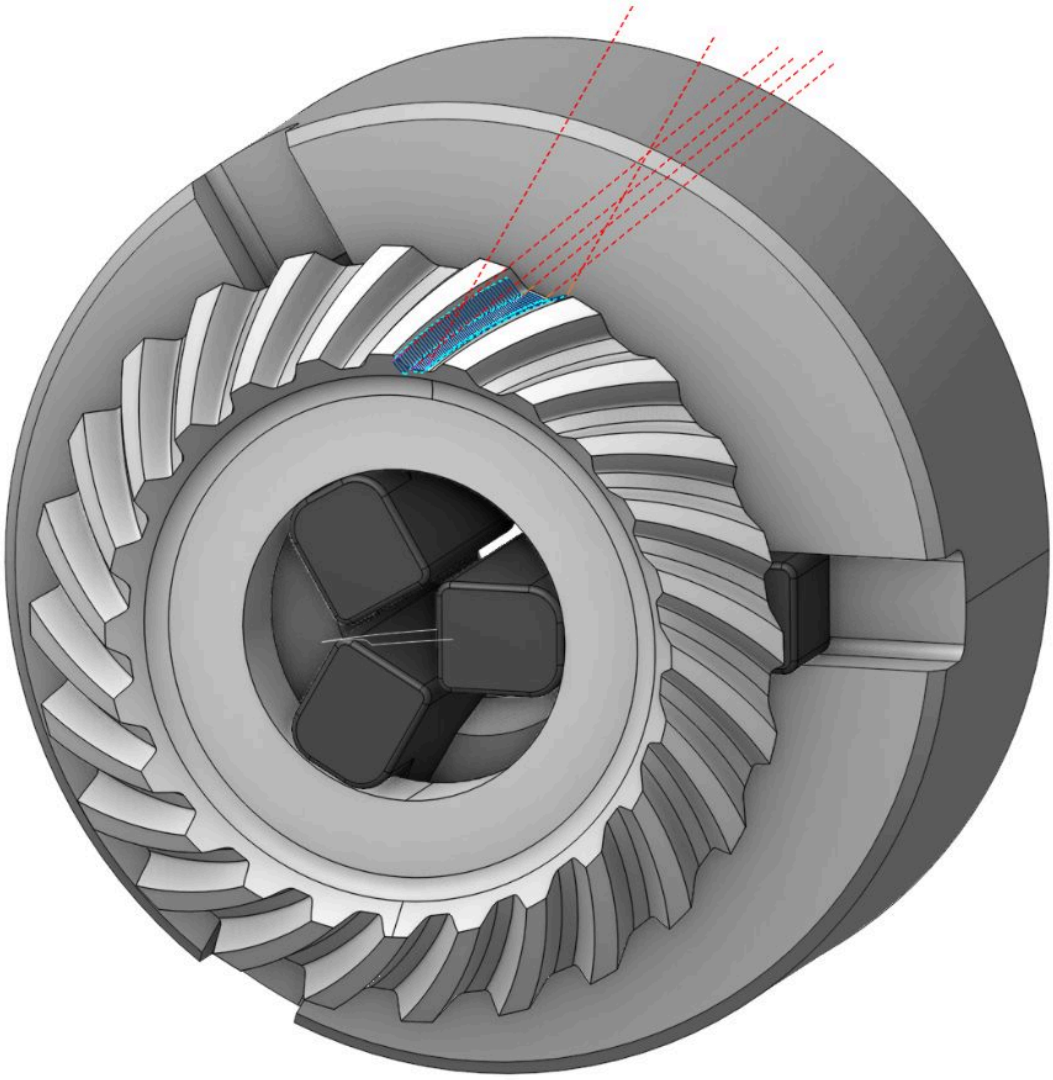
Setup:

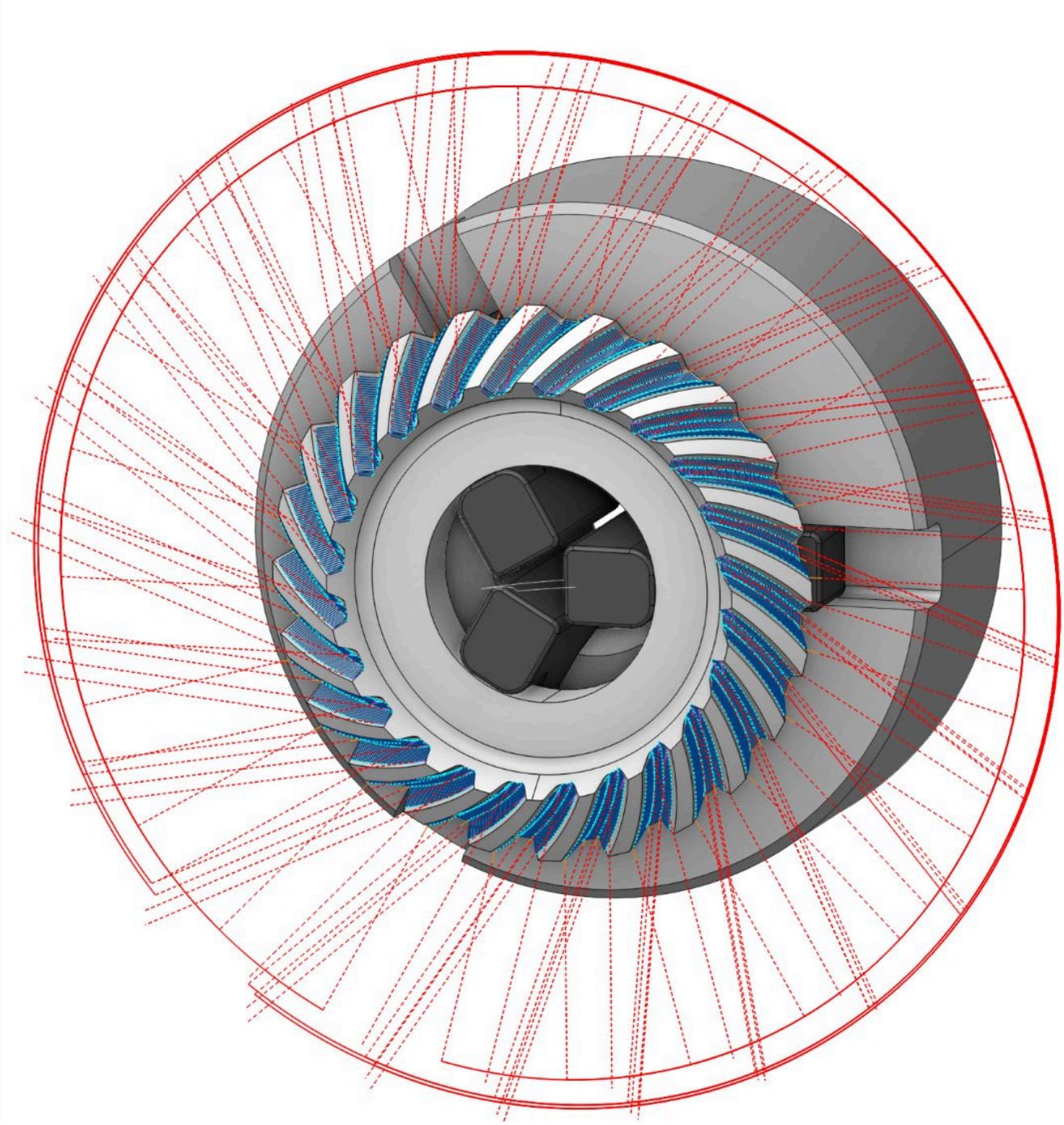
The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Multiply Group





Application area:

In this group, copying methods for trajectories are defined, which are then inherited by operations within this group. This approach allows operations to process individual detail elements, while the group ensures their automatic propagation to all standard elements according to established rules. The copying is carried out according to the rules of the Transformation tab. To sort operations, use Sequencing mode. Details

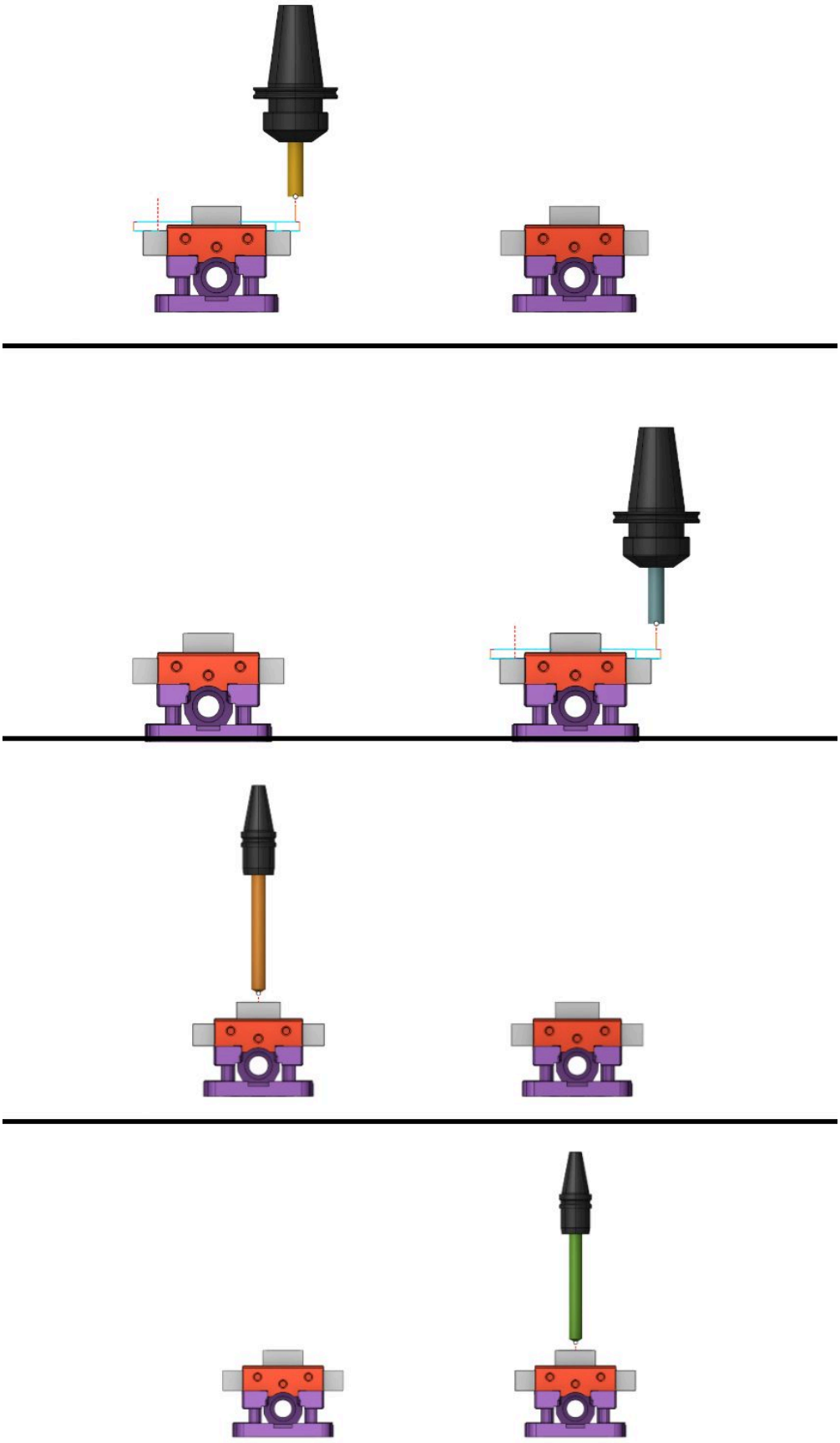
Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Sequencing mode



Application area:

Operations inside the part group define the machining sequience of this one part only. In fact it defines the residual workpiece material for every next operation. If the project contains more than one part then the real machining sequence can be redefined, for example to minimize the tool changes or the tombstone rotations. Use the button on the main panel to switch on the sequencing mode. In this mode the reordered operations tree is shown.

By default the operation are sorted automatically to minimize the tool changes. It's possible to change the machining order manually by drag-n-drop or with the arrow buttons on the sequencing panel. Arrow button moves the selected operation before/after the previous/next one. Sort button restores the automatical sorting. The rules for the automatic sorting are defined in the next popup menu.

The links between the operations can not be calculated before the real machining order is set. Therefore the links calculation was separated from the main calculation process. Full workflow for the multipart projects contains the next steps

- Placements of the part in the work space of the machine
- Design of the the machining process for every part. Calculation of the tool path without the links (approaches and returns).
- Simulation of the process with the disabled sequencing mode, to check the correctness of the workpiece change etc.
- Reordering of the operations in the sequencing mode.
- Calculation of the links between the operations.
- Simulation of the process in the sequencing mode to check the links and the tool changes between the operation.
- Tool path generation

Main element:

Sort.

This button is needed to automatically sort operations with certain prioritie. Sorting priority is performed by the location of groups in the list

▼ Click here to expand..
Tool. Group to sort by tool.

Side. Group to sort by side.

Part. Group to sort by part.

Feature. Group to sort by element of part.

By operations. Group to sort by operation.

Swiss lathe. Group to sort by Swiss-type operation.

Reset sorting. Reset automatically sort operations.

Delete group in tree. If a group is disabled, it is not displayed in sorting tree.

Increase sorting priority. Increases priority.

Decrease sorting priority. Decrease priority.

Move up / Move Down

Needed for manual sorting of a specific operation.

Link.

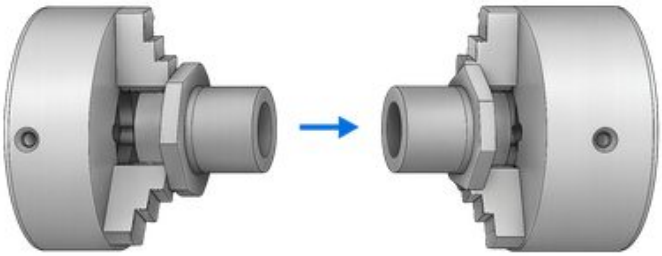
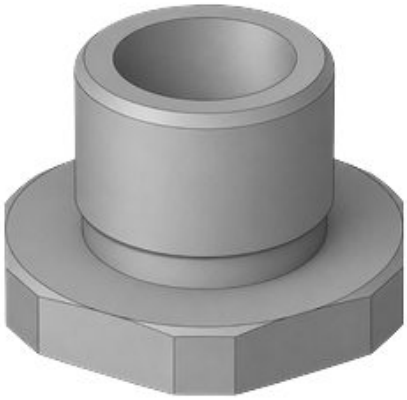
Calculates selected operations with link. Transitions between operations are displayed as lines

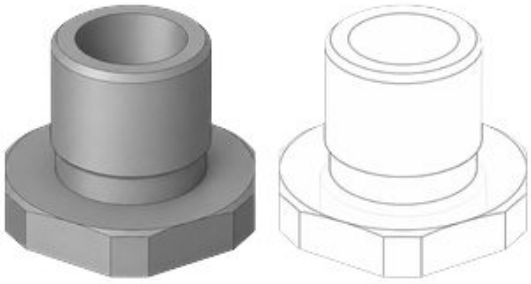
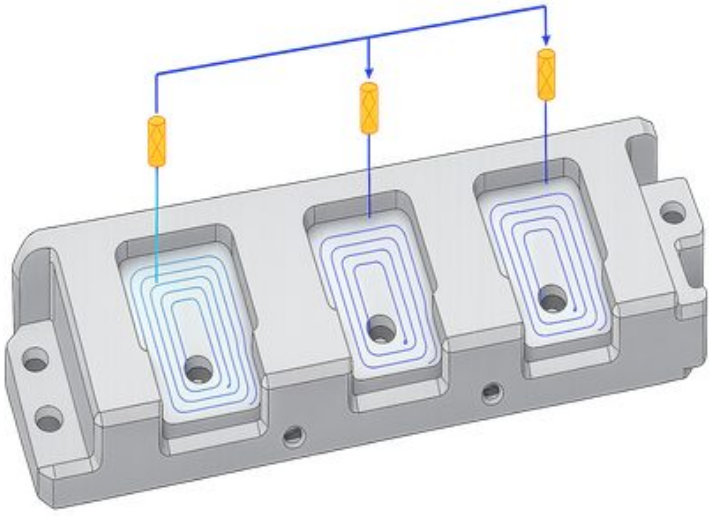
Unlink.

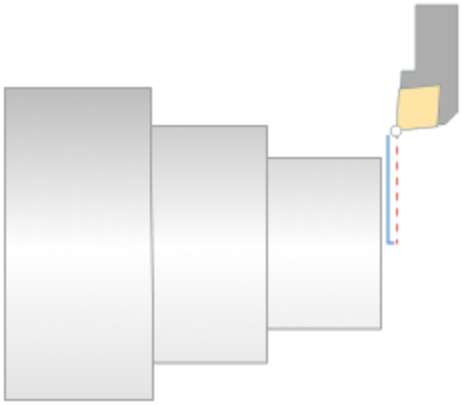
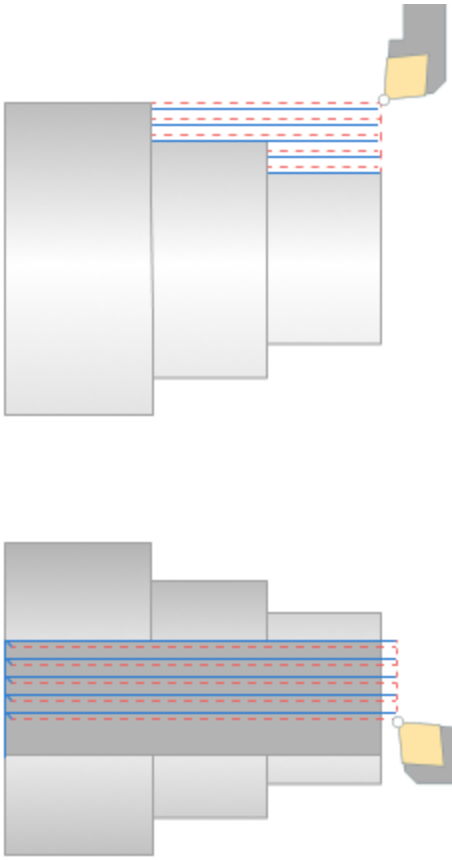
Calculates selected operations without link.

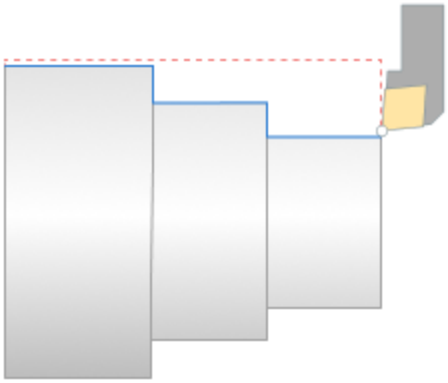
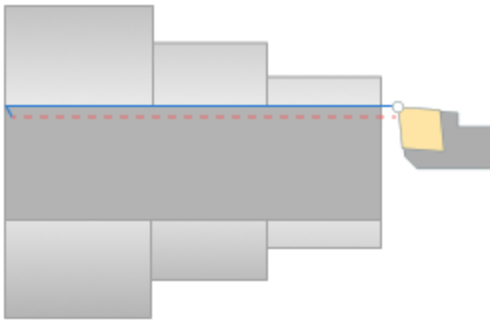
List of types of machining operations

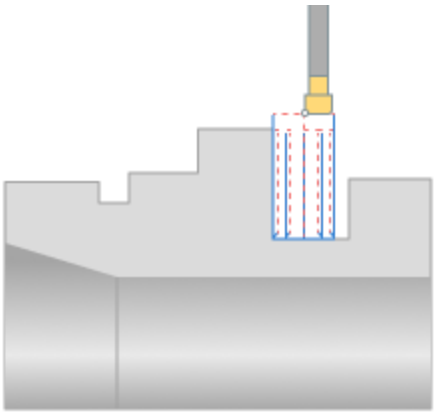
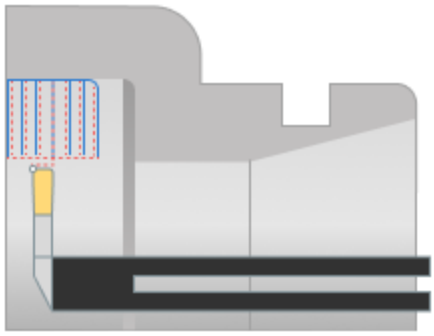
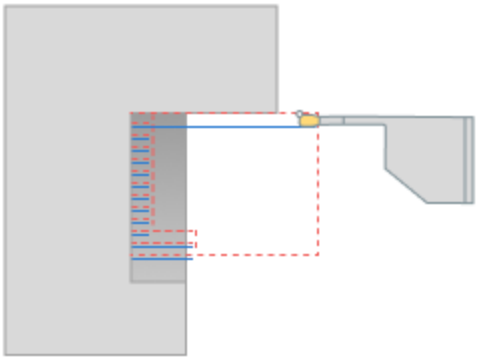
The following is the current list of operations implemented in the current version of the CAM system.

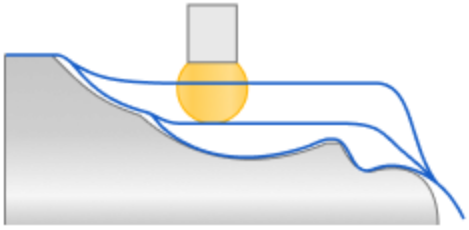
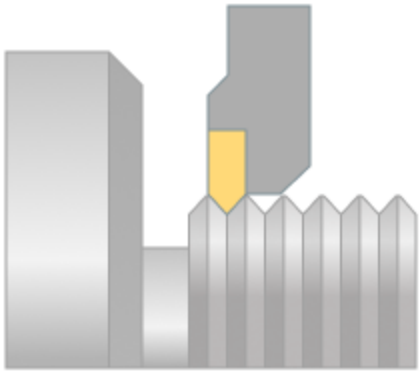
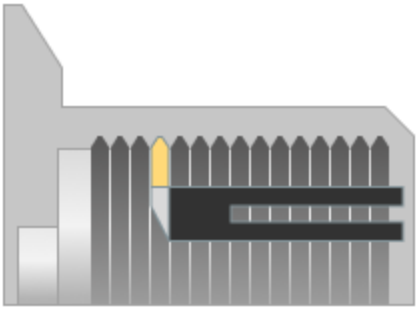
Structure	
Setup stage	
	This group is designed to define the stage of the manufacturing process where the part remains in a fixed position. It allows the part to be machined from different sides, while previous setups do not interfere with subsequent setups by hiding the part, workpiece, and other objects from previous setups. Within this group, you can add fixtures that will be considered in subprojects (Part, Copy of Part).
Part	
	This group is used to create multi-component projects: several different parts or standard parts, divided into operations based on technology (similar to a conveyor belt). The Setup tab in this group is used to configure the main project parameters, such as part positioning on the equipment, the part coordinate system, etc. A set of transformations for converting toolpath coordinates is also available for this group.
Copy of part	

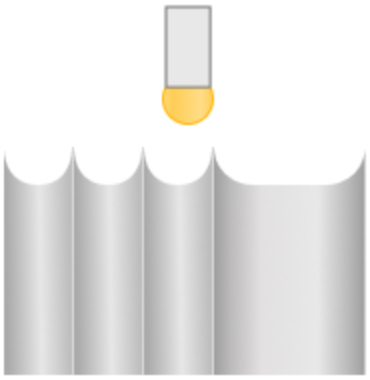
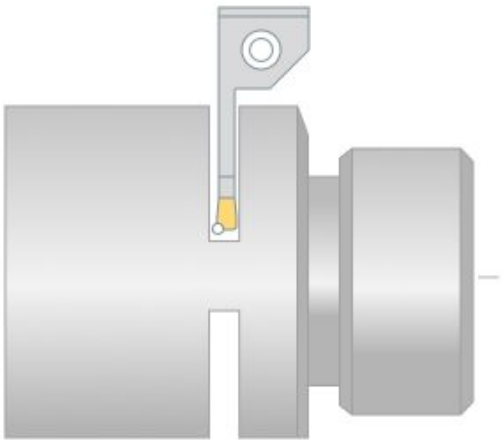
Structure	
	A special type of group designed for preparing a project with several identical parts. The workflow typically follows this: a prototype Part with the required operations is created; then a Copy of Part is created, which uses the prototype as a template. Copies are synchronized with the prototype — operations cannot be added or deleted within a copy: when an operation is created or deleted in the prototype, corresponding copies of the operations are automatically created or deleted in all copies. Copy operations do not calculate the toolpath themselves, but apply the prototype's toolpath to the copy's position; resetting the prototype calculation requires manually recalculating the copies. For copying, output can be configured via subroutine calls (if the postprocessor supports it). The Setup tab in Copy of Part is used to specify the position, workpiece, and coordinate system for the copy.
Multiply Group	
	A group that defines the methods for copying paths; these rules are then inherited by operations within the group. This allows operations to process individual features of a part, while the group itself automatically propagates the results to all standard features according to the defined rules. Copying is performed according to the rules in the Transformations tab.
Lathe	
Lathe facing	

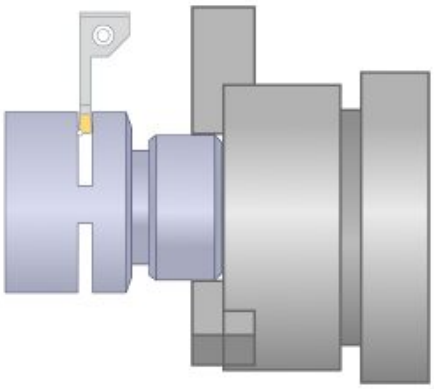
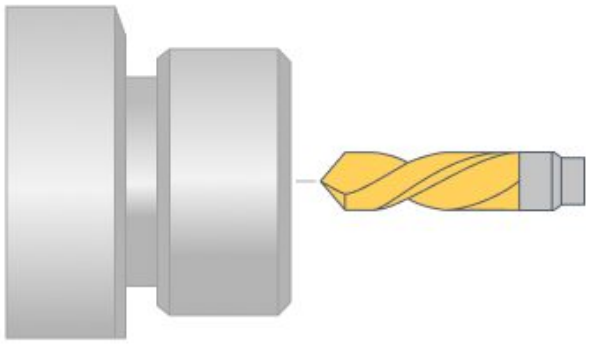
Lathe	
	The Lathe Facing operation is designed to prepare the uneven end surface of a workpiece (left or right). Material is removed layer by layer using vertical tool movements. It automatically generates a toolpath for machining the end surface based on the current workpiece and part. The operation is used for both roughing and finishing and is often applied before drilling or other turning cycles.
OD Roughing, ID Roughing	
	The operation is designed for the removing of the sizeable part of the workpiece. It can be used when the workpiece is much different of the part. The material is removed by the series of the parallel tool motions. The operation provides to remove a lot of workpiece volume in the shortest time.
OD Finishing, ID Finishing	

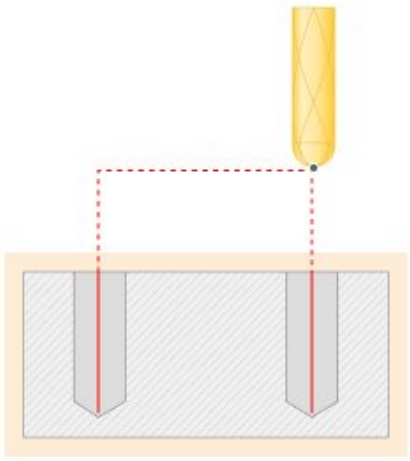
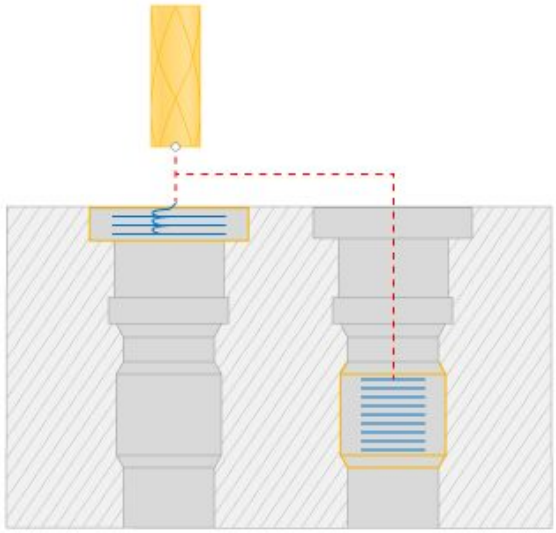
Lathe	
 	The operation is designed for the final finishing machining. The machining is performed by the offset motions along the part generatrix. It gives the best result if the part and workpiece have the little differences. The operation allows to generate the toolpath without the workpiece checking. It is also possible to make 4-axis turning
OD Grooving, ID Grooving, Face grooving	

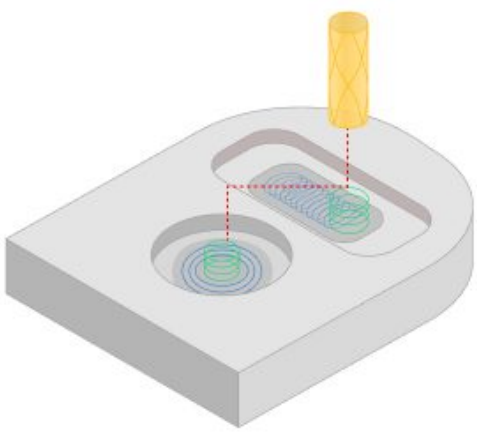
Lathe	
  	The operation is designed for the groove machining and other zones that can not be machined by other lathe operations. The cycle generates the tool path according to the groove tool possibility to cut by the front side. The tool path can combine the rough path for the workpiece volume removing and finish path for the shaping. The workpiece volume can be removed by some layers with the different strategies and cutting directions. There is the possibility to switch on the chip breaking and delays.
OD Adaptive turning	

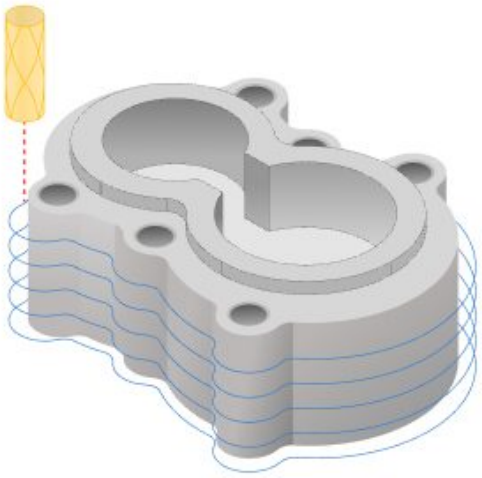
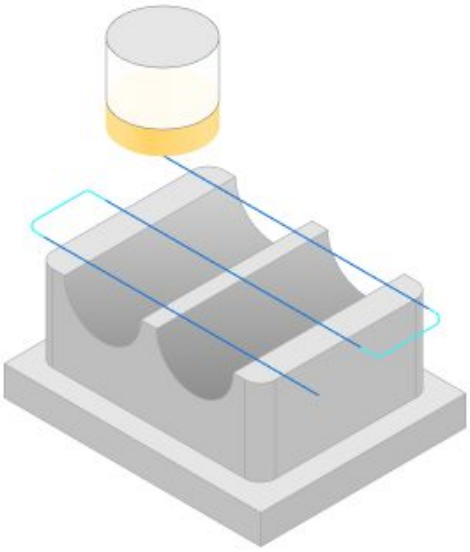
Lathe	
	An adaptive roughing turning operation that generates an optimized, smooth toolpath while maintaining a constant tool load. This increases tool life and significantly increases cutting speed (the process is more than twice as fast as traditional grooving). The operation uses an Adaptive strategy, similar to the Roughing Waterline strategy, and is designed for tools with round inserts.
OD Threading, ID Threading	
 	The operation is designed to make the different threads by turning cutter or thread chaser. There is the possibility to select the thread parameters from ISO or Imperial databases. The thread parameters can be set manually to make the special threads. The machining can be performed in few strokes. Different types of the approach engage retract and return are available
Profile threading	

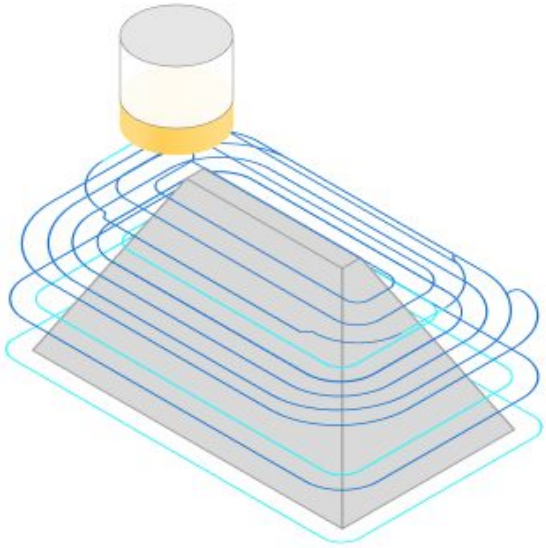
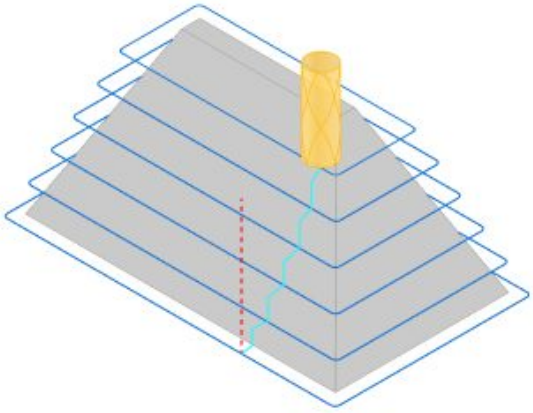
Lathe	
	The Profile Threading operation allows you to form threads with a profile shape different from the tool's shape. Material is removed from within the entire thread groove in a series of sequential passes—the placement of the passes is calculated based on the tool's shape and the shape of the thread groove.
Lathe part-off	
	The operation cut out the part with the additional chamfer or rounding machining. The size of a groove, the chip breaking parameters and the delay values can be set.
Part-off with takeover	

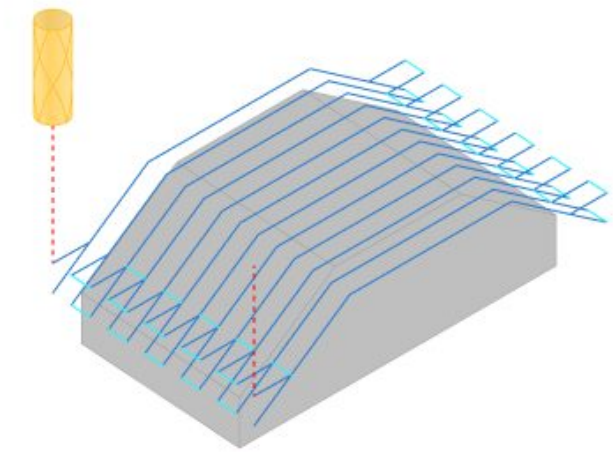
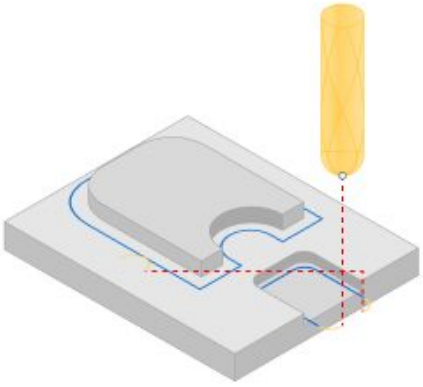
Lathe	
	The Part-off with Takeover operation is designed to cut a machined part from a remaining workpiece while simultaneously forming the end face (including chamfering or rounding). The area for cutting can be prepared in advance (for example, by cutting a groove). It differs from a standard Lathe part-off in that, upon completion of the operation, the workpiece is transferred (takeover) to another spindle.
Lathe hole machining	
	The operation is designed to generate the NC commands to machine the axial holes. The next cycles are supported: simple drilling, deep drilling with chip breaking or removing, threading by tap etc. There is the possibility to set the cycle output mode: Long hand or canned cycle.
Holes	
Hole machining	

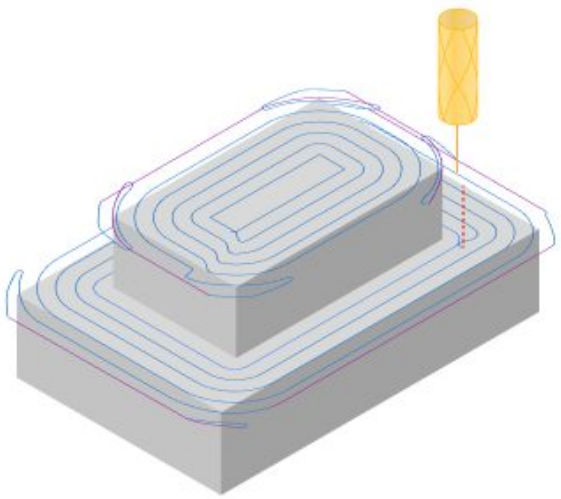
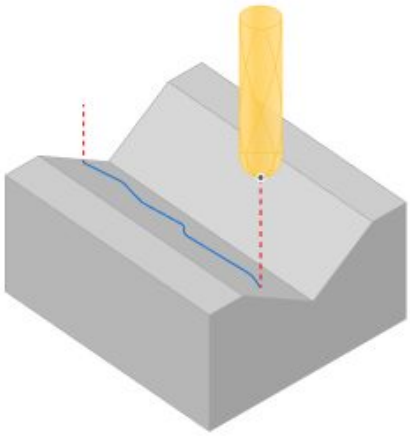
Holes	
	Creates a set of machining commands for holes. These include drilling, boring, centering, tapping or thread milling. The operation can be used both for hole machining and for preliminary drilling in tool plunge points in pocketing and waterline roughing operations. Hole machining operation can be used to machine holes that are positioned differently, i.e. holes whose axes are not normal to the same plane. Note that operation can machine holes that are not lying in orthogonal planes.
FBM Mill	
	When creating an FBM Mill, the team automatically recognizes known manufacturing features in a part and allows you to select predefined or saved machining procedures from the FBM Procedure Library. The system automatically generates the necessary operations for the identified features and organizes them according to current rules (for example, minimizing tool changes).
2D	
Pocketing	

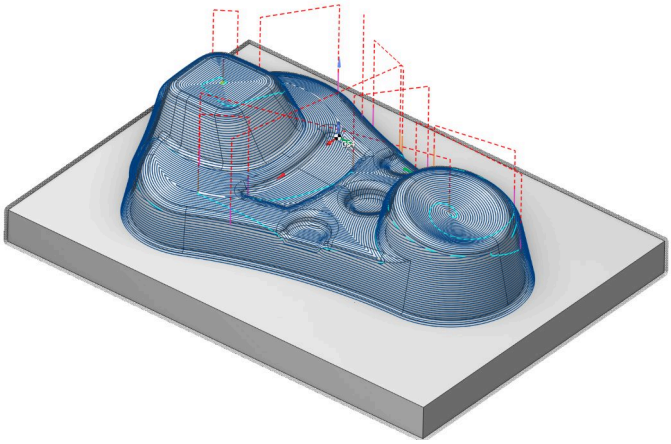
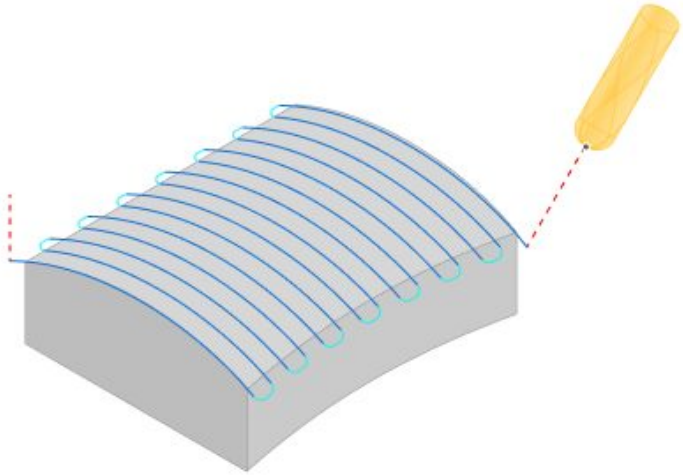
2D	
	Waterline removal of material inside the defined area or pocket. The shape of the area for pocketing is formed from curves created on the horizontal (XY) plane. This operation is used for the 2 & 2.5D machining of pockets and isolated areas, and also for preliminary material removal before engraving (2D finishing) operations.
Engraving	
	The operation is designed for the engraving of 2D geometry and inscriptions on flat areas. The image being engraved is formed from projections of curves onto the horizontal (XY) plane. Horizontal movements of the tool machine the main parts of the model's side edges. To create the sharp inner corners and for machining of smaller width areas, 3D milling is used. The operation is used for engraving of flat drawings and inscriptions and for finishing passes along side walls of pockets and for isolated areas during 2 & 2.5D machining.
2D contouring	

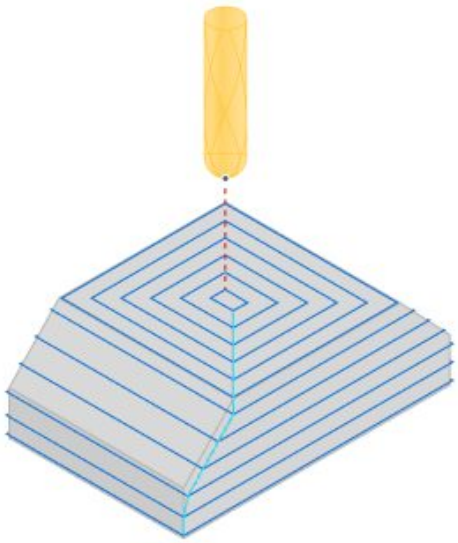
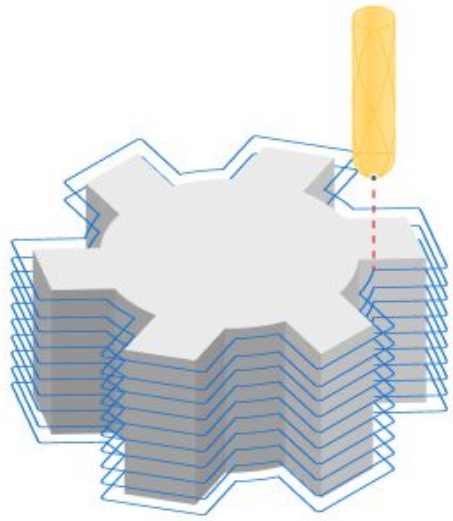
2D	
	For the machining of horizontal contours or curves projected onto the horizontal plane. The horizontal movements of the tool are created based on the geometry being machined. The tool center or the tool edge can follow the contour. The operation is used for creating parts with vertical sides or for a machining pass with a constant Z depth etc.
Basic	
Face milling	
	Face Milling operation removes stock on a given horizontal plane with one of the following strategies: One way, Zigzag, Optimized zigzag, Spiral.
Roughing waterline	

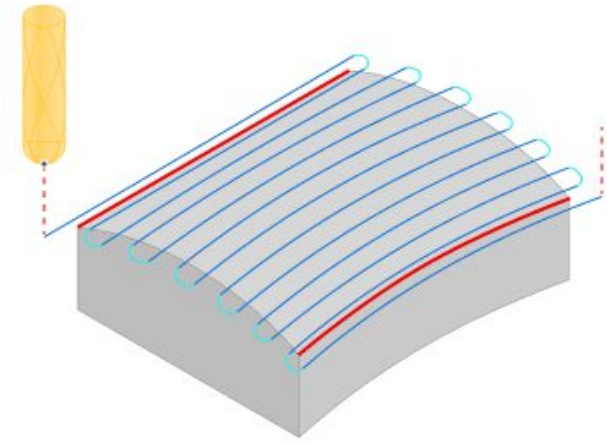
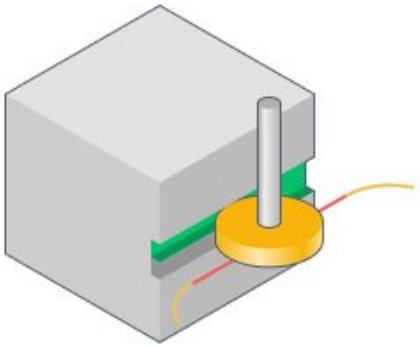
Basic	
	Waterline removal of stock material of a workpiece, which lies outside the 3D model. As in pocketing, the main part of the material is removed by the horizontal (XY) movements of the tool. The operation is often used for primary rough machining of complex models, which have considerable geometrical difference to the workpiece.
Finishing waterline	
	Waterline machining of surfaces of a volume model. Milling is performed by using horizontal movements of the tool. The operation gives a good result when machining models or their parts with their major surface areas that are close to the vertical. For machining of models of high complexity, it is recommended to use the waterline operation together with plane or drive.
Roughing plane	

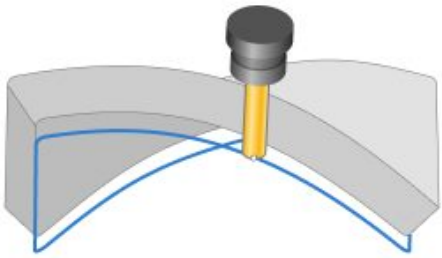
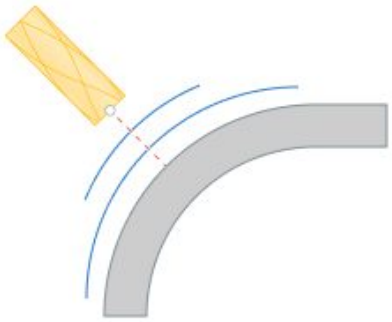
Basic	
	Plane removal of stock material of a workpiece, which lies outside the 3D model. The sections lie in vertical parallel planes. To limit the pressure on the tool, machining can be performed with small preset Z depths. The finished operation is usually closer to the finished model than using the waterline operation with similar parameters. The operation is normally used when it is necessary to obtain a roughed workpiece that does not differ much from the source model. It is also useful when milling soft materials.
2.5D contouring	
	The 2.5D contouring operation expands the capabilities for processing multi-level part contours. The operation works with a model composed of flat regions bounded by closed profiles at different heights; machining is performed through a series of horizontal passes (layers)—the number of passes and their Z-levels are determined by the working levels and step in the operation parameters.
Flat land	

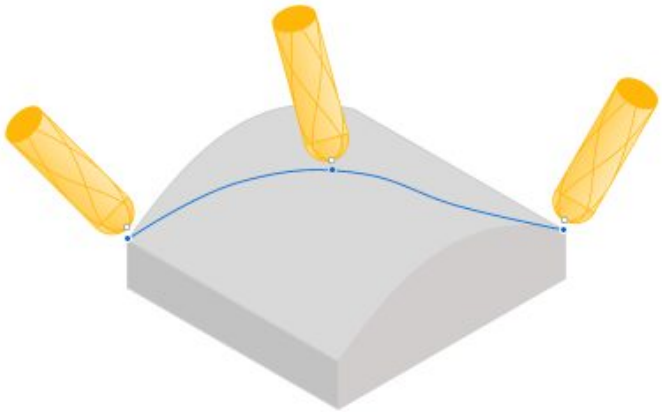
Basic	
	The operation allows to make a finish machining of flat horizontal surfaces of a part. The flat segments are recognized automatically. A tool toolpath consists of series of horizontal patches. All not horizontal segments of model for machining are inspected to avoid gouges during machining.
3D contouring	
	Generates a series of tool movements along freeform curves. The view of the toolpath in plane is similar to 2D contouring – tool movements are constructed with the tool center or edge passing along the contour. The Z coordinate at every point of the toolpath is calculated as a displacement based on the Z coordinate of the corresponding point on the curve. The operation can be used for machining of edges of parts of a die or for creation of a complex shaped groove etc.
Advanced	
Combo finish	

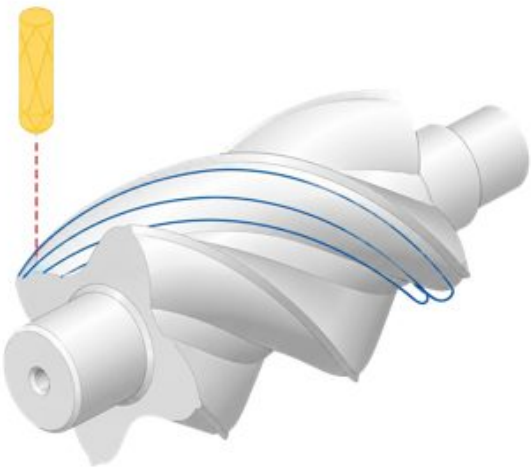
Advanced	
	Combines plane, waterline, and scallop finishing strategies.
Surfacing	
	The finishing operation allows machining of surface models with variety of strategies (parallel to plane, parallel to curve, morph and others) and tool axis orientation modes (fixed, normal to surface, to rotary axis, through point, through curve, etc).
Scallop	

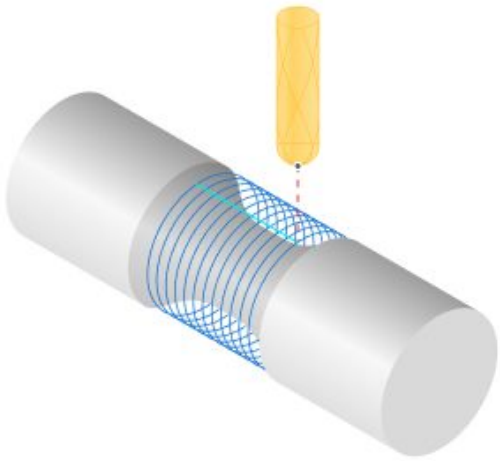
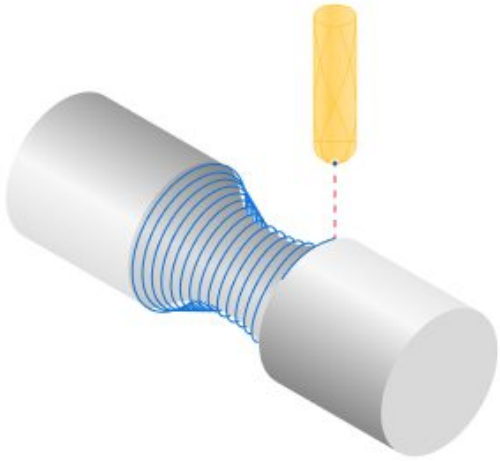
Advanced	
	Scallop (or 3d constant step-over) tool path starts with curves lying on the part surfaces and repeatedly offsets them inwards until the curves collapse. consistent step-over across the part surfaces is guaranteed The tool path is well suited for high speed machining of complex molds and sculptured models. Features: Lightning fast toolpath calculation. One entry, one exit. It is possible to machine a whole part with a single continuous spiral-like toolpath with only one entry and one exit points. High speed machining. It is possible to generate a toolpath with rounded corners.
Helical	
	3D Helical machining operation are useful for machining of cylindrical parts without an undercuts. The entire model is selected as the job assignment. The operation can generate a single-pass spiral like path for the entire model. If there are model areas that can not be processed without a transition, it will be processed after the processing of the current pass. The operation does not control the height of the scallop and does not ensure a uniform height change.
Morph	

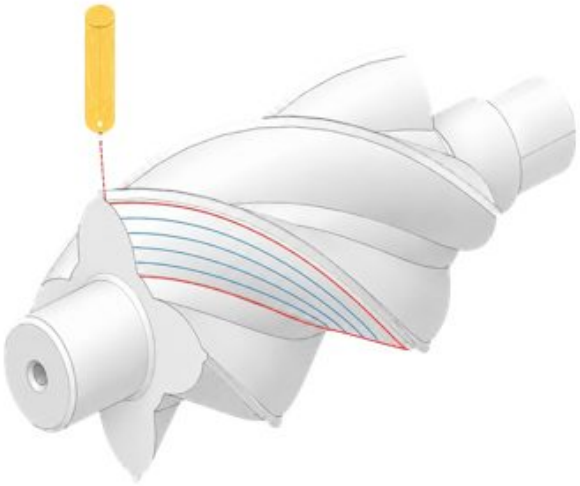
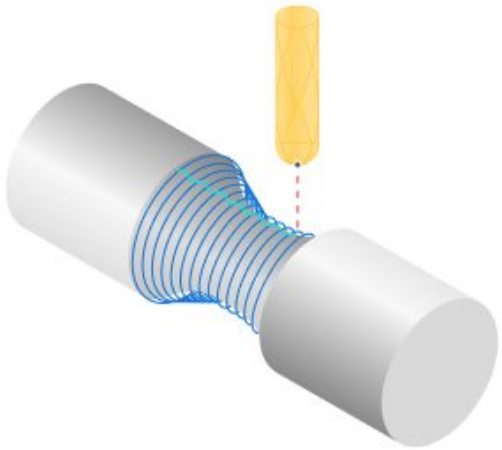
Advanced	
	Morph operation generates a toolpath that smoothly morphs between two specified curves with high speed links. Available strategies include: Across, Along, Spiral. 3 to 5 axis toolpath with the following tool axis orientation modes: Fixed, Normal to drive curves 4d/rotary axis/drive curves 5d/surfaces 5d. Benefits: Many operations for the machining of: turbine wheels, turbine blades, and screws, as well as complex channels etc. High speed links.
Undercut waterline	
	The Waterline Undercut operation is designed to create recesses, grooves, or cavities using an undercut milling tool - it generates a waterline specifically for undercut tools.
Swarf	

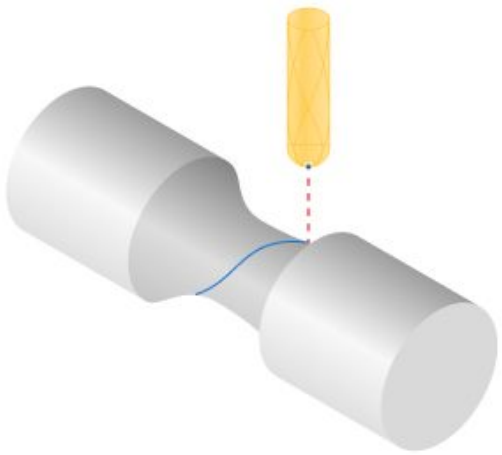
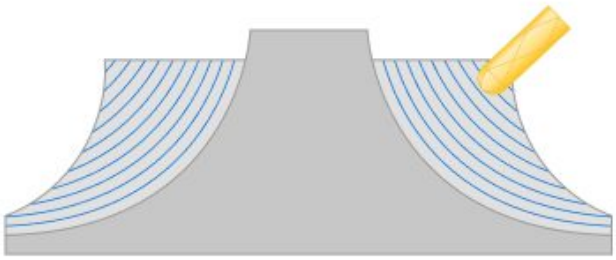
Advanced	
	The Swarf operation is a 5-axis operation for machining complex guided surfaces using the side face of a cutter. It is designed primarily for finishing (also supports roughing passes), allows for tool tilting (automatically or manually using vectors along the toolpath), and has two algorithms for defining the machining area: By Bottom Edge. By Two Curves.
5D Roughing waterline	
	A specialized 5-axis roughing operation for machining pockets with curved bottoms, based on the logic of the Waterline operation. The operation is assigned to "Floor surfaces" (usually the pocket bottom) and operates within a closed work zone (Job Zone).
5D by meshes	

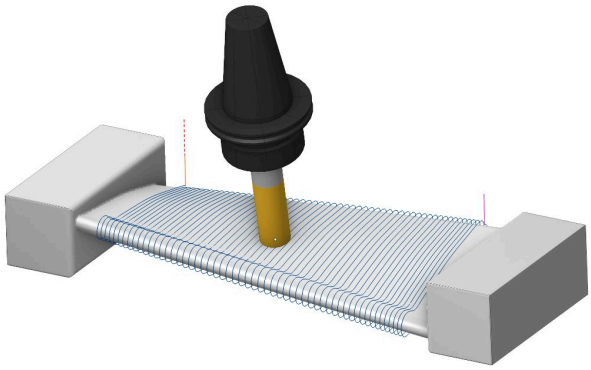
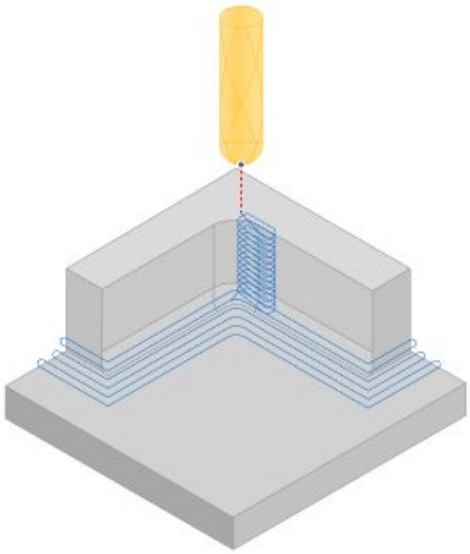
Advanced	
	A versatile 5-axis finishing operation for processing complex shapes and triangular meshes (mesh / STL). Suitable for finishing sculptures, scanned models, and complex surfaces, it supports Scallop, 3D Helical, and Waterline-like patterns, automatic/manual tool axis orientation, and automatic tool-holder collision avoidance.
6D Contouring	
	It generates a continuous multi-wave toolpath for spatial curves and surfaces with an extended set of tool axis orientation strategies and rotational axis constraint management.
Contour polishing	

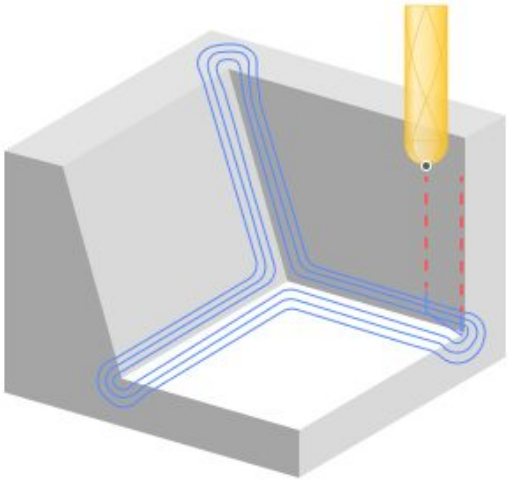
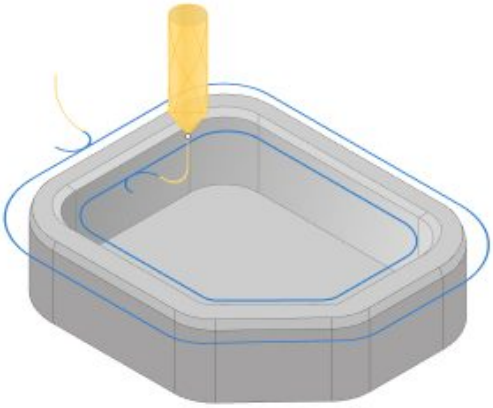
Advanced	
	Finishes part walls and edges along contours using a disc tool.
Rotary	
Rotary waterline	
	The rotary machining operation is used for the machining of the camshafts, crankshafts, worm shafts, paddles, decorate parts and so on. This operation is available if machine has at least one continuous rotary axis.
Rotary roughing	

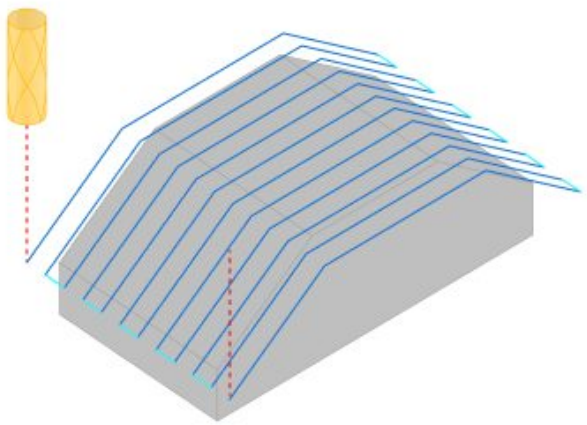
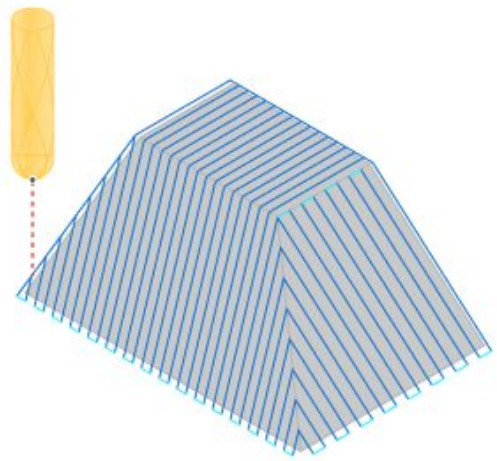
Rotary	
	Roughing rotary is a 4 axis toolpath that removes the workpiece material layer by layer. It is similar to the Roughing Waterline except that the machining layers are not planes, but cylinders around the rotary axis.
Rotary finishing	
	This is a 4-axis finishing operation for machines with at least one continuous rotating axis. It is used for finishing shafts and rotating bodies (common examples: camshafts, crankshafts, worm gears, blades, and decorative parts).
Morph 4D	

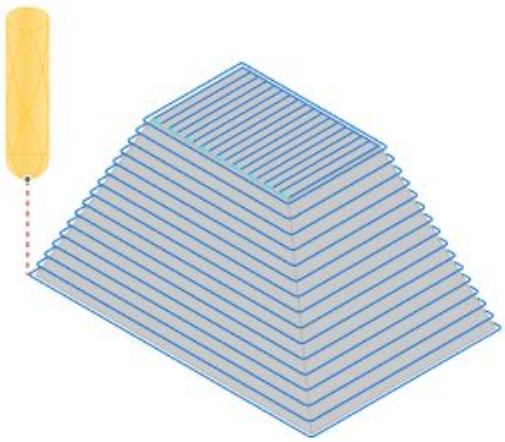
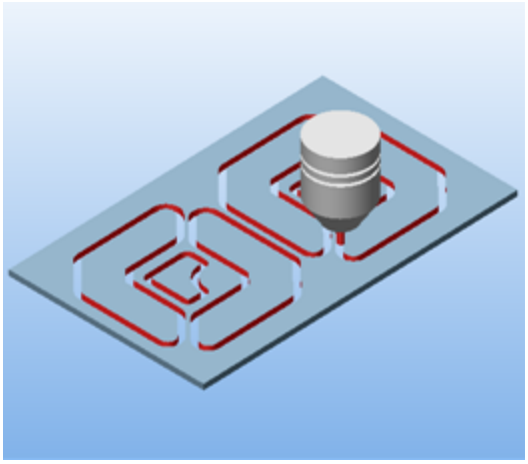
Rotary	
	This is a 4-axis finishing version of the Morph operation, adapted for machines with a continuous rotary axis (4x and 5x configurations). It generates a smooth toolpath between two specified curves (First/Second Curve) or between contact curves of selected Machining Surfaces, supports Sync Lines for improved morphing quality, and provides morphing strategies (Across, Along, Spiral, etc.).
4D Surfacing	
	A finishing operation adapted from 5D Surfacing for machines with a continuous rotary axis (4x and 5x configurations). This operation is designed for finishing complex surfaces with a clear rotation axis or extended along an axis: it generates surface passes, supports strategies such as Parallel to Curve, Morph Between Two Surfaces, and Spiral Between Two Surfaces, and provides control over the tool axis orientation (but disables some 5D strategies: Through Point, Through Curve, and Perpendicular to Toolpath).
4D Contouring	

Rotary	
	The 4D Contouring operation generates continuous toolpaths for machining spatial curves on surfaces with an explicit axis of revolution or extruded along an axis. It supports single passes along selected curves, multi-pass machining along isoparametric surface lines, and multi-level machining along a curve.
Turbomachinery Milling	
Impeller Machining	
	The Impeller Machining operation is designed for efficient 5-axis machining of blisks and impellers: it includes strategies for roughing the workpiece, finishing the blades and hub, and grinding/finishing fillets. The operation supports tool/holder/workpiece collision control and allows precise definition of machining areas: Blade Surfaces, Hub Surfaces (must be a surface of revolution), and Shroud Surfaces.
Blade Machining	

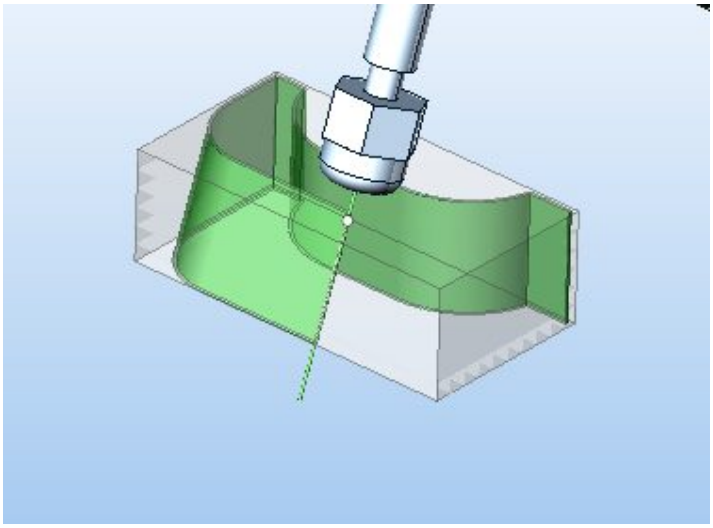
Turbomachinery Milling	
	Finishes blade airfoils, root zones, hubs, and edges.
Rest machining	
Corners cleanup	
	The rest machining operation takes the diameter of the previous tool as a parameter and generates passes where the previous tool would leave unmachined material.
Pencil	

Rest machining	
	The rest machining operation generates passes along inner corners of the part.
Chamfering	
	The Chamfering operation is used to create chamfers and fillets, or remove burrs along the edges of a part. It generates a toolpath for chamfering with cylindrical, conical, or ball-nosed cutters; the operation is most often used as a final step in chamfering/filleting, blunting sharp edges, and removing burrs.
Legacy operations	
Finishing plane	

[Legacy operations]	
	Plane machining of surfaces of a volume model. Passes lie in vertical-parallel planes. A good result can be achieved when machining flat areas and also areas close to the vertical that are perpendicular to the toolpath. Therefore, for machining of complex shaped models this operation is best used with the waterline or other plane operation, which has toolpaths perpendicular to the toolpath of the first operation.
Optimized plane	
	Two plane operations with mutually perpendicular toolpaths are created at a time for surface machining of a 3D detail. The default parameters of this operation are set so that every operation would machine only those surface areas of the model, where it can achieves an optimal result. This means that there will be a regular quality of machining on the entire model surface. Use of the optimized plane operation allows quality machining of models with difficult surface shapes, and also minimizes the machining time.
Complex	

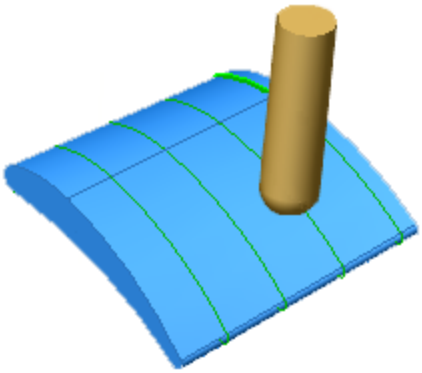
Legacy operations	
	Two operations are created: plane and waterline for surface machining of a 3D model. Parameters for the operations are set automatically so that the flat areas are machined using the plane operation and the areas close to vertical by the waterline. As a result, there would be a proportional quality of the entire surface of the machined detail. The complex machining provides easier conditions for the tool, this allows the use of longer tools with a smaller diameter. The operation allows performing quality machining for any surface angle, and also minimizes the machining time.
Cutting	
Jet cutting	
	The operation is used to carve the parts from the sheet. The outer contours and the contour of the holes can be defined by any closed or unclosed curve. The carving is performed by the tool motion along the part contours. The holes are cut in first and the outer contour is cut later.
4D Jet cutting	

Cutting



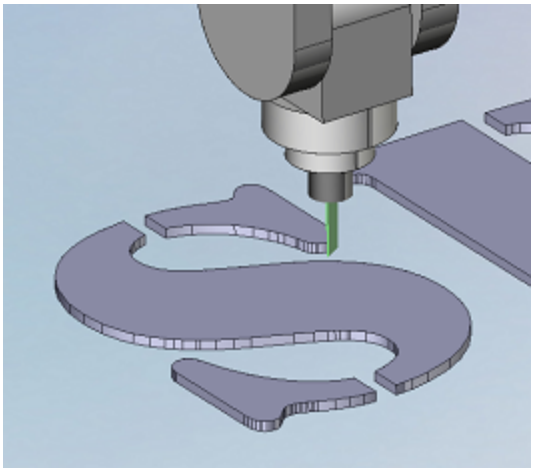
Jet cutting 4d operation can be used for hydro, laser, plasma etc. cutting types where the tool is a jet or a beam. It allows to machine simple elements and also more complex elements with inclined sides. Working contours are set the same way as in the Wire EDM operations, however, the resulting path is generated in the format of "point + normal" or "point + rotary axes of the machine."

Jet cutting 5D

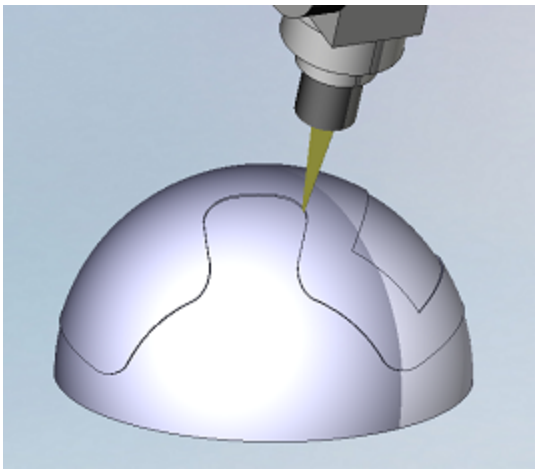
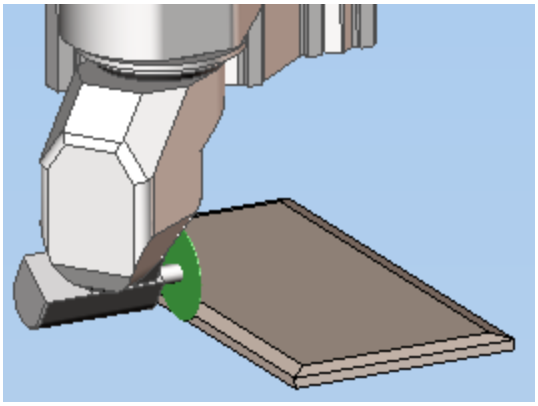
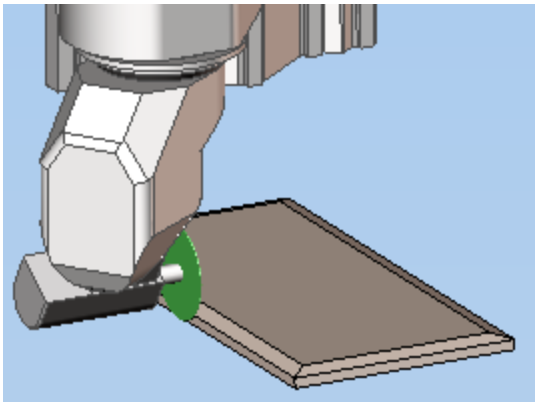


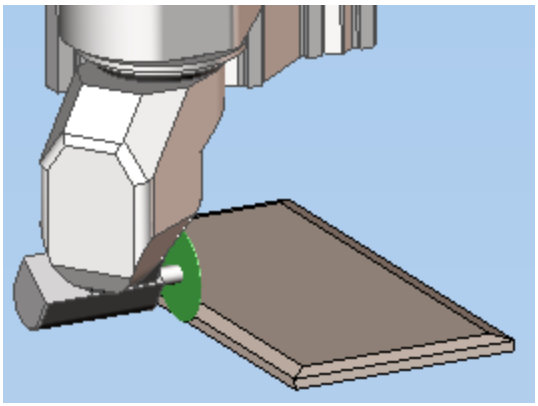
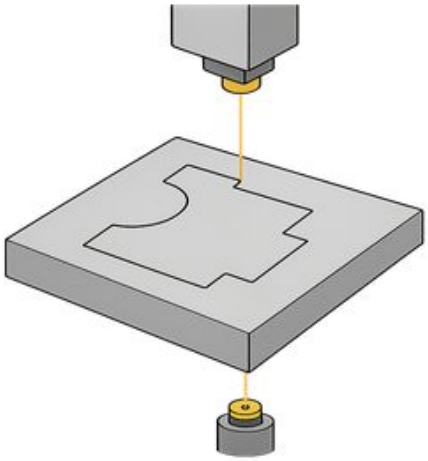
Operation Jet cutting 5D is designed for the cutting on the shaped spatial surfaces. It is based on the operation "5D contouring" excluding multipass machining feature unnecessary for this kind of application.

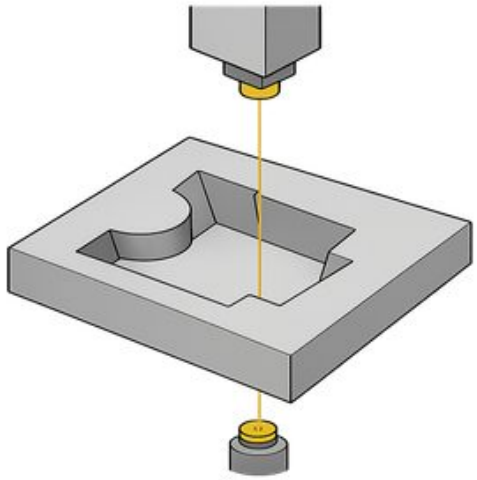
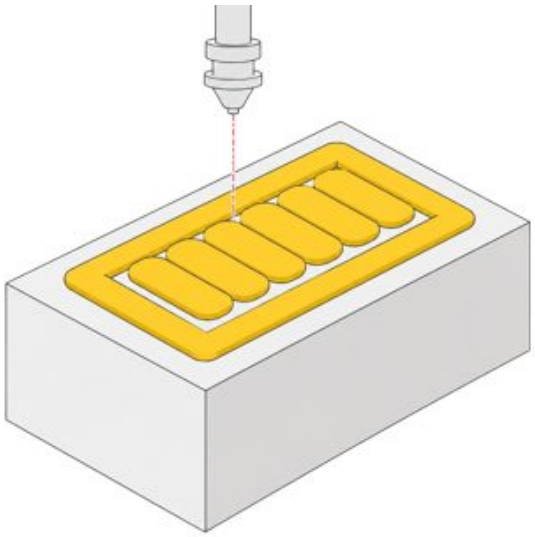
2D knife cutting

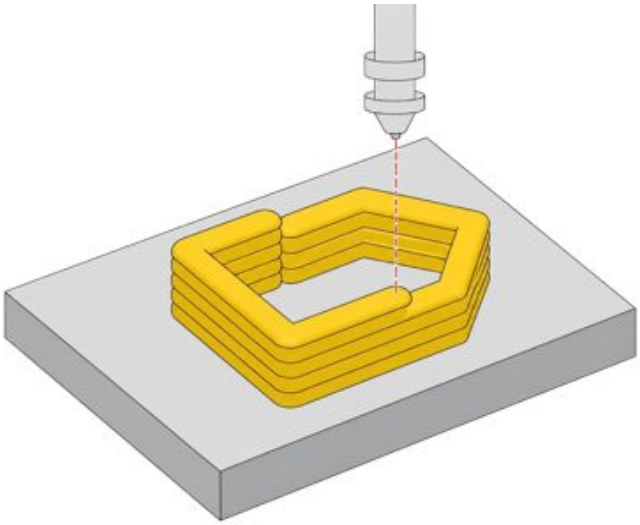
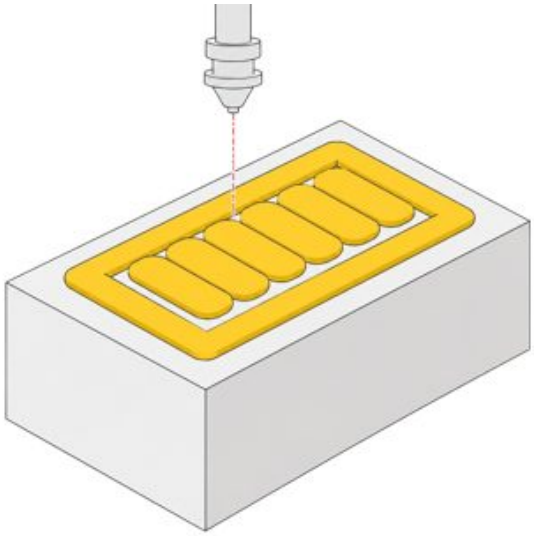


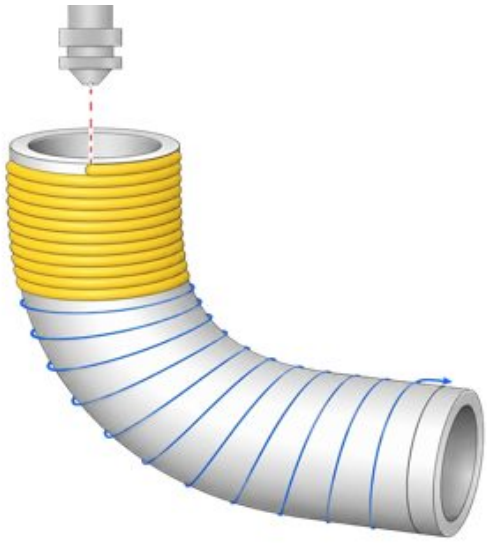
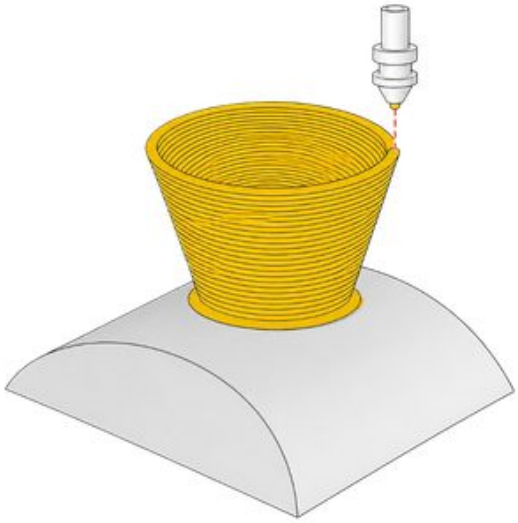
Knife cutting 2D operation designed to programming cutting of sheet material with the tool like knife (it can be knife, band-saw, disk-saw etc.). A special transition formed in the sharp corners of toolpath that avoids the bendings of the material because of the sharp turn of the knife. The operation based on 2D Contouring operation. The knife usage adds the additional requirements for the machine. The machine has to have, except the Linear X,Y,Z-axes, the additional rotary axis that rotates the tool around.

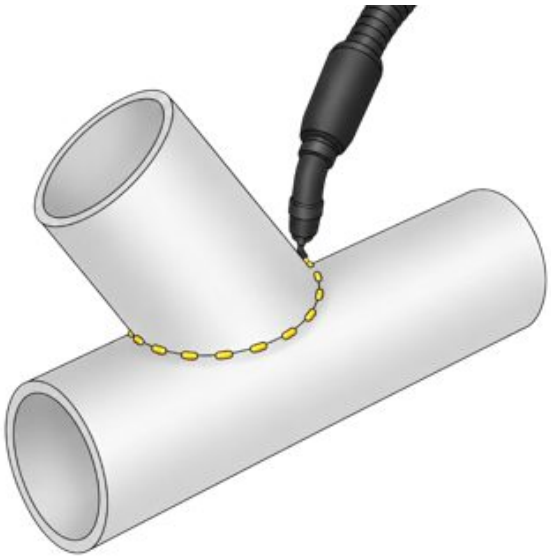
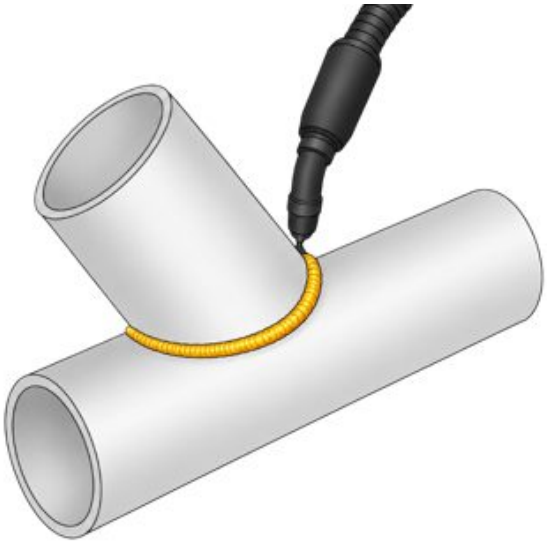
Cutting	
6D Knife cutting	
	Operation Knife cutting 6D is designed for the carving on the shaped spatial surfaces. It is based on the operation 5D contouring. A special transition formed in the sharp corners of toolpath that avoids the bendings of the material because of the sharp turn of the knife. In the every point of tool path the knife blade must be directed along the motion. It requires all 6 degrees of freedom. So active machine must have a minimum of three linear and three rotary axes. Very often the industrial robots are used for the knife cutting.
Disc tool	
Disc Roughing	
	An operation from the Disc tool machining group, designed for preparing a mass of stone (and similar) blanks with a disc tool: a series of cuts are made with a disc in the areas where material needs to be removed, then thin layers are knocked out manually (spalling), after which finishing is usually performed with a suitable tool.
Disc cutting 2D	
	An operation for cutting sheet materials with a circular tool (saw).

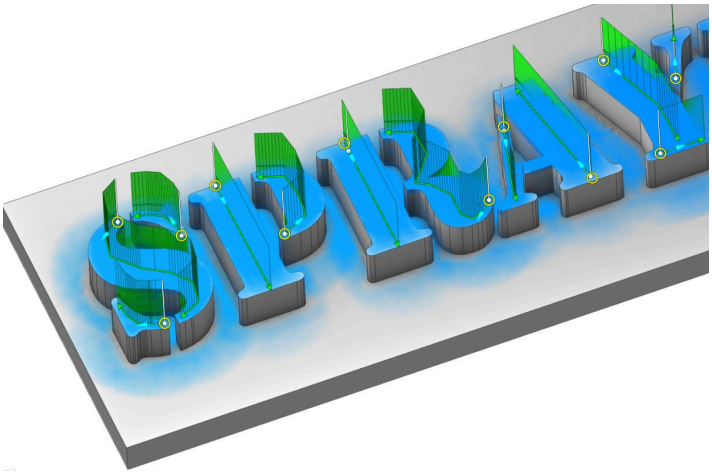
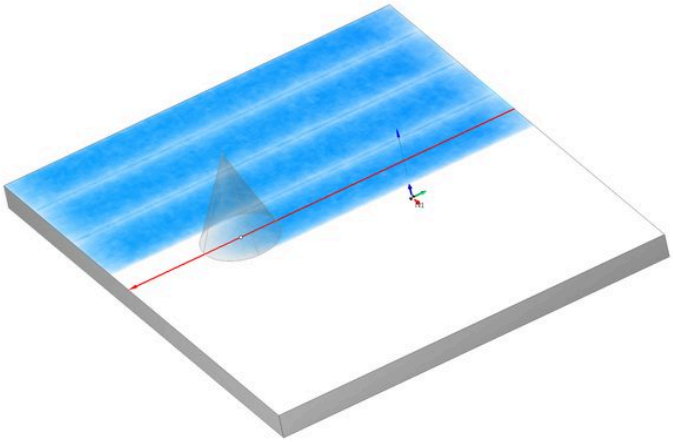
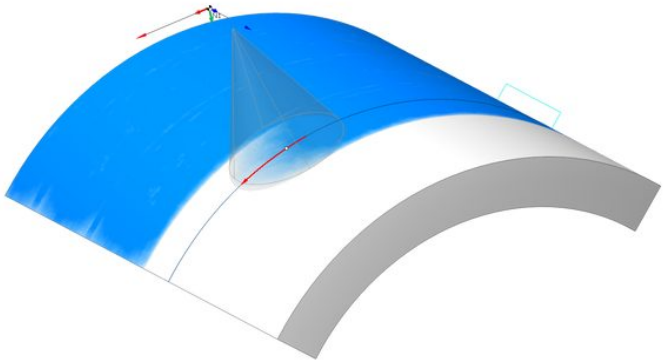
Disc tool	
Disc cutting 6D	
	The Disc cutting 6D operation is designed for sawing/cutting both flat and volumetric parts using a disc tool.
Wire EDM	
Wire EDM 2D Contouring	
	The Wire EDM 2d Contouring operation is designed for wire path generation along flat contour on 2d contouring as well as along flat contour with wire slope angle on taper or 3d contouring. Resulting wire path is based on contours, which lays in one plane.
Wire EDM 4D Contouring	

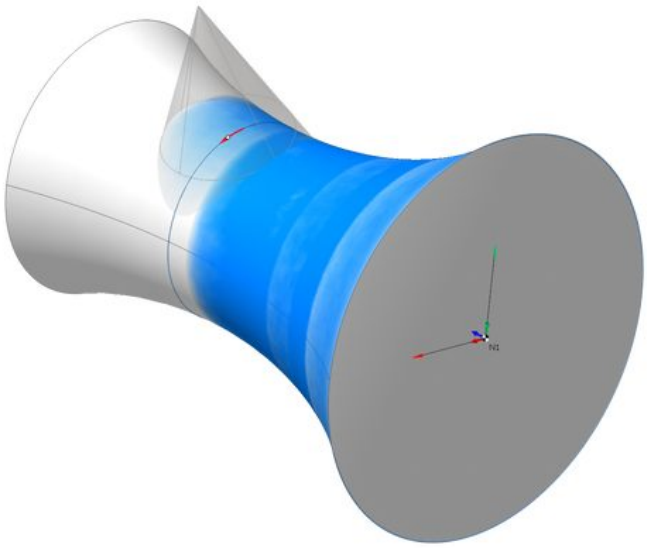
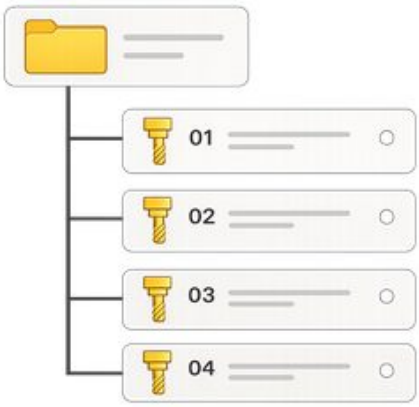
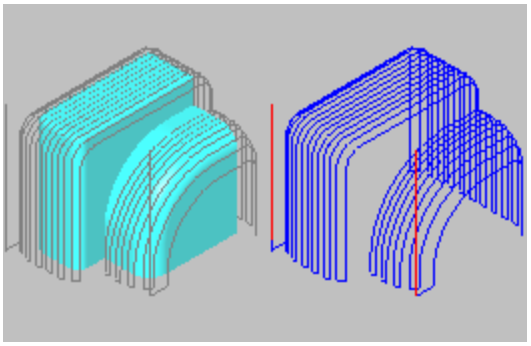
Wire EDM	
	The Wire EDM 4d Contouring operation is designed for wire path generation along two flat contours simultaneously. One of this contours is set moves of lower guide of wire EDM machine, to put it more precisely – moves in working (XY) contour plane. Second contour is set moves of upper guide of wire EDM machine – leading (UV) contour. Thus, in the operation upper and lower wire ends can to moves on different paths.
Additive	
Area cladding	
	It implements the concept of additive manufacturing, when, in contrast to a cutting the material is not removed, but added to the workpiece during the machining process. It allows, for example, to build on the surface of the workpiece the layer of material having specific characteristics: high hardness, strength, wear resistance, anti-friction properties, corrosion and heat resistance, etc. It allows also to restore the geometric dimensions of costly parts and tools, to repair blades, dies, molds, gears, shafts, etc. The interface of job zone definition and the set of parameters is similar to the pocketing operation. It allows using curves and edges of the 3D model to restrict the area in which you want to make a buildup of material. Depend on the selected base surface this area can be positioned on the plane, cylinder or on the revolution body. And when the "Project toolpath onto the part" option is enabled, cladding in general can be made on the surface of an arbitrary shape. Operation has Parallel and Offset strategies to fill the area. You also can

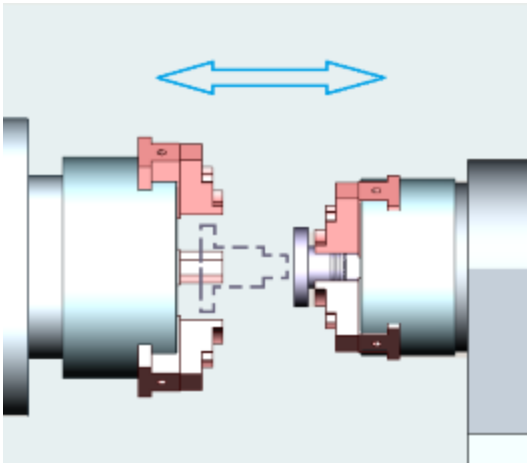
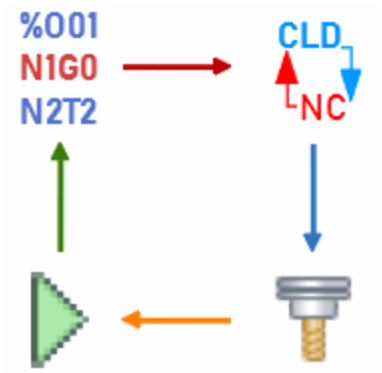
Additive	
	define total layer count and side angle for the walls.
Curve cladding	
	Additive operations that generates toolpath along curves defined inside job assignment from the bottom to top. It is useful for thin-walled models. Source curves can be placed on a plane, cylinder or body of revolution. And when the "Project toolpath onto the part" option is enabled, cladding in general can be made on the surface of an arbitrary shape. It can generate layer by layer like toolpath or helix spiral.
Cladding 3D	
	Additive operation that has 3D model at the input. It is similar to Roughing waterline operation except that it works from the bottom to top. It intersects source model layer by layer and generates toolpath to fill calculated intersection area for each level. Operation has Parallel and Offset strategies to fill the area.
Cladding 5D operation	

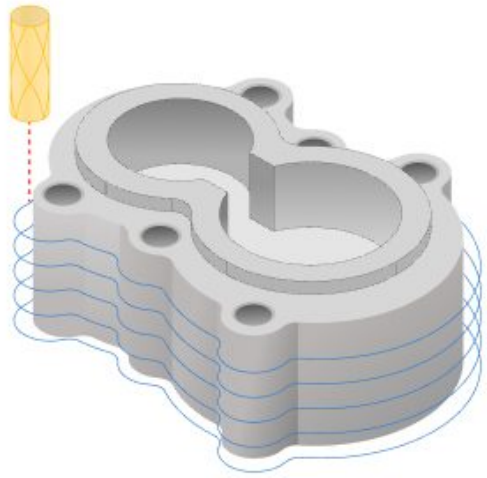
Additive	
	Cladding 5D operation allows you to build up a layer of material on the surface of a part on 3- or 5-axis machines. It is useful for processing thin-walled models. The operation allows surfacing of individual surfaces of the part with their subsequent milling. It can also serve as hardening of surfaces by surfacing material in the most loaded areas of the part. Spiral strategies and parameters have been added to the operation, which will make it possible to avoid passing the tool in the same place several times. Also, the operation can use the following strategies: Parallel to plane, Morph, Parallel to curve.
Non-planar slicing	
	The Non-planar slicing operation creates additive printing paths on an arbitrary (non-planar) substrate: it uses the non-planar slicer engine and generates non-planar print paths; if necessary, the classic planar strategy with additional parameters is also supported.
Welding	
Point welding	

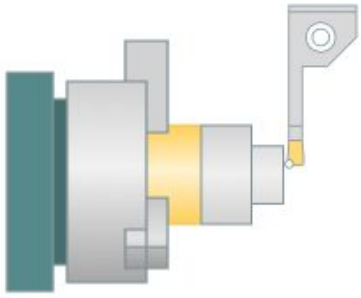
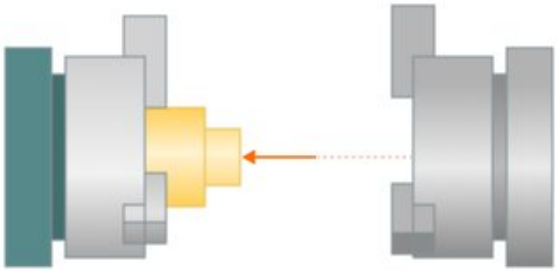
Welding	
	Used for spot welding - supports both tack welding and spot welding.
Welding 6D	
	It implements the functional of automatic weld seam geometry calculation without reference to a particular type of welding equipment (i.e., does not generate the specific commands to the laser, electric arc, gas burners, ultrasonic device, etc.). It is enough to add the edge between welded parts to the Job assignment and the system automatically calculates the angles in each curve point so that the welding head is held as close to the middle between the adjacent walls and do not collide with them. Then you can switch to the Simulation mode to see how the material is added to the place where the tip of the welding head is touching.
Spray	
Contour spraying	

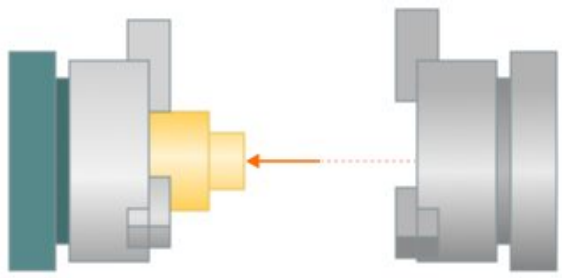
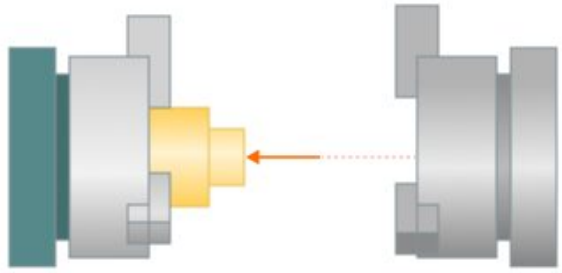
Spray	
	Contour spraying operation based on 6D Contouring operation. You can use this operation if you need more flexible control of a tool position in each point of the toolpath.
Surface spraying	
	Surface spraying operation based on Cladding 5D operation operation. You can use many useful strategies to create a toolpath for painting on surfaces
Morph spraying	
	Morph spraying operation based on Morph operation
Rotary spraying	

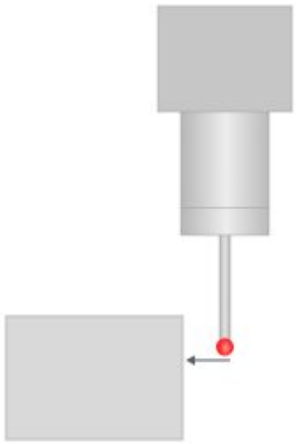
Spray	
	Rotary spraying operation based on Rotary finishing operation
Auxiliary	
Group	
	The Group is intended for systematization of different operations with similar parameters. It is possible to form a job list with tree structure by operations groups. If some group parameters are changed then similar parameters of all included operations will be changed too.
Multiply Group	
	A group that defines the methods for copying paths; these rules are then inherited by operations within the group. This allows operations to process individual features of a part, while the group itself automatically propagates the results to all standard features according to the defined rules. Copying is performed according to the rules in the Transformations tab.

Auxiliary	
Auxiliary operation	
	Auxiliary operations of CAM system designed to store the specific sequence of CLData commands (for specific types of machines, for particular company) into the named list, which can be saved and used many times in a process of work with the system. This, for example, may be such types of operations as clamping a chuck, tool interchange, approaching tail stock, part overturn, set of the active workpiece coordinate system G54-G59 and so on.
G-code based, G-code based lathe	
	Operations are intended to form a tool path on the basis of NC text, that is used as the job assignment, and the selected interpreter. The NC text can be written manually or can be loaded from an external file and edited, if necessary. Its application is also possible for the indexed and continuous machining on the 4 and 5-axes machining centers. All available simulation types are supported, including additive manufacturing to simulate material layer buildup. Using these operations you can perform direct control of the machine simulation using G-codes, check and optimize the NC program, convert the text of the NC from one controller to another, debug your own interpreter during its creation.
Make shaped cutter	

Auxiliary	
	To create a shaped milling tool in ENCY, draw the tool outline as a generator line in Drawing mode (you can obtain a curve from a model or create it using Sketch). Then, right-click in the tool list and select Save as tool. Select the shape type, name, and tool scheme. Once saved, the outline will be available in the tool library and can be selected in the Tools window for use in contouring/profile milling operations.
Move part	
Point Pick and place	
	Point Pick-and-Place is a variant of the Pick-and-Place operation in which the Job assignment is based on a set of point nodes. The list of points includes: Start point, Pick point, Move point (relative to Pick), Move point (relative to Place), Place point, and End point; these points are collected into a chain of trajectories (Pick → Move → Place, etc.).
Abstract pick-n-place - Legacy	
	Abstract Pick-and-place (general operation Pick-and-place) is the operation of controlling the gripper to move the workpiece within the working area of the machine.
Bar feeding	

Move part	
	Bar feeding is the initial feed operation for the bar into the spindle (it should be the first operation in a Swiss-lathe project). It positions and secures the workpiece, sets the bar extension parameters, the tool contact position, controls the initial clamp, and so on.
Sub spindle working	
	Sub spindle working is an adapted version of the pick-and-place operation for turn-milling machines with a subspindle. This operation allows you to: synchronize the main and subspindles. grasp the workpiece with the subspindle for further machining on both spindles. move the part between the main and subspindles.
Turn take over	

Move part	
	Turn take over is a part transfer/intercept operation between spindles (pick-and-place for turning projects). Its purpose is to synchronize the main and sub-spindles, intercept or return the part, automatically retrieve the Workpiece Connector and Setup CS from the next part of the project, integrate with the Part-off operation, manage clamps (CLData), and generate movement commands (Multigoto/Goto).
Place to next stage - Legacy	
	Place to next stage — a special type of Pick-and-place that takes the placement position from the next operation (Setup stage / Part). Used for robots and milling machines; for turning projects, Turn take over is preferred.
Probing	
Tool probing	

Probing	
	Tool probing is a cycle/operation for checking and calibrating tools: the length/radius of the tool is measured, the integrity/breakage of the tool is checked, and the initial dimensions of the tool are calibrated.
Part probing	
	Part probing (workpiece/part measurement) is a cycle for measuring the geometry of a workpiece: checking the dimensions of surfaces, holes, position and orientation of elements, determining the actual position of the part (offset/rotation compensation), checking angles, etc.
Turn tool probing	
	Turn tool probing is a tool measurement/calibration operation in the turning section: with its help, the system measures the length and/or radius of the cutter, checks the integrity/breakage of the tool and obtains real parameters for adjusting tool compensation in subsequent operations.

Probing	
Turn part probing	
	Turn part probing is the process of measuring the geometry of a workpiece on a turning fixture: shapes/dimensions (e.g., holes, ends, surfaces) are measured, the position and orientation of the part in the clamp is checked, dimensions are taken to compensate for deviations, and reports are generated.

Basic technology terms

This section explains basic technology terms: group, part, workpiece, machining result, tool, stock, angles, steps and machining methods, the tool magazine, and tool compensation.

Group

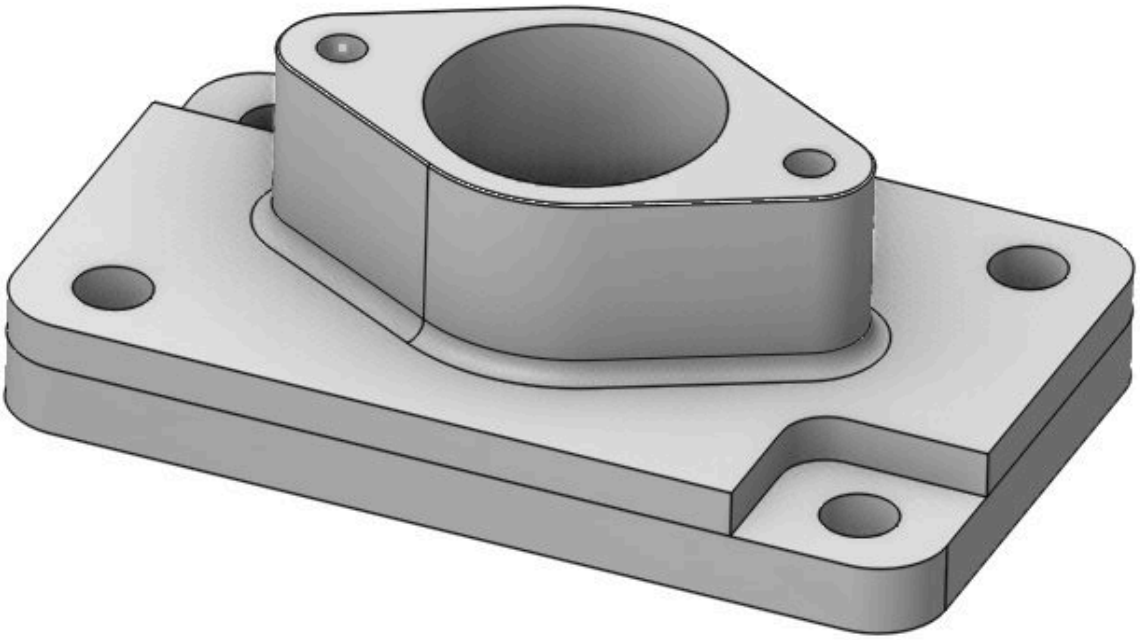
The group is intended for systematization of different operations with similar parameters. Creating a group does not result in any toolpath calculations, but parameters may be set for all operations in the group.

The operations group may include operations of any type, including operations groups. That is, using operations groups it is possible to form a job list with structure like a tree. Assignment and behavior of operations group are similar to the folder in a file system of the computer.

If some group parameters are changed then similar parameters of all included operations will be changed too (if operation from the group contains such parameters).

Operation is expedient to apply at machining parts by several operations with similar parameters. In this case it is possible to reduce time of set-up of operations parameters (avoiding repeated parameters setting).

Part



Application area:

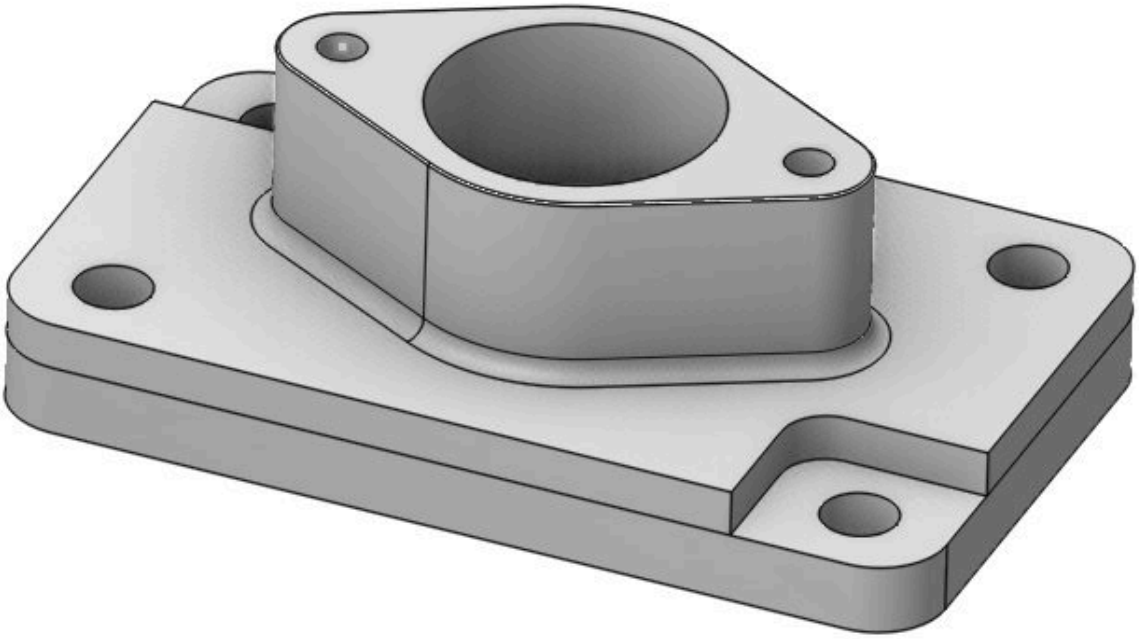
A Part is a group of geometrical elements that defines the space to check for gouges. A part is always to consist of surfaces. Those may be either specified explicitly or generated from curves by some way. For example, it is possible to define a revolution surface by its generating curve or a prism by its base curve.

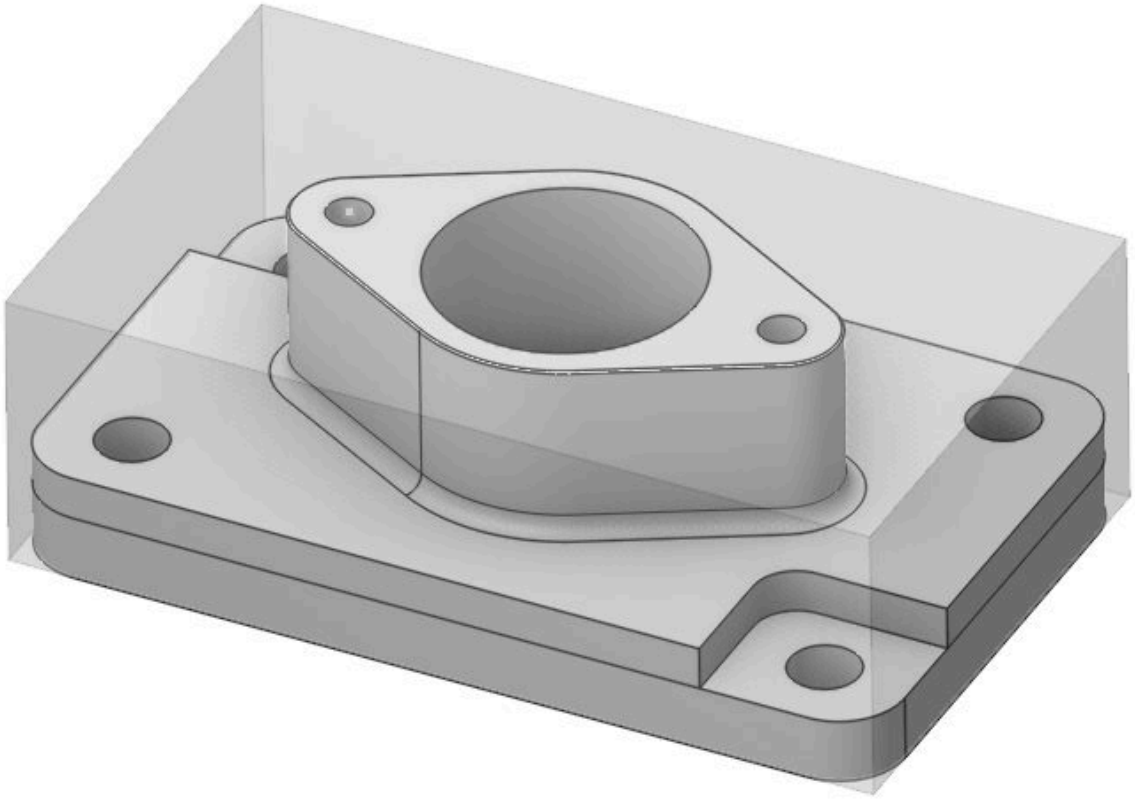
The part is usually specified in the root node of the machining tree. Thus all the operations use the same part by default since the default part of an operation is a reference to the part of the preceding operation. Optionally one can alter this rule by overriding the default part of an actual operation.

The surfaces of the part are used to eliminate undesirable part gouges while the surfaces to be machined are specified in the Job Assignment node.

Methods for determining the part are indicated here. Details

Workpiece





Application area:

A workpiece model of an operation defines the material to be machined. This means that it defines the initial shape of the workpiece from which the required finished component will be produced.

The roughing operations perform machining of the entire workpiece that lies outside of the model being machined. For the finishing operations the system machines surface areas that lie inside the workpiece area.

The initial shape of the workpiece is specified in the root node of the machining tree. It can be defined as a sum of the following elements:

- A box swept around the part;
- Box around model;
- Box around Job Assignment (only for 25D operations);
- Box around a geometrical model;
- Box by a midpoint and size, by an angular point and size, by two points);
- A cylinder swept around the part. The axis of the cylinder is the X axis.
- The cylinder around Job Assignment (only for 25D operations); calculated
- The cylinder around a geometrical model;
- The cylinder by central point, radius and altitudes;
- A prism extruded from the specified curves,
- A revolution body defined by its generating curves.
- A solid designed in a stand-alone CAD. All the surfaces should represent a bounded space in that case.
- A surface model designed in a stand-alone CAD can be converted into a solid by closing it faces to the specified Z level.

For elements A cylinder swept around the part radius of cylinder is calculated by part faces not a part box.

For all cylinders it is possible to be allocated along any of principal axis, to be described around box, a inscribed in box, with an axis in the middle of box or in an origin point of co-ordinates.

Also Auto, an definition type of workpiece or model which will be active if in the appropriate list of a model and workpiece it is added nothing has been added.

From the first operation to the last one the workpiece changes its shape as result of its machining by each operation. The intermediate workpiece is transmitted over operations since each operation has the default workpiece as the Machining result of the previous operation. By that way the capability to machine only the rest material is realized. If the result of a previous operation should be eliminated by toolpath calculation that always can be done by replacing the default workpiece with the Workpiece of the previous operation item.

Methods for determining the workpiece are indicated here. Details

Machining result

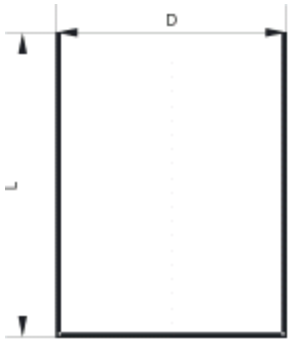
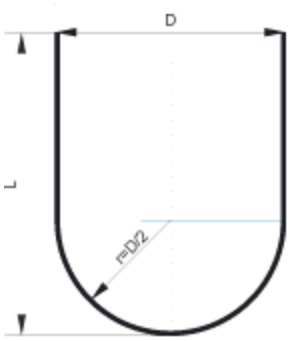
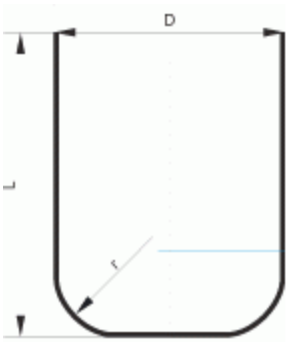
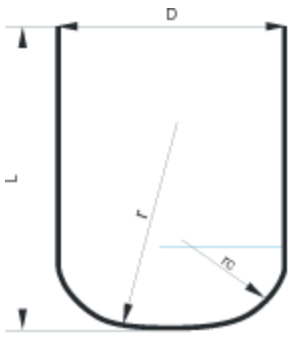
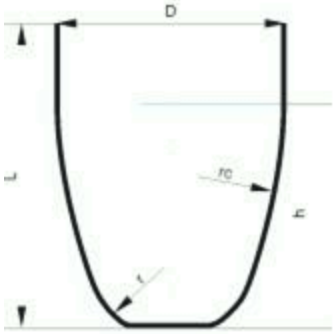
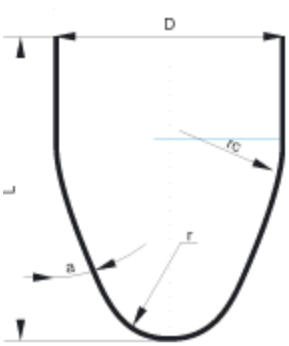
The Machining result is the material leaved by an operation after machining the workpiece. The node is introduced to allow a visual control of the rest material as well as for transparence of the Machining result of the previous operation item of the Workpiece node. The node is calculated automatically while its contents can not be modified. If the toolpath of an operation is not yet calculated then the machining result of the operation is its workpiece.

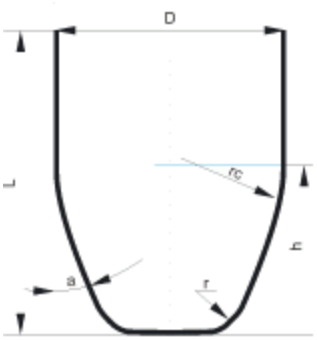
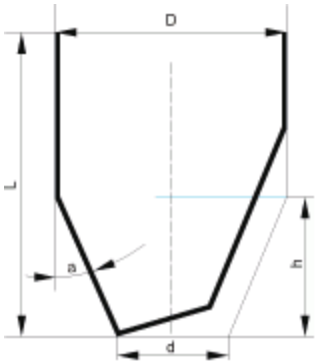
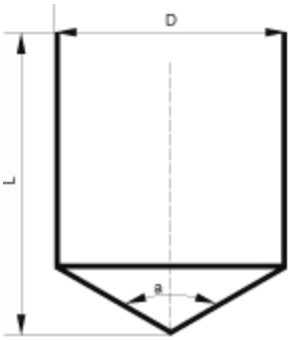
Tool

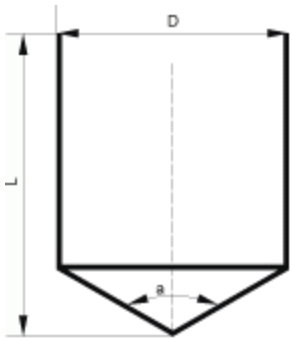
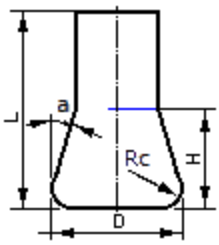
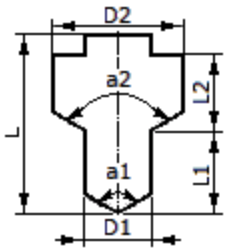
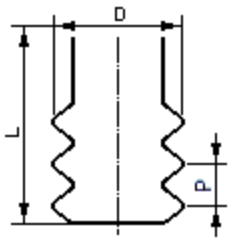
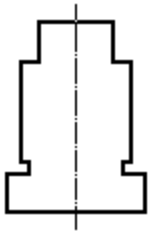
The toolpath calculation is performed with the tool tuning point taken into consideration. The tool parameters are set on the Tool tab of the operation parameters dialog.

Mill tool

Calculation of machining toolpath in milling operations is performed for various types of tools which differ by the form of the profiling part and the parameters used to describe it. Supported tool types with their geometrical parameters are listed in the table below:

Cylindrical mill: length, diameter. **L** - Length **D** - Diameter	Spherical mill: length, diameter. **L** - Length **D** - Diameter	Torus mill: length, diameter, rounding radius. **L** - Length **D** - Diameter **r = D/2** - Rounding radius
		
Double radial mill: length, diameter, rounding radius at the cylindrical part, rounding radius of the peak. L - Length D - Diameter Rc - Cylindrical-part rounding radius r - Peak rounding radius	Limited double radial mill: length, diameter, rounding radius at the cylindrical part, rounding radius at the peak, height. L - Length D - Diameter Rc - Cylindrical-part rounding radius r - Peak rounding radius H - Height	Conical mill: length, diameter, rounding radius at the cylindrical part, rounding radius of the peak, angle. L - Length D - Diameter Rc - Cylindrical-part rounding radius r - Peak rounding radius a1 - Angle
		

Cylindrical mill: length, diameter. **L** - Length **D** - Diameter	Spherical mill: length, diameter. **L** - Length **D** - Diameter	Torus mill: length, diameter, rounding radius. **L** - Length **D** - Diameter **r = D/2** - Rounding radius
Limited conical mill: length, diameter, rounding radius at the cylindrical part, rounding radius at the peak, angle, height. L - Length D - Diameter Rc - Cylindrical-part rounding radius r - Peak rounding radius a1 - Angle H - Height	Engraver: length, diameter, angle, height, peak diameter. L - Length D - Diameter a1 - Angle H - Height D1 - Peak diameter	Drill: length, diameter, grinding angle. L - Length D - Diameter a1 - Grinding angle
		
Center drill:	Undercut mill:	Two stage drill. L1 - First-stage length D1 - First-stage diameter a1 - First-stage angle L2 - Second-stage length D2 - Second-stage diameter a2 - Second-stage angle

Cylindrical mill: length, diameter. **L** - Length **D** - Diameter	Spherical mill: length, diameter. **L** - Length **D** - Diameter	Torus mill: length, diameter, rounding radius. **L** - Length **D** - Diameter **r = D/2** - Rounding radius
		
Threading mill:	Shaped tool:	
		

Besides the parameters that describe the shape of the mill, the user can also define for the tool:

- linear dimensions assignment units (millimeters or inches);
- rotation direction (CW or CCW);
- number of teeth;
- material;
- durability (in hours).

The NC program can be calculated for the end or the central tool programmed point. The end-programmed point implies a point on the tool rotation axis with the Z coordinate equal to the very bottom cutting point of the tool. The central programmed point – is the point on the tool axis with the Z coordinate equal to the top edge level of the profiled part of the tool (or the bottom edge of the cylindrical part).

Mill parameters can be defined on the Tool page.

Lathe tool

The tools used in turning operations can be divided into two categories: the lathe tools and the axial tools. Each category in turn can be divided into groups regard to the operations they use.

Lathe tools:

- Bore tools;
- Straight turn tools;
- External threading tools;
- Internal threading tools;
- External grooving tools;
- Internal grooving tools;
- Face grooving tools.

Axial turn tools:

- End mills;
- Drills;
- Center drills;
- Taps;
- Counterbores;
- Countersinks.

A tool consists of two parts: an insert and a holder, which should strictly correspond to each other. Optionally it is possible to set a freeform holder for an insert. To do that one need select the holder type. This holder type can be combined with any insert type of the given group. With a insert only a holder can be used however.

In addition to the geometrical parameters of a tool it is possible to define:

- the spindle position (left or right);
- the spindle rotation direction (clockwise or counterclockwise);
- the position of the tool tuning point;
- the tool orientation (radial, axial or at the specified angle);
- the support position (upper or lower).

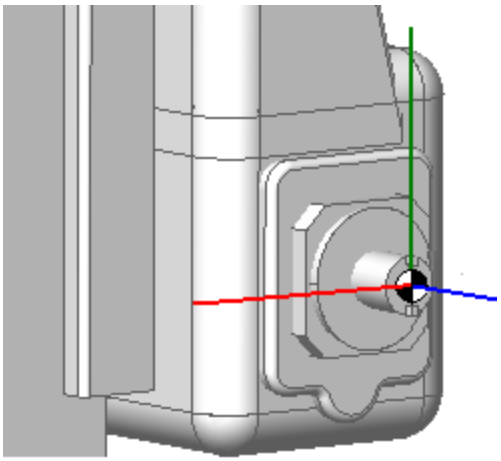
Tool overhang

Tool overhang is distance from tool base point to tooling point.

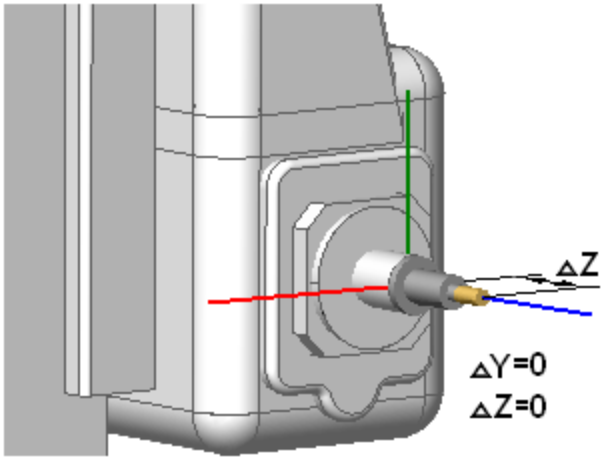
Tool base point is base point of instrument block (tool and tool holder) at machine coordinate system.

For mill machine tooling point lie along tool axis.

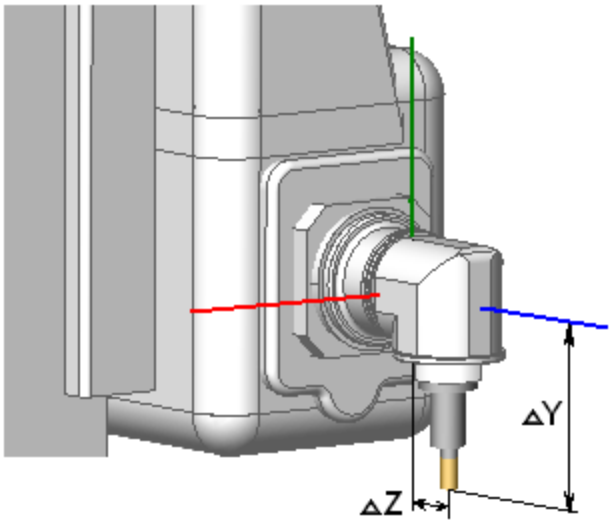
Some overhang definitions for mill machines are shows below.



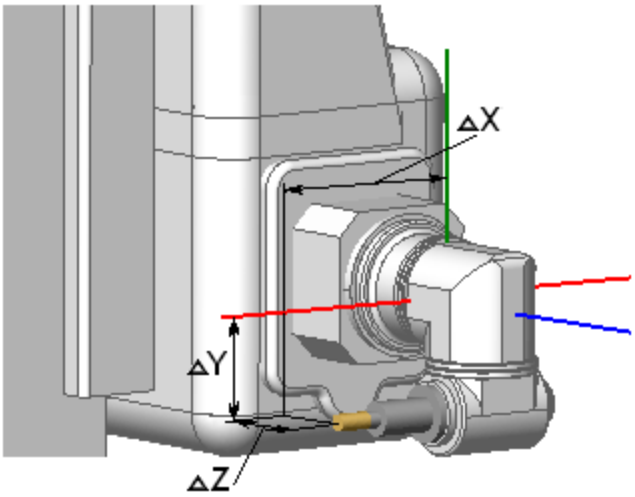
Tool base point



Axial tool overhang



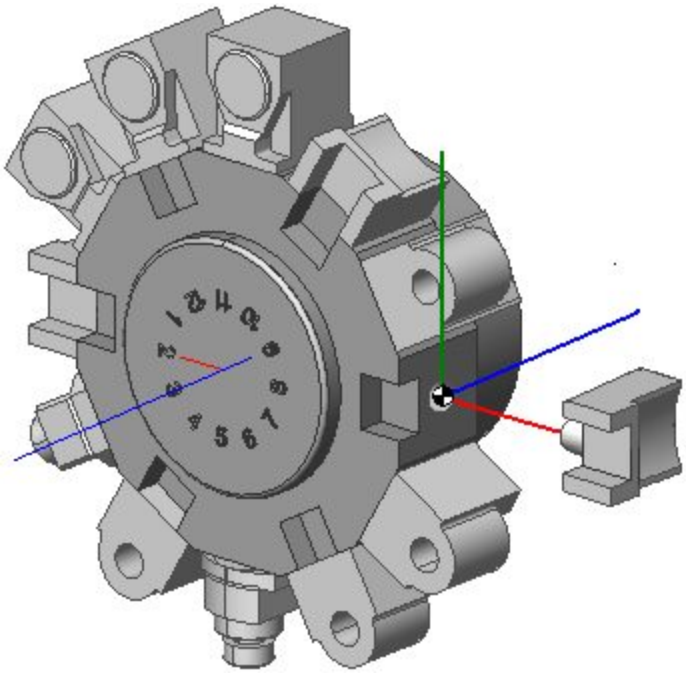
Overhang for rotary head



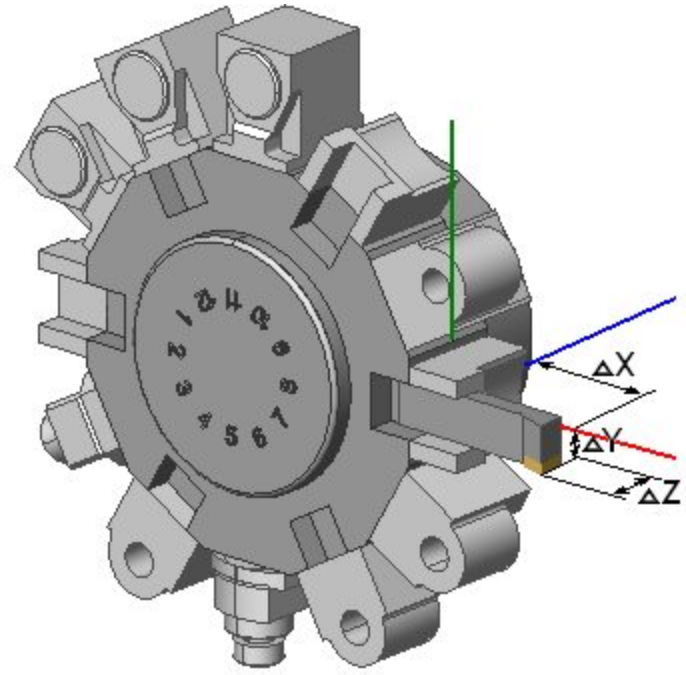
Overhang for 2 rotary heads

For lathe machines base point lie on turret base plane along tool fix system axis.

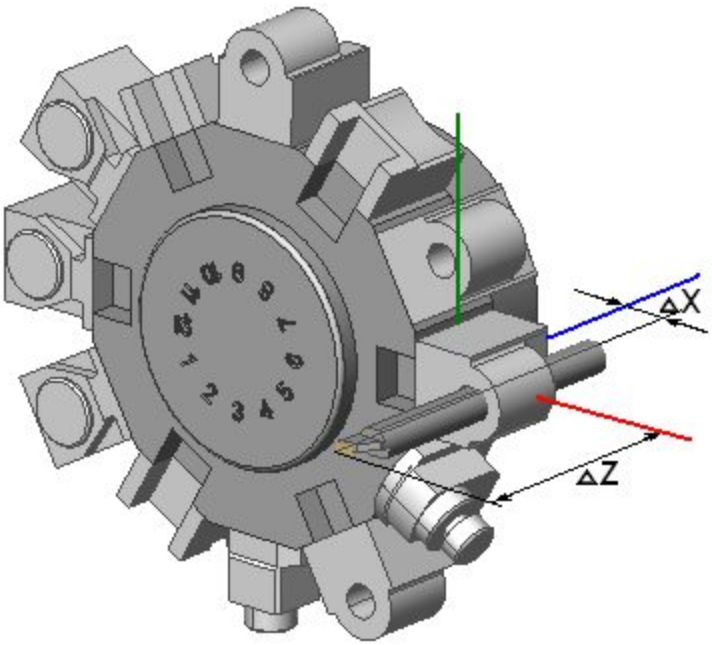
Some overhang definitions for mill-lathe machines are shows below.



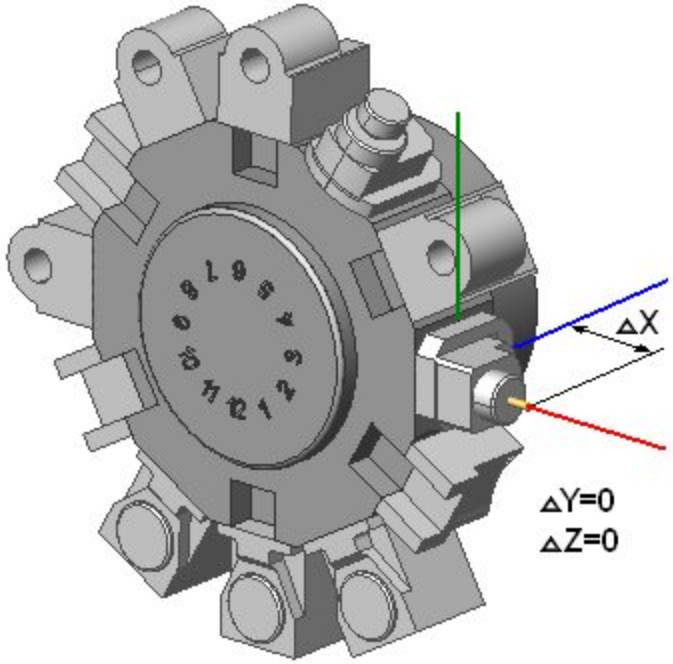
Base tool point on turret



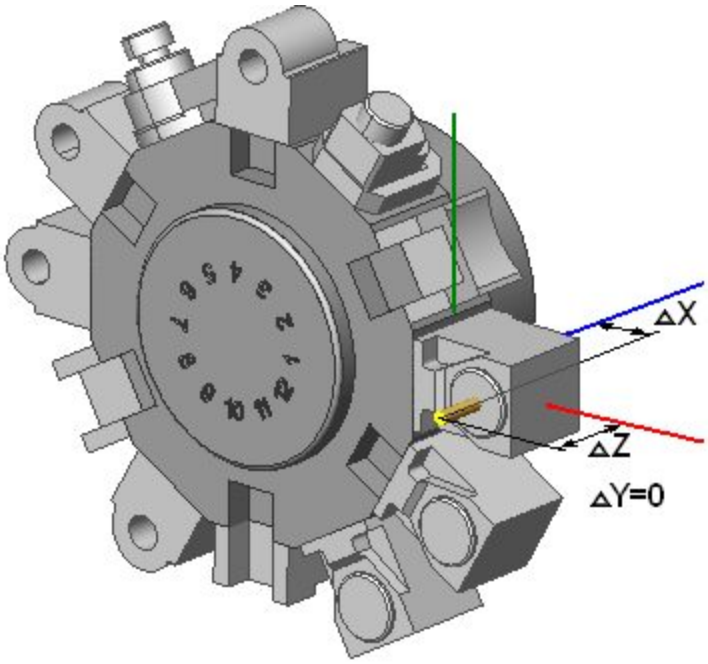
Straight turning tool overhang



Boring bit overhang



Base point for radial milling head



Base point for axial mill head

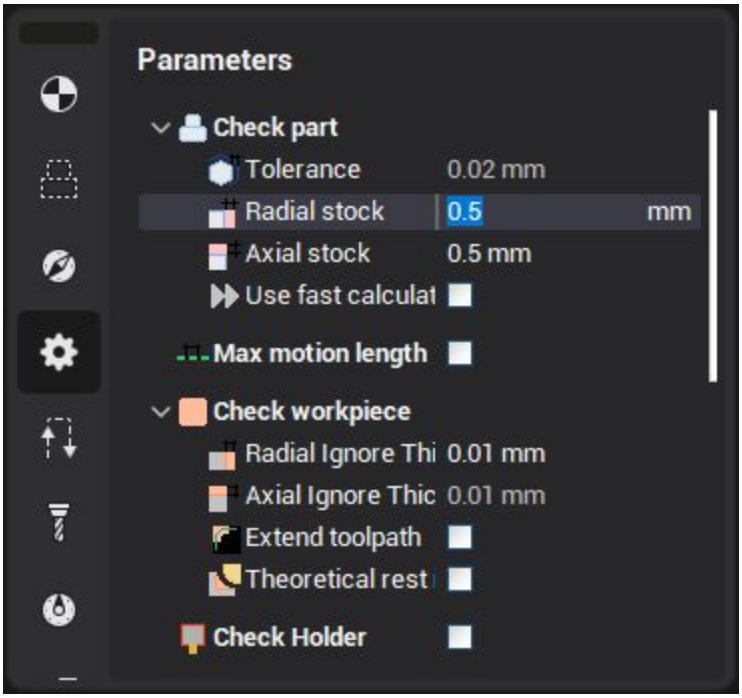
Stock

The stock is a layer of material on the detail surface, which needs to be left after an operation for further rest milling.

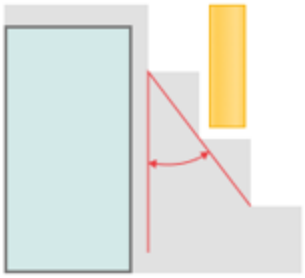
A positive stock defines the excess thickness of the material to be left, and negative – the material removed from the detail surface.

In reality, the minimum thickness of the remaining material is strictly equal to the stock only when the inner deviation value is zero. In other cases, the minimum layer thickness is less then the defined stock by the inner deviation amount.

The stock can be assigned in the Parameters page.



Relief Angle



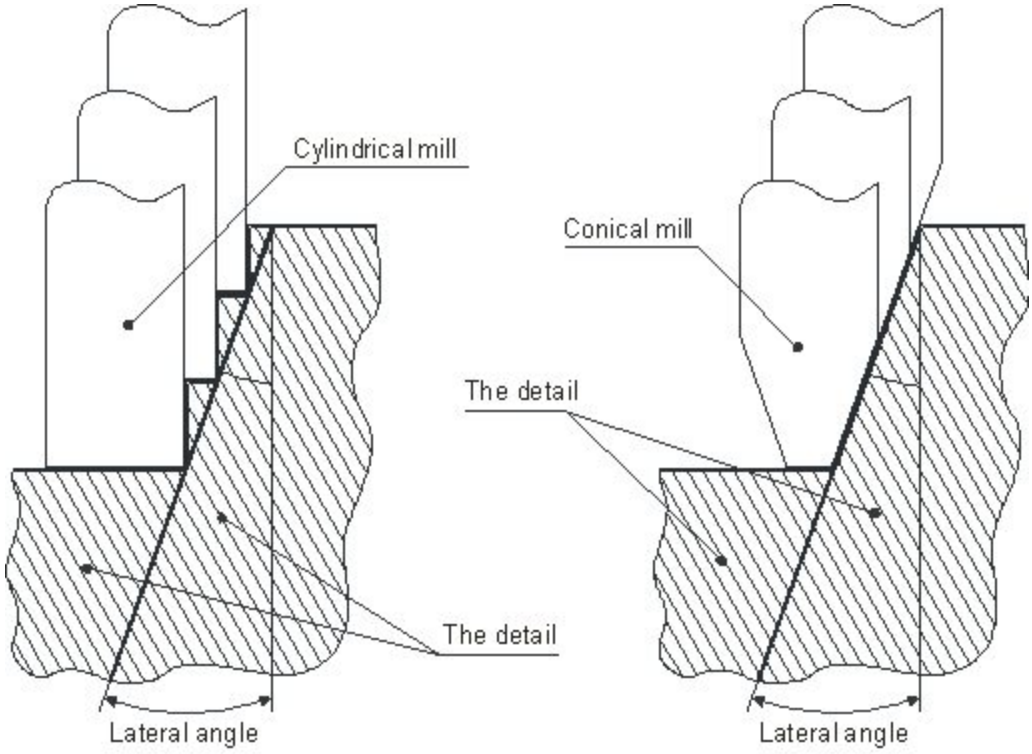
Application area:

When machining deep vertical walls, it is sometimes needed to restrict tool contact with the already machined part. For this purpose, in the operations that perform machining by layers, it is possible to assign a relief angle. When machining a vertical (or close to vertical) surface, the real stock will increase at machining layer as shown in the picture (below). The relief angle is allowed for all walls that are close to vertical, i.e. not only close for the model being machined, but also the restricting ones.

The relief angle value cannot be negative, nor can it be more than or equal to 90 degrees. Setting the relief angle too high will lead to a large layer of unmachined material at the lower layers.

Lateral Angle

The lateral angle defines the slope of the side surface of a (curve) model being machined in the engraving and pocketing operations. For finish machining (engraving) the system selects an engraving tool that has a conical angle equal to the lateral angle of the model.



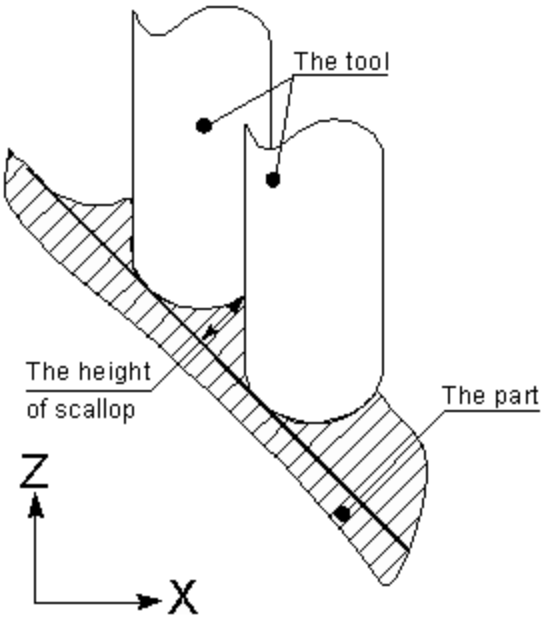
If the lateral angle is equal to zero, then the model being machined will represent an extrusion, the basis of which is defined by an area at the top machining level. If the angle is greater than zero, then a linear side surface is added to the top area, with an angle between the geometry and the vertical equal to the lateral angle.

The value of the lateral angle must be more than or equal to zero and less than 90 degrees.

The lateral angle can be defined in the Parameters page.

Adaptive Step (Selection step by scallop height)

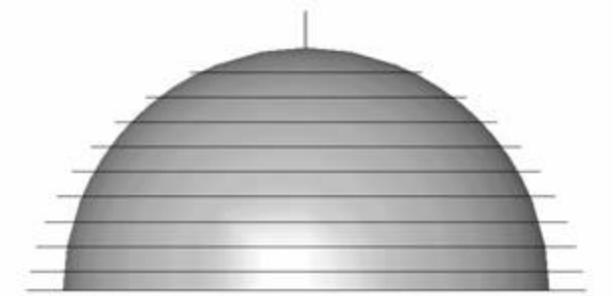
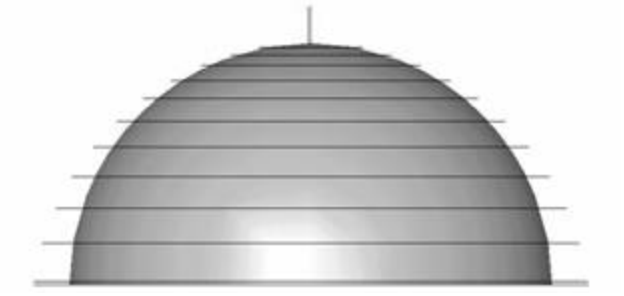
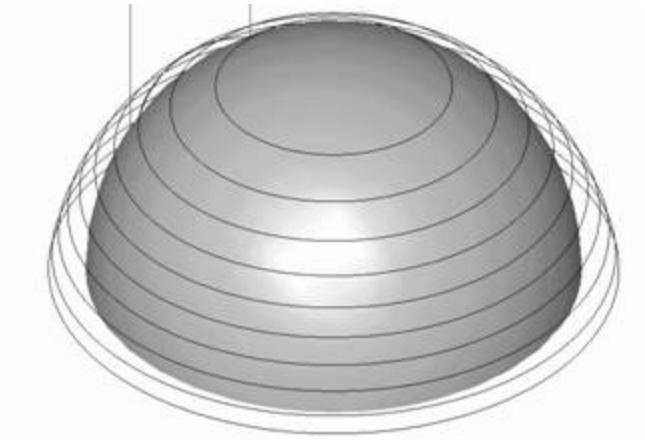
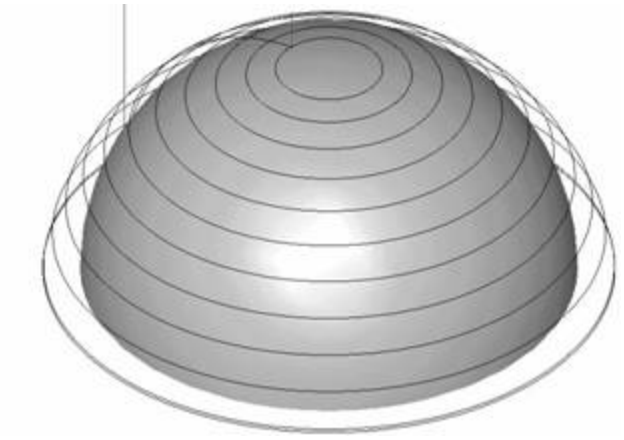
When machining surfaces, whose radius of curvature does not coincide with the profile radius of the tool, then there appears a scallop of residual material between the neighboring passes. Its height depends on the surface shape of the model being machined, the type and size of the tool and the distance between the neighboring passes of the mill.



In operations it is possible to assign the step by the maximum height of the scallop. With this, the distance between the neighboring tool passes will be automatically selected so that the height of the scallop does not exceed the defined.

The step value is calculated for every work pass such that the set height of the scallop is never exceeded, even in the worst case.

It is not hard to see that the step value in the vertical direction will decrease on surfaces that are close to the horizontal, and increase with increasing slopes of the surfaces. On vertical surfaces the step value approaches infinity. In order to avoid excessively large steps, it is necessary to assign the maximum vertical step value. If the calculated value exceeds the maximum, then the maximum allowed step value will be selected.

Machining with permanent step	Step is chosen by scallop height
	
	

The step value in the horizontal direction increases on flat surfaces and decreases on close to vertical. In order to avoid too many passes on areas close to the vertical, it is necessary to assign a minimum step value. If the calculated step is less than the minimum, then the minimum value will be used.

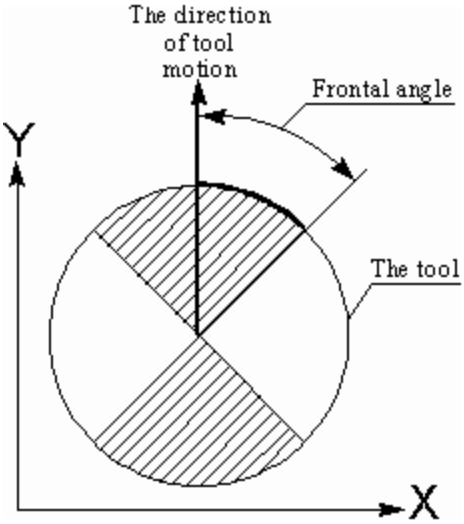
The maximum height of the scallop can be assigned by a positive value only. It is not advised to set the height of the scallop less than the machining tolerance.

The scallop value at step in the vertical direction can be assigned in the Parameters page, and in the horizontal – on the Strategy page.

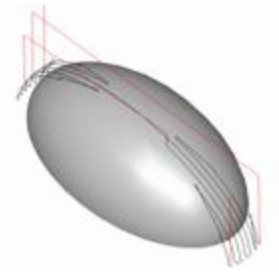
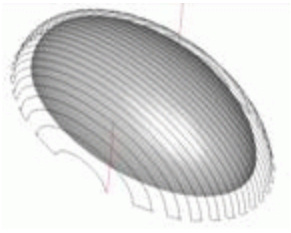
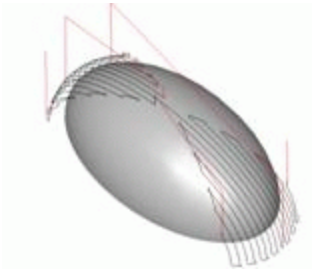
Frontal angle

The height of the scallop between neighboring work passes during the plane machining of inclined surfaces depends mostly on the angle between the normal to the surface and the tool movement direction. In most cases the smaller the angle between projections onto the horizontal plane of the normal and the tool movement direction, the smaller the height of the scallop.

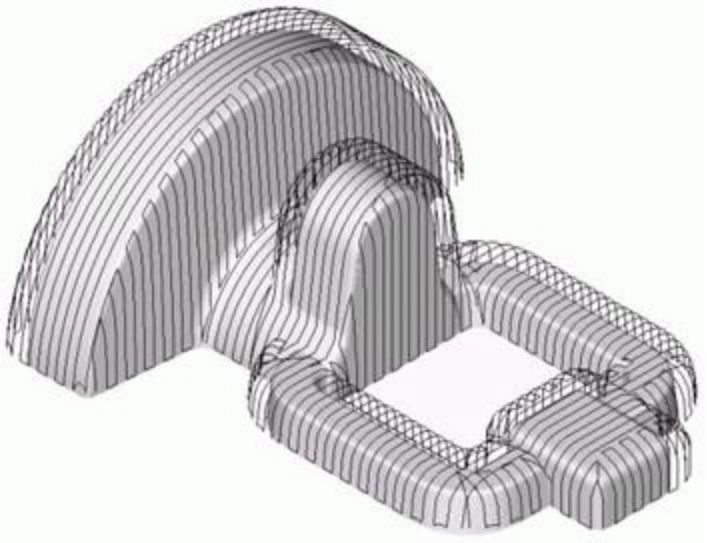
In order to obtain the optimal toolpath in plane finishing and drive operations, it is possible to define a limitation of the frontal angle. The frontal angle is the angle between projections onto the horizontal plane of the tool movement direction and the normal to the surface at the cutting point.



In the picture, there is a spherical mill – viewed from above. An area of the work pass will be included in the resulting toolpath only if the angle between projections onto the horizontal plane of the tool movement direction and the normal vector to the surface at the contact point is less than defined. For the spherical mill the following will also be true: any area of the pass will be included in the resulting toolpath only if the point of tool touching the surface lies inside the hatched sector.

		
Frontal angle = 90°	Frontal angle = 45°	Frontal angle = 15°

The result of machining by two mutually perpendicular plane operations with the frontal angle equal to 45 degrees is shown in the picture (below).



The frontal angle can be within the limits of 0 degrees (machine only perpendicular to the movement direction) to 90 degrees (without limitation). For two plane operations that are mutually perpendicular, the optimal value for the frontal angle will be equal to 45 degrees.

The frontal angle for the plane finishing and drive operations can be defined in the Strategy page.

Machining methods in drive operations

In the drive operations, the view of the tool toolpath in the plane is defined by the drive area, which is formed from the defined drive curves. The Z coordinate is calculated according to the condition by which the tool touches the surface being machined.

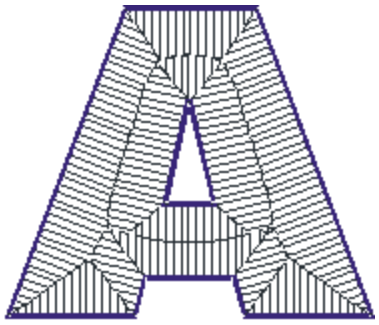
Projections of the tool work passes onto the horizontal plane are always located inside the drive area. The method defining the formation of these projections can be assigned by one of the following methods:

- On curve – performs only one pass along the area borders. That is, performs machining of the detail so that the tool axis is always on one of the curves, which limit the drive area.
- Along curve – the first pass is performed along the area borders, and all following passes – equidistant in the horizontal plane to the previous pass. In other words, when creating work passes the tool axis goes along the curves, which are equidistant to the area borders. The step between the neighboring passes is equal to the defined machining step.
- Across curve – the horizontal projection of every stroke represent a section, which starts on the border of the drive area, and is perpendicular to this point. The length of this section is selected so that the same area is not machined twice, and the point closest to it on the border assigns the directions of the passes for every point inside the area.

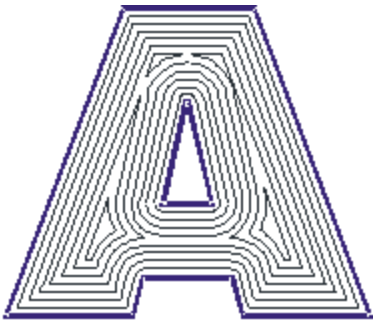
The number equidistant passes when machining along curve, and the length pf passes when machining across curves, in general is limited by the form and the dimensions of the drive area. Work passes are constructed until the entire model, which is inside the drive area, has been machined.

If the user activates the "width" mode, then using the selected method (along or across) only that width along the borders of the drive area will be machined. This means that the number of equidistant passes will be additionally limited by the width of the area being machined.

The view of the trajectories with different combinations of parameters set.



Across Curve



Along Curve



On Curve



Along Curve with
using width

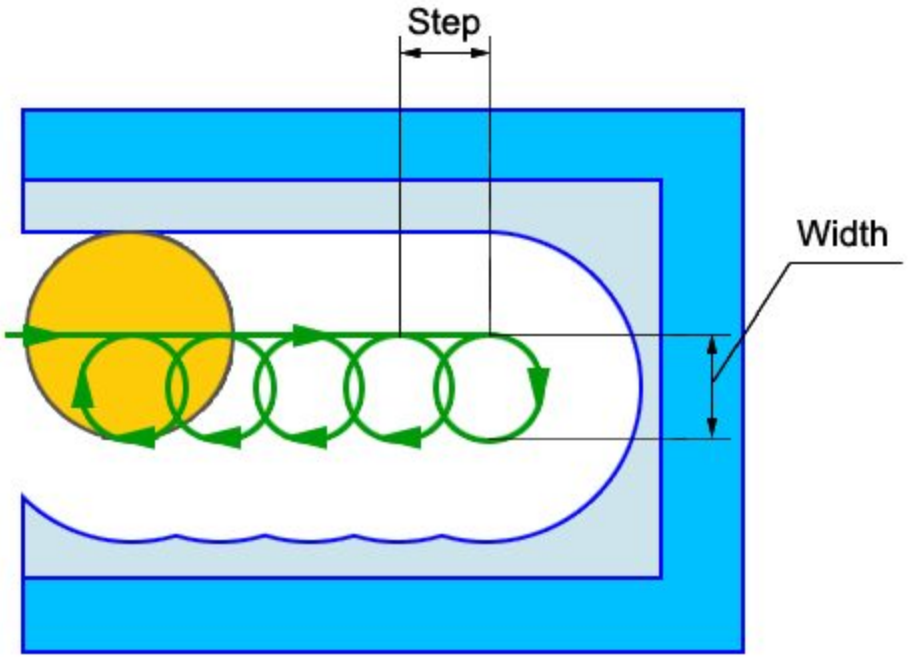


Across Curve with
using width

The machining method for the drive operations can be assigned in the Strategy page.

Trohoidal machining (Equidistant legacy)

High speed cuts is formed by the adding an additional circles to the tool path. This method allows to reduce the NC data much in comparison with the trochoid and at the same time secure the tool.



The width and the step of the high speed cuts path depends on the High speed cuts modes. CAM system gives 4 modes:

1. Do not use. With this mode the additional circles will not formed in any case.

Attention: If the pocket step more than the half of the tool diameter or smooth radius is assigned then unmachined islands can be remained. It is obligatory to make the visual check of the tool path in the Simulation mode.

1. For islands removing only. The mode generates the minimal quantity of the additional circles. The circles diameter is minimal and enough to remove the islands. The cuts step is equal to the tool diameter;
2. With pocket step. The mode guarantees the uniform tool load. The radius of the additional circles can not be less than the smooth radius. The step of the cuts is less or equal to the pocket step. The mode is recommended for high speed machining;
3. Reduce on the cleanup. The high speed cuts tool path is generated similarly to the third mode but the cuts step on the cleanup is equal to the cleanup step to reduce the tool vibration;

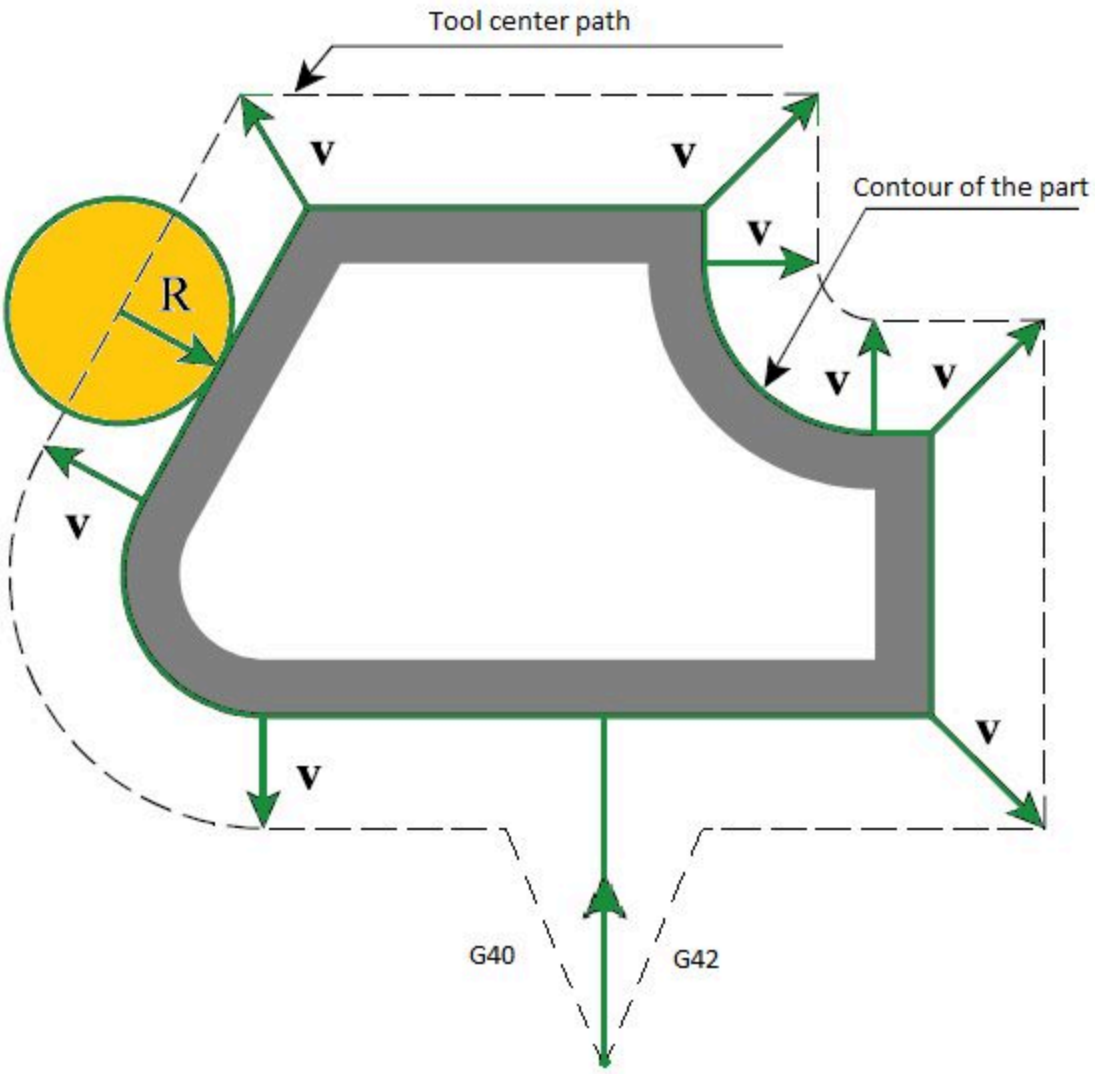
Attention: If machining step great then half of tool diameter or tool have corner radius, then can be rest of material islands. You must visually check this machining at Simulation mode.

Tool magazine

A tool magazine is a device on the machine for storing and quickly changing tools during the machining process of a part. The tool change may be executed automatically at the command in NC-program. There are machines that have several tool magazines (usually for multiple-spindle machines).

Tool compensation in mill operations

The Tool Radius Compensation is a feature of CNC controls that allows programming of a part by specifying the contour of the part, the tool radius and the side of machining instead of the tool center path. The tool center path is calculated by the CNC control itself based on the given data.



In order to correctly calculate the trajectory of the tool center CNC needs to know the direction in which to move the tool for each frame of the trajectory. There are special commands that set the direction of correction in the NC-program. Usually this command G41 - the correction to the left of the programmed path and G42 - the correction to the right on the programmed path. To disable the correction the command G40 is used. The correction value is usually stored in a table of tool correctors on CNC.

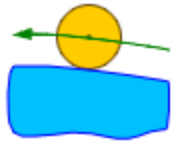
However, the use of radius correction imposes some restrictions on the geometry of the contour, programmed in the NC-code. If the contour has bad elements such as an arc whose radius is less than the value of correction, or frames whose length considerably less than the value of correction, the CNC can not always build the correct trajectory of the tool center. As a result, the trajectory may contain loops, which usually leads to overcuts. You should also pay particular attention to elements of the trajectory,

where the correction is switched on and off. In these areas, moving of the tool may differ significantly from the pre-programmed, because in one part of the correction is turned off and the other - is turned on. To avoid overcuts in these places usually add a special sections - compensation switch cuts.

CAM-system can generate a trajectory with tool radius correction. On the " Lead In/Lead Out " page is a panel where you can enable the use of correction, as well as to select the types of compensation switchlines.

There are 5 different types of the correction:

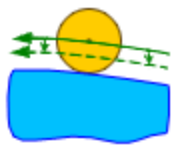
- Computer. In this case, the CAM-system itself calculates the trajectory with the size of the selected tool. In the program is generated a trajectory for the tool center. Correction turn on and of commands does not appear. This does not allow the operator to affect the correction at the machining time. However, the CAM-system when calculating the toolpath can correctly handle bad cases and remove the loops. That pretty much guarantee the accuracy of the trajectory and the absence of gouges. This type is installed by default.



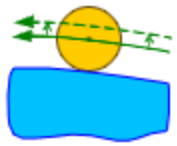
- NC control. In this case, the trajectory calculation is made without taking into account tool radius. In the NC-program displays a contour of the part, as well as the turn on and off compensation commands of the appropriate sign. This allows the operator to adjust the trajectory, taking into account tool radius actually used at the machining time. In the simulation mode to simulate the behavior of CNC uses the correction value is equal to the radius of the tool.



- Wear. The calculation is made given the size of the tool, as with options computer. But in the NC-program will also appear the compensation switch commands. Correction is directed towards the part. This allows the operator to compensate the tool wear, setting the value of correction as the difference between the source tool radius and the actually used tool radius. In the simulation mode for emulating the behavior of CNC the zero correction value is used.



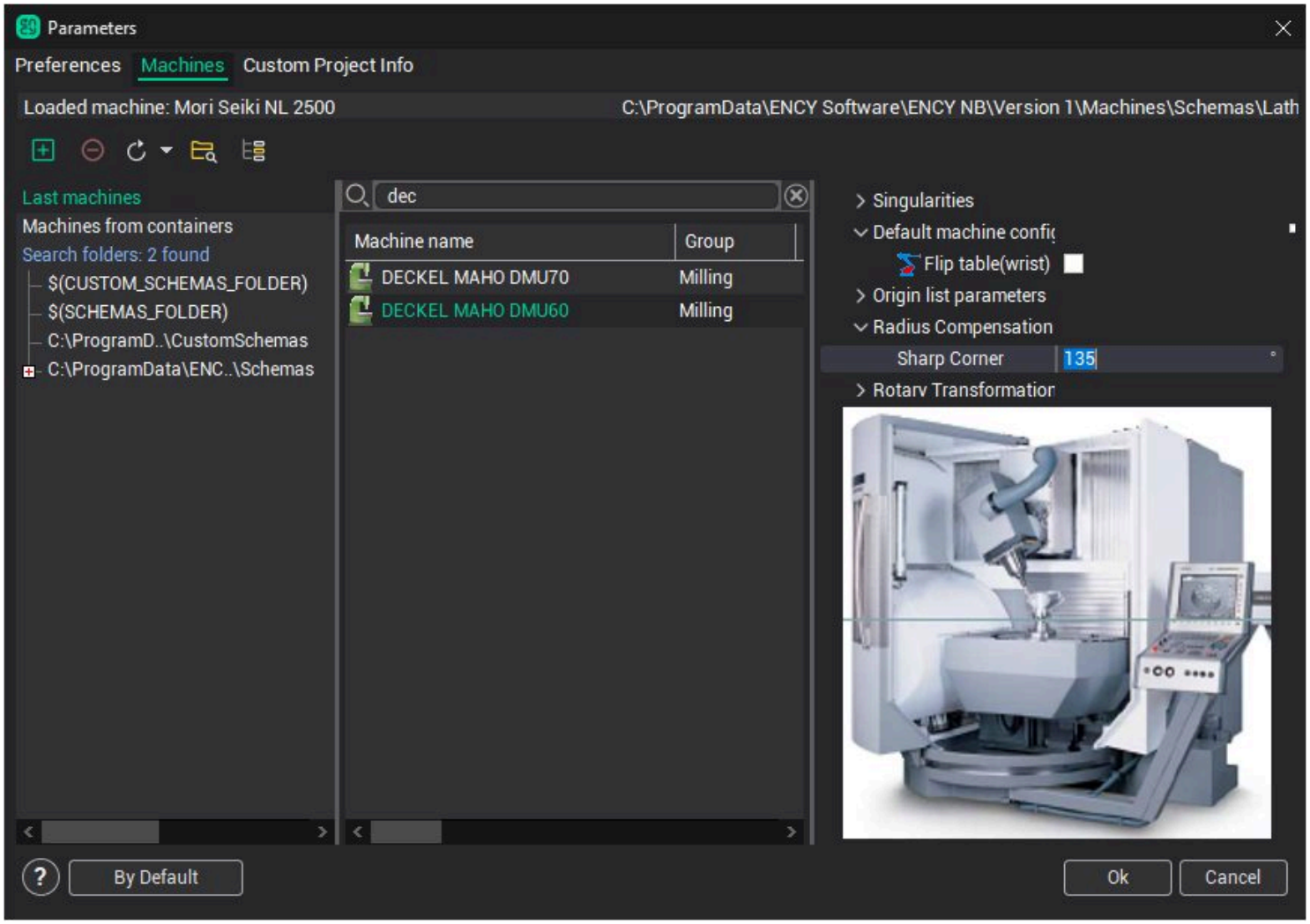
- Reverse wear. Similarly Wear type, but the sign of the correction is opposite (directed from the part).



Radius compensation simulation options

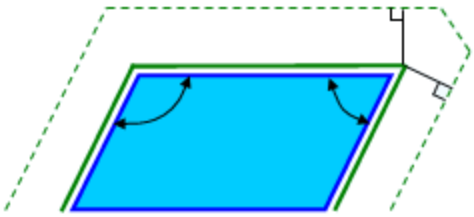
CAM system can calculate, view and simulate tool motion using compensation for the tool radius. When compensation is used, there are commands to turn compensation on and turn off included in the CLData. These are usually `G41`, `G42`, `G40` codes with a compensation number. CAM system will draw the path of the tool motion and can simulate the machining with compensation of the tool radius very similar to how it will happen on the machine.

Different NC machines can use different methods for applying / canceling compensation. CAM system have several options which can be used to 'tune' CAM system's tool radius compensation so that it matches those used by the machine control. These options are available in the `Machine:... Parameters` window on the Machines tab. There is a node called Control parameters –> Radius compensation a property editor, the properties are used for tuning the CAM system simulation of radius compensation.



Use these properties:

- Sharp corner – this value defines the method of rounding a corner. If the angle between the moves is greater than this value then the motion will be extended to intersect. Otherwise, if the angle is less, then each motion will extended by the value of the radius compensation and connected by a linear move. In the drawing below are shown an example where the "left" corner is greater than the sharp corner value, but the one on the "right" is less.



Related reference:

The motions of CRC switch on/off

Mill machining

Application area:

Mill machining provides operations for removing material and finishing part surfaces with a rotating cutting tool. Operations are organized into groups matching the operation selection window.

Operation groups:

- Holes. Provides operations for machining holes and generating machining processes from recognized manufacturing features. Details
- 2D. Contains operations for machining pockets, engraving, and following planar contours. The operation topics describe the geometry and parameters required to generate each toolpath. Details
- Basic. Contains face milling, waterline and plane machining, contouring, and flat-area finishing operations. These topics describe the common milling strategies and their operation settings. Details
- Advanced. Contains operations for surface finishing, undercut machining, multi-axis roughing, and machining spatial contours. The individual topics describe the available toolpath strategies and tool orientation settings. Details
- Rotary. Contains operations that use a rotary axis for machining surfaces and contours. The topics cover rotary roughing and finishing, waterline machining, morphing, surfacing, and contouring. Details
- Turbomachinery Milling. Contains specialized operations for machining impellers and individual blades. The operation topics describe the required blade and hub geometry, machining strategies, and toolpath settings. Details
- Rest machining. Contains corners cleanup, pencil, and chamfering operations. Details

Operation setup:

Common operation settings and topics covering machining with different numbers of axes are described in Operations setup. Details

Types of machining operations

A machining process is represented in the system as an ordered sequence of machining operations. The machining process may contain an arbitrary number of operations of different types. Every operation, depending on its type, has a set of rules for toolpath creation, which is characterized by their individual parameters. The number of available operation types depends on the configuration of CAM system being used.

Operations are basically divided into roughing and finishing. The Roughing operations provide removal of the workpiece material, which lies outside of the model being machined. Normally, the roughing operations are used for preliminary removal of stock material, where the shape and dimensions of the model being machined are quite different to the shape and dimensions of the workpiece. The finishing operations perform machining of the model's surfaces only, without area clearance. The Finishing operations are normally used for the final cutting of the model surface after previous machining, and without it, if the final model does not differ much from the workpiece or if the workpiece is made of soft material.

In the new operation creation window, machining of residual material is a separate group. This has been done for convenience only. Using the normal roughing or finishing operations with appropriate parameters set, it is possible to generate rest-machining toolpaths. The roughing operations in remachining perform stock removal of residual material, and the finishing ones, machine the model surface only in un-machined areas. Rest milling operations allow the user to optimize machining of complex details. They are best used with roughing or finishing rest milling using a tool with a different shape or a smaller diameter than the tool from the previous operations. The spatial transformation operation (offset toolpath) is also included into the rest milling operations group.

Depending on the type of machining, operations can be divided into milling and turning.

- The pocketing, engraving, 2D contouring and 3D contouringoperation use curves for machining at job assignment. And surfaces from model are not used for it.
- Other operation use solids, faces, meshes for machining. Curves may be used as auxiliary element at job assignment, for example, as cutting direction or machining boundary.

Holes

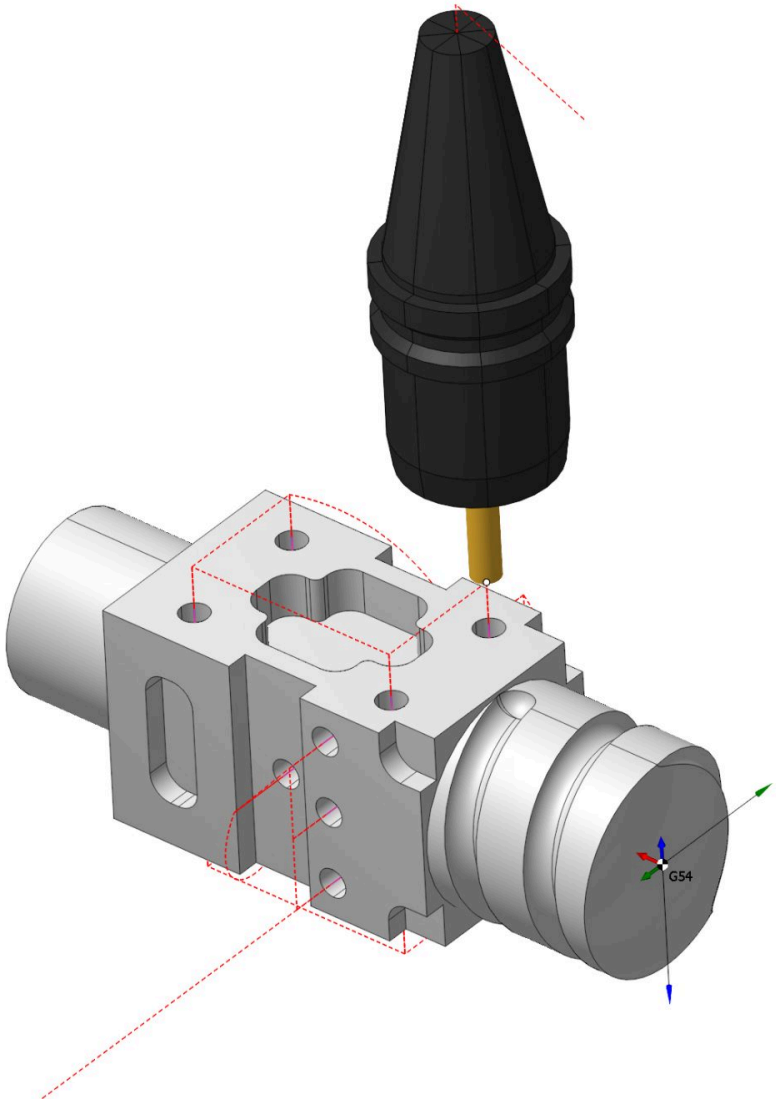
Application area:

The Holes group provides operations for machining holes and generating machining processes from recognized manufacturing features.

Operations:

- Hole machining. Creates hole machining toolpaths, including drilling, boring, and threading cycles. Details
- FBM Mill. Generates operations for recognized manufacturing features using machining procedures. Details

Hole machining



Application area:

The hole machining operations are designed for drilling, centering, boring, countersinking, tapping, thread milling and hole pocketing. It can machine holes that are not lying in the same plane and that are not lying in orthogonal planes. The operation can be used both for the machining of holes in a model, or for pre-drilling of the tool plunge points for the pocketing and operations.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Center. Create hole by center point.

Create. Create hole by coordinates input. Details

Recognize. Automatically recognize holes in the part. Details

Pattern. Create holes array by pattern. Details

Properties. Displays the properties of an element. You can also call this menu by double clicking on an item in the list. Details

Sorting. Sorting using some parameters to sort holes list. Details

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

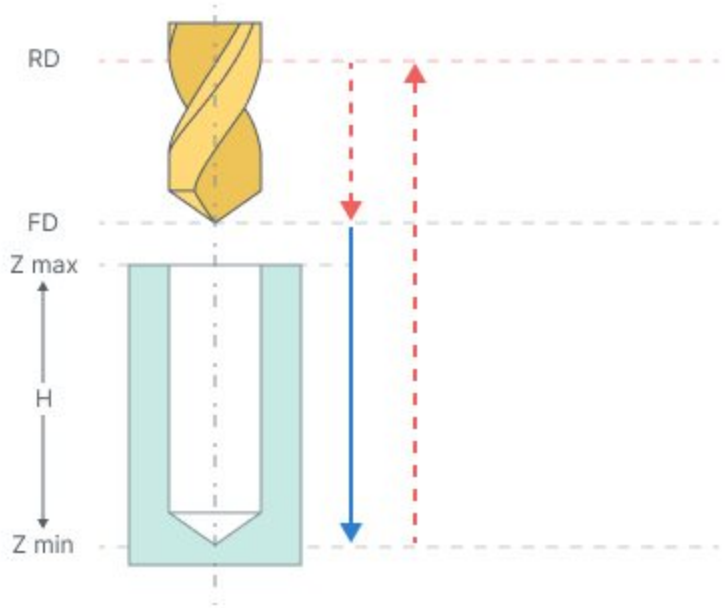
Drilling Type.

This group defines the basic parameters of various types of hole processing.

▼ Click here to expand..
Drilling Type:

Simple drilling. Drilling cycle drills holes with rapid approach to the safe level and rapid retract the safe plane level.

▼ Click here to expand..



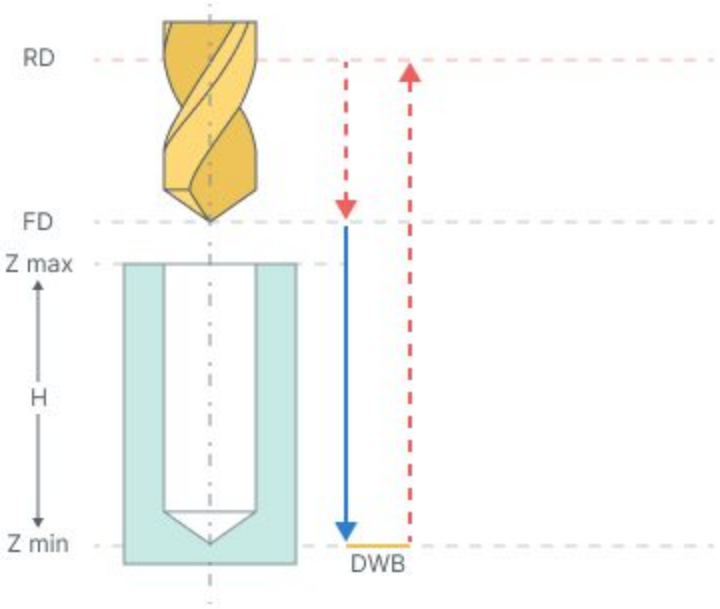
The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.

- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Tool return to the Rapid distance(RD) level at the rapid feed.

Drilling with dwell. Drilling cycle drills holes with rapid approach to the safe level, dwell at hole bottom level and rapid retract the safe plane level

▼ Click here to expand..

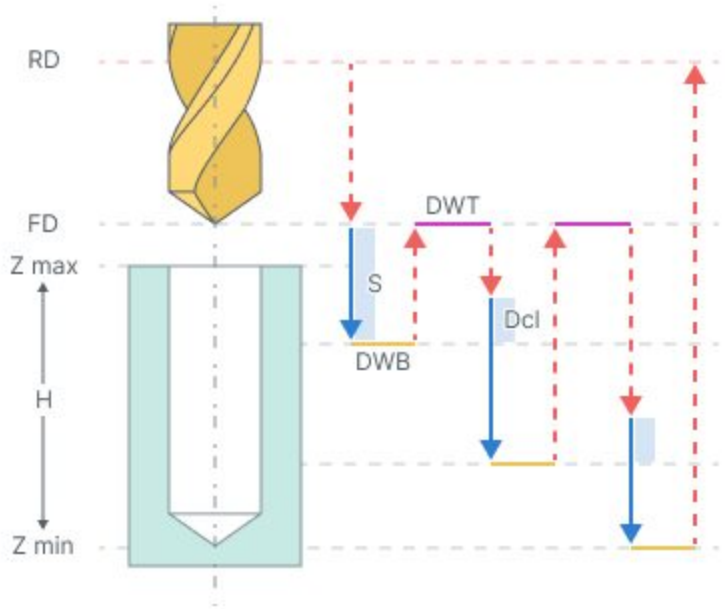


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Bottom Dwell (DWB) at the Z min level.
- Tool return to the Rapid distance(RD) level at the rapid feed.

Chip removing. Drilling with chip removing cycle performs tool motion to the hole center at the Rapid distance(RD) level and consequent cyclic drill with tool retraction to the Feed distance(FD) level.

▼ Click here to expand..

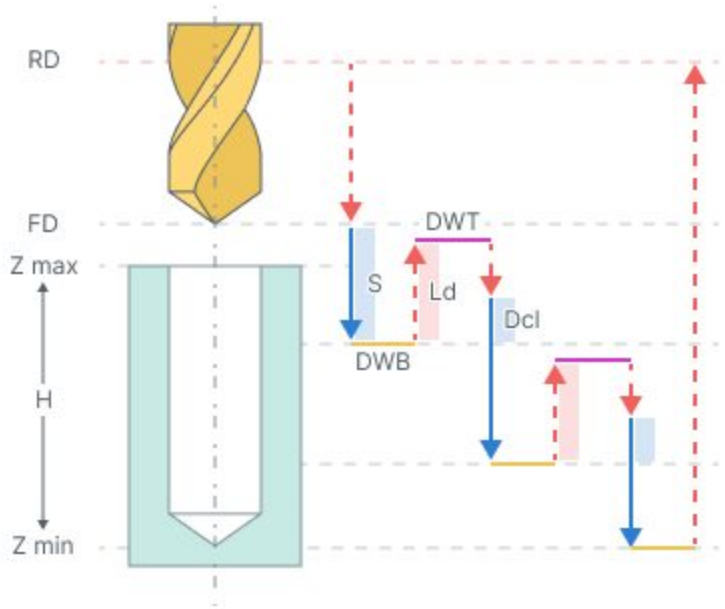


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Step (S).
- Bottom Dwell (DWB) at this level.
- Rapid return to Feed distance(FD) level.
- Dwell for the Top Dwell (DWT) time.
- Rapid motion to the previous depth level, with a Deceleration (Dcl).
- Work feedrate to the Deceleration (Dcl) with Step (S).
- Dwell for the Top Dwell (DWT) time.
- Repeat previous five iterations until the full hole depth is reached.
- Tool return to the Rapid distance(RD) level at the rapid feed.

Chip breaking. Drilling with chip breaking cycle performs tool approach to the hole center at the Rapid distance(RD). When cyclic drilling is performed with tool retraction for chip breaking.

▼ Click here to expand..

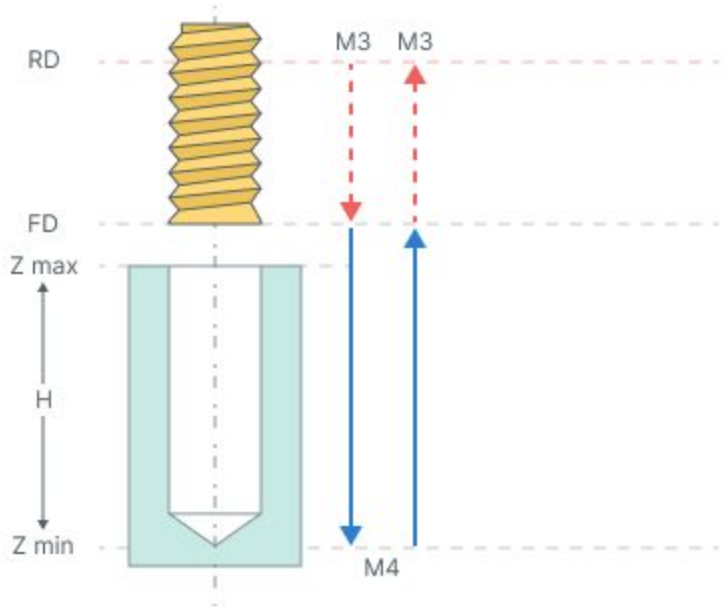


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Step (S).
- Bottom Dwell (DWB) at this level.
- Rapid tool retraction for the Lead-out (Ld).
- Dwell for the Top Dwell (DWT) at the bottom time.
- Rapid motion to the previous depth level, with a Deceleration (Dcl).
- Work feedrate to the Deceleration (Dcl) with Step (S).
- Bottom Dwell (DWB) at this level.
- Repeat previous five iterations until the full hole depth is reached.
- Tool return to the Rapid distance(RD) level at the rapid feed.

Tapping. Tapping cycle performs rapid approach to the Rapid distance(RD) level, thread tapping with subsequent retraction at work feedrate with reverse spindle rotation.

▼ Click here to expand..

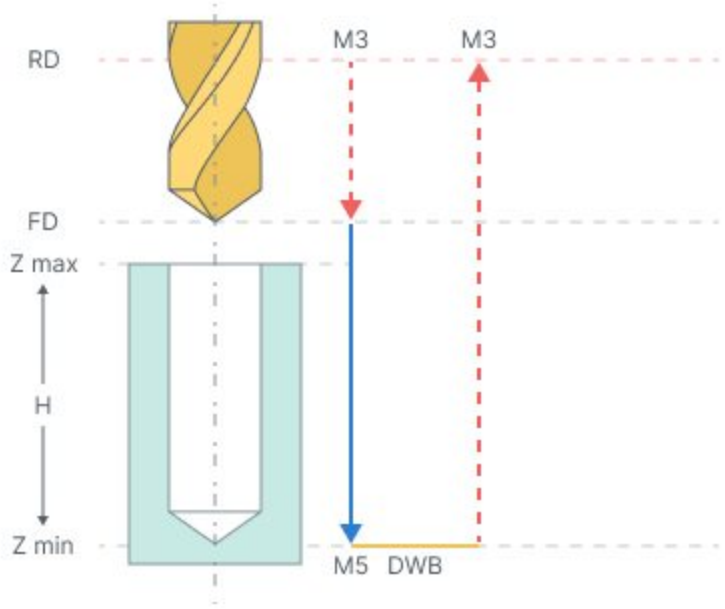


Drilling cycle G84 consist of the following steps:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level with M3(M4) and then Spindle reverse M4(M3).
- Tool return to the Rapid distance(RD) level at the rapid feed.
- Restore spindle rotation direction and speed.

Bore 5. Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and work feedrate retract to the Rapid distance(RD) level.

▼ Click here to expand..

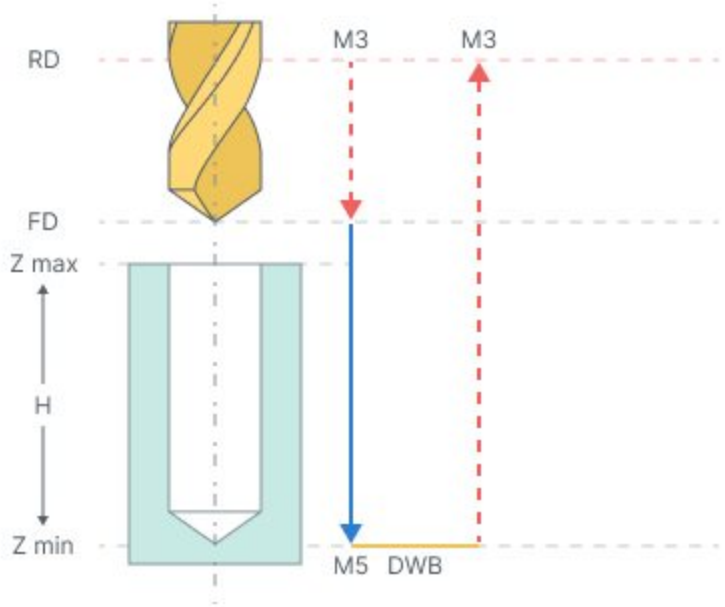


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Spindle stop (M5).
- Bottom Dwell (DWB) at this level.
- Tool return to the Rapid distance(RD) level at the rapid feed.
- Restore the spindle rotation direction and speed.

Bore 6. Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and rapid retract to the Rapid distance(RD) level.

▼ Click here to expand..

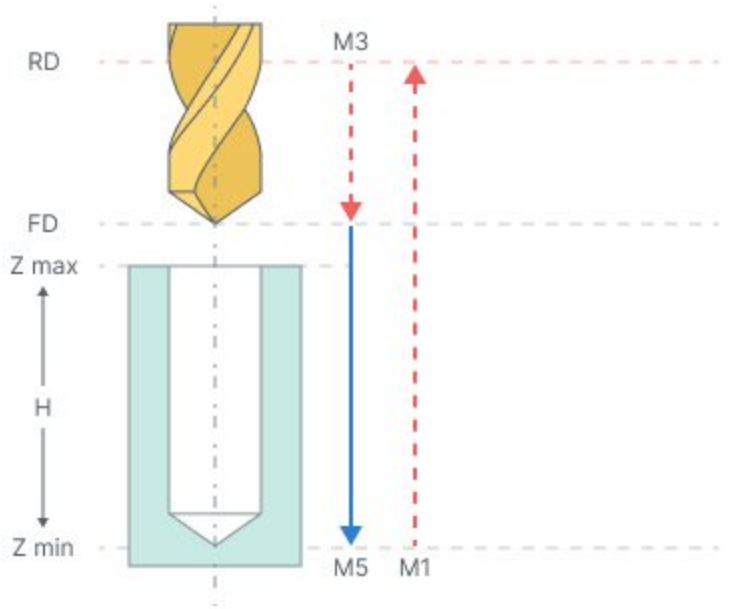


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Spindle stop (M5). If Oriented tool retract check box enabled, then spindle stops with the fixed angle of orientation and then the tool shifts slightly sideways in accordance with a given displacements.
- Bottom Dwell (DWB) at this level.
- Tool return to the Rapid distance(RD) level at the rapid feed.
- Restore the spindle rotation direction and speed.

Bore 7. Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and manual retract to the Rapid distance(RD) level.

▼ Click here to expand.

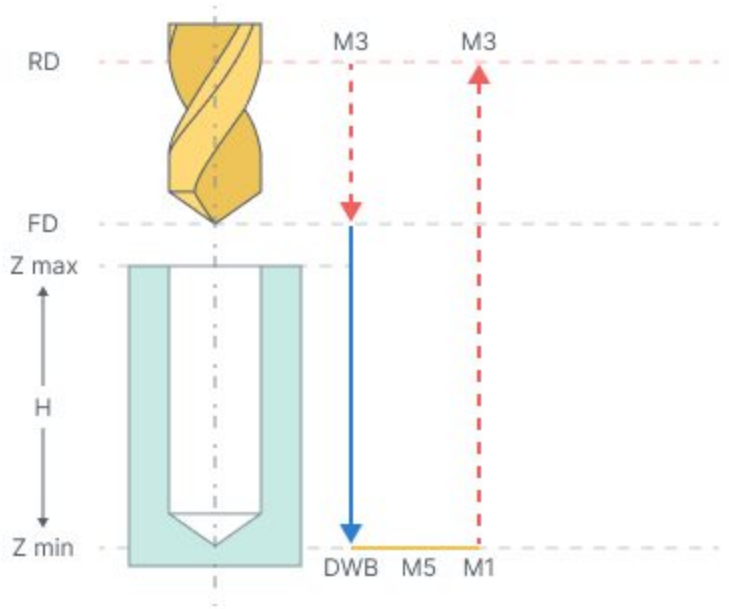


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Spindle stop (M5). If Oriented tool retract check box enabled, then spindle stops with the fixed angle of orientation and then the tool shifts slightly sideways in accordance with a given displacements.
- At the bottom of the pit, command M1 is called - wait for the operator. The operator can remove the drill manually.
- Tool return to the Rapid distance(RD) level at the rapid feed.
- Restore the spindle rotation direction and speed.

Bore 8. Drilling cycle type drills holes with rapid approach to the safe level, dwell at hole bottom level, spindle stop and manual retract to the safe plane level.

▼ Click here to expand..

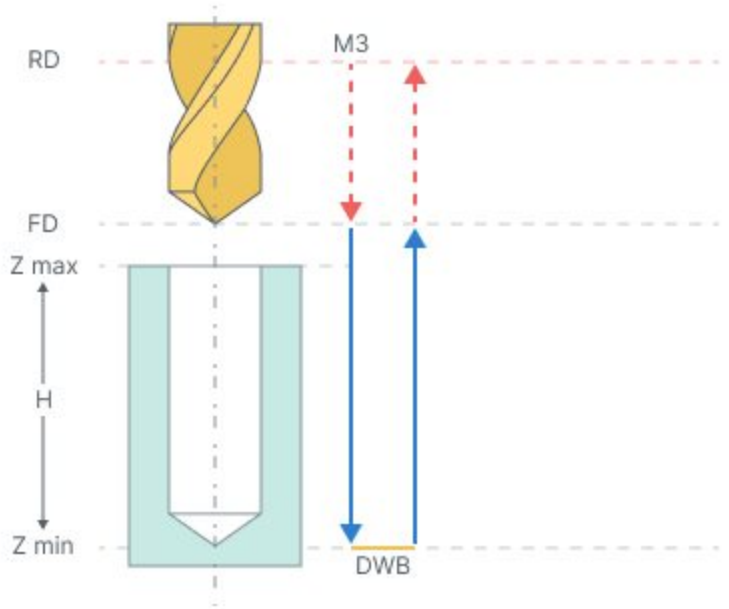


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Bottom Dwell (DWB) at the Z min level.
- Spindle stop (M5).
- At the bottom of the pit, command M1 is called - wait for the operator. The operator can remove the drill manually.
- Tool return to the Rapid distance(RD) level at the rapid feed.
- Restore the spindle rotation direction and speed.

Bore 9. Boring cycle type bores holes with rapid approach to the safe level, dwell at hole bottom level, spindle stop and manual retract to the safe plane level.

▼ Click here to expand.



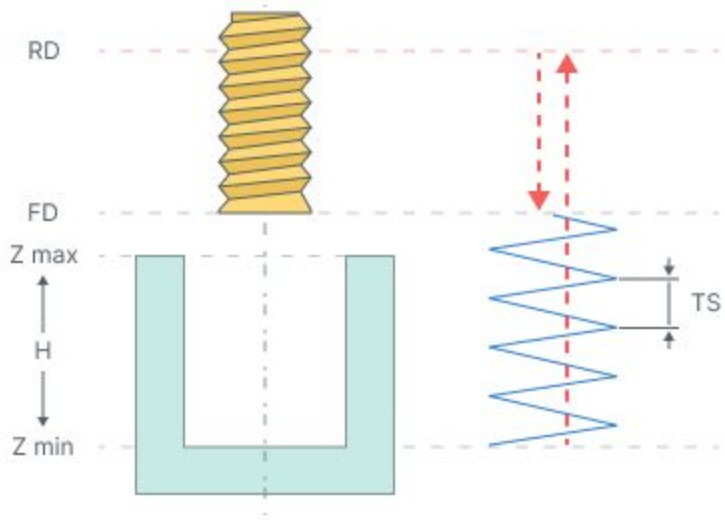
The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Bottom Dwell (DWB) at the Z min level.
- Tool return to the Feed distance(FD) level at the work feed.
- Tool return to the Rapid distance(RD) level at the rapid feed.

By spiral (thread milling). Thread milling cycle is used to machine external or internal threading or to machine hole by a helix. Spiral machining is used then hole diameter is larger than the tool diameter. The tool rotates around the hole axis and simultaneously travels along the axis. spiral diameter is chosen according to the hole and the tool dimensions. Machining can be done in several passes to mill holes of desired diameter.

This drilling type is kept for operations created in earlier versions and is not offered in new operations. New operations use the Helix spiral and Thread milling types.

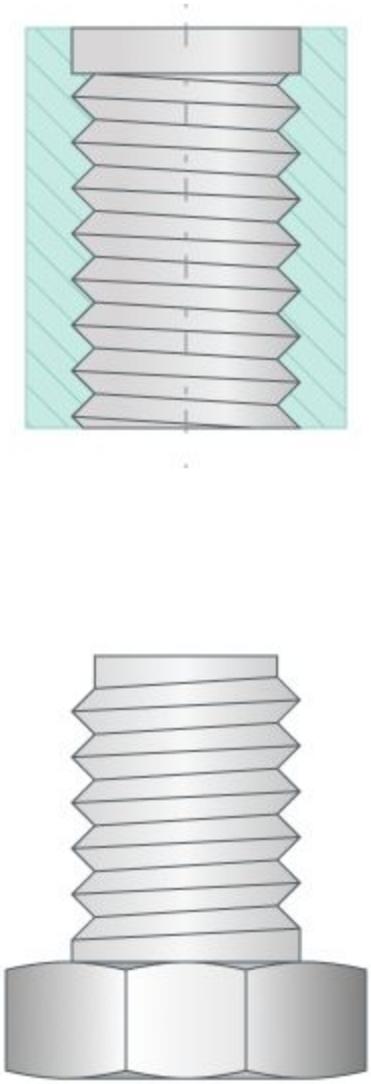
▼ Click here to expand..



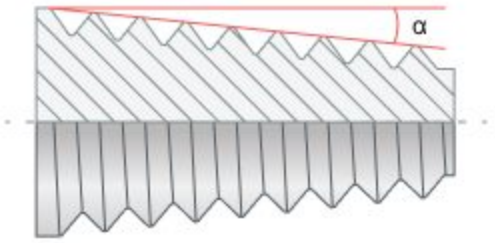
The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate travel to the spiral start with Thread step (TS).
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Return to the hole center.
- Tool return to the Feed distance(FD) level at the work feed.
- Rapid travel to the Rapid distance(RD) level.

Thread type. Parameter specifies whether the threading is External(OD) or Internal(ID).

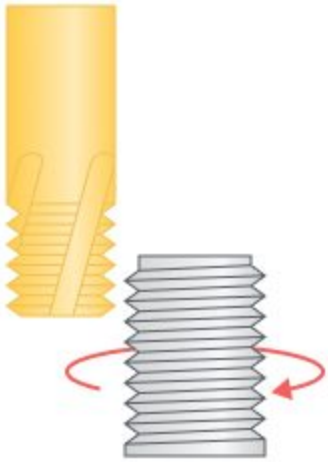


Taper angle. Necessary for cutting tapered threads. Through the parameter you can set the Taper angle.

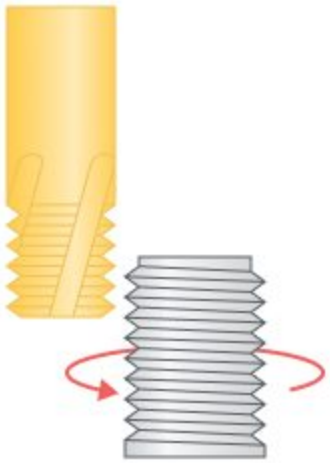


Thread direction (Spiral direction). Thread kind, right or left, is determined by the Thread direction. It can also determine the direction of milling Forward or Reverse.

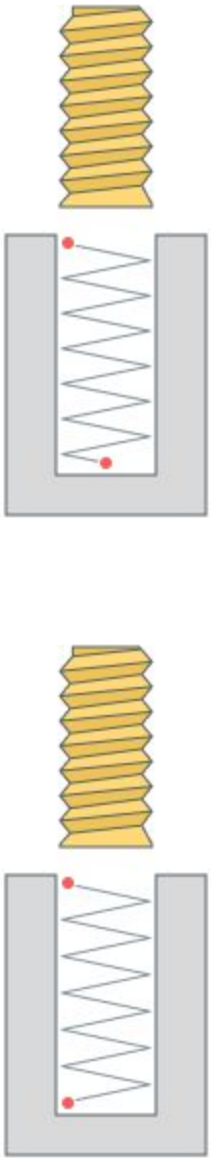
CW (Right). The Spiral is twisted right. The tool is rotating clockwise if watched from above. Use for thread milling.



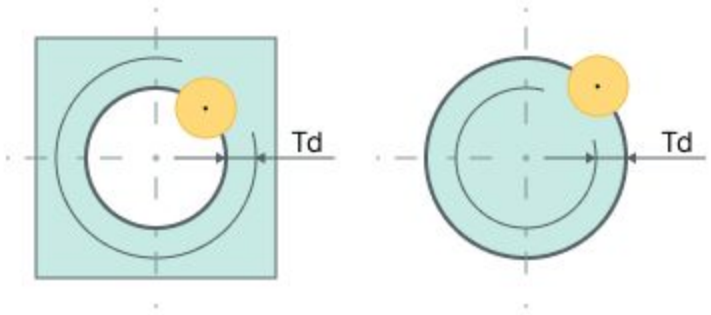
CWW(Left). The spiral is twisted left. The tool is rotating counter clockwise if watched from above. Use for thread milling.



Round thread count to integer (Round to integer) Try rounding to a smaller or larger number of revolutions.



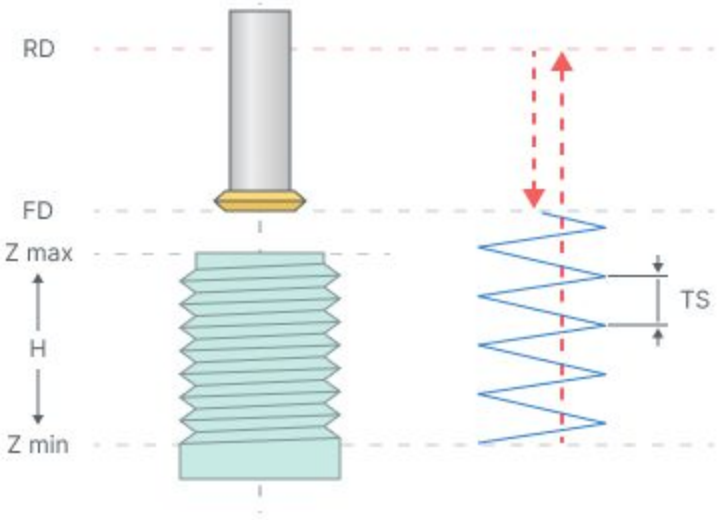
Thread depth (Td). Defines the distance between the inner and outer diameters of the thread. In all cases (inner and outer threads) the diameter that is defined in the job assignment defines the outer (maximal) diameter of the thread. The inner diameter of the thread is calculated as the difference between the outer diameter and the Thread depth. The Thread milling type has no such parameter: the thread depth is defined by the Minor diameter and Major diameter of the hole thread. Details



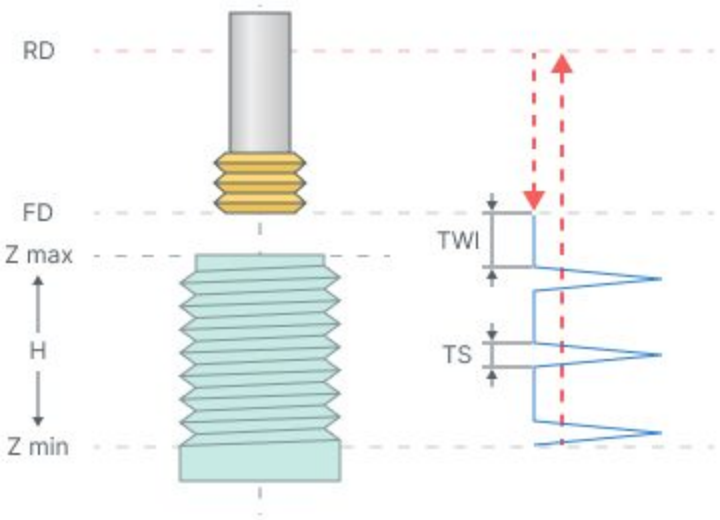
Start count. Number of thread starts.

Thread Mill Path. Parameter is used defines the toolpath type according to the used tool type. It can be one of the following: Continuous or Axial Divide.

Continuous. This toolpath type is used for the single-cutter tool, which forms only one thread turn with each turn of the spiral. Geometrically the trajectory is a continuous spiral with Thread step (TS).



Axial Divide. This type is used for multi-cutter tool which forms multiple thread turns for one turn of the trajectory spiral. The trajectory consists of subsequent spiral turns connected with rapid cutting-edge long transitions along the spiral axis with Thread step (TS) and Tool working length (TWL).

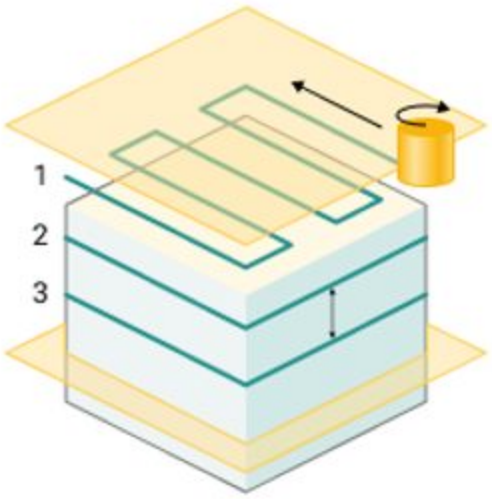


Turn count. Number of threads.

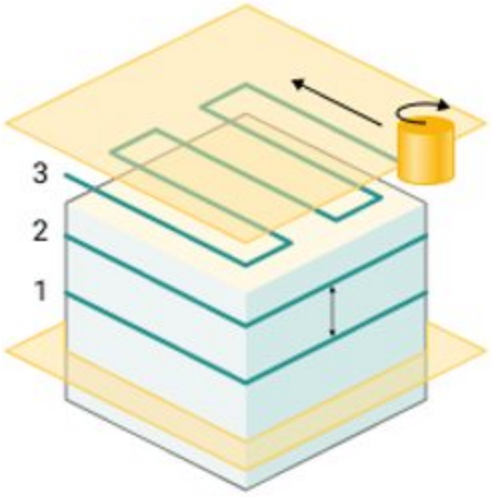
Tool working length (TWL). It specifies the length of transitions between adjacent spiral turns. This value must be calculated as the thread pitch multiplied on the coils number that can be created by the tool per one spiral turn (See Axial Divide hint).

Machining sequence (Direction) Processing order.

Top-down. Processing is carried out from top to bottom.



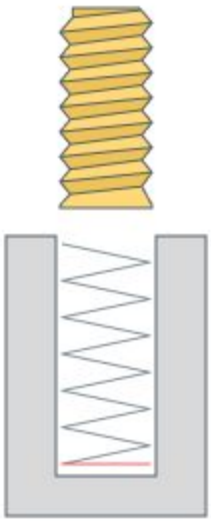
Bottom-up. Processing is carried out from bottom to top.



Rough passes. Toolpath is offset in plane by the 'Roughing passes' thickness amount.

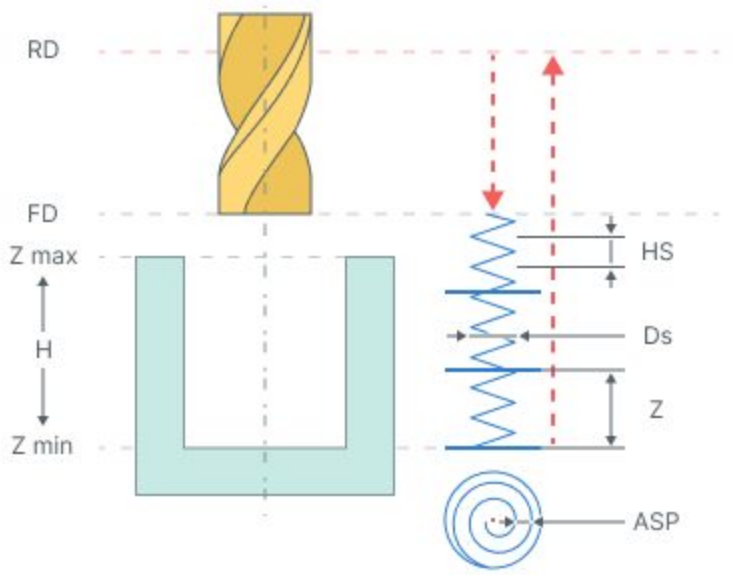
Finish pass. The pass becomes finishing. An additional roughing pass is added to it at a distance specified in the parameter.

Last circle pass. Adds a circular passage at the bottom of the hole.



Combined spiral. The cycle is used to machine holes which diameter is greater than the tool diameter. The pocketing is performed by layers. The tool cuts in along a spiral to each layer and then expands the hole to the desired diameter by moving along Archimedes spiral with finishing pass along the circle. The Archimedean spiral is approximated by the circle arcs.

▼ Click here to expand..



The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate spiral cut-in to the Axial stepover (Z). Helic diameter (Ds) is specified in the percents of the tool diameter. The Depth per revolution (HS) is defined by the Ramp angle a, mm (Hi) or tool pitch count.
- Archimedes spiral with Radial stepover (ASP) motion at that level until the tool axis has reaches the circle with diameter equal to hole diameter reduced by the tool diameter.
- Finishing pass along specified above circle without level change.
- Repeat previous three steps until desired hole depth is machined with travel to the next cut-in point without level change.
- Return to the hole center.
- Tool return to the Feed distance(FD) level at the rapid feed.
- Rapid travel to the Rapid distance(RD) level.

Helix spiral. The cycle mills a hole or a boss by a helix when the hole diameter is larger than the tool diameter. The tool moves along a spiral around the hole axis and simultaneously travels along the axis. The material can be removed by several rough passes and a finish pass.

▼ Click here to expand..
Machining side. Specifies whether the inner surface of a hole (ID) or the outer surface of a boss (OD) is machined.

Taper. Machines a tapered hole or boss. The parameter sets the taper angle.

Milling direction. Direction of the tool movement along the spiral. CW and CCW set it as seen from above, for both machining sequences. Climb and Conventional set the milling mode: the direction is calculated from the spindle rotation and the machining side.

Axial stepover. Axial distance the tool moves per spiral turn. It is set by the ramp angle of the toolpath or as a distance.

Round turn count to integer. Rounds the number of spiral turns to an integer. The axial stepover is then not kept exactly.

Initial angular position. Angular position of the spiral start around the hole axis, in degrees.

Machining sequence. Processing order: Top-down or Bottom-up. The resulting milling mode, Climb or Conventional, is shown in parentheses.

Bottom circle pass. Adds a circular pass at the bottom of the hole. With the Bottom-up sequence the circle is the first motion.

Compensation mode. Similar to the Output group in 2D contouring. Details

Rough passes. Thickness of the material layer that the rough passes remove before the final size. For internal machining it is measured from the final hole wall toward the axis, for external machining from the final boss surface outward. It is set as a distance or in % of the tool diameter; Off means no rough passes. The Step sets the radial distance between the rough passes as a distance or in % of the tool diameter; x (Count) sets the number of rough passes instead.

Finish pass. Adds a finish pass that removes the specified stock after the rough passes. The stock is added on top of the rough passes volume. It is set as a radial distance or in % of the tool diameter.

The following parameters are located on the Links/Leads tab.

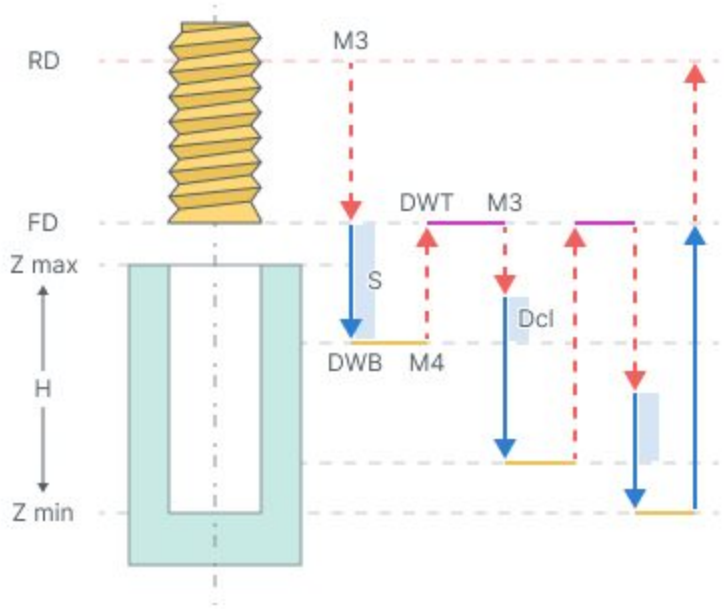
Radial safe distance. Used only for external machining (OD). Gap between the tool and the boss surface at the point where the tool starts the lead and ends the retract. It is set as a distance or in % of the tool diameter.

Engage/Retract. Defines how the tool enters the spiral and leaves it:

- Direct – the tool moves to the spiral along a straight line and leaves it the same way: from the hole axis for internal machining, from the radial safe distance for external machining;
- By Arc – the tool enters the spiral along an arc tangent to it and leaves it the same way. The Radius of the arc is set as a distance or in % of the hole radius; for external machining the percentage is taken from the boss radius. The Angle of the arc is set in degrees: while moving along the arc, the tool also moves along the axis following the spiral.

Tapping with chip removing.

▼ Click here to expand..

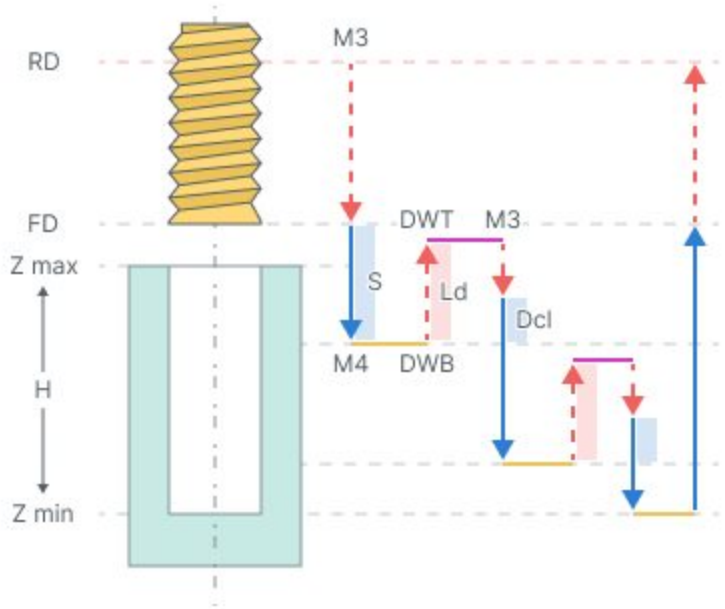


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Step (S).
- Restore spindle rotation direction and speed (M4)
- Bottom Dwell (DWB) at this level.
- Rapid return to Feed distance(FD) level.
- Dwell for the Top Dwell (DWT) time.
- Restore spindle rotation direction and speed (M3)
- Rapid motion to the previous depth level, with a Deceleration (Dcl).
- Work feedrate to the Deceleration (Dcl) with Step (S).
- Restore spindle rotation direction and speed (M4).
- Bottom Dwell (DWB) at this level.
- Repeat previous seven iterations until the full hole depth is reached.
- Tool return to the Feed distance(FD) level at the work feed.
- Tool return to the Rapid distance(RD) level at the rapid feed.

Tapping with chip breaking.

▼ Click here to expand..

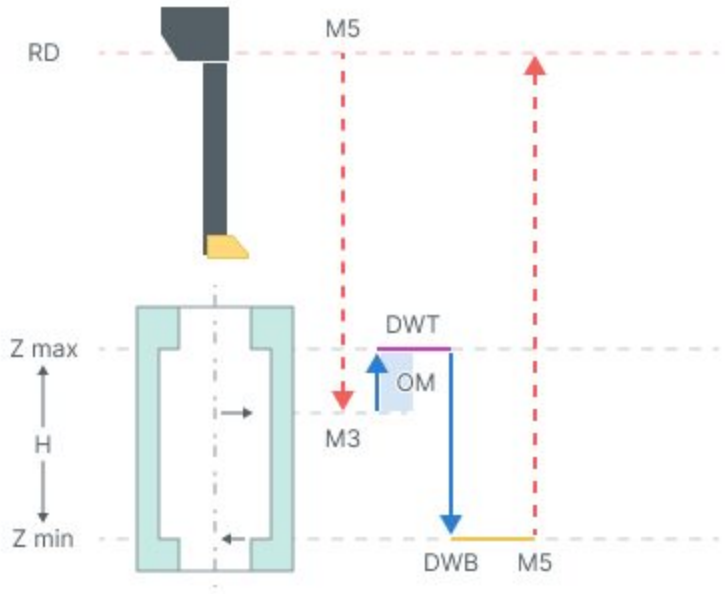


The cycle consists of:

- Rapid tool motion to the hole center at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid descend to the Feed distance(FD) level, that is located on the tab Links.
- Work feedrate motion to the Step (S).
- Restore spindle rotation direction and speed (M4)
- Bottom Dwell (DWB) at this level.
- Rapid tool retraction for the Lead-out (Ld).
- Dwell for the Top Dwell (DWT) time.
- Restore spindle rotation direction and speed (M3)
- Rapid motion to the previous depth level, with a Deceleration (Dcl).
- Work feedrate to the Deceleration (Dcl) with Step (S).
- Restore spindle rotation direction and speed (M4).
- Bottom Dwell (DWB) at this level.
- Repeat previous seven iterations until the full hole depth is reached.
- Tool return to the Feed distance(FD) level at the work feed.
- Tool return to the Rapid distance(RD) level at the rapid feed.

Groove boring. For this operation recommended to use the Oriented Tool Retract parameter.

▼ Click here to expand..



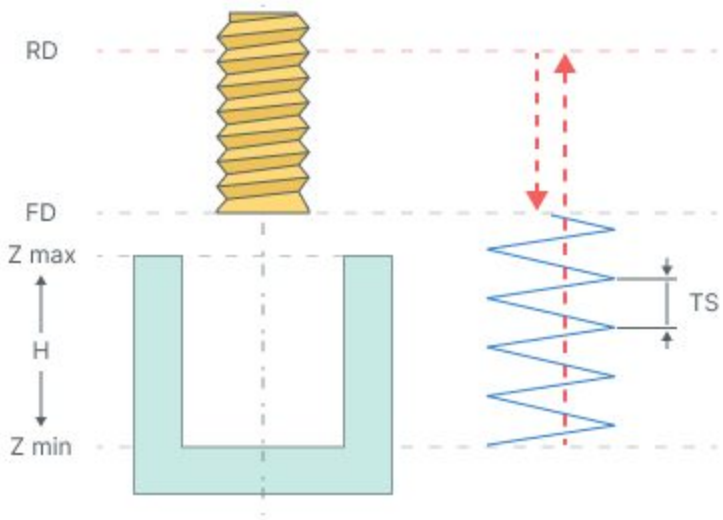
M3 - Spindle rotation (M3)

The cycle consists of:

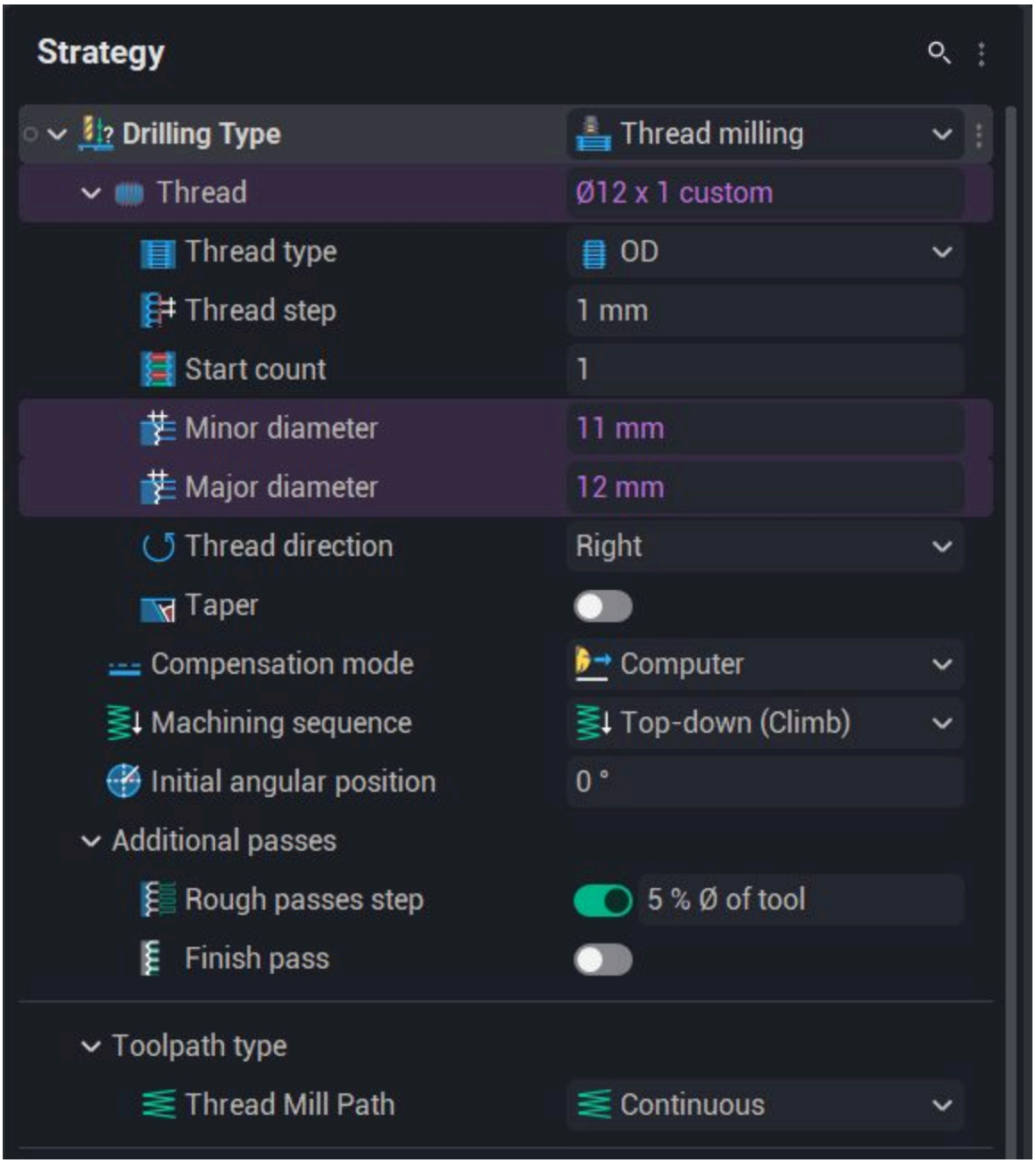
- Rapid tool motion at the Rapid distance(RD) level, that is located on the tab Links.
- Rapid tool motion to the Zmax - Opposite move length(OM).
- On spendle rotation (M3)
- Work feedrate motion to the Zmax + Opposite move length(OM).
- Dwell for the Top Dwell (DWT) time.
- Work feedrate motion to the Z min level, that is located on the tab Job assignment in the Properties hole. In absolute values. If you want to use relative values use the parameter H = Zmax - Zmin.
- Bottom Dwell (DWB) at this level.
- Spindle stop (M5).
- Tool return to the Rapid distance(RD) level at the rapid feed.

Thread milling. The cycle mills an internal or external thread with a thread mill. The tool moves along the thread spiral around the hole axis and simultaneously travels along the axis. The thread data (type, step, start count, diameters, direction and taper) is defined for each hole in the job assignment. Details The machining strategy is common for all holes of the operation.

▼ Click here to expand..



Thread. Thread of the job assignment holes: its designation and the source of the thread data. The group contains the same parameters as the Thread group of the Hole editing window. Details The group shows the holes selected in the job assignment list, or all holes when none is selected. If these holes have different values of a parameter, the parameter is highlighted in purple. A value entered in the group is applied to these holes.



Compensation mode. Similar to the Output group in 2D contouring. Details

Machining sequence. Processing order: Top-down or Bottom-up. The resulting milling mode, Climb or Conventional, is shown in parentheses. It depends on the thread direction, the thread type and the machining sequence.

Initial angular position. Used for multistart threads. Defines the angular position of the first thread start around the hole axis, in degrees.

Additional passes. The group defines the passes that divide the thread depth:

- Rough passes step – divides the thread depth into several radial passes. The step sets the distance between the passes as a distance or in % of the tool diameter; x (Count) sets their number. When the option is off, the depth is cut in one pass, apart from the finish pass;

- Finish pass – the last pass becomes the finish pass. An additional rough pass is added before it at the distance specified in the parameter.

Toolpath type. The Thread Mill Path group defines the toolpath according to the tool used:

- Path type – Continuous is used for a single-cutter tool, which forms one thread turn with each turn of the spiral; the toolpath is a continuous spiral. Axial Divide is used for a multi-cutter tool, which forms several thread turns per spiral turn; the toolpath consists of spiral turns connected with transitions along the axis;
- Turn count – used with Axial Divide. Number of spiral turns the tool makes before it moves along the axis to the next section. Usually 1. Set 2 or more for a fine thread, when the tool pitch is bigger than the thread step;
- Tool working length – used with Axial Divide. Length of the tool cutting part: after each section the tool moves along the axis by this length. It is set as a distance, in % of the thread step or Auto from tool: for a thread mill it is the tool pitch multiplied by the number of tooth rows, for other tools the flute length. The length is rounded to a whole number of thread steps.

The following parameters are located on the Links/Leads tab.

Radial safe distance. Used only for external threads (OD). Gap between the tool and the major diameter of the thread at the point where the tool starts the lead and ends the retract. It is set as a distance or in % of the thread depth.

Engage/Retract. Defines how the tool enters the thread spiral and leaves it:

- Direct – the tool moves to the thread spiral along a straight line and leaves it the same way: from the hole axis for an internal thread, from the radial safe distance for an external thread;
- By Arc – the tool enters the thread spiral along an arc tangent to it and leaves it the same way. The Radius of the arc is set as a distance or in % of the hole radius: the major radius for an internal thread, the minor radius for an external thread. The Angle of the arc is set in degrees: while moving along the arc, the tool also moves along the axis following the thread spiral.

Below are the parameters that may differ from the verification type

The parameters of the By spiral (thread milling), Helix spiral and Thread milling types are described in their sections above.

Top Dwell (DWT). Defines the delay at the top level of the hole .(See drilling type hint).

Bottom Dwell (DWB). Defines the delay at the bottom level of the hole. (See drilling type hint).

Step (S). In deep hole drilling refers to the distance or increment between successive passes of the drill bit when drilling deep holes (See drilling type hint).

Deceleration (Dcl). This value is required for the idle movement to the previous depth level. The tool doesn't reach the prior cut at a distance entered in this value. Once the tool reaches the specified value, the movement will occur during the working stroke until the next level (See drilling type hint).

Lead-out (Ld). Retraction value from previous cutting step (See drilling type hint).

Depth per revolution (HS). Defines the plunge helix step (See drilling type hint).

Helic diameter (Ds). Determines the helix diameter of the plunge (See drilling type hint).

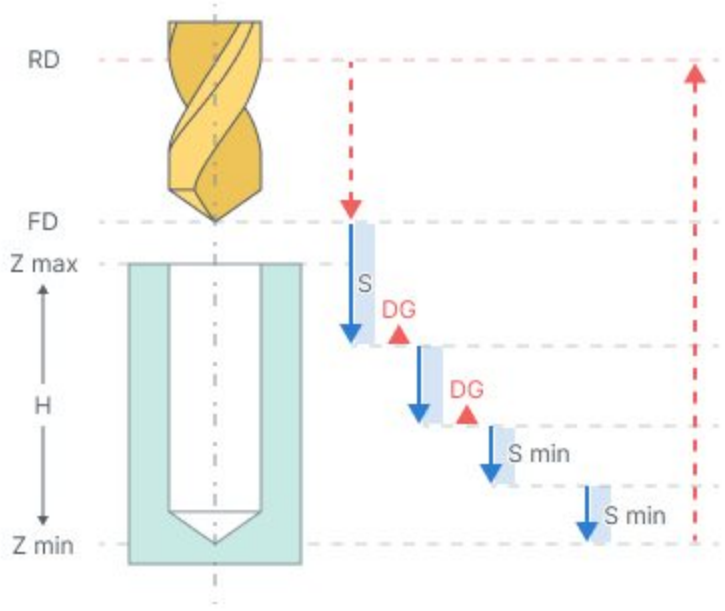
Radial stepover (ASP). Archimedes spiral with Radial stepover (ASP) motion at that level until the tool axis has reaches the circle with diameter equal to hole diameter reduced by the tool diameter (See drilling type hint).

Axial stepover (Z). Defines the distance between the horizontal layers for (ASP) (See drilling type hint).

Opposite move length (OM) Working movement length (See drilling type hint).

Thread step (TS) Defines the spiral step for the spiral hole machining or the thread pitch in the case of the thread milling (See drilling type hint).

Degression (DG). If enabled: the depth of every following peck is reduced on the defined value. The step reduction occurs until its value is not less than Minimal step. The subsequent steps will be equal to the minimum step.



If disabled: the step is constant. The final step is executed to cover the remaining depth of the hole.

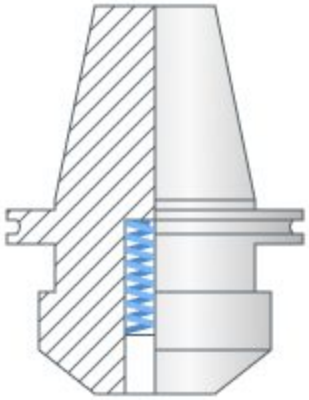
Minimal step (Smin). Minimal step is a percentage of the first step value.

Thread pitch. Defines the pitch in millimeters or inches. It depends on the current measurement units.

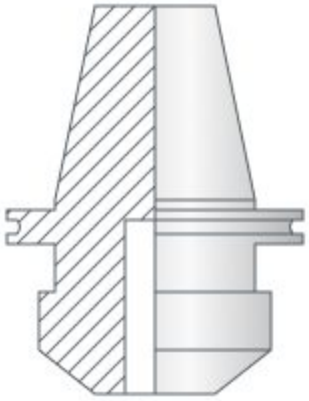
Spindle position (Initial angular position). Used for the multistart threads and defines the start Spindle position in degrees.

Socket type. Numerical controls in machining often offer various cycles tailored for different socket types, which can be categorized as either floating or fixed.

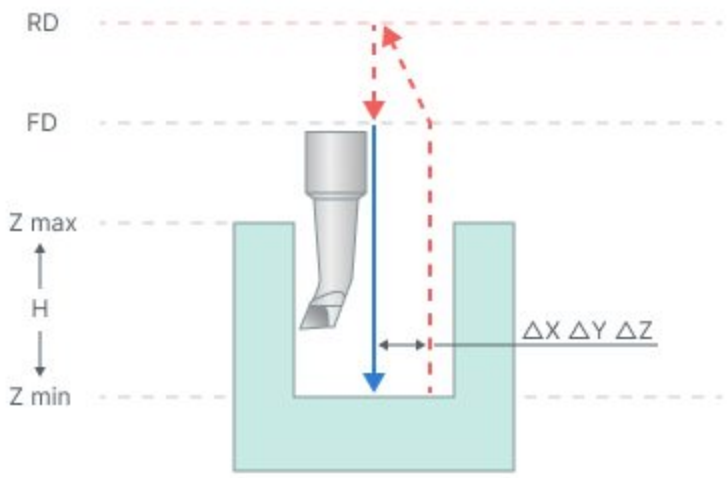
Floating.



Fixed.



Oriented tool retract. Options on the Oriented retract panel allow retraction without contact the tool with machined surface at the exit. To do this, after the final depth of hole is reached the spindle stops with a strictly defined angle and a slightly shifts to the side. Then tool returns to the top level with a stationary spindle.



Spindle orientation angle. Defines the start Spindle position in degrees.

ΔX, ΔY, ΔZ - Offset from hole center (See Oriented tool retract hint).

Compensation mode. Similar to the Output group in 2D contouring. Details

Milling type. Milling direction of the Combined spiral type.

Conventional. Use for Combined spiral. Parameter group works similarly to the Waterline roughing operation - Milling type. Details

Climb. Use for Combined spiral. Parameter group works similarly to the Waterline roughing operation - Milling type. Details

Round to integer. Used by the Combined spiral type. Try rounding to a smaller or larger number of revolutions.

Finish pass. Used by the Combined spiral type. The pass becomes finishing. An additional roughing pass is added to it at a distance specified in the parameter.

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

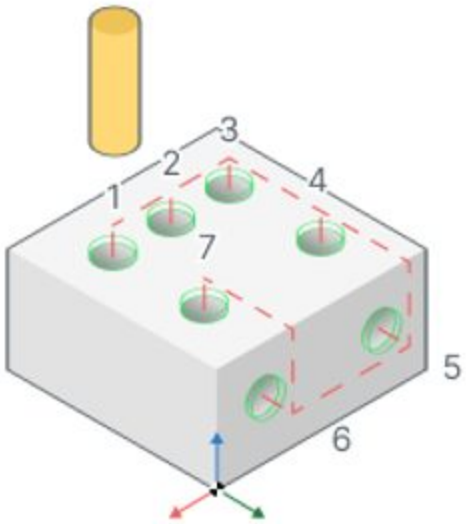
The parameters of this group are the same as in the 5D Surfacing operation. Details.

Hole sorting.

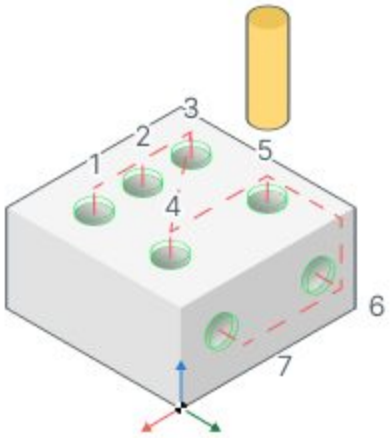
Hole processing order

▼ Click here to expand.

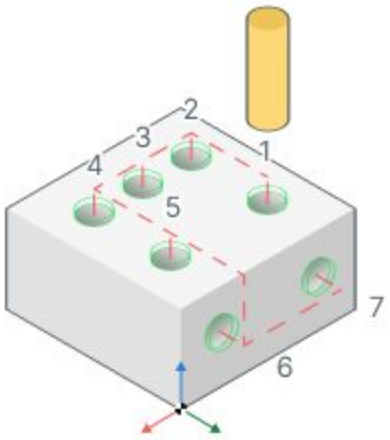
Optimal. The system minimizes the length of the idle motions.



By list. The order of the holes machining depends on the order in the Job assignment list.



Group holes with the same plane. Item selected then system will optimize machining order inside groups of holes that lie at the same planes.

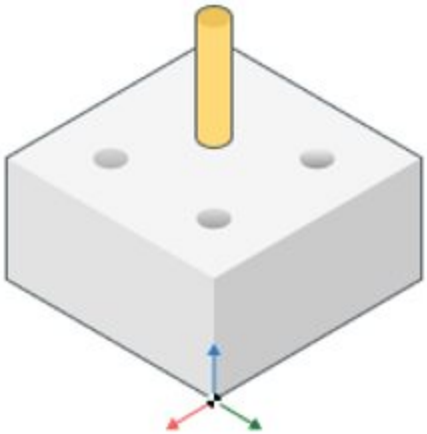


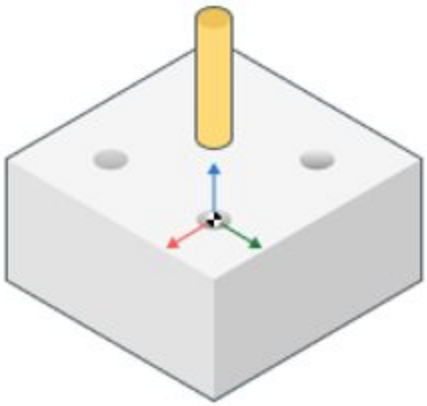
Use holes coordinate system.

▼ Click here to expand..

If enabled: ORIGIN command is output before the hole machining. So every hole is machined with it's own local coordinate system. Read the operation coordinate system topic for detailed information about ORIGIN. So the hole machining cycle is applied for the XY plane. Most of the CNC supports the cycles in this plane.

If disabled: All the holes are machined in a common coordinate system operations.





NC Code Format

Defines how the drilling operation is output.

▼ Click here to expand..

Long Hand. Only simple NC commands like ISO G0, G1 will be used in the CLData.

Canned Cycle. Canned cycles like ISO G81 will be used in the generated CLData.

Cycle format. For compatibility with older versions of postprocessors.

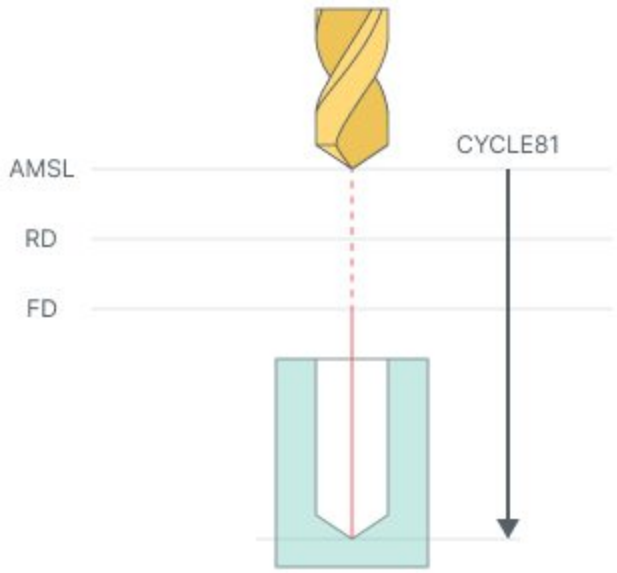
Default (as specified in system Setup). The cycle format will be used, which is specified in the system settings Details. The default setting in the system Setup window has a value EXTCYCLE.

EXTCYCLE (recommended). The new format of the cycle EXTCYCLE will be used. This cycle has an advanced set of parameters, including all machining strategies that are implemented in the system, and allows a realistic simulation of the tool movements according to the chosen strategy.

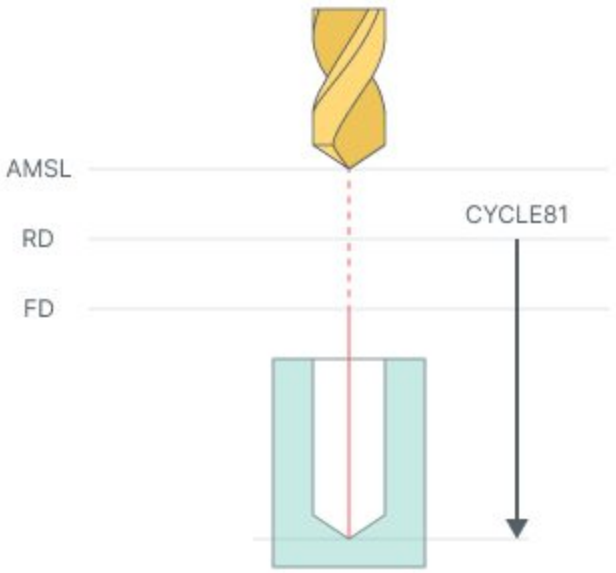
CYCLE (for old postprocessors). The old format of the cycle CYCLE will be used. This cycle cannot be used for some of the strategies available in the system (e.g., hole pocketing or machining by spiral). Also this cycle simulates any machining strategy only as a simple movement to the lower level of the hole. This format is required for compatibility with older versions of postprocessors, where EXTCYCLE technological command handler is not implemented.

Start cycle at air move safety level. Option defines the position at which the each cycle toolpath will start.

If enabled: the first cycle point is at the air move safety level which depends on the part top level or origin of CS. It can help you to reduce the number of intermediate link points when you want to have the modal cycles output mode at the each next movement (whithout redundant G80 cycle off commands).



If disabled: the the first cycle point located at the Rapid distance from the top level of hole.

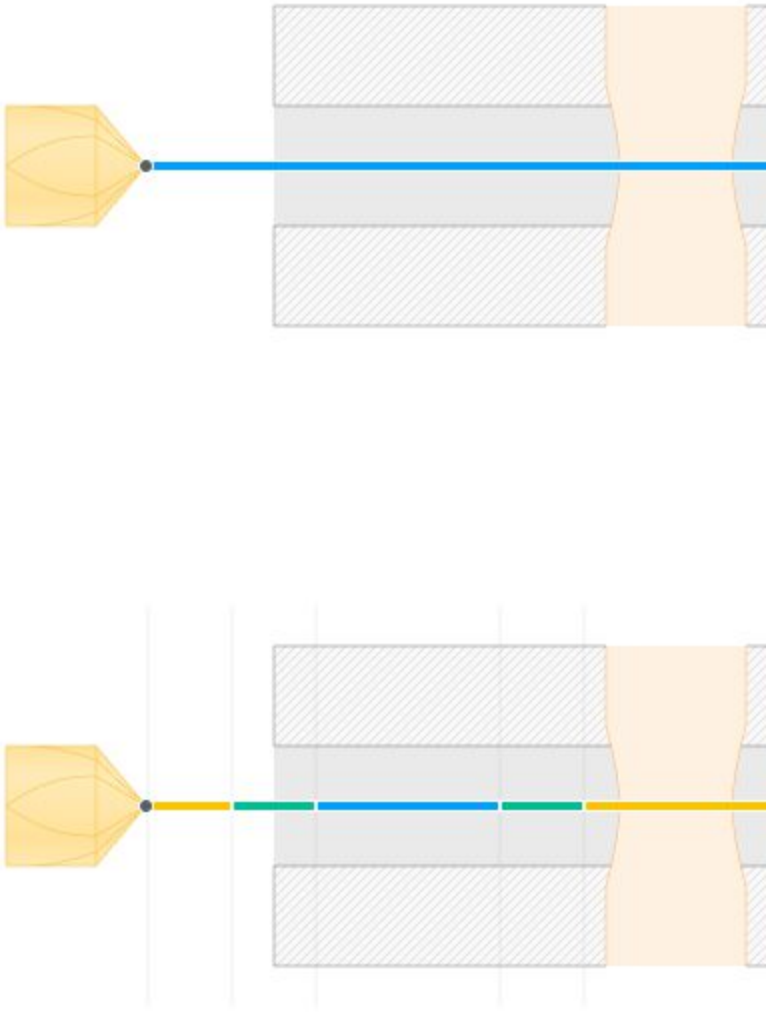


By Default. Canned Cycle.

Optimize feed in cross holes

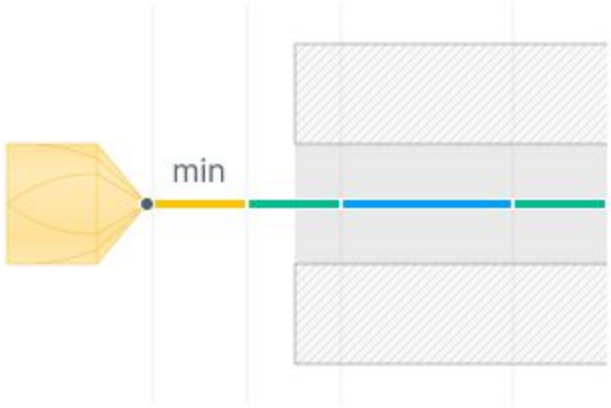
Programming feed rate reduction at hole intersections (work only with NC Code Format - Long Hand)

▼ Click here to expand..

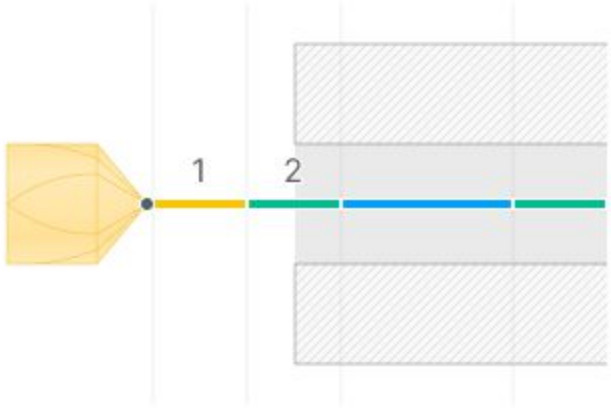


Blue - working feed Yellow - minimum feed Green - medium feed

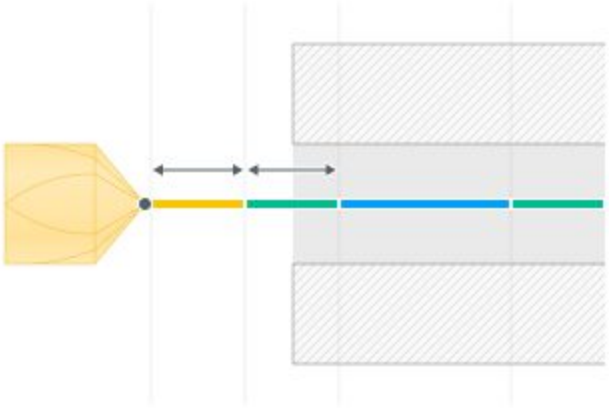
Min feed. The minimum allowable feed rate to which the speed is reduced when entering or exiting the hole's cutting zone



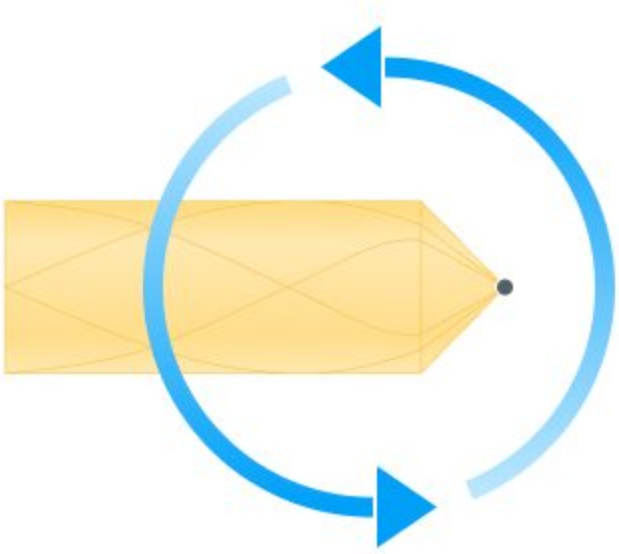
Step count. The number of steps before the main cut when entering or exiting the hole's cutting zone



Step distance. Step length

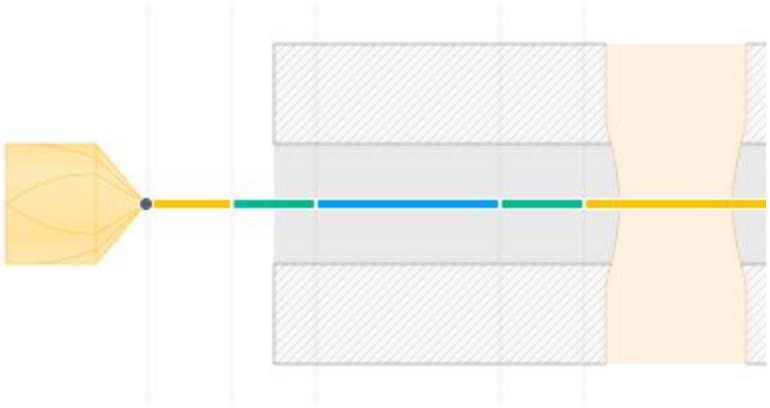


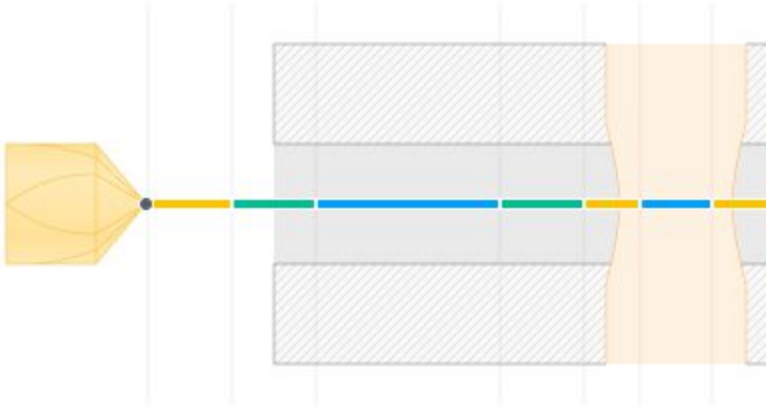
Reduce spindle speed. Reducing the spindle speed decreases the frequency and intensity of these vibrations, which lowers the stress on the cutting edges of the drill bit when entering or exiting the hole's



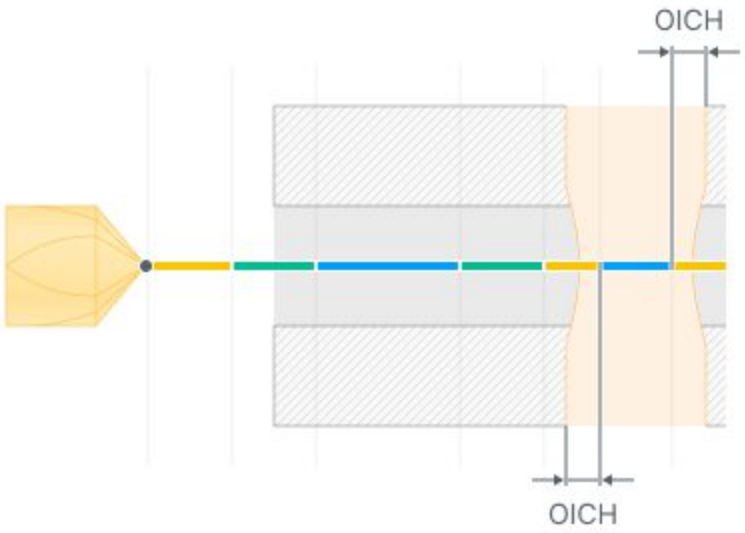
Always finish with reduced feed. If the trajectory ends at the working feed (not at the minimum), then steps are added to reduce it to the minimum (the trajectory is extended).

Void feed (disabled). This parameter allows you to set individual feed in cross holes.



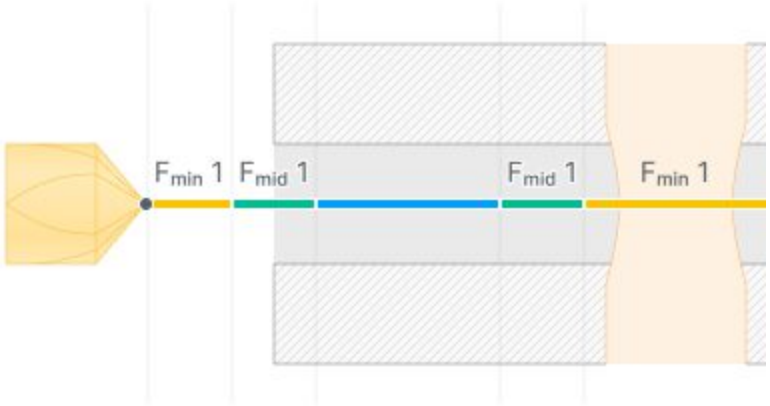


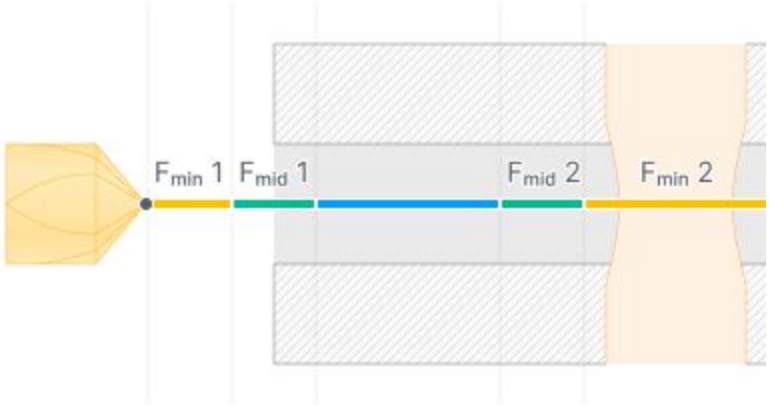
Offset inside cross-hole. Allows you to add distance in a cross hole from the walls of the crossing hole

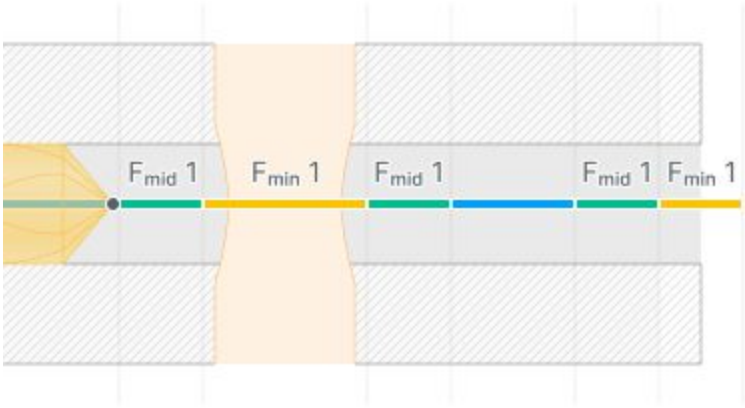


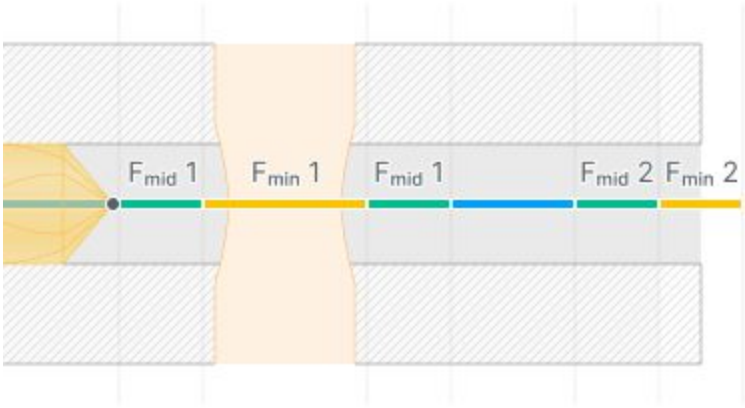
OICH - Offset inside cross-hole

Special part entry/exit feed. Allows you to make a separate feed/spindle speed for entering/exiting the part.

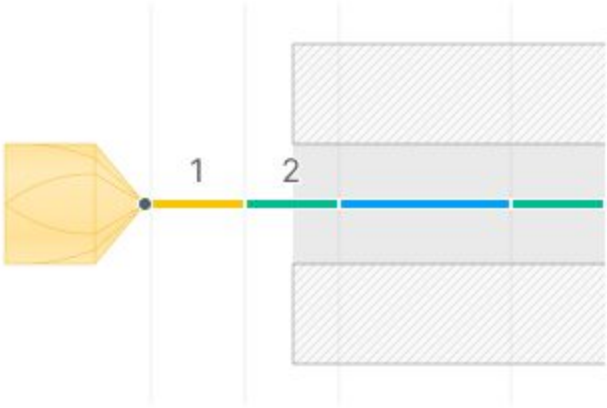

F_{min} 1
F_{mid} 1
F_{mid} 1
F_{min} 1

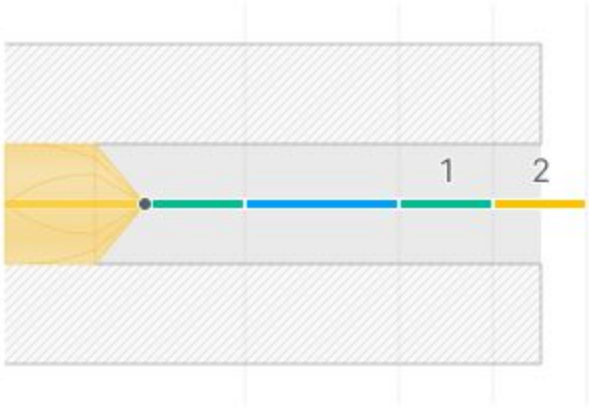

F_{min} 1
F_{mid} 1
F_{mid} 2
F_{min} 2



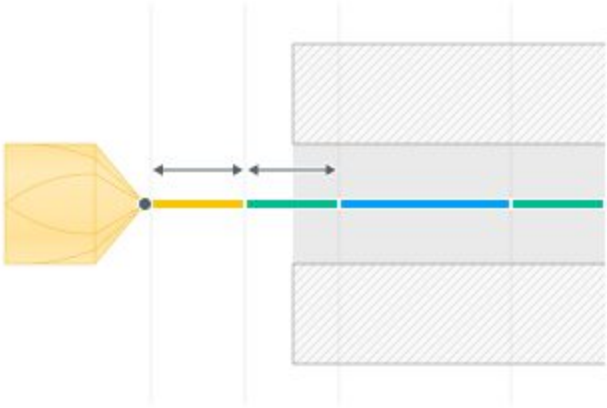


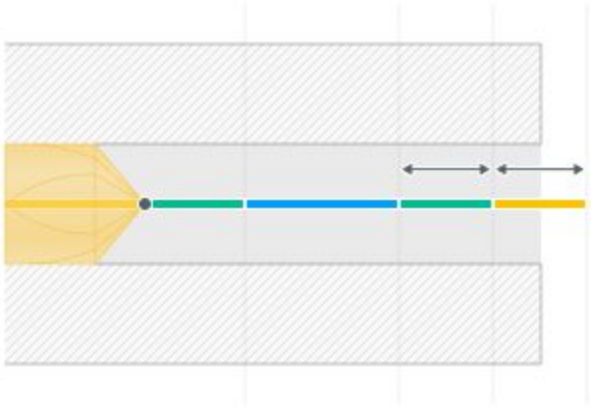
Step count. The number of steps before the main cut when entering or exiting the part.



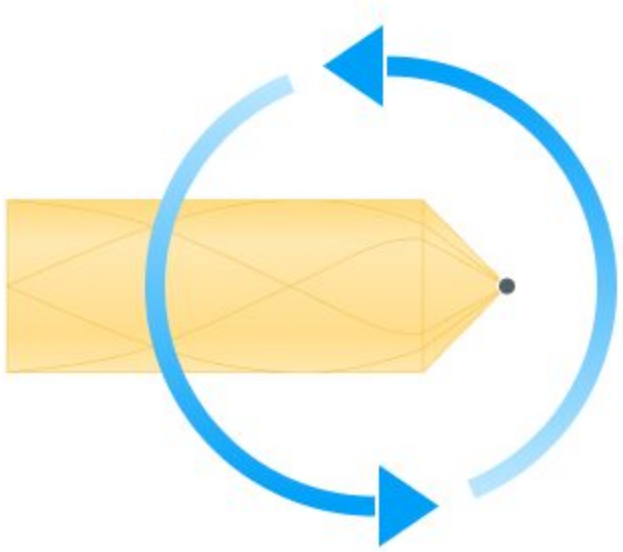


Step distance. Step length





Reduce spindle speed. Reducing the spindle speed decreases the frequency and intensity of these vibrations, which lowers the stress on the cutting edges of the drill bit when entering or exiting the part.



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

FBM Mill

Feature Based Machining

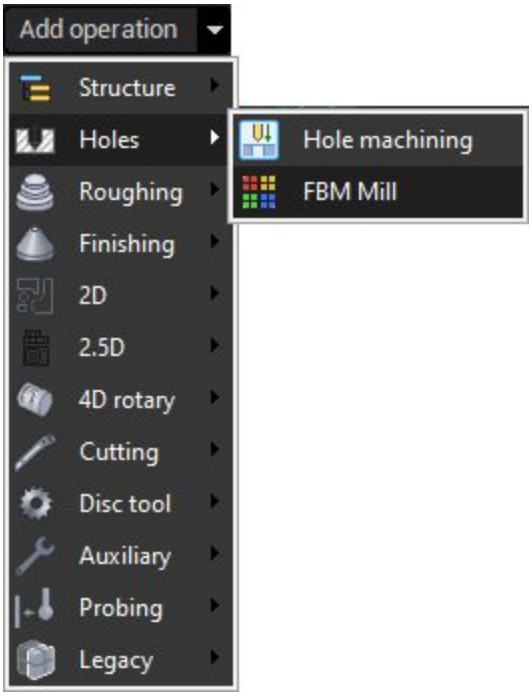
A Manufacturing Feature is a component of a part that can be manufactured with a standard machining process. Examples of features include holes, pockets, slots, grooves, chamfers.

The Feature Based Machining (FBM) is a CAM approach based on feature recognition and either automatic or automated generation of machining processes based on the information extracted from the recognized features and a Machining Knowledge Base.

FBM Mill group

The FBM Mill Group is a group operation in CAM system that allows you to do the Feature Based Machining.

When you create a new FBM Mill Group operation CAM system automatically recognizes all the known features in the part. After that you can easily select either predefined or stored machining procedures for the features from the FBM Procedure Library and CAM system will automatically generate the operations required to machine the features and sort them based on the current rules (one strategy is to sort operations in order to minimize the tool changes).



Creating a new FBM Mill group operation is as simple as creating a regular operation. Just select the corresponding item from the New.. menu.

FBM Machining Procedure

The FBM procedure is a machining sequence defined either for a feature or a feature class. There are two types of FBM procedures.

1. Atomic procedures
2. Complete procedures.

The Atomic procedure does only a small part of machining. Usually it consists just of one operation. The examples of atomic procedures include Spot Drilling, Rough Drilling, Chamfer Sinking, Chamfer Contouring.

The Complete procedure includes all the operations needed to machine a feature completely.

When you select a procedure for a feature in the UI only complete procedures are suggested to you. When you add operations to a selected procedure, only the atomic procedures are suggested.

A FMB procedure consists of four parts.

1. The procedure metadata such as type (atomic, complete), caption, icon, last modification date.
2. The Feature Constraints.
3. The Tool Query List
4. The Operation List

Feature Constraints

The Feature Constraints is the part that defines what type of a feature is the procedure for and what parameters should the feature meet.

The examples of feature types include Hole, Stepped Hole, Hole Groove.

Every feature type has its own set of parameters. For instance, a hole has such parameters as type (blind, through), diameter, height, tip angle, taper angle and others. A stepped hole is a composite feature. It consists of two or more holes and may include some grooves. A hole groove is a cutout in a hole that is defined by such parameters as type (round, square, trapezoid), height, diameter, taper angle, corner radius

In the Feature Constraints you define the constraints for the parameters the feature has to satisfy in order to be applicable to the machining procedure. The constraints can be expressed in the following ways.

1. Just a value. E.g. Diameter=10.
2. A range of values (Min..Max). E.g. Diameter=10..20.
3. Less Than (<) expression. E.g. Diameter<5.

4. Less Or Equal (<=) expression. E.g. Diameter<=5.
5. Greater (>) expression. E.g. Diameter>5.
6. Greater Or Equal (>=) expression. E.g. Diameter>=5.

When the user selects a feature in the UI to assign a machining sequence for it, CAM system scans through all procedures in the library and compares the feature against the feature constraints of every procedure. Every time it finds a match the procedure is added to the suggestion list. After that CAM system looks at the Tool Query Lists of the procedures and searches for the tools in the Tool library.

Tool query list

The Tool Query List consists of entries defining the rules of selection of the tools from the Tool Library for the operations of the machining procedure.

For each tool query the desired tool type and the constraints for the tool parameters are specified. The tool parameters constraints are expressed the same way as the feature constraints. The important thing is it is possible to use the references to the feature parameters in the tool constraints. A reference is defined by the full path to the parameter started with the # character. For example, for a hole feature the possible parameter references include #Hole.Diameter, #Hole.Height, #Hole.TipAngle and others. So for a hole machining procedure a usual tool query looks like:

- Tool.Type = Drill;
- Tool.Diameter=#Hole.Diameter;
- Tool.Length>=#Hole.Height.

It is possible also to use math operators and math functions in the constraints. For example, for a chamfer machining operation the tool query has the following constraints:

- Tool.Type = Conical Mill
- Tool.TaperAngle = Hole.TipAngle/2.

Operation list

The operation list is the list of machining operations to be used with the feature.

For every operation the following parameters are defined:

1. Subfeature Id.
2. Tool number.
3. Technological group.
4. Icon and caption.
5. Machining cycle type.
6. Machining cycle parameters.

7. The Subfeature Id is the Id of the subfeature of a composite feature which this particular operation machines. For example, a Stepped Hole machining procedure consists of operations that machine particular steps and grooves of this hole, so Subfeature Id of its operations can be Step1, Step2,...StepN, Groove1, Groove2,.. GrooveN.

The Tool number specifies the number of the tool in the Tool Query List which the operation uses. Several operations can use the same tool. In this case they will have the same tool number.

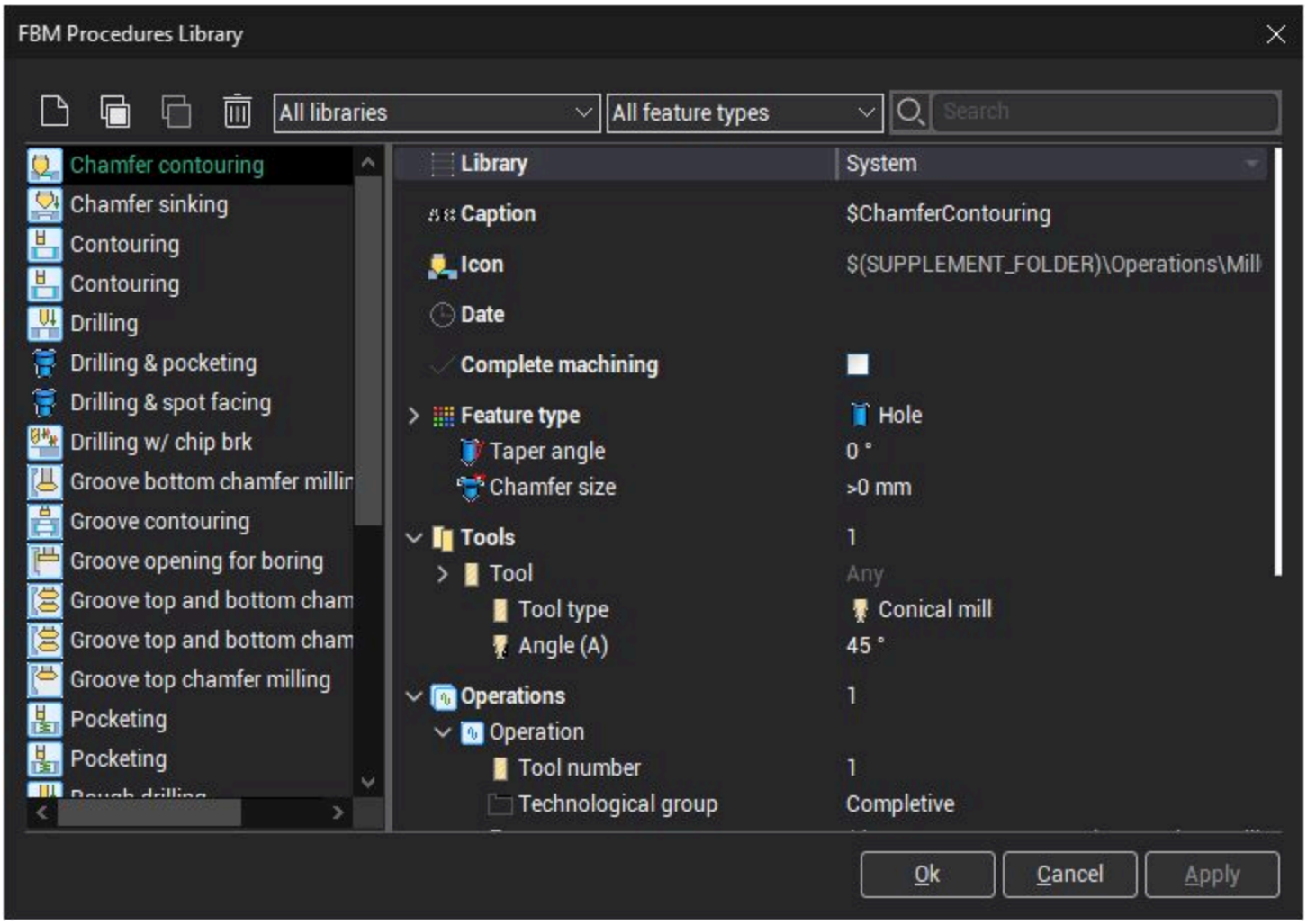
The Technological Group defines the class to which the operation belongs. There are the following technological groups:

- preparing;
- roughing;
- semi-finishing;
- finishing;
- completive

The Machining cycle type sets the machining cycle for the operation. For every feature type there are numerous machining cycles available. For example, for a hole feature there are such cycles as Simple Drilling, Drilling w/Chip Breaking, Drilling w/Chip Removing and others available.

The Machining cycle parameters are stored as a list of pairs Parameter=Value, so only the important parameters has to be specified, the other parameters will have the default values of the cycle.

FBM Procedure Library UI



The FBM Procedure Library is the storage for the machining procedures. There are four folders in the library:

- System;
- All users;
- Personal;
- History.

The System folder is read only, it comes together with the CAM system installation to provide the most universally used complete and atomic procedures.

The All Users folder contains procedures accessible to all users of the PC.

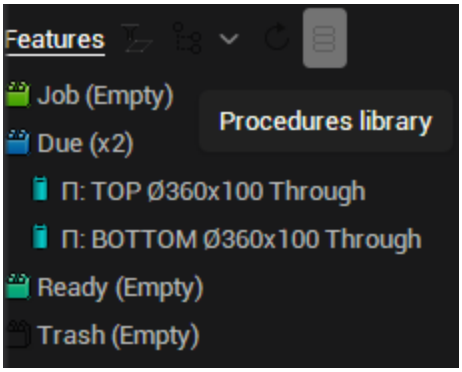
The Personal folder is for your own procedures.

The History folder contains procedures that are auto-saved with every project. In the course of work with CAM system FBM Mill the user dynamically creates new procedures directly in the UI simply by editing parameters of the existing procedures, inserting or removing operations into them. When the user saves

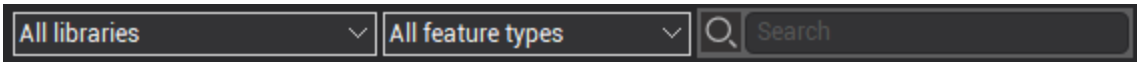
the project all the new and modified procedures are automatically saved to the History folder for the possible future reuse. The feature allows the user to grow his machining knowledge base effortlessly.

Opening the library

To open the library press the Procedures library button at the Features toolbar.



Filtering procedures



Use the Libraries drop-down to switch between libraries.

Use the Feature Type selector to filter procedures by feature type

Type text in the Search box to filter procedures by name.

Creating, Deleting, Copying procedures



- To create a new procedure press the New button.
- To delete a procedure, select it and press the Delete button.
- To copy a procedure press the Copy button, then press the Paste button.

Editing a procedure

The editing of a procedure is done in the Inspector. The inspector is slightly unusual though: the usual node expanding/collapsing works differently here:

- when the node is expanded, all the inner nodes are displayed. There is nothing new here.
- however, when the node is collapsed, the inner nodes that have overridden values are still visible.

By default all the nodes are collapsed, so you see only the overridden parameters. This makes sense as the overridden parameters define what the procedure is.

- To edit a parameter, expand the corresponding node in the inspector, enter the value, then collapse the node.
- To reset the parameter value to its default state, just clear its value by Selecting All and pressing either the BackSpace or Del key on the keyboard, then hit Enter

The value fields of parameters with default values are colored gray.

The parameters with invalid values are marked red.

Changing the library of a procedure

You can change the library of the procedure by selecting from the Library drop-down in the inspector.

Changing the type of a procedure

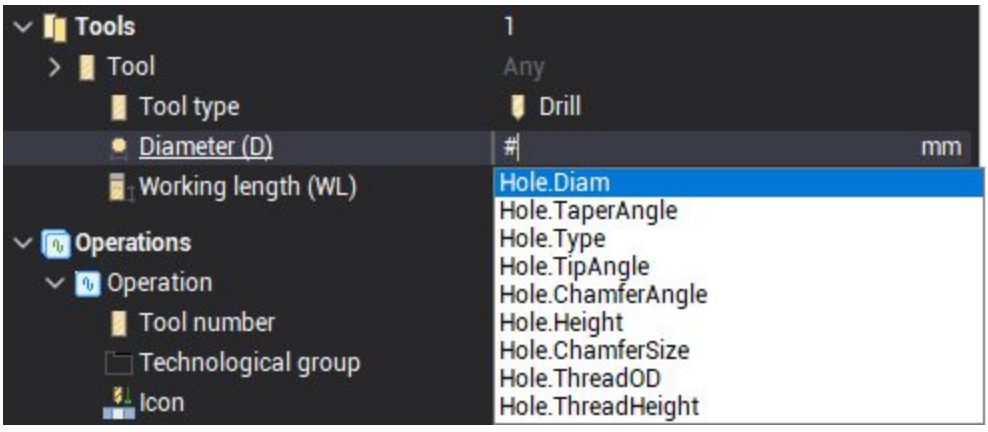
To switch between Atomic/Complete procedure types check/uncheck the Complete machining box in the inspector.

Editing the Feature Constraints

1. Select the Feature Type.
2. Expand the Feature Type
3. Specify the constraints for the feature parameters. The default value for a constraint is Any. It means that there are no constraints on the parameter.

Editing the tool queries

- To change the number of tools in the Tool query list enter the value in the Tools field and hit Enter.
- To reference a feature parameter in the tool query just type #. The parameter completion panel should appear. Select the needed parameter from it either with the mouse or with the Up/Down keys.



Editing the Operations

- To change the number of operations in the list just enter the value in the Operations field and hit Enter.
- For every operation you have to set the Tool Number, the Cycle type and the Cycle parameters. The default tool number of the operation is the same as its sequence number.

Applying/canceling the changes

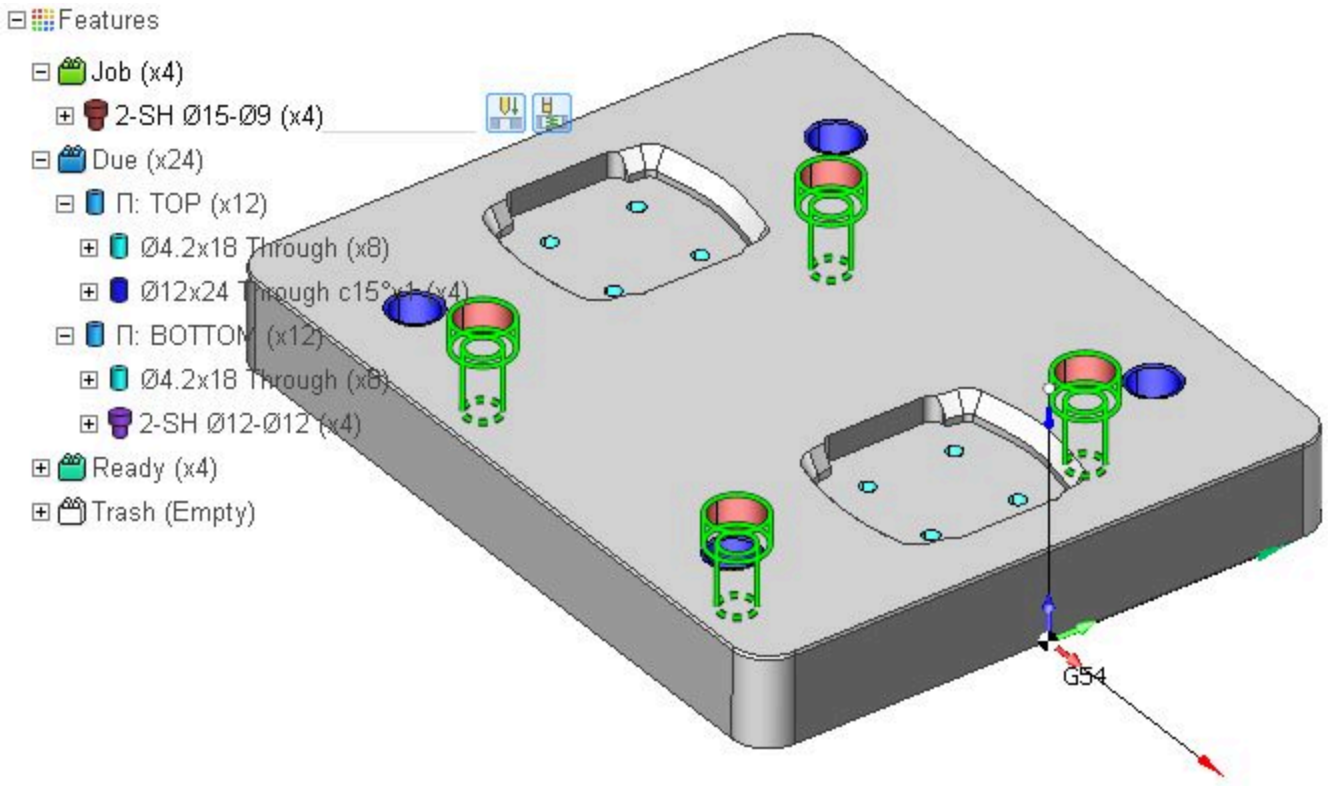
Edited procedures are marked with the bold font in the list.

- To apply changes press the Apply button at the bottom.
- To apply changes and close the FBM Procedure Library click Ok at the bottom.
- To cancel changes and close the FBM Procedure Library click Cancel at the bottom.

Feature Tree UI

After CAM system recognizes features, it sorts and organizes them into a hierarchical tree structure. At the highest level the Feature Tree consists of the four groups:

- Job;
- Due;
- Ready;
- Trash.



The Job features are the features that are machined in the current FBM Mill Group. In the graphic view you can easily distinguish them by the glowing lime outline.

The Due folder contains features that are not machined in any FBM Mill group yet.

The Ready folder contains features that are already machined in other FBM Mill group operations of the project.

The Trash folder contains the deleted features.

When one of those folders is collapsed then the features from it are not displayed in the graphic view. By default the Job and the Due folders are expanded while the Ready and the Trash folders are collapsed. So you see on the screen only the features that are subject to machining.

Counting features. Alternate features

Next to every node in the Feature tree the number of features of the node is shown in the parenthesis. You may notice that the number of features shown in the tree is bigger than the number of features you may expect. You also may notice that some of the features appear in the tree twice. This is not an error, but rather a feature. It happens because some features have alternate features. For example, a through hole can be drilled from two sides. In this case CAM system recognizes not one but two holes: one with the positive axis direction and the other with the negative axis direction. These are distinct features, however they are interconnected: when you machine one of the two alternate features, CAM system marks the other one as Ready automatically.

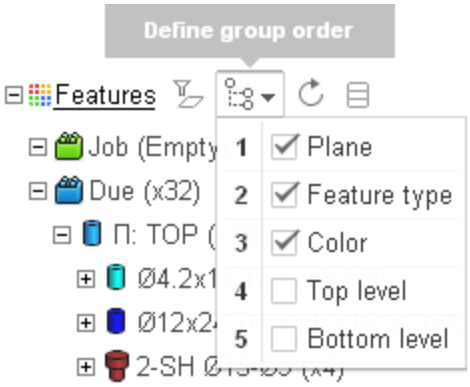
Changing the grouping scheme

CAM system groups features in accordance with the current grouping scheme. It is possible to group the features based on their:

- type;
- plane (principal orientation);
- top and bottom levels;
- color.

Furthermore, the features that are similar or identical are assigned the same distinct color and grouped together as well.

To change the grouping scheme of the Feature tree click the Grouping Scheme drop-down at the Features toolbar. Check/uncheck the items in the scheme to enable/disable them, drag the items to change their order. All the changes are immediately reflected in the tree.



Selecting features

There are two ways of selecting features.

1. By clicking on a node in the tree.
2. By clicking on a feature in the graphic view.

To select multiple features hold either Ctrl or Shift key.

The Action Bar



When you select a feature in the graphic view rather than in the tree view, the Feature Action Bar appears under the cursor. The action bar contains the frequently used commands for the features:

1. The Feature Machining Procedure combo.
2. The Visibility switch.
3. The Filter command.
4. The Delete button.

Showing/Hiding features

To show or hide the selected features you can either click on the feature icon in the tree or press the corresponding button in the Action Bar.

Filtering features

It is possible to filter features by plane. Just select a feature lying in the plane of interest and press the Filter By Plane button. As a result only the features that have the same plane orientation with the selected feature will be displayed, the other features will be filtered out both from the tree view and the graphic view. This command comes in handy when working with a 3d machine tool, when you only want to see features lying in the horizontal plane.

Deleting features

When you select a feature in the tree or in the graphic view, a small cross icon appears next to the first selected node in the tree. Press this icon to delete the selected features. You can also delete a feature by pressing the Delete button in the action bar

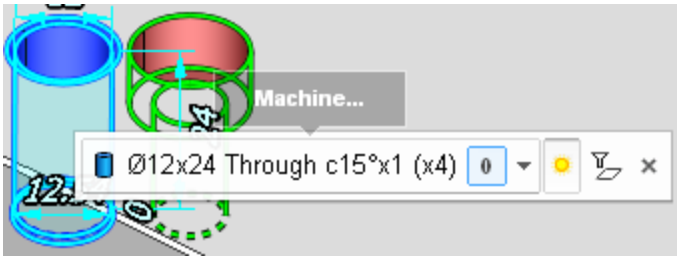
The Delete Features operation is context aware.

- When you delete a feature from the Job group, the feature is not deleted, it is just moved from the Job folder into the Due folder.
- When you delete a feature either from the Due or the Ready folder it is moved to the Trash folder.
- When you delete a feature from the Trash folder it is actually not deleted, but restored: it reappears in the Due folder again.

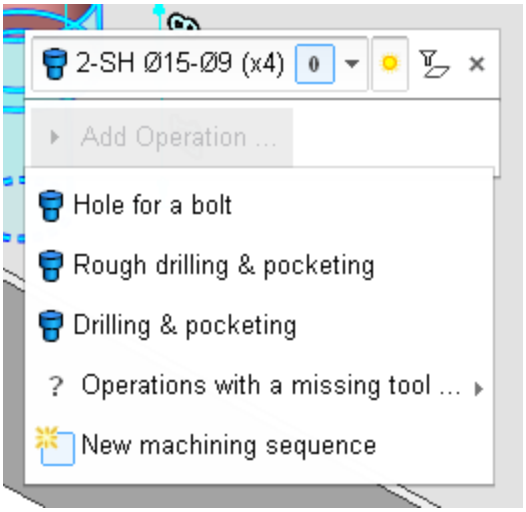
Assigning procedures to features UI

To select a machining procedure for a feature,

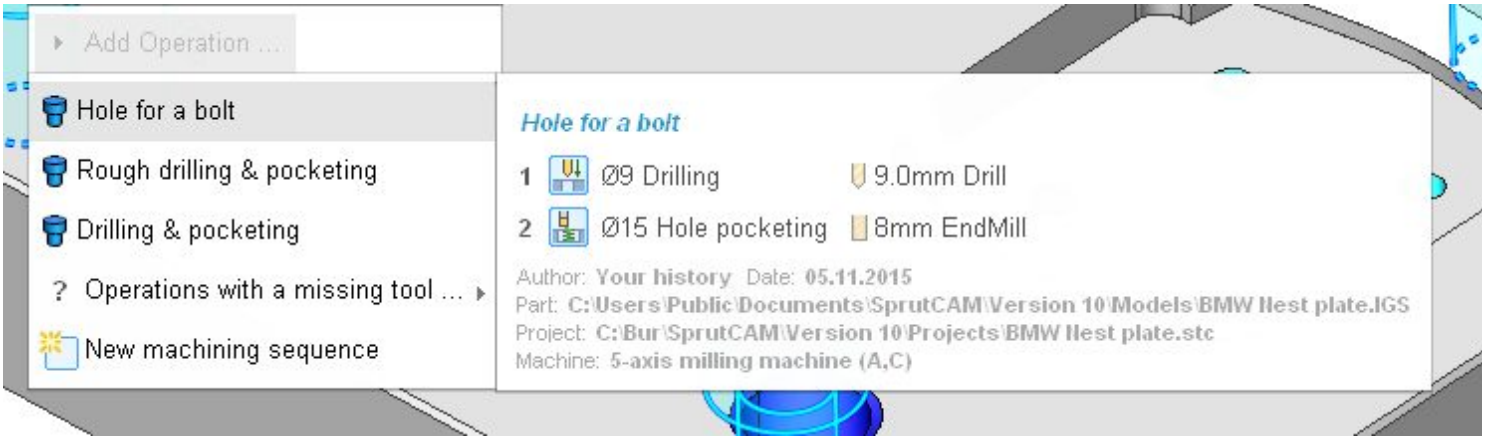
- Either click on the 0 icon in the tree. This icon appears when you highlight a node in the tree;
- Or click on the Machining procedure combo in the action bar.



- Then hover the cursor over the Add Operation⋯ menu and you'll see the list of the machining procedures applicable to the selected features.



- Hover the cursor over a procedure in the menu and you'll see the list of the constituting operations together with the tools selected for them.



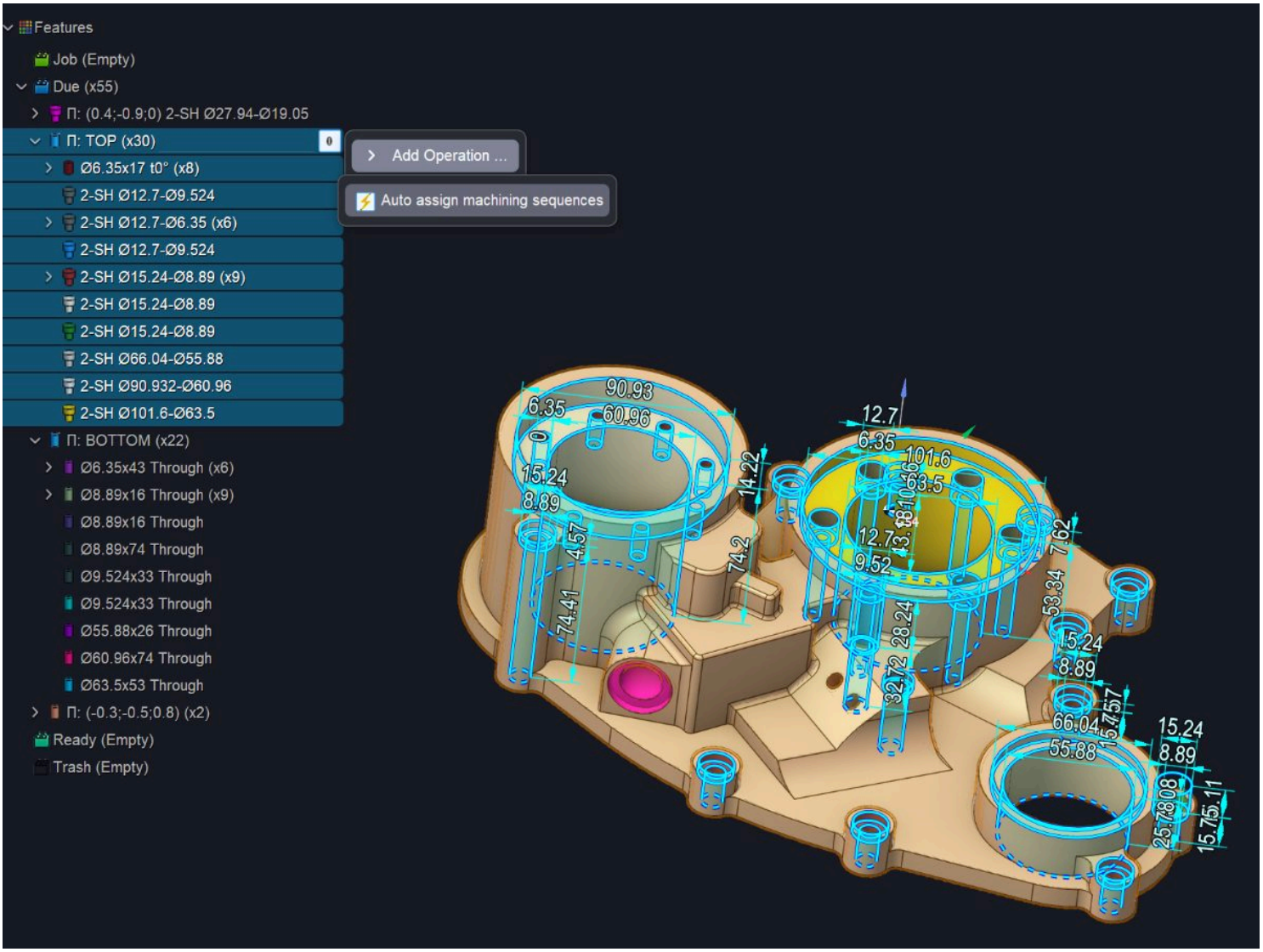
Along with that you will see some additional information about the origins of the suggested procedure: the author, the date of creation, the part it was created for, the project it was used in, the machine which was used in that project.

The operations for which CAM system can not find tools are put into a separate sub-menu, the Operations with a missing tool··· When you select a machining sequence from this menu, the tools missing in the Tool Library are created.

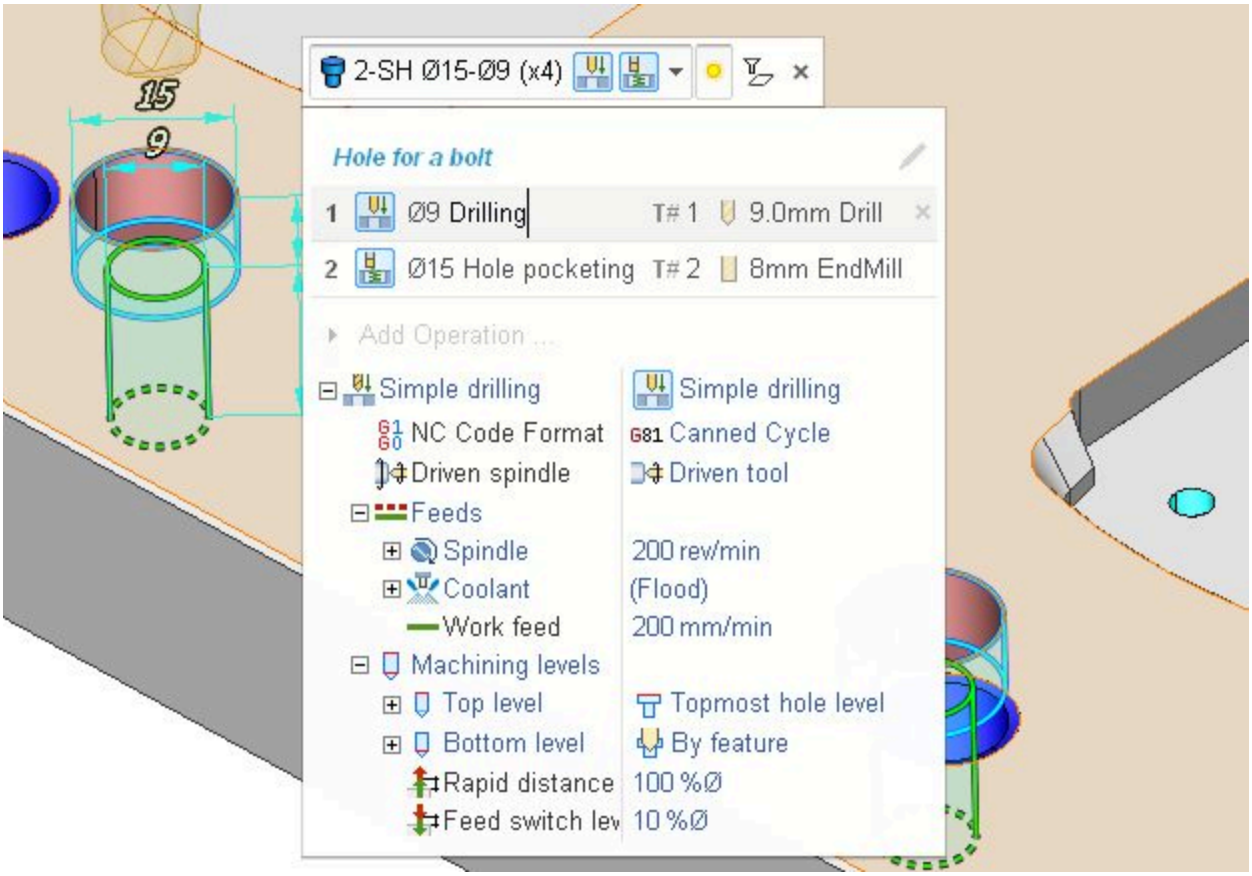
To create a new machining sequence rather than using an existing one, choose the New Machining Sequence command.

After you select the machining procedure for a feature or create a new one, the feature is moved to the Job folder, and CAM system updates the resulting operation list, while the selected procedure becomes available for editing.

The ability to automatically assign processing rules to a group of holes in a specific plane



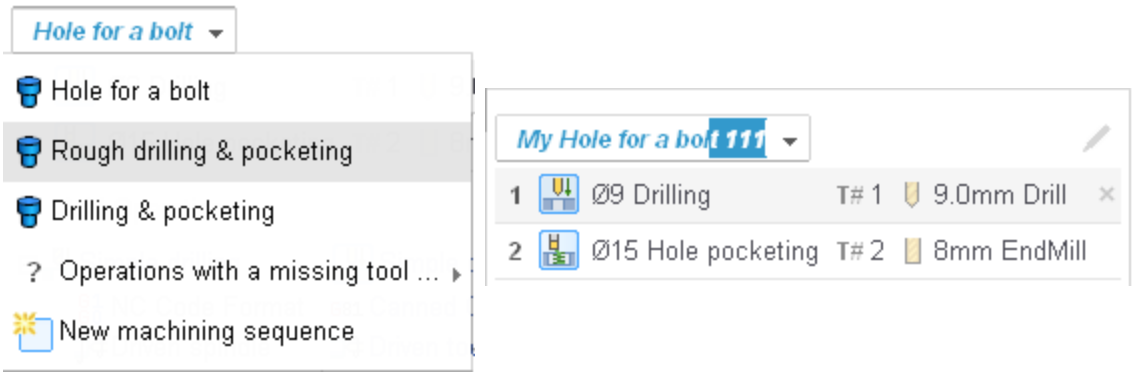
Procedure Editing UI



The editing of machining procedures is done in this little panel you can see above.

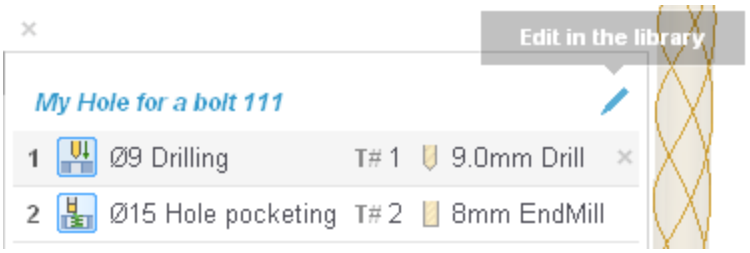
The Machining procedure selector

At the top of the panel there is the machining procedure selector (Hole for a bolt at the image). It's a combobox. You can quickly select a different procedure for the feature from it. In this combobox in the inner edit-box you can also change the name of the current procedure.



The Edit in the Library button

At the right of the procedure selector there is the Edit in the Library button. Clicking on it saves the current machining sequence to the Procedure Library and opens the Library with the procedure selected.



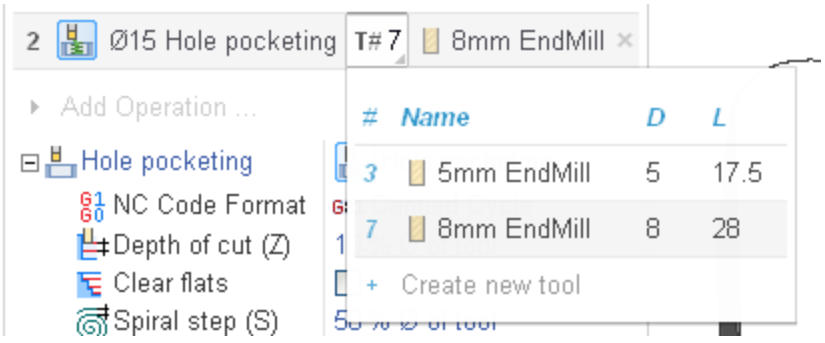
The operation list

Under the header there is the list of the operations constituting the machining sequence.



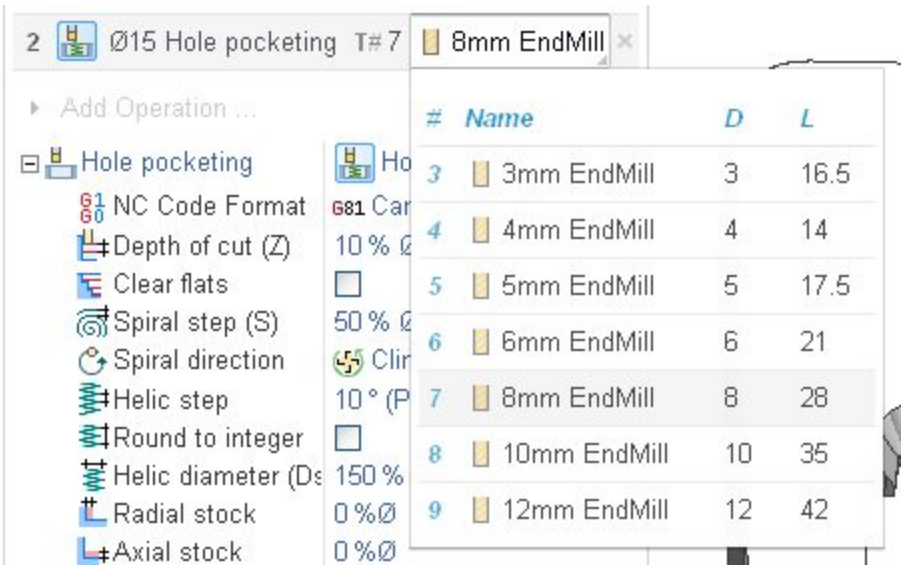
Each entry in the list consists of the following fields.

1. The sequence number. You can drag and drop operations by this number.
2. The Icon.
3. The subfeature prefix (present only if an operation machines a subfeature inside a composite feature).
4. The Operation name, It's an edit-box, you can change the operation name any time you want.
5. The tool number selector. This is a combobox. With it you select the tool from the Used Tool List for the operation of the machining sequence. Only applicable tools that satisfy the tool query are available for selection. The same tool can be used in several operations. In this case the number of operations the tool is used by is displayed in the parenthesis.



To add a new tool to the Used Tool List select the Create new tool command.

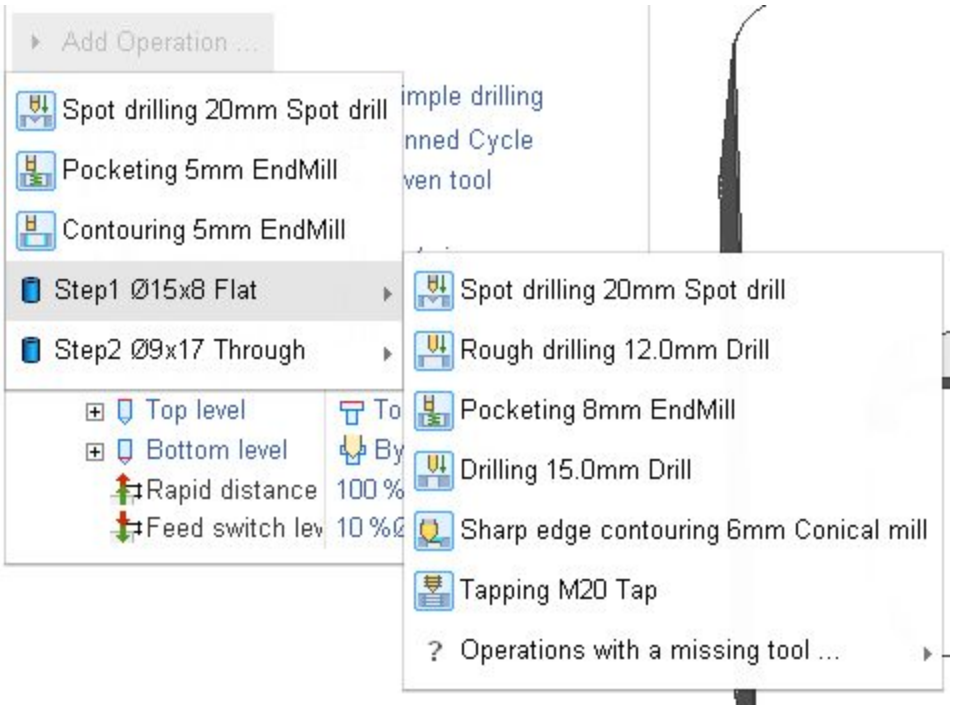
6.The tool selector. In this combobox you change the tool selected in the previous field. If the tool is used in several operations, all those operations are affected.



7.The Delete button.

The Add Operation⋯ menu

Under the operation list there is the Add Operation⋯ menu.

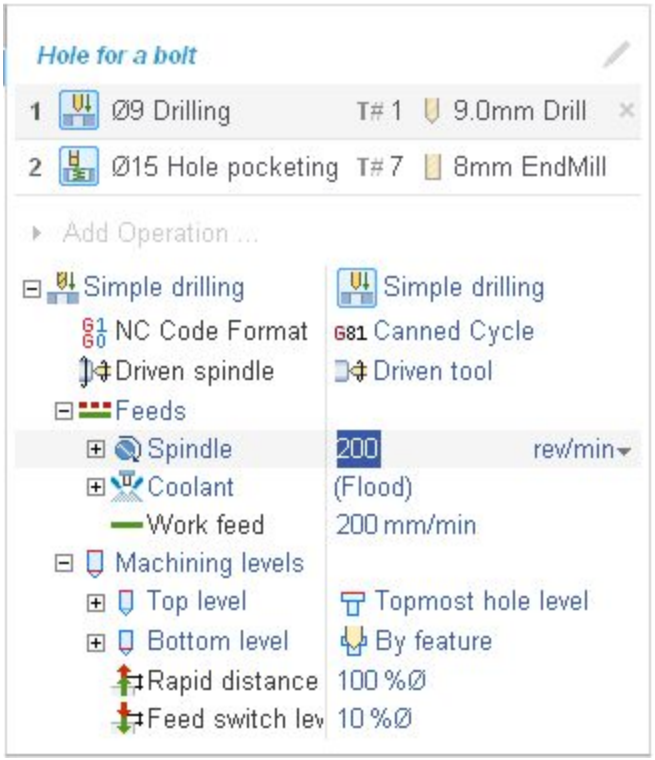


From this menu you can add new operation to the machining sequence. Only applicable operations are suggested in this menu. The operations with missing tools are put into a separate sub-menu. If the feature has subfeatures, operations for the subfeatures are available in separate submenus as well.

The operation parameters inspector

The parameters of the selected operation are displayed in the inspector ready for editing. The first parameter is the cycle type. You can select the cycle type from the list of the cycles available for the

selected feature. The rest of parameters are the parameters of the cycle.



Using the editing procedures UI to create new FBM procedures

The editing procedures UI is a very simple and efficient way of adding new machining procedures to the library. You just select a feature, select an applicable or create a new machining sequence for it, edit the machining process, then you click the Edit in the library button, make some adjustments to the procedure already in the library and that's it, the new procedure is ready.

With the History feature the process of expanding your machining knowledge base simplifies even further. When you save a project with FBM Mill group operations in it, all the machining sequences of the project are saved to the library automatically to the History folder.

Operations generation parameters

CAM system generates and updates operations inside the FBM Mill group automatically as you add or change machining sequences of the features. The order of the generated operations is the same as the order of features in the Job folder of the feature tree.

Two parameters placed at the Strategy tab of the FBM Mill group parameters affect the generation of operations.

1. Minimize tool changes.
2. Unite operations with the same tool.

Minimize tool changes

When the Minimize tool changes option is enabled, the operations inside the FBM Mill group are ordered in such a way that the operations which use the same tool are put together (of course, if it doesn't conflict with the order of operations of the machining procedures).

Unite operations with the same tool

This option is available only if the Minimize tool changes option is enabled. When enabled, it makes CAM system to merge operations that use the same tool and stand next to each other in the FBM Mill group into one operation. This merge allows even further optimization of the toolpath.

FBM Mill operation parameters

The operations inside the FBM Mill group are the special FBM Mill operations.

A FBM Mill operation allows to machine in one operation features lying in different planes, with different feature types and even with different machining cycles as long as the cycles use the same tool. The operation offers the following options to generate an efficient collision free toolpath.

Feature sorting methods

- By List,
- Optimal,
- Group by plane.

Identical Features processing methods

- Independent,
- Copy,
- Formalize As Subroutine.

Local Coordinate Systems of the features

Methods of transition between toolpaths

- Short,
- Orthogonal,
- Via Safe Surface.

2D

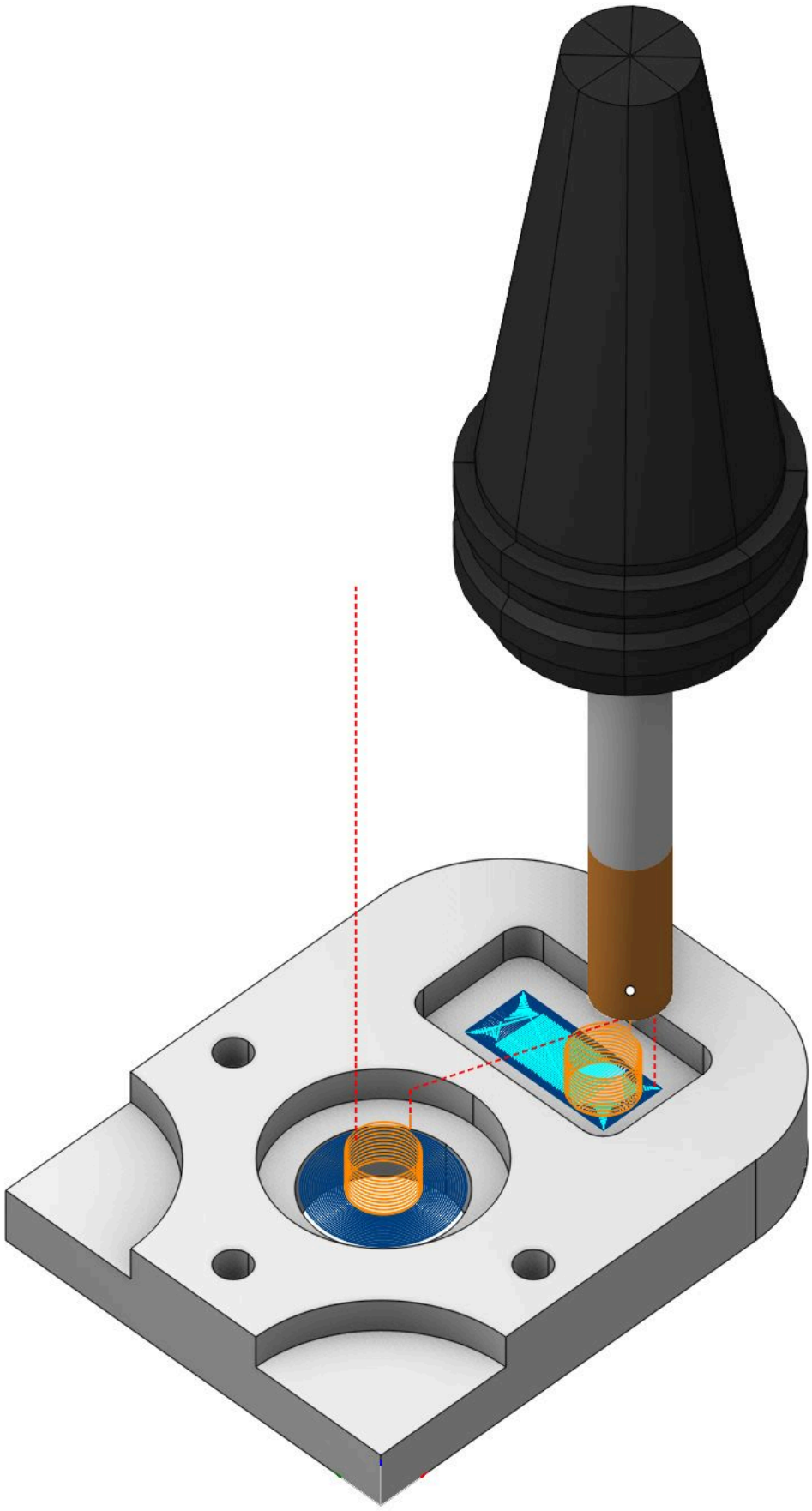
Application area:

The 2D group contains operations for machining pockets, engraving, and following planar contours. The operation topics describe the geometry and parameters required to generate each toolpath.

Operations:

- Pocketing. Removes material from pockets defined by the job assignment. Details
- Engraving. Machines engraved geometry and inscriptions. Details
- 2D contouring. Machines planar contours. Details

Pocketing



Application area:

The pocketing operation is used for 2/2.5D machining of pockets and islands, and also for preliminary material removal before engraving operations. As in the engraving operation, the model being machined is formed from a projection of curves onto the horizontal (XY) plane. The model is created by successive addition of curves or curve groups into the resulting area.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Base surface. This surface serves the purpose of projecting a trajectory onto it. Details

Add Boss. Allows you to add a boss. Details

Add Pocket. Allows you to add a pocket. Details

Add Inversion. Indicates a closed area, inside which the machining rules will be reversed, i.e. machined areas will become unmachined and vice versa. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Machining strategy:

This parameter allows the user to achieve a toolpath that appears parallel or equidistant or other when performing pocketing operations. This parameter group works similarly to the Waterline roughing operation. Details

Machining levels:

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

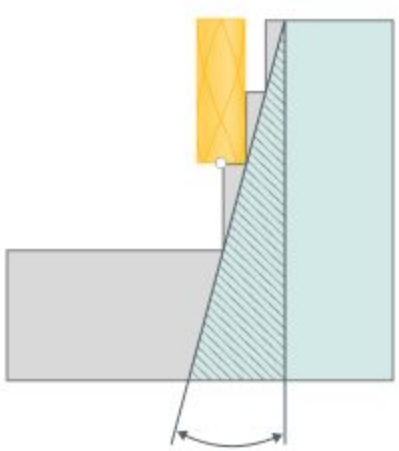
Depth Step:

Material removed in one pass along the tool axis between the Top and Bottom levels.

▼ Click here to expand..
Side angle. This refers to the inclined surface of the component (such as a casting slope), and creates a clearance from the vertex.

When the Side angle parameter is set to 0, the resulting machined feature is a simple extrusion. The profile of this extrusion is defined by the geometry sketched or selected at the top machining level (Z-plane).

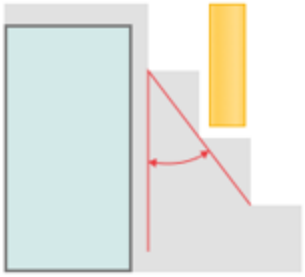
If the Side angle is set to a value greater than 0 (but less than 90), the CAM system automatically generates a tapered side wall. This creates a 3D feature where the sides are inclined relative to the vertical axis (Z-axis), with the angle of inclination precisely matching the specified lateral angle.



Relief angle. Inputting a positive value avoids the tool shank 'rubbing' as the machining depth increases.

When machining deep vertical walls, it is sometimes needed to restrict tool contact with the already machined part. For this purpose, in the operations that perform machining by layers, it is possible to assign a relief angle. When machining a vertical (or close to vertical) surface, the real stock will increase at machining layer as shown in the picture (below). The relief angle is allowed for all walls that are close to vertical, i.e. not only close for the model being machined, but also the restricting ones.

The relief angle value cannot be negative, nor can it be more than or equal to 90 degrees. Setting the relief angle too high will lead to a large layer of unmachined material at the lower layers.



Milling type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting.

Controls the sequence of toolpath passes during the surface machining. This parameter group works similarly to the Waterline roughing operation. Details

Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. This parameter group works similarly to the 2D contouring. Details

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

The parameters of this group are the same as in the 5D Surfacing operation. Details.

Transformations.

A dialogue for transforming the toolpath. This parameter group works similarly to the 2D contouring. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Distances. Distances required to switch feeds Details.

Allow plunges. Enabling plunges allows machining of closed pockets. When CAM software encounters a closed pocket that can not be machined from the outside, it forms a Plunge move. Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

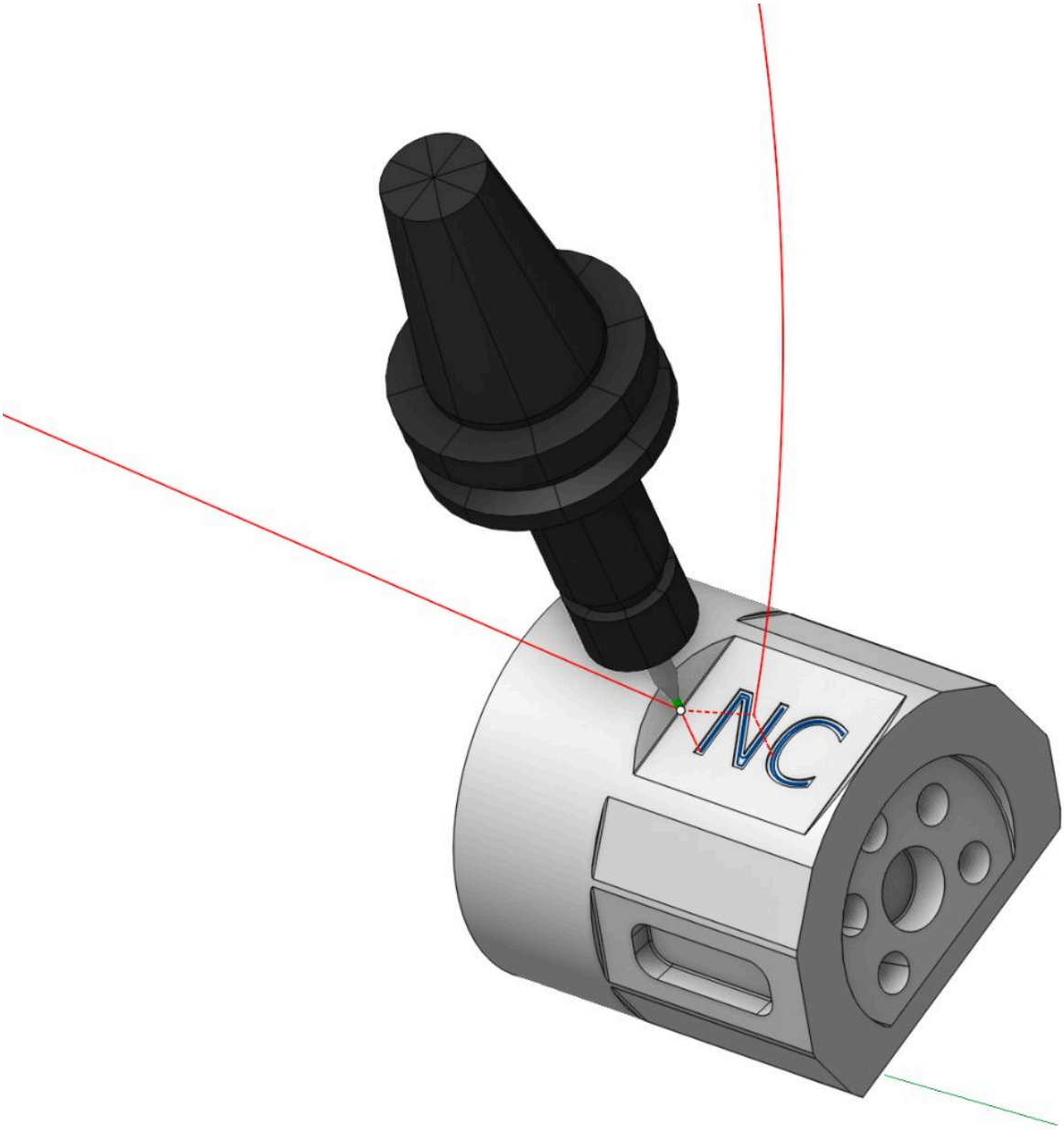
Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Holes (Drill points):

Drill points, including their coordinates, depth and tool diameter, are stored in an editable list and used for tool entry when the machining area cannot be approached from outside. Details

Engraving



Application area:

This operation is designed for the engraving of drawings and inscriptions on flat areas, and also for performing a finish pass along the side walls of pockets and contours using 2 / 2.5D machining.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Base surface. This surface serves the purpose of projecting a trajectory onto it. Details

Add Boss. Allows you to add a boss. Details

Add Pocket. Allows you to add a pocket. Details

Add Inversion. Indicates a closed area, inside which the machining rules will be reversed, i.e. machined areas will become unmachined and vice versa. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

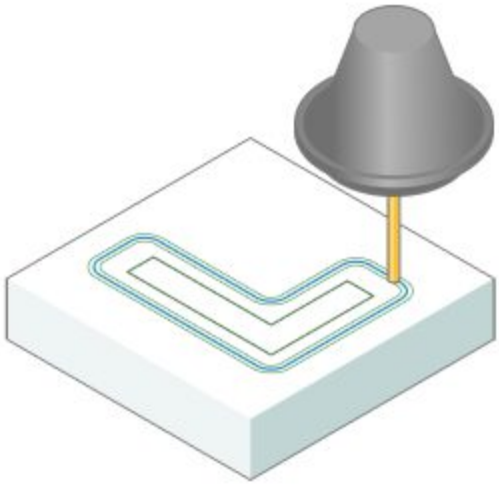
Restrictions. It allows you to restrict areas that should not be machined. Details

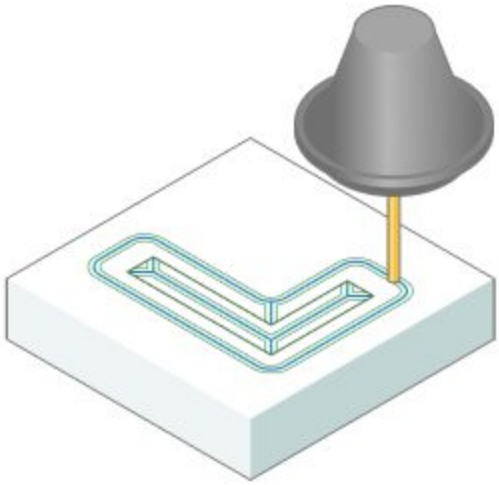
Strategy:

V Carving.

It adds additional passes to the trajectory to achieve a higher quality result, which is particularly effective for creating small elements.

▼ Click here to expand..





Machining levels:

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

Depth Step:

Material removed in one pass along the tool axis between the Top and Bottom levels. This parameter group works similarly to the Pocketing . Details

Milling type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting:

Controls the sequence of toolpath passes during the surface machining.. This parameter group works similarly to the 2D contouring. Details

Corners:

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

Some parameter group works similarly to the 2D contouring - Corners smoothing . Details

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

The parameters of this group are the same as in the 5D Surfacing operation. Details.

Job zone:

Enables additional modification of the working area initially selected on the Job assignment tab. This parameter group works similarly to the Plane finishing operation. Details

Transformations.

A dialogue for transforming the toolpath. This parameter group works similarly to the 2D contouring. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

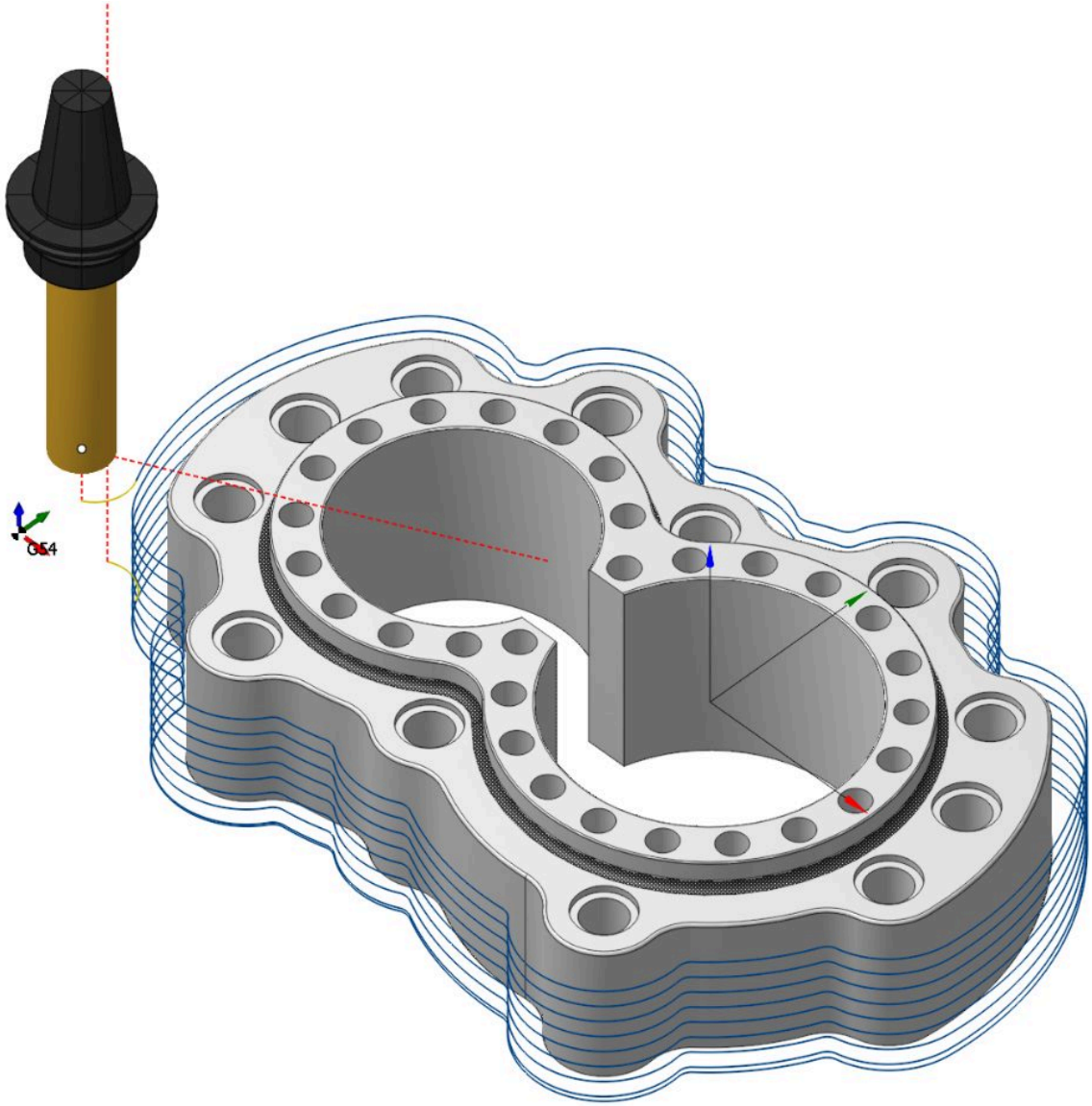
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

2D contouring



Application area:

The operation is designed for machining along horizontal contours or curve projections on the horizontal plane, cylinder or figure of revolution. It is also possible machining with the cylindrical and polar interpolation. "Project on part" option using allow to get a complex five-axis toolpath, when orientation of the tool is changing. Operation in conjunction with the "Tool contact point" parameter allows you to easily machine chamfers on the parts. The operation's list of processes could consist of several contours and curve projections. Every object can have its own machining method: either the tool center passes along the contour or by touching it with the left or right of the tool. If the contour is machined from right or left, then it is possible to define an additional stock for it. Positive stock is laid off towards machining. If the center of the mill follows the contour, then the stock value will be ignored, for it is impossible to define exactly which side the additional stock should be laid off.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Base surface. This is the surface on which the trajectory is constructed according to the curve projected onto it. Details

Curve. Set work order along curve . Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Top level. Specifying the top level by model elements. Details

Bottom level. Specifying the bottom level by model elements. Details

Add tabs. Adds tabs along the contour.Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list

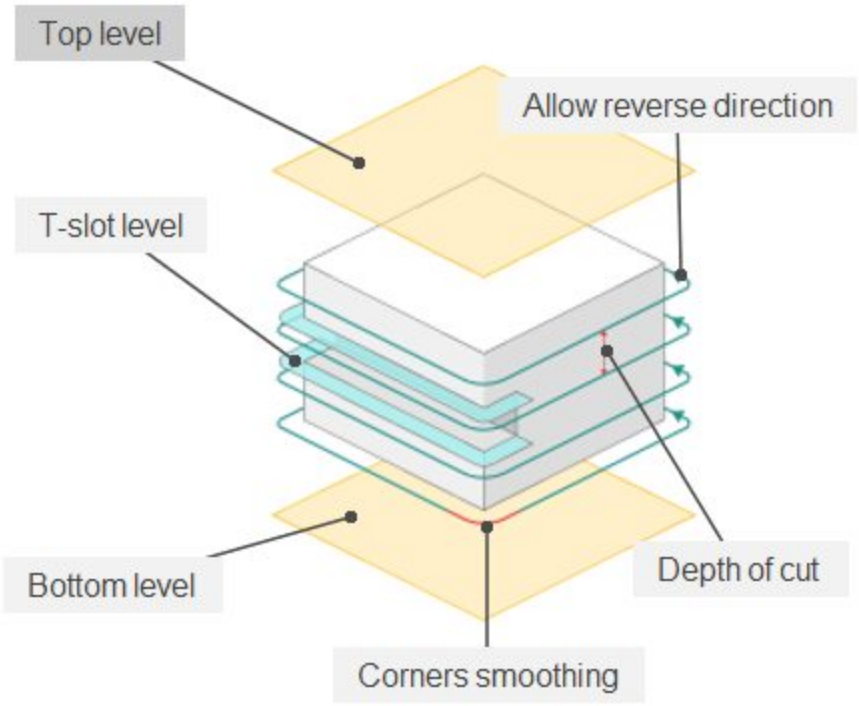
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Depth of cut.

The material removed in a single pass along the tool axis between Top level and Bottom level. By default, it equals % of the cutting tool diameter.

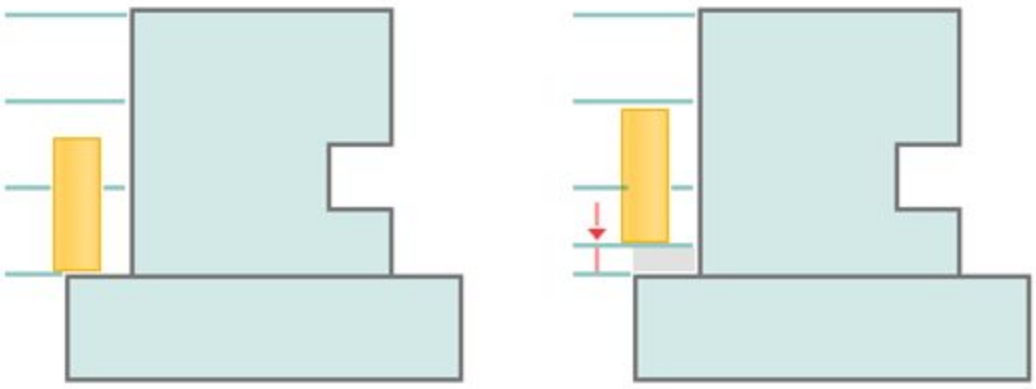
▼ Click here to expand..



Top level. The Top level for machining can be set dynamically or dimensonally. The default is based on the workpiece level.

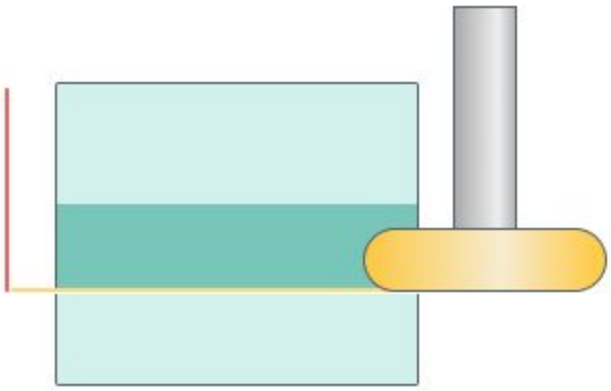
Bottom level. The Top level for machining can be set dynamically or dimensonally. The default is based on the workpiece level.

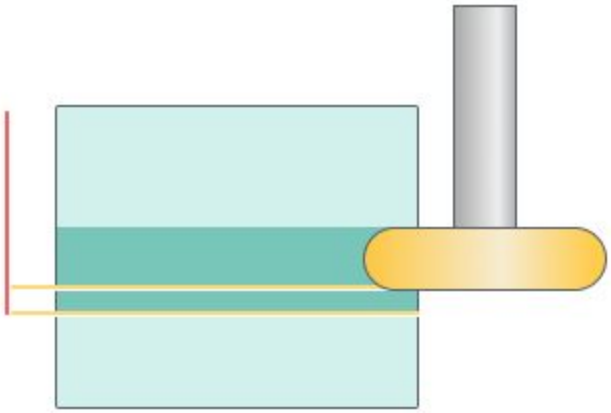
Z cleanup. The function adds an additional passage in front of the bottom level at a distance specified in the parameter.



T-slot level. Designed for undercut tools.

The first (top) pass is made at a distance of the tool height from the top level.



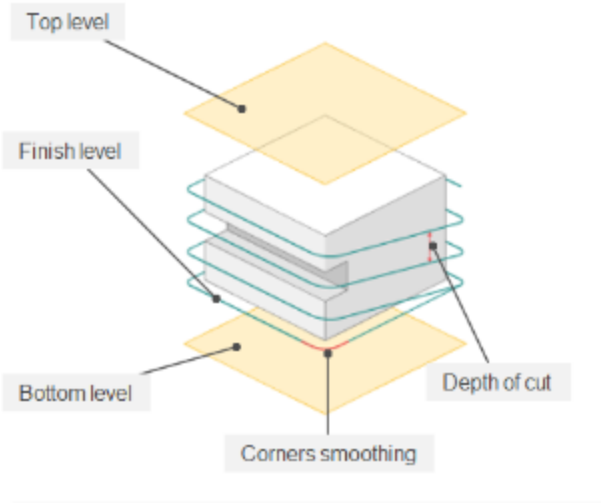


If the thickness of our tool is greater than the distance between the top and bottom levels, then the error will appear: Tool size is more than T-slot. When activating this option you need to check this condition.

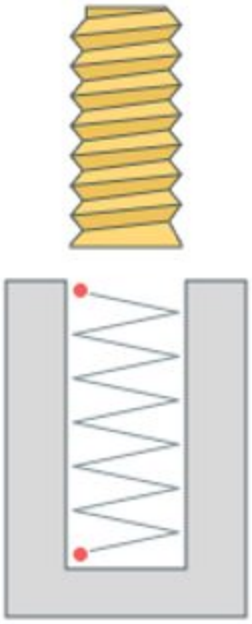
Helical machining.

Machining of a contour using a helical motion.

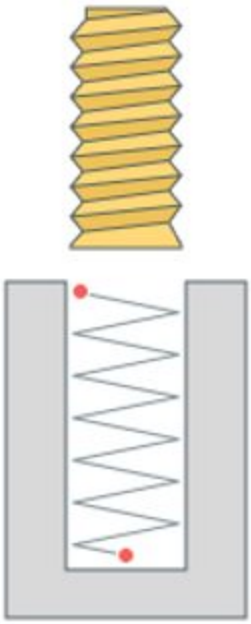
▼ Click here to expand..



Adjust step. If enabled: (by default) - adjusts the step to ensure the number of revolutions is an integer. If it's possible to round to a smaller or larger number of revolutions within the Max Step Deviation %, it adjusts the step for this specific number of revolutions. However, if the step would deviate beyond the acceptable deviation percentage, it completes the majority of revolutions with the initially set user step, and the first revolution is completed with a step that fulfills the remaining portion to complete a full revolution.



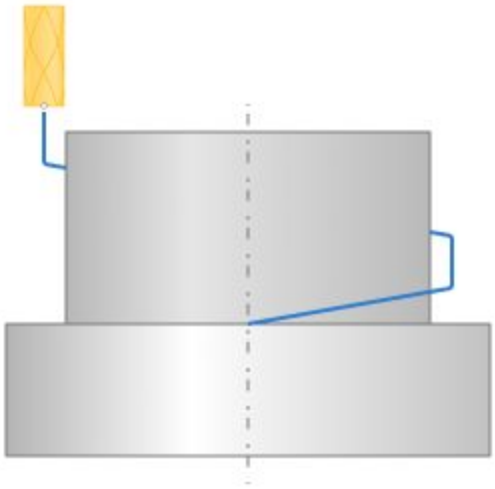
If disabled: maintains the step.

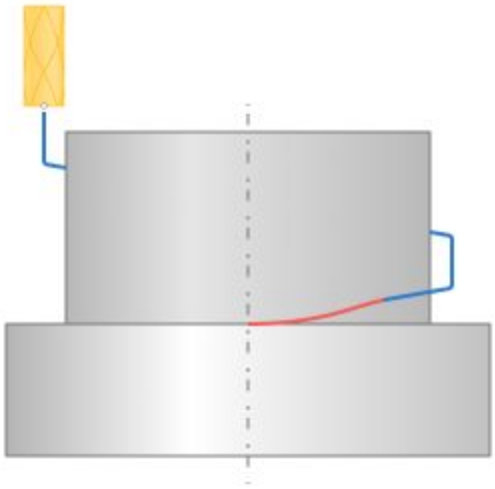


Max step deviation. The allowable deviation of the step when Adjust step is enabled. By default, it's set to 10%

Finish level. At the end of the helical machining, a final pass at full depth is added to remove any remaining 'ski-ramp'.

Smooth by radius. When choosing a Helical machining, a transition is created in the area where the slope of the spiral intersects with a flat area.

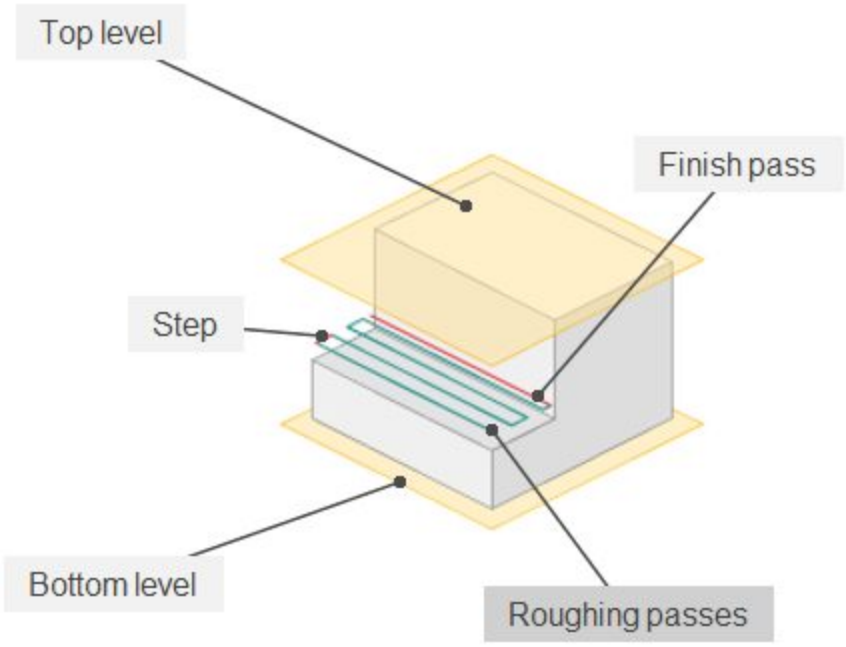




Roughining passes.

Toolpath is offset in plane by the 'Roughing passes' thickness amount. Additional passes are added to the contour by the value specified in the parameters with the specified number of steps.

▼ Click here to expand..

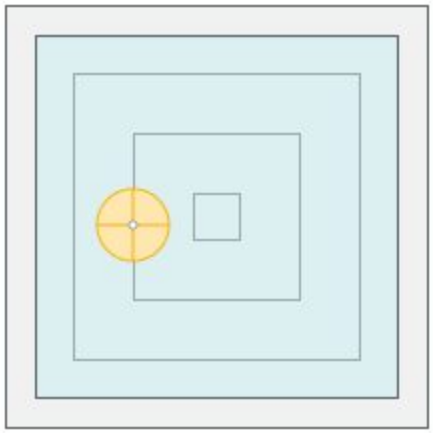


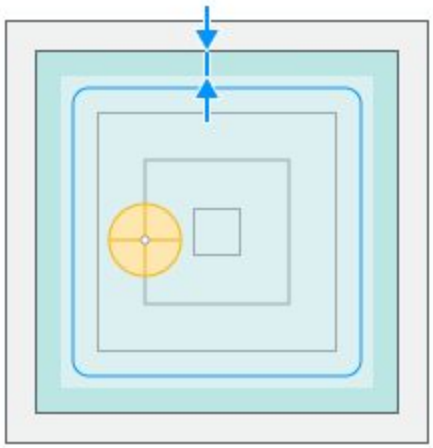
Step. Step between roughing passes.

Finish pass.

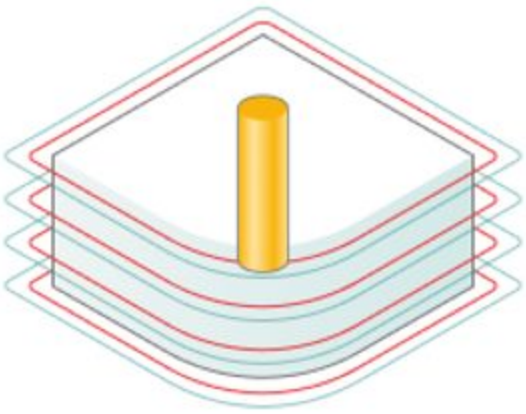
The contour specified in the Job assignment becomes finishing. An additional roughing pass is added to it at a distance specified in the parameter.

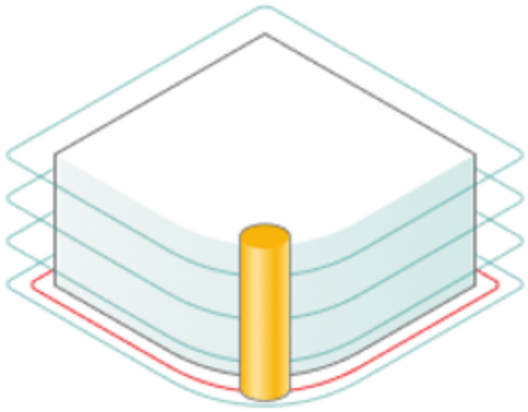
▼ Click here to expand..





At the bottom level only. The finish cut is only applied at final depth.





Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

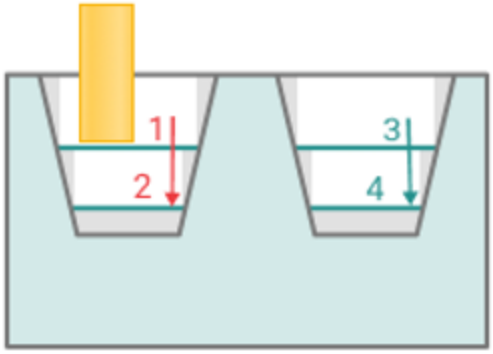
The parameters of this group are the same as in the 5D Surfacing operation. Details.

Sorting.

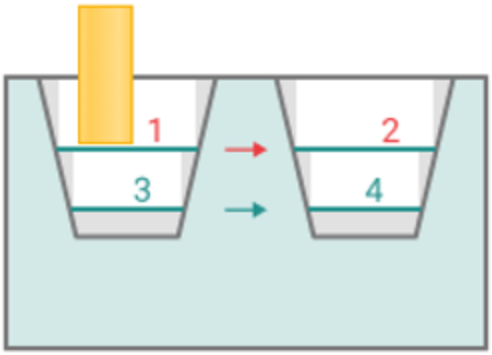
Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..

By cavities. All levels of the first profile are machined first. After that all levels of the next profile are machined.

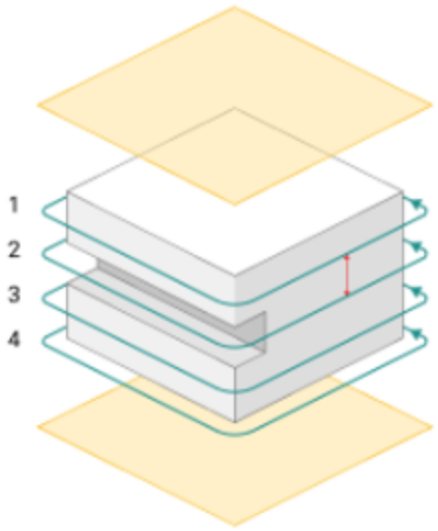


By layers. The first layer of all profiles is machined in the beginning. After that the next layer is machined.

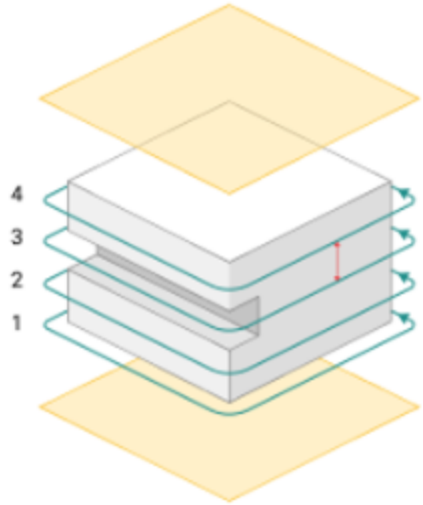


Machining direction. Determines where to start processing.

Downwards.



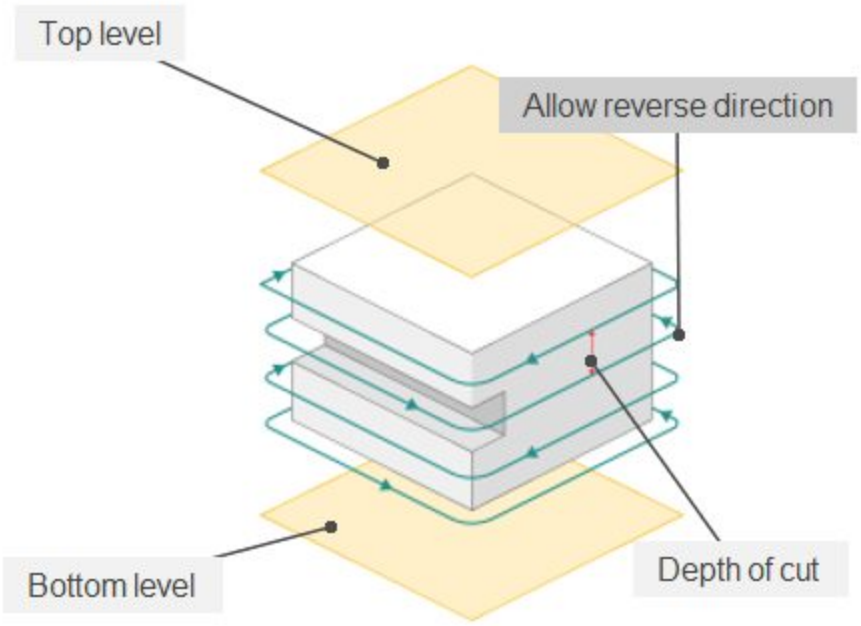
Upwards.



Idle moves minimization. The order of the profiles machining is sorted to minimize the idle motions.



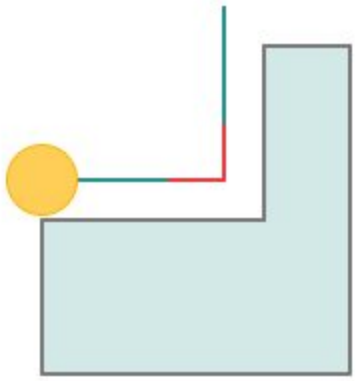
Allow reverse direction. The direction of machining (climb / conventional) alternates at each level.

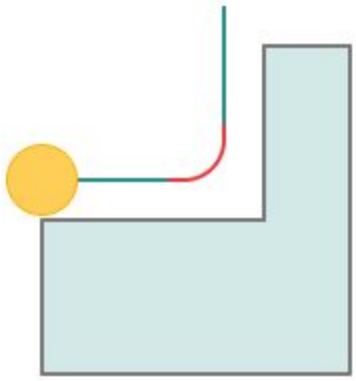


Corners smoothing.

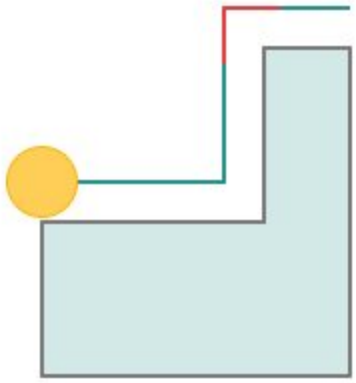
When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

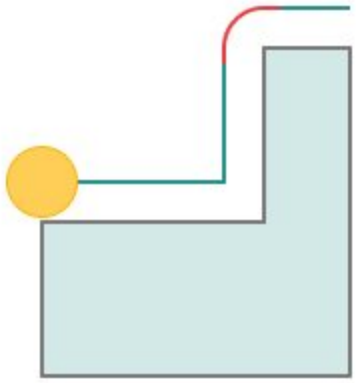
▼ Click here to expand..
Inner corners. Rolling the inner corner to the value entered in the parameter.





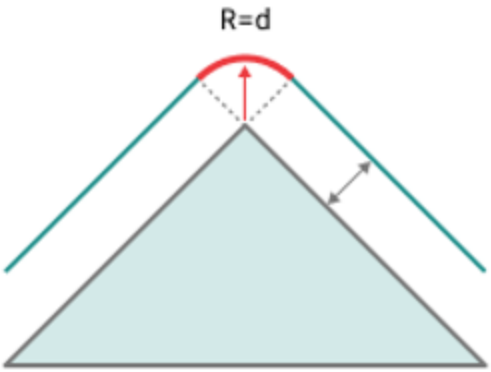
Outter corners. Rolling the outter corner to the value entered in the parameter.



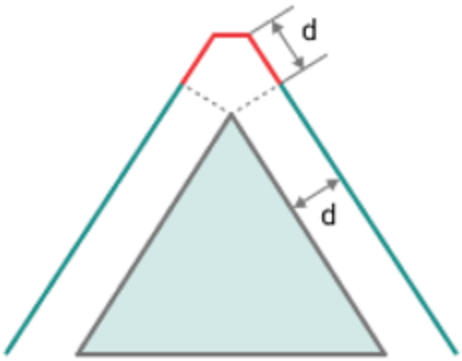


Corner roll type.

By arc. When rolling by arc, an arc whose radius is equal to the offset value will be inserted into the toolpath.



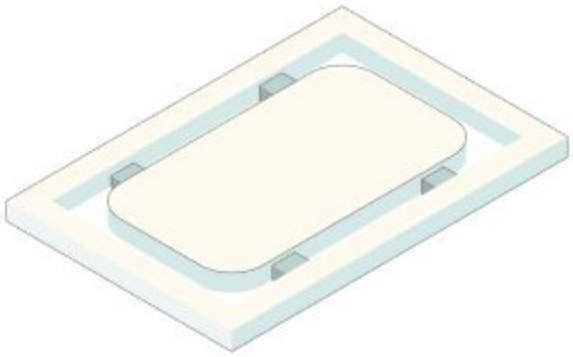
By tangent. In the tangent mode, linear moves are used to transit the corner. The tool path before and after the corner is extended by the offset length. If the extended path intersects then it is truncated to the intersection point, else a connection move is created.



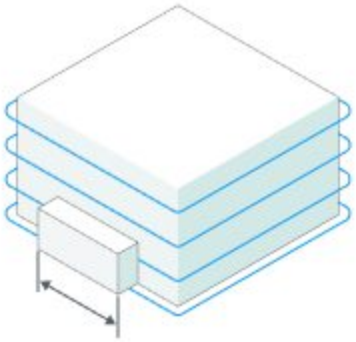
Tabs.

Tabs are needed to hold the part or fragments of the workpiece from falling out at the final stage of processing, which can lead to tool breakage or defective parts.

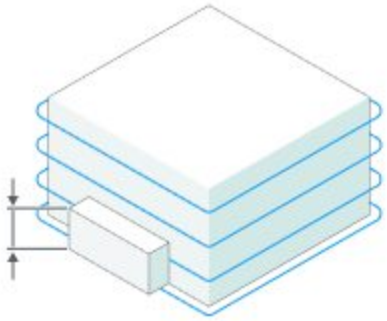
▼ Click here to expand..



Default tab size. Specifies the values for the default tab sizes.

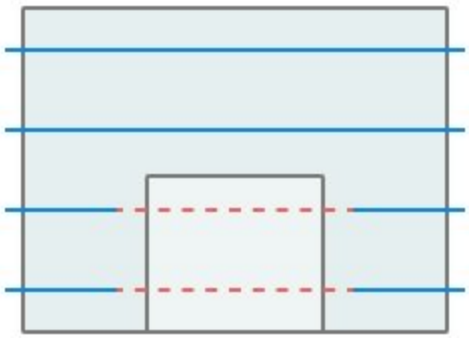


Default tab height. Specifies the values for the default tab sizes.

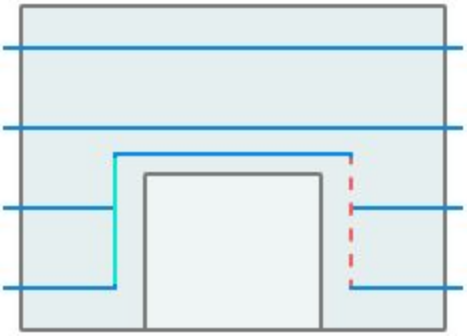


Go up. Method of climbing to the upper level of the tabs.

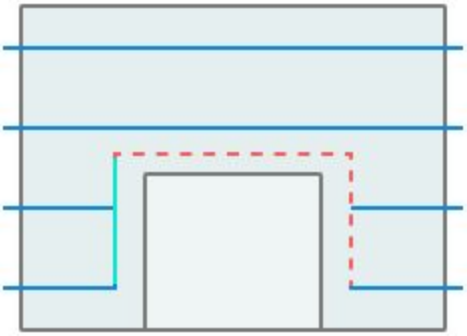
Off. There is no lift, rapid is activated. We go to the end of the tap right at the current level.

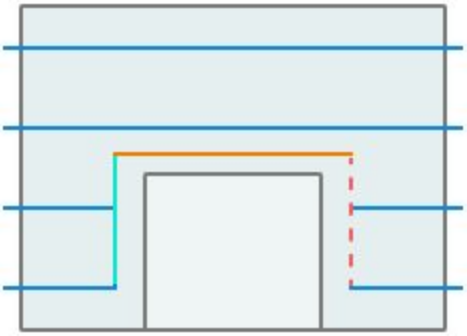


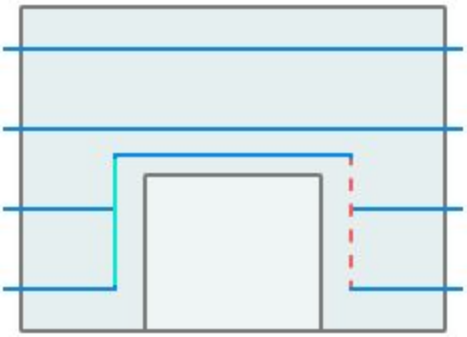
Rapid vertical. We lift at an accelerated speed to the height of the Go above the bridge.



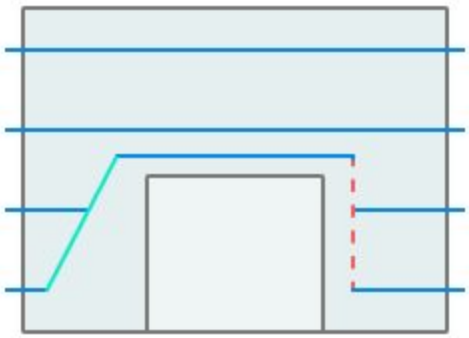
Go above. Method of horizontal movement over the taps. Can be rapid, on transition feed, on work feed.





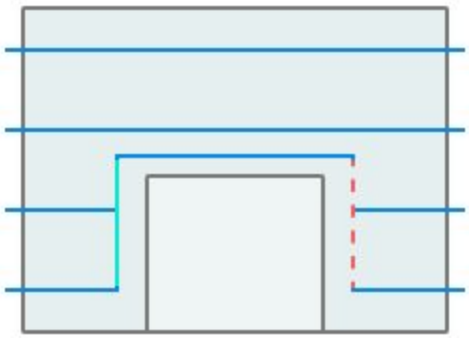


Go down. Drop method on Plunge feed.

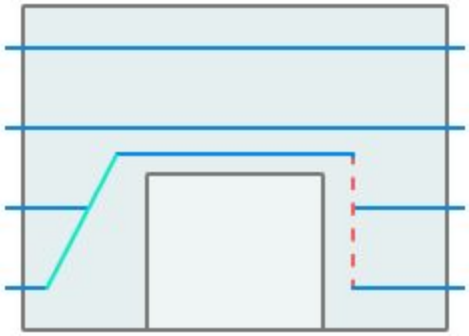


▼ Click here to expand..

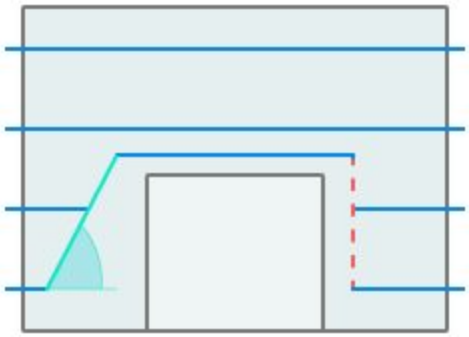
Vertical plunge. Plunge feed runs on vertical plunge.



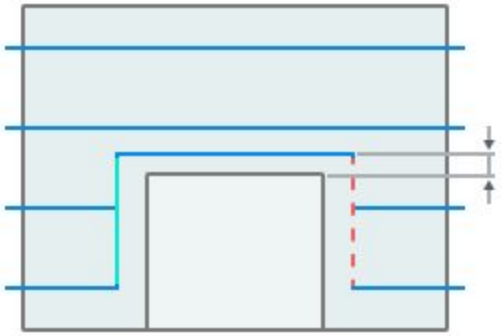
Plunge by angle. Plunge feed runs on plunge angle.



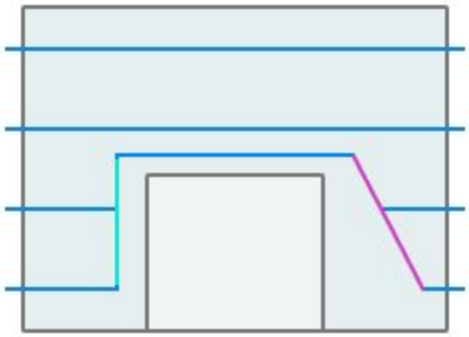
Plunge angle. Plunge feed runs on plunge angle.



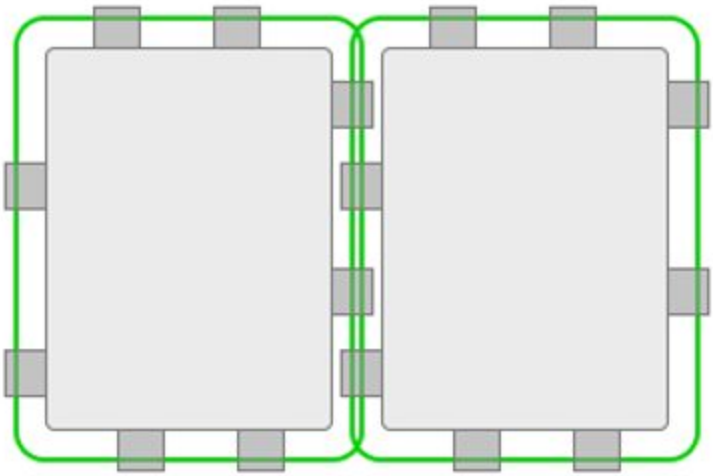
Gap above tab. The height of the gap from Tap height to the level at which the bridge bypass path will be located.

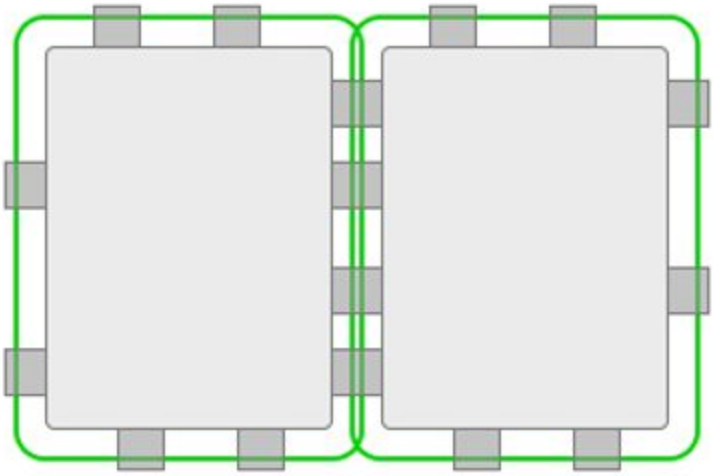


Return by angle. Lift on the Return feed at a given angle



Merge tabs of neighbor curves. This option allows you to take into account tabs from the current contour and the nearest ones.



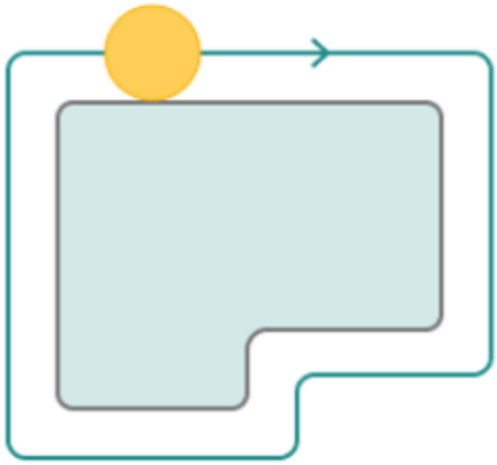


Output.

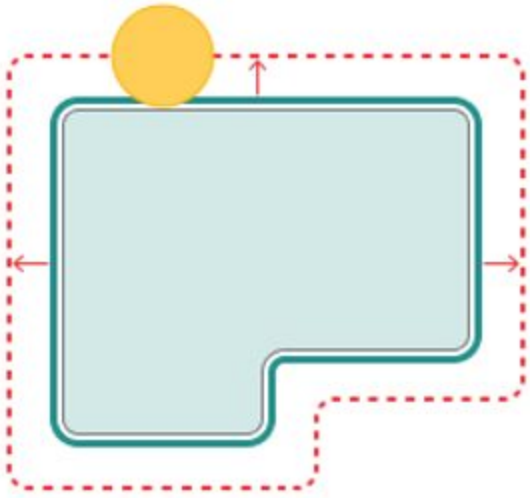
▼ Click here to expand..

Radius compensarion use.

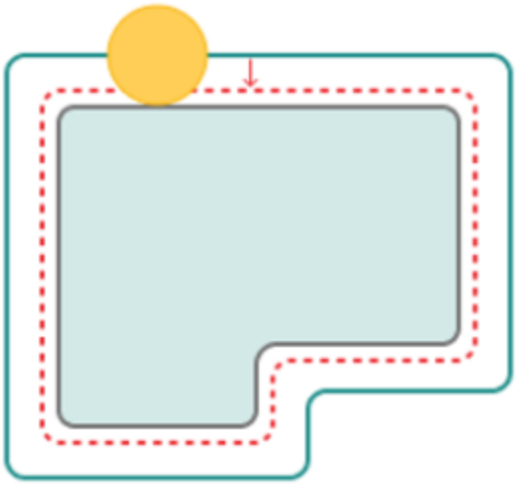
Computer. A toolpath offset by the cutter radius is calculated. No Cutter Radius Compensation (CRC) commands are included in the toolpath.



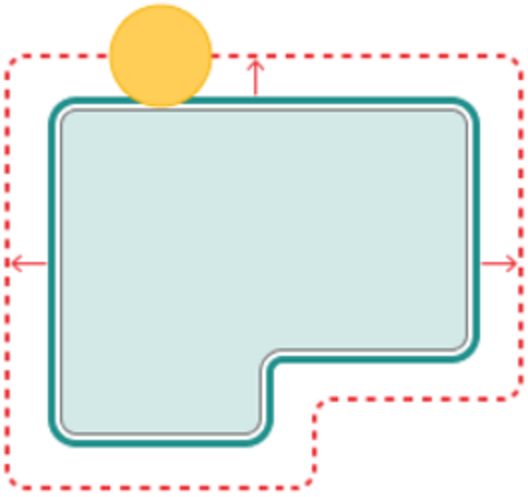
Control. A centreline toolpath is calculated. Cutter Radius Compensation (CRC) commands (e.g. G40 / G41 / G42) are included in the toolpath. In this case CAM system cannot check for cutter interference etc. so error messages can occur at the CNC controller if there is cutter interference e.g. tool radius larger than part radius.



Wear. A toolpath offset by the cutter radius is calculated. Cutter Radius Compensation (CRC) commands (G40 / G41 / G42) are also included in the toolpath. The cutter is moved closer to the part by the 'wear' value entered into the CNC controller. The 'wear' amount is the difference between the 'nominal' and the 'undersize' cutter. In the CAM system, the trajectory does not change visually.



Reverse wear. A toolpath offset by the cutter radius is calculated. Cutter Radius Compensation (CRC) commands (G40 / G41 / G42) are also included in the toolpath. The cutter is moved away from the part by the 'wear' value entered into the CNC controller. The 'wear' amount is the difference between the 'nominal' and the 'oversize' cutter. In the CAM system, the trajectory does not change visually.



Compensation radius. This value is used by the simulator for simulating the offset toolpath. This value has no effect on the calculated toolpath.

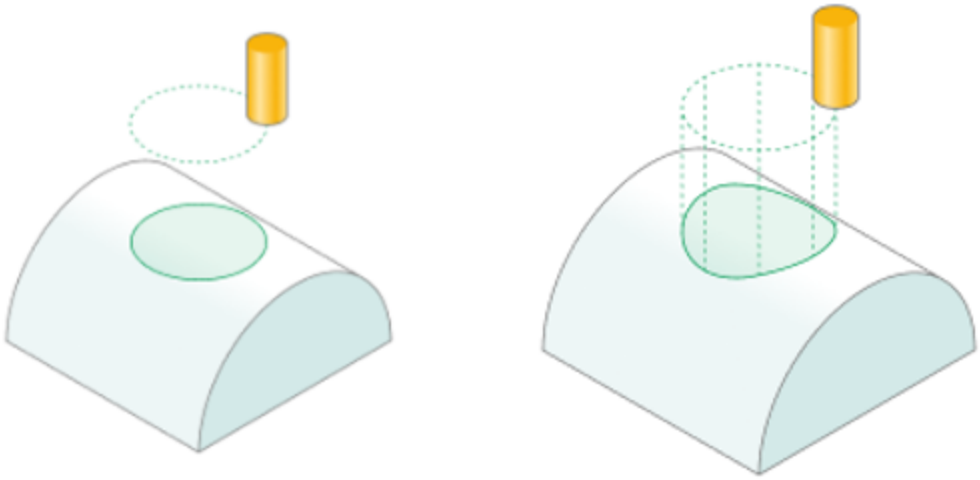
Transformations.

A dialogue for transforming the toolpath.

▼ Click here to expand..

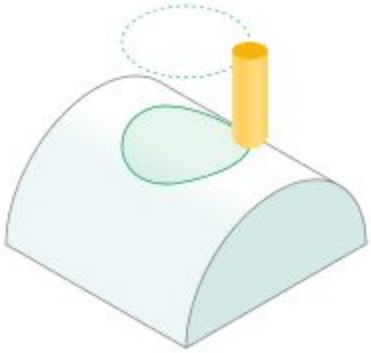
Project toolpaht onto the part. If enabled: The toolpath is projected onto the part model. The 'Top level' and 'Bottom level' values are incremental. If the 'Bottom level' value is set to 0, the tool will follow the 3D shape of the part. If the 'Bottom level' value is set to -1, the tool will cut into the 3D part shape at a depth of 1.

If disabled: The toolpath is not projected onto the part. The (absolute) 'Top Level' and 'Bottom Level' values are used.

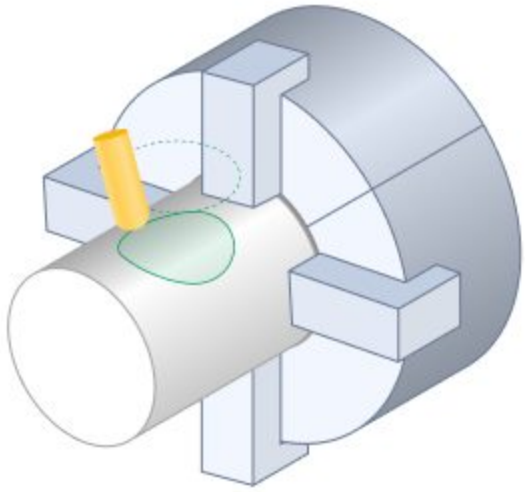


Tool axis orientation. Various methods of tool orientation.

Fixed. Processing is carried out along 3 axes.



Normal to surface 4D. Processing is carried out along 4 axes. The part axis must be located along the global X axis. Work with Project toolpaht onto the part.



Normal to surface 5D. The machining process can be carried out using 4 or 5 axes. Work with Project toolpaht onto the part.



Normal to base surface. Processing is carried out along 3 axes. You need to specify the base surface on the Job assignment tab. Differs from Fixed in that the 3rd axis can be rotary.



By Default. Equals Normal to base surface.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details

Distances. Distances used in Links rules. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Basic

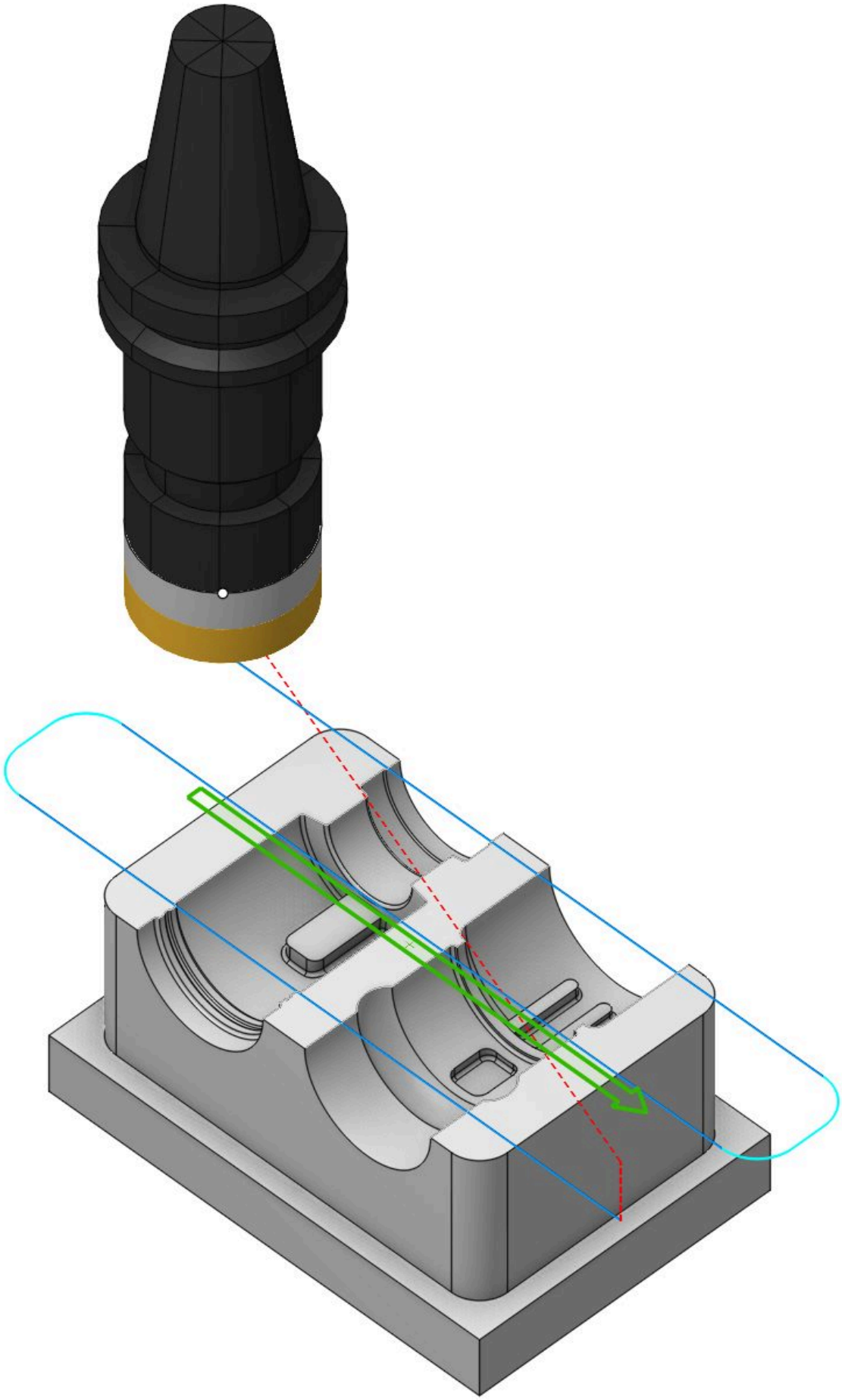
Application area:

The Basic group contains face milling, waterline and plane machining, contouring, and flat-area finishing operations. These topics describe the common milling strategies and their operation settings.

Operations:

- Face milling. Machines planar faces. Details
- Roughing waterline. Removes stock in horizontal layers. Details
- Finishing waterline. Finishes surfaces using waterline passes. Details
- Roughing plane. Removes stock using plane passes. Details
- 2.5D contouring. Machines contours at different levels. Details
- Flat land. Finishes flat areas of the model. Details
- 3D contouring. Machines spatial curves. Details

Face milling



Application area:

A facing operation removes workpiece material on a specified horizontal plane using one of the following strategies:

- One pass
- Parallel,
- Zigzag,
- Optimized zigzag,
- Spiral.

The geometry of the part is ignored, only the shape of the workpiece is taken into account during the operation calculation process. The part is not controlled for undercuts.

Material can be removed in multiple passes. In this case, the lower processing level is determined by default by the highest point of the part, and the upper level is determined by the highest point of the workpiece. The number of passes is calculated automatically depending on the distance between the upper and lower processing levels, the size of the finishing and the depth of the roughing passes. It is also possible to set the number of passes manually. In this case, the processing step will be recalculated.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

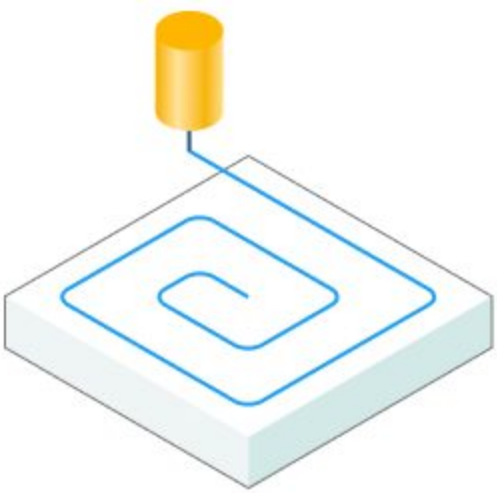
Strategy:

Strategy.

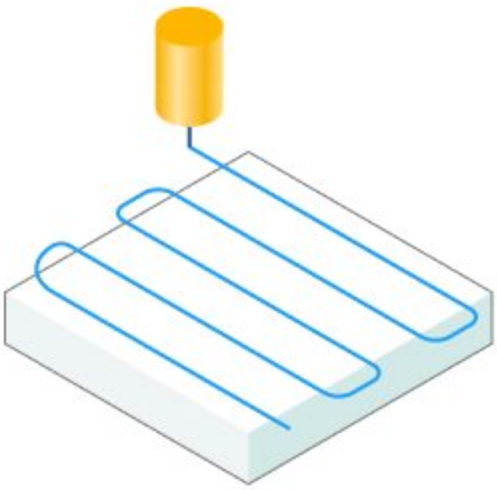
This group allows the user to modify the primary parameters of the parallel passes strategy.

▼ Click here to expand..
Type of Machining Strategy:

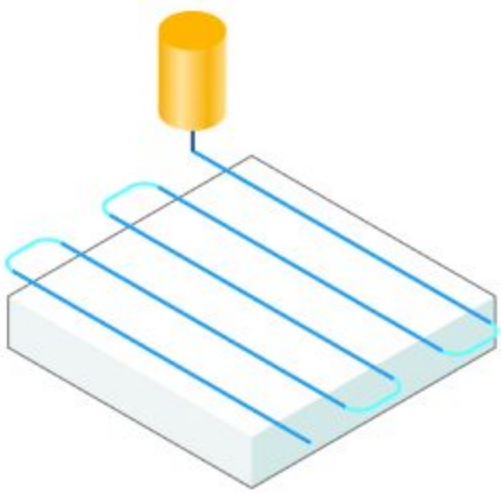
▼ Click here to expand..
Spiral. The tool rolls into the workpiece using the so-called roll-in technique, then moves along the boundary of uncut material removing it like a round lawn mower. The maximum allowed width of cut and the tool engagement angle are never exceeded. The strategy is very good for removing large volumes of material in minimal time.



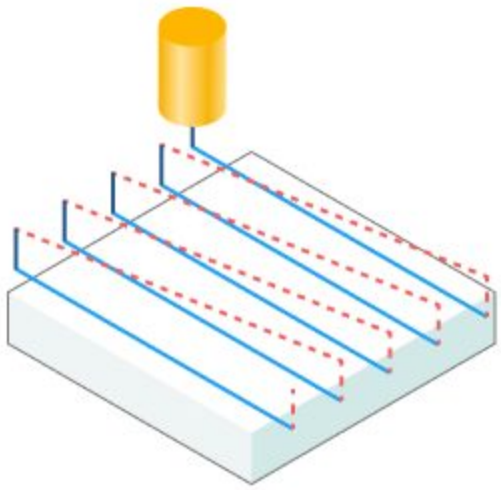
Optimized zigzag. The tool moves along zigzag trajectory. The link moves between zig- and zag- passes are also made in-material on the work feed to reduce machining time. The strategy is good for roughing and semi-finishing.



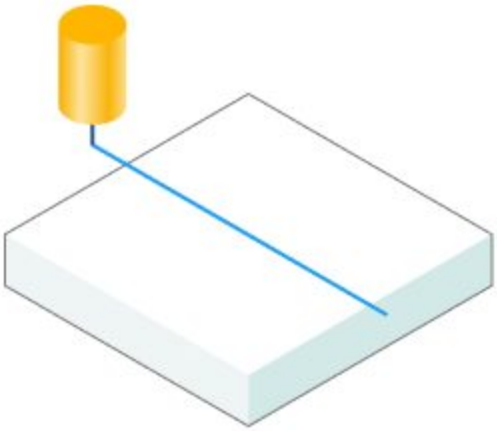
Zigzag. The tool moves along zigzag trajectory. The link moves between zig- and zag- passes are also made in-material on the work feed to reduce machining time. The strategy is good for roughing and semi-finishing.



One way. The tool moves in parallel passes always in one direction depending on the milling type (climb/conventional). A work pass starts and ends outside the workpiece boundary. The link moves are made on the safe plane. With this strategy the best possible surface finish can be achieved.

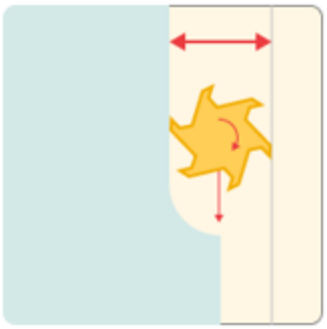


One pass. The tool makes exactly one pass in the center of the workpiece. The work pass starts and ends outside the workpiece boundary. Use this strategy when the diameter of the tool is bigger than the width of the workpiece.

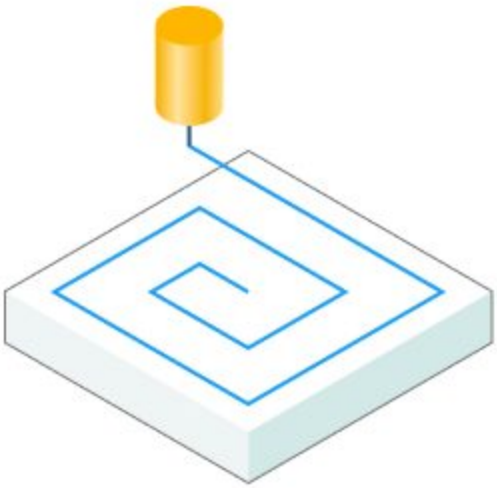


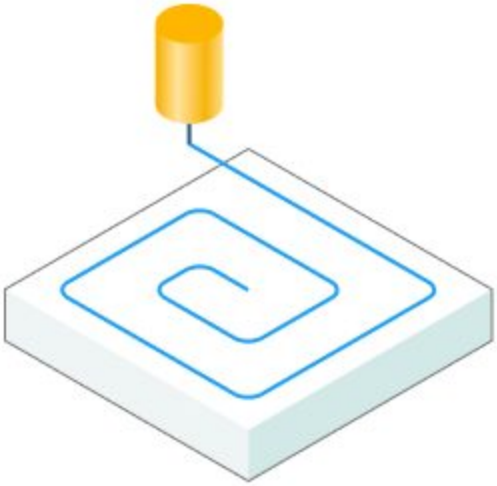
Here are the key parameters, the set of which may vary depending on the chosen strategy in the strategy section:

Step. The maximum allowed width of cut.

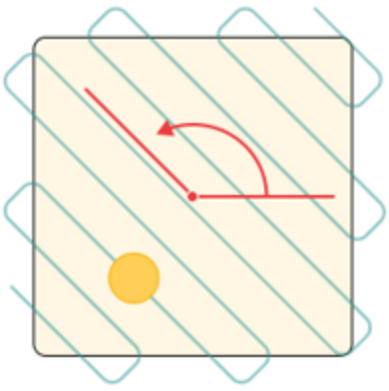


Smooth corners. Sharp corners in the toolpath are smoothed using the given radius.

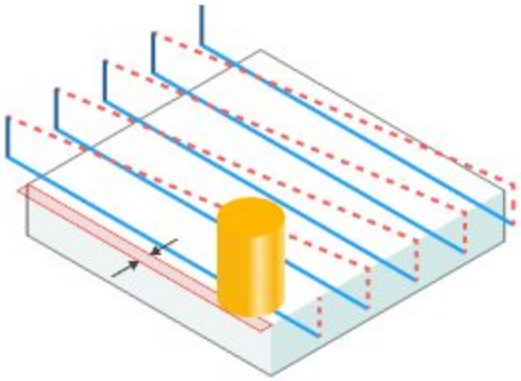




Angle. Assigns the cut angle direction for the plane operations.



Finall pass overlap. How much the periphery of the tool overlaps the workpiece at the final pass.



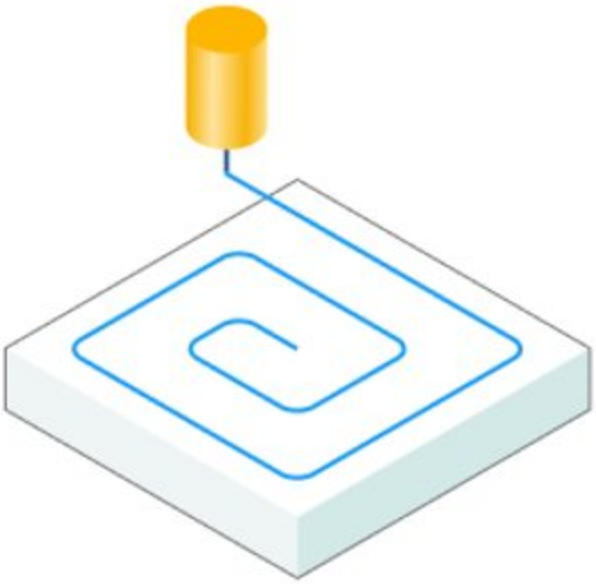
Milling type

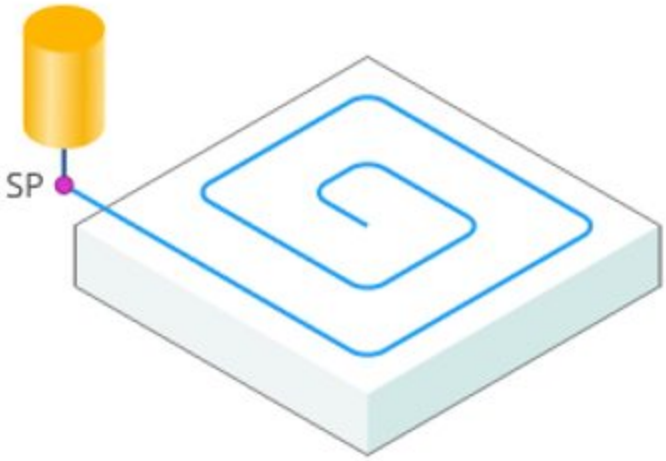
Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting.

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Start Point. The point defining the beginning of the plunge.

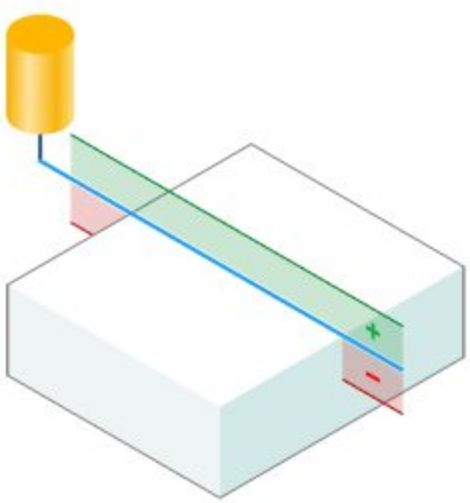




Machining levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

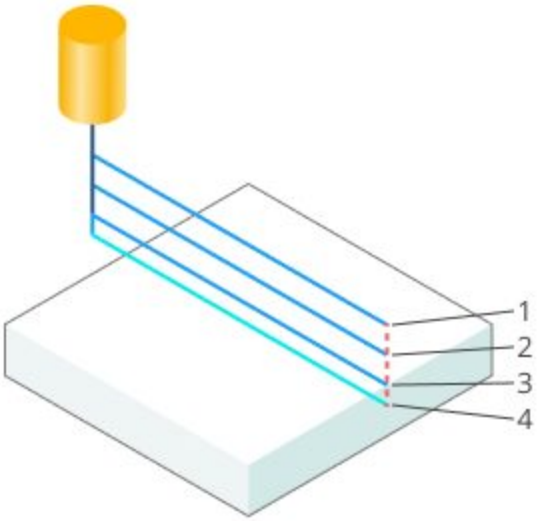
▼ Click here to expand..
Axial stock. Stock to leave on the final level. Use positive values to leave excess material for subseqent machining or negative values if you want to remove material from the Part (gouge).



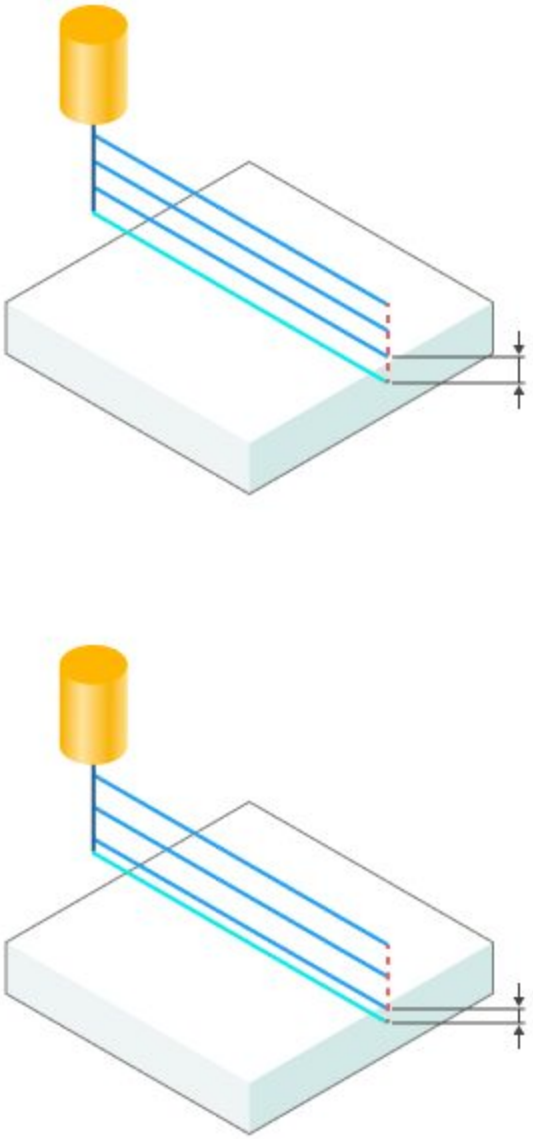
Nb. of passes

The number of machining passes. By default this is computed automatically based upon the top and the final levels for machining, the Cleanup height and the Depth Step.

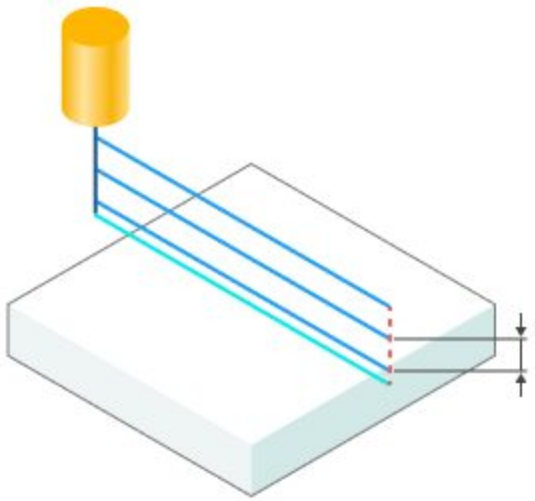
▼ Click here to expand..



Cleanup height. The depth of the final pass can be controlled. Using a smaller final pass amount usually enables a better surface finish.



Depth step. The maximum allowed stepover for the roughing passes.



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

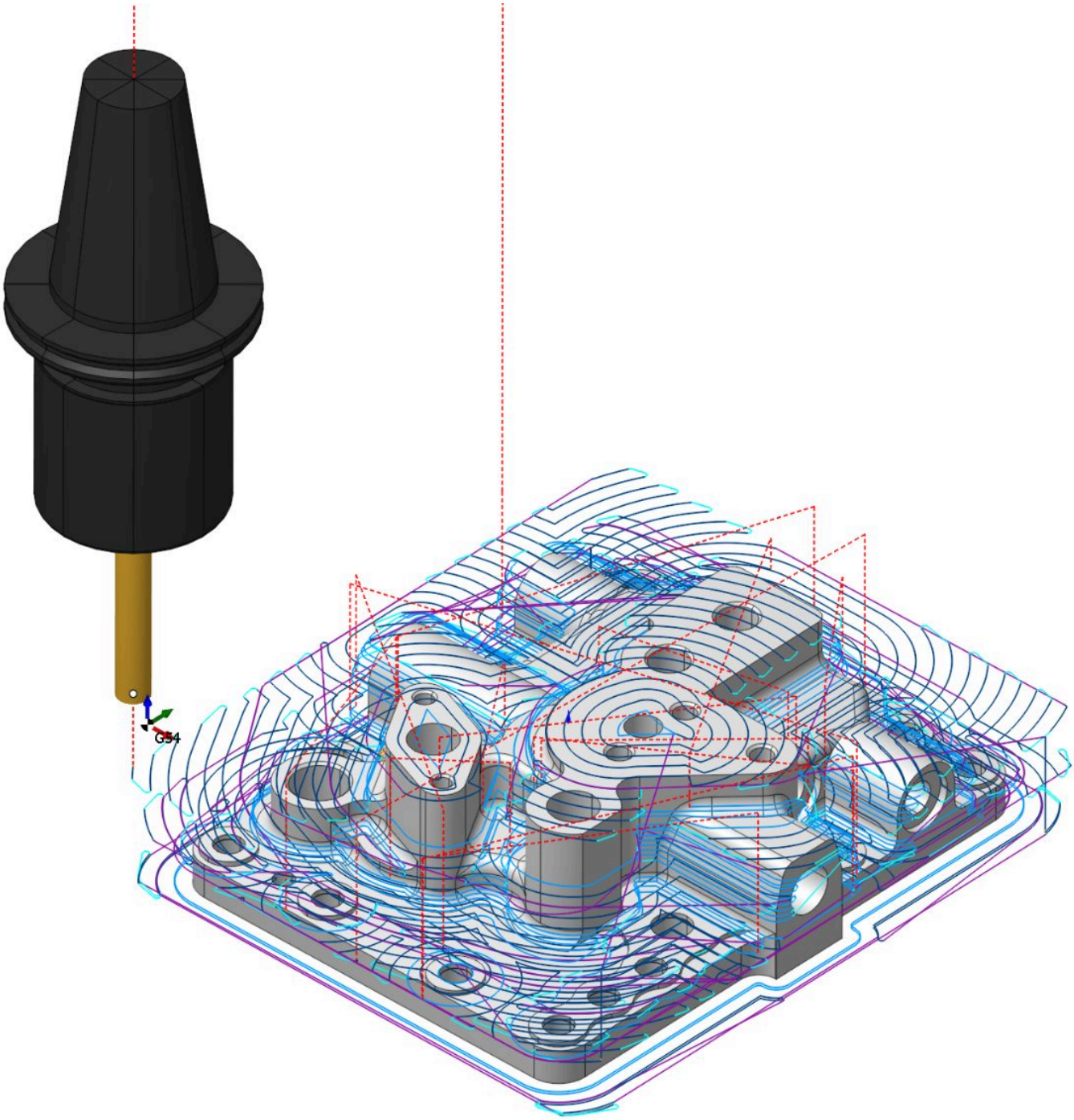
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Roughing waterline



Application area:

The waterline roughing operation is used for the initial rough machining of complex-shaped models that differ significantly from the original workpiece. A model being machined by the waterline roughing operation is assigned by a set of solid bodies, surfaces and mesh objects. For every geometrical object or a group of objects, an additional stock, which during machining will be added to the main stock of the operation, can be defined.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

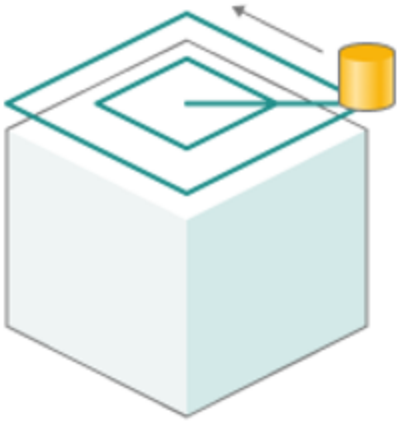
Strategy:

Machining strategy:

This parameter allows the user to achieve a toolpath that appears parallel or equidistant or other when performing pocketing operations.

▼ Click here to expand..
Type of Machining Strategy:

▼ Click here to expand..
Equidistant. Processing along curves obtained by offsetting the part contour. Details



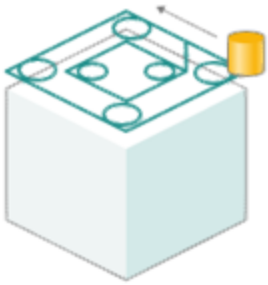
Parallel. Processing with parallel passes. Details



Adaptive. The strategy is used to effectively remove large volumes of material with high feedrates, maximal cutting depths (up to the flutes length) and relatively shallow cutting widths (5% to 30% of the tool diameter). Details



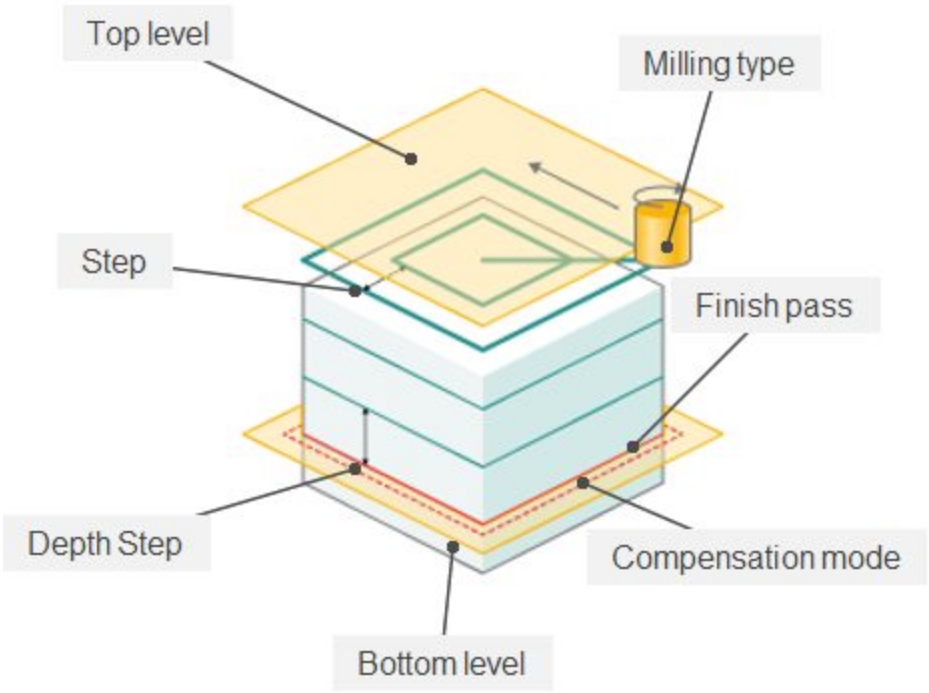
HPC. The high performance cutting strategy is designed for the efficient removing of material in the open and closed pockets. Details



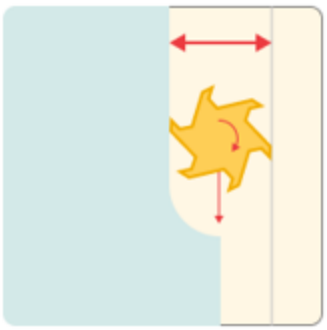
Deep HPC. Equidistant machining with additional looping motions at corners. Details



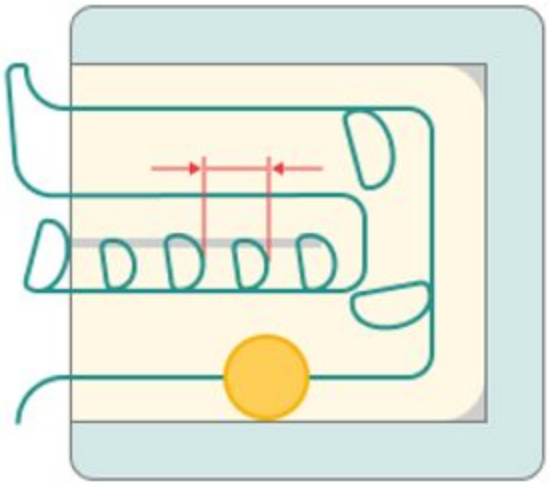
You also need to read about how to choose the pocketing strategy. Details
Here are the key parameters, the set of which may vary depending on the chosen strategy in the Machining strategy section:



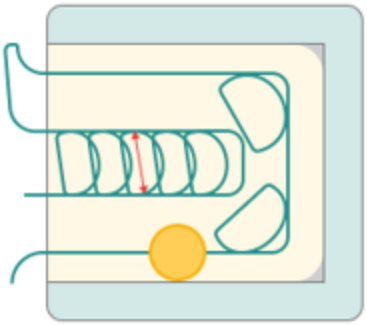
Step. The maximum allowed width of cut.



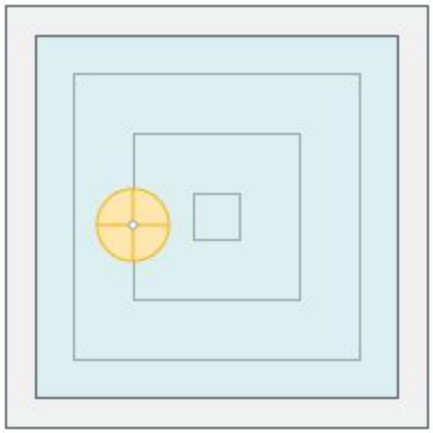
HSC step. Define the step of the trochoidal arcs as either; a percentage of the current tool diamater or, as a set distance. If the high speed cuts are disabled then unmachined areas are possible.

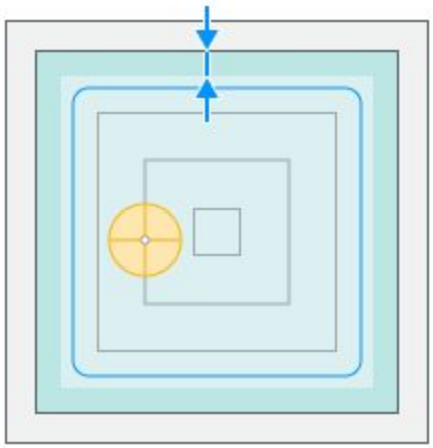


Big arcs. Expands arcs between the passes of the trajectory.



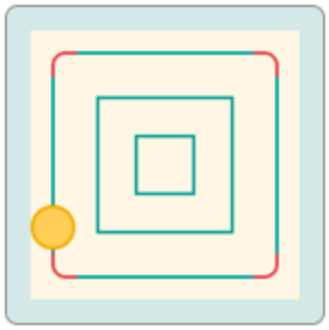
Finish pass. For some strategies, a finishing pass is added. Before the finishing pass, an allowance equal to the value specified in the parameter is added.



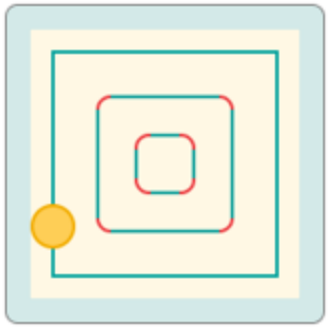


Compensation mode. Similar to the Output group in 2D contouring. Details

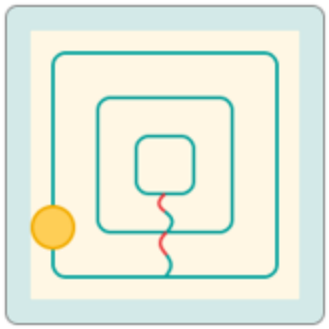
Finish rounding radius. The parameter is necessary for rounding corners during the finishing pass.



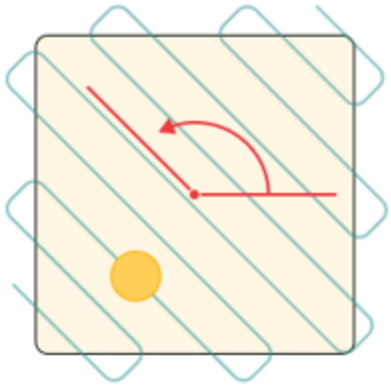
Rough rounding radius. The parameter is necessary for rounding corners during the roughing pass.



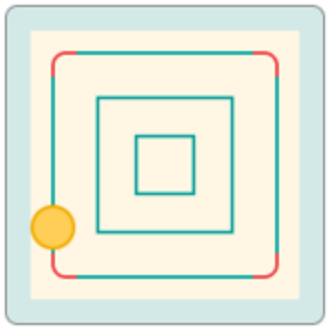
Linking radius. Linking radius for the 'links' and also the transitions for the trohoidal arcs.



Angle. Assigns the cut angle direction for the plane operations.



Corners smoothing. Virtually, it is used in most operations and provides toolpath 'rounding' when machining inner corners. This reduces vibrations and increases machining speed. Rough passes inside the contour do not have rounding.

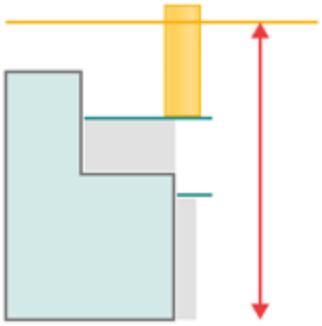


Machining levels:

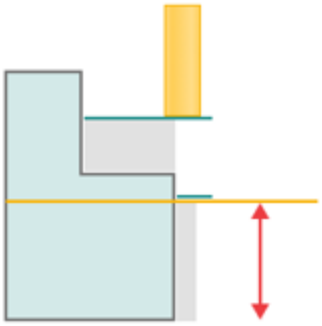
It defines the range along tool axis for the machining.

▼ Click here to expand..

Top level. The level at which the final pass is made. By default it is the top most level of the machined part.



Bottom level. Drag the plane with the mouse or drag the plane centre circle and use 'Smart snap' to align with a feature or, click on the dimension in the graph view to set the desired bottom level for machining. If the text in the field is green in colour, the level is calculated automatically as the top of the part, wokpiece and fixtures. To return to the automatic mode, simply clear the field and press 'Enter'.

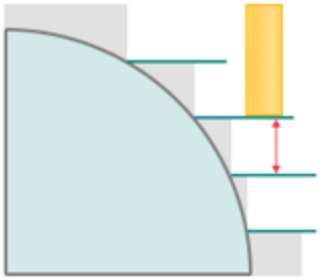


Depth Step:

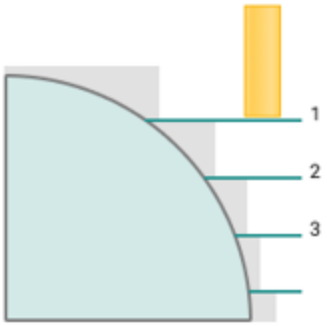
Material removed in one pass along the tool axis between the Top and Bottom levels.

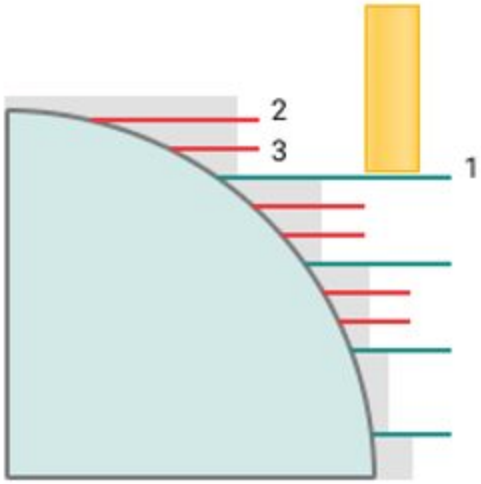
Here are the key parameters, the set of which may vary depending on the chosen strategy in the Machining strategy section:

▼ Click here to expand..

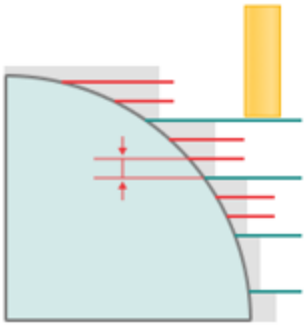


Step up. Machining is performed strictly from top to bottom.

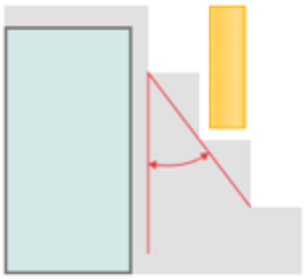




Step up value. Distance from the already machined layer to the overlying one.



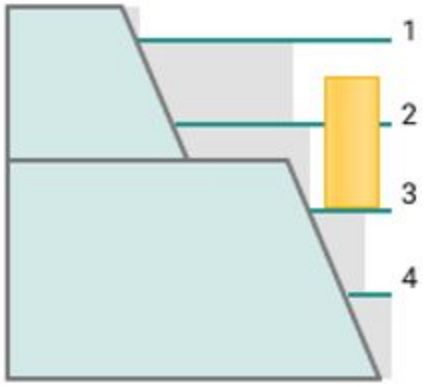
Relief angle. Inputting a positive value avoids the tool shank 'rubbing' as the machining depth increases.
Details

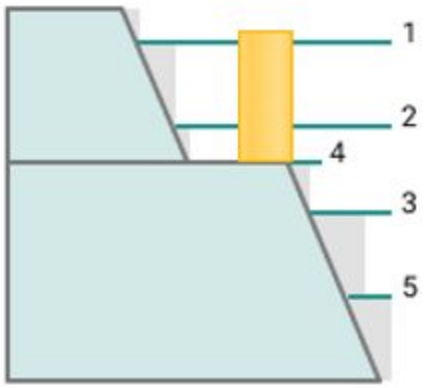


Clear flats.

Extra passes are applied for flat faces.

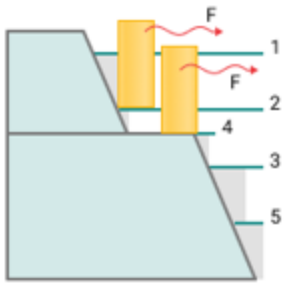
▼ Click here to expand..



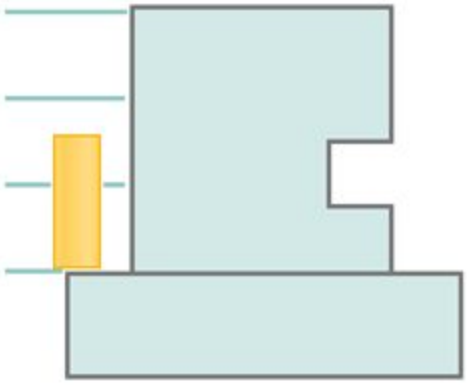


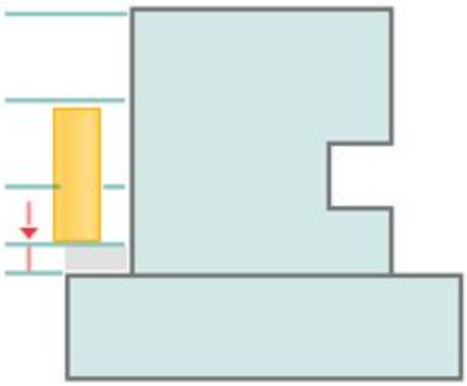
Here are the key parameters, the set of which may vary depending on the chosen strategy in the Machining strategy section:

Flats on finish feed. Outputs the Clear flats path on the finishing feed. Doesn't work with Cleanup height.



Cleanup height. The function adds a new pass at the distance specified in the parameter from the Clear flats pass, when the Clear flats option is enabled.





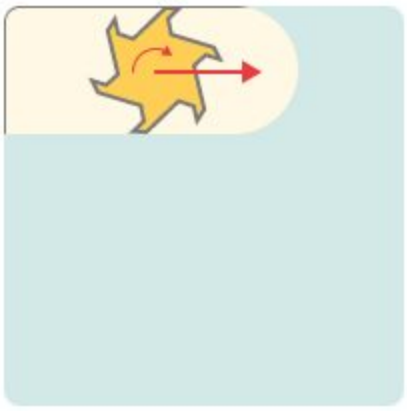
Milling type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

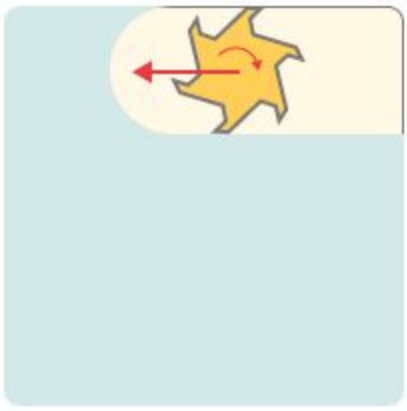
▼ Click here to expand..

Climb. In machining refers to a cutting strategy where the tool moves in the same direction as the rotation of the workpiece. It typically results in less tool wear and a smoother finish compared to

"Conventional" cutting, where the tool moves against the rotation of the workpiece. Climb cutting is often preferred when precision and surface finish are critical.

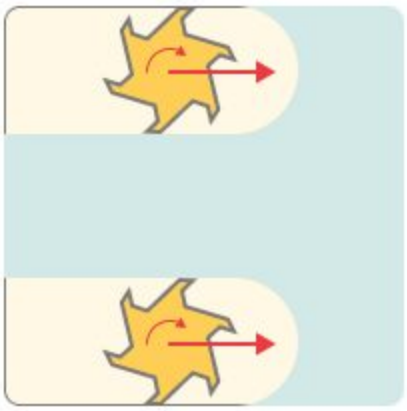


Conventional. Cutting in machining refers to a strategy where the tool moves against the rotation of the workpiece. This cutting method is used in machining operations, but it can result in more tool wear and a rougher finish compared to "Climb" cutting, where the tool moves in the same direction as the workpiece rotation. Conventional cutting is sometimes chosen when other factors, such as tool life or chip evacuation, are more important than surface finish.



Both. Refers to the practice of using both "Climb" and "Conventional" cutting strategies during a machining operation. This approach can be employed to optimize tool performance, surface finish, and other machining factors depending on the specific requirements of different parts of the workpiece or

different stages of the machining process. It allows for greater flexibility and control in achieving desired results in machining.

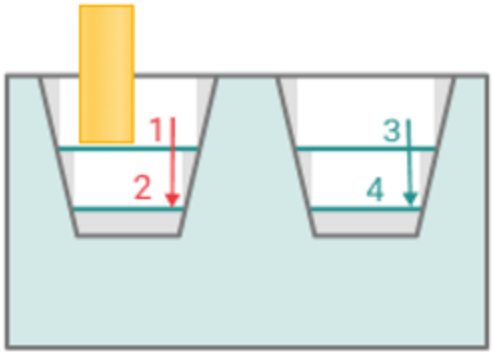


Sorting.

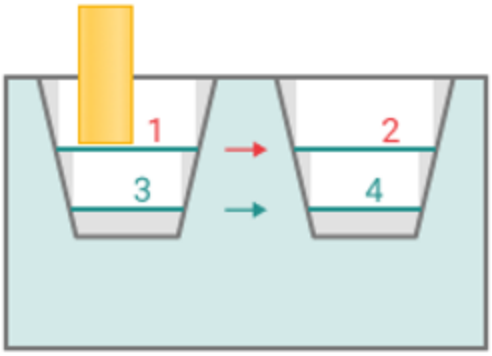
Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Here are the key parameters, the set of which may vary depending on the chosen strategy in the Machining strategy section:

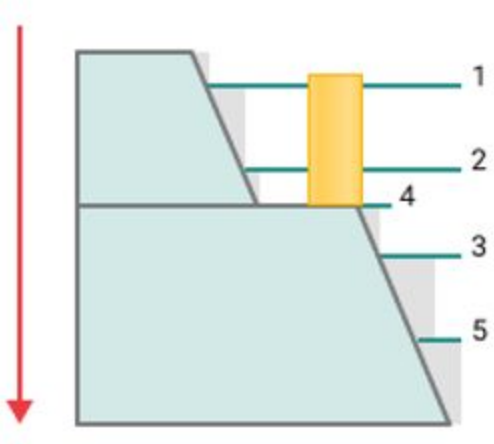
By cavities. All levels of the first profile are machined first. After that all levels of the next profile are machined.



By layers. The first layer of all profiles is machined in the beginning. After that the next layer is machined.



Downwards Only. Sort the machining to be strictly from the top to bottom. This can create a non-optimal toolpath. Enable this option only if you encounter errors with the level sorting. It can sometimes eliminate the problems.



Machining Order. Defines the machining sequence in the curve machining operations.

Direct. Sort the passes in such a way that the intersection begins on the left side of the arrow (parameter Angle), when looking in the direction of the arrow. It works only with the Parallel strategy.

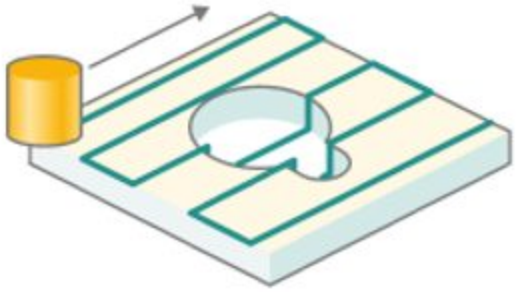
Inverted. Sort the passes in such a way that the intersection begins on the right side of the arrow (parameter Angle), when looking in the direction of the arrow. It works only with the Parallel strategy.

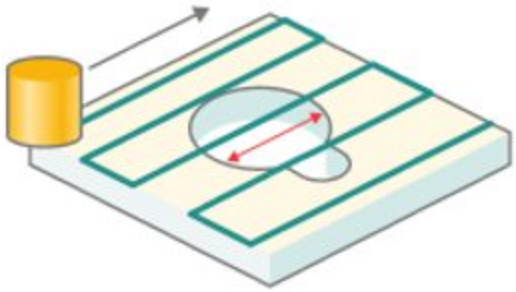
Optimal. Sort the passes in such a way that the intersection begins at the shortest distance (which reduces the number of transitions between passes). This sorting method is applicable only with the Parallel strategy.

Trimming

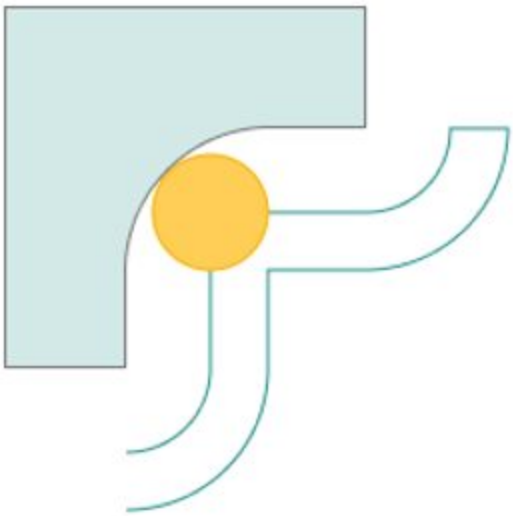
Allows for the skipping of surfaces not intended for machining and allows for the extension or trimming of the toolpath.

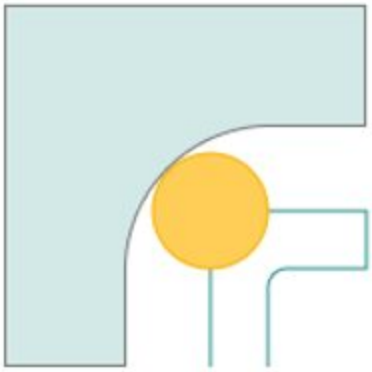
▼ Click here to expand..
Hole capping. Quite often a geometrical model has holes the machining of which has been performed earlier or are required to be machined in a later operation. In this case, when machining the surface that has the hole, it can be ignored. The ignored holes can be of any shape (not always round). The defined size describes the diameter of a disk that can cover the hole.





Trim waste rolls. The cutter 'rolls' around the sharp outer corners of a model. The tool path is smooth but not always optimal.





Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the

final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Distances. Distances required to switch feeds Details.

Allow plunges. Enabling plunges allows machining of closed pockets. When CAM software encounters a closed pocket that can not be machined from the outside, it forms a Plunge move. Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

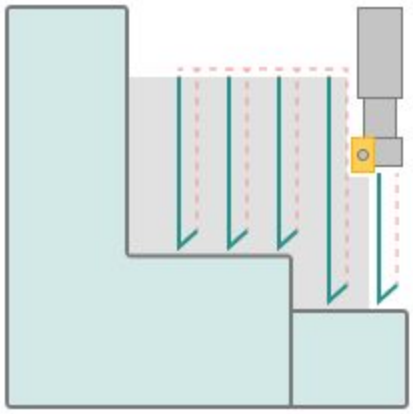
Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Holes (Drill points):

Drill points, including their coordinates, depth and tool diameter, are stored in an editable list and used for tool entry when the machining area cannot be approached from outside. Details

Plunge roughing

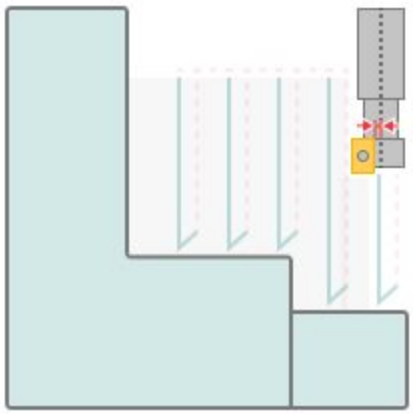


Application area:

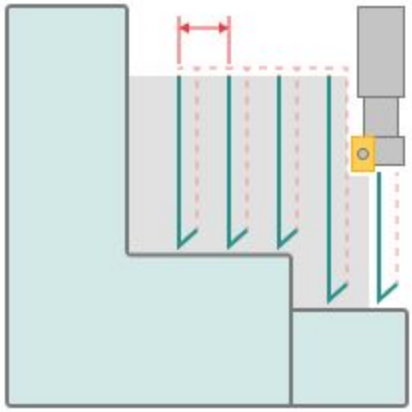
This option is available on the Parameters tab in the Roughing waterline operation. Details

Plunge roughing option generates vertical passes from Roughing Waterline, Plane and Pocketing operations result. This option requires a tool that is designed to handle axial cutting.

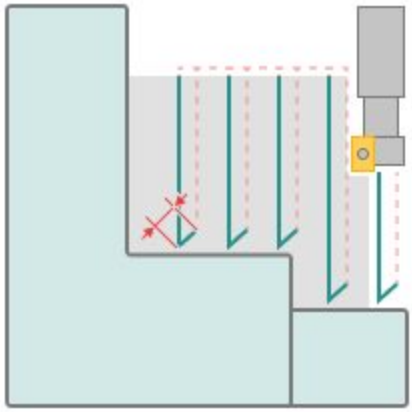
Core radius. Size of the central non-cutting part of the tool.



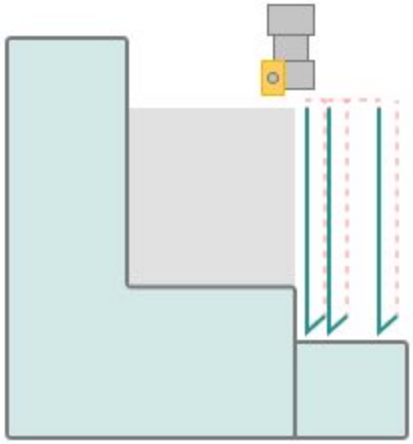
Step. Step between passes.



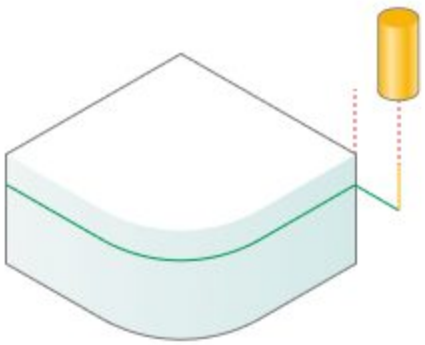
Pull back distance. Distance the tool pulls back away from the model.



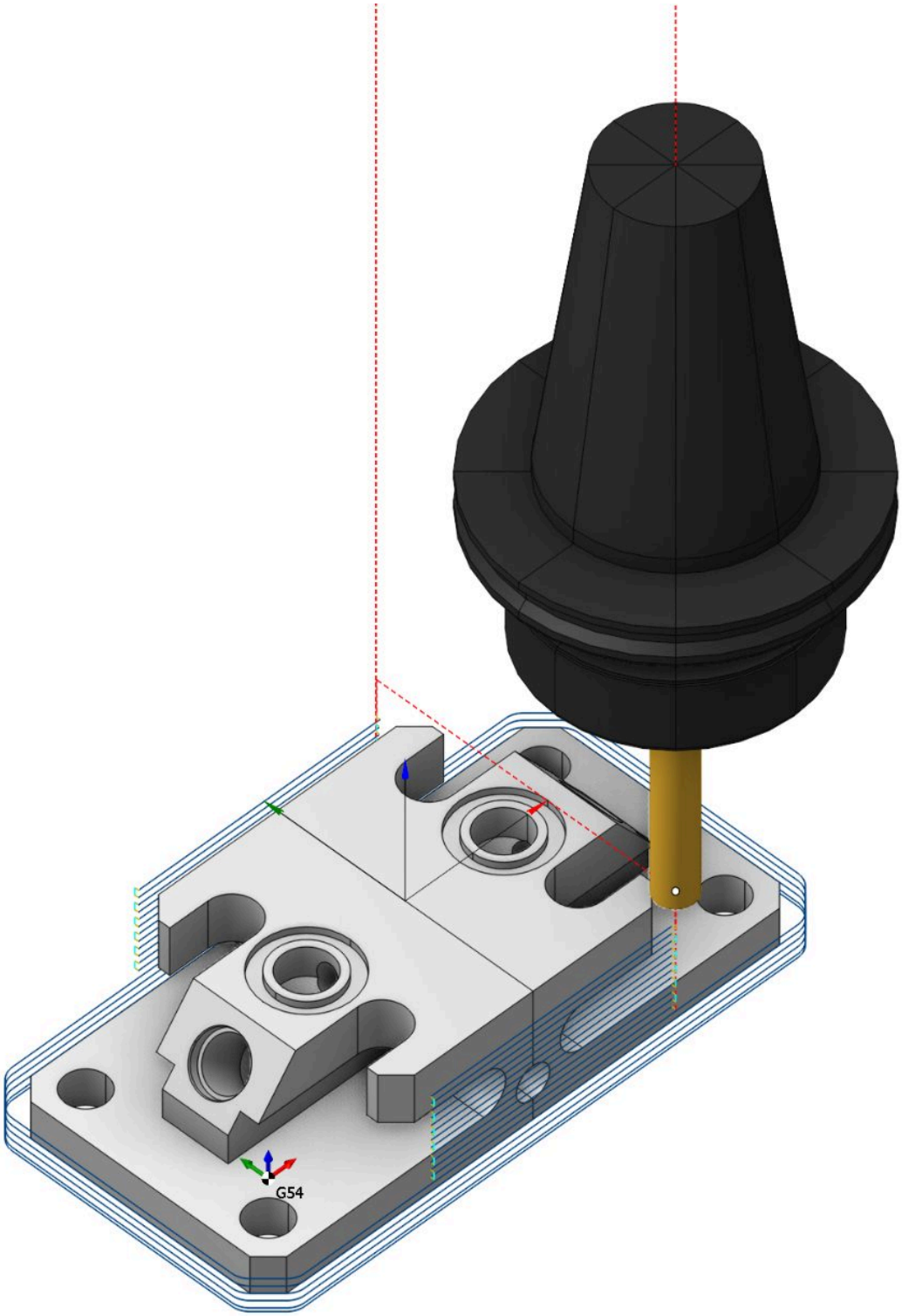
Additional pass. Creates an additional pass if can't process the item on the current pass. This function is useful when plunge option cannot process corner areas. Step will be ignored for additional passes.



Feed distance. Distance where vertical motion switch to work feed.



Finishing waterline



Application area:

The waterline finishing operation gives a good result when machining models or their parts that have their main surface areas close to vertical. For finish machining of flat areas, the user should use Plane operations. Details.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

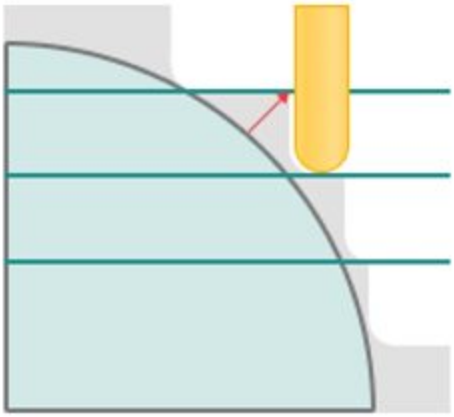
Strategy:

Depth Step.

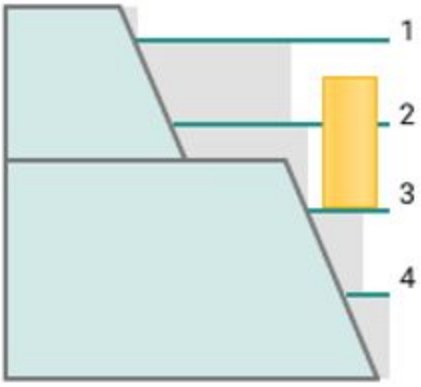
Material removed in one pass along the tool axis between the Top and Bottom levels.

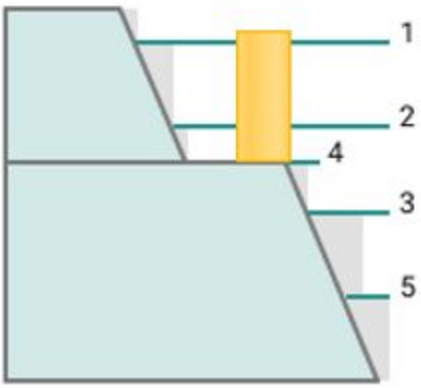
▼ Click here to expand..
Scallop. If disabled: distance between passes is defined only in the parameter Depth Step.

If enabled: the step of machining is calculated automatically based on the maximum given height of scallop. Where necessary the tool passes will be generated more closely to each other to achieve the desired scallop height.



Clear flats. Extra passes are applied for flat faces.





Machining levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

Milling type

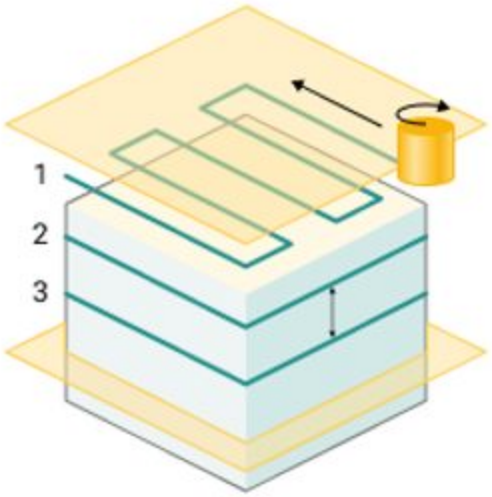
Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting.

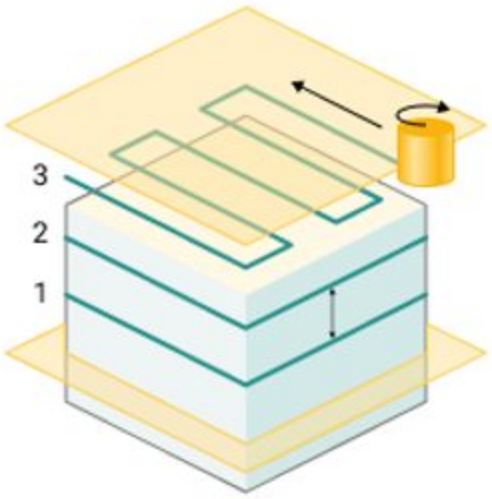
Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..

Downwards. For models, which have surface areas close to the vertical, machining downwards is recommended.



Upwards. Machining upwards is advised for use on models with form-creating surfaces which are closer to horizontal.



Job zone.

Enables additional modification of the working area initially selected on the Job assignment tab. Some parameter group works similarly to the Plane finishing operation. Details

Trimming.

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. This parameter group works similarly to the Waterline roughing operation. Details

Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can

configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

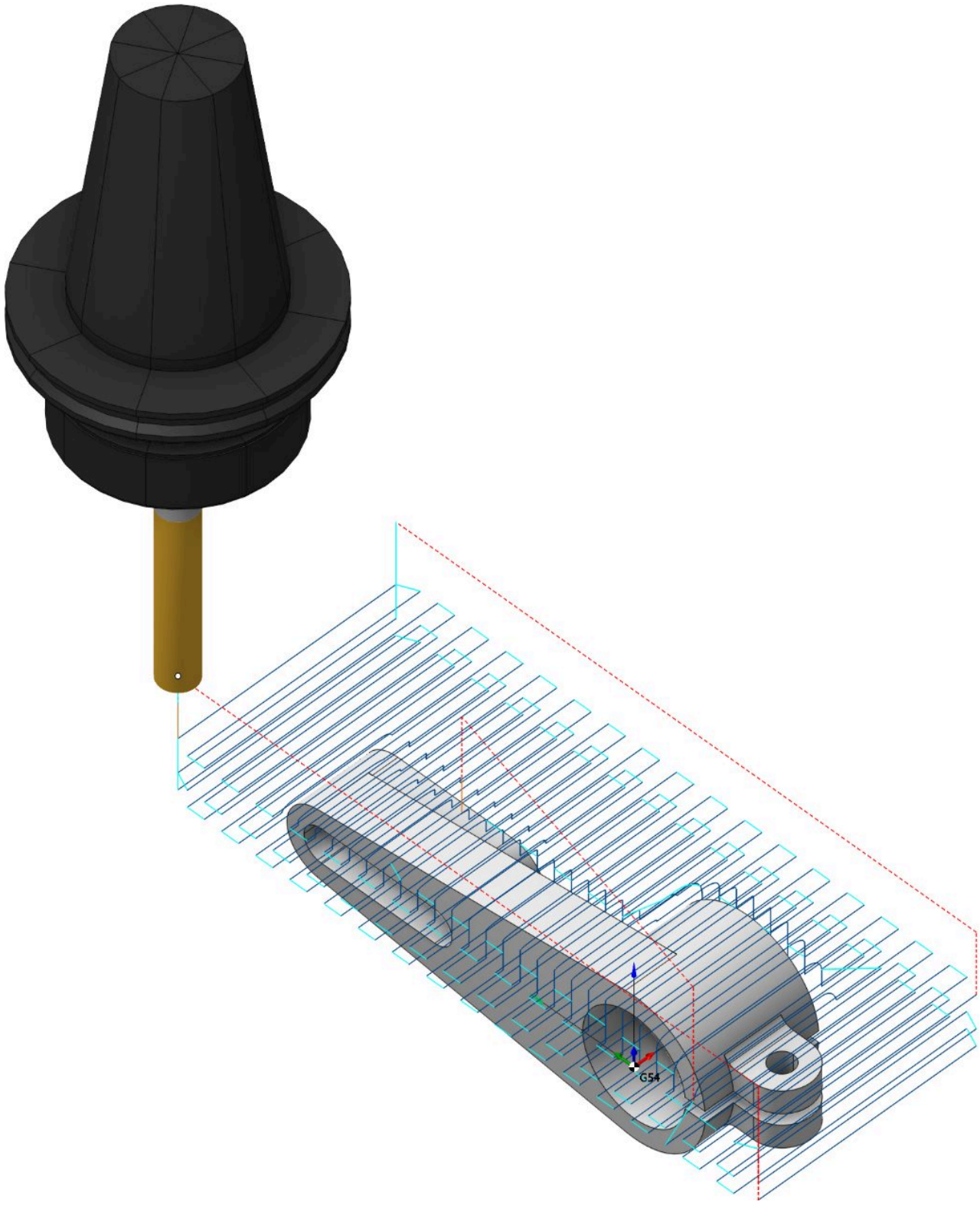
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Roughing plane



Application area:

The machining results of the plane roughing operation are usually closer to the source model when compared to the waterline strategy using similar parameters. This operation is used for machining

models with significant differences to the defined workpiece model prior to rough machining, and for milling soft materials.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Step.

The maximum allowed width of cut. This parameter group works similarly to the Plane finishing operation. Details

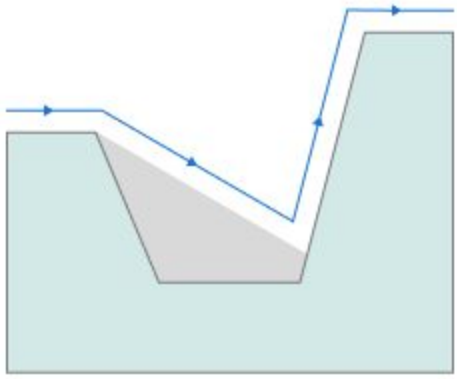
Depth of cut.

Material removed in one pass along the tool axis between the Top and Bottom levels.

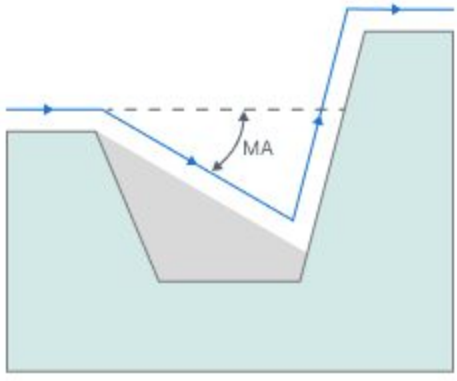
▼ Click here to expand..

Descent. The option allows limiting the downward movement of the tool. Such restrictions are particularly relevant if the tool, due to the specifics of its cutting part or because of the poor machinability of the workpiece, can only engage with the material at a limited angle.

Without checking. The downward movement of the tool inside the material is not being controlled. As a result, the finished part contains imperfections solely due to the geometry of the tool.

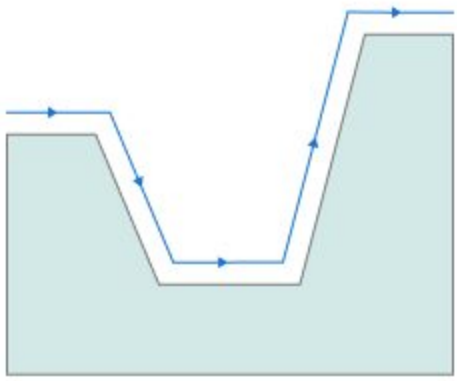


Max angle. The tool's movement within the material downwards is performed at an angle no greater than the specified value.

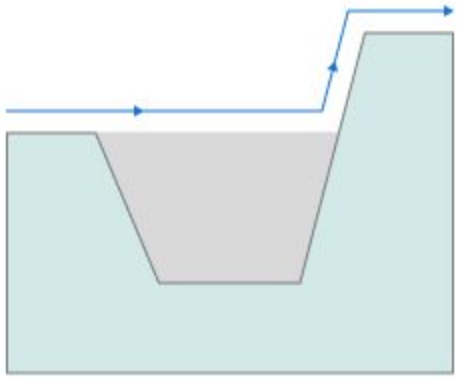


MA - Max angle

Clear shadow. Additionally, the remaining material in the grooves is further processed. This is achieved through reciprocating movements along the path while simultaneously cutting at a specified angle.



Go up only. The tool's downward movement inside the material is completely eliminated. The material remaining in the grooves is not further processed. This method is advisable for convex details. Parts with grooves or pockets require subsequent refinement.



Machining levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

Milling type

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

Trimming.

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. Some parameter group works similarly to the Plane finishing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

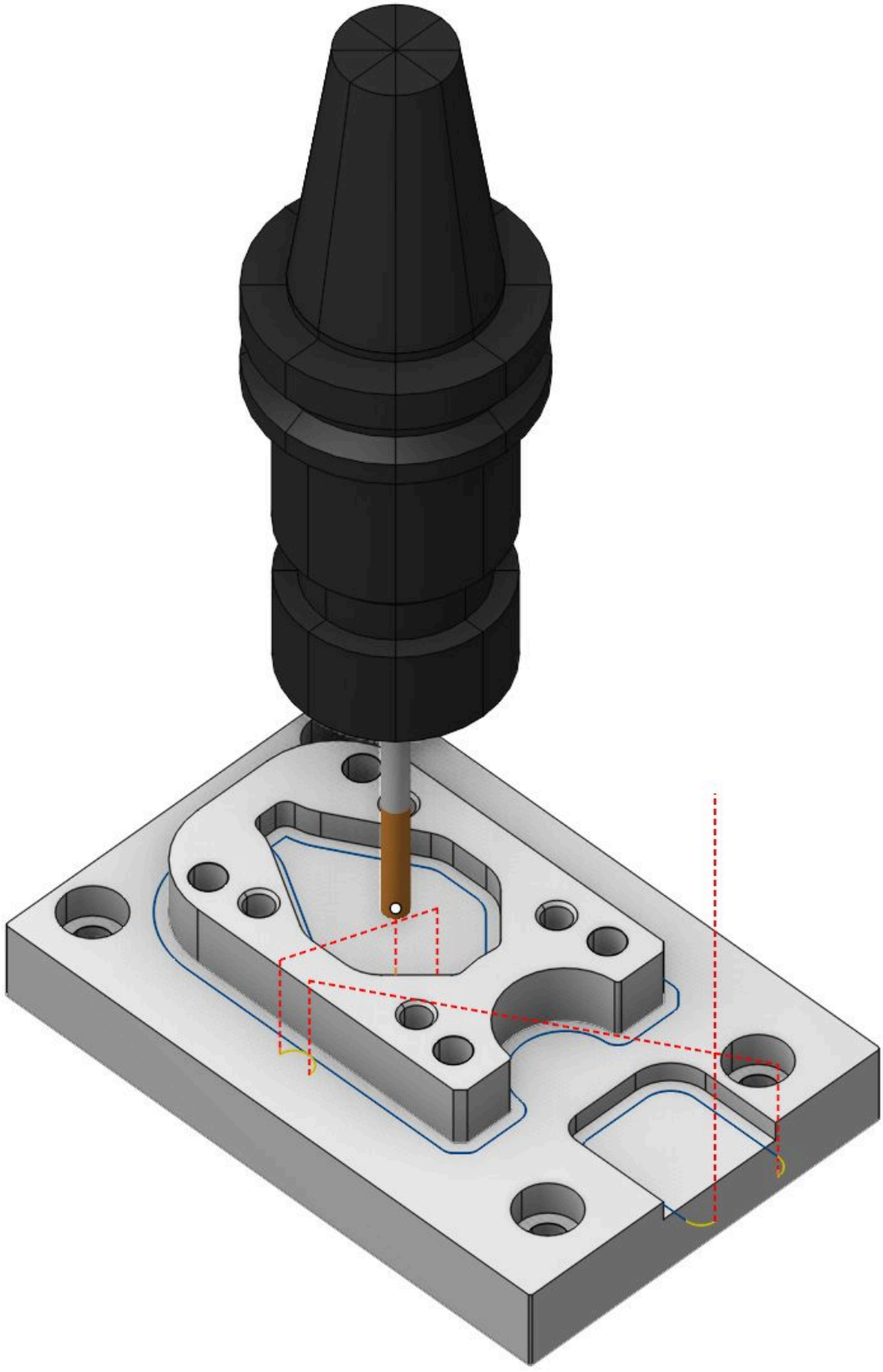
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

2.5D contouring



Application area:

This operation extends the capabilities of the 2D contouring operation in terms of machining multi-level contours of parts. Main the functionality of the 2D contouring operation was preserved.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Curve. Set work order along curve . Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Add tabs. Adds tabs along the contour.Details

Delete. Removes an item from the list

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

The main parameters of the operation Strategy are described in the 2D contour section. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details

Distances. Distances used in Links rules. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

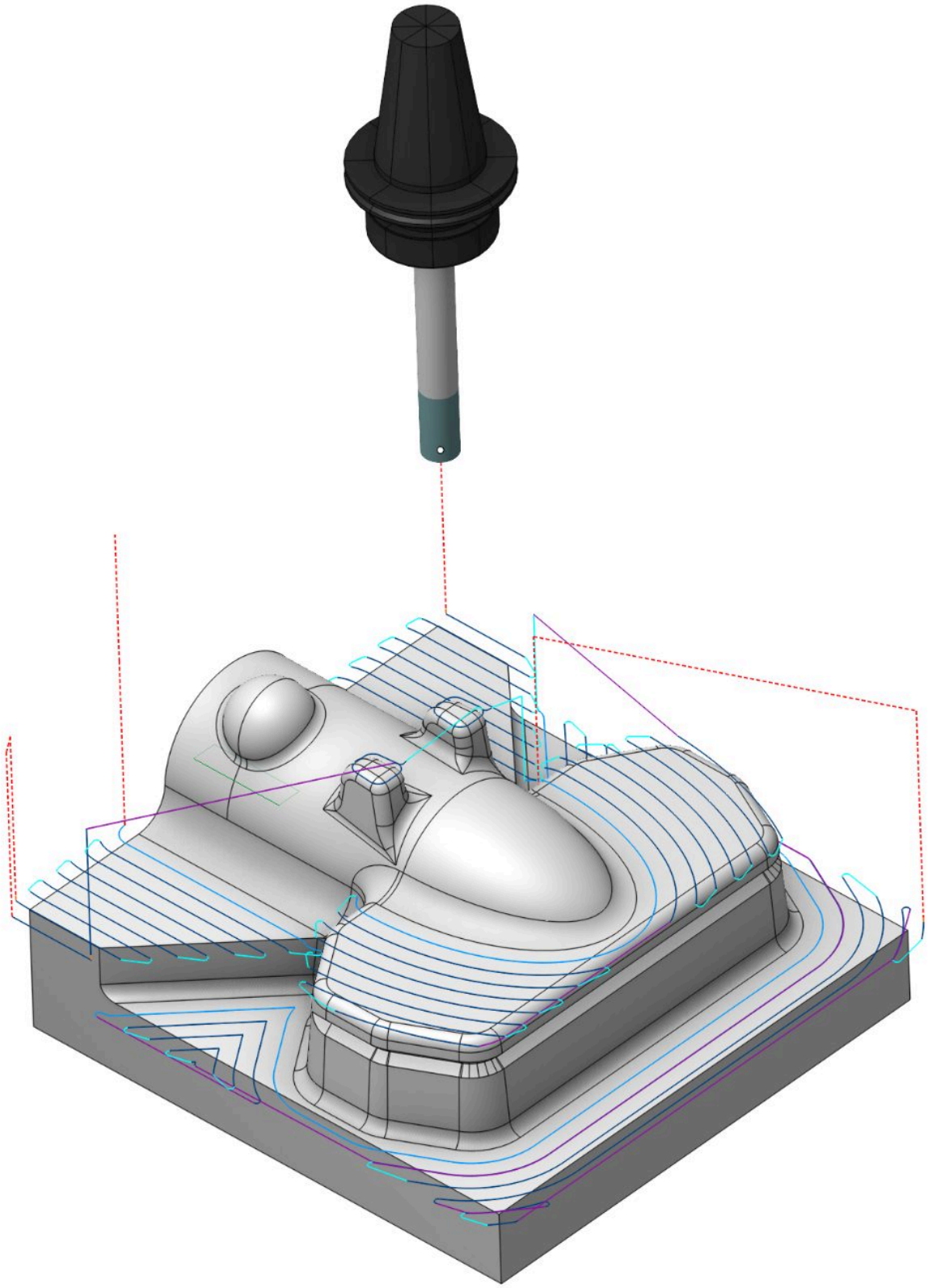
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Flat land



Application area:

The operation is expedient to machine a model with horizontal flats. Machining consists of series of horizontal tool passes on miscellaneous levels.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Machining strategy:

This parameter allows the user to achieve a toolpath that appears parallel or equidistant or other when performing pocketing operations. This parameter group works similarly to the Waterline roughing operation. Details

Machining levels:

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

Flats on finish feed

Outputs the Clear flats path on the finishing feed. This parameter group works similarly to the Waterline roughing operation. Details

Milling type

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Multilayer

▼ Click here to expand..
Step along tool axis. Step into depth of cut.

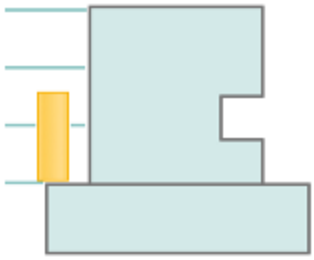
Plunge height.

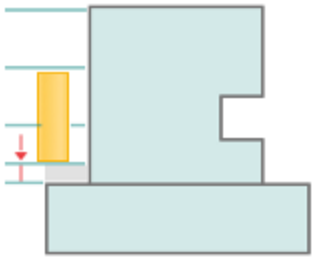
From previous level. Processing is carried out from the lower flat level to the upper level of the workpiece with a step Step along tool axis.

From level. Processing is carried out only on flat levels.

мм. Machining is carried out from a flat level to the value specified in Plunge height with a step Step along tool axis.

Cleanup height. The function adds an additional passage in front of the bottom level at a distance specified in the parameter.





Trimming

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. This parameter group works similarly to the Waterline roughing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Distances. Distances required to switch feeds Details.

Allow plunges. Enabling plunges allows machining of closed pockets. When CAM software encounters a closed pocket that can not be machined from the outside, it forms a Plunge move. Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

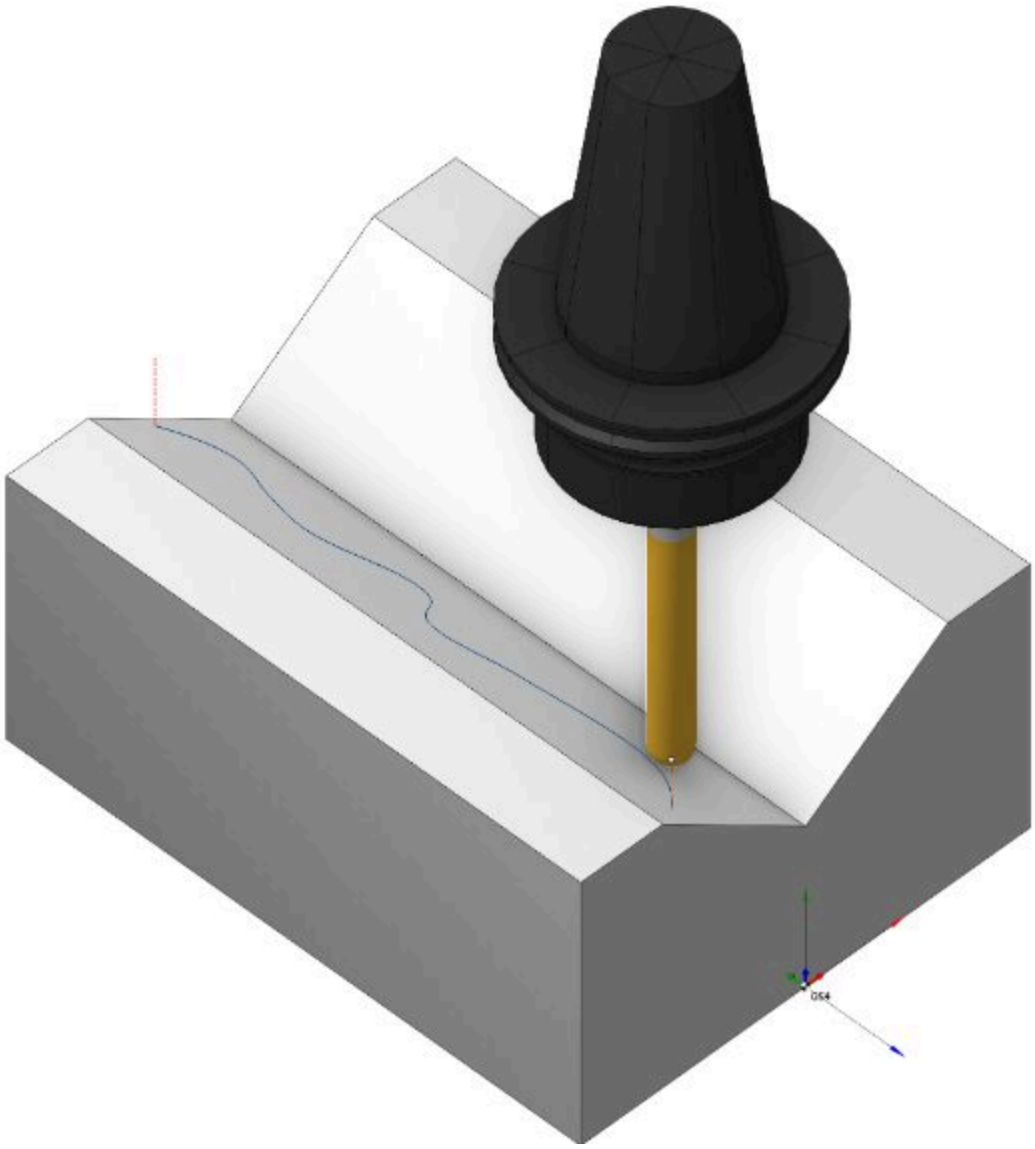
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

3D contouring



Application area:

This operation is designed for performing machining along any spatial curves. Machining is performed in a series of 3D passes of the tool. The passes are created offset in Z from each other based on the Z step value. The number of passes and their Z displacement value depends on the machining levels and the step value entered on the parameters page.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Curve. Set work order along curve . Details

Add tabs. Adds tabs along the contour.Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

The main parameters of the operation Strategy are described in the 2D contour section. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details

Distances. Distances used in Links rules. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Advanced

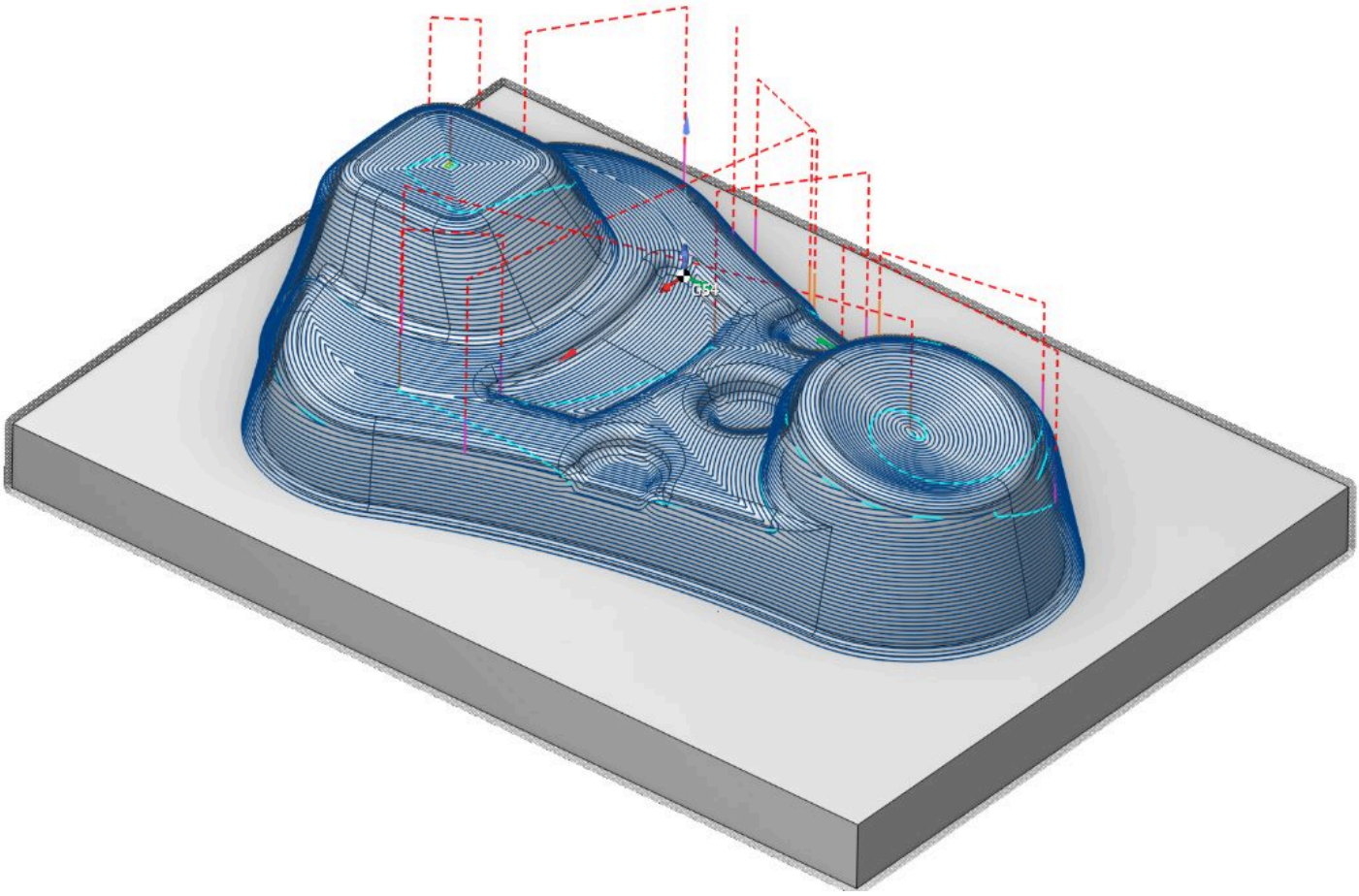
Application area:

The Advanced group contains operations for surface finishing, undercut machining, multi-axis roughing, and machining spatial contours. The individual topics describe the available toolpath strategies and tool orientation settings.

Operations:

- Combo finish. Combines plane, waterline, and scallop finishing strategies. Details
- Surfacing. Finishes surfaces using configurable strategies and tool orientations. Details
- Scallop. Generates finishing passes with a consistent step-over across surfaces. Details
- Helical. Generates a helical finishing toolpath. Details
- Morph. Generates passes that morph between specified curves. Details
- Undercut waterline. Machines undercut areas using waterline passes. Details
- Swarf. Machines surfaces with the side of the tool. Details
- 5D Roughing waterline. Generates five-axis roughing toolpaths. Details
- 5D by meshes. Generates toolpaths for machining meshes. Details
- 6D Contouring. Machines spatial contours with control of tool orientation. Details
- Contour polishing. Finishes part walls and edges along contours using a disc tool. Details

Combo Finish



Application area:

This operation can replace outdated legacy operations and address associated issues.

Advantages:

- Support for generating helices for waterline passes (helical machining).
- In combined strategies, gentle and steep areas are processed using different strategies. This ensures a uniform scallop pattern on the surface and improves cutting conditions.
- Combined strategies feature smoothing/merging of machining zones based on the Smooth distance parameter. This helps produce a more solid toolpath with fewer unnecessary passes and transitions.
- Intelligent toolpath merging for combined strategies has been implemented.
- The calculation process is fully parallelized.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Machining surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Restrict surfaces.

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. For a surface set, Axial stock and Radial stock are added to operation stocks; radial equals axial by default. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

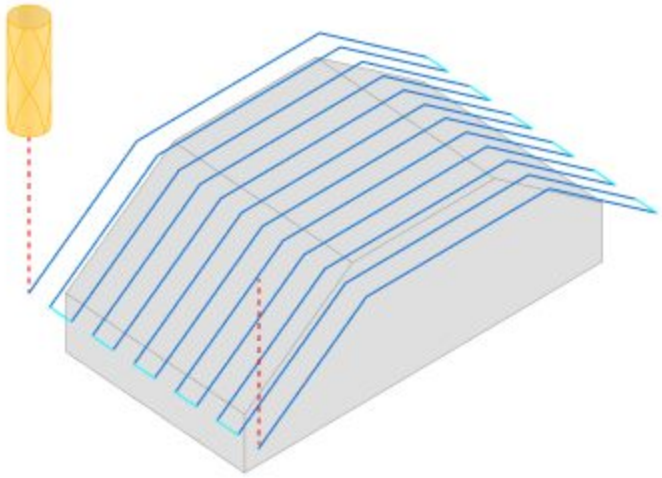
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

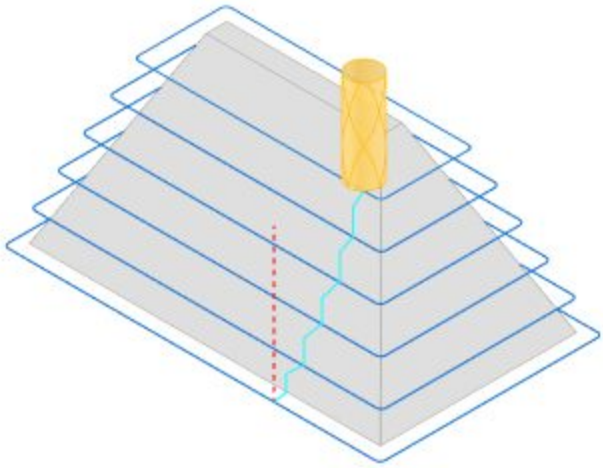
Strategy.

This parameter enables the user to create a toolpath with different strategies.

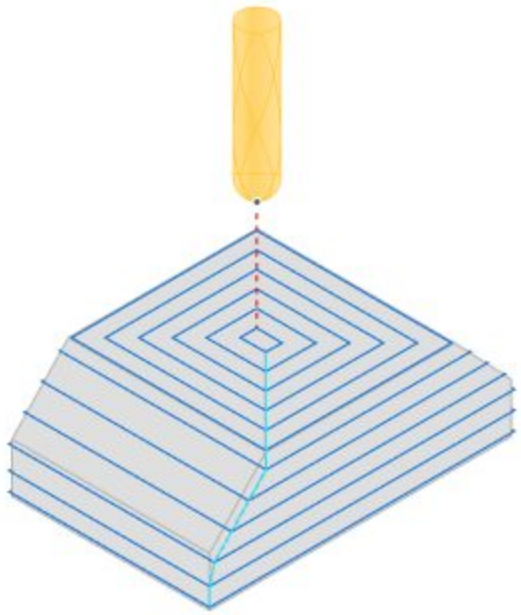
▼ Click here to expand..
Plane. The plane finishing is designed for the machining of smooth areas of a model's surfaces, and also for areas close to vertical, whose (steep) trajectories are along the toolpath.



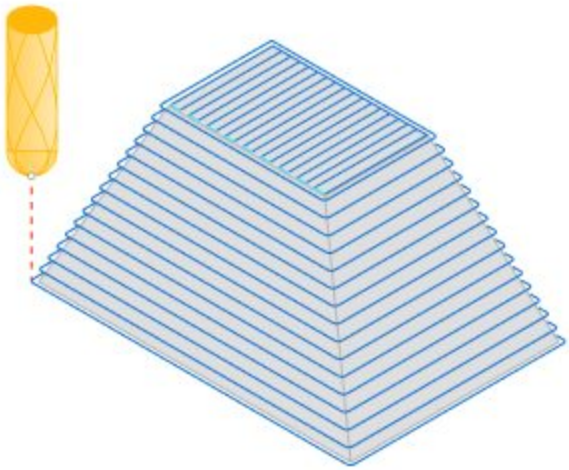
Waterline. The waterline finishing gives a good result when machining models or their parts that have their main surface areas close to vertical.



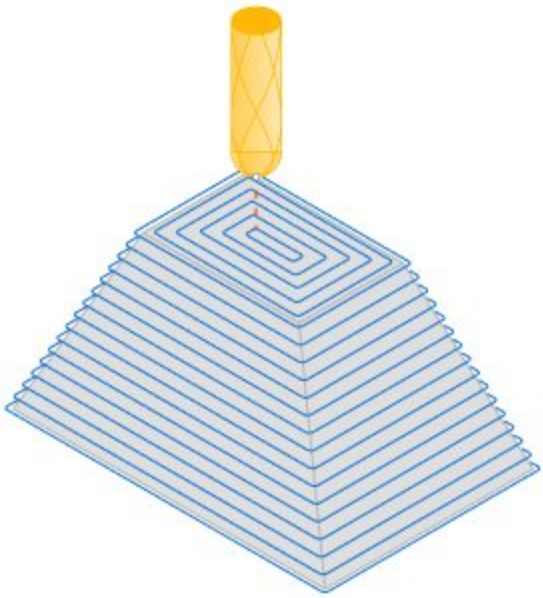
Scallop. The Scallop toolpath starts from the curves lying on the part surfaces and is generated by repeatedly offsetting those curves inwards until the curves collapse. This method creates a toolpath similar to equidistant strategy.



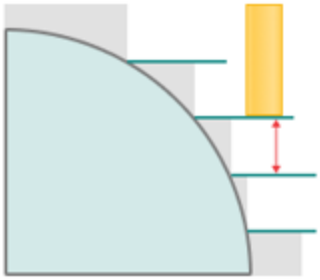
Plane + Waterline. The plane toolpath machines only areas with the slope angle less than the Split slope angle while the waterline toolpath machines areas where the surface slope angle is greater than the Split slope angle. By default the split slope angle is set to 45 degrees. This strategy ensures that shallow areas are machined with the plane toolpath, and steep areas are machined with the waterline toolpath.



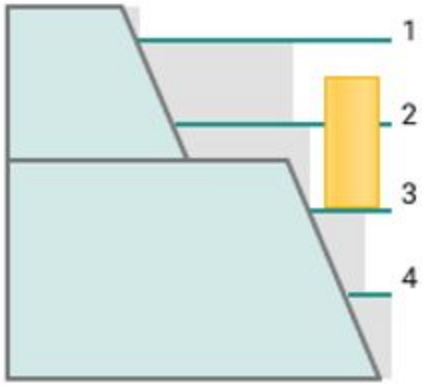
Scallop + Waterline. The plane toolpath machines only areas with the slope angle less than the Split slope angle while the waterline toolpath machines areas where the surface slope angle is greater than the Split slope angle. By default the split slope angle is set to 45 degrees. This strategy ensures that shallow areas are machined with the scallop toolpath, and steep areas are machined with the waterline toolpath.

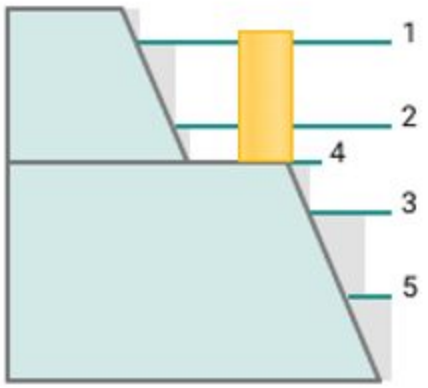


Step. Material removed in one pass along the tool axis between the Top and Bottom levels.

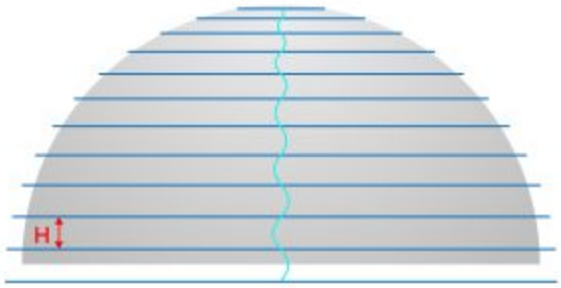


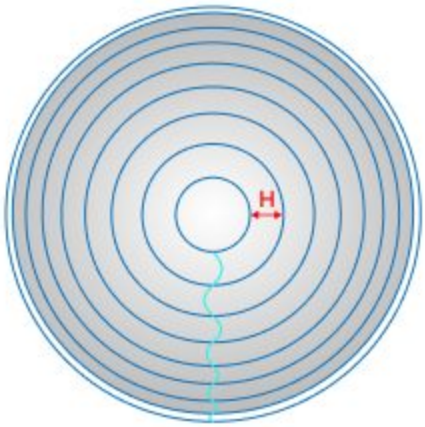
Clear flats. Adds waterline passes at horizontal flat levels. Enabled by default, shown except for Plane, and affects the waterline part of the toolpath.





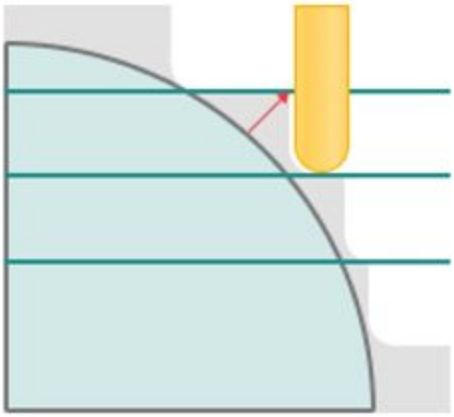
Adaptive step. Keeps the step along the surface rather than in projection, adding passes on inclined areas so the surface distance does not exceed Step. Not available for the Scallop strategy.



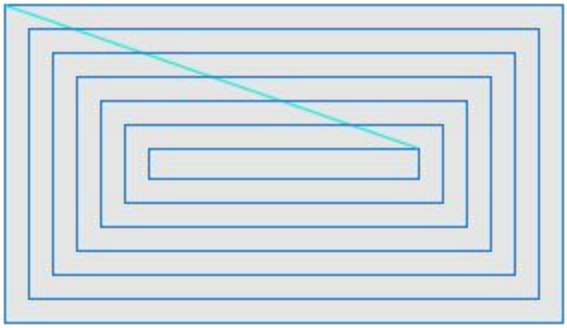


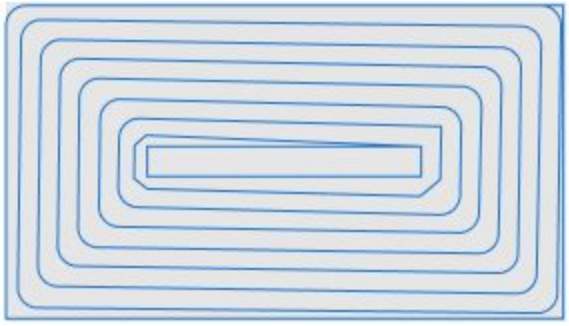
H - Crest height / step

Scallop. Limits the scallop height between neighbouring passes; the Step remains the maximum. Not available for the Scallop strategy and ignored when Adaptive step is enabled.

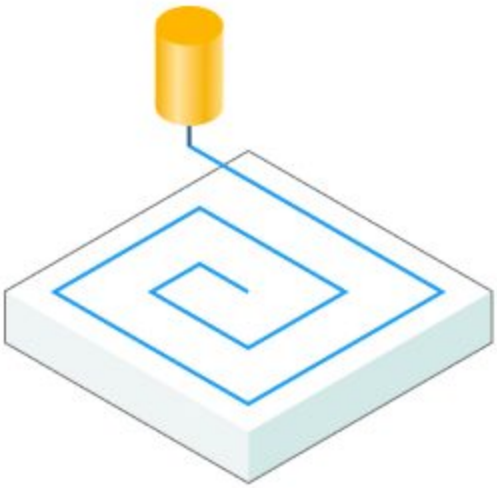


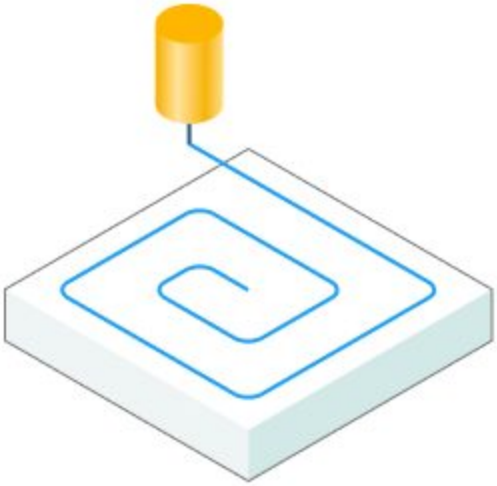
Spiral Machining. Setting the flag configures the toolpath as a continuous spiral to minimize links.



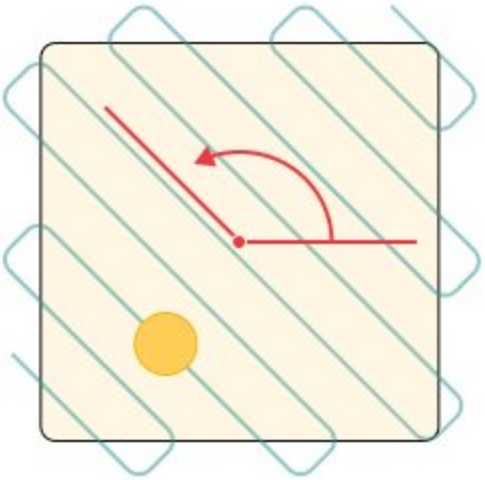


Corners smoothing. Rounds sharp corners with the specified Radius to reduce deceleration. Works with all strategies.





Angle. Assigns the cut angle direction for the plane operations.



Milling type.

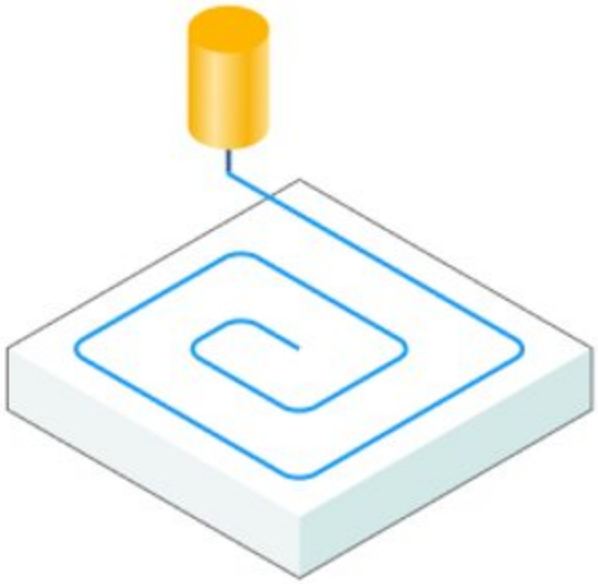
Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

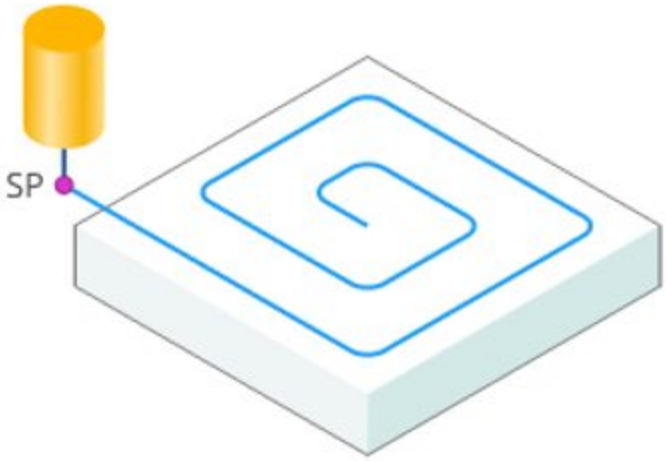
Sorting.

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..

Start Point. The point defining the beginning of the plunge.

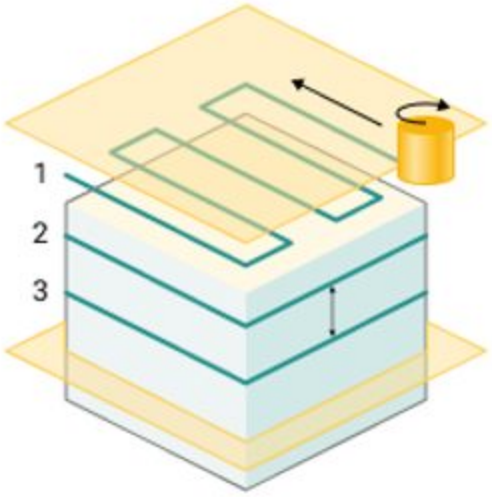




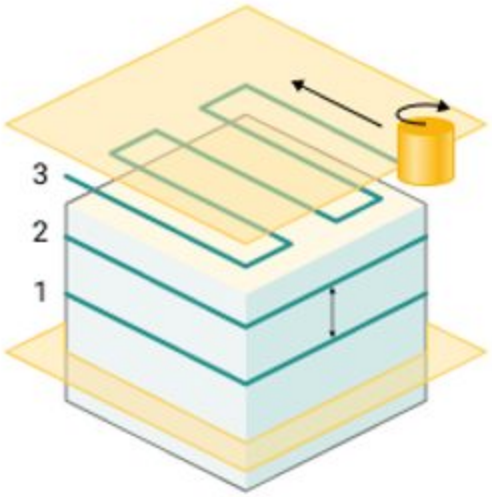
Start point shift. Shifts each next closed pass start to avoid one seam. Default 300 %. Not applied with Spiral machining.

Machining order.

Downwards. For models, which have surface areas close to the vertical, machining downwards is recommended.



Upwards. Machining upwards is advised for use on models with form-creating surfaces which are closer to horizontal.



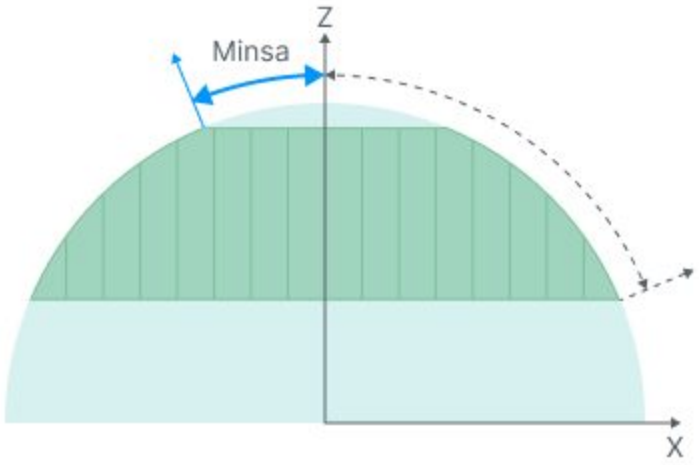
Job zone.

Enables additional modification of the working area initially selected on the Job assignment tab.

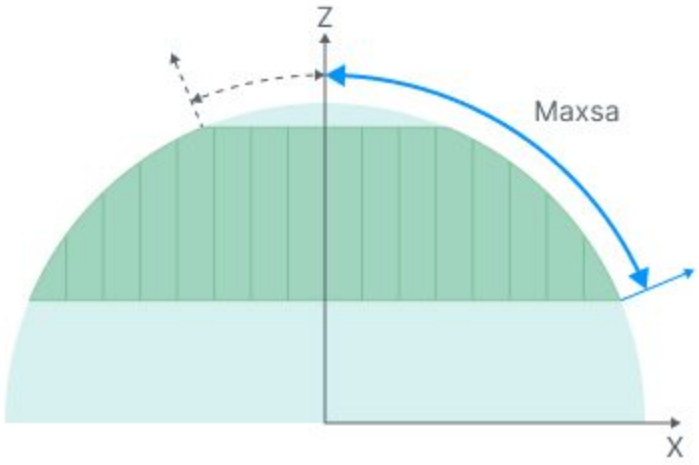
▼ Click here to expand..

Surface roll type. Surface only (default) machines selected surfaces only; With edges also rolls over surface edges. Takes effect when Check part is enabled.

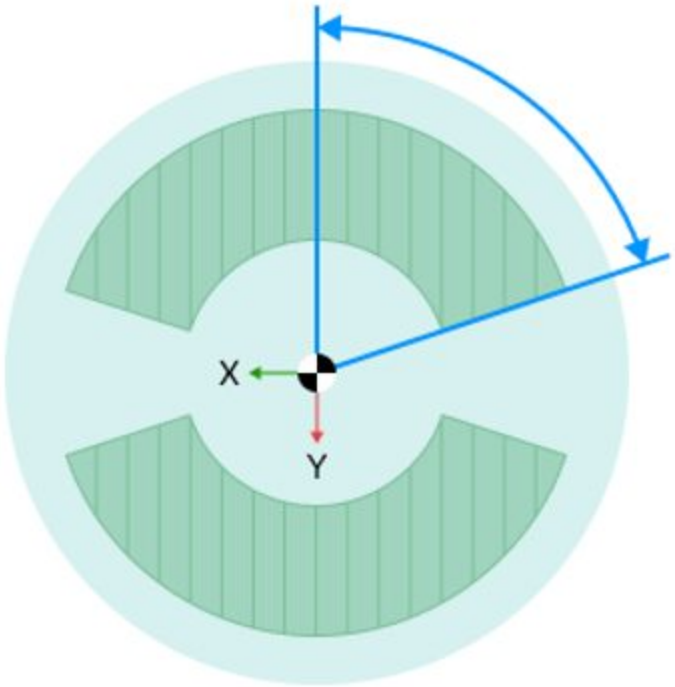
Min slope angle. The default value is 45 degrees; when disabled, no minimum limit is applied. Not shown for Scallop. In the system it is possible to perform selective machining of the model surface areas, depending on the angle between the normal to the surface and the vertical axis Z. The range being machined can be defined by the minimum and maximum slope angles.



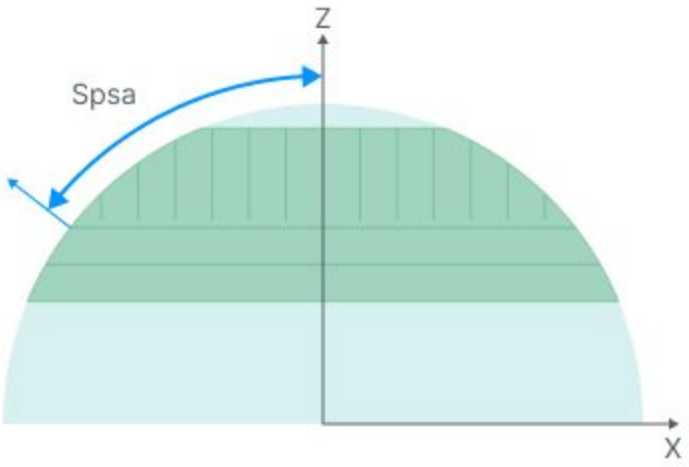
Max slope angle. The default value is 45 degrees; when disabled, no maximum limit is applied. In the system it is possible to perform selective machining of the model surface areas, depending on the angle between the normal to the surface and the vertical axis Z. The range being machined can be defined by the minimum and maximum slope angles.



Frontal angle. The frontal angle is the angle between projections onto the horizontal plane of the tool movement direction and the normal to the surface at the cutting point. Details



Split slope angle. Angle between vertical and horizontal passages.



Spsa - Split slope angle

Merge distance. Sets the distance used to smooth and merge adjacent machining regions in the combined strategies (Plane + Waterline, or Scallop + Waterline). Neighbouring areas located within the specified distance are joined into a single continuous region, which eliminates short, fragmented passes and produces a smoother, more uniform toolpath.

Machining levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

Trimming.

Hole capping closes openings with a default diameter of 2 x tool diameter. Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. Some parameter group works similarly to the Waterline roughing operation. Details

Parameters:

For this operation, Check workpiece includes Radial ignore thickness, Axial ignore thickness, and Theoretical rest material.

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

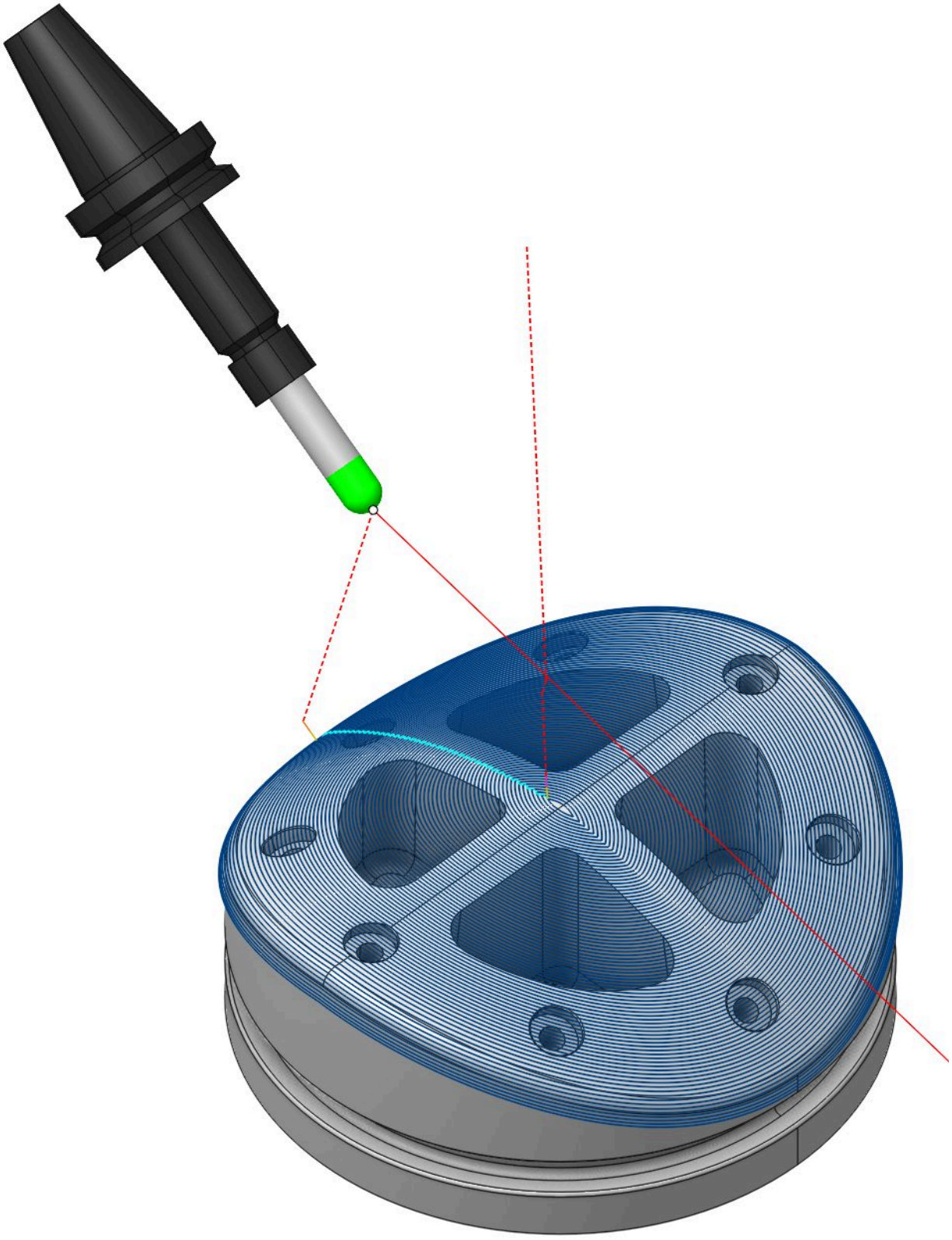
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Surfacing



Application area:

Surfacing operation is the operation from the finishing group and enables machining the surfaces of the model with a variety of strategies (parallel to planes, parallel to curve, morph, etc.) and tool axis orientation modes (fixed, normal to surfaces, to rotary axis, through point, through curve, etc.).

Job assigment:

Machining Surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

First Curve. For the Parallel to Curve strategy, this feature specifies curves for parallel passes computations. Additionally, in the Morphing Between Two Curves strategy, it specifies the First Curve . You can select one or multiple, not necessarily connected, curves or edges.

Second Curve. The system uses it in he Morphing Between Two Curves strategy. It specifies the Second Curve . You can select one or multiple, not necessarily connected, curves or edges.

Tilt Curve. This is t he curve that the tool axis vector aims at when setting Tool Orientation specification Through Curve.

First Surface. This feature s erves as the First Surface in a Morph Between Two Surfaces strategy, facilitating the morphing process to create toolpath lines.

Second Surface. This feature s erves as the Second Surface in a Morph Between Two Surfaces strategy, facilitating the morphing process to create toolpath lines.

Surfaces for projection. These are the s urfaces intended for machining onto which the toolpath transfers when enabling the Project Toolpath onto the Part option in the Strategy tab.

Job Zone. Use the job zone to trim the passes outside the specified 2d containment areas. The plane of the containment area is defined by the initial tool orientation. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that s hould avoid machining in the current operation. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Strategy:

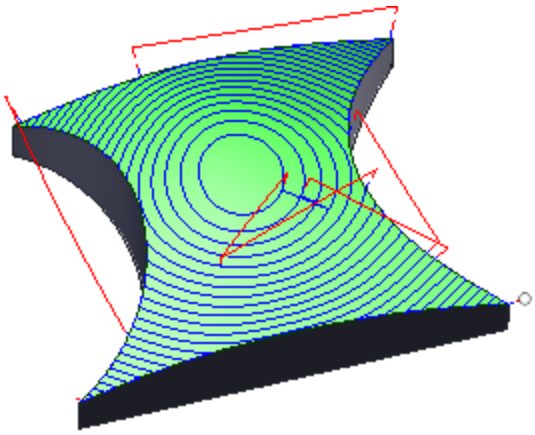
This parameter allows the user to achieve a required toolpath:

▼ Click here to expand..

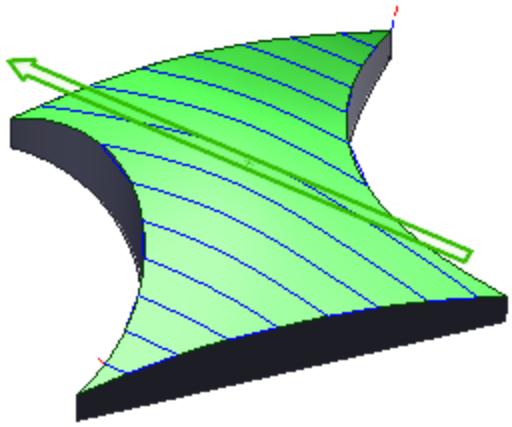
Type of Strategy:

▼ Click here to expand..

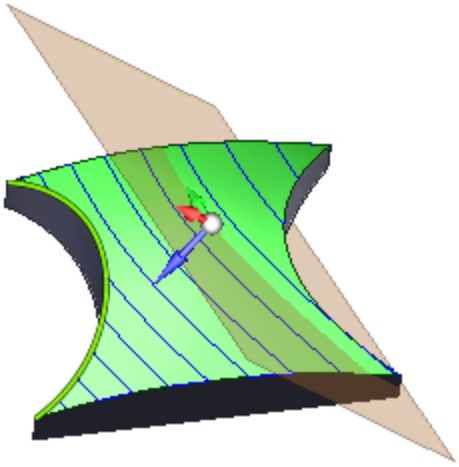
Parallel to Horizontal Plane. The system generates the tool passes as section lines of the Machining Surface by horizontal planes (planes are perpendicular to the tool axis), similar to the strategy in a Finishing Waterline operation. Details.



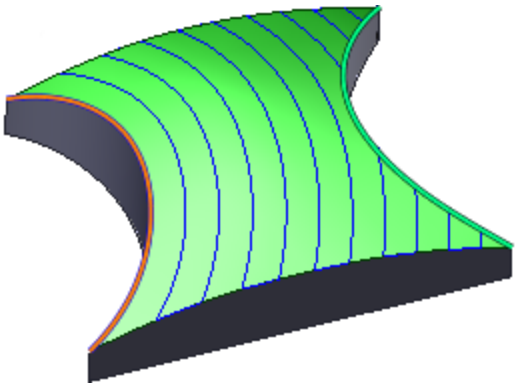
Parallel to Vertical Plane. The system generates the tool passes as section lines of the Machining Surface by vertical planes (planes are parallel to the tool axis), similar to the strategy in a Finishing Plane operation. You can change the angle of the plane around the tool axis. Details.



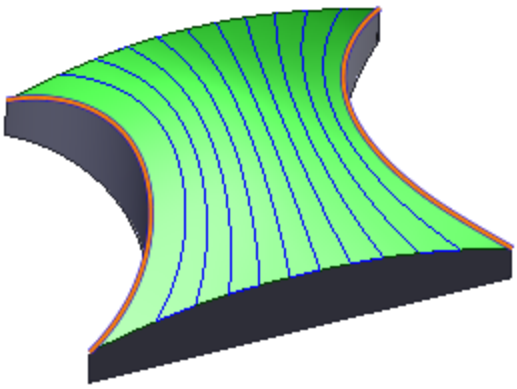
Parallel to 3d Plane. The system generates the tool p asses as section lines of the Machining Surface by planes positioned at adjustable angles in space.



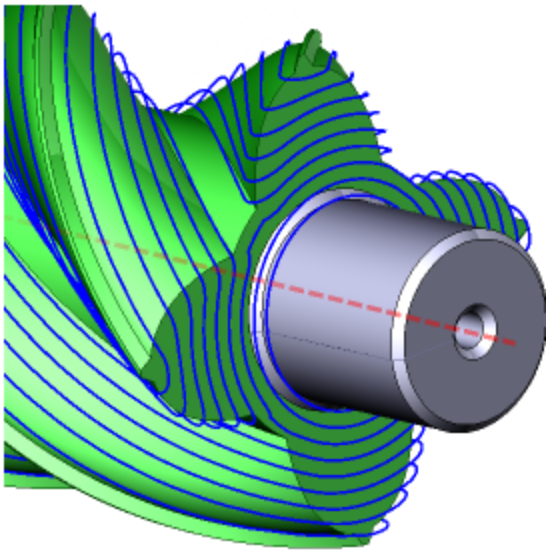
Parallel to Curve. The system generates the tool p asses on Machining Surface at a specified Step parallel to the First Curve defined in the Job Assignment.



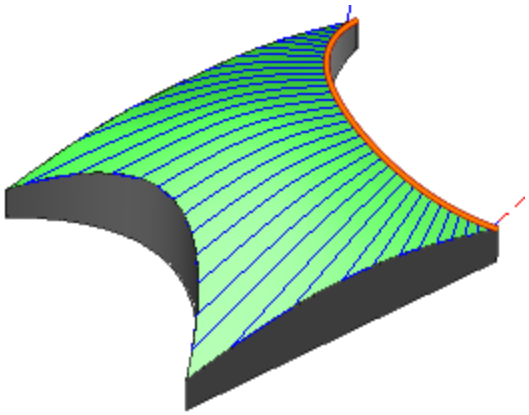
Morph Between Two Curves. The system generates the tool p asses as geometric points arrays on the Machining Surface, maintaining a constant ratio between the distance from the point to the First Curve and the distance to the Second Curve for each pass.



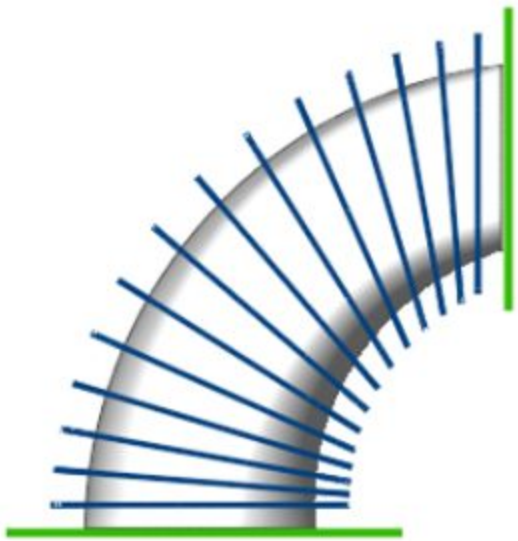
Around Rotary Axis. The system generates the tool p asses as sections of the Machining Surface at a specified Step by employing cylinders with an axis specified as the Rotary Axis.



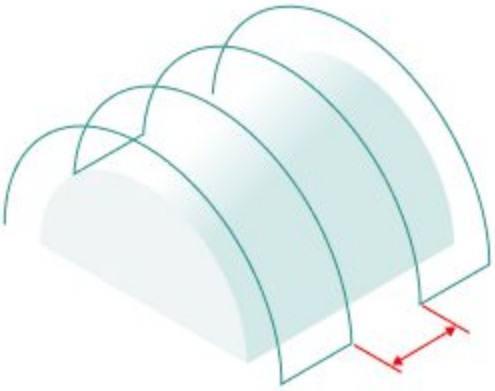
Across Curve. The system generates the tool p asses as section lines of the Machining Surface with planes perpendicular to the First Curve specified in the Job Assignment.



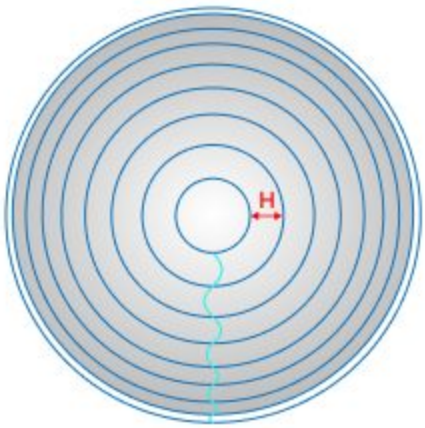
Morph Between two surfaces. The system generates the tool p asses as section lines of the Machining Surface by surfaces formed through a smooth transition from the First Surface to the Second Surface, at intervals according to the Step.



Step. The maximum allowed width of cut.

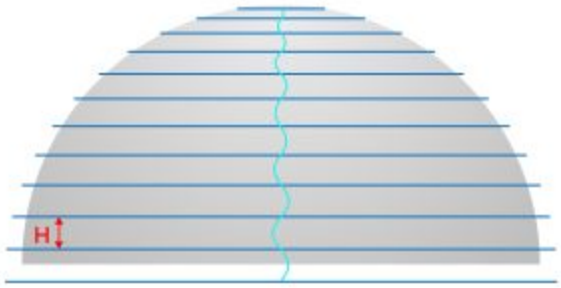


Adaptive Step. If enabled: This parameter adjusts the distances between tool passes according to the curvature radii of the surface and the tool to maintain uniform scallop height after surface machining. Details.



H - Crest height / step

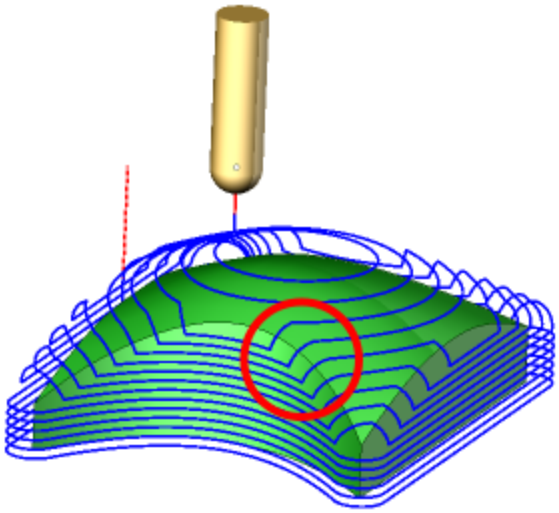
If disabled: This is the fixed vertical distance between consecutive tool passes.



Calculate Based on Tool Center. Flagging switches the mode that involves first offsetting the surfaces for machining by the tool profile for 3-axis machining or along the surface normal by the tool radius for 5-axis machining, and then calculating toolpath curves on these offset surfaces. In another mode, the system initially generates tool path as curves on the machined surfaces. Then, it positions and orient the tool relative to the calculated curves to keep the tool's contact point with the machined surface constant.

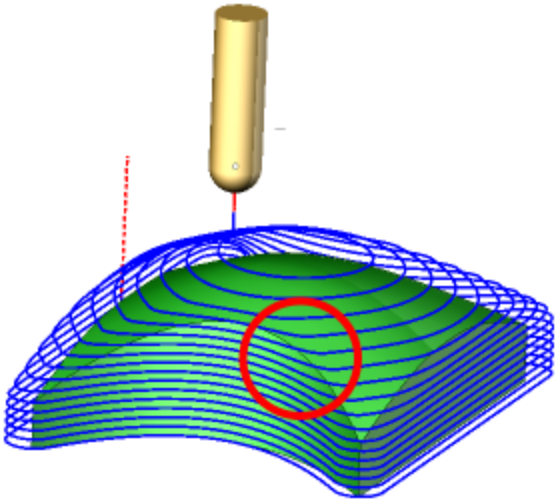
Comparison of the toolpath based on the center and the tool contact point:

Calculation based on tool contact point:



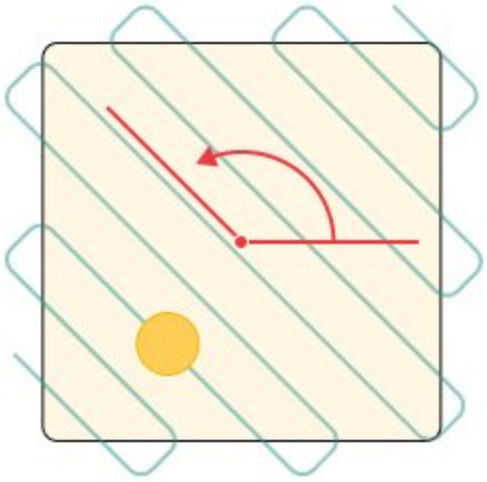
In this mode the work passes are generated by calculating curves on the machining surfaces as the first step and then positioning and orienting the tool relative to the calculated curves in such a way that the point of contact of the tool with a machining surface stays the same. It is desirable, for example, when smooth surfaces are machining, or doing flank milling.

Calculation based on tool center:

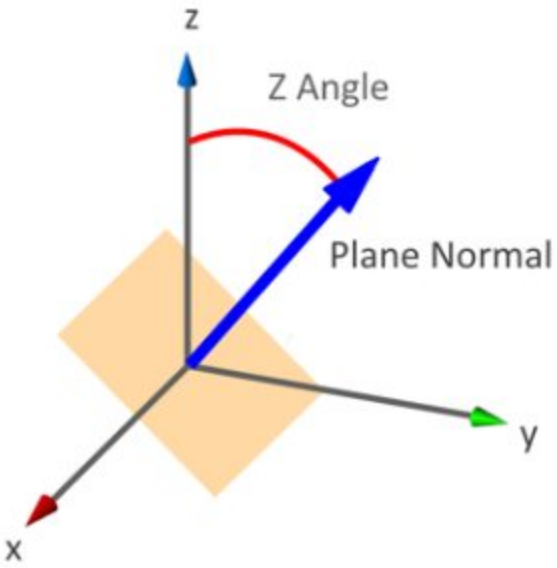


In this mode the work passes are generated in such a way that at first the machining surfaces are being offset either by the tool shape itself for 3 axis machining or by the tool radius along the surface normal for 5 axis machining and only then the sections of the offset surfaces are calculated. For example, for the Parallel to plane strategies it means that the generated work passes all lie on parallel planes. Another advantage of this mode is that not only the surfaces themselves but the edges between machining surfaces are taken into account.

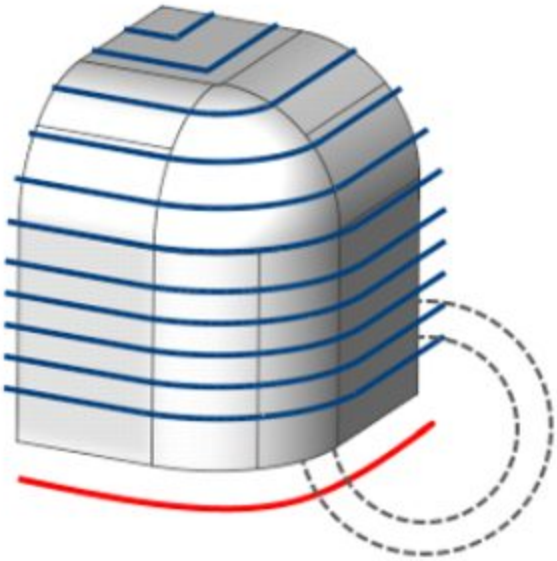
XY Angle. This is the r otation angle of the section planes around the tool axis.



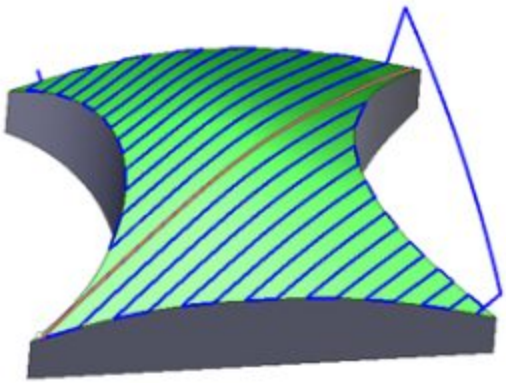
Z Angle. This is the a ngle between the Z-axis and the normal to the section plane.



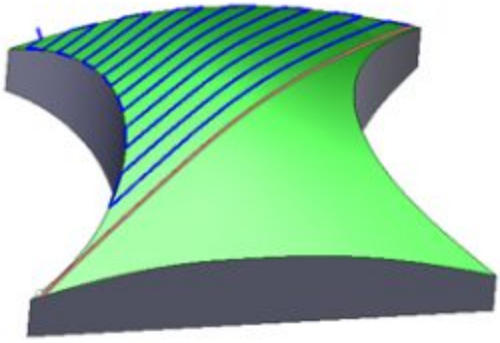
Use 2d Curves. When flagging, the system generates the toolpaths relative to curves projected onto the XY plane in Parallel to Curve and Morphing between Two Curve s strategies.



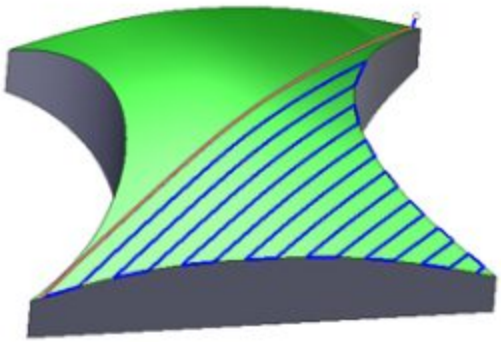
One Side Machining. In the Parallel to Curve and Morph between Two Curve s strategies, it controls the placement of the toolpath relative to the curves specified in the Job Assignment. Without the flag, the system generates toolpath on both sides of the curve.



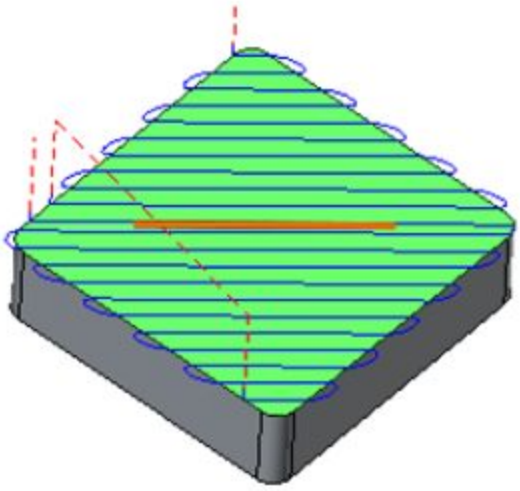
Left. The system generates passes on the left side of the curve specified in the Job Assignment.



Right. The system generates passes on the right side of the curve specified in the Job Assignment.



Extend Curves to Infinity. Setting the flag allows the system to extend the passes parallel to the selected lines up to the edges of the Machining Surface when the curves are smaller than the Machining Surface.



Rotary Axis. This is t he axis of section cylinders in the Around Rotation Axis strategy.

Milling Type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

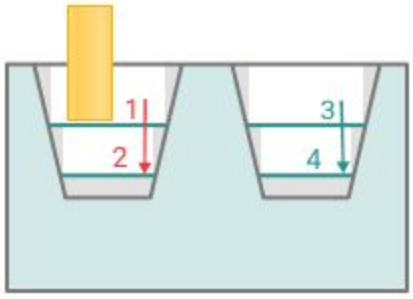
Sorting:

Controls the sequence of toolpath passes during surface machining.

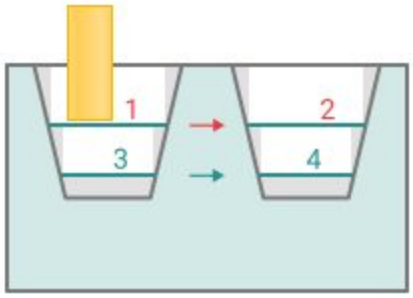
▼ Click here to expand..

By regions. Determines whether working passes are grouped by region or by layer.

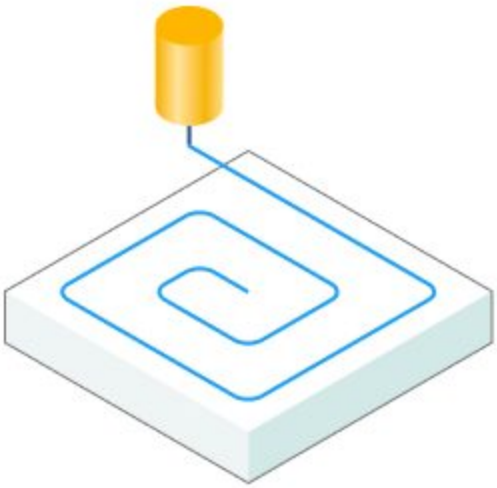
ON. All layers of one region (cavity) are machined before moving to the next region.

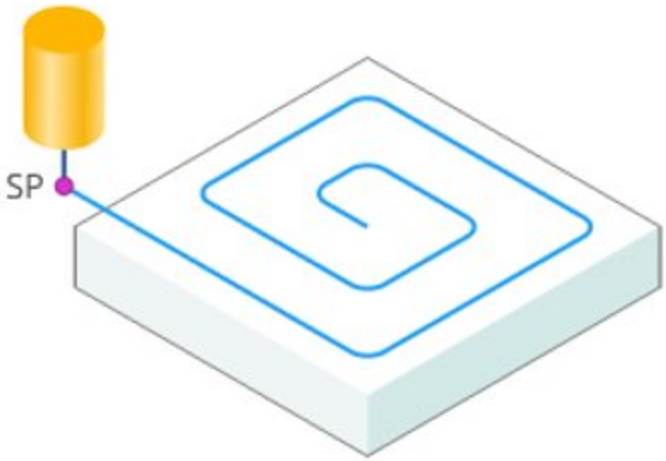


OFF. One layer is machined across all regions before moving to the next layer.



Start Point. Activating the flag permits the input of coordinates marking the starting point of working passes in surface machining.

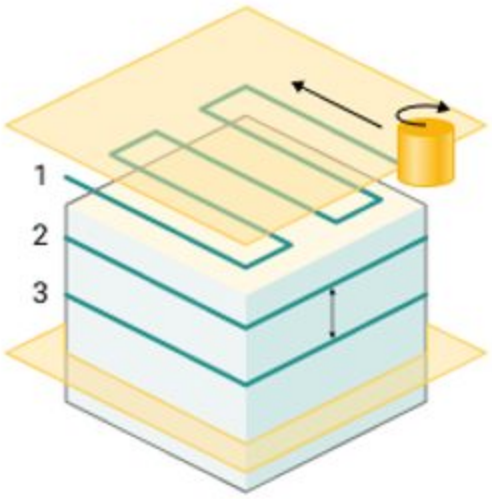




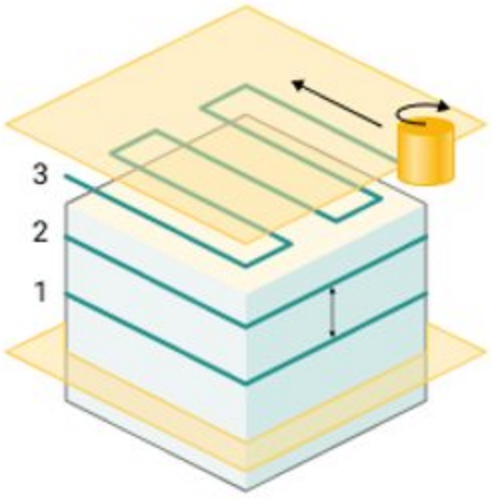
Machining Order. Controls the execution order of the tool's working passes.

▼ Click here to expand..

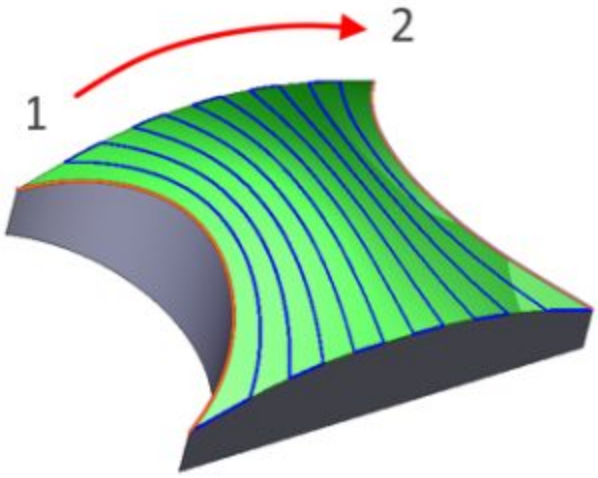
Downwards. Working passes proceed from top to bottom, from the nearest pass to the furthes.



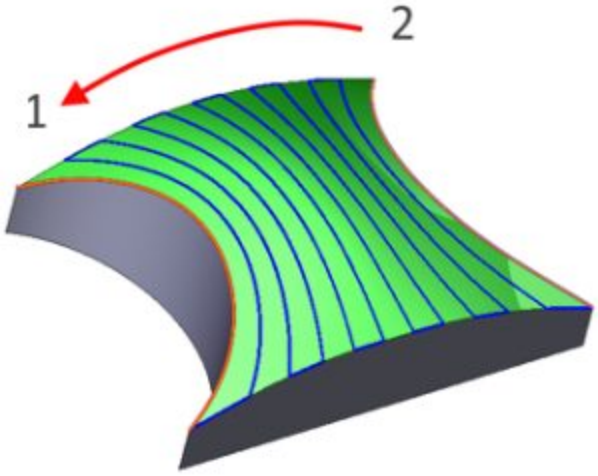
Upwards. Working passes proceed from bottom to top, from the most distant pass to the closest one.



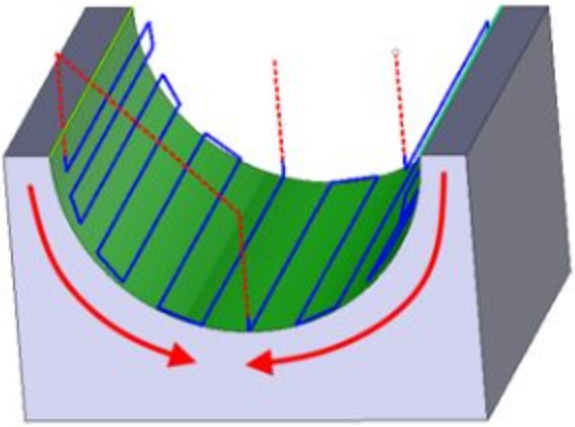
From First to Second. Operation executes passes from the First towards the Second Curve or Surface.



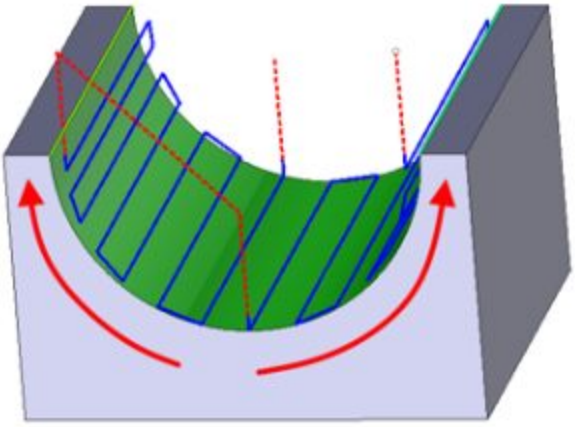
From Second to First. Operation executes passes from the Second towards the First Curve or Surface.



From Outside in. Working passes proceed from the periphery towards the midpoint of the Machining Surface.



From Inside Out. Working passes proceed from the midpoint towards the periphery of the Machining Surface.

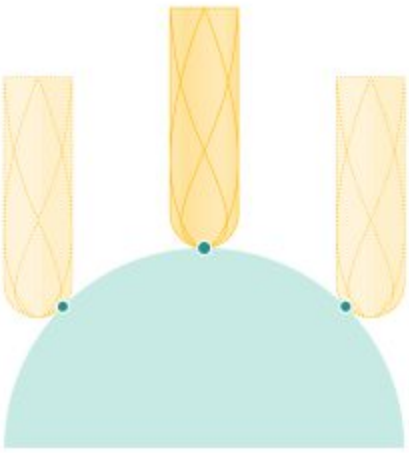


Tool contact point.

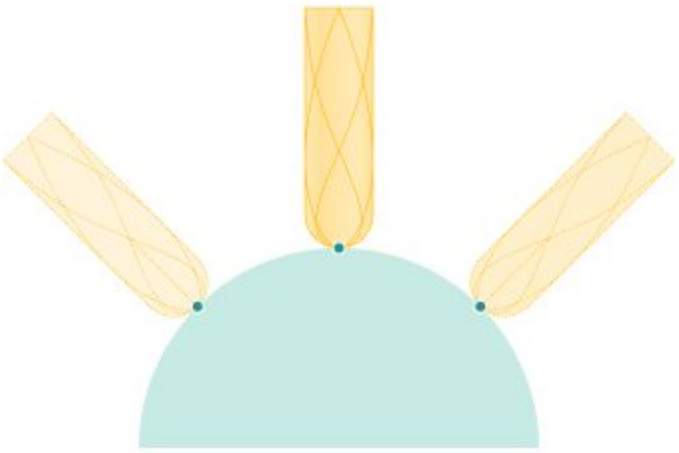
Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

▼ Click here to expand..

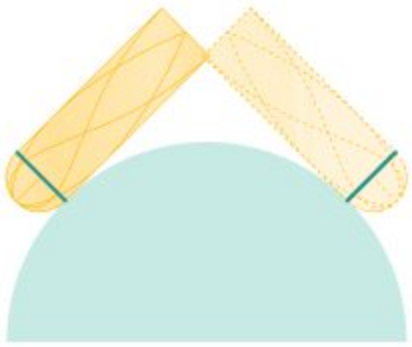
Auto. The point of contact of the cutting tool moves along the cutting edge of the tool without changing the axis vector of the tool.



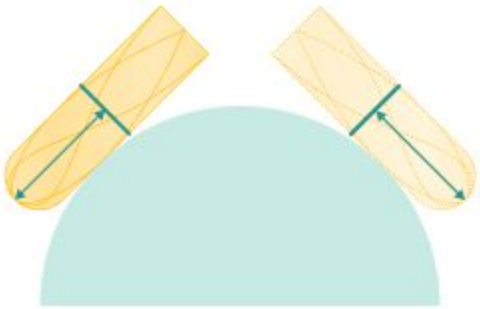
End. The point of contact of the cutting tool is located at the top of the cutting edge of the tool.



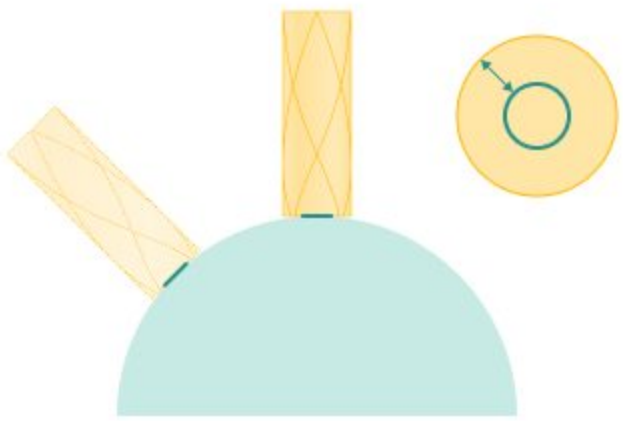
Center. The processing is carried out on the outer radius of the cylinder, which is located at a certain distance from the top of the cutting edge.



Height. The processing is done on the outer radius of the cylinder, which is located at a certain distance from the top of the cutting edge. The distance can be adjusted through the nested parameter Value.



Parameter. The processing is done with the flat part of the cutting tool, which is located at a certain distance from the top of the cutting edge. The distance can be adjusted through the nested parameter Value.



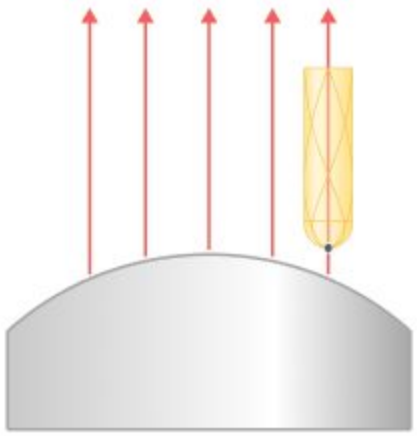
Value. Parameter determining distances.

Tool Orientation:

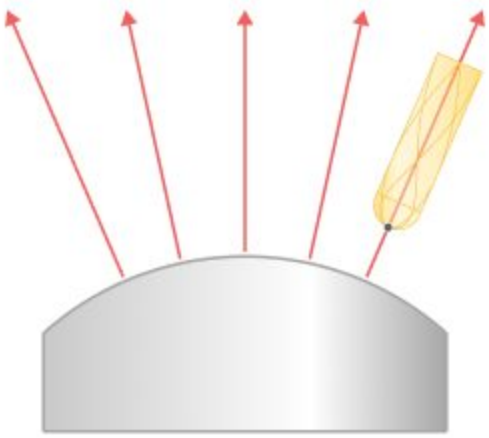
Controls tool axis orientation.

▼ Click here to expand..

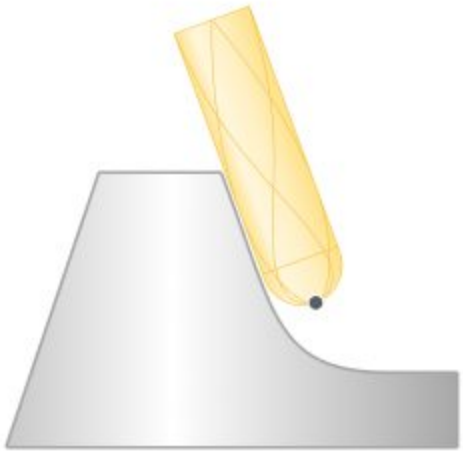
Fixed. Throughout the toolpath, the tool maintains a fixed orientation, defined by the initial positions of the machine's rotary axes as configured in the Setup tab.



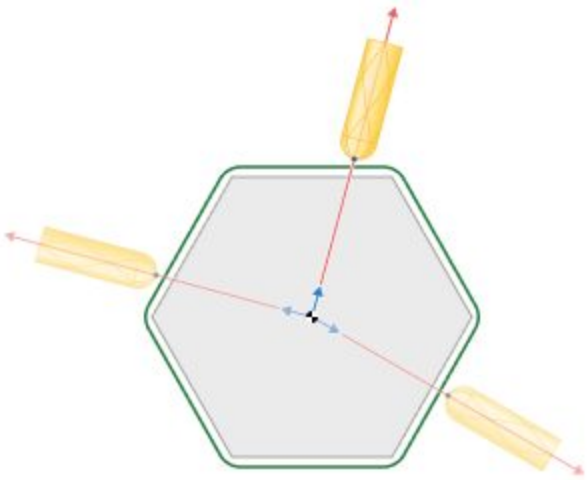
Normal to Surface. T he system orients t he tool axis normal to the Machining Surface.



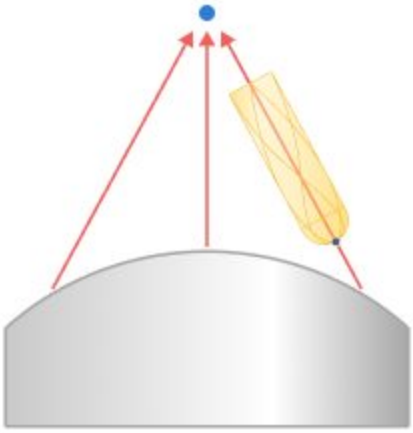
Flank. The tool's side part machines the surfaces.



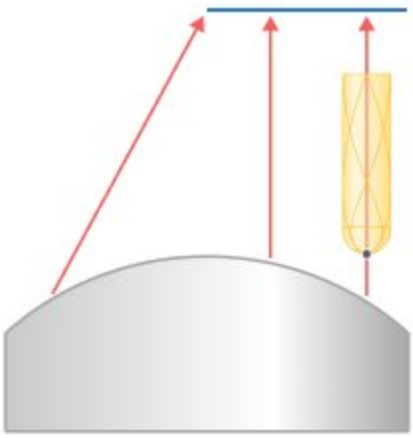
To Rotary Axis. The system directs tool's axis vector from the designated Rotary Axis.



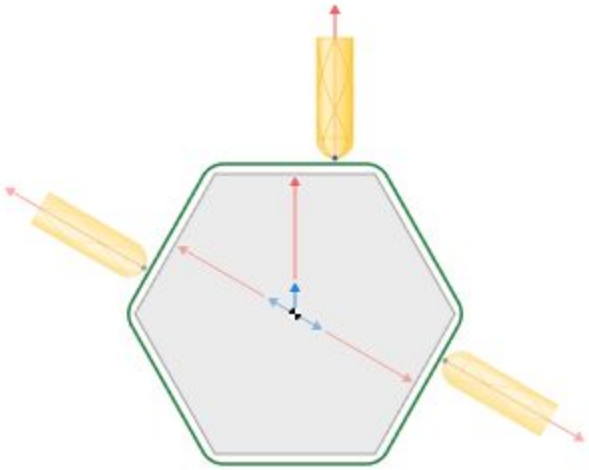
Through Point. The system directs tool's axis vector towards the designated point (pole).



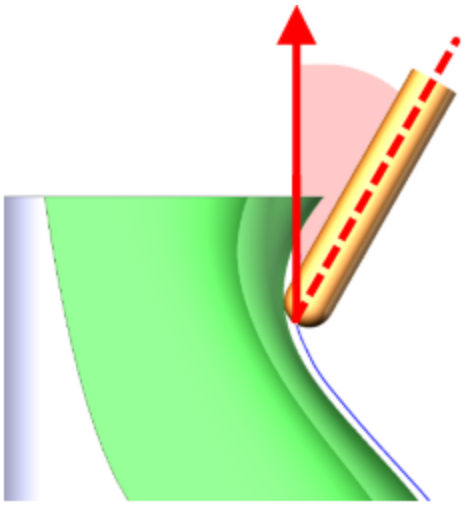
Through Curve. The tool axis vector targets the nearest point on the Tilt Curve specified in the Job Assignment.



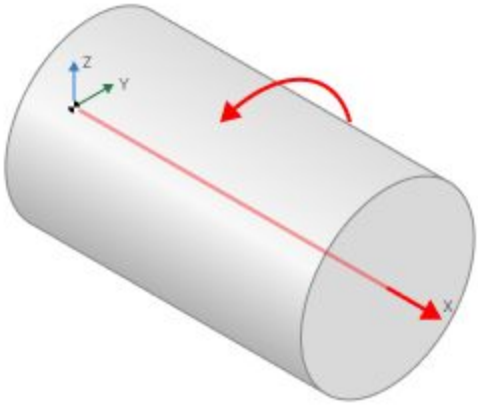
Perpendiculat to Toolpath. The tool's axis stands perpendicular to the tangent of the trajectory while being parallel to the Rotary Axis.



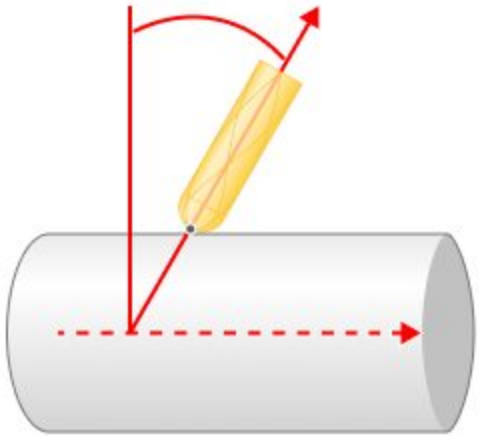
Vertical Clearance Angle. When the flag is set, the system automatically tilts the tool to the specified angle at areas where the Machining Surface displays a steep or negative incline (with the surface normal pointing down relative to the tool's axis), making it possible to machine simple areas with undercuts.



Rotary Axis. Setting the flag locks one of the components of the tool orientation vector, maintaining a constant Side Angle between the tool axis and the perpendicular to the specified axis (or the axis itself if the Tool Orientation is set as Perpendicular to Toolpath).

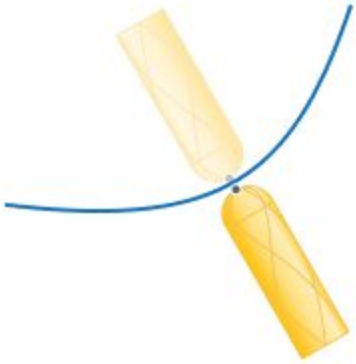


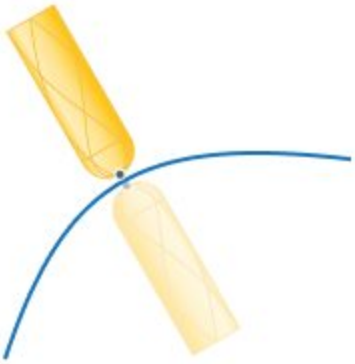
Side Angle. Specifies the inclination angle value of the tool axis relative to the axis designated as the Rotary Axis.



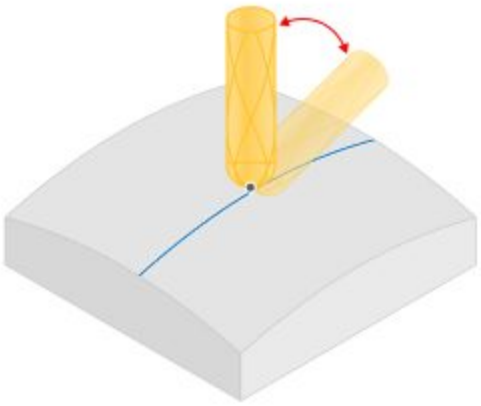
End Side Angle. Specifies the inclination angle value of the tool axis relative to the axis designated as the Rotary Axis at the end of the toolpath.

Inverse Tool Axis Direction. Alters the tool axis vector direction. Use this parameter if CAM system incorrectly attempts to machine surfaces from the opposite side to the desired one or if it fails to generate a toolpath..



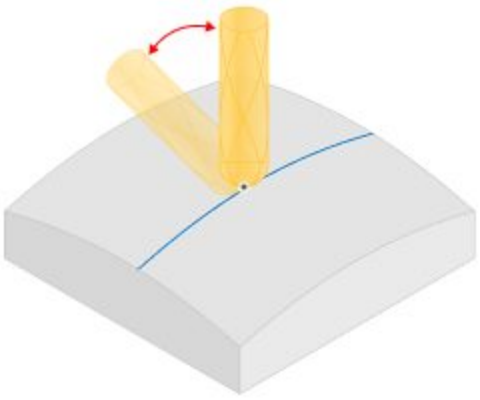


Lead Angle. It is a n additional tool tilt angle within the plane of motion direction.



End Lead Angle. It is a n additional tool tilt angle within the plane of motion direction at the end of the toolpath.

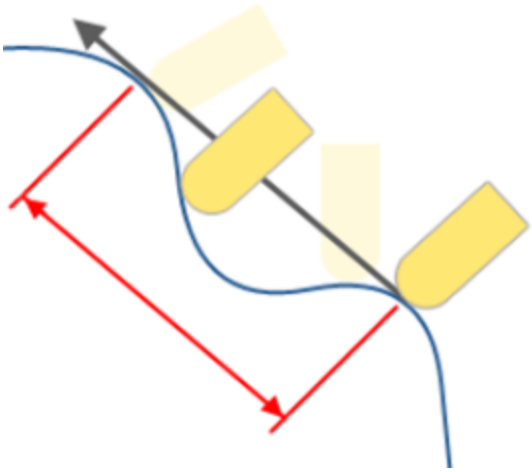
Lean Angle. It is a n additional tool tilt angle within the plane ortogonal to the motion direction.



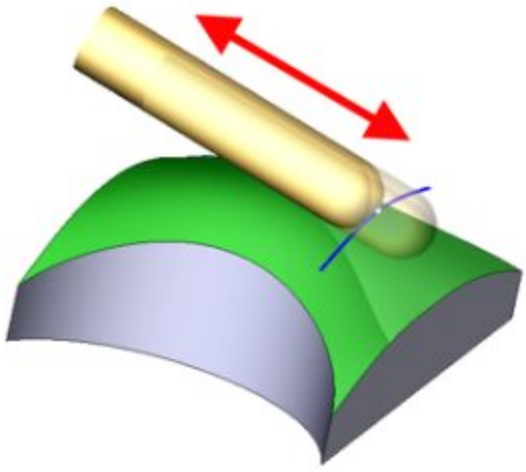
End Lean Angle. It is a n additional tool tilt angle within the plane ortogonal to the motion directionat the end of the toolpath.

Inverse Lead Angle. When flagging, t he lead angle sign becomes the opposite when the direction of tool movement changes.

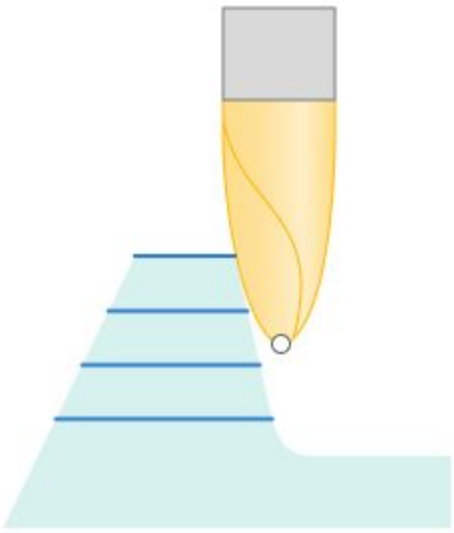
Normal Blend Distance. The distance along the toolpath where the tool's tilt angle remains constant.

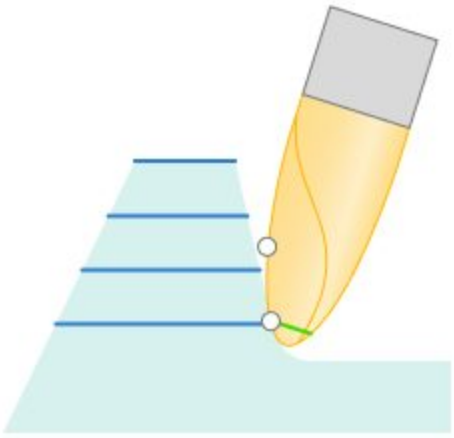


Axial Displacement. The feature shifts the contact point of the tool and the Machining Surface along the tool's axis.

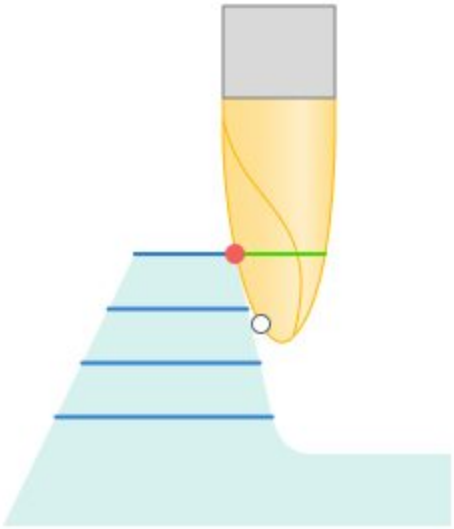


Contact Points. C hange the tool's contact point from the initial line to the final line.

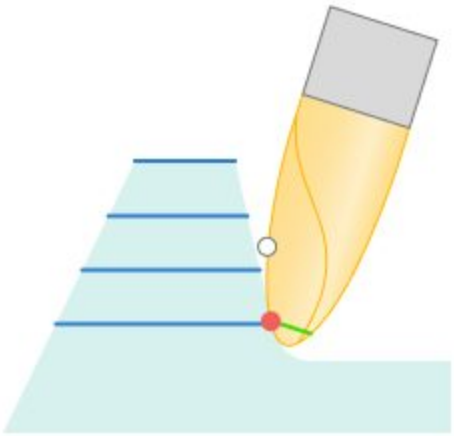




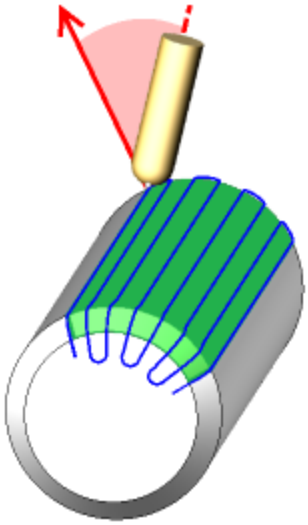
Start param. Initial point of contact



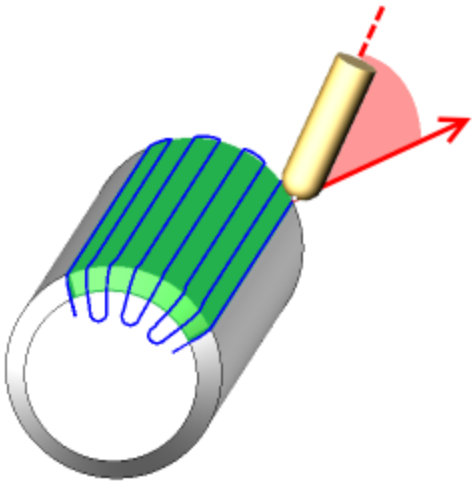
End param. End point of contact



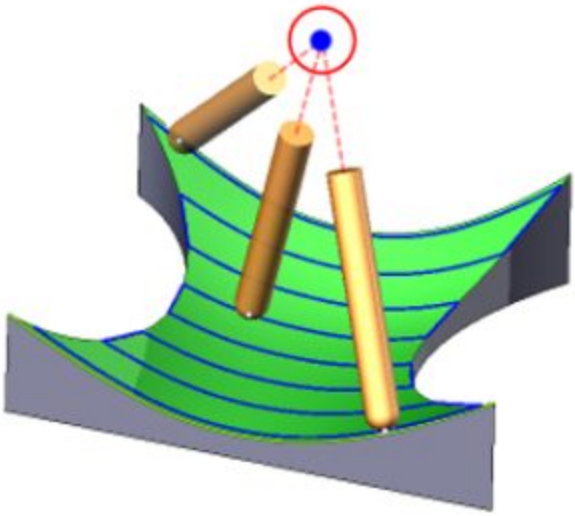
Lean Angle 1. If the Tool Orientation is set as To Rotary Axis it is the angle by which the tool deviates in the positive direction from the initial vector in a plane perpendicular to the Rotary Axis.



Lean Angle 2. If the Tool Orientation is set as To Rotary Axis it is the angle by which the tool deviates in the negative direction from the initial vector in a plane perpendicular to the Rotary Axis.



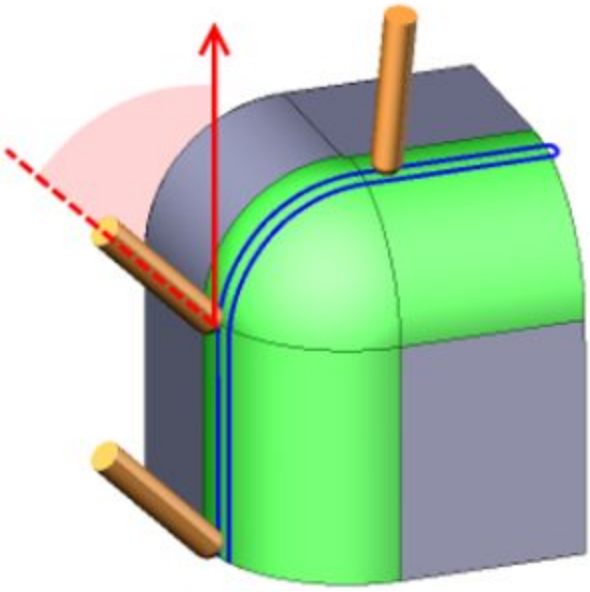
Origin. Specifies the pole coordinates if the Tool Orientation is set as Through Point.



Limit Rotation Angles:

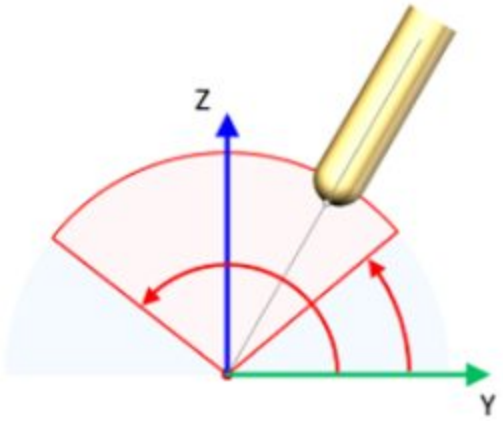
When the flag is set, it specifies the limit values of the tool's angular position relative to the specified axis.

▼ Click here to expand..

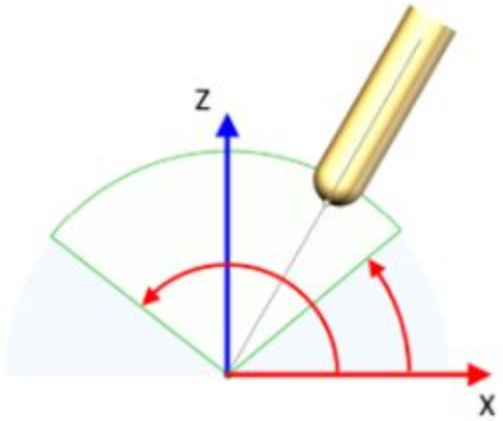


▼ Click here to expand..

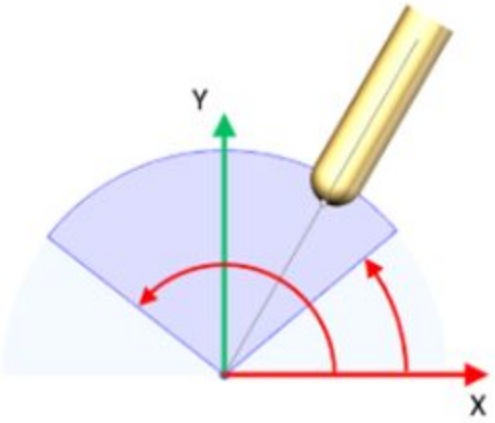
Around X (YZ). Activating the flag defines the tool's tilt limit values relative to the X axis within the YZ plane.



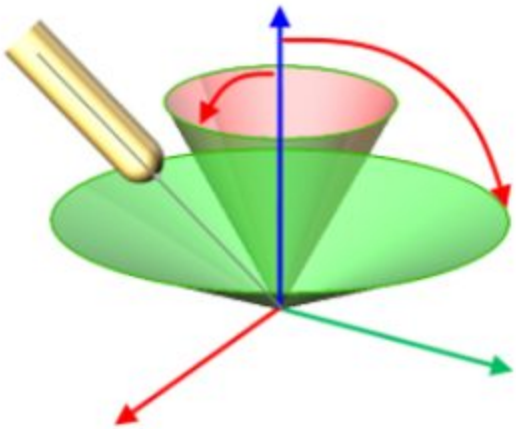
Around Y (XZ). Activating the flag defines the tool's tilt limit values relative to the Y axis within the XZ plane.



Around Z (XY). Activating the flag defines the tool's tilt limit values relative to the Z axis within the XY plane.



Conical limit. Activating the flag establishes maximum and minimum tilt angles of the tool relative to the designated axis, constrained by two cones with specified angles.

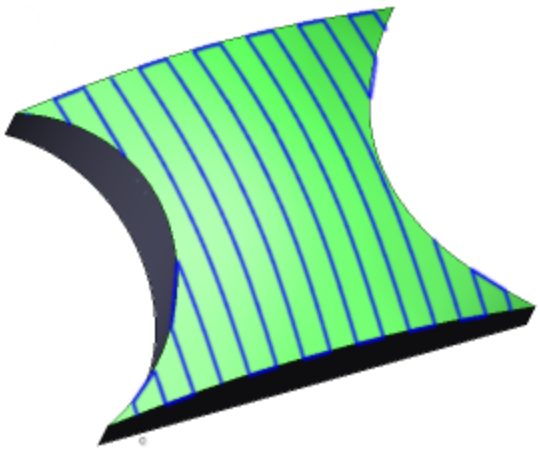


Job zone.

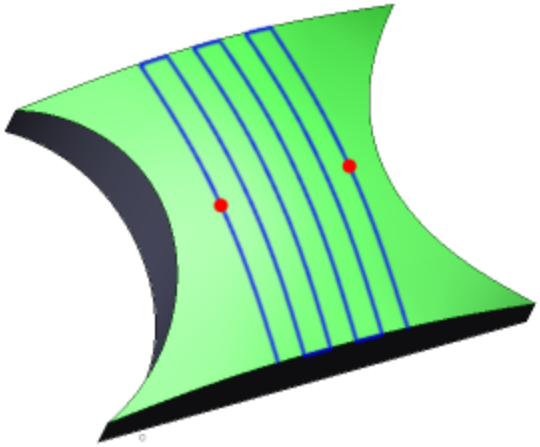
Enables additional modification of the working area initially selected on the Job assignment tab.

▼ Click here to expand..
Margins. For strategies that run parallel to planes, there are constraints on the positioning of slicing planes.

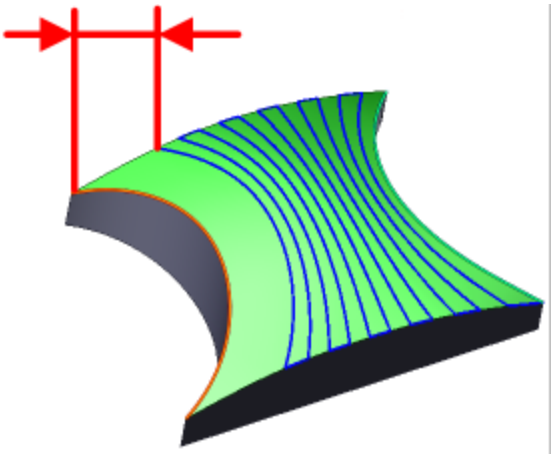
Full. There are no constraints on the formation of slicing planes.



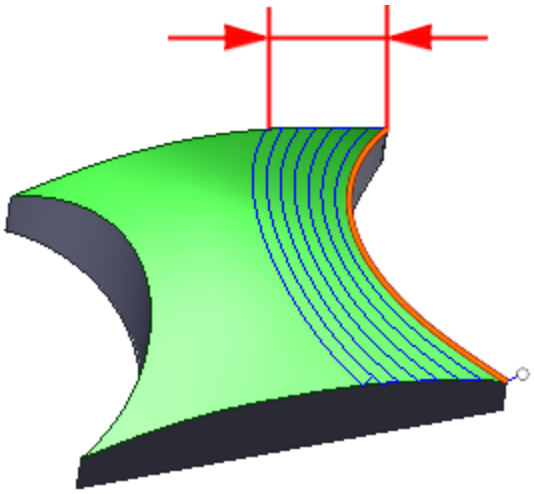
Points. Arrange slicing planes between two designated red points. Use the mouse to drag these points to the appropriate locations in the graphic window.



Start Margin. I n curve or surface-based strategies, the parameter determines the offset from the First Curve or Surface, i where the toolpath formation begins.



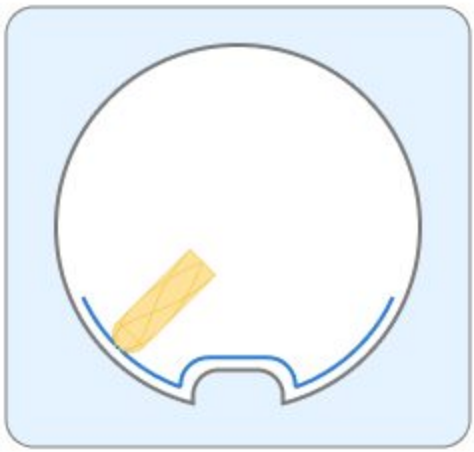
Zone Width. In curve or surface-based strategies, the parameter determines the distance from the Start Margin to t he last machining pass.



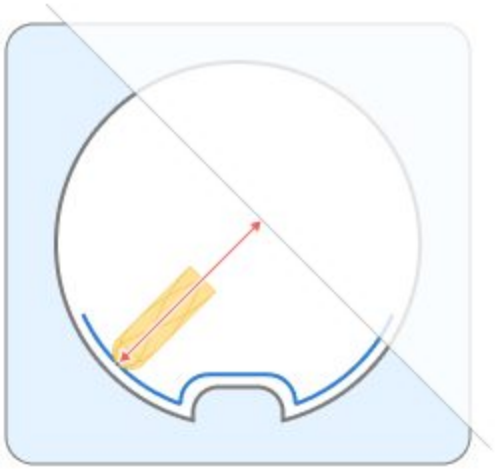
End Margin. The parameter defines the offset from the Second Curve or Surface, up to which the toolpath is formed.

Limit Job Zone. In the Across Curve strategy, it allows using for the trajectory only the section of the Machined Surface closer to the current point on the curve.

Machining from inside. The parameter allows you to process the internal surfaces of the part (project the trajectory onto the internal surfaces of the part)



Max retract height. The projection zone is defined. Projection will not be performed on surfaces located below this level. Checks for tool-part collisions only below a specified height



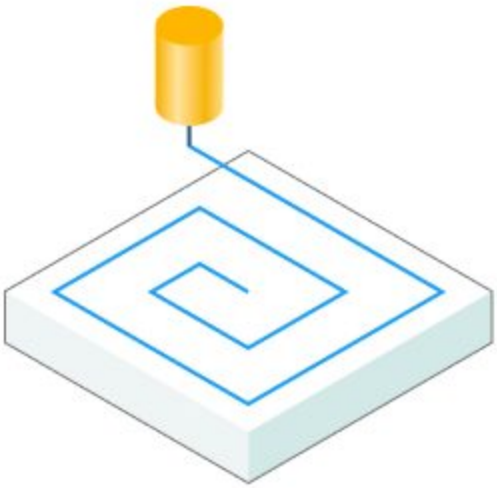
Some parameter group works similarly to the Plane finishing operation. Details

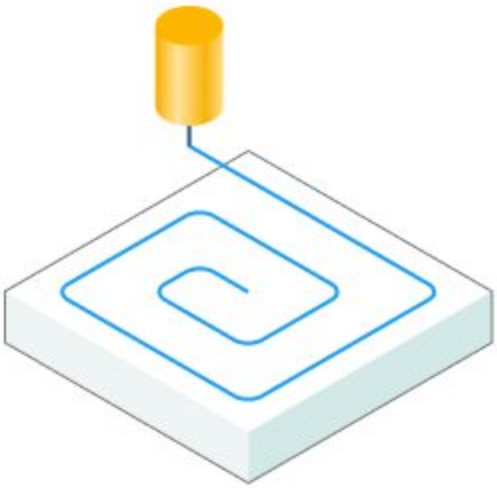
Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

▼ Click here to expand..

Blend Distance. Smoothing occurs within this distance inside the sharp corner.



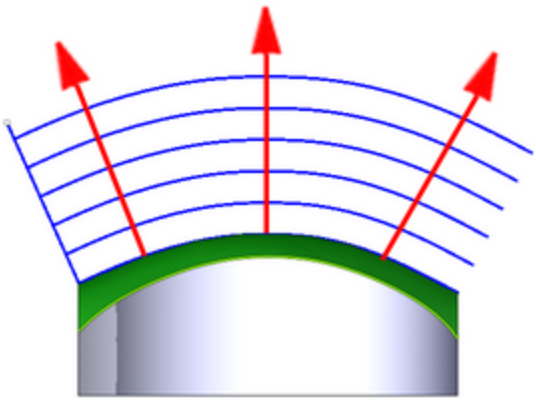


Roughing Passes.

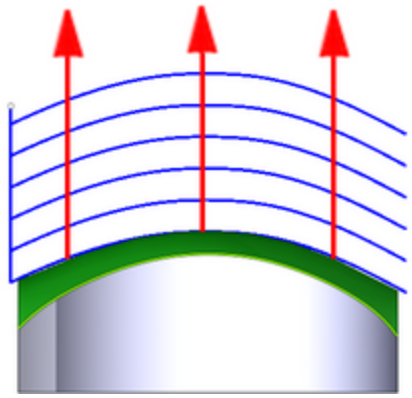
Toolpath is offset in plane by the Roughing passes thickness amount. Additional passes are added to the contour by the value specified in the parameters with the specified number of steps.

▼ Click here to expand..
Orientation of roughing passes:

▼ Click here to expand..
By Surface Normal. Calculating roughing paths involves offsetting the finish paths along the surface normal direction.

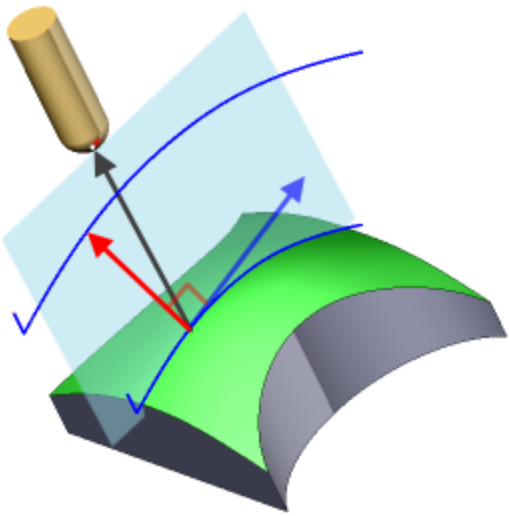


Along Tool Axis. Calculating roughing paths involves offsetting the finish paths along the tool's axis.

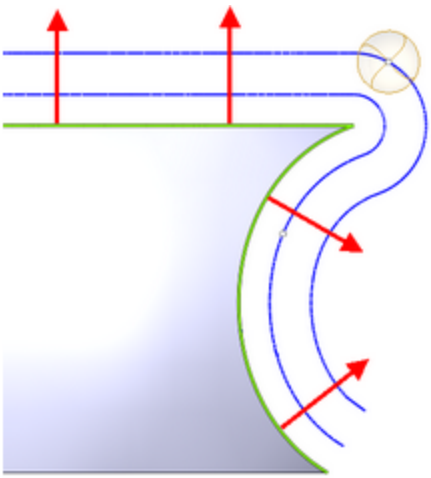


Tool orientation

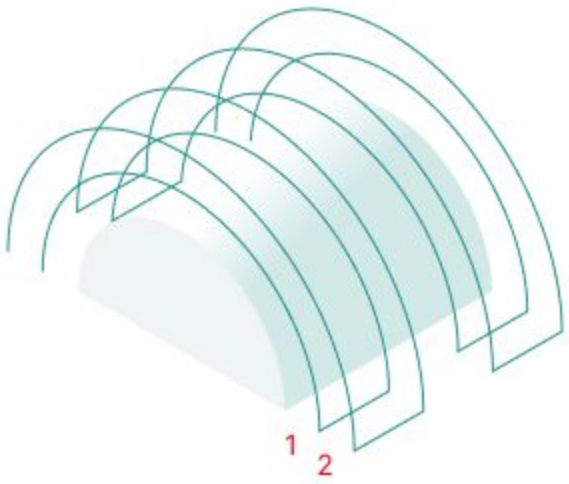
In Tool Plane. Calculating roughing paths involves offsetting the finish paths within the plane of tool motion. This approach is effective for the strategy Parallel to Vertical Plane and when surfaces are machined with the end part of the tool.



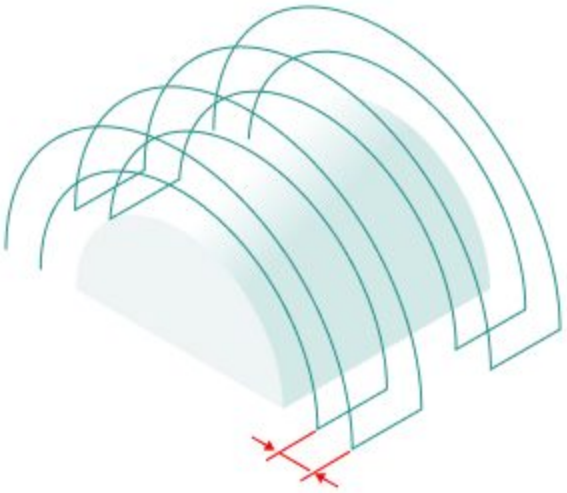
Perpendicular to Tool Axis. Calculating roughing paths involves offsetting the finish paths within the plane perpendicular to the tool axis,. This approach is effective for the strategy Parallel to Horizontal Plane.



By Default. Displays the default calculation method adopted in the system.
Count. It is the count of rough passes.

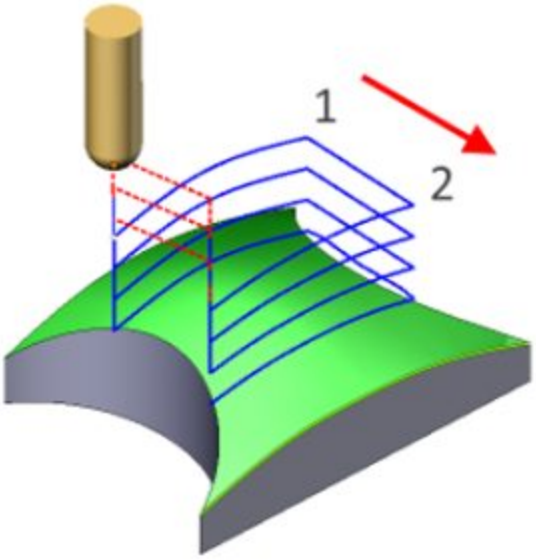


Step. It is the distance between rough passes.

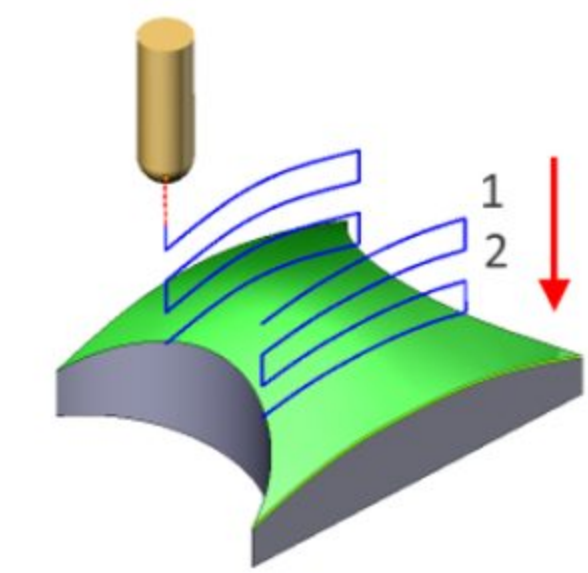


Sort By. Specifies the execution order for rough passes.

Levels. Rough passes are executed in the order of the layers' arrangement.



Strings. Rough passes are executed in the order of the strings' arrangement.

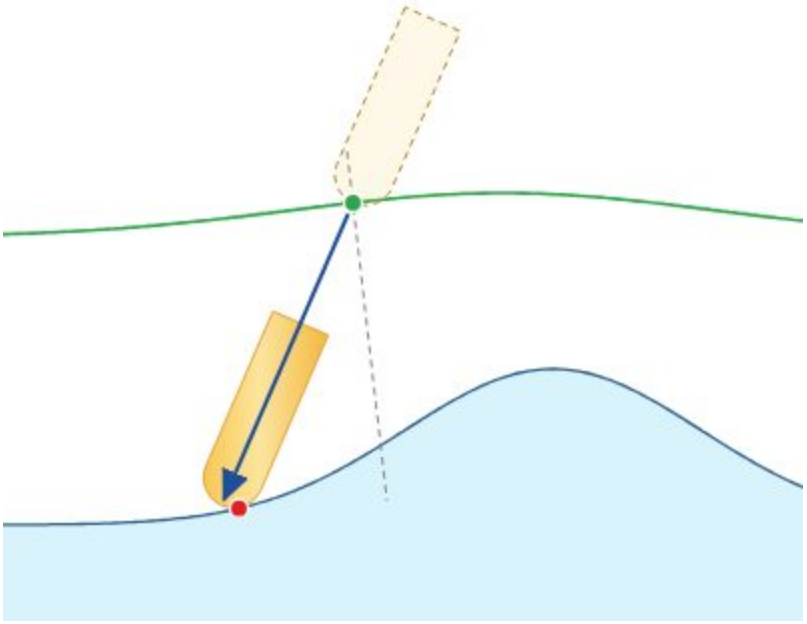


Project Toolpath onto the Part:

By setting the flag, the system generates tool paths for processing the surfaces identified as Surfaces for Projection in the Job Assignment by mapping the path derived from the Machining Surface onto the Surfaces for Projection. If no Surfaces for projection are selected, the toolpath is projected onto the machining surfaces themselves.

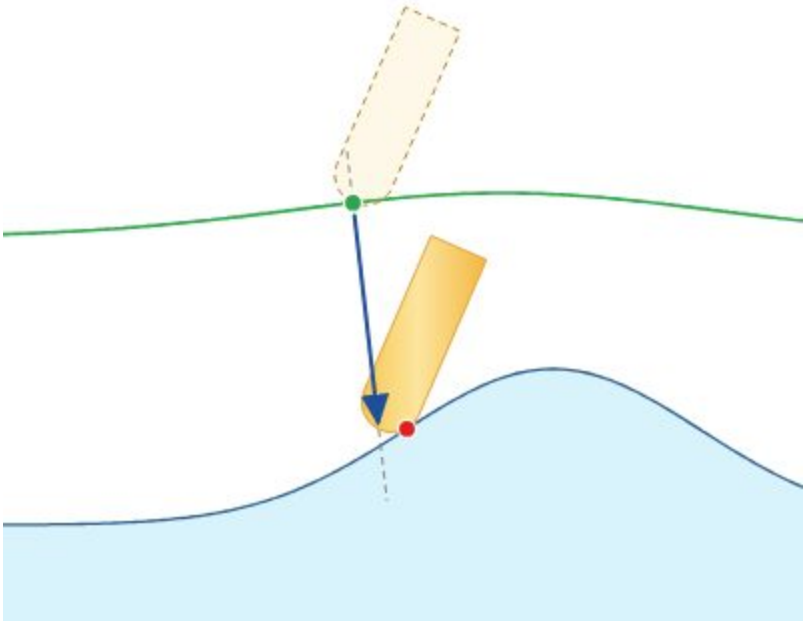
▼ Click here to expand..
Type. Determines the projection direction.

Along tool axis. Each toolpath point moves along the tool axis until the tool contacts the surfaces.

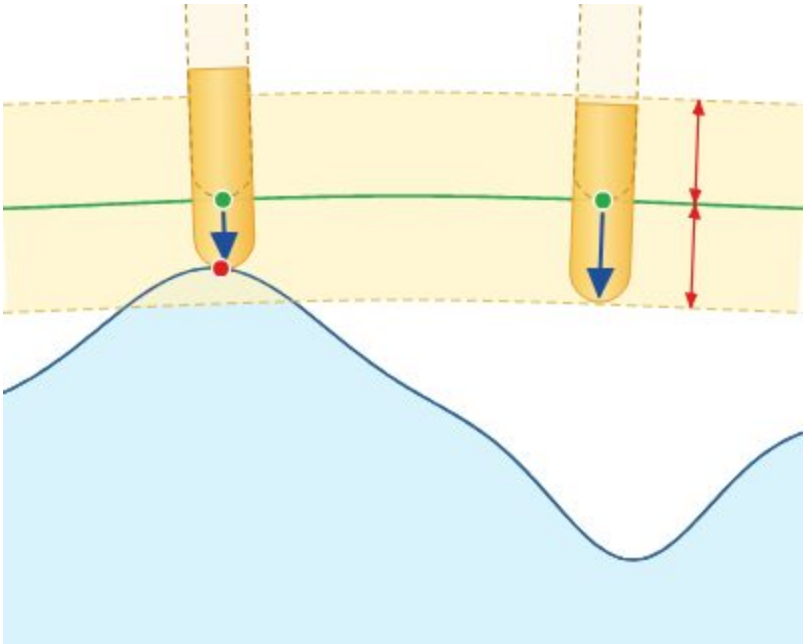


By surface normal. The tool axis is determined by the tool orientation settings, and the tool moves along the normal of the machining surface. If Surfaces for projection are selected in the job assignment, contact with them is detected over the entire tool body, including the taper and undercut. The search

extends by Max deviation in both directions from the original point along the normal. If no Surfaces for projection are selected, projection works as in earlier versions.



Max deviation. Defines how far the toolpath can move from its original position along the surface normal. When Surfaces for projection are selected, contact is searched for only within this range. Set the value to at least the greatest distance between the machining surfaces and the surfaces for projection. The default of half the tool diameter may be insufficient.

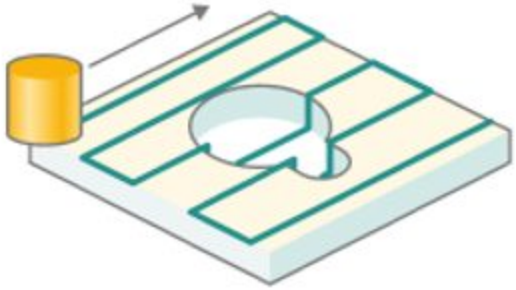


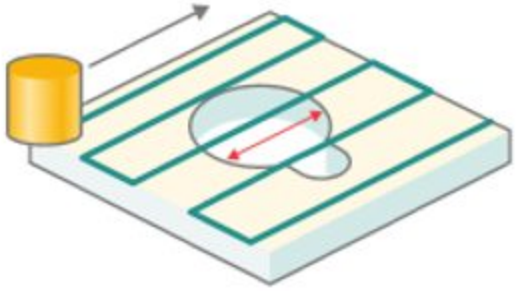
Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath.

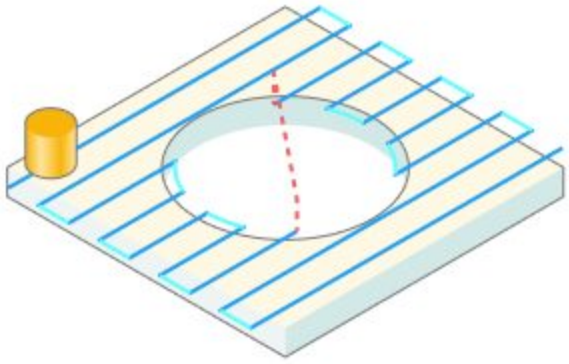
▼ Click here to expand..

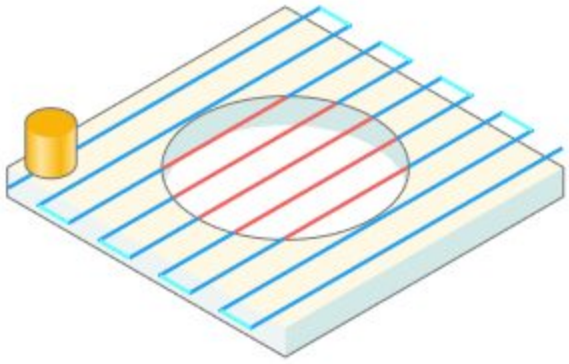
Hole Capping. Activating this option allows the system to skip holes with diameters that are equal to or smaller than the specified dimension.



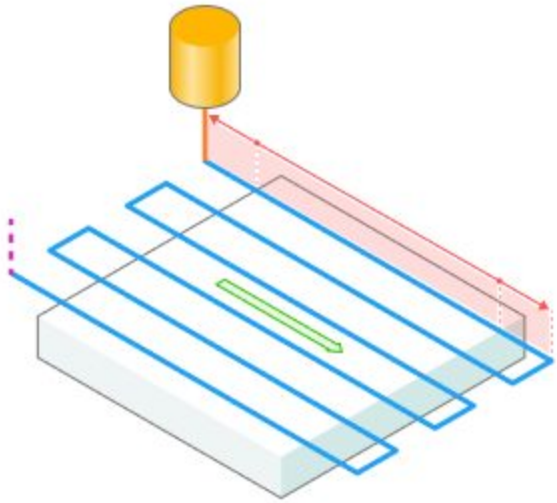


Gap capping. By enabling this feature, the system can close toolpath gaps smaller than the specified dimension.

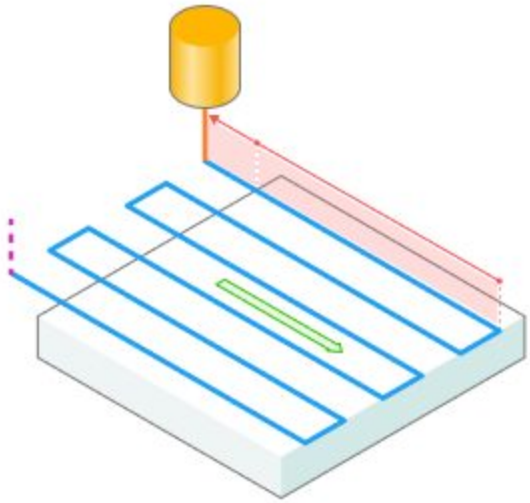




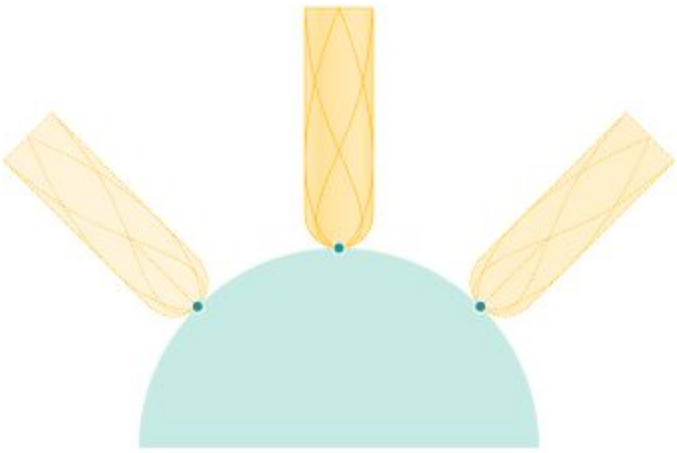
Extend (+)/Trim (-) passes. With this setting, users can extend or trim the initial and final segments of the toolpath.



Start. Entering a positive value, the system extends the start of the working passes of the toolpath by the specified amount. Entering a negative value shortens the beginning of the passes.



End. Entering a positive value, the system extends the end of the working passes of the toolpath by the specified amount. Entering a negative value shortens the end of the passes.



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Scallop



Application area:

The Scallop toolpath starts from the curves lying on the part surfaces and is generated by repeatedly offsetting those curves inwards until the curves collapse. This method creates a toolpath similar to equidistant strategy So basically it is an equidistant toolpath except the offsetting occurs t in 3d space on the machining surfaces, not on a plane. The toolpath achieves a consistent scallop height regardless of the steepness of machining surfaces. Another advantage is the minimal amount of linking moves together with respected climb/conventional milling type. The operation is best suited for semi-finishing and finishing.

Job assignment:

Machining Surfaces. Select various surfaces of the part as the working task. The system detects the starting curves as the curves of contact of the cutter with those surfaces.

Tilt Curve. This is t he curve that the tool axis vector aims at when setting Tool Orientation specification Through Curve.

Job Zone. Use the job zone to trim the passes outside the specified 2d containment areas. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that should avoid machining in the current operation. The system generates restricted or cutted toolpath depending on the option you choose. Details.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

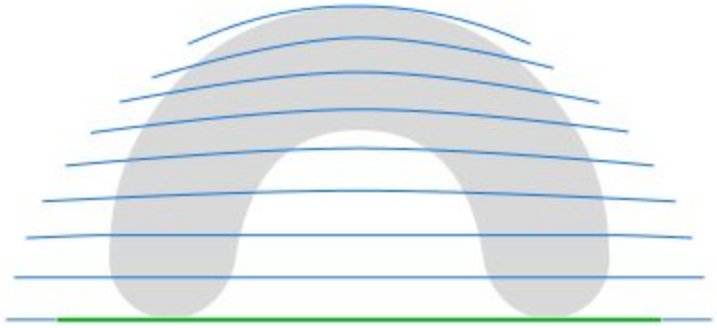
Strategy:

Strategy:

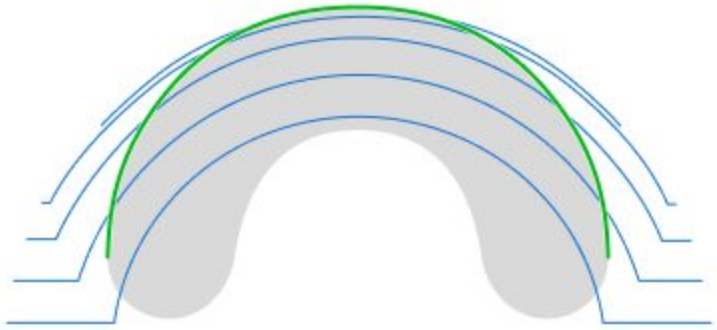
These parameters allows the user to achieve a required geometry of toolpath:

▼ Click here to expand..
Start From. Parameters determining the construction of the initial toolpath curves.

Bottom. The toolpath begins at the Bottom Level of machining.

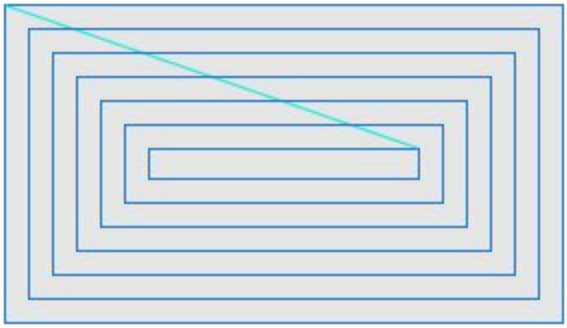


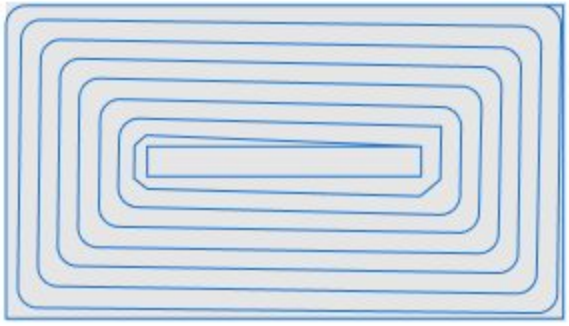
Top of Vertical Faces. The process begins with the silhouette curves, omitting the vertical walls.



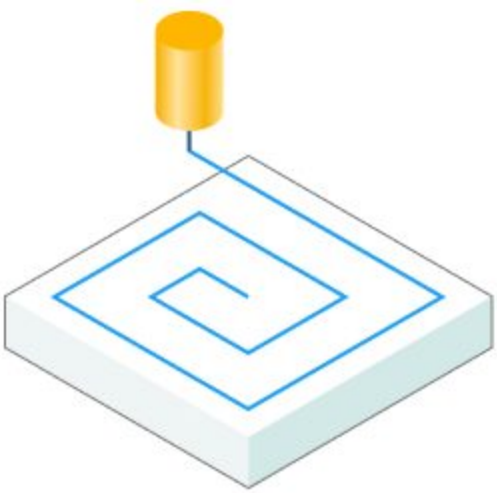
Step. The maximum allowed width of cut

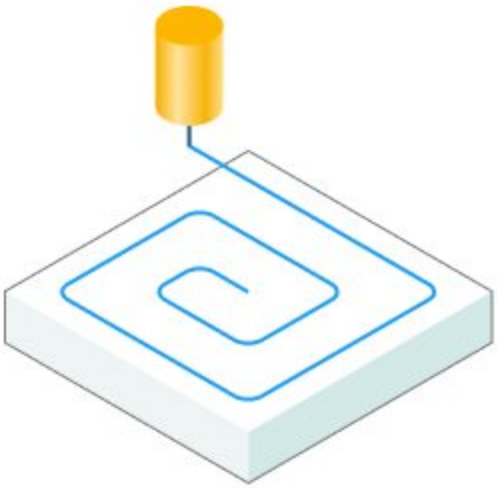
Spiral Machining. Setting the flag configures the toolpath as a continuous spiral to minimize links.





Smooth Corners. Setting the flag smooths sharp corners in the toolpath.





Motph Passes. Activating the flag alters the strategy for creating the toolpath, resulting in a path with a consistent step. Also, w hen unprocessed "islands" exist within the Machining Surfaces, setting the flag enables the generation of toolparhes with smooth transitions from the "island" contours to the outer contours of the Machining Surfaces.

Milling Type:

This feature enables users to select the necessary milling direction (climb or conventional) during the toolpath calculation process.

The parameters of Milling Type are the same as in the Waterline Roughing operation. Details.

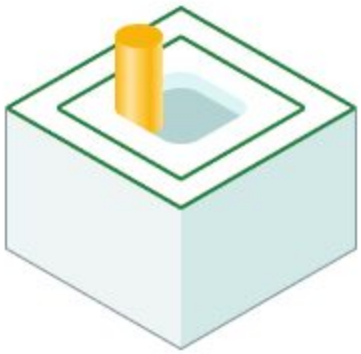
Sorting:

Controls the sequence of toolpath passes during surface machining.

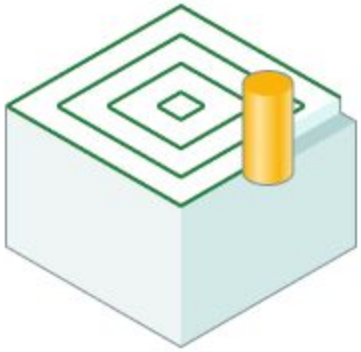
▼ Click here to expand..

Machining Direction. Controls the direction of tool motion on toolpath curves.

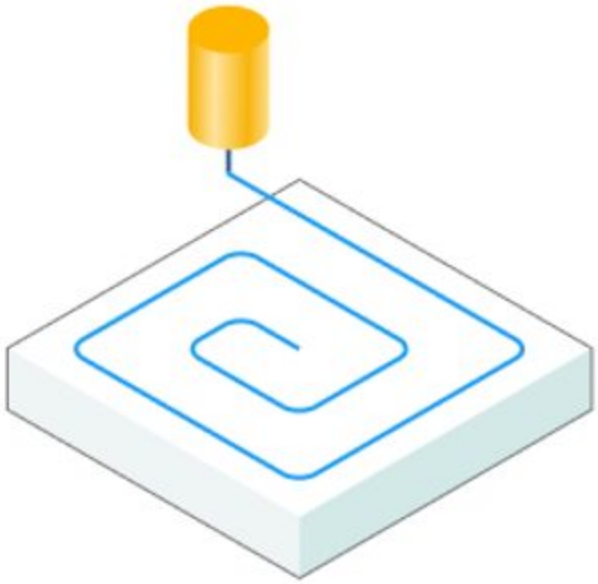
Inside-out. The tool transitions from the central toolpath curves to the outer ones.

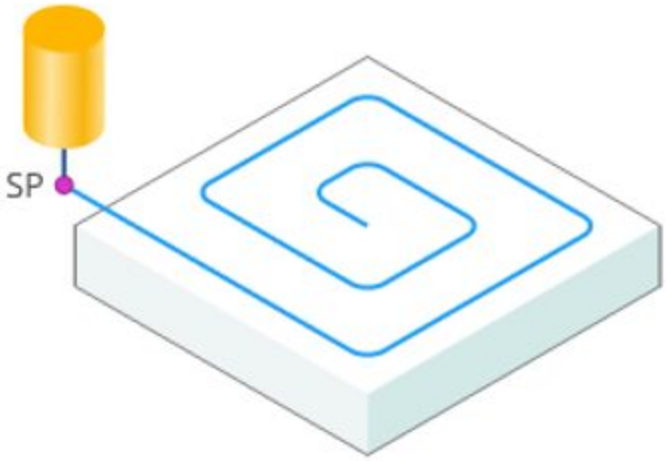


Outside-in. The tool transitions from the outer toolpath curves to the central ones.



Start Point. The point defining the beginning of the plunge.





Machining Levels:

It defines the range along tool axis for the machining. Specify the Bottom Level as in the Waterline Roughing operation. Details.

5 Axis Conversion:

You can control the tool's position when executing operations on a five-axis machine. Details.

The parameters of Tool orientation are the same as in the 5D Surfacing operation. Details.

Limit Rotation Angles:

When the flag is set and the 5 Axis Conversion option is on, it specifies the limit values of the tool's angular position relative to the axes. The parameters of Limit Rotation Angles are the same as in the 5D Surfacing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

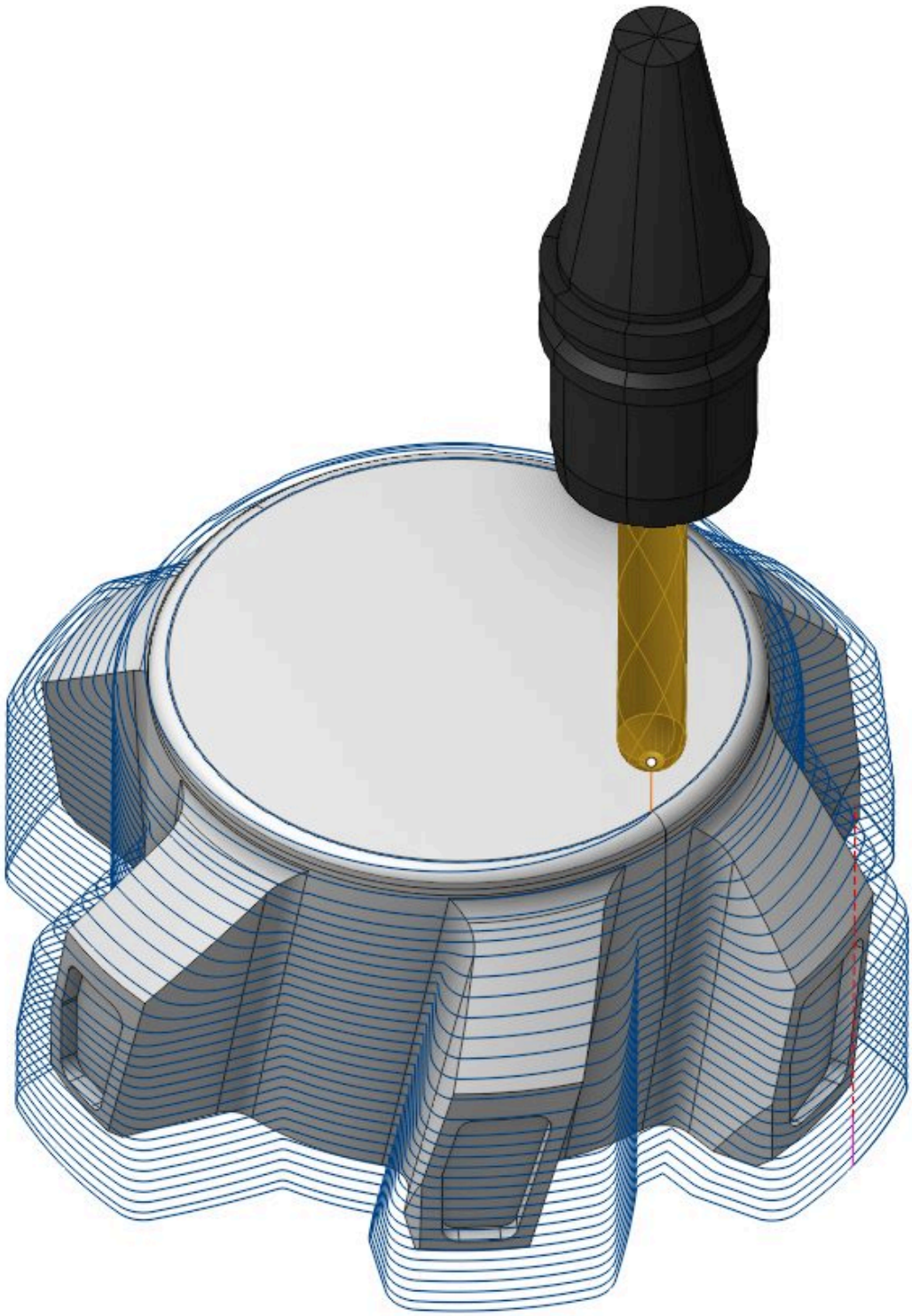
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

3D Helical



Application area:

The operation generates continuous helical passes with the given vertical stepover between the top and the bottom level. It is effective when machining slope or vertical surfaces.

Job assignment:

Machining Surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

Job Zone. Use the Job Zone to form the passes within or outside the specified 2d containment areas. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that s hould avoid machining in the current operation. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

These parameters allows the user to achieve a required toolpath:

Milling Type:

This feature enables users to select the necessary milling direction (climb or conventional) during the toolpath calculation process.

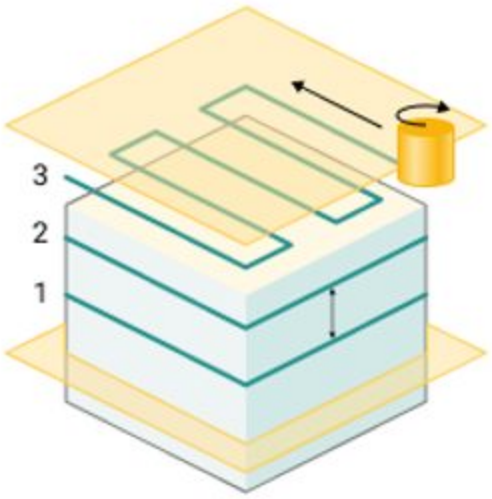
The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Sorting:

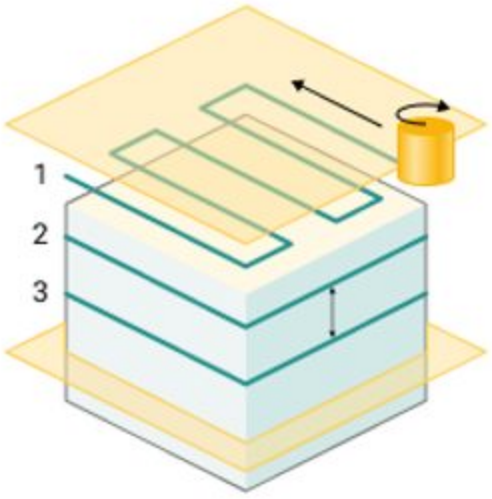
Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Start from. Specifies the starting level for tool motion:

Bottom. The tool moves from the bottop to the top of the Job Assignment areas.



Top. The tool moves from the top to the bottom of the Job Assignment areas.



Step.

Material removed in one pass along the tool axis between the Top and Bottom levels. It i s a vertical distance between any two adjacent complete helix turns. This parameter group works similarly to the Waterline Roughing operation. Details.

Adjust Step:

The parameter changes the tool passes to allow machining to the defined level.

Smooth Radius:

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

Machining Levels:

It defines the range along tool axis for the machining. These parameters are the same as in the Waterline roughing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

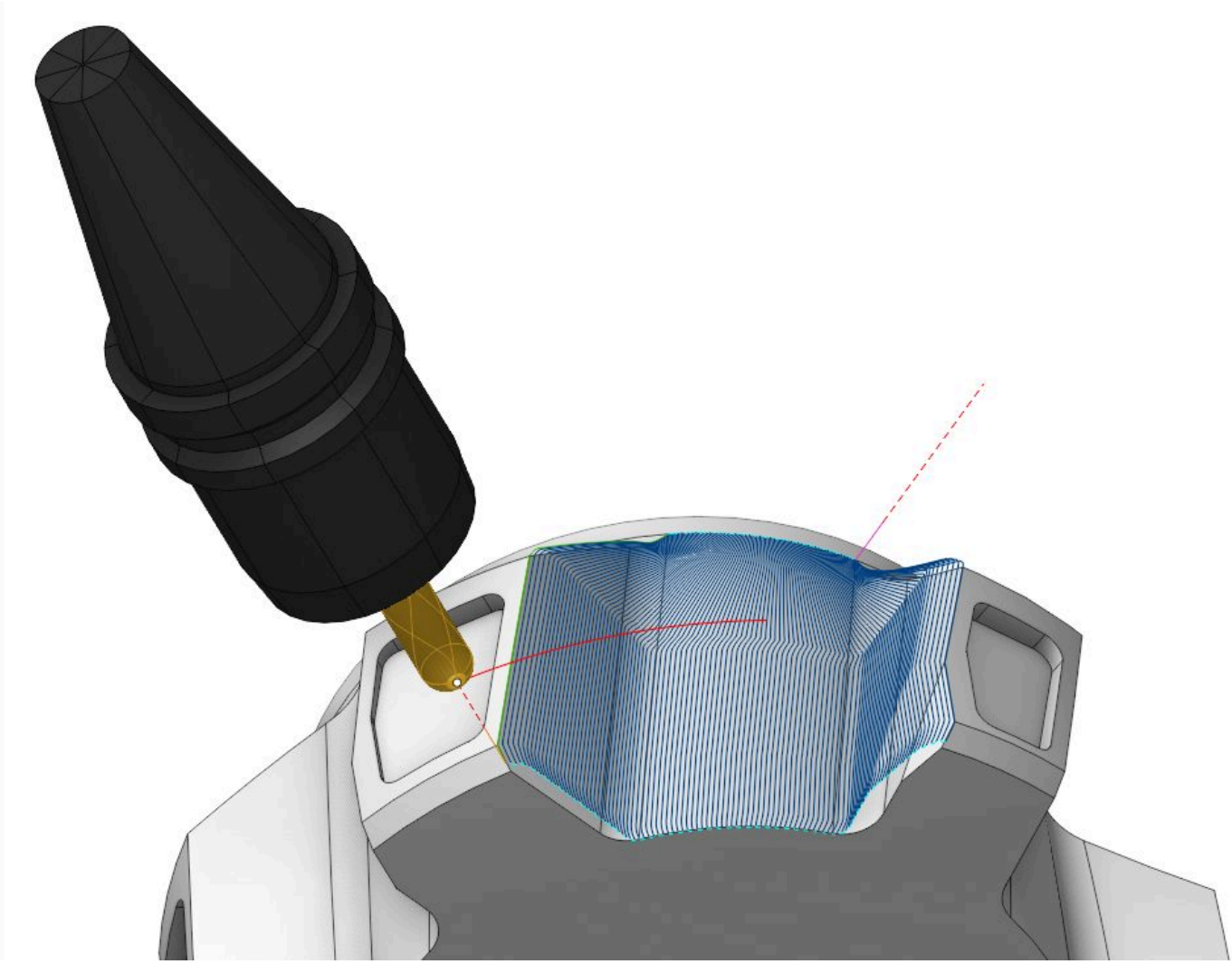
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Morph



Application Area:

The system generates toolpath passes as lines spaced at a specified distance, each shaped by a smooth transition from the first defined curve to the second

Job Asssignment:

Machining Surfaces. The system uses the Machining Surfaces to restrict the area of machining. Morph generates passes only where a tool has contact with those surfaces. Morph generates passes not between two curves you set in the job assignment, but between two Machining Surfaces contact curves which are closest to those two curves you specify as the First and the Second curve.

First Curve. It is t he starting line from among the two lines between which the system generates the toolpaths.

Second Curve. It is t he finishing line from among the two lines between which the system generates the toolpaths.

Sync lines are used to improve the quality of morphing in difficult cases, especially when machining closed contours. A Sync Line specifies two corresponding points of the First and the Second Curve. You can use any types of curves as Sync Lines: edges, 3D curves, 2D geometry curves. They do not have to start and end precisely on the First and Second Curve. You can draw sync lines at the 2D geometry tab.

Tilt Curve. This is t he curve that the tool axis vector aims at when setting 5 Axis Conversion specification Through Curve.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

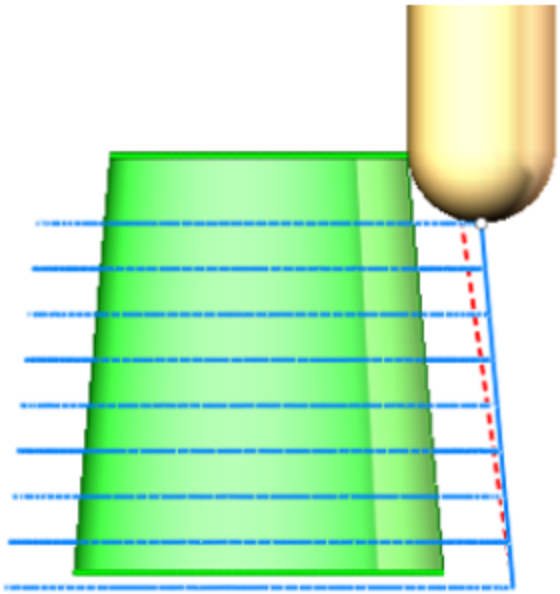
Strategy:

Strategy:

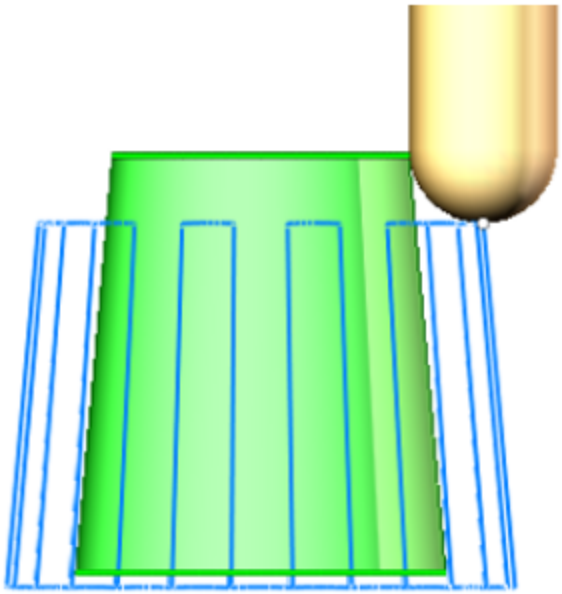
These parameters allows the user to achieve a required toolpath:

▼ Click here to expand..
Toolpath Direction:

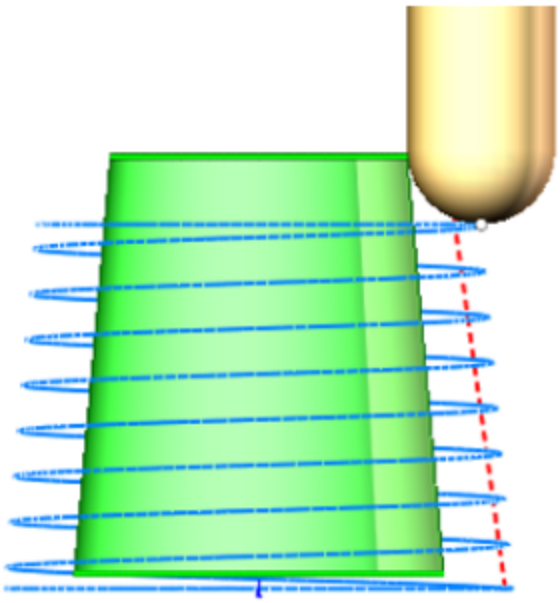
Along. The system generates tool passes parallel to the First and the Second Curves. This toolpath is calclulated based on Across toolpath. So if it look not good you can always switch to Across to find possible reasons out.



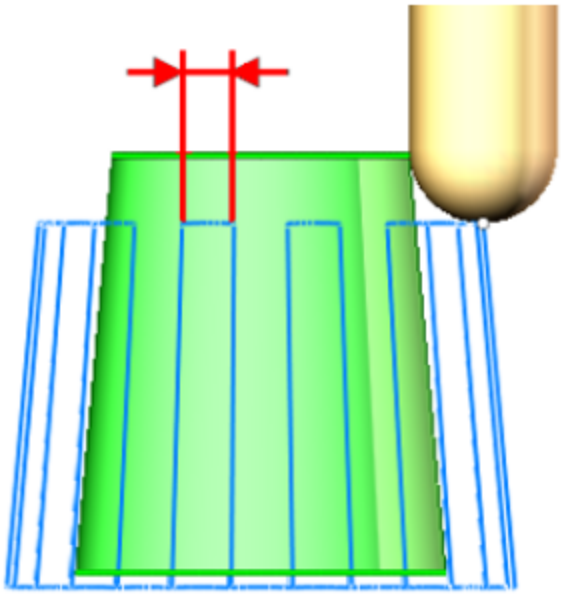
Across. The system generates tool passes by connecting corresponding points on the First and the Second Curves.



Spiral. The resulting toolpath is one smooth spiral that morphs between the First and the Second Curves. This toolpath is calclulated based on Across toolpath. So if it look not good you can always switch to Across to find possible reasons out.



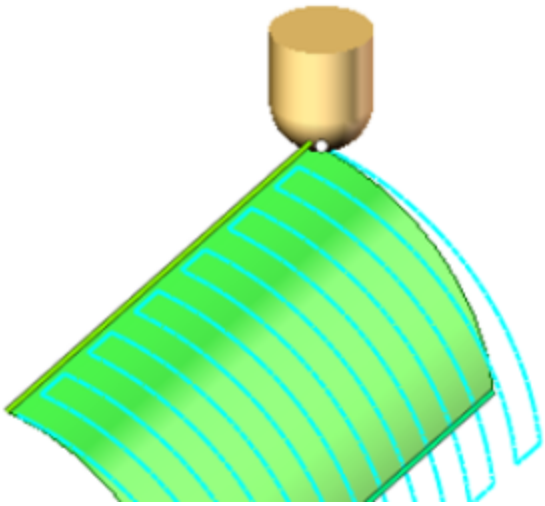
Step. The maximum allowed width of cut. Due to the nature of the toolpath the stepover varies from point to point, but it will never exceed this value.



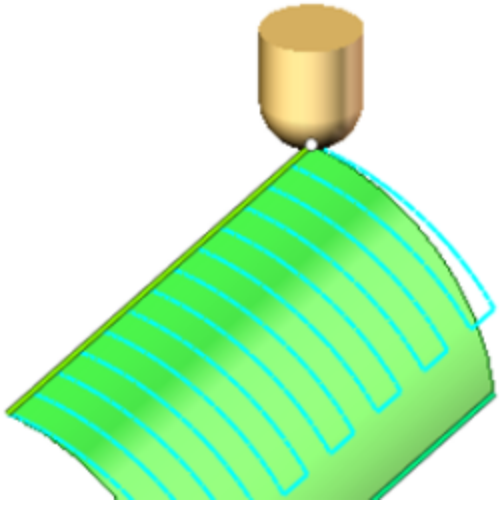
Tool Containment. This feature defines methods to develop toolpath curves, considering the tool's interaction with the Machining Surfaces.

▼ Click here to expand..

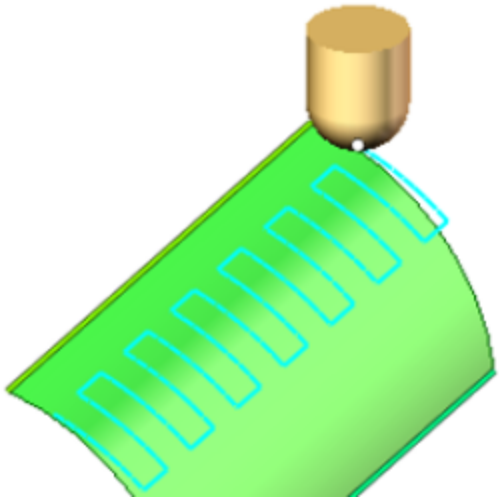
Contact. The system generates tool passes where the tool has contact with Machinig Surfaces, between two contact curves (point curves of Machining Surfaces and tool contact) which are closest to those two curves you specify as the First and the Second curve. Use this option if the First and Second curves are the boundary edges of Machining Surfaces.



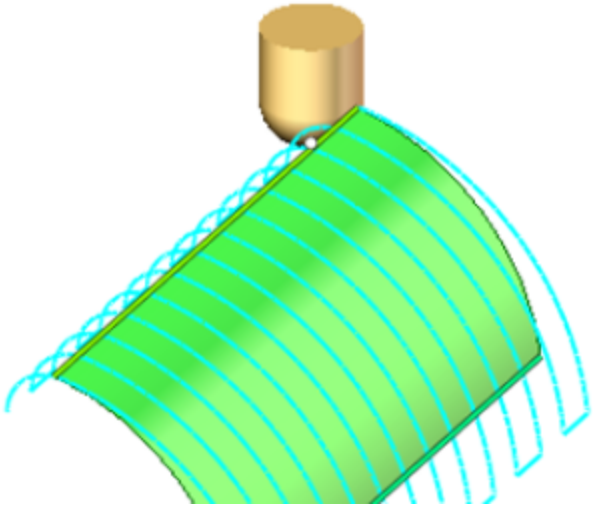
Center. The toolpath forms between the motion lines of the tool center on the First and Second Curves within the Machining surface area, ensuring no gouging occurs with the tool's current orientation.



Inside. The system shifts t he machining area's borders inward from the First and Second Curves by the radius of the tool.



Outside. The system shifts t he machining area's borders outwards from the First and Second Curves by the radius of the tool.



Milling Type:

This feature enables users to select the necessary milling direction (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Sorting:

Controls the organization of the toolpath during surface machining

Start Point. Activating the flag permits the input of coordinates marking the starting point of working passes in surface machining.

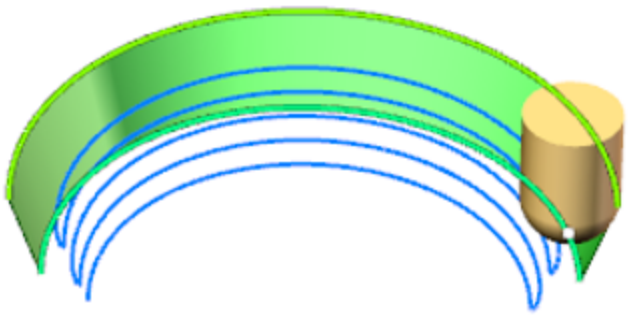
Machining from inside.

The parameter allows you to process the internal surfaces of the part (project the trajectory onto the internal surfaces of the part). This parameter group works similarly to the 5D Surfacing operation. In the section Job zone. Details

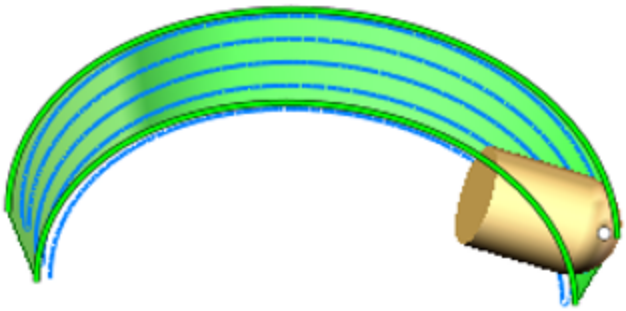
Tool Axis Orientation:

Controls the tool axis orientation during the machining process.

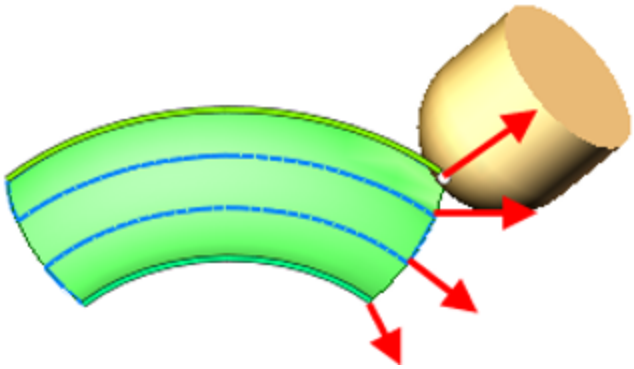
▼ Click here to expand..
▼ Click here to expand..
Fixed. The tool orientation is fixed throughout the toolpath and is determined by the initial position of the machine rotary axes, which is set at the Setup tab in the inspector.



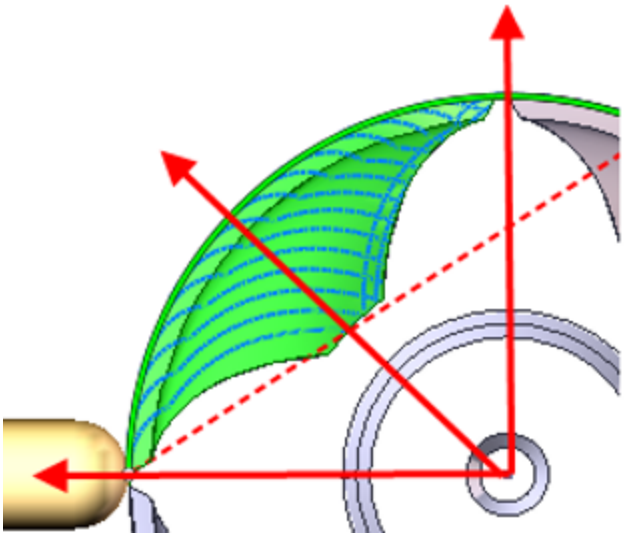
Normal to Curves. The tool is positioned by normal to the surface that is formed by connecting corresponding points of the First and the Second drive curves. Additionally, Lean angles may be applied to deflect the tool from this direction.



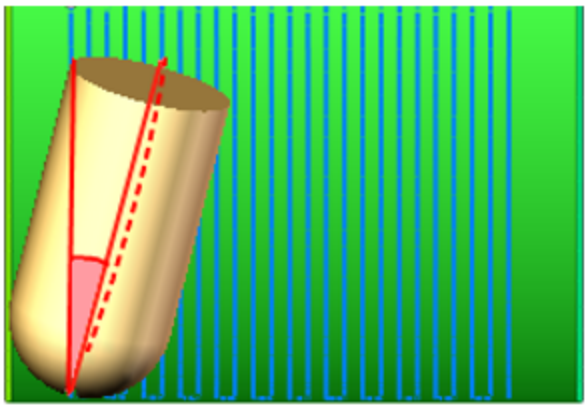
Normal to Surfaces. The tool is positioned by normal to the Machining Surfaces. The performance of the function depends on the type and accuracy of the elements in the 3D model of the part. Use it on your own risk.



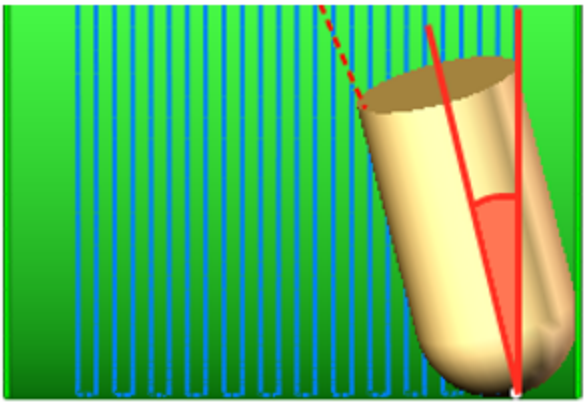
To Rotary Axis. The principal tool axis orientation is computed by subtracting the tool center point from the projection of this point onto the rotary axis. The Side Angle between the tool axis and the rotary axis can be either computed automatically or specified directly.



Lean angle 1. It is the angle by which the tool is inclined from the normal towards the Second Curve near the First Curve.



Lean angle 2. It is the angle by which the tool is inclined from the normal towards the First Curve near the Second Curve.



Alternate front side. Check this box if CAM system mistakenly tries to machine surfaces from the side opposite to the desired one or there is no toolpath at all.

Rotary Axis. Setting the flag locks one of the components of the tool orientation vector, maintaining a constant Side Angle between the tool axis and the perpendicular to the specified axis (or the axis itself if the Tool Orientation is set as Perpendicular to Toolpath). The parameters of Rotary axis are the same as in the 5D Surfacing operation. Details.

Side Angle. Specifies the inclination angle value of the tool axis relative to the axis designated as the Rotary Axis. The parameters of Side Angle are the same as in the 5D Surfacing operation. Details.

5 Axis Conversion:

You can control the tool's position when executing operations on a five-axis machine. Details.

The parameters of tool 's position are the same as in the 5D Surfacing operation. Details.

Limit Rotation Angles:

When the flag is set and the 5 Axis Conversion option is on, it specifies the limit values of the tool's angular position relative to the axes. The parameters of Limit Rotation Angles are the same as in the 5D Surfacing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the

final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

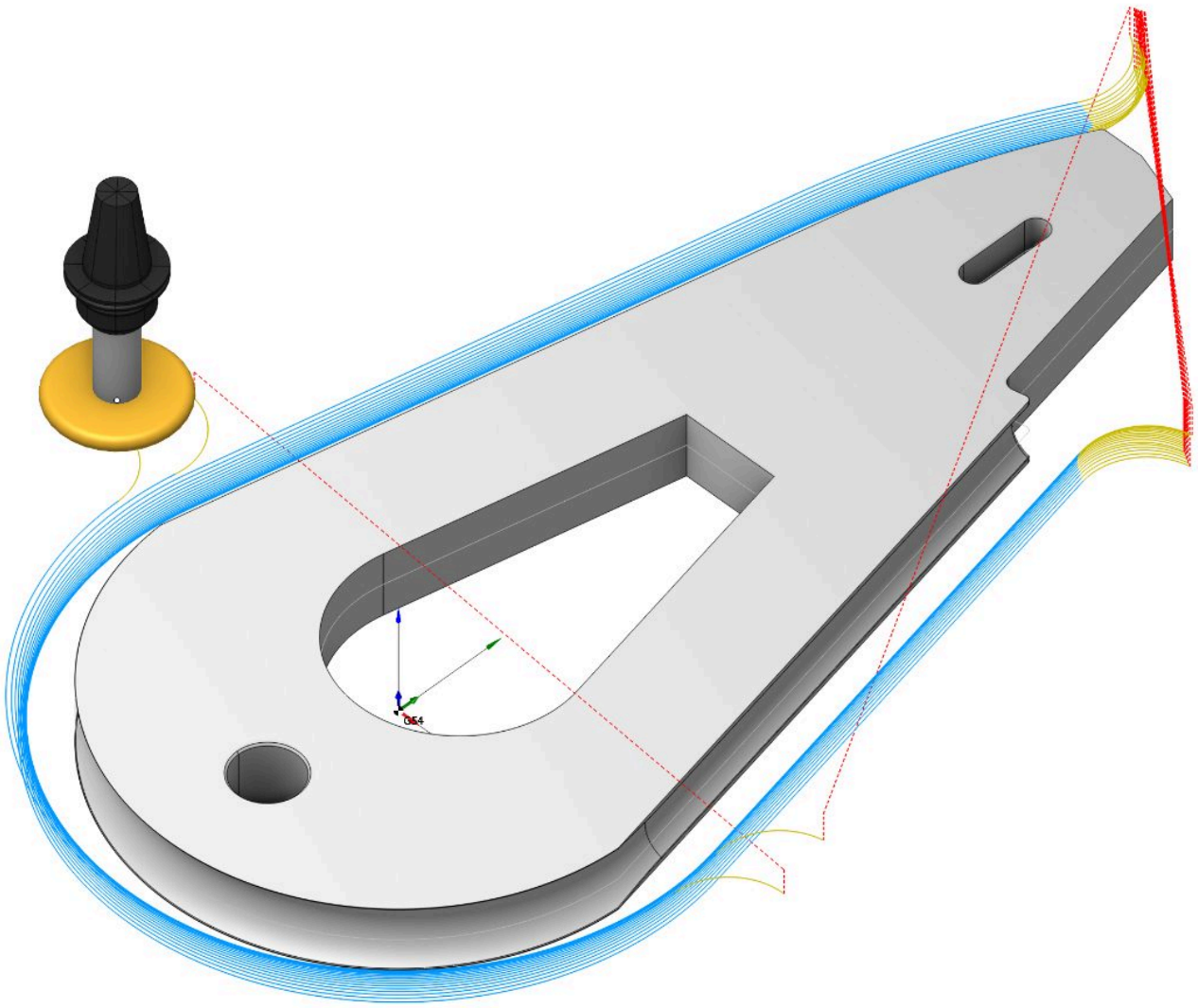
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Undercut waterline



Application area:

Waterline undercut operation is required for creating recesses, grooves, or cavities using a milling tool. This operation generates a waterline trajectory for undercut tools.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

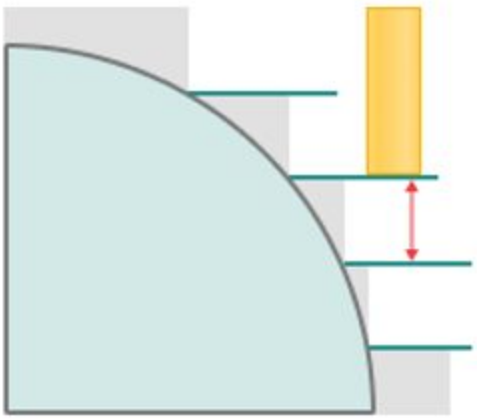
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

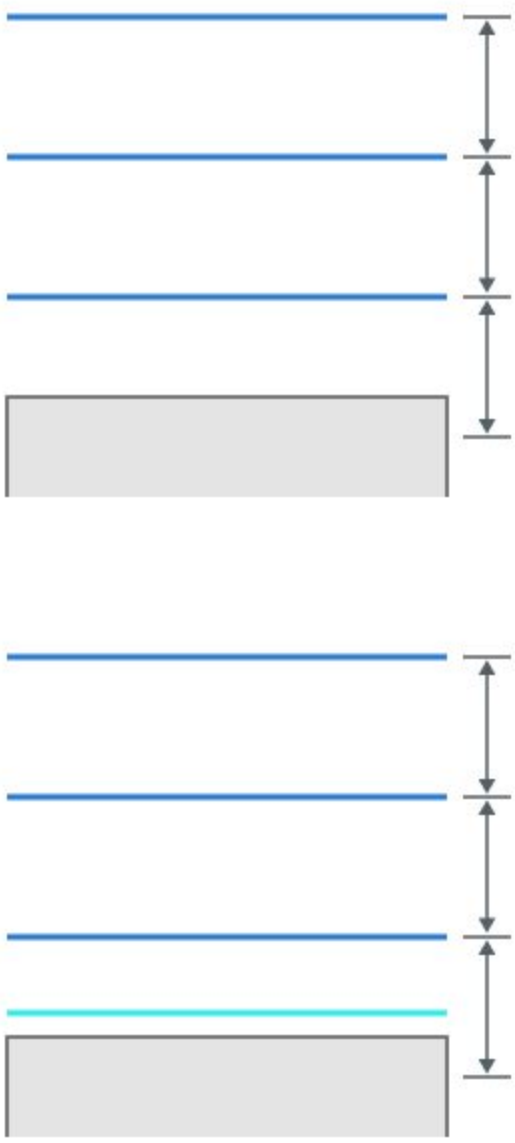
Depth Step.

Material removed in one pass along the tool axis between the Top and Bottom levels.

▼ Click here to expand..



Adaptive step. Additional pass on the upper or lower level. Necessary for cleaning up parts.



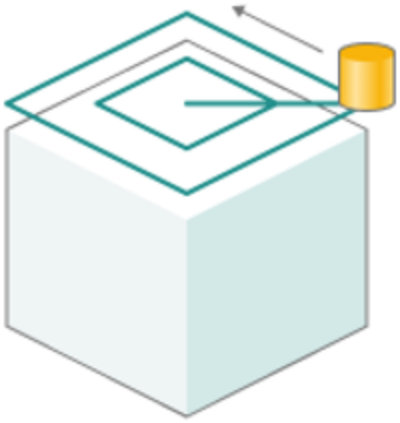
Roughing passes.

Toolpath is offset in plane by the 'Roughing step' thickness amount.

▼ Click here to expand..

Machining strategy:

Equidistant. Processing along curves obtained by offsetting the part contour. Details

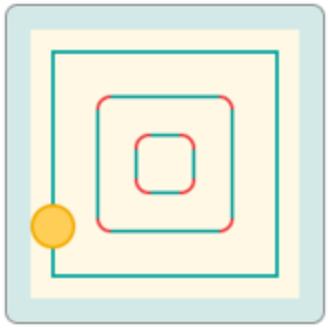


Adaptive. The strategy is used to effectively remove large volumes of material with high feedrates, maximal cutting depths (up to the flutes length) and relatively shallow cutting widths (5% to 30% of the tool diameter). Details

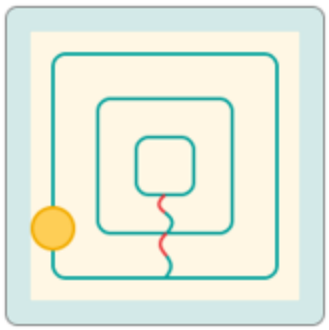


Roughing step. Step between roughing passes.

Rough rounding radius. The parameter is necessary for rounding corners during the roughing pass.



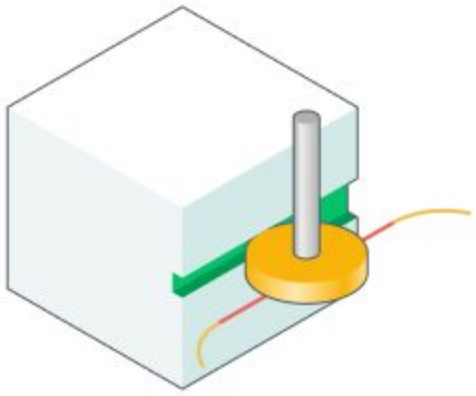
Linking radius. Linking radius for the 'links' and also the transitions for the trohoidal arcs.



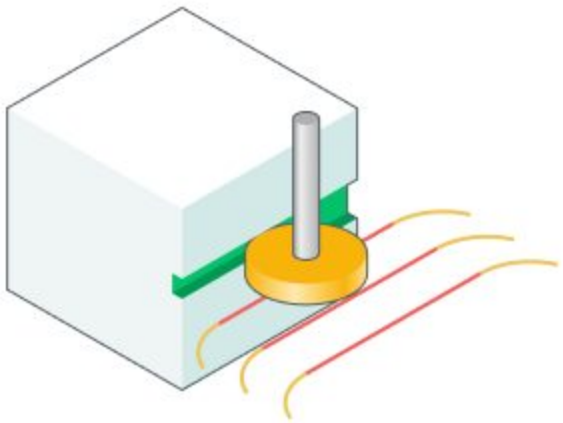
Step count. Number of roughing passes.

Sorting. This tab is necessary for sorting the displayed passages. This parameter group works similarly to the Waterline roughing operation. Details

Check workpiece. If enabled: It adds the number of passes based on the values of the Roughing step and Step count parameters, taking into account the thickness of the workpiece. If the number of passes exceeds the thickness of the workpiece, the additional passes that do not cut the workpiece are not executed.



If disabled: It adds the number of passes based on the values of the Roughing step and Step count parameters, disregarding the workpiece.



Machining levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

Milling type.

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting.

Controls the sequence of toolpath passes during the surface machining. This parameter group works similarly to the Waterline finishing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

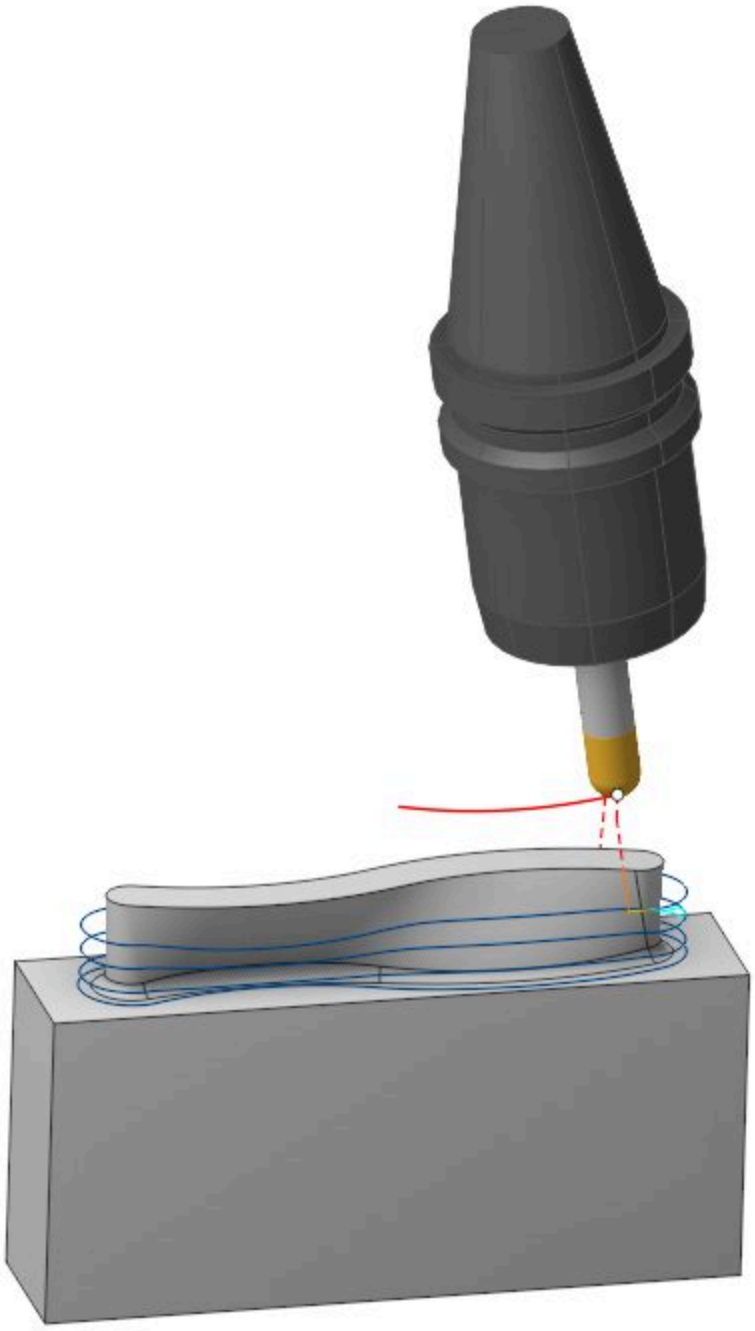
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Swarf



It is the distance between radial passes

Application Area:

This operation machines complex ruled surfaces using the mill's side face. It classifies as a five-axis operation with adjustable tool inclination, either automatically or manually by altering vectors along the tool's trajectory. Typically used for finishing, the operation also supports roughing passes.

Job Assignment:

There are two algorithms for calculating the operation: one based on the By bottom edge and another based on the By two curves. The chosen strategy determines the work assignment and a subset of parameters on the Strategy tab.

By bottom edge.

Bottom Edge. It is the curve defining the lower boundary of the surface being machined. It's parameters are the same as in the Parameters of Curves in the 5D contour and 6D contour operations.

By two curves.

First curve. Defines the lower curve of the workpiece surface.

Second curve. Defines the upper curve of the workpiece surface.

Sync lines. D efines the connection of the lower and upper curves, with its help the surface sections are separated. The more synchronization lines, the higher the accuracy.

Support meshes. D efines the work surface, required for models with a complex shape. Added to the automatically generated surface, can replace it if the "use automatic meshes" checkbox is disabled

Parameters available for the two strategies.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

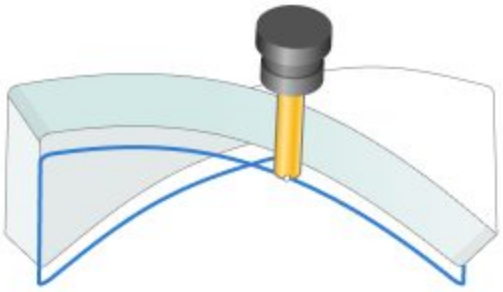
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

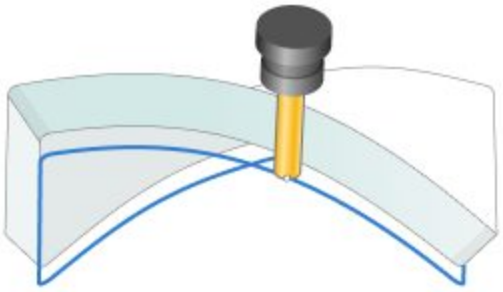
Strategy:

This parameter group allows the user to achieve a required toolpath:

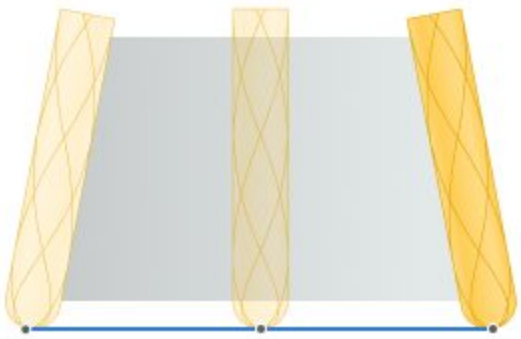
▼ Click here to expand..



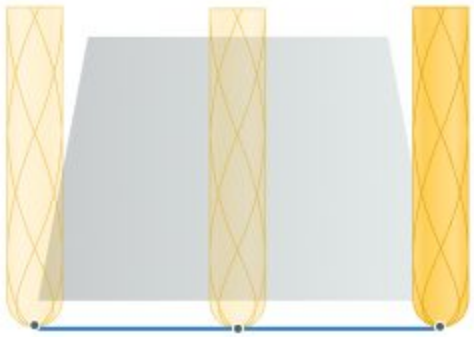
Automatic strategy. Calculates the most optimal path of the tool with the least amount of cuts and defects.



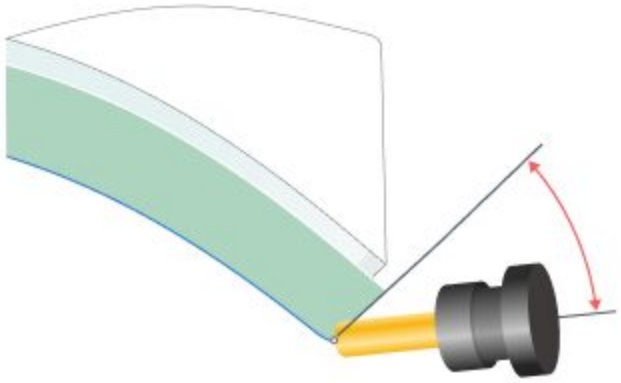
Shortest distance strategy. Calculates the tool path with the smallest distance between the upper and lower curve.



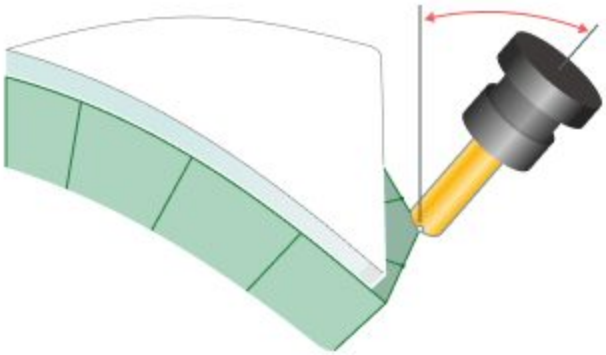
Normal to guide curve strategy. Calculates the tool path perpendicular to the main (lower) curve.



Lean angle. It is a n additional tool tilt angle within the plane ortogonal to the motion direction.



Use Custom Vectors. By default, system positions vectors to follow the designated surface. By setting the flag, the user can manually adjust the tool's tilt at any chosen points in the graphical window. The parameters of this group are the same as in the Parameters of Curves in the 5D contour and 6D contour operations. Details.



Use alternative side. C hanges the processing side to the opposite

Use automatic meshes. T he parameter responsible for using an automatically generated mesh between two curves. Often does not follow the curves of the surface being processed.

Milling Type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Tool contact point.

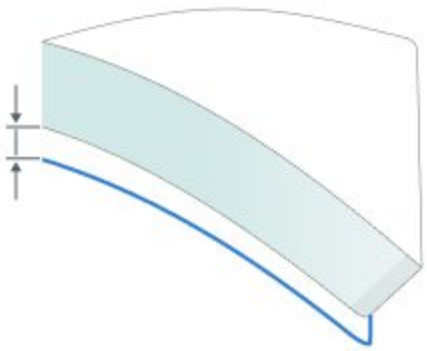
Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

The parameters of this group are the same as in the 5D Surfacing operation. Details.

Breakthrough:

Enables the tool to be shifted along its axis by a specified distance when undercuts are found. The tool will not shift if there are no undercuts and the surface below restricts it.

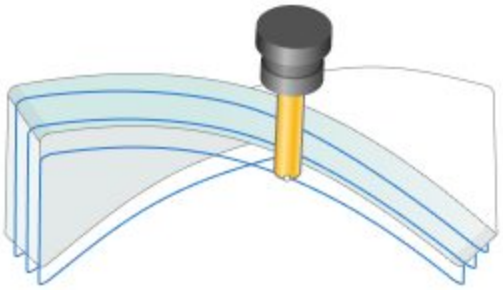
▼ Click here to expand..



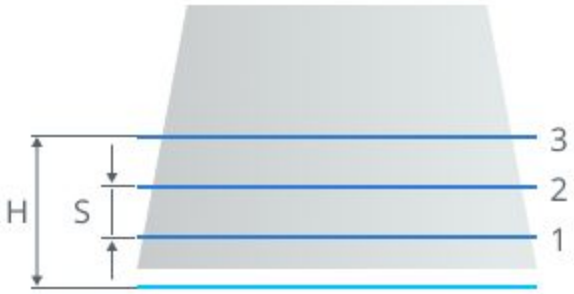
Axial passes:

The flag setting includes additional passes along the machined surface.

▼ Click here to expand..

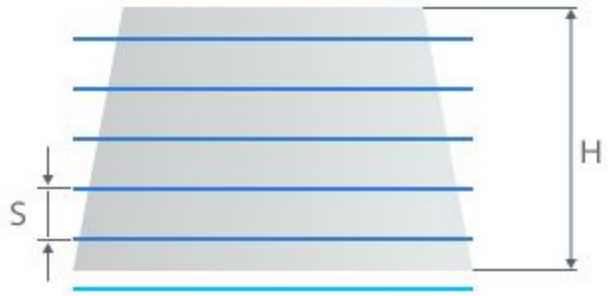


Custom. The user sets the Step and the number of Pass count.



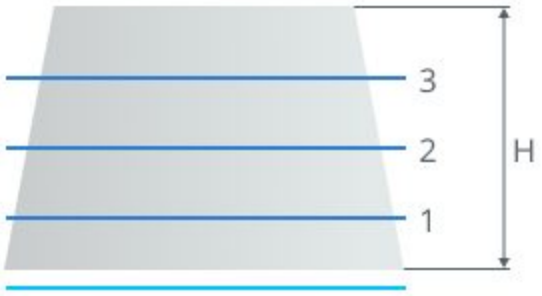
S- Step H - The height parameter. Calculated based on the `*Step*` and `*Pass count*`.

By step. The system automatically determines the upper level. The step between passes is specified by the user in the parameter.



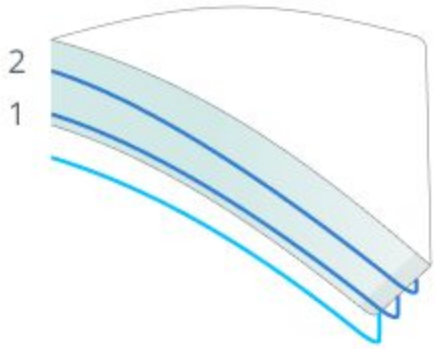
S - Step H - The height parameter. Calculated automatically based on the model

By count. The system automatically determines the upper level. Step based on the number of Pass count.

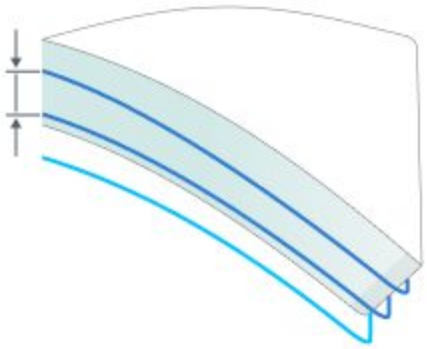


H - The height parameter. Calculated automatically based on the model

Pass count. It is the count of axial passes.



Step. It is the distance between axial passes.

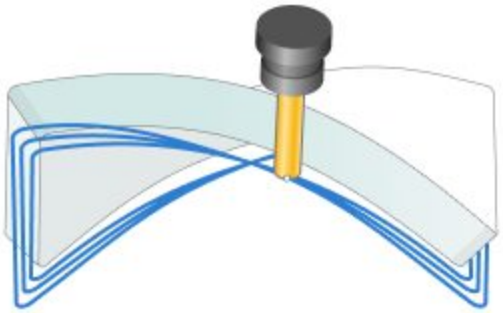


Contact Points. C hange the tool's contact point from the initial line to the final line. This group works similarly 5D Surfacing operation. Details

Radial passes:

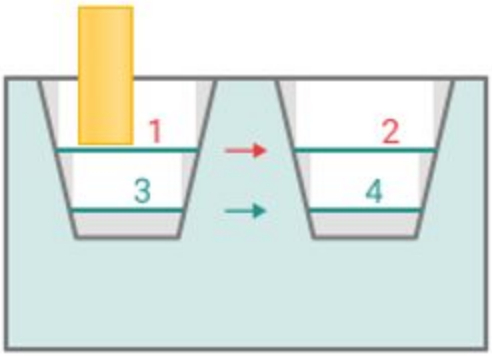
The flag setting includes additional rough passes perpengicular to the machined surface.

▼ Click here to expand..

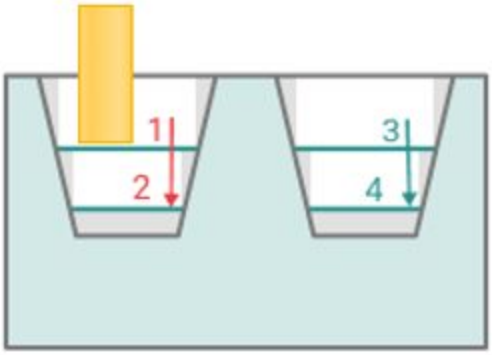


Setting the flag s pecifies the execution order for rough passes:

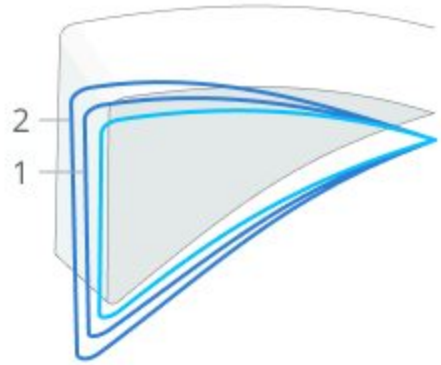
Levels. Rough passes are executed in the order of the layers' arrangement.



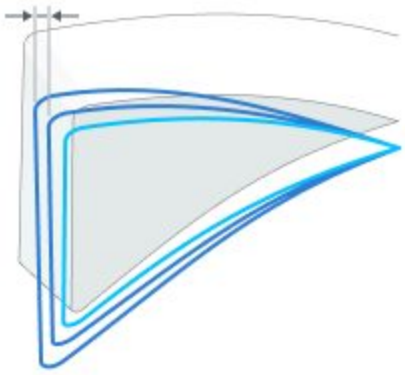
Strings. Rough passes are executed in the order of the strings' arrangement.



Count. It is the count of radial passes.



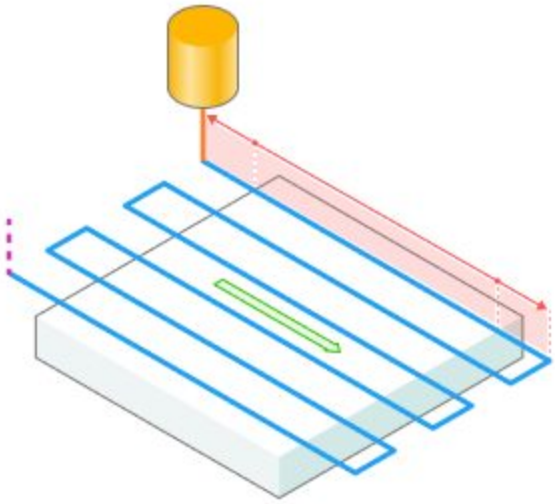
Step. It is the distance between radial passes.



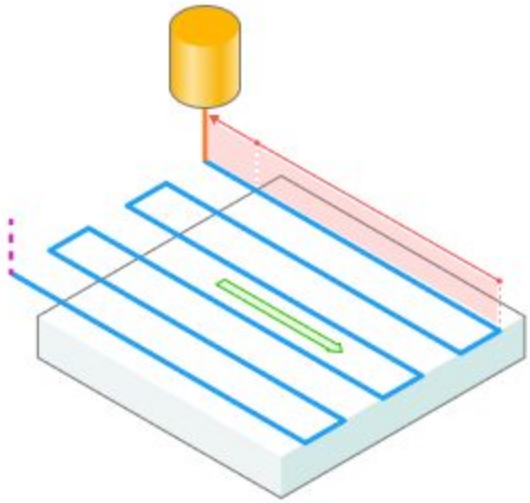
Extend (+)/Trim (-) passes.

With this setting, users can extend or trim the initial and final segments of the toolpath.

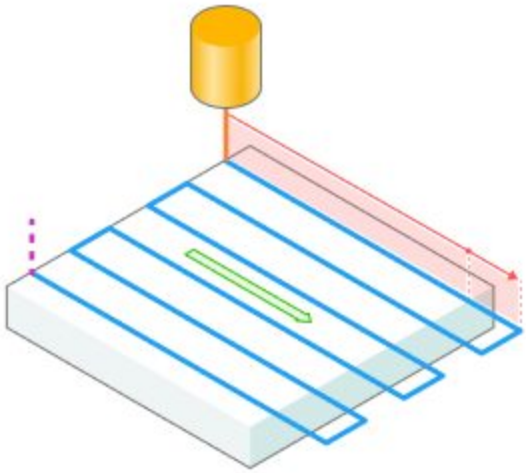
▼ Click here to expand..



Start. Entering a positive value, the system extends the start of the working passes of the toolpath by the specified amount. Entering a negative value shortens the beginning of the passes.



End. Entering a positive value, the system extends the end of the working passes of the toolpath by the specified amount. Entering a negative value shortens the end of the passes.

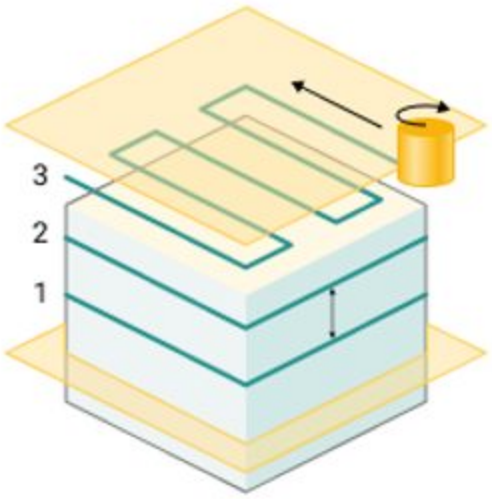


Sorting.

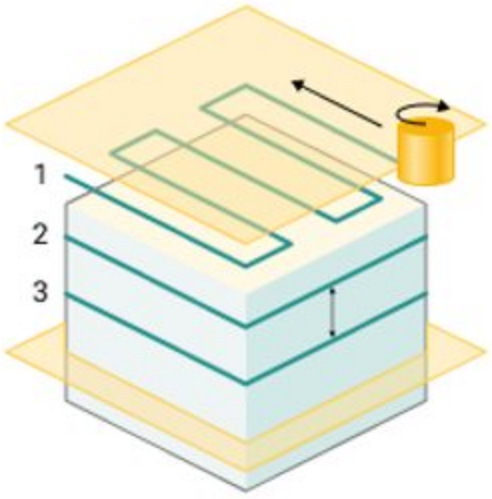
Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Machining Order. Determines the order in which passages are formed.

Upwards. The passages are formed from bottom to top



Downwards. Passages are formed from top to bottom



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

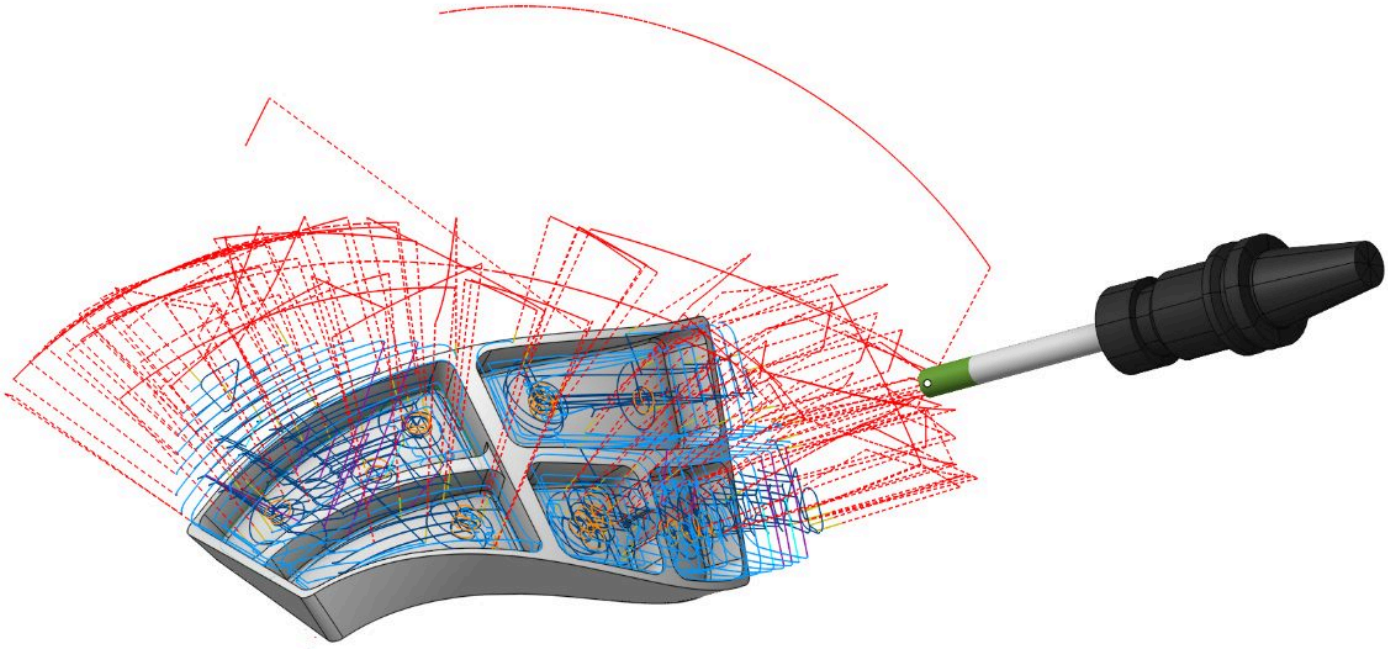
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

5D Roughing waterline



Application area:

Specialized processing for pockets with curved bottoms, based on the Waterline operation.

Job assignment:

Floor surfaces. Control surface. Typically defined by the pocket bottom.

Job Zone curve. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. In this operation, only a closed work zone is applicable. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Strategy.

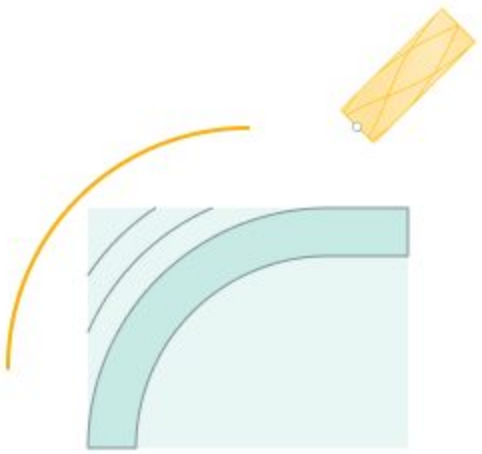
This parameter allows the user to achieve a toolpath that appears adaptive or equidistant when performing pocketing operations. This parameter group works similarly to the Waterline roughing operation. Details

Machining levels.

It defines the range along tool axis for the machining.

▼ Click here to expand..

Auto top level. If the flag is set, the upper level is automatically calculated based on the blank.



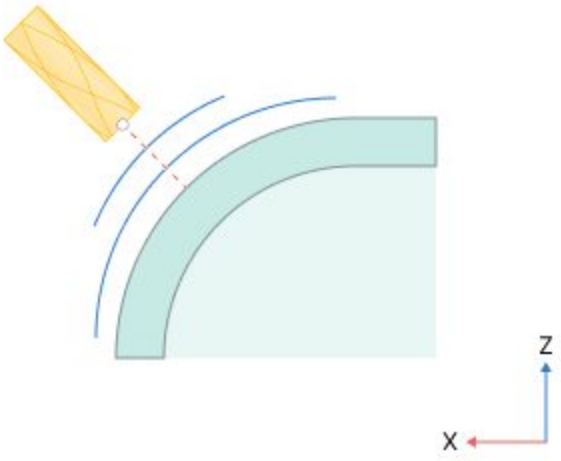
Some parameter group works similarly to the Waterline roughing operation. Details

Face side.

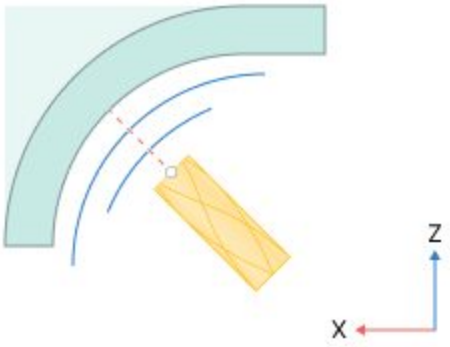
Determines the machining direction from the bottom surface of the pocket.

▼ Click here to expand..

Front. The tool can only process from the side with the positive direction of the surface normal vector.



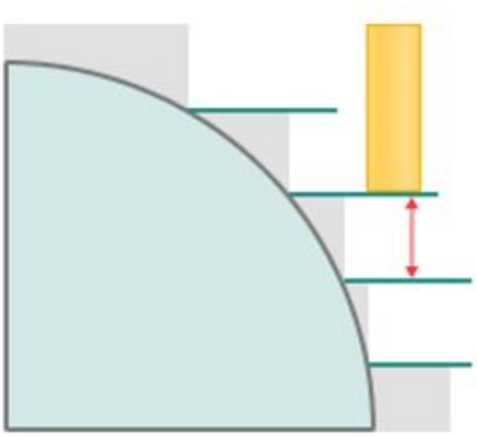
Back. The tool can only perform processing from the side opposite to the surface normal vector.



Depth Step.

Material removed in one pass along the tool axis between the Top and Bottom levels.

▼ Click here to expand..



Milling type.

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or

return occurs toward it. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Allow plunges. Enabling plunges allows machining of closed pockets. When CAM software encounters a closed pocket that can not be machined from the outside, it forms a Plunge move Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

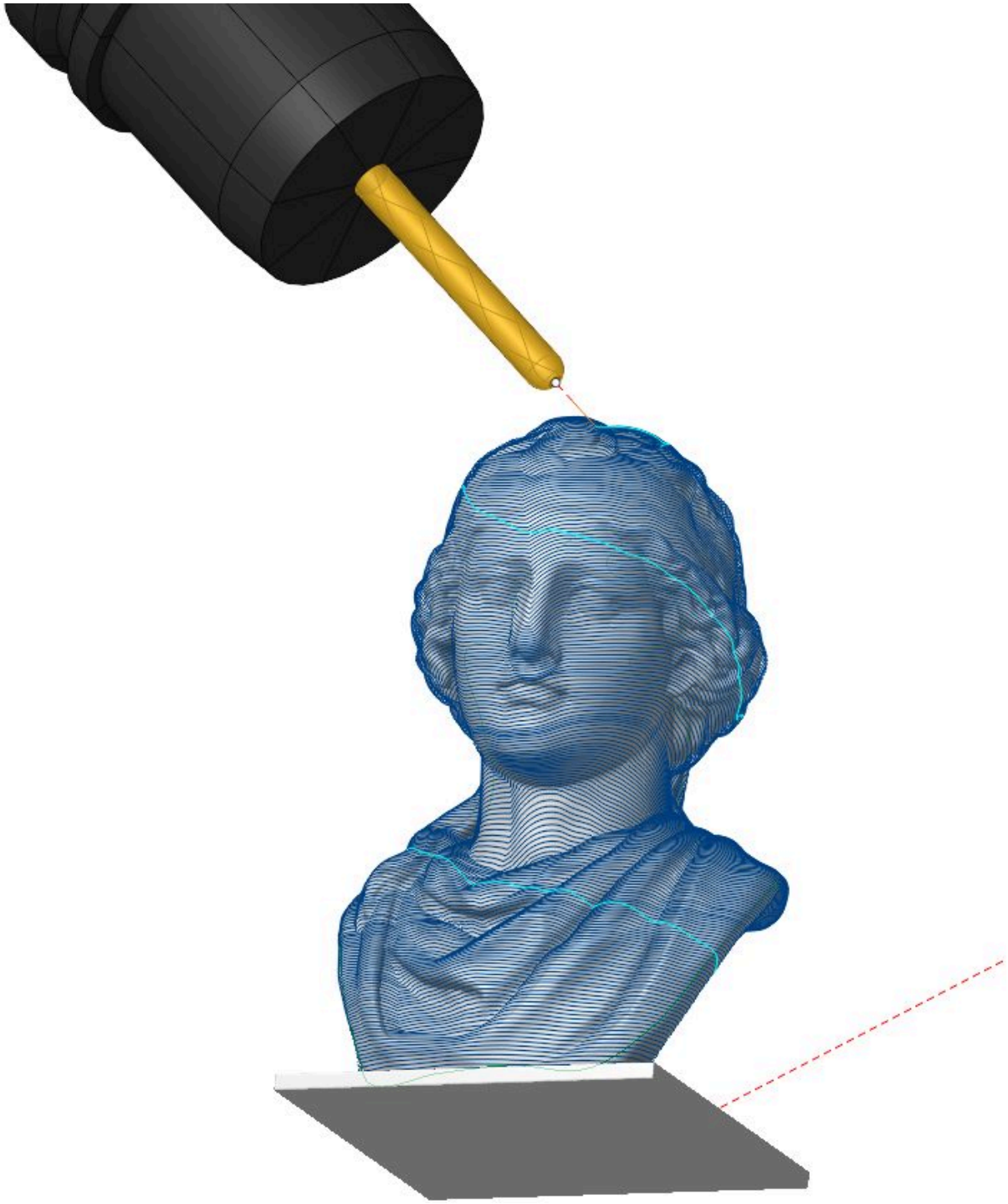
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

5D by meshes



Application Area:

A versatile 5 axis finishing operation with a powerful set of strategies, tool axis orientation modes and automatic tool and holder collision avoidance. Perfect for finishing of complex shapes such as sculptures or scanned STL models. Easy to use. The operation allows machining mesh models similarly to Scallop, 3D Helical and Waterline strategy from 5D Surfacing. The operation allows generating machining toolpath on the 5 axis machines and robots. The operation supports surface models and triangular mesh models. You can use spherical, conical mills with a spherical end and lollipop mills.

To start machining just create the operation and press Run. The operation will generate a scallop pattern on the whole part starting from the bottom level. The tool will stay in the vertical orientation unless there is a collision with the part, otherwise it will be tilted to the side.

Workflow:

1. Define the toolpath Strategy. The default strategy is Scallop as it allows to machine the whole part with a consistent stepover.
2. Choose the tool axis orientation. The default option is Fixed, because together with the automatic collision avoidance it doesn't require any additional setup and achieves a smooth and predictable toolpath.
3. Define the collision avoidance strategy. The default is Side Tilting, as it is works well with Scallop, Helical and Waterline patterns.
4. Optionally, define the job zone
5. Generate the toolpath

Job assignment

Machining Surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

Tilt Curve. This is t he curve that the tool axis vector aims at when setting Tool Orientation specification Through Curve.

Job Zone. Use the Job Zone to trim the passes outside the specified 2D containment areas. You can select the normal to the area of Job Zone. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that s hould avoid machining in the current operation. Details

Start Curves. Using a closed curve, such as a spline, you can select a model segment for machining. You can specify the machining side and configure vectors to define the tool axis orientation.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

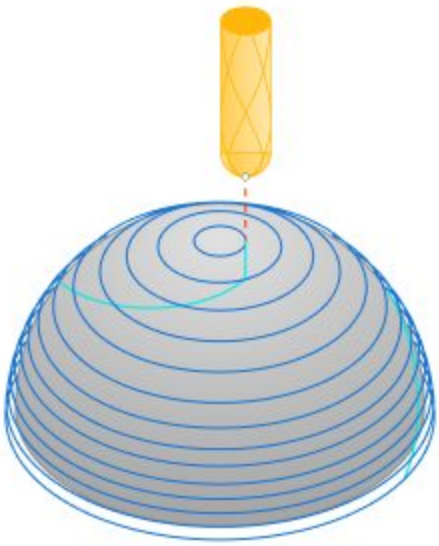
Strategy:

Strategies:

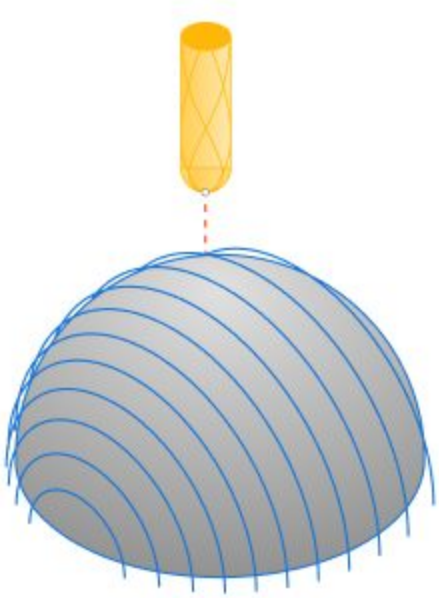
This parameter allows the user to achieve a required toolpath:

▼ Click here to expand..

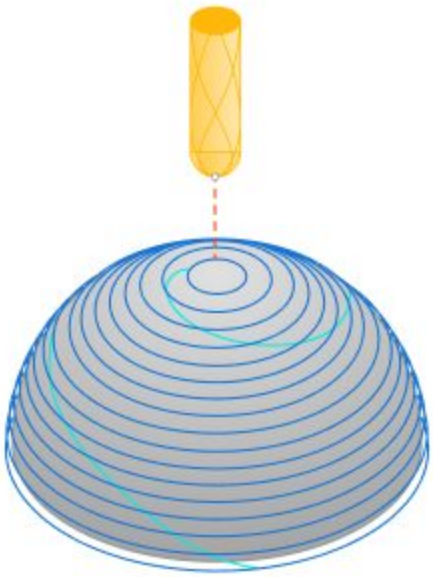
Waterline. The system generates the tool passes as section lines of the Machining Surface by horizontal planes (planes are perpendicular to the tool axis), similar to the strategy in a Waterline Finishing operation Details.



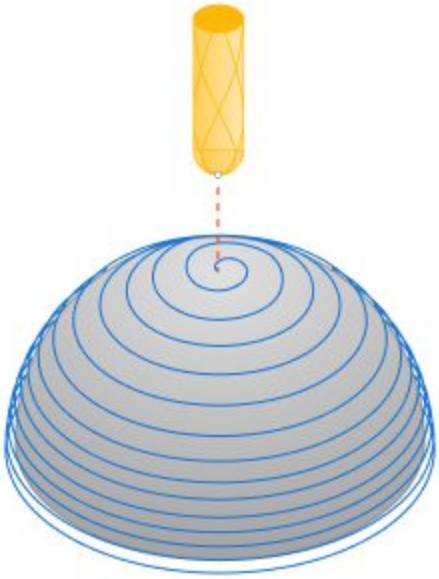
Plane. The system generates the tool passes as section lines of the Machining Surface by vertical planes (planes are parallel to the tool axis), similar to the strategy in a Plane Finishing operation. Details.



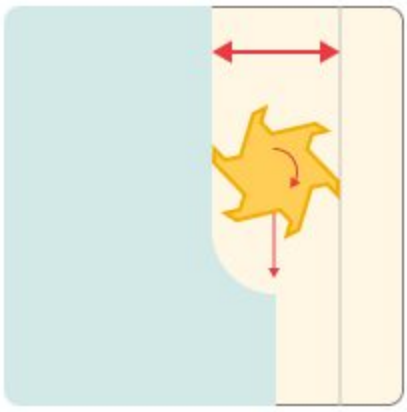
Scallop. The Scallop toolpath starts from the curves lying on the part surfaces and is generated by repeatedly offsetting those curves inwards until the curves collapse. The system generates toolpath as in the Scallop finishing operation. Details.



Helical. The strategy generates continuous helical passes as in the 3D Helical operation. Details.

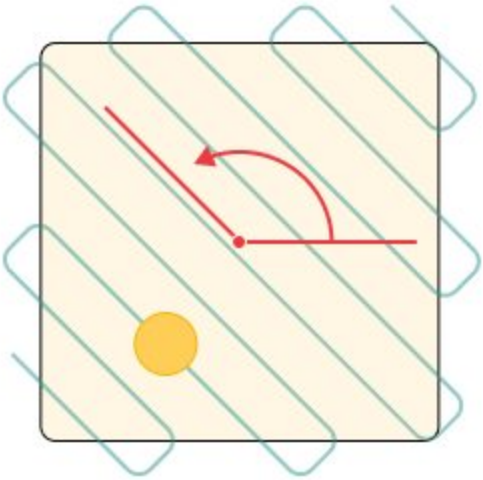


Step. The maximum allowed width of cut.



S - Step

XY Angle. This is the r otation angle of the section planes around the tool axis.

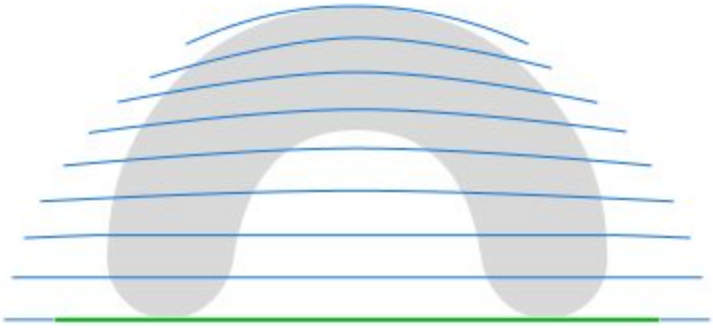


Start From:

This group of parameters appears when the Scallop strategy is active and determines the construction of the initial toolpath curves.

▼ Click here to expand..

Bottom. The toolpath begins at the Bottom Level of machining. within the Job Assignment.



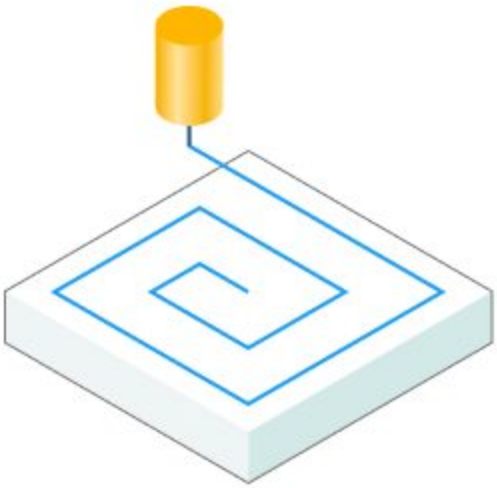
Job Boundary. The toolpath begins at the curves of the Job Zone.

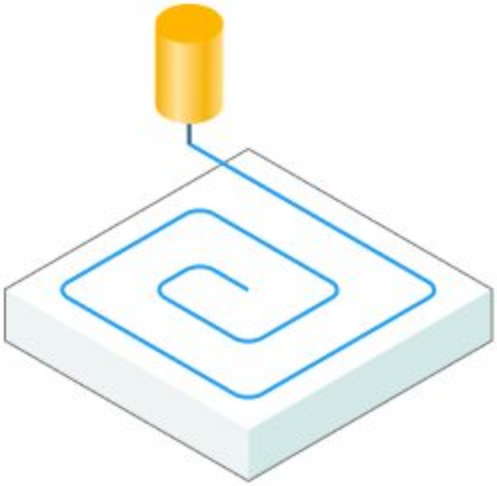
Other parameters of this group are the same as in the Scallop Finishing operation. Details.

Smooth Radius:

This parameter appears when the Helical strategy is active and a llows for the smoothing of inner toolpath corners using a designated radius.

▼ Click here to expand..



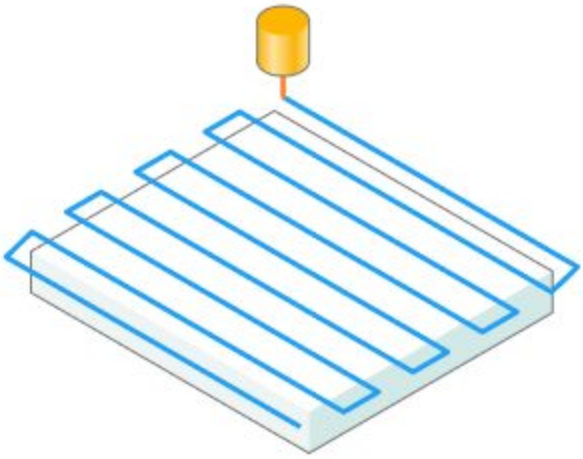


Job zone:

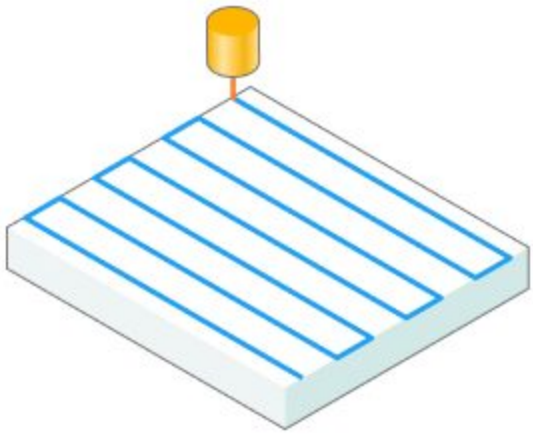
Enables additional modification of the working area initially selected on the Job assignment tab.

▼ Click here to expand..
Restrict Job Zone:

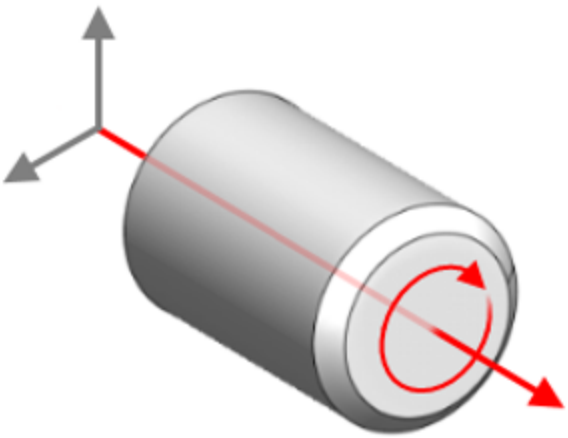
With Edges. Upon this parameter's activation, the system machines both the surfaces specified in the Job Assignment and their boundary edges.



Surface Ony. Upon activation of this parameter, the system processes exclusively the surfaces specified in the J ob Assignment, excluding their bounding edges.



Rotary Axis. Specifies the origin and direction for tracking this group's parameters. The Rotary Axis is defined by its Origin and Direction. You can easily set the desired parameters of the rotary axis both in the inspector and with the mouse in the graphic view.



WCS X (Global CS X). The X axis of the specified coordinate system is the Rotary Axis.

WCS Y (Global CS Y). The Y axis of the specified coordinate system is the Rotary Axis.

WCS Z (Global CS Z). The Z axis of the specified coordinate system is the Rotary Axis.

WCS (0,0,1) (Global CS (0,0,1)). The vector of the Rotary Axis gets its definition from direction cosines relative to each axis.

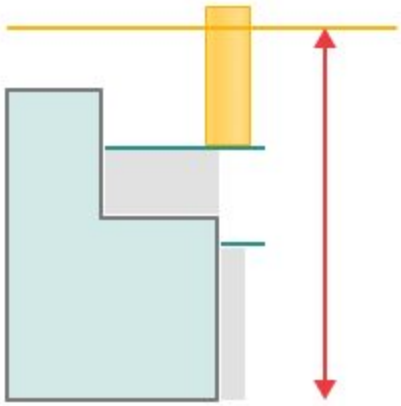
Direction. Specify the direction cosines relative to each axis.

Origin. Specify the origin coordinates for tracking this group's parameters.

Base CS. Specify the coordinate system to configure the initial position and orientation of the Rotation Axis.

Tool axis. The vector of the Rotary Axis is the tool axis.

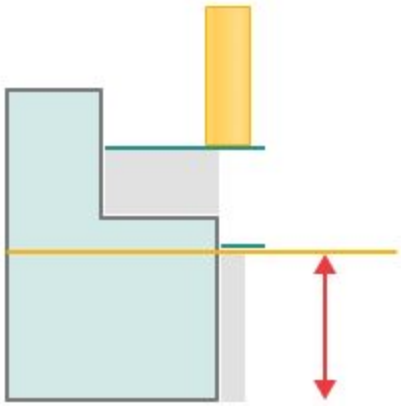
Top level. The highest distance from the Origin where tool passes occur. The level at which the final pass is made. By default it is the top most level of the machined part.



mm. Specify desired dimension from the Origin to the Top Level in the text field in the inspector. You can also drag the plane with the mouse or drag the plane centre circle and use 'Smart snap' to align with a feature or, click on the dimension in the graph view.

Auto. The system determines the Top Level from the highest points in the designated areas of the Job Assignment.

Bottom Level. The lowest distance from the Origin where tool passes occur.

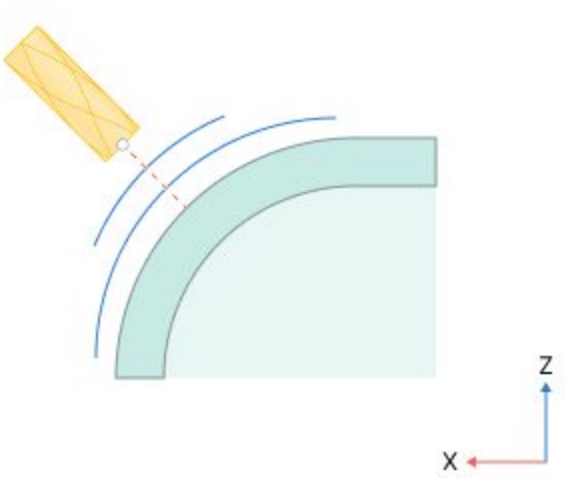


mm. Specify desired dimension from the Origin to the Bottom Level in the text field in the inspector. You can also d rag the plane with the mouse or drag the plane centre circle and use 'Smart snap' to align with a feature or, click on the dimension in the graph view.

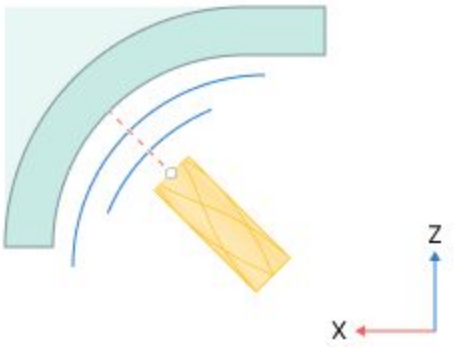
Auto. T he system determines the Bottom Level from the lowest points in the designated areas of the Job Assignment.

Machining Side. Identifies the permissible side of the surface for machining. Details.

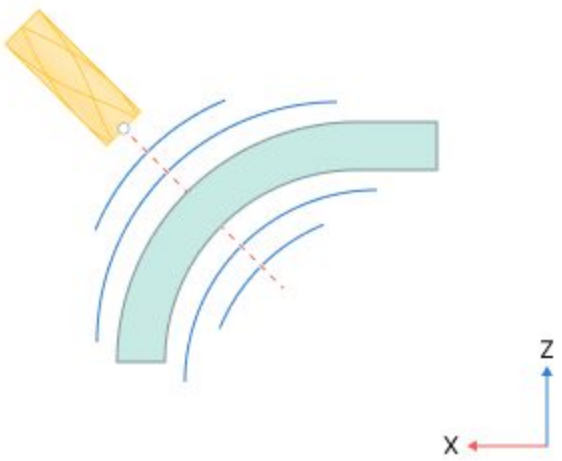
Front. The tool can only process from the side with the positive direction of the surface normal vector.



Back. The tool can only perform processing from the side opposite to the surface normal vector.



Both. The tool can perform processing from both sides of the surface.



By Default. Utilizes the machining method set as the default in the system.

Milling Type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Sorting.

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Machining Direction. Controls the direction of tool motion on toolpath curves. The parameters of this function are the same as in the Scallop Finishing operation. Details.

Start from. Specifies the starting level for tool motion. The parameters of this function are the same as in the 3D Helical operation. Details.

Change Direction. Allows for the modification of pass execution order - from bottom to top or top to bottom.

Tool Orientation:

Controls tool axis orientation. The principal parameters of this function are the same as in the 5D Surfacing operation. Details.

▼ Click here to expand..
Singularity avoidance angle. With this option, the tool deviates from the fixed axis at a specified angle consistently, even when there are no collisions. As a result, the "singularity" of the rotating table is

avoided.

Avoid Collisions:

With the Check Holder function, this feature detects segments of the toolpath where the tool holder collides with the part and modifies those segments according with the specified strategy. Details

Limit Rotation Angles:

This option specifies the limit values of the tool's angular position relative to the axes. The parameters of Limit Rotation Angles are the same as in the 5D Surfacing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

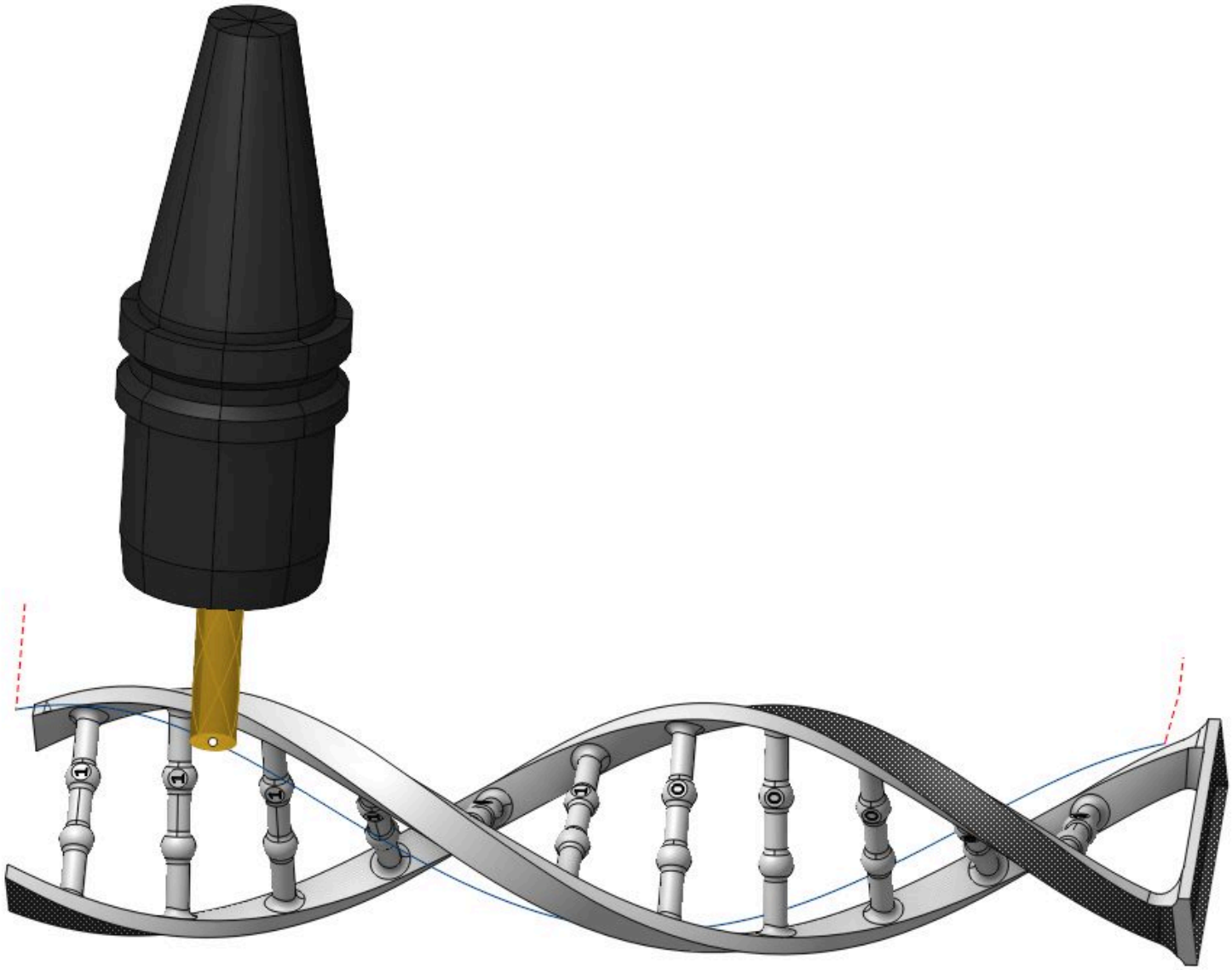
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

5D contour and 6D contour



Application area:

This operation generates a continuous 5-axis toolpath for any spatial curves, It allows for both a single toolpath construction along a chosen curve and passes along several isoparametric lines of a selected surface, as well as conducting multi-level processing along a specified curve. It offers a wide selection of tool orientation strategies, adjustable automatically or manually by editing vectors at each point along the curve.

For the curves specified in the Job Assignment, user can modify the tool orientation vectors at each point, change the tool's movement direction along the curve, alter the machining side, compensate for the tool radius, select the interpolation method for changing the tool orientation vector from point to point, and set the stock.

Job assignment:

Edge/Curve on Surface. Job assignment is formed with the edges of the part or any curves. The tool's current orientation determines vectors at the curve points.. The tool contact point will move along the specified curve based on the curve properties and the chosen tool orientation strategy. Details

5D Curve. The system generates the 5-axis tool path as the projection of the arbitrary curves on the part surface. In this case the tool contact point will move in touch with the surface. The passes can be shifted from the curve by the stock or by the levels. The tool axis is located perpendicular to the surface. Details

Drive Faces. One can u se them for creating work passes along isoparametric lines. The isoparametric curves of the surfaces depends on its way of creation and reflect the topology of the surface. Different fillets can be machined this way very well. Details

Recognize elements. It allows recognizing the following constructive elements: Circle, Rectangle, Rounded Rectangle, Slot, Hexagon, and Keyhole. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Tool contact point.

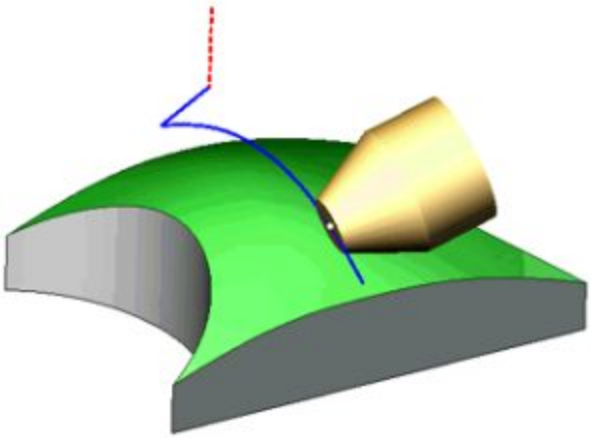
Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

The parameters of this group are the same as in the 5D Surfacing operation. Details.

Tool Orientation:

Controls tool axis orientation. The principal parameters of this function are the same as in the 5D Surfacing operation. Details.

▼ Click here to expand..
Incline Shape Tool. Tilts the tool so its working side touches the machining surface during toolpath traversal.



Limit Rotation Angles:

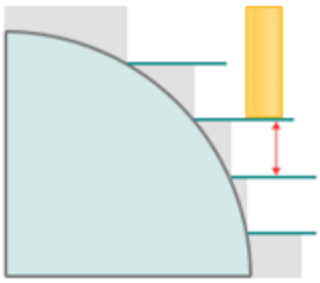
This option specifies the limit values of the tool's angular position relative to the axes. The parameters of Limit Rotation Angles are the same as in the 5D Surfacing operation. Details.

Passes:

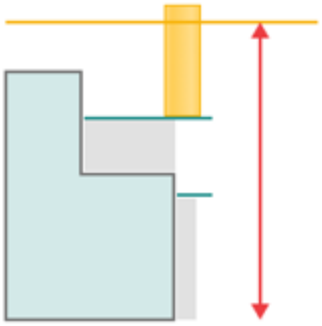
You can create additional work passes relative to the specified curves. The passes shifts in the direction normal to the surface. If the Flank tool orientation is selected, you can choose the direction of the shifts.

▼ Click here to expand..

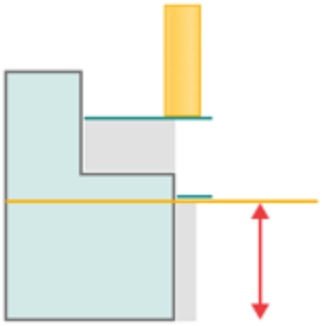
Step. Material removed in one pass between the Top and Bottom Levels. You can specify the Step both as the thickness in length units or the percent of the tool diameter as well as the passes count.



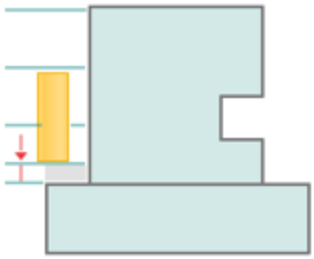
Top Level. Defines the maximal stock measured from the specified curve that must be removed by the rough passes.



Bottom Level. Defines the distance from the machined surface to the last pass. If the value is positive, the tool will follow a path outside the workpiece material at the specified distance. If the value is negative, the tool will cut the part material at the specified depth.



Z Cleanup. The function adds a new pass at the distance specified in the parameter from the Bottom Level.



Direction. With the Flank tool orientation strategy active, you can select the direction for laying out rough passes.

Along Surface Normal. The layout of passes follows the direction normal to the surface.

Along Tool Axis. The layout of passes follows the vector of tool axis.

Corners smoothing:

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. The principal parameters of this function are the same as in the 2D Contouring operation. Details.

Continous axial shifting:

With the Flank tool orientation strategy is in use, you can create a toolpath with smooth changes in the tool's axial displacement along the path relative to the specified curve.

▼ Click here to expand..
Start Shift Value. It is t he tool's axial displacement at the start of the toolpath.

End Shift Value. It is t he tool's axial displacement at the end of the toolpath.

Machining from inside.

The parameter allows you to process the internal surfaces of the part (project the trajectory onto the internal surfaces of the part). This parameter group works similarly to the 5D Surfacing operation. In the section Job zone. Details

Sorting:

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Toolpath uniting mode. Controls the order of pass execution in the multi-pass machining of different profiles on the part. Parameters of this feature are the same as in the 2D Contouring operation. Details.

Idling minimization. The order of the profiles machining is sorted to minimize the idle motions. Parameters of this feature are the same as in the 2D Contouring operation. Details.

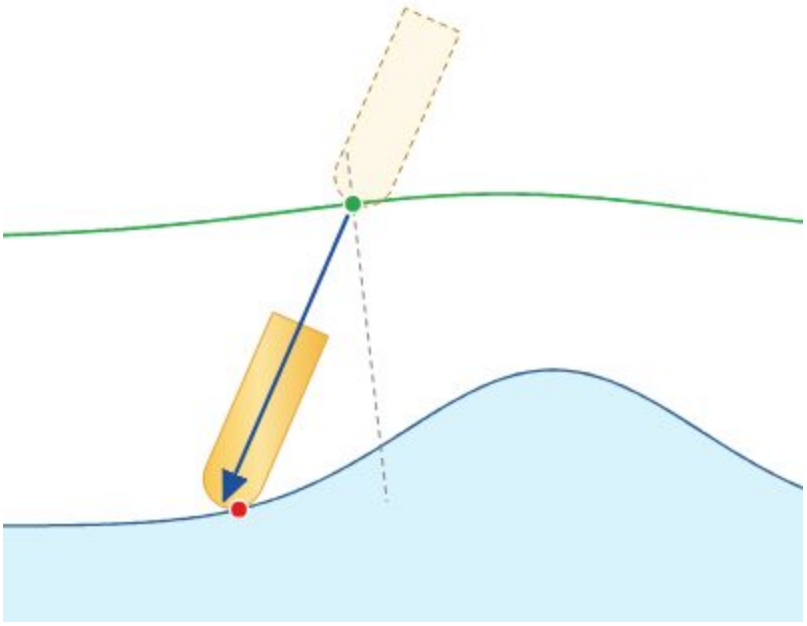
Milling type. Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. The parameters of the Milling Type are the same as in the Waterline Roughing operation. Details.

Project toolpath onto the part with offset:

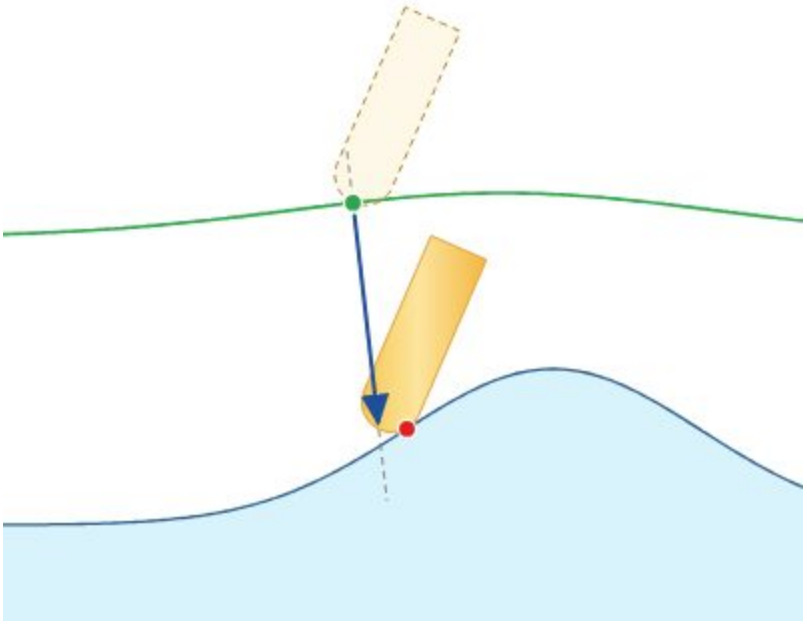
Determines the method of toolpath creation relative to the surface of the part. Parameters of this feature are the same as in the Strategy tab, in the Transformations item of the 2D Contouring operation. The toolpath is projected onto the model defined on the Part tab. If no part is defined, the job assignment geometry is used. For both projection types, the tool axis is determined by the Tool orientation settings at each point of the original toolpath and remains unchanged during projection. Details.

▼ Click here to expand..
Type. Determines the projection direction.

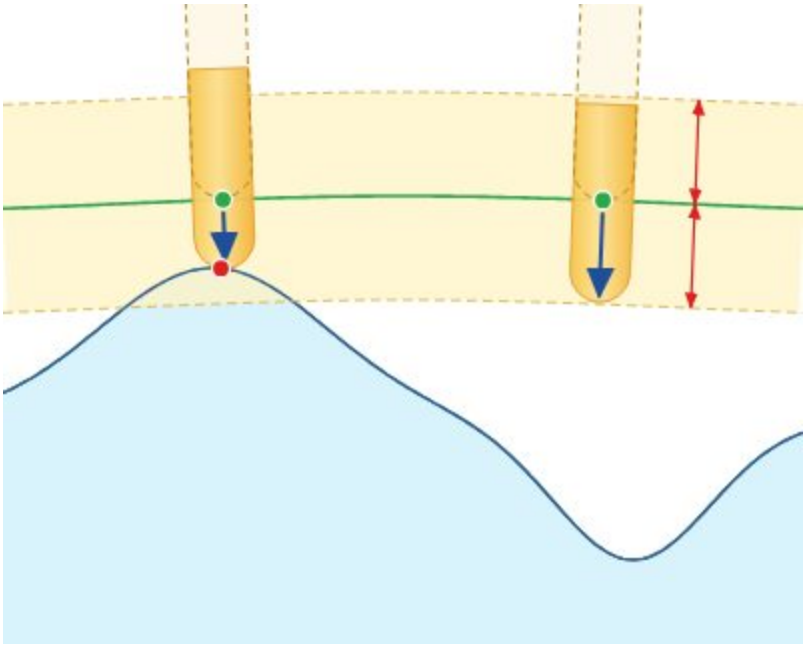
Along tool axis. The tool moves along its axis until it contacts the part. This preserves the projection behavior of earlier versions.



By surface normal. The tool retains its axis tilt and moves along the normal of the surface used to generate the passes until it contacts the part. Contact is detected over the entire tool body, including its side, rather than only at the tool end. The search extends by Max deviation on both sides of the surface. If the part is outside this range, the tool moves by the full Max deviation beyond the surface, opposite to the normal.



Max deviation. Defines the contact search distance from the toolpath surface along the normal in both directions. Set it to at least the greatest distance from this surface to the part. Otherwise, where the part lies farther away, the tool cannot reach it and moves only by the specified Max deviation.



Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

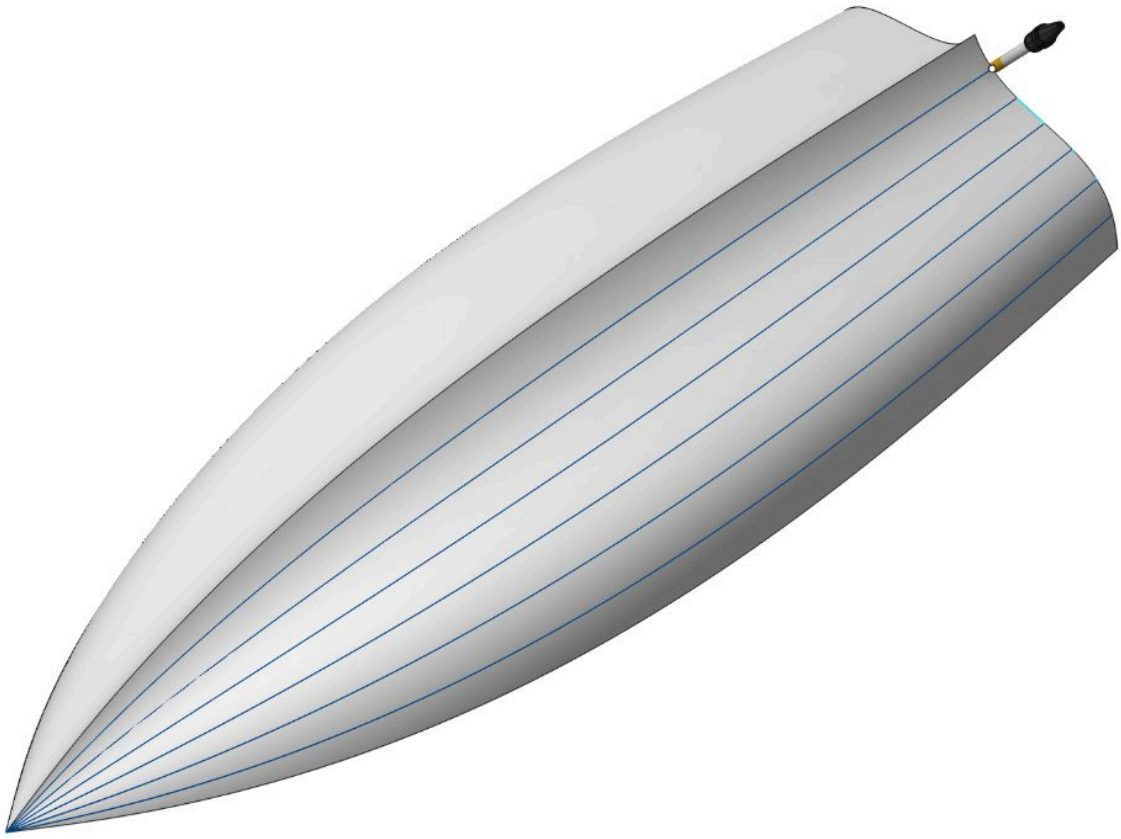
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Parameters of Drive Faces



Application Area:

You can u se Drive Faces for creating work passes along isoparametric lines. The isoparametric curves of the surfaces depends on its way of creation. Frequently they reflect the topology of the surface and so is very useful for machining as work passes. Different fillets can be machined this way very well. Include surfaces in the Job Assignment of the 6D Contouring operation if you plan to use their isoparametric lines. To do it, it's necessary to select the desired surfaces and click the Drive faces button. If some faces with the analogous parameterization are selected and added to the list by one click on the Drive faces button then they will be machined together.

Item Properties:

Base Info:

Name. Specify a name for the item's appearance in the Job Assignment list.

Stock. This feature allows to shift both rough and finishing passes from the machined surface by the specified value. If the value is positive, the tool will follow a path outside the workpiece material at the specified distance. If the value is negative, the tool will cut the part material at the specified depth.

Selected Count. The number of selected items.

Drive Face's Parameters:

Alternate Streamline. This feature changes the passes direction to the orthogonal ones.

Alternate Front Side. This feature changes the machined side of surface to the back one

Step Method. You can select a method for setting distances between isoparametric curves.

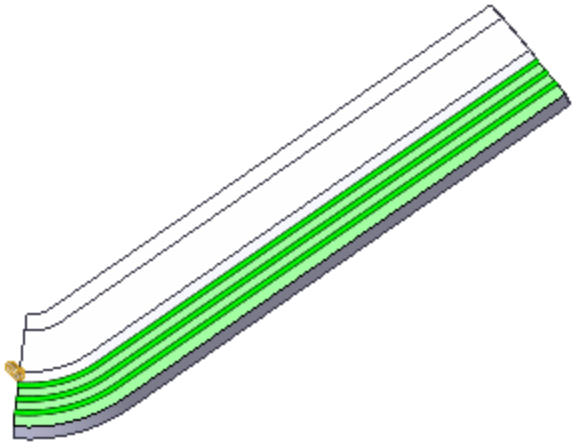
Number. Input in the text field the number of isoparametric lines within the Drive Face.

Distance. Input in the text field the distance between the isoparametric lines.

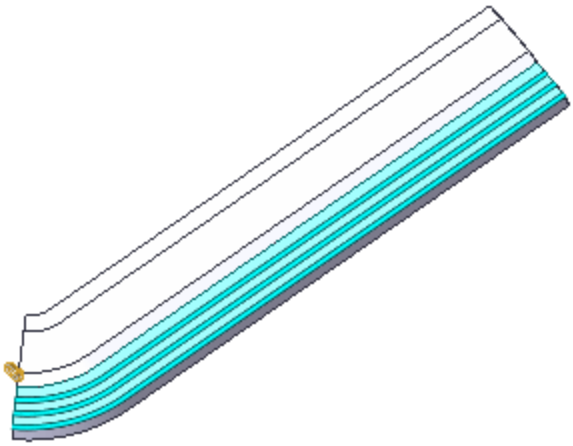
Scallop. Input in the text field the desired value of scallop.

Curve Distribution. This option controls the way CAM system produces isoparametric curves on the Drive Face:

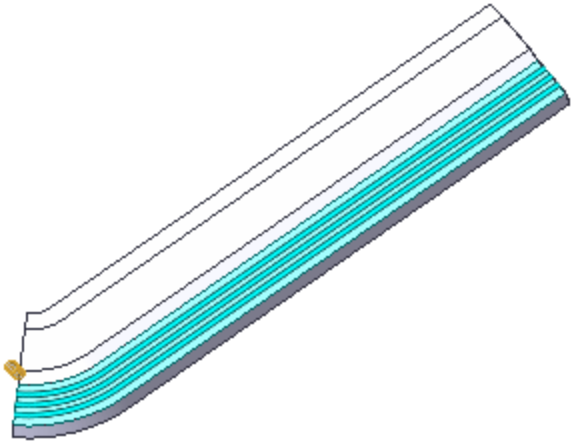
▼ Click here to expand..
From Start. The first curve is placed on the left edge of the surface and other curves are arranged to the right of the first one.



From End. The first curve is placed on the right edge of the surface and other curves are arranged to the left of the first one.



From Center. The first curve is placed in the middle of the surface, other curves are arranged to the left and to the right of it.



Auto. Curve placing method is defined by CAM system based on the machining method and tool axis orientation.

Item Stuff.

It is the list of model's constructive eliments in the item.

Delete. You can delete the selected constructive eliment from the item.

Clear. You can delete all the constructive eliment from the item.

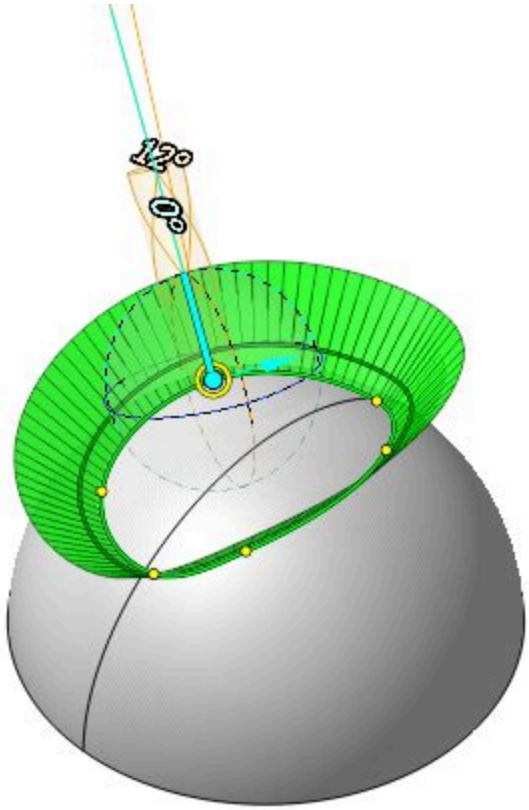
Parameters of Curves

Application Area:

The 6D Contouring operation allows for path creation using two varieties of Job Assignment lines:

5D Curve. The system generates the 5-axis tool path as the projection of the arbitrary curves on the part surface. In this case the tool tip will move in touch with the surface. Specify the surfaces or body to be projected on as the part of the operation.

▼ Click here to expand..

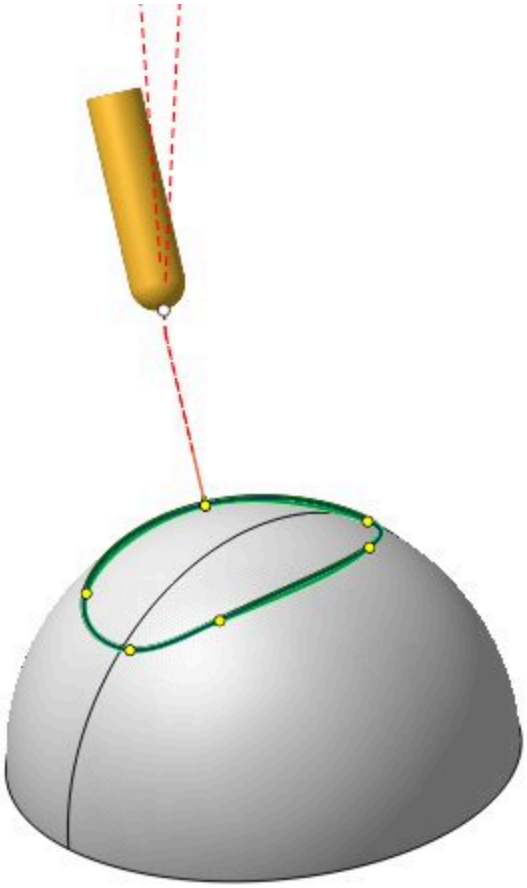


Edge/Curve on Surface. Job assignment is formed with the edges of the part or any curves. If the edges if not highlighted when the mouse pointer goes under it then check the edges selection button on the filter panel (the same applies to curves). It must be down.



If the edge selection filter button is down but the edges is still not highlighted then sew the faces. If the set of the connected edges must be machined in one pass, then it must be selected from the screen together and added to the Job Assignment list by the one click on the Edge\Curve on surface button. In this case the edges are joined to the one item and machined as a whole. If the edges are added to the list separately then it is machined separately even if it is connected. Press and hold the Ctrl to select some objects from the screen at once. This option is most commonly used when it's necessary to mill surfaces with the side part of the mill. In this case, you can select the lower edges of the surface to be machined as the Job Assignment.

▼ Click here to expand..



Item Properties:

Base Info:

Name. Specify a name for the item's appearance in the Job Assignment list.

Stock. This feature allows to shift both rough and finishing passes from the machined surface by the specified value. If the value is positive, the tool will follow a path outside the workpiece material at the specified distance. If the value is negative, the tool will cut the part material at the specified depth.

Selected Count. The number of selected items.

Project Curves Parameters:

Alternate Direction. This feature alters the toolpath direction along the curve.

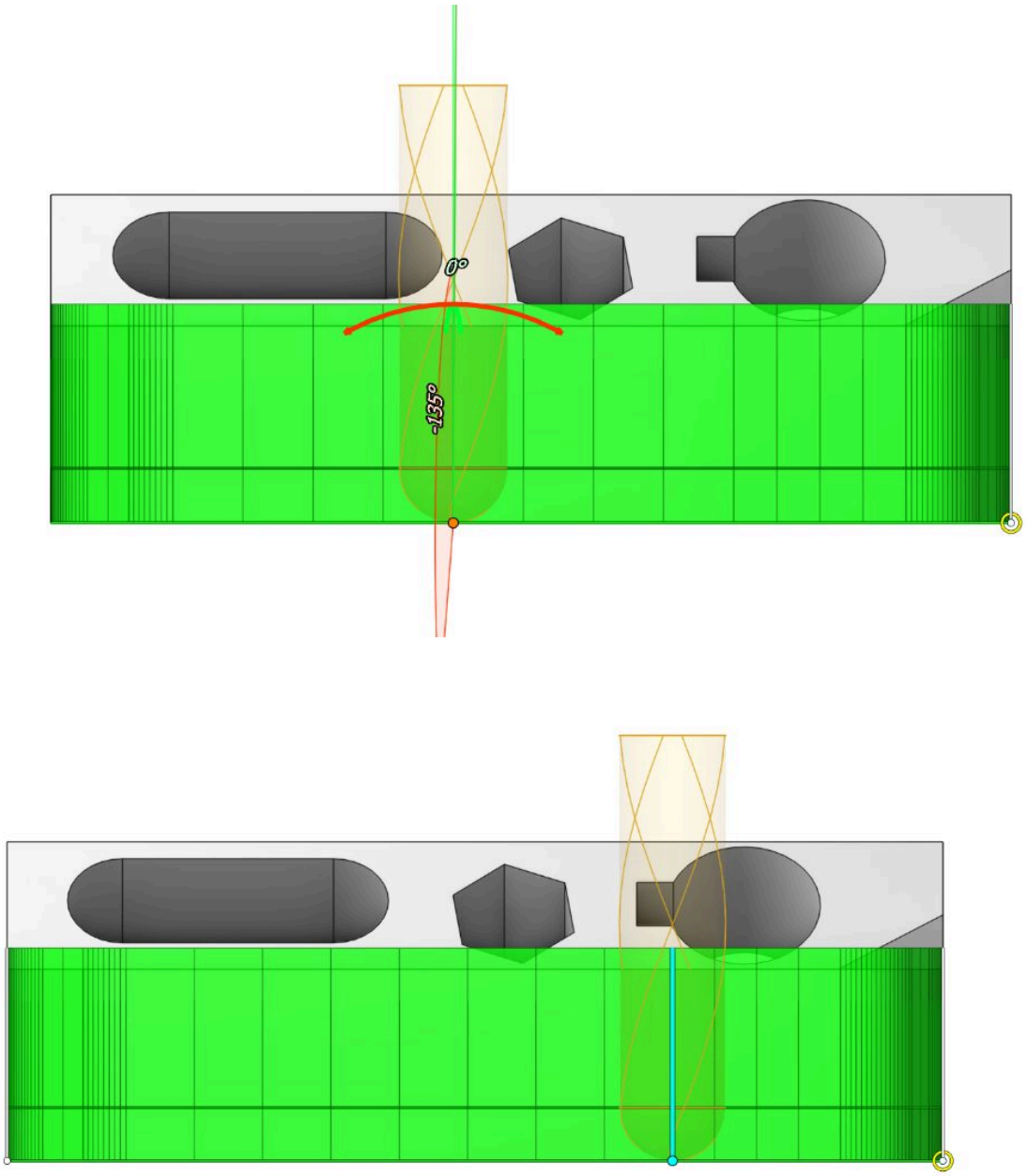
Alternate Front Side. An edge/curve might relate to two different surfaces. Pressing this button allows switching between one of the two surfaces.

Properties:

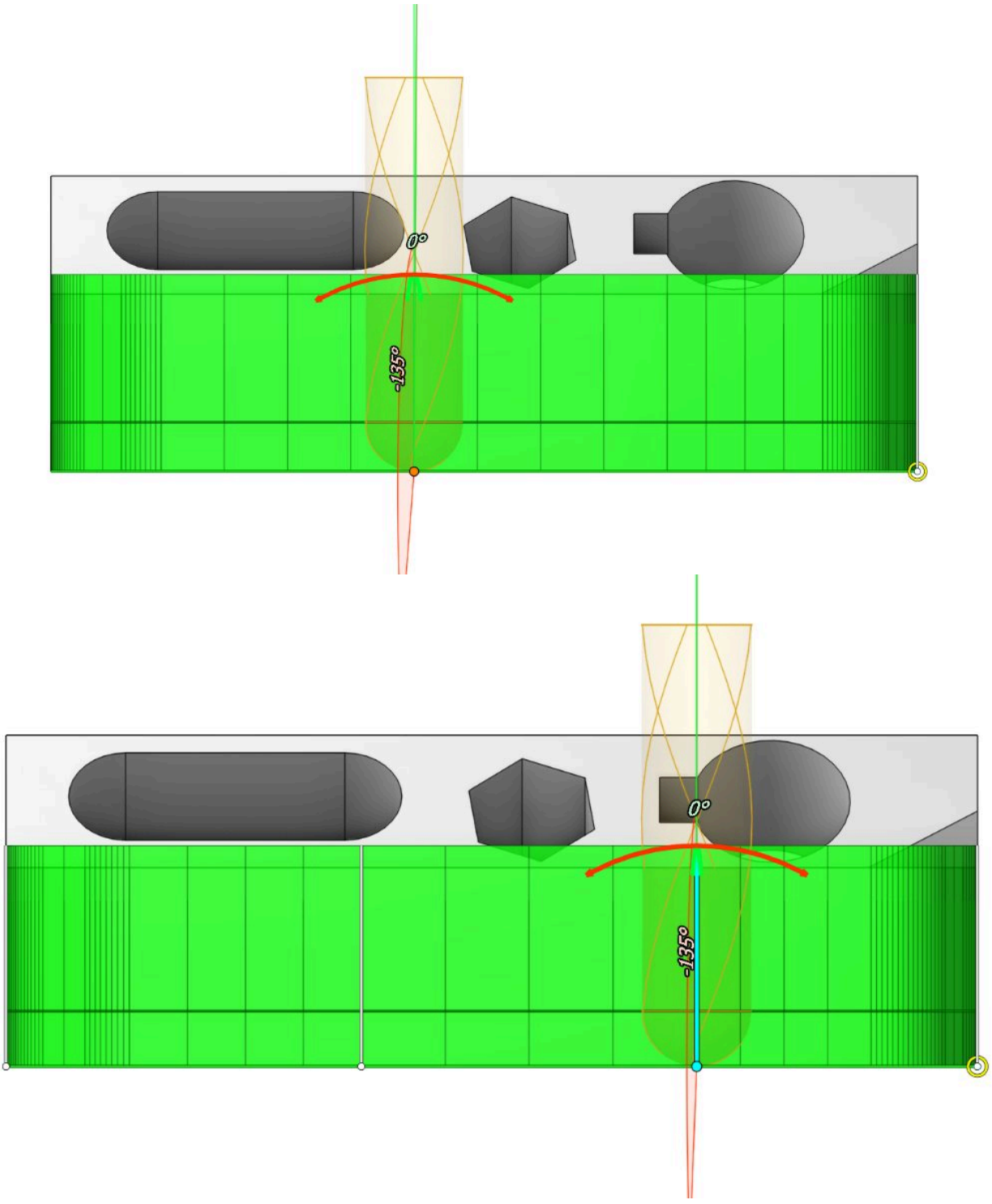
An important advantage of the 6D Contouring operation of CAM system is its comprehensive set of features for adjusting the tool movement parameters along the curve. You can edit these parameters interactively in the graphical window or in the dialog window.

▼ Click here to expand..
In the graphical window, you can adjust vectors that determine the tool orientation at points along the selected curve. This can be done in the following ways:

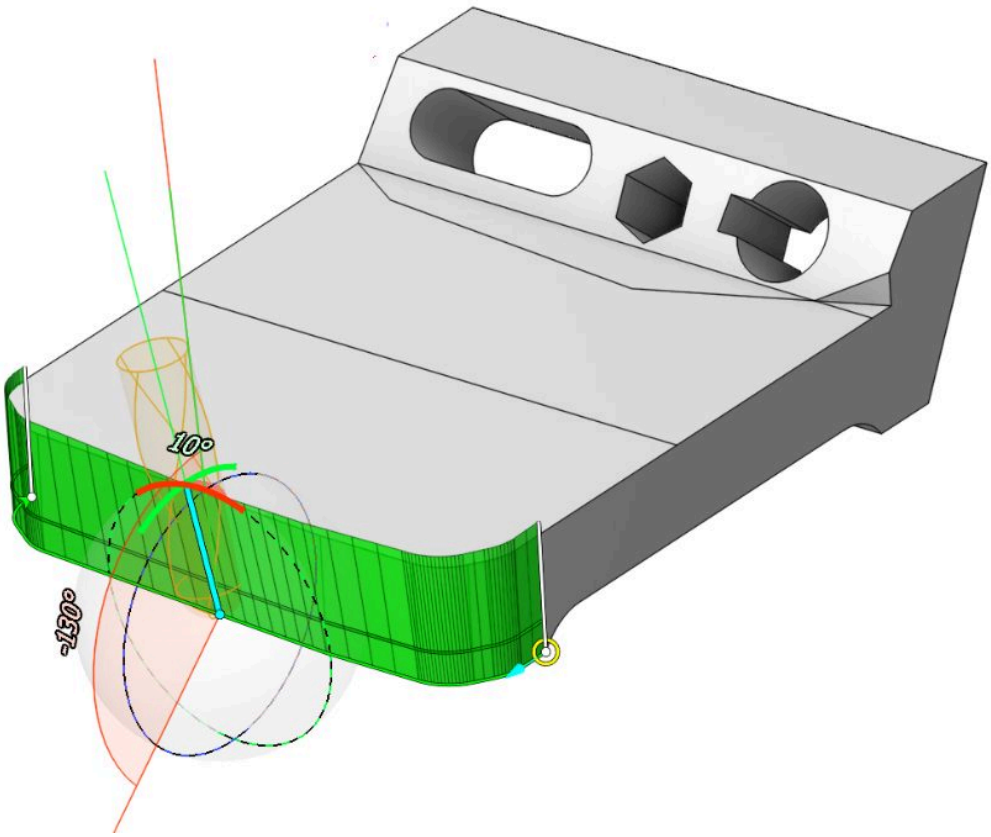
- clicking on the point on the curve and holding the left mouse button, you can move the vector along the contour.



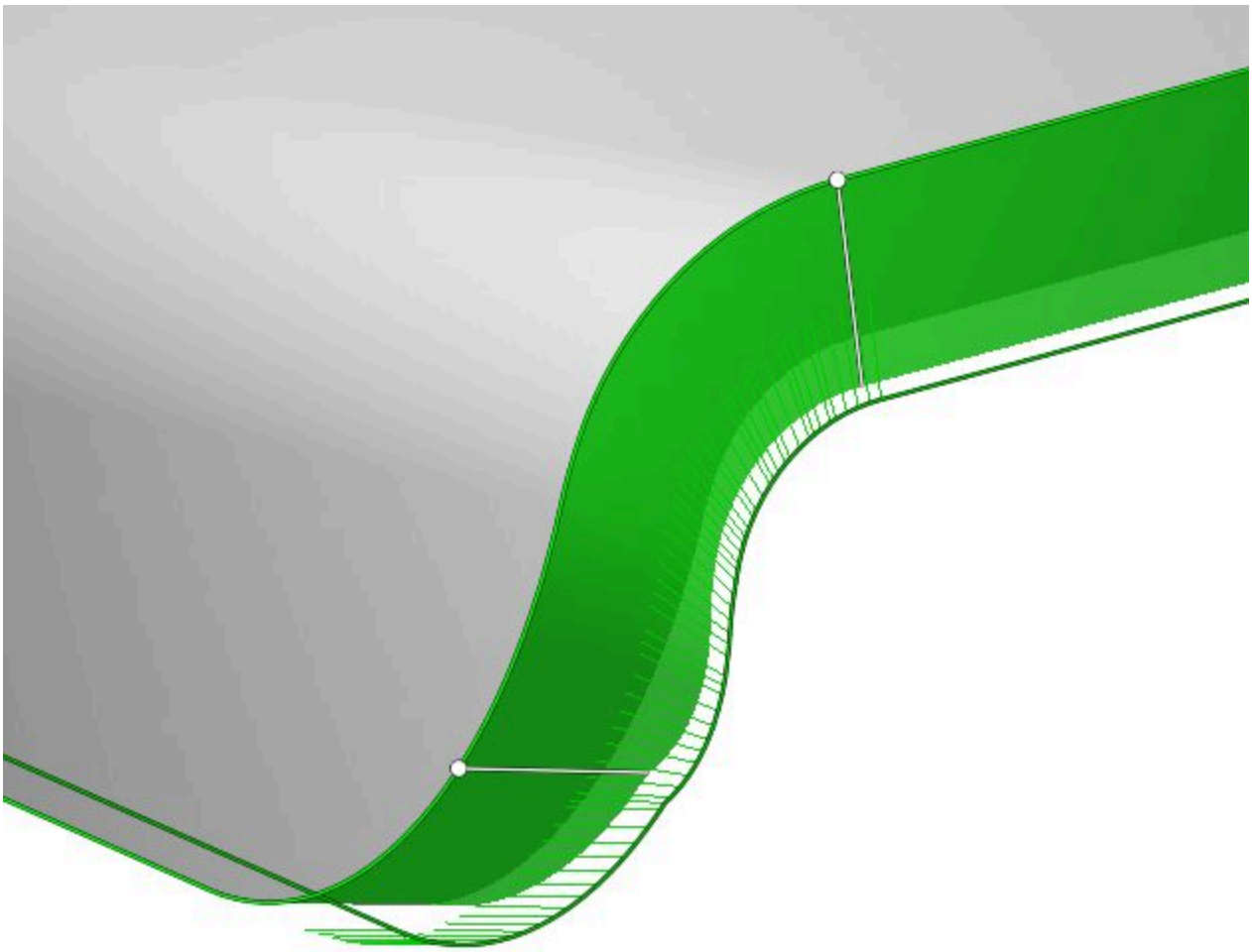
- if you do the same with the Ctrl key pressed, the copied vector will move, the main one will remain in place.



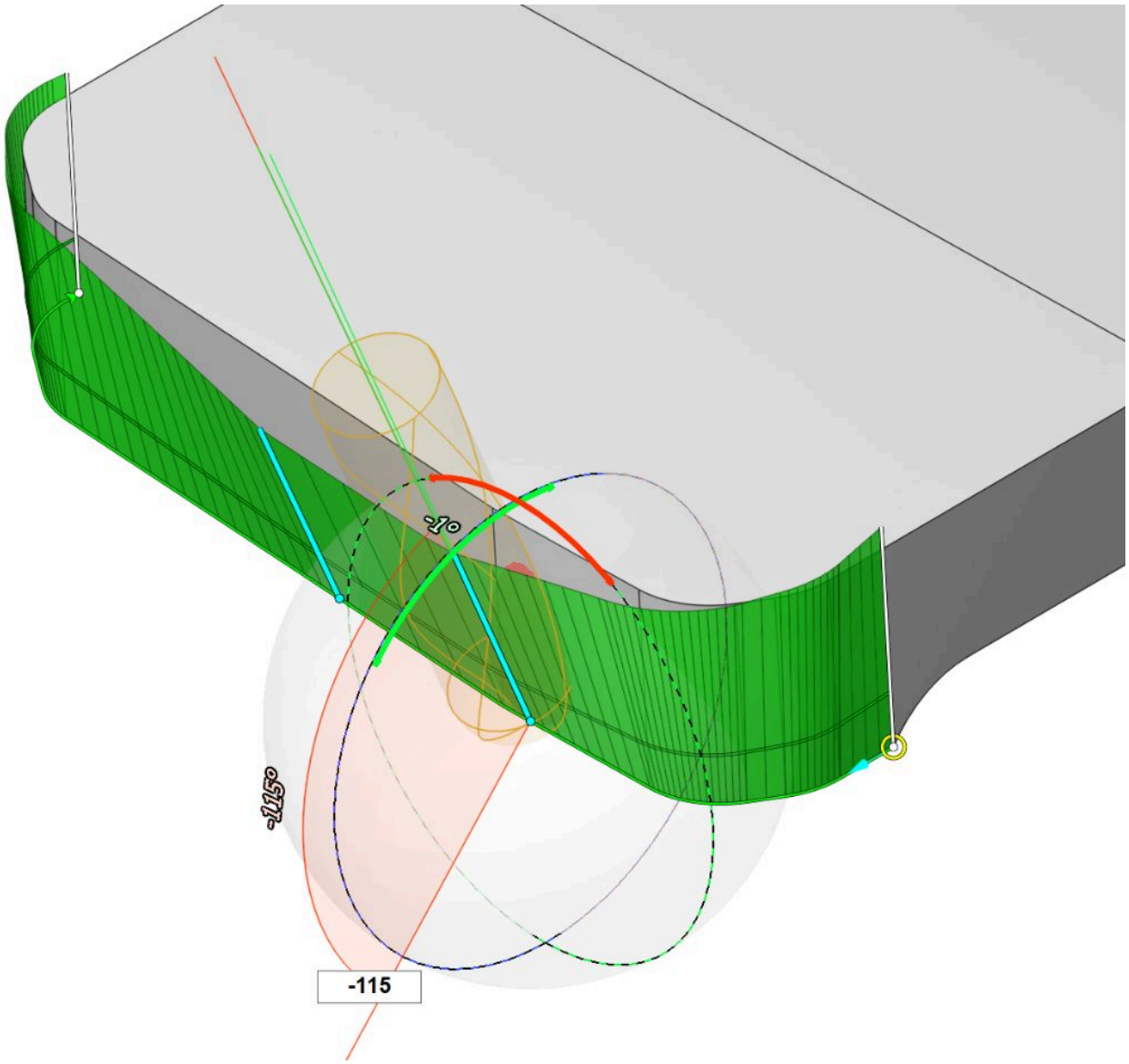
- by clicking on the vector, you can edit the direction through the interactive sphere, it has two circles along and across the original surface (or changing the value of two angles).



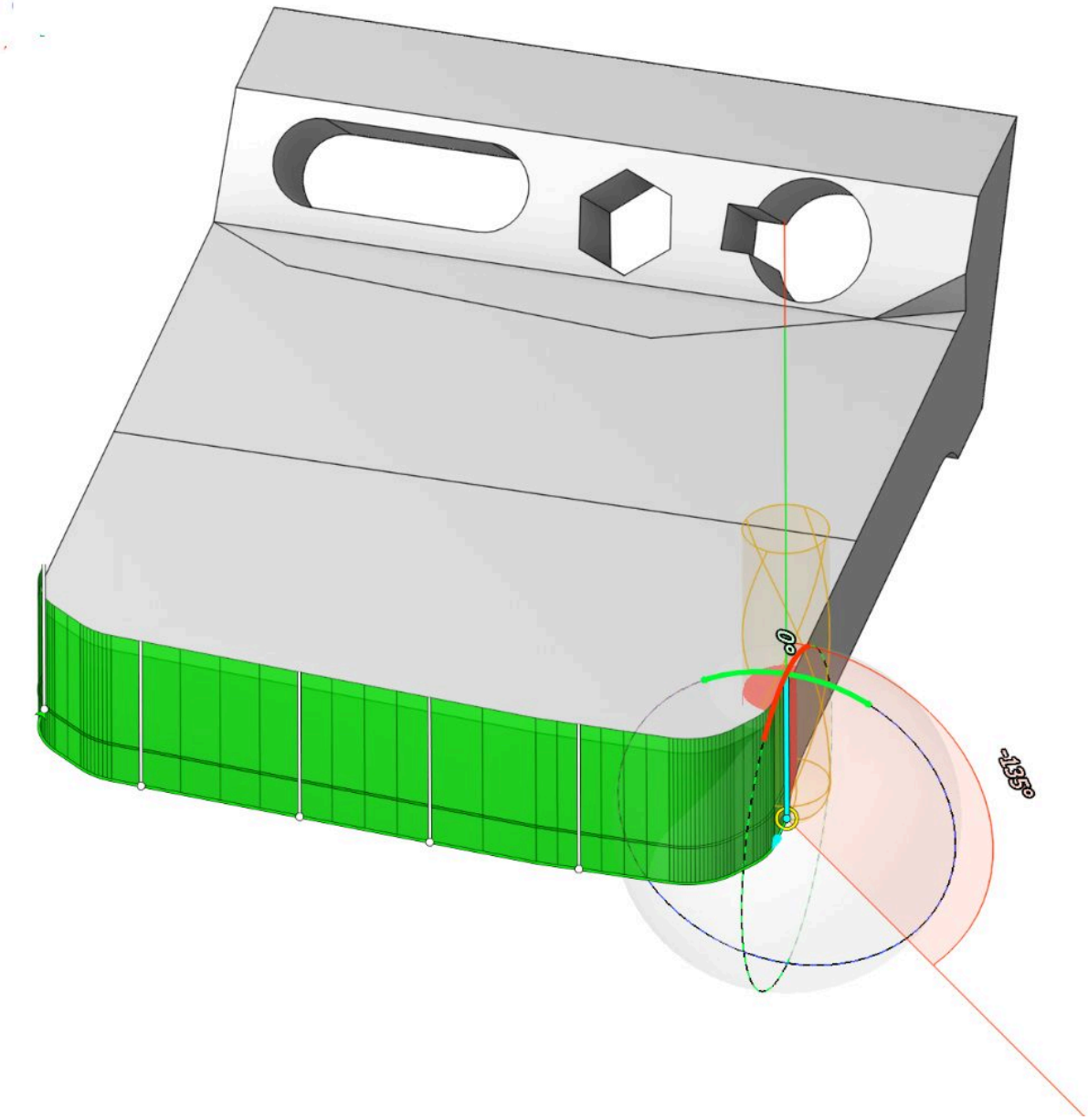
- holding Shift, the left mouse button has the ability to tilt the vector strictly to the point on the surface of the part.
- delete the vector by Del button on your keyboard.

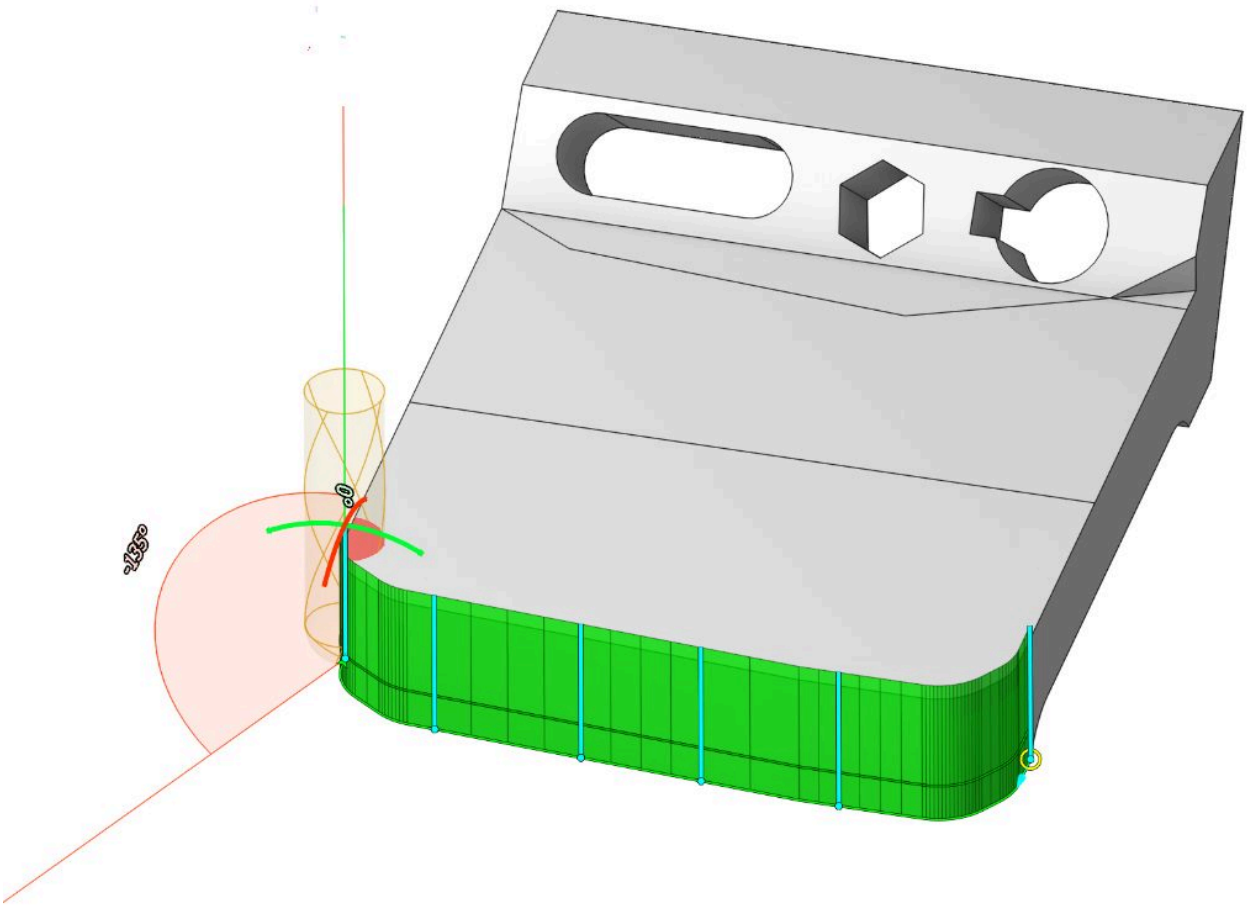


- Multi-select functionality expanded: Modifying a vector's angle via size adjustment or semi-circle rotation now propagates the angle change to all selected vectors. (using Ctrl/Shift)



- To select a group of vectors, Shift-click the first vector, then Shift-click the last vector. All vectors between them will be selected.


0°
-135°



By dragging the circular point, you can modify the beginning of the toolpath. Clicking the blue arrow allows for a change of the direction of the tool's path along the curve.

When you click on a curve, a pop-up window appears, enabling you to specify following parameters:



Lead in. Configure the tool axis direction and the length of engage. Upon activation of this feature, a yellow arrow and a line segment originating from a yellow dot become visible. Dragging this segment adjusts both the engage length and direction. By rotating the yellow arrow, the user can modify the tool's orientation when approach.

Lead out. Configure the tool axis direction and the length of retract. Upon activation of this feature, a yellow arrow and a line segment originating from a yellow dot become visible. Dragging this segment

adjusts both the retract length and direction. By rotating the yellow arrow, the user can modify the tool's orientation when retutrn.

Compensated. Adjusts for contact point compensation by switching the vector direction from along the tool axis to along the normal to the tool surface at the contact point.

Normal Interpolation Mode. Defines the rule for vector direction variation between points.

Relative to Curve. The vector change is based on accounting for the curve's tangent.

Linear. Vector modification follows a linear rule.

Smoothed. Modifies vector using a spherical principle.

Normal Blend Distance. The distance between two points along the toolpath where the tool's axis angle remains constant.

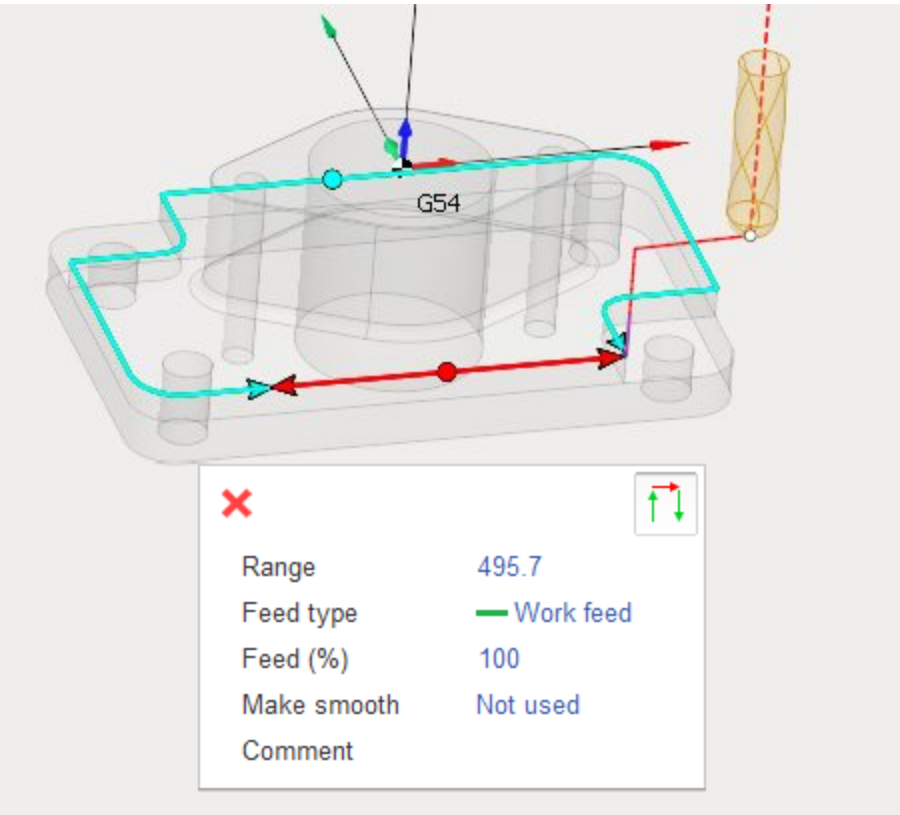
Set Tool Vectors Automatically. This feature appears when operating 5D Curve, CAM system will attempt to automatically set the vectors normal to the surface.

Use custom vectors. This feature appears when operating Edge/Curve on Surface. By default, system positions vectors to follow the designated surface. To enable vector editing in the graphical window, select the curve and click the Use Custom Vectors button.

Alternate Front Side. This feature appears when operating Edge/Curve on Surface. An edge might relate to two different surfaces. Pressing this button allows switching between one of the two surfaces.

Convert to Spline. If the curve is not a spline, pressing the corresponding button can transform the curve into a spline.

Feed Control Mode. This feature enables the creation of a contour segment with specific feed values. To use this feature, press the relevant button and then click on the curve segment where you wish to set the desired feed mode. In the pop-up window, select the Add New Feed command. You can move the processing segment to any location on the curve by dragging the point. Expand this segment along the curve by dragging the arrows. Double-clicking on the processing segment again brings up a pop-up window where you can enter the necessary feed control parameters:



Range. Specify the length of the contour segment where feed value adjustment occurs.

Feed Type. Determine the type of motion - Work feed or Rapid feed.

Feed (%). You can configure the feed value on the editable contour as a percentage of the Feed Type value determined in the Speeds tab.

Make Smooth. Facilitate smooth changes in feed rate.

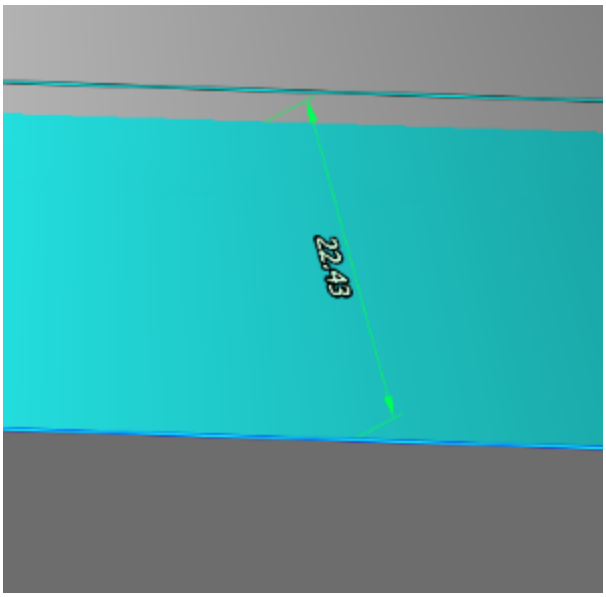
Not Used. Do not perform feed rate smoothing.

By (%) of Range. Modify the feed over a distance half the Step's length, calculated as a percentage of the Range, to achieve the Feed (%) value at the Range's midpoint.

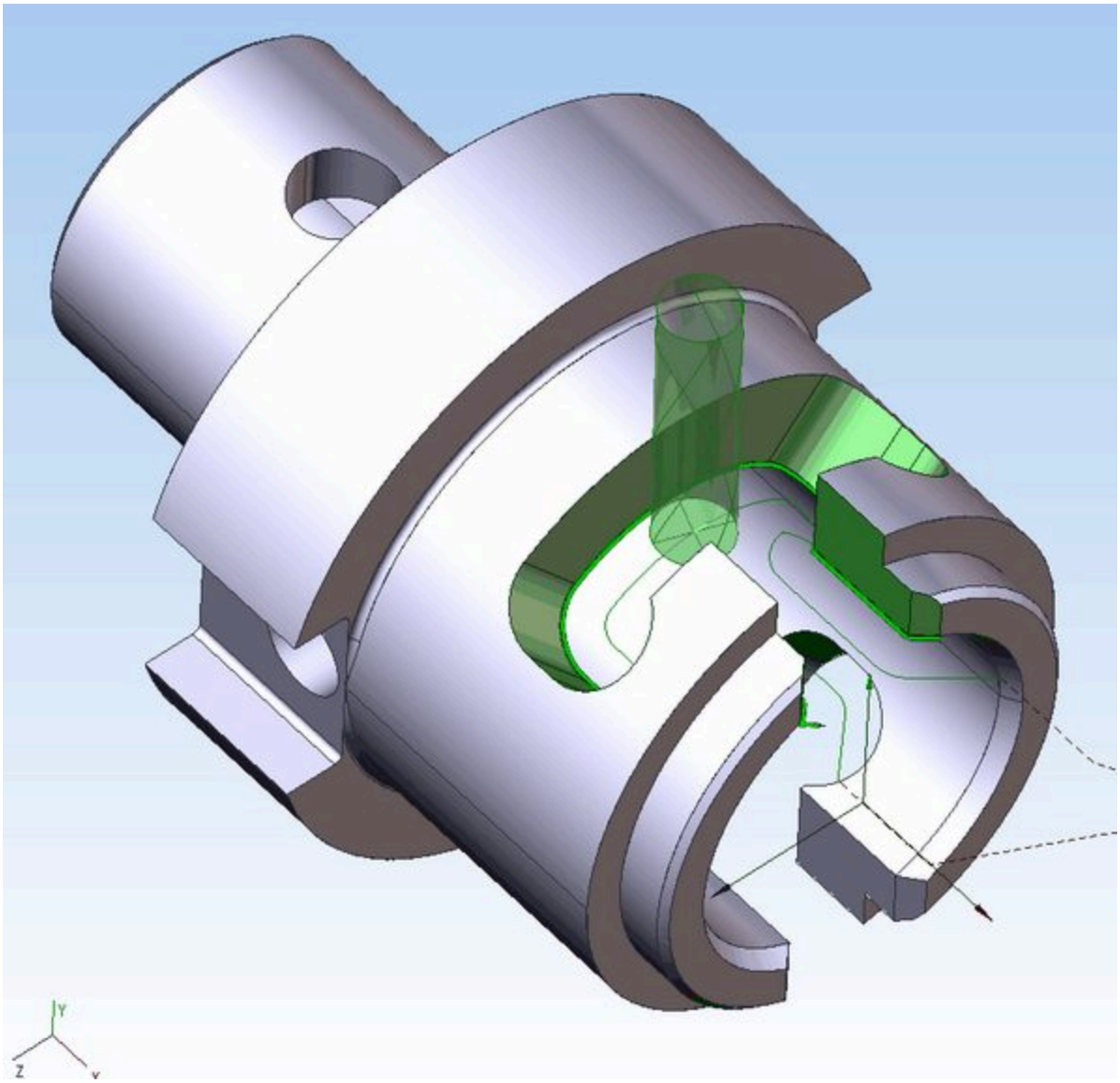
By Feed Step. Modify the feed stepwise using the Step value to achieve the specified Feed (%) at the Range's midpoint.

By pressing the Delete button you can remove the segment currently being modified in the feed mode configuration. Press the button Feed Control Mode once more to exit the mode for editing feed settings.

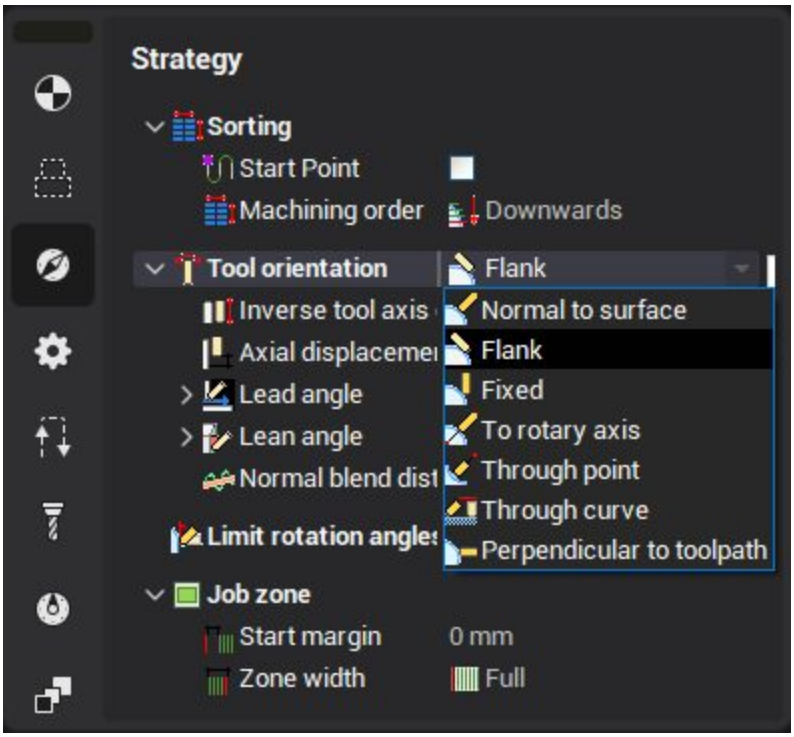
When you click on the curve, a dimension appears that enables vector Height adjustment. This only affects the convenience of working with them.


22.43

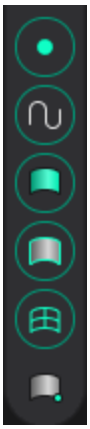
5-axis milling of the ruled surfaces by the flank of the mill



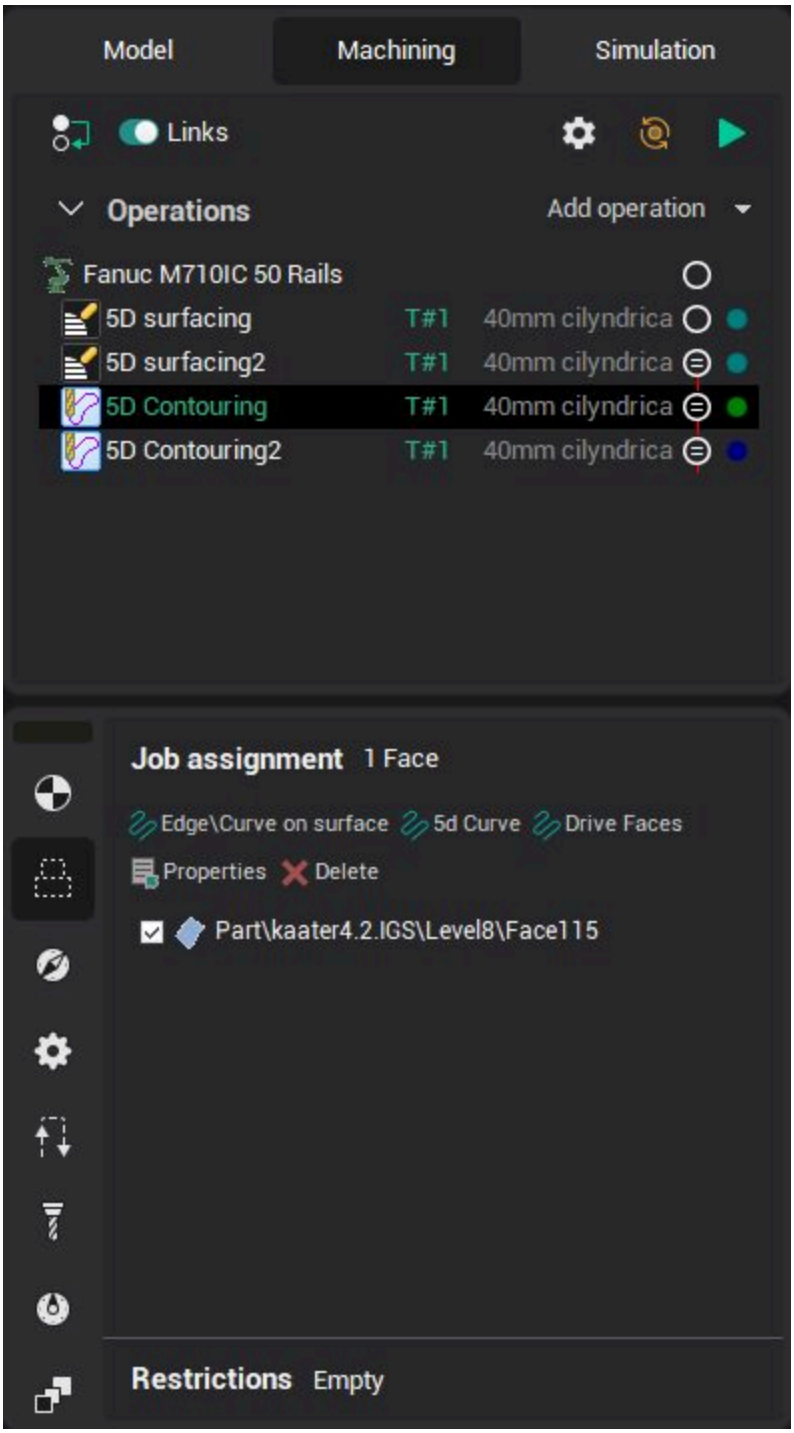
Set the Contact tool type parameter to the Flank position to machine the ruled surfaces by the flank of the mill. The Inverse tool axis direction tick changes the tool axis direction to the opposite one. Axial displacement defines the additional shift along the tool axis.



Job assignment is formed with the edges of the part or any curves. If the edges if not highlighted when the mouse pointer goes under it then check the edges selection button on the filter panel (the same applies to curves). It must be down.

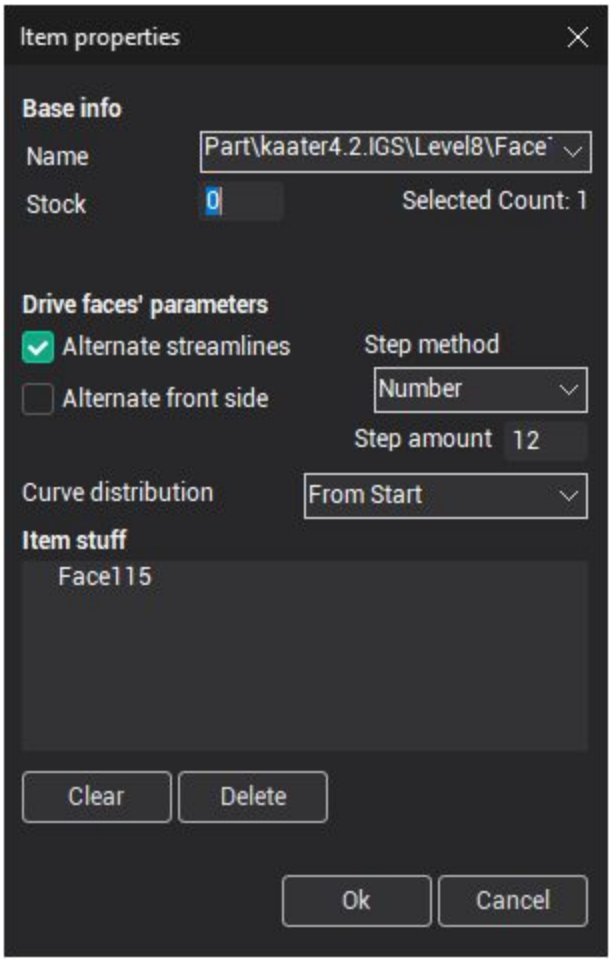


If the edge selection filter button is down but the edges is still not highlighted then sew the faces.



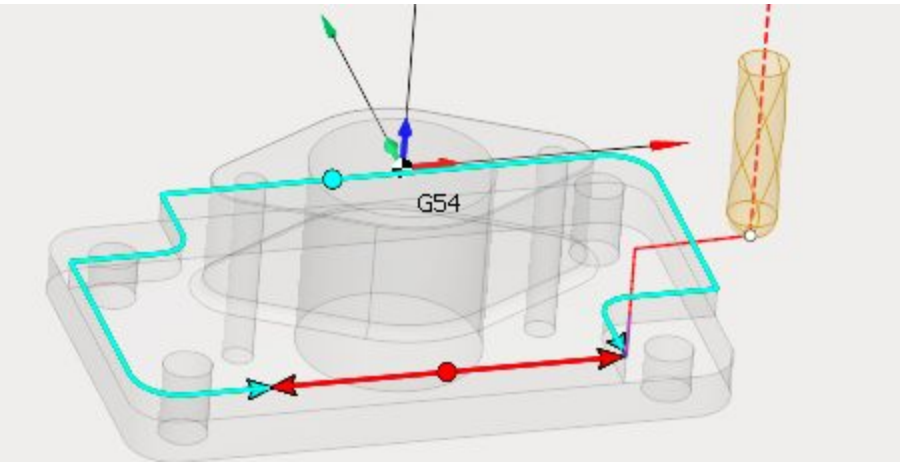
If the set of the connected edges must be machined in one pass, then it must be selected from the screen together and added to the job assignment list by the one click on the `Edge\Curve on surface` button. In this case the edges are joined to the one item and machined as a whole. If the edges are added to the list separately then it is machined separately even if it is connected. Press and hold the Ctrl to select some objects from the screen at once.

The direction and side of the machining can be changed item properties window. Select the item and click properties button to open it.

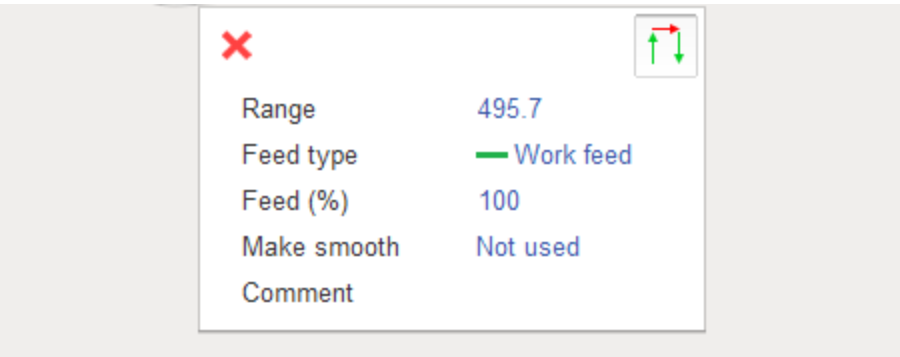

Item properties
Base info
Name
Part\kaater4.2.IGS\Level8\Face
Stock
0
Selected Count: 1
Drive faces' parameters
Alternate streamlines
Alternate front side
Step method
Number
Step amount
12
Curve distribution
From Start
Item stuff
Face115
Clear
Delete
Ok
Cancel

Feeds control feature

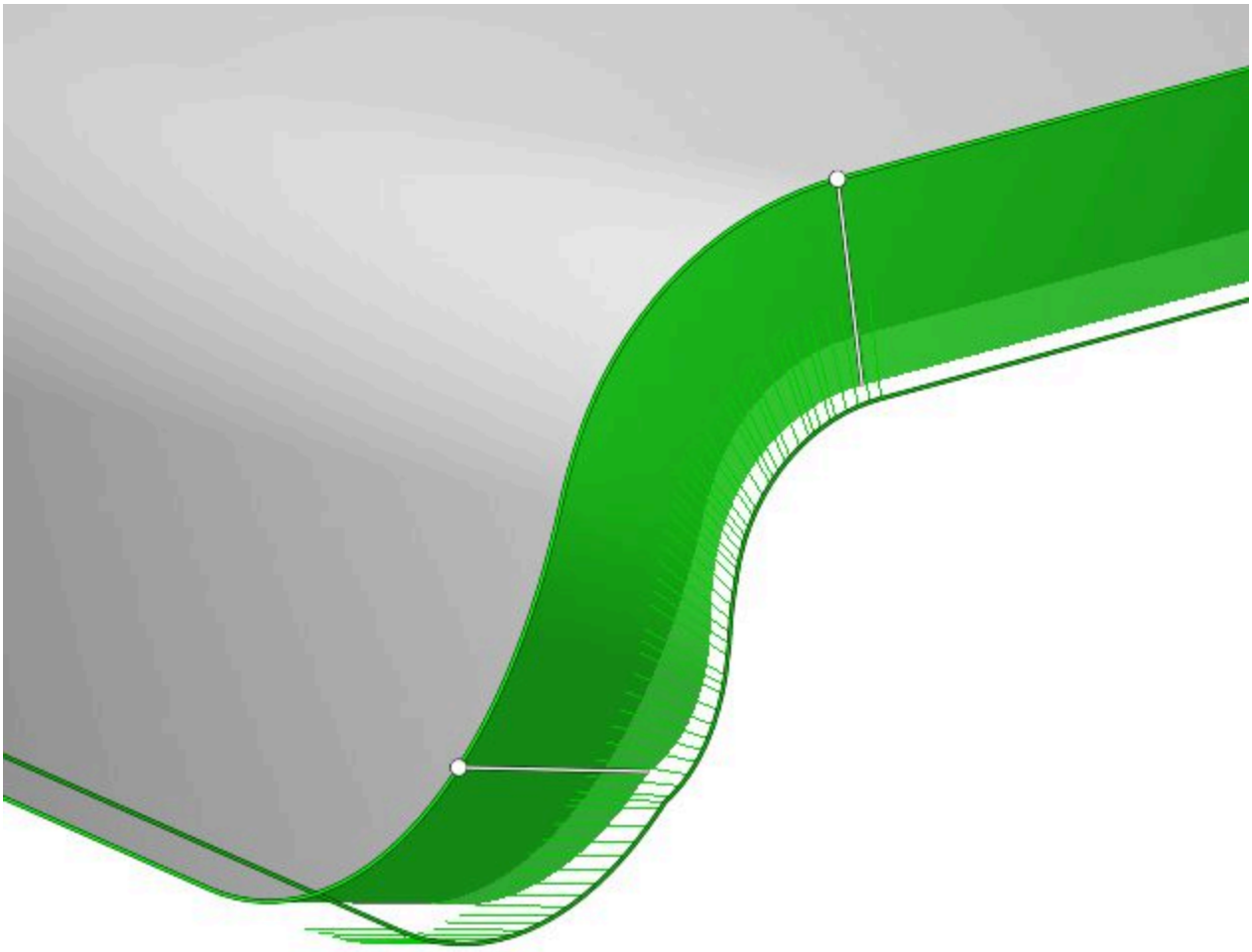
This function allows you to create feed value contour. In order to have the ability to edit feed value you need to click Feed control mode.



In the pop-up panel, you can control the range, feed type, speed and interpolation type.



Using custom vectors feature



Vectors allow to set the orientation of the tool at any point of the contour. In order to have the ability to edit vectors you need to click Use custom tool vectors.

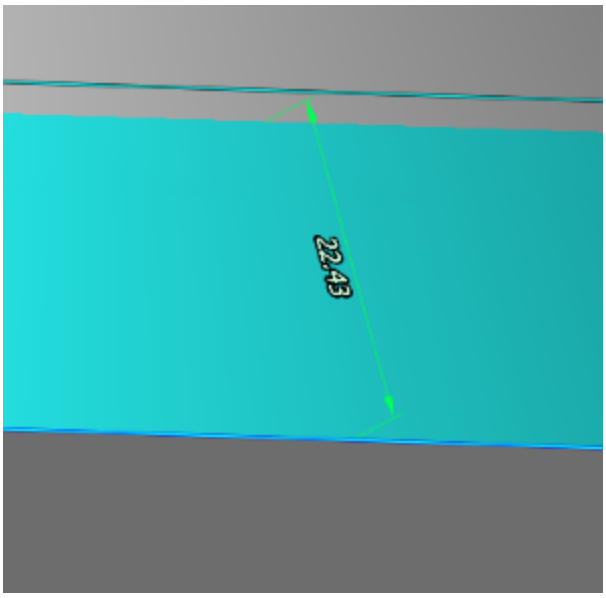
If a 5D Curve was added to the Job assignment, instead of an Edges, then by default there is available only one Fixed vector. By pressing the same button, CAM system will try to set the vectors automatically.



Using Custom vectors allows you to edit the original surface to be processed. Editing methods:

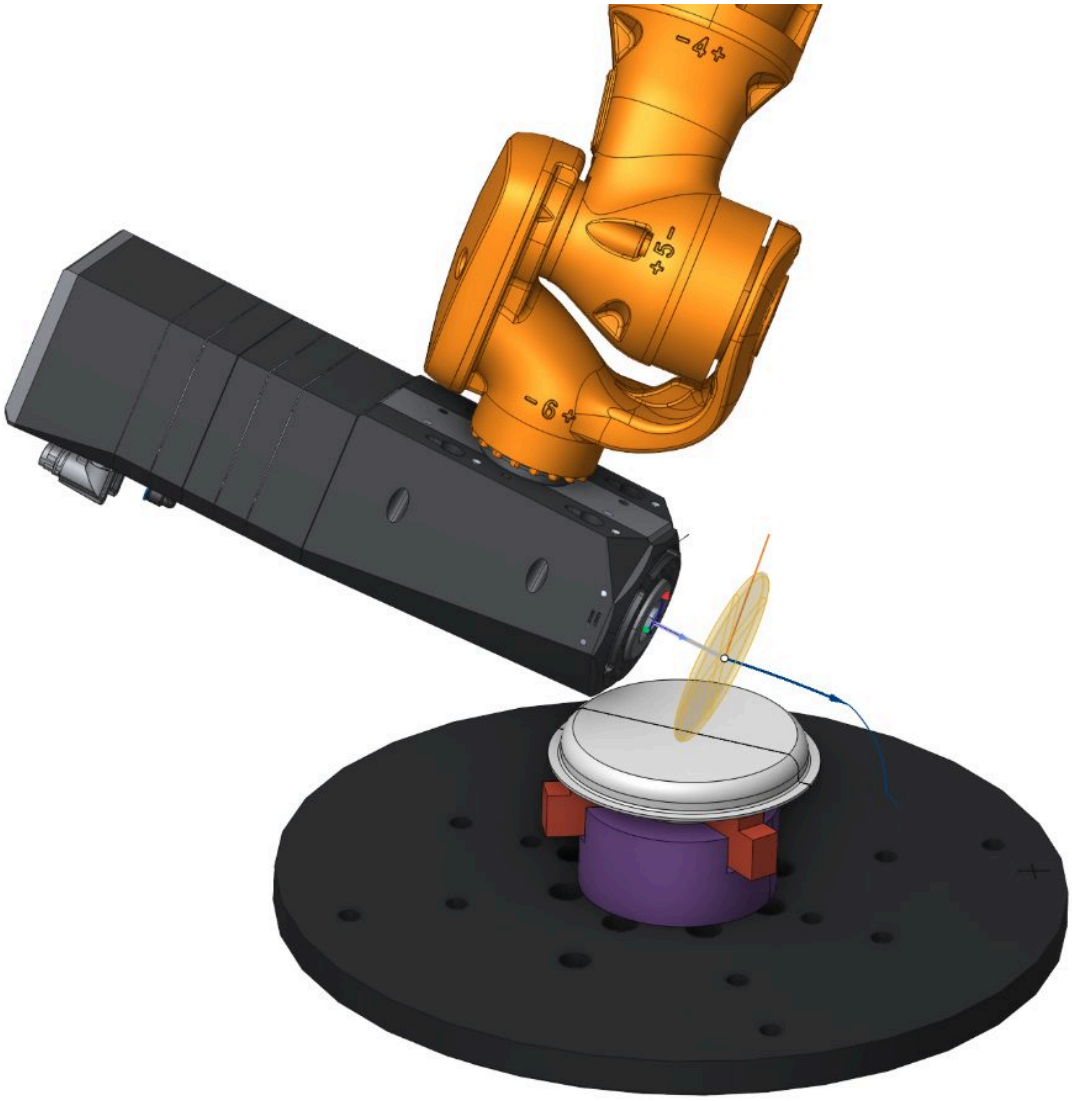
- holding the left mouse button, you can move the vector along the contour.
- if you do the same with the Ctrl key pressed, the copied vector will move, the main one will remain in place.
- by clicking on the vector, you can edit the direction through the interactive sphere, it has two circles along and across the original surface (or changing the value of two angles).

- holding Shift, the left mouse button has the ability to tilt the vector strictly to the point on the surface of the part.
- delete the vector by Del button on your keyboard.



If to click on the contour, a size appears with which you can edit the height of the vectors. This only affects the convenience of working with them.

Contour Polishing



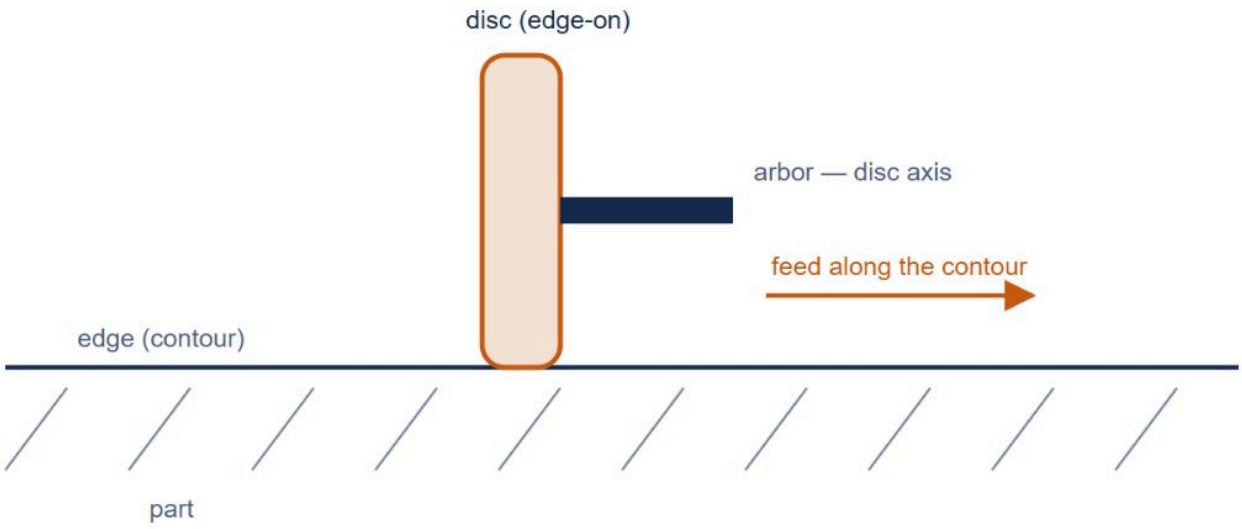
Application area:

The operation is designed for finishing the walls and edges of a part along specified contours using a disc tool. Contact is formed on the disc rim. The operation supports three-dimensional contours and undercut areas.

Machining principle.

The disc is positioned across the machined edge. The feed follows the contour, while the tool rotation produces polishing motion across the edge. At each toolpath point, the position of the disc is determined by contact with the part.

Segments without contact, or where the part obstructs the disc, are trimmed from the working pass. The remaining passes are connected by links over the safety surface.



Top view. The contour axis points towards the viewer. The disc contacts the edge with its rim; feed is directed along the contour and polishing motion is directed across the edge.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

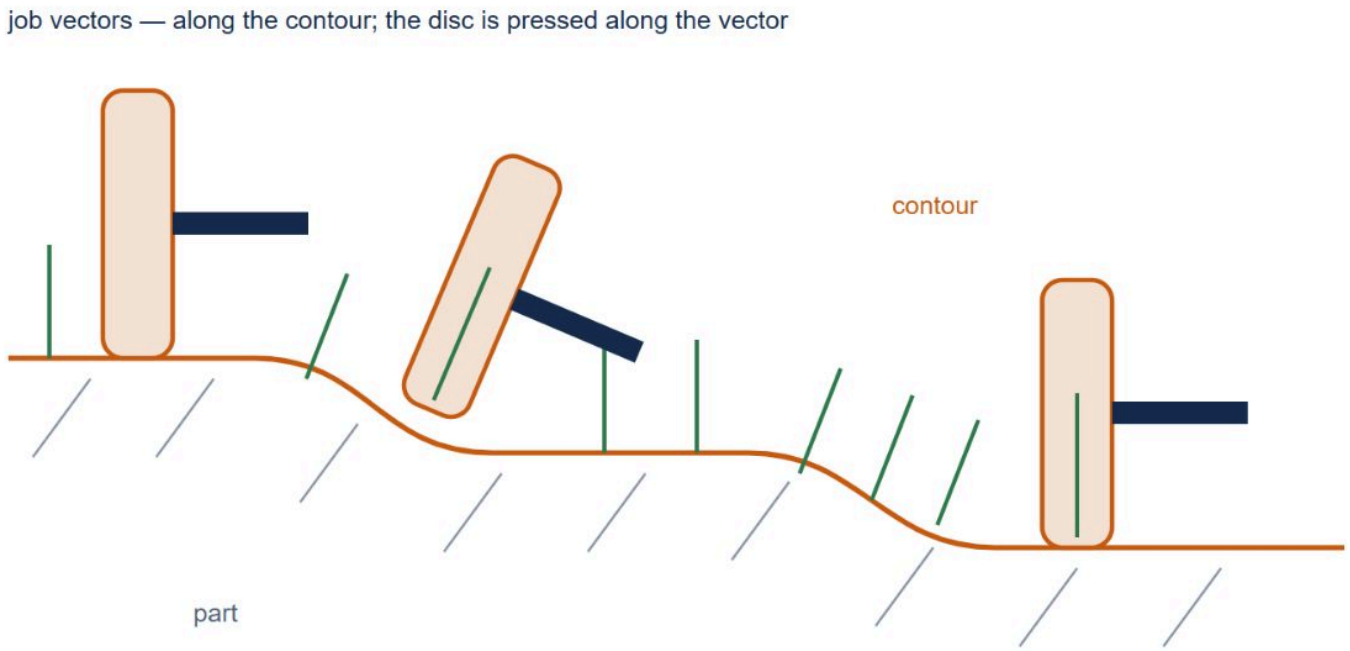
Job assignment:

5D Curve. The system generates the 5-axis tool path as the projection of the arbitrary curves on the part surface. In this case the tool contact point will move in touch with the surface. The passes can be shifted from the curve by the stock or by the levels. The tool axis is located perpendicular to the surface. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details



Top view. Job vectors are perpendicular to the contour axis and define the side from which the tool contacts the part. Reversing the vectors changes the machining side.

Strategy:

Contour axis.

Defines the reference direction perpendicular to the job vectors. In Roll mode, this direction also determines the fixed tool axis.

▼ Click here to expand..
Direction. Selects X, Y, Z or Custom. The default direction is Z. Custom defines an arbitrary vector.

Origin. Defines the X, Y and Z coordinates of a point on the axis. The point is used to position the graphical representation of the axis.

Base CS. Defines the coordinate system in which the axis is specified: Global or WCS.

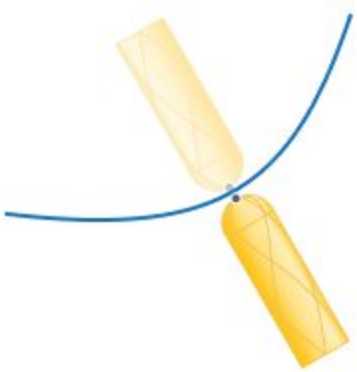
Tool contact point.

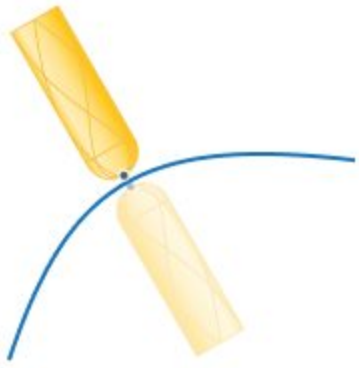
Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations. The principal parameters of this function are the same as in the 5D Surfacing operation. Details.

Tool orientation

Controls tool axis orientation. The principal parameters of this function are the same as in the 5D Surfacing operation. Details.

Inverse tool axis direction. Alters the tool axis vector direction. Use this parameter if CAM system incorrectly attempts to machine surfaces from the opposite side to the desired one or if it fails to generate a toolpath.





Parameters:

Parameters. Contains the main settings for controlling how the part, workpiece, fixtures, and other geometry affect the generated toolpath. Details

Links/Leads:

Links/Leads. Controls approach and return movements, transitions over the safe surface, and the distances used when approaching or retracting from the working path.

▼ Click here to expand..

Consists of:

Approach/Return. Defines the Approach and Return rules and the Tool change position. These settings control tool movements to the beginning of the working path and away from its end. Details

Safe motions. Defines the Safe surface and Safe level used for transitions between working passes and for approach and return movements. Details

Distances. Provides Max retraction distance to control retraction along the tool axis and Feed distance to define the feed-switching distance along the tool axis. Details

Feeds/Speeds:

Feeds/Speeds. Defines the spindle speed, rapid feed, and feed rates for different parts of the toolpath. Spindle speed can be specified in revolutions per minute or as a cutting speed. The defining value is underlined; the other value is calculated from it using the tool diameter. Details

Transformations:

Transformations. Defines coordinate transformations applied to the toolpath calculated for the operation. Details

Part:

Part. Defines the geometry used to check the toolpath for gouges. Details

Workpiece:

Workpiece. Defines the material to be machined in the operation. Details

Fixtures:

Fixtures. Defines workholding devices, such as chucks, grips, and clamps, and other restricted geometry considered during toolpath calculation. Details

Rotary

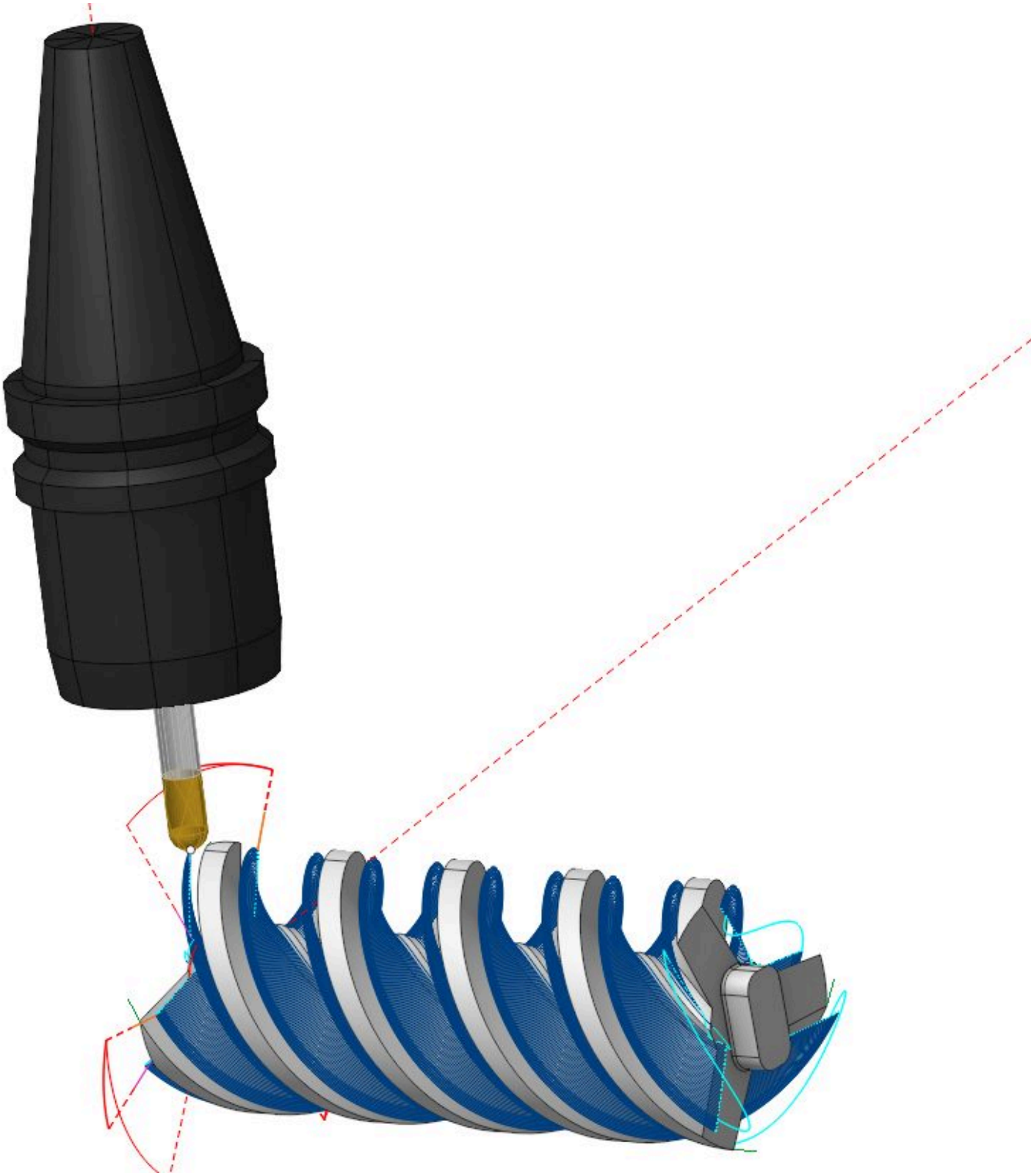
Application area:

The Rotary group contains operations that use a rotary axis for machining surfaces and contours. The topics cover rotary roughing and finishing, waterline machining, morphing, surfacing, and contouring.

Operations:

- Rotary waterline. Generates waterline passes around a rotary axis. Details
- Rotary roughing. Removes stock in layers around a rotary axis. Details
- Rotary finishing. Finishes surfaces around a rotary axis. Details
- Morph 4D. Generates four-axis morphing toolpaths. Details
- 4D Surfacing. Generates four-axis surface machining toolpaths. Details
- 4D Contouring. Generates four-axis contouring toolpaths. Details

Rotary waterline operation



Application Area:

This operation is based on the 5D surfacing operation and preconfigurated for a rotary machining of models like a screw and body rotation. It is available for 4x and 5x configrations, but it locks many five-axis machining strategies, tool incline angles, singularity avoidance functions, and axis deviation. It only

comes pre-set with the Around Rotaty Axis strategy and the Tool Orientation is set to To Rotary Axis only. So, t he system generates the tool p asses as sections of the Machining Surface at a specified Step by employing cylinders with an axis specified as the Rotary Axis. The operation will process the entire model when the Job Assignment is empty.

Job Assignment:

Machining Surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

Surfaces for Projection. These are the s urfaces intended for machining onto which the toolpath transfers when enabling the Project Toolpath onto the Part option in the Strategy tab.

Job Zone. Use the Job Zone to trim the passes outside the specified 2D containment areas. You can select the normal to the area of Job Zone. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that s hould avoid machining in the current operation. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

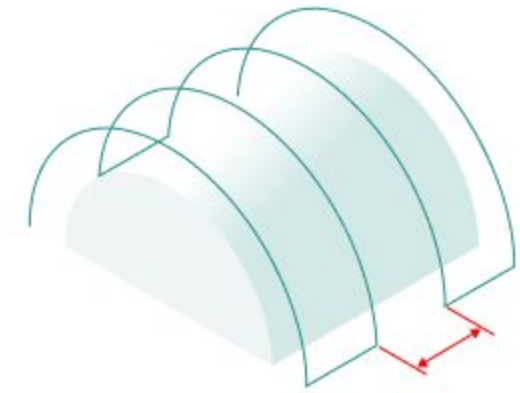
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Step:

The maximum allowed width of cut.

▼ Click here to expand..

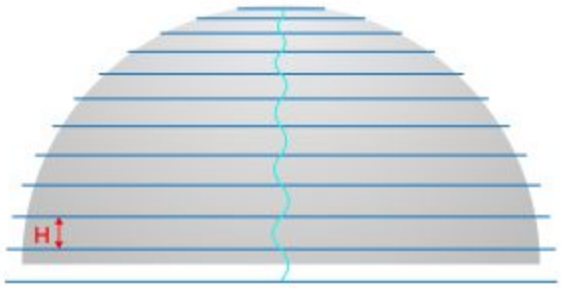


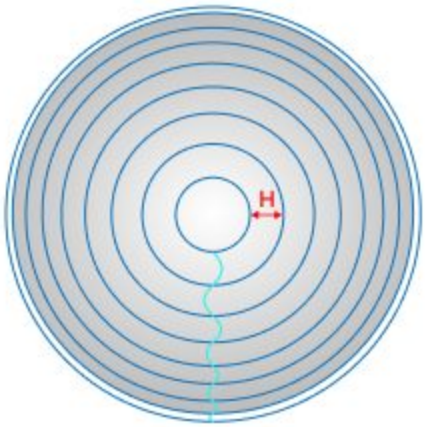
Adaptive Step:

▼ Click here to expand..

If enabled: This parameter adjusts the distances between tool passes according to the curvature radii of the surface and the tool to maintain uniform scallop height after surface machining. Details.

If disabled: This is the fixed vertical distance between consecutive tool passes





H - Crest height / step

Milling Type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details

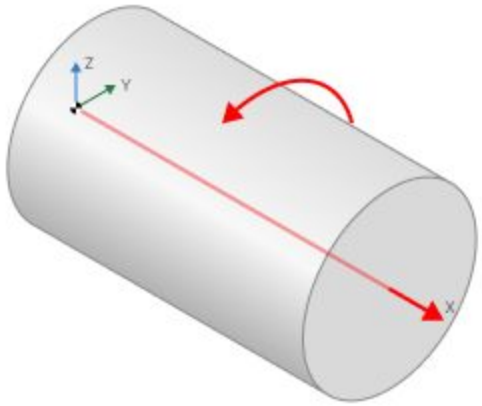
Sorting:

Controls the sequence of toolpath passes during surface machining. The parameters of this group are the same, as in the 5D Surfacing operation. Details.

Rotary Axis.

This is t he axis of section cylinders used to construct the tool path lines.

▼ Click here to expand..



Job zone:

Enables additional modification of the working area initially selected on the Job assignment tab.

▼ Click here to expand..

R min. The smallest radius from the Rotary Axis, where tool passes occur.

R max. The largest radius from the Rotary Axis, where tool passes occur.

Corners smoothing:

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. The parameters of this group are the same, as in the 5D Surfacing operation. Details

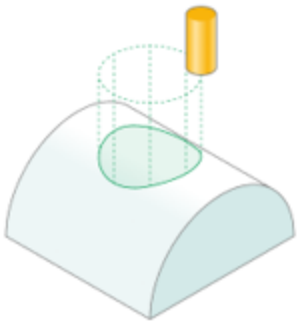
Roughing Passes.

Toolpath is offset in plane by the Roughing passes thickness amount. Additional passes are added to the contour by the value specified in the parameters with the specified number of steps. The parameters of this group are the same, as in the 5D Surfacing operation. Details

Project Toolpath onto the Part:

By setting the flag, the system generates tool paths for processing the surfaces identified as Surfaces for Projection in the Job Assignment by mapping the path derived from the Machining Surface onto the Surfaces for Projection.

▼ Click here to expand..



In this instance, identify auxiliary geometry surfaces, obtainable through extra constructions in the CAD module, as the Machining Surfaces instead of the part surfaces needing direct milling. Arrange these surfaces so that projecting the toolpath from them is straightforward. Declare the milling surfaces as the Surfaces for Projection.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of this group are the same, as in the 5D Surfacing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

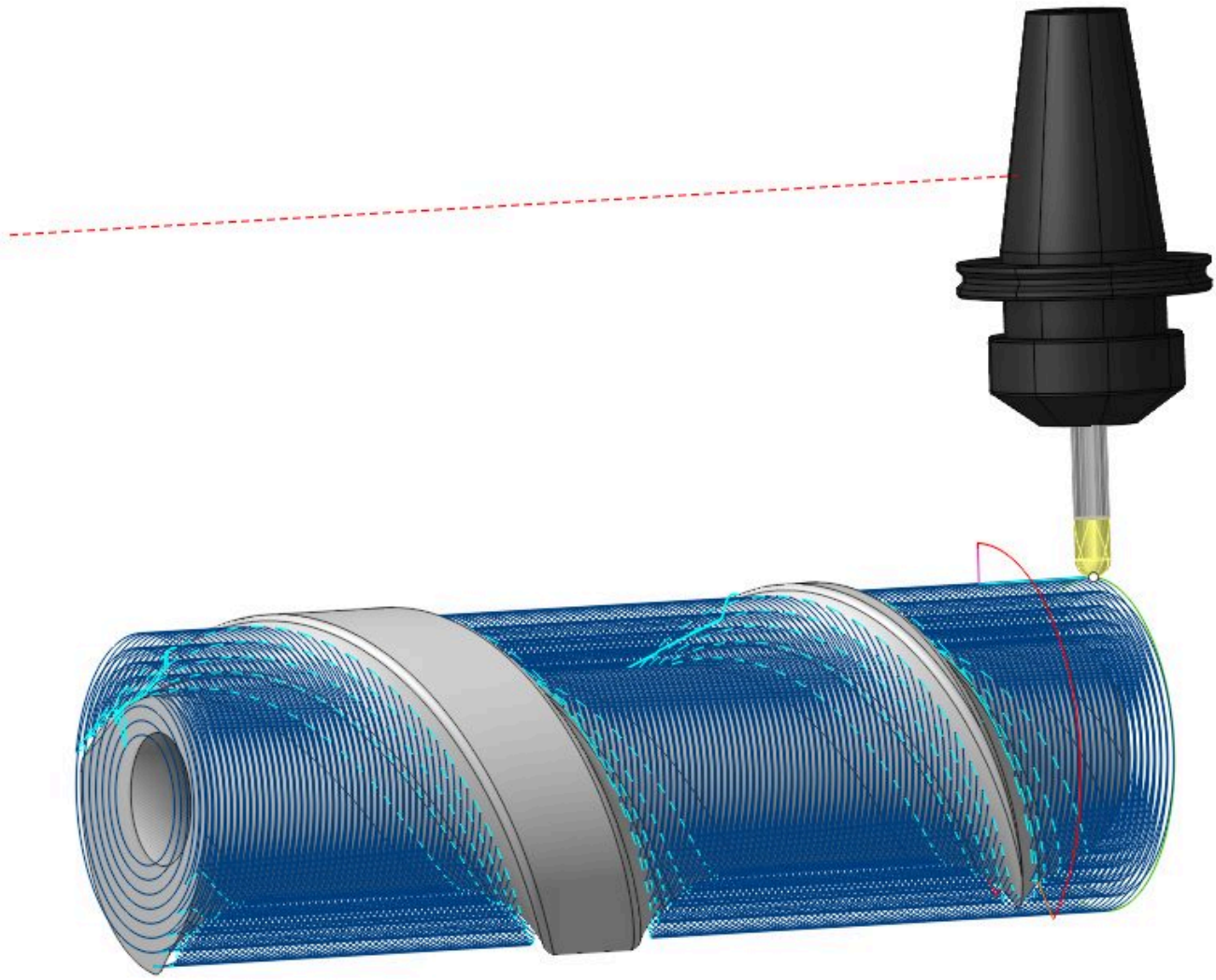
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Rotary roughing



Application Area:

Rotary Roughing is a continuous rough four-axis machining operation that removes the workpiece material layer by layer. It is similar to the Roughing Waterlineoperation except that the machining layers are not planes, but cylinders with their axis aligned along the Rotary Axis.

Job Assignment:

Spiral curve. The curve needed to indicate the angle of the spiral for the strategy Side angle.

Top Level. Specifying the top level by model elements. This level defines the maximum radius on the model where machining happens. Details

Bottom Level. Specifying the bottom level by model elements. This level defines the minimum radius on the model where machining happens. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

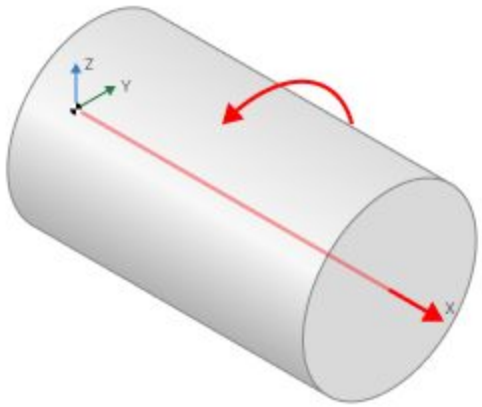
Strategy.

Job zone.

It enables additional modification of the working area initially selected on the Job Assignment tab.

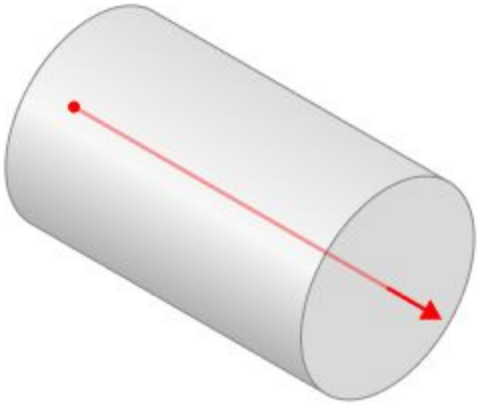
▼ Click here to expand..
Rotary axis. The system directs tool's axis vector from the designated Rotary Axis. The rotary axis position can be defined in the properties inspector. You can adjust the rotary axis interactively using the arrow or select it from the drop-down menu.

If the mode is set to the WCS X (GlobalCS X), WCS Y (GlobalCS Y) or WCS Z (GlobalCS Z), then the axis of rotation will be oriented along the corresponding axis of the coordinate system.



Direction. If mode is WCS (GlobalCS) (0,0,0) then axis vector must be defined manually in the fields X,Y,Z of custom direction group. Specify the direction cosines relative to each axis.

Origin. The Origin field defines the point that acts as the coordinate start for the chosen rotary axis.



Base CS. The Base Coordinate System (Base CS) changes the coordinate system between the global and local coordinate systems of the part.

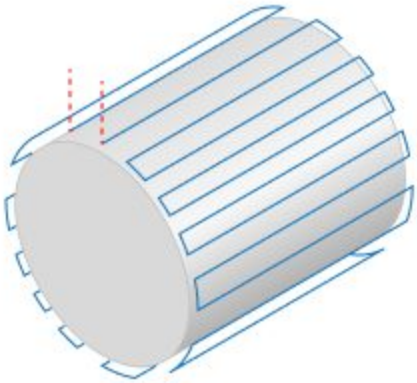
Some parameters works similarly to the Rotary finishing . Details

Trajectory form.

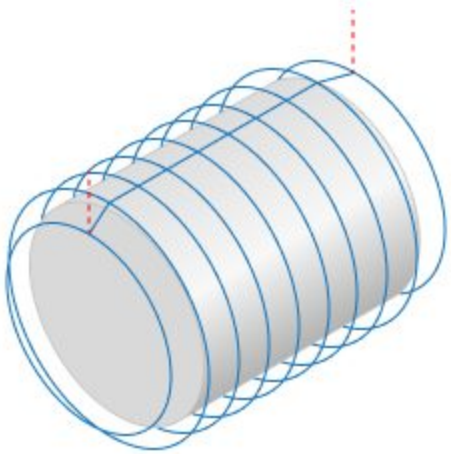
The trajectory output format.

▼ Click here to expand..
The trajectory can be in one of the following forms:

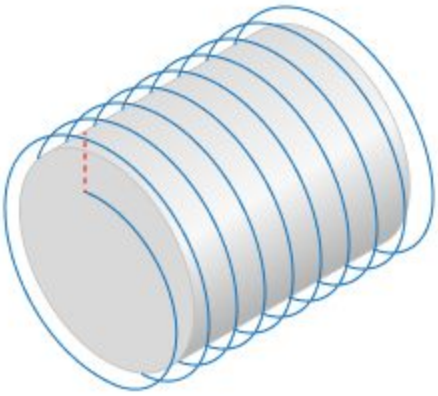
Linear. When you select this option, the work passes form as intersection lines between the machined surfaces and planes through the Rotary Axis..



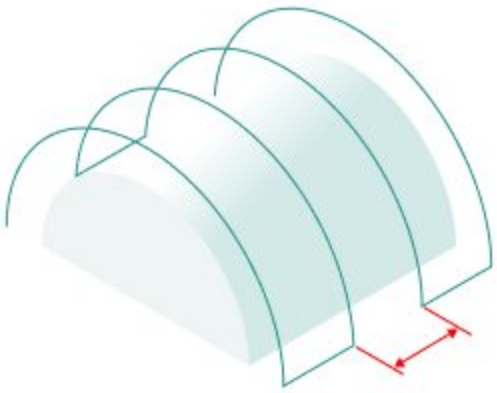
Circular. When you select this option, the work passes form as intersection lines between the machined surfaces and planes perpendicular to the Rotary Axis.



Spiral. Activating this option shapes the toolpath into a continuous helical line. The pattern is well suited for machining parts like screws and impellers.

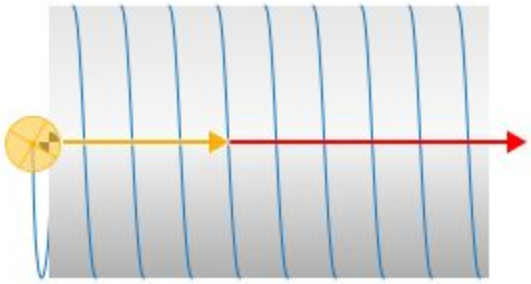


Step. It is the distance between tool passes along the rotary axis.

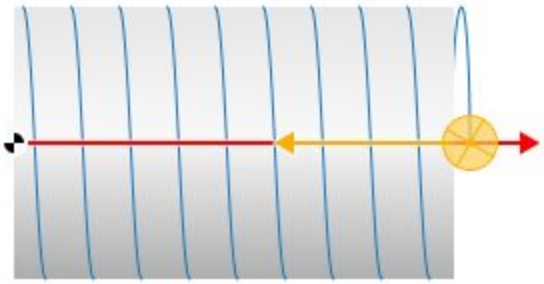


Axial direction. Depending on the feature of the Trajectory Form parameter, it either defines the direction of the tool passes themselves or the sequence of several tool passes. In the case of a spiral whose step matches the machining Step, this parameter influences the direction of the spiral's twist.

Forward. The tool passes is ordered in the Rotary Axis vector direction.



Backward. The tool passes are ordered in the reverse direction of the Rotary Axis vector.



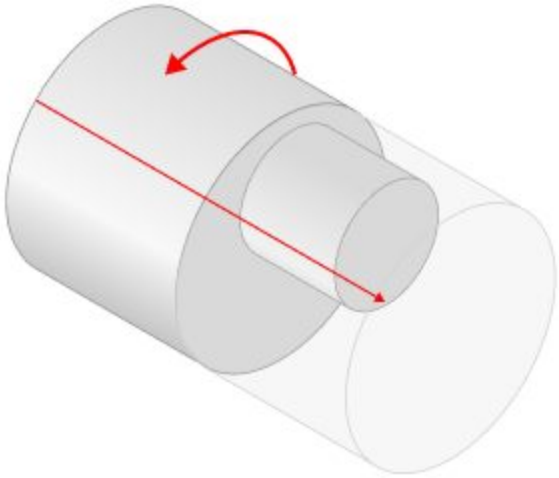
Spiral step. It defines parameters of the Spiral toolpath. The spiral step value can be positive or negative. The sign defines the torsion direction.

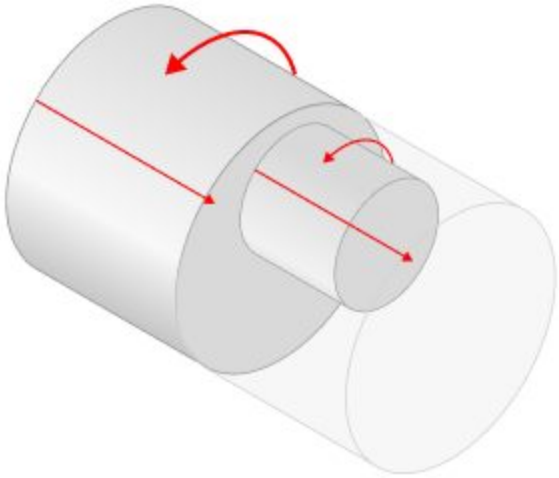
Same as the Basic Step. The toolpath is a single helix line.

mm, %Diameter, Angle. You can set it as an arbitrary value either as an absolute amount, as a percentage of the tool diameter, or as the angle of spiral twist in degrees. When the spiral step is not equivalent to the machining Step, a sequence of spiral paths occurs, spaced by the machining Step.

Angle by curve. Automatically determines the helix angle based on the curve from the Job assignment.

Use flexible axis. When setting the flag, the system automatically determines the axis around which the working strokes are formed in each section as the average of the curvature centers in the section. This leads to a consistent trajectory that maintains continuity and corrects the orientation of the tool axis relative to the part. The option is suitable for asymmetric parts such as crankshafts





Milling Type.

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

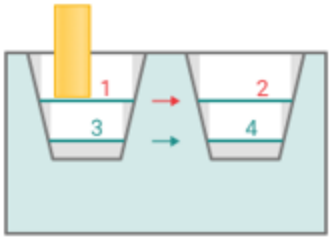
The parameters of this group are the same as in the 5D Surfacing operation. Details.

Sorting.

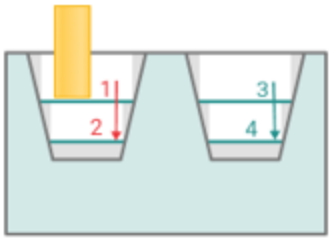
Controls the sequence of toolpath passes during surface machining.

▼ Click here to expand..

By layers. The first layer of all profiles is machined in the beginning. After that the next layer is machined.



By strings. All levels of the first profile are machined first. After that all levels of the next profile are machined.

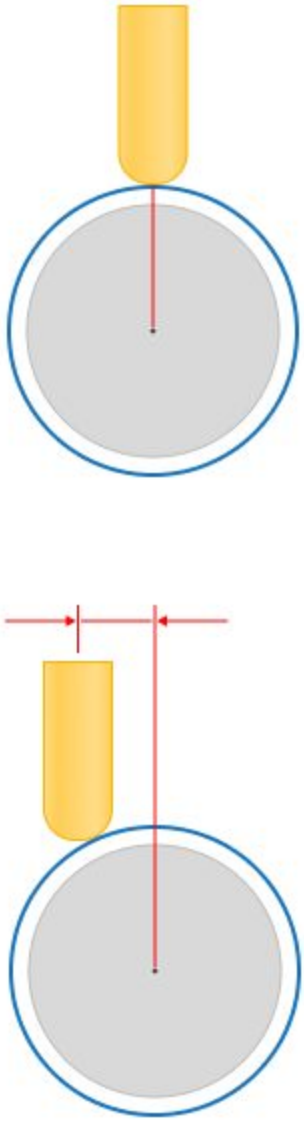


Offset.

▼ Click here to expand..

If enabled: The point towards which the tool is oriented can be offset from the Rotary Axis by the specified value. It can be specified to improve the cutting conditions. The offset can be defined as absolute value or as the percents of the tool diameter. It can be positive or negative.

If disabled: Offset is off — tool orientation is aligned with the Rotary Axis.



Machining levels:

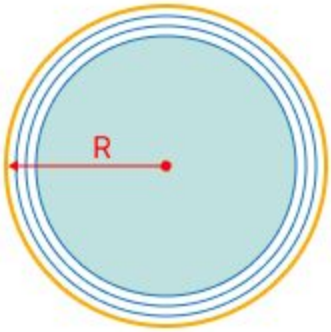
It defines the range along tool axis for the machining.

▼ Click here to expand..

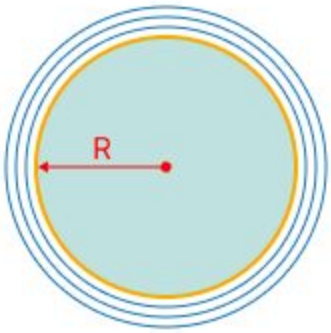
Depth Step. Material removed in one pass along the tool axis between the Top and Bottom levels. If the step is defined as the number of passes, the system will perform as many roughing passes as specified. If the step is defined by distance or as a percentage of the tool diameter, the system calculates the number of passes by dividing the allowance by the roughing Depth Step.

Uniform Step. Activating the flag causes the system to arrange depth passes to provide an even cutting depth within the allowed material removal.

Top level. The radius at which the first pass is made. By default it is the top most level of the machined part.



Bottom level. The radius at which the final pass is made. Drag the cylinder with the mouse and use 'Smart snap' to align with a feature to set the desired bottom level for machining. If the text in the field is green in colour, the level is calculated automatically as the bottom of the part, wokpiece and fixtures. To return to the automatic mode, simply clear the field and press 'Enter'.



R - Bottom radius / level

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

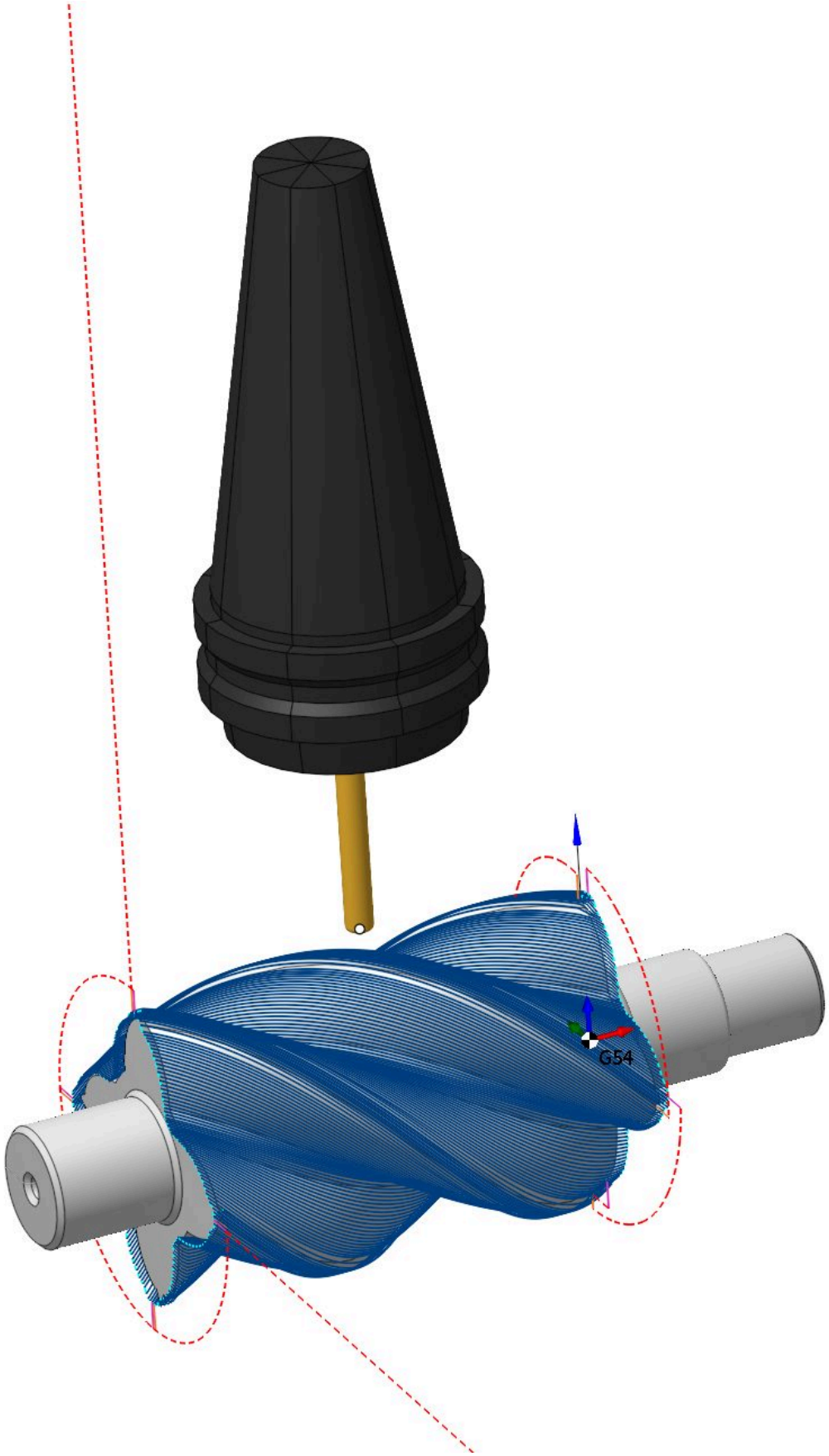
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Rotary finishing



Application area:

The rotary machining operation is available if machine has at least one continuous rotary axis. It is used for the machining of the camshafts, crankshafts, worm shafts, paddles, decorate parts and so on. The main peculiarity of the operation is that it uses the 4th rotary axis together with the linear axes. The 5th axis (when it exists) is fixed. Sometimes the 5th axis can be used also. The Rotary machining operation gives the possibility to machine some surfaces of the part or the part as a whole. In the first case the required surfaces must be specified in the job assignment. In the second case the job assignment must be empty.

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

Spiral curve. The curve needed to indicate the angle of the spiral for the strategy Side angle.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

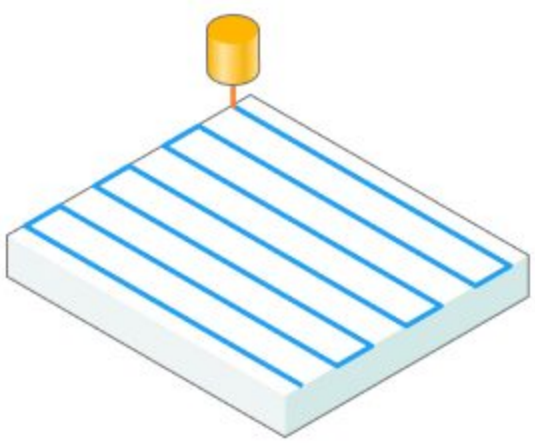
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy.

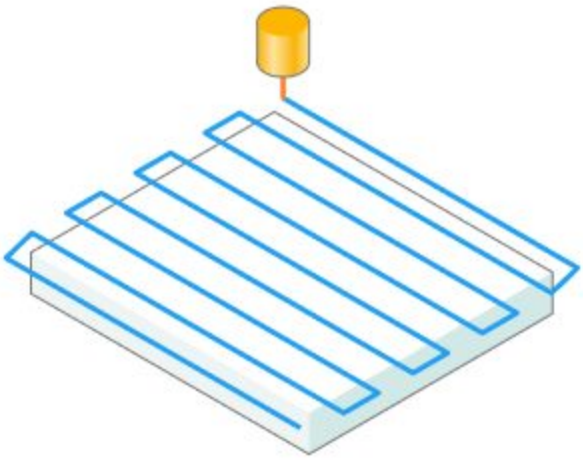
Job zone.

It enables additional modification of the working area initially selected on the Job assignment tab.

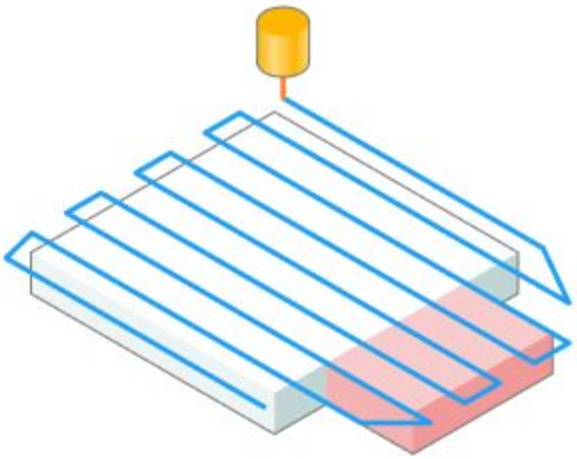
▼ Click here to expand..
Surface only. Machining occurs only on the selected surfaces.



With edges. The resulting toolpath features areas of machining selected surfaces as well as tool rolling sections on edges of selected surfaces.

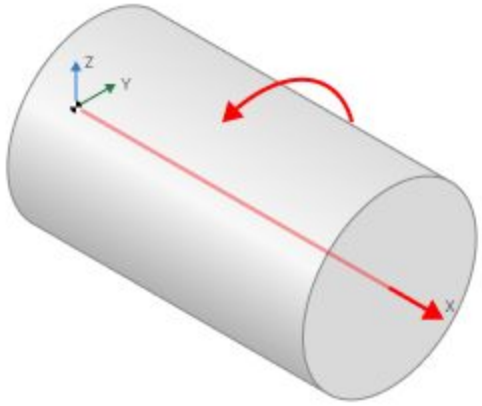


With restriction model. The resulting toolpath features areas of machining selected surfaces, tool rolling sections on edges of selected surfaces, and areas with surfaces that form constraints.



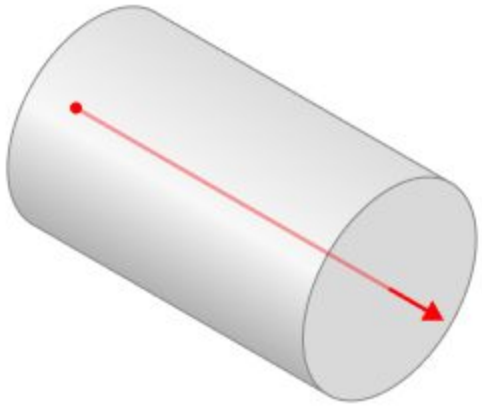
Rotary axis. The system directs tool's axis vector from the designated Rotary Axis. The rotary axis position can be defined in the properties inspector. You can adjust the rotary axis interactively using the arrow or select it from the drop-down menu.

If the mode is set to the Along X, Along Y or Along Z, then the axis of rotation will be oriented along the corresponding axis of the coordinate system.



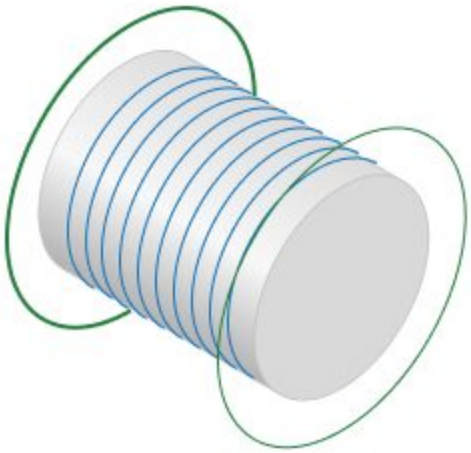
Direction. If mode is Custom(WCS) then axis vector must be defined manually in the fields X,Y,Z of custom direction group. Specify the direction cosines relative to each axis.

Origin. The Origin field defines the point that acts as the coordinate start for the chosen rotary axis.

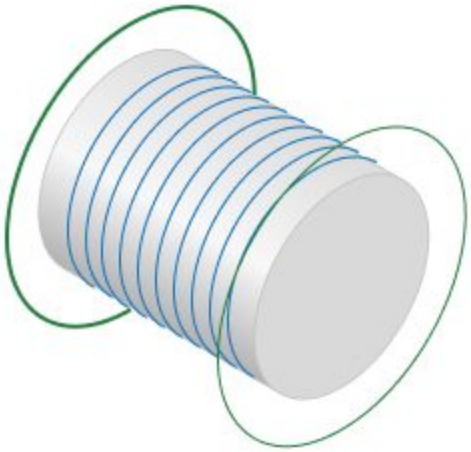


Base CS. The Base Coordinate System (Base CS) changes the coordinate system between the global and local coordinate systems of the part.

Min. axial position. The minimum distance from the Origin along the axis of rotation. By default, it is taken from the workpiece.

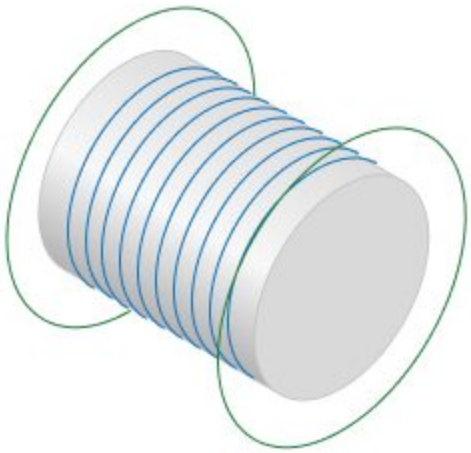


Max. axial position. The maximum distance from the Origin along the axis of rotation. By default, it is taken from the workpiece.

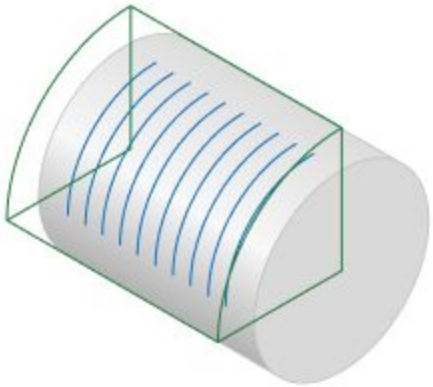


Angular domain. This is an area defined by Start Angle and Finish Angle within which machining occurs. To change the job zone sizes you can mouse drag corresponding side of the visualized sector or specify the exact numerical value in the text box that appears next to dimension line.

Full circle.



0...90.



Start angle. The initial angle of the sector.

Finish angle. The final angle of the sector.

Machining from inside. The parameter allows you to process the internal surfaces of the part (project the trajectory onto the internal surfaces of the part). This parameter group works similarly to the 5D Surfacing operation. In the section Job zone. Details

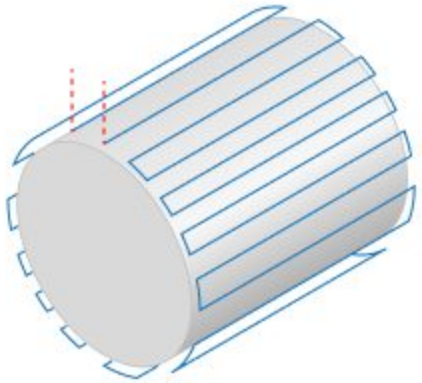
Trajectory form.

The trajectory output format.

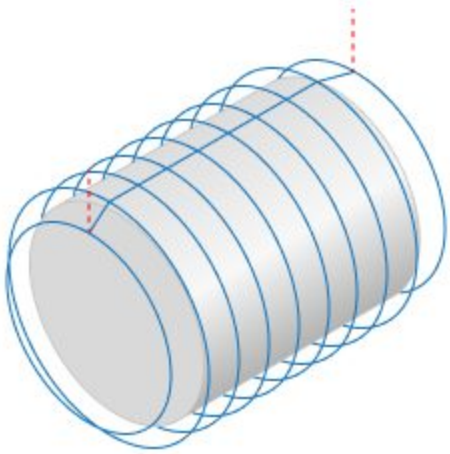
▼ Click here to expand..

The trajectory can be in one of the following forms:

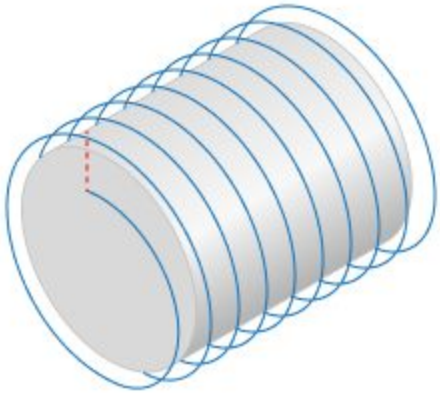
Linear. When you select this option, the work passes form as intersection lines between the machined surfaces and planes through the Rotary Axis..



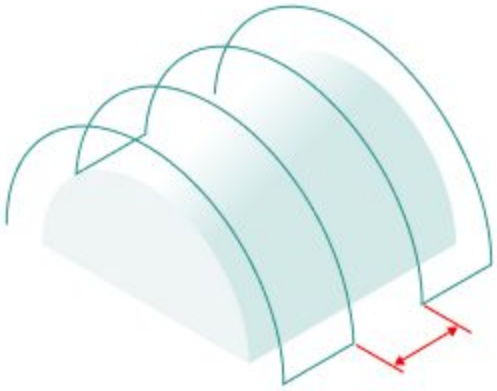
Circular. When you select this option, the work passes form as intersection lines between the machined surfaces and planes perpendicular to the Rotary Axis.



Spiral. Activating this option shapes the toolpath into a continuous helical line.

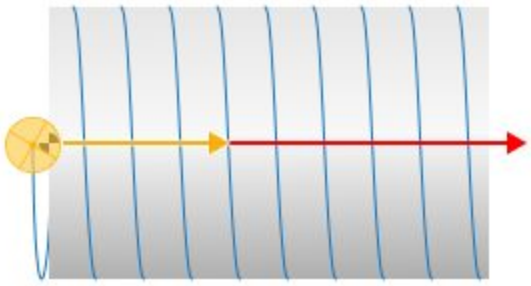


Step. Step along the rotary axis.

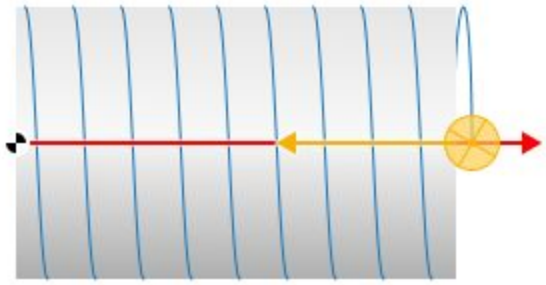


Axial direction. Depending on the feature of the Trajectory Form parameter, it either defines the direction of the tool passes themselves or the sequence of several tool passes. In the case of a spiral whose step matches the machining Step, this parameter influences the direction of the spiral's twist.

Forward. The tool passes is ordered in the Rotary Axis vector direction.



Backward. The tool passes are ordered in the reverse direction of the Rotary Axis vector.



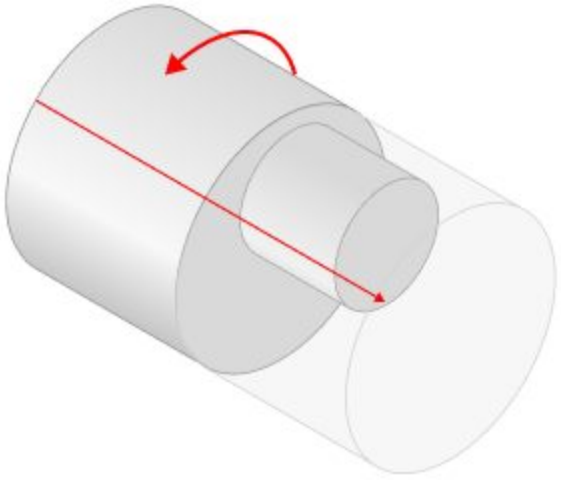
Spiral step. It defines parameters of the Spiral toolpath. The spiral step value can be positive or negative. The sign defines the torsion direction.

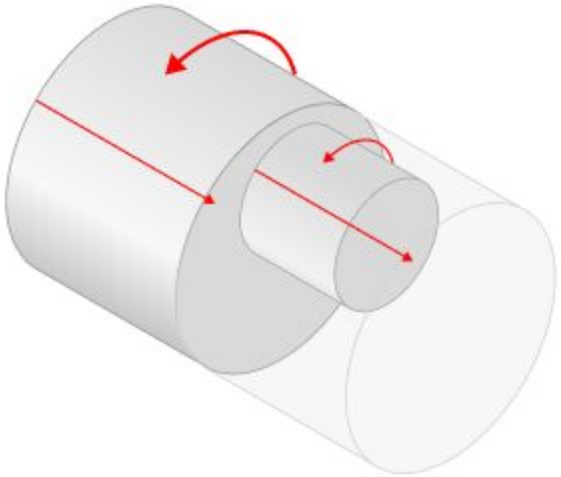
Same as the Basic Step. The toolpath is a single helix line.

mm, %Diameter, Angle. You can set it as an arbitrary value either as an absolute amount, as a percentage of the tool diameter, or as the angle of spiral twist in degrees. When the spiral step is not equivalent to the machining Step, a sequence of spiral paths occurs, spaced by the machining Step.

Angle by curve. Automatically determines the helix angle based on the curve from the Job assignment.

Use flexible axis. When setting the flag, the system automatically determines the axis around which the working strokes are formed in each section a s the average of the curvature centers in the section. This leads to a consistent trajectory that maintains continuity and corrects the orientation of the tool axis relative to the part. The option is suitable for asymmetric parts such as crankshafts



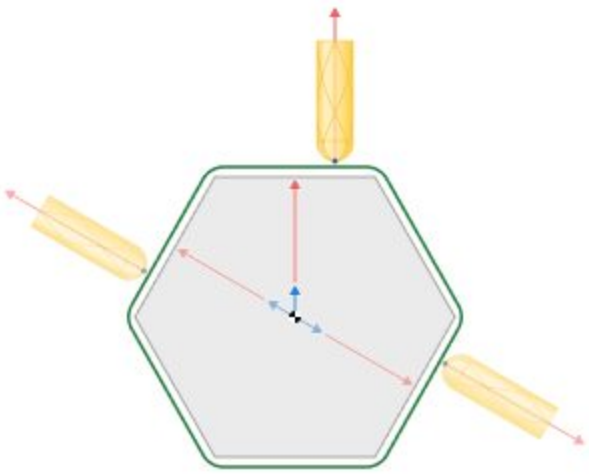


Tool orientation.

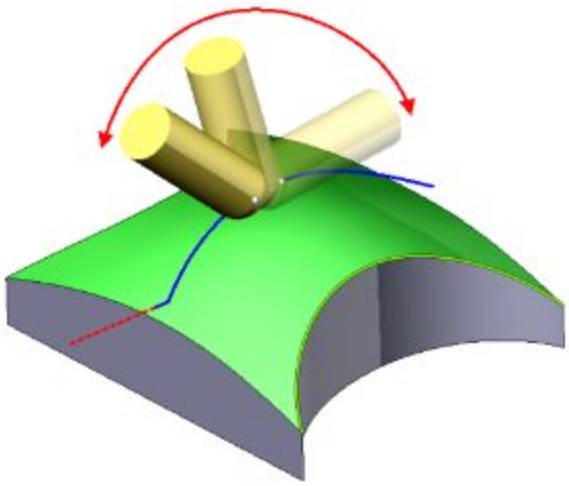
Controls tool axis orientation.

▼ Click here to expand..

By Normal. The tool axis is parallel to the surface normal in every tool path point in the plane perpendicular to the Rotary Axis.



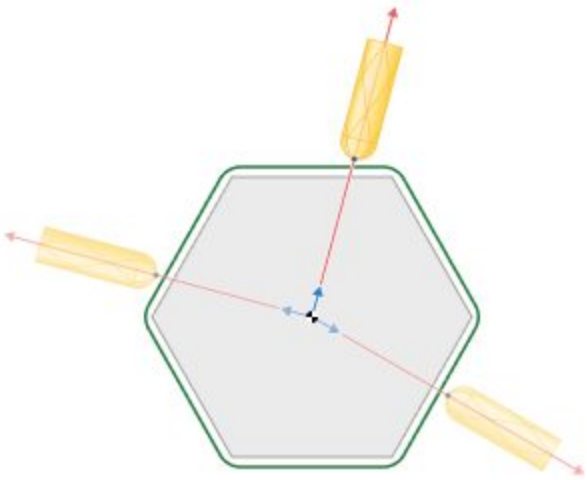
Lead angle. It is a n additional tool tilt angle within the plane of motion direction. It allows to improve the cutting conditions in the contact point. It can be positive or negative.



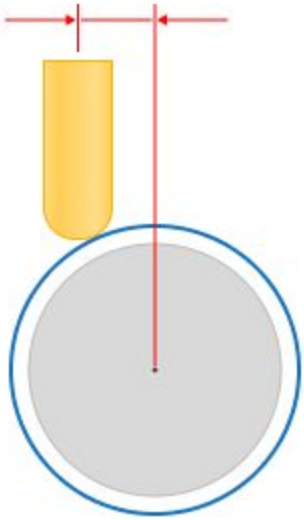
Side angle. Specifies the inclination angle value of the tool axis relative to the axis designated as the Rotary Axis in the plane perpendicular to the motion direction.

Normal blend distance. Parameter for controlling normal vector smoothing. Available when Use new calculation method is enabled. Details

To axis. The tool axis is intersect the rotary axis in every tool path point. So the tool axis does not depends on the surface normal.



Offset. The point towards which the tool is oriented can be offset from the Rotary Axis by the specified value. It can be specified to improve the cutting conditions. The offset can be defined as absolute value or as the percents of the tool diameter. It can be positive or negative.



Side angle. Specifies the inclination angle value of the tool axis relative to the axis designated as the Rotary Axis in the plane perpendicular to the motion direction.

Milling type.

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

▼ Click here to expand..

Both. Tool rotation around the Rotary Axis reverses with each pass, minimizing auxiliary travel lengths and reducing overall machining time.

Follow. Tool rotation direction around the Rotary Axis follows the tool's own rotational direction, adjusted to make the tool and workpiece movements compatible for climb milling.

Counter. Tool rotation direction around the Rotary Axis depends on the tool's own rotational direction, adjusted to make the tool and workpiece movements compatible for conventional milling.

CW. When viewed from the positive direction of the axis, the tool rotates around the Rotary Axis in a clockwise direction, independent of the tool's own rotation.

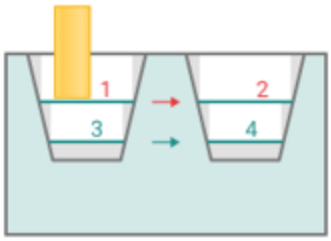
CCW. When viewed from the positive direction of the axis, the tool rotates around the Rotary Axis in a counterclockwise direction, independent of the tool's own rotation.

Sorting.

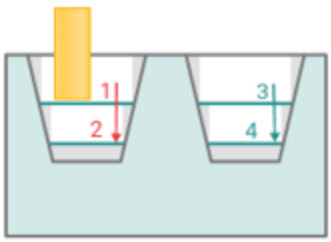
Controls the sequence of toolpath passes during surface machining.

▼ Click here to expand..

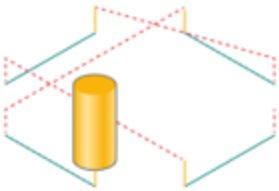
By layers. The first layer of all profiles is machined in the beginning. After that the next layer is machined.

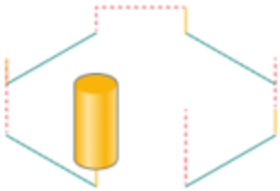


By strings. All levels of the first profile are machined first. After that all levels of the next profile are machined.



Idling minimization. If the flag is set, the system chooses each subsequent pass to minimize the links distance. When the flag is disabled, it processes all passes in a strict order determined by the machining directions and trajectory type.



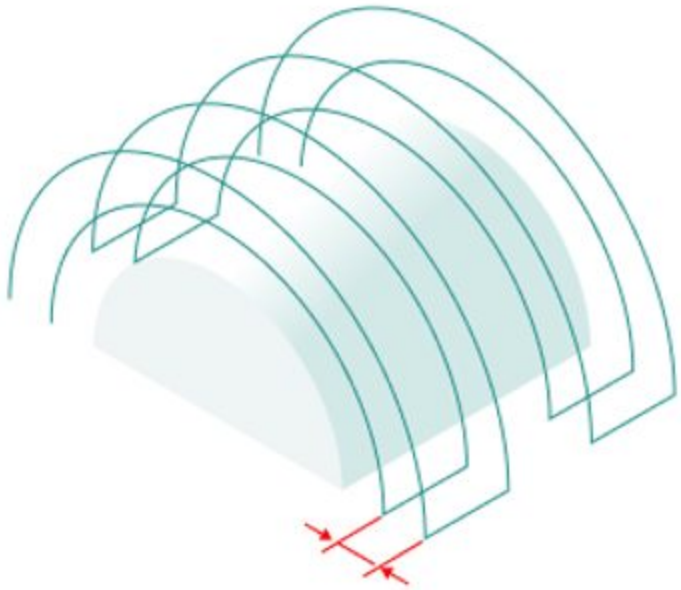


Passes.

Defines the settings for rough and finish machining passes.

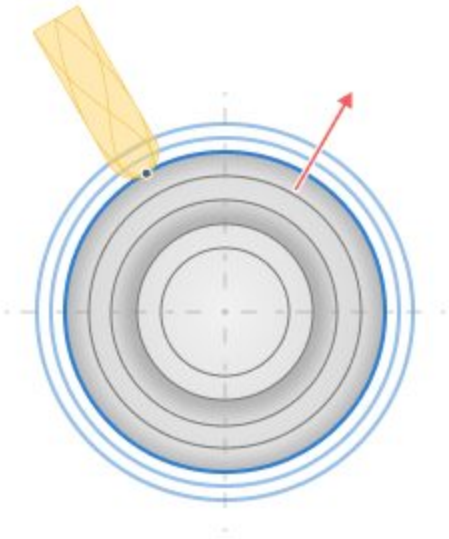
▼ Click here to expand..
Rough passes. Setting the flag triggers the addition of roughing passes.You should specify the overall allowance for these passes in the text field.

Step. The step between rough passes. If the step is defined as the number of passes, the system will perform as many additional passes as specified. If the step is defined by distance or as a percentage of the tool diameter, the system calculates the number of passes by dividing the allowance by the roughing Step.

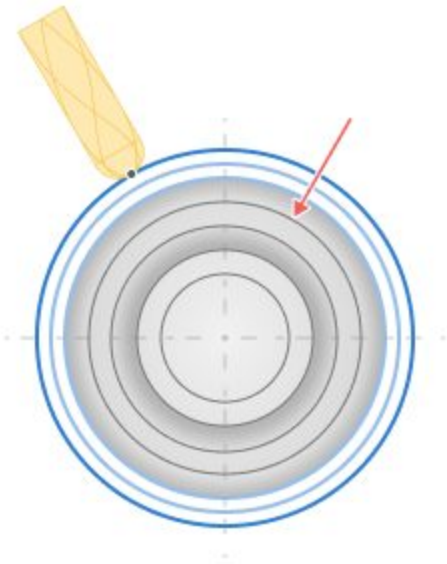


Passes order. For additive processing allows you to control the order of passages.

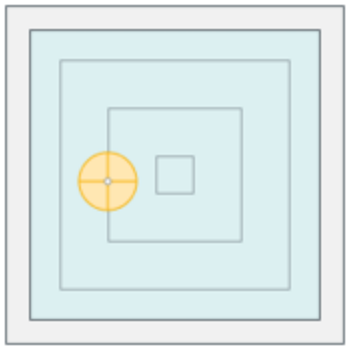
Insude to outside. The tool transitions from the central toolpath curves to the outer ones.

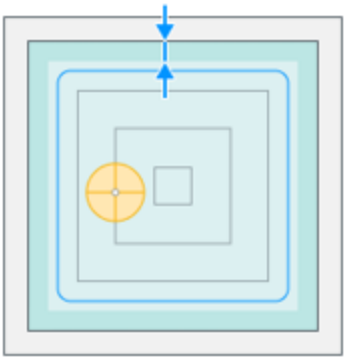


Outside to insude. The tool transitions from the outer toolpath curves to the central ones.



Finish pass. When the flag is set, the contour defined in the Job Assignment becomes the tool's finishing pass. The system adds an additional roughing pass at the entered distance. If Rough passes are in use, the final rough pass will take place at this entered distance.





Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

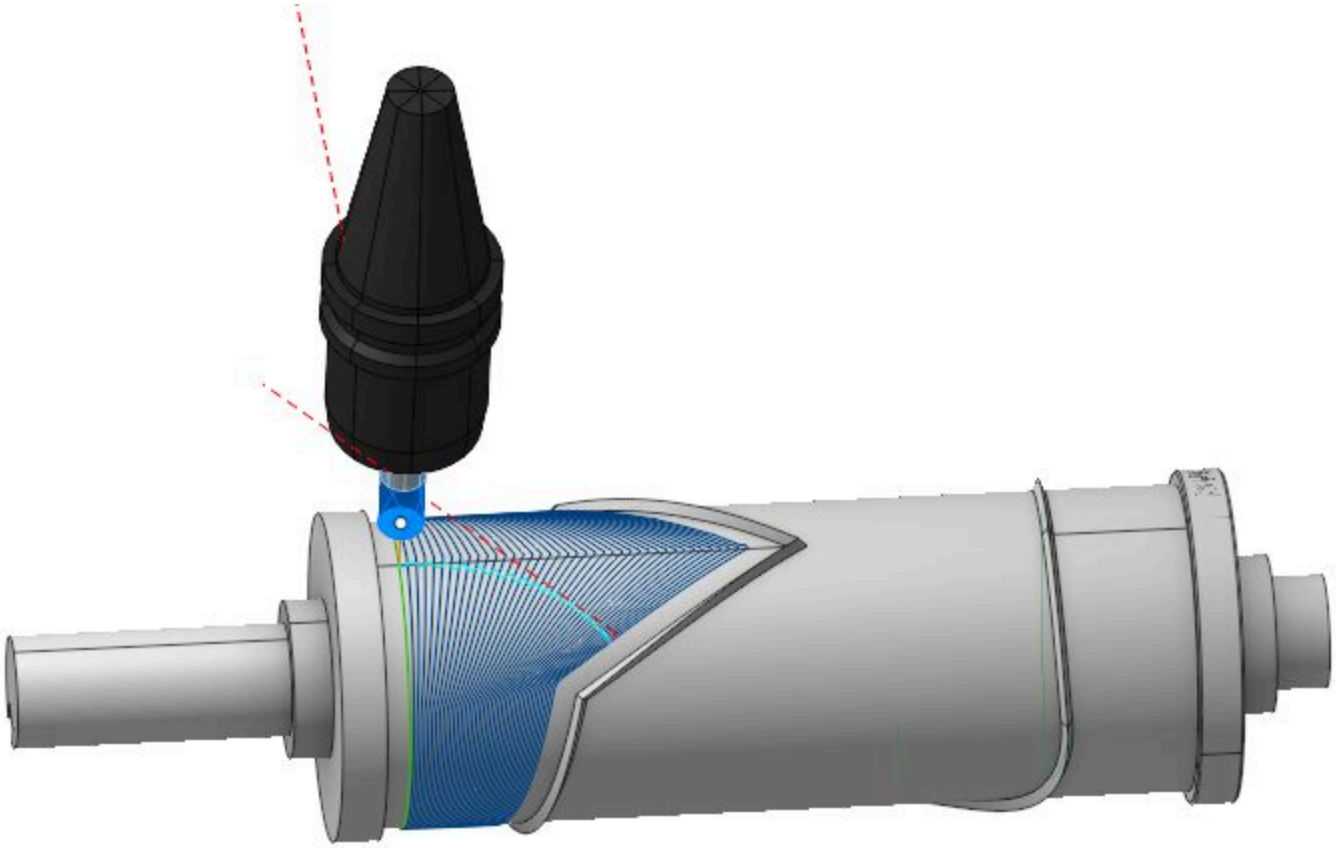
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Morph 4D



Application Area:

This finishing operation is for 4x and 5x configurations and adapts Morph operation to rotary machining of complex surfaces with a distinct axis of rotation or the surfaces extended along the axis. Tool orientation set as default To Rotary axis, also the Fixed strategy is available, other tool orientation strategies are blocked. In the To Rotary axis strategy Lead Angle is available only. Safe surface set as Cylinder. For holder collision available Trim toolpath method only.

Job Asssignment:

Machining Surfaces. The system uses the Machining Surfaces to restrict the area of machining. Morph generates passes only where a tool has contact with those surfaces. Morph generates passes not between two curves you set in the job assignment, but between two Machining Surfaces contact curves which are closest to those two curves you specify as the First and the Second curve.

First Curve. It is t he starting line from among the two lines between which the system generates the toolpaths.

Second Curve. It is the finishing line from among the two lines between which the system generates the toolpaths.

Sync lines are used to improve the quality of morphing in difficult cases, especially when machining closed contours. A Sync Line specifies two corresponding points of the First and the Second Curve. You can use any types of curves as Sync Lines: edges, 3D curves, 2D geometry curves. They do not have to start and end precisely on the First and Second Curve. You can draw sync lines at the 2D geometry tab.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Strategy:

These parameters allows the user to achieve a required toolpath. The parameters of this group are the same as in the Morph operation. Details.

Milling Type:

This feature enables users to select the necessary milling direction (climb or conventional) during the toolpath calculation process. The parameters of thus group are the same. as in the Waterline Roughing operation. Details.

Sorting:

Controls the organization of the toolpath during surface machining

Start Point. Activating the flag permits the input of coordinates marking the starting point of working passes in surface machining.

Tool Axis Orientation:

Controls the tool axis orientation during the machining process. The strategies To Rotary axis (where one can use only the Side Angle) and Fixed are available in this operation. These parameters are the same as in the 5D Surfacing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

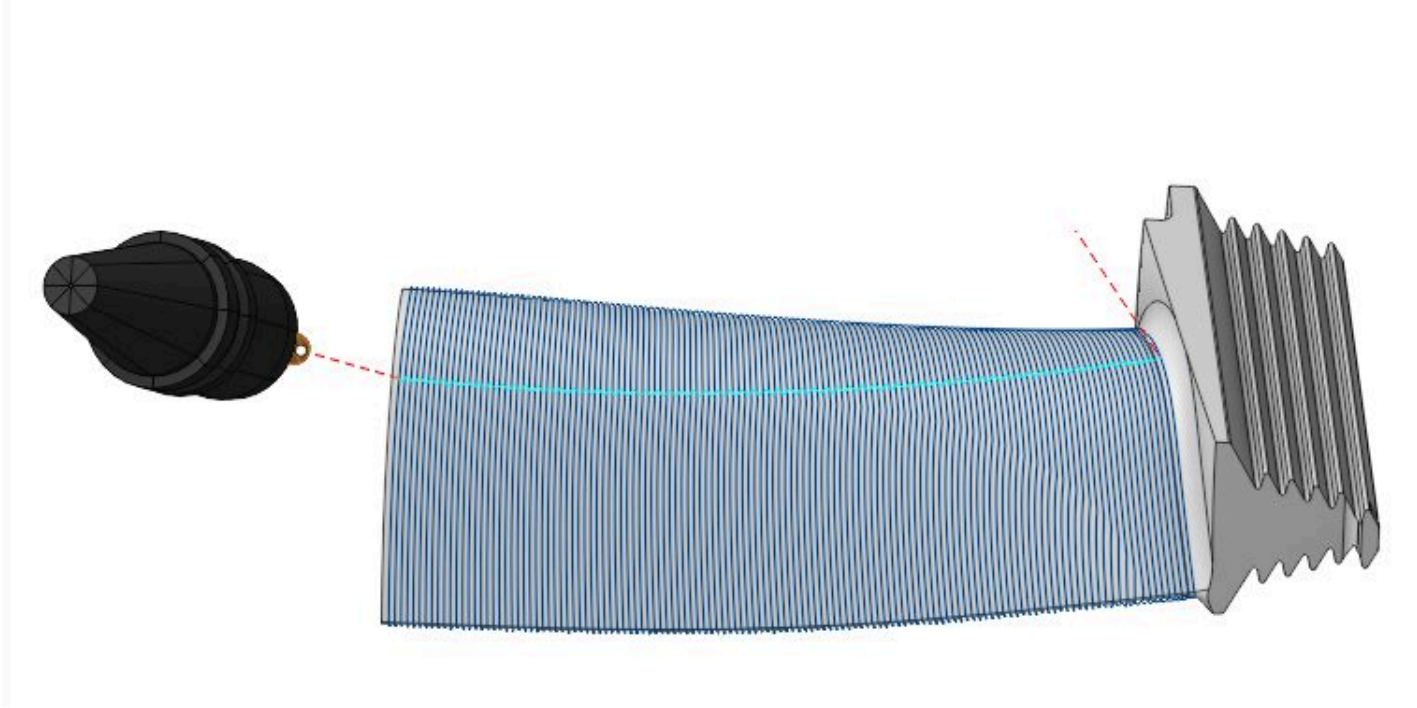
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

4D Surfacing



Application Area:

This finishing operation is for 4x and 5x configurations and adapts 5D Surfacing operations to rotary machining of complex surfaces with a distinct axis of rotation or the surfaces extended along the axis. It does not support the Through Point, Through Curve, and Perpendicular to Toolpath tool orientation strategies. In this operation incline angles, avoid singularity and axis deviation features are blocked. Safe surface set as Cylinder.

Job Assignment:

Machining Surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

First Curve. For the Parallel to Curve strategy, this feature specifies curves for parallel passes computations. Additionally, in the Morphing Between Two Curves strategy, it specifies the First Curve . You can select one or multiple, not necessarily connected, curves or edges.

Second Curve. The system uses it in he Morphing Between Two Curves strategy. It specifies the Second Curve. You can select one or multiple, not necessarily connected, curves or edges.

Tilt Curve. This is the curve that the tool axis vector aims at when setting Tool Orientation specification Through Curve.

First Surface. This feature serves as the First Surface in a Morph Between Two Surfaces strategy, facilitating the morphing process to create toolpath lines.

Second Surface. This feature serves as the Second Surface in a Morph Between Two Surfaces strategy, facilitating the morphing process to create toolpath lines.

Surfaces for projection. These are the surfaces intended for machining onto which the toolpath transfers when enabling the Project Toolpath onto the Part option in the Strategy tab.

Job Zone. Use the job zone to trim the passes outside the specified 2d containment areas. The plane of the containment area is defined by the initial tool orientation. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that s hould avoid machining in the current operation. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Strategy:

This parameter allows the user to achieve a required toolpath. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Milling Type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Sorting:

Controls the sequence of toolpath passes during surface machining. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

The parameters of this group are the same as in the 5D Surfacing operation. Details.

Tool Orientation:

Controls tool axis orientation. This operation features tool orientation strategies such as Normal to Surface, Fixed, Flank, and To Rotary Axis only. It locks options like Lean Angle and Vertical Clearance Angle. Other parameters of these strategies are the same as in the 5D Surfacing operation. Details.

Limit Rotation Angles:

When the flag is set, it specifies the limit values of the tool's angular position relative to the specified axis. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Job zone:

Enables additional modification of the working area initially selected on the Job assignment tab. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Corners smoothing:

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Roughing Passes:

Toolpath is offset in plane by the Roughing passes thickness amount. Additional passes are added to the contour by the value specified in the parameters with the specified number of steps. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Project Toolpath onto the Part:

By setting the flag, the system generates tool paths for processing the surfaces identified as Surfaces for Projection in the Job Assignment by mapping the path derived from the Machining Surface onto the Surfaces for Projection. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

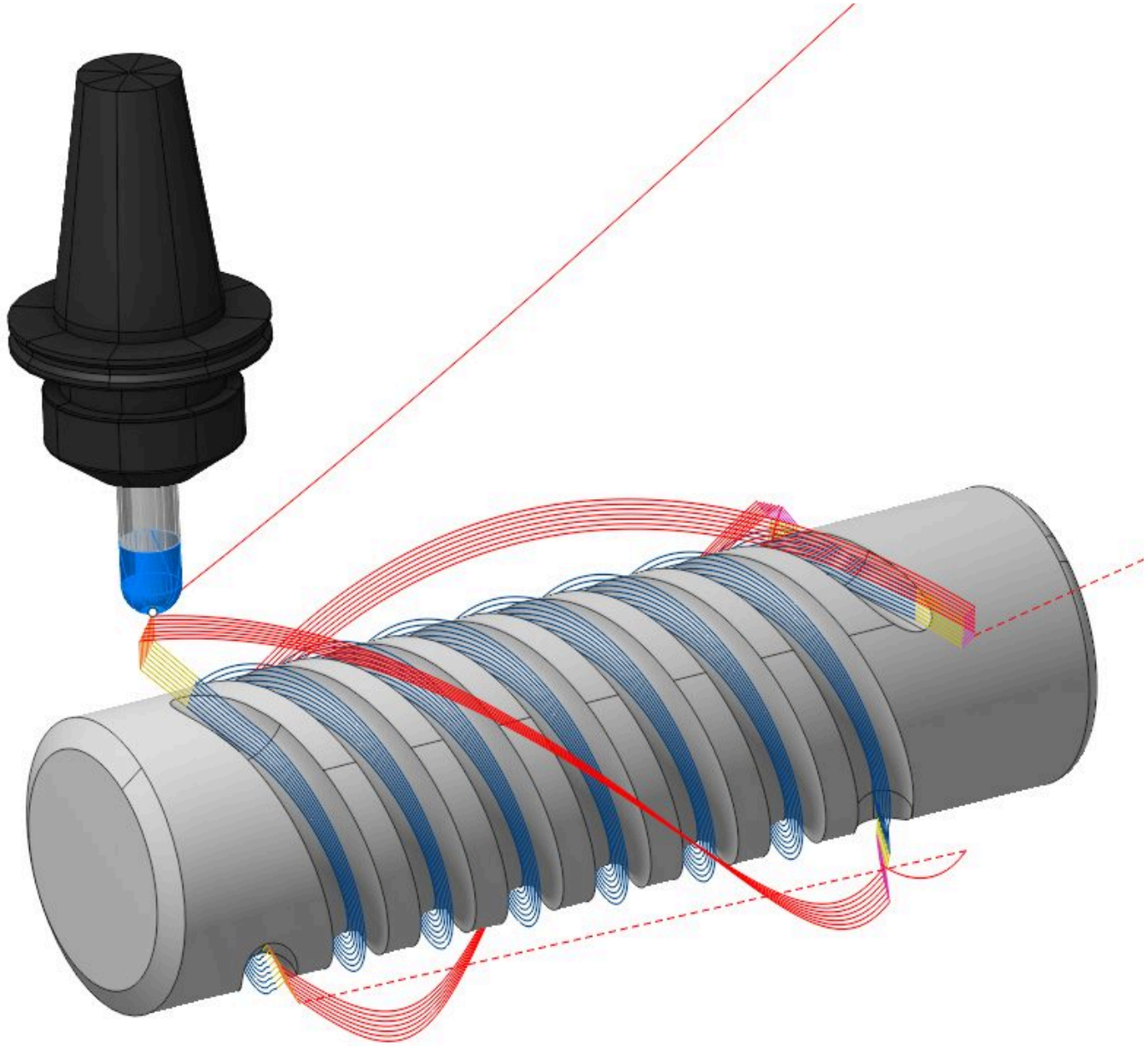
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

4D Contouring



Application Area:

The operation generates continuous toolpathes for processing spatial curves on surfaces with a clear axis of rotation or elongation along an axis. It enables the creation of both single toolpathes along selected curves and multiple passes along various isoparametric lines of the chosen surface, as well as facilitates multi-level machining along selected curves. Operation based on the 6D Contouring operations operation and perconfigurated for rotary machining and available for 4x and 5x configurations. Tool orientation set as default To Rotary axis. The tool orientation strategies Through point and Through curve, as well as the options Advanced axes limits control, Lean angles, Side angles are blocked. Safe surface set as Cylinder.

For the curves specified in the Job Assignment, user can modify the tool orientation vectors at each point (but only in the plane perpendicular to the Rotary Axis), change the tool's movement direction along the curve, alter the machining side, compensate for the tool radius, select the interpolation method for changing the tool orientation vector from point to point, and set the stock.

Job assignment:

Edge/Curve on Surface. Job assignment is formed with the edges of the part or any curves. The tool's current orientation determines vectors at the curve points.. The tool contact point will move along the specified curve based on the curve properties and the chosen tool orientation strategy. The parameters of this group are the same as in the feature 5-axis milling of the profile on the surface / Parameters of Curves. Details.

5D Curve. The system generates the 5-axis tool path as the projection of the arbitrary curves on the part surface. In this case the tool contact point will move in touch with the surface. The passes can be shifted from the curve by the stock or by the levels. The tool axis is located perpendicular to the surface. The parameters of this group are the same as in the feature 5-axis milling of the profile on the surface / Parameters of Curves. Details.

Drive Faces. One can use them for creating work passes along isoparametric lines. The isoparametric curves of the surfaces depends on its way of creation and reflect the topology of the surface. Different fillets can be machined this way very well. The parameters of this group are the same as in the feature 5-axis milling along the isoparametric curves / Parameters of Drive Faces. Details.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

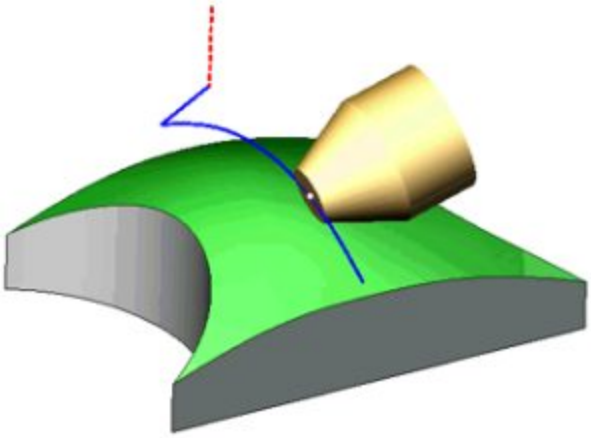
The parameters of this group are the same as in the 5D Surfacing operation. Details.

Tool Orientation:

Controls tool axis orientation. The principal parameters of this function are the same as in the 5D Surfacing operation. Details.

▼ Click here to expand..

Incline Shape Tool. Tilts the tool so its working side touches the machining surface during toolpath traversal.



Limit Rotation Angles:

This option appears when the Flank tool orientation strategy is active and specifies the limit values of the tool's angular position relative to the axes. The parameters of this group are the same as in the 5D Surfacing operation. Details.

Passes:

You can create additional work passes relative to the specified curves. The passes shifts in the direction normal to the surface. If the Flank tool orientation is selected, you can choose the direction of the shifts. The parameters of this group are the same as in the 6 D Contouring operation. Details.

Corners smoothing:

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. The principal parameters of this function are the same as in the 2D Contouring operation. Details.

Continous axial shifting:

With the Flank tool orientation strategy is in use, you can create a toolpath with smooth changes in the tool's axial displacement along the path relative to the specified curve. The parameters of this group are the same as in the 6 D Contouring operation. Details.

Sorting:

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Toolpath uniting mode. Controls the order of pass execution in the multi-pass machining of different profiles on the part. Parameters of this feature are the same as in the 2D Contouring operation. Details.

Idling minimization. The order of the profiles machining is sorted to minimize the idle motions. Parameters of this feature are the same as in the 2D Contouring operation. Details. .

Milling type. Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. The parameters of the Milling Type are the same as in the Waterline Roughing operation. Details.

Project toolpath onto the part with offset:

Determines the method of toolpath creation relative to the surface of the part. Parameters of this feature are the same as in the Strategy tab, in the Transformations item of the 2D Contouring operation. Details

Type and Max deviation work as in the 5D contour and 6D contour operations. Details

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Turbomachinery Milling

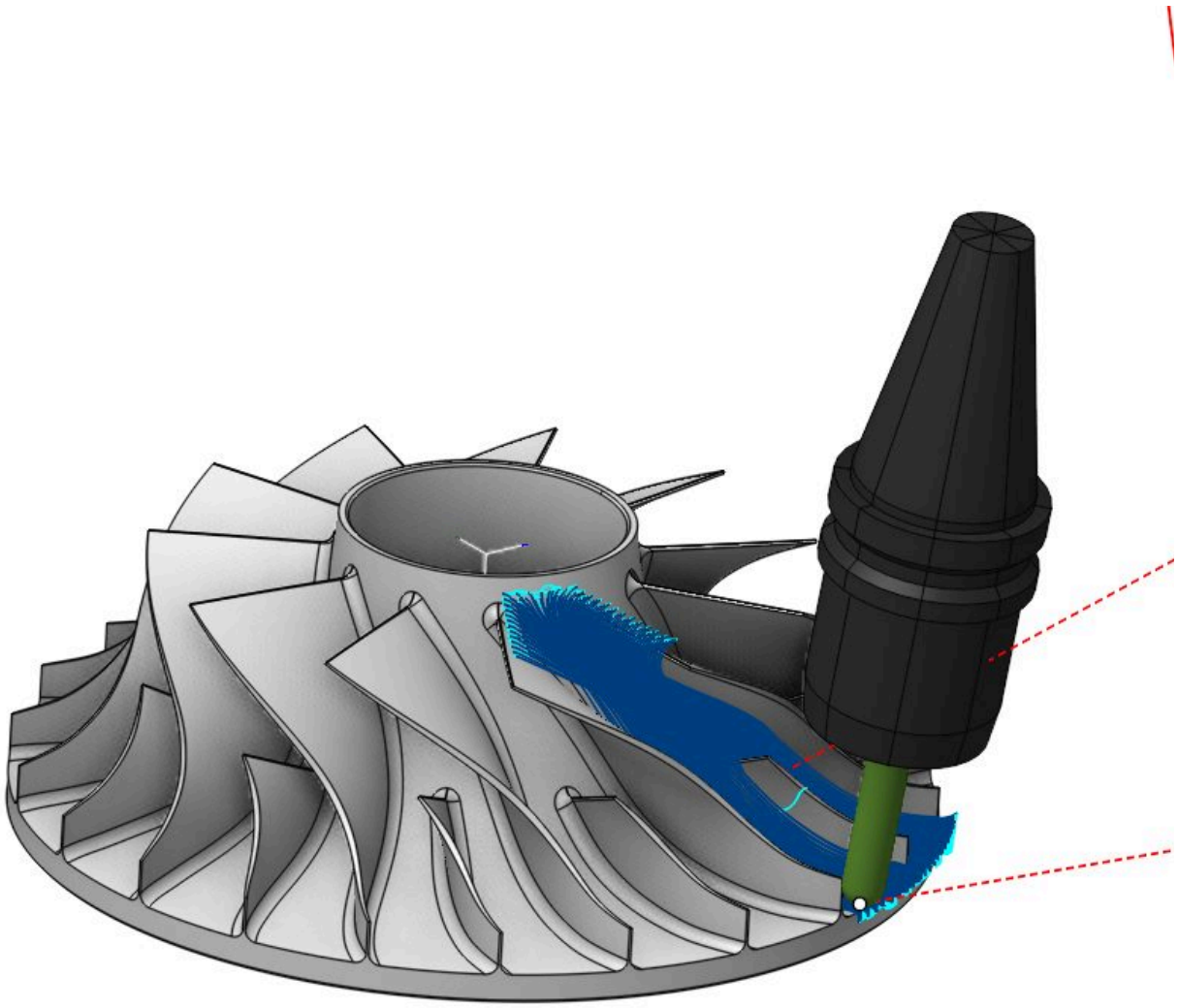
Application area:

The Turbomachinery Milling group contains specialized operations for machining impellers and individual blades. The operation topics describe the required blade and hub geometry, machining strategies, and toolpath settings.

Operations:

- Impeller Machining. Provides roughing and finishing strategies for impeller blades, hubs, and fillets. Details
- Blade Machining. Finishes blade airfoils, root zones, hubs, and edges. Details

Impeller Machining



Application Area:

The operation makes the process efficient for five-axis machining of parts such as monocycles and impellers, integrating strategies for generating the most suitable toolpathes for rough machining of monocycles, finishing of blades and hubs, and refinement of fillets. It also provides additional functionalities to avoid collisions with the tool, holder and the blank.

Job Assignment:

Blade Surfaces. It serves to define blade wall surfaces for machining. If blade geometry is not specified, the system generates a toolpath for machining surfaces enclosed between the Hub and the Shroud surfaces. For rough machining, select the surfaces of two adjacent blades. During finishing, the system machines the selected Blade Surfaces. The geometry selection should include blade working surfaces, upper and lower surfaces, and fillets.

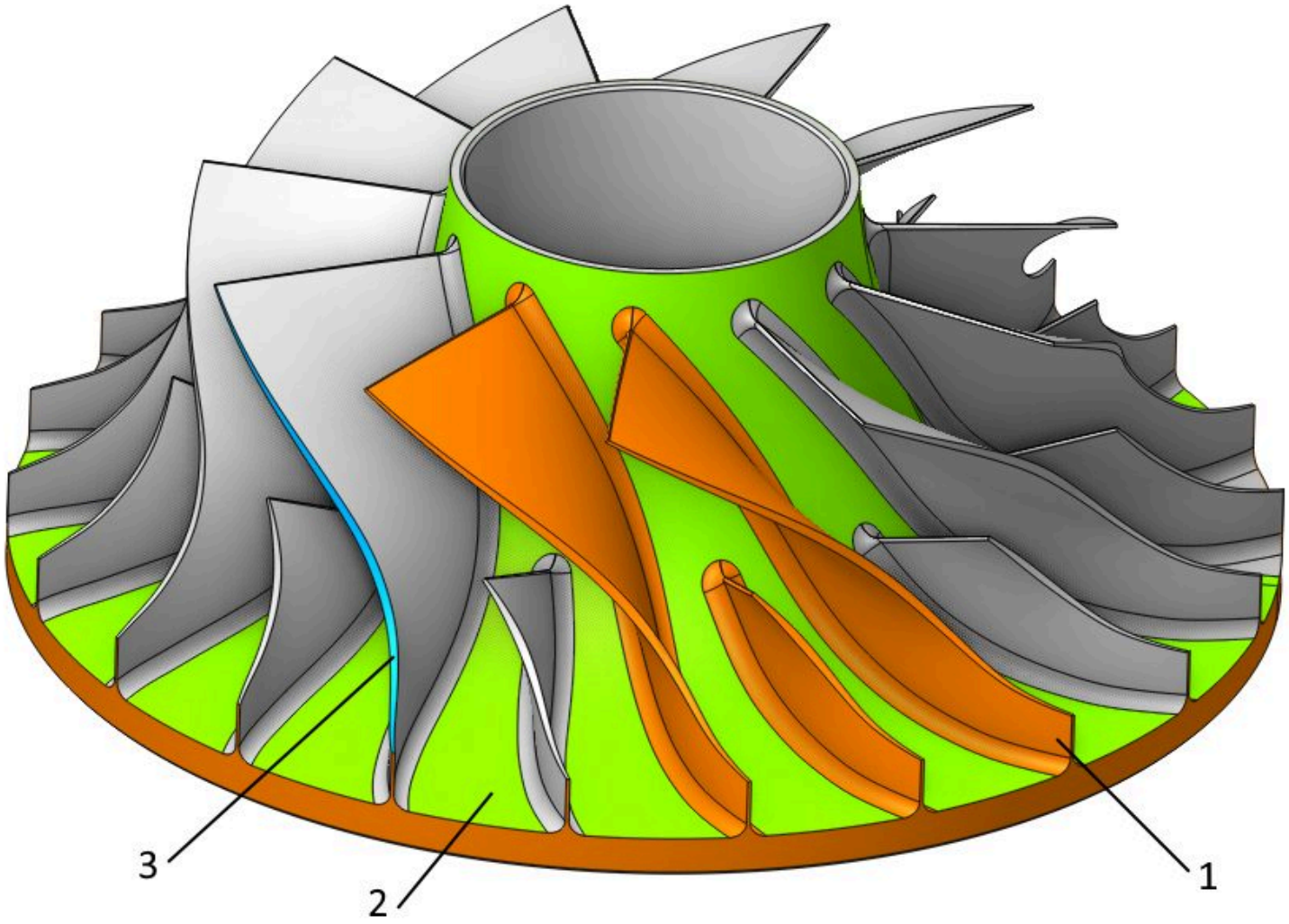
Hub Surfaces. I t identifies the axis of rotation and shapes the space between the hub and the shroud for toolpath formation. The hub must be a surface of rotation. For the finishing process, it is enough to pick any hub surface that has a span from the leading edge to the trailing edge of the blade.

Shroud Surfaces. It shapes the space between the hub and the shroud where the toolpath forms and serves as the base curve for certain strategies. It's essential to select surfaces that limit the lateral outline of the impeller.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details



1-Blade Surfaces, 2-Hub Surfaces, 3-Shroud Surfaces.

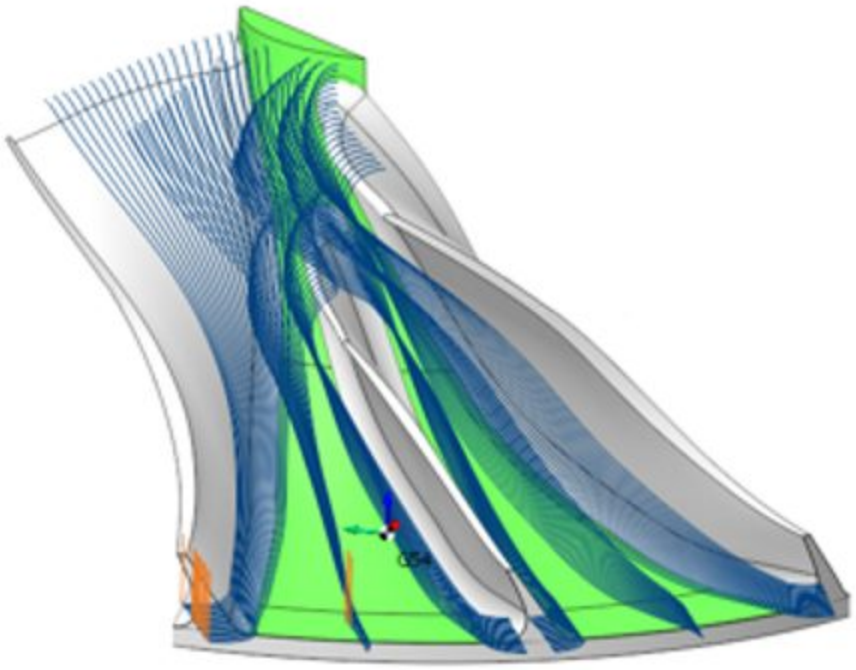
Strategy:

Machining:

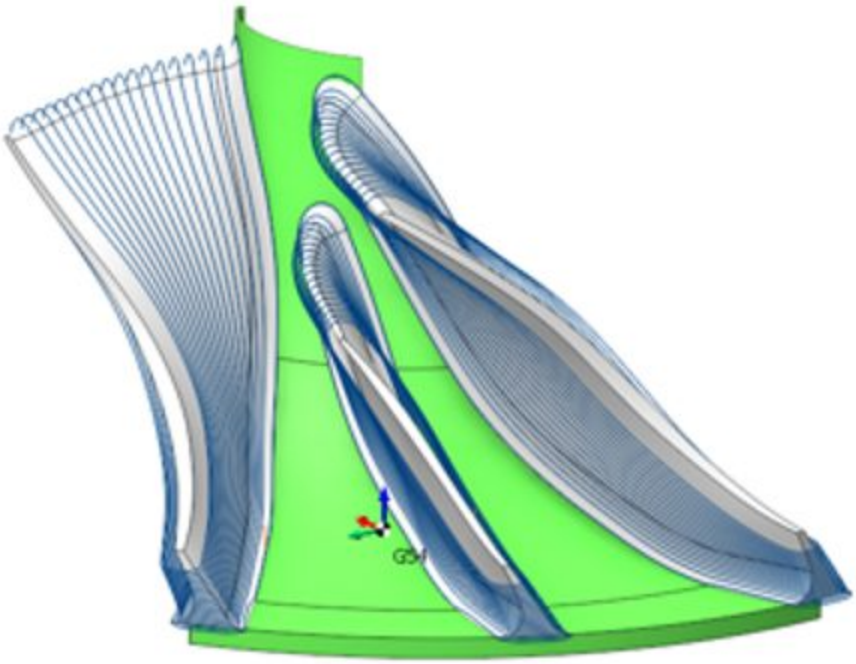
Specify the type of machining and the surface to process.

▼ Click here to expand..

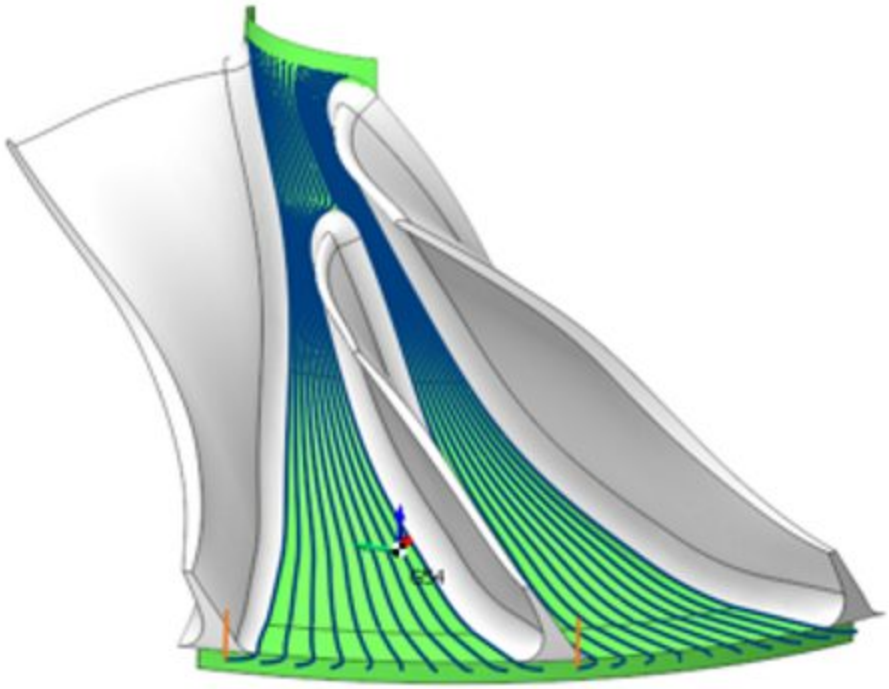
Roughing. Executes rough machining on the impeller blank within the area defined by the Hub Surfaces, Shroud Surfaces, and the two adjacent selected Blade Surfaces.



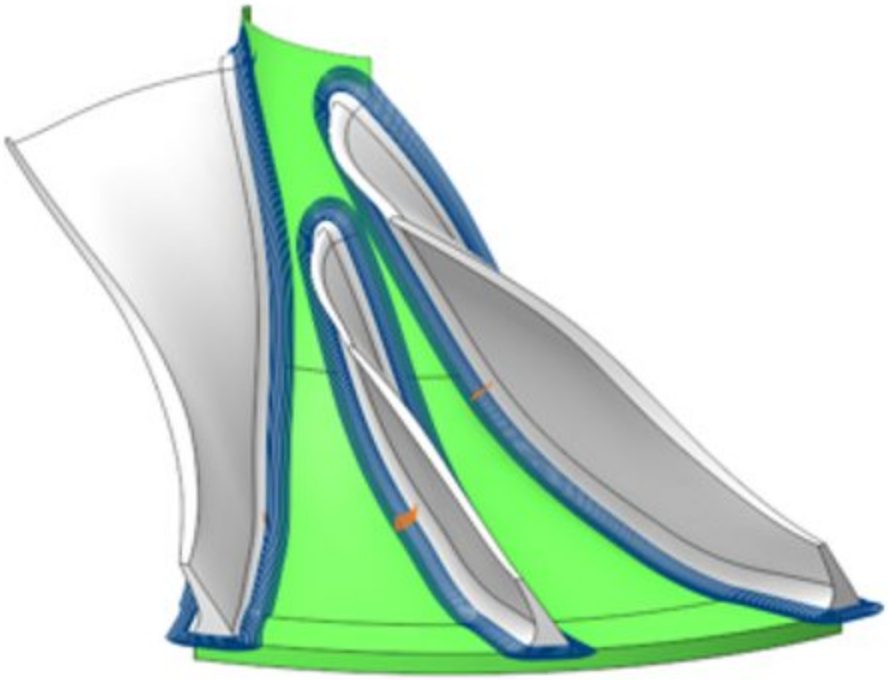
Blade Finishing. Performs finish machining on the selected Blade Surfaces. Set the Hub Surfaces and Shroud Surfaces first. The process will focus only on theselected Blade Surfaces if any are chosen. If none are selected, the machining will cover all the impeller's blades.



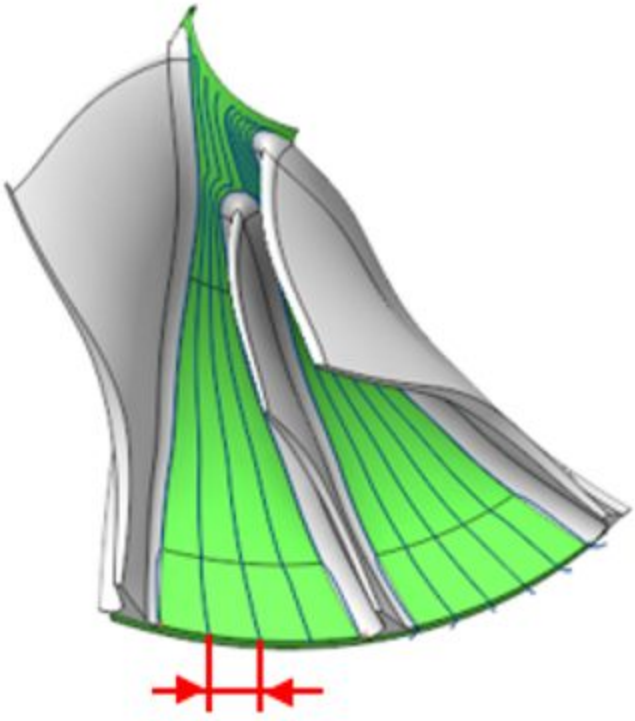
Hub Finishing. Performs finish machining on the selected Hub Surfaces in area between the selected Blade Surfaces.



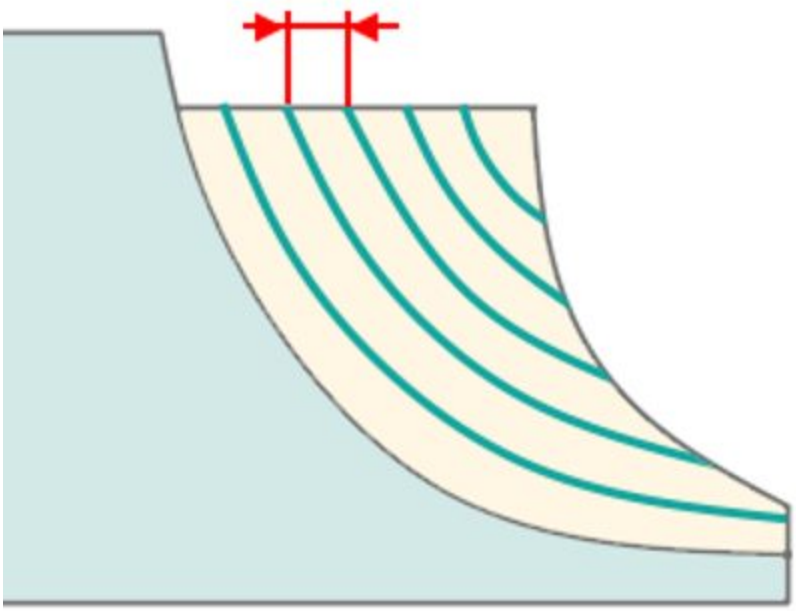
Fillet Finishing. Executes fine machining of the fillet surfaces between the blade and hub. Set the Hub Surfaces and Shroud Surfaces first. The process will focus only on the area of selected Blade Surfaces if any are chosen. If none are selected, the machining will cover all the impeller's fillet surfaces.



Step. The maximum allowed width of cut.



Depth step. The distance between roughing passes.



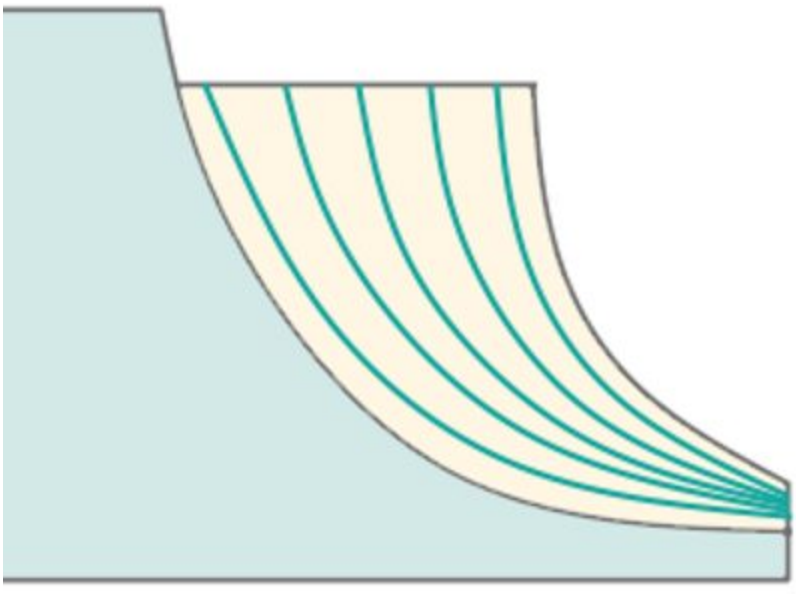
Count. It is the count of passes on the fillet surface when the Fillet Finishing strategy is active. This number of passes extends on each side from the middle of the fillet.

Strategy:

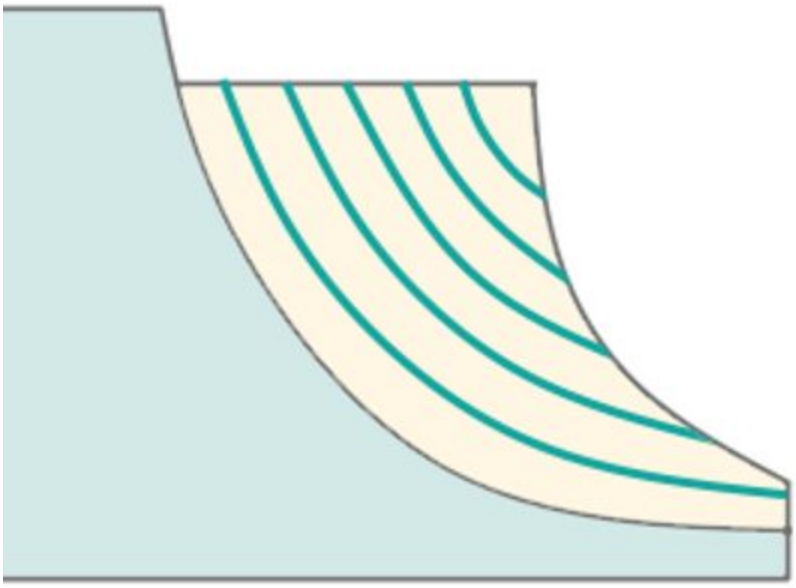
This parameter allows the user to achieve a required toolpath:

▼ Click here to expand..

Morph Between Shroud and Hub. The tool paths are the lines that result from a smooth transition from the generating lines of the shrood to those of the hub. They pass through a specified Step (for finishing) or a Roughing Step (for roughing).



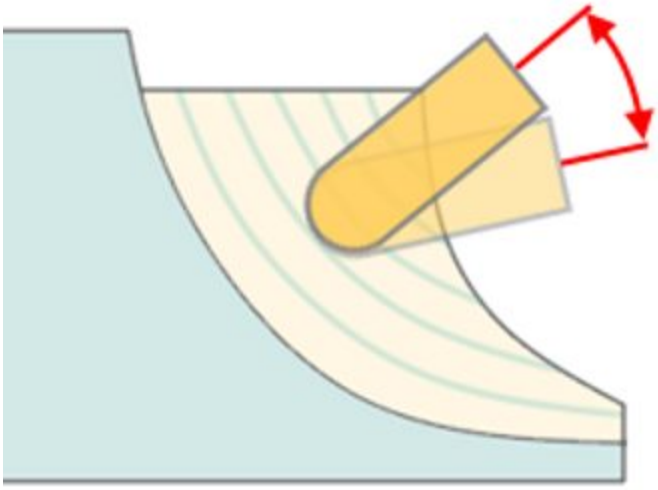
Offset from Hub. The system creates the toolpaths on surfaces that offset the Hub Surfaces by a specified Step for finishing or a Roughing Step for roughing.



Tool Orientation:

Controls tool axis orientation.

▼ Click here to expand..
Lead Angle. It is a n additional tool tilt angle within the plane of motion direction.



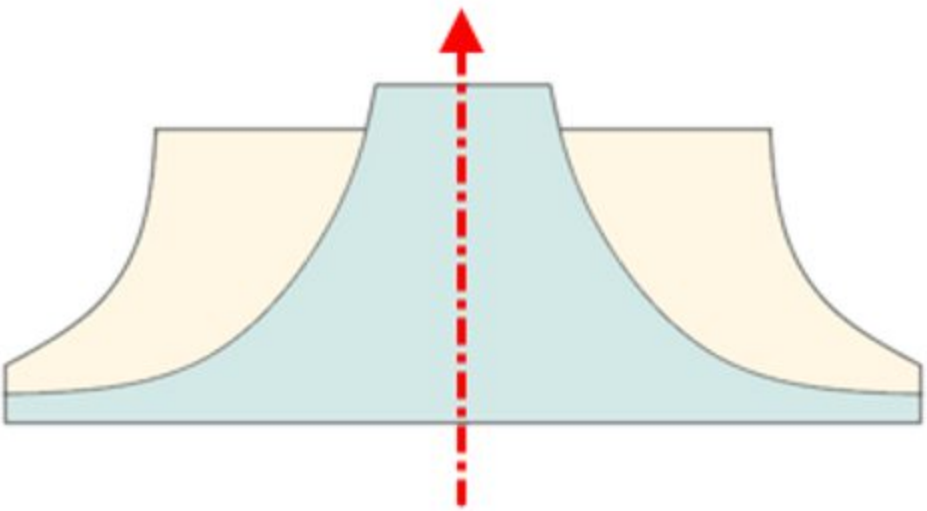
Avoid Collisions:

With the Check Holder function, this feature detects segments of the toolpath where the tool holder collides with the part and modifies those segments according with the specified strategy. Details.

Rotary Axis:

This feature permits the manual designation of the impeller's axis. Select one of the Workpiece CS or Global CS axes, that the impeller's geometric axis should pass through.

▼ Click here to expand..



Sorting:

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..

Start Point. Activating the flag permits the input of coordinates marking the starting point of working passes in surface machining. The parameters of this option are the same as in the 5D Surfacing operation. Details.

Milling Type. Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. The parameters of the Milling Type are the same. as in the Waterline Roughing operation. Details.

Trim rolls.

The cutter 'rolls' around the sharp outer corners of a model. The tool path is smooth but not always optimal. This parameter works similarly to the Waterline roughing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

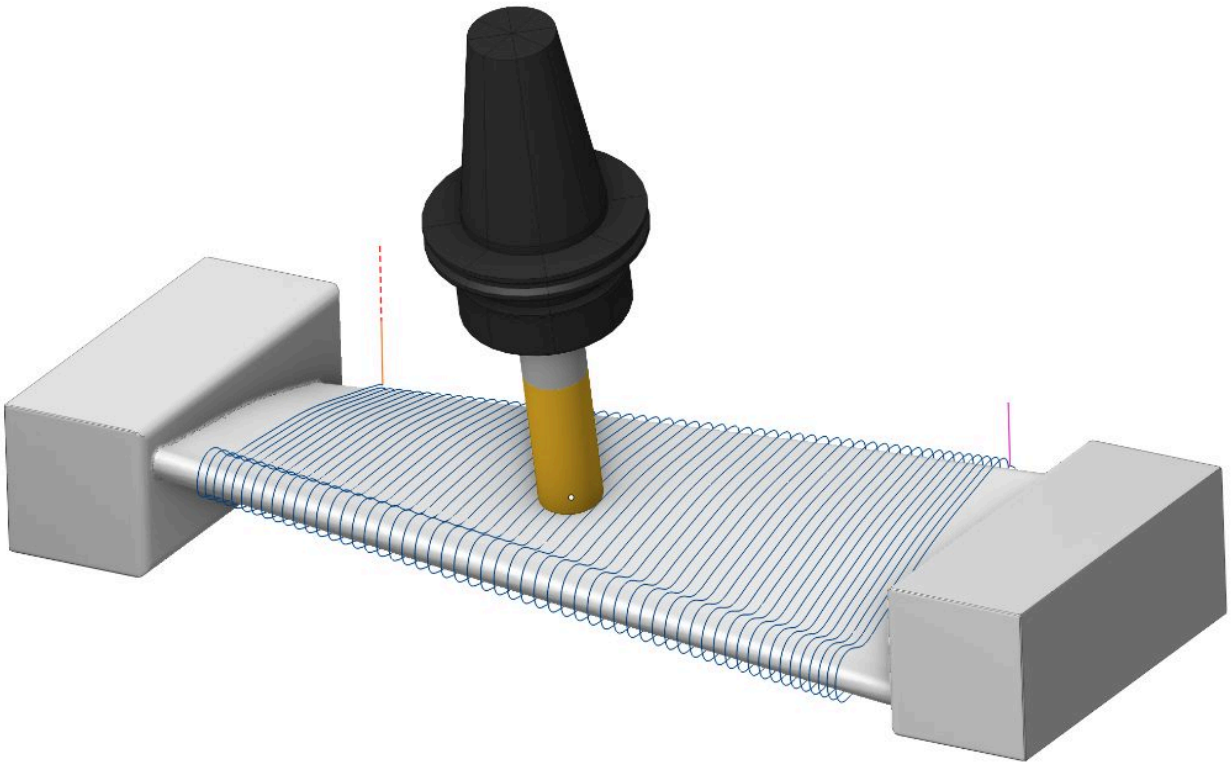
Workpiece:

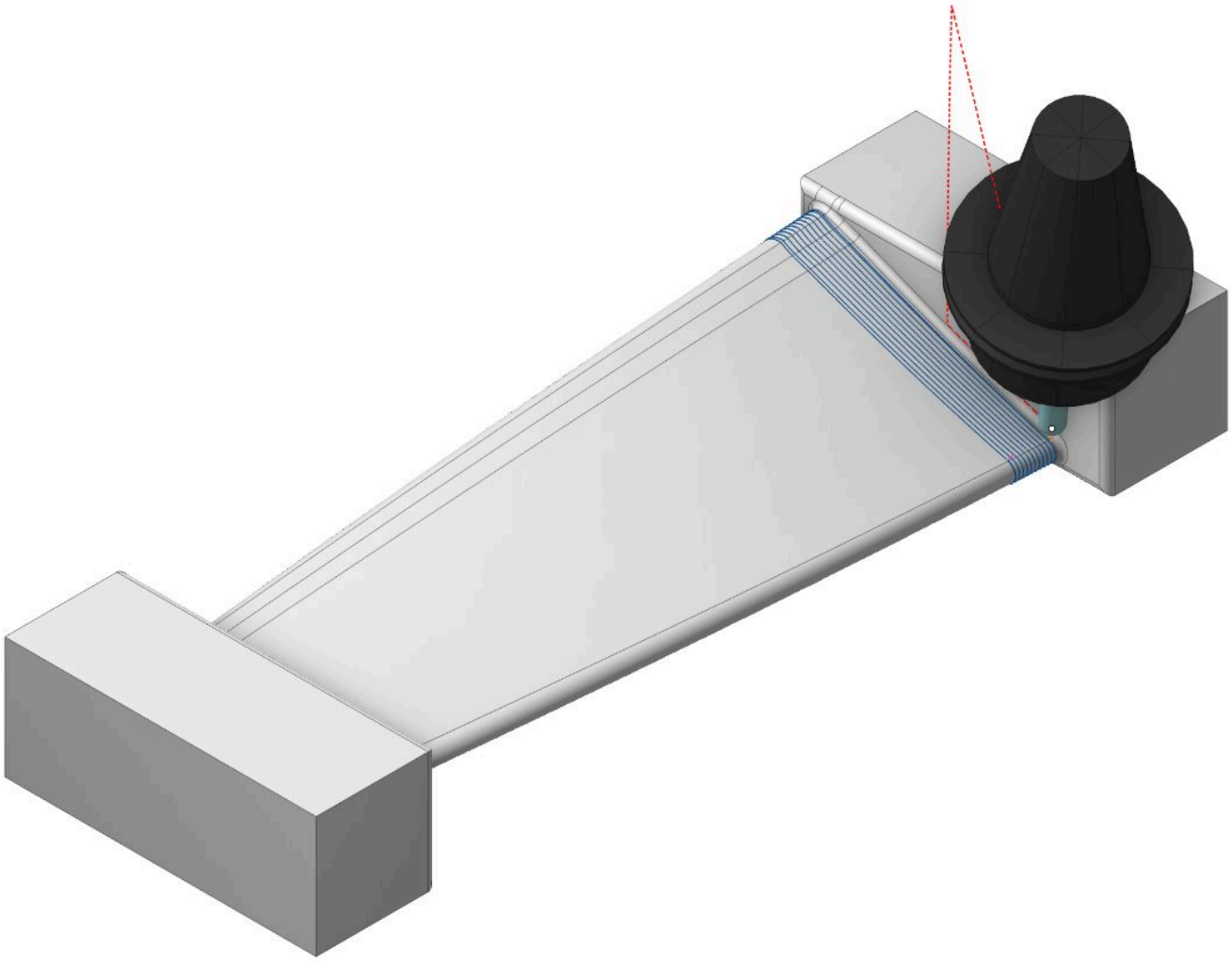
A workpiece model of an operation defines the material to be machined. Details

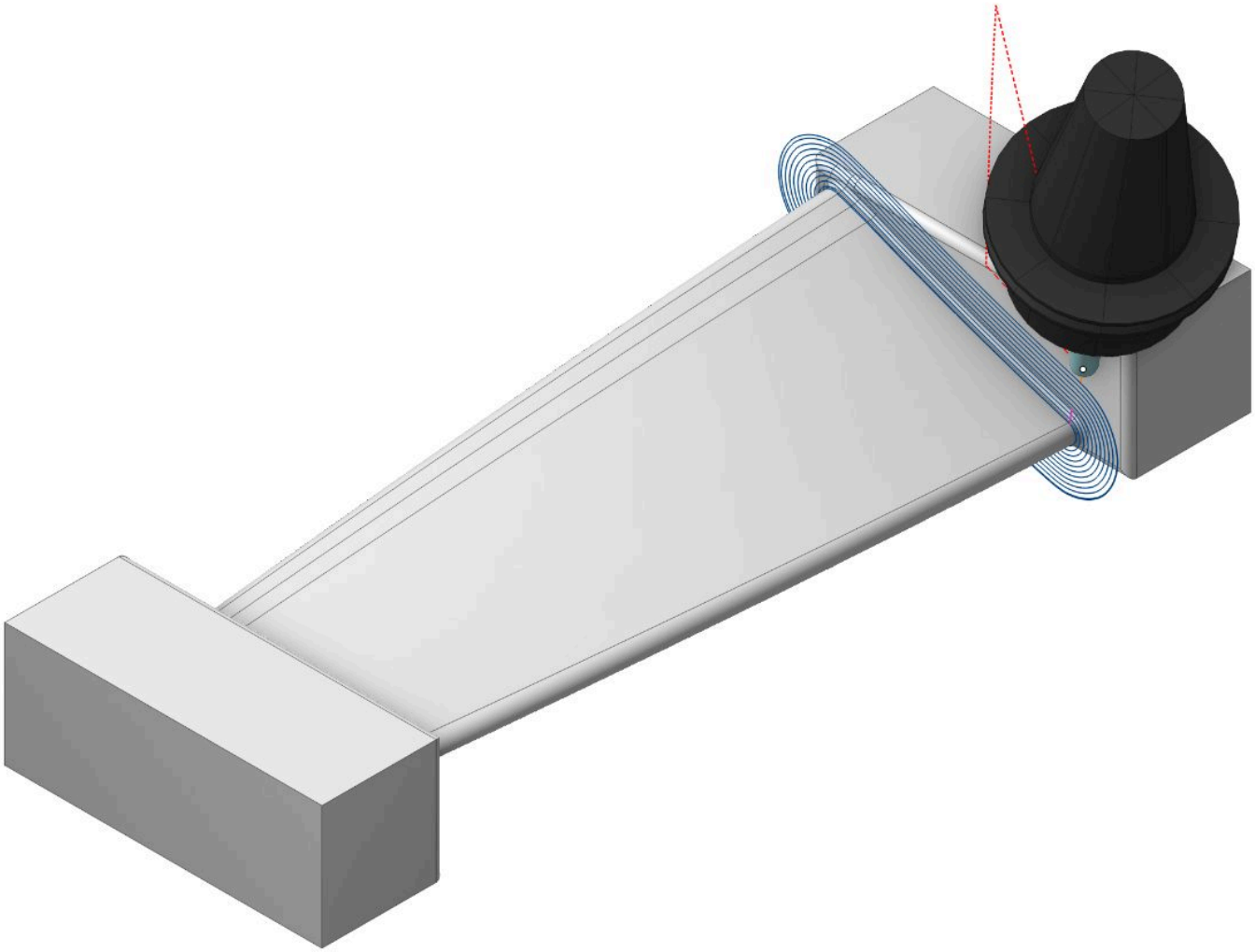
Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Blade Machining







Application area:

The Blade Milling operation is a specialized multi-axis finishing operation for machining single blades of compressors, turbines, and impellers. The operation machines the blade airfoil, the hub (platform) surfaces with fillets, the blade root zone, and the leading and trailing edges, using a rotary-axis machining model: toolpath passes are built around a user-defined rotary axis and projected onto the part surfaces.

What it is used for typical scenarios:

- Finishing of the airfoil profile with a continuous Spiral pass, separate Circular loops, or longitudinal Linear passes between the edges.
- Machining of the hub (flow-path) surfaces and fillets adjacent to the blade.
- Additional cleanup passes in the root zone of the blade next to the hub.
- Dedicated longitudinal machining of the leading and trailing edges with control of the tool contact point height.

- Compensation of thin-blade deflection by a smoothly varying finishing stock (Bend compensation stock).

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

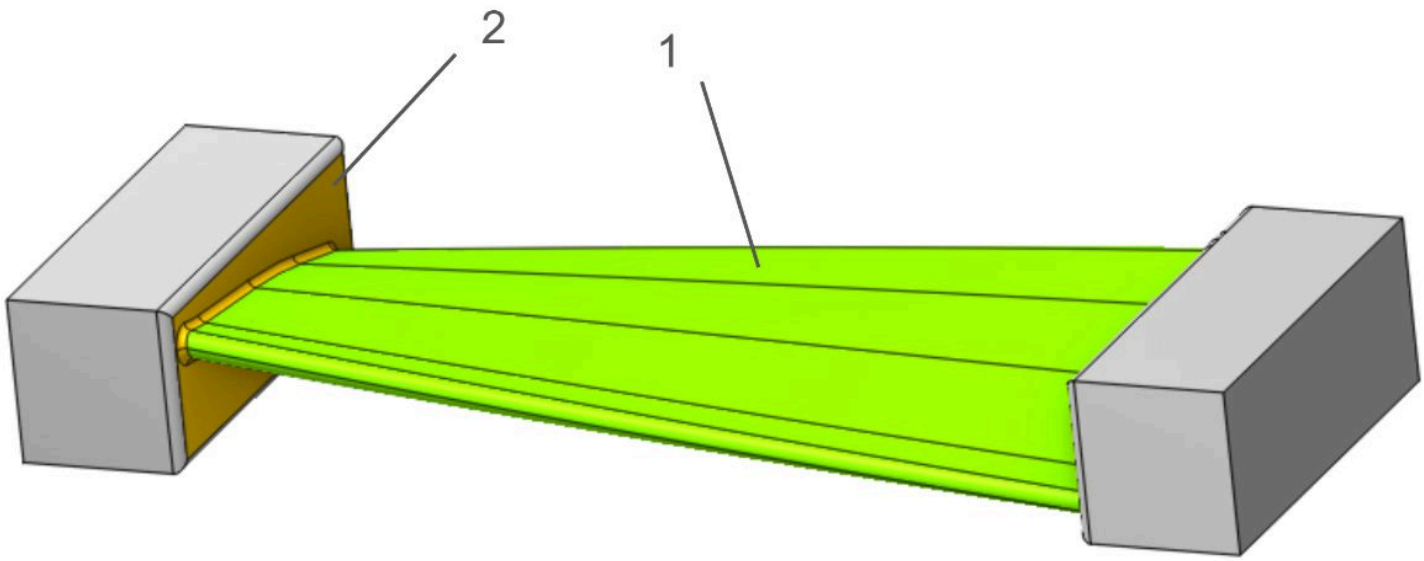
Blade surfaces. The set of faces forming the blade airfoil (pressure side, suction side, and edge surfaces). Required when the Target zone is Blade, Root zone of blade, or Edges.

Hub surfaces. The set of faces forming the hub (platform, flow path) together with the fillets between the hub and the blade. Required when the Target zone is Hub. For the Root zone of blade the hub surfaces act as the limiting geometry of the machined area; they may be omitted — the calculation then runs without the hub limitation.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details



1-Blade Surfaces, 2-Hub Surfaces

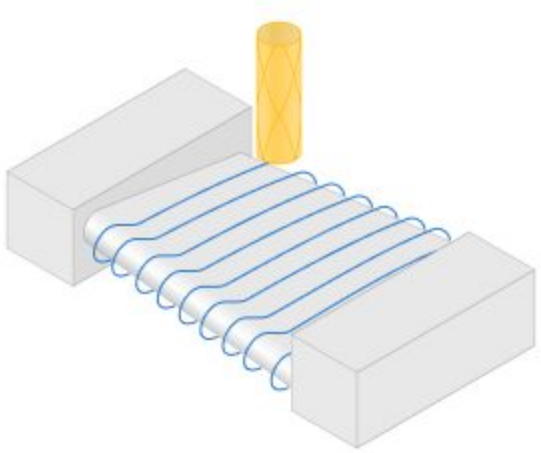
Strategy.

Target zone.

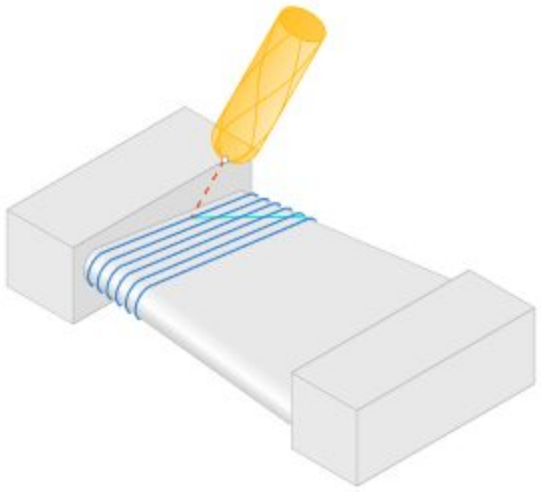
The Target zone parameter selects which structural element of the blade is machined. It is the main switch of the operation: it defines the machining strategy, the required geometry groups, and the set of visible parameters.

▼ Click here to expand..

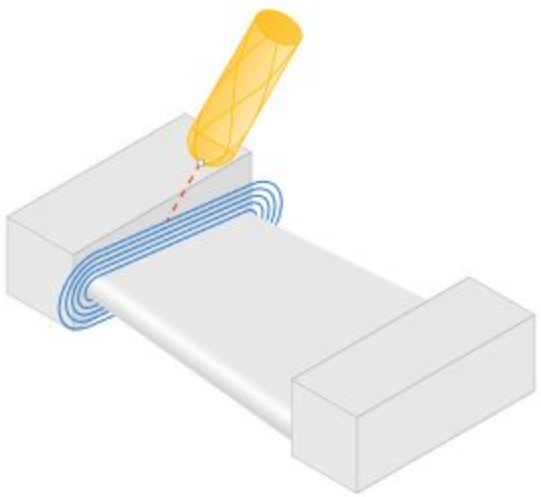
Blade. Machining of the airfoil profile. Passes are generated around the rotary axis over the whole working range of the blade and projected onto the Blade surfaces. All three trajectory forms are available (Spiral, Circular, Linear).



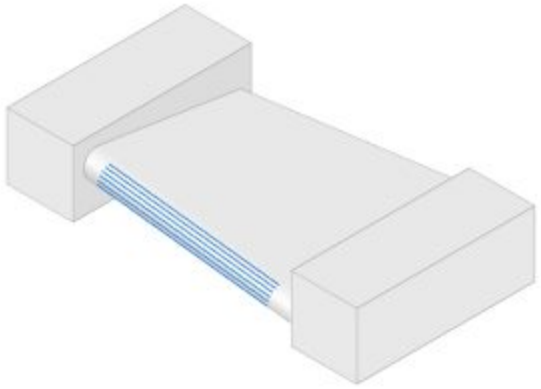
Root zone of blade. Additional passes over the lower part of the blade surfaces adjacent to the hub. The machined band is limited by the Zone height value measured from the hub. The zone is intended for cleanup with a different (usually smaller or tapered) tool after the main airfoil machining.



Hub. Machining of the hub surfaces and fillets: equidistant offset passes are built successively from the contour at the blade outwards with the constant Step over the Hub surfaces. The working band along the blade axis is set by the Zone height parameter. Outside the outer hub contour the passes are extended automatically; on the extension the next-pass feed is applied.



Edges. Dedicated machining of the leading edge (LE) and the trailing edge (TE) by longitudinal passes. The edges are detected automatically as curvature peaks of the airfoil cross-sections; the passes run along each edge and are distributed across the edge according to the Pass count and Arc length parameters. The height of the tool contact point can be controlled along the pass - Tool contact point.



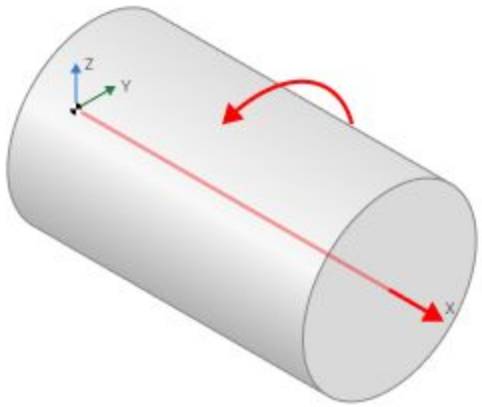
Job zone.

It enables additional modification of the working area initially selected on the Job assignment tab.

▼ Click here to expand..

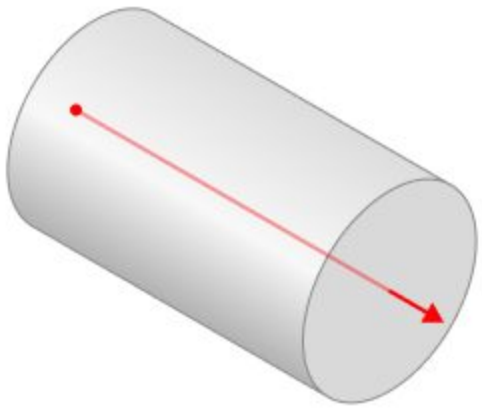
Rotary axis. The system directs tool's axis vector from the designated Rotary Axis. The rotary axis position can be defined in the properties inspector. You can adjust the rotary axis interactively using the arrow or select it from the drop-down menu.

If the mode is set to the Along X, Along Y or Along Z, then the axis of rotation will be oriented along the corresponding axis of the coordinate system.



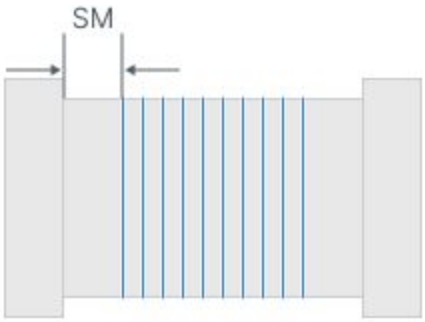
Direction. If mode is Custom(WCS) then axis vector must be defined manually in the fields X,Y,Z of custom direction group. Specify the direction cosines relative to each axis.

Origin. The Origin field defines the point that acts as the coordinate start for the chosen rotary axis.



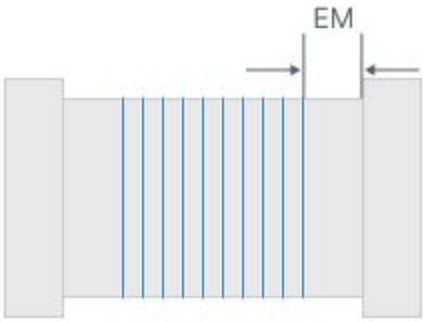
Base CS. The Base Coordinate System (Base CS) changes the coordinate system between the global and local coordinate systems of the part.

Start margin. The offset of the machining start from the blade root.



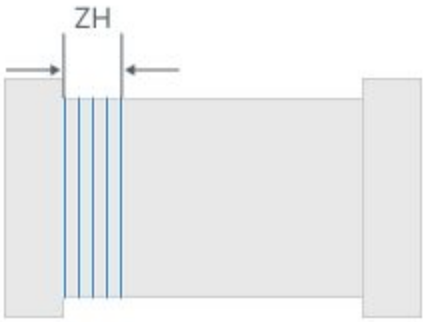
SM - Start margin

End margin. The offset of the machining end from the blade tip.



EM - End margin

Zone height. For the Root zone of blade and Hub target zones: the height of the machined band measured from the hub along the blade axis.



ZH - Zone height

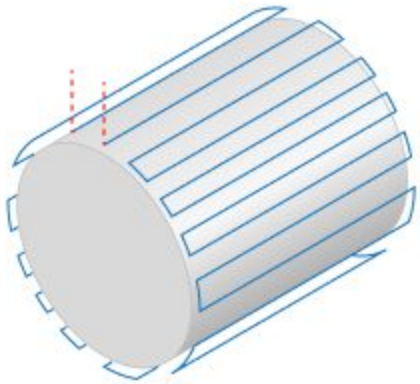
Trajectory form.

The trajectory output format.

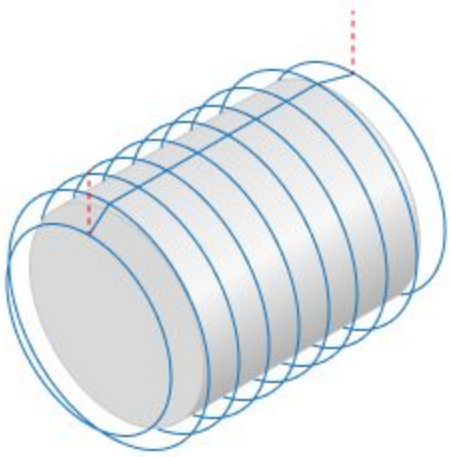
▼ Click here to expand..

The trajectory can be in one of the following forms:

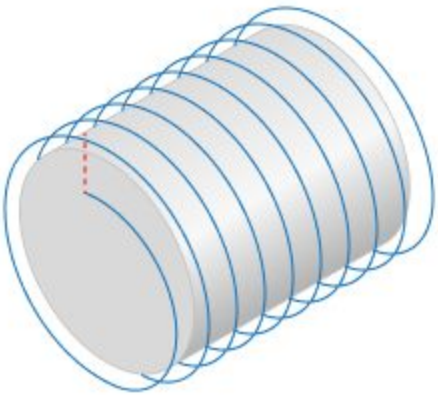
Linear. When you select this option, the work passes form as intersection lines between the machined surfaces and planes through the Rotary Axis..



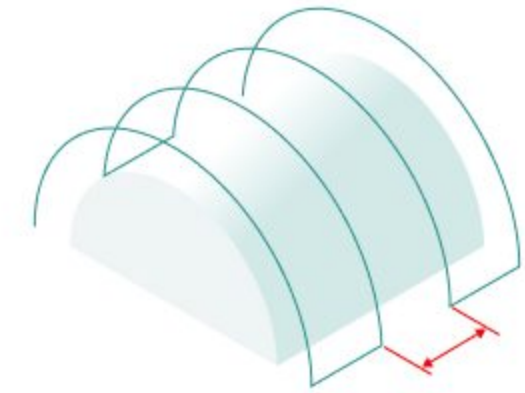
Circular. When you select this option, the work passes form as intersection lines between the machined surfaces and planes perpendicular to the Rotary Axis.



Spiral. Activating this option shapes the toolpath into a continuous helical line. The pattern is well suited for machining parts like screws and impellers.

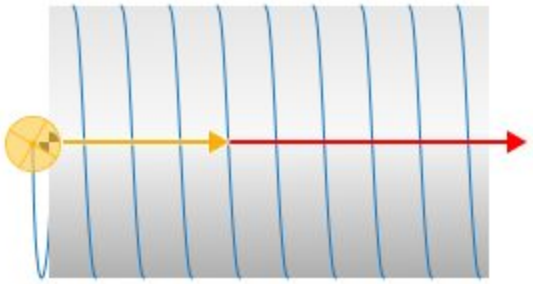


Step. It is the distance between tool passes along the rotary axis.

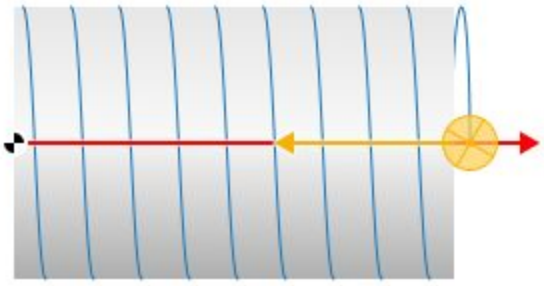


Axial direction. Depending on the feature of the Trajectory Form parameter, it either defines the direction of the tool passes themselves or the sequence of several tool passes. In the case of a spiral whose step matches the machining Step, this parameter influences the direction of the spiral's twist.

Forward. The tool passes is ordered in the Rotary Axis vector direction.

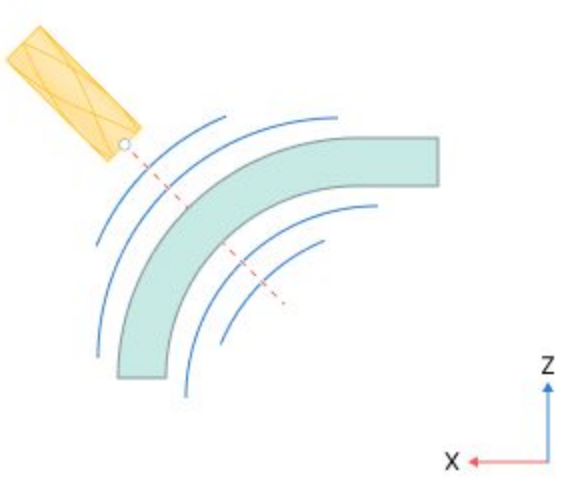


Backward. The tool passes are ordered in the reverse direction of the Rotary Axis vector.

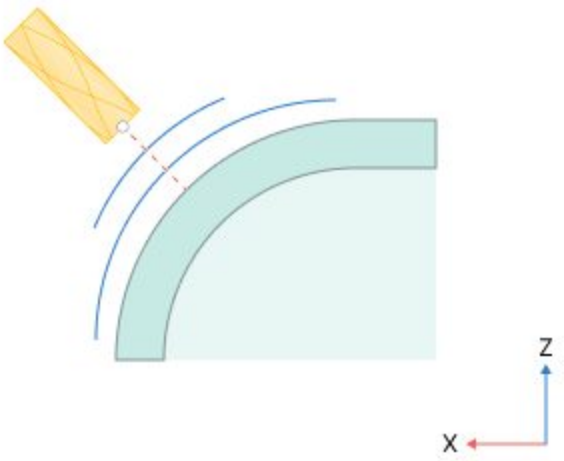


Side. For Linear and Circular modes in the Blade zone: selects which side of the airfoil is machined. In Circular mode with a single side selected the loops become open arcs limited by the edge margins.

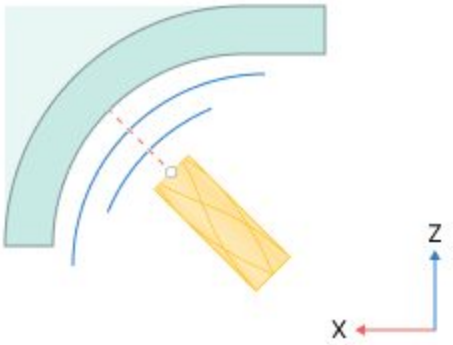
Both. The tool can perform processing from both sides of the surface.



Convex (suction side). The tool can only process from the side with the positive direction of the surface normal vector.



Concave (pressure side). The tool can only perform processing from the side opposite to the surface normal vector.



LE margin. Offset from the leading-edge peak along the airfoil arc; default equals tool diameter.



TE margin. Offset from the trailing-edge peak along the airfoil arc.



Tool contact point.

Change the tool's contact point.

▼ Click here to expand..
One point. A fixed contact point held constant along the whole pass. Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations. This group works similarly 5D Surfacing operation - Tool contact point. Details

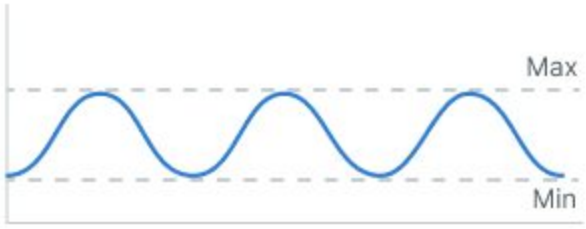
Changing the tool contact point along the pass

Two point. C hange the tool's contact point from the initial line to the final line. This group works similarly 5D Surfacing operation - Contact Points. Details



Changing *the tool contact point along the pass*

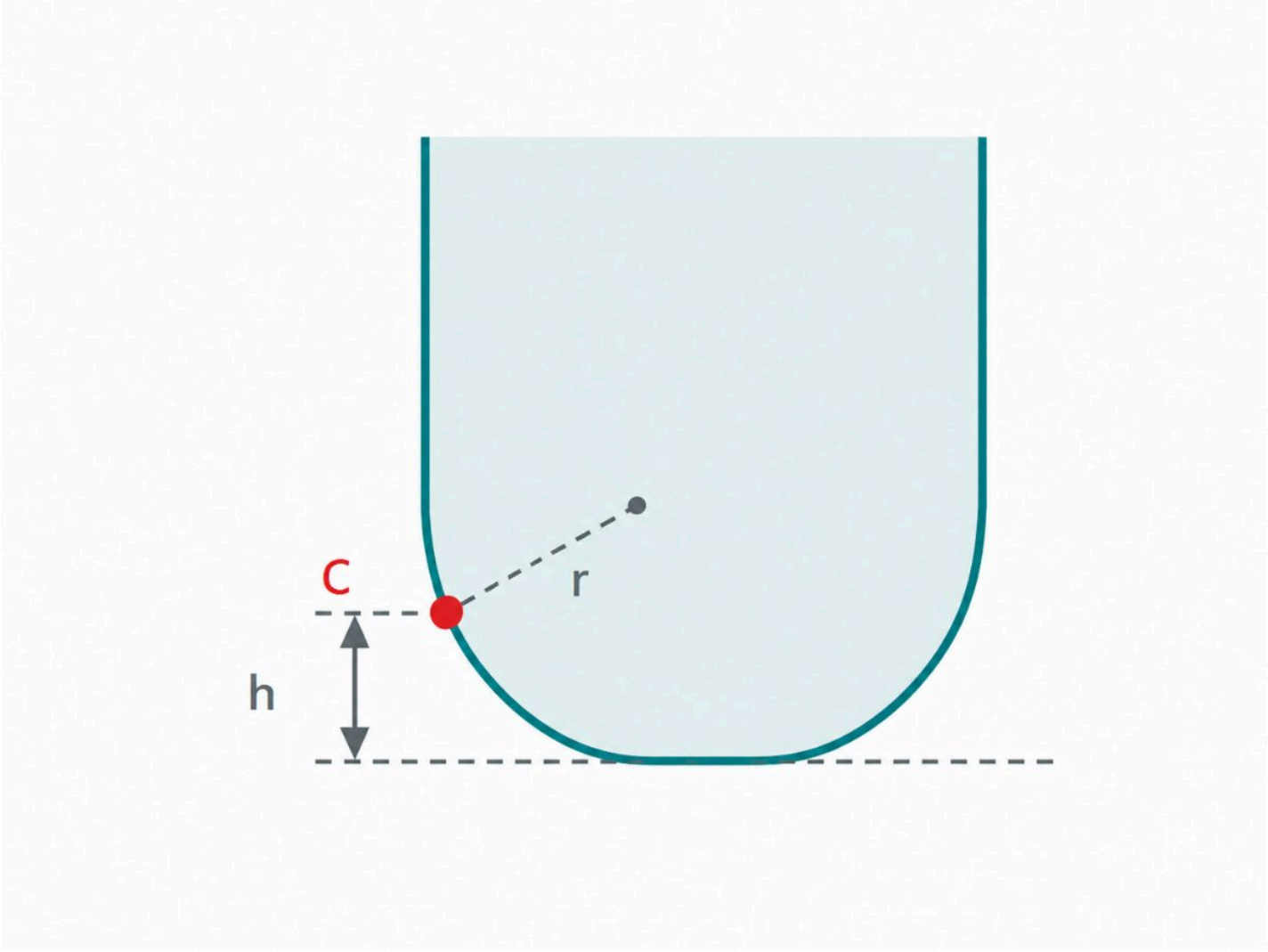
Wave. The contact point oscillates sinusoidally between the Start point and End point values with the specified Wave period (the wavelength measured along the pass, default — 20 linear units).



Changing the tool contact point along the pass

Wave period. Wave distribution gives the most uniform wear of the tool corner radius.

> Note.
>
> The lead angle in every point of the pass is computed from the current contact height h and the tool corner radius r as:
>
> lead = arccos(1 - h/r)
>
> The Lead angle field of the Tool orientation group is hidden while the block is active.

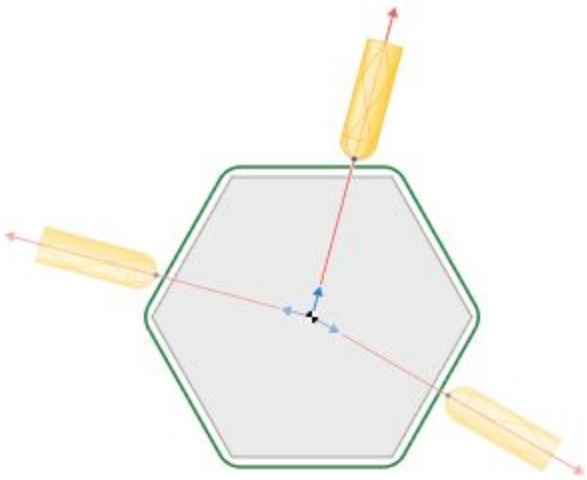


Tool orientation.

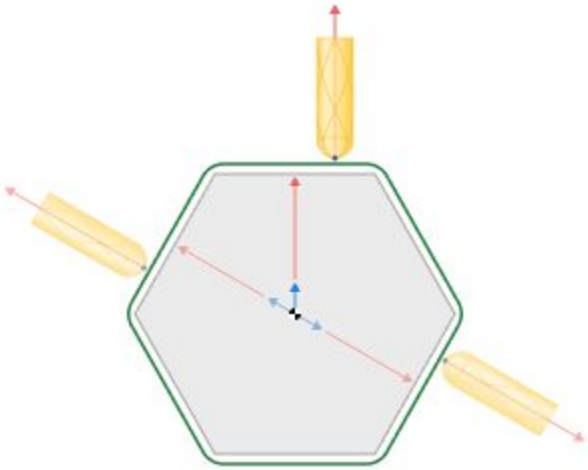
The Tool orientation group controls the direction of the tool axis relative to the machined surface.

▼ Click here to expand..

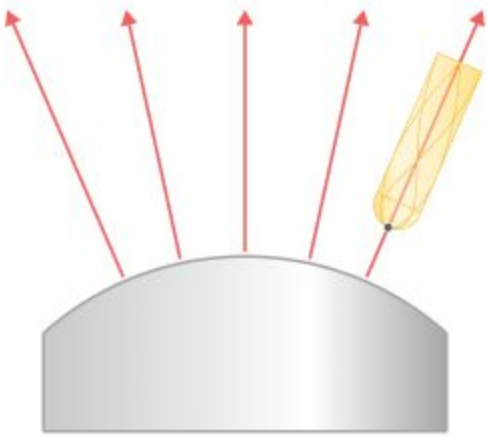
To Rotary Axis. The system directs the tool axis vector from the designated Rotary axis. The axis is defined in the Job zone group.



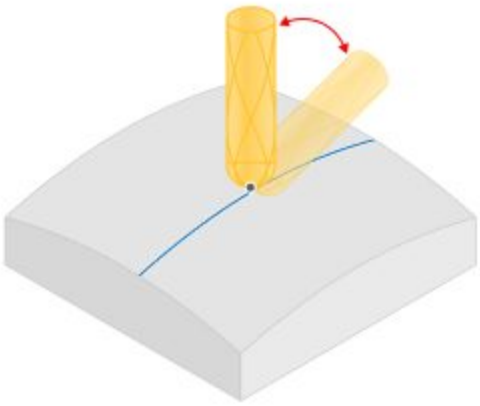
In Rotary Axis Plane. The tool axis lies in the plane perpendicular to the Rotary axis: the surface normal is taken without its component along the axis. Side angle sets the tilt along the axis.



By Surface Normal. The system orients the tool axis normal to the machined surface. This is the default orientation mode.



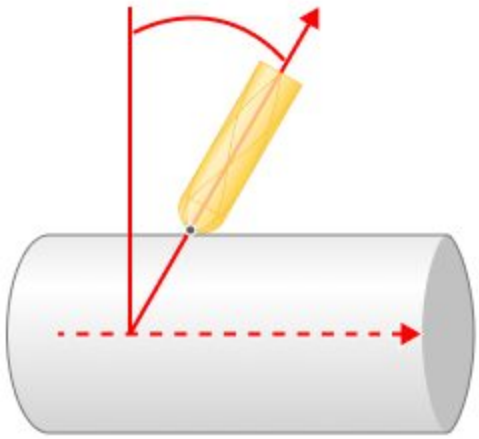
Lead angle. Specifies an additional tool tilt angle within the plane of motion direction. Available in In Rotary Axis Plane and By Surface Normal modes. The default value is 5°.



Auto min lead angle. When ON, automatically increases the lead angle where the surface rises behind the tool and the rear (heel) part of the cutter bottom would engage the surface. The effective angle is the greater of the specified Lead angle and the detected surface rise plus Back angle; the automatically increased angle is limited to 60°. Where no increase is required, the specified Lead angle is retained. When OFF, this automatic adjustment is disabled. The option applies to the Blade target zone and has no effect on a spherical tool, which has no flat bottom.

Back angle. Specifies the additional heel clearance angle added to the detected surface rise. The value is edited in the Auto min lead angle row and defaults to the Lead angle value.

Side angle. Specifies the inclination angle of the tool axis relative to the axis designated as the Rotary axis. The value in this row defines the inclination at the start of the working zone. The default value is 0°.



End angle. Specifies the inclination angle of the tool axis relative to the Rotary axis at the end of the working zone. Expand Side angle to access this field. By default, it equals Side angle. If the values differ, the inclination changes linearly along the blade axis between them.

The corresponding orientation and angle settings are described for the Surfacing operation. Details

In the Edges target zone, enabling Tool contact point hides the Lead angle field. The lead angle is then calculated from the specified contact height on the tool corner radius.

Edge normal smoothing.

The Edge normal smoothing block smooths the contact normal of the tool in the approach zone near the leading and trailing edges. On thin edges the surface curvature changes abruptly, which causes a sharp "dive" of the tool axis when the pass crosses the edge; the smoothing blends the normal over a distance before the edge and removes the jerk of the rotary axes.

Bend compensation stock.

The Bend compensation stock block adds a smoothly varying stock along the blade to compensate the elastic deflection of a thin blade. Under the cutting force the free end of the blade bends away from the tool, so a uniform finishing pass leaves an uncut layer near the tip. The block leaves an intentional, smoothly growing stock in the flexible region; this stock is removed by a final light pass with small cutting forces. The maximum additional stock at the peak position, in linear units (default — 0.05).

▼ Click here to expand..
Start position. The relative position (0⋯1) along the working zone where the additional stock begins (default — 0.3). The range 0⋯1 corresponds to the working zone between the resolved start and end

margins.

Peak position. The relative position where the additional stock reaches Max stock (default — 0.8).

End position. The relative position where the additional stock ends (default — 1.0).

Profile. The shape of the stock variation between the positions: Linear, Parabolic (default), or Sine.

> Note.
>
> The block is available for the Blade and Root zone of blade target zones. In the Hub and Edges zones it is ignored.

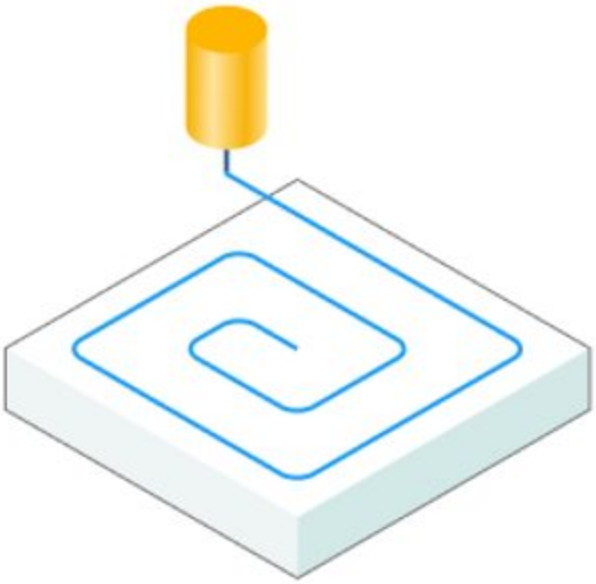
Milling type.

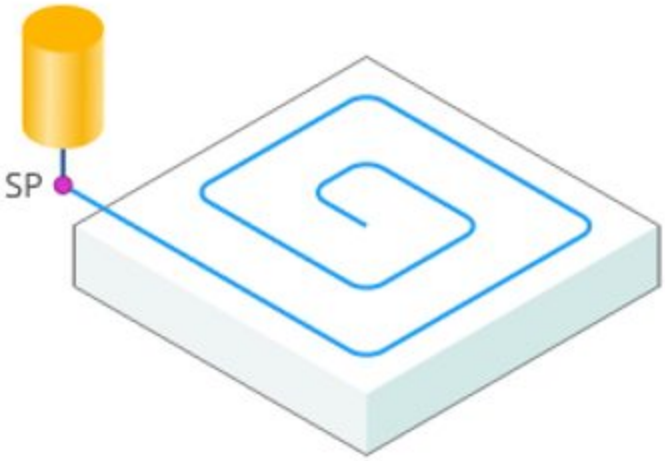
Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting.

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Start Point. The point defining the beginning of the plunge.



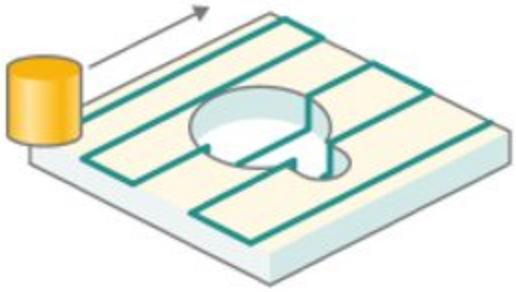


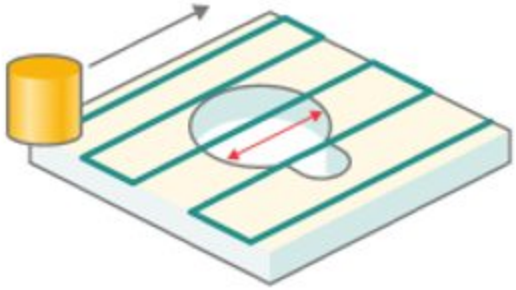
Hole capping:

Hole capping. Allows holes in the model to be ignored during surface machining. The toolpath is generated as if the holes were closed, allowing working passes to continue across them.

The holes may be circular or non-circular. The specified size represents the diameter of a disk that can completely cover a hole.

▼ Click here to expand..





Parameters:

Parameters. Contains the main settings for controlling how the part, workpiece, fixtures, and other geometry affect the generated toolpath. Details

Links/Leads:

Links/Leads. Defines tool movements before, between, and after working passes, including approach from the tool change position, engagement, retraction, links between passes, and return to the tool

change position. The settings control the sequence, shape, and distances of these movements.

▼ Click here to expand..
Consists of:

Approach/Return. Controls tool movements from the tool change position to the beginning of the engage path and back. Details

Links. Defines the rules for entry and exit links and links between working passes. Details

Leads. Adds engage and retract motions to working passes and defines their shape and dimensions. These motions use their corresponding feed rates. Details

Safe motions. Defines the type of Safety Surface and its distance from the workpiece. The surface is used for links between machining areas or toolpaths and for approach and return movements. Details

Advanced links settings. Controls additional aspects of link generation for five-axis machining, taking rotary-axis movements into account. Details

Distances. Defines the distances used by the linking rules, including feed switching and retraction distances. Details

Feeds/Speeds:

Feeds/Speeds. Defines the spindle speed, rapid feed, and feed rates for different parts of the toolpath. Spindle speed can be specified in revolutions per minute or as a cutting speed. The defining value is underlined; the other value is calculated from it using the tool diameter. Details

Transformations:

Transformations. Defines coordinate transformations applied to the toolpath calculated for the operation. Details

Part:

Part. Defines the geometry used to check the toolpath for gouges. Details

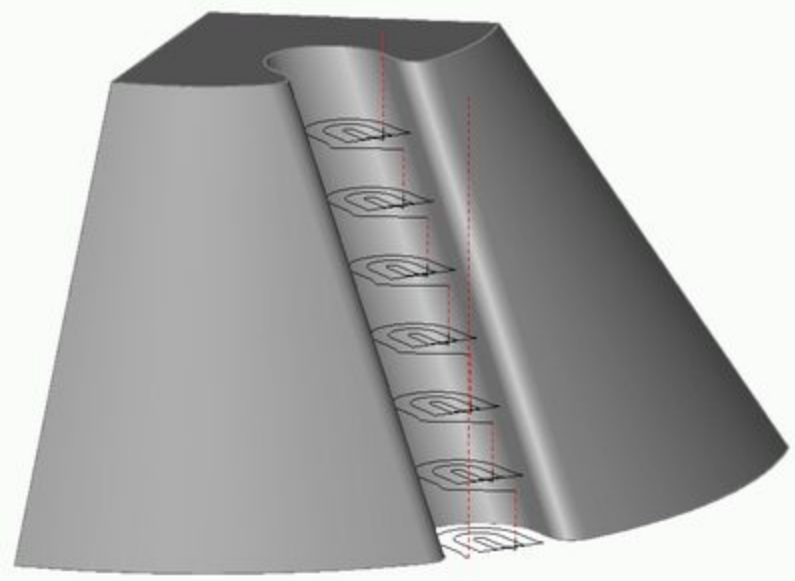
Workpiece:

Workpiece. Defines the material to be machined in the operation. Details

Fixtures:

Fixtures. Defines workholding devices, such as chucks, grips, and clamps, and other restricted geometry considered during toolpath calculation. Details

Rest machining



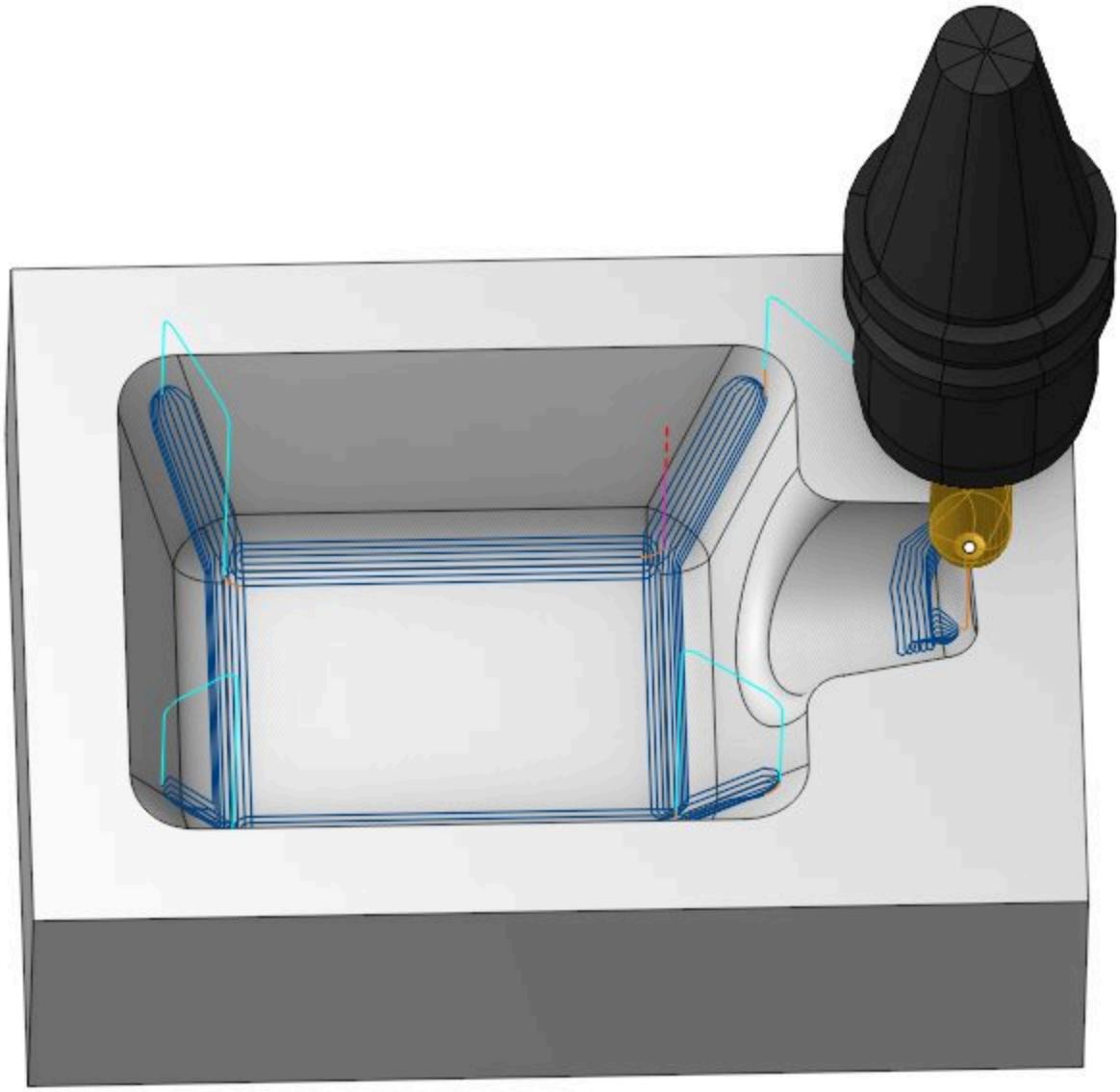
The rest milling operations allow the user to perform machining only in areas where there is unmachined stock material left after previous machining operations. It is also possible to machine remaining material after 'virtual' machining performed using a user-defined list of tools. These operations are designed for rough or finish rest milling using tools that have a different shape or with a smaller diameter than those that were used for previous operations. The application of rest milling can considerably decrease the machining time for complex shaped models, thereby reducing costs.

The rest milling operations are identical to the other machining operations, the only differences are the default parameters assigned. These defaults are: a smaller diameter tool is selected, and the workpiece is set as 'residual material'. During rest milling, the roughing operations perform removal of the 'volume' residual material, and the finishing operations machine the model surfaces only in areas that are unfinished.

Rest milling of residual material is enabled in the system by selecting the workpiece as material that is left after all previous operations. Calculation of the workpiece and selection of unmachined areas is performed automatically by the system. This approach has some valuable benefits when compared to the widely used method of Machine after a tool of any size. Firstly, it can correctly calculate the residual material, even if incompatible definitions for previous operations are used. It also means that all previously used operations and their characteristics (tool types etc.) are considered when creating the rest machining strategy.

All machining characteristics for the rest milling operations are similar to the normal machining operations of the same type.

Corners cleanup



Application Area:

It iis an operation from the rest machining group. It takes the diameter of the previous tool as a parameter and generates passes where the previous tool would leave unmachined material.

Job Assignment:

Machining Surfaces. The passes are generated only on places where the tool contacts the machining surfaces. If the machining surfaces are missing, the toolpath is generated within the whole part.

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details.

Tilt Curve. This is t he curve that the tool axis vector aims at when setting 5 Axis Conversion specification Through Curve.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

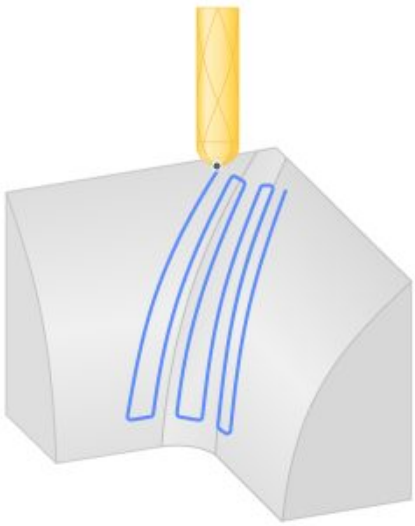
Strategy:

Strategy:

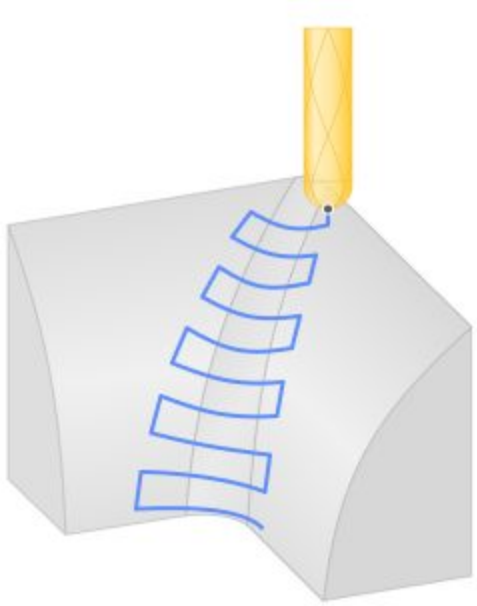
This group of parameters allows the user to achieve a required toolpath:

▼ Click here to expand..
Toolpath Direction.

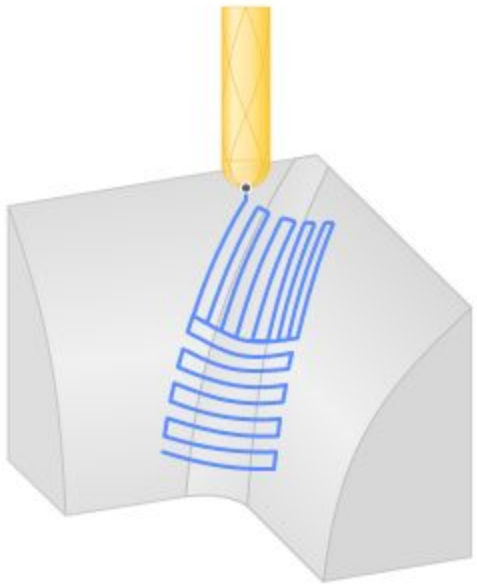
▼ Click here to expand..
Along. The system generates the passes along the inner corners.



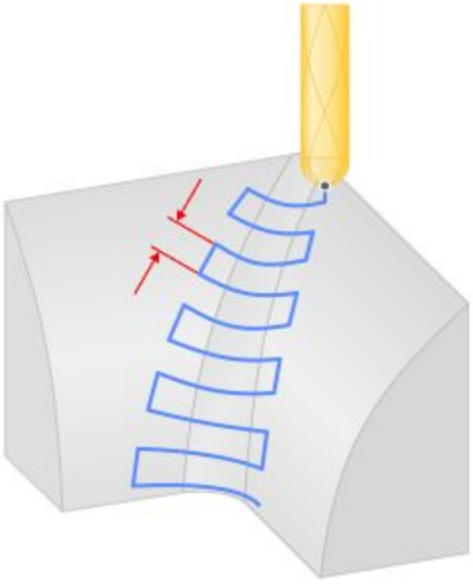
Across. The system generates the passes across the inner corners.



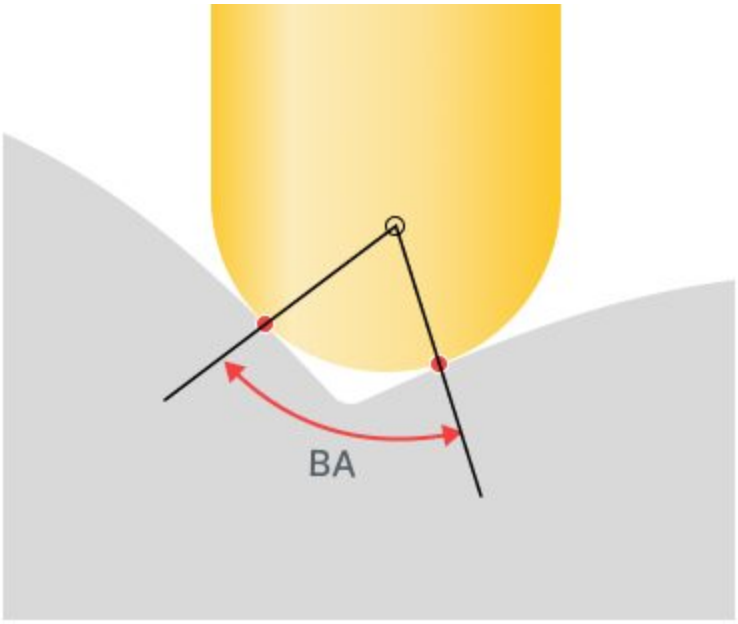
Combined. The system generates the along passes for shallow areas, and generates the across passes for steep areas.



Step. The maximum allowed width of cut.

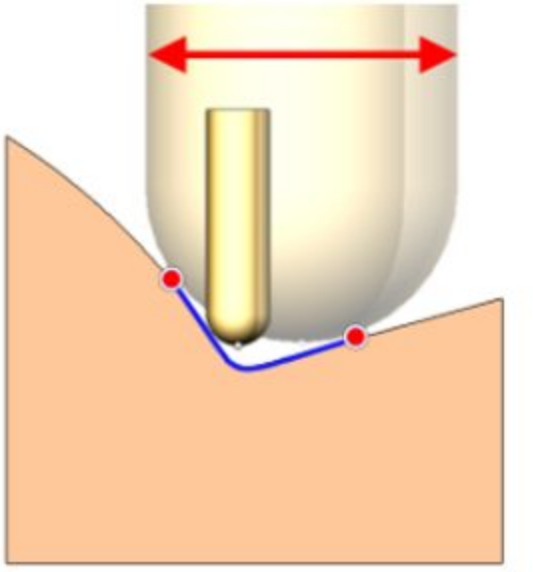


Bitagency Angle. This parameter is used to limit the edges detect by the operation. It is the angle between two points where the part's surfaces and the tool's cutting edges simultaneously make contact. The tool will operate within zone inside this angle.

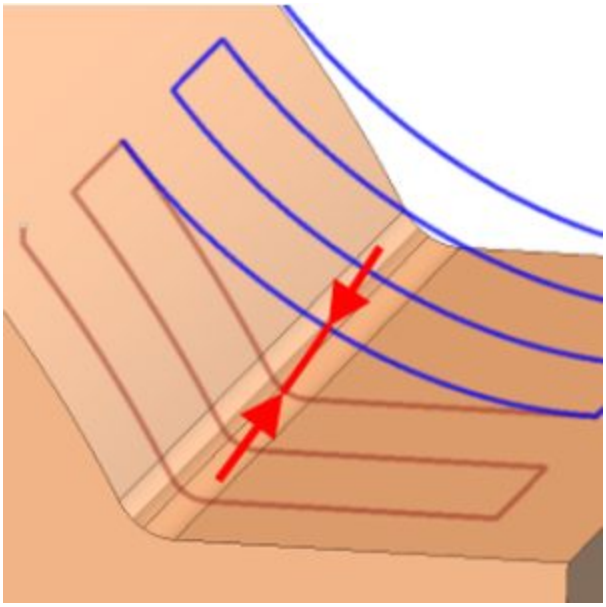


BA - Bitangency angle

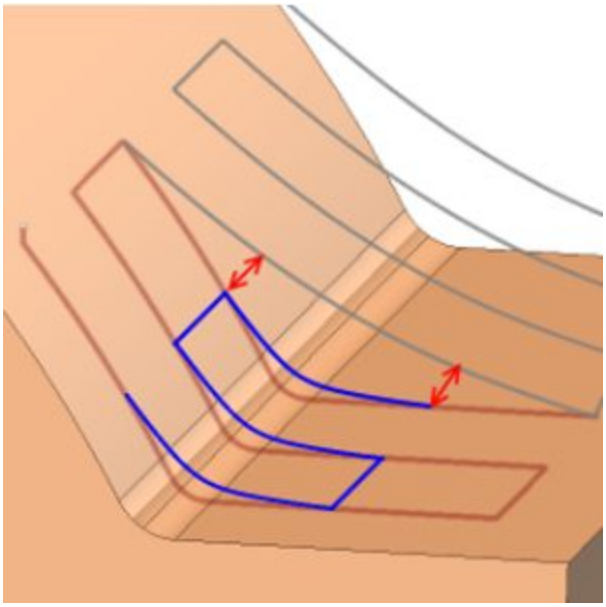
Previous Tool Diameter. Diameter of a sphericall mill used for rest material calculation.



Max Cut Depth. The maximal allowable depth of cut.



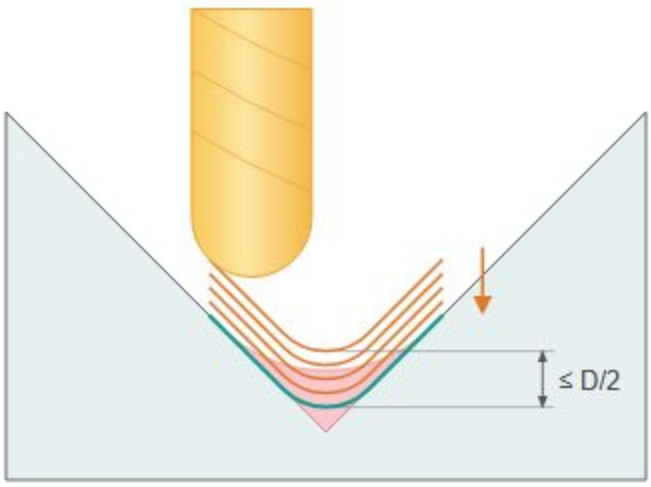
Min Cut Depth. The minimal allowable depth of cut.



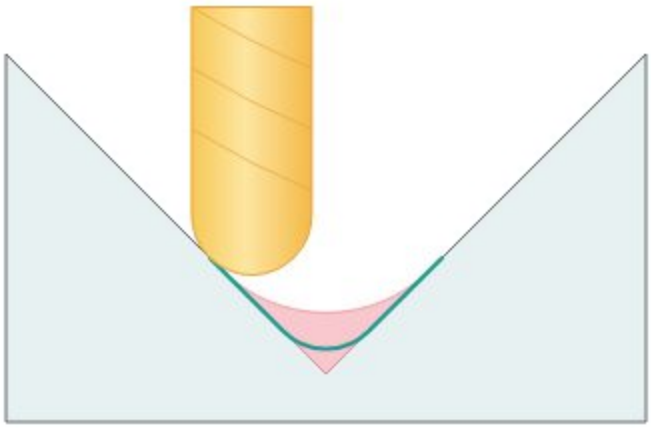
Gradual plunge. Controls how the tool enters the rest material at the start of each region machined across the corner.

▼ Click here to expand..

ON. Before the first pass of the region, copies of that pass are machined from the top down. The copies are offset upwards along the Z axis at intervals defined by Step. The highest copy is no more than half the tool diameter above the first pass. The tool enters the corner material gradually, layer by layer, followed by the first pass and the remaining passes. The toolpath is longer, and some copies may run in the air above the corner walls.



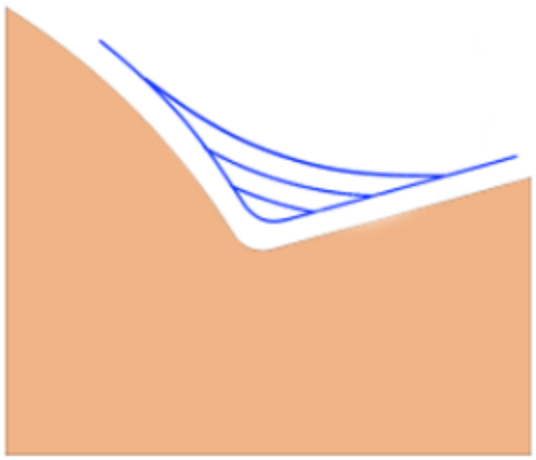
OFF. Machining of the region starts directly with its first pass. No additional passes are stacked above the start of the corner. The toolpath is shorter, but at the start of the region the tool removes the rest material to its full depth at once.



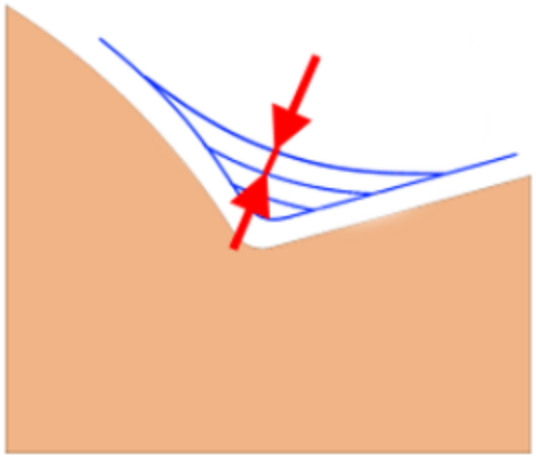
With Roughing checked and Unite Type set to By Strings, gradual plunge is enabled and disabled in the same way.

Clear the check box when the additional passes above the start of the corner are not needed, for example, when there is little rest material in the corner and the tool can remove it in one pass.
Roughing. Check the box to enabled the roughing passes.

▼ Click here to expand..

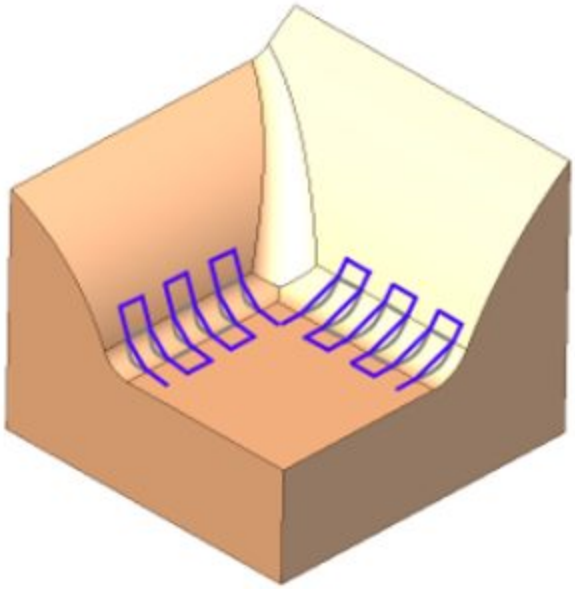


Roughing Step. The distance between roughing passes

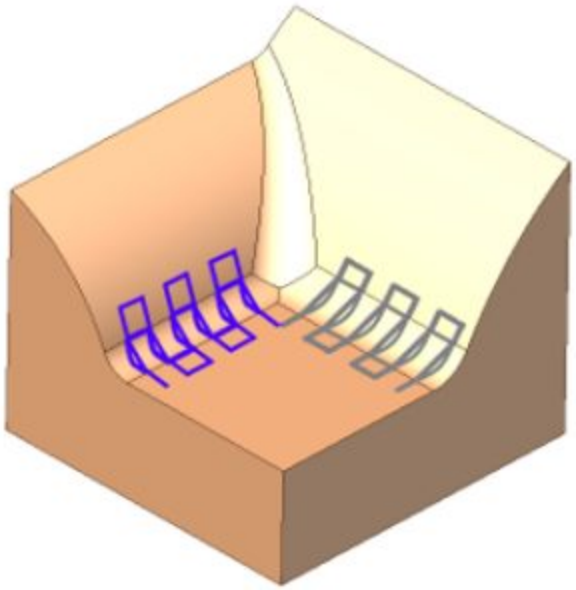


Unite Type. Controls the sequence of toolpath passes during the roughing process.

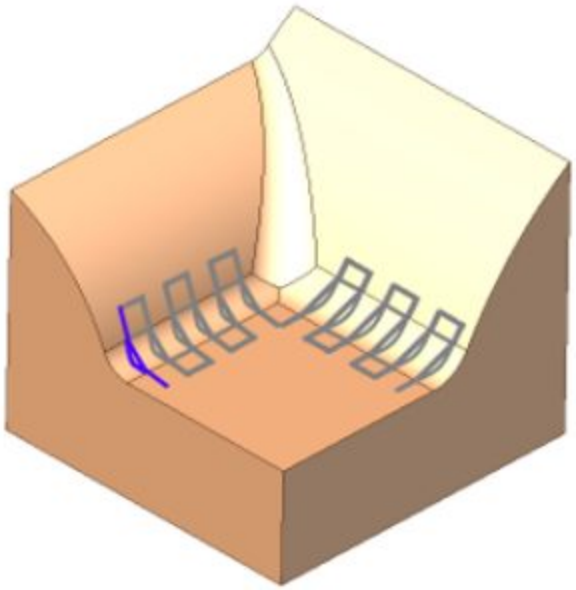
By Levels. Each level undergoes machining in every region before the operation advances to the subsequent level.



By Regions. The tool fully processes one region before advancing to machine the next region.



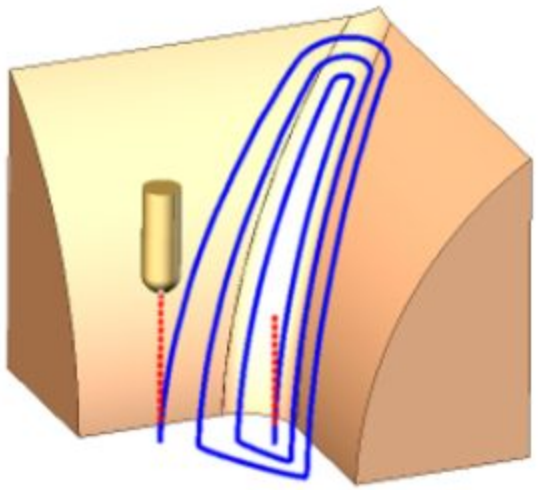
By Strings. The tool completes machining all levels of a string before moving on to the next string.



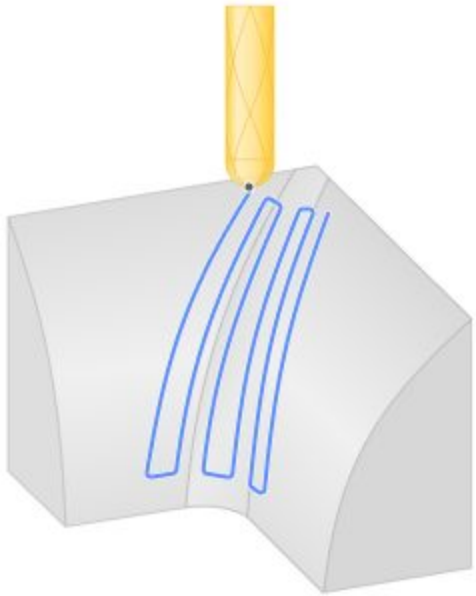
Along Linking Type. This parameter appears if the machining strategy includes along passes.

▼ Click here to expand..

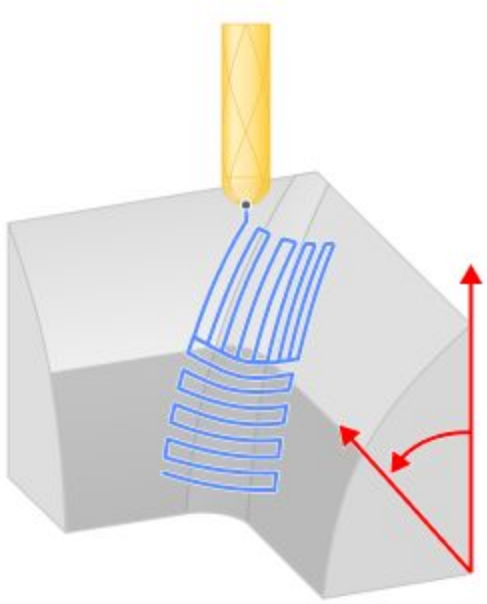
Spiral. The tool machines a single region following a continuous spiral toolpath.



Linear. The tool processes a single region by making passes that follow the corner line, with links between each pass.



Steep/shallow split angle. This parameter appears when the strategy of combined passes is selected, and demarcates steep from shallow regions.



Milling type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting:

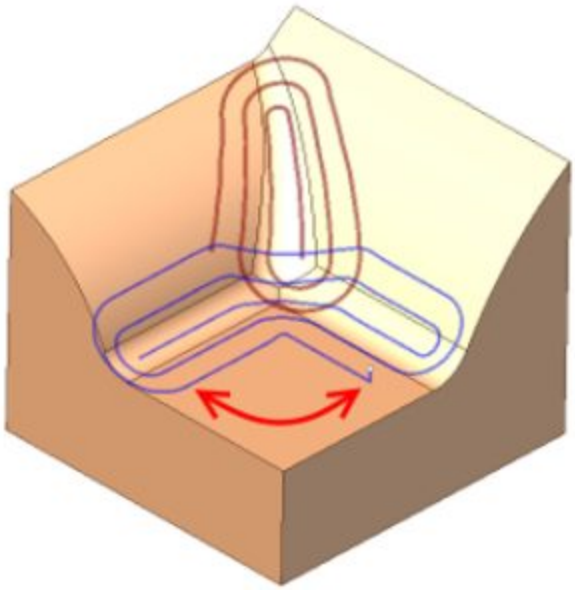
Controls the sequence of toolpath passes during the surface machining.

Connection:

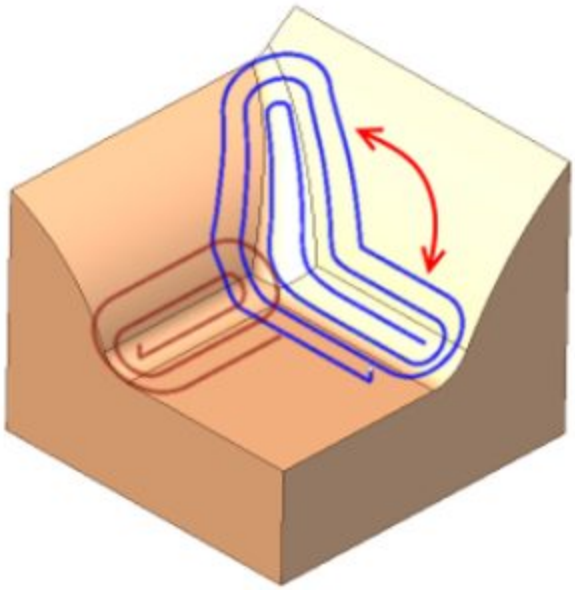
The parameter controls the consolidation of toolpaths for several regions if possible.

▼ Click here to expand..

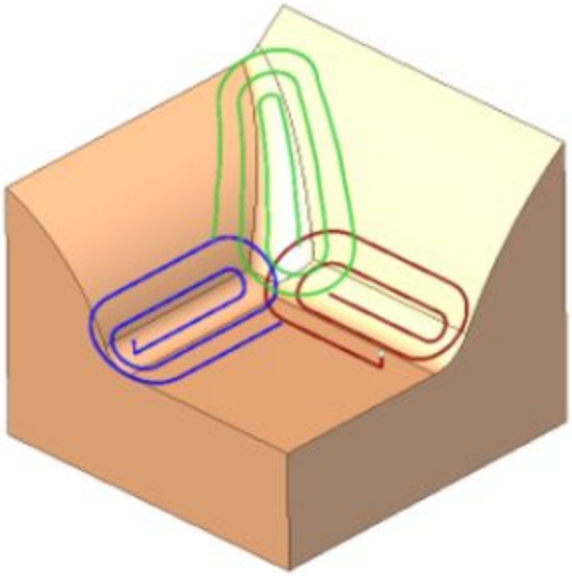
Constant Z. Passes are connected preferably in a plane.



Angular. Passes are connected preferably by the smoothest angle.



Off. Every region is machined separately.



Machining Levels:

It defines the range along tool axis for the machining. Details. Specify the Bottom Level as in the Waterline Roughing operation. Details

5 axis conversion:

Controls tool axis orientation. Details. The principal parameters of this function are the same as in the 5D Surfacing operation. Details

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

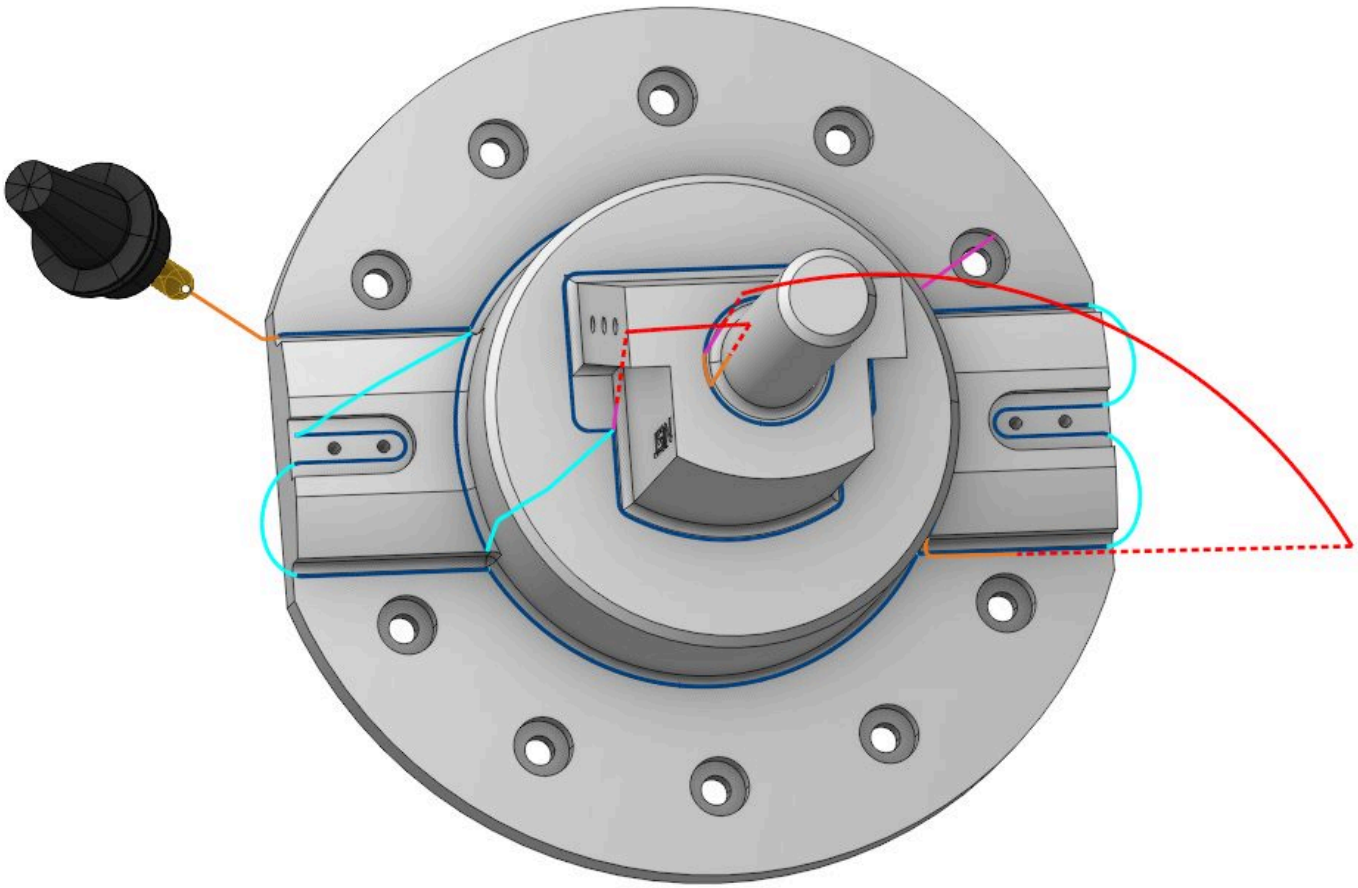
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Pencil



Application Area:

The rest machining operation generates passes along inner corners of the part.

Job Assignment:

Machining Surfaces. The passes are generated only on places where the tool contacts the machining surfaces. If the machining surfaces are missing, the toolpath is generated within the whole part.

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details.

Tilt Curve. This is t he curve that the tool axis vector aims at when setting 5 Axis Conversion specification Through Curve.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

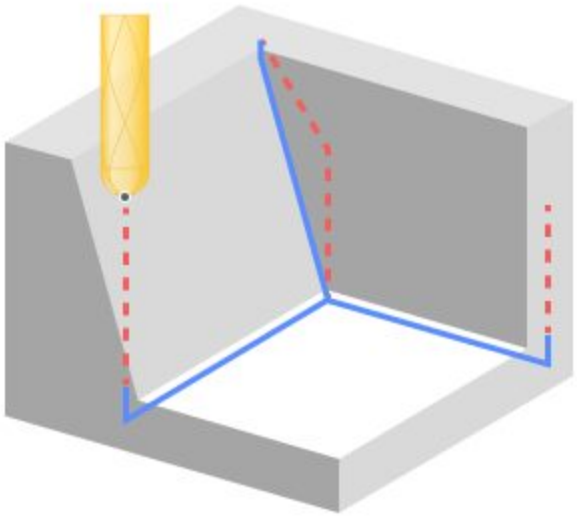
Strategy:

Strategy:

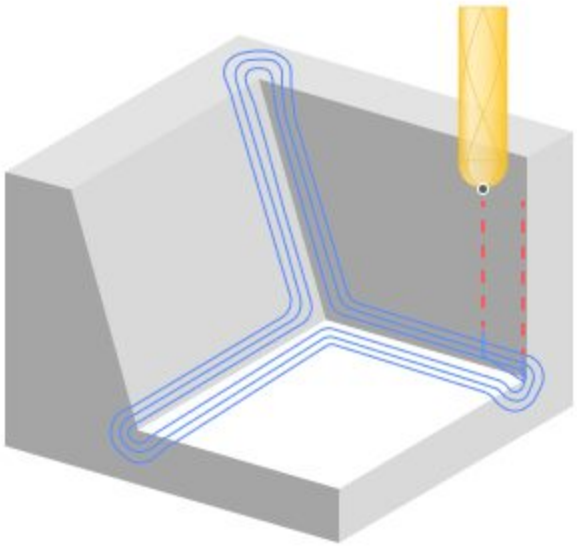
This group of parameters allows the user to achieve a required toolpath:

▼ Click here to expand..
Angle Processing Method:

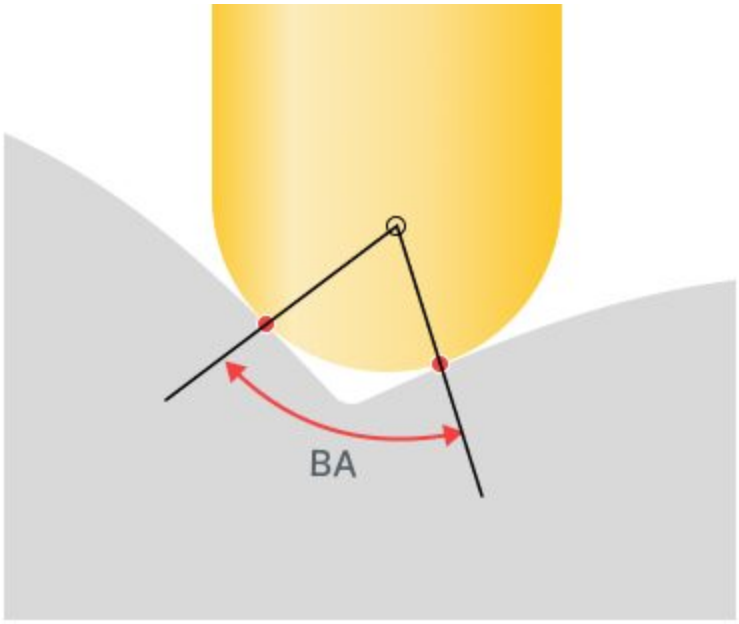
One pass. The system generates a single pass along every inner corner.



Parallel passes. The system g enerates multiple passes during the processing of a single inner corner. Each additional pass means a lateral shift of a one-pass strategy toolpath across the machining surface by a Step value. The maximum allowed width of cut

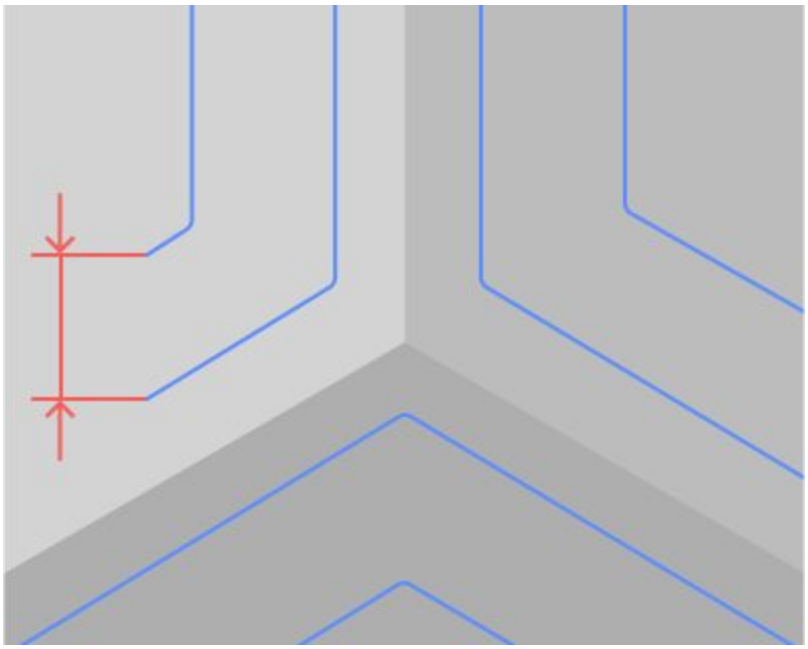


Bitagency Angle. This parameter is used to limit the edges detect by the operation. It is the angle between two points where the part's surfaces and the tool's cutting edges simultaneously make contact. The tool will operate within zone inside this angle.

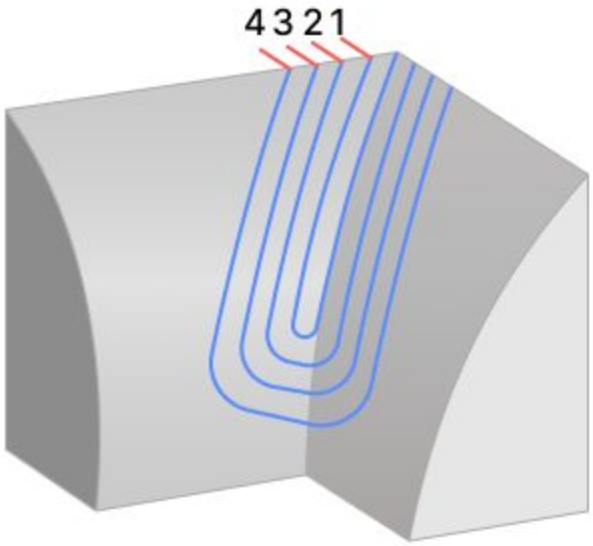


BA - Bitangency angle

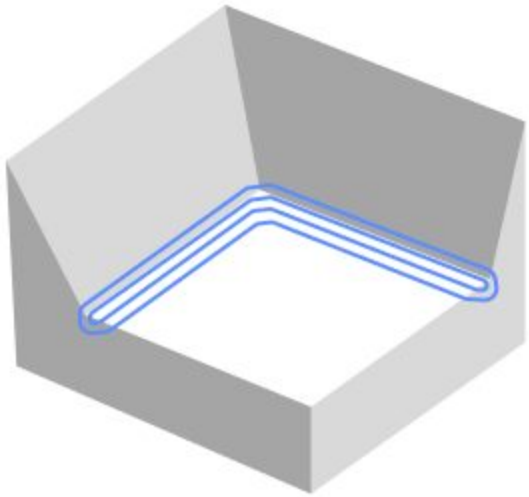
Step. The maximum allowed width of cut.



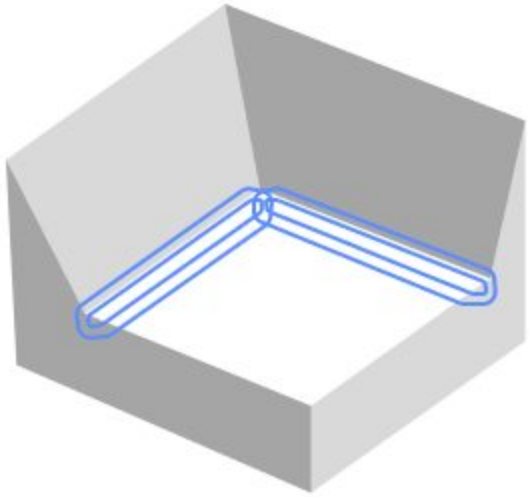
Nb. of passes. The number of passes along the corner. This number of passes extends on each side from the middle of the corner.



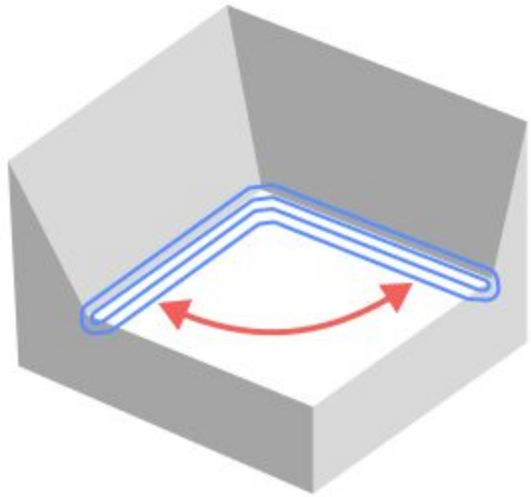
Machine by strokes. When the option is disabled, CAM system attempts to consolidate the machining of multiple corners in areas permitting a smooth transition.



Activating the flag separates the toolpath into regions, allowing for the independent processing of different corners wherever a smooth transition between corners can occur.



Connection angle. It is the sharpest angle between connected passes. If the angle between the corners being machined is greater than this value, the toolpath for these corners will be separated.



Milling type:

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

Sorting:

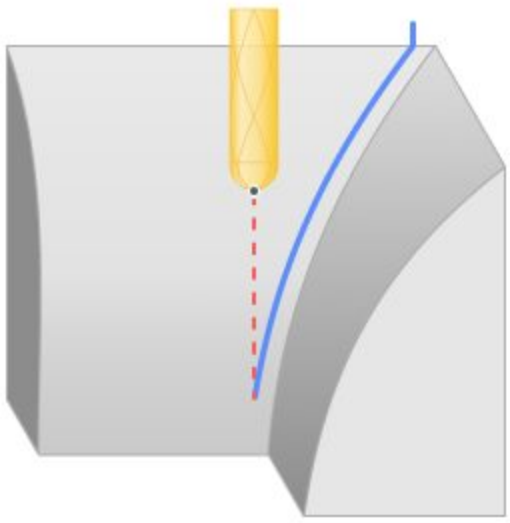
Controls the sequence of toolpath passes during the surface machining.

Machining Slopes:

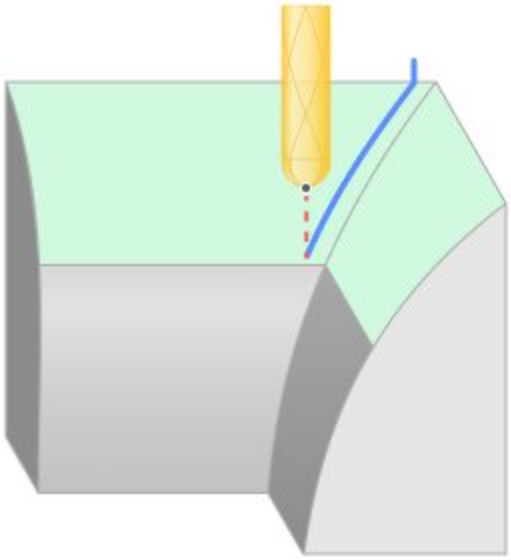
Determines areas for machining based on the gradient of the corners generatrix lines.

▼ Click here to expand..

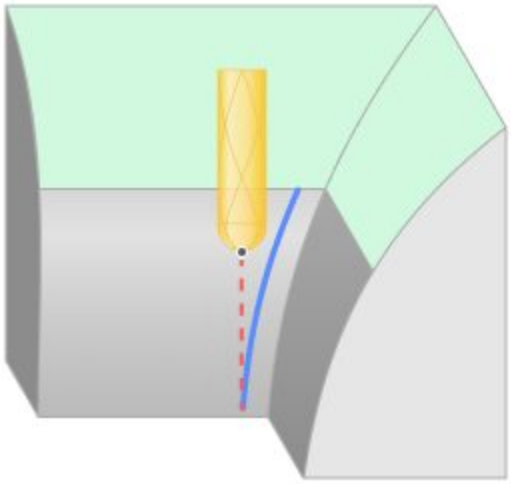
All. The system p rocesses both shallow and steep areas.



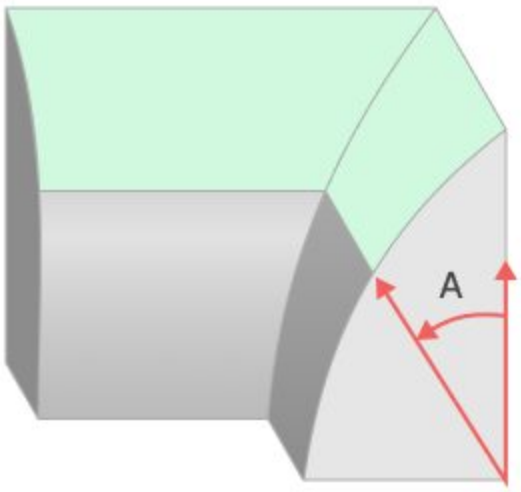
Shallow. The system p rocesses only the shallow areas.



Steep. The system p rocesses only the steep areas.



Steep/shallow split angle. The angle that separates the shallow from the steep areas.



Machining Levels:

It defines the range along tool axis for the machining. Details. Specify the Bottom Level as in the Waterline Roughing operation. Details

5 axis conversion:

Controls tool axis orientation. Details. The principal parameters of this function are the same as in the 5D Surfacing operation. Details

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

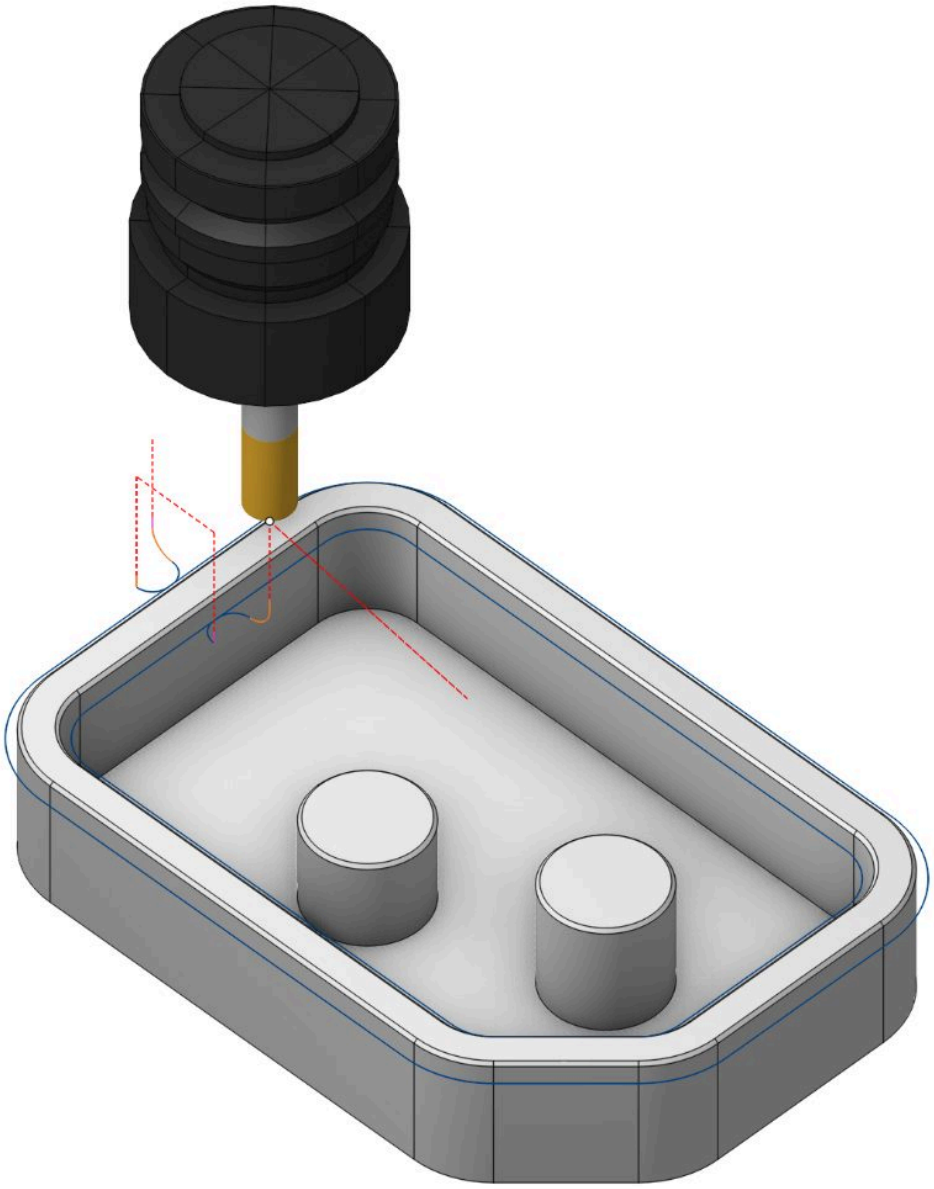
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Chamfering



Application area:

The operation generates a toolpath for chamfer removal using cylindrical end, conical, and spherical mills. It is most commonly used as a final step to form the chamfers and fillets specified in the 3D model, to blunt sharp edges, or to remove burrs along the edges of the part.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Add sharp edge. This feature enables the creation of a chamfer removal toolpath based on the edges of the 3D model. When selecting a surface instead of edges, it instantly adds all the surface's sharp edges into the Job assignment.

Add face. This feature enables processing of existing chamfers on a 3D model by selecting surfaces designed as chamfers.

Recognize. This feature enables automatic recognition and addition of chamfers into the Job assignment. The system classifies recognized elements into four types: sharp edges, CAD chamfers, fillets, and hole chamfers.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

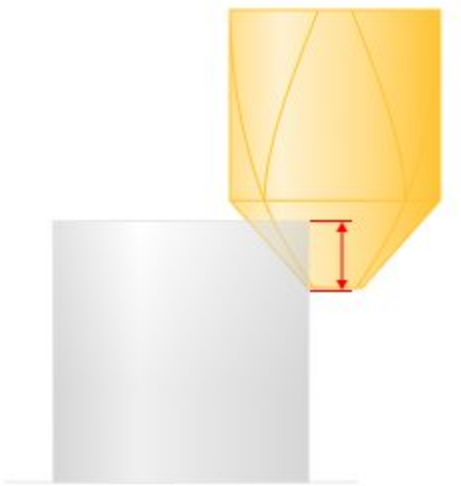
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

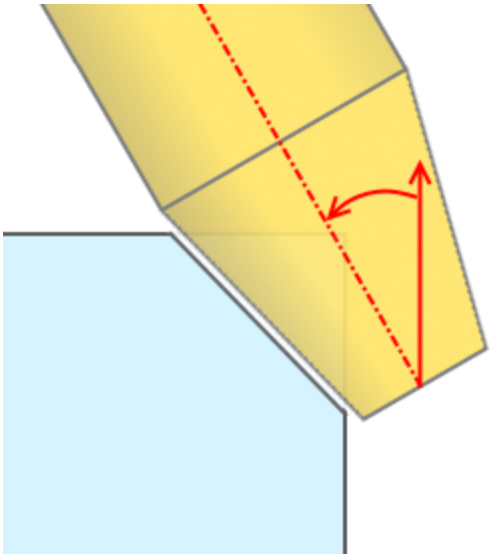
Strategy.

This is the set of parameters that define the geometric features of the machining process.

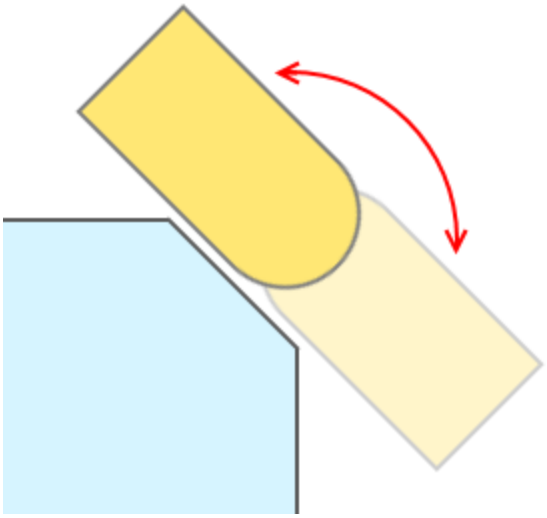
▼ Click here to expand..
Chamfer depth. Specifies the blunting depth for sharp edges. If a "CAD Chamfer" is being processed, the tool follows the chamfer's surface.



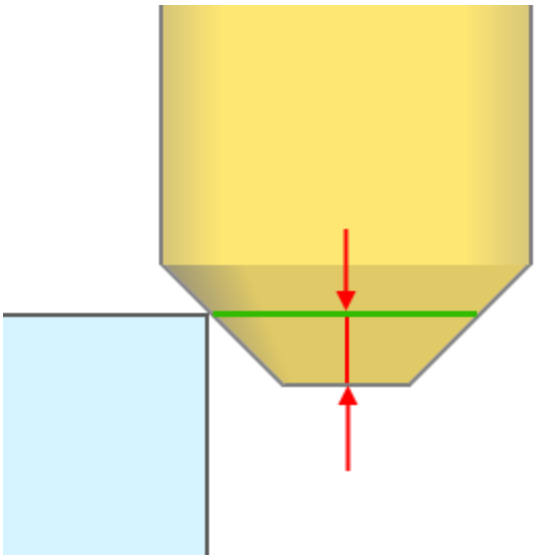
Incline tool. This feature allows tilting the tool to remove the chamfer with the tool's side edge.



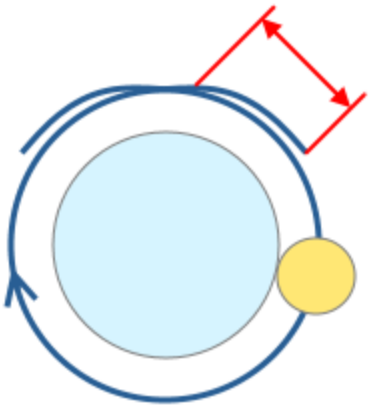
Inverse tool axis direction. This parameter enables the inversion of the tool axis vector direction. You can also reach this objective by altering the side through the direction arrow in the graphical window, thus the tool axis direction shifts for every curve on its own.



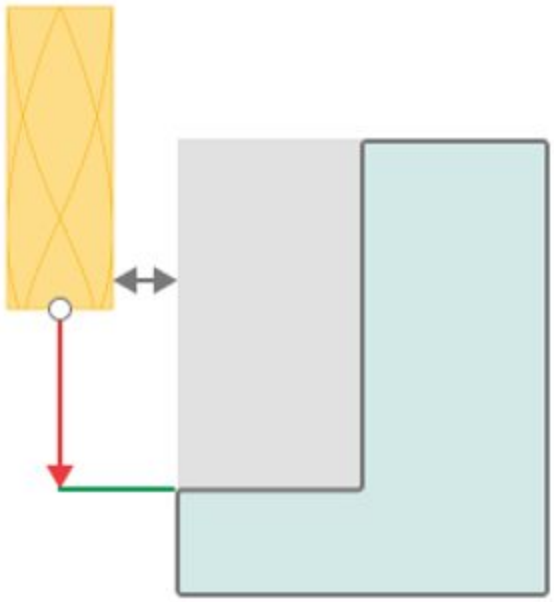
Tool contact point. This is the point on the tool that should primarily engage with the geometric elements specified in the operation's Job assignment. The contact point is determined by the distance from the tool tip. User can also adjust it interactively in the graphical window.



Overlap. This feature enables toolpath extension in closed chamfers to prevent burrs at the junction.



Safe distance. Specifies the least acceptable distance between the tool's non-cutting section and the workpiece to avoid unintended contact with the workpiece elements. It matters in chamfering operations within recesses, pockets, and similar areas.



Milling type.

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

▼ Click here to expand..

Curve direction. The tool's feed direction matches the Job assignment vector direction.

Sorting.

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..

Idle moves minimization. The order of the profiles machining is sorted to minimize the idle motions.



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

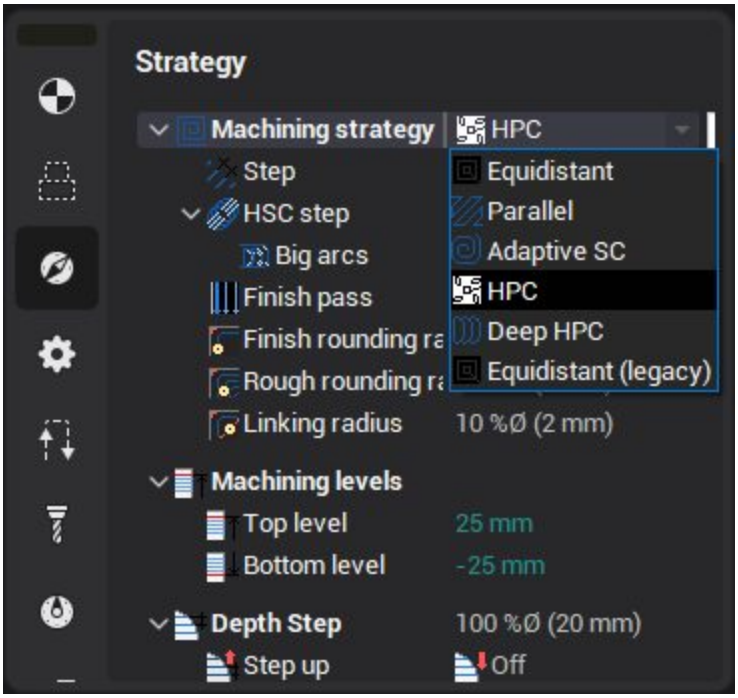
High performance cutting (HPC)

The high performance cutting (CAM system HPC) strategy is designed for the efficient removing of material in the open and closed pockets.

This strategy is available in the following operations:

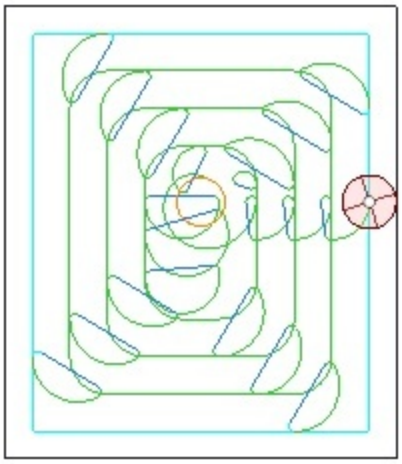
- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

The strategy is enabled by selecting the corresponding option in the Machining strategy drop-down:



All the following options are available when the HPC strategy is selected.

New trochoidal arcs

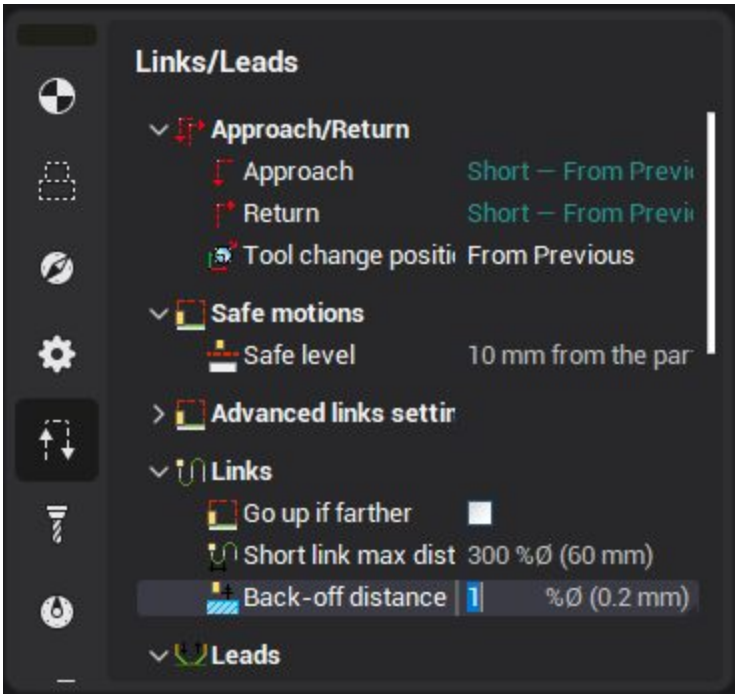


Tool path length has been reduced by up to 20% compared to the legacy version of the program.

New trochoidal arcs avoid tool overload without requiring feed rate reduction.

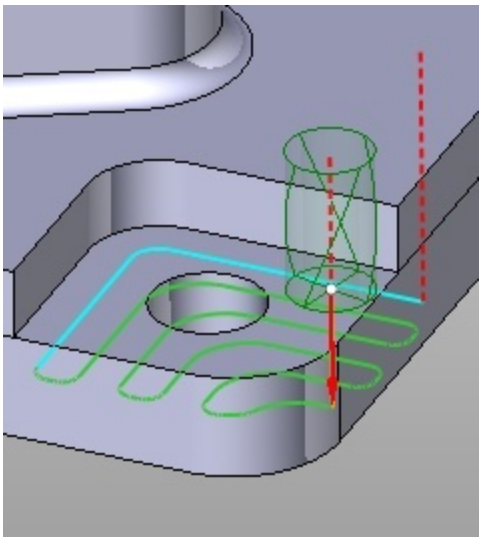
It makes for a much smoother machining process.

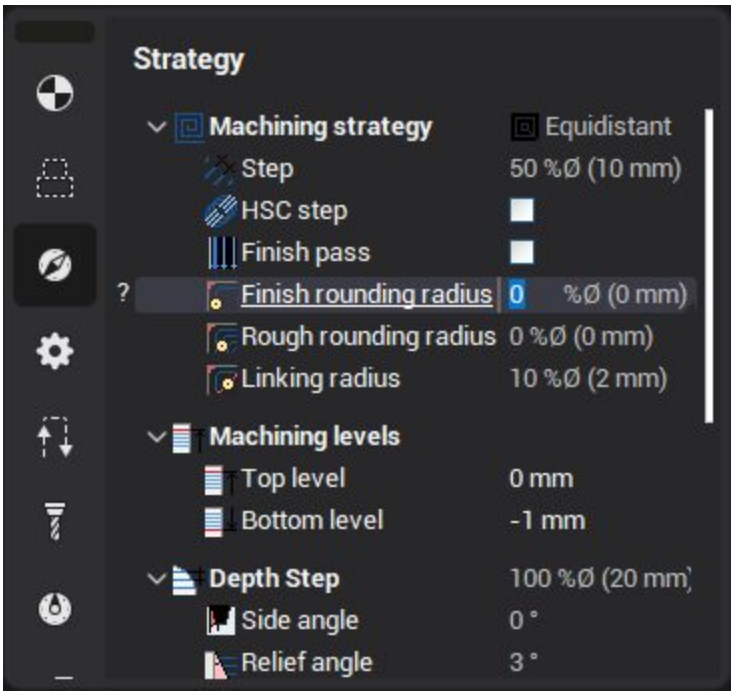
Back-off distance parameter



The tool can be lifted above the already machined surface when it moves to the next trochoidal arc start position.

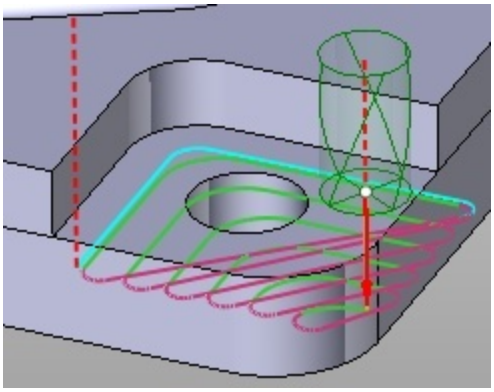
Rounded links in zigzag mode





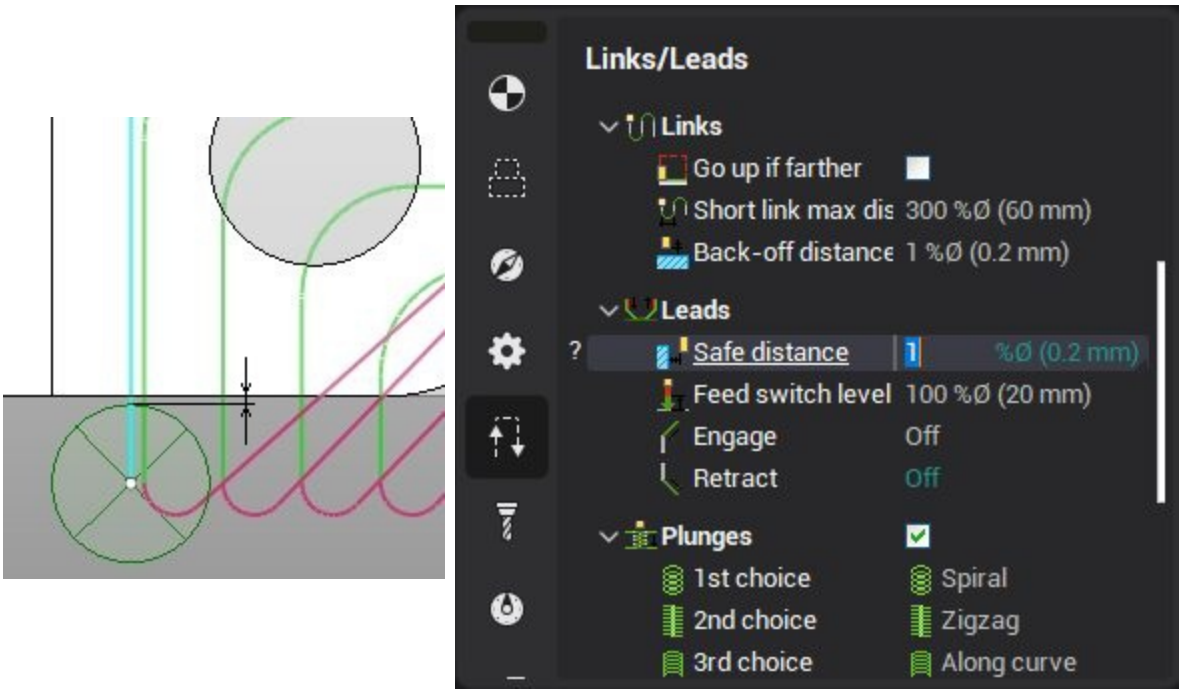
The 'Finish rounding radius', 'Rough rounding radius' and 'Linking radius' value is used for rounding of the links.

Links on the same Z-level



In the climb and conventional mode, the tool goes directly to the next path without retraction to the safe level. If a rapid motion is performed over an already machined surface, then the "Tool back-off distance" is used. "Idle radius" is also used to make the motion smooth.

Safe distance

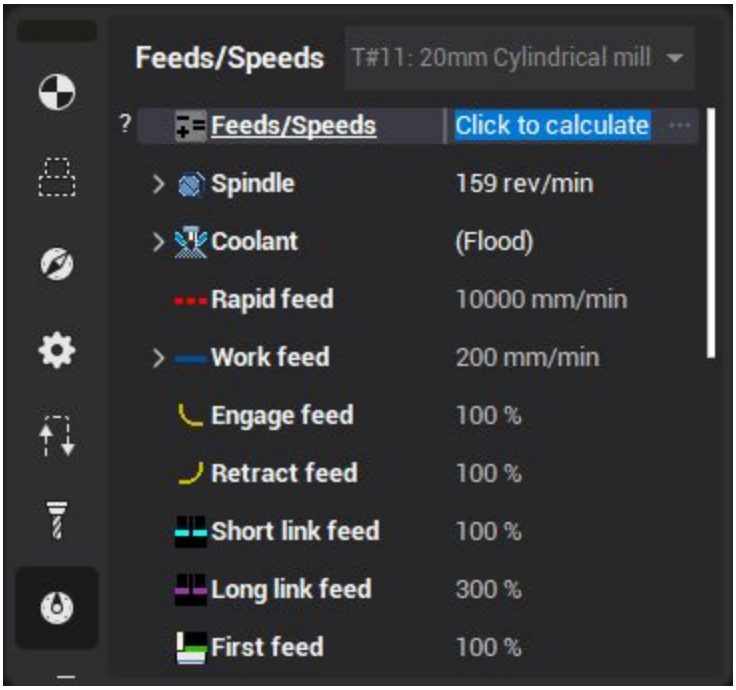


Safe distance is used to move the tool down/up from/to the safe surface.

The vertical motion is performed at this distance from the workpiece. So there is no longer the need to enable the approaches/retractions to exclude the rapid feed collisions.

If you use a pre-drilled hole to plunge when roughing, the pre-drill tool diameter must be greater than the mill tool diameter by at least double the safe distance amount, otherwise the pre-drilled holes will not be detected.

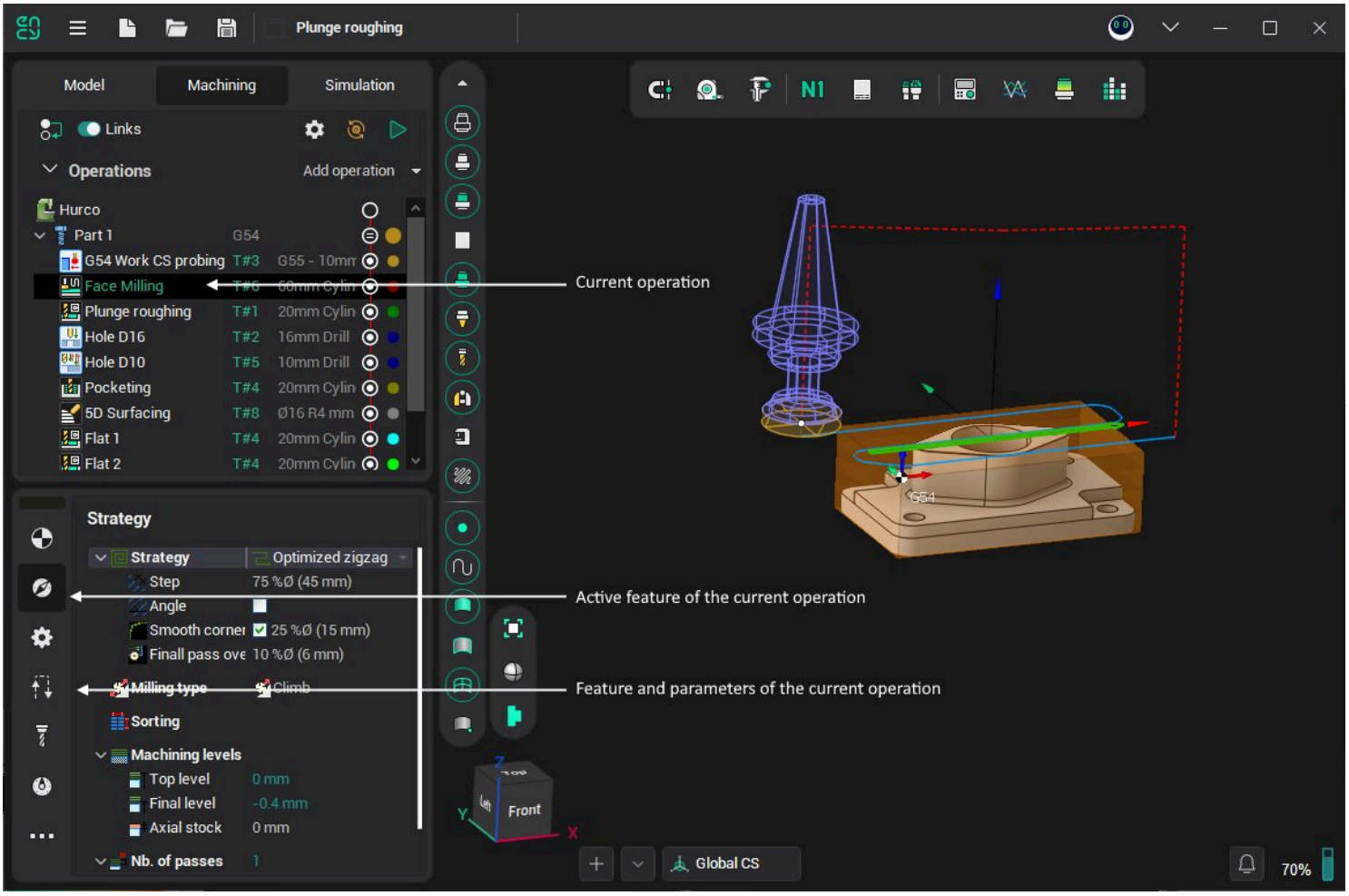
Rapid feed links



The link moves can be calculated using either the next feed or the return feed values. If the link length is less than the 'short link' distance, then the 'next feed' value is used, else the 'return feed' value is used. The return feed is set to 300% of the work feed by default, which is a non-cutting feed. If cutting is detected during a 'return feed' move when simulated, this move will be marked with an error.

Operations setup

The parameters of an operation define what is to be machined and the way it is to be machined. Selecting a parameter node inside the operation tree changes the bottom side of the tab to display the tools used to define and edit the parameter properties.



Using design and machining features in Job Assignment

This section describes using design and machining features in Job Assignment, including creating operations for selected geometry and drawing spline curves.

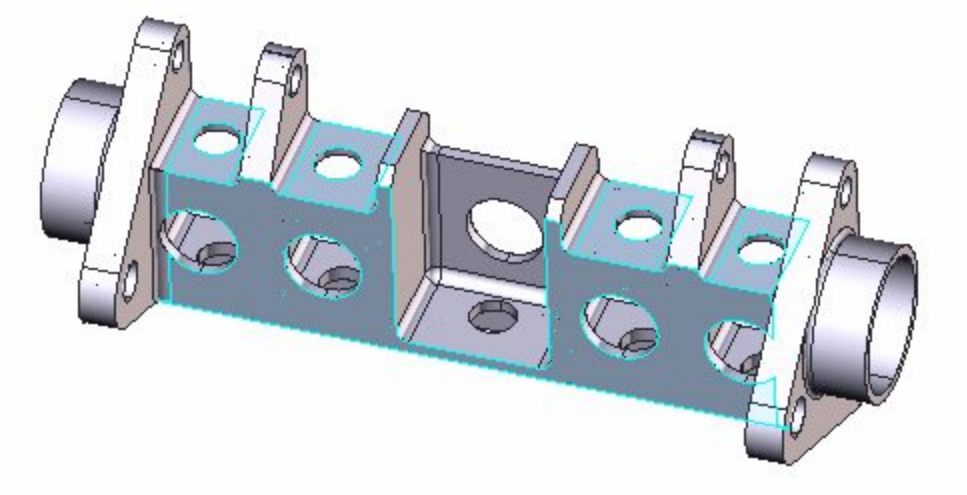
Creating operations for selected features

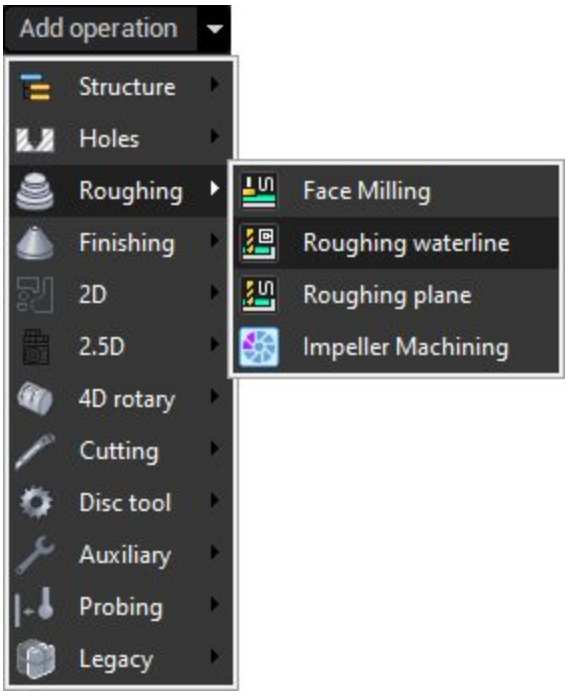
Creating technology for a prismatic part consisting of many simple shapes is a time consuming process. To simplify this task CAM system offers the possibility to create operations for selected geometry. This is very straightforward. At first you select what you want to machine in the graphic view, and than you create the operation which will machine the selected geometry. Based on the operation type created CAM system automatically recognizes machining features like pockets and holes in the selected geometry and puts them into the Job Assignment of the just created operation. After that CAM system positions the machine tool axes to alight the live tool axis with the Z direction of features local coordinate system. Than it selects the most appropriate tool and parameters of the operation based on the attributes of the machined features.

E.g. if you double click on a hole in the part, CAM system automatically measures it and highlights all the holes with the same diameter. Now if you create a hole machining operation CAM system automatically puts all the selected holes into the Job Assignment of the created operation and sets the drill diameter equal to the diameter of the holes. If you create a `3d Hole machining operation` and selected holes are placed in different planes, CAM system will create not one hole machining operation but several operations according to the number of planes in which holes are positioned.

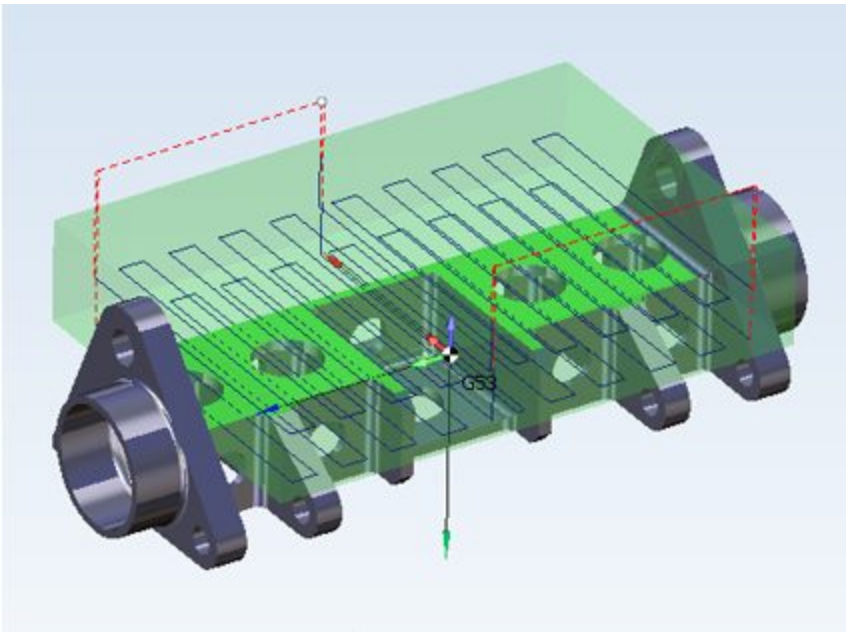
Example of fast creating pocketing operation:

We will machine the part depicted below on a 3+2 milling machine. At first we want to rough open pockets. Just pick the bottom faces of the pockets to be machined at the screen and select the Roughing waterline operation from the Create operation drop down list.



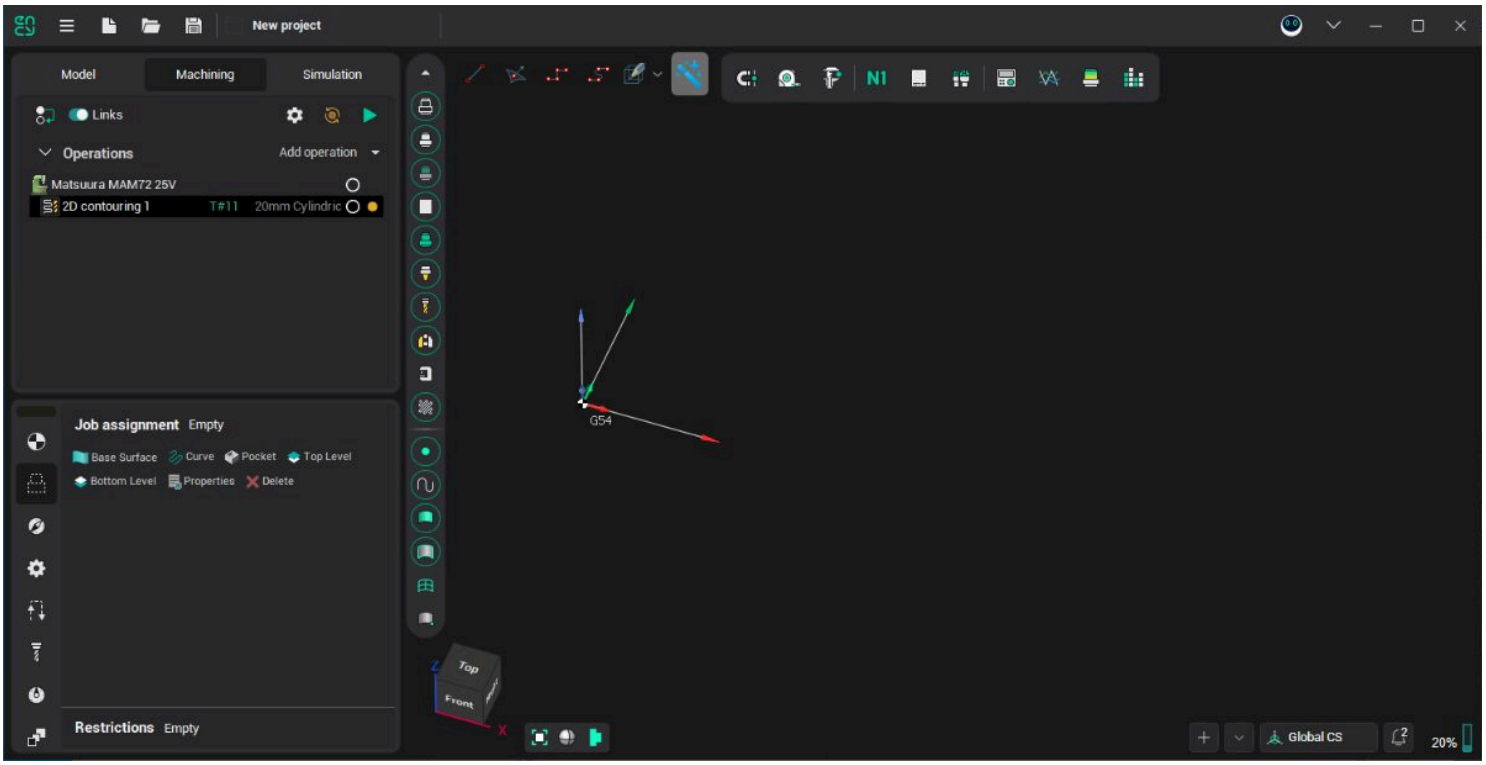


CAM system automatically creates two roughing operations with pockets in the job assignment. The generated toolpath can be seen at the picture below:



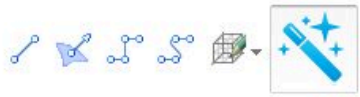
Spline curves drawing

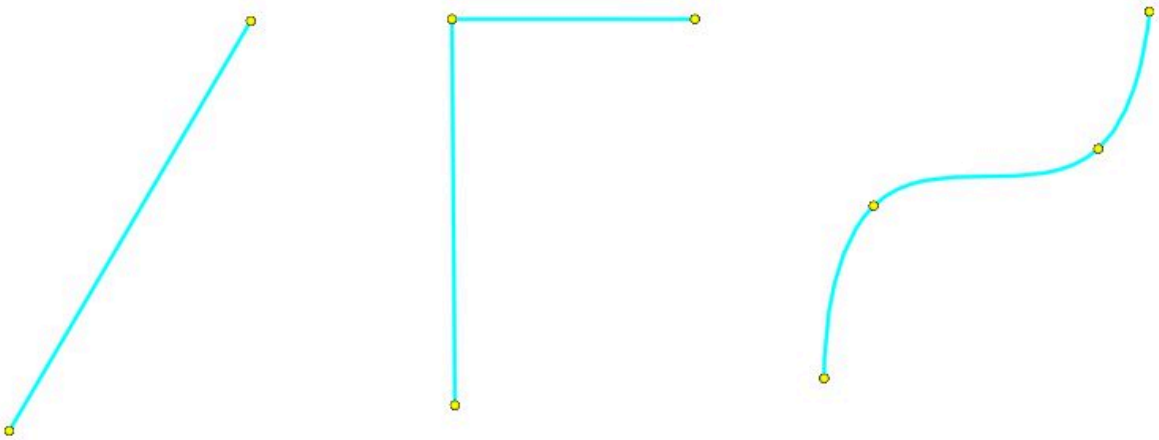
This feature is experimental, and still under the development. Live curve drawing toolbar added to the graphical window, which is visible while Technology mode is active.

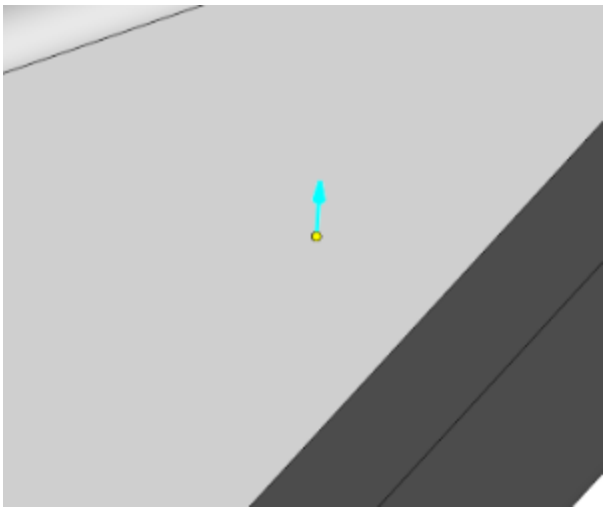


It allows to draw auxiliary curves:

- straight lines
- polygons
- splines.





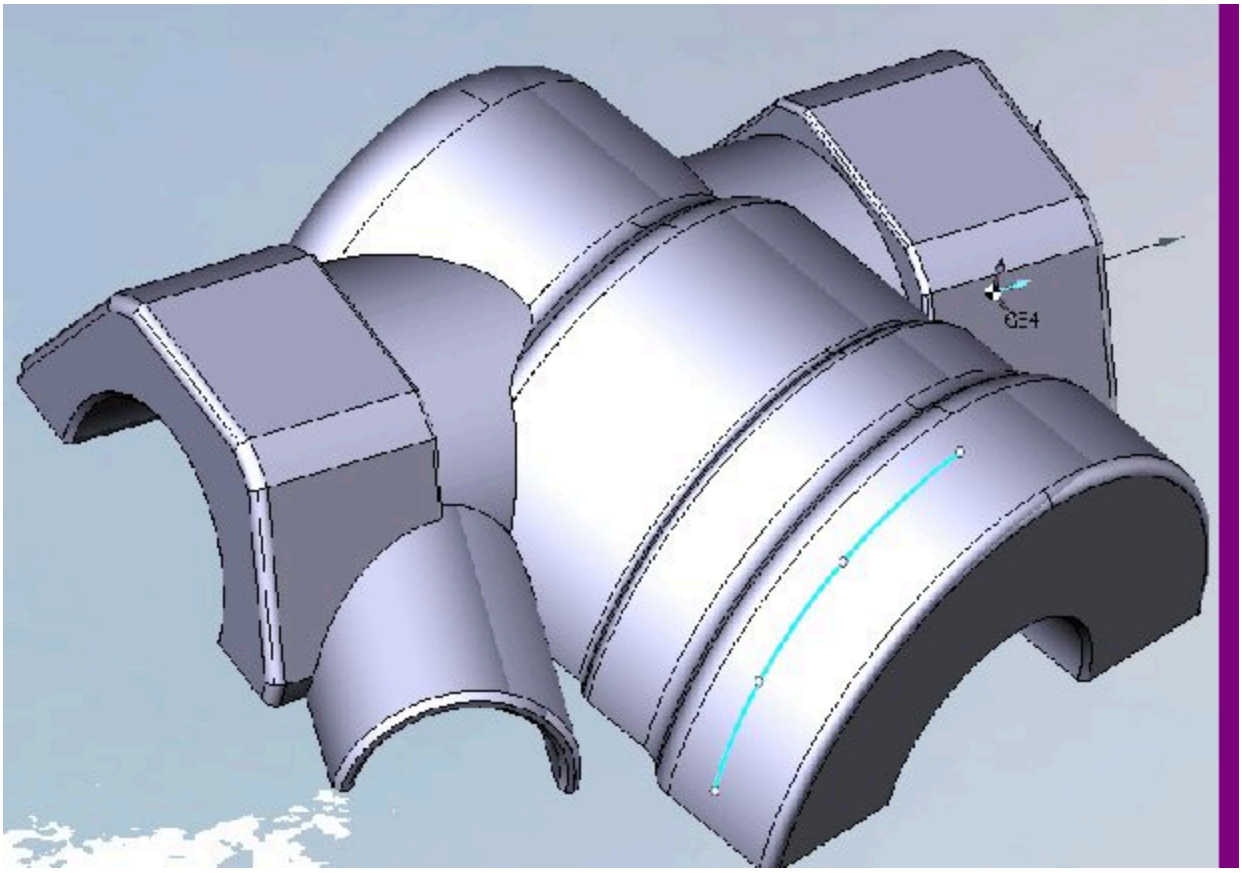


Directly on the surfaces of objects that are visible on the screen. Nodes of these curves can be dragged, deleted and added just by simple mouse click.

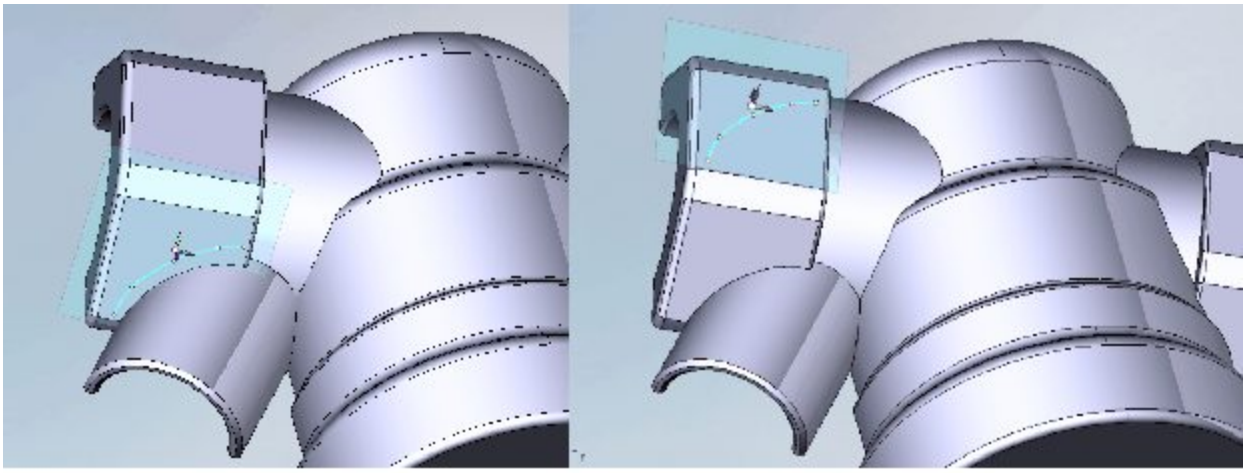
There are 3 drawing modes:

1. Free. Points are selected by the next rule
2. In case, there is no snap object under the cursor, and it is the 1st point. The point will have undefined Z.
3. In case, there is no snap object under the cursor, and it isn't the 1st point. The point will have the same Z as previous point (we will drawing at plane);
4. In case we snap surface under cursor surface's, the nearest point will be selected.

5. On surface. Points will be selected from surface only;



6. On plane. Points will be selected from plane only. All drawing curves are connected to this plane and will transform with plane transformation too (translate or rotate plane).



Job assignment for the 2.5D machining operations

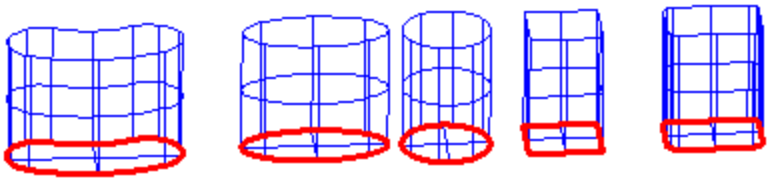
For the creation of CNC programs for 2/2.5D machining consisting of flat areas, pockets, covers etc., it is not always best to construct a 3D model. On the other hand, however, it is handy to be able visualize the depth of the geometry. CAM system allows the construction of such models using 2D contours and automatically displays the volume model.

The Volume model is formed from 2D contours located at different heights, limited by closed contours and the walls between them. Open (unclosed) contours and points can also be used when constructing a visual model; for further reference, see below.

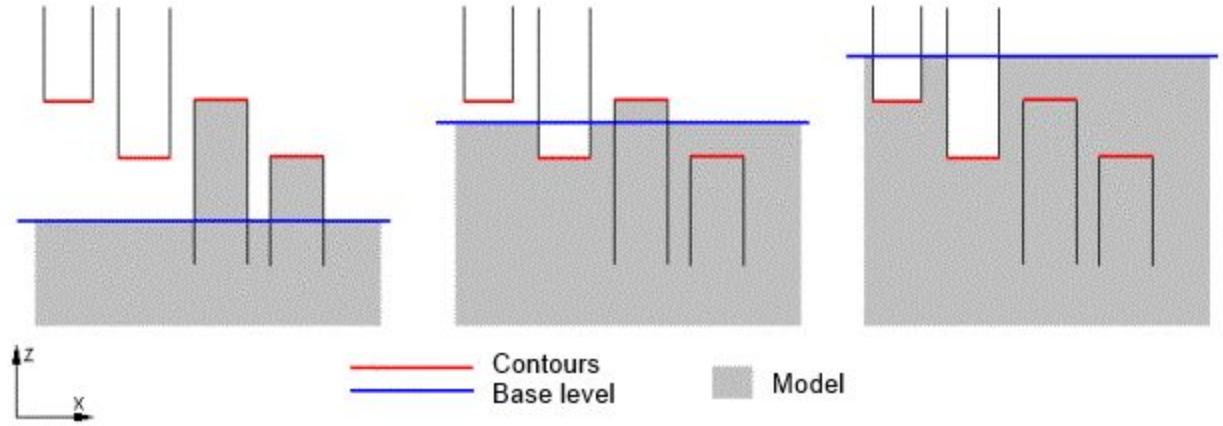
The '3D model' is constructed from 2D (flat) contours lying on different levels. There are two methods to add such areas: a 'cover' – "adds" material from the very bottom to the area level, and a 'hole' – removes the entire material from the top to the area level. This means that the 'side wall' for a cover exists below the 'area' level, and for a hole – above the area level. In order to construct a model you can also define the Base level, the space below which represents a body of infinite depth from which, by placing 'holes' the user can obtain a model.

All level values are defined by the absolute Z coordinate of the current coordinate system.

For example, imagine a situation when someone is building figures from sand. The base level is comparable to the sand level, and the construction tools are the cans with the bottom form, defined by contours. The contours can be different shapes, for example as these shown below.



By using these forms the user can either press out holes, or by filling it with sand and turning it over, construct covers. The closed end of the can is the start; the open end extends endlessly (endlessly down for covers and up for holes).

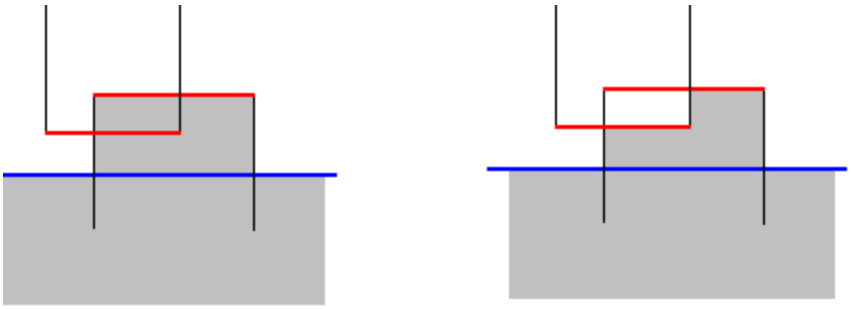


Of course, if hole-forms are located in empty space (or above the base level without other constructions), then they cannot press anything out, Likewise, cover-forms, which are located inside material (or below the base level without holes) cannot fill anything. Whereas, cover-forms located above the sand level always fill a cover, and hole-forms always press out a hole in the sand, if it exists.

When creating a figure from sand, the creation sequence is important. For example, in order to obtain a step cover, one should first create an integral cover, and then press out a step. If one tries to press out a step in the emptiness first, and then fill a cover, then the correct result will not be achieved!

The examples above show the two different results.

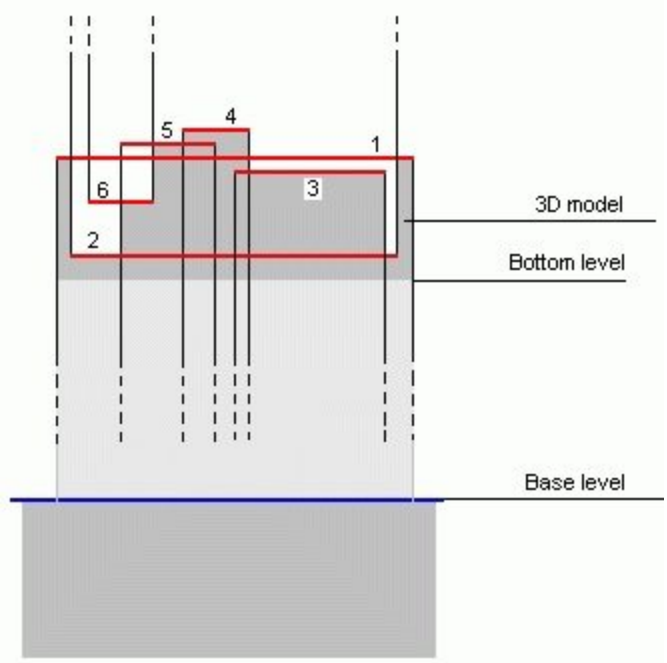
Left – hole created first, then the cover. Right – cover created first, then the hole.



It is obvious that in the first case the hole did not reach the "sand" level, i.e. there was nothing to press out, and so the cover was untouched. In the second case, the cover was created, and then the hole pressed out a 'part' of the cover.

By default, the base level is located endlessly below the zero plane of the system. Most models can be built without the base level. For example, the user needs to create a cover for the outer border of the model at the required level, the subsequent construction of the model will be performed inside that cover. When drawing, the 3D model will not be shown below the level defined by the Bottom level parameter of the operation (defined on the Parameters page), any part of the model located below that level will not affect the machining operation.

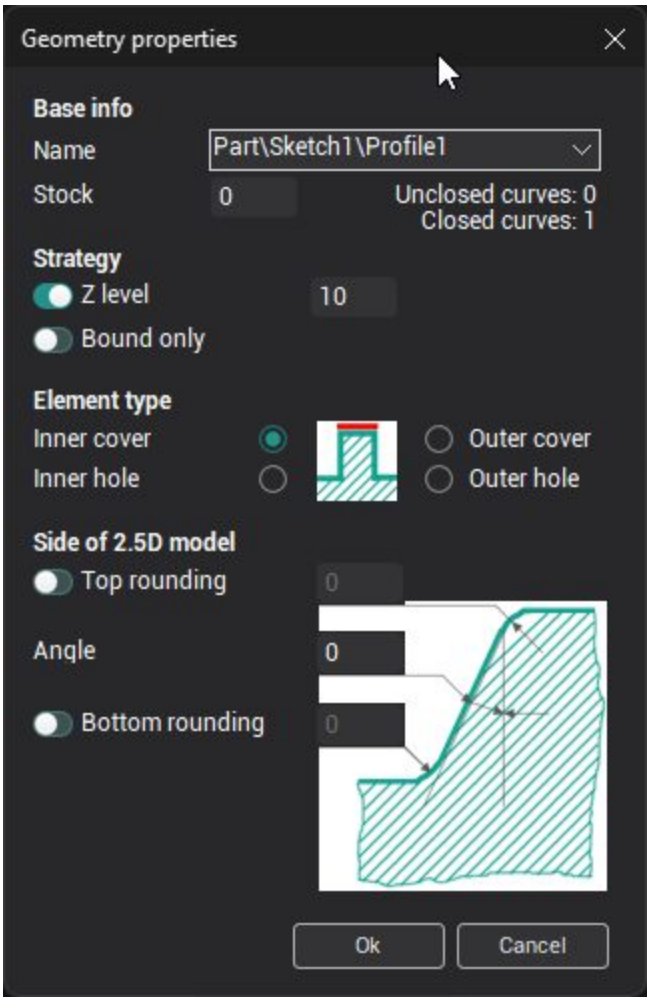
An example of such model is shown on the picture.



The numbers define the sequence of actions:

1. creates a cover for the outer profile of the model;
2. presses out a hole;
3. creates a cover inside that hole;
4. creates a cover higher than the level of the first cover;
5. another cover, which intersects with the one created in article 4;
6. presses out a hole in the last cover.

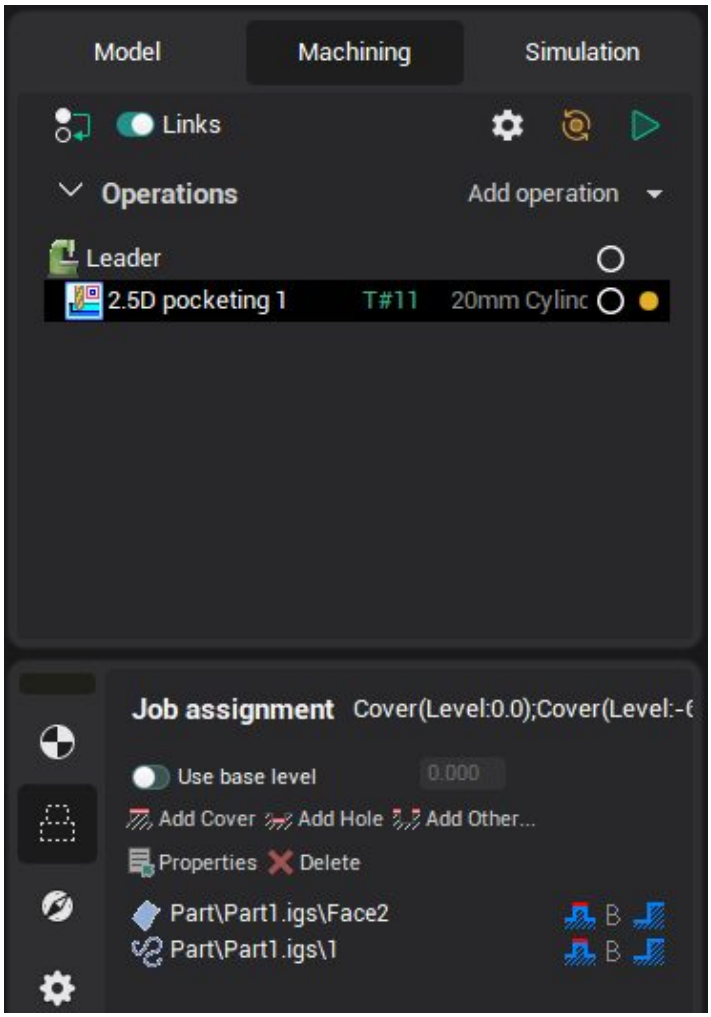
The properties window can be used to change the item parameters. The window can be opened by double click on the item or from the pop-up menu. The dialog is shown below:



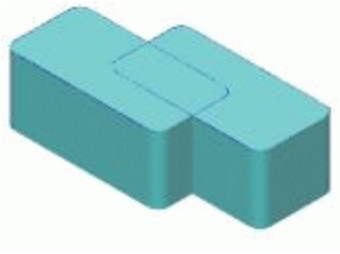
It is possible to set the wall shape in the window also. The angle defines the wall slope in degrees. The top and bottom fillets can be defined also.

Let's practice some techniques for the visual model construction.

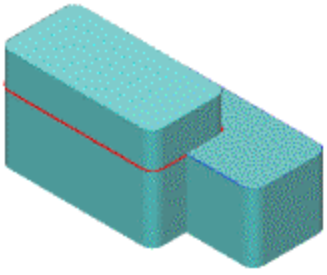
In the "2D Geometry" mode construct two rectangular intersecting contours; Switch to the "Machining" tab and select the "2.5D area pocketing" operation; Open the "Model" window and add the two contours into the machining list of that operation. All constructions of a visual model are performed by using the commands and parameters available on the panel shown to the right. As was said above, any contour can form both a cover and a hole. In many cases the use of a base level is not required. The user can form the visual model by using an outer contour that forms the body, and adding or removing pockets (holes) or covers; Activate the first contour, set the type to "Cover", and assign the contour level to "0". If the contour level has not been defined, then it is considered, that its level coincides with the maximum Z coordinate of that contour; Activate the second contour and set the same parameters as they are for the first one. The bottom level of the model is defined by the "Bottom level" parameter in the "Parameters" window; Assign its value equal to -20.

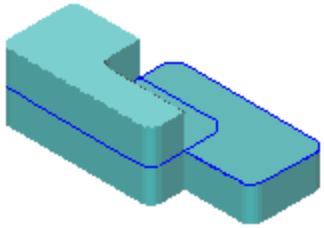
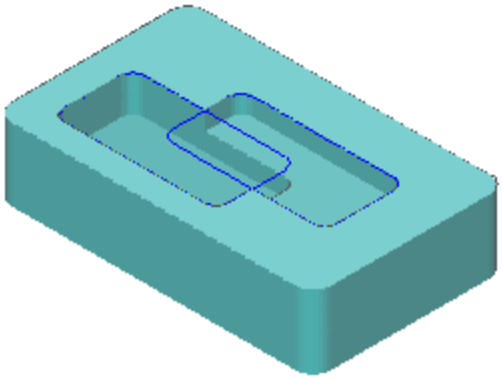


Put a tick in the Preview box. The  button must be pressed. There should be a model similar to that shown in this picture: We can represent parameters and their conditions in a chart: 1. Contour 1 cover level 0 2. Contour 2 cover level 0 Thus, we have created one variant. Using both contours as covers, we have created a model. Both contours lie on the zero level, the bottom part of the model is at -20;



The next step – selects the first contour and in the "Level" field set the value equal to 10. As the result, the contour will be located on the level 10; the visual model will change accordingly. 1. Contour 1 Cover level 10 2. Contour 2 Cover level 0

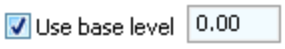
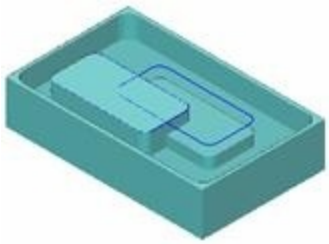


The same contour can be used several times. 1. Contour 1 Cover level 10 2. Contour 2 hole level 0 3. Contour 2 Cover level 0 The result is shown on the picture	
Add one more contour with number 3 to the list 1. Contour 3 cover level 5 2. Contour 1 hole level 0 3. Contour 2 hole level -10 In this case, the contour 3 had first pulled the cover to level 5, then contour 2 pressed out the hole to level 0 and finally, contour 1 pressed the hole to level -10.	

Some examples for the use of Base level.

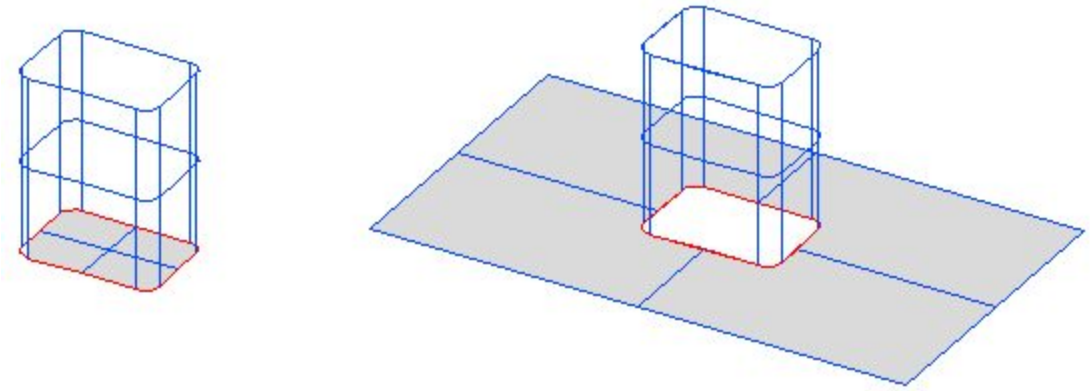
Above we used the analogy of the Base level as the equivalent to the sand surface from which different forms can be built, i.e. starting from that level, we always have the possibility to press out holes.

Example for the use of Base level.

Activate the Base level mode  Make the base level 0; Create a list of contours with these parameters: 1. Contour 3 hole level -10 2. Contour 1 cover level 0 3. Contour 2 hole level -5 The result should be similar to the one shown.	

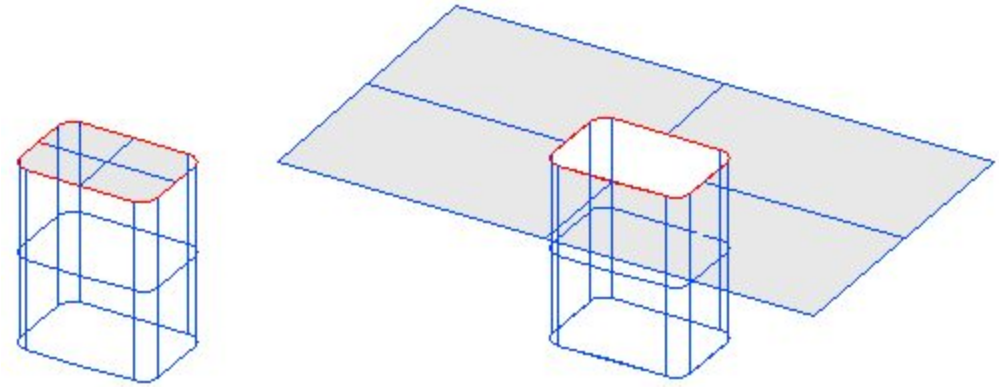
Model example with use Outside parameter.

Until now we have been dealing with pressing holes and pulling covers with the Inside parameter. Now we shall learn the application of the Outside parameter. Once again, if we make a comparison to the construction of sand figures, i.e. represent contours as shapes for working with sand, then, when we select the Outside parameter, we invert the work area of the tool. The pictures below show the different forms created by the same contour using the Inside (left) and Outside (right) modes.



Inside Outside

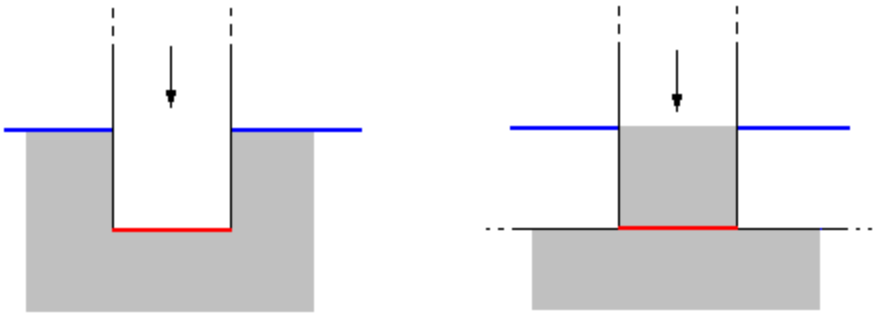
Forms for pressing out holes



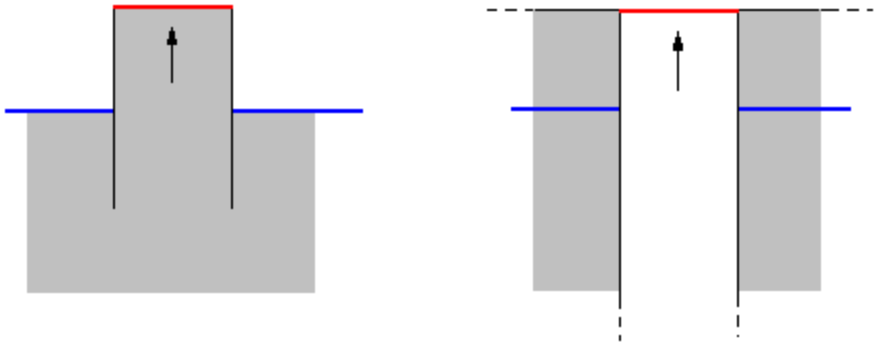
Inside Outside

Forms for pulling covers

See what happens when switching between modes



In the left picture there is the pressing of a hole by a contour set as Inside, on the right the pressing of a hole by a contour set as Outside.



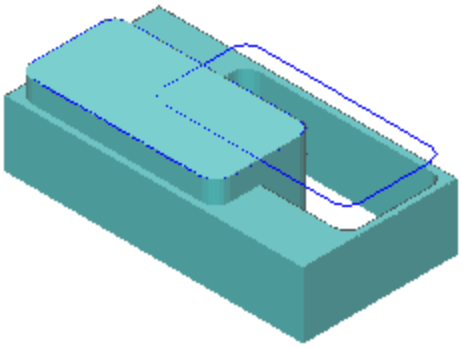
Cover pulling is shown in this picture. Left is Inside parameter, right is Outside.

Here is an example of using Outside parameter.

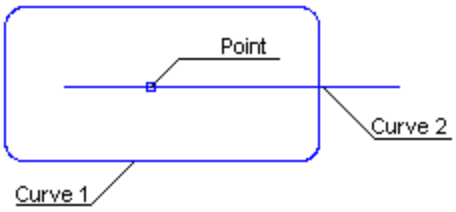
1. Create a list of contours with parameters defined as below;

1.Contour 1	Cover	level 0	
2.Contour 1	Cover	level -5	Outside

The result should be similar to the one shown below. Besides using closed contours, points and open (unclosed) contours can be used for the creation of a model.



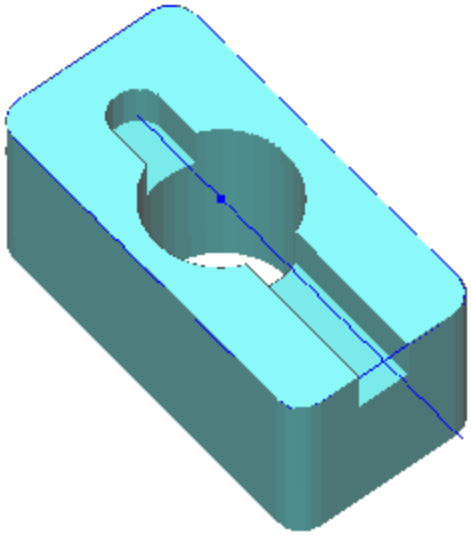
1. In 2D geometry create one closed contour, one open and one point, similar to the picture below:



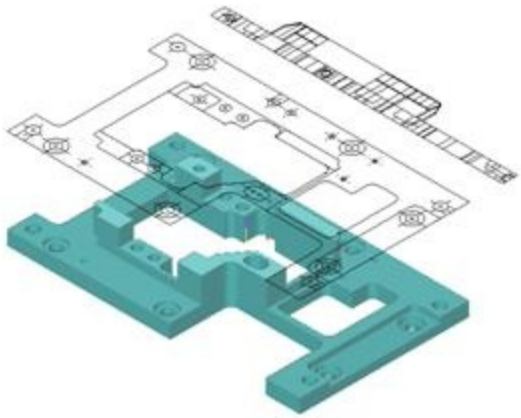
1. Create a list of elements with the parameters defined below:

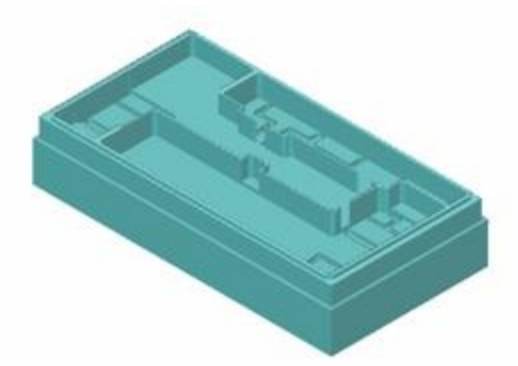
1. Contour 1	Cover	level 0	
2. Point 1	Hole	level -20	additional stock 8
3. Contour 2	Hole	level -5	additional stock 3

The result should be similar to the picture below:



Examples of visual model construction for 2.5D machining.





Operations for 2/2.5-axes milling

The peculiarity of 2D and 2,5D processing operations – non obligation of presence 3D model of the part. The work task, and also the part, the billet and rigging may be formed only because of such curves, and in some cases because of points. The use of surfaces under the certain circumstances also possible. For example, the operation of Hole machining on adding the holes it assumes in work task the use of cylindrical surface. Or in the operations of processing curves in the work task is possible to indicate "walls" – i.e. such surfaces which are perpendicular to the plane of the current operation of machining. Besides, in all the operations, where the setting of curves, the edges of 3d model may be used as curves.

2.5D machining operations

These operations are designed for creating NC programs for models, which have pockets, ledges, flat areas etc., for which it is not always efficient to construct a surface/solid model.

The visual model is formed from flat areas limited by closed profiles, located at different heights that have walls between them. Open (unclosed) profiles and points can also be used in the construction of the visual model.

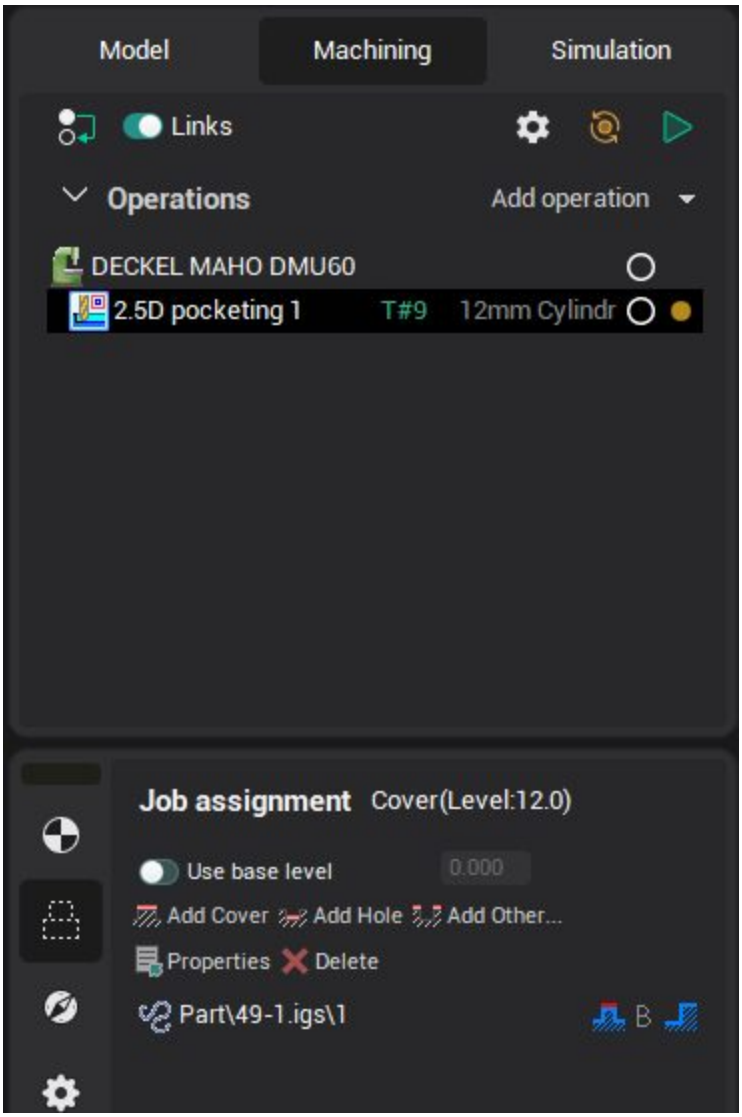
In the 2.5D operations, the system allows the user to visually create the geometry of a model by using flat profiles. If the parameters of a profile are changed, the displayed model is updated automatically. The operation processes the list, which consists of an arbitrary number of profiles and curve projections. For every object there is an individual machining method.

If the user defines a workpiece or restricted areas, then only those areas of the defined profiles that lie inside the workpiece and outside the restricted areas will be machined. And if no workpiece or restricted areas are defined, then the system will perform machining of all the defined profiles without any limitations.

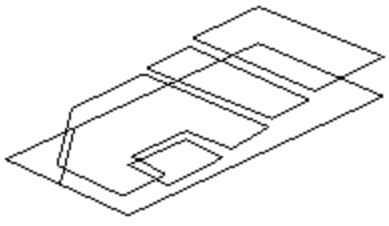
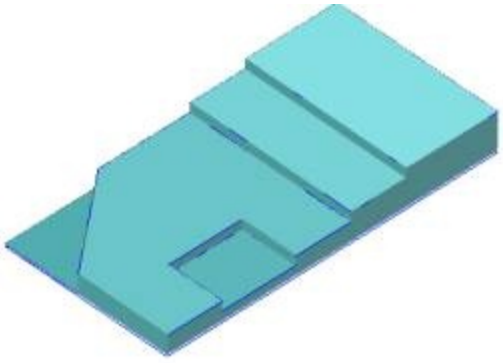
Machining is performed by a series of horizontal passes of the tool. The passes differ from each other only by the depth at which they are located. The number of such passes and their Z depth depend on the machining levels and the step defined on the parameters page in the Operation parameters window.

In the same window, the user can set up the machining tolerance and the stock.

All 2.5D operations use 2D curves to define the model. Formation of the model is performed in the Model window, where one can assign parameters for either a group of elements or a single element.



The system dynamically displays the 3D model in the graphic window, updating as any alterations to the parameters for elements are made in the Model window.

	
Curves that assign the model	Created model

Operations for the 3-axes milling

The peculiarity of the operation for 3-coordinates processing is the opportunity of simultaneous displacement on three axles direction of cutting tool relatively to the part. For setting these operations in CAM system it is required the presence of three-dimensional models of processing parts.

Operations for 4-axes and 5-axes milling

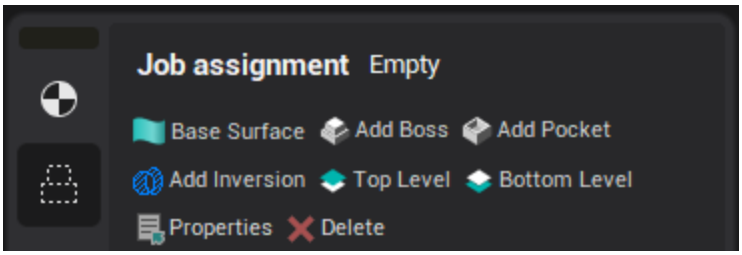
The modern milling machines and production centers allows to make the machining using five axes simultaneously. The machines design can be different but the principle of 5-axis machining is as follows: the positional relationship between the part and tool is describes by three linear and two angular dimensions. CAM system can generate the five axis toolpath virtually for any machine configuration. For more information refer to the 5 axis machining topic.

Next are the types of multi-axes machining.

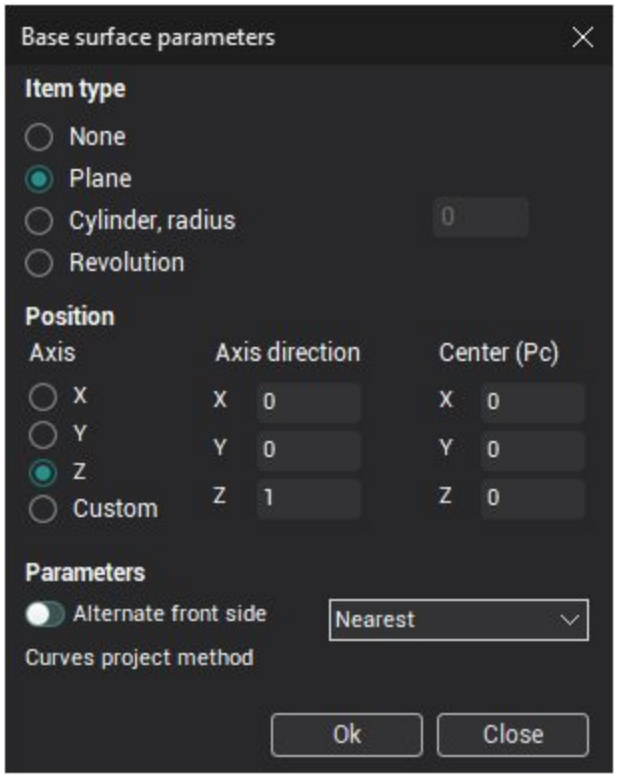
1. Indexed five axes milling. In this case the angular dimensions are fixed in required position at the start of machining. After that the simple three axes milling is performed with using of three linear dimensions.
2. Rotary machining with using of the cylindrical interpolation. Generally, this kind of machining imply the simultaneous motions by the three axes: two linear and one rotary ones. The machining is performed on the cylindrical surface and the machined elements are defined by the unrolled planar curves.
3. Continues four-axes milling of the solids of revolution can be created with using of the existent solid model or unrolled planar curves and Engrave or Pocketing operation. It is necessary to define the base surface as the cylinder of any solid of revolution. Particularly this way allows to machine the pocket on cylinders.
4. Continues five-axes machining of the curves and surfaces is performed with using of the 5D contouring operation.
5. Continues four-axes milling is of the free-form solids and surfaces is performed with using of the Rotary machining operation.
6. Holes machining performs the machining of the holes with the free axes direction.

4-axis milling with engraving and pocketing operations

Engraving and pocketing operations allow to define the job assignment as the set of the closed contours. Everyone of it defines the Pocket or the Boss of the model.



Besides, it is possible to set the Base surface in the current operations, on which one of the defined contours will be projected. The base surface is a plane, cylinder or the solid of revolution.



The contours of the job assignment can define the boss and pockets directly on the base surface or to be an unrolled curves of it. In contrast to the Cylindrical interpolation the machine can not have the rotary axis that rotates the workpiece.

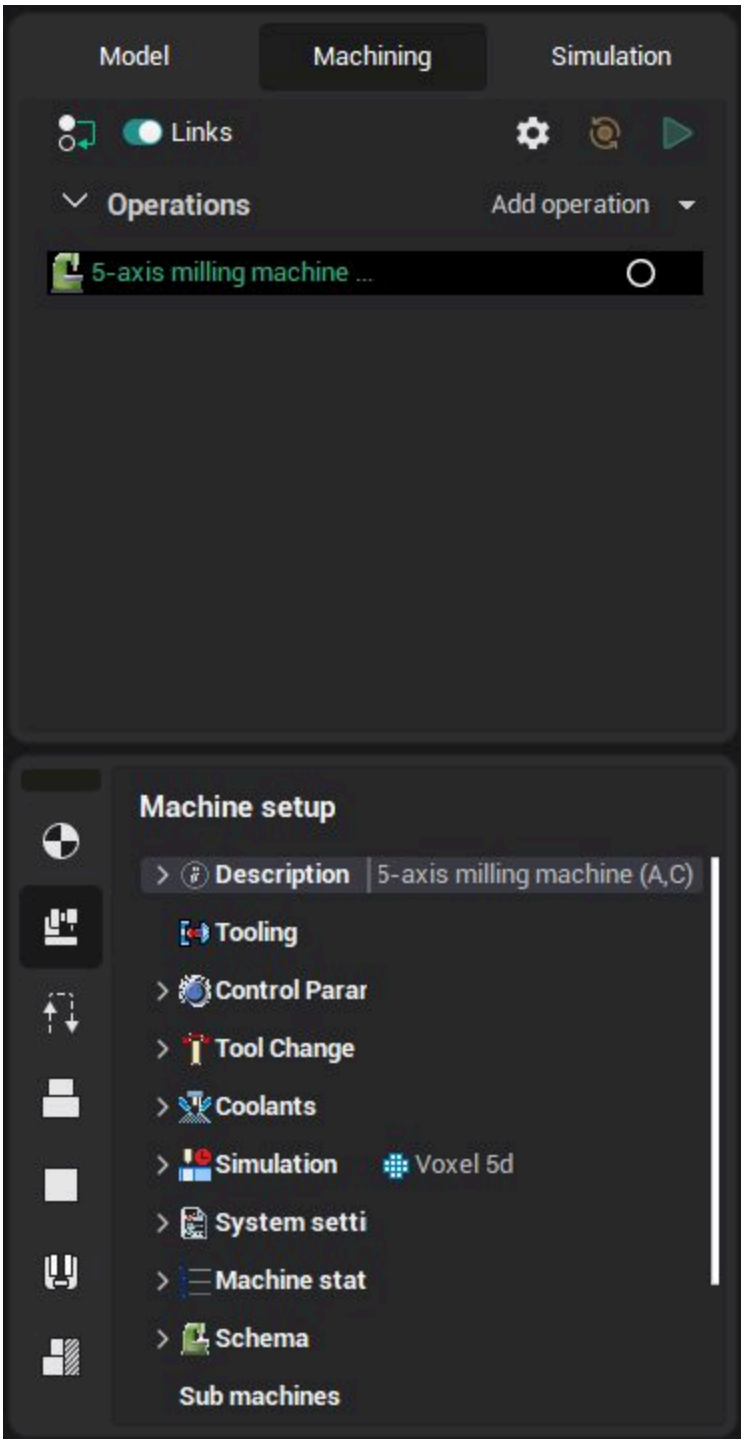
5 axis machining

CAM system can generate toolpath for 5 axis milling machines based on the machine kinematics and the parameters of the used control. CAM system virtually supports any machine-control configuration. The generated toolpath contains commands for positioning the XYZ axes of the machine (GOTO, CIRCLE, etc) and the commands for positioning the machine rotary axes (MULTIGOTO). Generally for the most controls the rotary coordinates are the actual positions of the corresponding machine axes, while the XYZ coordinates are the coordinates of the tooling point relative to some workpiece coordinate system. The main difference between 5 axis controls is in the way the controls update the tooling point and the workpiece coordinate system after repositioning of rotary axes.

Most controls have two modes for the five axis machining. Those are the default "3 axis" mode and "Tool Center Point Management (TCPM)" mode. The controls behave differently in these modes. Anyway the behavior of the control is composed of the three options.

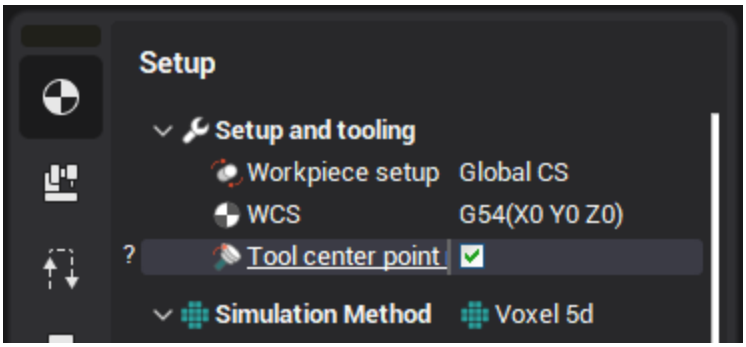
1. Does the tooling point rotate together with the tool head rotation or it stays fixed.
2. Does the workpiece zero point (G54) rotate together with the rotary table or it stays fixed.
3. Does the axes of the workpiece coordinate system rotate with the rotary table or they stay fixed.

In CAM system both of the options are available in the machine configuration file under the ControlData section and in the machine properties inspector under the Control Parameters section.



The options are as follows.

1. TCPM mode is available option make the tool center point management option available in continuous five axis milling operations.



When enabled the tool center point management option forces CAM system to enclosure the work passes of the generated toolpath with the INTERP 5AXIS ON, INTERP 5AXIS OFF commands, and to generate the CLDATA of the work passes with regard to the options set under the TCPM 5Axis compensation mode section.

2. TCPM mode default state option specifies the default state of the TCPM mode option in five axis milling operations.

3. Indexed 5Axis compensation mode group contains the set of five axis compensation options for the default "3 Axis" mode of work of the control.

4. TCPM 5Axis compensation mode group contains the of the five axis compensation options for the TCPM mode of the control.

5. 5 Axis tooling point compensation option is responsible for the rotation of the tooling point together with the rotary head.

6. 5 Axis workpiece zero point compensation option is responsible for the rotation of the workpiece zero point rotation to them with the rotary table.

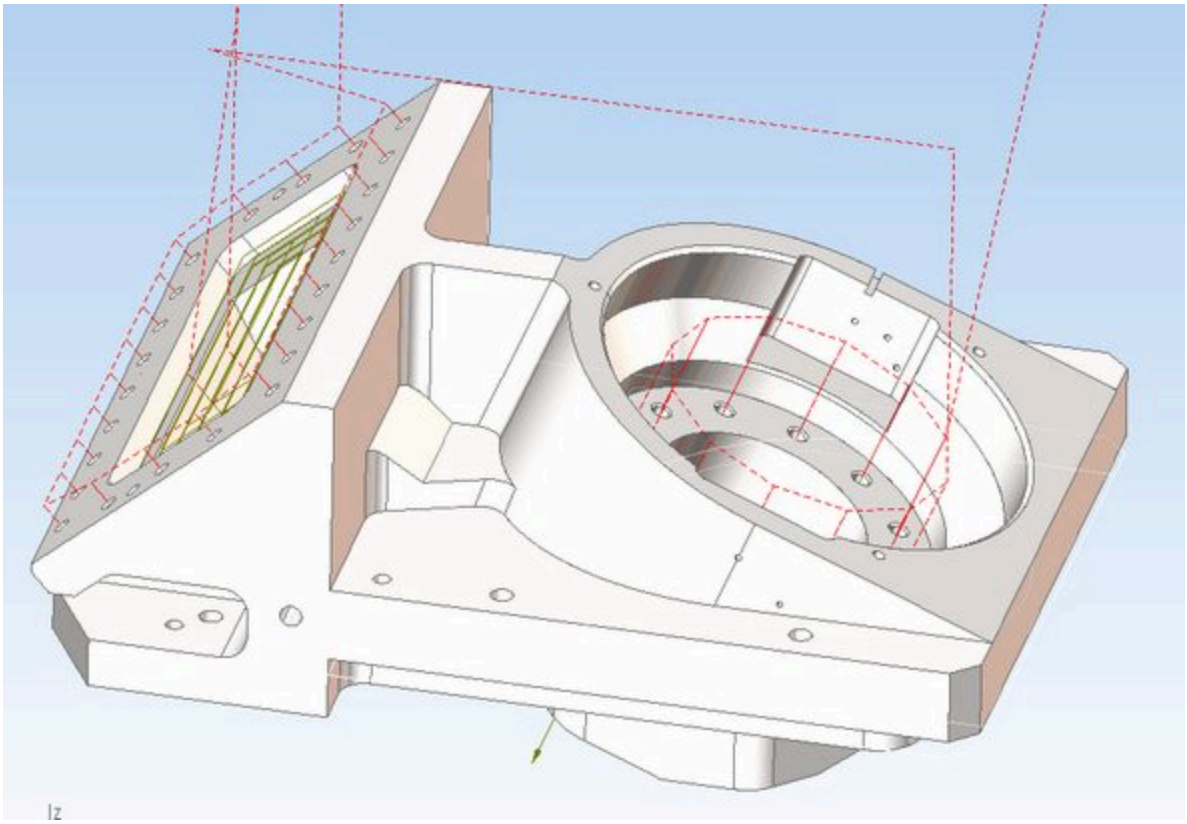
7. 5 Axis coordinate system compensation option is responsible for the rotation of the coordinate axes of the workpiece coordinate system together with the rotary table.

Most of modern controls have the TCPM mode available, and in this mode all the three 5 axis compensation options are enabled. This makes possible the generation of the five axis toolpath independent on the machine kinematics, the workpiece setup and the tool length.

However in the default "3 Axis" mode most of controls have all the three 5 axis compensation options disabled. That makes them behave as regular 3 axis controls knowing nothing about the machine kinematics. In order to make possible the generation of an indexed 3+2 toolpath those controls require mandatory use of local coordinate systems (Heidenhain Plane function, CYCLE 19). For more information refer to the Tool orientation topic.

Advanced axes limits control feature available for multiaxis operations.

Indexed 5-axes milling

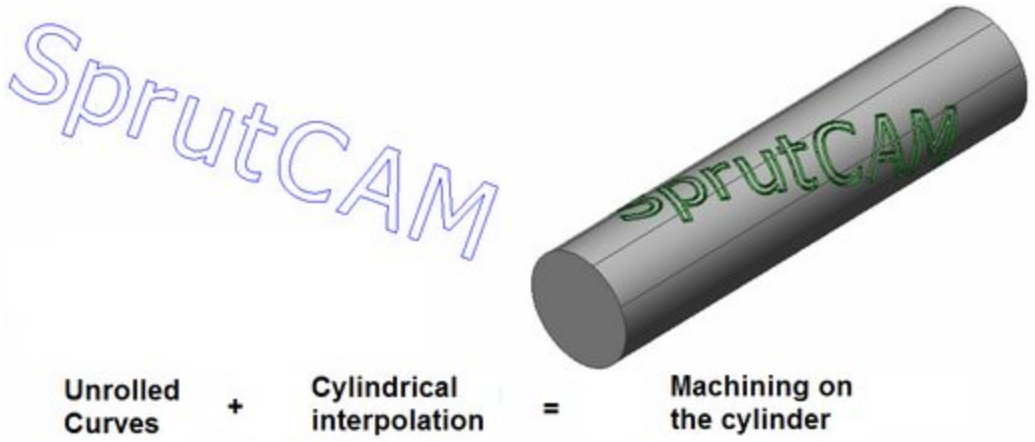


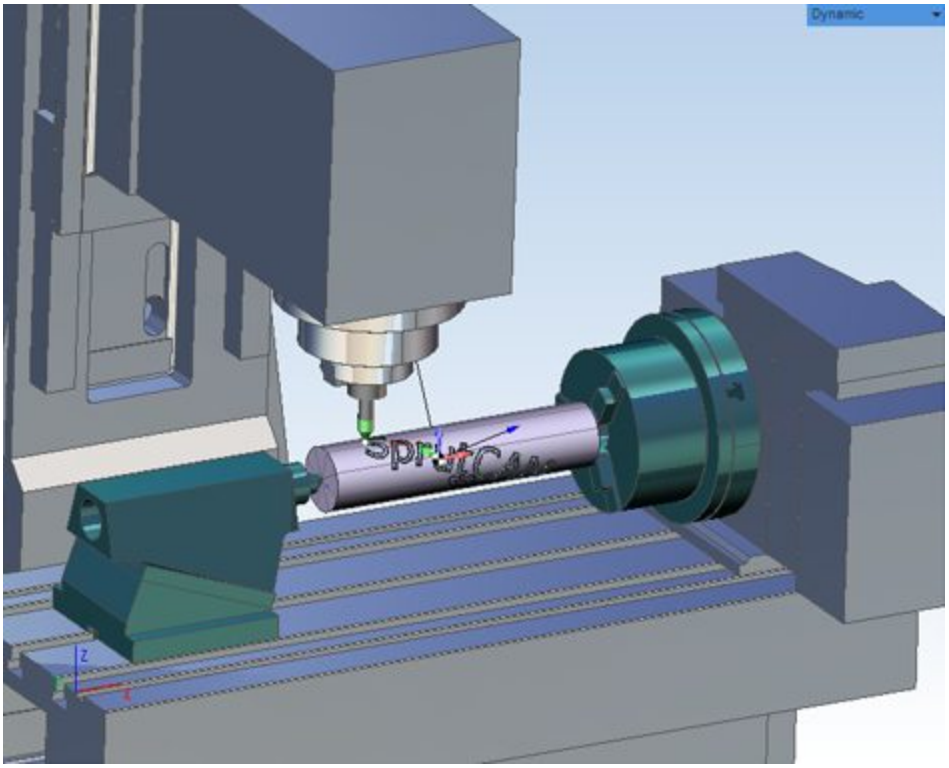
All 2D and 3D operation can be used for the indexed machining on the 4 and 5-axes machining centers. It needs to define the positions of the rotary axes or the local coordinate system.

Advanced axes limits control feature available for multiaxis operations.

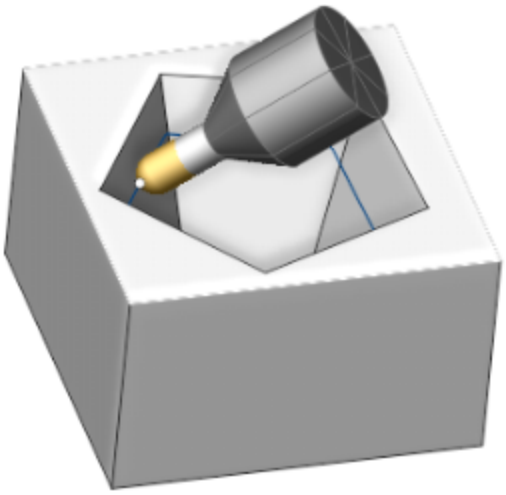
Rotary machining using cylindrical interpolation

`2D contouring`, `2.5D walls machining` and `2.5D pocketing` operations can generate the tool path for the continuous 4-axis milling. In this case it is needed the Cylindrical interpolation to be used.





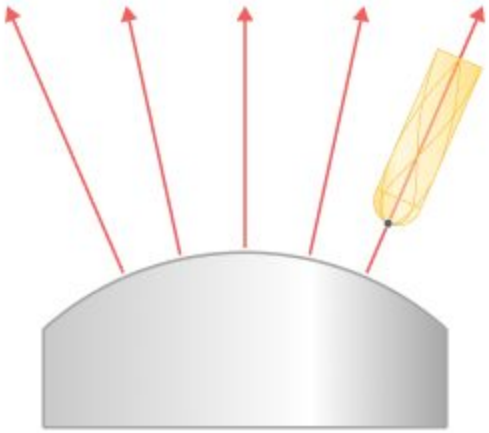
5 axis tool path conversion



This option converts 3 axis toolpath into 5 axis for such operations as: Scallop, Morphand Geodesic. This feature allows to avoid collisions and use minimal length tool. 5 axis tool path conversion can be used with spherical mill tool.

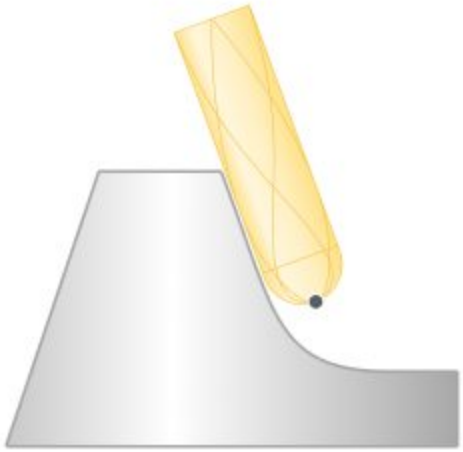
5 axis toolpath conversion options:

Normal to surface



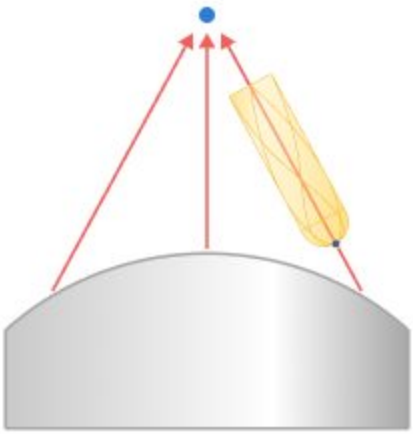
The tool is oriented by normal to machining surfaces. Additionally the lead and lean tool angles can be applied to further tilt the tool along or to the side from the cutting direction.

Flank



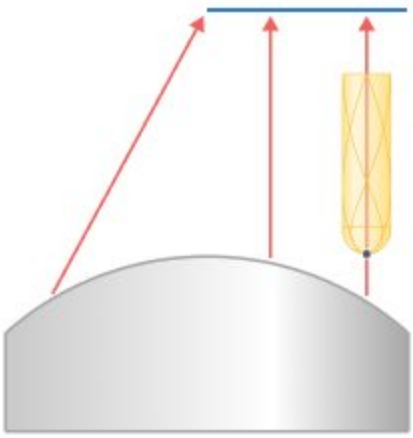
The tool contacts machining surfaces with the peripheral part (cylindrical part for the cylindrical mills). Additionally lead and lean angles can be applied. The strategy can be used for swarf milling.

Through point



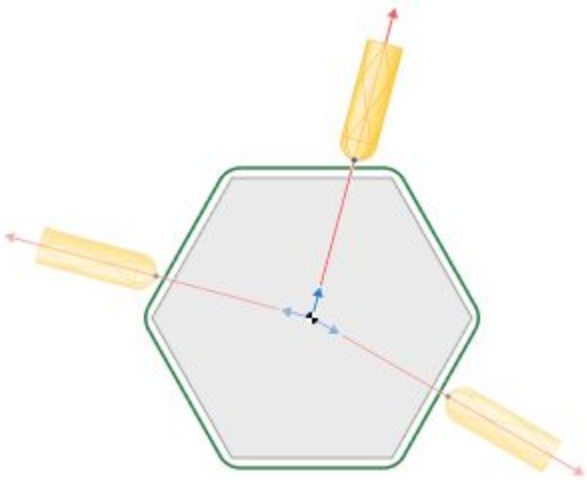
The tool axis is oriented to the specified point.

Through curve



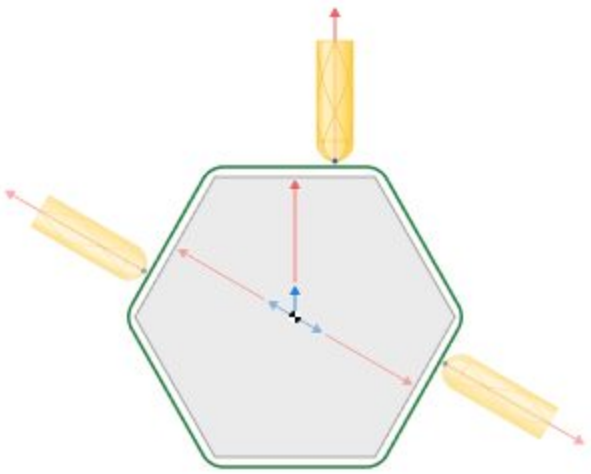
The tool axis is oriented to the nearest point of the specified Tilt curve. Additionally lead angles can be applied.

To rotary axis



The tool axis is directed to the rotary axis, as in the rotary machining. Additionally the side angle to the rotary axis and lean angles can be specified

Perpendicular to toolpath



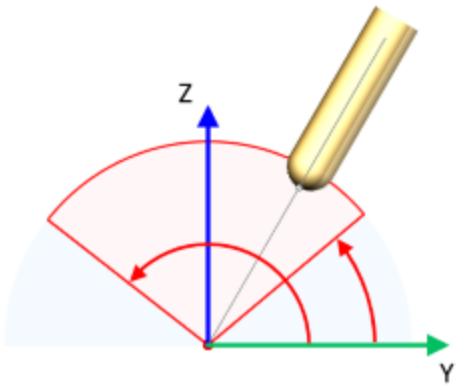
The tool axis is oriented to the perpendicular to toolpath and perpendicular to the rotary axis. Additionally the side angle to the rotary axis and blend distance can be specified.

4 axis machining with the Rotary axis

The rotary axis feature allows to transform a 5 axis toolpath into a 4 axis toolpath by locking one of the components (X, Y, Z) of the tool axis direction.

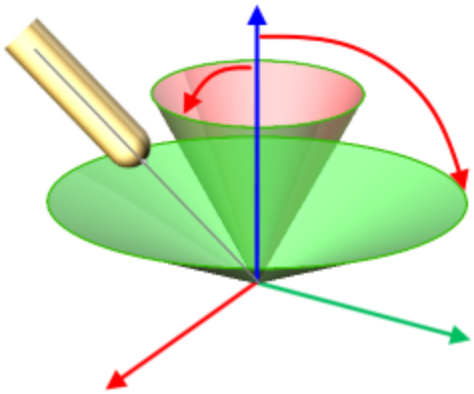
Limit Rotation angles

Limit around axis



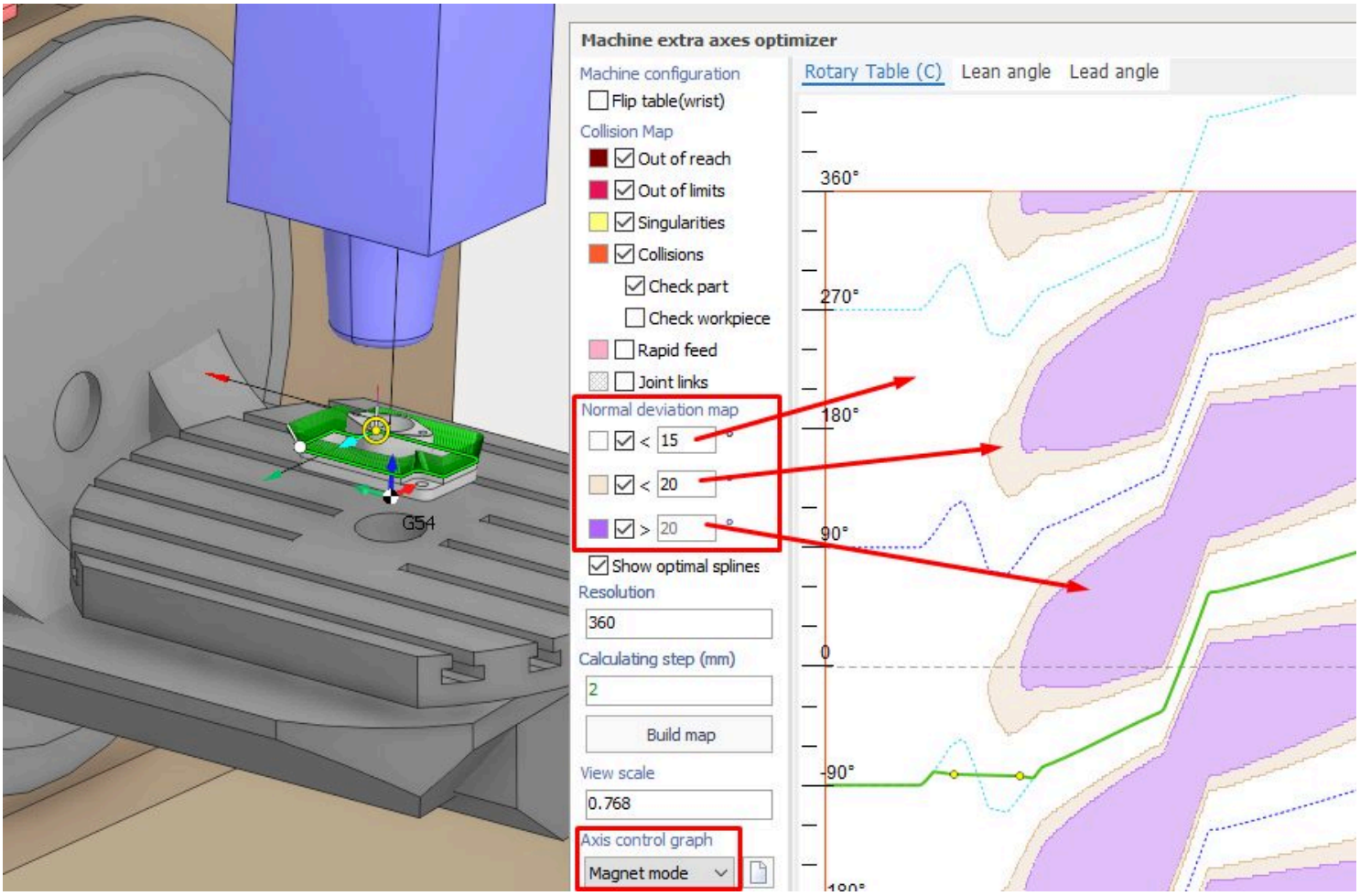
With this option you can limit the tool on the XY plane, XZ plane or YZ plane between two angles.

Conical limit

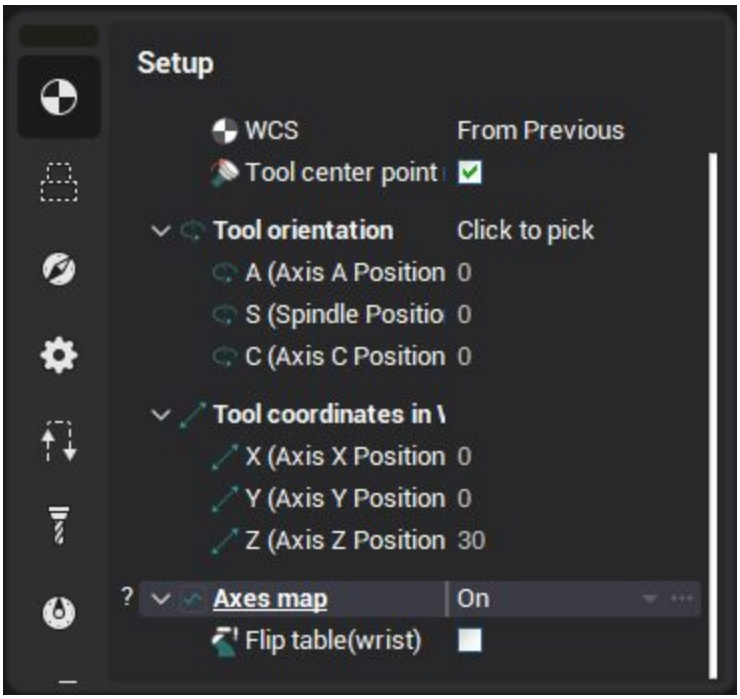


With this option, you can limit the tool at an angle along one axis between two cones.

Axes map for 5-axis machines



The Axes Map feature can be used to optimize the operation trajectory for 5-axis machines, similarly to robot extra axes optimizer. This feature is disabled by default, to enable it use the Axes map combobox on the Setup inspector tab. To open the special window for toolpath optimization, use the ellipsis button located to the right of the combobox. Currently, there are two types of axes map available for 5-axis machining: the C axis map and the maps for lead/lean angles.



"C axis" map

The rotary axis of the machine, which is responsible for rotating around the Z axis of the tool, is often named with the C letter (and complementary is the A or B axis). The 5-axis trajectory defines the tool orientation along itself and by default the optimal values are selected for the machine rotary axes to achieve the specified tool orientation in each toolpath point. However in some cases it can lead to undesirable trajectories with obstacle collisions or uncontrollable abrupt spikes in the motion of rotary axes due to singularities. With this feature, the axis map spline can be used to directly specify the value of one of the rotary axes along the toolpath. The value of the secondary axis is selected automatically to minimize the deviation from the user defined tool normal. The common user interface parts are described in the Robot axes map documentation page, below is the information about special C axis editing features.

Deviation map

When the C axis map tab is selected, the main part of the window displays the 2D color map showing the minimum possible deviation of the tool normal from the optimal (specified in job assignment) for each possible C axis value (vertical axis) and each toolpath point (this is the horizontal axis). The zones with the same color have the minimum deviation within the same specified range. The ranges can be customized with the Normal deviation map panel on the left. The map can also contain zones with the obstacle collisions.

Optimal splines

The dotted splines on this map (which can't be edited) correspond to the case when the C axis value is optimal in each toolpath point. Those optimal values are not unique, they differ by the shift of 360

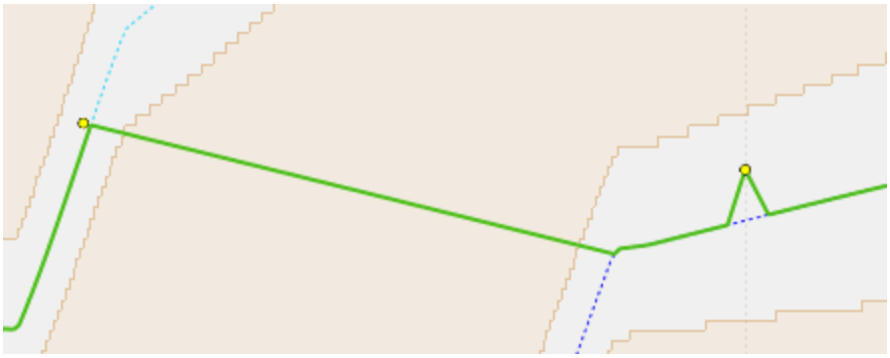
degress or correspond to the different fixed value of the Flip table parameter. The visibility of the optimal splines can be turned on/off with the checkbox below the "Normal map deviation" panel.

Spline editing modes

Three modes for the C axis spline editing are available. The first two modes are the same as in the robot axes map. In each mode the trajectory is defined by the custom control points. Use double click to create a new control point and drag it to adjust the position.

1. Spline mode - the axis trajectory is defined by the smooth line passing through the specified waypoints.
2. Polyline mode - the same as previous, but the control points are connected with the straight segments.
3. Magnet mode - the axis trajectory is also defined by the control points, but between them is generated by special algorithm. The goal is to achieve the trajectory which is the least different from the optimal splines. Let's consider the two consecutive points.

- If they are more close to each other than any of the optimal splines they are connected with the straight line segment.
- If they lie near the same optimal spline, then they are first smoothly connected to this spline and between those connection points the trajectory matches the optimal one. If some point is very close to the optimal spline, it is not connected to it, rather it "defines" which optimal spline to use in its neighbourhood.
- If the points are close to different optimal splines, then the algorithm searches the closest points between the two splines and connects them with the straight line. Then the first point is connected with the left closest found point, and the second - with the right closest (as they correspond to the same optimal splines). Below is the axis trajectory example for the third described case.



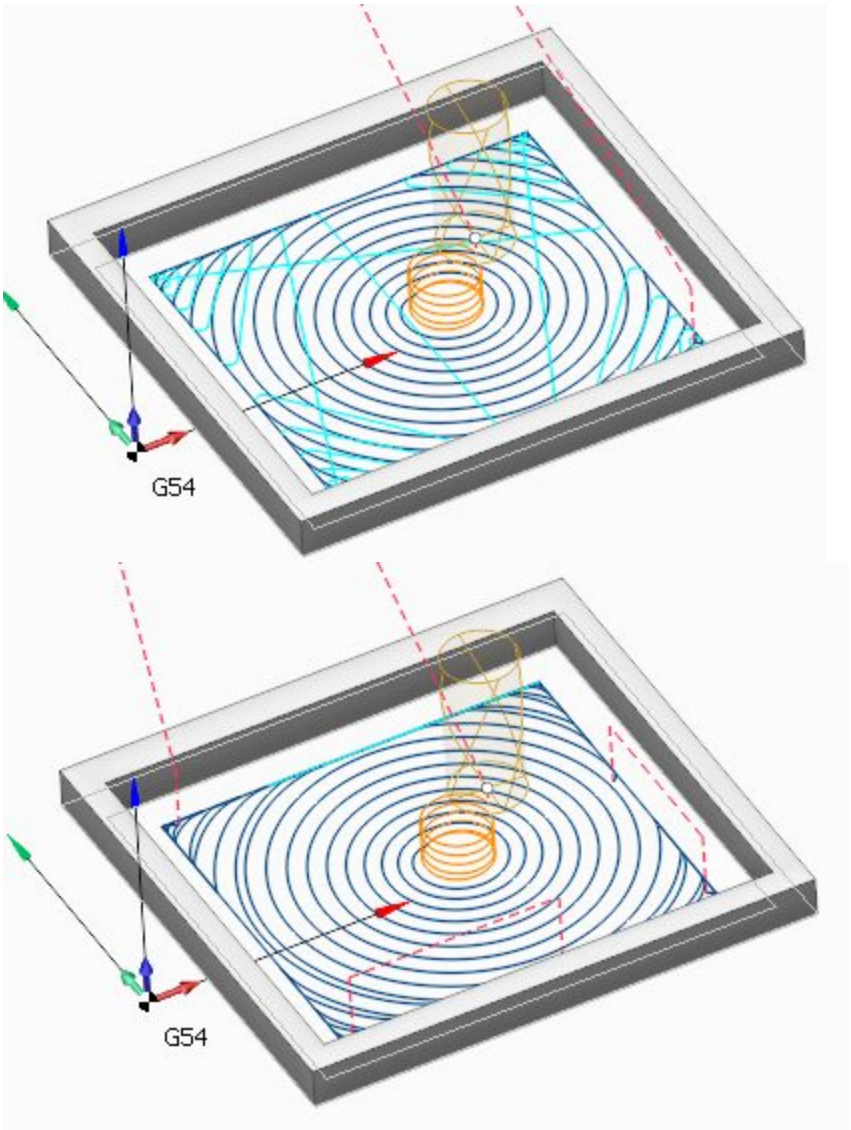
Lead/Lean angle map

The Lead/Lean angle are the important parameters in 5-axis machining which define the inclination of the tool along the toolpath and to the side (in the perpendicular plane) of the toolpath, respectively. Switch to the Lead or Lean angle tab to control the values of these parameters along the operation toolpath. The user interface is the same as for the robot extra axes optimizer.

Adaptive

The strategy is used to effectively remove large volumes of material with high feedrates, maximal cutting depths (up to the flutes length) and relatively shallow cutting widths (5% to 30% of the tool diameter). Such parameters are possible as the specified tool engagement angle (which is defined as the width of cut, or step) is guaranteed to never be exceeded by the strategy.

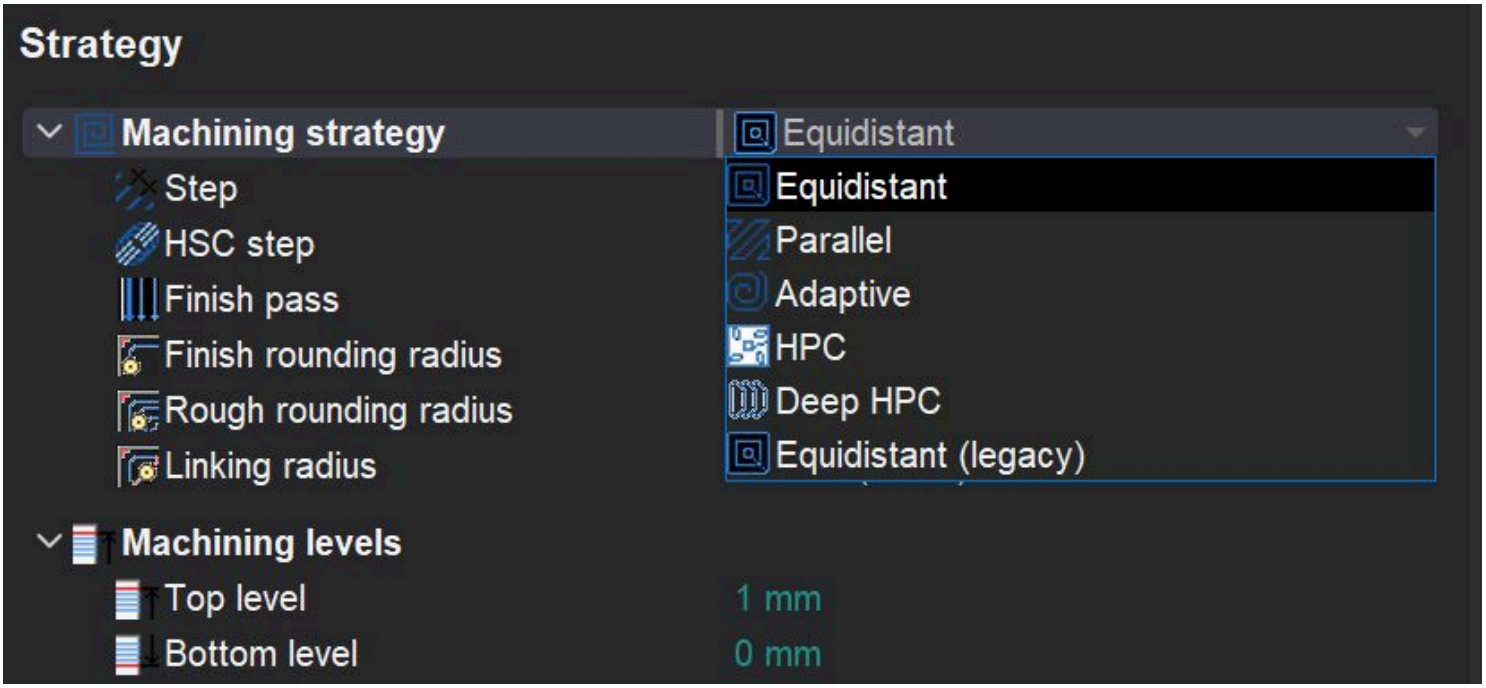
The material is removed in spiral-like fashion. There are no sharp corners in the toolpath. Smoothness of the toolpath is precisely controlled by the dedicated parameters for the roughing rounding radius, the finishing radius and the linking radius. Linking is done preferably in the working plane with an additional small Z clearance which helps fight heat buildup. The tool engages material using the so called 'Roll-In techniqe' which prolongs tool life. Both climb and mixed (climb and conventional) milling is available. For the mixed milling the width of cut and the feedrate of conventional passes can be set separately from the climb passes.



The strategy is available in the following operations:

- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

The strategy can be enabled by selecting the corresponding option from the Machining strategy drop-down:



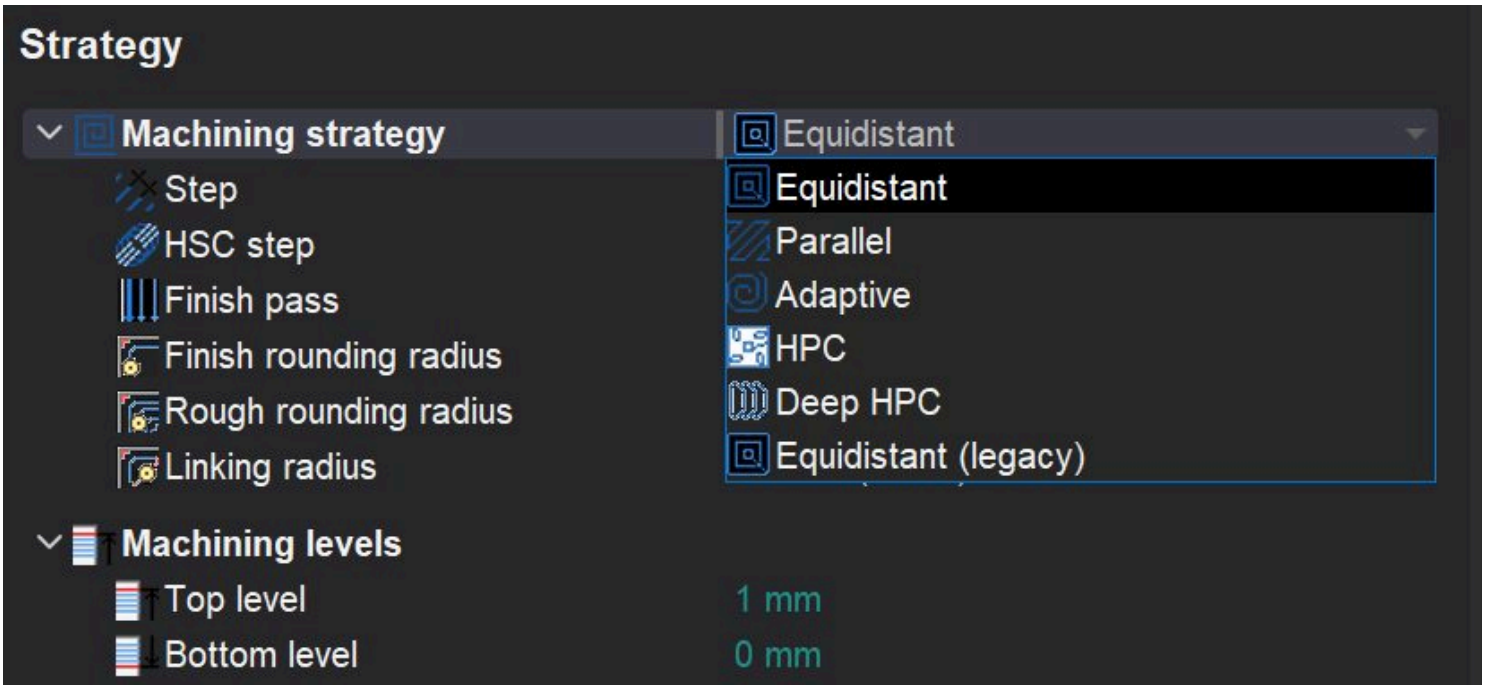
Pocketing strategies

The pocketing strategies are designed for the removing of material in the open and closed pockets.

These strategies are available in the following operations:

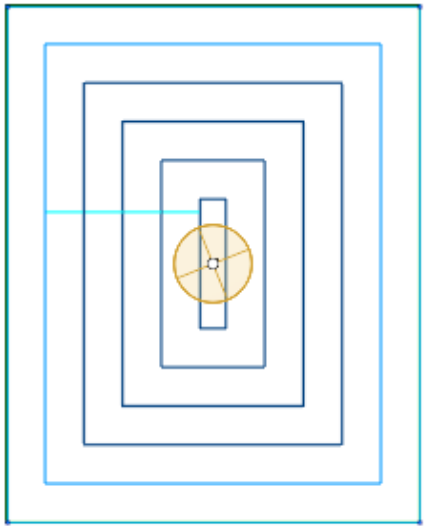
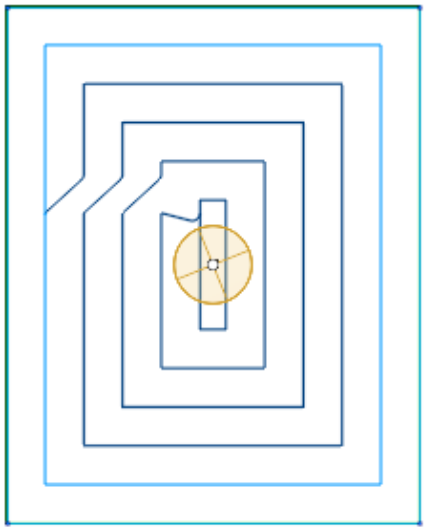
- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

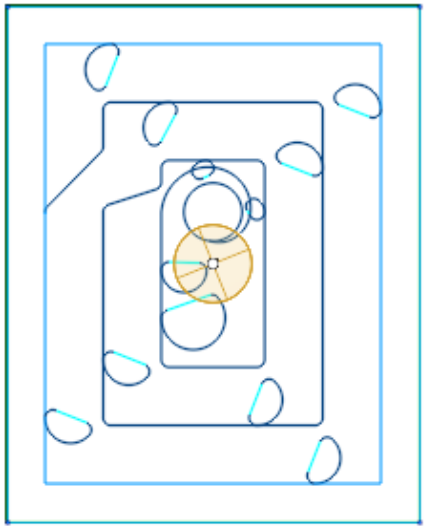
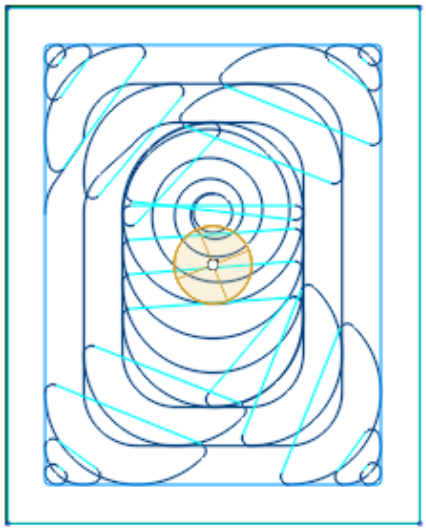
Seven strategies are available to select in the Machining strategy drop-down:

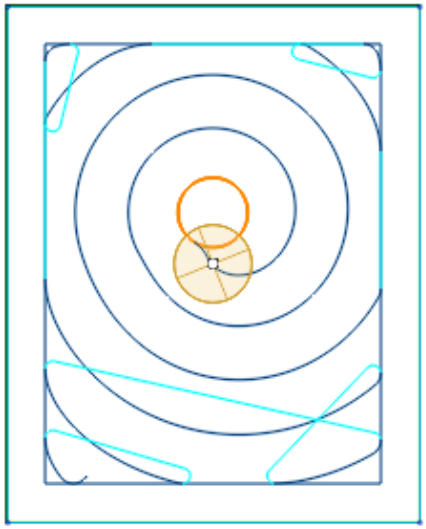
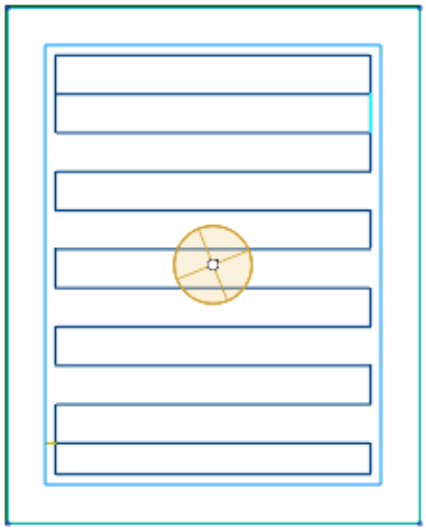


There are 6 strategies. Some items are optional and require an additional license. So many strategies are the result of long-term development. Every strategy has its own advantages and disadvantages, so no one of them can't be removed from the system.

Strategy		
Equidistant (legacy)	Advantages Fast calculation Simple tool path	Disadvantages Residual unmachined islands are possible if the step is more than 50% Uneven tool load and chip thickness Many Z motions to/from the safe plane

Strategy		
		
Equidistant 	Advantages It's possible to define the safe distance The most of the links are performed without the climbing of the safe plane Links rounding is available	Disadvantages Residual unmachined islands is possible if the step is more than 50% Uneven tool load and chip thickness
HPC (high performance cutting)	Advantages All advantages of the equidistant strategy Special arc is added to remove the residual unmachined islands	Disadvantages Uneven tool load and chip thickness The special arc's radius can be too small, that gives the uneven feed rate.

Strategy		
		
Deep HPC 	Advantages All advantages of the HPC strategy The even tool load	Disadvantages Tool path is longer than the HPC strategy Idle motions are possible Unstable calculation. Sometimes the tool load can be greater than required.
Adaptive	Advantages The even tool load The perfect tool path for the open pockets	Disadvantages The length of tool path can be longer than the length of the DeepHPC strategy with the same parameters. It's actual for the big closed pockets.

Strategy		
		
Parallel 	Advantages Fast calculation Simple tool path	Disadvantages A lot of Z-motions in the complex pockets

Features of Adaptive strategy

The strategy is used to effectively remove large volumes of material with high feed rates, maximal cutting depths (up to the flute's length) and relatively shallow cutting widths (5% to 30% of the tool diameter). Such parameters are possible as the specified tool engagement angle (which is defined as the width of cut, or step) is guaranteed to never be exceeded by the strategy.

The material is removed in spiral-like fashion. There are no sharp corners in the toolpath. Smoothness of the toolpath is precisely controlled by the dedicated parameters for the roughing rounding radius, the finishing radius and the linking radius. Linking is done preferably in the working plane with an additional small Z clearance, which helps fight heat buildup. The tool engages material using the so called 'Roll-In technique' which prolongs tool life. Both climb and mixed (climb and conventional) milling is available. For

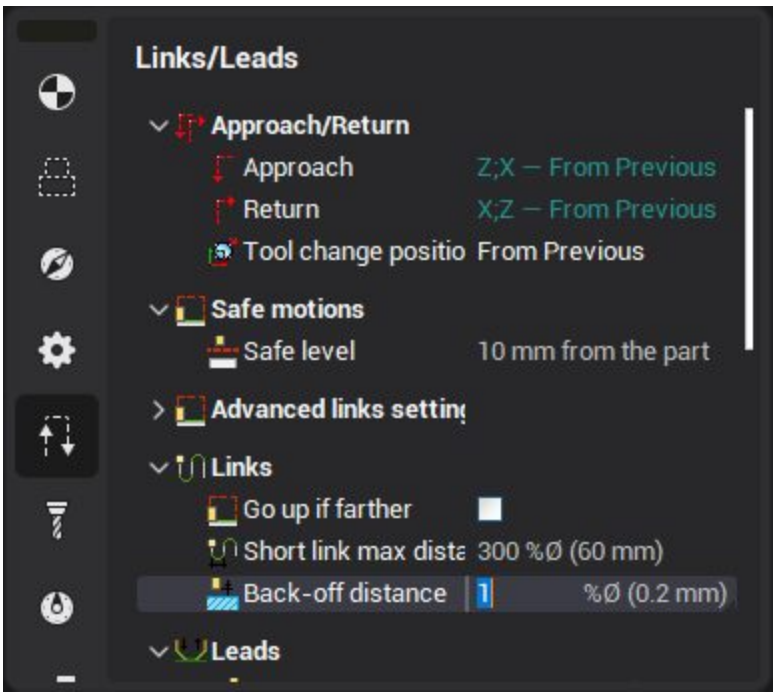
the mixed milling, the width of cut and the feed rate of conventional passes can be set separately from the climb passes.

How to choose the pocketing strategy

1. The choice number one is Adaptive This strategy is not set as default, only because it requires the additional licensing. So we strongly recommend purchasing it. All other variants must be tested only if this strategy is not available or gives the improper toolpath.
2. If Adaptive is not possible, and you need the even tool load, then try Deep HPC strategy.
3. If even tool load is not necessary and the machining step is more than 50% of the tool diameter then try HPC strategy
4. If even tool load is not necessary and the machining step is less than 50% then try Equidistant strategy.
5. Use Parallel strategy at your own discretion.
6. Use Equidistant (legacy) if all other strategies give improper toolpath.

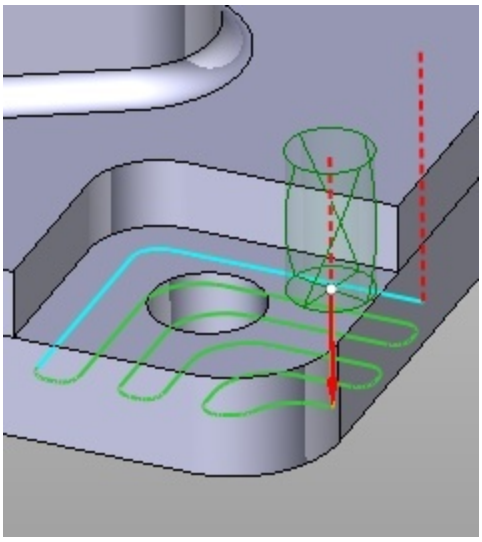
Tool path parameters

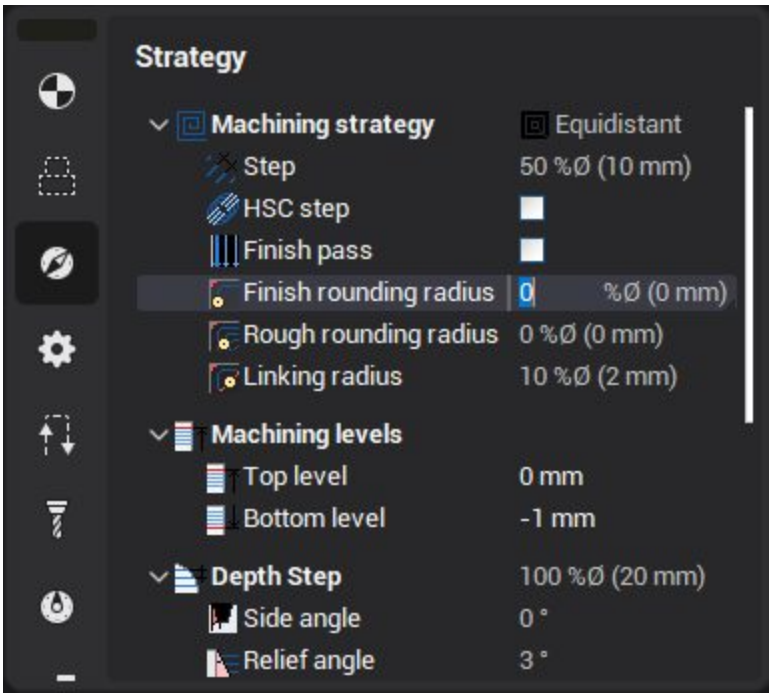
- Back-off distance parameter



The tool can be lifted above the already machined surface when it moves to the next trochoidal arc start position.

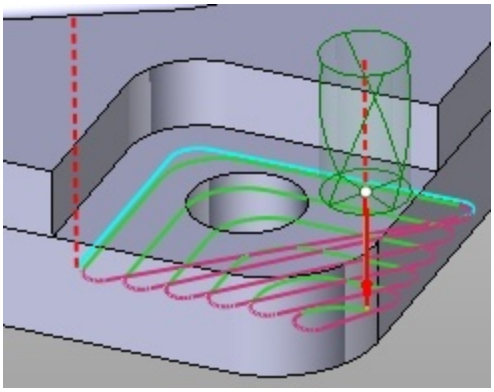
- Rounded links in zigzag mode





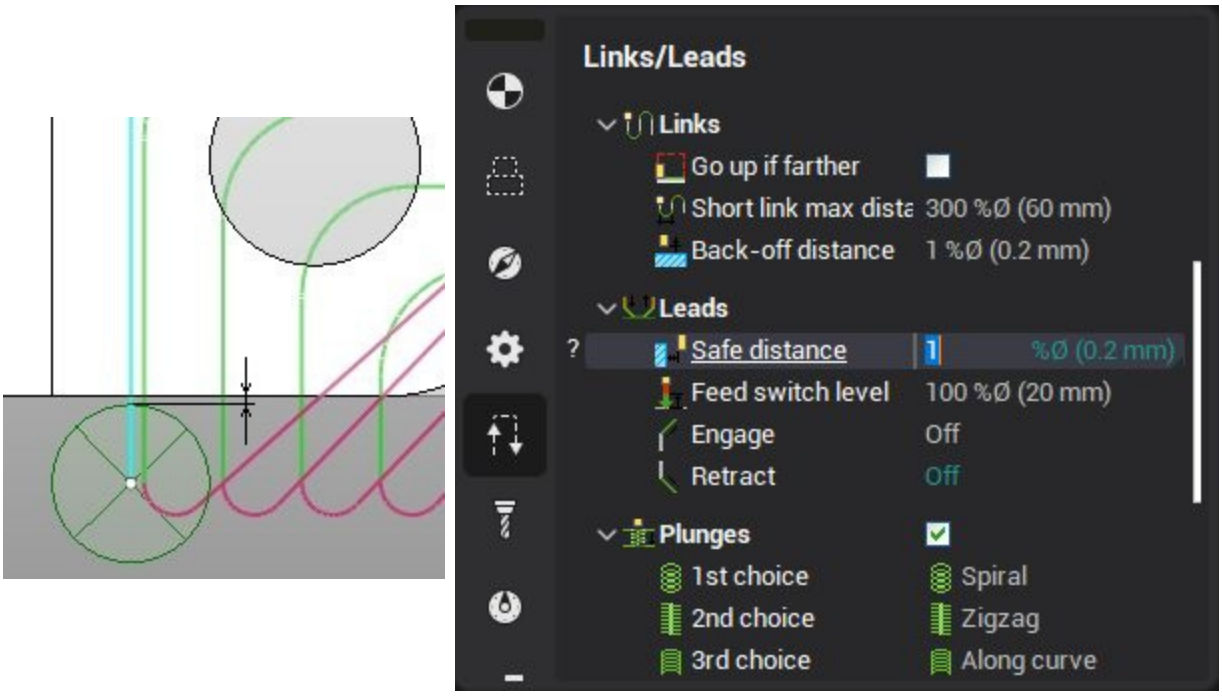
The 'Finish rounding radius', 'Rough rounding radius' and 'Linking radius' value is used for rounding of the links.

- Links on the same Z-level



In the climb and conventional mode, the tool goes directly to the next path without retraction to the safe level. If a rapid motion is performed over an already machined surface, then the "Tool back-off distance" is used. "Idle radius" is also used to make the motion smooth.

- Safe distance

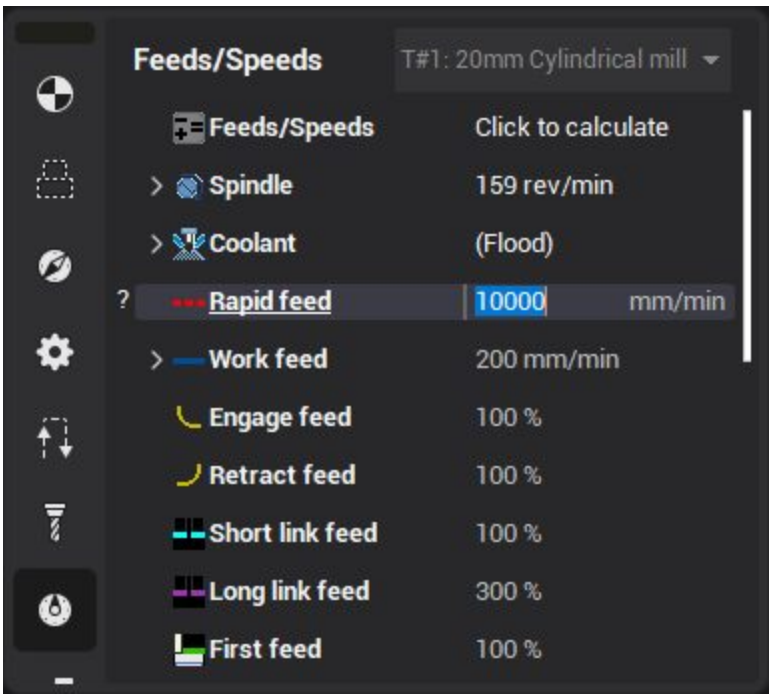


Safe distance is used to move the tool down/up from/to the safe surface.

The vertical motion is performed at this distance from the workpiece. So in version 10 there is no longer the need to enable the approaches/retractions to exclude the rapid feed collisions.

If you use a pre-drilled hole to plunge when roughing, the pre-drill tool diameter must be greater than the mill tool diameter by at least double the safe distance amount, otherwise the pre-drilled holes will not be detected.

- Rapid feed links



The link moves can be calculated using either the next feed or the return feed values. If the link length is less than the 'short link' distance, then the 'next feed' value is used, else the 'return feed' value is used. The return feed is set to 300% of the work feed by default, which is a non-cutting feed. If cutting is detected during a 'return feed' move when simulated, this move will be marked with an error.

Lathe machining

Lathe machines used for turning workpieces of metal and other materials similar in shape to the figure of revolution. Such operations as turning and boring of cylindrical, conical and form surfaces, cutting of thread, facing, hole drilling, enlarging and reaming performed on lathes.

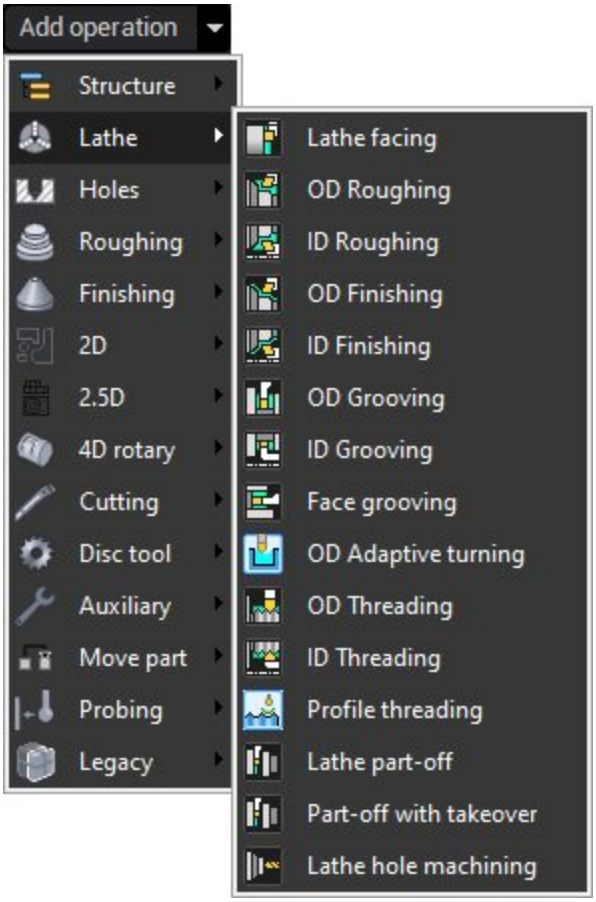
The workpiece receives rotation from spindle, cutter – the cutting tool – moves with rest sledge from traverse shaft or lead screw which receive rotation from feed gearing.

CAM system allows to design NC-programs for lathe machines with CNC.

Types of lathe machining operations

CAM system represents the machining process as a sequence of operations. The sequence can contain any number of operations of various kind. Each operation uses it's particular methods to form toolpath and accepts individual set of parameters. The list of available operations is defined by the system configuration.

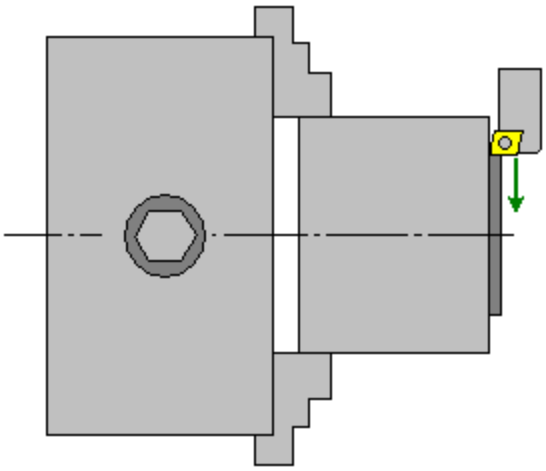
All lathe operations placed in one group "Lathe" inside the list of all operations.



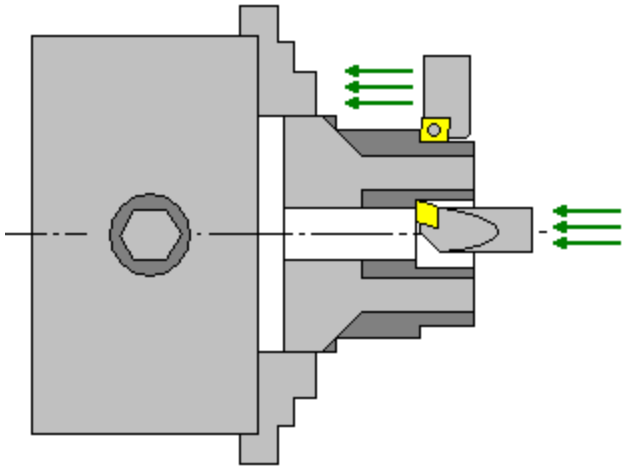
Lathe machining operations

In CAM system lathe machining is grouped by operations:

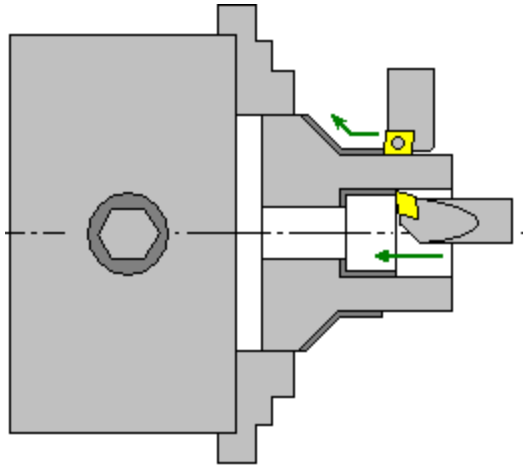
- Lathe facing



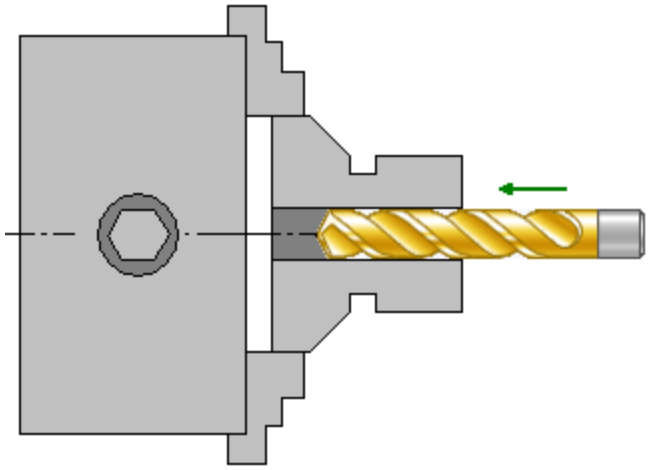
- OD Roughing, ID Roughing



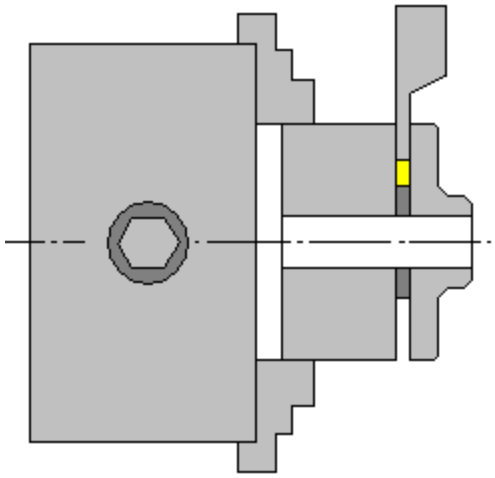
- OD Finishing, ID Finishing



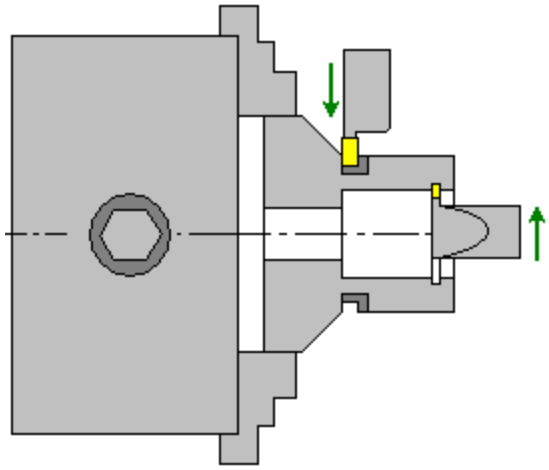
- Lathe hole machining



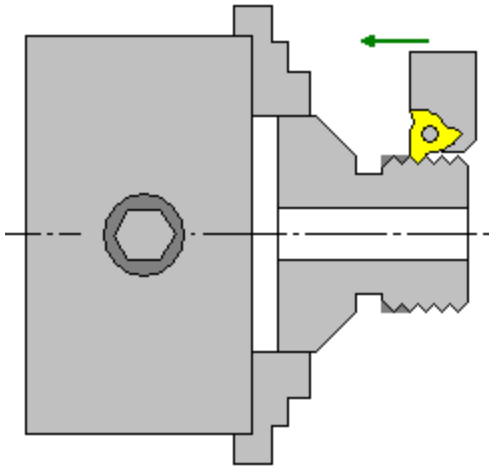
- Lathe part-off



- OD Grooving, ID Grooving, Face grooving

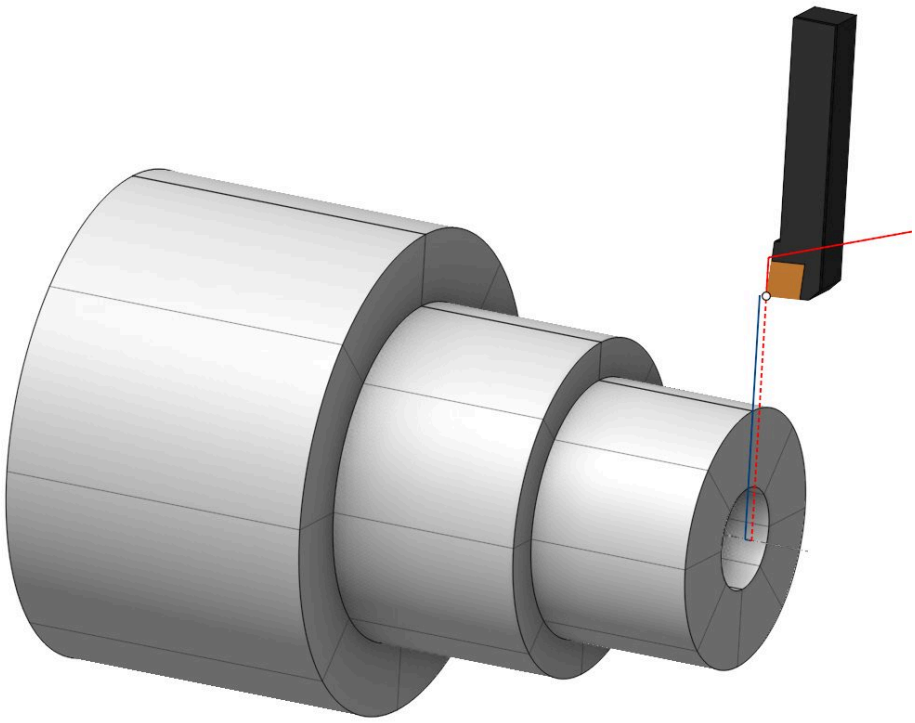


- OD Threading, ID Threading



All operations, which were inherited from the previous version moved to the Legacy group. See documentation for the previous version of CAM system to get more info about these operations.

Lathe facing



Application area:

The operation is designed to prepare the uneven face of workpiece (left or right). The material is removed by the vertical tool motions stroke by stroke.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Facing. This allows programming the machining of the end face of a workpiece with a single command. The trajectory is formed in the same way as when specifying Contour or Contour Repeat cycles. However, the trajectory is automatically calculated based on the current blank and workpiece, ensuring correct machining of either the left or right end face. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

CycleParams:

This parameter group works similarly to the OD Roughing and ID Roughing operations. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

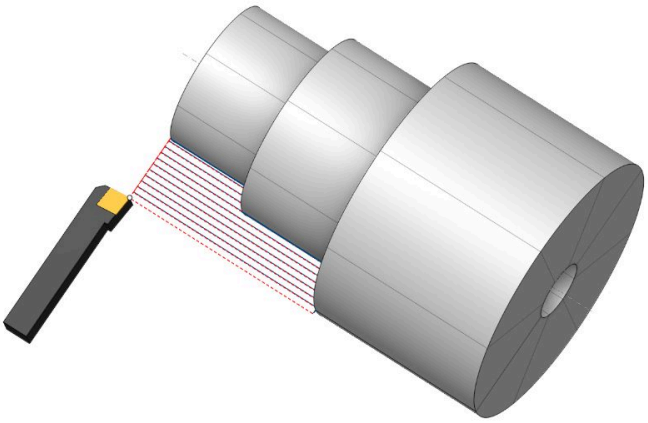
Workpiece:

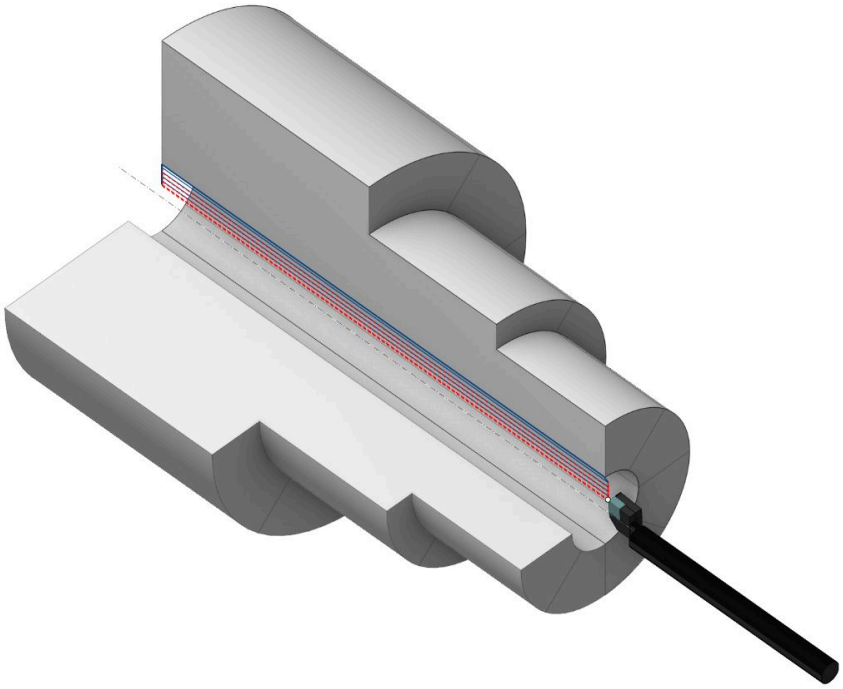
A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

OD Roughing and ID Roughing





Application area:

OD roughing, and ID roughing operations are designed to get the intermediate part by removing a lot of the workpiece volume that is located outside of the part and fixtures. The tool removes material by the series of the parallel strokes. The strokes can be parallel or perpendicular to the revolution axis depending on using cycle type and its parameters.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Roughing. The roughing allowance is removed by tool movements along the axis of the workpiece (when processing surfaces elongated in the direction of the Z-axis) or perpendicular to it (when processing surfaces elongated in the direction of the X-axis). In other words, a trajectory is formed similar to ISO G71/G72, but it is output to the control program in an expanded form, without the use of a cycle. Details

Roughing cycle. Similar to rough machining, but the operation is output to the control program in the form of a cycle, similar to ISO G71/G72. The roughing cycle allows the creation of one of the ISO G71 or ISO G72 cycles based on the specified contour of the workpiece. Switching between G71 and G72 is achieved by changing the Direction of tool movements (along the axis of the workpiece Axial or perpendicular to it Radial) in the parameters. Details

Offset Roughing. The roughing allowance is removed by tool movements equidistant to the contour of the workpiece. In other words, a trajectory is formed similar to ISO G73, but it is output to the control program in an expanded form, without the use of a cycle. Details

Offset cycle. Similar Offset Roughing, the operation is output to the control program in the form of a cycle, similar to ISO G73. Details

Zigzag. This process is used for machining enclosed areas from both sides, using a neutral tool. The tool is engaged by plunging downwards and performs a horizontal movement to remove the next layer. The tool re-engages by plunging again and makes a horizontal pass in the opposite direction. This way, the entire roughing allowance is removed layer by layer through horizontal passes with alternating directions. Details

Profile. This is the simplest type of cycle, where a single pass is generated, and its trajectory is identical to the specified initial geometric contour of the workpiece. Only minor modifications to the contour can be performed, including approach and retract linking, offsetting by specified allowances, excluding shaded areas that cannot be reached by the designated tool, and compensating for the tool radius. Details

Facing. This allows programming the machining of the end face of a workpiece with a single command. The trajectory is formed in the same way as when specifying Contour or Contour Repeat cycles. However, the trajectory is automatically calculated based on the current blank and workpiece, ensuring correct machining of either the left or right end face. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

CycleParams:

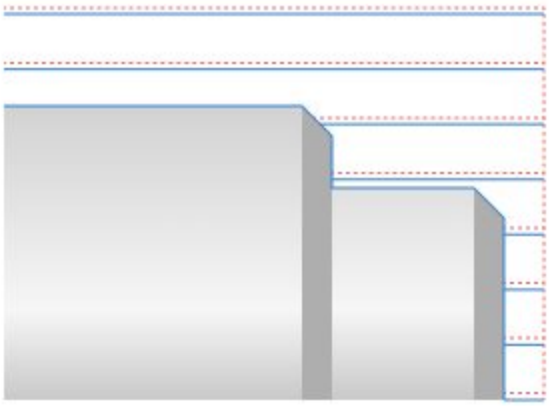
Cycle.

You can select one of the available turning cycles described in the Job assignment. When you switch a cycle in this section, the corresponding element in the Job assignment automatically changes. The set of parameters varies for each turning cycle.

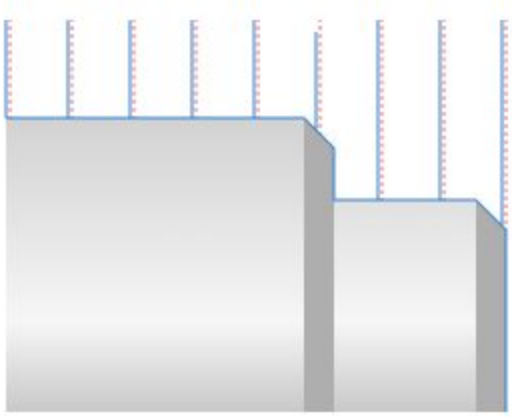
▼ Click here to expand..
Here are the key parameters, the set of which may vary depending on the chosen strategy in the Cycle section:

Direction. Property switch between G71 and G72.

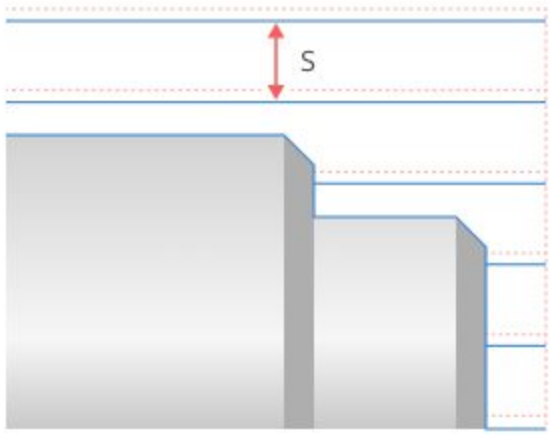
Axial. Turning is carried out with moves parallel to the axis of the part. (G71).



Radial. Turning is carried out with moves perpendicular to the axis of the part (G72).

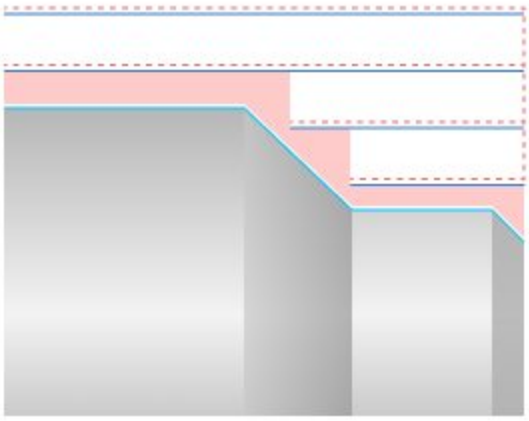


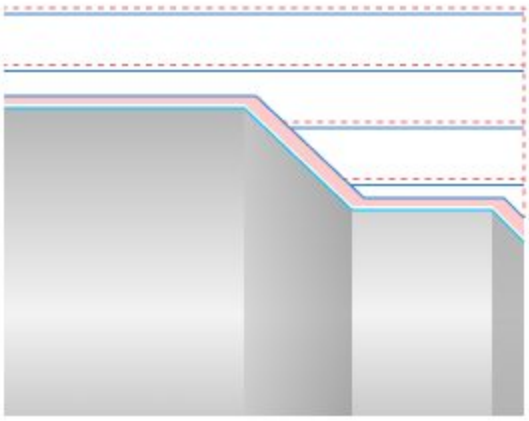
Step. The distance between tool passes is equal to the cutting depth.



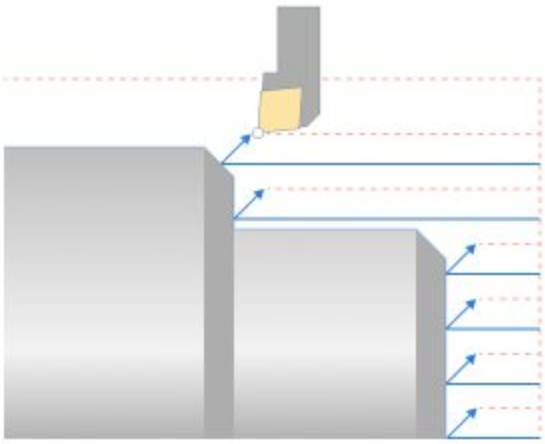
S - Step

Overlap. When the flag is set, the system performs a semi-finish pass to remove the ridges remaining after the roughing passes.

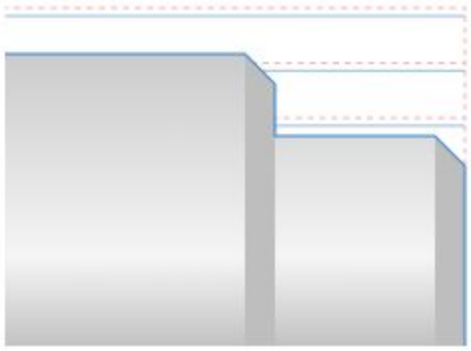


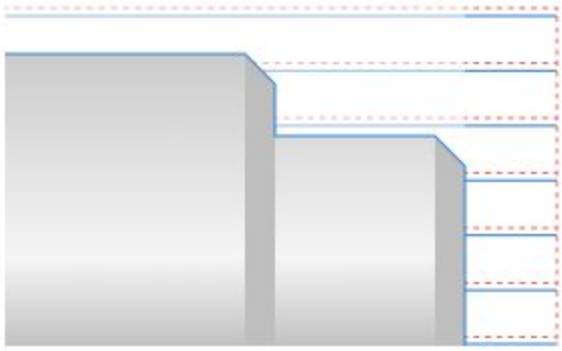


Safe distance. Specifies the distance, the tool retracts after a roughing pass. At this distance from the machined surface, the tool returns to its original position.

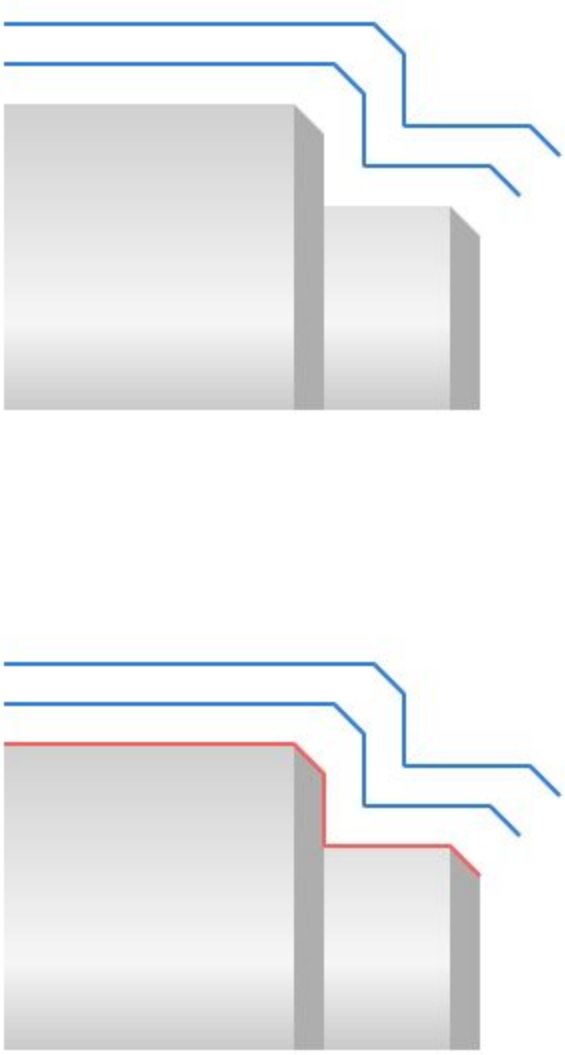


Leads in passes. When the flag is set, the tool performs the Engage in each working pass by the amount specified on the Links tab.





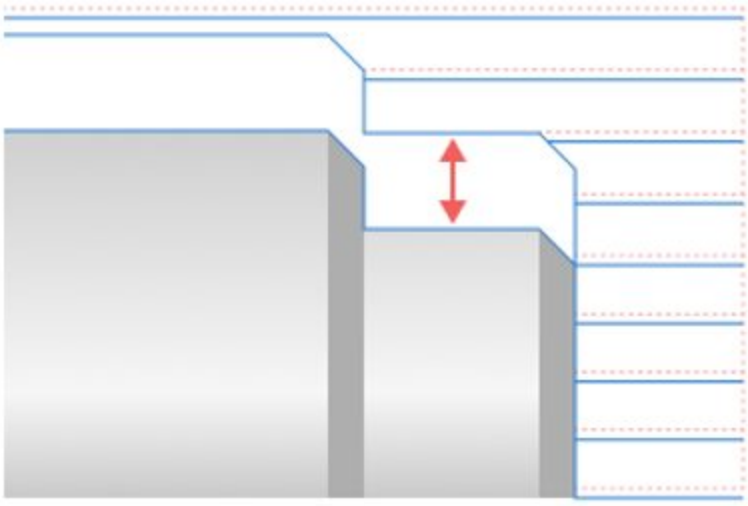
Finish pass. When the flag is set, the system performs a finishing pass in the same operation.



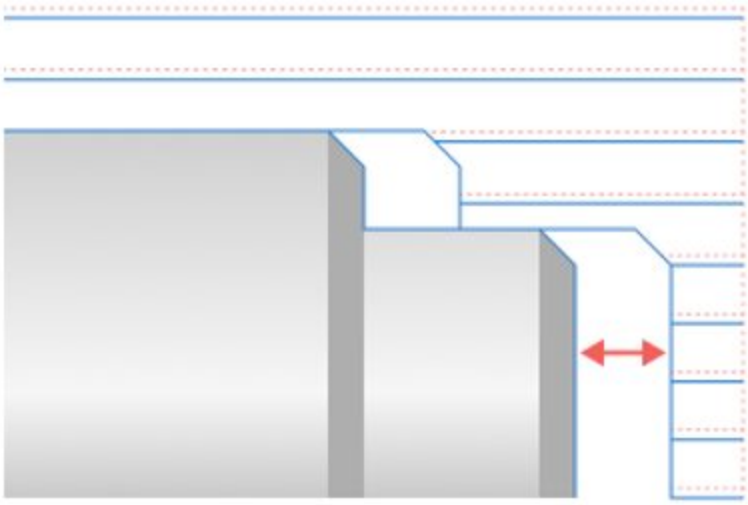
Stocks. This parameter leaves this allowance for subsequent machining. If the Finish pass flag is set , the system removes this allowance in the same operation.

▼ Click here to expand..

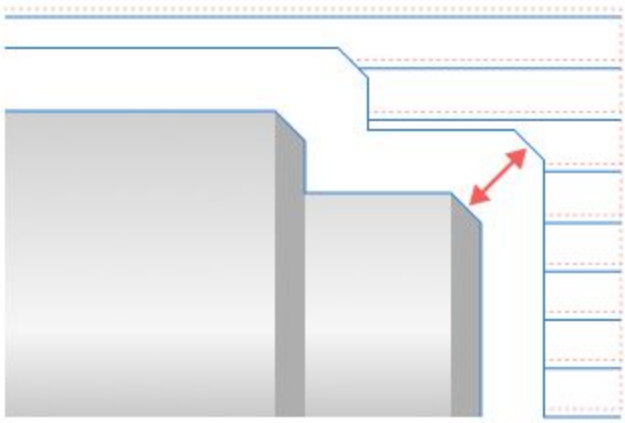
Radial. A stock of a specified size appears on cylindrical surfaces. On conical and spherical surfaces, the allowance decreases in accordance with the angle of the surface normal. When setting both the radial and the axial stocks simultaneously, they combine quadratically on these surfaces.



Axial. A stock of a specified size appears on faces. On conical and spherical surfaces, the allowance decreases in accordance with the angle of the surface normal. When setting both the radial and the axial stocks simultaneously, they combine quadratically on these surfaces



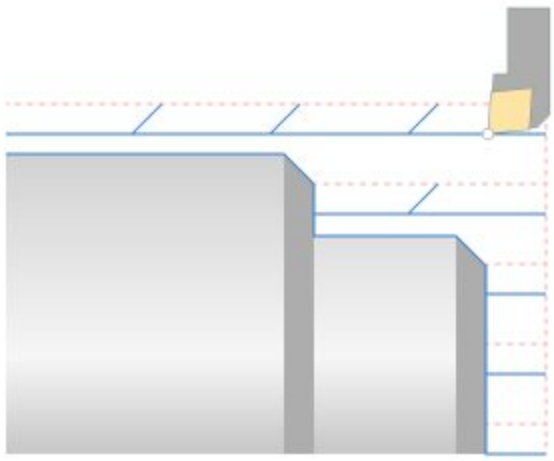
Profile. The toolpath with this allowance passes equidistantly to the contour of the part by a specified amount. All allowances combine when you set them simultaneously.



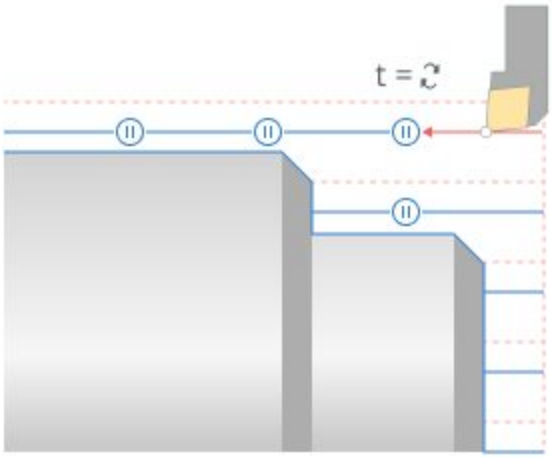
Chip breaking. Flagging activates the mode of kinematic chip breaking in which, after a specified cutting length, the cutter retracts to stop the formation of continuous chip. Then the cutter engages again and continues cutting. The retract distance is set in the Safe distance parameter.

▼ Click here to expand..
Dwell. Allows you to break chips not by retract, but by stopping the feed of the cutter.

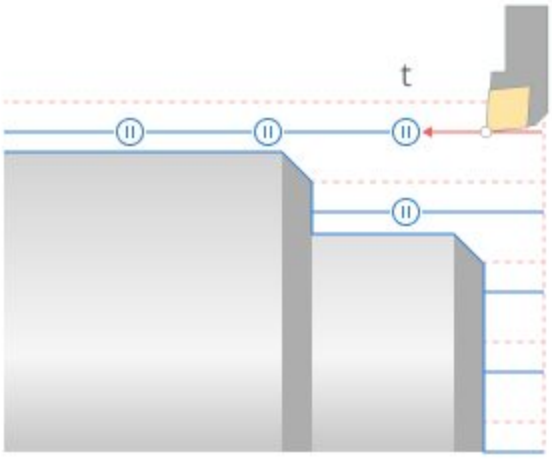
Off. When chip breaking by retract, the Dwell parameter is in the Off position.



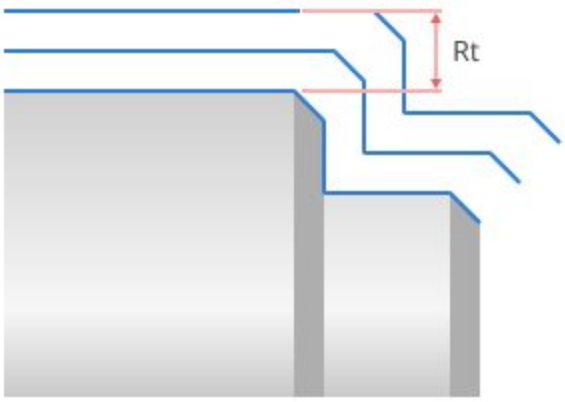
Revolutions. Stopping the cutter feed for a specified number of spindle revolutions performs the chip breaking.



Seconds. Stopping the cutter feed for a specified time in seconds performs the chip breaking.

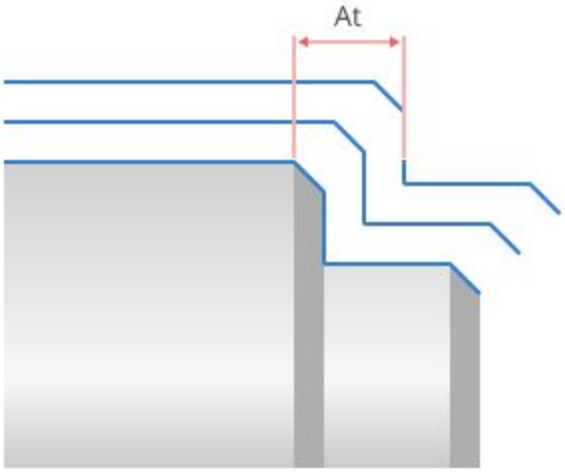


Radial thickness. The distance of relief in the X axis direction. Similar to the U parameter in the first block of the ISO G73 cycle.



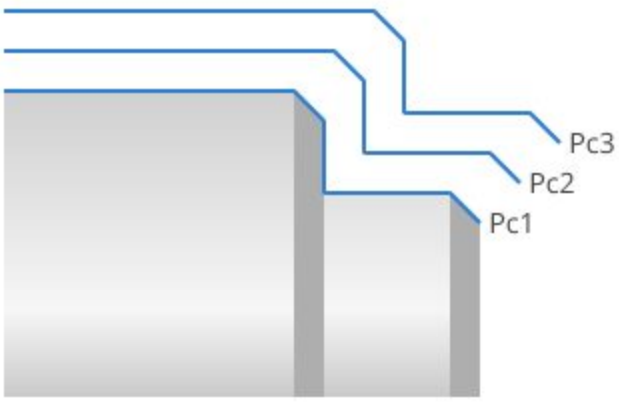
Rt - Radial thickness

Axial thickness. The distance of relief in the Z axis direction. Similar to the W parameter in the first block of the ISO G73 cycle.



At - Axial thickness

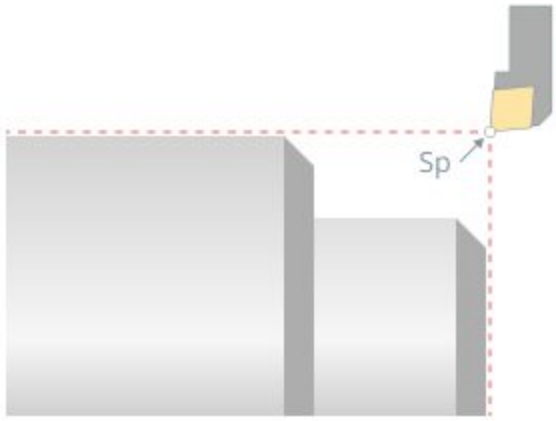
Pass count. The number of tool passes required to remove the rough stock. Similar to the R parameter in the first block of the ISO G73 cycle.



Pc1, Pc2, Pc3 - Passes 1–3

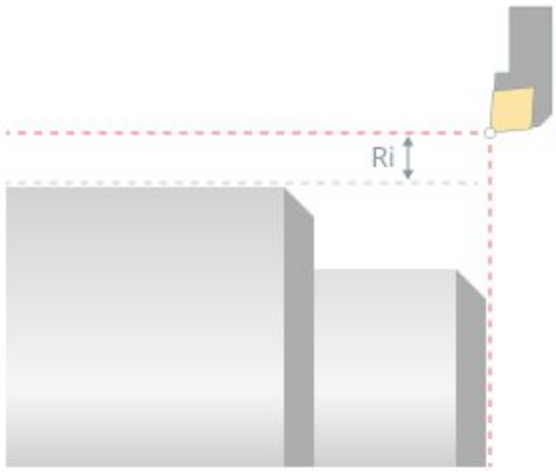
Start point. The tool is approaches to this point, from which the execution of the G 71, G 72, G 73 cycles begins, and to which the tool returns upon completion of the cycle. If the increments are zero, then the Start point is the intersection point of the lines drawn through the Engage and Retract points .

▼ Click here to expand..



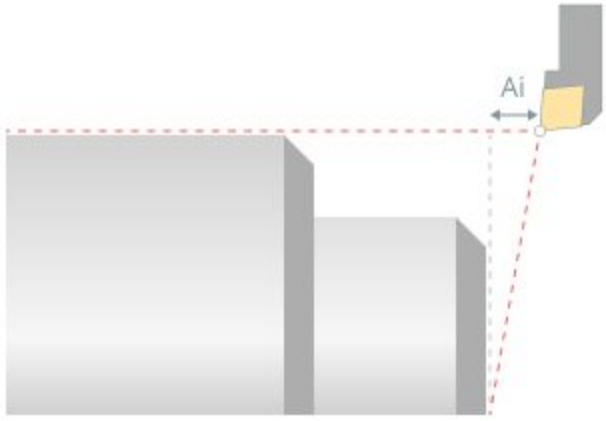
Sp - Start point

Radial increment. Shifts the Start point along the X-axis in the direction corresponding to the increment sign.



Ri - Radial increment

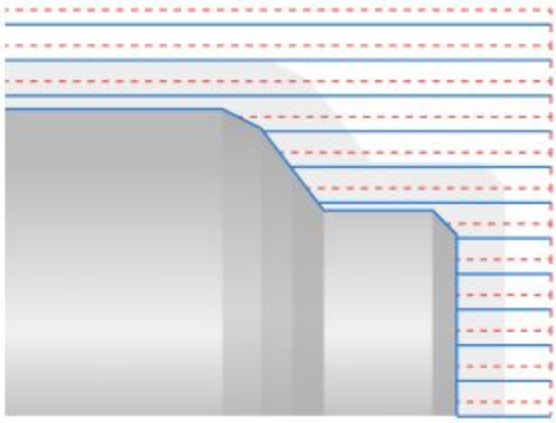
Axial displacement. Shifts the Start Point along the Z-axis in the direction corresponding to the increment sign.

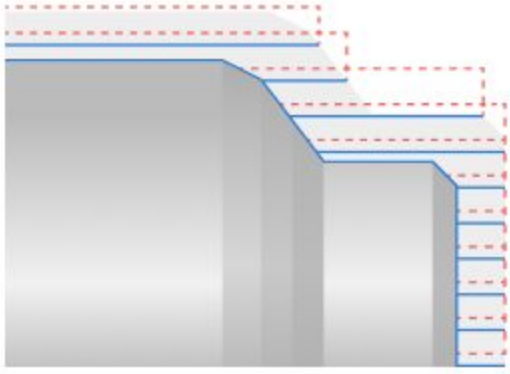


Ai - Axial increment

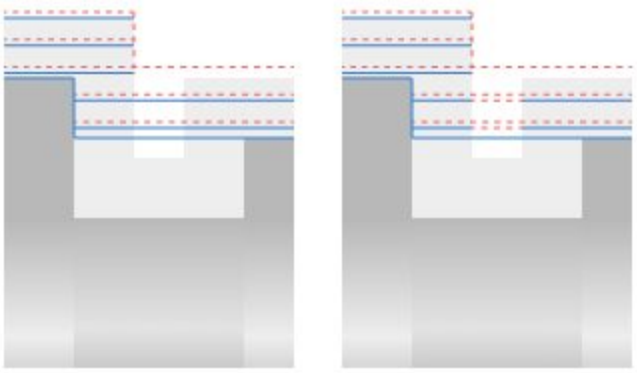
Check workpiece. When the flag is set, the system takes into account the current state of the workpiece. This significantly reduces machining time, especially if the shape of the allowance is complex. Details

▼ Click here to expand..

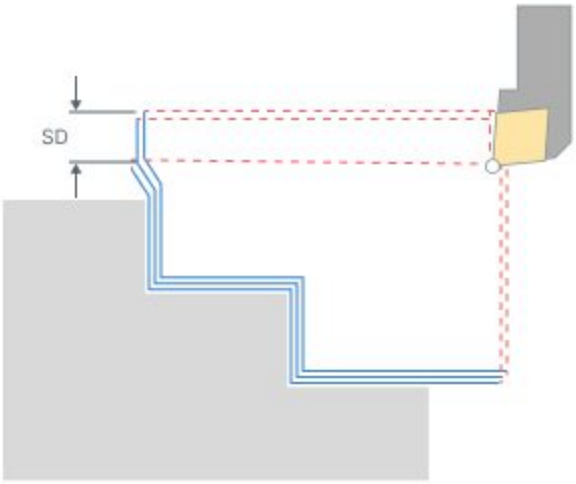




Ignore gaps. Allows you to set the maximum distance of tool pass through the air at the working feed. If the length between two processing areas is less than this distance, the system will treat them as one processing area.

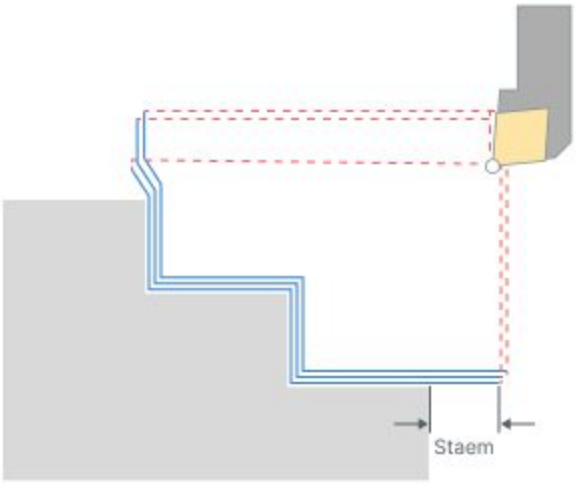


Safe distance. Specifies the distance of tool retract after a rough pass. At this distance from the machined surface, the tool returns to its initial position.



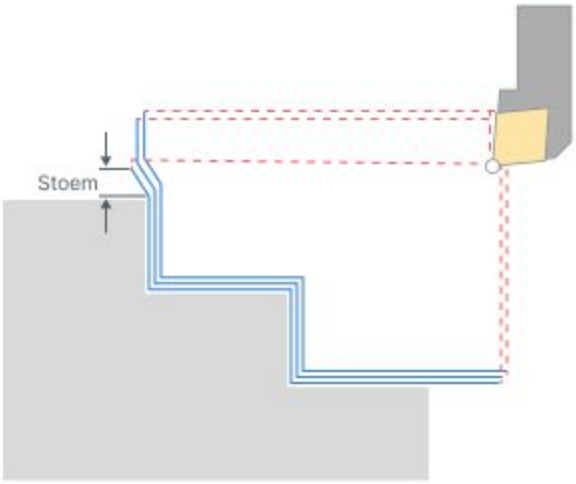
SD - Safe distance

Start entry amount. This parameter extends the beginning of the toolpath of the working passes.



Staem - Start entry amount

Stop entry amount. This parameter extends the end of the toolpath of the working passes.



Stoem - Stop entry amount

Max traversal distance. Specifies the maximum distance that the tool can pass on the traversal feed. If the distance between areas exceeds this parameter, the system would treat them separately.
Compensation: This parameter controls tool offsets and the method of outputting the tool nose radius correction. Details

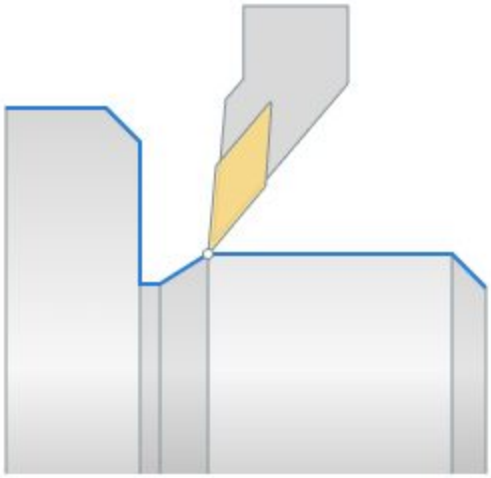
▼ Click here to expand..
Length corrector. Controls the output to the program of tool offset numbers that are specified in the Tools – Tool Properties – Numbers.

First. If the First parameter is selected, the system outputs the 1st corrector's number to the NC program.

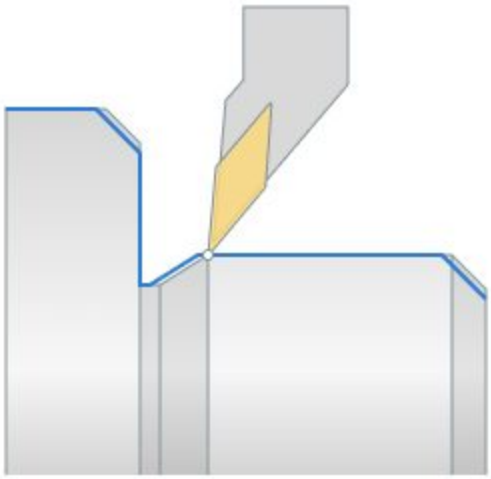
Second. If the Second parameter is selected, the system outputs the 2nd corrector's number to the NC program.

Compensation. Controls tool nose radius compensation.

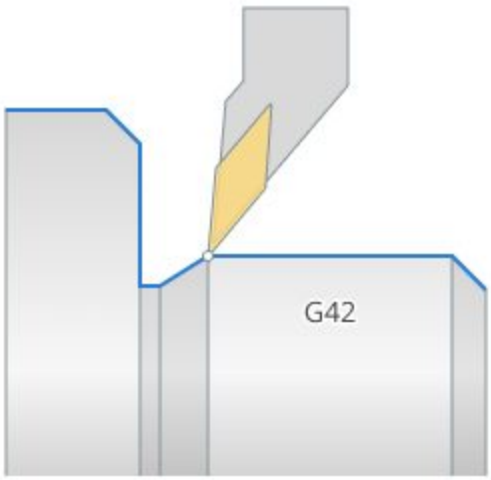
Off. The contour of the toolpath corresponds to the contour of the part and is output to the NC program without considering the tool nose radius. When using this mode on complex contours, the undercuts or incomplete machining are possible, so it is necessary to be carefully when machining simulation.



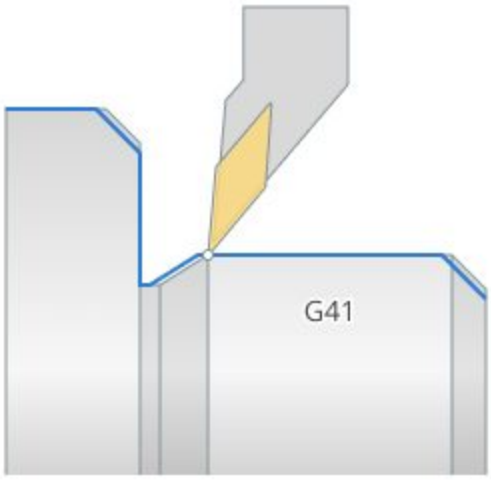
Computer. When traversing segments of the part contour that are not linear sections parallel to the coordinate axes, the toolpath shifts by the tool nose radius.



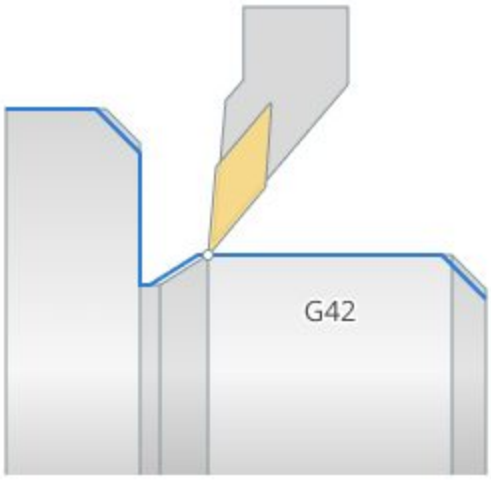
Control. The contour of the toolpath corresponds to the contour of the part and is output to the NC program without considering the tool nose radius. The system includes the commands for enabling and disabling correction (ISO G41/42/40) before the engage and retract frames respectively.



Wear. The system transforms the part contour geometry to the toolpath as in the Computer mode, but also includes commands for enabling and disabling correction, as for the Control mode. The tool moves closer to the part by the tool wear value entered in the NC console.



Reverse wear. It differs from the Wear mode in the direction of correction - G42 instead of G41, or vice versa. The tool moves away from the part by the tool wear value entered in the NC console.

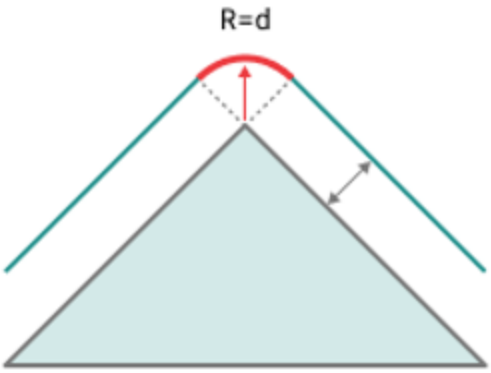


The parameters for the Zigzag cycle works similarly to the OD Grooving, ID Grooving and Face grooving operations. Details

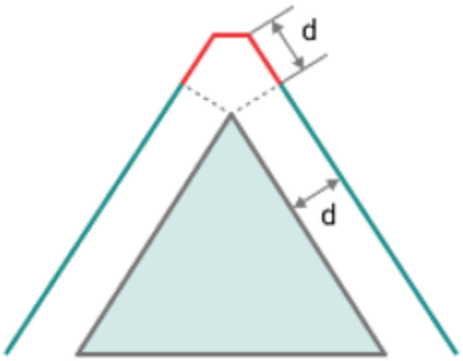
Roll by arcs.

▼ Click here to expand..

If enabled: When rolling by arc, an arc whose radius is equal to the offset value will be inserted into the toolpath.



If disabled: In the tangent mode, linear moves are used to transit the corner. The tool path before and after the corner is extended by the offset length. If the extended path intersects then it is truncated to the intersection point, else a connection move is created.



Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

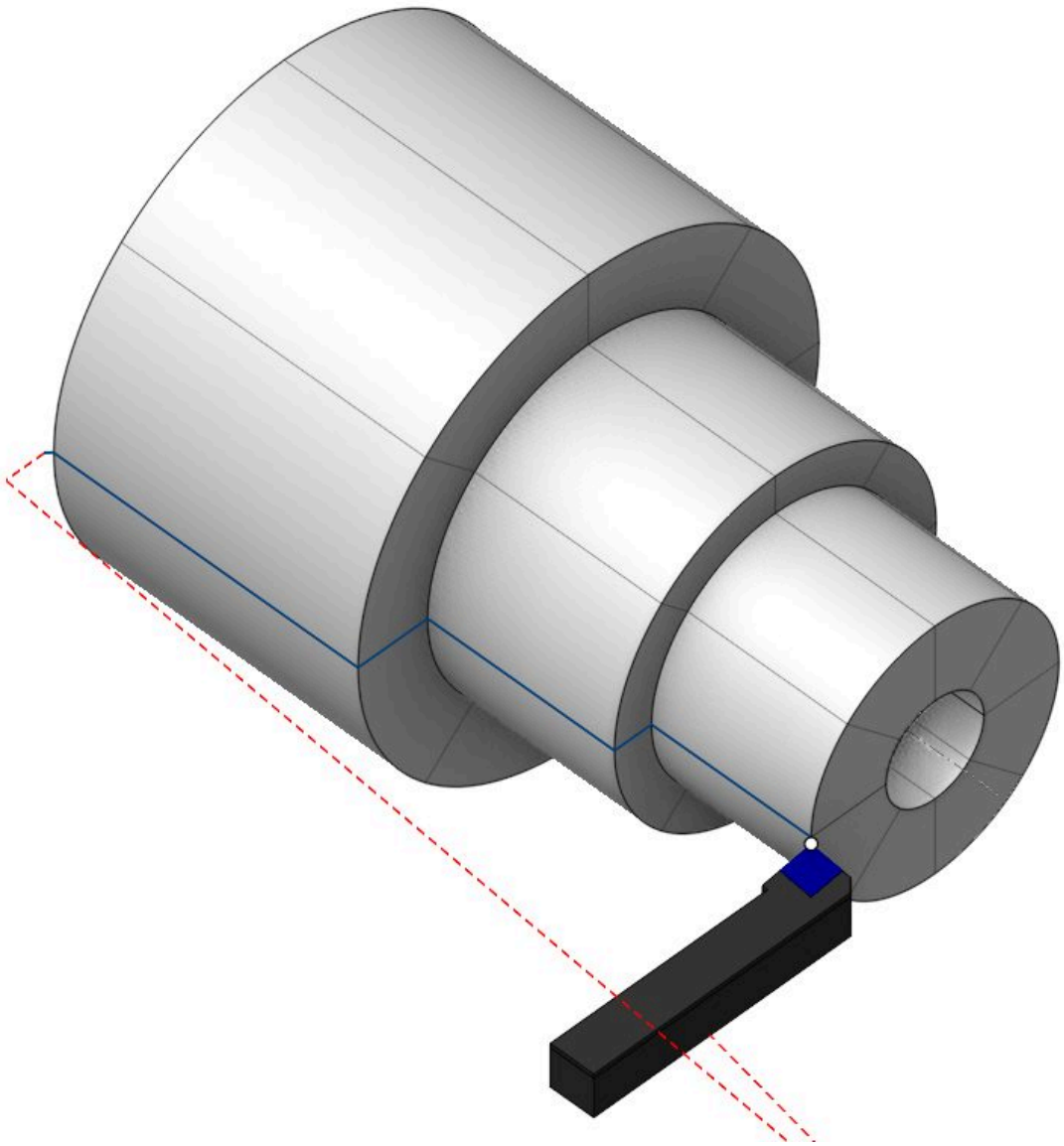
Workpiece:

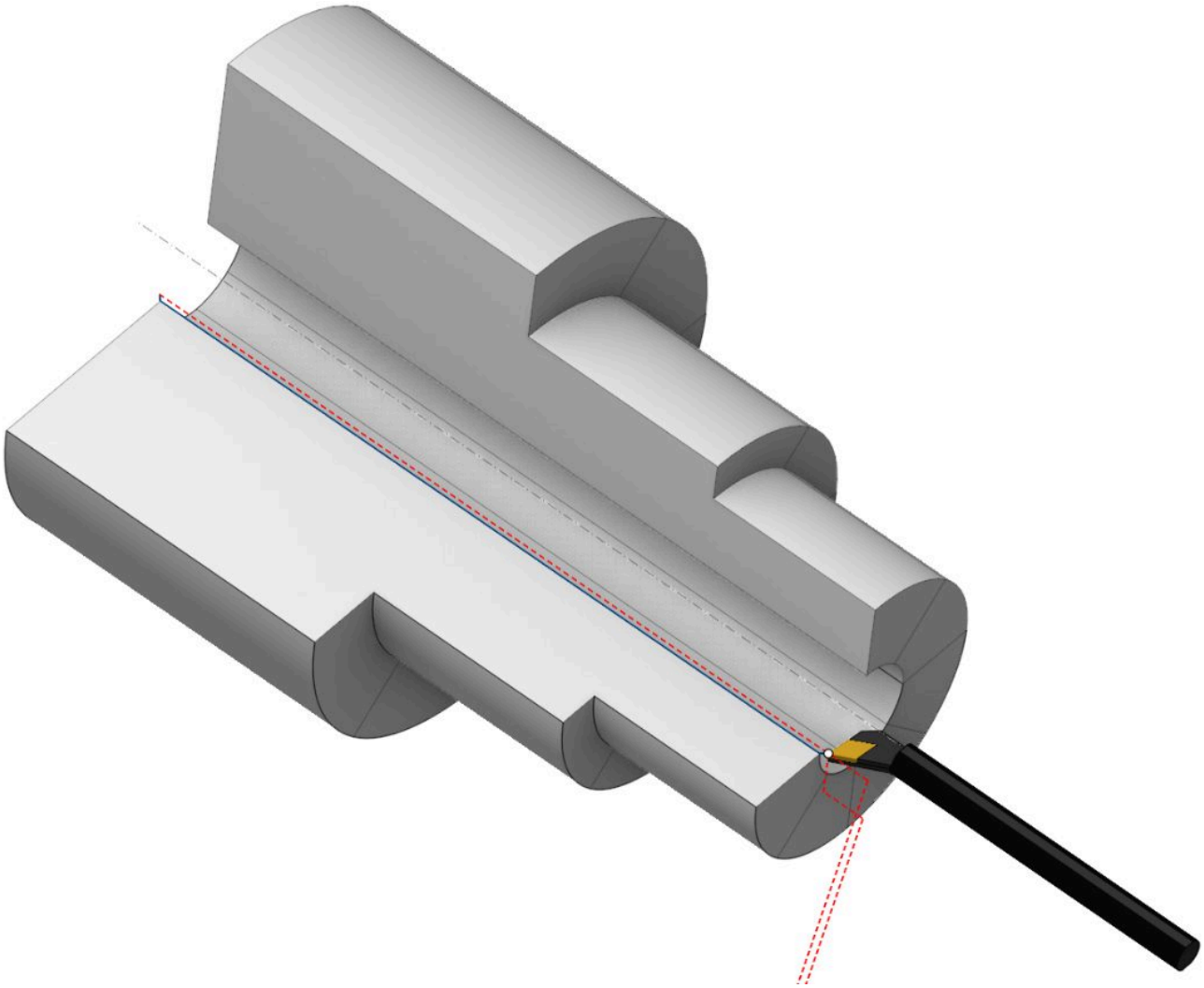
A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

OD Finishing and ID Finishing





Application area:

The lathe finishing operations are designed for the removing of a small stock volume that is remained after the previous machining. The machining is performed by the series of the offset strokes to the part generatrix.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Roughing. The roughing allowance is removed by tool movements along the axis of the workpiece (when processing surfaces elongated in the direction of the Z-axis) or perpendicular to it (when processing surfaces elongated in the direction of the X-axis). In other words, a trajectory is formed similar to ISO G71/G72, but it is output to the control program in an expanded form, without the use of a cycle. Details

Roughing cycle. Similar to rough machining, but the operation is output to the control program in the form of a cycle, similar to ISO G71/G72. The roughing cycle allows the creation of one of the ISO G71 or ISO G72 cycles based on the specified contour of the workpiece. Switching between G71 and G72 is achieved by changing the Direction of tool movements (along the axis of the workpiece Axial or perpendicular to it Radial) in the parameters. Details

Offset Roughing. The roughing allowance is removed by tool movements equidistant to the contour of the workpiece. In other words, a trajectory is formed similar to ISO G73, but it is output to the control program in an expanded form, without the use of a cycle. Details

Offset cycle. Similar Offset Roughing, the operation is output to the control program in the form of a cycle, similar to ISO G73. Details

Zigzag. This process is used for machining enclosed areas from both sides, using a neutral tool. The tool is engaged by plunging downwards and performs a horizontal movement to remove the next layer. The tool re-engages by plunging again and makes a horizontal pass in the opposite direction. This way, the entire roughing allowance is removed layer by layer through horizontal passes with alternating directions. Details

Profile. This is the simplest type of cycle, where a single pass is generated, and its trajectory is identical to the specified initial geometric contour of the workpiece. Only minor modifications to the contour can be performed, including approach and retract linking, offsetting by specified allowances, excluding shaded areas that cannot be reached by the designated tool, and compensating for the tool radius. Details

Facing. This allows programming the machining of the end face of a workpiece with a single command. The trajectory is formed in the same way as when specifying Contour or Contour Repeat cycles. However, the trajectory is automatically calculated based on the current blank and workpiece, ensuring correct machining of either the left or right end face. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

CycleParams:

This parameter group works similarly to the OD Roughing and ID Roughing operations. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations

per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

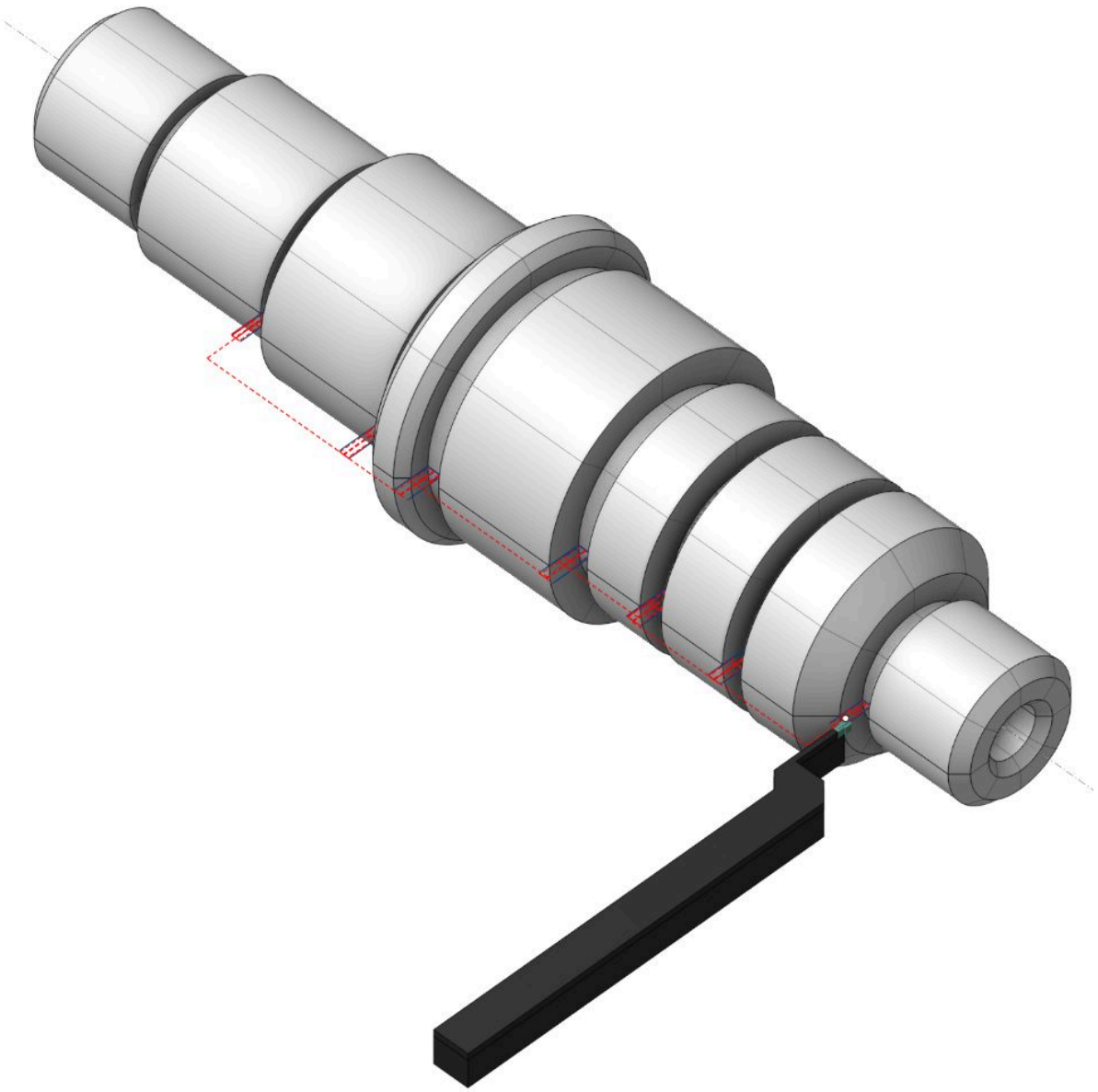
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

OD Grooving, ID Grooving and Face grooving



Application area:

This operation turns grooves or closed areas that are inaccessible for processing by other types of turning operations. Rough machining consists of a sequence of passes perpendicular to the axis of the part. Finish passes are the movements that closely follow the contour of the part.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Slotting. This cycle efficiently machines simple rectangular grooves with a minimum number of actions. Details

Grooving . This cycle creates a complex toolpath for turning all types of grooves and undercuts. The cycle identifies the lowest point of the groove and processes both its sides with different cutting edges of the tool. Details

Zigzag. The tool removes the rough stock layer by layer. In each layer, it plunges downwards to the cutting depth and performs horizontal movement. Thus, horizontal passes with alternating directions remove the entire rough stock, layer by layer. Details

Profile. The system forms a single pass, the toolpath of which is identical to the geometric contour of the part. User can add engage/retract; shifting by stock values; tool unreachable gray zones excluding; tool radius compensating to the toolpath. Details

Properties. Displays the properties of the selected element. You can additionally add allowance. You can call the Properties menu by double-clicking on the selected element.

Delete. Removes the selected item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Cycle parameters:

Cycle:

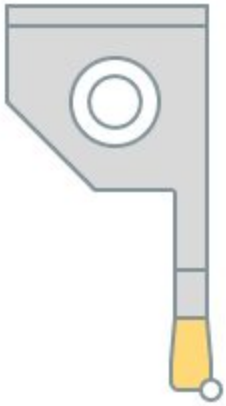
You can select one of the available turning cycles described in the Job assignment. When you switch a cycle in this section, the corresponding element in the Job assignment automatically changes. The set of parameters varies for each turning cycle.

▼ Click here to expand..
Machined Side. Specifies the sides of the groove to process. Details

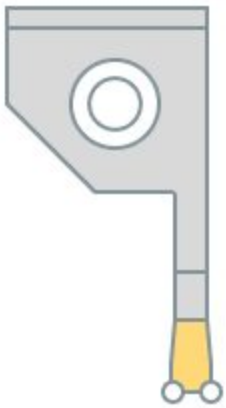
▼ Click here to expand..
1st tool tip. The system forms the toolpath from the specified Rough passes direction to the side the tool can touch with its first setting point.



2st tool tip. The system forms the toolpath from the specified Rough passes direction to the side the tool can touch with its second setting point.



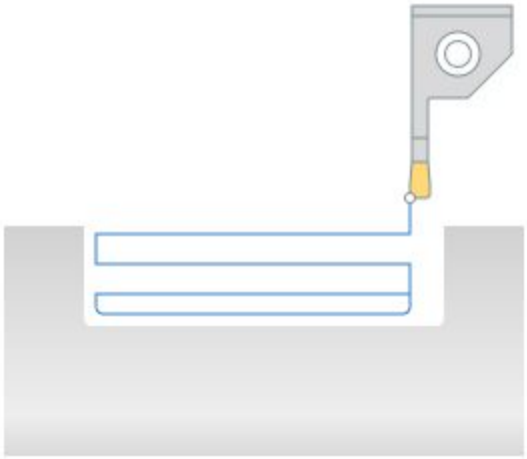
Both tool tips. The system forms the toolpath to machine both sides of the groove.



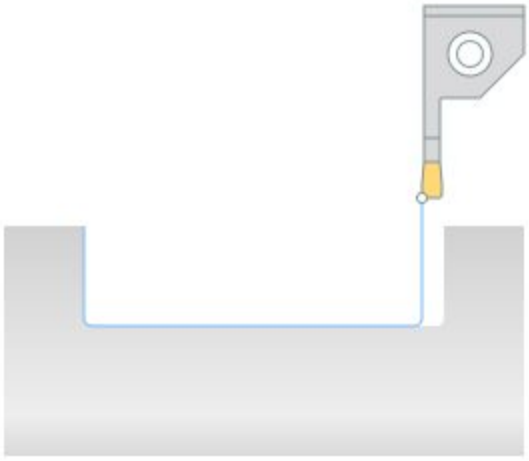
Tool path. This parameter allows choosing for which passes – roughing, finishing, or both – the system is generating the toolpath. Details

▼ Click here to expand..

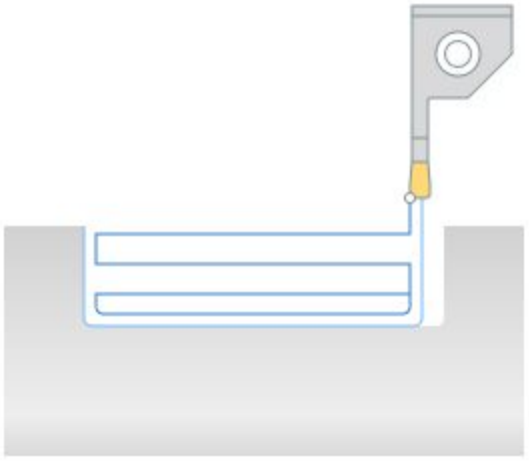
Rough only. The system generates the toolpath only for rough machining. The parameters window shows only the parameters that affect this toolpath.



Finish only. The system generates the toolpath only for finish machining. The parameters window shows only the parameters that affect this toolpath.

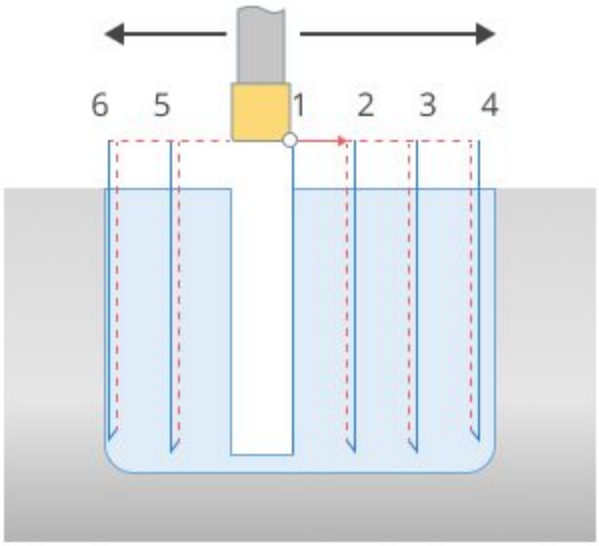


Rough and finish. The same tool first performs rough passes, then – finishing ones.

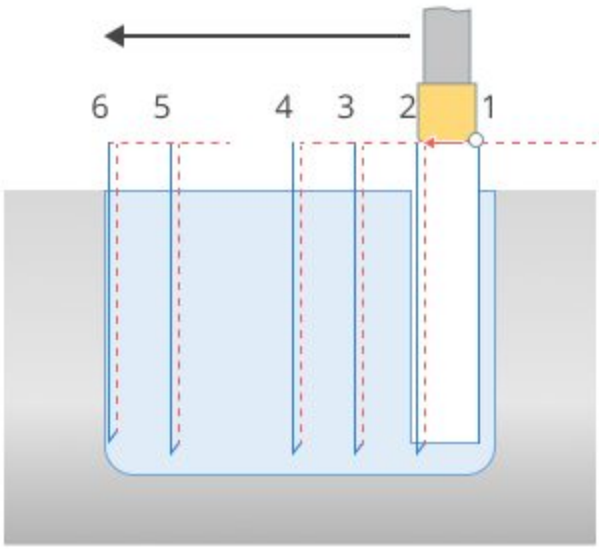


Rough passes direction . Specifies the direction in which the tool performs passes during rough machining of the groove.

▼ Click here to expand..
From center. Rough passes start at the deepest part of the groove and move outward to the groove sides.

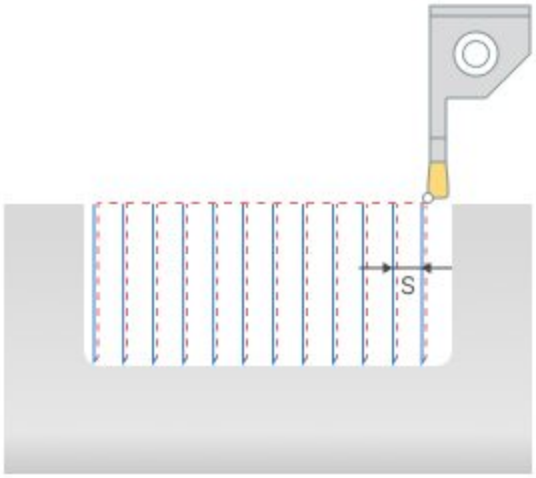


Forward. Rough passes proceed in one direction – from the starting point of the Job Assignment to the end.



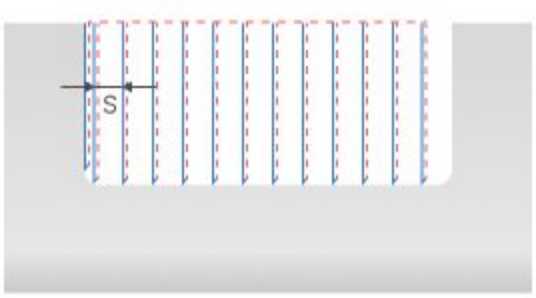
Rough step. The distance between tool passes is equal to the cutting depth.

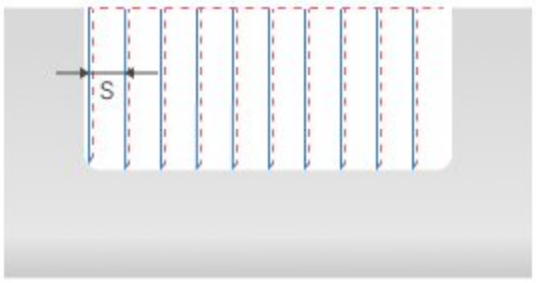
▼ Click here to expand..



S - Rough step

Adjust step. When the flag is set, the system automatically adjusts the step for consistent cutting force on each plunge.

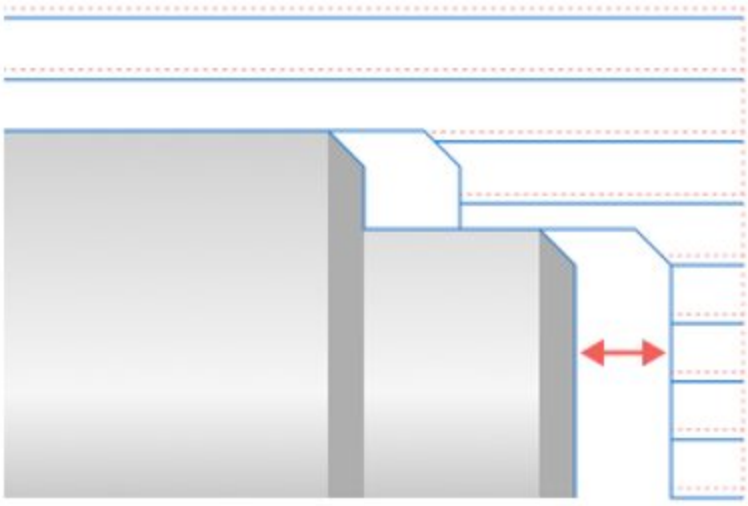




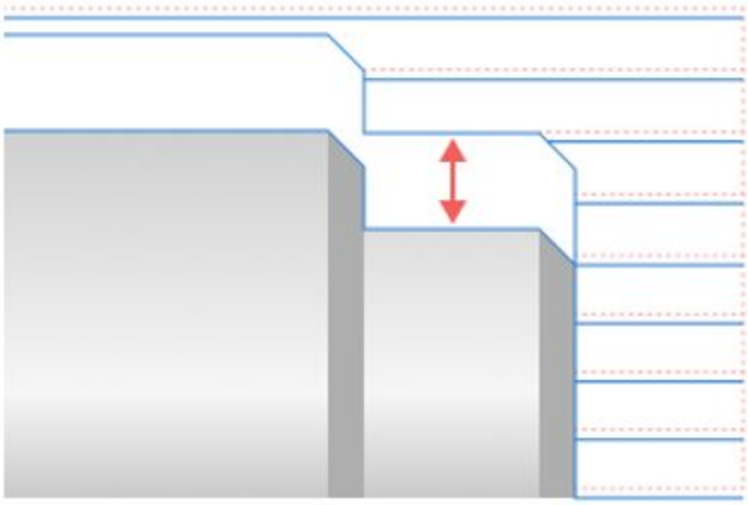
S - Rough step

Max step deviation. Defines the maximal deviation of the adjusted step from the defined step. This value can be defined in the length units or in the percents of the tool width.
Rough stock. This parameter leaves this allowance for subsequent machining. If the Finish pass flag is set, the system removes this allowance in the same operation.

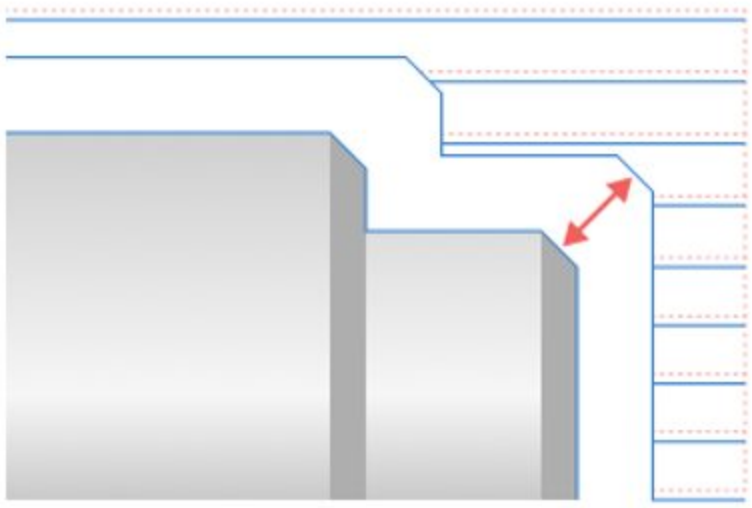
▼ Click here to expand..
Side stock. A stock of a specified size appears on faces. On conical and spherical surfaces, the allowance decreases in accordance with the angle of the surface normal. When setting both the radial and the axial stocks simultaneously, they combine quadratically on these surfaces.



Bottom stock. A stock of a specified size appears on cylindrical surfaces. On conical and spherical surfaces, the allowance decreases in accordance with the angle of the surface normal. When setting both the radial and the axial stocks simultaneously, they combine quadratically on these surfaces.



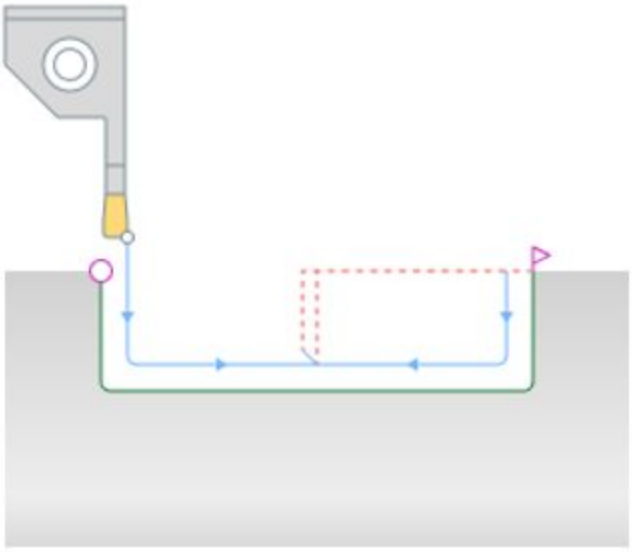
Profile. The toolpath with this allowance passes equidistantly to the contour of the part by a specified amount. All allowances combine when you set them simultaneously.



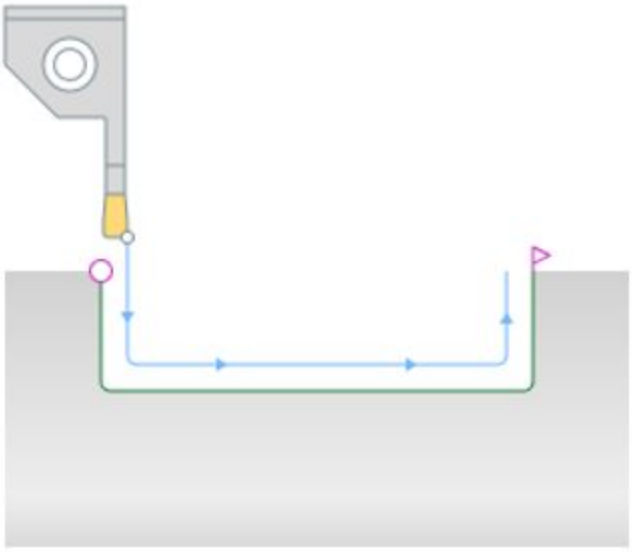
Finish pass direction. Determines the sequence of movements during the finishing pass.

▼ Click here to expand..

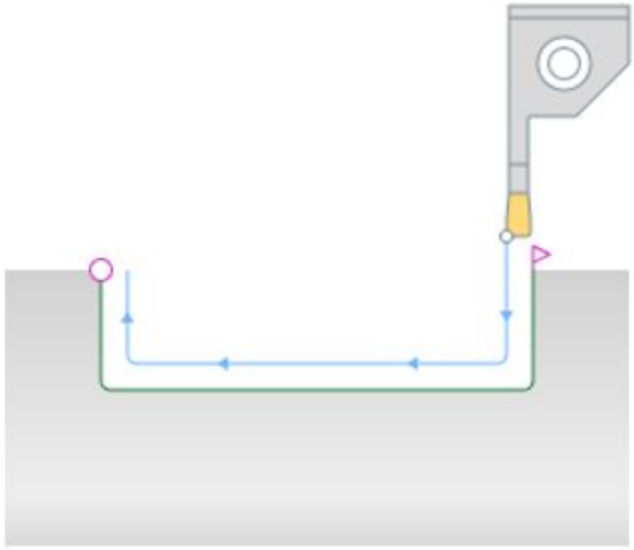
To center. The tool passes from the periphery to the center during the finishing. It processes the sides of the surface to be machined using different strokes.



Forward. The tool processes finishing in one stroke, from the starting point of the Job assignment to the ending point.



Backward. The tool processes finishing in one stroke, from the end of the Job assignment to the start point.



Canned cycle. This parameter determines the output of a groove canned cycle in the NC program.

▼ Click here to expand..

Do not use. The toolpath does not format as an ISO G75 canned cycle in the NC program.

Use canned cycles. The tool path formats as an ISO G75 canned cycle in the NC program.

Separate 1st cut. The tool's first pass formats as a separate ISO G75 canned cycle, while subsequent passes format as a second separate ISO G75 canned cycle..

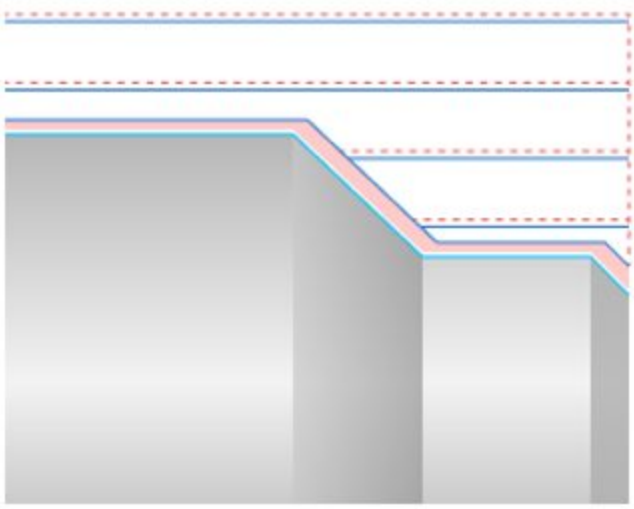
Separate every cut. Every tool pass formats as a separate ISO G75 canned cycle.
Multilayer. This technique facilitates the multi-layer removal of rough material. Upon flag activation, the tool makes cross passes to remove the first layer, then moves to the second layer, and continues accordingly.

▼ Click here to expand..
Return Type. This specifies the elevation level for the tool during idle movements between layers in a single groove. The Safe Distance parameter sets this level.

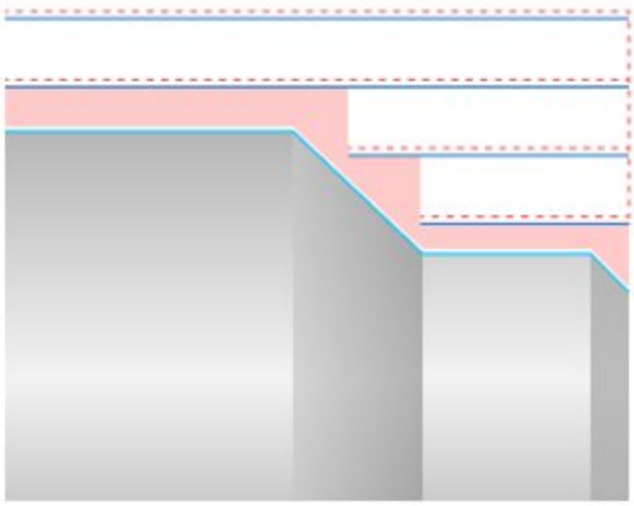
Count. Specify directly the number of layers that split the groove's depth.

Overlap. The system performs a semi-finish cuts to remove the ridges remaining after the roughing passes. Details

▼ Click here to expand..
On. When the flag is set, the system performs a semi-finish pass to remove the ridges remaining after the roughing passes.



Off. The additional tool path is not generated



Auto. In Auto mode, the Overlap option is enabled when the Multilayer option is enabled.
Chip breaking. The system controls the mode of kinematic chip breaking in which, after a specified cutting length, the cutter retracts to stop the formation of continuous chip. Then the cutter engages again and continues cutting.

▼ Click here to expand..
Off. The system has disabled the chip breaking mode.

On first plunge. The system activates the kinematic chip breaking mode only during the first rough pass of each machining layer.

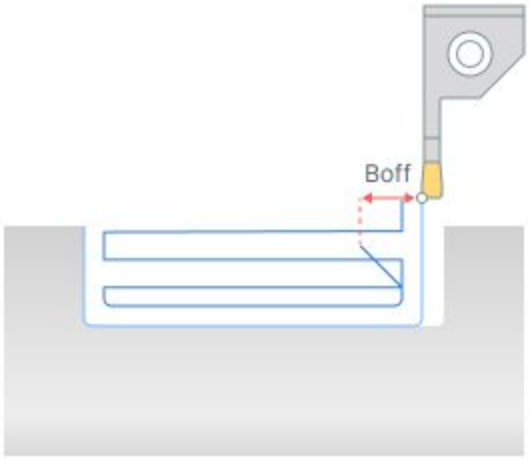
Break count. Specify the number of cutter stops directly.

Return (% of Plunge). Specify the cutter's return distance after stopping for chip breaking as a percentage of the Step Depth.

On every plunge. The system activates the kinematic chip breaking mode for every rough pass.

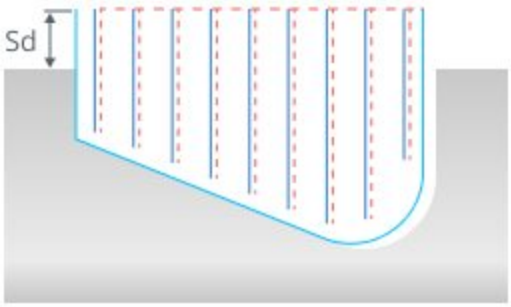
Delay at the bottom. This feature controls feed stop at the bottom of the groove to create an even groove bottom. Activating this feature introduces a stop duration.

Back off. The system steers the cutter away from the processed surface when retracting after rough cuts, by pulling the tool aside, with the retraction distance measured perpendicularly to the direction of the cut.



Boff - Back off

Safe distance. The distance from the top level of the groove to the level of rapid movements. You can edit the safety distance by dragging a point on the graphical screen or by directly input in the options window.



Check workpiece. When the flag is set, the system takes into account the current state of the workpiece. This significantly reduces machining time, especially if the shape of the allowance is complex. The parameters group works similarly to the OD Roughing and ID Roughing operations. Details

Compensation: This parameter controls the method of outputting cutter width compensation.

▼ Click here to expand..

Off. The system doesn't take into account the width of the insert, and outputs toolpath to the NC program without geometric transformations. When using this mode on complex contours, the undercuts or incomplete machining are possible, so it is necessary to be carefully when machining simulation.

Computer. The system takes into account the width and the tip radius of the cutter insert, and builds the toolpath according to the first setting point of the cutter.

Use 2 nd corrector. Machining the right side of the groove uses the second setting point. All other passes use the first setting point.

Roll by arcs.

The system rounds the external corners of the toolpath by arcs with a radius equal to the tool nose radius. The parameters group works similarly to the OD Roughing and ID Roughing operations. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

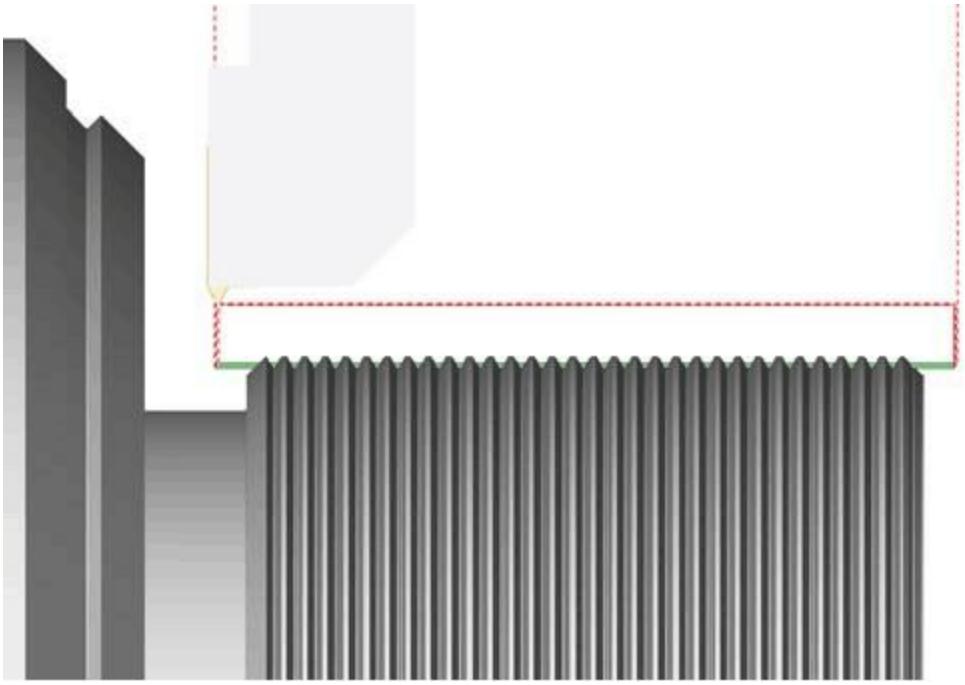
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

OD Threading and ID Threading



Application Area:

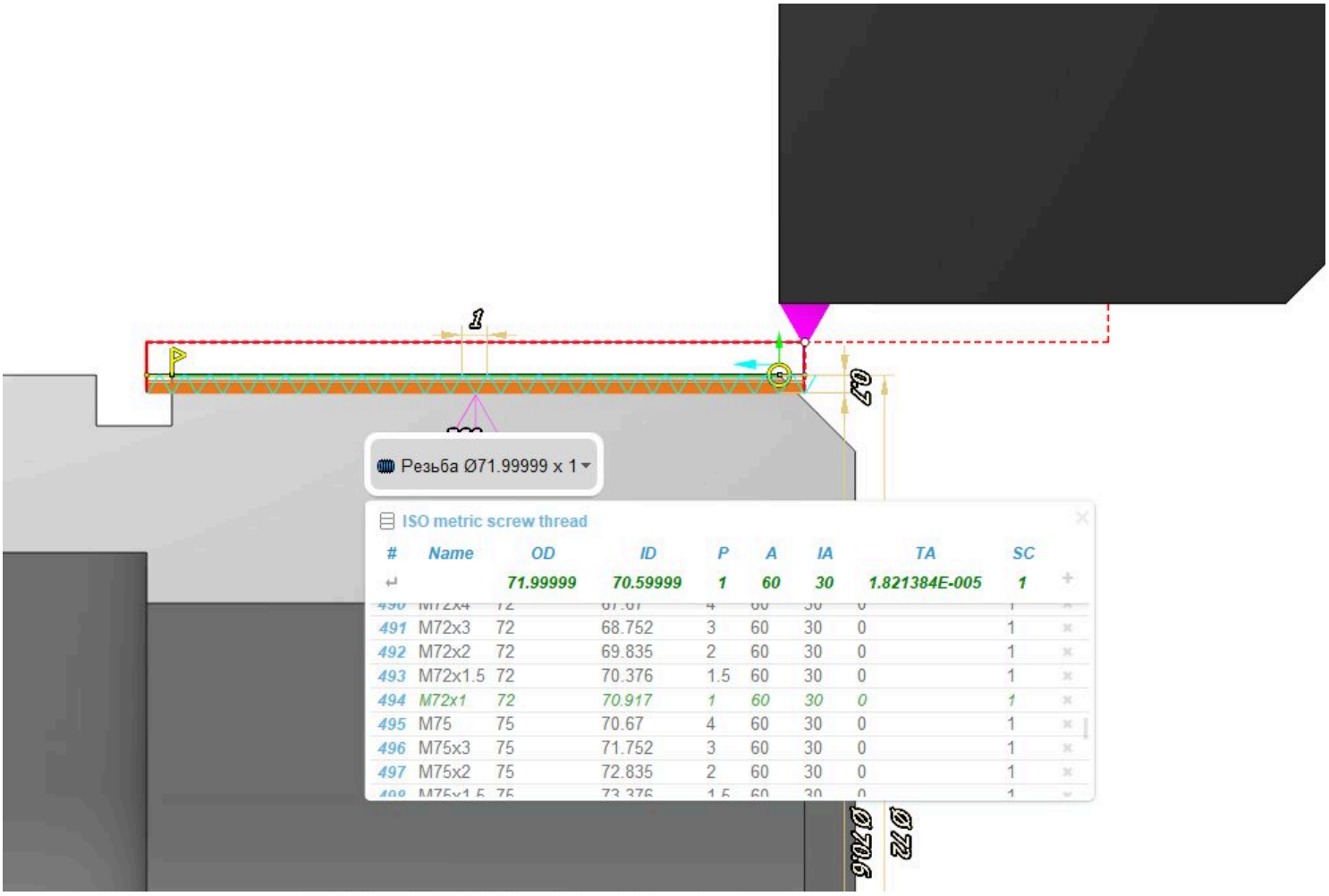
Lathe threading operations are designed for the threading by turn cutter or thread chaser. Helical surface forming is performed by the simultaneous spindle rotation and linear tool motion. Operation allows to thread as standard types of a thread (metric, inch, pipe threads etc.), and threads with a non-standard tooth profile depend on tool shape (diameters, step and angles can be difened manually). To produce a thread with desired specifications, one must take three main actions: identify the positions of the start and end points in relation to the workpiece, configure the geometric parameters of the thread, and select a strategy for its processing. The thread location is defined in the Job Assignment window. Where are a few steps to define the thread position: select the simple elements on a part geometry (curves and faces) where you want to make a thread, press the button with desired cycle type, then define the curve segment from the start to the end point. Each operation considers only one item from the Job Assignment. To process multiple threads, even with identical parameters, create as many operations as needed. By default thread diameters are taken from selected geometry. If you need to change it you can use properties of cycle or much easier to use graphical preview and threads table. Lathe threading cycles visualization exists. Interactive representation of the thread appears on the screen just after adding one of the threading cycles. Dimensions for thread pitch, depth top and bottom diameters, profile angles can be edited directly in the graphical screen. The simulation mode allows the checking of the thread shape. The cylindrical grooves are displayed instead the helical surface; because the simulation feature.

Job Assignment:G01 X70 Z5.0 F1.0 M08

Threading. This option determines the location of the threaded section on the part. When you select this item, the system automatically generates the thread line on the part profile. You can interactively modify this line in the graphical window by moving the start and end points, changing the thread direction arrow,

adjusting the threaded groove side arrow, and shifting the line to alter the external or internal diameter of the thread. Thread type - inner or outer - is specified by contour machining side (Perpendicular arrow near contour start point). Thread type - left or right - is specified by contour machining direction (Parallel arrow near contour start point) and spindle rotation direction. Contour approach and retract areas allows to set prolongation or chamfer for tool output. This feature activates continuous cylindrical, taper or face threading mode (G23/G33) with constant step. In this mode synchronization between tool movement and spindle rotation is enabled. All tool movements will processed in thread mode until the interpolation switching or rapid toolpath command will be detected. If tool moves parallel to the spindle rotation axis, cylindric thread will be formed. If tool moves both parallel and perpendicular to spindle rotation axis simultaneously, taper thread will be formed. It is possible to form special face thread, if tool moves perpendicular to spindle rotation axis. In this case groove looks like spiral of Archimedes will be formed at face. G32/G33 command does not generate any moves, so all working tool moves, approaches, retracts, transitions to the next passes should be specified separate parameters in the Strategy and Links tabs.

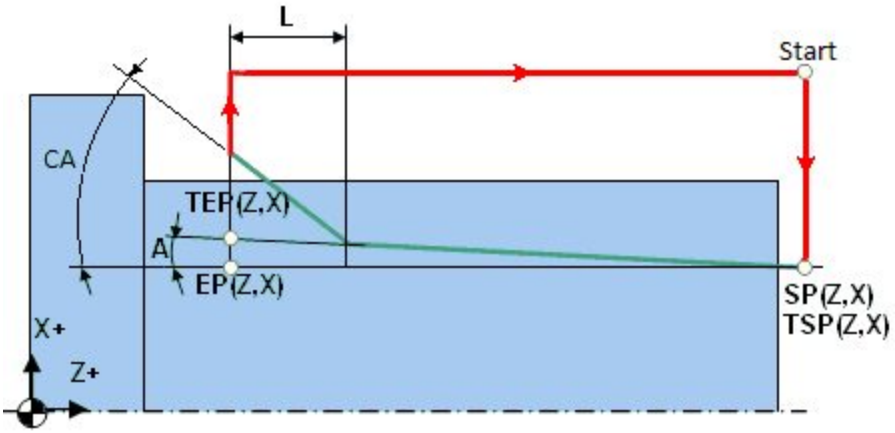
▼ Click here to expand..



> Example of NC program

```
G00 X60.0 Z10.0 M08 (Approach to Start point)
G00 X29.4           (Approach to start of pass 1)
G32 Z-52.0 F2.0     (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X28.9               (Approach to start of pass 2)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X28.5               (Approach to start of pass 3)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X28.1               (Approach to start of pass 4)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.8               (Approach to start of pass 5)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.56              (Approach to start of pass 6)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.36              (Approach to start of pass 7)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.26              (Approach to start of pass 8)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X200.0 Z150.0 M09   (Retract)
```

Single thread cycle. This option also serves to define the position of the threaded section on the part. The system will output the G92 cycle to the NC program. It generates closed set of moves for one threading pass. Picture below shows processing schema. Before calling this cycle tool is in Start point. Cycle is called by one frame of NC-program, defines thread start point, step, taper size, chamfer size and others. As a result of this frame the tool goes from Start to TSP point, thread to TEP point and returns to Start point. Usually threading is processed by several passes, so NC-program consists several cycle calls with various thread diameters. Therefore, it can be used for both standard-shaped threads and free-form threads.

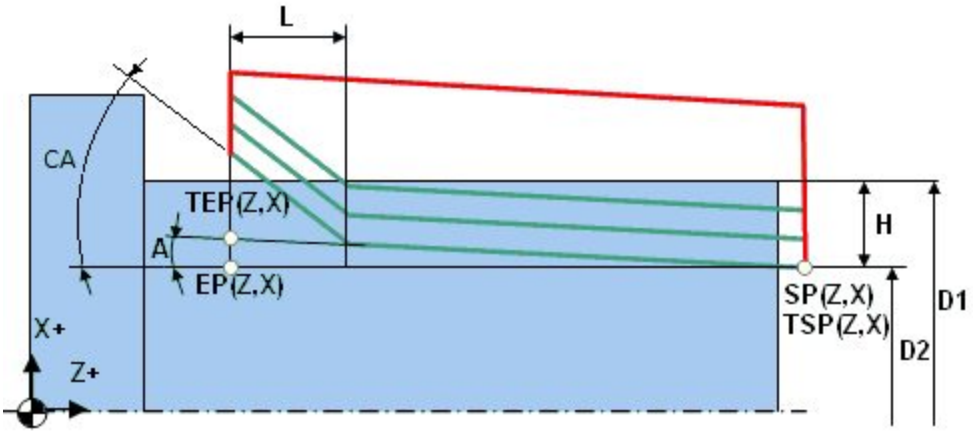


▼ Click here to expand..

> Example of NC program

```
X60.0 Z20.0 M08
G01 Z10.0 F1.0         (Approach to Start point)
G92 X29.4 Z-52.0 F2.0 (Calling cycle for one threading pass)
X28.9                  (Modal calling G92 cycle with another diameter value)
X28.5                  (Modal calling G92 cycle with another diameter value)
X28.1                  (Modal calling G92 cycle with another diameter value)
X27.8                  (Modal calling G92 cycle with another diameter value)
X27.56                 (Modal calling G92 cycle with another diameter value)
X27.36                 (Modal calling G92 cycle with another diameter value)
X27.26                 (Modal calling G92 cycle with another diameter value)
G00 X200.0Z150.0M09    (Retract)
```

Thread Cycle. This option also serves to define the position of the threaded section on the part. The system will output the ISO G76 multipass thread cycle to the NC program. It uses a single frame of the NC-program to set all parameters necessary for machining a standard form thread. Required depth is reached automatically by generating several passes. Among the parameters of the cycle there are start (TSP) and finish point (TEP) coordinates, taper angle (A, for taper threads), size of chamfer for tool out (L, CA), profile angles, thread depth (H), passes count, plunge strategy and others. You can configure these parameters interactively in the graphic window on the J ob Assignment tab or enter their numerical values on the Cycle Parameters tab. See NC control documentation for more information.



▼ Click here to expand..

> Example of NC program

```
G01 X70 Z5.0 F1.0 M08            (Approach to start point)
G01 X70 Z5.0 F1.0 M08            (Approach to start point)
G76 P010060
G76 X57.4 Z-24.0 P1.3 Q0.35 F2.0 (Calling G76 multipass thread cycle)
G00 X200.0 Z150.0 M09            (Retract)
```

Cycle Prarameters:

Cycle.

You can select one of the available turning cycles described in the Job Assignment. When you switch a cycle in this section, the corresponding element in the Job Assignment automatically changes. The set of parameters varies for each turning cycle.

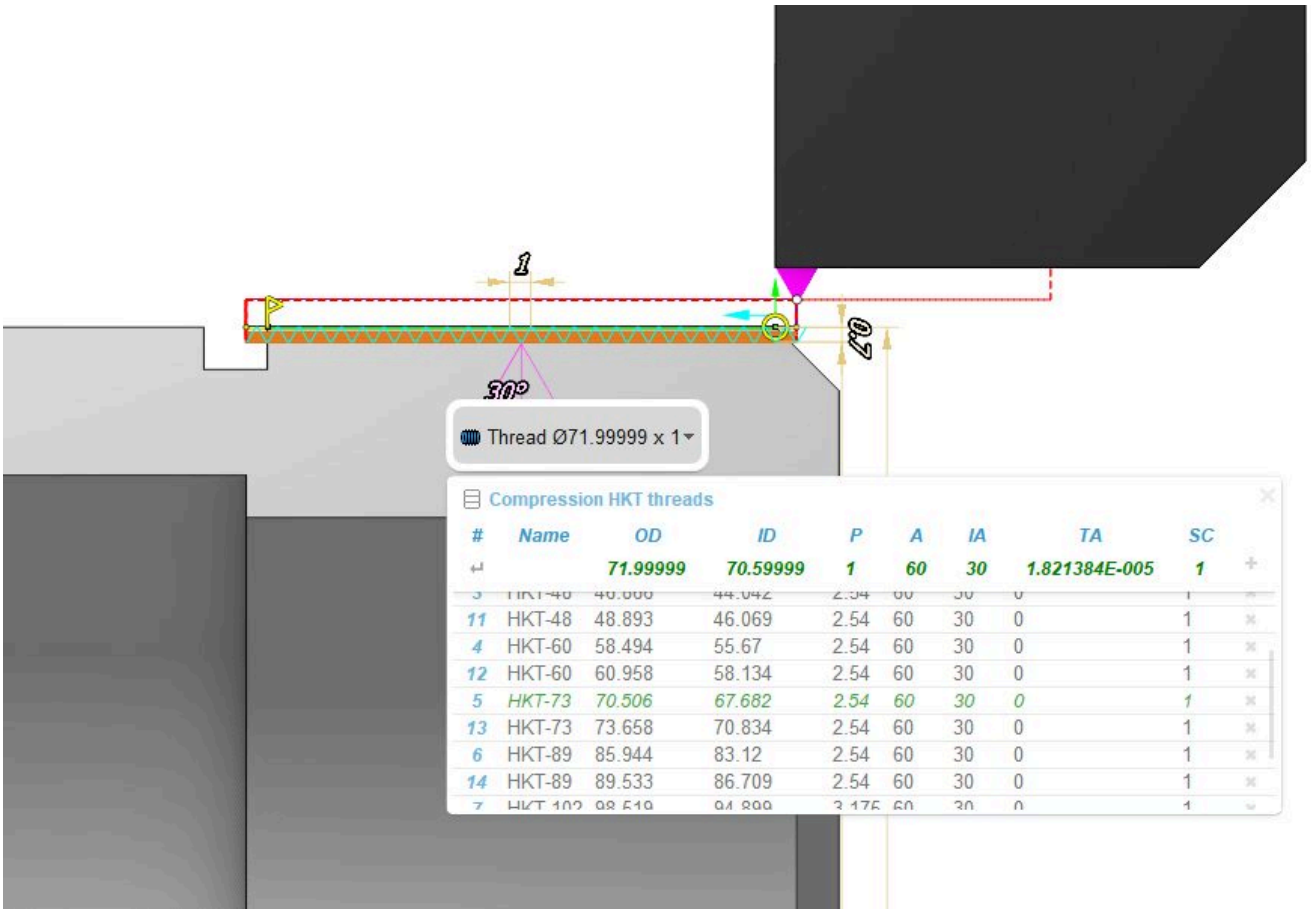
▼ Click here to expand..

Clearance Stock. Specifies the distance from the thread to the transitions level. The cycle goes through this level while moving from the end of the previous pass to the start of the next one.

Thread Form. The group configures the geometric parameters of the thread.

▼ Click here to expand..

Thread Libray. Select thread parameters from the database if the parameters for the thread conform to standard ones. There is no need to enter all parameters manually. Just establish the type of thread and select its designation from the dropdown list. The database will automatically complete the remaining parameters with standard values. You can access the thread parameter table from the Job Assignment tab by clicking the thread profile with the left mouse button. Pop-up action bar with the thread name, that appears when you select cycle item, allows to open threads table quickly and select one of the standard threads. The list of threads can be modified in this panel intuitively.



The operation with standard thread types remains identical in both cases:

▼ Click here to expand..
Click on the table header to select the desired thread type. Implement a database with your thread type by entering the file path in the "Add new file" field. The following fields appear in the table:

#. Record Number

Name. Thread Type Designation

OD. Outer Thread Diameter

ID. Inner Thread Diameter

P. Thread Pitch

A. Thread Profile Angle

IA. Inclination Angle - defines the inclination of one side of the thread profile relative to the vertical

TA. Taper Angle (for tapered threads)

SC. Number of Starts (for multi-start threads)

+. Allows you to add a new row. A new row appears after the row where the cursor is located.
Symbolic Name. It's just an optional symbolic designation of the thread type and size, M10 for example. You can use it to quick search the thread in the threads' table. You can enter your own designation for non-standard threads.

Thread Pitch. The distance between adjacent identical points of the thread profile on the one helical line. Specify the thread pitch either in length units or as the number of threads per length unit.

Depth. Defines thread profile height (difference between outer and inner diameters). This value must have positive value.

Thread Angle. Specifies t he angle between two adjacent sides of the groove profile.



Inclination Angle. Specifies the inclination of one profile side of the thread groove relative to the vertical.



The Thread Angle and Inclination Angle parameters define angle of tool plunge at each pass. if plunge mode is Flank or Alternative Flank. The thread profile formed during machining largely depends on the tool's shape and position. The profile angles set on the Thread Form page are used for toolpath calculation, but they do not synchronize with the tool orientation, so it is essential to monitor the tool properties specified on the Tool tab.

Chamfer. The tool exits the workpiece at the end of the cutting stroke. This feature guarantees an accurate completion of the screw thread surface if there is no exit groove for the cutting tool on the relevant side of the thread.

Auto by Retract. The system defines the tool exit using exit segments that they set interactively in the graphical interface of the Job Assignment tab.

Length. Specifies the length of the thread exit in length units.

% of Lead. Specifies the length of the thread exit as the percent of the Thread Pitch.

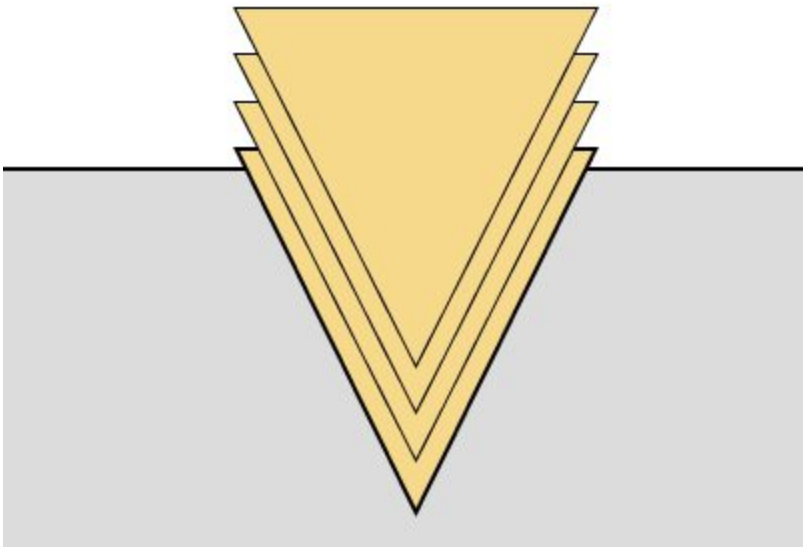
Angle. Specifies the angle of the exit section relative to the horizontal.

Number of starts. Specifying two or more starts results in the cutting of a multi-start thread. Various cycle types use Number of starts differently. In ISO G76 numbers of starts send to cycle as a parameter. But some machines have not this parameter in cycle. In this case it is possible to create multistart threads by making the same cycle with another Spindle start angle parameter value. Another way is using ISO

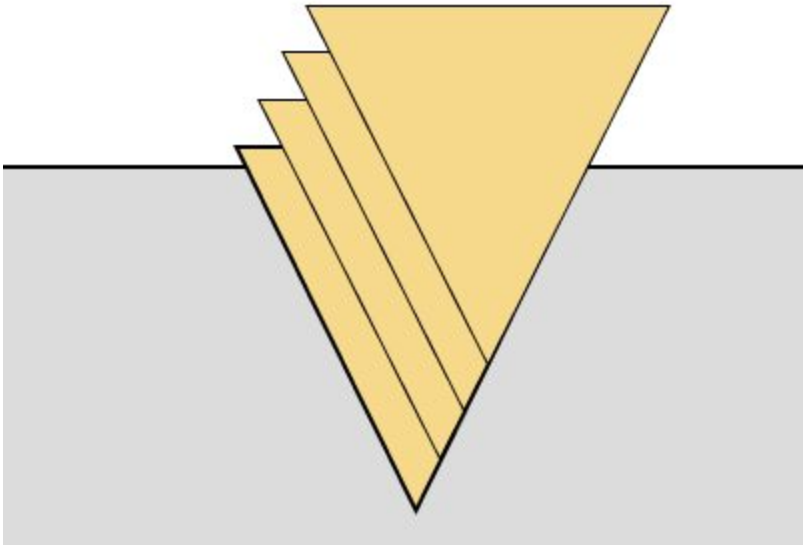
G92 or ISO G32/33. In this case operation automatically generates passes with different spindle start angles.

Spindle Start Angle. Specifies the angle from the X-axis where the thread's helical line cutting starts.
Sequence. Defines plunge mode for each pass. Practically thread is processed by several passes. It allows to improve surface quality and reduce tool loading.

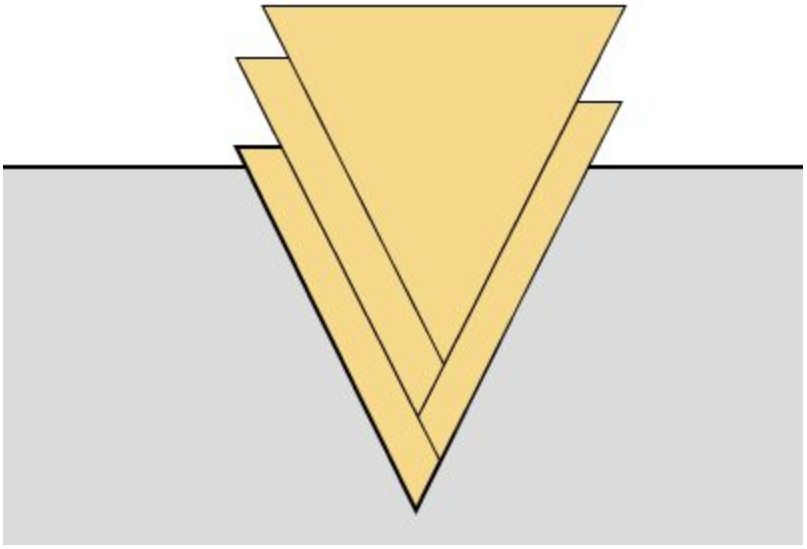
▼ Click here to expand..
Radial. The direction of plunge is perpendicular to the axis of rotation.



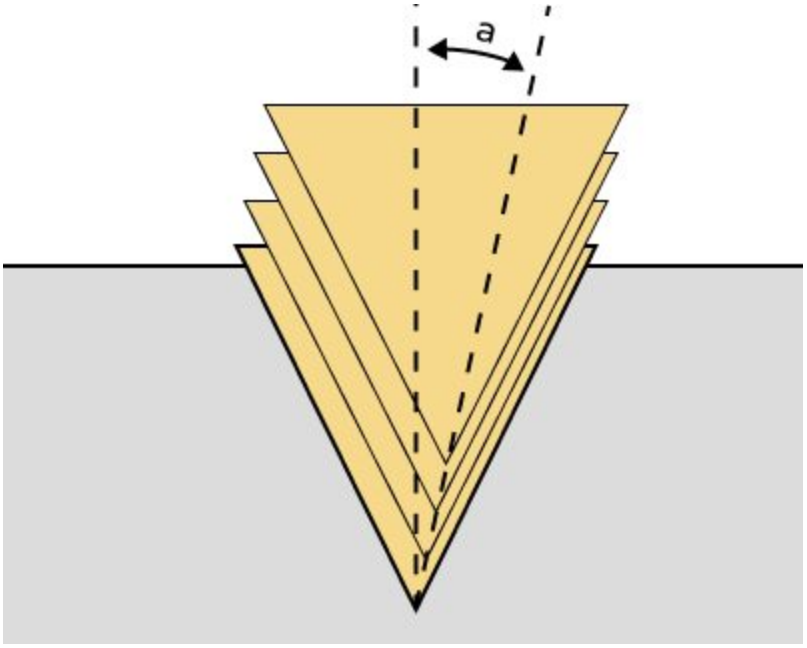
Flank. The plunge is made along one side of the thread groove.



Alternate Flank. Plunge is made alternately along the two lateral sides of the thread groove.

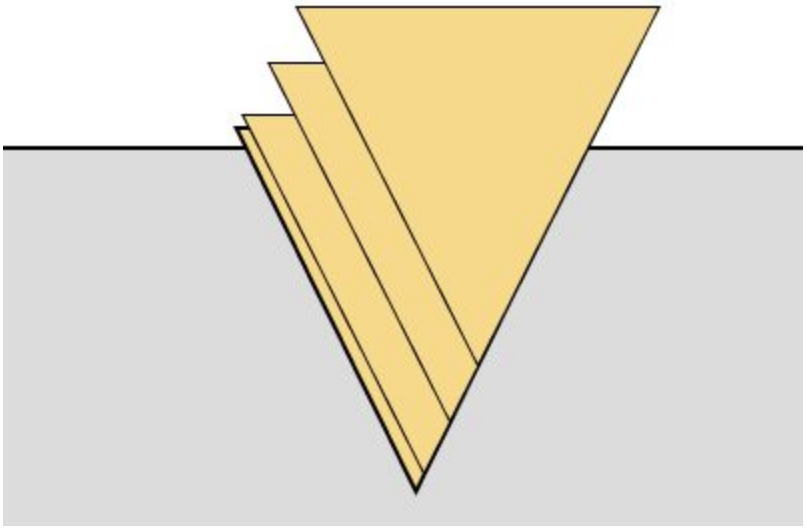


Modified flank. Plunge is made with angle, specified at Angle parameter.

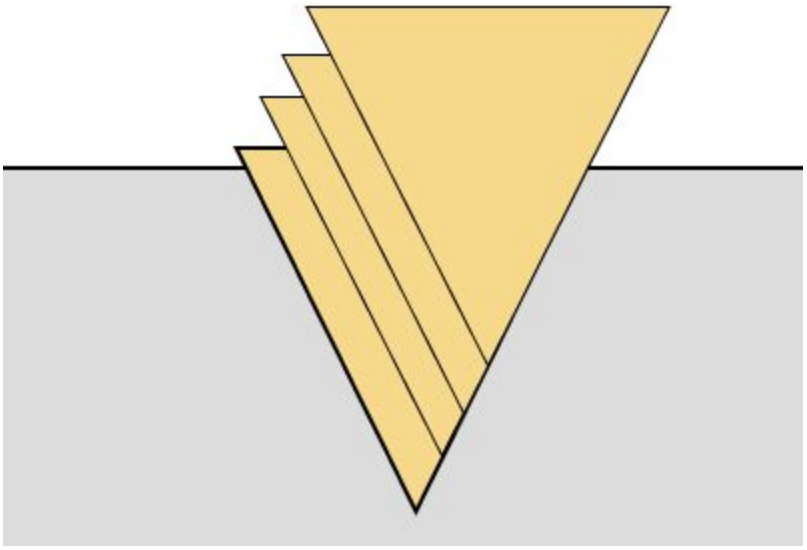


Determine Cut Depth From. Controls the method for setting cutting depths. When cutting depths are equal across different layers, advancing to the next layer usually alters the area of material removed by the tool, and consequently, changes the load on the tool. To maintain a consistent tool load, one can calculate the cutting depth based on keeping the area of material cut by the tool constant.

Equal Area. In this mode cutting depth decreases at each level t o ensure a constant removal area of the layer.



Equal depth. The cutting depth is constant at each layer.



You can set the number of rough passes by directly specifying their Number or by indicating the Amount of First Cut.

Number of Cuts. Specify the count of roughing passes.

Amount of First Cut. Specify the cutting depth for the first pass.

Minimal Cut Depth. If the calculated depth on any pass becomes less than this value, then the minimal cut depth will be used.

Finish Pass Count. Specify the count of finish passes. Usually, finishing passes aim to enhance the quality of the formed surface. In threading operations, there are two types of finishing passes: the direct

finishing pass and burnishing. You can enable or disable the finishing pass, but it is always one, with its depth specified in the relevant field. Burnishing, unlike the finishing pass, is performed with no cutting depth and effectively duplicates the toolpath of the previous pass.

Finish Pass Depth. Defines finish pass stock.
Compensation. This parameter controls the method of outputting cutter width compensation.

▼ Click here to expand..
Length corrector. Controls the output to the program of tool offset numbers that are specified in the Tools – Tool Properties – Numbers.

First. If the First parameter is selected, the system outputs the 1st corrector's number to the NC program.

Second. If the Second parameter is selected, the system outputs the 2nd corrector's number to the NC program.

Roll by arcs.

The system rounds the external corners of the toolpath by arcs with a radius equal to the tool nose radius. The parameters group works similarly to the OD Roughing and ID Roughing operations. Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

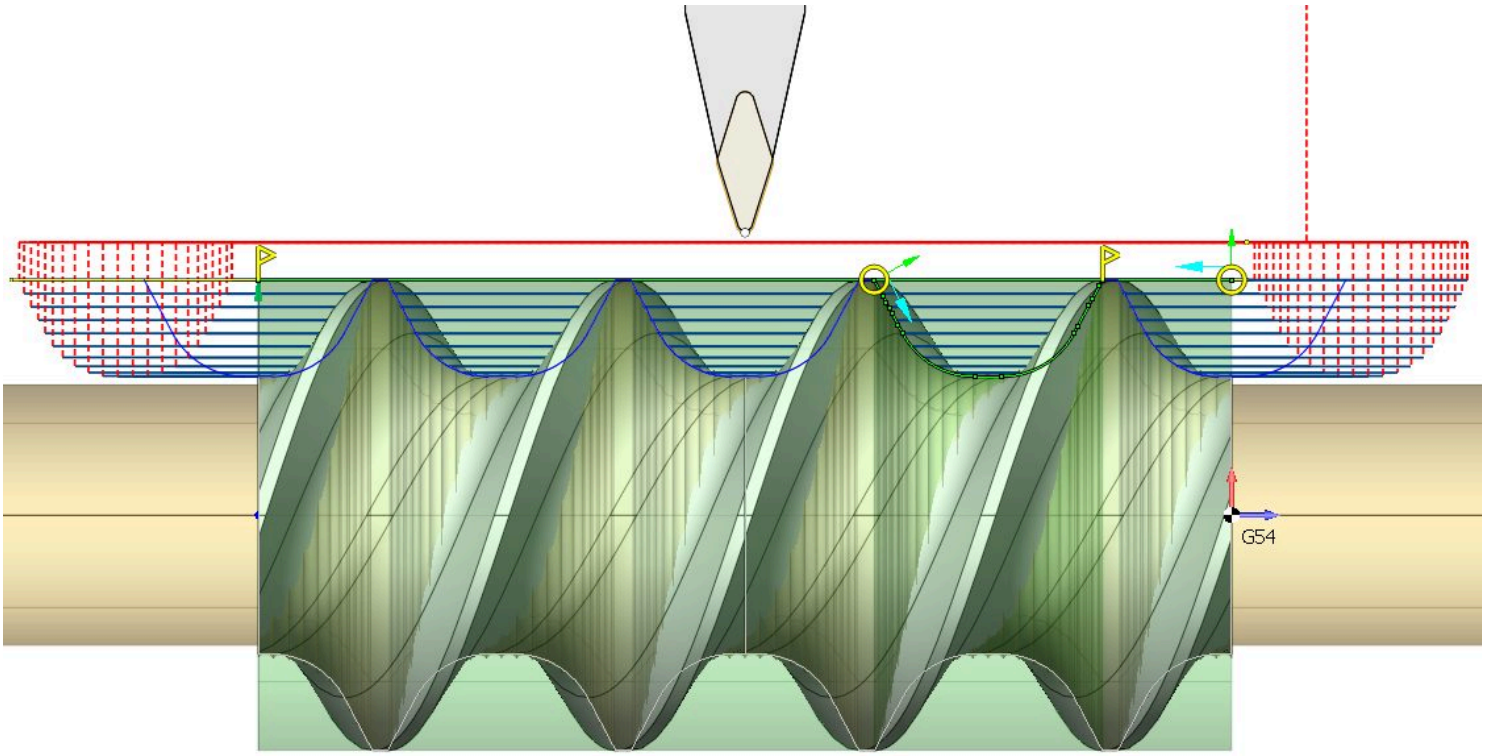
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

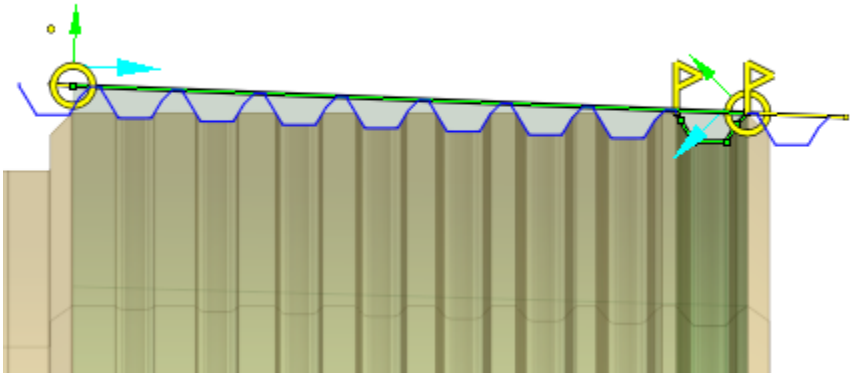
As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

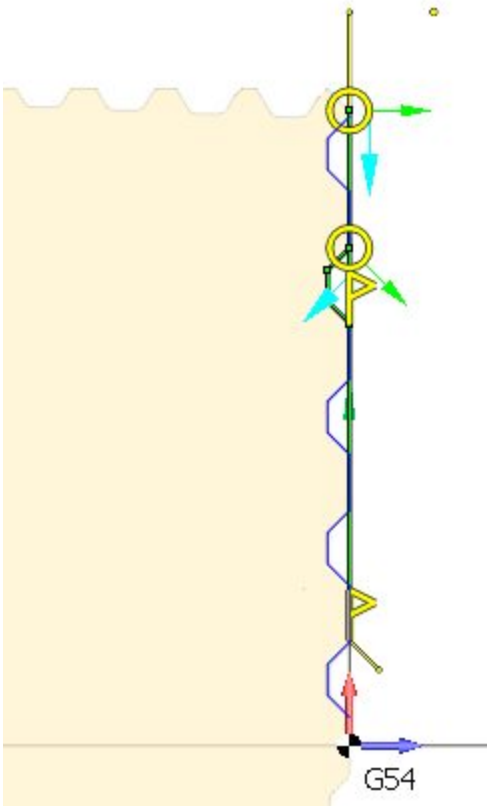
Profile Threading



Application Area:

The profile threading operation allows you to make threads with a profile shape different from the tool's profile. This is achieved by removing material inside the entire thread groove with a series of consecutive passes, the mutual arrangement of which is calculated taking into account both the shape of the tool and the shape of the threaded groove itself. You must define two items in the Job Assignment: an item that determines the position of the beginning and end of the thread on the part and an item that defines the shape of the thread groove. Depending on the orientation of the given profiles, you can get a cylindrical, conical or face thread (archimedean spiral).



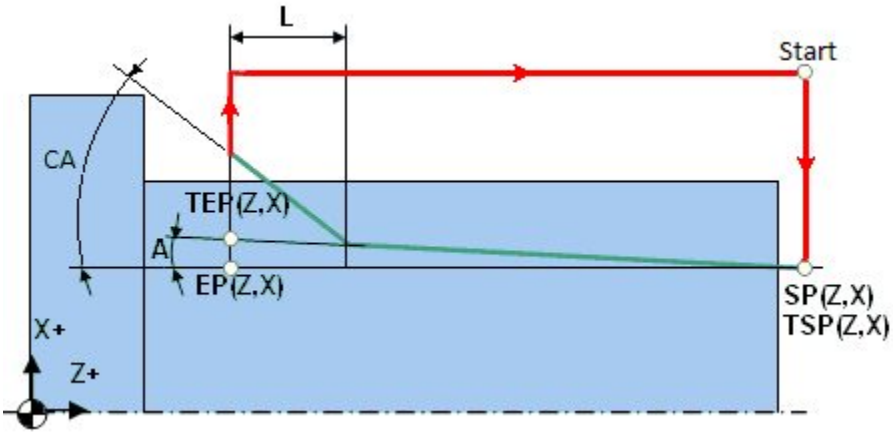


Job Assignment:

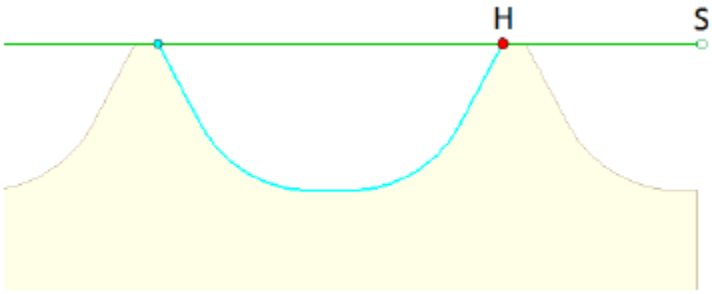
Threading. This option determines the location of the threaded section on the part. When you select this item, the system automatically generates the thread line on the part profile. You can interactively modify this line in the graphical window by moving the start and end points, changing the thread direction arrow, adjusting the threaded groove side arrow, and shifting the line to alter the external or internal diameter of the thread. Thread type - inner or outer - is specified by contour machining side (Perpendicular arrow near contour start point). Thread type - left or right - is specified by contour machining direction (Parallel arrow near contour start point) and spindle rotation direction. Contour approach and retract areas allows to set prolongation or chamfer for tool output. Each operation considers only one item from the Job Assignment. To process multiple threads, even with identical parameters, create as many operations as needed. This feature activates continuous cylindrical, taper or face threading mode (G23/G33) with constant step. In this mode synchronization between tool movement and spindle rotation is enabled. All tool movements will processed in thread mode until the interpolation switching or rapid toolpath command will be detected. If tool moves parallel to the spindle rotation axis, cylindric thread will be formed. If tool moves both parallel and perpendicular to spindle rotation axis simultaneously, taper thread will be formed. It is possible to form special face thread, if tool moves perpendicular to spindle rotation axis. In this case groove looks like spiral of Archimedes will be formed at face. G32/G33 command does not generate any moves, so all working tool moves, approaches, retracts, transitions to the next passes should be specified separate parameters in the Strategy and Links tabs.

Single thread cycle. This option also serves to define the position of the threaded section on the part. The system will output the G92 cycle to the NC program. It generates closed set of moves for one

threading pass. Picture below shows processing schema. Before calling this cycle tool is in Start point. Cycle is called by one frame of NC-program, defines thread start point, step, taper size, chamfer size and others. As a result of this frame the tool goes from Start to TSP point, thread to TEP point and returns to Start point. Usually threading is processed by several passes, so NC-program consists several cycle calls with various thread diameters. Therefore, it can be used for both standard-shaped threads and free-form threads.



Threaded groove. This feature identifies the contour that determines the thread groove profile. The system will repeat this contour along the Threading or the Single pass thread at intervals equal to the Thread Pitch specified in the Cycle Parameters tab. You can interacively input the Thread Pitch in the graphical window. It is important to check that the groove's direction and side correspond to the Threading's or the Single pass thread's similar parameters. In the figure, the green curve marks the start (point S) and end points of the thread. The blue curve outlines the shape of the threaded groove. The point marked in red (point H) on this contour aligns vertically with the start of the thread at point S. Since the profile-defining curve can be anywhere relative to the start point of the thread, it is essential to correctly position point H.



Cycle Prarameters:

Cycle.

You can select one of the available turning cycles described in the Job Assignment. When you switch a cycle in this section, the corresponding element in the Job Assignment automatically changes. The set of parameters varies for each turning cycle.

▼ Click here to expand..
Clearance Stock. Specifies the distance from the thread to the transitions level. The cycle goes through this level while moving from the end of the previous pass to the start of the next one.

Thread Form. The group configures the geometric parameters of the thread.

▼ Click here to expand..
Thread Libray. Select thread parameters from the database if the parameters for the thread conform to standard ones. There is no need to enter all parameters manually. Just establish the type of thread and select its designation from the dropdown list. The database will automatically complete the remaining parameters with standard values.

Symbolic Name. It's just an optional symbolic designation of the thread type and size, M10 for example. You can use it to quick search the thread in the threads' table. You can enter your own designation for non-standard threads.

Thread Pitch. Specify the thread pitch either in length units or as the number of threads per length unit.

Number of starts. Specifying two or more starts results in the cutting of a multi-start thread. Various cycle types use Number of starts differently. In ISO G76 numbers of starts send to cycle as a parameter. But some machines have not this parameter in cycle. In this case it is possible to create multistart threads by making the same cycle with another Spindle start angle parameter value. Another way is using ISO G92 or ISO G32/33. In this case operation automatically generates passes with different spindle start angles.

Spindle Start Angle. Specifies the angle from the X-axis where the thread's helical line cutting starts.

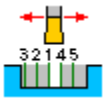
Thread Roughing. When setting the flag, the tool removes material from the thread grooves over several passes. This is essential for achieving a quality surface and reducing tool load. The system performs roughing passes by moving the tool sequentially with a fixed step along the rotational axis, dividing the process into multiple layers.

Bottom Stock. This is the stock left at the bottom of the thread groove for removal by subsequent finishing passes.

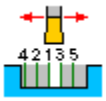
Side Stock. This is the stock left at the sides of the thread groove for removal by subsequent finishing passes.

Machining Direction. Defines the order of rough cuts.

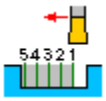
Bidirectional. The Bidirectional machining is performed from the groove center to the groove flanks at first moving to the right end then to the left one.



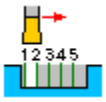
Alternate. The Alternate machining starts from the groove center, then advances to the flanks by alternating the forward and backward tool strokes



Forward. The Forward mode machines a groove from front to back. B y default it is direction from right to left.



Backward. The Backward mode machines a groove from back to front. B y default it is direction from left to right.

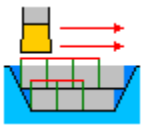


Rough Step. It is the distance between two adjacent rough cuts along a thread. Specify either the distance between tool travels or the number of tool travels. The smaller the step between the tool travels is the smaller the tool load is but machining time grows.

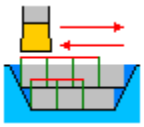
Multilayer. This feature allows to remove a rough material by several layers.

Off. This value corresponds to the single-layer machining.

Monodirectional. This feature activates multilayer machining, p ass directions are identical on each layer.



Zigzag. This feature activates multilayer machining, a root pass direction correspond the Machining Direction, at the each following layer the pass direction is opposite to the previous layer.



Depth Step. Defines the maximal distance to plunge the tool on each layer. It can be defined either by depth directly, or by number of layers.

Thread Finishing. Upon flag installation, the system creates finishing passes along the profile contour defined in the Job Assignment.

Finish step. It is the distance between two adjacent passes, calculated by the profile length. The step can be defined by count of passes or by distance directly.

Machining Direction. Defines the order of finish cuts.

Forward. Finishing is conducted heading towards the spindle.



Backward. Finishing is conducted away from the spindle.



Bidirectional. Finishing passes are arranged from the periphery of the thread groove to the center



Compensation. This parameter controls the method of outputting cutter width compensation.

Length corrector. Controls the output to the program of tool offset numbers that are specified in the Tools – Tool Properties – Numbers.

First. If the First parameter is selected, the system outputs the 1st corrector's number to the NC program.

Second. If the Second parameter is selected, the system outputs the 2nd corrector's number to the NC program.

Roll by arcs.

The system rounds the external corners of the toolpath by arcs with a radius equal to the tool nose radius. The parameters group works similarly to the OD Roughing and ID Roughing operations. Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

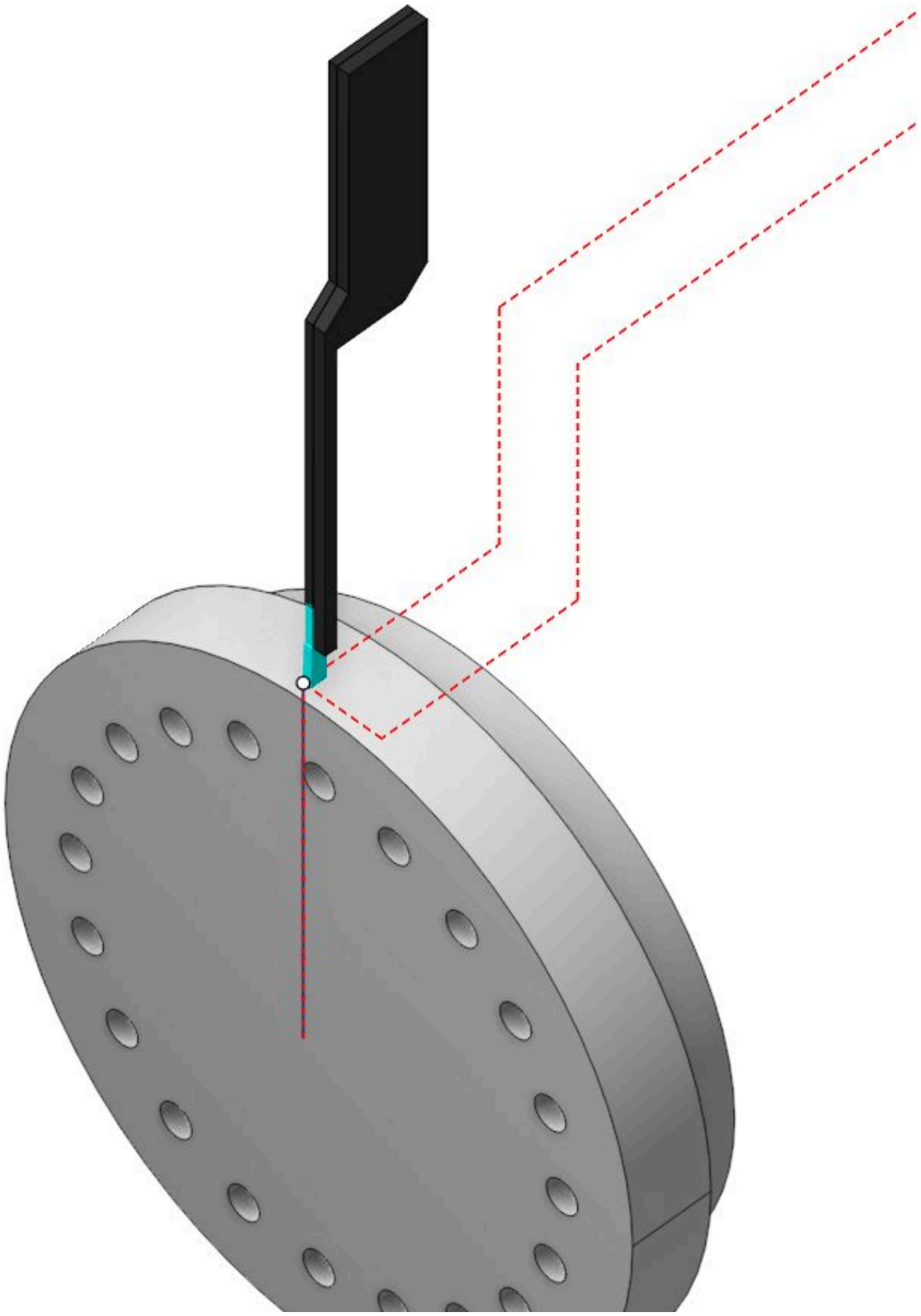
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Lathe part-off





Application area:

The purpose of this operation is to cut off the processed part from the leftover workpiece material, while simultaneously forming the back end of the part, including creating a chamfer or radius on it. One can prepare the workpiece before parting off, for example, by machining a groove

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Part-off . Executes the cut off the processed part from the leftover material of the workpiece, while also machining the back face of the part . The system generates the working cycle's contour as a vertical line connecting the outer and inner diameters of the workpiece on the part's right or left side.

Profile. The system forms a single pass, the toolpath of which is identical to the geometric contour of the part. User can add engage/retract; shifting by stock values; tool unreachable gray zones excluding; tool radius compensating to the toolpath. Details

Grooving cycle. The system outputs the part-off operation into the NC program as either an ISO G75 or ISO G74 cycle.

Slotting. This cycle efficiently machines simple rectangular grooves with a minimum number of actions. Details

Grooves . This cycle is similar to Roughing, but it outputs the machining process to NC program as a cycle similar to ISO G71/ G72. Details

Zigzag. The tool removes the rough stock layer by layer. In each layer, it plunges downwards to the cutting depth and performs horizontal movement. Thus, horizontal passes with alternating directions remove the entire rough stock, layer by layer. Details

Properties. Displays the properties of the selected element. You can additionally add allowance. You can call the Properties menu by double-clicking on the selected element.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

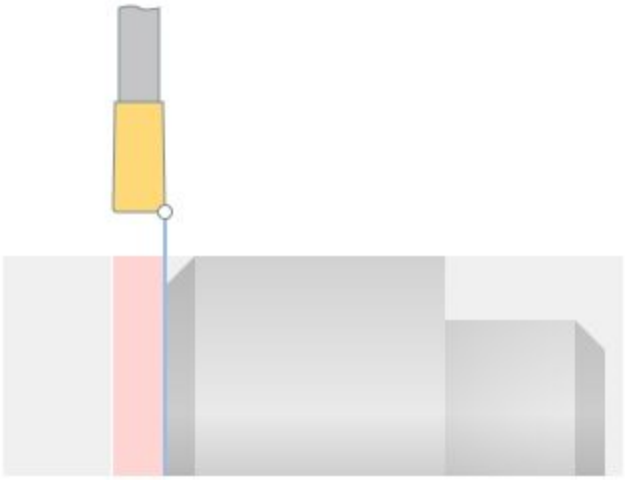
CycleParams:

Cycle.

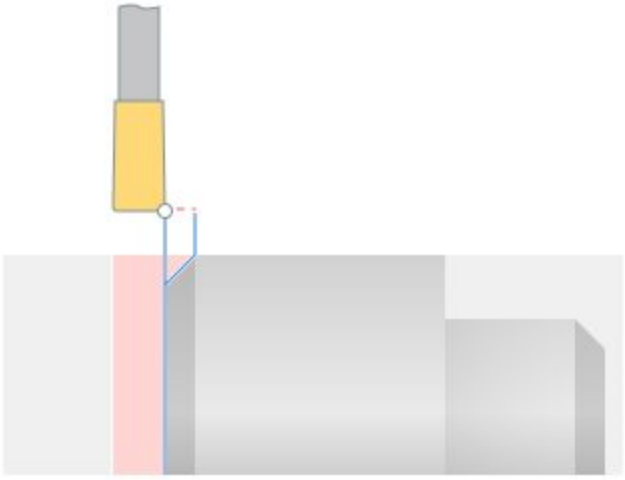
You can select one of the available turning cycles described in the Job assignment. When you switch a cycle in this section, the corresponding element in the Job assignment automatically changes. The set of parameters varies for each turning cycle.

▼ Click here to expand..
Smooth sharp edge. This feature specifies techniques for shaping the outer edge on the rear face of the machined part.

▼ Click here to expand..
Off. The outer edge on the rear face of the part retains its original condition after previous operations.



Chamfer . The system forms a chamfer on the outer edge of the part's rear face.

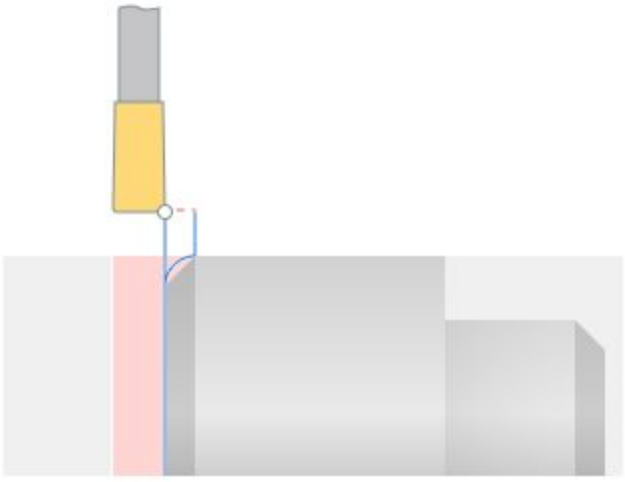


Size. Input the size of the chamfer's leg.

Angle. Input the chamfer angle. Measure the angle from the vertical.

Make preparatory pass. Prior to the primary machining pass, the cutter makes a relief groove with a depth exceeding the chamfer dimension to avoid tool's side action during subsequent chamfer formation.

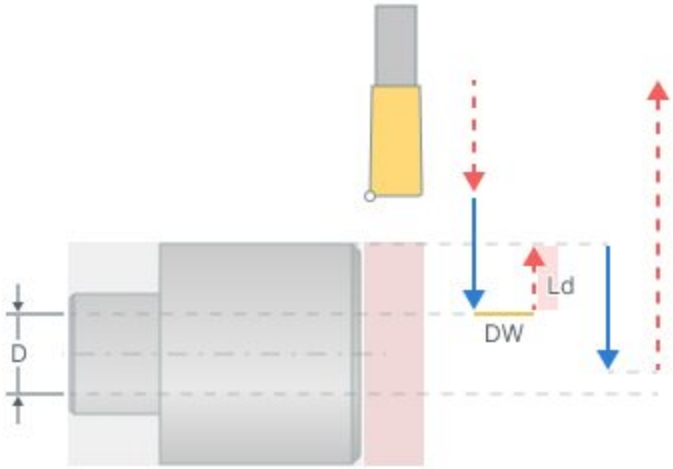
Rounding. The system forms a rounding on the outer edge of the part's rear face.



Size. Same as in Chamfer.

Make preparatory pass. Same as in Chamfer.
Reduce speed at the end. This option allows you to set the feed and spindle speed at the end of cut. For example, for cutting off heavy parts.

▼ Click here to expand..



Diameter (D) . This parameter specifies the part diameter where speeds changes begin (See Reduce speed at the end hint).

Lead out (Ld). Upon reaching the specified Diameter, the tool idle retracts by the Lead out amount in radial terms, the new Spindle Speed is then set, and the tool resumes its movement towards the workpiece center with the new Spindle Speed and feed rate (See Reduce speed at the end hint).

Spindle speed. The revolution rate of the spindle, set at the end of the cut.

Delay(DW). The dwell during which speed changes occur (See Reduce speed at the end hint).

Chip breaking. The system controls the mode of kinematic chip breaking in which, after a specified cutting length, the cutter retracts to stop the formation of continuous chip. Then the cutter engages again and continues cutting.

▼ Click here to expand..
Depth step . Specify the cut depth after which the cutter retracts for chip breaking.

Lead out. Specify the cutter's return distance after stopping for chip breaking in the length units.

Delay. This feature controls feed stop. Activating this feature introduces a stop duration.

▼ Click here to expand..
Off. The feed does not stop neither at the bottom, nor at every layer.

At the bottom . Feed stop occurs at the end of the entire cut's working pass.

At every layer . Feed stop occurs at each Step depth when Chip breaking mode is active.

Return to top level. When this flag is set, the tool returns to the start of the Job assignment, after completing the cut. Otherwise, the tool remains at the end of the cut off pass.

Counterspindle back off . This feature specifies the distance for the counterspindle move back after finishing the cut off.

▼ Click here to expand..
Off. The system does not control counterspindle.

mm. The c ounterspindle back off is set in length units .

% width of tool. The counterspindle back off is set as a percentage of the cutting tool width.

Counterspindle axis . The machine axis name controlling the counterspindle movement.

Compensation. This parameter controls the method of outputting the cutter width and the tool length compensation.

▼ Click here to expand..
Insert width compensation. Determines the method of outputting the cutter width.

On. The system takes into account the width of the cutter insert.

Off. The system doesn't take into account the width of the insert. When using this mode on complex contours, the undercuts or incomplete machining are possible, so it is necessary to be carefully when machining simulation.

Some parameter group works similarly to the OD Roughing and ID Roughing operations. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

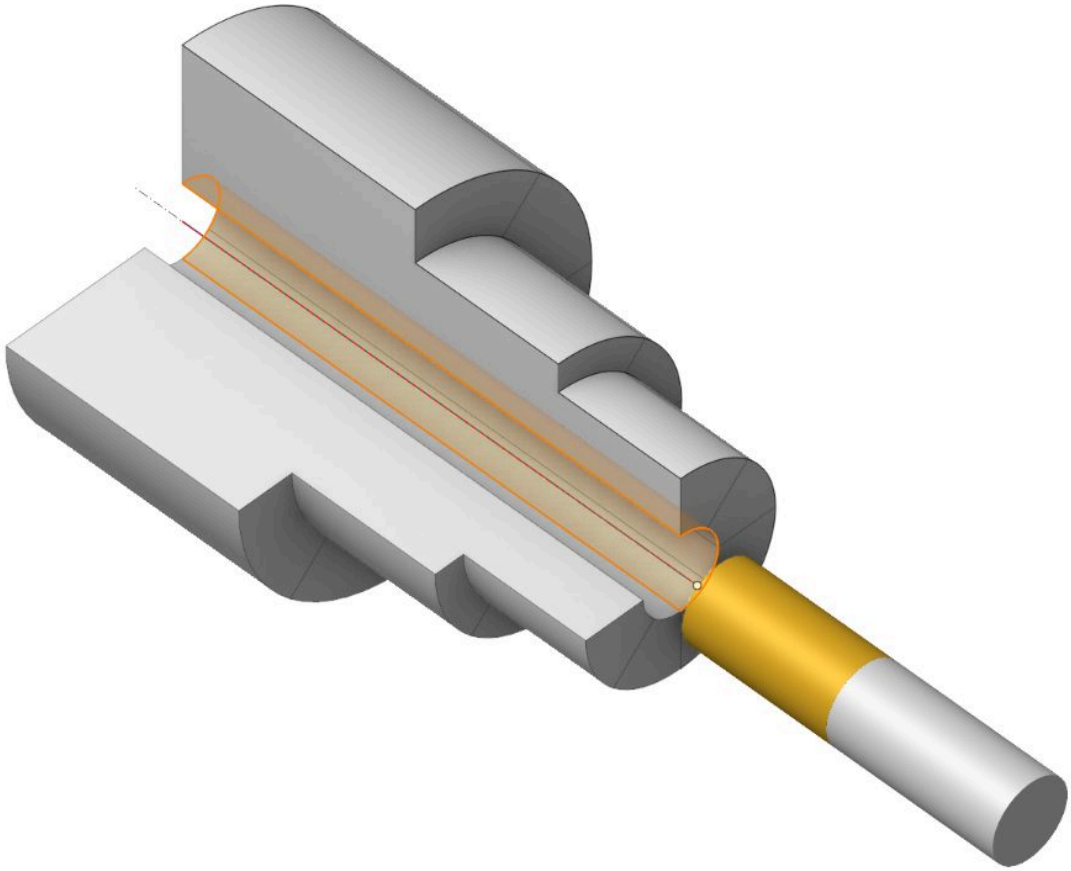
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Lathe hole machining



Application area:

The hole machining operations are designed for drilling, centering, boring, countersinking, tapping, thread milling and hole pocketing. It can machine holes that are not lying in the same plane and that are not lying in orthogonal planes. The operation can be used both for the machining of holes in a model, or for pre-drilling of the tool plunge points for the pocketing and operations.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Center. Create hole by center point.

Create. Create hole by coordinates input. Details

Recognize. Automatically recognize holes in the part. Details

Pattern. Create holes array by pattern. Details

Properties. Displays the properties of an element. You can also call this menu by double clicking on an item in the list. Details

Sorting. Sorting using some parameters to sort holes list. Details

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

This parameter group works similarly to the Hole machining operation. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations

per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

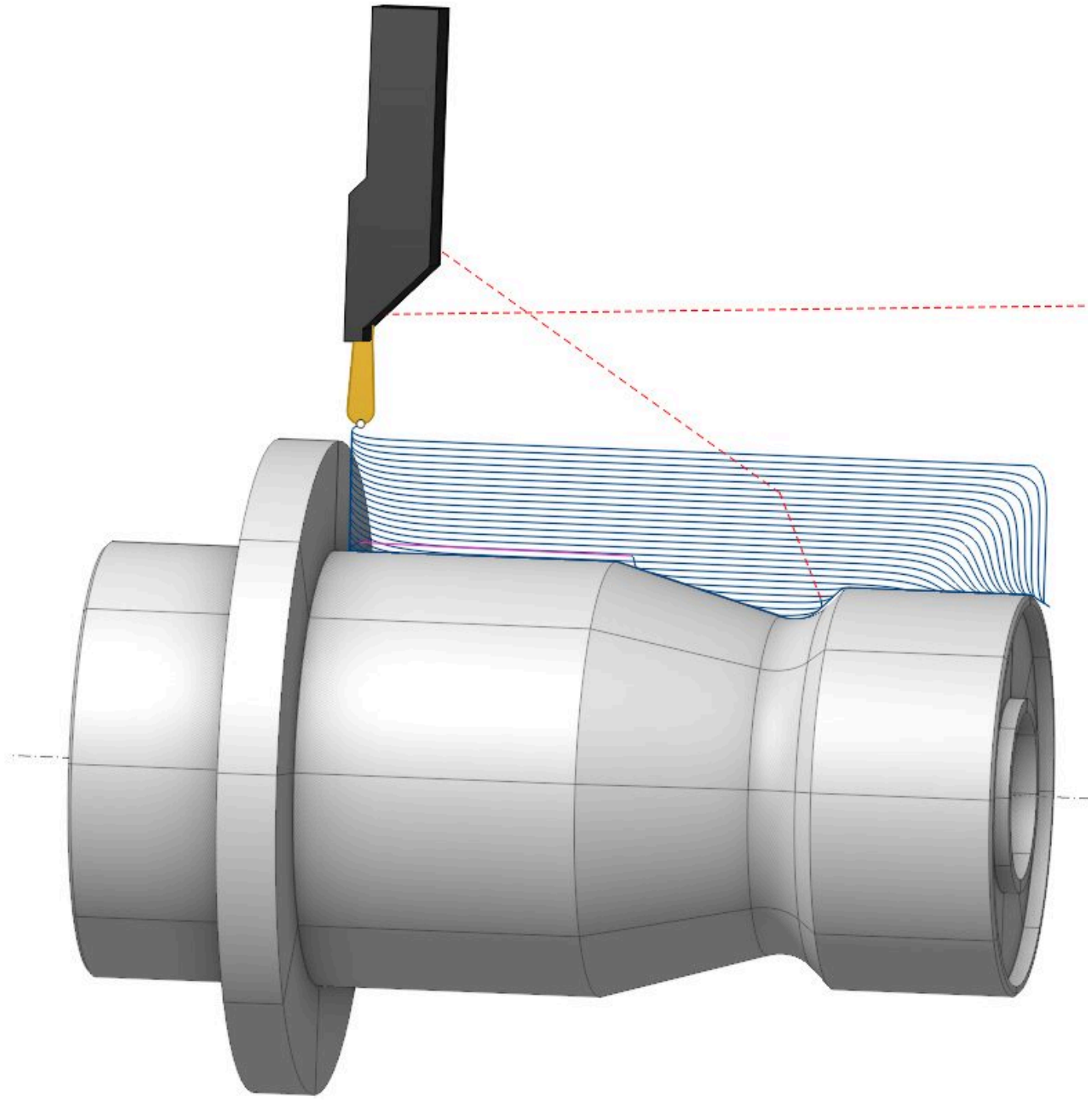
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

OD Adaptive turning



Application Area:

The operation generates an optimized smooth trajectory that maintains constant tool load, extending tool life and significantly increasing processing speed through higher cutting feed rates. The turning process executes more than twice as fast compared to the traditional Grooving cycle. This cycle adopts the Adaptive strategy from the Waterline Roughing operation, with all parameters mirroring this strategy. It uses tool with round insert.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job Assignment:

Adaptive. The process uses high feed rates with a small step, rounded passes, and avoids sharp edges in the toolpath. It incorporates a roll-in technique for entering the tool, which is less stressful on the tool. The method supports processing in both forward and backward directions.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

CycleParams:

Cycle.

Only an Adaptive cycle is available for this operation.

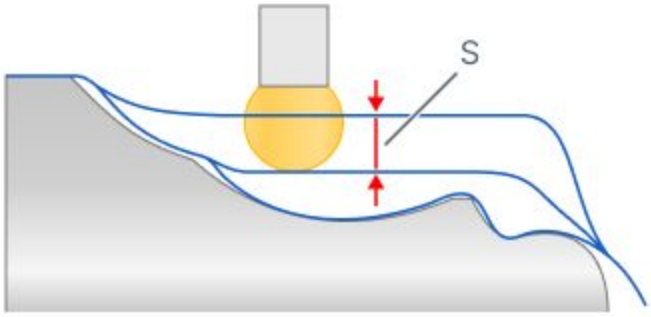
▼ Click here to expand..

Tool Path.

Here are the key parameters, the set of which may vary depending on the chosen strategy in the Cycle section:

▼ Click here to expand..

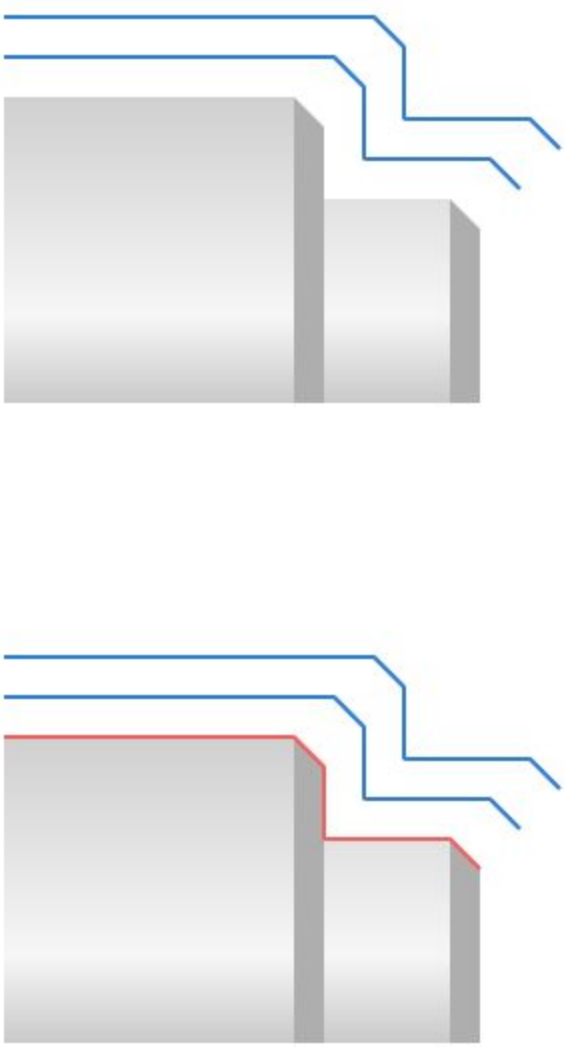
Step. The distance between tool passes is equal to the cutting depth.



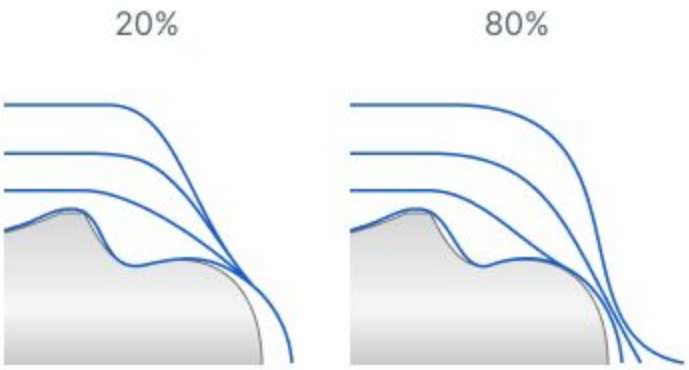
S - step

Backward Stepover. When Bidirectional turning mode is active, you can set the backward direction step as a percentage of the forward direction Step.

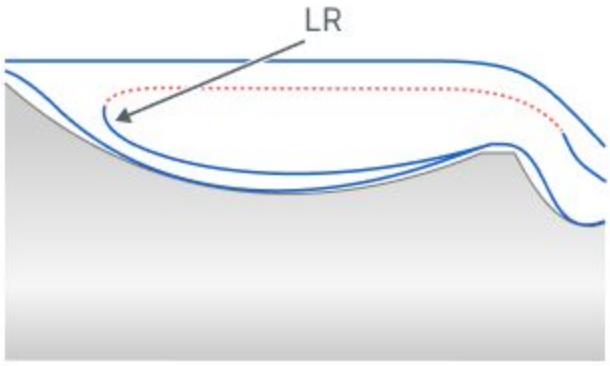
Finish pass. The contour specified in the Job Assignment becomes finishing. An additional roughing pass is added to it at a distance specified in the parameter..



Rough rounding radius. Specifies the radii for rounding off the entry and exit zones during rough passes.

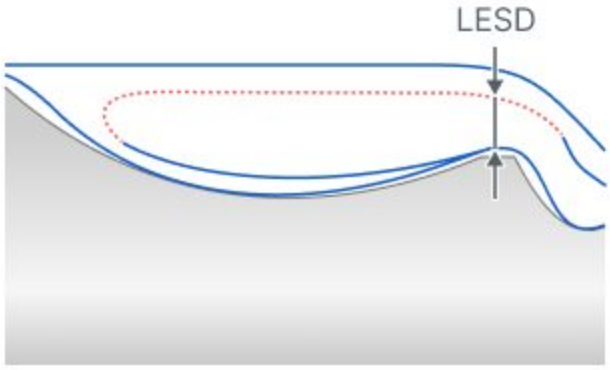


Linking radius. Linking radius t hat smooths the transitions between work passes.

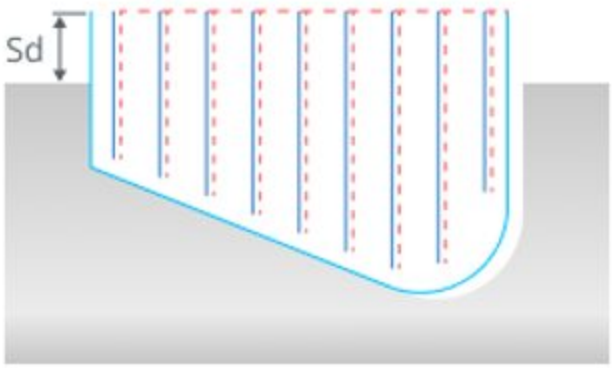


LR - Linking radius

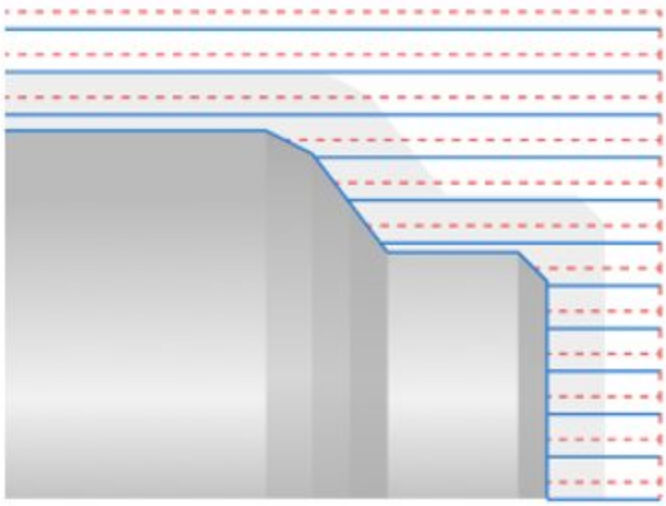
Lead Safe Distance. Specifies the minimal distance from unprocessed parts of the workpiece for rapid tool movement between machining areas.

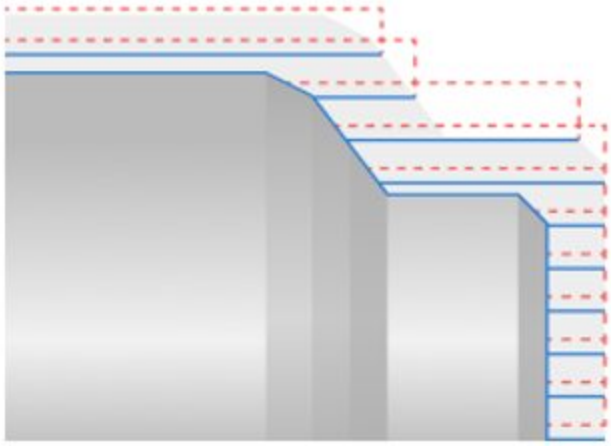


LESD - Lead Safe Distance
Safe distance. The distance from the top level of the groove to the level of rapid movements. You can edit the safety distance by dragging a point on the graphical screen or by directly input in the options window.



Check workpiece. When the flag is set, the system takes into account the current state of the workpiece and removes all toolpath segments that really do not cut the workpiece. This significantly reduces machining time, especially if the shape of the allowance is complex.

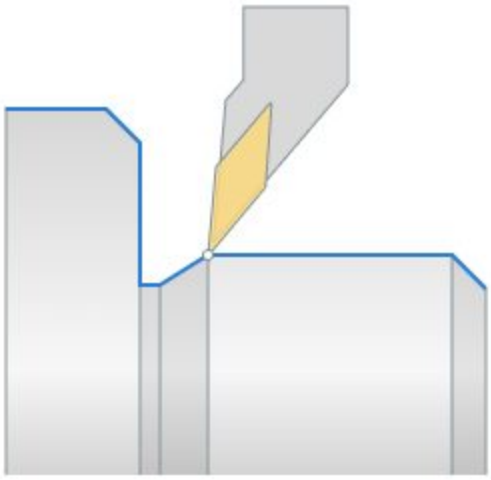




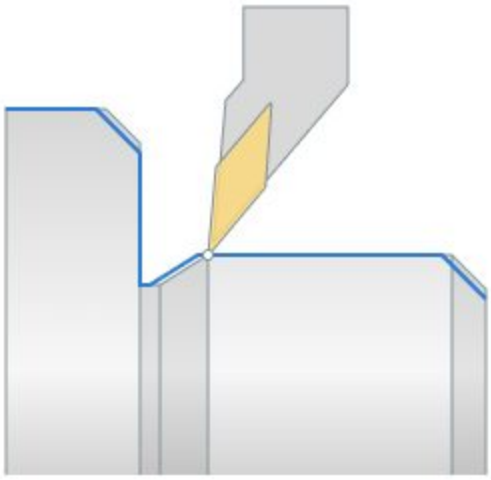
Compensation. Controls the method of outputting cutter insert parameters.

Insert Width Compensation. This parameter controls the method of outputting the tool nose radius correction. Details

Off. The contour of the toolpath corresponds to the contour of the part and is output to the NC program without considering the tool nose radius. When using this mode on complex contours, the undercuts or incomplete machining are possible, so it is necessary to be carefully when machining simulation.



Computer. The system takes into account the width and the tip radius of the cutter insert, and builds the toolpath according to the first Tooling Point of the cutter.



Direction.

Defines direction of work passes.

▼ Click here to expand..

Bidirectional. Operation performs work passes in forward and backward directions relative to the starting point of the Job Assignment.

Forward. Work passes proceed towards the spindle.

Backward. Work passes proceed away from the spindle.

Roll by arcs.

The system rounds the external corners of the toolpath by arcs with a radius equal to the tool nose radius. The parameters group works similarly to the OD Roughing and ID Roughing operations. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

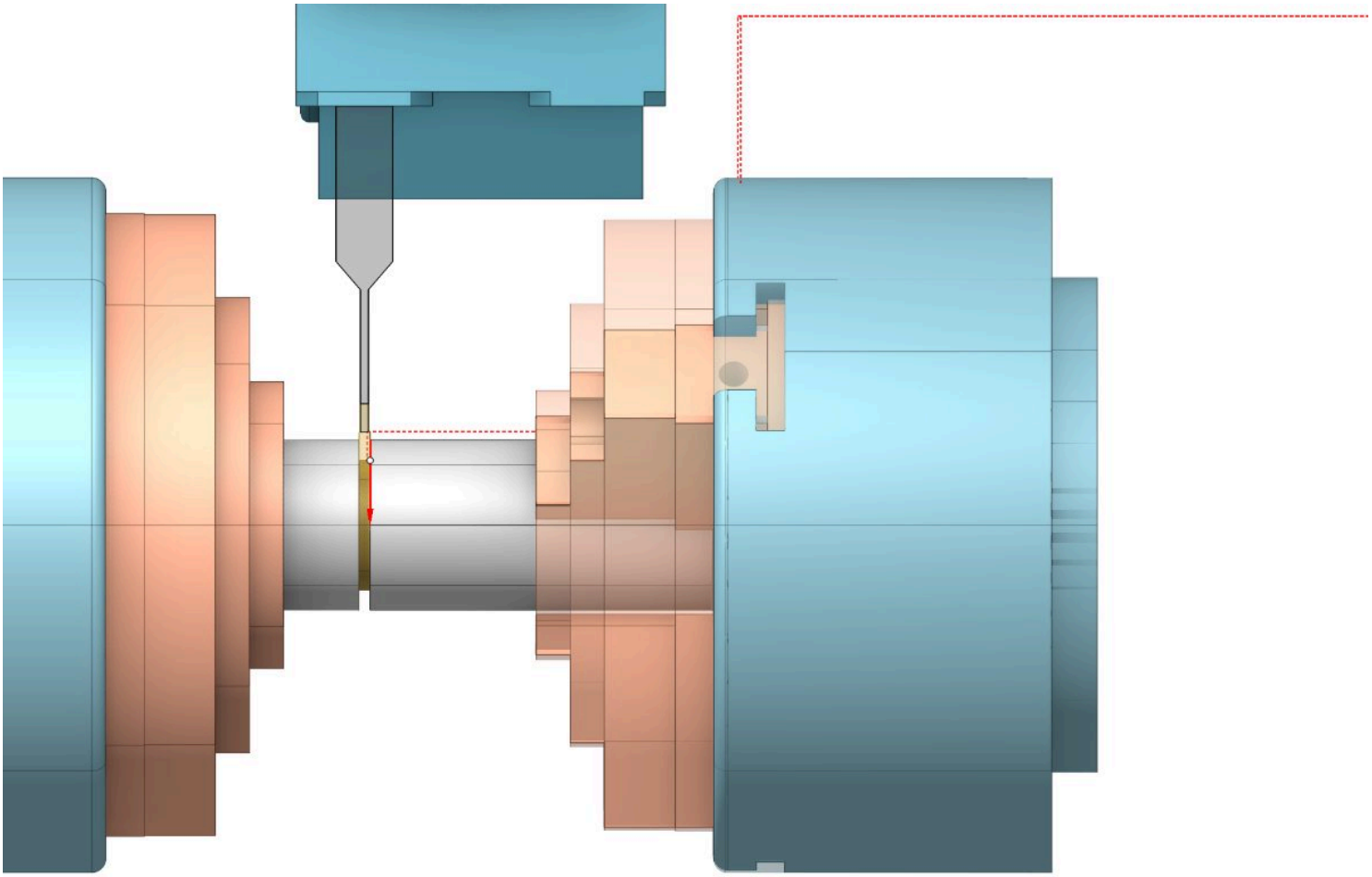
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Part-off with takeover



Application area:

The purpose of this operation is to cut off the processed part from the leftover workpiece material, while simultaneously forming the back end of the part, including creating a chamfer or radius on it. One can prepare the workpiece before parting off, for example, by machining a groove. Differs from Lathe part-off operation in that at the end of the operation the workpiece is moved to another spindle.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Part-off. Executes the cut off the processed part from the leftover material of the workpiece, while also machining the back face of the part. The system generates the working cycle's contour as a vertical line connecting the outer and inner diameters of the workpiece on the part's right or left side.

Profile. The system forms a single pass, the toolpath of which is identical to the geometric contour of the part. User can add engage/retract; shifting by stock values; tool unreachable gray zones excluding; tool radius compensating to the toolpath. Details

Grooving cycle. The system outputs the part-off operation into the NC program as either an ISO G75 or ISO G74 cycle.

Slotting. This cycle efficiently machines simple rectangular grooves with a minimum number of actions. Details

Grooves. This cycle is similar to Roughing, but it outputs the machining process to NC program as a cycle similar to ISO G71/ G72. Details

Zigzag. The tool removes the rough stock layer by layer. In each layer, it plunges downwards to the cutting depth and performs horizontal movement. Thus, horizontal passes with alternating directions remove the entire rough stock, layer by layer. Details

Properties. Displays the properties of the selected element. You can additionally add allowance. You can call the Properties menu by double-clicking on the selected element.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

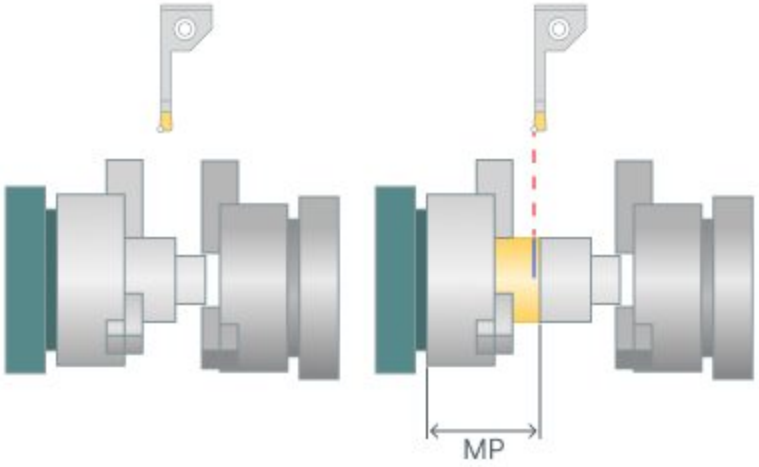
CycleParams:

Some parameter group works similarly to the Lathe part-off operation operations. Details

Strategy:

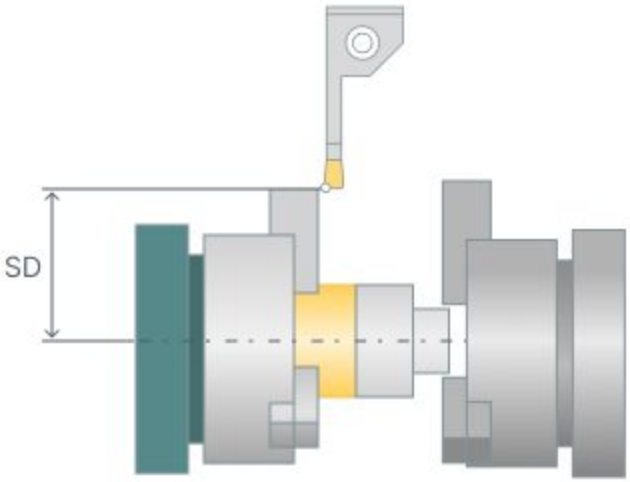
▼ Click here to expand..
Takeover. Pick and place options

Move part. Allows pulling the part up to its cut-off point



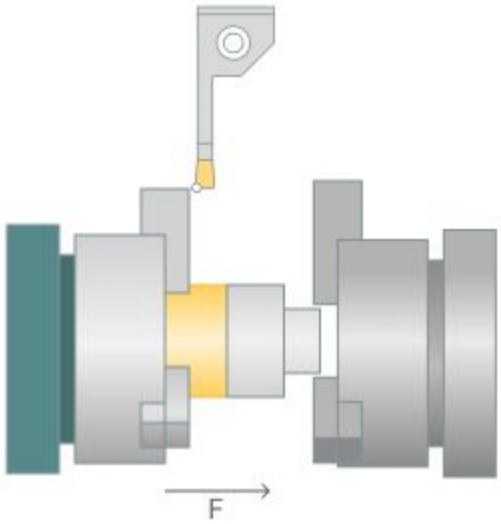
MP - Move part

Safe Diameter. Determines the X-coordinate of the cutter prior to counter-spindle movement



SD - Safe Diameter

Feed. Feed rate at which the part is pulled out



F- Feed

Some parameter group works similarly to the Turn take over operations. Details

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

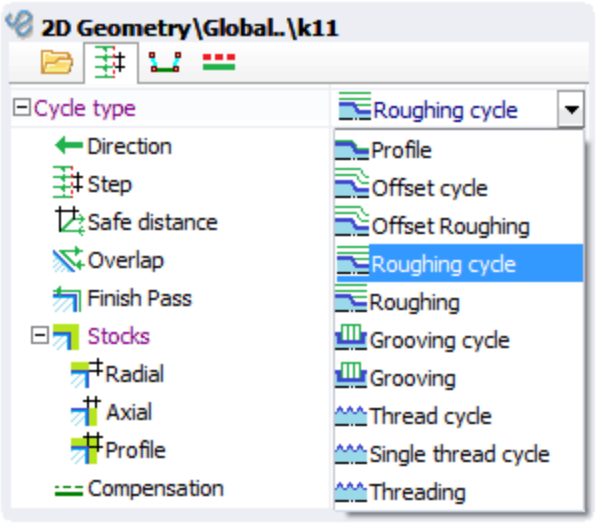
A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

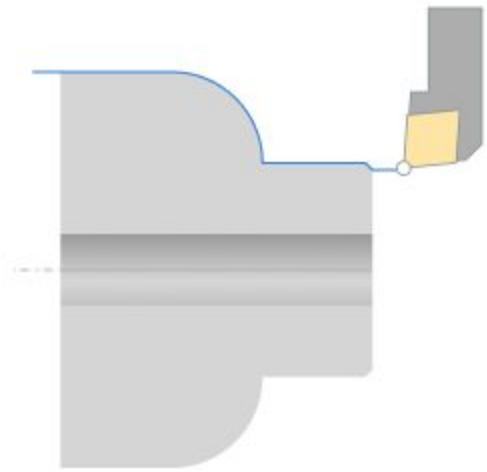
Lathe cycles

Cycle type defines the tool path generation way based on the defined profile. The next types of the job assignment element for the lathe operations are available.



- In the "Profile" mode the defined profile is output into the tool path without any additions.
- Offset cycle generates the cycle ISO G73 based on the defined profile.
- Offset Roughing generates the same tool path like the ISO G73, but it is output in the expanded mode. This mode can check current workpiece state.
- Roughing Cycle generates the one of the cycles ISO G71/G72 based on the defined profile.
- Roughing generates the same tool path like the ISO G71/G72, but it is output in the expanded mode. This mode can check current workpiece state.
- GroovingLathe mode generates the same tool path like the ISO G74/G75, but it is output in the expanded mode. This mode can check current workpiece state. In the future versions this mode will be improved and will check the real profile configuration.
- Thread cycle - is the multi-pass threading cycle that outputs the tool path in the ISO G76.
- Single-thread cycle - threading that can be performed in several passes. Every pass is generated as the separated cycle in the ISO G92 format.
- Threading - threading with the expanded output format. So every motion is performed as the separate block. Work motions are synchronized with the spindle rotation. This mode is enabled by the ISO G32/G33. Expanded format allows to make the specific threads like the threads with the different taper, spirals on the face etc.
- 4-axis turning cycle allows to machine hard-to-reach areas of a part with just one operation by continuously changing of the tool inclination angle in the process of moving along a contour (using the 4-th axis of the machine, if available).

Profile cycle

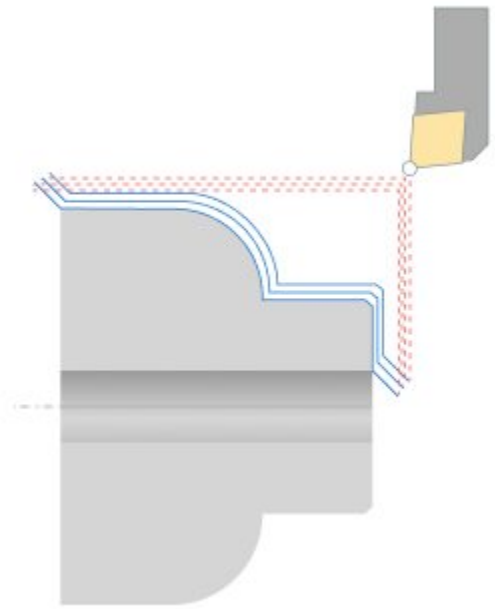


Profile cycle it is the simples cycle where the toolpath is the same with source geometrical contour defined in job assignment. Only few changes could be done:

- engage/retract joining;
- shifting by stock values;
- tool unreachable gray zones excluding;
- tool radius compensating.

See job item properties page for detailed description of parameters.

Offset Roughing

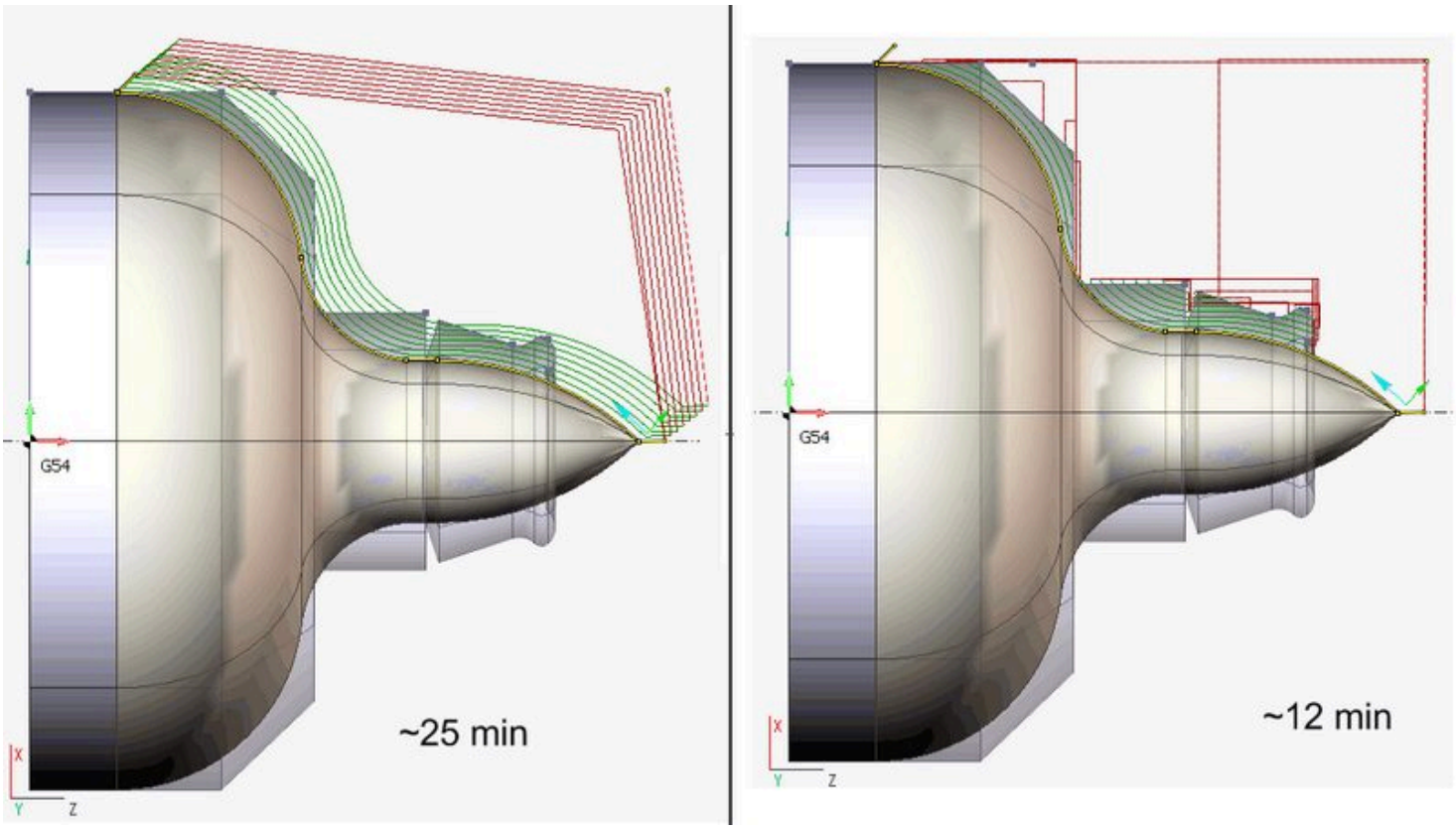


Offset Roughing generates the same tool path like the ISO G73, but it is output in the expanded mode.

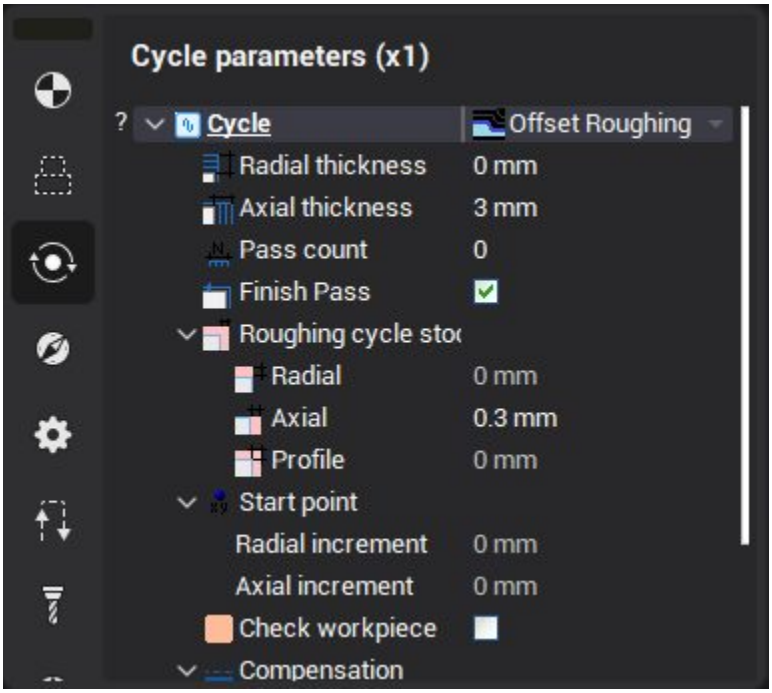
This mode can check current workpiece state.

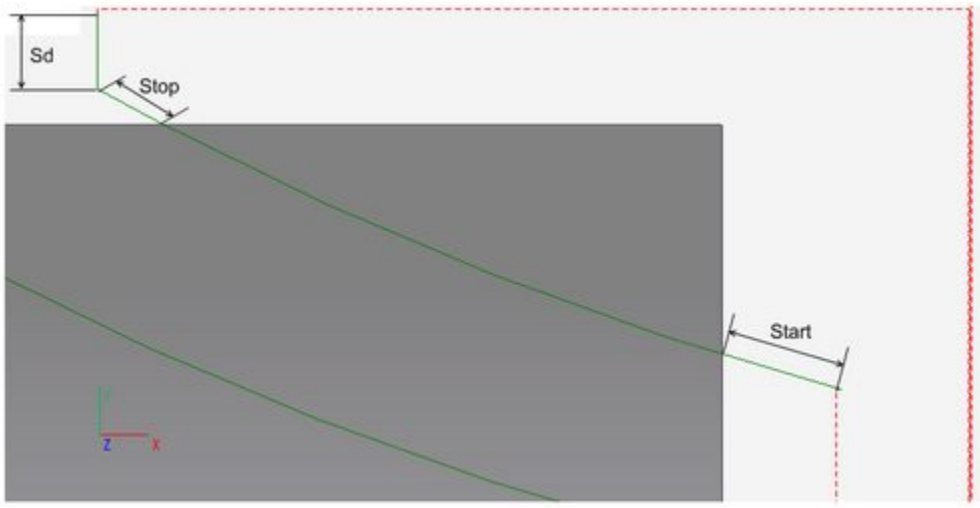
Checking the current state of the workpiece can reduce machining time. In the picture is represented the item. Machining takes approximately 25 minutes.

The use of algorithms based on the current state of the workpiece will reduce the processing time of this part by 50%, excluding the tool passes through the air at the working feed.



The workpiece checking parameters can be defined in the properties window.





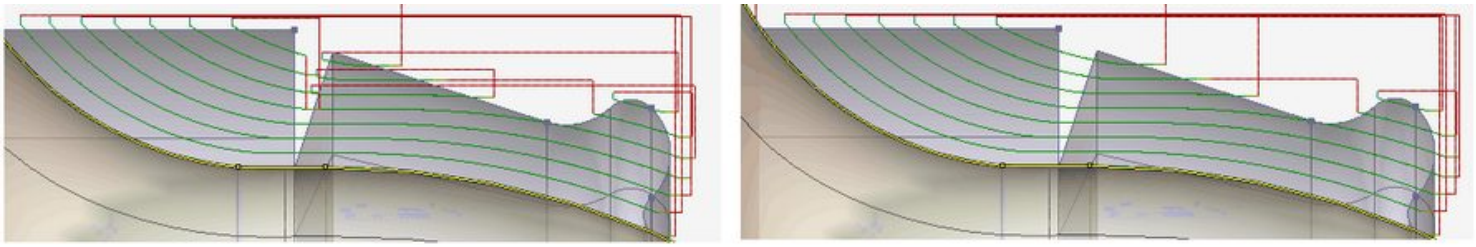
Sd — Safe distance

Start — Start entry amount

Stop — Stop entry amount

In some cases it is advisable to ignore the fact that the tool passes through the air, if the distance of this pass is quite small. Ignore thickness parameter allows to specify the maximum distance of the pass through the air at the working feed.

If the distance between two areas is less than Ignore thickness parameter, the system will treat them as one area.

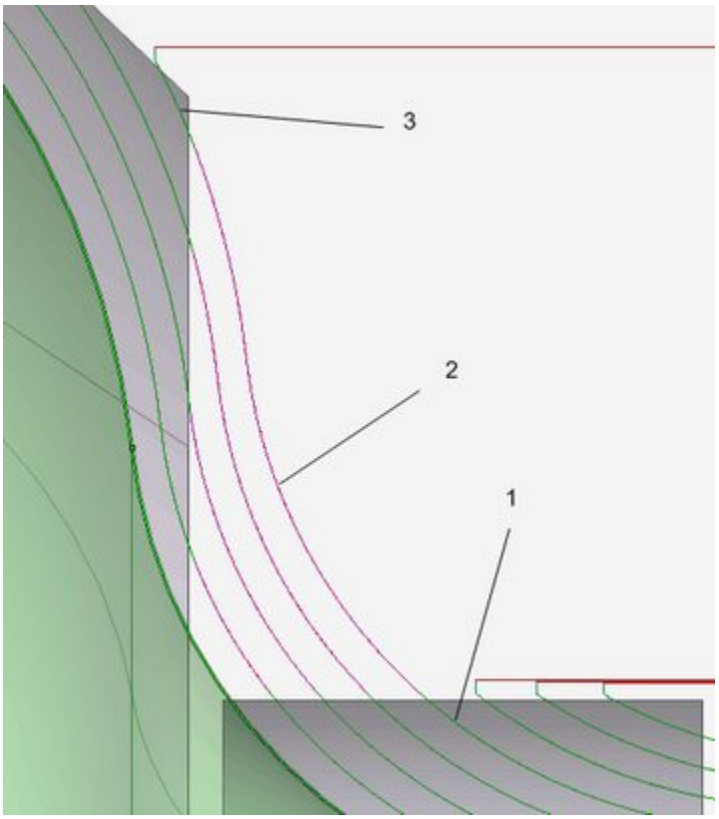


On the left side of the picture Ignore thickness parameter is less than the distance between the areas of machining, the system treats them separately.

On the right side of the picture Ignore thickness parameter more than the distance between areas of machining, the system treats them as one area.

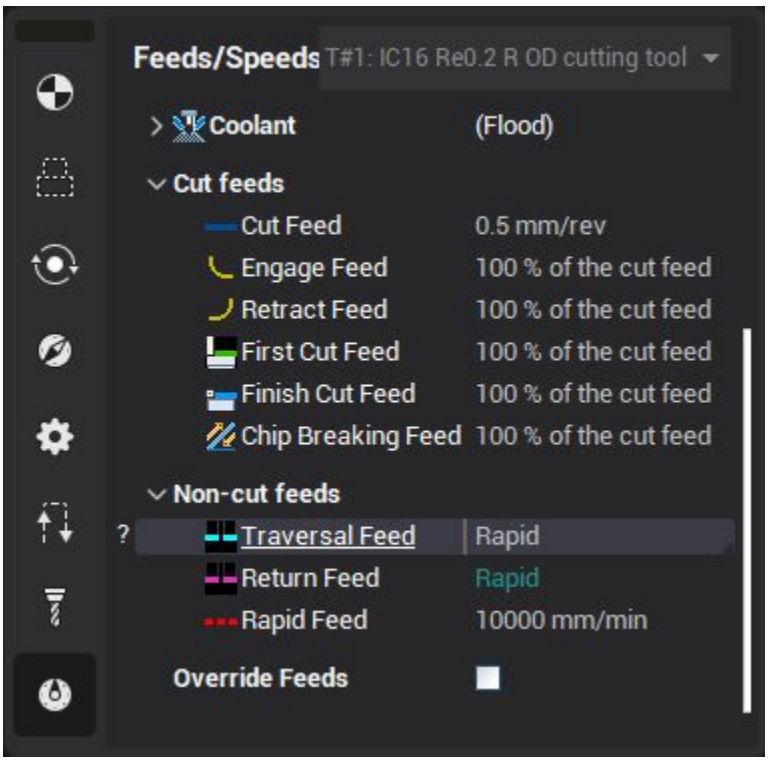
In some cases it is advisable to treat several areas as one, but move to the second area on the high feed with enabled interpolation (traversal feed).

Tool machines the first area (1) at the working feed, then moves to the second area (3) at traversal feed by the profile (2).



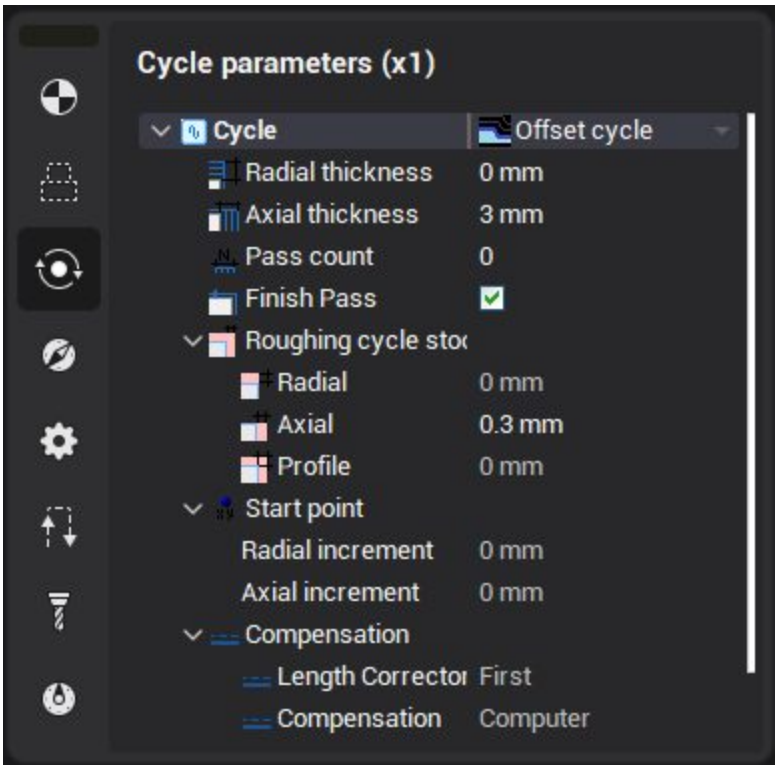
The Maximum traversal distance specifies the maximum distance that the tool can pass on the traversal feed. If the distance between areas is greater than this parameter, the system would treat them separately.

The system provides the ability to set the feed values for a particular profile.



Offset cycle (ISO G73)

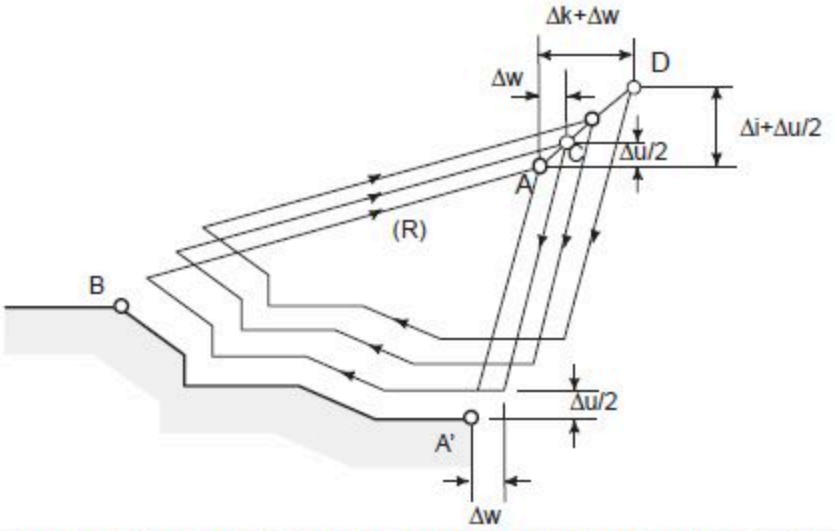
Offset cycle generates the canned cycle G73 based on the defined profile. The cycle parameters can be defined in the properties window.



If "Finish pass" check box is set then the finishing cycle ISO G70 is generated just after the rough cycle.

The extract from the Fanuc Operator's manual about cycle G73 is shown below.

This feature allows you to perform repeated processing along a contour with a gradual shift of the contour. By this cutting cycle, it is possible to efficiently cut workpiece whose rough shape has already been made by a rough machining, forging or casting method, etc.



The pattern commanded in the program should be as follows.

A→A'→B

G73 U (Δi) W (Δk) R (d) ;

G73 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t) ;

N (ns)·····
·····
F
S
T
N (nf)·····;

The move command between A and B is specified in the blocks from sequence number ns to nf.

Δi : Distance and direction of relief in the X axis direction (Radius designation). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5135, and the parameter is changed by the program command.

Δk : Distance and direction of relief in the Z axis direction. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5136, and the parameter is changed by the program command.

d : The number of division. This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated. Also, this value can be specified by the parameter No. 5137, and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter/radius designation)

Δw : Distance and direction of finishing allowance in Z direction

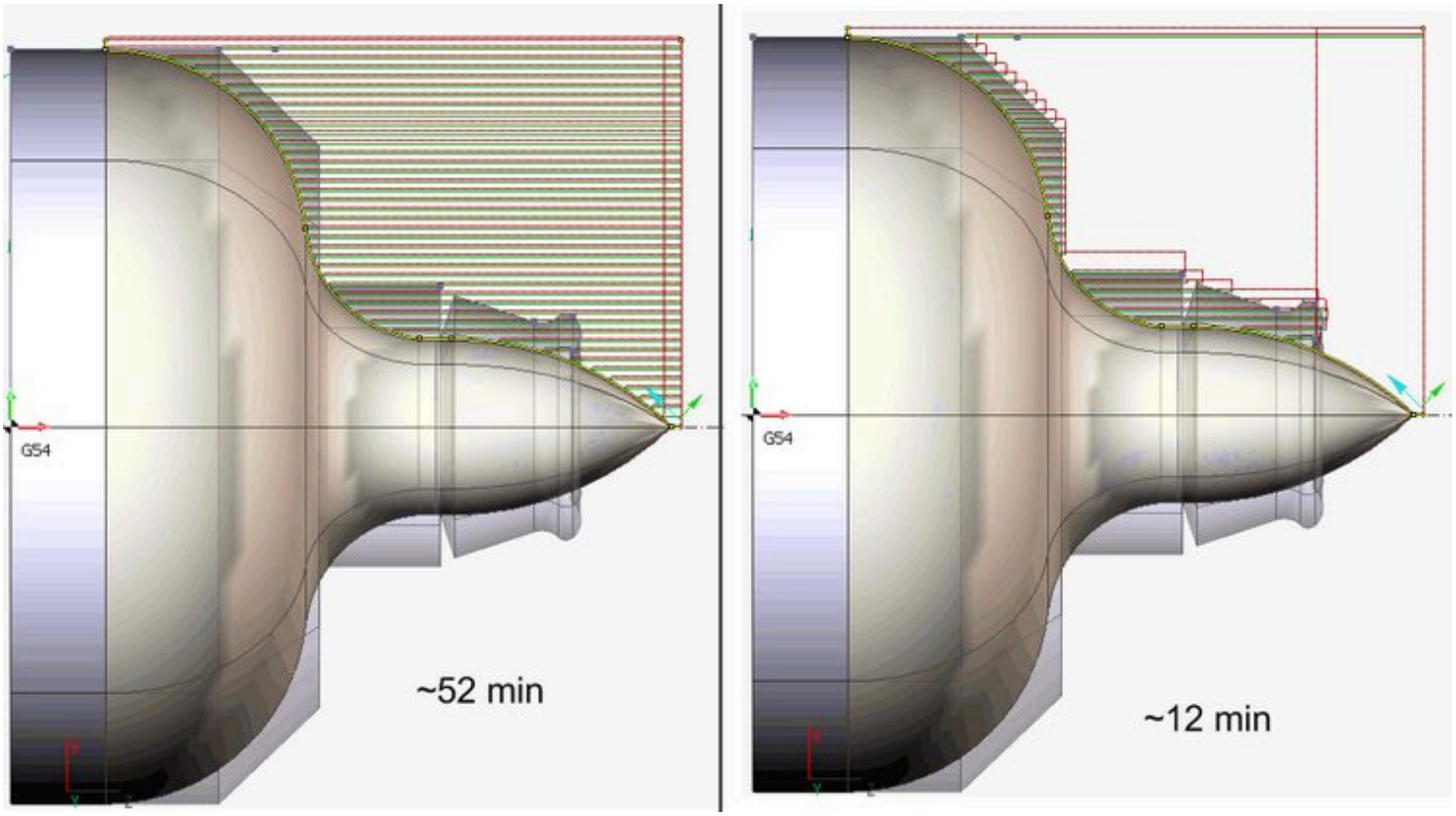
f,s,t : Any F, S, and T function contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S, and T functions in this G73 block are effective.

Roughing

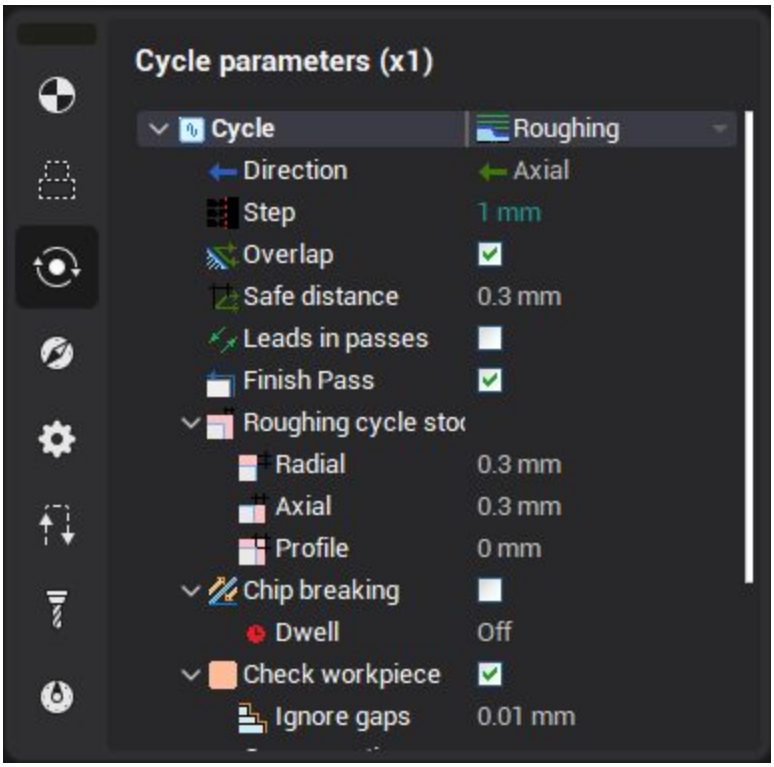
Roughing based on ISO G71/G72, but it is output in the expanded mode.

This mode can check current workpiece state.

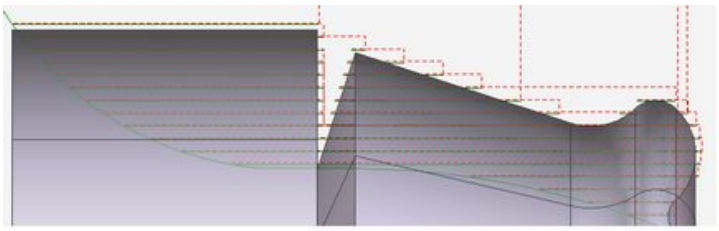
Checking the current state of workpiece can reduce machining time. In the picture is represented the item. Machining takes approximately 52 minutes. The use of algorithms based on the current state of workpiece will reduce the processing time of this part by 80%, excluding the tool passes through the air at the working feed.

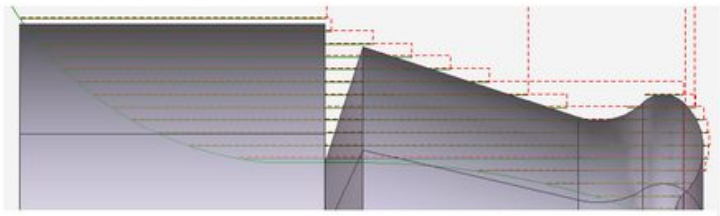


The workpiece checking parameters can be defined in the properties window..



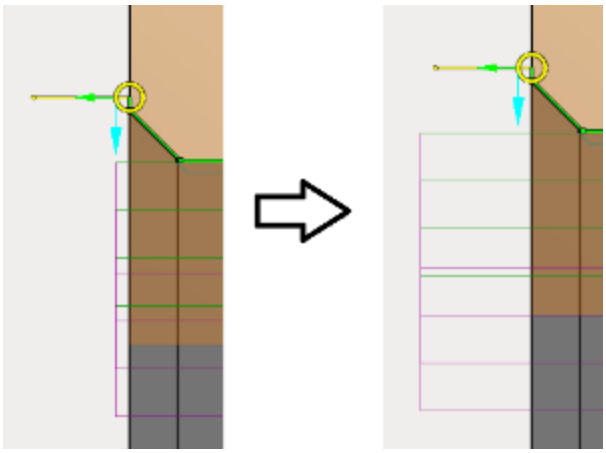
In some cases it is advisable to ignore the fact that tool passes through the air, if the distance of this pass is quite small. Ignore thickness parameter allows to specify the maximum distance of the pass through the air at the working feed. If the distance between two areas is less than Ignore thickness parameter, the system will treat them as one area.



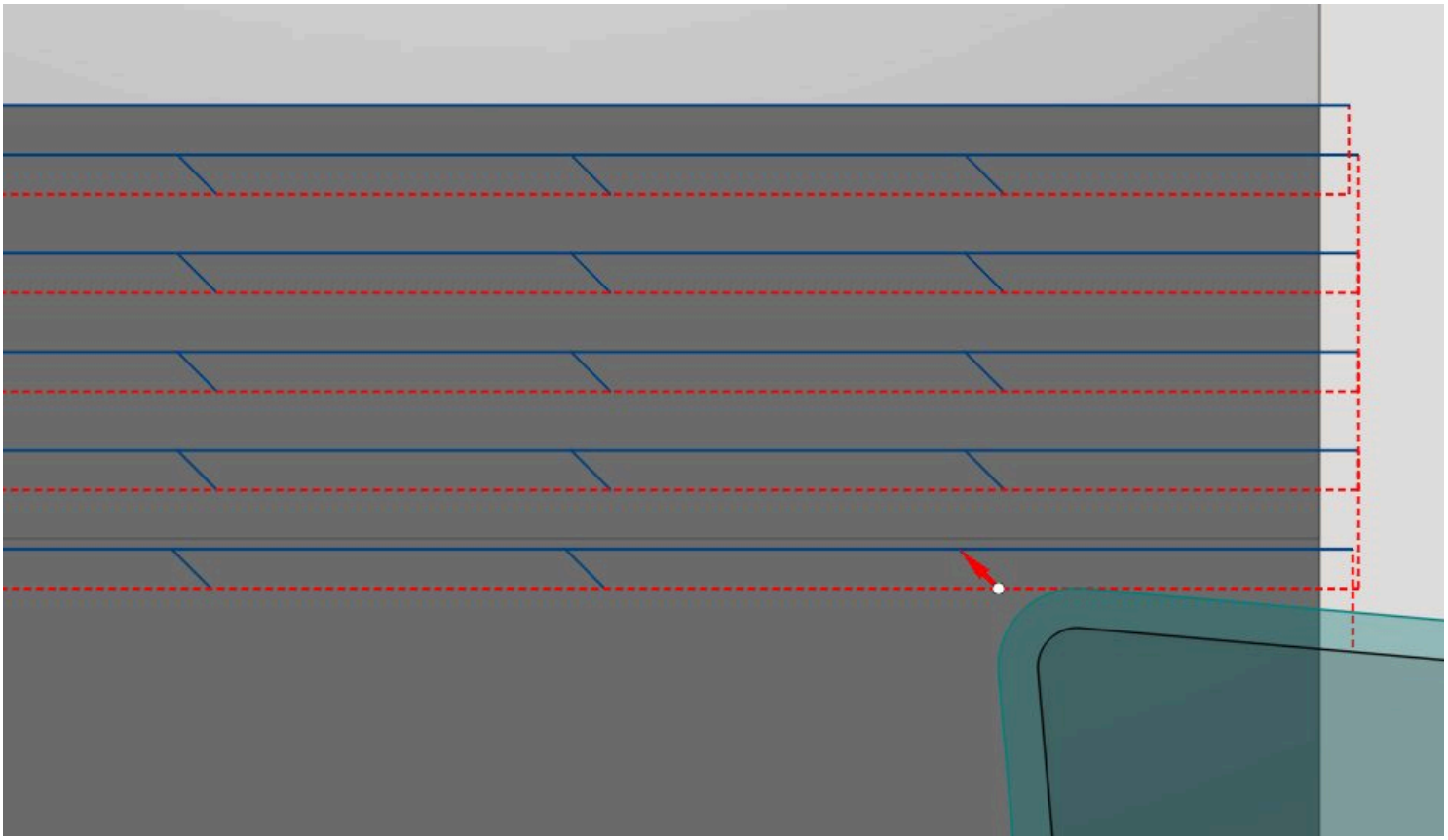


In the left side of the picture Ignore thickness parameter is less than the distance between the areas of machining, the system treats them separately. On the right side of the picture Ignore thickness parameter more than the distance between areas of machining, system treats them as one area.

Also this mode have Leads in passes parameter, which allows you to use engage/retract in passes from the Links/Leads tab. This is quite useful with enabled check workpiece parameter.

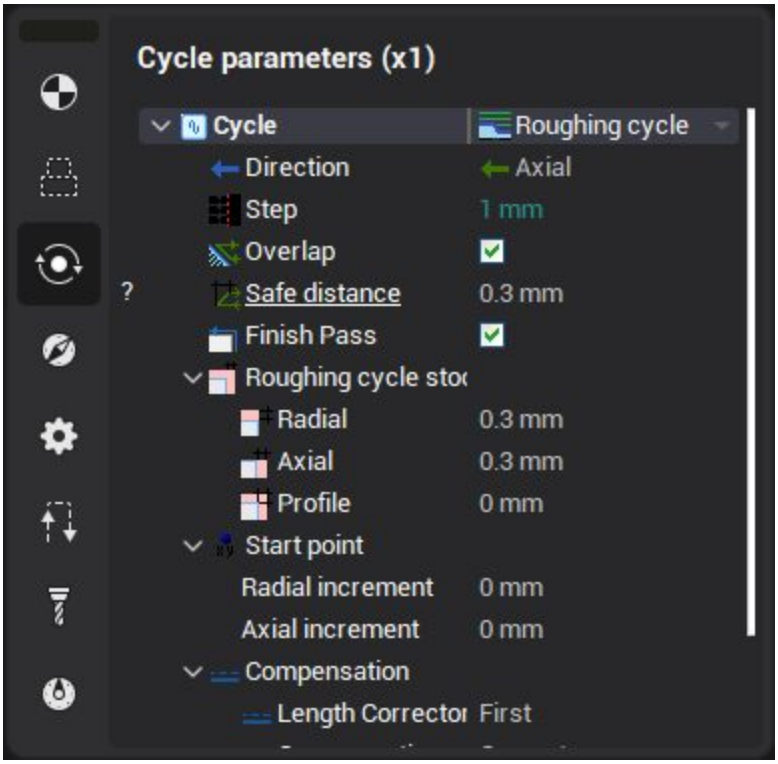


Chip breaking parameter helps to adjust length for cut before chip breaking. The lenght of the retract sets by safe distance parameter. Dwell parameter sets delay in seconds or revolutions instead of retract in chip breaking. Chip breaking works in overlap mode too. Chip breaking feed is set on the Speeds tab.

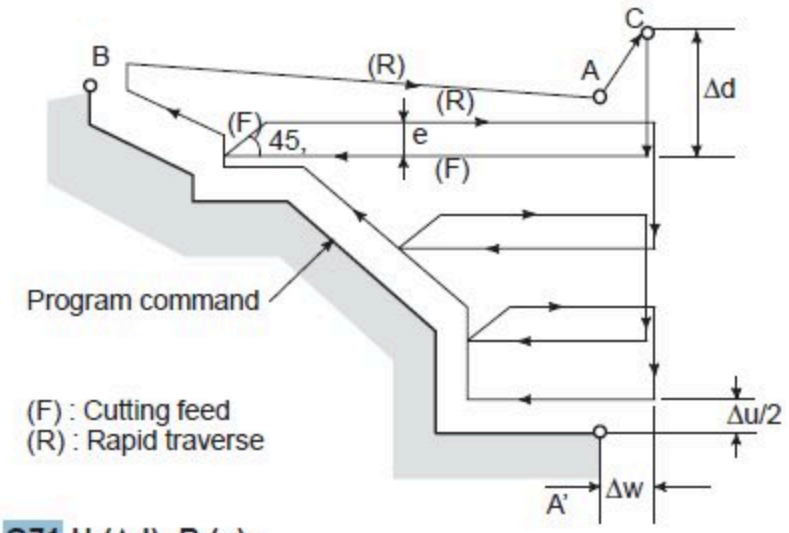


Roughing cycle

The roughing cycle generates one from both canned cycles ISO G71 or ISO G72 based on the defined profile. The cycle parameters can be defined in the properties window. Direction property switch between G71 and G72. If "Finish pass" check box is set then the finishing cycle ISO G70 is generated just after the rough cycle.



The extract from the Fanuc Operator's manual about cycles G71/G72 is shown below.



G71 U (Δd) R (e) ;
G71 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t)

N (ns)·····
·····
F____
S____
T____
} The move command between A and B is specified in the blocks from sequence number ns to nf.
N (nf)·····;

Δd : Depth of cut (radius designation)
Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

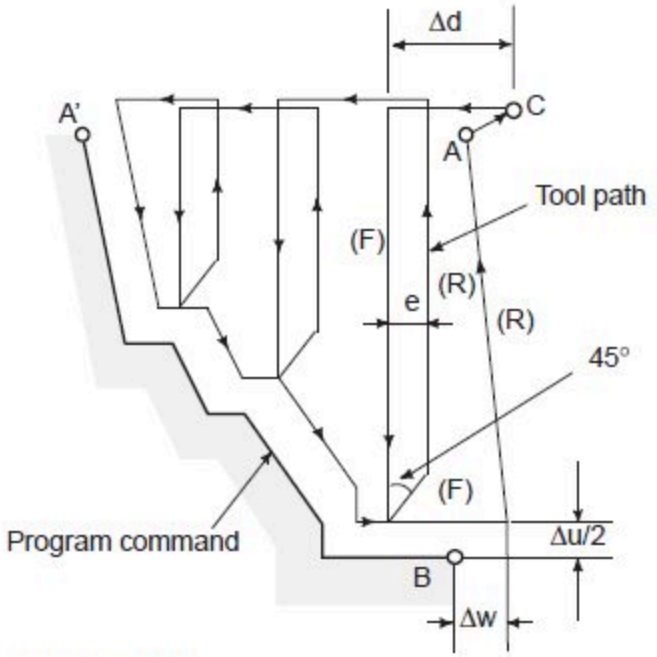
ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).

Δw : Distance and direction of finishing allowance in Z direction.

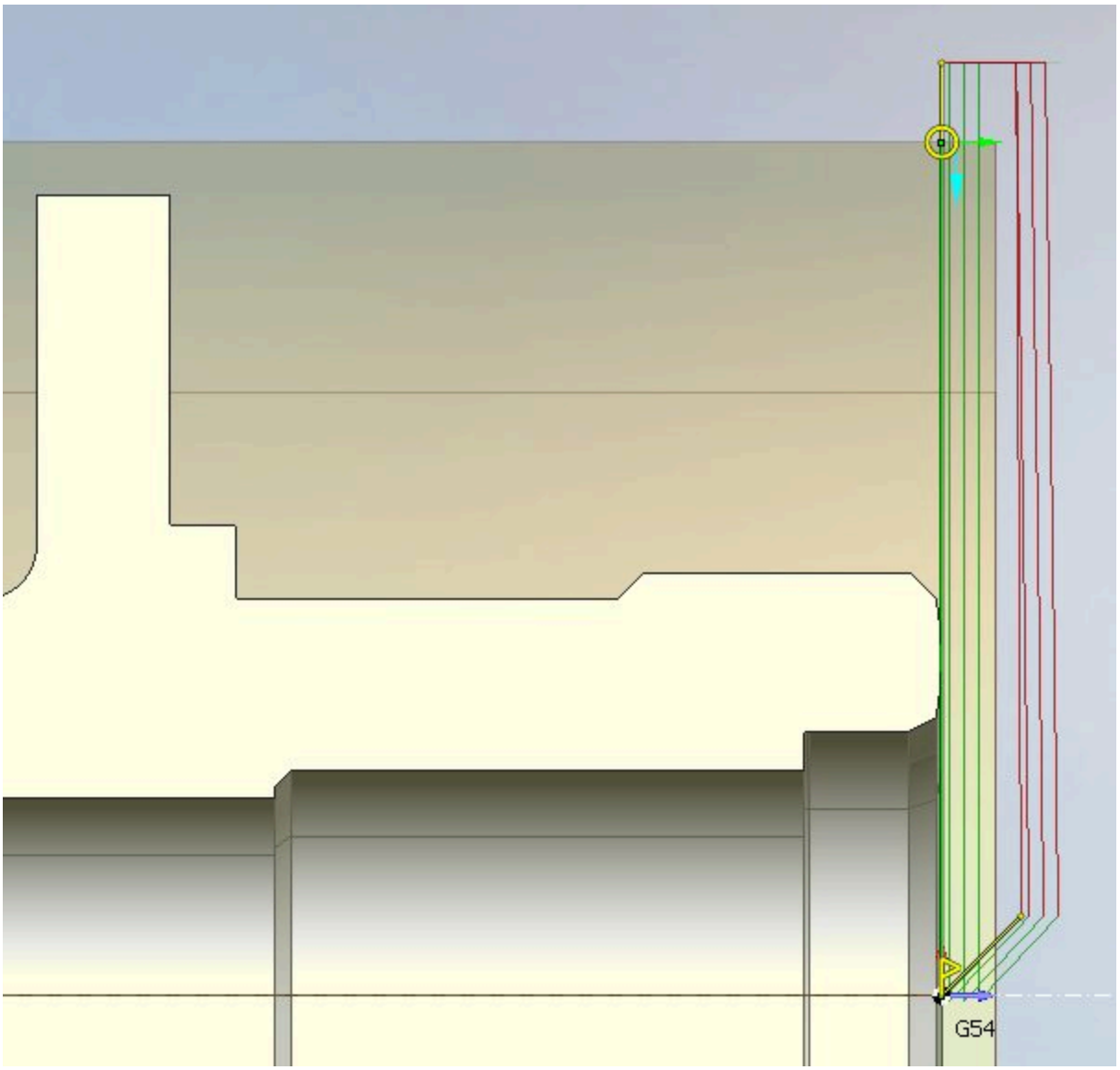
f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.



G72 W(Δd) R(e) ;
G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
The meanings of Δd, e, ns, nf, Δu,Δw, f, s, and t are the same as those in G71.

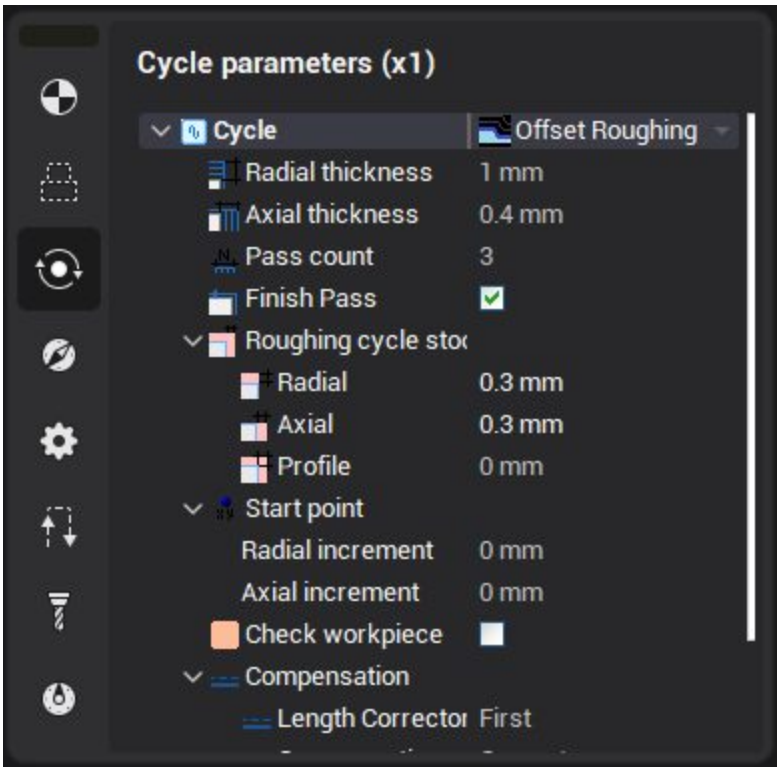
Facing

Facing cycle based on Offset roughing or simple Profile machining cycles. It just defines another source geometrical contour as right or left face of the part with extending it to the workpiece bounds.



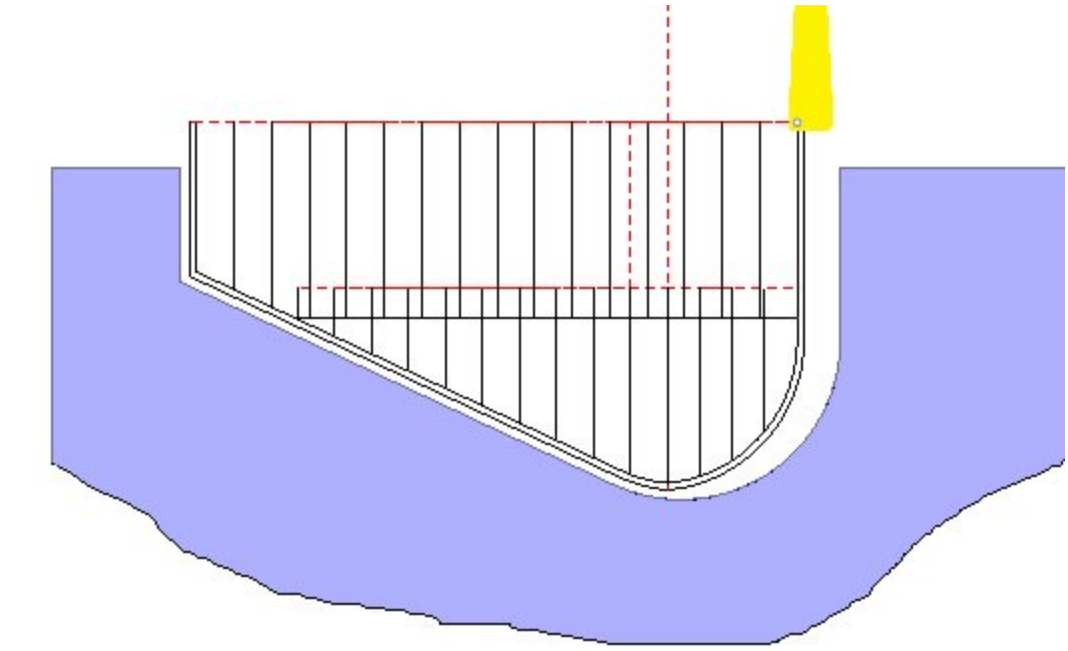
It designed to simply machine front and back axial faces of the part and frequently it is the first operation that prepare surface of the workpiece for the following machining.

Parameters of this item depend on the active cycle type. By default it is Offset roughing so you can redefine Axial thickness, pass count and stocks etc. See documentation of exact cycle type to get more detailed info about parameters of each cycle.

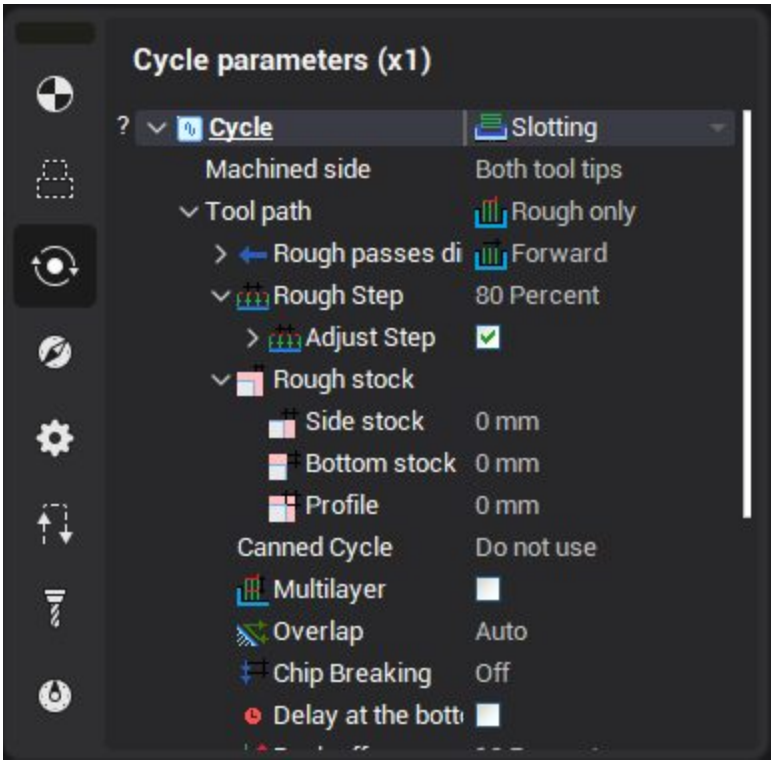

Cycle parameters (x1)
Cycle
Offset Roughing
Radial thickness
1 mm
Axial thickness
0.4 mm
Pass count
3
Finish Pass
Roughing cycle stoc
Radial
0.3 mm
Axial
0.3 mm
Profile
0 mm
Start point
Radial increment
0 mm
Axial increment
0 mm
Check workpiece
Compensation
Length Corrector
First

Grooving cycle

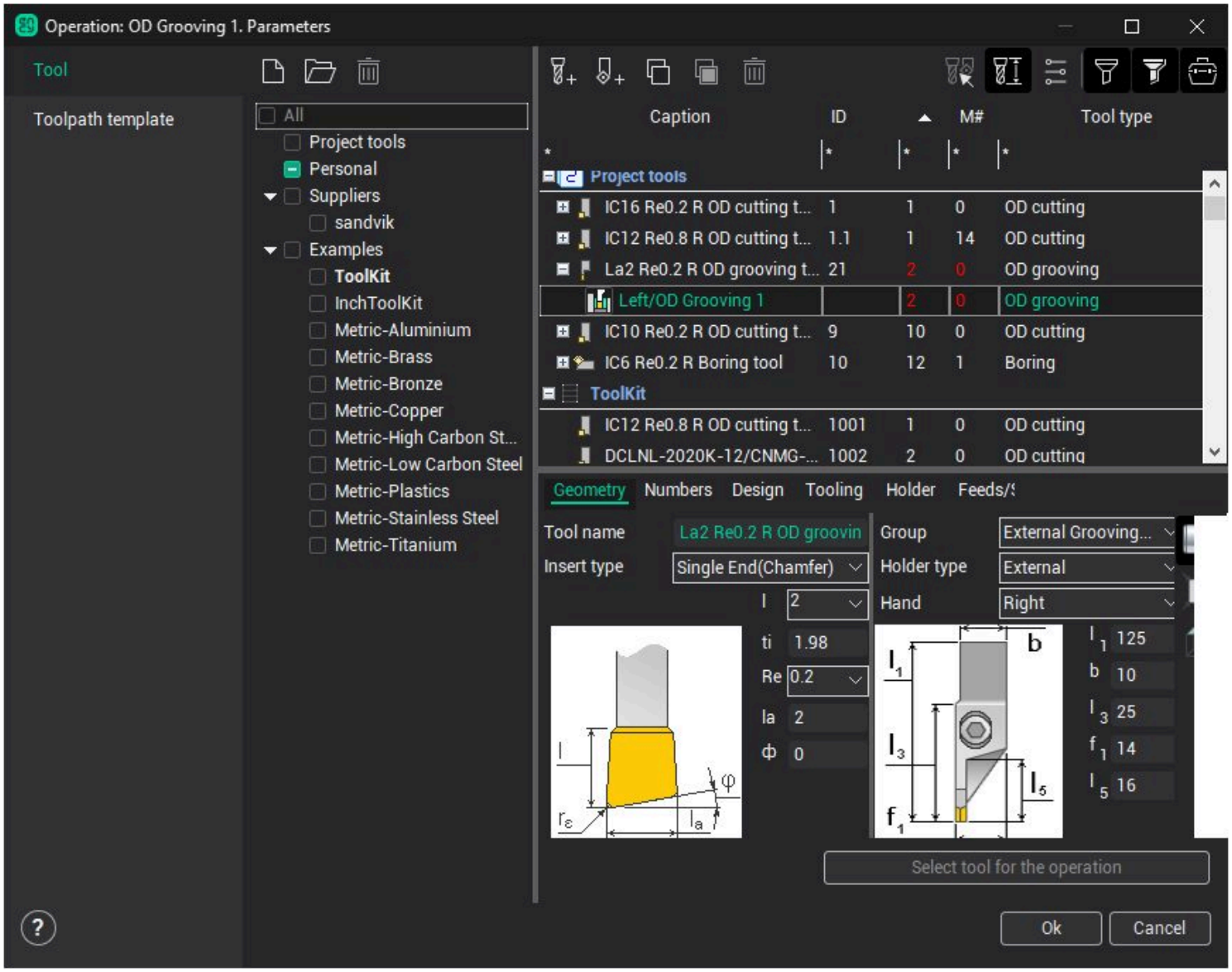
Advanced grooving element generates the complex tool path for the any kinds of the lathe grooves. It allows to machine the outer radial, inner radial, face or inclined grooves.



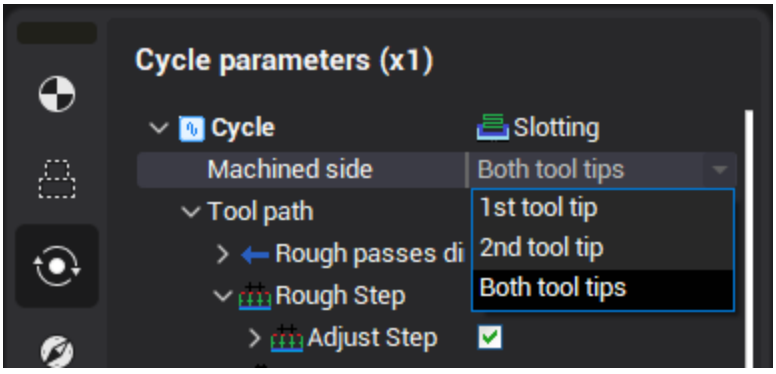
The cycle parameters can be defined in the properties window.



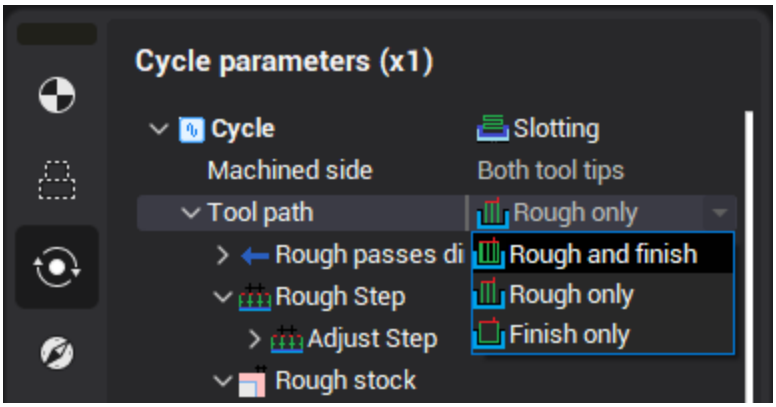
The groove cycle defines the bottom point of the groove contour and machines the both sides of the groove by the different tool tips. every side is machined from the end point of the profile to the bottom point. The tool tips are defined in the Tool dialog that is shown below.



Machined side parameter allow to machine only one side of the groove. If 1st tool tip is selected then the only side that is touched by the first tool tip will be machined.

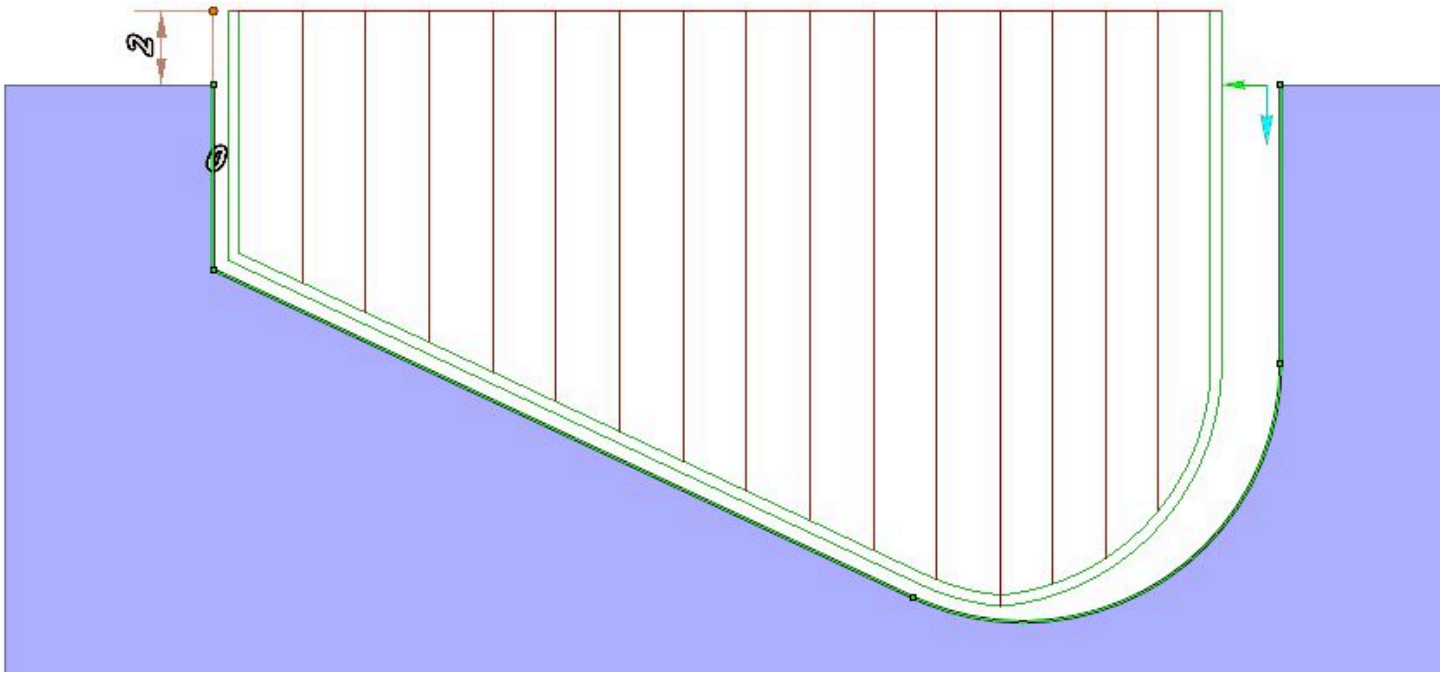


Advanced grooving cycle generates rough and finish passes. Tool path parameter defines what passes must be generated.



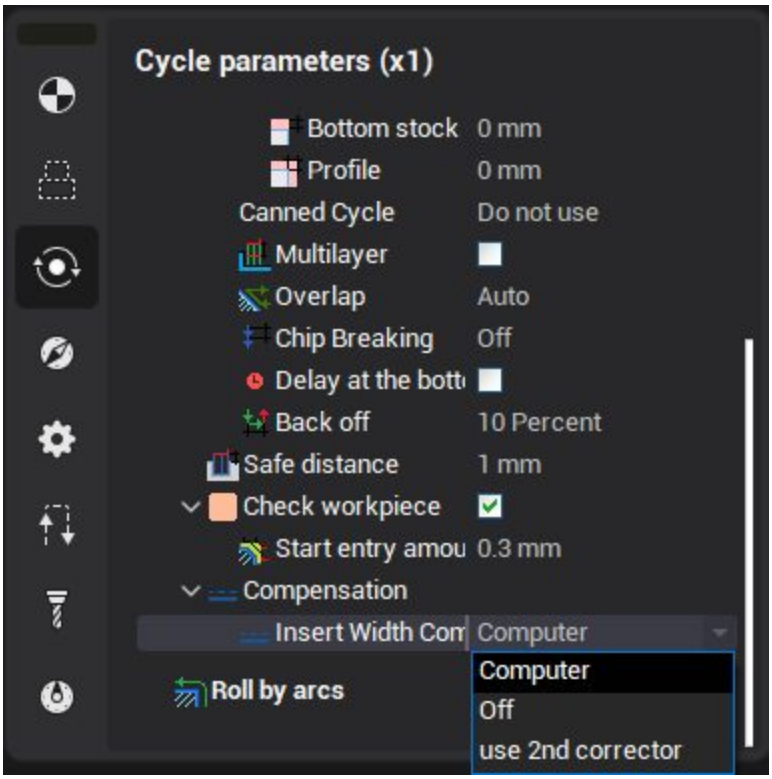
If Finish only is selected then finish pass only will be generated and the dialog will show only the parameters for the finish path. If Rough only is selected then the rough path will be generated and the parameter for the rough pass only is shown in the dialog. If Rough and finish is selected then all parameters are shown.

Safe distance parameter defines the distance from the groove top to the level of the rapid motions. To edit the safe distance it is possible to drag the point in the graphical window or input the value in the dialog.



Parameters of the finish path

Insert width compensation defines the correctors using.

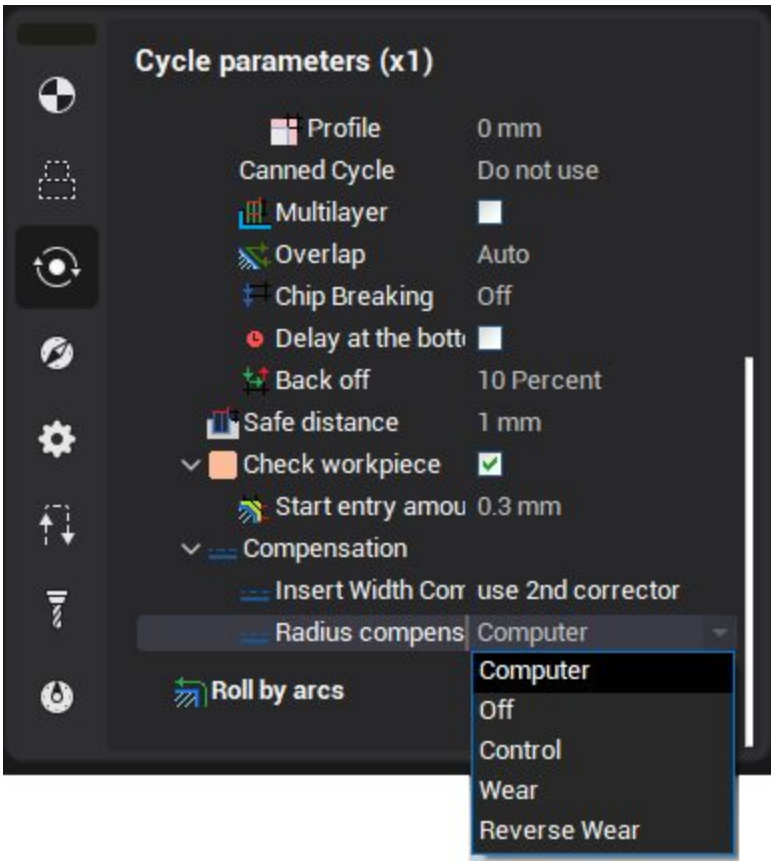


Computer mode generates the tool path for the first corrector only.

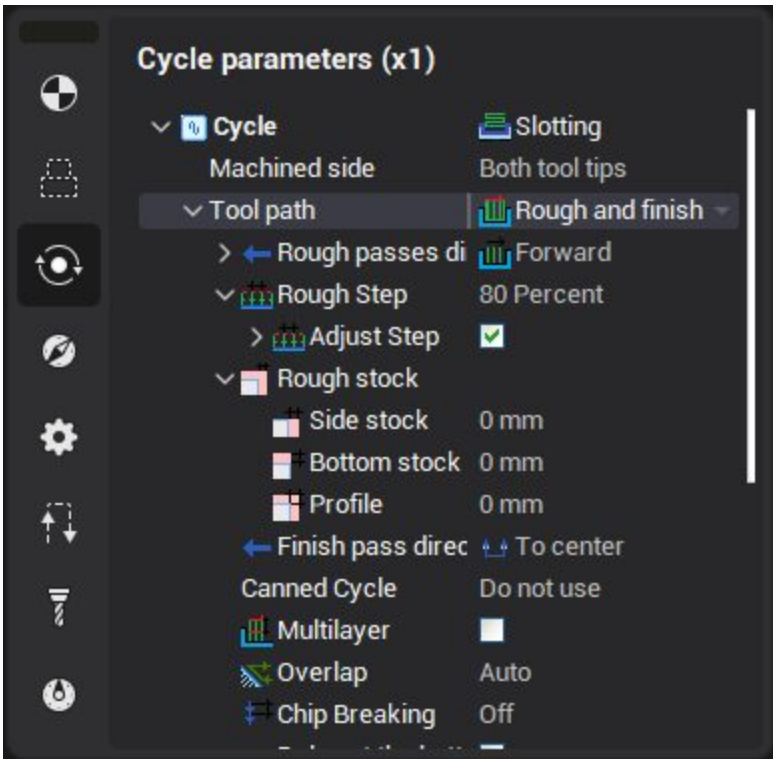
Off mode is not realized.

Use 2nd corrector mode generates the tool path for the first and second correctors.

Radius compensation parameter is described here.



Parameters of the rough path



Rough step defines the distance between the rough passes. It can be defined in the percents of the tool width or in the units of the length. If option Adjust step is defined then CAM system automatically changes the step for the equal force on every plunge. Max Step deviation defines the maximal deviation

of the adjusted step from the defined step. This value can be defined in the length units or in the percents of the tool width.

Canned cycle option allow to use or not use the canned cycles in the tool path.

Rough stock defines the additional stock for the rough passes.

Multilayer option is necessary to generate the rough passes in some layers. It is possible to define the layers count or the depth of the layer.

If Overlap option is enabled then the additional tool path is generated from the end of the rough cut to the end of the previous rough cut. In Auto mode, the Overlap option is enabled when the Multilayer option is enabled..Overlap option would be inactive if toolpath uses the canned cycles.

Chip breaking option can be enabled on the first rough cut or on every rough cut. It is possible to define the number of the breaks or the step for the chip breaking. The return distance for the chip breaking can be defined in the length units or in the % of the plunge step.

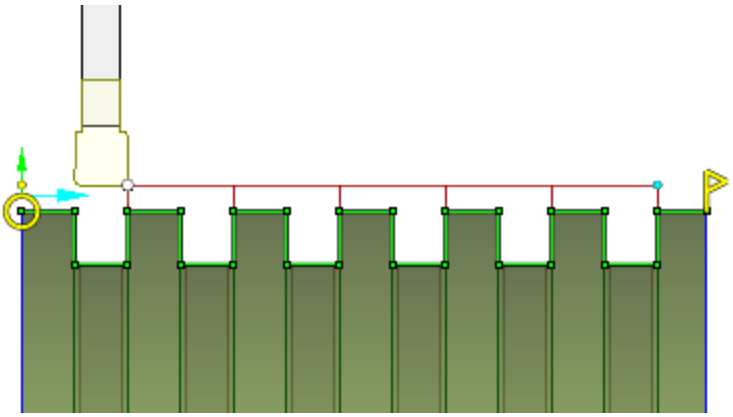
If Delay at the bottom option is enabled then the Delay command is generated in the end of the rough cut. The delay time can be defined in seconds or in the turns of the part.

Back off distance can be defined in the length units or the percents of the rough cut step.

The Check workpiece feature can significantly reduce the machining time. That portion of the moves on the working feed, which is outside the workpiece, is replaced by rapid movements. The parameter Start entry amount determines the distance from the workpiece at which to switch the feed.

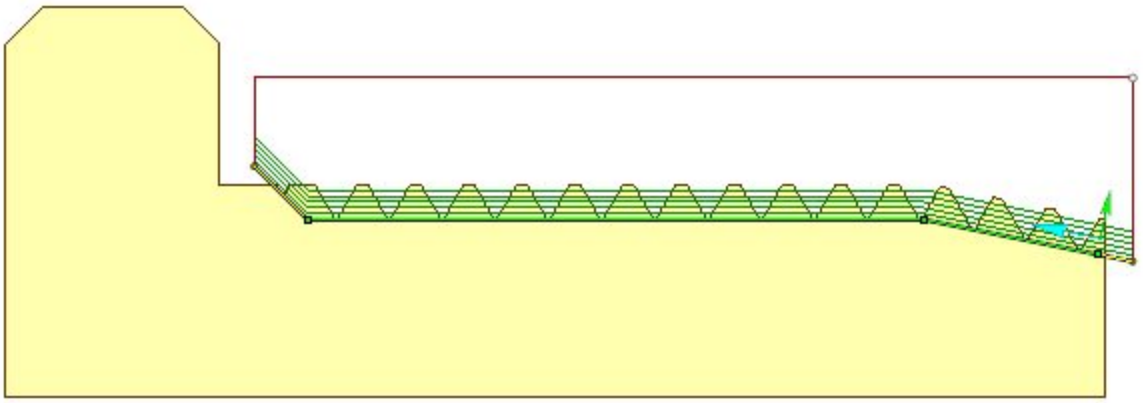
Slotting cycle

The Lathe slotting cycle is the same with the Grooving cycle just more finely tuned for rectangular grooves machining. So rectangular grooves now can be cleared in one click with minimal tool motions.



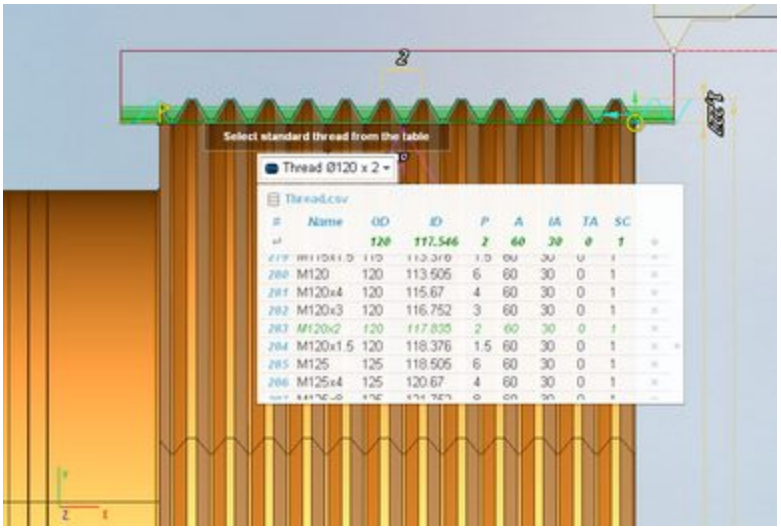
Thread cycles

Thread cycle allows to generate passes (one ore more) for creating thread with specified parameters.



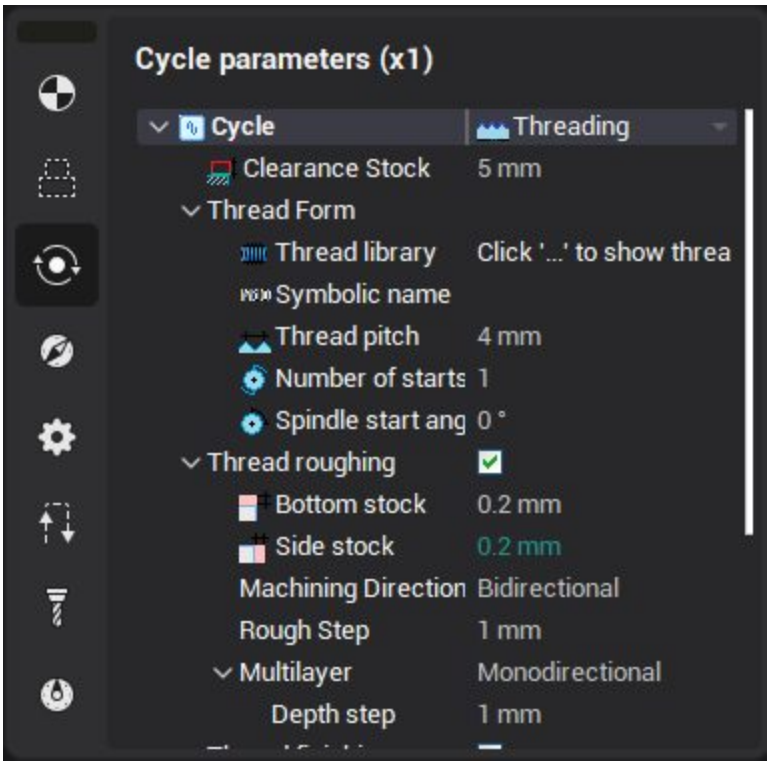
Threading area is specified by geometrical contour. Contour defines thread bottom diameter (inner diameter for external thread and outer diameter for inner thread). Second diameter is calculated by Depth parameter. Thread type - inner or outer - is specified by contour machining side (Perpendicular arrow near contour start point). Thread type - left or right - is specified by contour machining direction (Parallel arrow near contour start point) and spindle rotation direction. Contour approach and retract areas allows to set prolongation or chamfer for tool output.

Lathe threading cycles visualization exists. Interactive representation of the thread appears on the screen just after adding one of the threading cycles. Dimensions for thread pitch, depth top and bottom diameters can be edited directly in the graphical screen.



Pop-up action bar with the thread name allows to open threads table quickly and select one of the standard threads. The list of threads can be modified in this panel intuitively.

The Properties window for the job assignment item has the following parameters.



The Clearance stock is the distance from the thread to the transitions level. The cycle goes through this level while moving from the end of the previous pass to the start of the next one.

Thread profile is specified by tool form, parameters in Thread form group and for a free form thread only you additionally need to specify a threaded groove profile as a separate Job assignment's item.

Symbolic name it's just an optional symbolic designation of the thread type and size, M10 for example. You can use it to quick search the thread in the threads' table.

Thread pitch can be set by two cases. In the first case lead is defined as the distance between two same points of the profile, located on the neighboring threads. In the second case lead is specified by count of turns per length unit.

Operation allows to create multistart threads by editing Number of starts and Spindle start angle parameters. Various cycle types use Number of starts differently. In ISO G76 numbers of starts send to cycle as a parameter. But some machines have not this parameter in cycle. In this case it is possible to create multistart threads by making the same cycle with another Spindle start angle parameter value. Another way is using ISO G92 or ISO G32/33. In this case operation automatically generates passes with different spindle start angles.

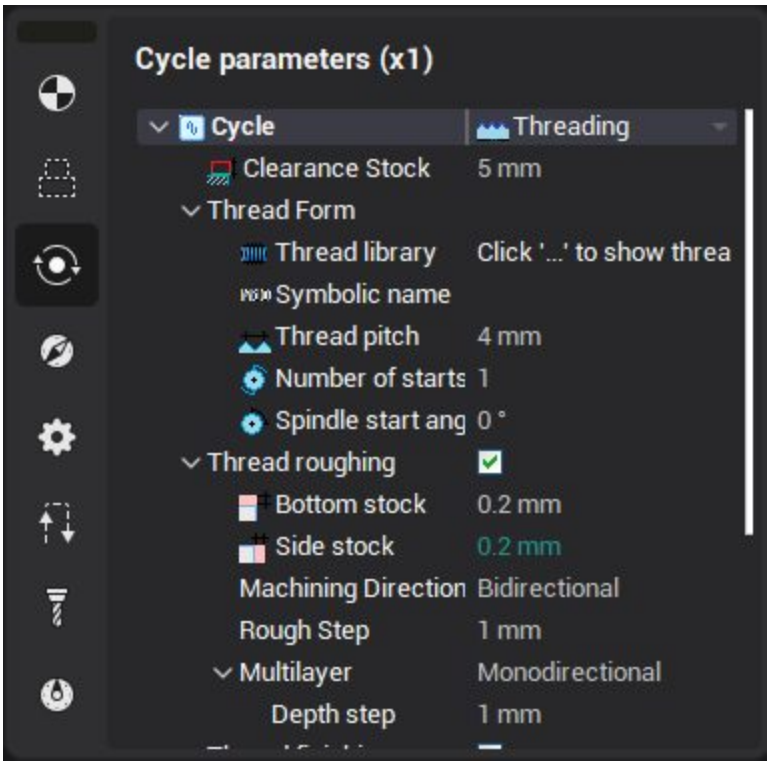
Parameters in Sequence, Roughing and Finishing groups defines number and mutual arrangement of passes. They are different for the type of a thread profile, therefore, see the relevant topics:

- Standard form thread;
- Free-form thread.

Thread toolpath can be generated in various output formats: Multipass thread cycle (ISO G76), Single pass thread cycle ISO G92 or Advanced (expanded) thread machining.

Standard form thread

The Properties window for the job assignment item of the standard form threading operations has the following parameters.



Thread profile is specified by tool form and parameters in the Thread form group.

Value in Depth field defines thread profile height (difference between outer and inner diameters). This value must have positive value. Direction of this value calculates automatically and depends from contour Machining side parameter.

If plunge mode is Flank or Alternate Flank, then Thread angle and Inclination angle parameters defines angle of tool plunge at each pass.

Thread angle	Inclination angle
	

Parameters in Sequence group defines numbers of starts and plunge mode for each pass.

Value in Sequence combobox defines plunge mode. The following types of strategies are available:

- Radial. The direction of plunge is perpendicular to the axis of rotation.



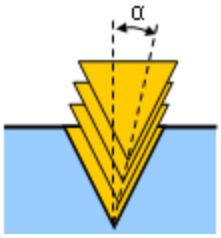
- Flank. The plunge is made along one side of the ledge.



- Alternate flank. Plunge is made alternately along the two lateral sides of the ledge.



- Modified flank. Plunge with angle, specified at Angle parameter.



Practically thread is processed by several passes. It allows to improve surface quality and reduce tool loading.

It is possible to specify the number of passes by setting a count directly or by setting the first pass depth. In the last case the number of starts calculates automatically from the thread profile depth.

If cutting depth is constant, then the plunge to the next layer leads to increasing machining area and tool loading. It is possible to calculate cutting depth to provides constant machining area and tool loading. The Determine cut depth from parameter can accept two values: equal area and equal depth.

Equal area	Equal depth
	

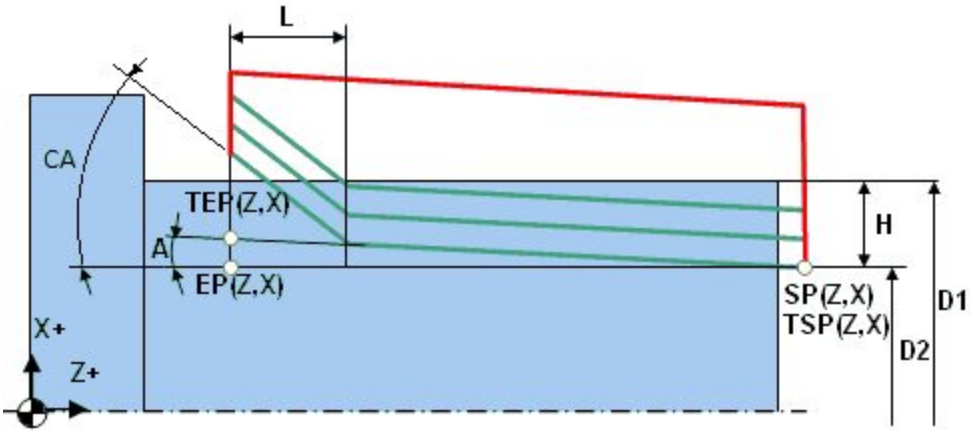
In Equal area mode cutting depth decreases at each level. It is possible to set Minimal cut depth. If calculated depth becomes less than this value, then the minimal cut depth will be used.

To ensure the cleanliness of the surface last pass is performed with very small stock, and then the smoothering of the finished profile is performed several times without any stock. Finish pass depth parameter defines finish pass stock, Finish pass count parameter defines count of passes along the ready profile, taking with finishing pass.

Threading cycle types

The toolpath of threading operations can be generated using various output formats. We consider each of them separately.

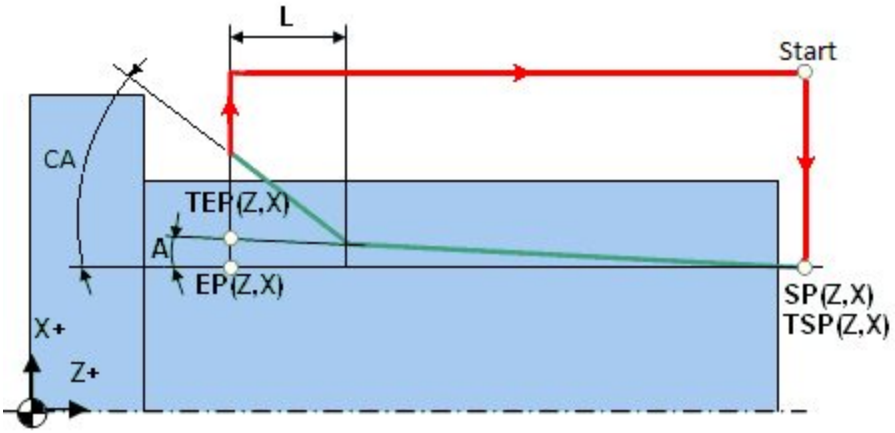
Multipass thread cycle (ISO G76) allows you to use a single frame of the NC-program to set all parameters necessary for machine to make a standard form thread. Required depth is reached automatically by generating several passes. Among the parameters of the cycle there are start and finish point coordinates, taper angle (for taper threads), size of chamfer for tool out, profile angles, thread depth, passes count, plunge strategy and others. See NC control documentation for more information.



Example of NC program

```
G01 X70 Z5.0 F1.0 M08            (Approach to start point)
G01 X70 Z5.0 F1.0 M08            (Approach to start point)
G76 P010060
G76 X57.4 Z-24.0 P1.3 Q0.35 F2.0 (Calling G76 multipass thread cycle)
G00 X200.0 Z150.0 M09            (Retract)
```

Single pass thread cycle ISO G92 (can be G92, G78, G21 and others in different NC controls) generates closed set of moves for one threading pass. Picture below shows processing schema. Before calling this cycle tool is in Start point. Cycle is called by one frame of NC-program, defines thread start point, step, taper size, chamfer size and others. As a result of this frame the tool goes from Start to TSP point, thread to TEP point and returns to Start point. Usually threading is processed by several passes, so NC-program consists several cycle calls with various thread diameters. Therefore, it can be used for both standard-shaped threads and free-form threads.




```
                    Example of NC-program
X60.0 Z20.0 M08
G01 Z10.0 F1.0          (Approach to Start point)
G92 X29.4 Z-52.0 F2.0 (Calling cycle for one threading pass)
X28.9                   (Modal calling G92 cycle with another diameter value)
X28.5                   (Modal calling G92 cycle with another diameter value)
X28.1                   (Modal calling G92 cycle with another diameter value)
X27.8                   (Modal calling G92 cycle with another diameter value)
X27.56                  (Modal calling G92 cycle with another diameter value)
X27.36                  (Modal calling G92 cycle with another diameter value)
X27.26                  (Modal calling G92 cycle with another diameter value)
G00 X200.0Z150.0M09     (Retract)
```

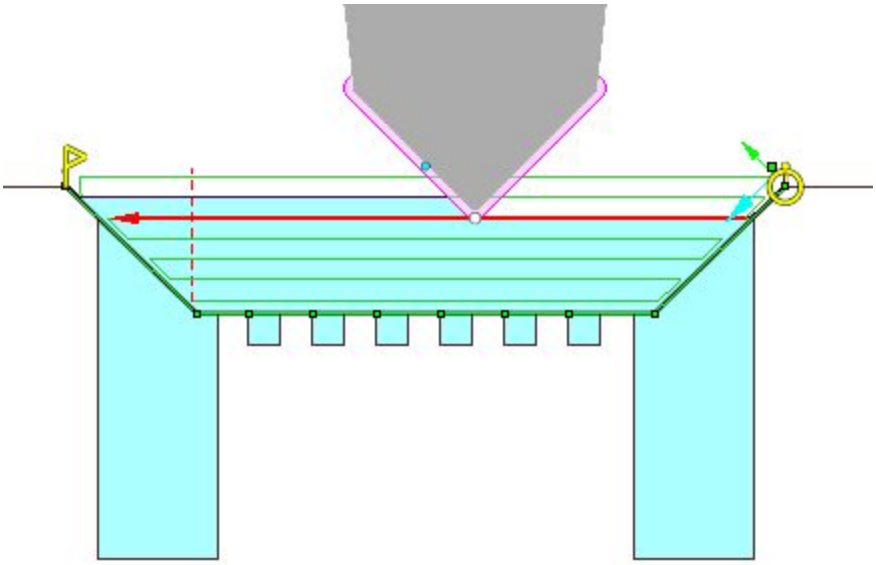
Advanced (expanded) thread machining is processed by using ISO G32/G33 (can be different in various machines). This command activates continuous cylindrical, taper or face threading mode with constant step. In this mode synchronization between tool movement and spindle rotation is enabled. All tool movements will processed in thread mode until the interpolation switching or rapid toolpath command will be detected. If tool moves parallel to the spindle rotation axis, cylindric thread will be formed. If tool moves both parallel and perpendicular to spindle rotation axis simultaneously, taper thread will be formed. It is possible to form special face thread, if tool moves perpendicular to spindle rotation axis. In this case groove looks like spiral of Archimedes will be formed at face.

G32/G33 command does not generate any moves, so all working tool moves, approaches, retracts, transitions to the next passes must be programmed in NC program directly. It can be used for both standard-shaped threads and free-form threads.

```
                    Example of NC program
G00 X60.0 Z10.0 M08 (Approach to Start point)
G00 X29.4           (Approach to start of pass 1)
G32 Z-52.0 F2.0     (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
```

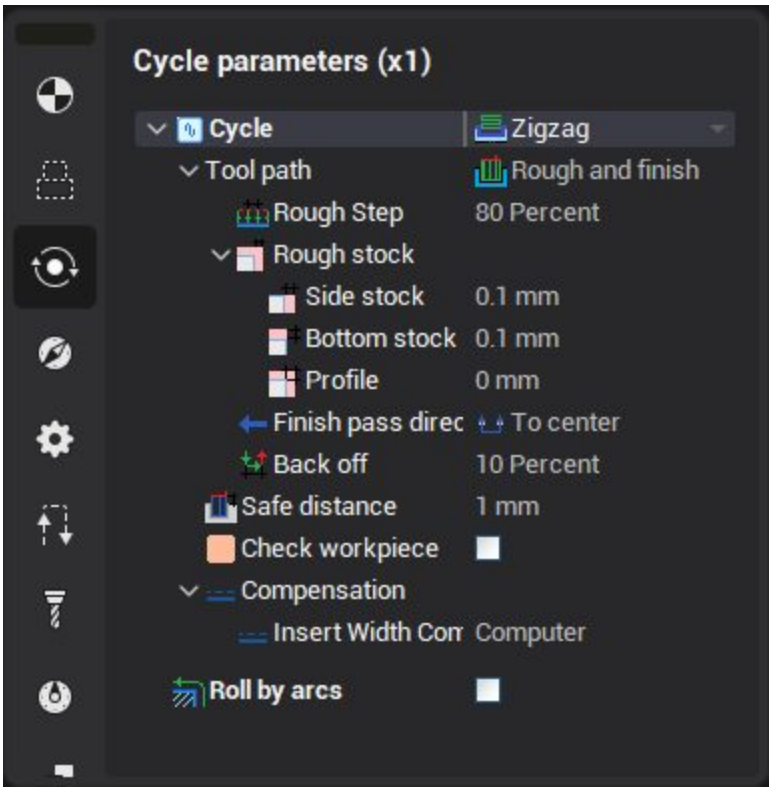
```
X28.9               (Approach to start of pass 2)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X28.5               (Approach to start of pass 3)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X28.1               (Approach to start of pass 4)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.8               (Approach to start of pass 5)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.56              (Approach to start of pass 6)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.36              (Approach to start of pass 7)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X27.26              (Approach to start of pass 8)
G32 Z-52.0          (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X200.0 Z150.0 M09   (Retract)
```

Zigzag cycle

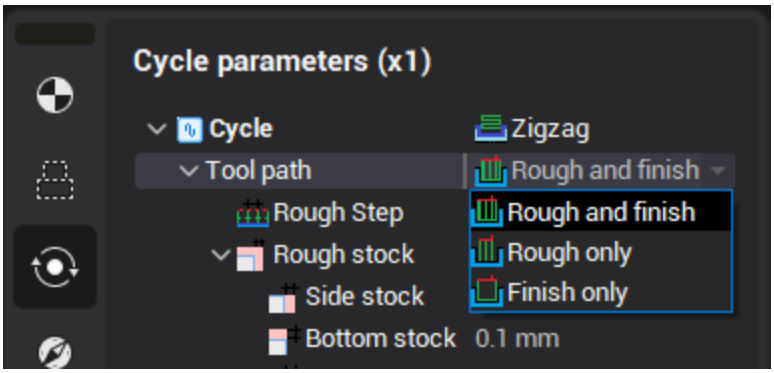


Zigzag lathe cycle generates toolpath useful for machining areas which are closed from both sides with neutral tool. The tool is lowered down layer by layer, and performs horizontal moves with changing direction for each layer.

Common parameter of the cycle



Zigzag cycle generates rough and finish passes. Tool path parameter defines what passes must be generated.

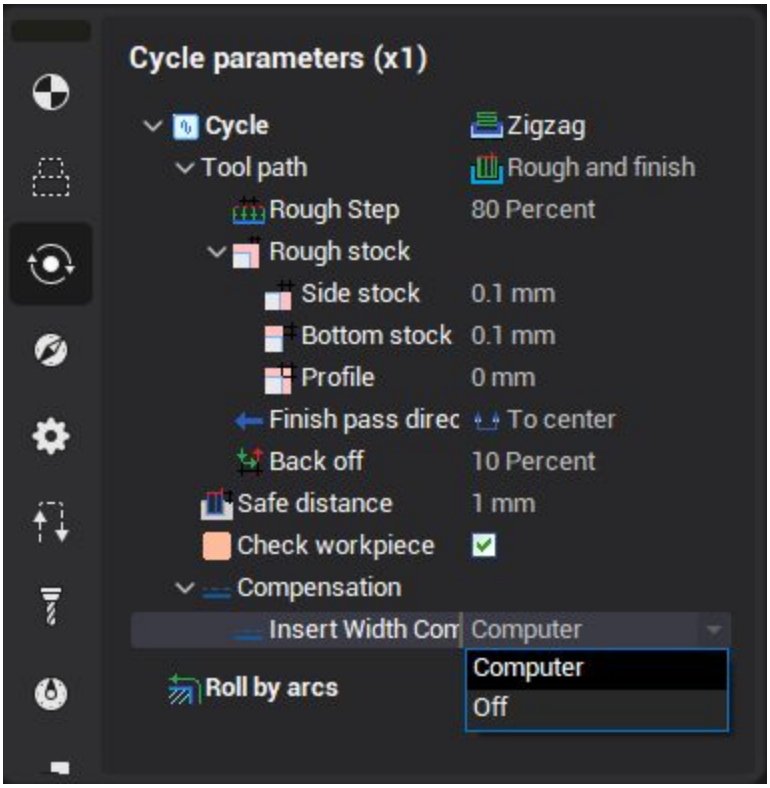


If Finish only is selected then finish pass only will be generated and the dialog will show only the parameters for the finish path. If Rough only is selected then the rough path will be generated and the parameter for the rough pass only is shown in the dialog. If Rough and finish is selected then all parameters are shown.

Safe distance parameter defines the distance from the groove top to the level of the rapid motions. To edit the safe distance it is possible to drag the point in the graphical window or input the value in the dialog.

Parameters of the finish path

Insert width compensation defines the correctors using.



- Computer mode generates the tool path for the first corrector only.

Parameters of the rough path

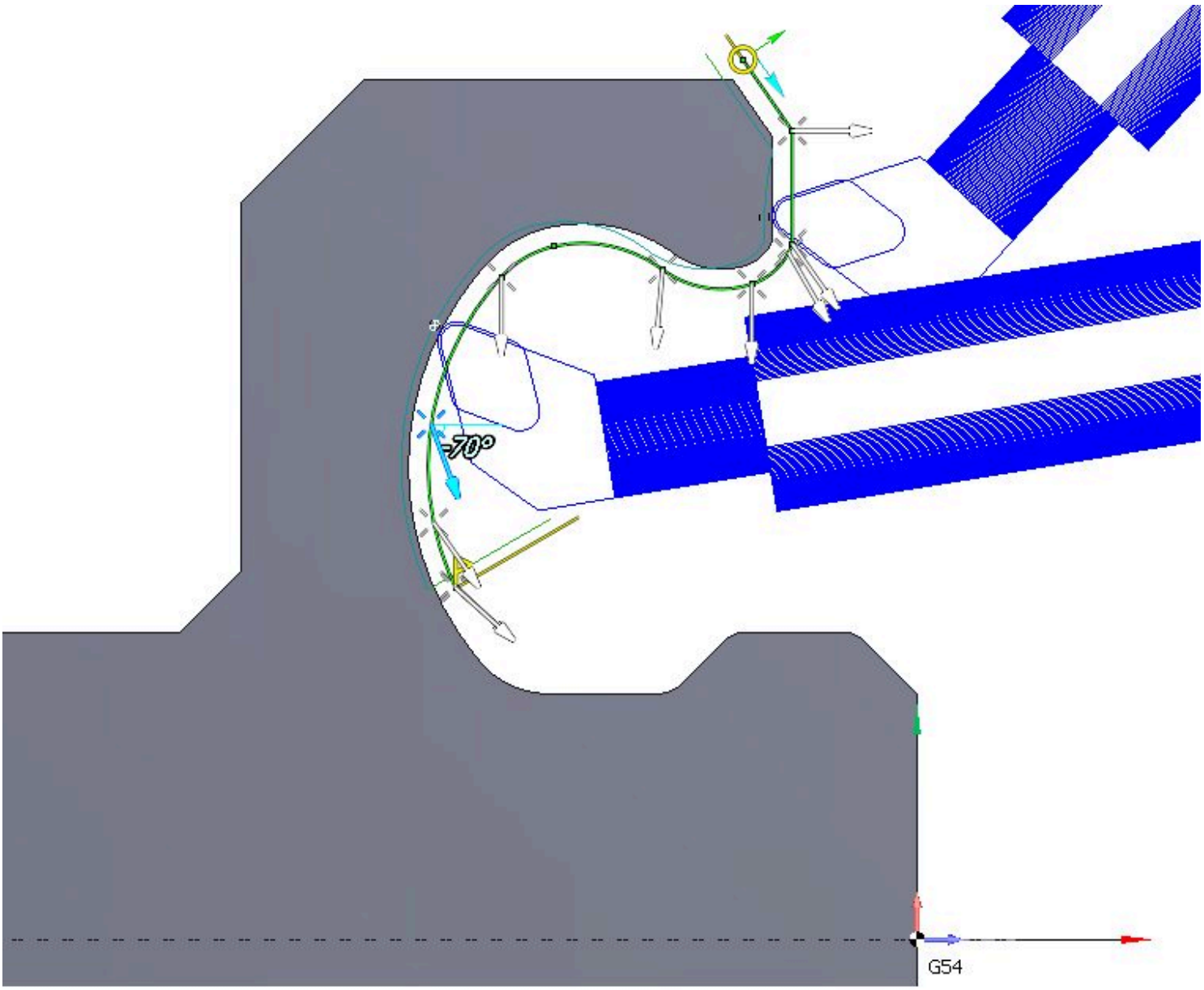
Rough step defines the distance between the rough passes. It can be defined in the percents of the tool width or in the units of the length.

Rough stock defines the additional stock for the rough passes.

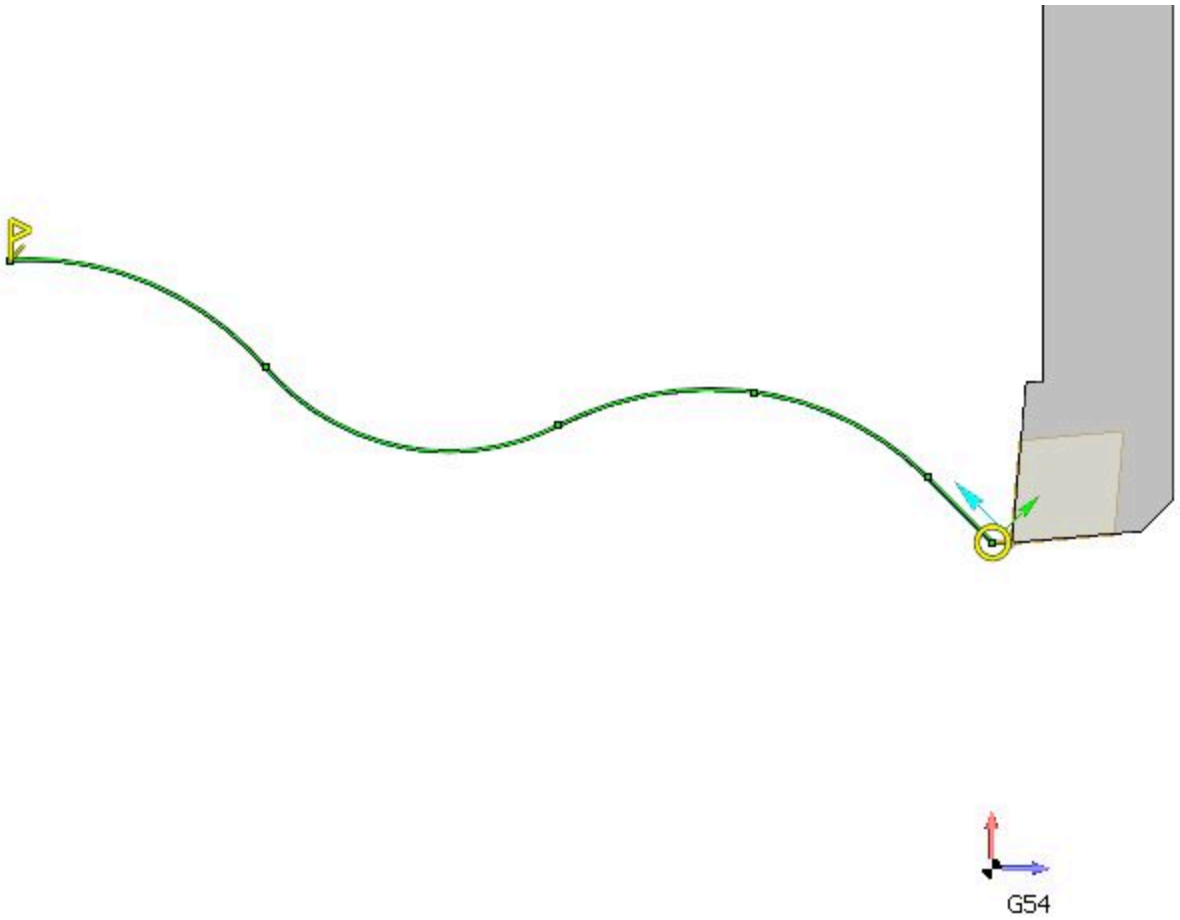
Check workpiece option removes all idle parts of the working motions that really do not cut the workpiece.

4-axis turning

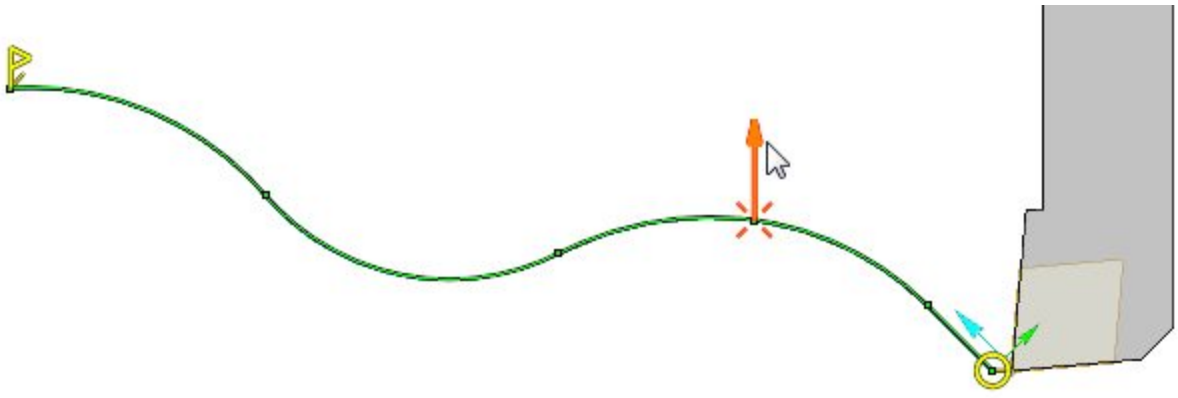
4-axis turning cycle allows to machine hard-to-reach areas of a part with just one operation by continuously changing of the tool inclination angle in the process of moving along a contour (using the 4-th axis of the machine, if available). This reduces the number of tool changes, thus increasing productivity. In addition to the defined contour geometry to calculate the toolpath information about the tool inclination angles in some points of the contour also is used. The user can specify any number of angles of the tool, which can be set at an arbitrary point of the contour.



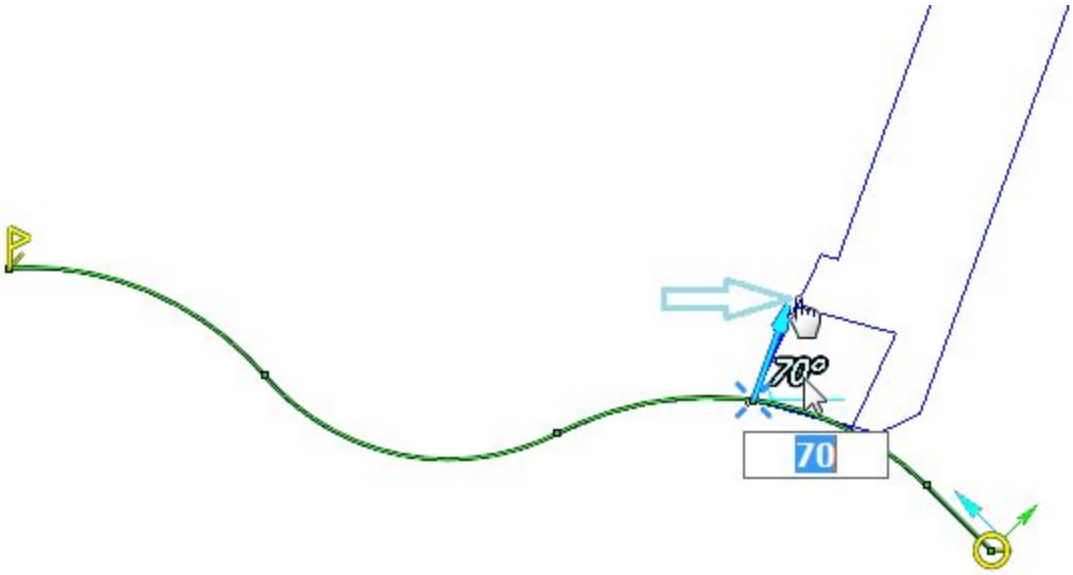
Almost all of the parameters for this type of machining can be specified interactively in a graphical window. Immediately after adding the "4-axis turning" item with some contour just the specified contour with start and end markers will appear. There will be no of additional visible tool inclination vectors. In this case, after calculating the toolpath angle of the tool will remain the same, so what, he specified in the operation's initial state properties inspector.



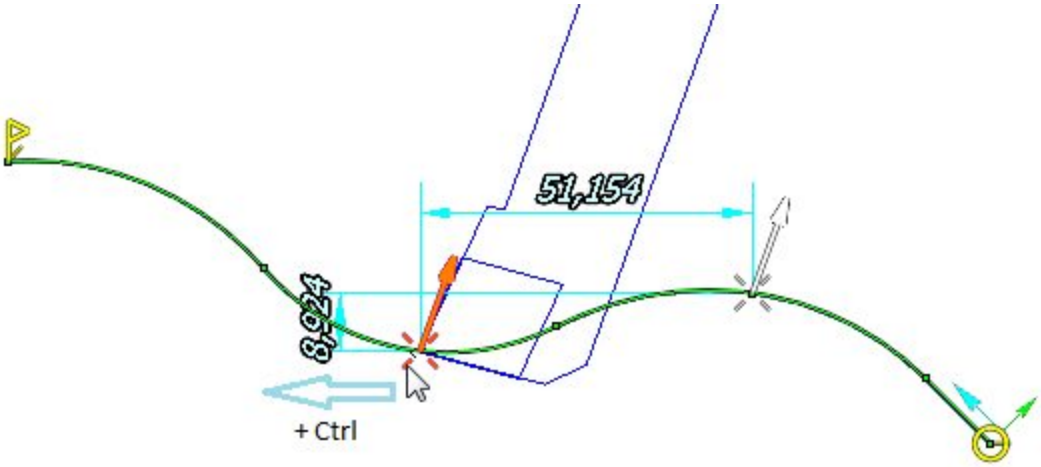
However, if you move the mouse cursor in the area of the nodal points of the contour then tool inclination vectors will be highlighted. By default, they are invisible, because the angle of the tool in these points is already corresponds to these vectors.



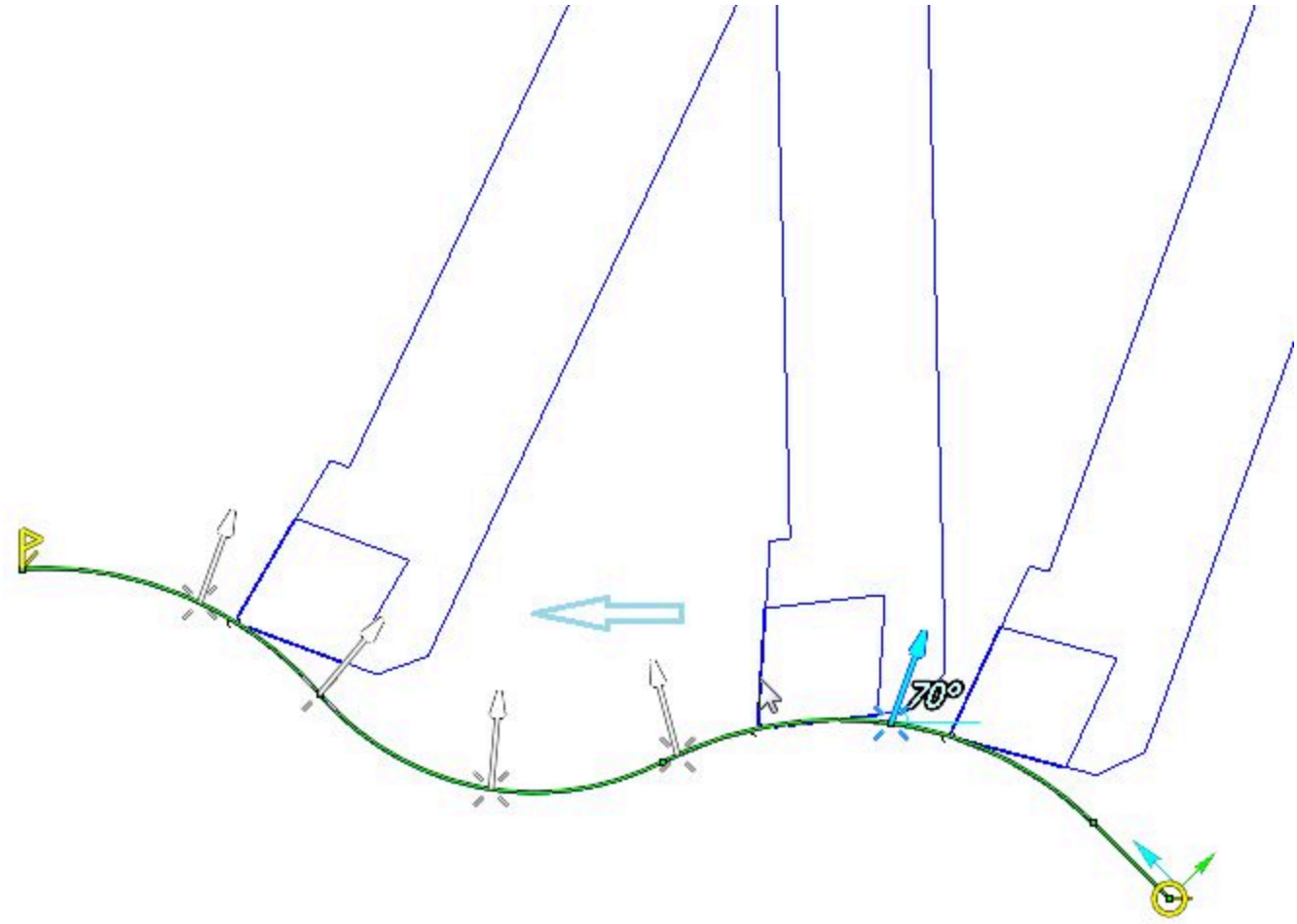
If you want to change the tool angle in the highlighted point, you can clutch and drag the vector. In this case the display will show angular dimension, which determines the angle of the tool relative to the axis of rotation, as well as the profile of the tool. Desired angle of the tool at a point can be achieved either directly by dragging with the mouse or by entering a specific value of the angle in the field that appears when you click at dimension's digits.



Now if you click anywhere on the screen then this vector becomes unselected and will remain visible. This way you can add new vectors. To define the inclination vector in the intermediate not nodal point of the contour you can clutch the cross at the bottom of vector and drag it. Vector will move behind the mouse pointer along the contour. If hold down Ctrl at this time, then the copy of the original vector will be changed.

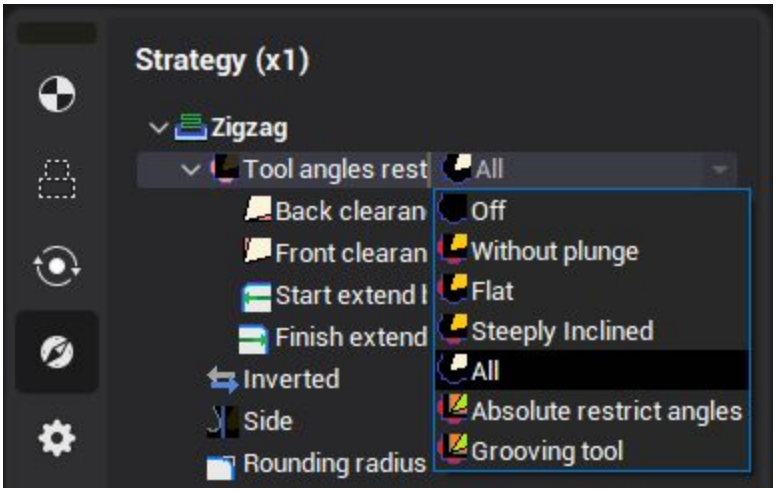


The slope between two given vectors vary continuously from the first vector to the second. It can be checked by clutching the tool profile, that appears when you select a vector, and drag it along the contour. The tool will move following the mouse pointer along the whole contour. The slope of the tool, will also vary continuously, and will conform to the slope of vector at the current point. Thus, If you turn on the visibility of part or workpiece, you can easily control the position of the tool for the presence of collisions, as well as to achieve optimal cutting angle.

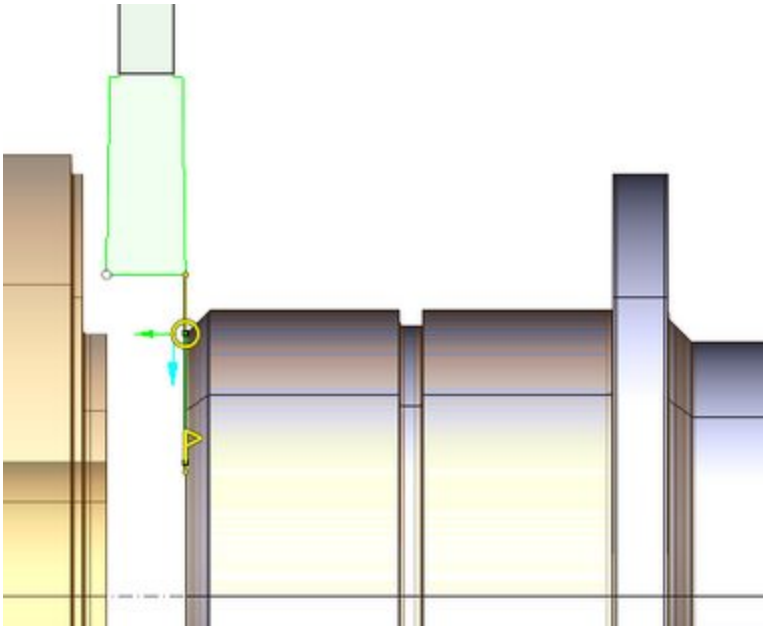


To delete unwanted inclination vectors of the tool you can select them on the screen and press the Del key on your keyboard.

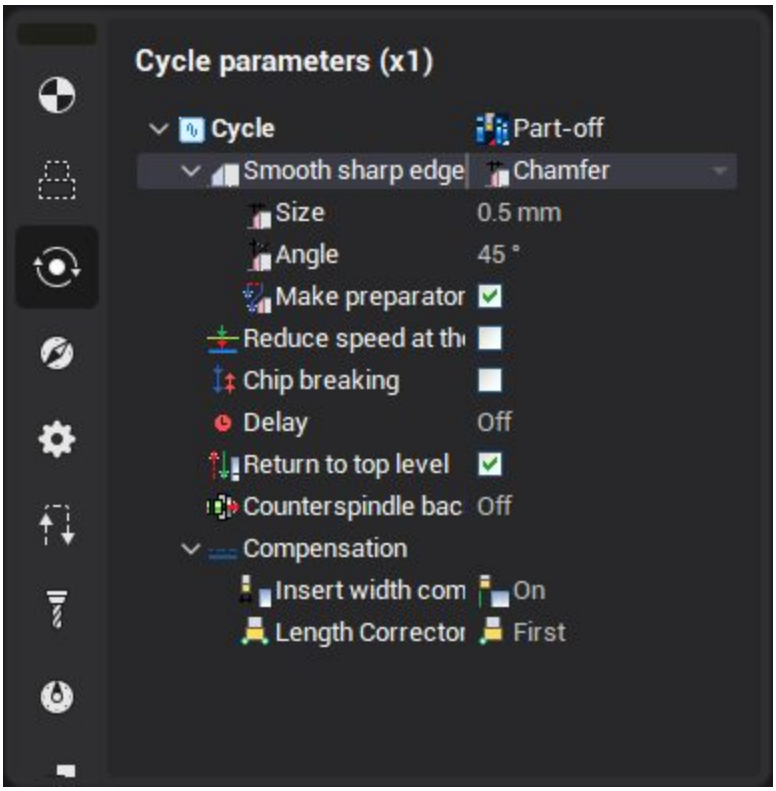
The remaining machining parameters for the "4-axis turning" item have no differences from other job assigment items of the lathe operations. They can be edited in the properties window of the job assignment item.



Parting-off cycle



The Lathe part-off cycle designed to make possible simply separate ready machined part from the remaining workpiece with the simultaneous finishing of the opposite face of a part. The working contour determined automatically as maximal (left or right) side of the part from highest to lowest point of the workpiece.



You can define parameters of the chamfer or rounding to get smooth edge at the finished face of a ready part. It is also possible to enable chip breaking while parting off and delay at the bottom or every peak.

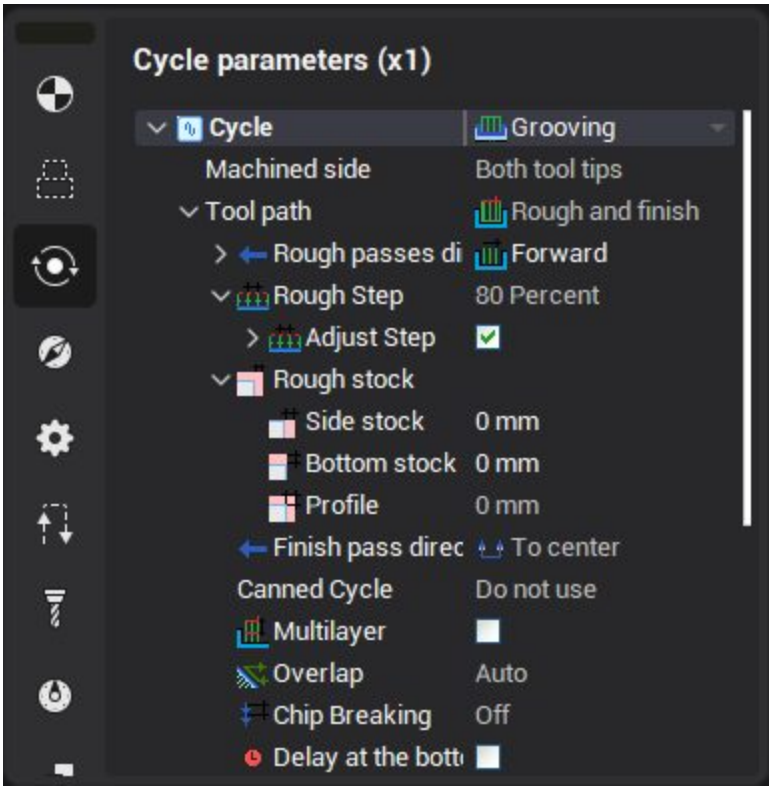
Return to the top level can be disabled if you want to use current tool as a stopper for the bar ejected from a feeder.

If insert width compensation is enabled, it can automatically add tool width to the axial coordinate of machining face. Actual tool length corrector left or right can be simply switched.

Counterspindle back off parameter allows to define distance which counterspindle must move back after finishing the cut. Axis name that makes axial moves of the counterspindle must be determined here.

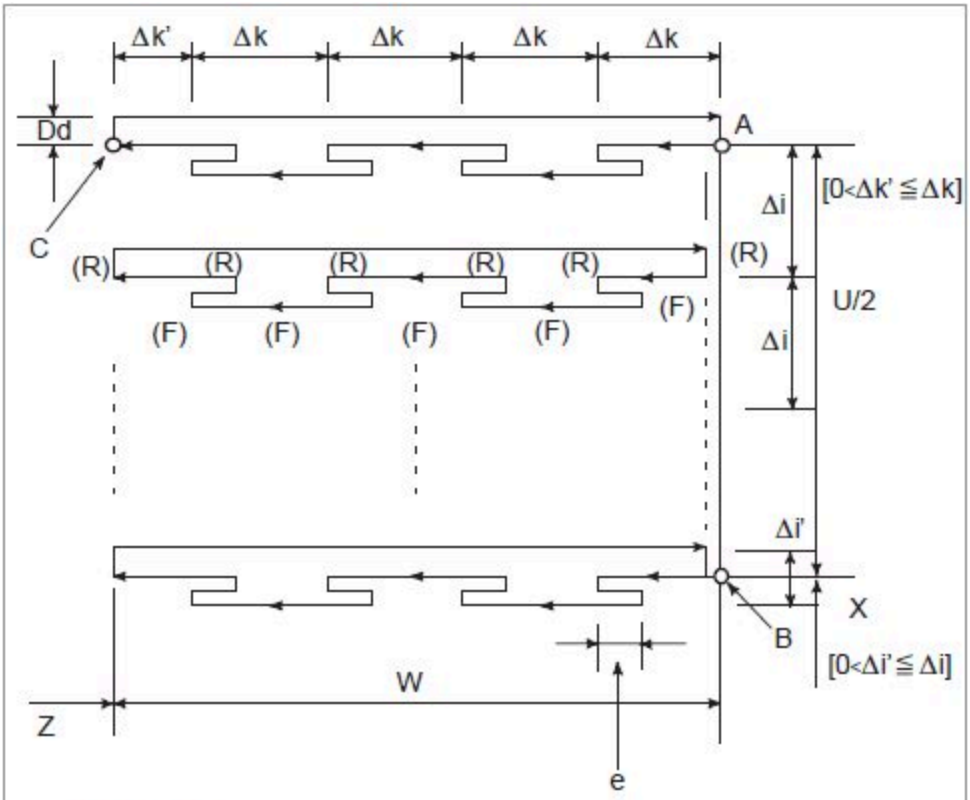
Peck groving/drillling cycle

The grooving cycle generates one from both canned cycles ISO G74 or ISO G75 based on the box of the defined profile. The cycle parameters can be defined in the properties window. Direction property switch between G74 and G75.



The extract from the Fanuc Operator's manual about cycles G74/G75 is shown below.

The following program generates the cutting path shown in the picture below. Chip breaking is possible in this cycle as shown below. if X(u) and P are omitted, operation only in the Z axis results, to be used for drilling.



G74R (e) ;

G74X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F (f) ;

e : Return amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5139, and the parameter is changed by the program command.

X : X component of point B

U : Incremental amount from A to B

Z : Z component of point C

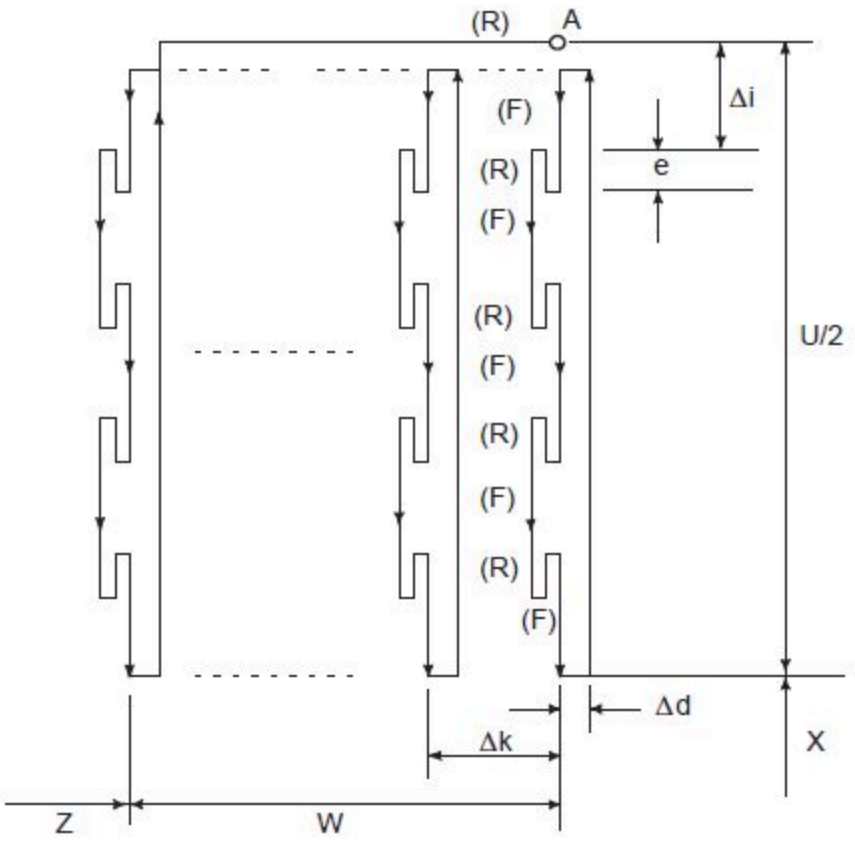
W : Increment amount from A to C

Δi : Movement amount in X direction (without sign)

Δk : Depth of cut in Z direction (without sign)

Δd : Relief amount of the tool at the cutting bottom. The sign of Δd is always plus (+). However, if address X (U) and Δi are omitted, the relief direction can be specified by the desired sign.

f : Feed rate

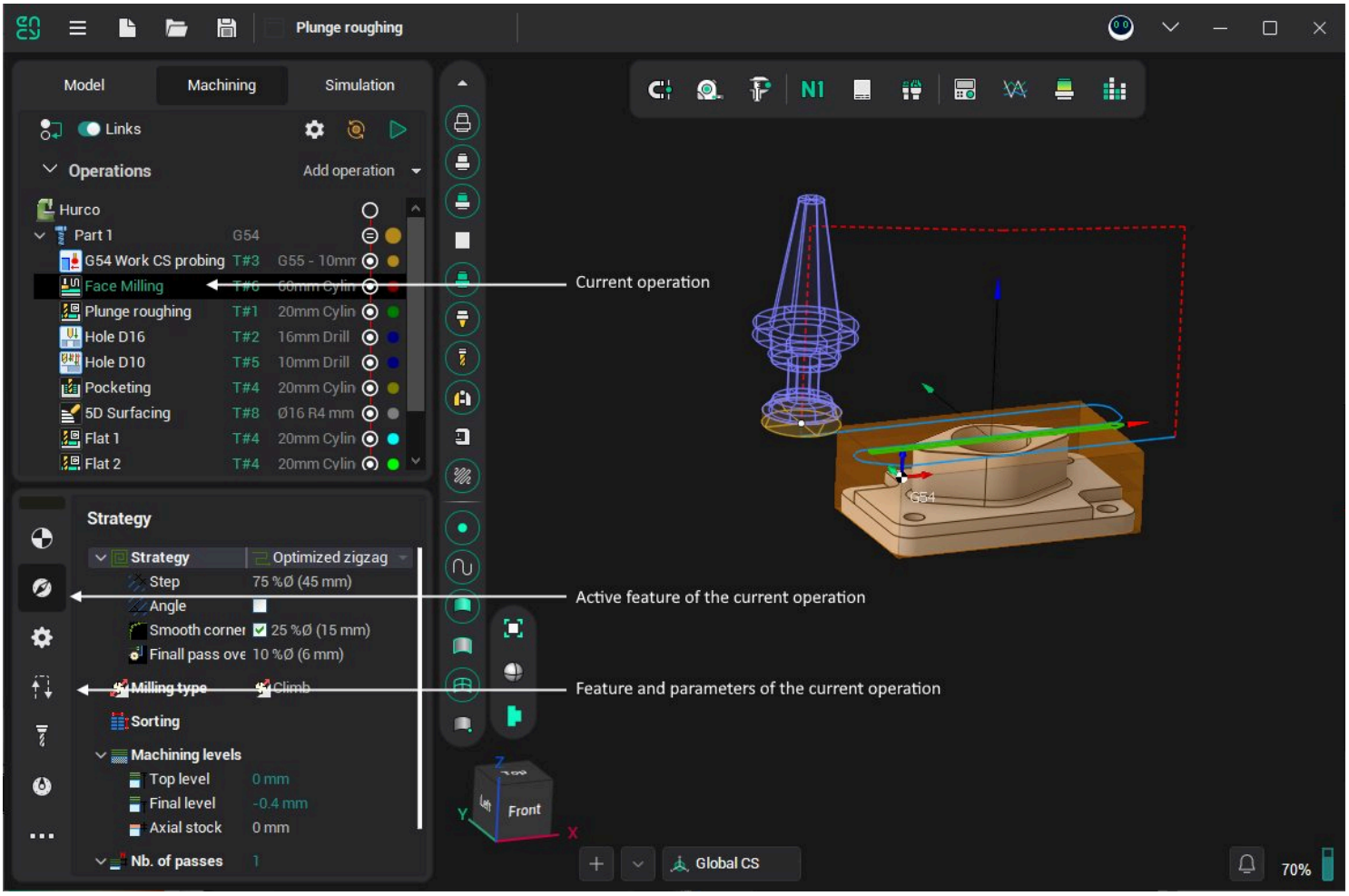


G75R (e) ;

G75X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F(f) ;

Operations setup

The parameters of an operation define what is to be machined and the way it is to be machined. Selecting a parameter node inside the operation tree changes the bottom side of the tab to display the tools used to define and edit the parameter properties.



Job assignment of the lathe operations

The Lathe operations lets generate toolpath along the specified contour. Operations use the simplified path calculation algorithm which frequently does not consider geometrical sizes of the tool holder and its admissible cutting directions. Tool nose radius and its plunge angle are considered only. Current workpiece state also is not considered if not enabled. Any curve (imported, drawn in 2DGeometry mode or turn generatrix of the part) can be specified as processing contour. Key difference of operation from other operations is possibility of editing specified contour in a Job assignment formation mode. The specified contour can be issued as standard cycles for stock removing, machining of grooves or thread. This possibilities allows engineer to generate necessary toolpath.

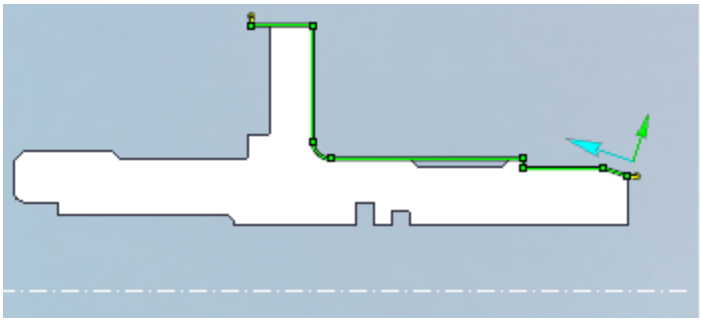
Job assignment element adding ways:

1. Adding existing curve. To do this select the desired curve (or set of curves) in the graphic window and click "Add" button for desired item (cycle) type.
2. Adding generatrix of surfaces group. To do this select the desired surfaces in the graphic window and click "Add" button for desired item (cycle) type.
3. Adding generatrix of the whole part. To do this click "Add" button for desired item (cycle) type without select any geometrical elements in the graphic window.

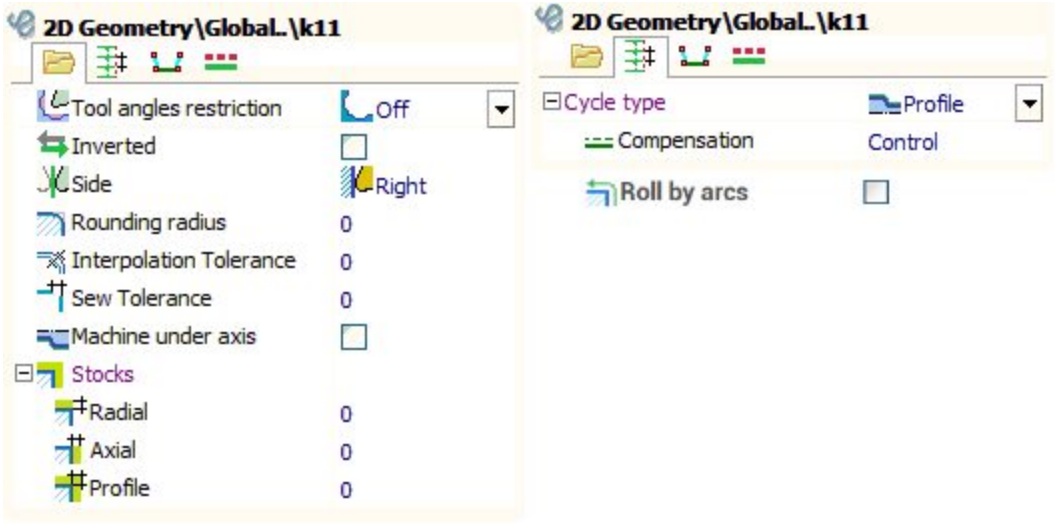
Job assignment element parameters

Basic Parameters:

After adding contour to the job assignment elements list it displays in a graphical window. Arrows defines machining direction and machining side are displayed opposite to a contour start point. It is possible to change machining direction by clicking left mouse button to blue arrow. It is possible to change machining side by clicking left mouse button on to yellow arrow.



Contour parameters window is available by clicking right mouse button to any of this arrows or by clicking elements 'Parameters' button.



Cycle type defines the way of forming toolpath based on specified contour.

Rounding radius defines arc radius, with which all outer sharp corners of the contour will be rounded. If this value is zero corners will not be rounded.

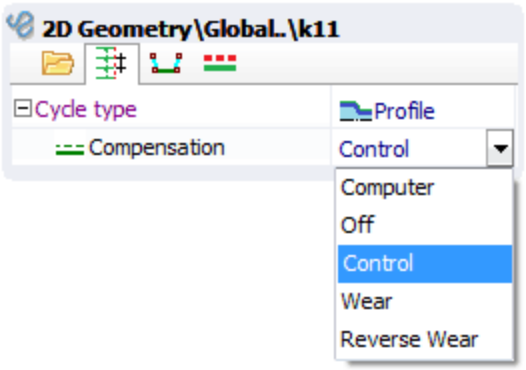
Interpolation tolerance allows to change contour by uniting set of its segments by arcs or lines. In this case, the resulting contour will be different from the original by not more than the specified value. This function is very convenient for splain curve processing.

Stocks allows to move contour at the radial, axial and equidistant direction. Stocks positive (+) direction is defined by contours machining side.

Sew tolerance is used when create the turn generatrix from the set of curves and surfaces.

Roll by arcs allows to more accurately determine the toolpath along the contour by adding arcs to the outer corners of the toolpath. This function is convenient for a tool with a large radius.

Tool radius compensation modes:



If correction is turned off (Off) NC program uses original contour without any transformations. Using this mode at complex contours can appear lines or defects, so it is necessary to be carefully when machining simulation.

If the correction mode is " Computer " then equidistant based on tool nose radius will be built to the original contour, and then it shifts to tooling point. Usually this leads to shift in the inclined segments, increasing external radiuses and decreasing internal radiuses of contour circle arcs by the tool nose radius.The starting points are also shifted by the tool nose radius.

If correction mode is "Control" NC program uses original contour without any transformations. Correction commands are included before approach and retract frames (ISO G41/42/40).

In the Wear and Reverse wear modes contours geometry transforms like in " Computer " mode. But correction commands are included before approach and retract frames too, like in the "Control" mode. Reverse wear mode differs from wear mode by correction direction.

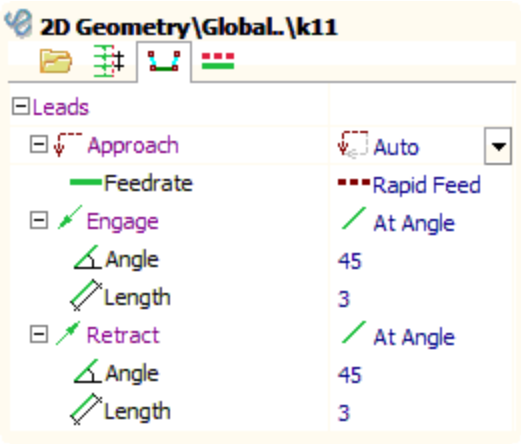
"Machine under axis" parameter allows to create toolpath below rotation axis. So, all radial coordinates becomes negative.

If "Rounding radius" parameter enabled and compensation mode is "Wear" or "Computer" then in outer corners of a contour the arcs with radius equal to tool nose radius will be formed. Sometimes it helps reduce the vibration of the machine from the abrupt change in direction of movement of the tool.

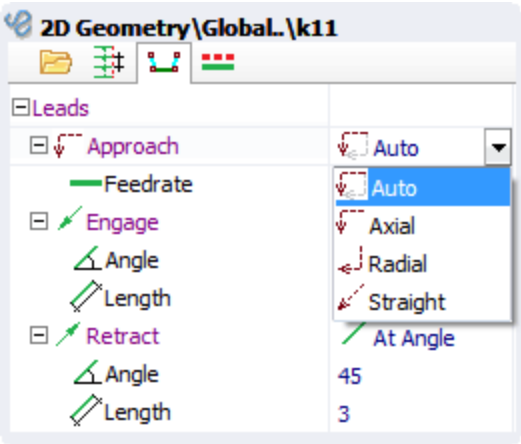
Engage, retract and transition between the elements.

Before the beginning and in the end of each contour one special element is added. These elements are called respectively the engage and retract. When tool nose radius compensation is used, engage is compensation switch on cut, retract is compensation switch off cut. Engage and retract geometrical

parameters can be set in a graphical field. Both an end point and length of a cut can be set here. Engage/retract can be processed by the line or by the arc. Click to the selected engage one more time on the screen for switching arc/cut. Engage and retract geometrical parameters also can be set in the next window.



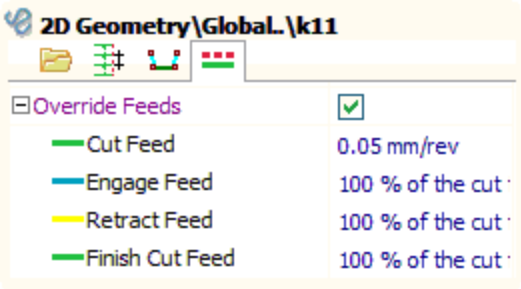
If there are several elements in operation transitions between them will be processed by rapid feed. The way of the approach to the first point of a contour from last point of the previous element is defined by an approach mode.



In axial mode approach is processed by axial direction first, and then by radial direction. In radial mode approach is processed by radial direction first, and then by axial direction. "Straight" mode generates transition by straight line. "Auto" mode chooses the radial or axial approach depending on last point of the previous contour and the side of a processed contour. Transition processing feed can be specified also.

Feed control

By default each contour machining is processed by feed specified at operation parameters. To specify feed for each element of job assignment use element parameters window.

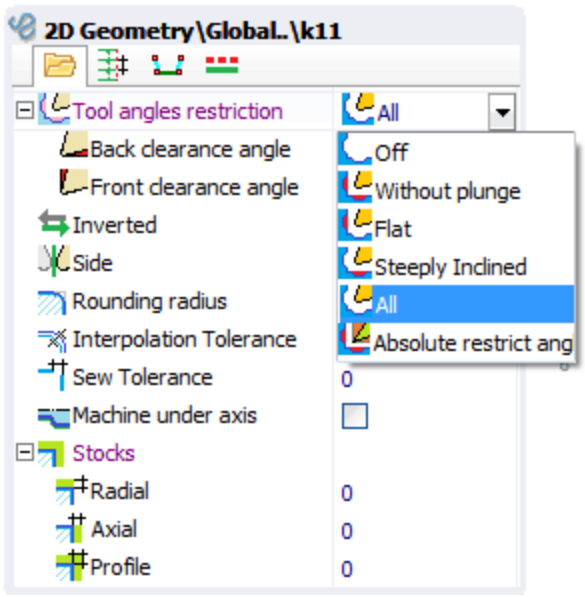


If "Override feeds" checkbox is not checked then contour feed is specified by operation parameters, else contour feed is specified in this window.

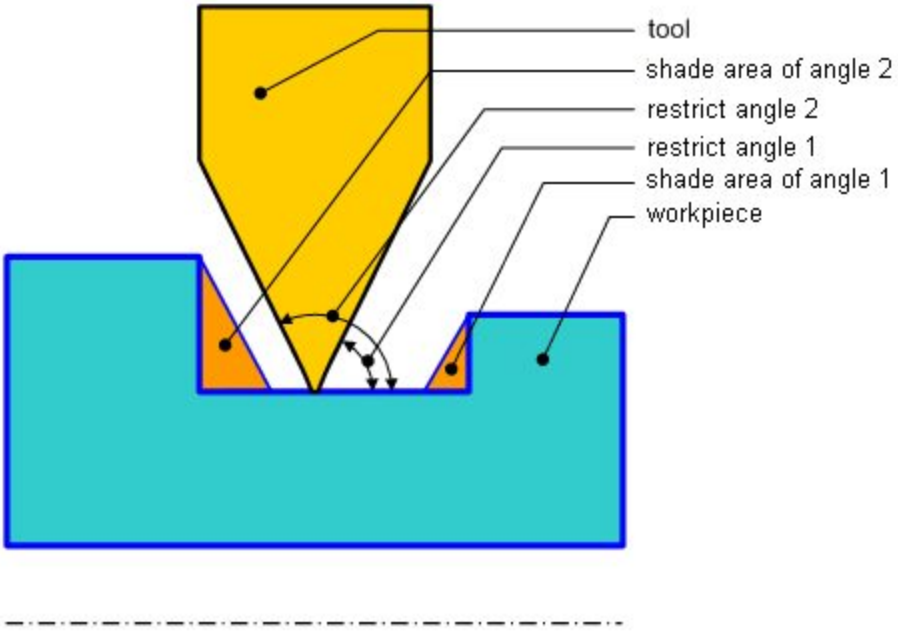
Tool angles restrictions

It is necessary to consider tool angles restrictions at grooving to prevent overlines and tool breakages. This angles can be set at parameters window. There are 6 tool angles restrictions mode.

Tool angles restrictions is turn off by default for simple contours. Contour correction by tool angles is not processed in this case. If contour have grooves, overlines becomes very possible.

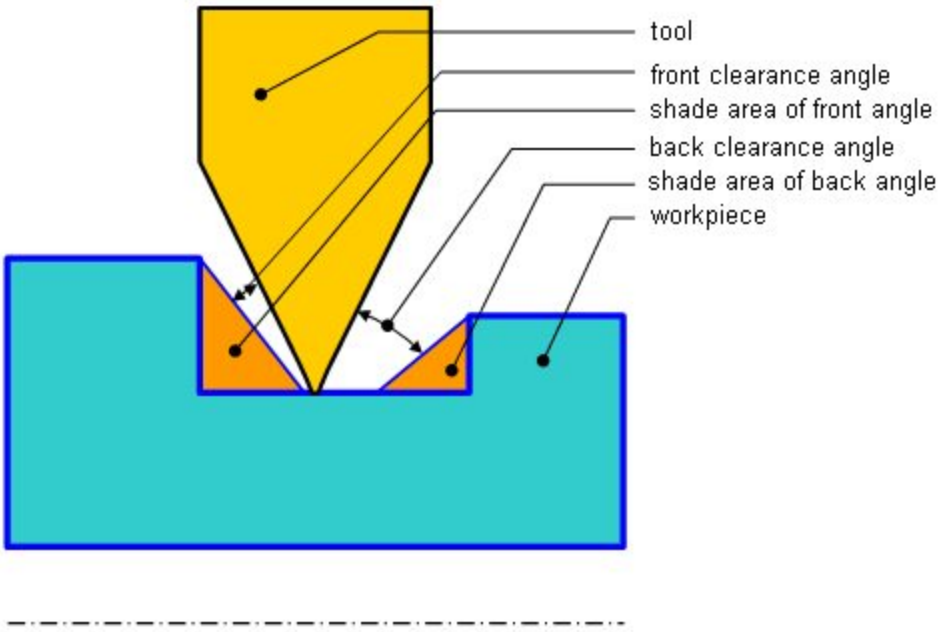


Absolute restrict angles mode allows to correct contour independently of tools real angles. These angles are measured from the horizontal line and are shown in the following figure.



"Without plunge" mode allows to prevent tool plunge to face and cylindrical grooves. When the outer groove is processed in the left spindle this mode is equal to set first angle to 0 and second angle to 90 degrees.

"All" mode allows to plunge to face and side grooves so far as tool allows. Back clearance angle and front clearance angle define absolute angles relative to tool real angles.



"Flat" mode allows tool to plunge to cylindrical grooves, but not to face grooves. When the outer groove is processed in the left spindle this mode is equal to set second absolute angle to 90 degrees and back clearance angle is used.

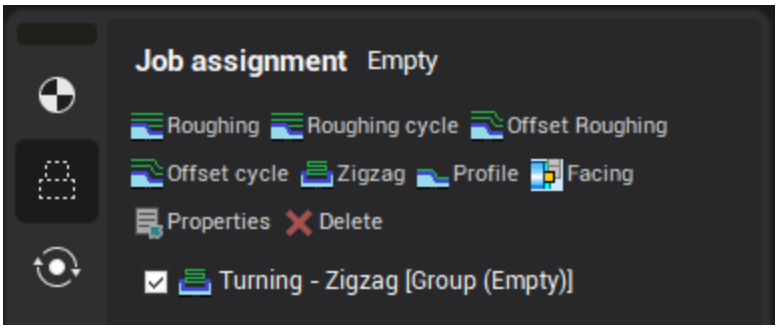
"Steeply inclined" mode allows tool to plunge to face grooves, but not to cylindrical grooves. When the outer groove is processed in the left spindle this mode is equal to set first absolute angle to 0 and front

clearance angle is used.

Contour editing modes

Activation of contour editing mode

To activate the contour edit mode, you need to switch to the job assignment, and add an element of the job assignment, if necessary.



Checkbox to the left of the elements allow to hide elements that do not require editing. If the window has at least one visible item and the job assignment visibility button is pressed , then the contour editing toolbar will appear in graphic window .

Contour elements selecting methods

Mouse click allows to select segment or contour vertex. Selected objects are marked by color. Rectangle selection (mouse down in first vertex of rectangle and mouse up in second) marks all elements entering in this rectangle. Double click to line selects all lines lies at the same straight. Double click to arc selects all arcs with same radius. Double click to chamfer selects all chamfers with same size. If "Ctrl" is pressed all elements connects selected element with previous selected element will be marked.

Contour editing operations

1. It is possible to drag vertexes.
2. It is possible to drag selected items by equidistant. It is necessary to select elements, not vertexes.
3. There are 2 ways to change arc radius. To change arc radius by the first way drag marked arc to the new position. Arc center moves so that its sibling elements don't change their position. To change arc radius by the second way mark arc by clicking them. Arc center will be shown. After that you can change sibling elements, but arc center will be constant.
4. It is possible to delete selected items by 'Del' key or button
5. It is possible to delete constructive element by button or "Shift+Del". Selected elements will be deleted, sibling elements will be connected.
6. To add chamfer select vertex and press corresponding button at the panel.

7. It is possible to undo and redo actions by clicking corresponding buttons at the panel.

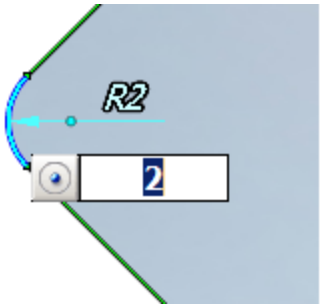
Assignment of exact dimensions

After selecting item dimension lines becomes visible. Click to dimension to set exact value. It is possible to use mouse wheel. If element has been changed, dimensions relative to elements original position are shown too.

System tries to recognize grooves automatically at selecting element. If groove is recognized its width shows. It is possible to select shifting groove side by clicking to green arrows.

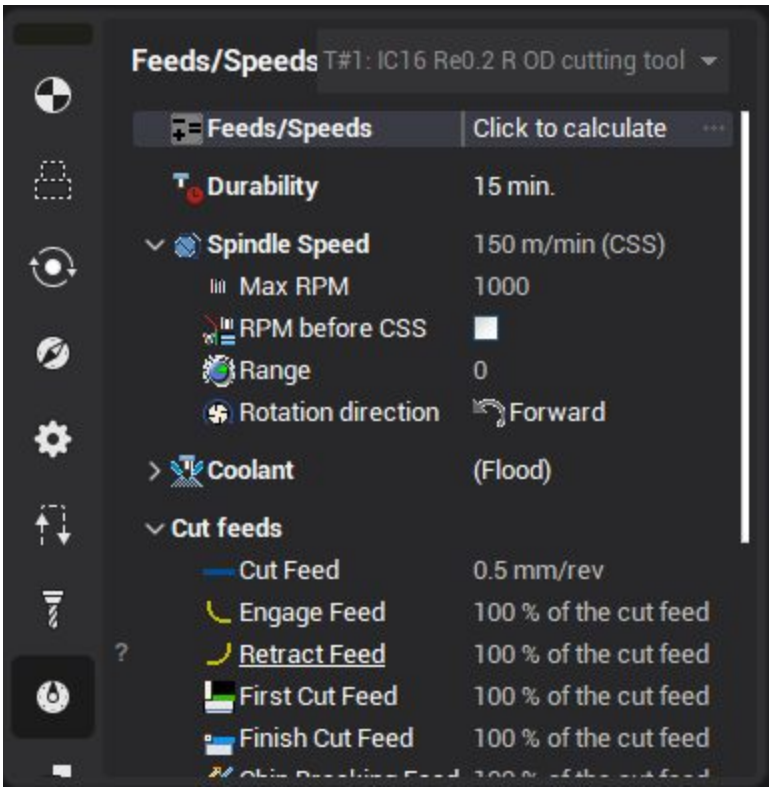


On arc radius editing it is possible to set changed object (center coordinates or sibling elements) mode by clicking button in value editing field



Cutting conditions of lathe operations

The feed rates and spindle speed of the current lathe operation are defined on the federates page of parameters window. The is shown below:



Speed

There are two ways to define the spindle rotation speed: Directly rotations per minutes RPM and with constant surface speed CSS. In the first way the rotations speed is constant. In the second way the NC control calculates the rotations speed depends on the current machining diameter. In this case the maximum spindle speed restricts the RPM. It is also possible to call calculator for speeds and feeds.

RPM before CSS

If the tool is located in the tool change position and spindle is enabled in CSS mode then spindle has slow rotations because the tool is too far from the workpiece axis. At the first rapid motion the spindle has huge acceleration because the tool goes to the axis. It is not good for the machine. To resolve this problem some machine builders recommends enable spindle in RPM mode and switch it to the CSS mode in the point with the work feed. So "RPM before CSS" enable the spindle in RPM mode on the rotations that must be equal to the RPM in the first work point. And after approach it switch to CSS mode.

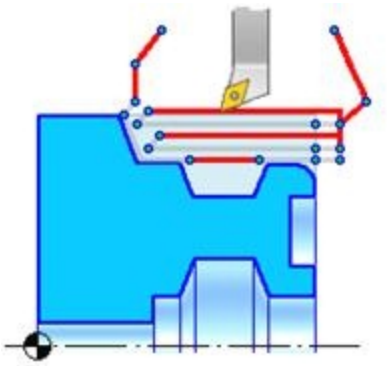
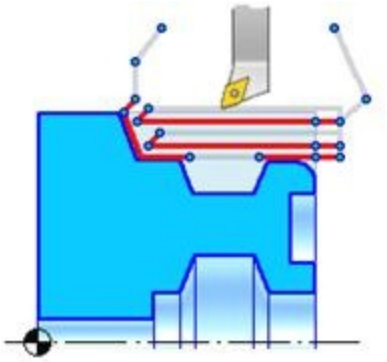
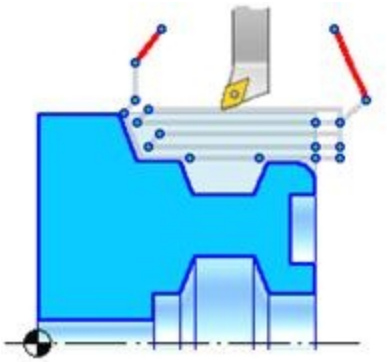
Range

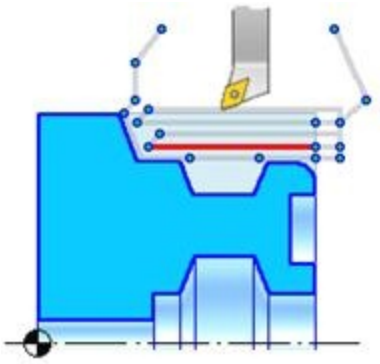
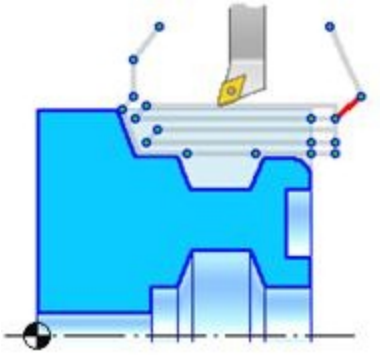
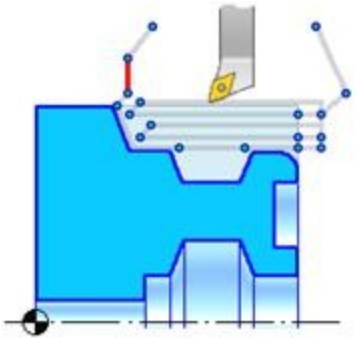
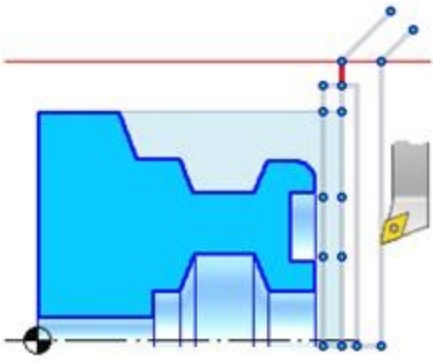
Range is the speeds range specific to the machine (with mechanical gear box).

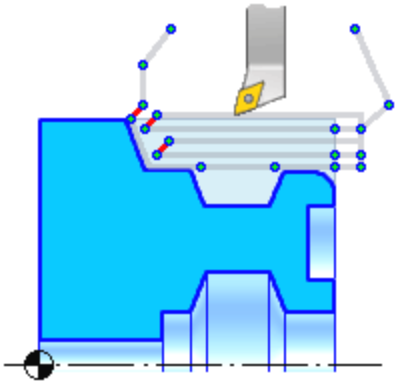
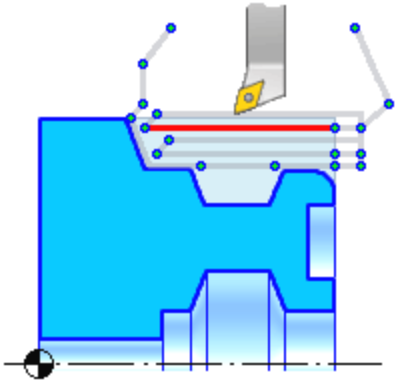
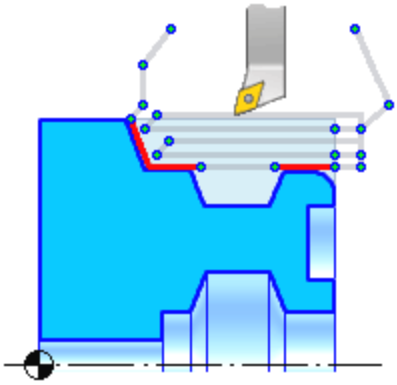
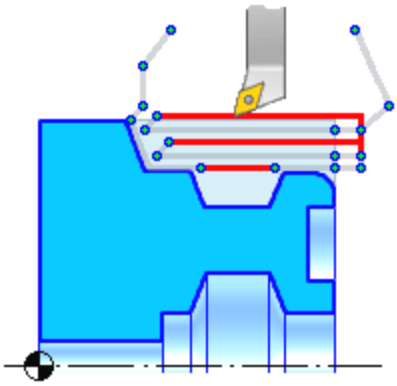
Feed

It is possible to set the feed for the different tool path segments. Depends on the operation type the tool path can contains the different segments, so the count of the feeds is changed.

All feed types are listed in the table below:

	All auxiliary feed All feeds where the shaving is not performed. Used to move the tool in work path start.
	All work feeds All feeds where shaving is performed.
	Rapid feed Used when moving throw the safe planes. The tool path segments that have the rapid feed are shown as red. To switch to the rapid feed the RAPID command of CLData is used.

	Cut feed Defines the feed of the work motion. It is the main feed. All other feeds can be defined as a percent of the cut feed.
	Approach feed Defines the feed for the tool engage in the cutting zone to the start point of the first approach from the Start point.
	Return feed Defines the feed for the tool return from the last point of the last path to the Finish point.
	Engage feed Defines the feed that is used in the engage to the first point of the work path. It is equal to the work feed as default.

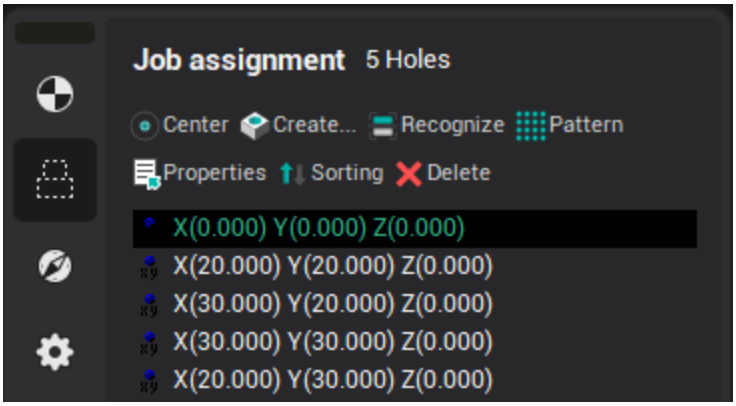
	Retract feed Defines the feed that is used on the retraction from the surface after the work stroke. It is equal to the work feed as default.
	First cut feed Defines the feed that is used to remove the first layer from the workpiece. If the workpiece has the crust then the first feed must be less than the cut feed.
	Finish cut feed In the rough operations defines the feed of the motion along the machining surface to get the required roughness.
	Traversal feed Defines the feed that is used to make the motion to the next work pass.

The feeds values can be defined in the defined in:

- `mm/min` – defines the feed value as a motion length per minute
- `mm/rev` – defines the feed value as the tool motion length per one spindle rotation;
- `in % of the cut feed` – defines the feed in the percent of the cut feed value. For instance if the value is 100% then the feed is equal to the cut feed. If the value is 50% then the feed is equal to the half of the cut feed.

Note: If the cut feed is changed than the all feeds defined as a percent of the cut feed will be modified.

Job assignment for hole machining operation



- Center — Create hole by center point
- Create... — Create hole by coordinates input
- Recognize — Automatically recognize holes in the part
- Pattern — Create holes array by pattern
- Properties — Properties of the selected items
- Delete — Delete selected items

When defining the parameters for the hole machining operation it is possible to define the data for holes to be drilled. In the hole machining operation, the holes list defines the number, sequence and parameters of the holes to be machined. The order can be altered by mouse dragging.

Each hole is defined by the coordinates for it's center, the diameter and also the value of the upper and bottom levels. There are two methods to define the center coordinates of holes: by coordinates or by a geometrical "point" object.

Regardless of the center definition method used, the depth of the hole is defined directly on the Model page. The holes specified by coordinates are marked with the sign while the holes defined by center point are marked with the icon. To define the top and the bottom levels, it is necessary to select the desired points from the list on the right and enter the Zmax and Zmin values.

Hole Machining operation supports two ways to specify drilling direction for each hole center. Use the Inverse normal field of the Hole editing window or specify the normal in the graphical window.

- Zmax – defines the Z coordinate of the top of the hole. The coordinate can be defined either directly or calculated automatically. When calculating automatically, transition to work feed is performed using the safe distance from the workpiece.
- Zmin – defines the Z coordinate of the bottom of the hole. The coordinate can be defined either directly or calculated automatically. When calculating automatically, the coordinate is taken from the model being machined.

In addition to the ZMin parameter operation can have the Drill tip compensation parameter specified. This parameter can be one of the following:

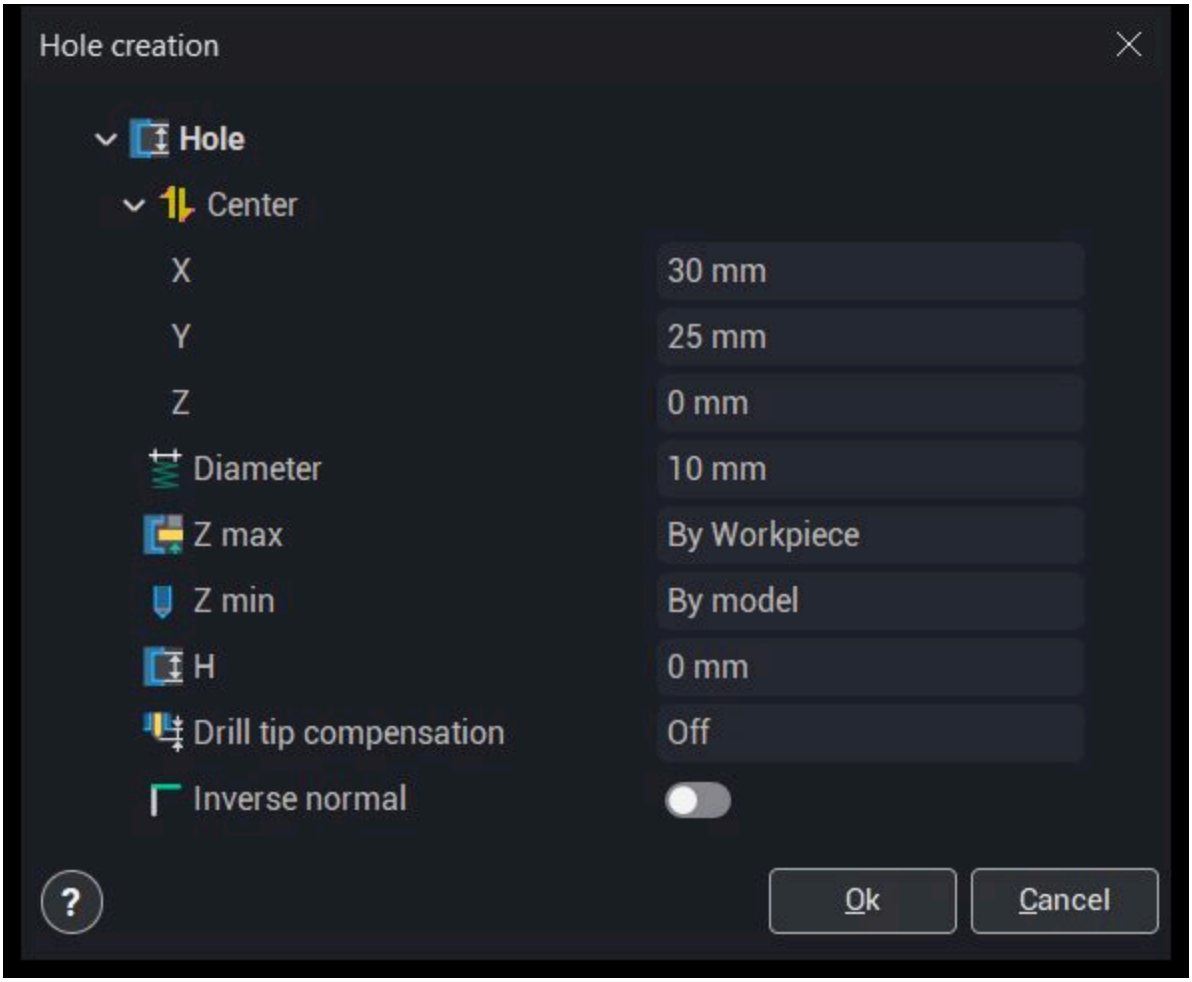
- Off – the drill tip descends to the ZMin level.
- Drill tip – drill descends below the ZMin level to the value of the tapered part of the drill, thus providing cylindrical drilling area to the ZMin depth.
- Length – drill top descends below the ZMin level to the specified value.

Select holes in the holes list and use the context menu Export selected in DXF item to export the list into the DXF-file.

To sort holes with different parameter values use the Sorting.

Defining holes by coordinates

The window that opens is designed for parameter assignment of new or existing holes. The Create command opens the Hole creation window. To edit the parameters of an existing hole, double click the required hole in the list or use the Properties command: the Hole editing window opens. Details



To create a hole the user should define the values of its parameters (position, diameter, depth) and press the Ok button. The created hole will be automatically added to the list.

When hole's parameters are being altered, the changes are displayed in the graphic window.

Center. Coordinates X, Y and Z of the hole center.

Diameter. Diameter of the hole.

Z max - top level mode:

- By workpiece - the top level is defined as the highest coordinate of the workpiece edge along the tool travel path;
- By feature -the top level is defined as the highest coordinate of the geometric element selected as the Job Assignment along the tool travel path;

- Default – the default mode is used; parentheses indicate this mode. See the default settings below;
- Manual – user set value manually.

Z min - bottom level mode:

- By feature -the bottom level is defined as the most remote coordinate of the geometric element selected as the Job Assignment along the tool travel path;
- By model - the bottom level is determined by the most remote coordinate of the hole position on the part in the tool travel direction;
- Default – the default mode is used; parentheses indicate this mode. See the default settings below;
- Manual – user set value manually.

H. Depth of the hole: the distance between Z max and Z min. It is set as a length or in % of the hole diameter (% D).

Drill tip compensation - choose the way the hole depth is specified:

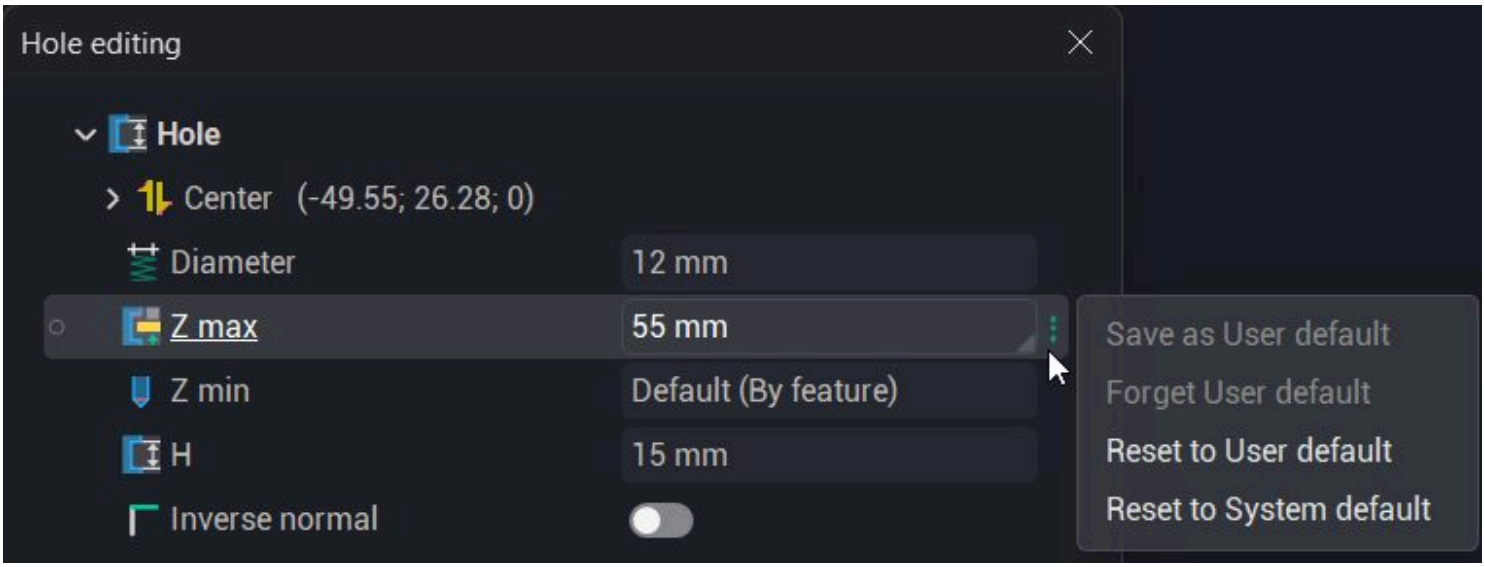
- Off – last tool path point matches the drill tooling point;
- Drill tip – last tool path point matches the drill tip point;
- Length – same as Off but the drill travels the specified value down from the drill tooling point.

Inverse normal. Reverses the drilling direction of the hole. The direction of an existing hole is changed in the Hole editing window.

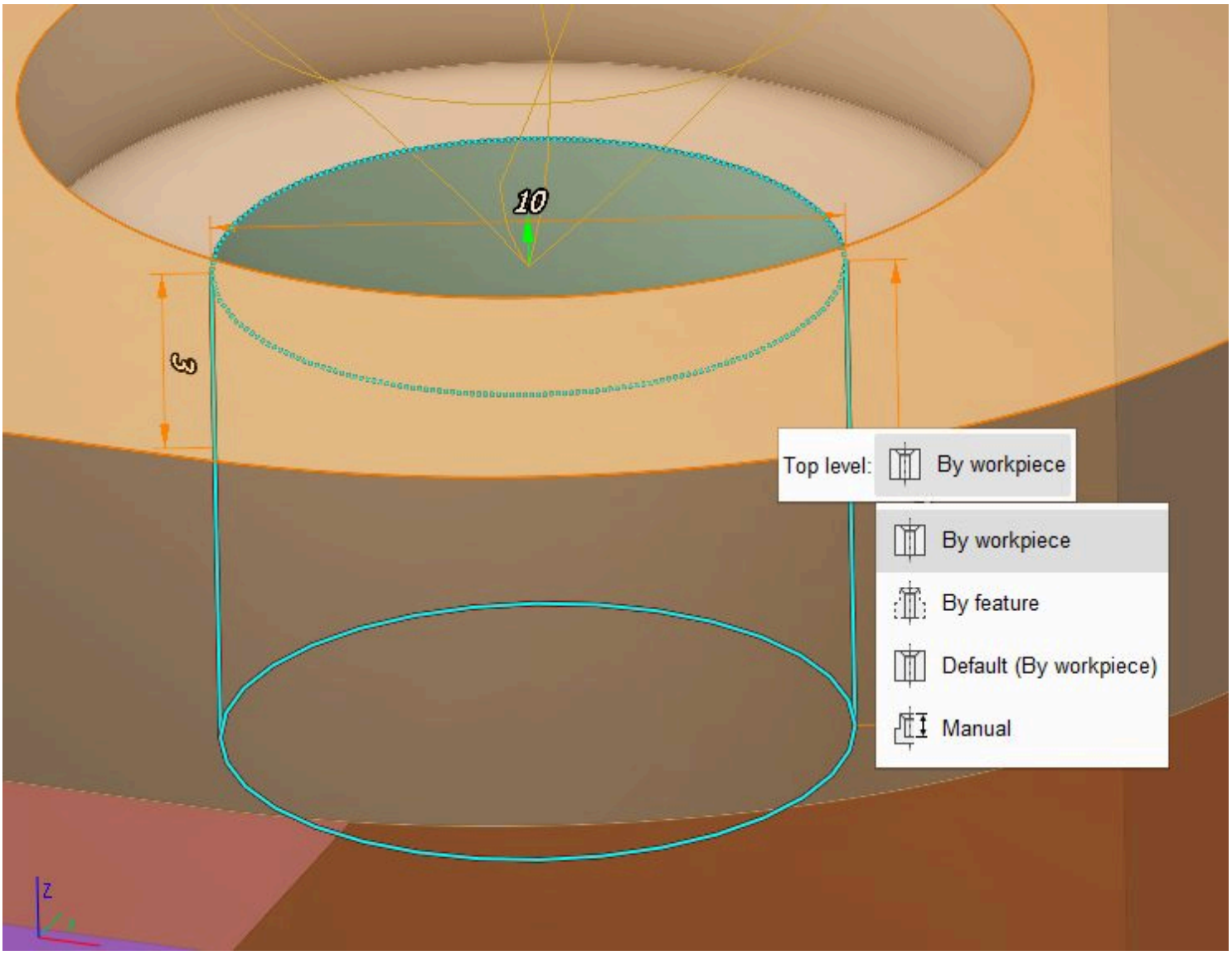
To change the default modes of Z max and Z min, hover over the Z max or Z min line and click the ... button on the right. The menu contains the following commands:

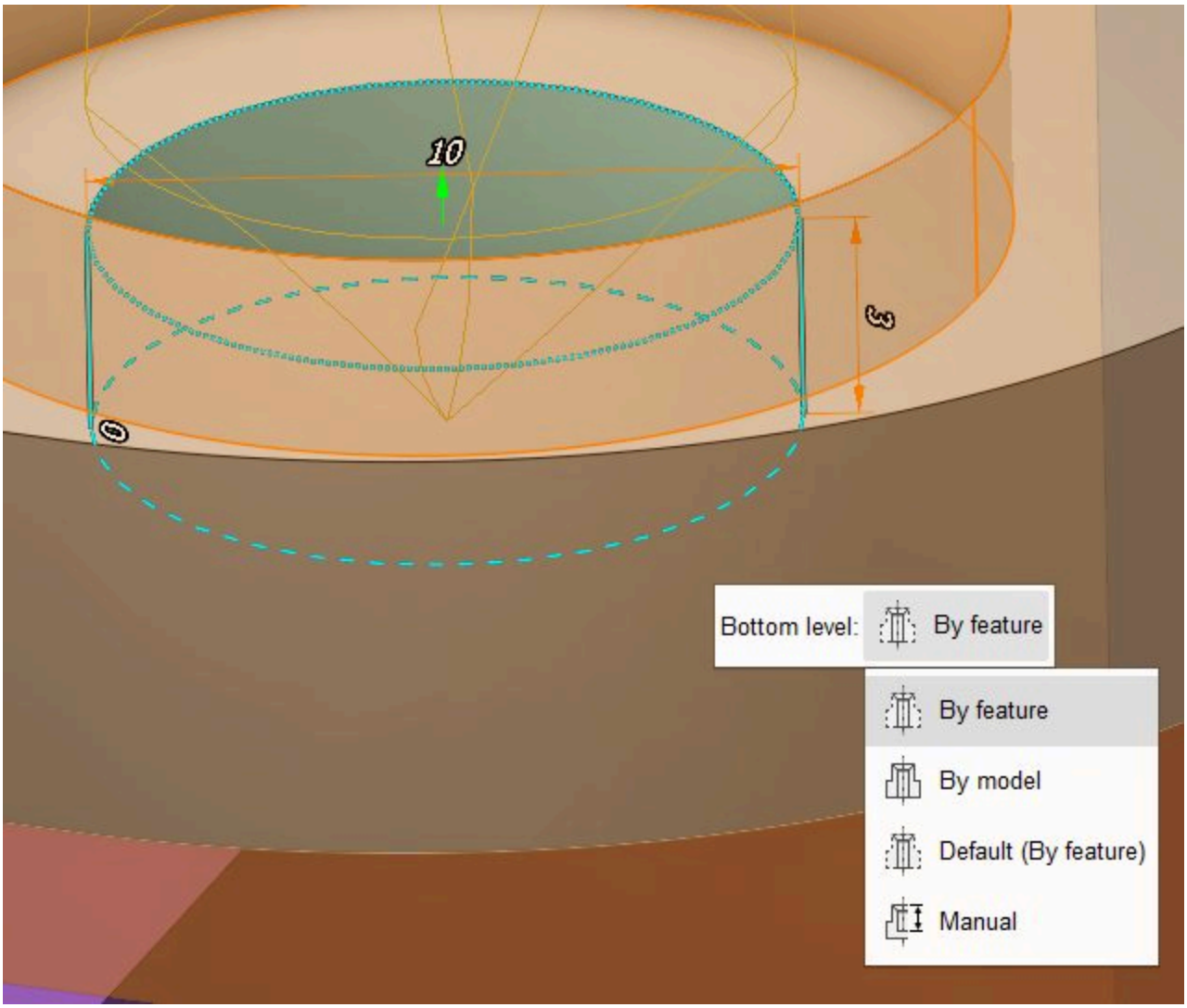
- Save as User default – the current mode of the level becomes the default mode. Available when the level uses a mode that can be the default: By Workpiece or By feature for Z max, By feature or By model for Z min;
- Forget User default – the default mode returns to the system default;
- Reset to User default – sets the Default mode for the level;
- Reset to System default – sets the system default mode for the level.

Commands that do not change anything are unavailable. These settings will be applied for the whole system, not just for a current project.



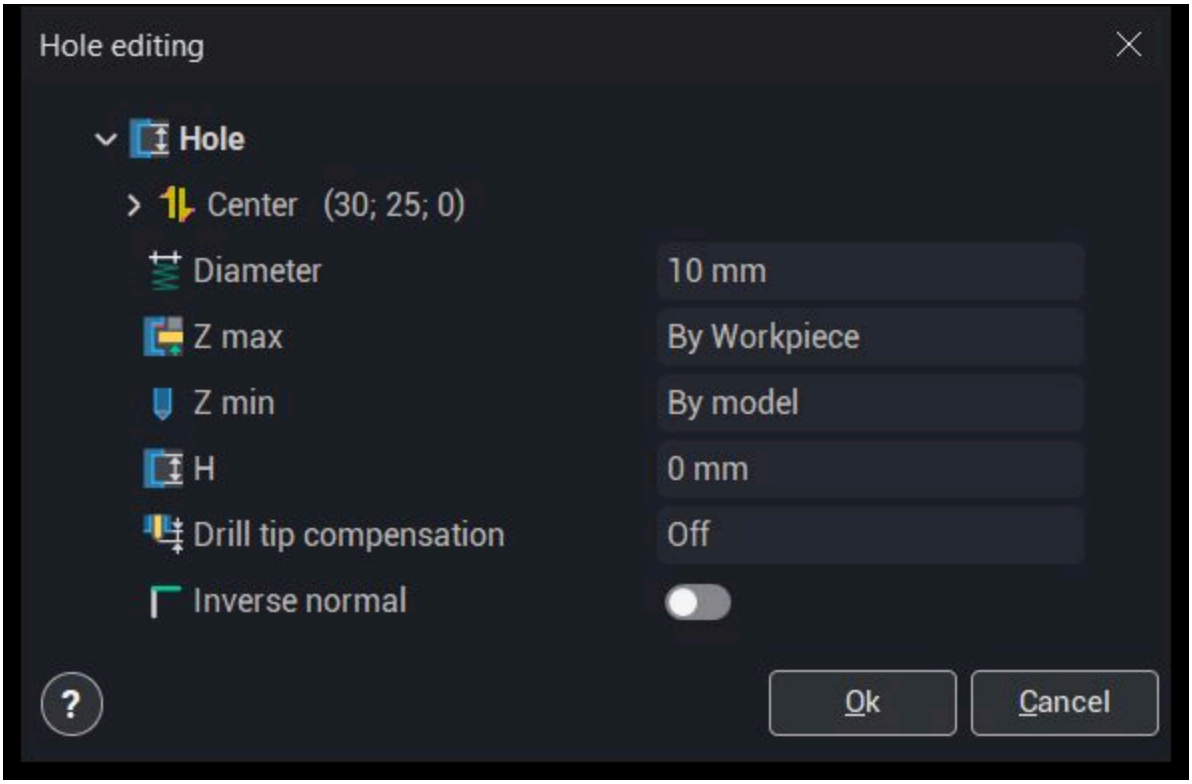
Also you can set Z max and Z min mode in graphics window. Click on top or bottom level and you see action menu. Each mode is displayed differently:


10
6
Top level:
By workpiece
By workpiece
By feature
Default (By workpiece)
Manual
Z


10
3
0
Bottom level:
By feature
By feature
By model
Default (By feature)
Manual

Defining holes by using a geometrical point object

The window that opens is designed for parameter assignment of new or existing holes. To edit the parameters of an existing hole, double click the required hole in the list or use the Properties command: the Hole editing window opens.

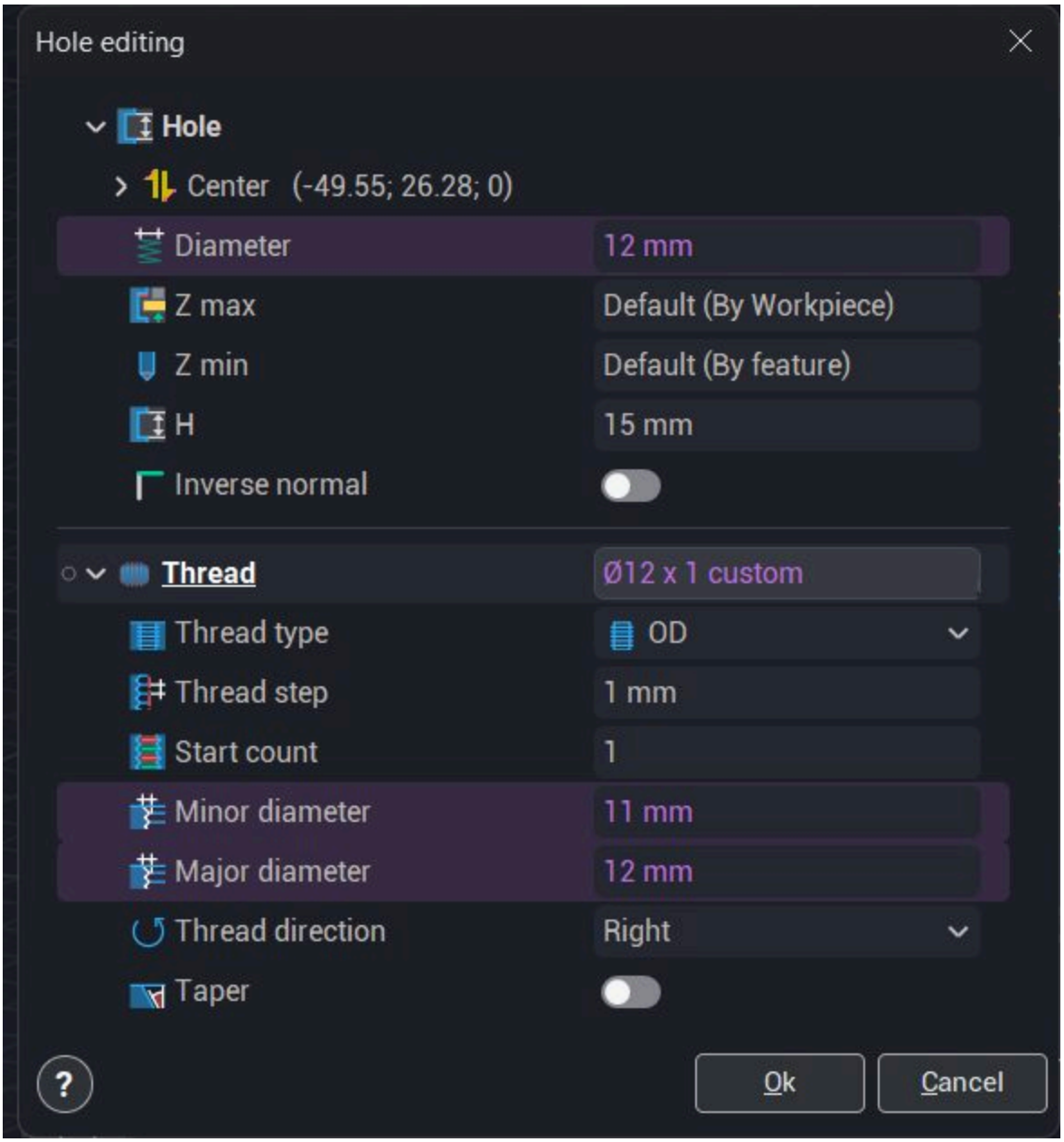


To create a hole the user should define the values of its parameters (position, diameter, depth) and press the Ok button. The created hole will be automatically added to the list.

When hole's parameters are being altered, the changes are displayed in the graphic window.

The Hole editing window contains the same parameters as the Hole creation window. Details The coordinates of the hole center are shown in one line as *(X; Y; Z)*. Expand the line to edit them.

To edit several holes at once, select them in the list and use the Properties command. If the selected holes have different values of a parameter, the parameter is highlighted in purple. A value entered in the window is applied to all selected holes.

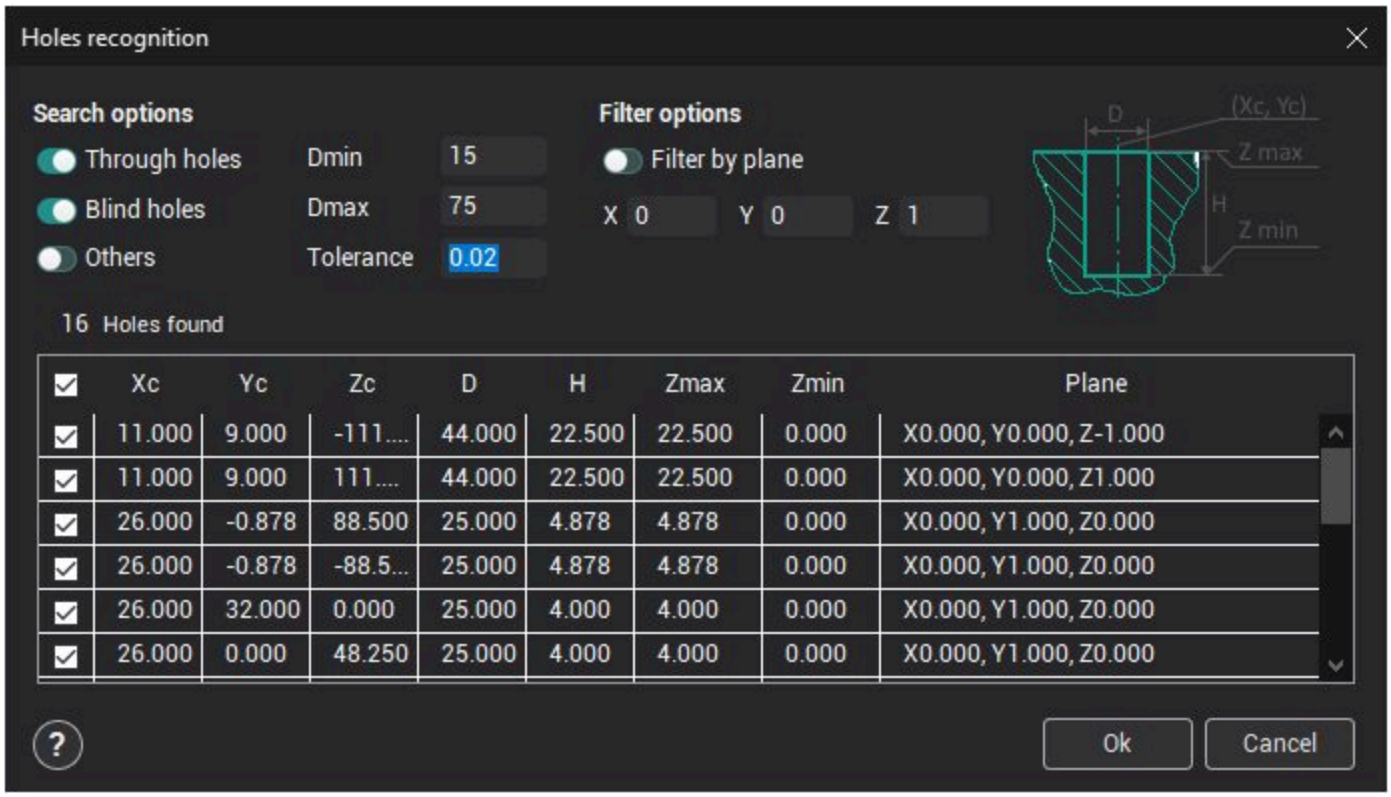

Hole editing
Hole
Center (-49.55; 26.28; 0)
Diameter
12 mm
Z max
Default (By Workpiece)
Z min
Default (By feature)
H
15 mm
Inverse normal
Thread
Ø12 x 1 custom
Thread type
OD
Thread step
1 mm
Start count
1
Minor diameter
11 mm
Major diameter
12 mm
Thread direction
Right
Taper
?
Ok
Cancel

Automatic hole recognition

The holes are found in the part. When a hole is found, it will be automatically added to the holes list.

Hole recognition is performed according to the selected search options. Only those holes that lie within the defined range will be added to the list. All holes are divided into three types:

- Through – holes, which go through the model, or with a bottom level that is lower than the bottom machining level of the operation.
- Blind – holes, the end of which lie in the model between the top and the bottom levels for the operation.
- Others – holes, for which only the center coordinates, can be defined but not the diameter and/or the depth of the hole. Such holes might have a variable diameter e.g. with facets, or just be curves.

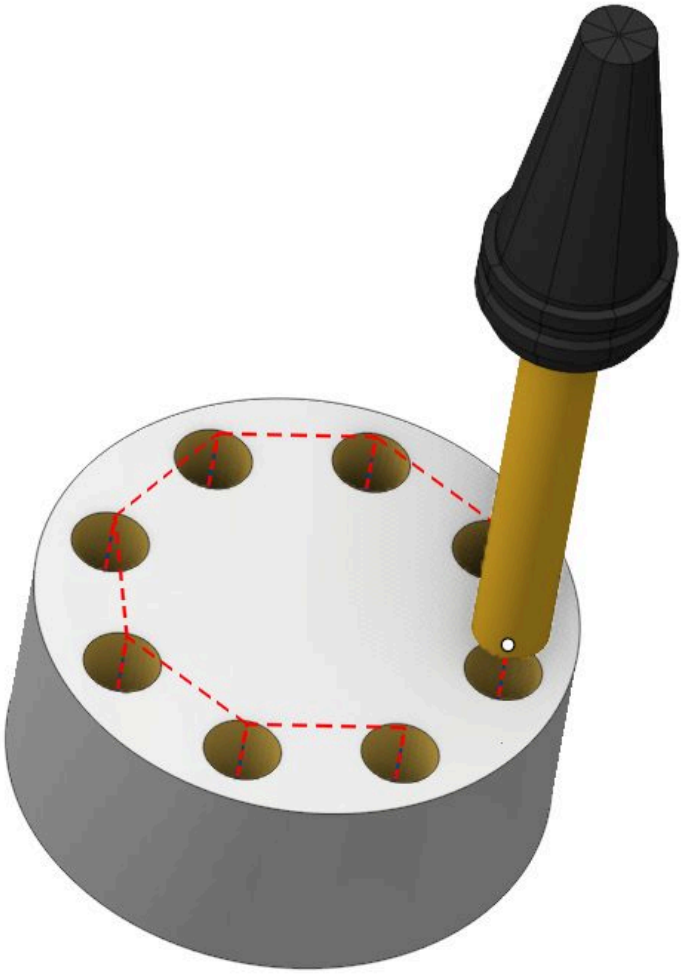


Parameters for the holes found in a search operation can be edited. When the parameters for a hole are being edited, the hole is highlighted in the graphic area. The parameters for holes can be altered by left clicking on it in the search window and typing the new values.

When the Ok button is pressed, all holes selected with a tick will be added to the holes list. Left clicking on the heading of the first column will activate or deactivate all holes.

FNOT PRINT>

Creating hole pattern



Application area:

In the CAM system, it is possible to set various types of hole templates. It is not necessary for a template to have a geometric construction of the hole.

Using certain rules, you can define the following templates:

- Linear
- Circular
- Angular
- Concentric

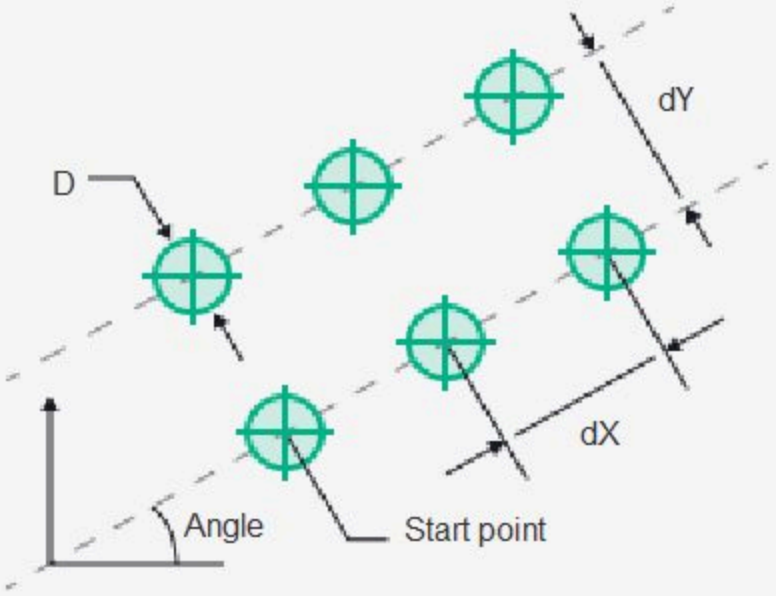
- Parallelogram

The system uses five types of pattern: Linear, Circular, Angular, Concentric and Parallelogram.

- On the Linear page user can create linear holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

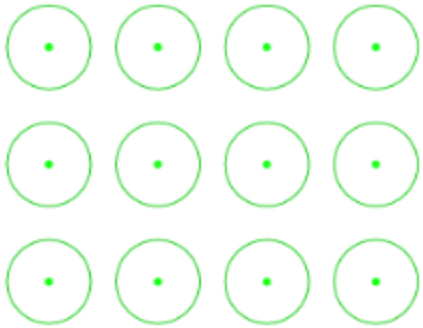


Columns	2	D	20
Rows	2	dY	10
Angle	0	dX	10

Start point

X	0	Zmin	Auto		
Y	0	Zmax	Auto	H	0

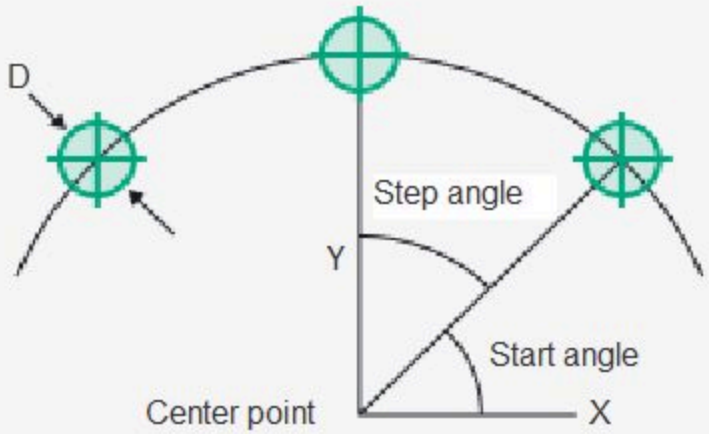
Ok | Cancel | Apply



- On the Circular page user can create circular holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

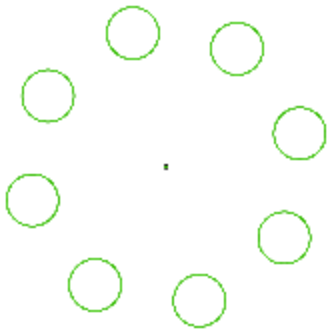


Radius	53	D	20
Number	8	Start angle	14
Step angle	45		

Start point

X	-14	Zmin	0		
Y	0	Zmax	0	H	0

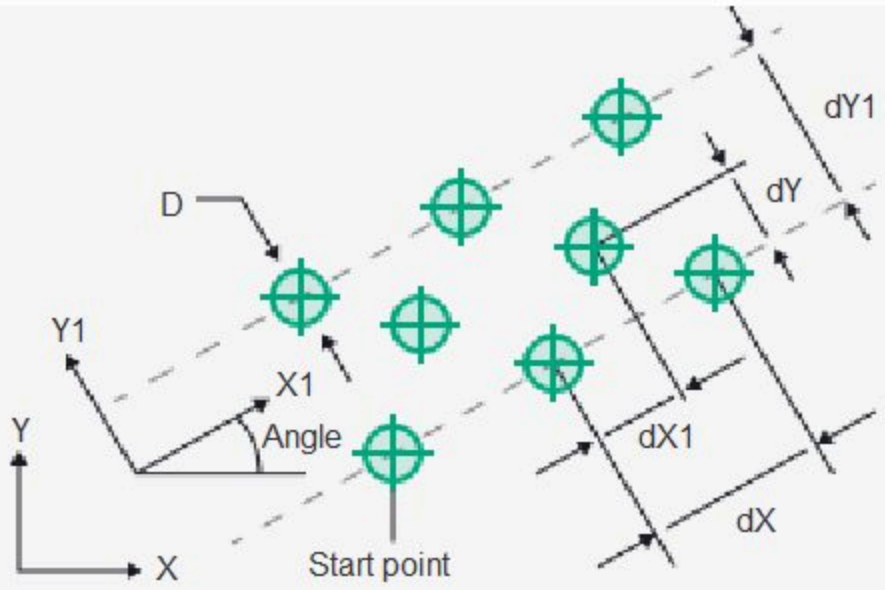
Ok | Cancel | Apply



- On the Angular page user can create linear holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

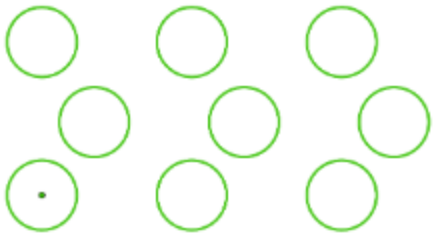


Columns	3	dX	20
Rows	3	dY	20
Angle	0	dX1	10
D	20	dY1	10

Start point

X	-14	Zmin	0		
Y	0	Zmax	0	H	0

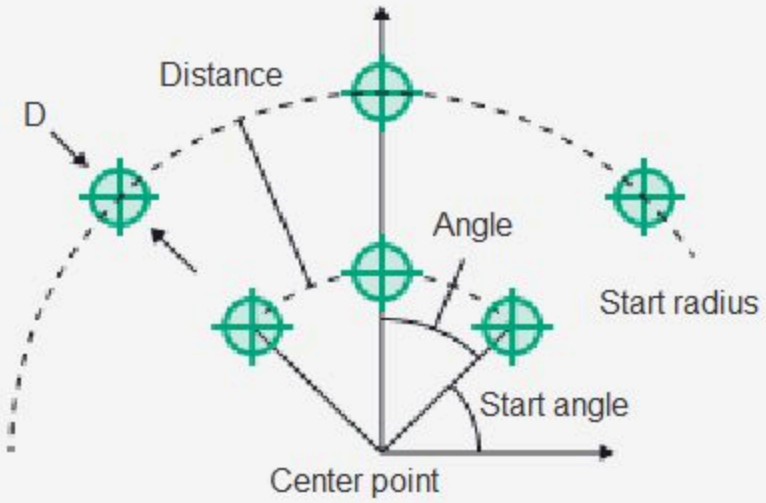
Ok | Cancel | Apply



- On the Concentric page user can create concentric holes pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram

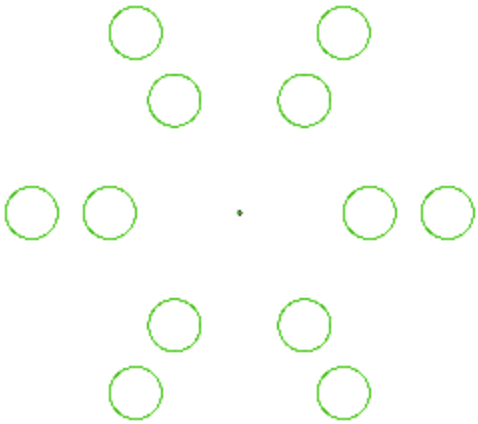


Start radius	50	D	20
Circles num.	2	Start angle	0
Holes num.	6	Distance	30
Angle	60		

Start point

X	-14	Zmin	0		
Y	0	Zmax	0	H	0

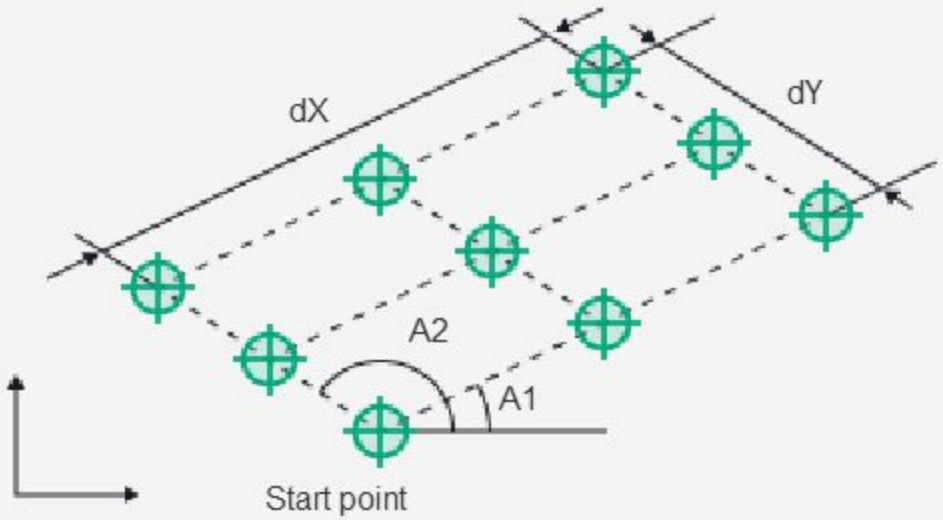
Ok | Cancel | Apply



- On the Parallelogram page user can create parallelogram pattern:

Holes pattern

Linear | Circular | Angular | Concentric | Parallelogram



Columns	5	D	20
Rows	3	Steps	Absolute
Angle1	2	dX	153
Angle2	130	dY	100

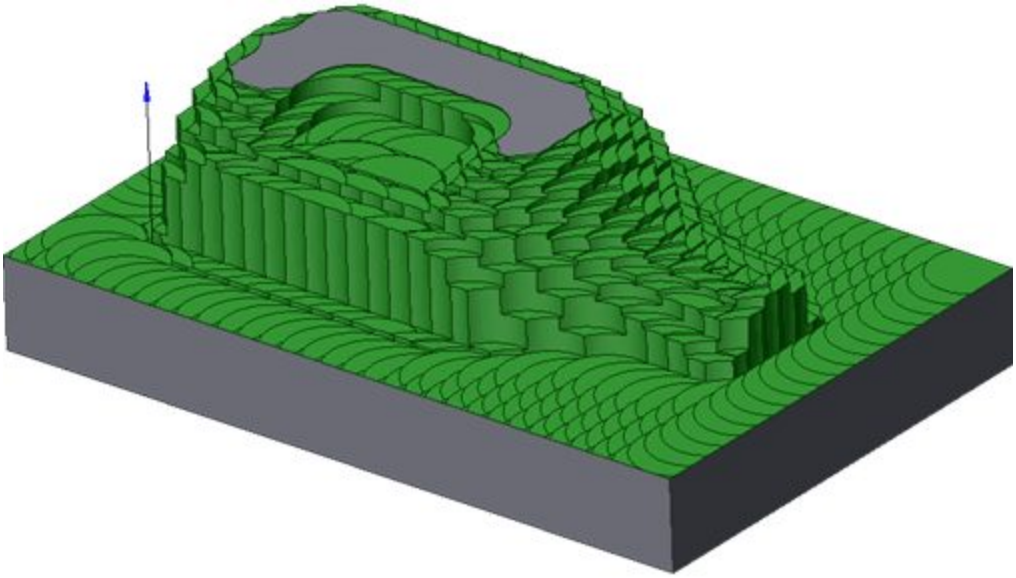
Start point

X	-14		Zmin	0	
Y	0		Zmax	0	H 0

Ok | Cancel | Apply



Using the hole patterns together with automatic determination of hole levels allows someone perform roughing machining of the part by the axial plunging strategy.



Mill-turn Machining

Algorithm of preparing NC program for lathe-milling processing center is similar for other types machines, but it has some features. This features will be described in current part.

1. First of all, a machine is chosen, on which a treatment will be processing. CAM system can program several types of lathe-milling processing center, which have fundamental differences in structure, including Swiss lathes machines.
2. If a machine is supply with a machining turret for it a setup tooling is forming.
3. After this describe a processing of part, workpiece and fixtures and its fixing mode. You can define several parts in one project, see Multi part projects for more.
4. Next the point of tool interchange is determined.
5. After that different operations may be created, both turning and milling so long as the billet will be manufactured. For getting an objective view in simulation mode, while setting a cutting tool it is necessary to set holder and overhang.
6. Some turning machines do not support standard cycle of processing holes during work with the driving tool. In this case it is necessary to use Hole machining with the expanded style of toolpath. This operation may generate standard cycles in expanded state.
7. If machine had not equipment with Y axis, a polar interpolation may be used for face plane milling.
8. For milling on cylinder surface by radial tools a cylinder interpolation may be used.
9. If detail has repeating elements, this way advisability to use that possibilities of the system as multiplication around an axis.
10. After calculation of every operation a trajectory is checking for a correctness in simulation mode.
11. Before finally generation of NC program it is necessarily to check operations parameters in summary table.
12. Check an accuracy in setting numbers of tools. System doesn't control if in different operations various tools set on the same numbers.
13. Necessarily to check turning tools point in all operations. With wrong turning point simulation works correctly, but NC-program is generate with serious slip which can bring to tool fracture or even machine breakage.
14. Switch to control condition of cutting mode, check direction of spindle rotation, heat-removing and correctness of feeding values.

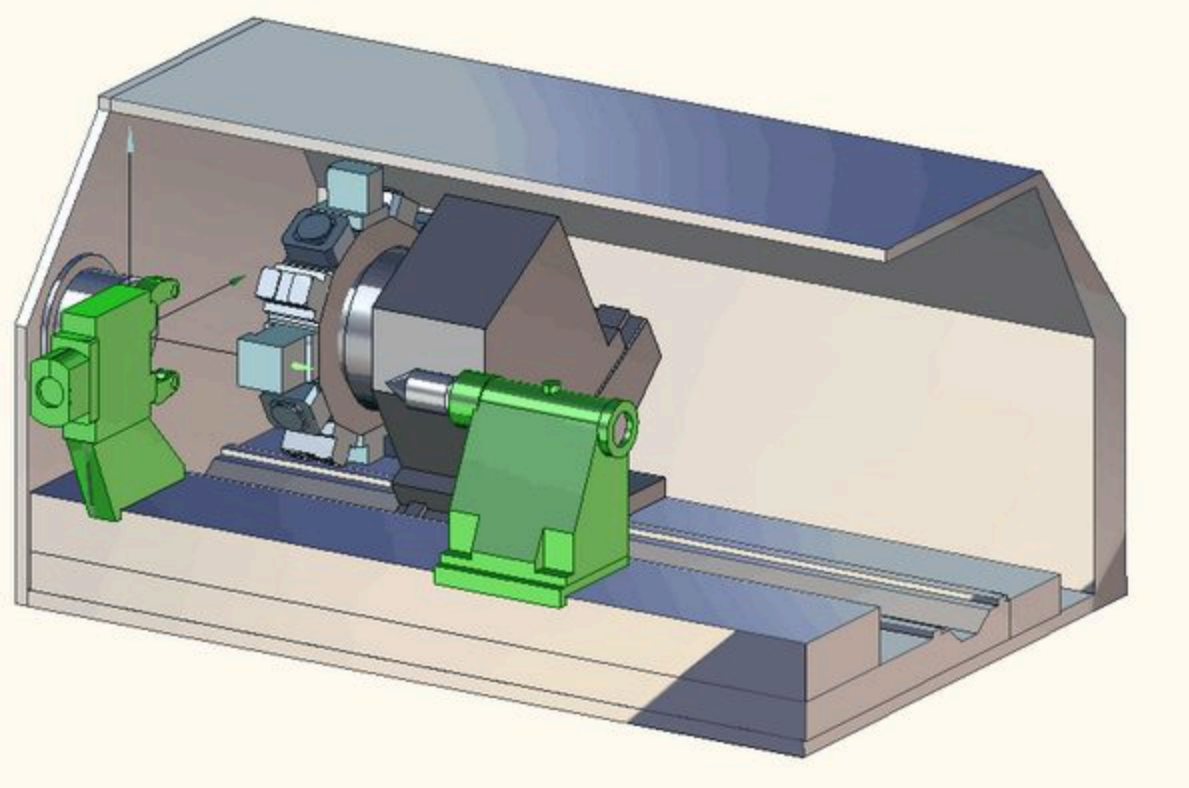
15. After any settings changing and recalculating make sure in absence of exclamation marks.

16. Generate a NC-program.

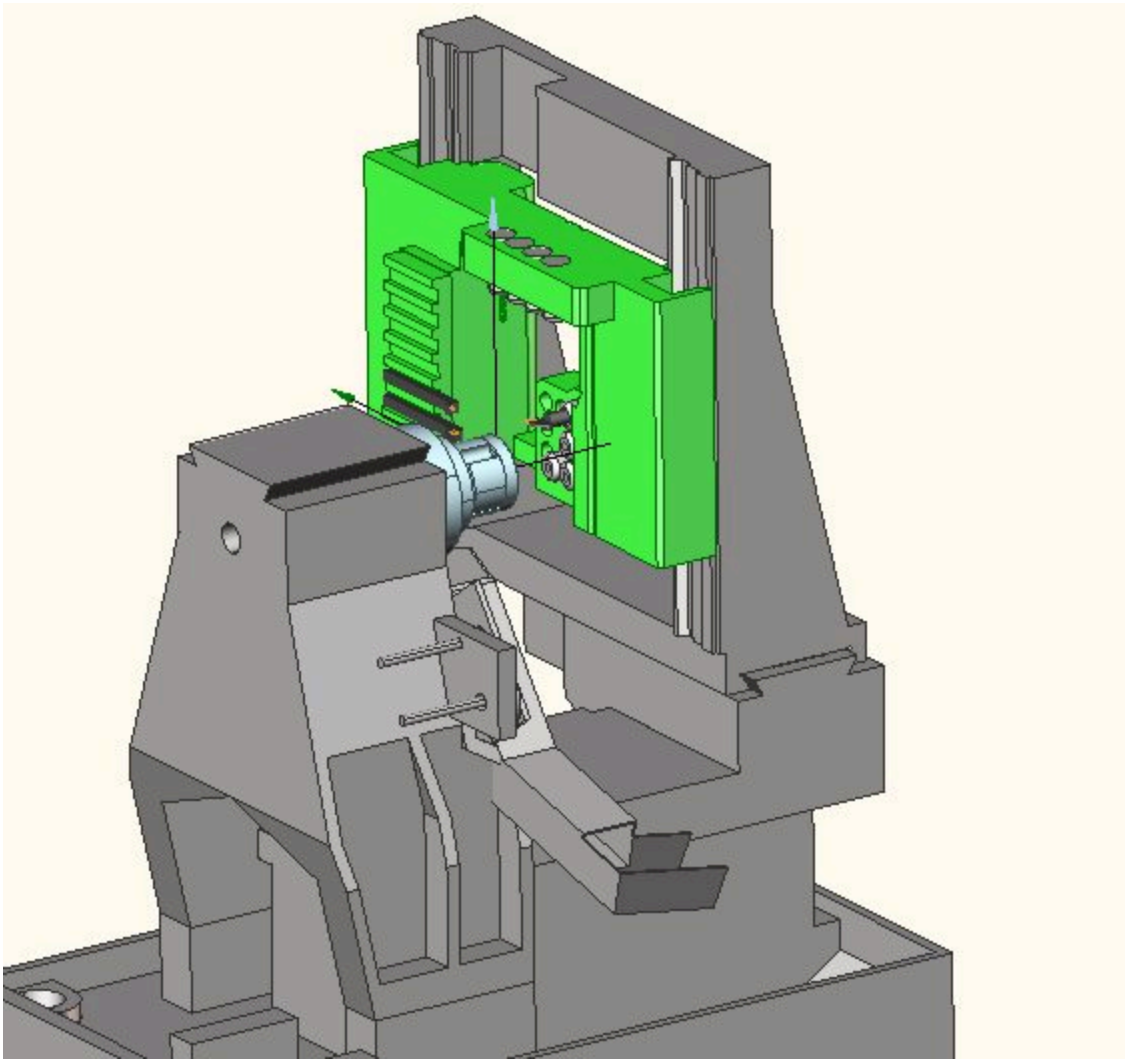
Lathe-milling machines types

CAM system make possible to generate NC-programs for turret lathe processing centers, turning machines as "Swiss-type" and many others.

Mori Seiki NL 2500 machine's scheme:



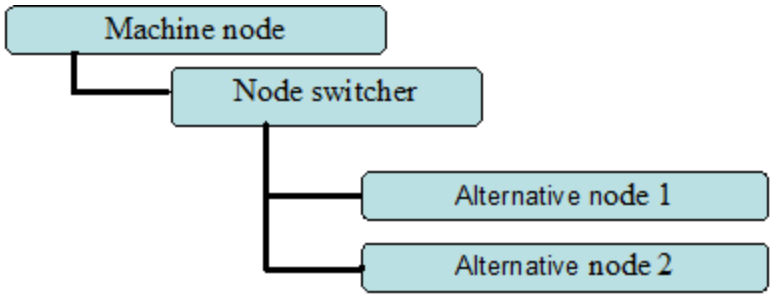
"Swiss-Type" Po Ly gim mini-88Y machine's scheme:



Using of machines kinematics allows to realize a visual inspection of collision all executing agency such as poppethead, collar plate, tool turrent and a tool prominent from it.

Setting-up tooling

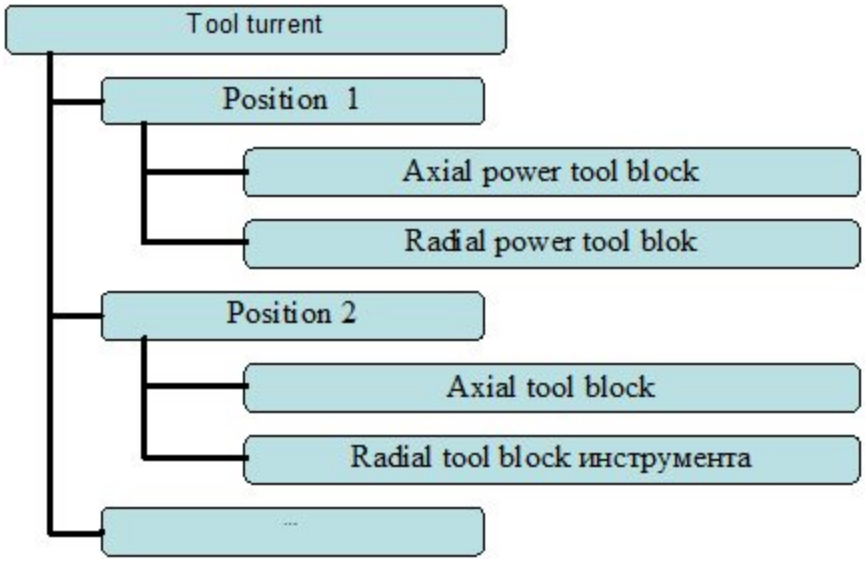
The kinematics of many machines may be appreciably changed by its changeover. For a possibility of quick and controlled machine changing in kinematics provided special node-switcher.



Node switcher at kinematic scheme of machine

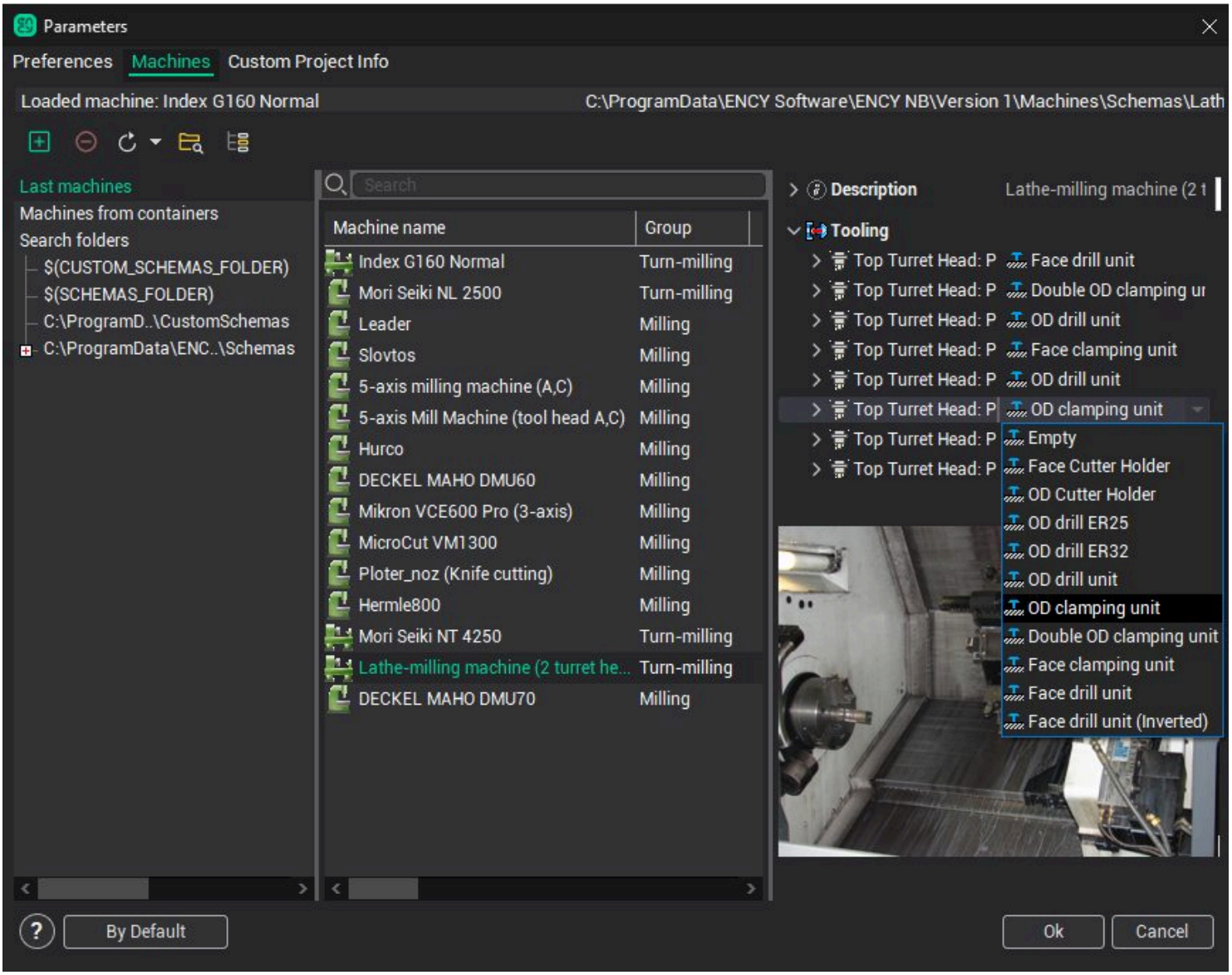
Node – switcher make possible to join to parent node of machine one from alternatives nodes. If in scheme presents a node-switcher, it shows in tree like drop-down list, from which one of alternatives node may be chosen.

For example, on turning-revolver center in tool turrent position one from several types of blocks may be attaching. Blocks make available to set axial or radial immovable (turning) tool, and axial or radial driving tool. Moreover block may have different dimension-type.



Sample of description for tool turrent

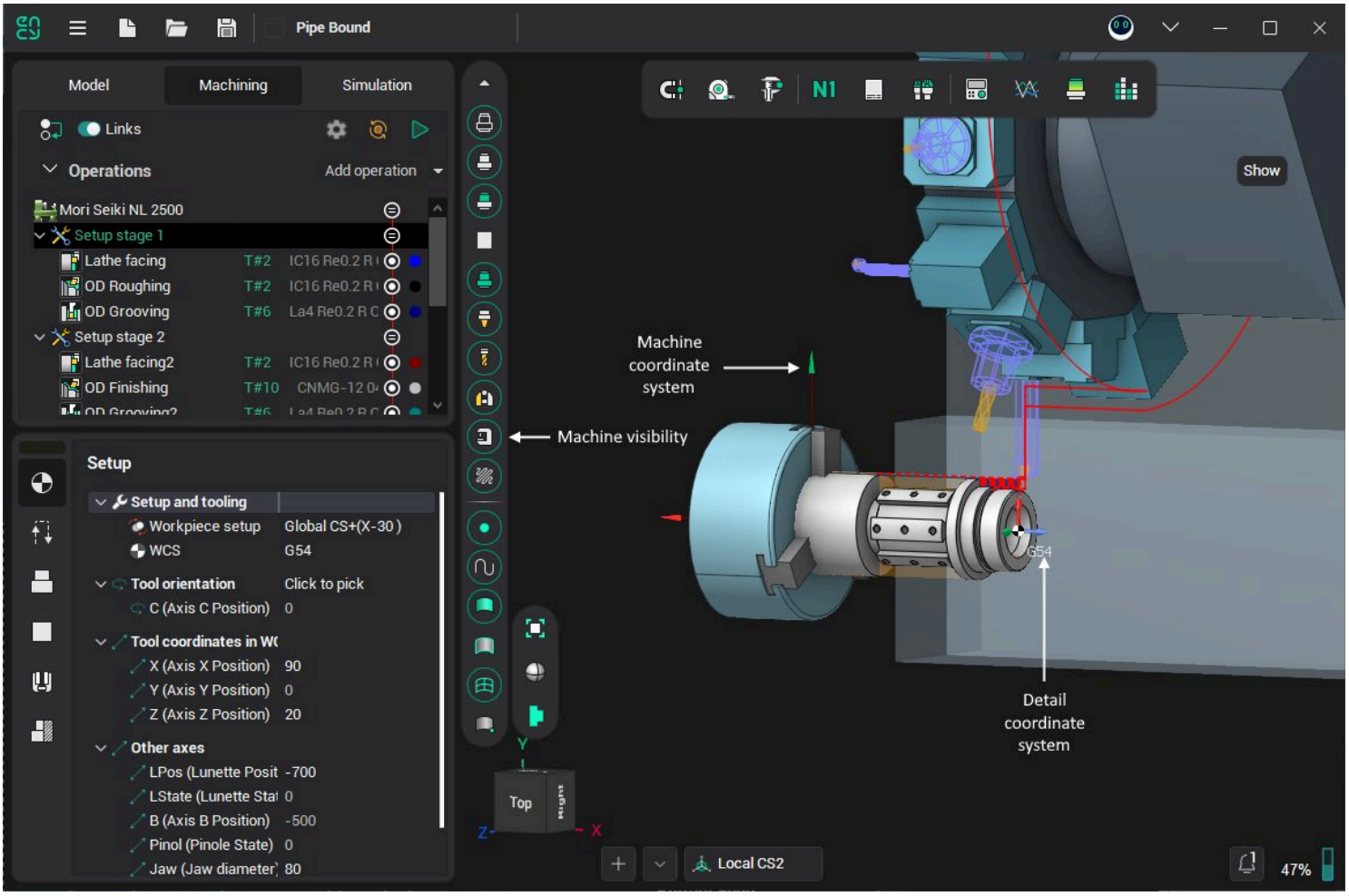
Block settings realized in chose machine window by double click on machine or by button Parameters


Parameters
Preferences
Machines
Custom Project Info
Loaded machine: Index G160 Normal
C:\ProgramData\ENCY Software\ENCY NB\Version 1\Machines\Schemas\Lath
Last machines
Machines from containers
Search folders
$(CUSTOM_SCHEMAS_FOLDER)
$(SCHEMAS_FOLDER)
C:\ProgramD..\CustomSchemas
C:\ProgramData\ENC..\Schemas
Search
Machine name
Group
Index G160 Normal
Turn-milling
Mori Seiki NL 2500
Turn-milling
Leader
Milling
Slovtos
Milling
5-axis milling machine (A,C)
Milling
5-axis Mill Machine (tool head A,C)
Milling
Hurco
Milling
DECKEL MAHO DMU60
Milling
Mikron VCE600 Pro (3-axis)
Milling
MicroCut VM1300
Milling
Ploter_noz (Knife cutting)
Milling
Hermle800
Milling
Mori Seiki NT 4250
Turn-milling
Lathe-milling machine (2 turret he...
Turn-milling
DECKEL MAHO DMU70
Milling
Description
Lathe-milling machine (2 t
Tooling
Top Turret Head: P
Face drill unit
Top Turret Head: P
Double OD clamping ur
Top Turret Head: P
OD drill unit
Top Turret Head: P
Face clamping unit
Top Turret Head: P
OD drill unit
Top Turret Head: P
OD clamping unit
Top Turret Head: P
Top Turret Head: P
Empty
Face Cutter Holder
OD Cutter Holder
OD drill ER25
OD drill ER32
OD drill unit
OD clamping unit
Double OD clamping unit
Face clamping unit
Face drill unit
Face drill unit (Inverted)
By Default
Ok
Cancel

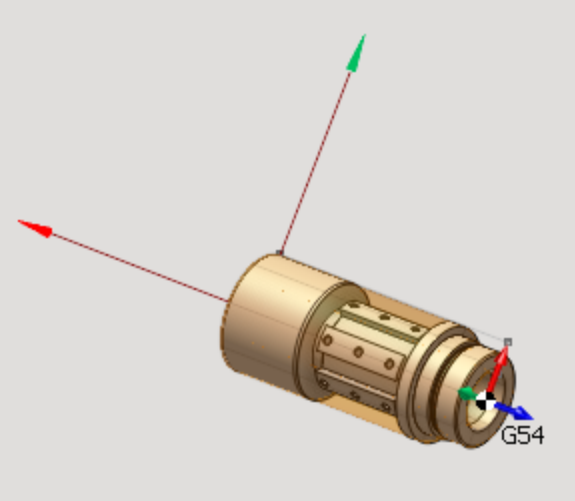
Positioning of part

After choosing a machine and after setting a capstan on it is necessary to set up position of detail. As a rule, on lathe-milling machines bodies of revolution are processing. Detail may be imported from CAD system, or setting by generatrix, built in 2D geometry mode. If the detail is importing then its generatrix restoring on 3D model base. This process does not always work correctly. When errors exists, recommended to import generatrix as a curve and setup the detail with it.

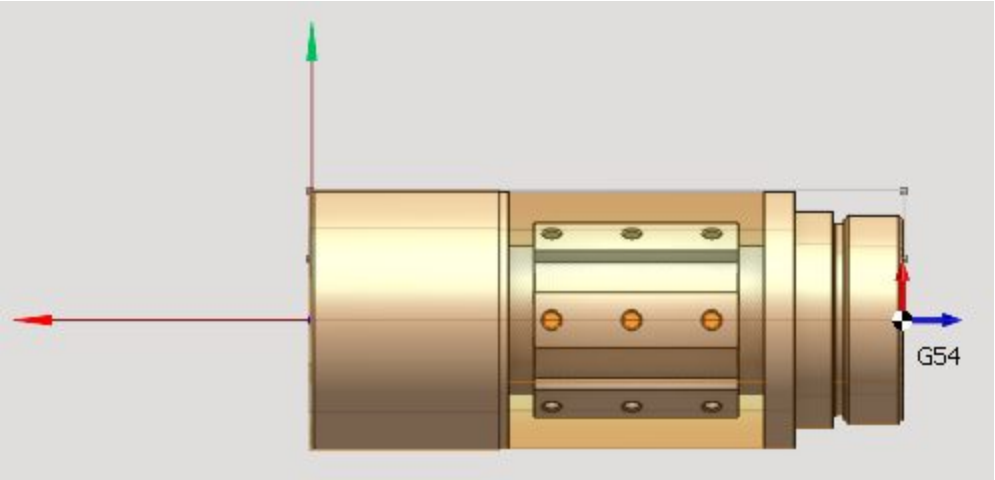
Necessary to discriminate part coordinate system and machine coordinate system. Detail coordinate system attached to detail and e.g. rotate with the detail when a spindle was rotated. Machine coordinate system is shown in left bottom corner of the screen when the machine is visible.



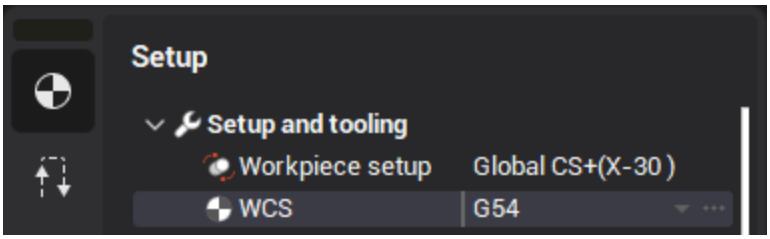
When visibility of machine is OFF auxiliary axles show a part coordinate system.



At setup detail for processing on lathe machines necessary to align detail axle with X axle. There are two ways to do this: by moving the detail or by creating new local coordinate system. Routine is to set an origin of coordinates on right butt of detail.

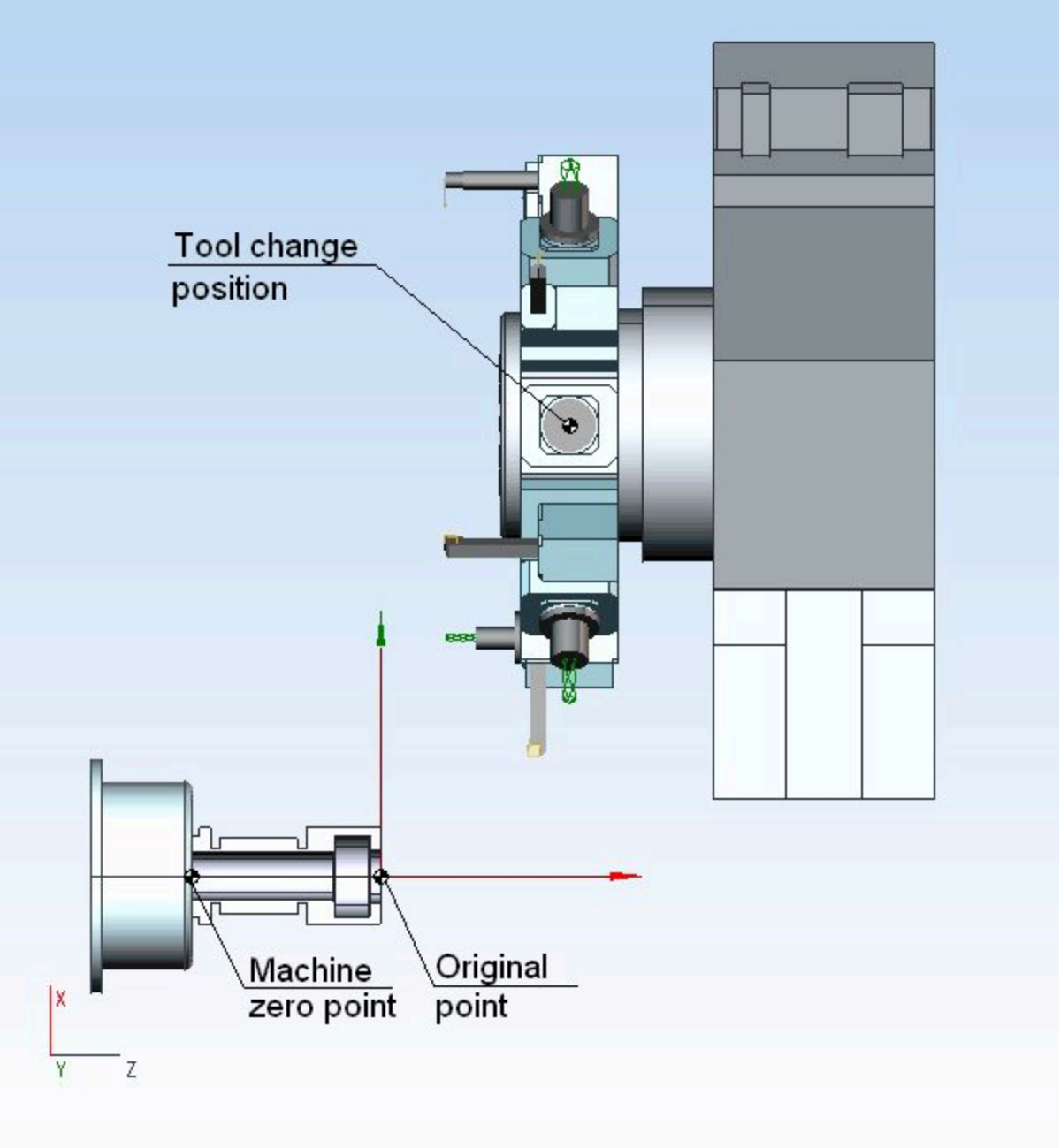


For created coordinate system become a set, necessary to indicate it in machine parameters or active operation. After this the system will superpose zero points of part and machine.



Machine zero point usually sets on a spindle nose. As result a detail will appear inside of machine. For a correct visualization necessary to set a detail zero point in machine coordinate system.

Tool change position



The coordinates of the point of tool interchange are counted off from the 0 of the machine. In CAM system, as a rule, 0 of lathe Z-direction is located on the spindle nose. So, the coordinate of Z of the point of tool interchange may be calculated by the following formula:

Z tool interchange = length of the part + overhang of the longest axed tool + safe distance

The position of tool interchange determines the distance of the center of the turret block from the axle of spindle, so:

X tool interchange = billet radius + overhang of the longest radial tool + revolver radius + safe distance

In some schemes of the machine the point of affixment on the revolver head may be dislocated in X-direction on some distance. In this case the definition of the point of interchange are chosen experimentally.

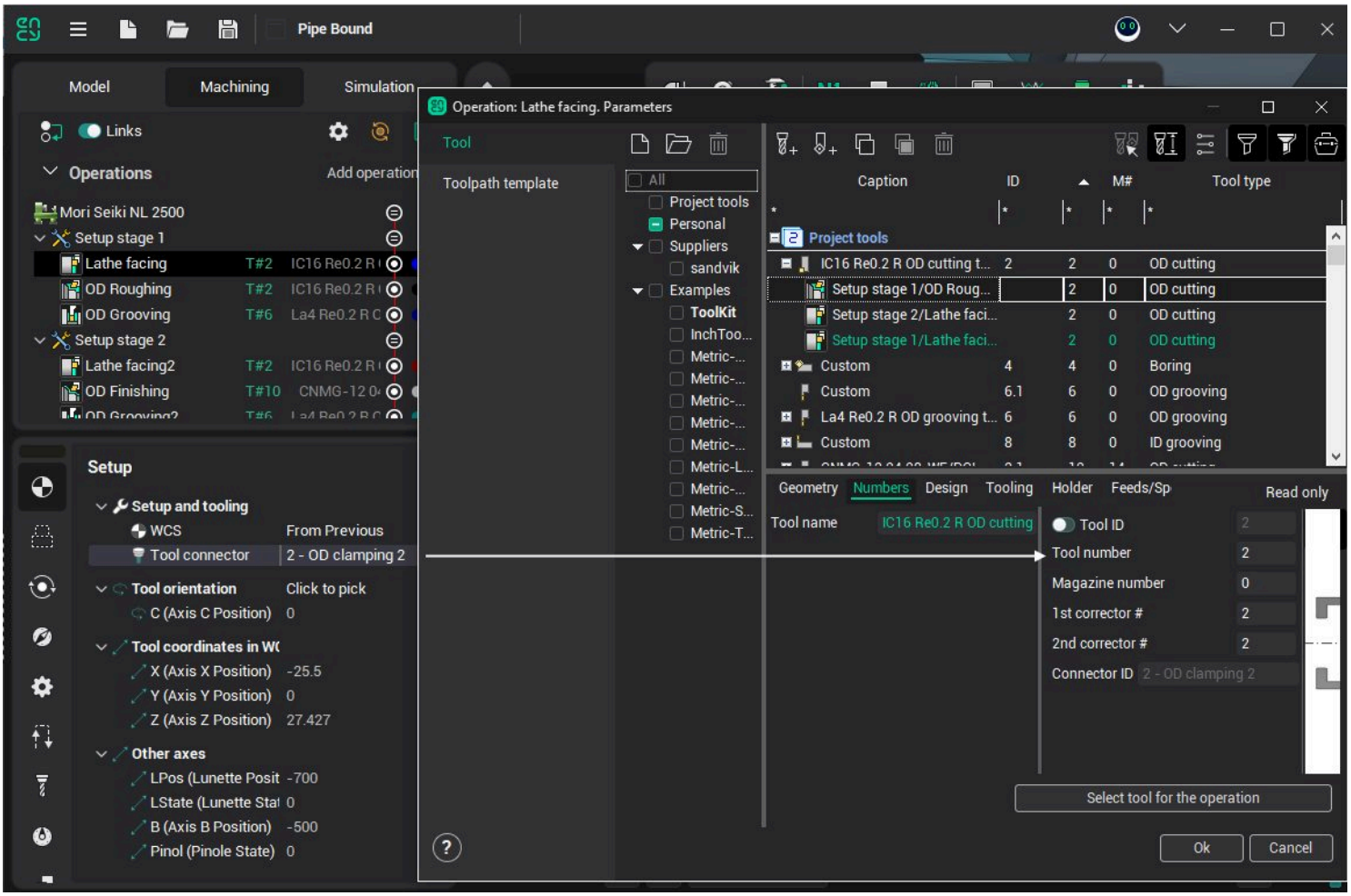
It is necessary to note that during the forming of axled or radial relieves, the point of tool interchange influences the NC program. For example radial relief to the point of interchange is executing in two steps, the first step move the tip of the tool in X-direction so that the center of a revolver head is found in the coordinate X tool interchange. So the following coordinate will be included in the NC program.

X=X tool interchange - the tool overhang by the axle - the radius of revolver head

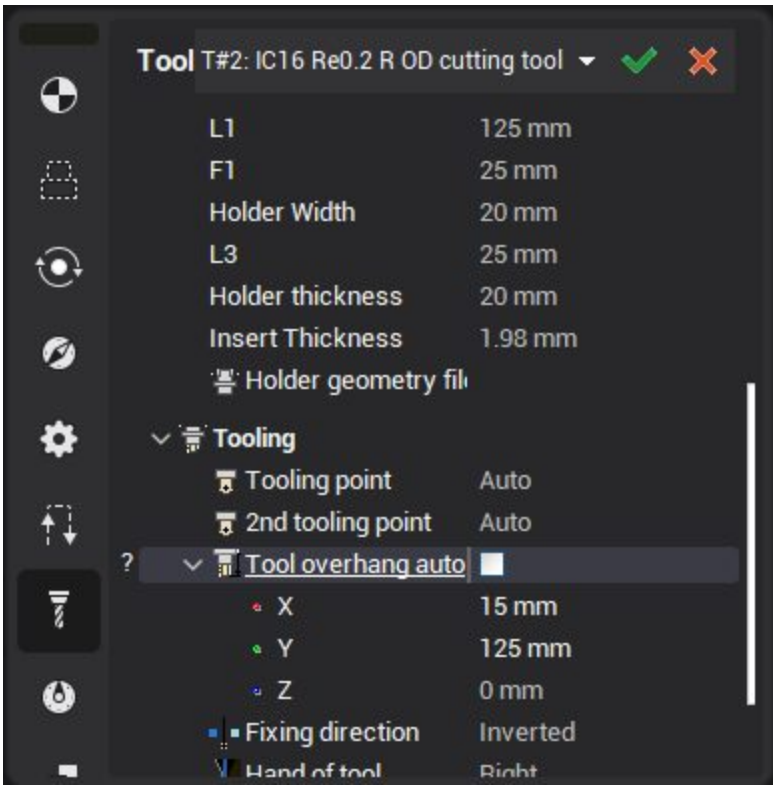
The following step is executing by the command **GOHOME** (G28), in other words it doesn't give coordinates.

Positioning of tool

After setup positioning of part and defining tool change point you can create new operation. One of main feature of operation is operation tool. For turret machine tool connector position (turret position number) define tool number for NC program output. So when you change tool connector then tool number is changed also. But otherwise when you change the tool number, tool connector number is not changed.

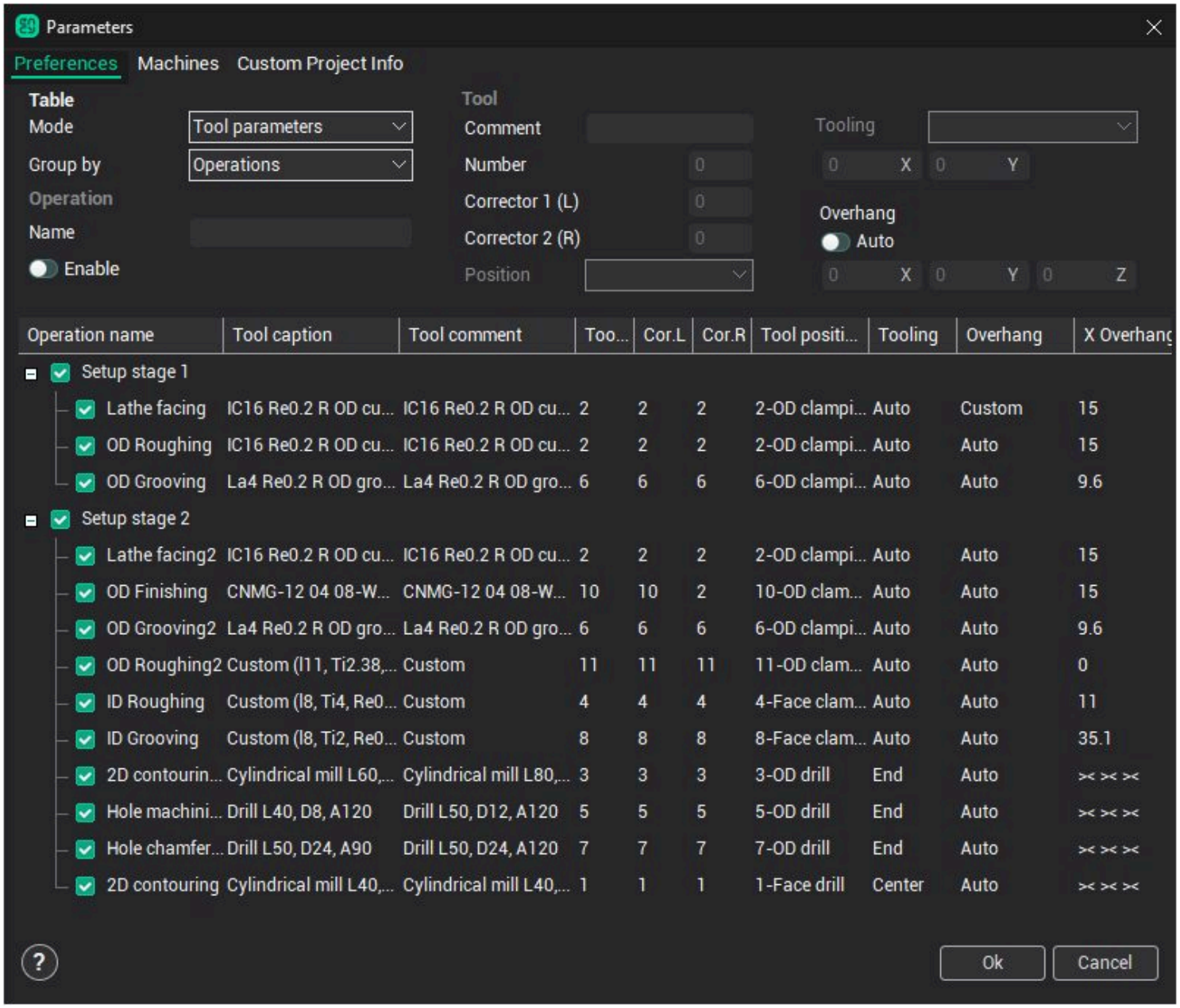


Tool overhang have action to NC program for move tool to tool change point. Tool overhang value influence to tool definition point at turret coordinate system. Setup tool overhang at window Tool at Technology mode.


Tool
T#2: IC16 Re0.2 R OD cutting tool
L1
125 mm
F1
25 mm
Holder Width
20 mm
L3
25 mm
Holder thickness
20 mm
Insert Thickness
1.98 mm
Holder geometry fil
Tooling
Tooling point
Auto
2nd tooling point
Auto
Tool overhang auto
X
15 mm
Y
125 mm
Z
0 mm
Fixing direction
Inverted
Hand of tool
Right

Obligatory testings before the final generation

The final control of parameters is executed in summary window of operations. To open the window choose root operation (the machine) and press the button Parameters.



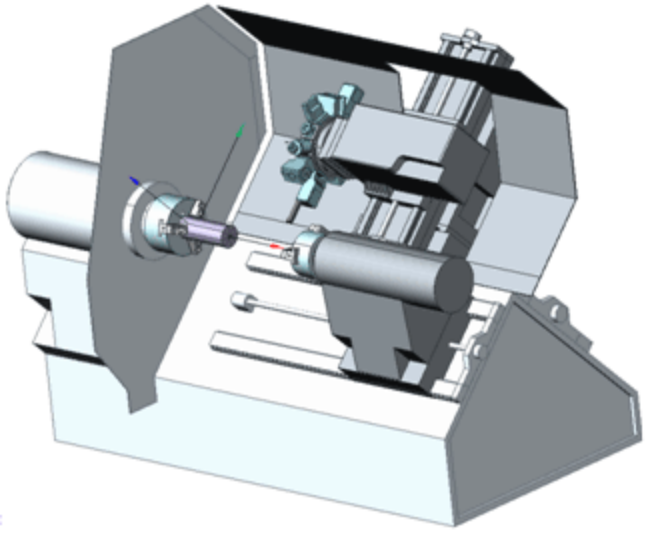
The window allows to edit one-type parameters of several operations at once. For that it is necessary to indicate the required operations holding down the button [Shift] or [Ctrl], after that input required values into the fields of input, located in the right part of the window. The window has two regimes: the regime of tools' editing and regime of editing of cutting regimes.

1. Check up the accuracy of installed tool numbers. The system doesn't control if in different operations under the same numbers different tools are installed.
2. It is obligatory to check up the tuning point of the tool in all the operations. If it is not correct simulation works correctly, and NC program is generating with a serious slip, such fact may lead to

tool breakage or may be even the machine.

3. Switch to the regime of control of cutting conditions, check up the direction of spindle rotation, cooling and accuracy of supply values.
4. After any changes of parameters and trajectory recalculation once more model the process of working and be sure in absence of exclamation marks.

Counter spindle machining



Application area:

Counter spindle machines allow machining of parts in single fixing, reducing idle time and increasing machining precision. Also, counter spindle presence eliminates need for locating tools. There are machining centers with two machining turrets in addition to counter spindle. These machines optimize idle time and virtually replace two machines. CAM system can create NC-programs for counter spindle machining for both lathe and lathe-milling machining technologies.

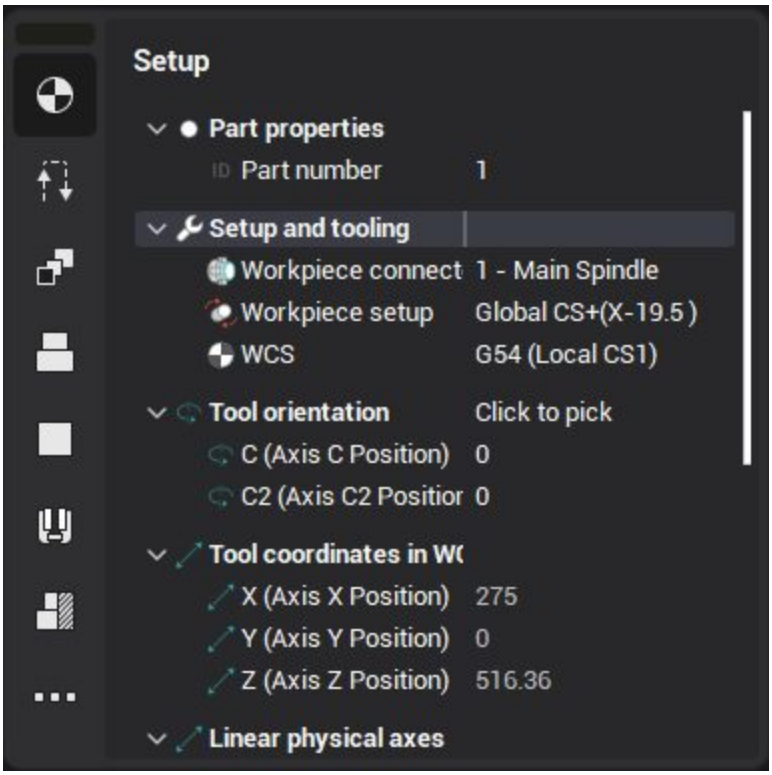
The peculiarities of working with machines equipped with a sub-spindle include:

The main project settings typically include:

What is necessary to know before developing counter spindle machining process:

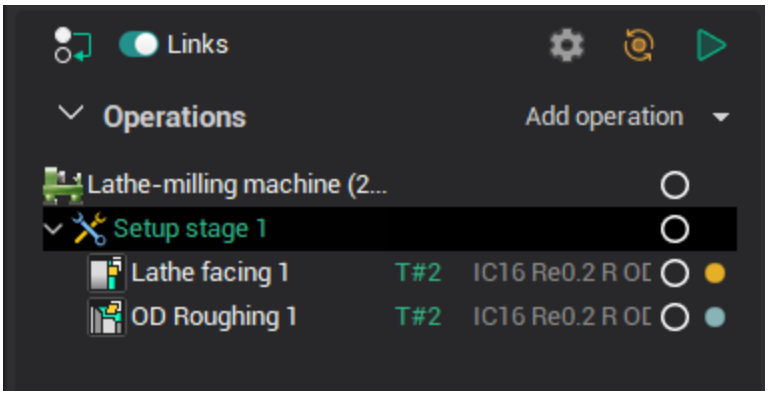
1. Spindle in which the machining will start.
2. Workpiece coordinate system (position in which the workpiece will be set). In the properties of this coordinate system specify it as Machine coordinate system, G54 for example. It is recommended to bind workpiece coordinate system to the non-fixed workpiece end.

First create the Setup Stage or Part operation which defines the initial position of the workpiece. Workpiece coordinate system and spindle where workpiece is fixed are specified on the Setup tab in the Workpiece connector, Workpiece setup and WCS fields respectively.



After defining workpiece location the exposed part elements should be machined.

Please note: To avoid mistaking active spindle in consequent operations specify `From the previous operation` *value for the Current workpiece holder field.*

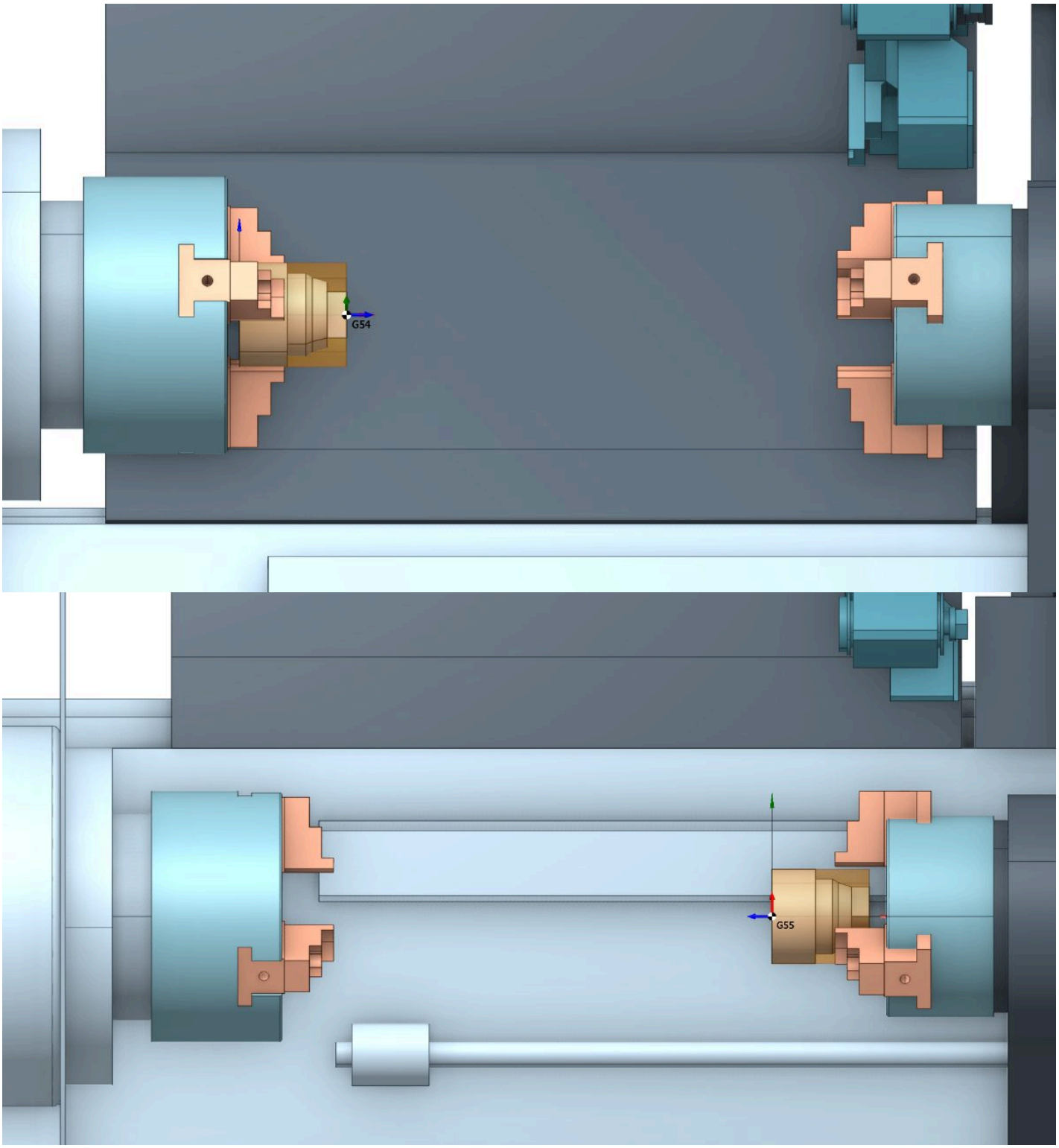


Workflow in lathe-milling projects

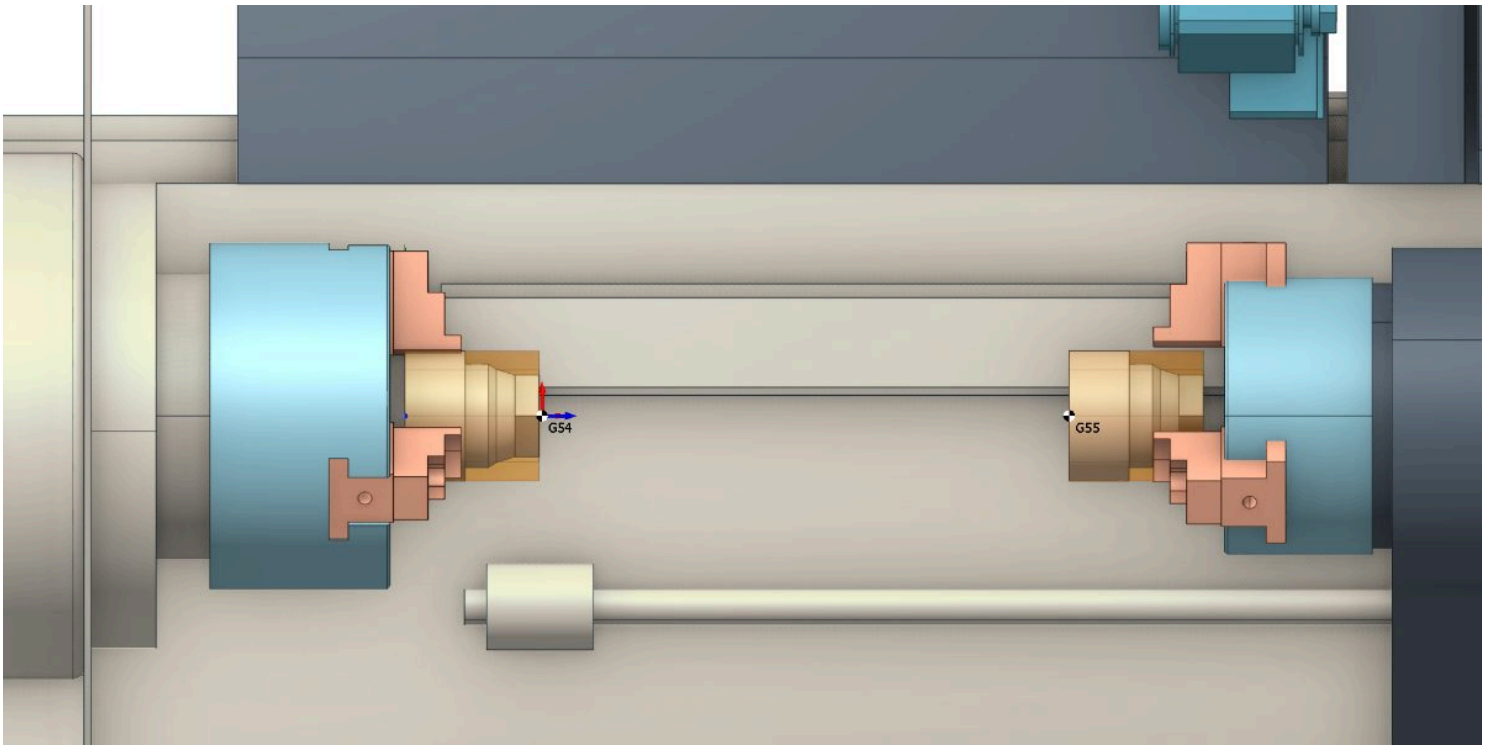
To refix workpiece from one spindle to another create a Stage or Part operation. For more information on the multi part projects see Multi part projects. Details

For a lathe-milling project, the following scenarios are typical:

- Setup 1 - Setup 2. Transfer of the workpiece from setup 1 to setup 2 for additional machining operations such as milling or drilling.



- Part 1 - Part 2. Machining of two different parts (Part 1 and Part 2) on the same machine, possibly with different operations and tools, to create complex or adjacent components.

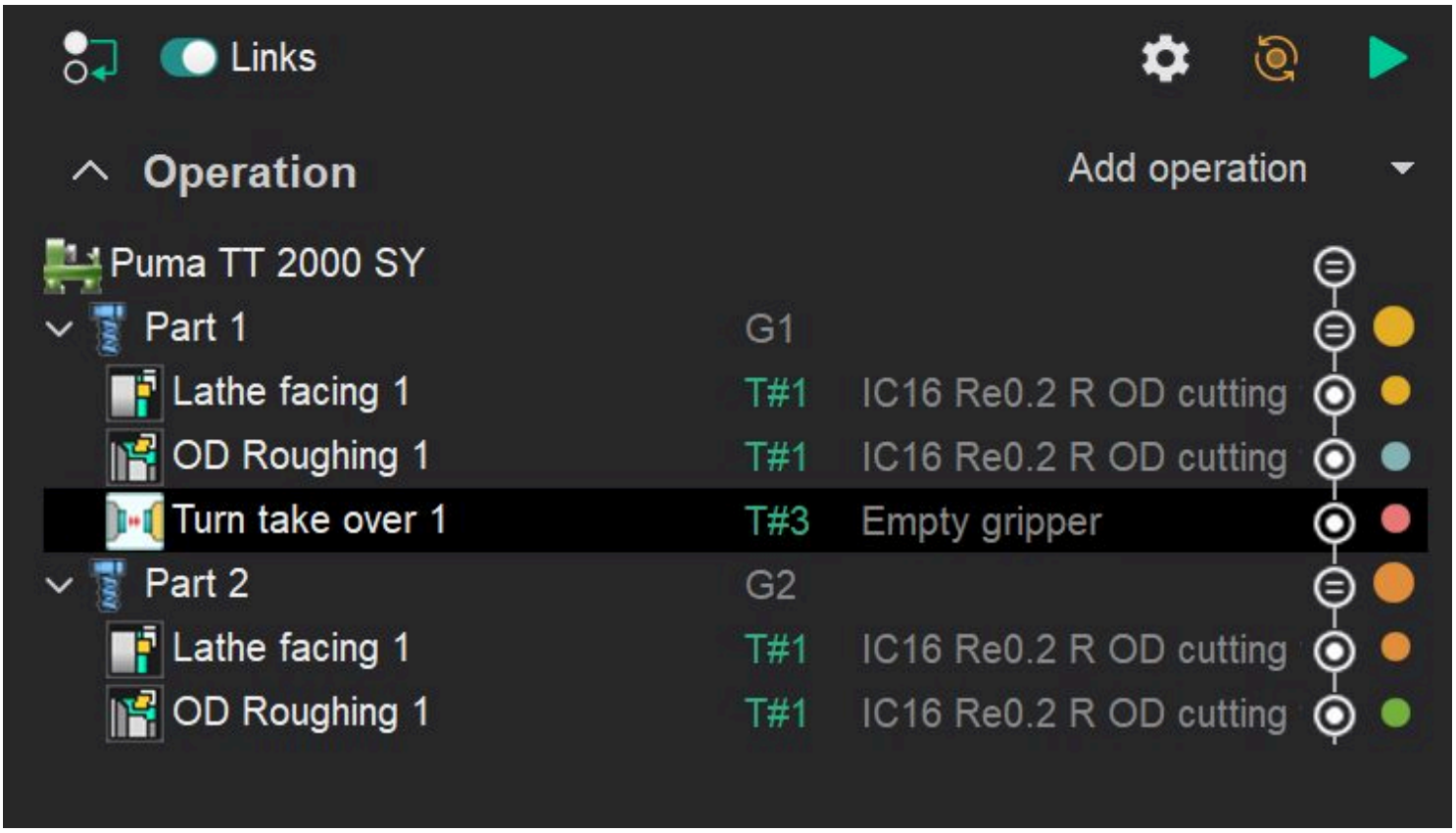


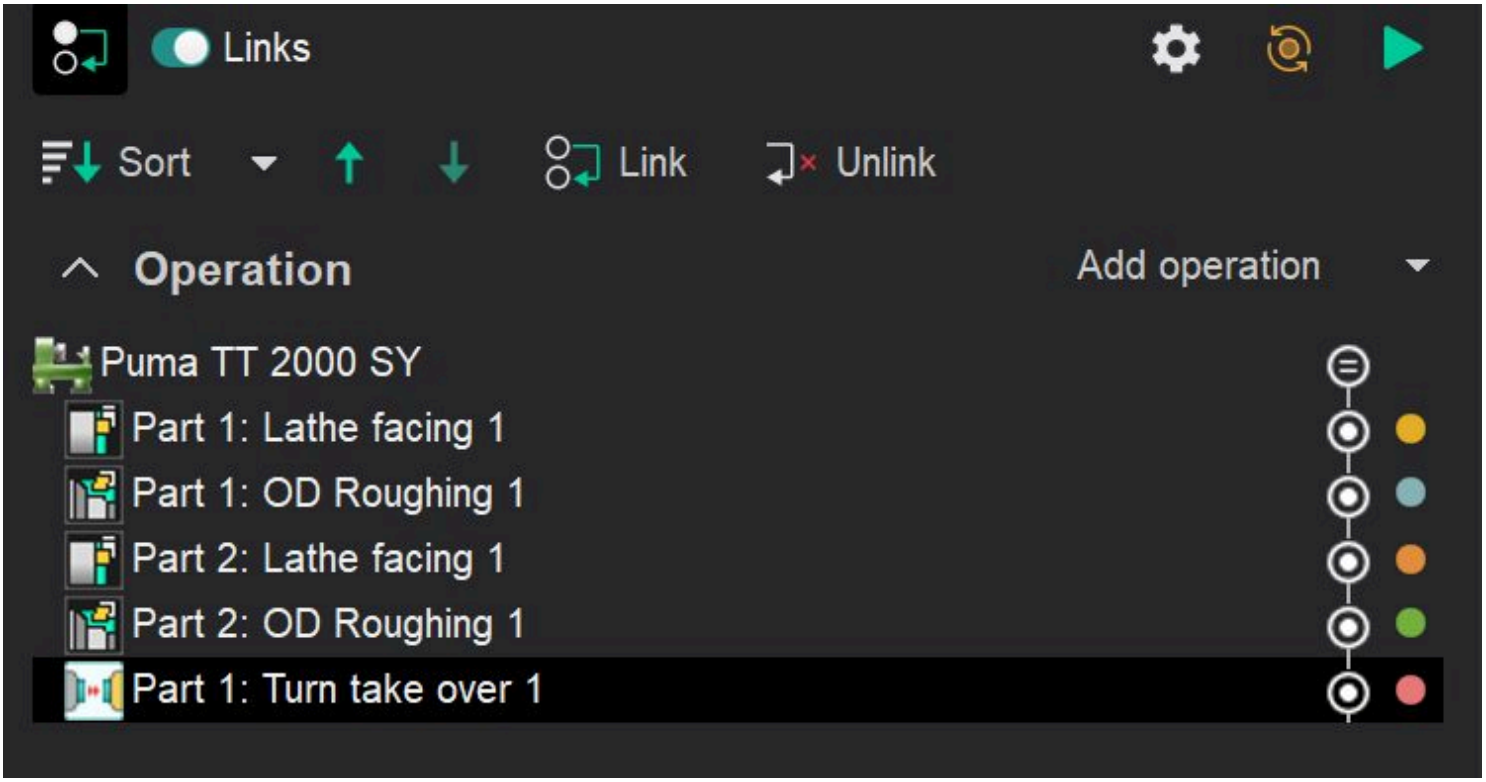
After refixing create operations to machine the rest surfaces. Operations can be either lathe or lathe-milling. These scenarios reflect the flexibility and multifunctionality of lathe-milling machines, allowing for various machining operations to be performed on the same equipment.

Move part.

To simulate the transfer of the part from the main to counter spindle you can use the Turn take over operation. Details

If the project involves multiple parts, don't forget to use Sequencing and Links to move this operation to the end of the manufacturing process.

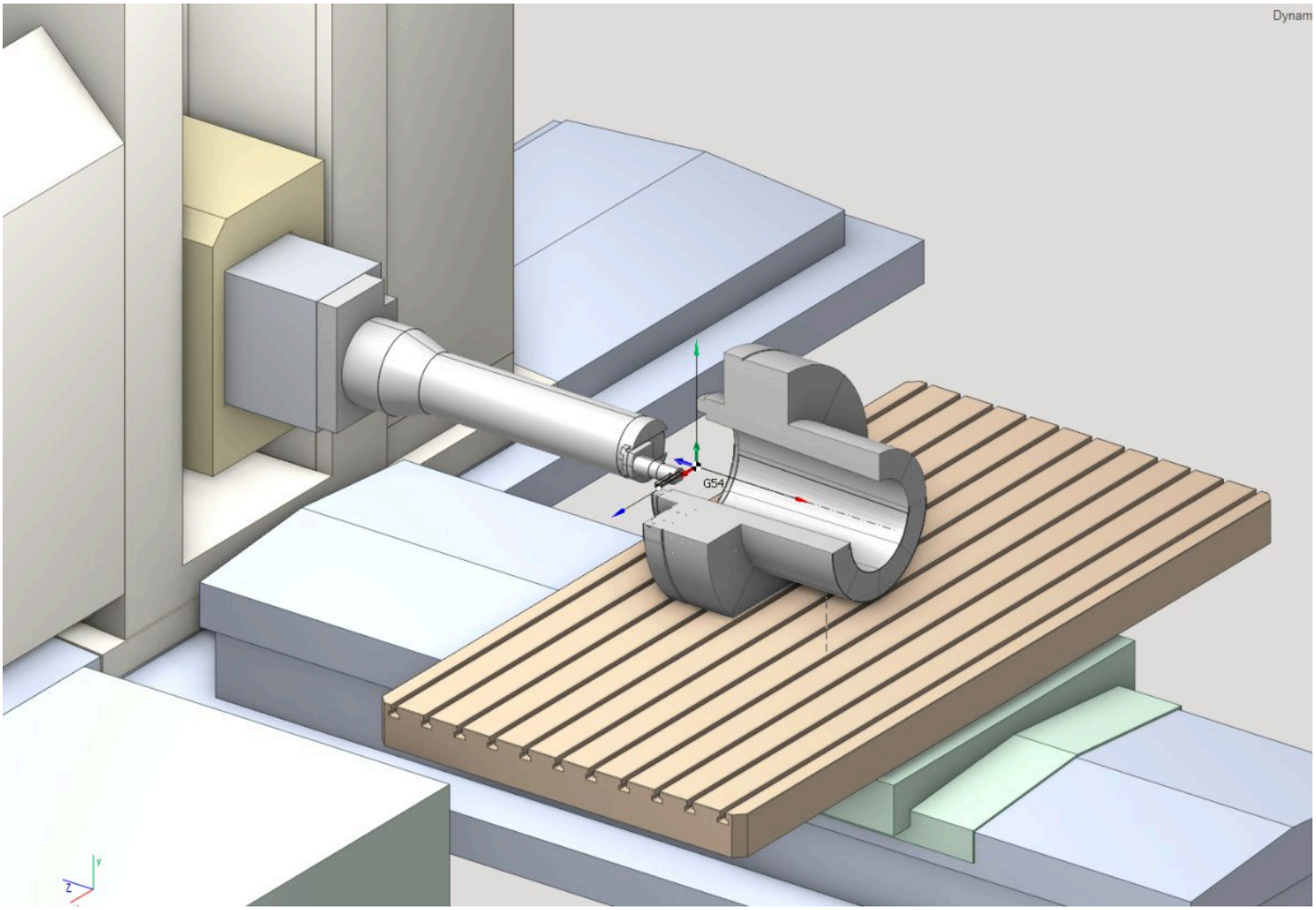




It is assumed that this operation, Turn take over, simulates the ejection of a part (semi-finished product) from the spindle, in which a new workpiece must be fixed. Then the process of interception of the part follows.

U-axis turning

About machine tools with U-axis



U-axis turning feature is designed to machine the large unbalanced parts. On these machine tools the workpiece is placed STATIONARY and the tool is rotated. U-axis is used to change the machining radius.

Machine schema requirements

The example of the schema with U-axis is included in standard package (Machine name "LR521"). It has the special section TrevisanSubMachine in the list of submachines. In this section the name of U-axis is defined. More details about submachines you can read here.

Trevisan submachine definition example

```
<TrevisanSubMachine>
   <!--turning-->
   <ToolNode>AxisU</ToolNode>
   <WrkNode>Workpiece1</WrkNode>
   <XAxisID>AxisX</XAxisID>
   <YAxisID>AxisY</YAxisID>
```

```
    <ZAxisID>AxisZ</ZAxisID>
    <ToolAxisID></ToolAxisID>
    <UAxisID>AxisU</UAxisID>
    <OriginG54BaseNode>AxisW</OriginG54BaseNode>
    <ApproachRule>G53 Z0; G53 B; LHPos;U;XY; G53 W;Z;</ApproachRule>
    <ReturnRule>Z10;XY </ReturnRule>
  </TrevisanSubMachine>
```

Project and tool path

The example of project is included into the standard package (project name "U-axis turning.stcp"). The turning axis always is going through the workpiece zero (G54-G59). Define the correct Origin (G54-G59) and B-axis position before create the lathe operation. CAM system calculates the generatrix, based on the defined axis, and draw the solid of revolution.

Generated CLData will contain the command to switch the interpolation axes X/U. This command must be analyzed in the postprocessor. Tool tip point also depends on this mode.

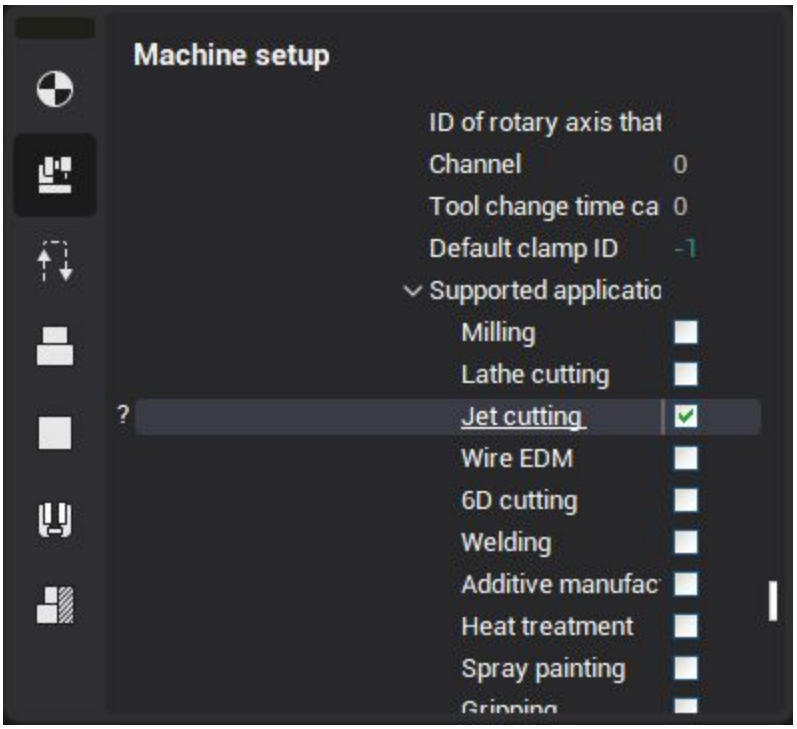


Machining on cutting machines

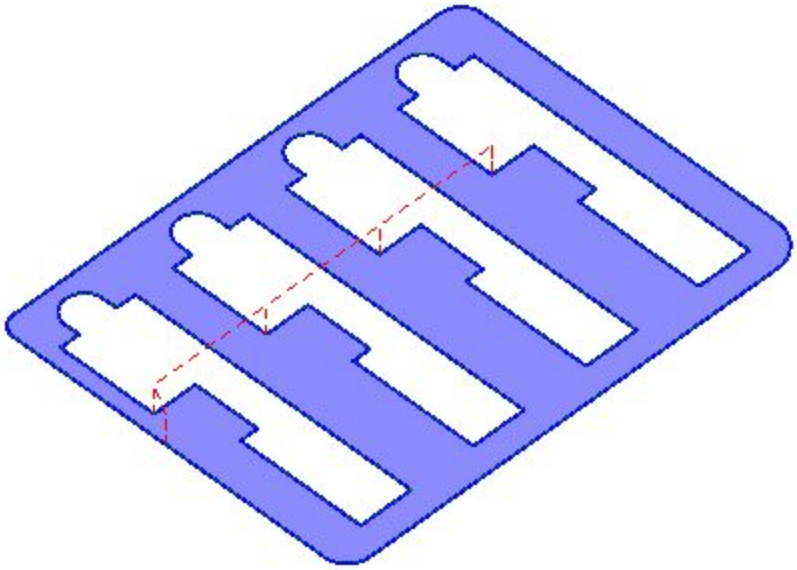
The cutting machines – equipment intended for the cutting of flat parts from different types of sheet material. The equipment is divided in two groups by the way of cutting. The most widely distributed machines are based on such methods:

- Plasmic cutting uses compressed electric arc which is blew by gas. Blowing the arc, gas is warming and decomposing to positive and negative charged particles (ionizing). The charged particles are transformed into compact current of plasma with the temperature up to 15000 C.
- Laser cutting is one of the most modern technology which allows to make patterns from any shift material at contour. At the heart of this technology there is the work of in-focus laser beam. Such instrument is suitable for cutting of different types of sheet materials. The main value of in-focus laser beam – the smooth surface of the cut, and high accuracy of the cutted lines.
- Oxygen cutting – a method of cutting hardware, based on the properties of metals, heated till the temperature of inflammation, to burn in technically pure oxygen. At the oxygen cutting, oxygen stream burning through metal and cutting it, is directed to metal, heated up to 1200-1300 C.
- Hydroabrasive cutting is based on the appliance of water stream of high pressure (400 MP) mixed with pomegranate sand. The big power of the stream support cutting, exactly speaking, destruction of solid structure on molecular level. This method allows qualitatively cut not only any kinds of metal and fusions, including intractable (non-rusting and heat-resistant steel, solid and titanic fusions) but also granite, ceramics, fire-proof and bullet-proof glass, rubber, paper, felt, composite and other materials).

Cutting operations are available in the list of operations in the "Cutting" group if the selected machine supports the appropriate type of machining only (has in its structure tool connector with enabled "Support / JetCutter" property). To make it available in the machine in which it is disabled by default, you should enable "Supported applications - Jet cutting" check box in the tool properties section of the machine parameters, as shown in the figure below.


Machine setup
ID of rotary axis that
Channel 0
Tool change time ca 0
Default clamp ID -1
Supported applicatio
Milling
Lathe cutting
?
Jet cutting
Wire EDM
6D cutting
Welding
Additive manufac
Heat treatment
Spray painting

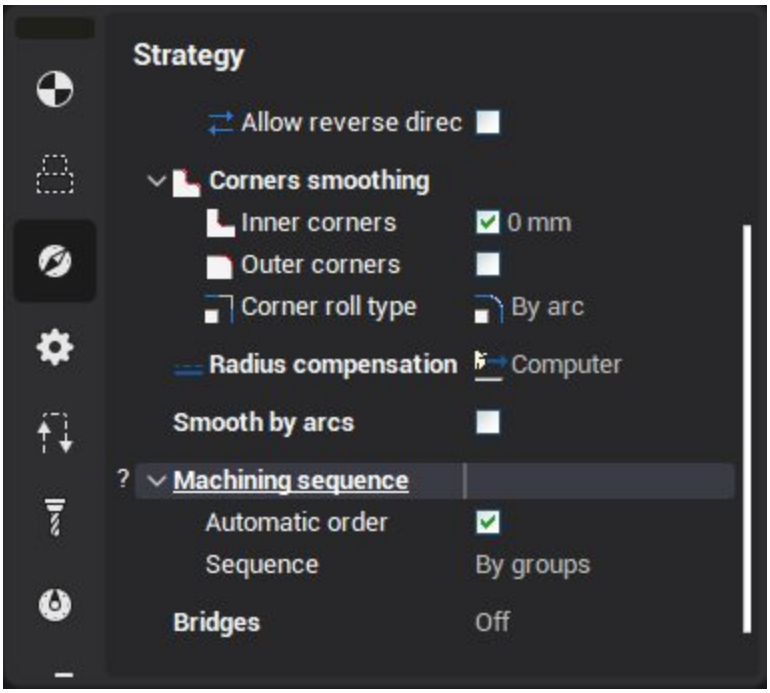
Jet cutting



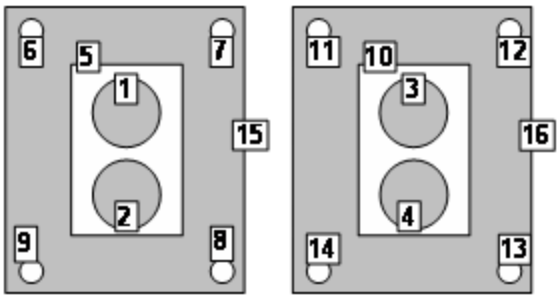
The operation is intended to carve the details from the sheet workpiece. The contours of the detail are defined by the curves projections.

The main differences from the 2D contouring is: the machining order, the definition bridges and the step over strategy. At the first the inner contours are machined. The outer contour is always machined at the last. The reason of this rule is the next. In the outer contour is machined at the first then it is impossible to carve the inner holes because the detail is not fixed yet.

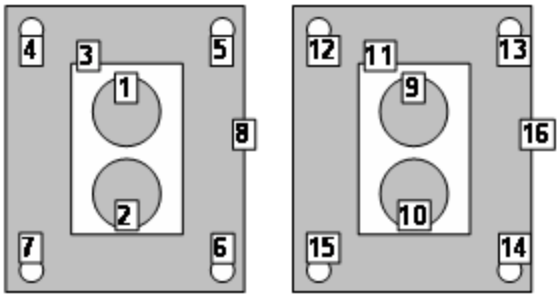
Use the panel below to define the machining order. The panel is located on the Strategy page of the parameters window.



If the option is ticked then the contours with the maximal nesting is machined at the first. The contours with the progressively less nesting are machined at the nest. And the outer contours are machined at the last.



If the option is off then the machining is performed by groups. The contours are machined from with the maximal nesting ones to the outer ones. After that the next group is machined. This machining has not much idle motions.



Every model item (contours) has the machining parameters. The contour can be machined from the left or from the right side. It is possible to set the tool.

Every object can have its own machining method: either the tool center passes along the contour or by touching it with the left or right of the tool. If the contour is machined from right or left, then it is possible to define an additional stock for it. Positive stock is laid off towards machining. If the center of the mill follows the contour, then the stock value will be ignored, for it is impossible to define exactly which side the additional stock should be laid off.

If in the operation there is a workpiece or restricted areas that have been defined, only those areas of the defined contours will be machined, which lie within the workpiece and outside the restricted areas. If neither a workpiece nor restricted areas are defined, then the system will machine all the defined contours without any limitations.

The operation tool is the cylindrical milling cutter or the jet. It is assumed that the tool length is unbounded. So the machining levels (top, bottom and safe levels) are not defined.

If the operation is performed using a local coordinate system or if using a swivel head then the system performs machining using the XY plane of the local coordinate system, and all work passes are

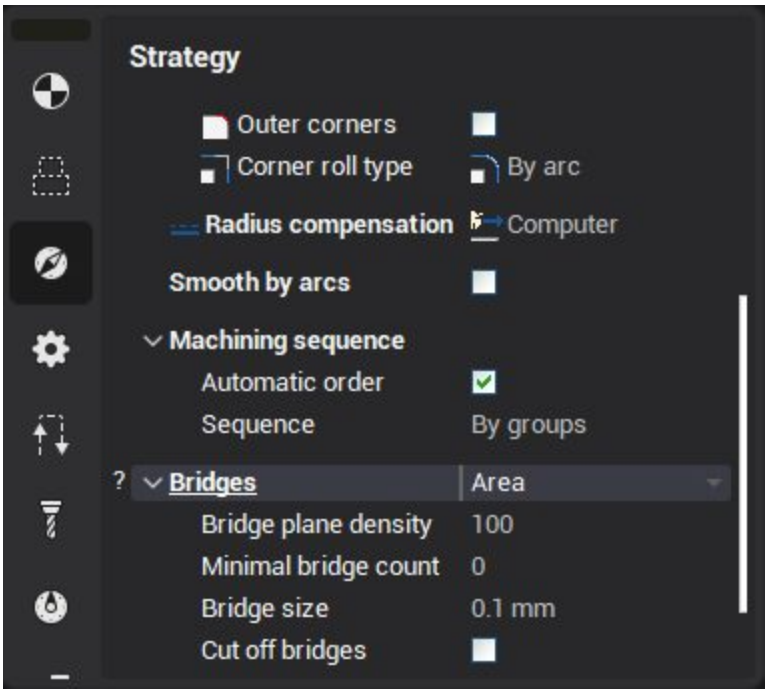
consequently parallel to the XY plane of the local coordinate system.

The start point for machining an open curve corresponds to its first or last point (depending on the settings used on the Model page and Inverse tick, and also the Allow reverse direction setting). For closed curves, if the initial point has not been defined on the Model page, approach to the first machining point is performed to an external corner or to the longest section automatically, to optimize the tool movements.

When the joining of the resulting toolpaths is calculated, the approach type selected will be added at the beginning of each toolpath and the retraction type at the end. The toolpath joining sequence depends on a combination of the settings of: curve/offset, compensation.

If Allow reverse direction is selected, then the cutting order will be set with regard to the Idling Minimization setting. The side of contour machining will not change. Otherwise the contours are machined in the order that they appear on the Model page. It is possible to define a start point for each of the profiles being machined.

Modes of definition of quantity of bridges:



- None – a mode of processing without bridges.
- Count – is defined concrete quantity of bridges. The equal quantity of bridges for all curves (for not closed curves a quantity of bridges on 1 less) will turn out.
- `Length (L)` – quantity of bridges is defined as length of a curve divided into the size specified in parameter C (for not closed curves a quantity of bridges on 1 less). The quantity of bridges for each curve varies proportionally to its length.
- `Area (A)` – quantity of bridges is defined as the curve area divided into the size specified in parameter C (the opened curves are processed without bridges). The quantity of bridges for each

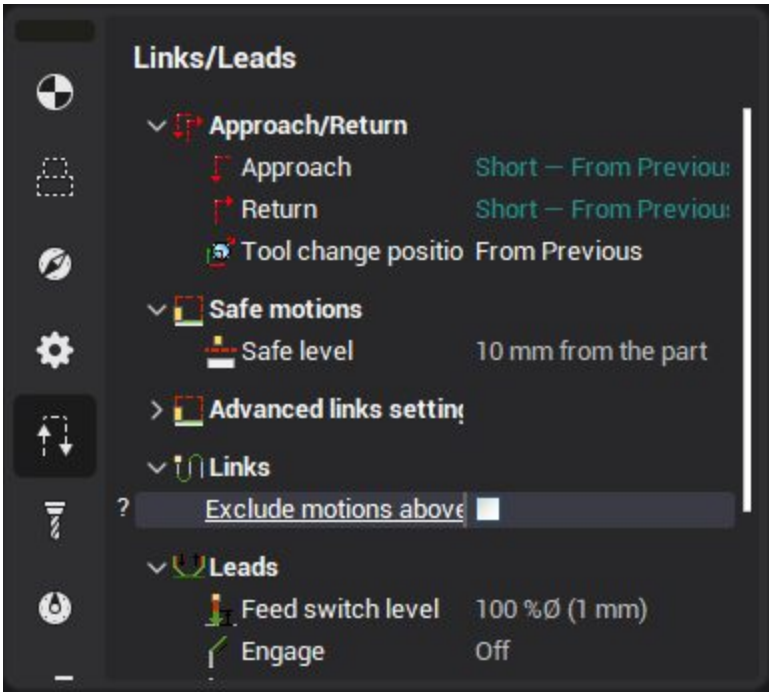
curve varies proportionally its areas.

Parameter of Minimal count defines is minimum possible quantity of bridges for a separate curve. If under settlement formulas the quantity of bridges turns out smaller the quantity of bridges is accepting equal to parameter of Minimal count.

Parameter the Bridge size defines length of a bridge (it is calculate along a curve).

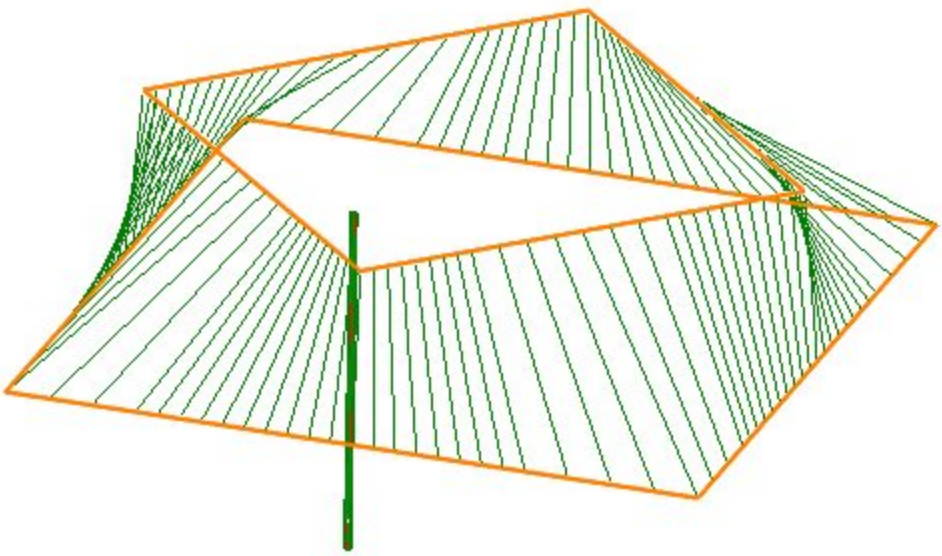
The flag a Cut off bridges allows to leave bridges and to process them after processing of all curves. At the established flag after processing of curves the command an additional stop (OPSTOP) will be deduced, then will be made machining of bridges.

Exclude motion above machining parts establishes how to carry out transitions between contours. At the switched off strategy transitions will be made at safe level on the shortest distance between points. At included transitions are made so that the tool did not pass over contours already cut out earlier.



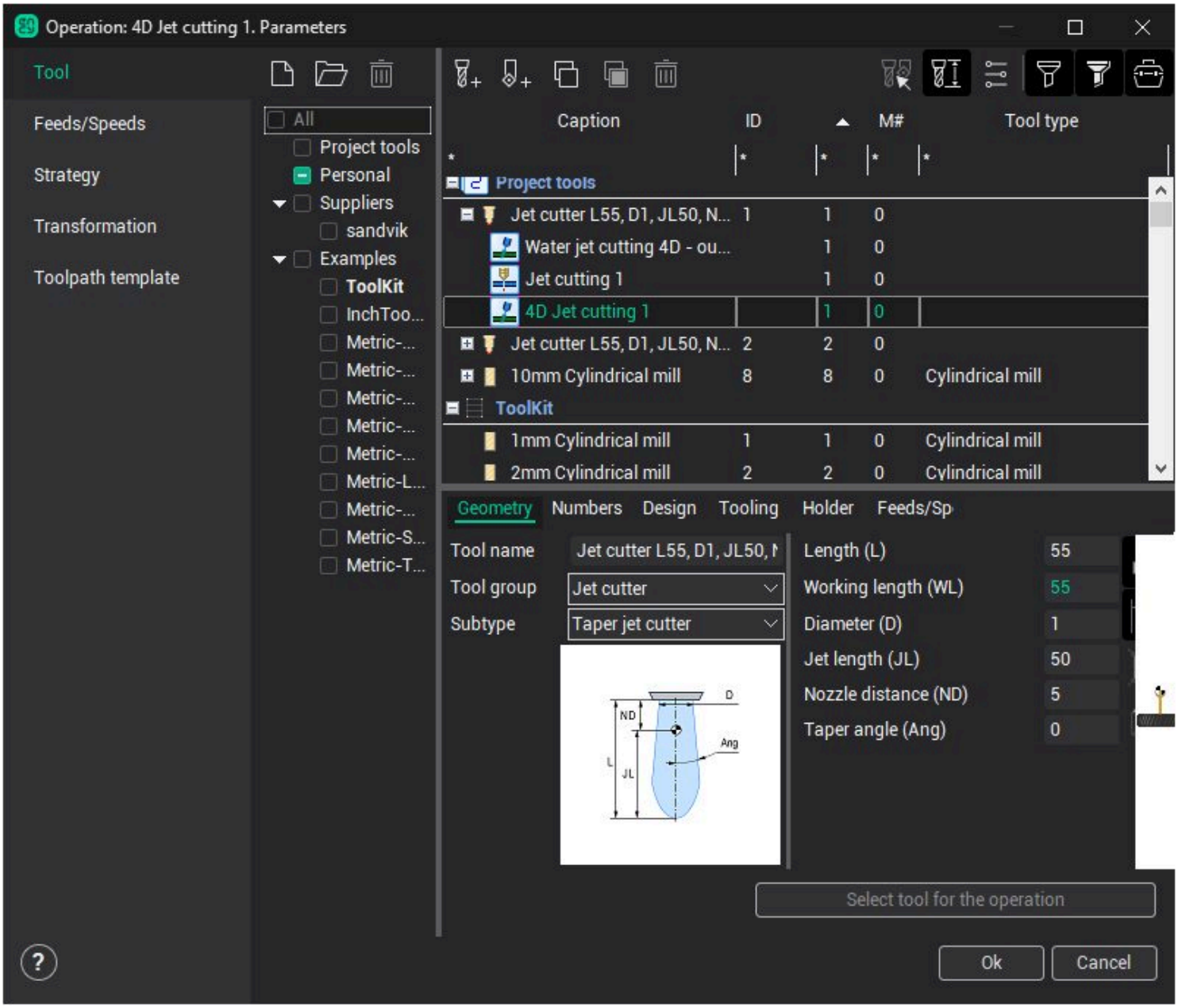
4D Jet cutting

Cutting 4D operation allows you to create not only purely 2D toolpath when the tool axis is oriented vertically, as well as more complex 4D toolpath when the tool axis is tilted according to the wall angle of the working geometry. At the same time, the tip of the tool is always positioned in the same plane, so the Z coordinate remains constant within the same contour. Accordingly, the processed geometry can be defined either as a flat curve (2D case), either two curves (upper and lower, 4D case), and the synchronization line also. They connect these contours and determine the tool angle in each position, as shown below. Working geometry is specified in the Job assignment window.



The tool of this operation by default is a Jet cutter that have properties listed below.

- Jet length - the length of the jet or beam that removes material in simulation mode.
- Diameter - jet or beam diameter at the exit point of the nozzle.
- Taper angle, if it has tapering.
- Nozzle distance - distance from the lower edge of the nozzle to the tool contact point, i.e. a point which will move on the upper level of the working contour defined in job assignment. By default, this point is the same with the tuning point of the tool, that is output to an NC code.

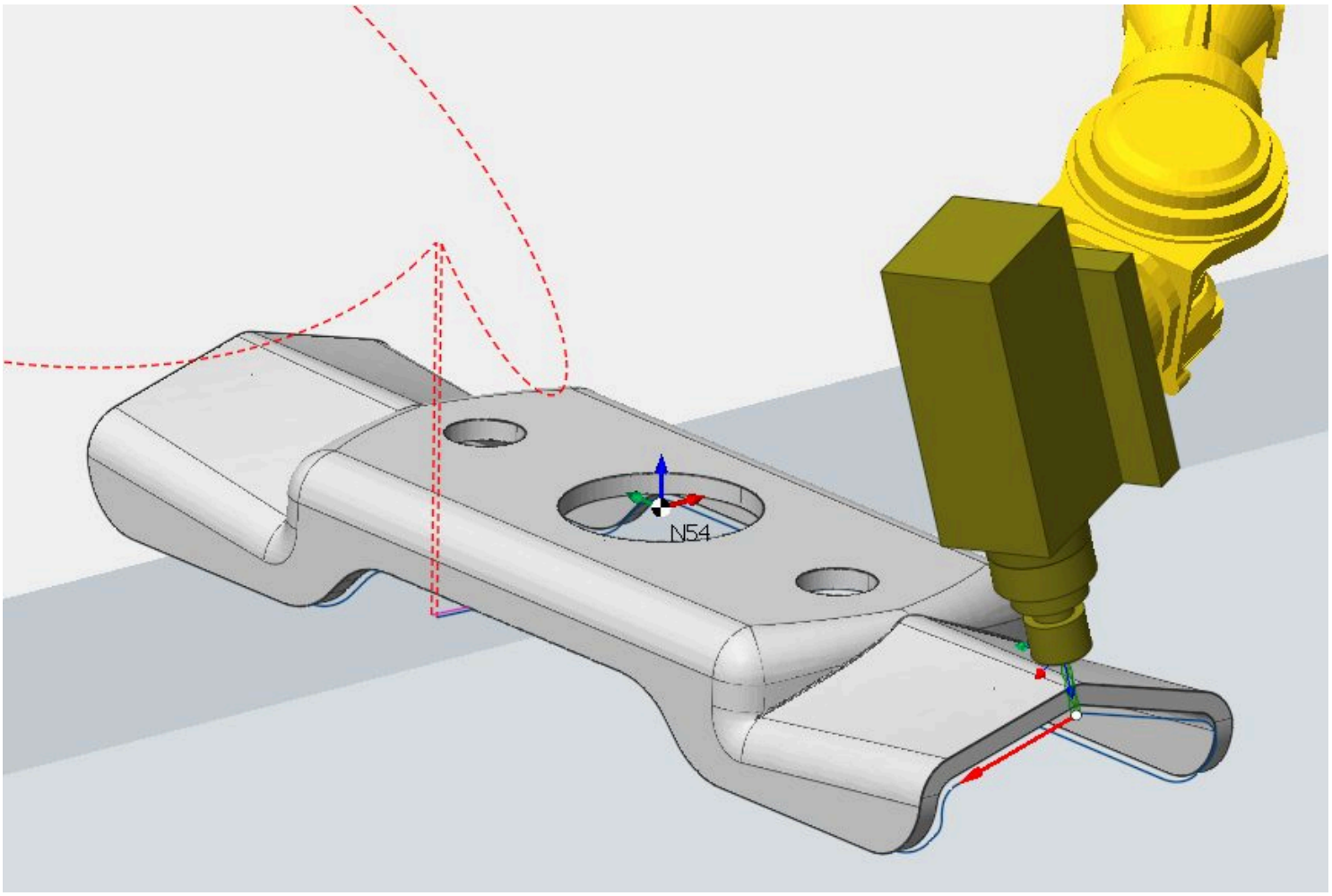


Cutting conditions definition window allows to specify parameters of the each offset pass individually (if you need more than one pass).

- Feedrate
- Cutting condition code
- Offset distance for exact level
- Offset code or radius corrector number.

On the Strategy tab of parameters window you can set parameters such as order and direction optimization, multi pass machining options, bridges etc.

Jet cutting 5D



Operation "Jet cutting 5D" is designed for the cutting on the shaped spatial surfaces. It is based on the operation "5D contouring" excluding multipass machining feature unnecessary for this kind of application.

The "way of definition of the machining profiles" and other parameters, which are not described in this chapter, is written in the description of the operation 5D contouring.

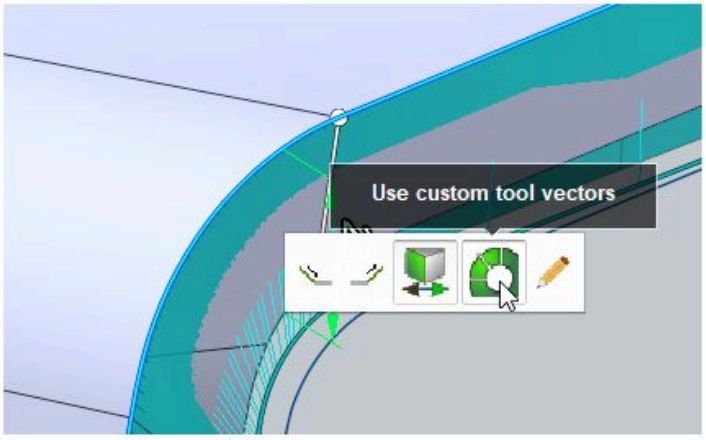
The operation requires at least 5 degrees of freedom. So active machine must have a minimum of three linear and two-three rotary axes. Very often the industrial robots are used for the cutting. If the machine schema doesn't support all degrees of freedom, then the generated tool path will be incorrect.

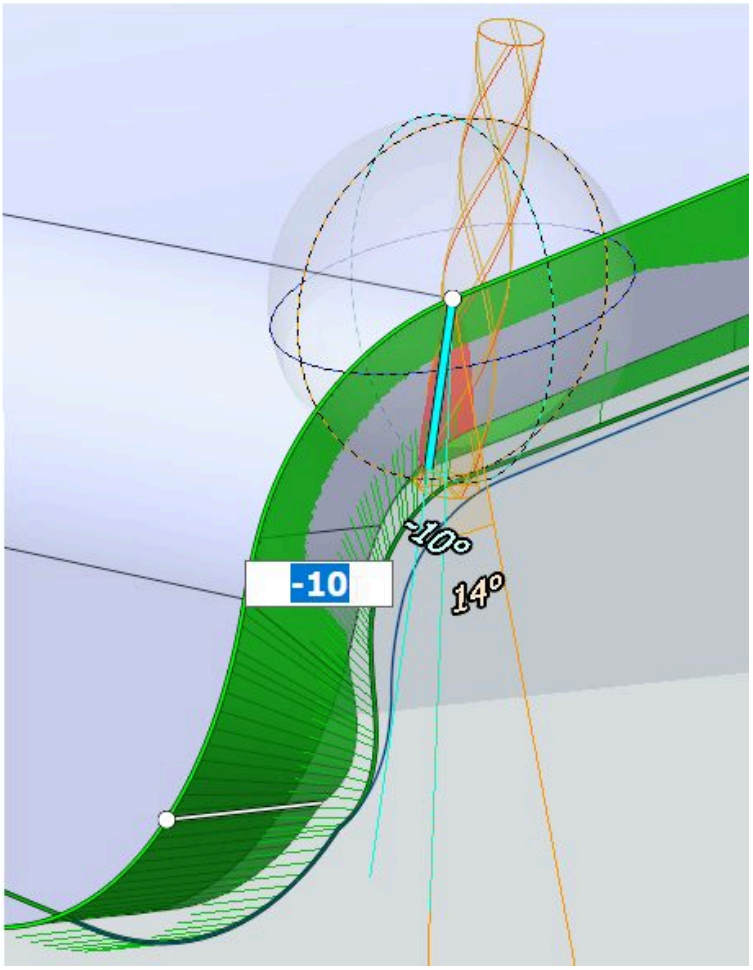
In the job assignment you should define the contours along which cutting will be performed:

- edges of the model then the tool orientation vectors will be calculated from the neighboring faces of the edge,
- simple curves drawn in the CAD,
- spatial spline curves drawn in the CAM directly.

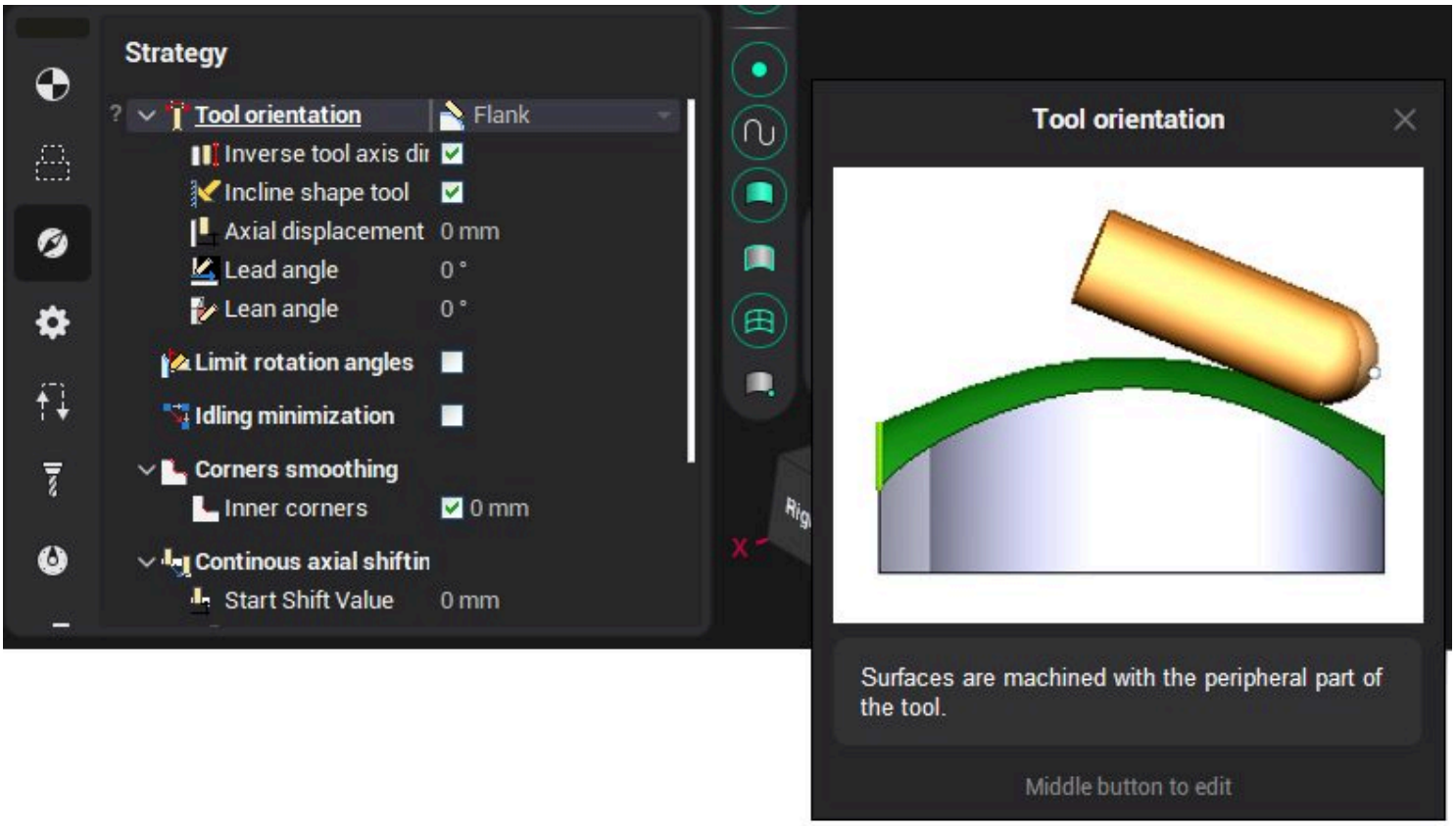
In the last two cases the tool orientation angles will be calculated from the nearest faces of the part.

After the contours are set, you can switch to the custom vector editing mode, and adjust the exact tool angles at each of the curve points in the graphics window directly.





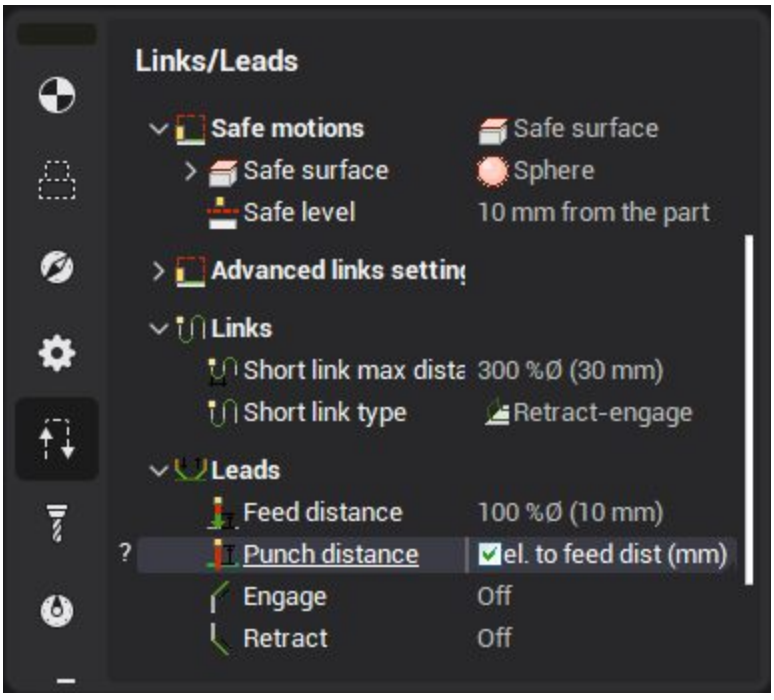
Also you should define the correct tool orientation law in the parameters. The operation contains a rich set of strategies. Click the smart hint button ? next to the each parameter to get additional information.



Also in Links/Leads tab you can use punch distance. This distance allows the tool to be retracted to the distance required for punching.

There is two value:

1. mm (entered value is calculated in mm)
2. rel. to feed dist (the entered value is calculated relative to Feed distance)



This parameter is also available in Plasma Operations.

Operations setup

This section describes how to configure jet cutting operations, from assigning the geometry to selecting machining conditions and strategy settings. It covers contour selection, machining direction, cutting side, and the properties of job assignment items.

The topics also explain settings shared by wire EDM and 4D jet cutting, including synchronization of upper and lower contours, approaches and returns, bridges, and compensation. Use this section when preparing the job assignment and adjusting the cutting sequence and toolpath behavior.

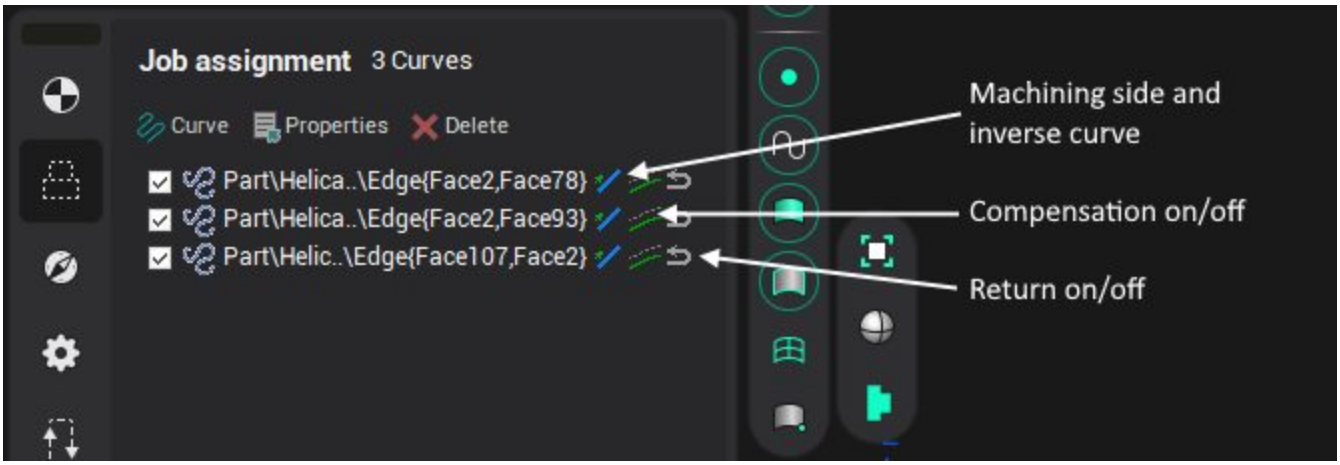
Jet cutting job assignment

The Jet cutting operation allow the user to create a toolpath from a selected curve. At a very basic level, using this operation, a curve can be transformed into an NC program. Curves that are to be used for creating a toolpath should be added into operation's Job assignment curve list. To define a circle model element can be added as a point with a stock. To add the point (circle center) as a model element add it in the Job assignment dialog and assign it an additional stock equal to the radius of the circle.

If several curves are selected for machining and idling minimization mode is deselected on the Strategy page in the Operation parameters window, then the order of their machining will correspond to the order in the list. To change the sequence of the geometrical objects in the list use the mouse dragging.

The machining direction for a selected curve can be reversed by clicking in the Inversion column.

Jet cutting operation assumes plasma jet, fluid jet, or laser beam as the cutting tool. The toolpath can pass either at left to the curve or at right to the curve or along the curve.

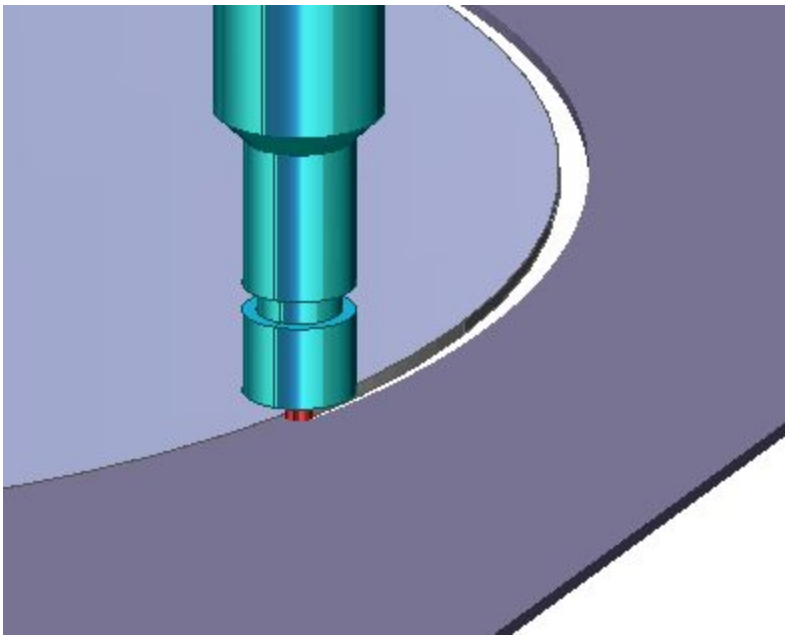


If toolpath passes Along the curve then NC-program will represent the curve itself. The tool radius compensation command is output into the program if the appropriate flag is checked. This is the simplest way of converting a custom CAD-defined curve to a CNC-code.

For toolpath passing At the left or At the right of the curve the appropriate offset curve is output into the NC-program. The offset value is specified in the Curves properties panel. If compensation is switched off then the offset amount is an arithmetic sum of the used tool radius and the operation stock value. If the operation stock is less than zero, the offset amount will be reduced by its value.

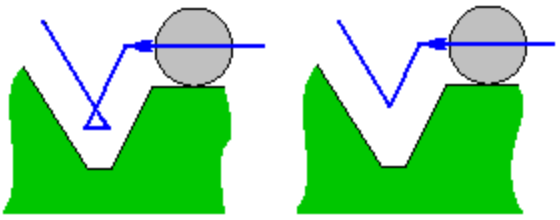
Enabling the Compensation forces the generated toolpath to contain the `G41`, `G42`, `G40` commands with appropriate correctors' numbers. A typical series of actions to generate a toolpath with turned on compensation is: check the Compensation box in the Curves properties panel, than select the side of machining (left or right) in the same dialog, than enable the tool Radius compensation on the Tool tab of the operation parameters panel and set the appropriate compensation value.

The Jet cutting operation's toolpath is the movement path of the central axis of the cutter.



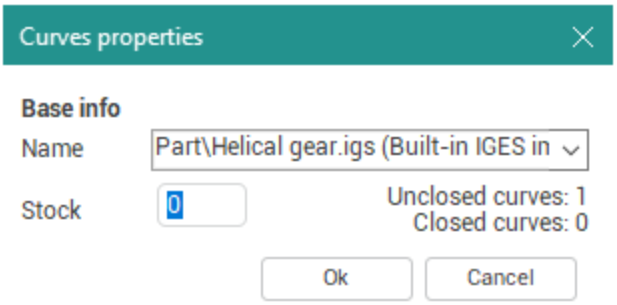
If a tool is required to return along an already machined contour then click in the With return column and the tool will travel along the selected contour, and then travel back along the contour to the original point. The picture above shows an example of machining along a contour with return. The stepover from contour to contour is performed around the workpiece at the same level as the work contours.

Modern CNC controls allow the programmer to use the actual component coordinates, and the operator enters the tool radius into the control. This allows the operator to for example, adjust for stock material for roughing/finishing operations. This can sometimes lead to situations where the offset toolpath calculated by the CNC control cannot be produced and causes error messages. An example is when using a tool that has a radius which will cause the offset contour to overlap. In the picture to the left the offset toolpath calculated by a CNC system can be seen, and to the right by a CAM system. Some older CNC's do not have the ability to calculate an offset toolpath. In these cases the task of offset toolpath calculation regarding the tool radius, the gap or the stock have to be performed in a CAM system..



The point on a curve to start machining is defined automatically by default. Where are some ways to change the start point. Open the Curve properties window by double click on item or from the popup menu.

This window contains the detailed information about item.

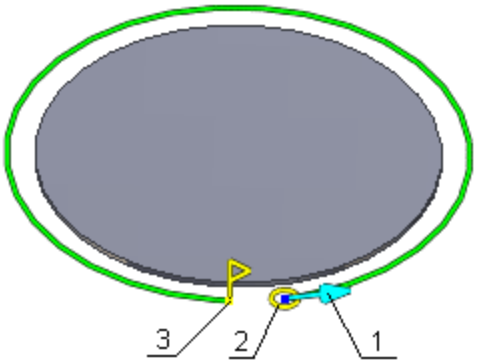


The start point can be input by coordinates or selected from the list. Along with geometry elements the list contains the following constants:

- Auto – the curve is being machined from the begin to the end;
- Curve begin – the machining will start from the curve begin;
- Curve end – the machining will stop in the end of curve;
- Custom – the point is defined by coordinates in the fields X and Y.

If the defined point does not lay on the curve then the machining will start from the point that is nearest to the defined ones.

The start point can be defined interactively by mouse from the screen. It is possible if the item contain one curve or one point only. If the machining parameters can be defined interactively then the picture below will appear near the selected item.



Where:

1. machining direction,
2. tool profile sketch,
3. end point.

The start point is shown by the tool (1) and machining direction (2).

There are two kind of the end point. – if the curve is closed. – if the curve is unclosed.

When you change start and end points positions point-snapping mode is activated and the end point mark looks like .

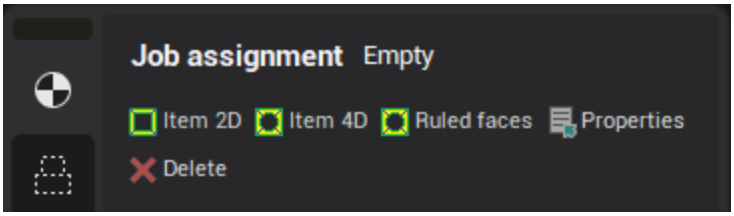
To specify the start point move the mouse pointer over the tool (2), then press and hold the left button and drag the start point to the desired position. To specify the end point use the same method on the end point sign (3).

The machining direction can be altered by clicking on the arrow (1).

Job assignment of EDM and Jet cutting 4D operations

Job assignment for the wire EDM machining operations have a list of job assignment items. These items are machining geometry and also technology parameters. Job items can be viewed in short or full form. Each item may be a single contour or a folder which contains several contours.

The short view is a list of job assignment items, you can see it below:



The operation `4D Contouring` allows to add 2D contour. The operation `2D Contouring` can not add 4D contour.

The following functions are available:

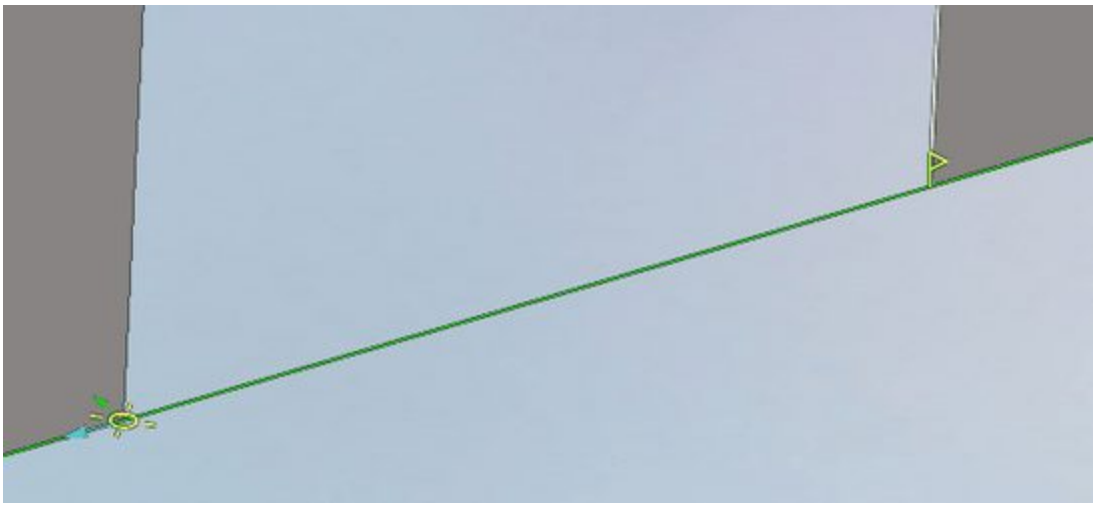
- `Wire EDM item 2D` – add selected item as 2D job assignment.
- `Wire EDM item 4D` – add selected item as 4D job assignment. One of contour will be taken as upper, the second one will be taken as lower contour. This button is available only for 4D Contouring
- Properties – opens a window with full view of the job assignment item properties. Several items can be edited at one time, just use the standard Windows keys combinations to select them.
- Delete – deletes the selected items from the list.

For call parameters window and delete items it is possible to use buttons on the pop-up panel:



It is possible to select several items with several parameters.

The contour can be closed and open. Start and end points of the contour can be changed in the graphics window:



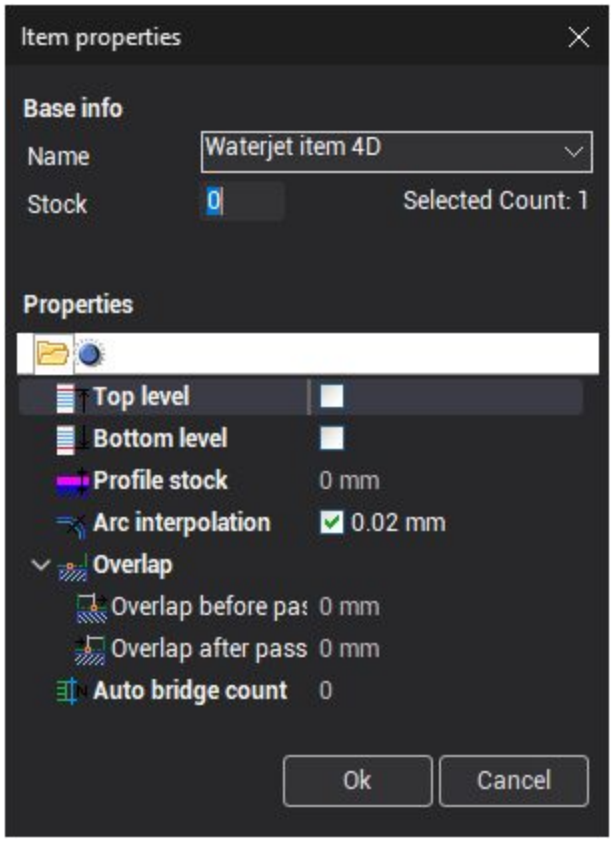
The system allows you to edit the contour directly in the graphics window. Editing principles are similar to those used in Lathe operations.

2D job assignment item properties

Each element of the job assignment has a set of properties.

To view or edit the properties of 2D job assignment item select the item and click the Properties button, or double-click the item.

This is the item properties dialog:



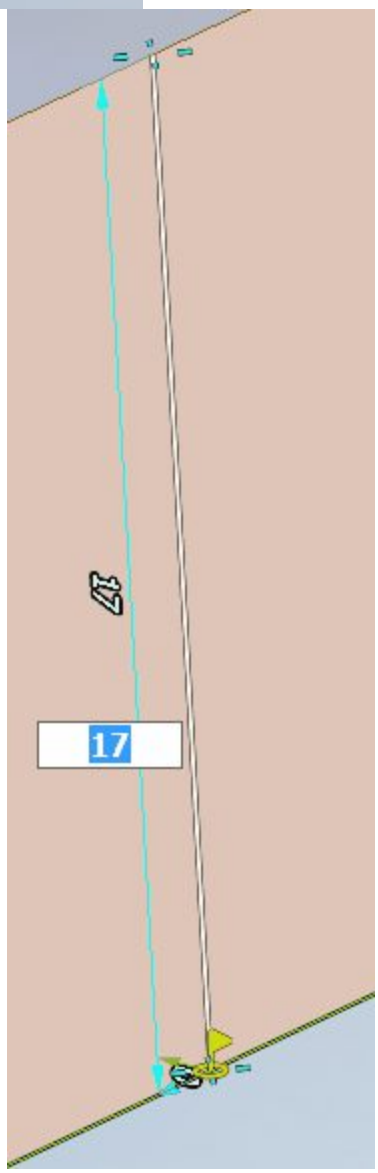
- Top level – plane for the top guide of the EDM-machine.To set the top guide level via the graphic window move the top level sign:



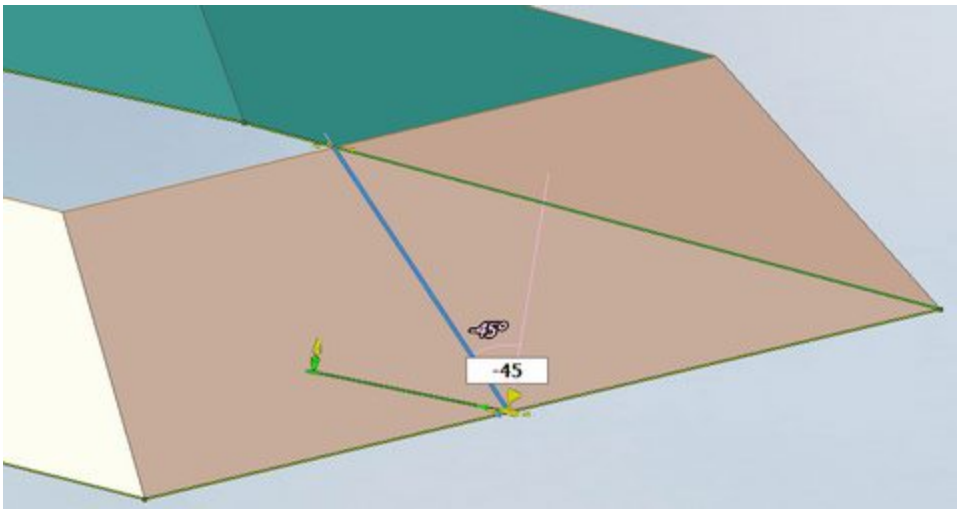
- Bottom level – plane for the bottom guide of the machine.To set the bottom guide level via the graphic window move the bottom level sign:



To set the exact level of the top or bottom guide click the sign of the level and input the value:



- Profile stock – additional stock for the resulting contour. The value of the stock can be either positive or negative.
- Overlap before pass – value of overlap at the beginning of the job assignment item.
- Overlap after pass – sets the value of overlap at the end of the job assignment item.
- Auto bridge count – number of bridges. Bridges properties can be set in the graphic window.
- Taper parameters – if turned off the wire will be positioned at normal to the XY plane and the result of the machining will be a cylindric surface. To machine a conical surface turn on the feature and set the taper angle value and direction for each contour. To set the taper angle click the synchronization line and input the value.

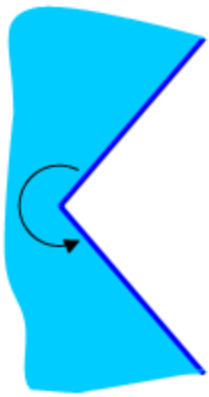
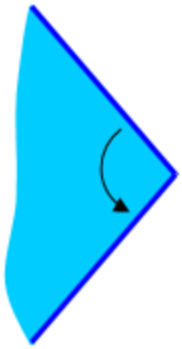
- 

When taper is turned on the taper angle value will be output into each NC frame (for example `G01 X30 Y45 A5`). Turning the taper feature on enables additional parameters.

- Taper application can be one of the following values:
- All passes – taper will be applied for all passes of the contours.
- Apply after pass – taper will be applied after the pass number set in the `Pass #` field. Taper will be disabled for the N passes, passes starting with N+1 will have taper enabled.
- Cancel after pass – taper will be canceled after the pass number set in the `Pass #` field. Taper will be enabled for N passes, passes starting with N+1 will have taper disabled.
- Corners rolling – Modern EDM NC-controllers support automatic rolling of sharp corners in the wire path. CAM system can use this feature of NC-controller. Corners rolling panel contains properties that are used to setup corners rolling feature of the NC. When the feature is enabled the output of contour coordinates into the NC-program are the same (the contour is not changed). However, in the NC-frames where the corners rolling is required additional words defining the rolling radii are output. Rolling radii can be defined separately for the top and bottom contours. For example, `G01 X95.24 Y53.09 R1.5 R5.3` – the first R word defines rolling radius for the bottom contour, the second R word defines rolling radius for the top contour 5.3.

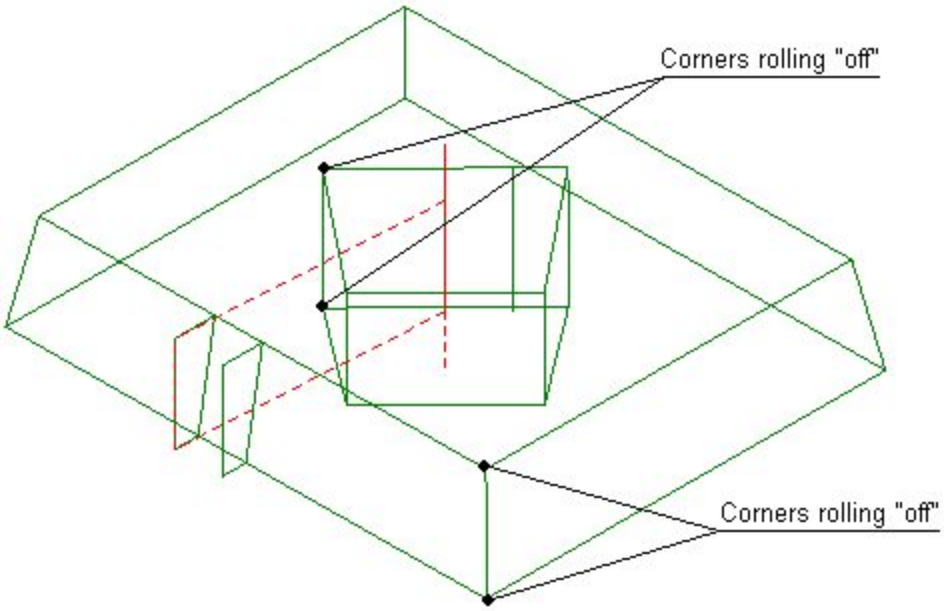
Remark: CAM system simulation supports visualization of the wire path modified by the NC-controller corners rolling.

CAM system supports various modes of corners rolling which can be set separately for inner(Inner corners rolling) and outer corners(Outer corners rolling). Whether the corner is inner or outer is determined by the angle value inside the part. Inner corner has the angle of 180° or more, outer has the angle less than 180°.

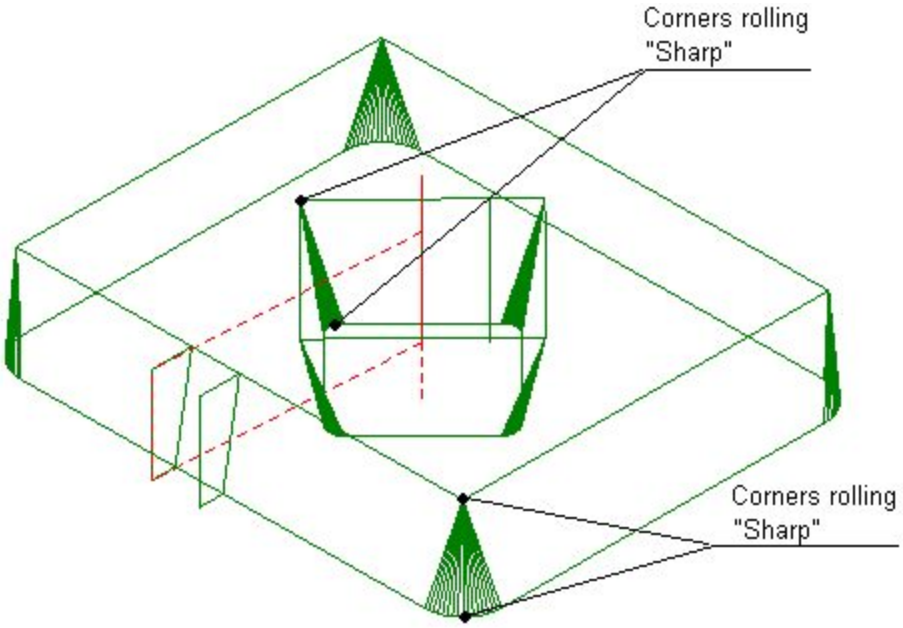
Inner corner	Outer corner
	

The following modes of rolling are supported:

- Off – in this mode corners rolling feature is disabled, rolling radii are not output into the NC-program.



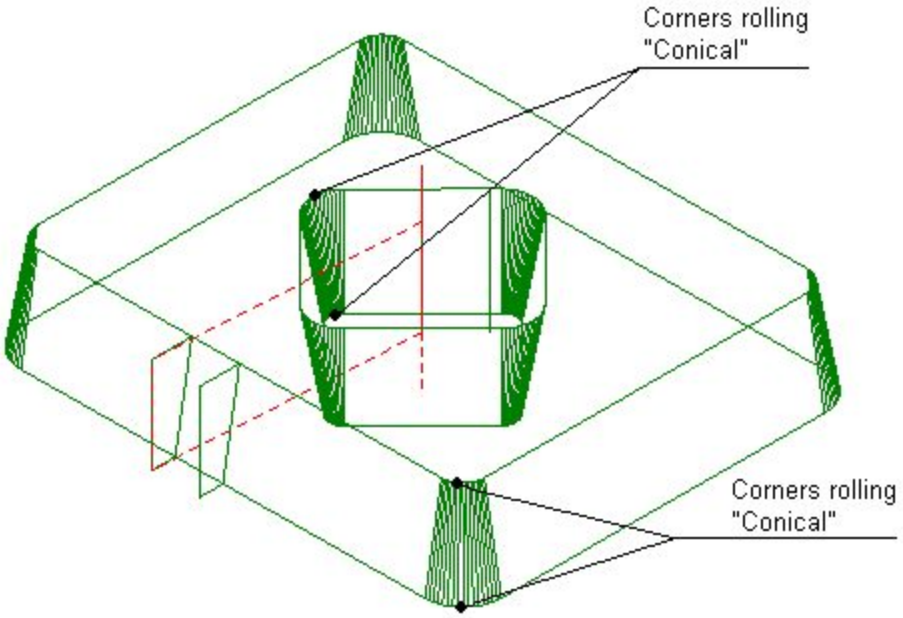
- Sharp – in this mode only the bottom contour corners are rolled, top level corners are not rolled. Therefore only the Bottom radius input field is enabled.



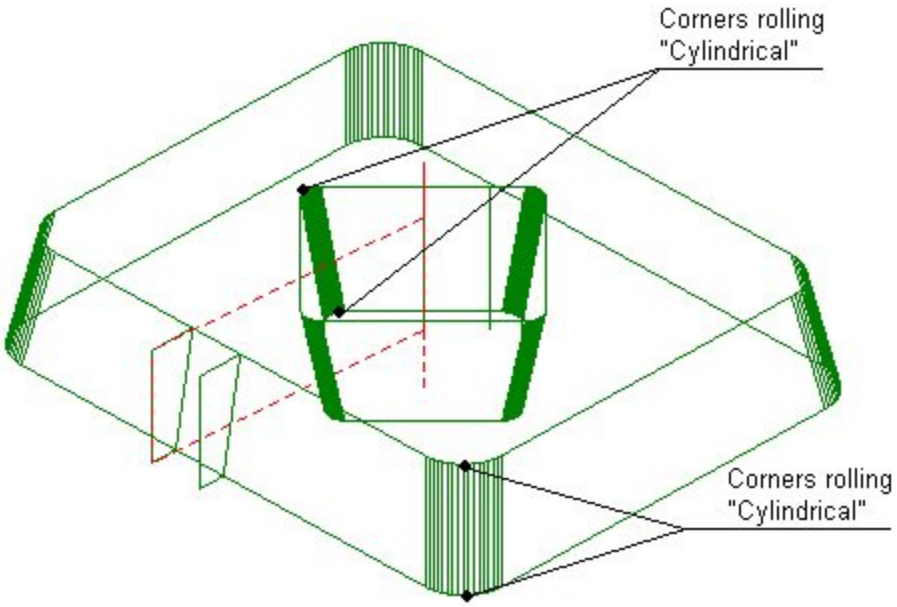
- Conical – in this mode the bottom contour corner rolling radius is set in the appropriate input field. Top contour rolling radius is defined a sum of bottom radius and a value depending on the taper angle and difference of top and bottom contour levels:

$$Rtop = Rbottom \pm h \cdot tg\ \alpha$$

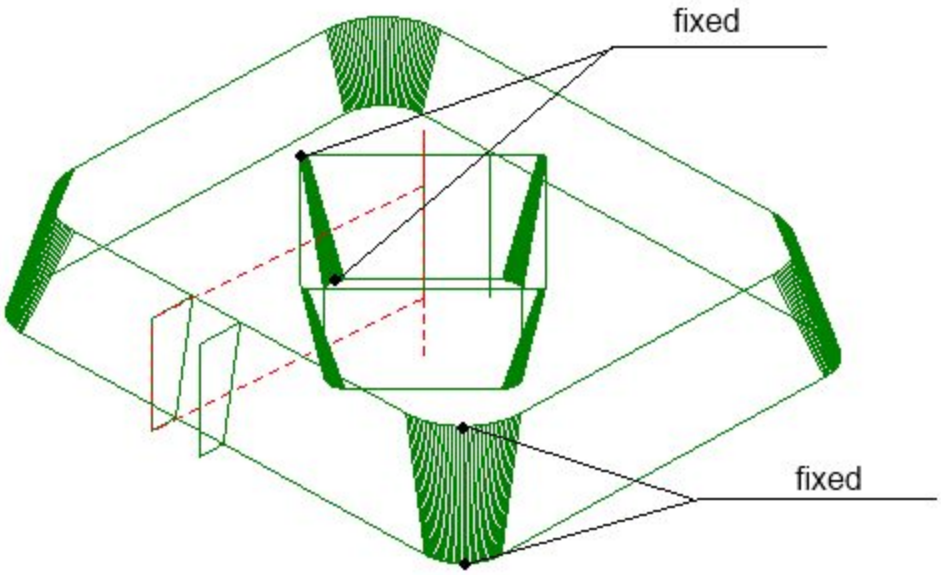
Thereby, a conical surface is machined on the part in the place of corner rolling.



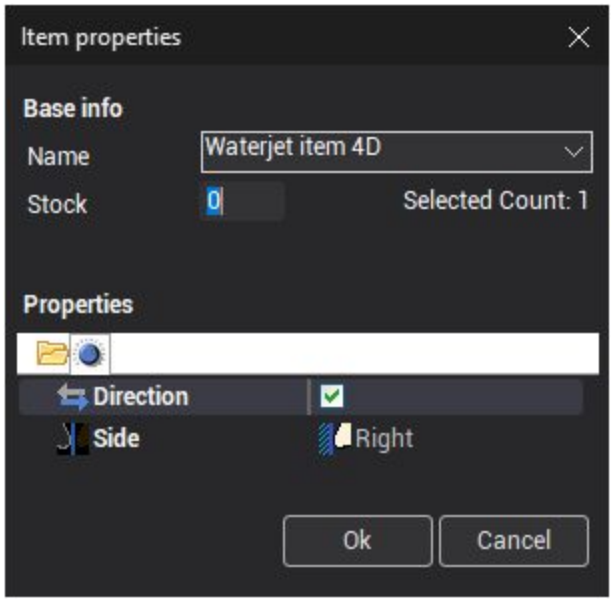
- Cylindrical – corner rolling radius is always equal for the top and bottom levels and is input in the Bottom radius field. Thereby, a cylindrical surface is machined in the place of corner rolling.



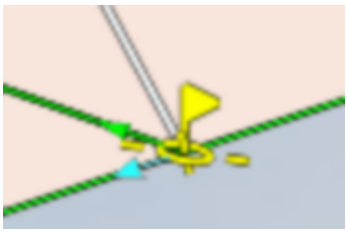
- Fixed – radii of rolling at top and bottom levels are defined independently in the respective fields and can be set to arbitrary positive values.



Use the second tab of the properties dialog to specify direction and side of the machining for each job assignment item.



In the graphic window direction is shown with a sky-blue arrow, machining side is shown with a lime arrow.



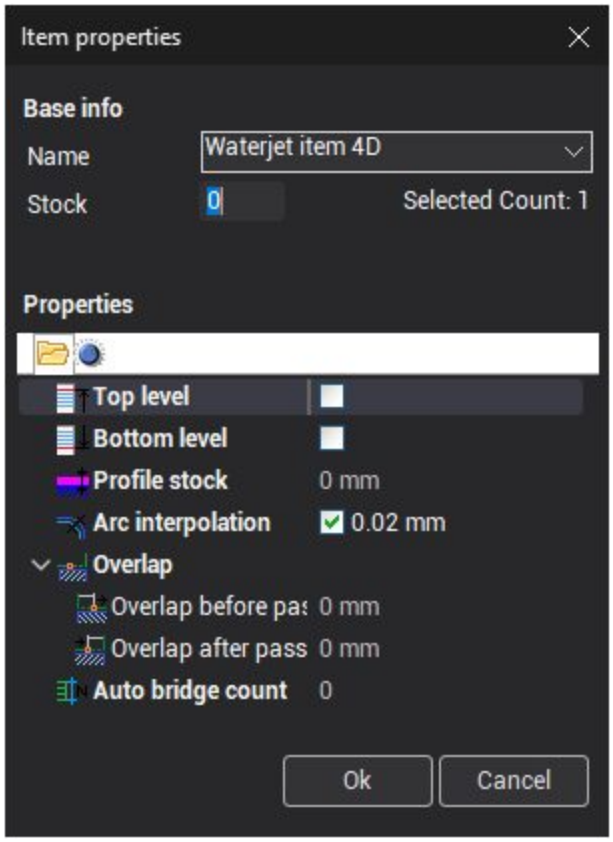
This parameters can be modified either via the properties dialog or by clicking in the arrow in the graphic window.

4D job assignment item properties

Each element of the job assignment has a set of properties.

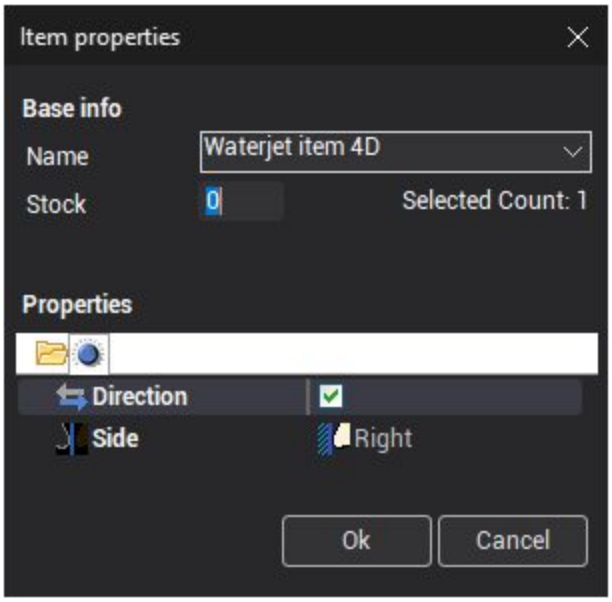
To view or edit the properties of 4D job assignment item select the item and click the Properties button, or double-click the item.

This is the item properties dialog:



All properties are the same as the 2D job assignment item properties.

Use the second tab of the dialog to set direction and side of machining for each job assignment item.



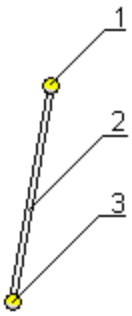
Direction and Side properties are the same as the 2D job assignment item properties.

- Swap chains – swaps the top and bottom levels.
- Inverse bottom chain – inverses the direction of the bottom level contour.
- Inverse top chain – inverses the direction of the top level contour.

Synchronization lines

In the Wire EDM 2d Contouring operation synchronization lines are shown as two points connected by a line. Synchronization lines can be used to define taper angle.

In the Wire EDM 4d Contouring operation in addition to moving synchronization lines line points positioning can be changed. To move the line position the mouse pointer at the middle of the line(2), and to move the points position the mouse at the point itself (1 or 3).



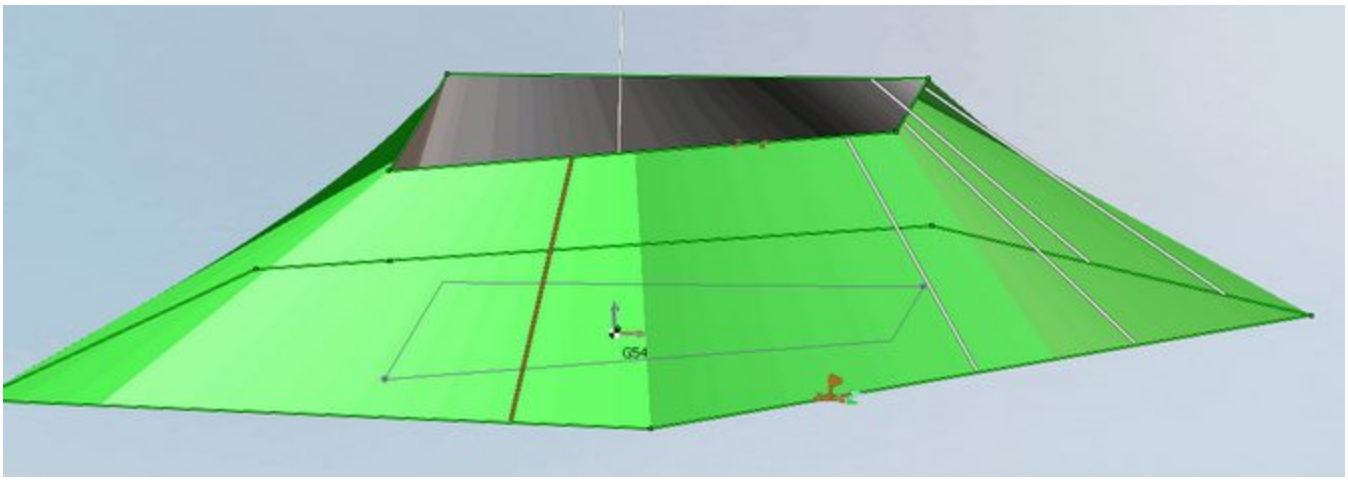
Synchronization lines can be copied and deleted in the graphic window.

To copy select the item and holding the [Ctrl] and left mouse button drag the item to desired position.

To delete an item select it by clicking it and press the [Del] button.

To create a new synchronization line position the mouse pointer over a fracture in the job assignment element and drag the created line to desired position.

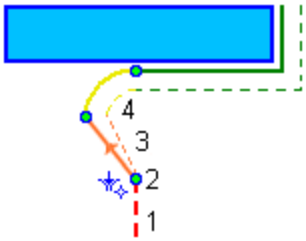



G54

Approaches/returns

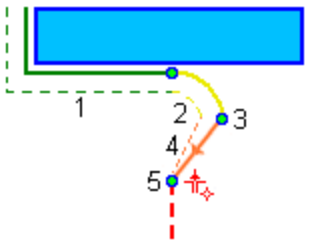
Lead-in and lead-out are the parts of the tool path, defined at the start and the end of each contour tool path. These are used for the correct machining at the start and end of a contour. These moves are used to enable various interpolation functions such as compensation, taper, multi axial interpolation, etc. To enable these features, it's needed to include one or two additional moves.

Approach to start point has these steps:



1. Approach at rapid feed to wire load point.
2. Wire loading, setup mode of cut and mode of correction and interpolation.
3. First part of lead-in – linear move from wire load point. On this step enabled modes are turned on.
4. Second part of lead-in – move to the start point of machining contour. It is necessary for composite lead-in, for example, "line and arc" lead-in.

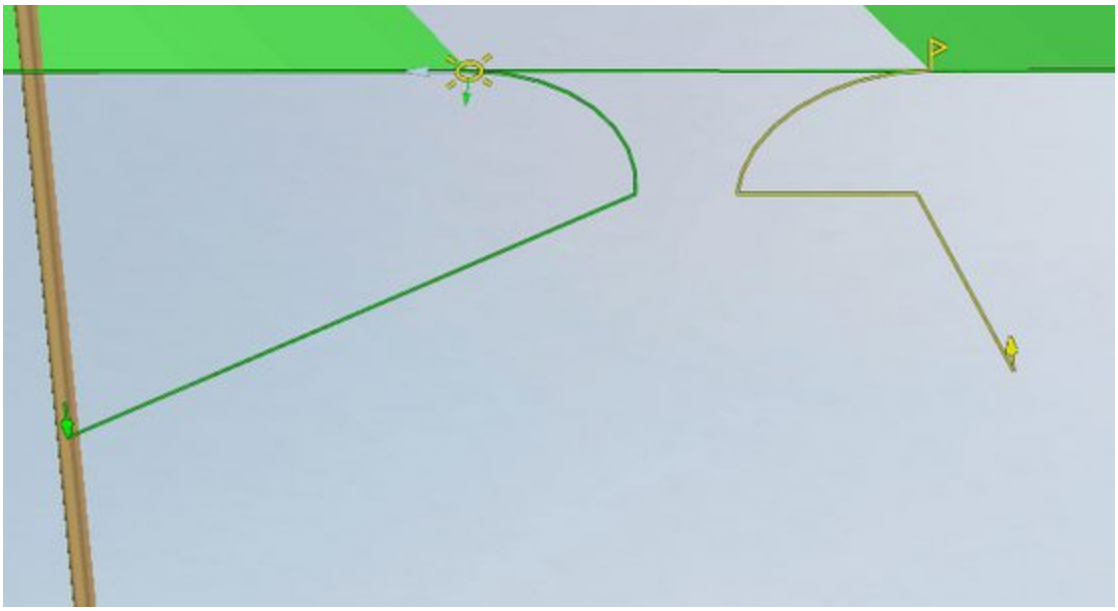
Retract from end point does the sequence in reverse:



1. Move to end point of machining contour.
2. First step of lead-out – non-linear move. It is necessary for composite lead-out, for example, for "line and arc" lead-out.
3. Turn off correction and interpolation.
4. Second step of lead-out – linear move.
5. Wire break point.

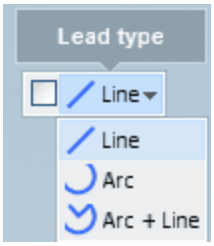
The lead-in/lead-out parameters are defined in the graphics window.

Approach\returns markers are available for each element of the job assignment and becomes available after the calculation of the operation. After changing the parameters it is necessary to recalculate the operation. The approach is green, return is yellow.



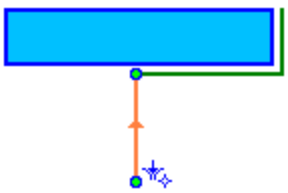
Approaches\returns markers are fully interactive. It is possible to move them and specify dimensions. Dimensions can be set as relative to other elements, as well as relative to the coordinate system.

It is possible to select lead mode in the pop-up panel:

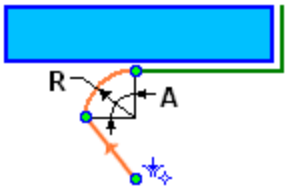


`Lead-type` – this panel is used to setup the lead type. There are several available lead types in the drop-down list:

- Line – linear motion from wire load point to start point of machining contour. The length of the linear move is determined by the position of the wire load point.



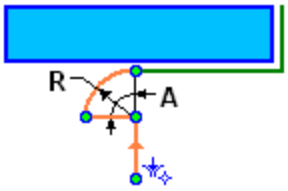
- Arc – lead-in has a linear motion from the wire load point to an arc start point. The arc move is tangent to the start of the machining contour.

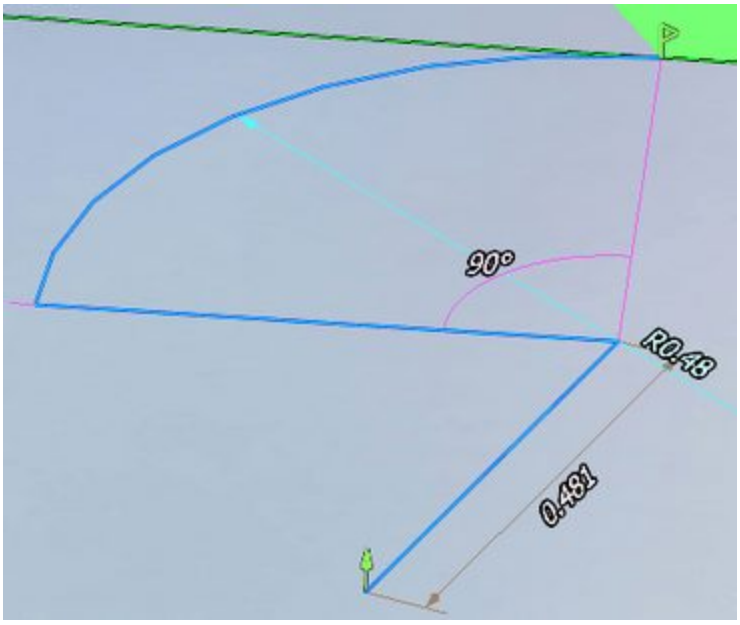


R, A - R — radius; A — approach arc angle

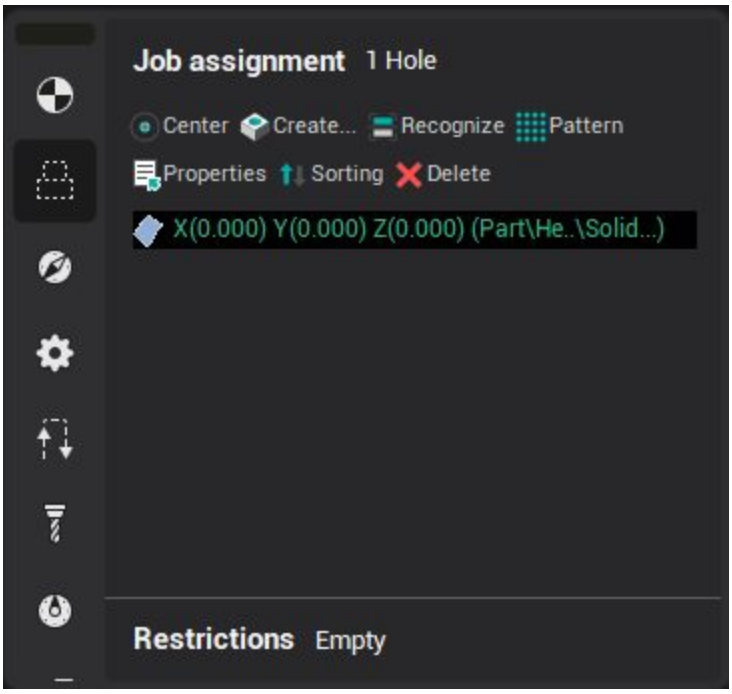


- Line and arc – the first linear motion moves from the wire load point to the arc center point, then to the arc start point. Then the arc moves tangent to the start point on the contour.





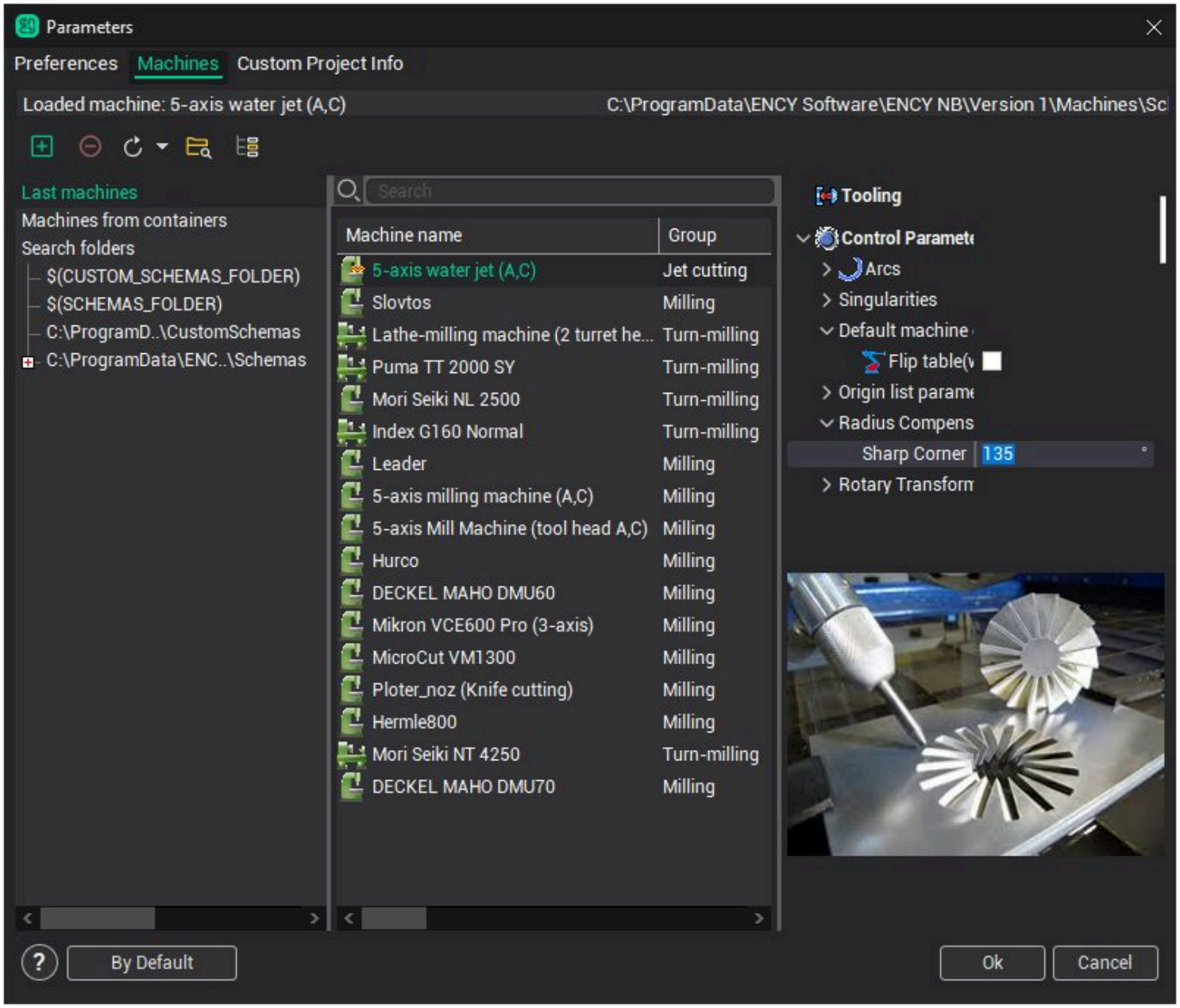
Attention: All wire load or wire break points that are used when an operation is calculated, can be viewed on the Technology panel Holes. Also, you can export this list of points to use in another CAM system project or other application. The export command is accessed from the main menu of CAM system or from the context menu of hole list `Export selected in DXF...`.



Wire radius compensation options

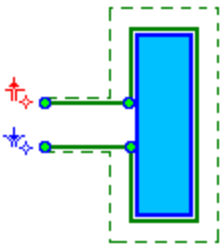
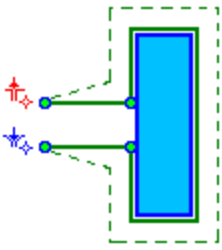
CAM system can calculate, view and simulate wire motion using compensation for the wire radius. When compensation is used, there are commands to turn compensation on and turn off included in the CLData. These are usually `G41`, `G42`, `G40` codes with a compensation number. CAM system will draw the path of the wire motion and can simulate the machining with compensation of the wire radius.

Different NC machines can use different methods for applying / canceling compensation. CAM system have several options which can be used to 'tune' CAM system's wire radius compensation so that it matches those used by the machine control. These options are available in the `Machine:... Parameters` window on the Machines tab. There is a node called Control parameters –> Radius compensation a property editor, the properties are used for tuning the CAM system simulation of radius compensation.

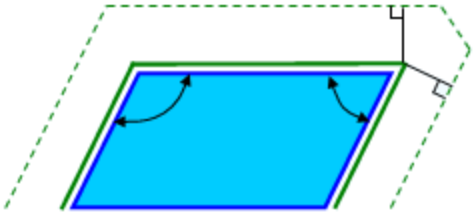


Use these properties:

- Normal approach – used for tuning the motion on approach and retract.

When **Normal approach** is on. Start and end machining point stay on normal to contour.	When **Normal approach** is off. Start and end machining points are shifted by radius compensation value.
	

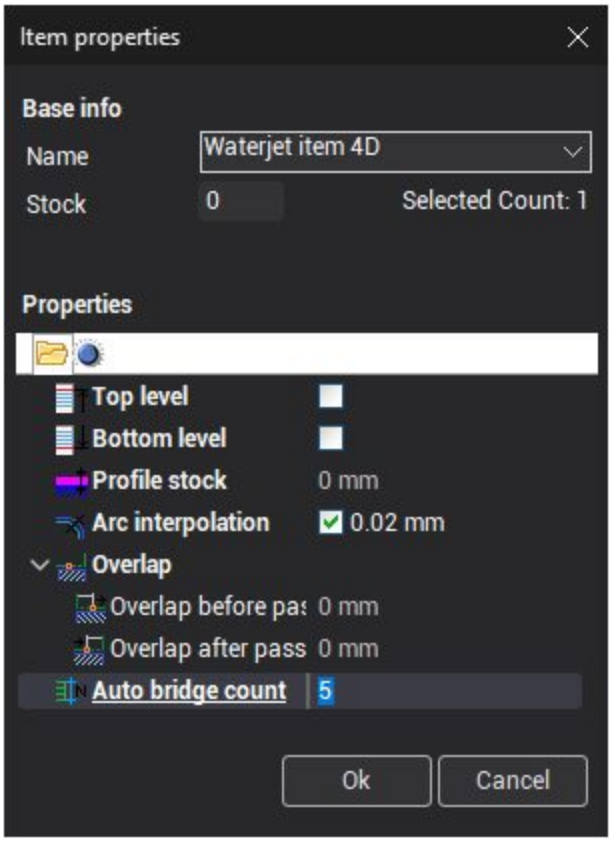
- Sharp corner – this value defines the method of rounding a corner. If the angle between the moves is greater than this value then the motion will be extended to intersect. Otherwise, if the angle is less, then each motion will extended by the value of the radius compensation and connected by a linear move. In the drawing below are shown an example where the "left" corner is greater than the sharp corner value, but the one on the "right" is less.



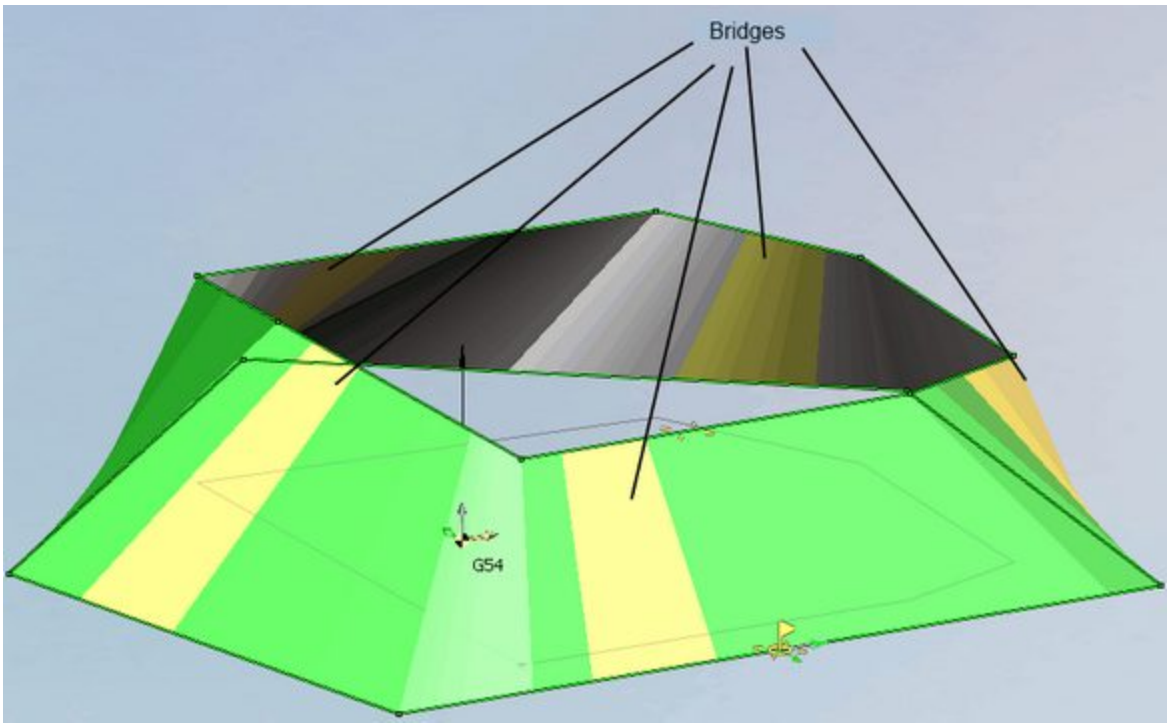
Bridges

Bridges are parts of contour that should be cut after the contour itself.

Bridges can be either disabled or set automatically. Use the Auto bridge count property to change the number of bridges.

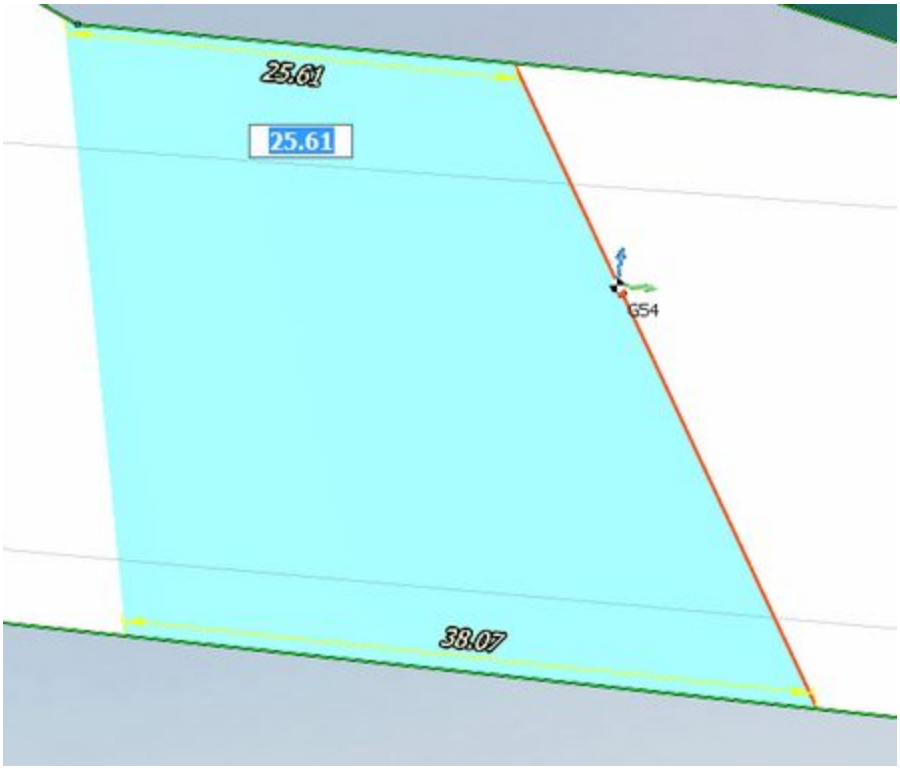


Auto bridges are placed at equal distance from each other. Change the parameters of bridges in the graphics window.



Click a bridge and drag it to a desired position.

The size of bridge can be set by the dimension line, either by dragging the edge of the bridges on the required distance. The edges of the bridge are synchronization lines.



Synchronization lines can be copied and deleted in the graphic window.

To copy select the item and holding the [Ctrl] and left mouse button drag the item to desired position.

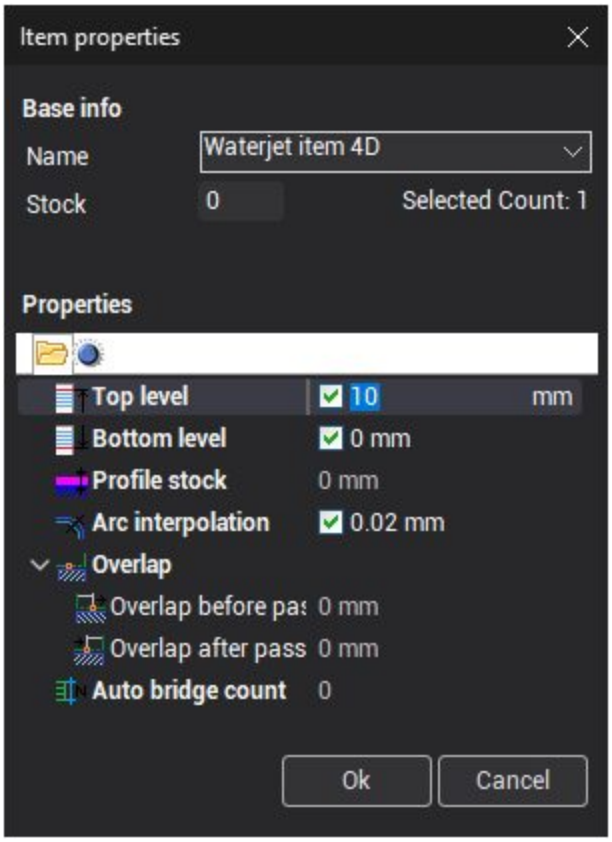
To delete an item select it by clicking it and press the [Del] button. It is also possible to use button on the pop-up panel:



It is possible to select several bridges with several parameters.

Multiselect feature

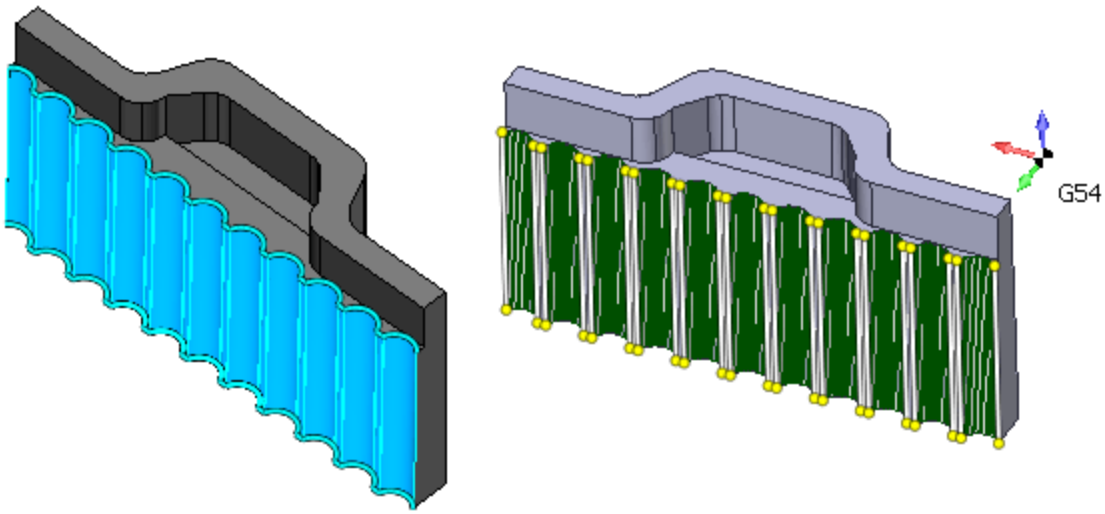
Use the floating actionbar to select items that have common attributes. Select any item and activate the activate the multiselect option.



Pop-up panel will show properties of the selected item. Use checkboxes to filter selection based on the values of properties of selected element. If a property is checked only items with equal value of that property are selected. For example to check all items that have stock of 0mm select one such item and check the Profile stock property.

Wire EDM feature

Wire EDM feature is a chain of ruled surfaces with top and bottom edges lying in horizontal planes. CAM system automatically recognizes top and bottom curves of the element, also it places synchronization lines in appropriate parts of contours.

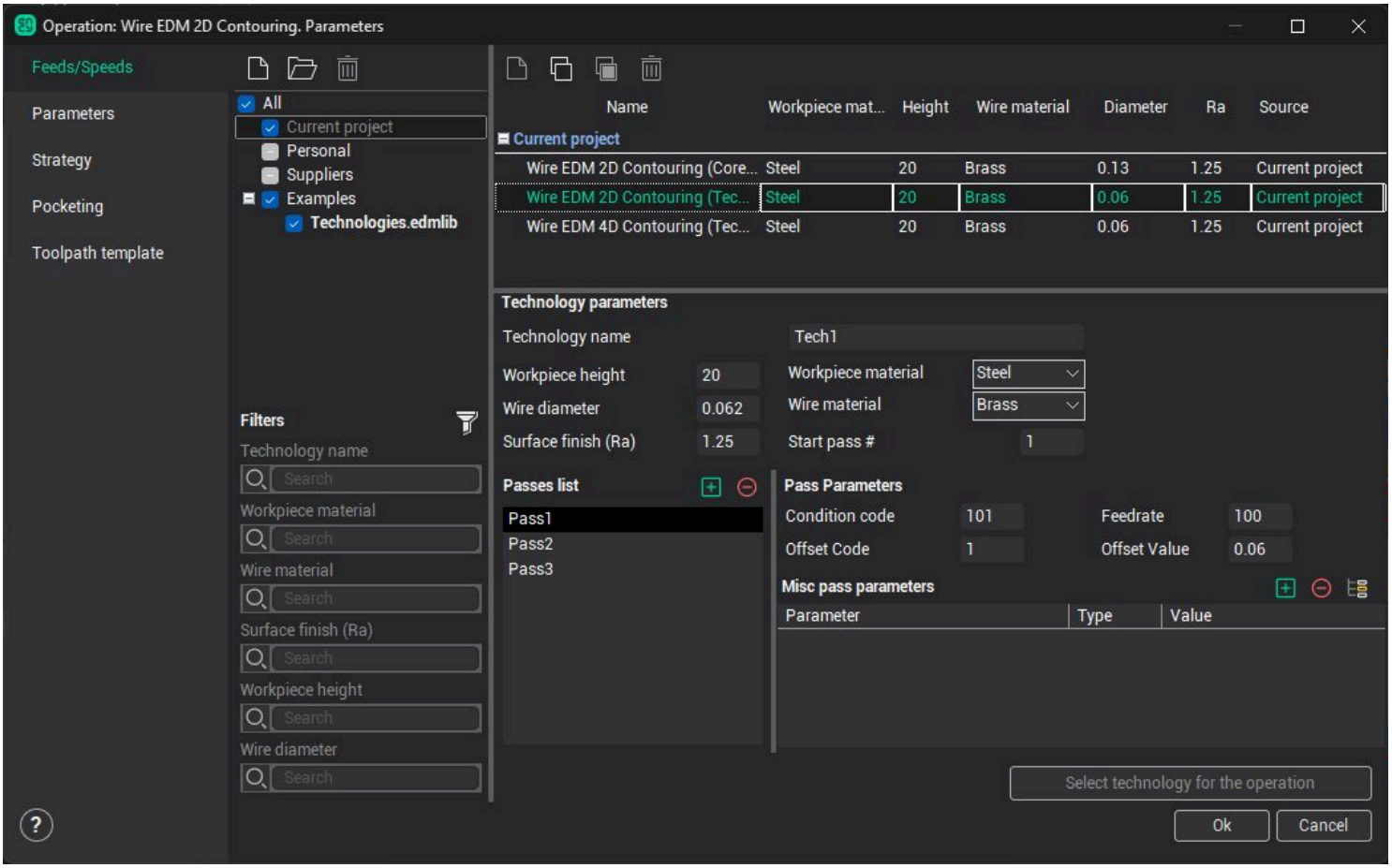


For a convenient and rapid creation of new Wire EDM operation to machine a 3D model select any of faces belonging t constructive elements you want to machine and create a new operation by selecting it in the drop-down list of the New button. CAM system auto detects constructive elements you marked, adds them to the job assignment and setups properties of the operation according to parameters of constructive elements.

To add a constructive element to existing operation select a surface belonging to the constructive element in the graphic window and press one of buttons Add cap or Add hole. CAM system will setup operation parameters according to the constructive element properties if the job assignment was empty.

Machining conditions of EDM and Jet cutting 4D operations

The Feeds/Speeds page of the operation parameters window can be used to specify machining conditions of the Wire EDM operation. It also allows you to work with condition libraries.



The left panel of the Feeds/speeds page shows the list of storages for edm libraries. The storage is either a library file (*.edmlib) or the Current project.

The storages grouped to folders. There is a checkbox next to each item. It toggles the visibility of this storage in the right panel.

The top toolbar contains the following buttons.

- New library> – creates a new library. It will shows save file dialog where you should point a path and name for new *.edmlib file. You should place your libraries outside Examples or Suppliers folder, otherwise they will be read only.
- Open library – add existing library to the list. It will show standard file open dialog where you should select one *.edmlib file. All *.edmlib files placed inside standard folder ($(COMMON_APPDATA)\Libraries\WireEDM) will be added to the list automatically.

-  Remove library – removes selected library from the list. It will not be deleted from the hard drive physically.

Present days NC-controls can to support various assignments of machining conditions, but many of them is using following algorithm. At the same time many parameters is exists and they defines specific machining conditions. Definite power characteristics (frequency, current strength, generator operating regime, etc), wire feed speed and wire offset can be assigned in subject to height and material of workpiece, diameter and material of wire and surface roughness. Usually equipment producers puts in NC-controls already defined table of process parameters or gives means to infill this table. Every set of parameters is named by definite code. Then in corresponding registers easily puts this codes when G-code is builds. NC-control compares codes with specific process parameters automatically.

A specific set of machining conditions (power, frequency, current strength, generator operating regime, offsets, etc.) associated with defined workpiece and equipment parameters (workpiece material, height, wire material and diameter) we call Machining technology.

Right top part of the window displays machining technologies list for storages selected on the left panel. Right bottom part contains parameters of selected machining technology. Each library contains the list of machining technologies. The project contains the list of operations, but each operation contains single machining technology. So both of technologies from libraries and operations can be shown in one list. The current operation in the technology list is shown in green.

To apply selected technology for the current opeartion there is as big button at the right bottom corner of the window. It will copy all content from the selected technology into the technology of current operation.



You can use buttons on the top middle toolbar of the window to manipulate technologies.

-  New technology – adds new technology to the list. It can be unuvailable if read-only storages selected in the list (Current project or library from Examples folder). Create your own library in custom folder to be possible to edit it.
-  Copy technology to clipboard – copies technology to clipboard to be possible to paste it to any other library.
-  Paste technology from clipboard - allows to paste thechnology from clipboard to selected library. The library should not be read only.
-  Delete selected technology – deletes active technology from the list.

Each machining technology contains following properties:

- Technology name – unique text identifier of process technology.
- Workpiece height – height of processed workpiece.

- Wire diameter – diameter of wire for selected technology.
- Surface finish (Ra) – roughness of surface, that will be provided by the selected technology.
- Workpiece material – material of workpiece, for which selected technology is assigned.
- Wire material – material of wire, whereby machining is does.
- Start pass # - the number of the pass from which machining should start. Default value is 1. However, cases can appears, when necessary to do machining by selected technology, but not from the very outset. For example, this necessity can arise in case of cutting separation of one part section to some operations. In that case in the strategy of first operation the number of passes is defined less then technology has. In the next operation the starting pass is defined different from 1. Closest sequence of passes with its numbers is shown on the Passes display list info panel on the Strategy tab of operation parameters window.
- Pass list / Pass Parameters – list of passes with process parameters for every pass.

The Pass term is one pass, that wire is does along the contour. You can use  buttons to add or remove passes. Following list of parameters is assigned for each pass.

- Condition code – is a value, that is specific for every machine and it is corresponding to number of register in NC-control. Its code defines process conditions. Usually registers C, E, S is used. For more information about the codes of process conditions see manual of used machine.

- Offset Code – is a number of wire offset register (number of compensation radius). For more information see manual of used machine.

- Offset Value – is a wire offset value for selected offset code. The value takes into account wire radius, overburning value and special stock for every pass. The value is send to postprocessor and can be used for initialization of registers, which is responsible for wire offset. Usually this registers is H and D. The offset value is used for compensation modeling with general stock jointly.

- Feedrate – is a rate of wire feeding. The value is measured in mm per minute or inch per minute subject to system settings. Many of wire EDM machines is not use feedrate, but the value is available if its will be needed for specific NC-control. This value is used also by system for cutting time calculation.

- Misc pass parameters – is an array of additional parameters of the pass. Every parameter is presented by line like Parameter – Type – Value. There Parameter is text description of it. Type is a type of parameter, it can to be Integer or Float. Value is a numerical value of the parameter. This parameters array and other parameters of the pass is sent to the postprocessor with PPFUN WEDMConditions(56) command and can be used for specific purposes in each specific case. Input of names and values is performed by mouse click on corresponding cell.

-  Add parameter – adds a new line to the table of additional parameters of selected pass.

-  Remove parameter – removes active line from additional parameters table.

-  – copies additional parameters table of selected pass to all passes of active technology.

It is possible to import technologies from external sources and export them outside the CAM system. To do this just select desired techologies in the list and the click Export selected technologies to XML popup menu item on right mouse button. XML is structured textual format that can be changed by any text editor or processed by any other program. After that you can import this XML to the system again using corresponding popup menu.



Machining technology library

Technology 1

- Workpiece material: steel.
- Workpiece height: 20.
- Wire material: brass.
- Wire diameter: 0.25.
- Surface finish (Ra): 1.25.

Passes

Name	Condition code	Offset code	Offset	Feedrate	Additional parameter 1	Additional parameter 2	...	Additional parameter N
Pass 1	101	1	0.75	8	0	0		0
Pass 2	102	2	0.5	10	0.5	0.3		0.7
Pass 3	103	3	0.125	3	1	0.6		1.4
...								

Technology 2

- Workpiece material: steel.
- Workpiece height: 40.
- Wire material: brass.
- Wire diameter: 0.5.
- Surface finish (Ra): 0.8.

Passes

Name	Condition code	Offset code	Offset	Feedrate	Additional parameter 1	Additional parameter 2	...	Additional parameter N
Pass 1	101	1	1.25	5	2.3	0		0
Pass 2	102	2	0.75	7	7.8	0		0
Pass 3	103	3	0.5	2	0	0.6		0
...								

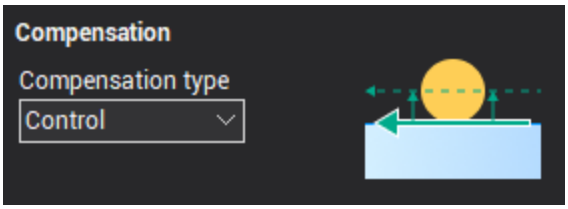
...

Strategy of EDM and Jet cutting 4D operations

Alteration of the many available parameters for the machining strategy are made in the Parameters –> Strategy window. This window is accessed by clicking the Parameters button which is located in the Machining mode. On the Strategy tab there are many panels with input fields and explanatory images. The composition of these panels are determined by the type of current operation.

Wire EDM machining operation of contours includes the following set of parameters:

- Compensation type – determines the way in which the offset of the wire is performed on a given contour.



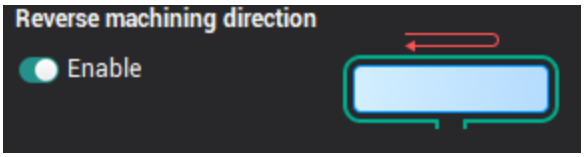
The following compensation types are available:

- Computer – the system itself calculates the corrected wire toolpath and the codes to enable compensation are not output in the G-code. In the registers responsible for compensation, the values of the offset are not added.
- Control – the system outputs into the G-code the codes to enable compensation, and does not offset the wire. The registers responsible for the value of compensation, record the values of the offset for each pass. Compensation is calculated by the CNC control.
- Both – the system outputs into the G-code the toolpath with provision for offsets already, but into G-code are outputted codes enable of compensation also. Registers that are responsible for the value of the compensation aren't filled.
- Reverse Both – correction is calculated similarly in the Both regime, but the direction of the compensation changes to the opposite.
- Off – wire offset values entered for the contour are not produced. Codes for compensation into the G-code are not output. Registers that are responsible for the amount of the compensation system are not used.

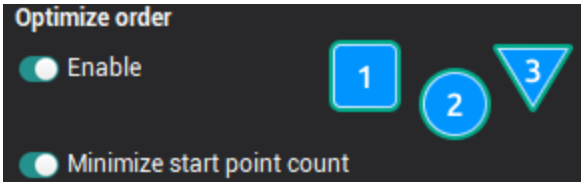
The value of compensation for each pass is defined as an Offset value, in the `Feeds/Speeds` tab, plus the stock value of the operation. For the compensation types Computer, Both and Reverse Both the value is used to construct an equidistant path, and for the Control type the value is entered into the register with a number equal to the Offset code, specified for the corresponding pass in the `Feeds/Speeds` tab.

The direction of compensation can be set for each contour individually within the Job assignment of an operation.

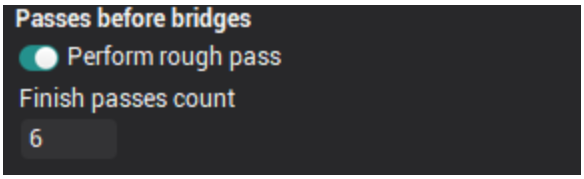
- Reverse machining direction – if you enable this option, the system will choose the direction for the pass, which provides the smallest length of the toolpath. If the option is disabled, then the direction of the pass will always correspond to that specified in the Job assignment for the contour.



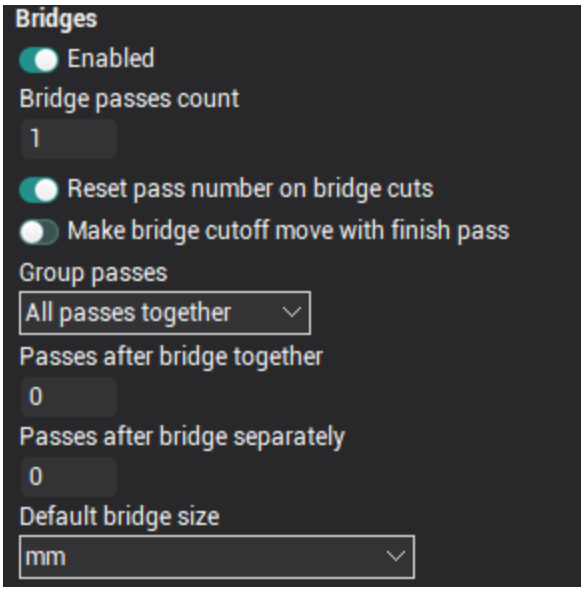
- Optimize order – this strategy determines the order of contours passes when the job assignment has more than one contour. The length of transitions between the contours will be minimal if the Optimize order option is enabled. If this option is disabled, then the order of the passes would be consistent with the order of the contours in the Job assignment.



- Passes before bridges this panel determine the number of passes that will be performed for each contour of the job assignment, to trim bridges. If the formation of the bridges is disabled, then these parameters determine the total number of passes for each contour. If the option Perform rough pass is included, then one rough pass for each contour will be executed, as well as the number of passes as defined in the Finish pass count field. When you turn this option off, rough and finish cuts to trim the bridges are not made. Approximate sequence of passes, depending on the set parameters displayed on Passes display list.



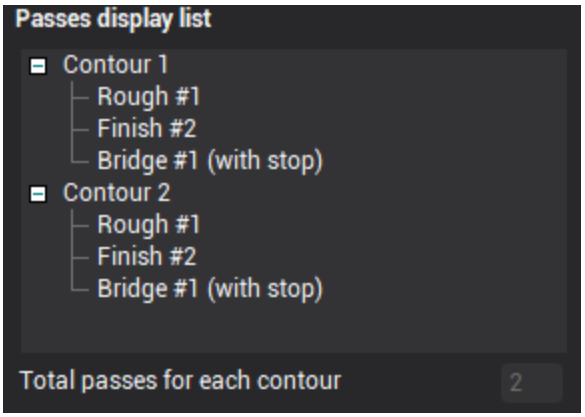
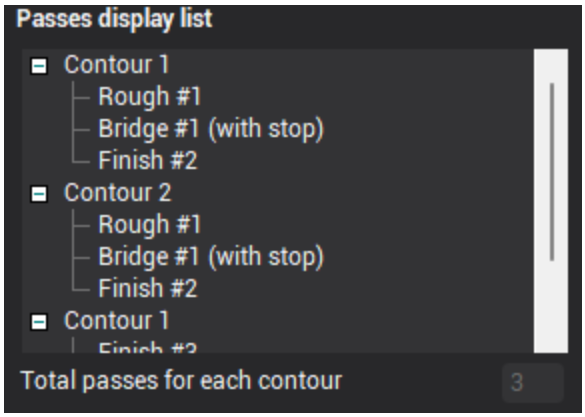
- Bridges. In some cases, for example, if the job assignment is a series of closed contours, passage of the full contour details may lead to an undesirable deposition of parts of the workpiece. The system provides a set of parameters that allow to keep the special sections without machining on the workpiece, these are called bridges. When the wire approaches such zones, the system can be add a Stop command position, to allow additional steps to fix certain parts of the workpiece, then the bridges can be automatically trimmed. Location of the bridges can be specified for each contour individually in the Job assignment section. In the Bridges section it is possible to configure the number of passes for cutting bridges, the number of passes for cleaning the contour after clipping of the bridges, and the parameters determining the sequence of these passes. If the Enabled is not selected then no bridges are cut and no clean cut after cutting bridges is available, accordingly, all fields on the panel are unavailable.



The Bridge pass count field sets the number of passes which will be performed for each bridge cutting on each contour. W hen the option Reset pass number on bridge cuts is enabled, then the count rates for the passes that define the cutting conditions for the bridge cutting moves are reset to the start value, ie from the value that is set on the `Feeds/Speeds` tab in the Start pass # field, otherwise, the count rates of the bridge cut passes will continue. For example: if the last contour pass prior to the bridge cutting was #2, the first bridge cutting pass would be #3 and the next #4 etc.

If the option `Make bridge cutoff move with finish pass` is enabled, then the bridge(s) will be cut on the final pass followed by the lead out move, then, if the bridge pass count is greater than one, the subsequent bridge cuts will be performed. If `Make bridge cutoff move with finish pass` is disabled, on the last contour pass, the lead out move will be performed leaving the bridge, and then the bridge cutting move will be preformed.

The fields Passes after bridge together and Passes after bridge separately together determine the number of finishing passes after bridge trimming that will be performed along the length of each contour as a 'clean up' pass. The difference between these two options is only affected if there is more than 1 contour feature. Example sequences for both types of final passes are shown in the pictures below:

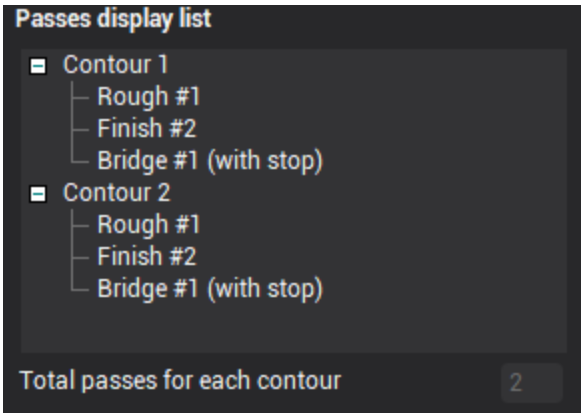
The sequence of execution of "together" passages:	The sequence of execution of "separate" passes:
	

In the drop-down Group passes menu You can choose the way of grouping different types of passes for when working with several contours.The following options are available:

- All passes together – all roughing passes, bridge cutting passes and finish passes are performed for each contour, only when completed is the next contour started.
- Bridges and finish together – all rough passes for all contours are performed, then all bridge cutting and finish passes are performed together for each contour.
- `Rough, bridges and finish separately` – first, all contours rough passes are run without bridge cutting, then the bridge cutting is performed for all contours without finishing, and finally, the finish cuts for all contours are performed.

The approximate sequence of the passes, depending on the selected parameters, are displayed in the Passes display list.

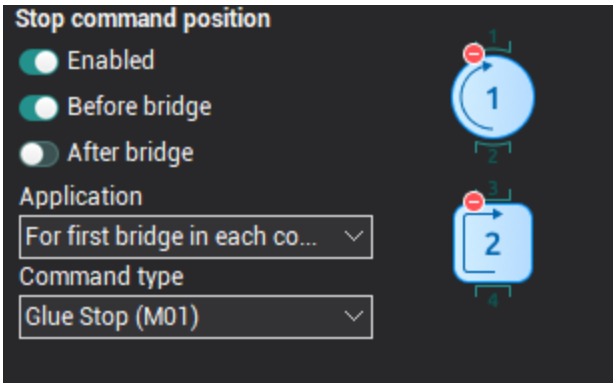
- Passes display list. On the Strategy tab for the Wire EDM operation there are many available options for contour machining that control the manner and the order of processing for the contours of the job assignment. To better understand the impact of a particular parameter on the machining sequence, there is an information panel available called Passes display list. When you change a value or parameter which influences the strategy process, this also changes the contents of this information panel. The main area of the panel takes the form of two-level tree type display. In this tree, the top-level displays the contours, and the lower level displays the types of and number of passes that defines the cutting conditions. In the bottom of the panel is a box that display the total number of passes which are made for each contour using the current settings.



Note: The information panel Passes display list only displays information and all of the fields are read-only. Alteration of the information displayed can only be made using the parameter options that are available in the main window. The information shown in the panel is approximate and may not correspond to the exact sequence of machining since its formation does not take into account the actual geometry of the contours that are in the job assignment of operations. By default, the list always contains two abstract contours.

Note: When specifying the number of passes in the strategy, these should be closely monitored so that the number of passes in the field Total passes for each contour coincides with the number of passes defined for the cutting conditions on the [Feeds/Speeds](#)tab. If there is a discrepancy in the number of passes, then the machined contours may have material remaining upon completion.

- Stop command position. The options located on this panel allow control over the output of the stop commands in the G-code for the bridge cutting passes. The stop commands are only output into the G-code when the Enabled option is selected. The next two parameters define the time of output for the stop command. The Before bridges option enables the stop command which will be output after the bridge approach move, prior to its cutting. The After bridge option enables the stop command which will be output after the cutting of the bridge but before the lead out move from the endpoint to the wire cut point. These parameters operate independently, ie they can be set simultaneously.



The drop-down list Application determines how the bridges should use the technological stops. The options include:

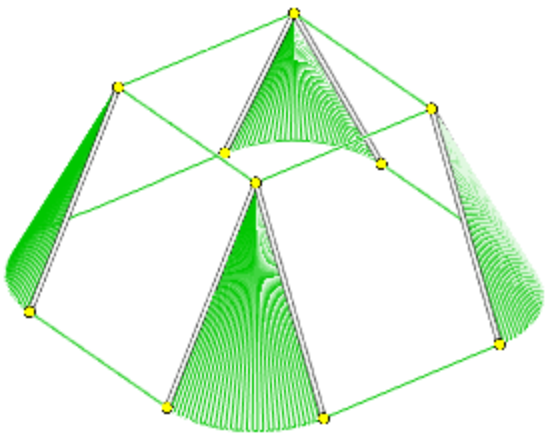
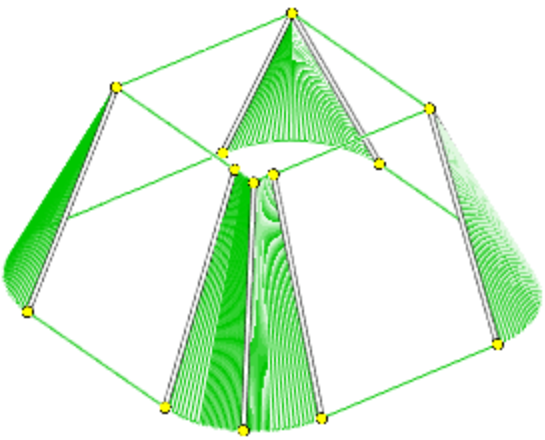
- For every bridge – stop command will be output for the bridge cutting move for every bridge specified in the operation.
- For first bridge in each contour – stop command will be output only for the first bridge of each contour.
- For first bridge in operation – stop command output only when cutting the first bridge of the operation.

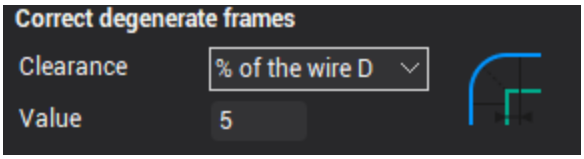
Command type this parameter defines a specific type of output stop command, and can take one of two values:

- `Glue stop (M01)` – the "optional" or "additional" stop allows, in contrast to the usual stop command (M00), when the switch on the CNC control panel is selected, it allows the operator to decide whether the process should stop. Typically, this command corresponds to the auxiliary code `M01`.

- `Stop (M00)` – this command causes an unconditional interruption of the G-code execution. Usually, it corresponds to the auxiliary code `M00`.

- Correct degenerate frames this feature is available in the Wire EDM 4D Contouring operation. Sometimes parts of a given contour can equate to a near zero or zero length. In the example below, the arcs in the left hand figure on the lower contour cause the geometry on the upper contour to degenerate to zero. Often a CNC control cannot handle such cases because they lack the geometric information required in the degenerated frame to be able to calculate for example an offset path, therefore, these cases should be avoided. This is achieved either by manually specifying correspondences in the job assignment, or, by using the correcting function for degenerate frames. In the latter case, the system automatically detects on a contour any very small lengths and "extends" them by an optional amount, as shown in the figure below right.

Contour with degenerate frames:	Contour with a corrected frame:
	



In the Correct degenerate frames section you can enable or disable the function and enter the amount on which to extend the degenerated elements. In the Clearance dropdown the following items are available:

- Off – when you select this item, the correction degenerated frames is disabled.
- Distance – this enables the function. The fixed amount of the required extension is entered into the Value field. The value relates to the currently selected units (millimeters or inches).
- `% of the wire D` – this enables the function. The amount of the required extension is entered into the Value field as a percentage of the current wire diameter. The value relates to the currently selected units (millimeters or inches).

Knife cutting

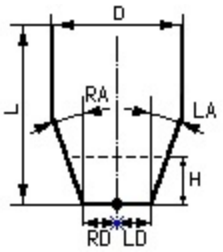
CAM system has 2 special operations for the knife cutting:

- 2D knife cutting
- 6D Knife cutting

These operations must be purchased additionally as a option. "Knife cutting 6D" can be included into the configuration that supports the continues 5axis milling. (see configurations). Both operations are available in the trial version.

The knife must be directed along the motion. It requires the additional rotary axis on the machine. if the active machine schema doesn't support this axis, then knife cutting operations give the wrong result.

Cutting tool - "knife"

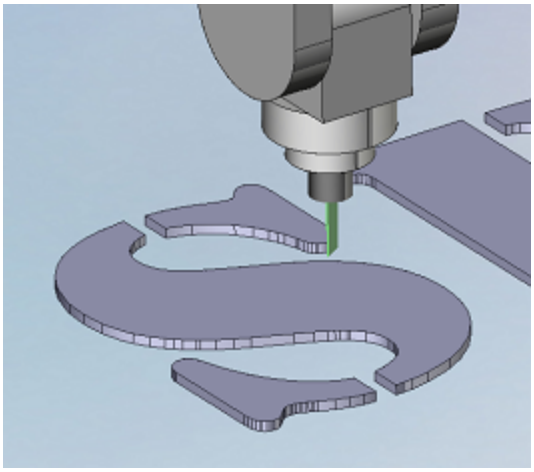


The knife dimensions are defined by 7 parameters:

RD - distance from the tool tip to the right cutting edge LD - distance from the tool tip to the left cutting edge RA - inclination angle of the right cutting edge LA - inclination angle of the left cutting edge D - blade width L - blade length H- the depth of cutting by default. This value is constant for every knife tool, so it's saved in the tools database.

Knife thickness is not used while calculation. It is calculated automatically for the visualization purposes.

2D knife cutting



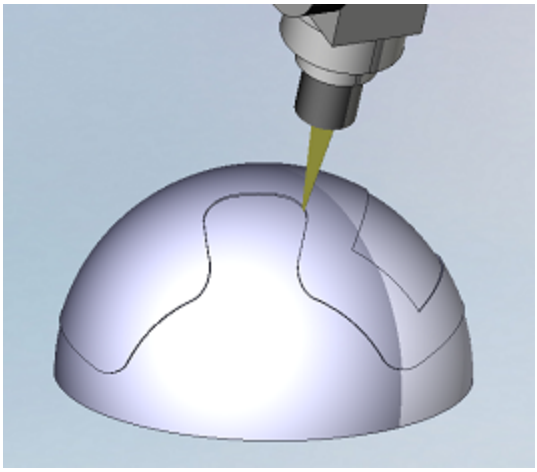
Operation "knife cutting 2D" is designed for the programming of sheet workpiece cutting. It is based on the "2D contouring".

The "way of definition of the machining profiles" and other parameters, which are not described in this chapter, is written in the description of the operation 2D contouring.

The knife usage adds the additional requirements for the machine. The machine has to have, except the Linear X,Y,Z-axes, the additional rotary axis that rotates the tool around. This axis must be defined in the machine schema. If it is absent, then the generated tool path will be incorrect.

The knife behavior in the sharp corners are defined by parameter "corner retraction".

6D Knife cutting



Operation "Knife cutting 6D" is designed for the carving on the shaped spatial surfaces. It is based on the operation "5D contouring".

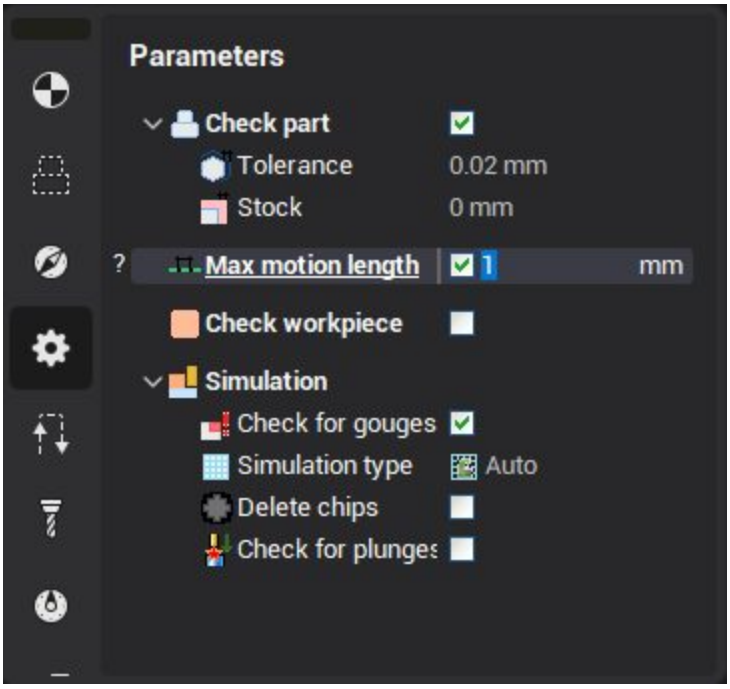
The "way of definition of the machining profiles" and other parameters, which are not described in this chapter, is written in the description of the operation 5D contouring.

In the every point of tool path the knife blade must be directed along the motion. It requires all 6 degrees of freedom. So active machine must have a minimum of three linear and three rotary axes. Very often the industrial robots are used for the knife cutting. If the machine schema doesn't support all degrees of freedom, then the generated tool path will be incorrect.

The knife behavior in the sharp corners are defined by parameter "corner retraction".

Knife corner retraction

In every point of tool path the knife blade is rotated to be directed along the motion. The maximal rotation in one cut is limited by "max motion rotation". The maximal transition is also limited by "max motion length". (see picture below)



If the motion needs more rotation then few blocks will be generated to rotate the tool in the point.

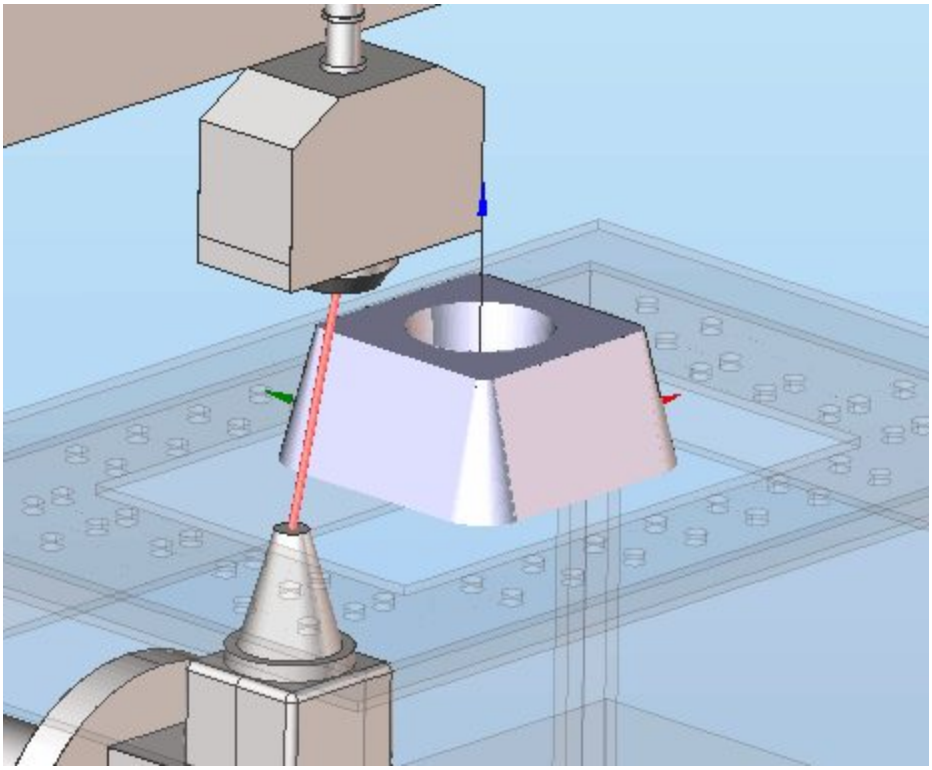
Wire EDM machining

The main principle of wire EDM cutting is the application of electrical discharges (sparks) on a part, that are generated as a result of pulsating current passing between a charged wire and the workpiece (electrodes) at a frequency ranging from 50 hertz to hundreds of kilohertz. The workpiece and wire are separated by a dielectric liquid. As result of the electrical discharges, microparticles are removed from the wire and workpiece and carried from the spark gap by the dielectrics stream. Moreover, the dielectric is a decomposition catalyst, under the high temperature of a discharge, the dielectric liquid is gasified around the spark gap and these vapour's that cannot immediately escape cause a complementary 'micro explosion'.

From a physical chemistry point of view, the high thermal temperatures destroy the crystal lattice of the workpiece, and the metal ions then lose contact with surface. When viewed, this process looks like the charged wire is eroding the metal and slowly sinks into the workpiece. The brass wire creates an opening in the workpiece which creates the required contour.

Wire EDM machining is used when traditional machining is difficult or uneconomic because of wastage or difficult to machine (hardened) metals.

CNC Wire EDM machines are used for cutting sheet metals by electro-erosion. Machines are available with 2 or 4 axes that can cut curvilinear contours with either vertical or tapered sides.



CAM system can to create NC-programs for wire EDM machines with either 2 or 4 axes:

- Wire EDM 2d Contouring,

- Wire EDM 4d Contouring.

Contouring operations are used for the wire path generation utilising a 2D contour for 2 axis contouring or simultaneously along two contours: upper and lower, for 4 axis contouring. it is possible to process several contours in one operation. The list of contours that are required for machining are selected in the Job assignment. In job assignment, the user can also specify the required machining direction. The compensation direction is calculated automatically depending on the contour type and the machining direction. For every contour it is possible to define start and stop points for machining, load and part wire points and draft angles, etc.

For the start and finish points of every contour, it is possible to create additional approach and retraction moves. Use of these additional moves helps ensure the correct machining of the start and finish points. The Approach and Retraction methods and their parameters are defined on the Lead In/Lead Out page of the operation parameters window.

To ensure a good surface finish, it is sometimes necessary to do several passes along a contour. CAM system allows various different methods for these passes: rough, finish, bridges, cutoff, etc. Enabling and disabling of these passes and defining of their additional parameters are made on the Strategy page in the operation parameters window. There are various options available for this purpose: optimised path direction, selected compensation type, etc.

Most modern Wire EDM NC-systems allow the loading into the CNC-control of a table of predetermined cutting modes for each pass along a contour. During the machining process one of these table sets is selected by including in the g-code a special instruction for that cutting mode. CAM system supports this method for cutting mode assignment in Wire EDM operations. Modification of these machining conditions tables is made in the Feeds/Speeds page on the operation parameters window. On this page it is possible to edit the machining conditions for each pass including the wire offset value and several other parameters: the height and material of the workpiece, the diameter and material of the wire, and the required surface finish. The tables of machining conditions can to be selected from machining technology libraries or defined in every operation manually. Working with machining technology libraries is undertaken in the same window on a separate tab.

Specific properties for each operation are defined on the Parameters page of the operation parameters window. As a consequence, this windows looks different for Wire EDM 2d Contouring and Wire EDM 4d Contouring.

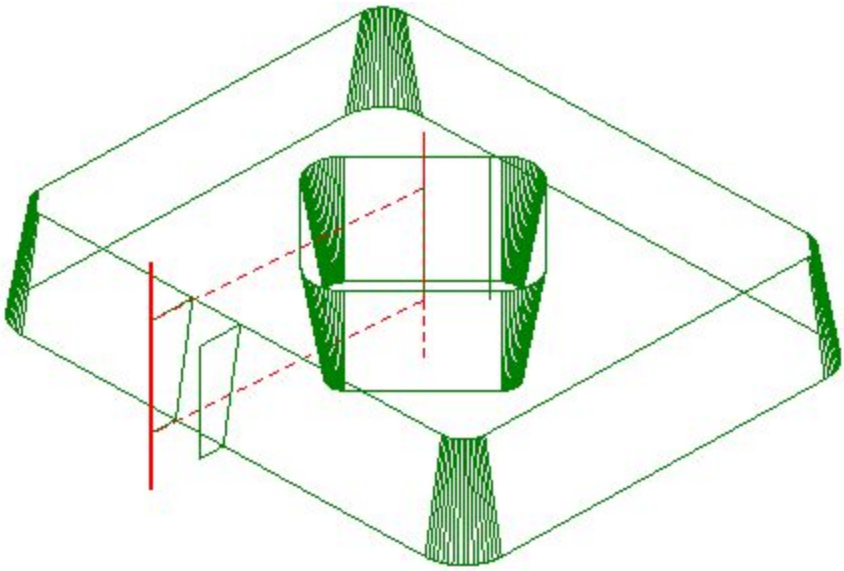
When machining the part with repeated items (holes of the same type, grooves, etc.) it is advisable to use Transformation, this reduces calculation time and decreases the time needed for NC data debugging. The function is available in the operation parameters window.

Wire EDM machining operations

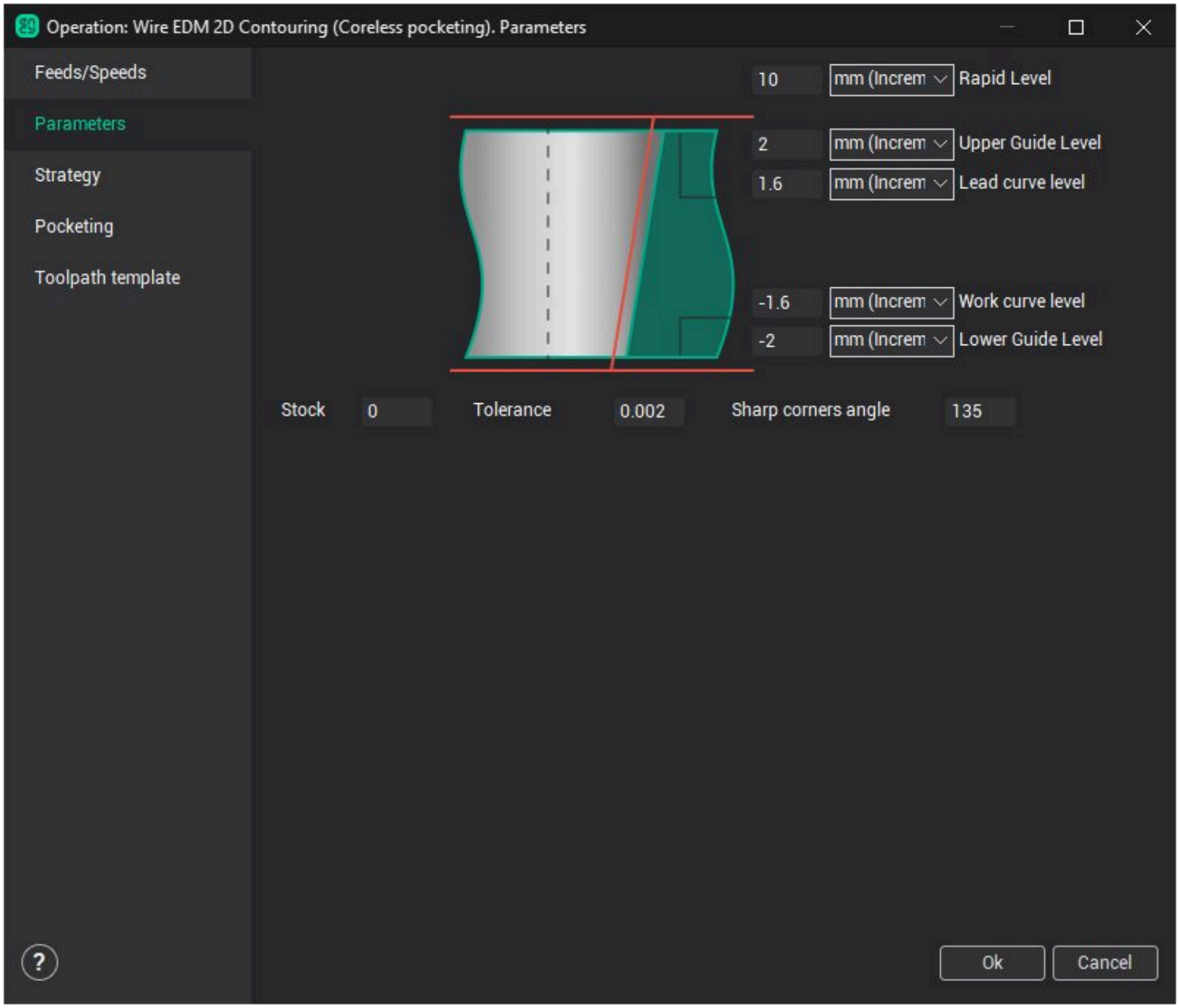
This section describes wire EDM operations: 2D and 4D contouring and wire EDM coreless pocketing.

Wire EDM 2d Contouring

The `Wire EDM 2d Contouring` operation is designed for wire path generation along flat contours as well as along flat contours with wire slope angle (taper) or 3d contouring. The resulting wire path is based on contours that lie on a single plane, unlike with the 4d contouring operation where contours must be selected for the lower plane (working contours plane XY) and the upper plane (leading contours plane UV).



Specific options for every operation are defined in the operation parameters window on the Parameters page.



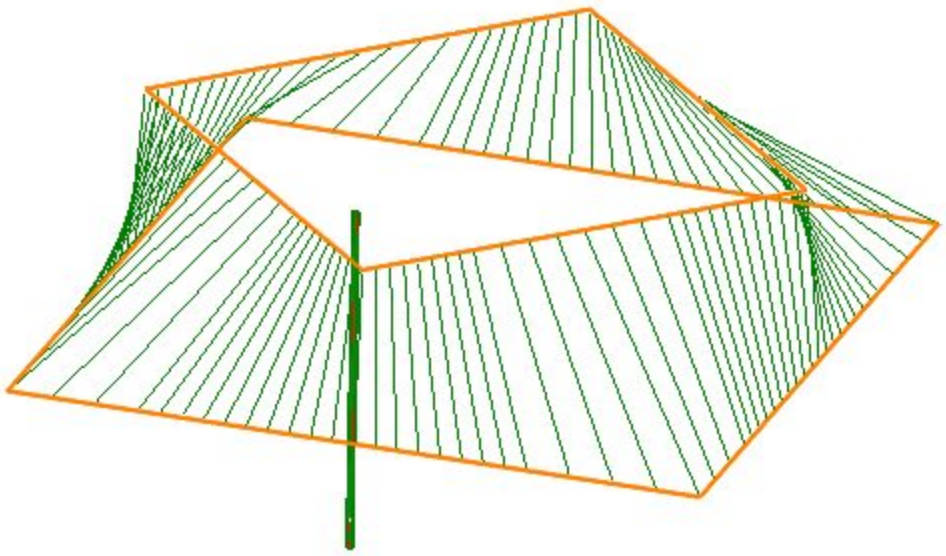
There are the following operation properties available:

- Working levels:
- Rapid level – defines the Z level where the rapid moves are performed. Its must be positioned above all working levels so that the rapid moves are performed at a height that is clear of fixtures etc.
- Upper Guide – defines the Z level for the upper guide of the wire EDM machine.
- Lead curve level – For 4d-machining this defines the Z level of the upper leading (UV) contour. For 2d-machining it defines the height at which the "virtual" upper contour will be produced.
- Work Curve level – Usually the Z level of the working (program) contour.
- Lower Guide level – defines Z level for the lower guide of the wire EDM machine

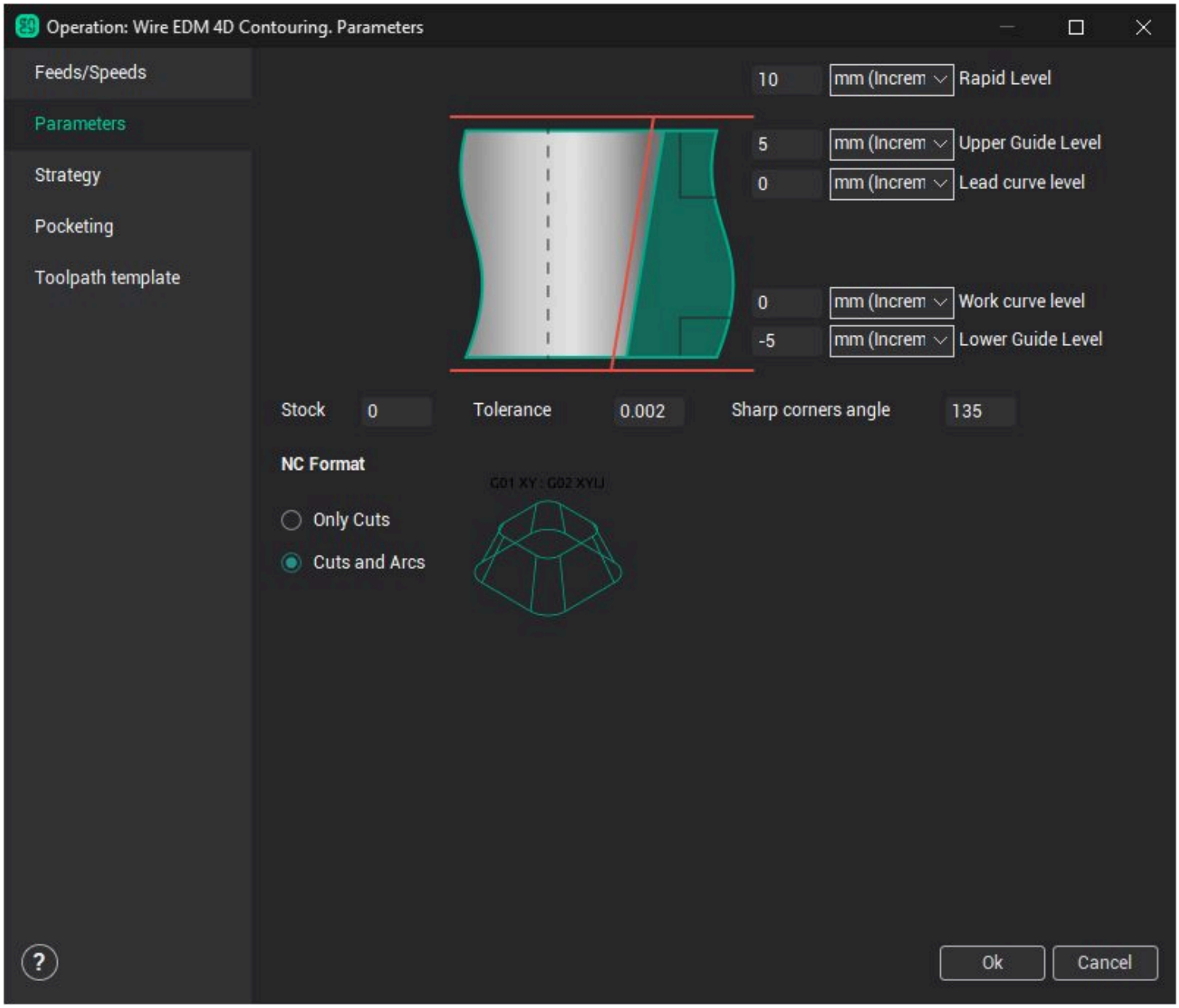
- Stock – value of the additional remaining stock that is to be used for all contours in operation. Calculation method for the value is dependant on the selected Compensation type from the Strategy page. Compensation value for each pass is the sum of Offset Value from `Feeds/Speeds` page and the Stock value. The system will create an equidistant curve based on this value in the Computer, Both and Reverse Both compensation types. In Control type the value will placed in the register with the Offset Code number from `Feeds/Speeds` page.
- Tolerance is a calculation tolerance that defines the maximum deviation of the wires approximate path from the ideal. If the tolerance is set too high, then the calculation time could be excessive, conversely, if the tolerance is set too low, then unacceptable gouges may start to appear on the part.
- Sharp corners angle permits the determination of which corners are sharp. If the modulus of an angle is greater than defined value, then it is defined as obtuse and will not be rolled.

Wire EDM 4d Contouring

The `Wire EDM 4d Contouring` operation is designed for wire path generation along two flat contours simultaneously. One of these contours moves the lower guide of the wire EDM machine, to put it more precisely – it moves the working (XY) contour plane. The second contour moves the upper guide of the wire EDM machine – the leading (UV) contour. Thus, in operation the upper and lower wire ends can move on different paths.



Specific options for every operation are defined in the operation parameters window on the Parameters page.



There is available following operation properties:

- Working levels:
- Rapid Level – defines the Z level where the rapid moves are performed. Its must be positioned above all working levels so that the rapid moves are performed at a height that is clear of fixtures etc.
- Upper Guide Level – defines the Z level for the upper guide of the wire EDM machine.
- Lead curve level – defines the Z level of the upper leading (UV) contour.
- Work curve level – Usually the Z level of the working (program) contour.
- Lower Guide Level – defines Z level for the lower guide of the wire EDM machine

- Stock – value of the additional remaining stock that is to be used for all contours in operation. Calculation method for the value is dependant on the selected Compensation type from the Strategy page. Compensation value for each pass is the sum of Offset Value from `Feeds/Speeds` page and the Stock value. The system will create an equidistant curve based on this value in the Computer, Both and Reverse Both compensation types. In Control type the value will placed in the register with the Offset Code number from `Feeds/Speeds` page.
- Tolerance is a calculation tolerance that defines the maximum deviation of the wires approximate path from the ideal. If the tolerance is set too high, then the calculation time could be excessive, conversely, if the tolerance is set too low, then unacceptable gouges may start to appear on the part.
- NC Format – on this panel can be selected one of two formats for the G-code:
- Only Lines – all arcs in the source contours will be approximated to linear segments (Lines). The G-code will contain only linear moves. NC-blocks in this case simultaneously contain coordinates of the lower contour (X, Y) and the upper contour (U, V). NC-block will look like the following way:

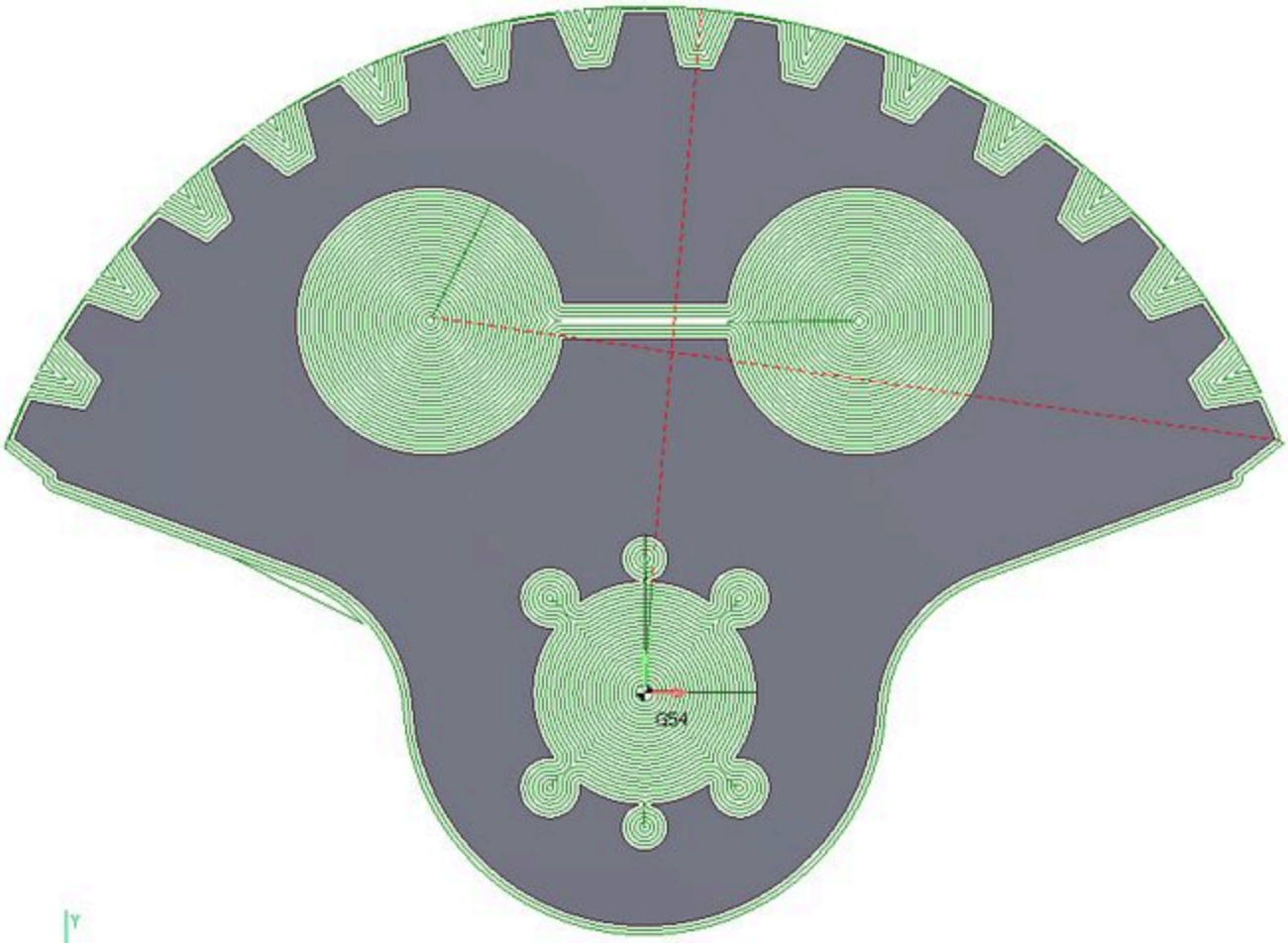
```
G01 X65.852 Y-89.422 U-3.902 V19.616
```

- Lines and Arcs – G-code can contain linear segments and arcs. Usually the NC-block in this format consist of two parts, which are separated by colon. On the left part of the block are the moves for the lower contour and on a right are the moves for the upper contour. NC-block will look like the following way:

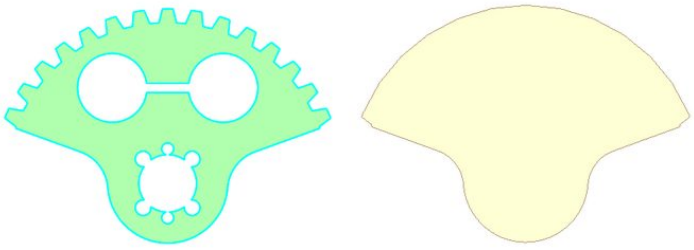
```
G03 X60. Y90. I-30. J0. : G03 X60. Y70. I-10. J0.
```

Wire EDM coreless pocketing

Wire EDM coreless pocketing is designed to remove material from areas without falling pieces of metal. The wire path usually begins in the pre-machined holes are located closer to the center of the workpiece. It consists of equidistant or zigzag passes, subsequently removed all of the material inside the pockets.



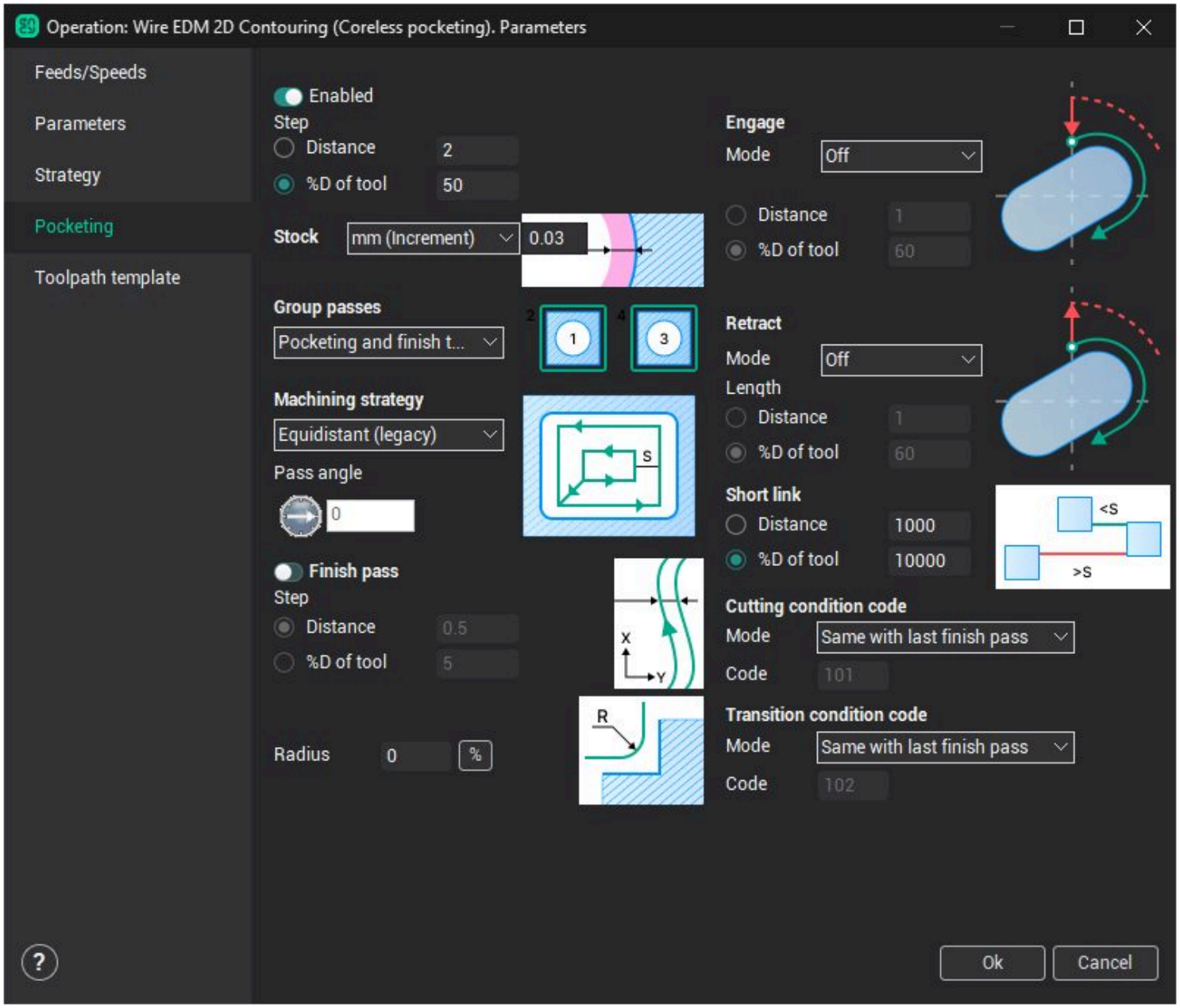
When using a pocketing you must specify in the job assignment closed contours only. This contours can be nested in each other and describe the islands as well as holes (pockets). In addition to the job assignment items workpiece must be specified too for the correct pocketing wire path calculation.

	
Job assignment	Workpiece

Wire EDM pocketing is not a separate operation of CAM system. It is made as an additional option, which is included in existing 2D and 4D EDM contouring operations. Thus, the wire EDM pocketing performs

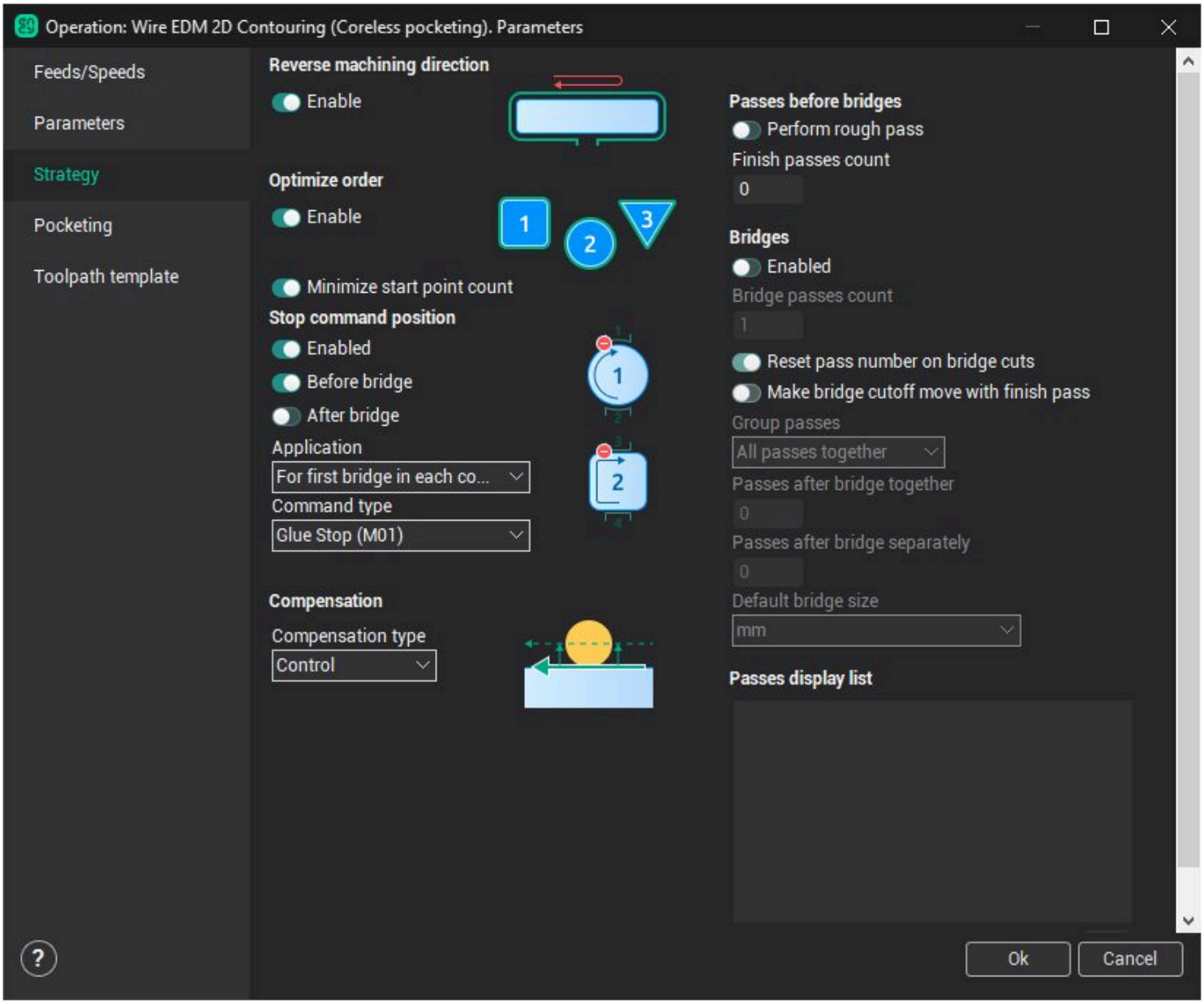
removal of the rough material, and then passes along the contours perform finish machining of the part using special cutting conditions.

To enable a pocketing set the appropriate checkbox in the wire EDM operation parameters window on the Pocketing page.



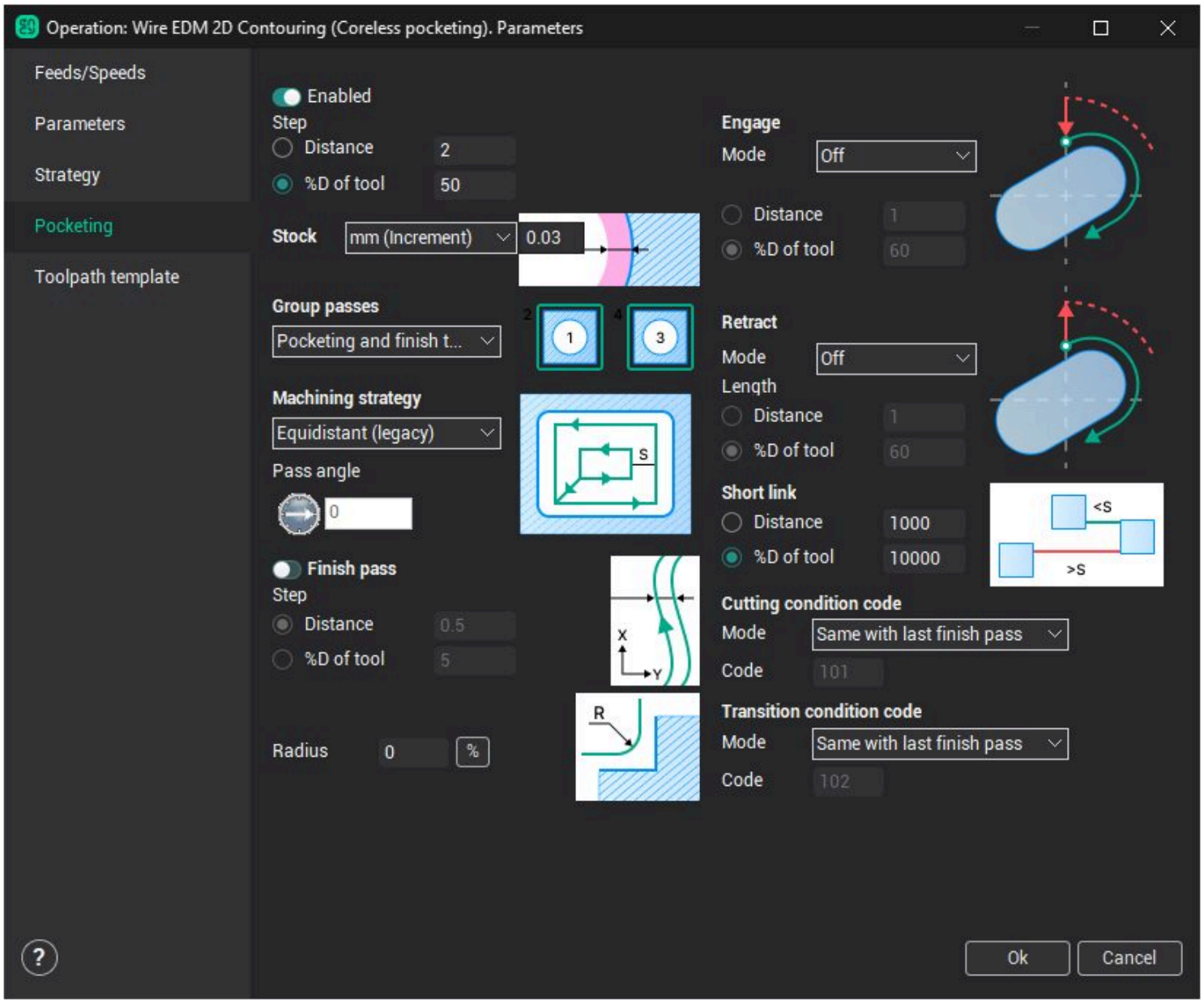
When this checkbox is enabled the parameters editing panels become available (such as machining strategy and step, ways to group passes, etc.). Many of these parameters are similar to the corresponding parameters in milling operations.

If you need to perform only rough material pocketing without performing finish passes along the contour, then you should turn off the checkbox "Perform rough passes" on the "Strategy" page of the operation parameters window.


Operation: Wire EDM 2D Contouring (Coreless pocketing). Parameters
Feeds/Speeds
Parameters
Strategy
Pocketing
Toolpath template
Reverse machining direction
Enable
Optimize order
Enable
1
2
3
Minimize start point count
Stop command position
Enabled
Before bridge
After bridge
Application
For first bridge in each co...
Command type
Glue Stop (M01)
Compensation
Compensation type
Control
Passes before bridges
Perform rough pass
Finish passes count
0
Bridges
Enabled
Bridge passes count
1
Reset pass number on bridge cuts
Make bridge cutoff move with finish pass
Group passes
All passes together
Passes after bridge together
0
Passes after bridge separately
0
Default bridge size
mm
Passes display list
Ok
Cancel

Pocketing parameters

To enable a pocketing set the appropriate checkbox in the wire EDM operation parameters window on the Pocketing page.



This stock is relative to the

Machining strategy allows you to define the shape of the wire path. When choosing the "equidistant" strategy the shape of the working passes will repeat the form of the original contours. If "parallel" strategy selected, then the working passes will be parallel lines. In the latter case, you can optionally specify the angle of these parallel lines in the plan.

Machining step determines the distance between two adjacent work passes. For convenience, the step value can be defined in several ways.

- Distance. The value is given an absolute value and does not change when you change other settings.

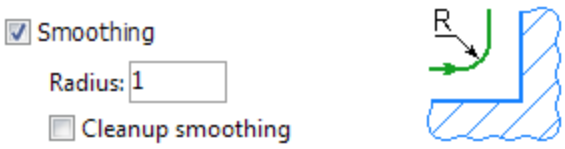
- %D of wire. The actual step size is specified percentage of the wire diameter and, therefore, varies by changing the wire.

If you enable Finish pass a little additional stock will be left on the part. It will be removed by the last finishing pass, which always runs along the initial contour (at equidistant). This allows, for example, to remove the scallops after parallel roughing passes (if you specify a parallel machining strategy) and get higher quality of finished surface. This stock is relative to the base stock for finishing.

The Stock parameter allows you to specify the value of the material, which should be left to machine with finishing passes along the contour. Stock can also provide the necessary reserve to compensate the difference between the wire diameter and the size of the burned material. It can be defined in several ways.

- Absolute. In this case, the stock specified in the current units of the system (millimeters or inches) and is relative to the source contour specified in the job assignment.
- Increment. When using the relative method of defining the stock, it is also specified in current units (millimeters or inches), however it is measured based on the number of finishing passes along the contour and the offset given to them. In addition the radius of the wire is added.
- %D of wire. Stock, as in the Absolute case, is relative to the source contour specified in the job assignment. However, its value is given as a percentage of the wire diameter.

The Smooth radius provides a rounding of the wire path when machining internal corners. This reduces vibration and increases machining speed. The "Radius" specifies the radius of the circular arcs that are added to the trajectory in the corners. Separately, you can point to the need to perform smoothing on the last pass.



If the operation includes both the rough passes (pocketing) and finishing passes along the contour, you can additionally specify in what order should perform the machining. Group passes parameter is designed for this. If you select "Pocketing and finish together" moving on to the next contour will not be made until the fully machined this contour. If you select "Pocketing and finish separately", the first rough pocketing of all pockets will be made, and only then finishing passes along the contours will be performed.

The Short link parameter defines a method for constructing the transition when connecting work passes. When the connecting points of the wire path are located at the distance greater than specified value, then the transition is performed on the rapid feed with the breaking of wire. Otherwise, the transition is performed on the shortest distance without breaking the wire at the special specified transition feed (transitions cutting condition code).

Machining conditions for wire EDM pocketing can be set separately for working passes and transitions. As with contouring cutting conditions are determined by a special code of machining conditions. According to it, the CNC control of the machine determines specific cutting parameters from the table. There are two ways to specify the cutting condition code.

- Condition code. In this case, given a specific number of code.
- Same with the first finish pass. When selecting this option condition code is automatically taken from the cutting conditions table for finishing passes along contours and will be same with the code of the first finishing pass.

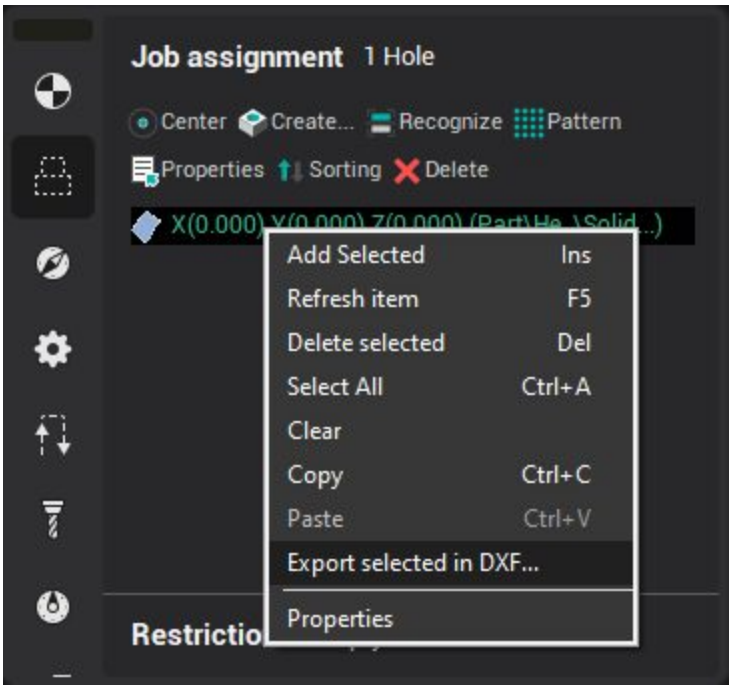
Approach and Return options allow you to specify the mode and the value of the approach to the beginning of the pass, or return from the end of the pass for wire EDM pocketing separately. In the Mode field you can select the method to construct it.

- Off. Approach and return are not built.
- Normal. Constructed by normal to the working pass.
- Tangent. Constructed by tangent to the working pass.

If the approach / return is enabled then its size can be set either in the current system units (millimeters or inches) or a percentage of the wire diameter.

Wire thread points

In the wire EDM machining operations trajectory typically begins from the pre-machined holes in the workpiece. If the holes already exist in the model of workpiece specified for operation, then the location for the entry of the wire will be determined automatically in places where there is no material. If the workpiece specified simply as solid piece of material without holes, it is possible to explicitly specify the holes through which should thread the wire. To do this you need to select desired wire EDM operation in the "Technology" mode and then select the "Holes" tab in the lower left corner of main window.



The holes can be added to this list by manually entering the coordinates, by selecting of geometric primitives, or automatically when recognition of 3D model. Working with this window is no different from filling the list of holes for the Hole machining operation.

In addition, wire EDM operations support the mode of reverse engineering. If the operation cannot find suitable hole in this list at the calculation time, it automatically creates a new hole for drilling and adds it to the list. Then the holes from this list can be exported to an external file (DXF). Later this file with holes can be loaded into another project or another application, in which is performed the design of workpiece for machine this part.

To export a list of holes you can use the pop-up menu of this window or `File / Export / Export holes` main menu item.

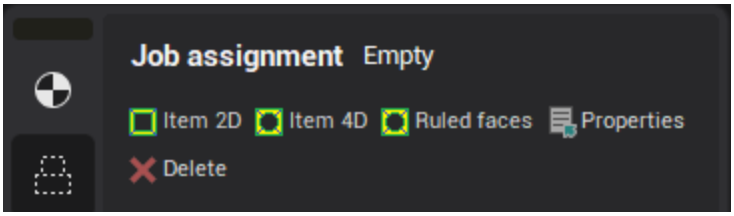
Operations setup

This section describes operations setup for EDM and Jet cutting 4D operations, including job assignment and the properties of its 2D and 4D items and approaches/returns.

Job assignment of EDM and Jet cutting 4D operations

Job assignment for the wire EDM machining operations have a list of job assignment items. These items are machining geometry and also technology parameters. Job items can be viewed in short or full form. Each item may be a single contour or a folder which contains several contours.

The short view is a list of job assignment items, you can see it below:



The operation `4D Contouring` allows to add 2D contour. The operation `2D Contouring` can not add 4D contour.

The following functions are available:

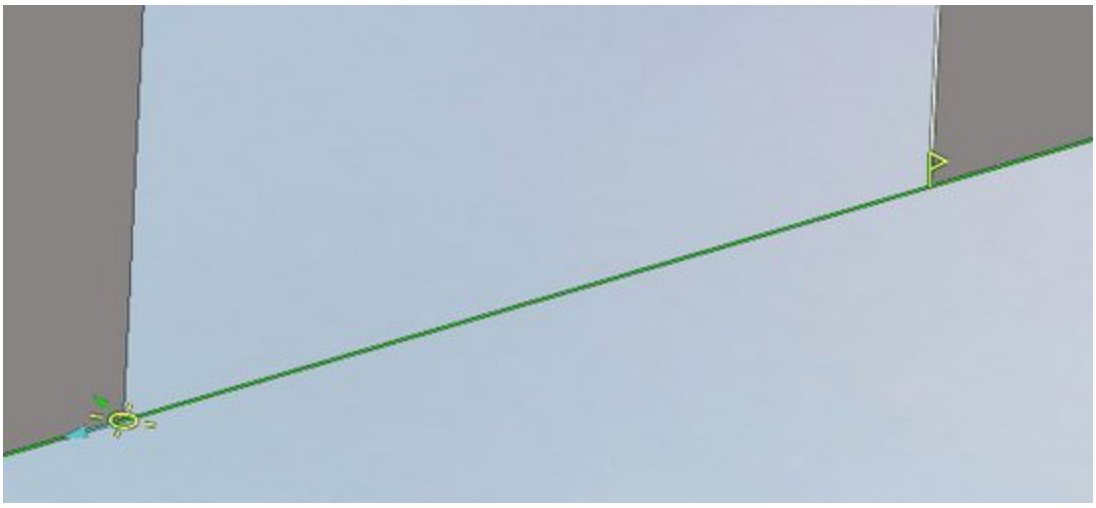
- `Wire EDM item 2D` – add selected item as 2D job assignment.
- `Wire EDM item 4D` – add selected item as 4D job assignment. One of contour will be taken as upper, the second one will be taken as lower contour. This button is available only for 4D Contouring
- Properties – opens a window with full view of the job assignment item properties. Several items can be edited at one time, just use the standard Windows keys combinations to select them.
- Delete – deletes the selected items from the list.

For call parameters window and delete items it is possible to use buttons on the pop-up panel:



It is possible to select several items with several parameters.

The contour can be closed and open. Start and end points of the contour can be changed in the graphics window:



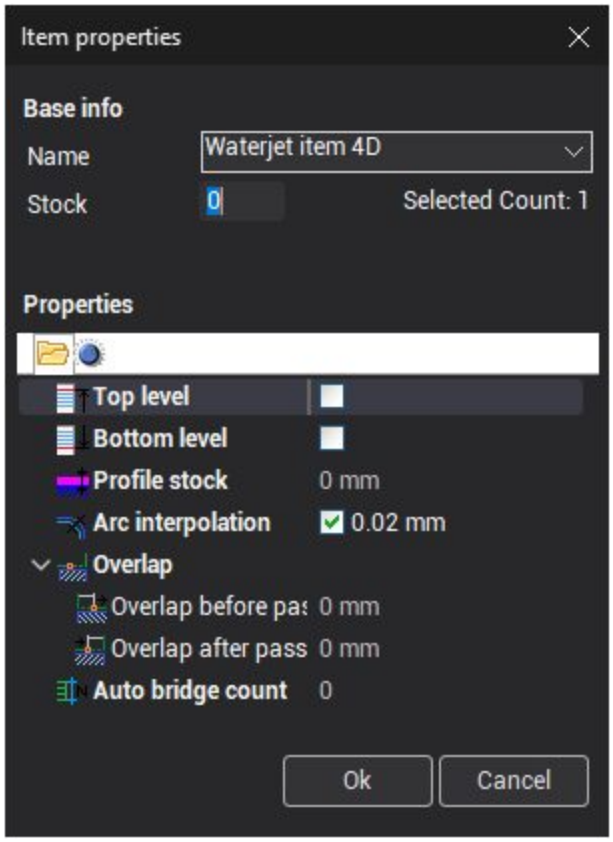
The system allows you to edit the contour directly in the graphics window. Editing principles are similar to those used in Lathe operations.

2D job assignment item properties

Each element of the job assignment has a set of properties.

To view or edit the properties of 2D job assignment item select the item and click the Properties button, or double-click the item.

This is the item properties dialog:



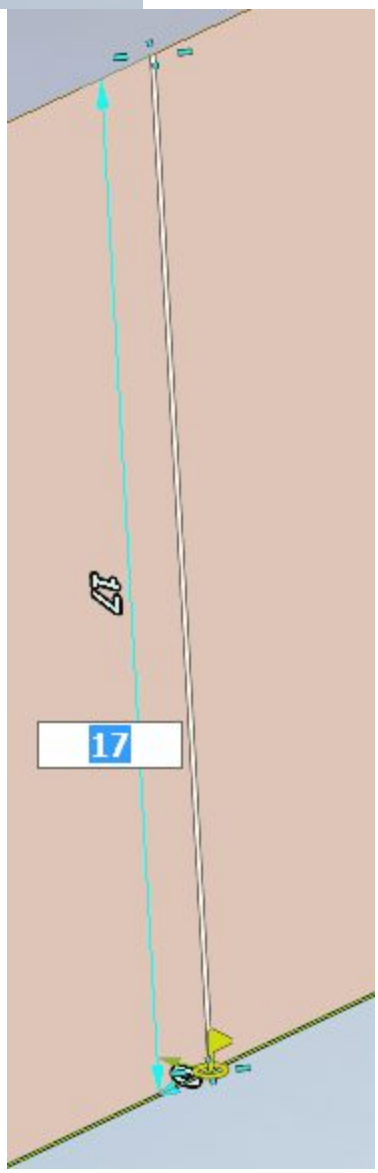
- Top level – plane for the top guide of the EDM-machine.To set the top guide level via the graphic window move the top level sign:



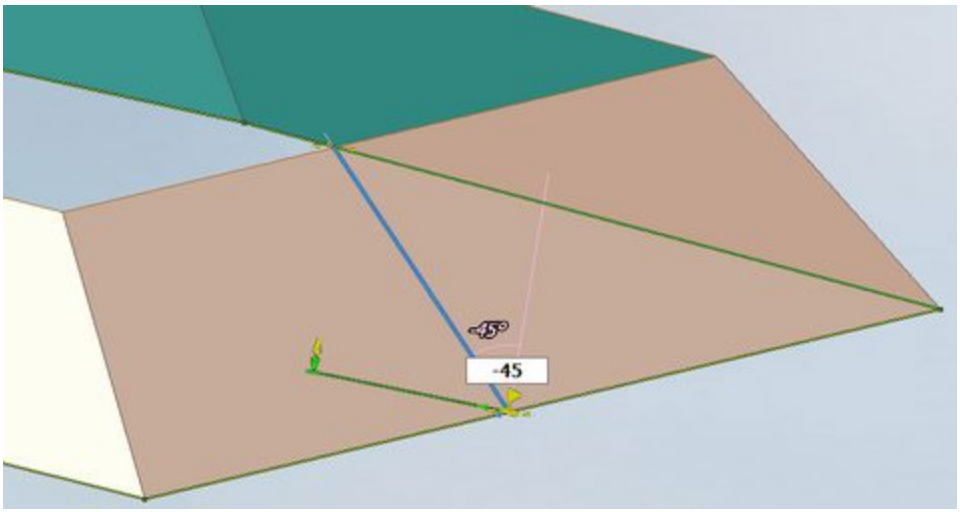
- Bottom level – plane for the bottom guide of the machine.To set the bottom guide level via the graphic window move the bottom level sign:



To set the exact level of the top or bottom guide click the sign of the level and input the value:



- Profile stock – additional stock for the resulting contour. The value of the stock can be either positive or negative.
- Overlap before pass – value of overlap at the beginning of the job assignment item.
- Overlap after pass – sets the value of overlap at the end of the job assignment item.
- Auto bridge count – number of bridges. Bridges properties can be set in the graphic window.
- Taper parameters – if turned off the wire will be positioned at normal to the XY plane and the result of the machining will be a cylindric surface. To machine a conical surface turn on the feature and set the taper angle value and direction for each contour. To set the taper angle click the synchronization line and input the value.

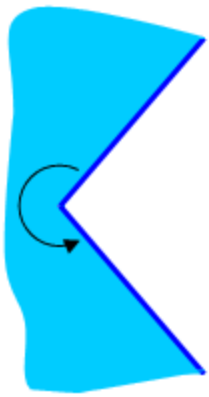
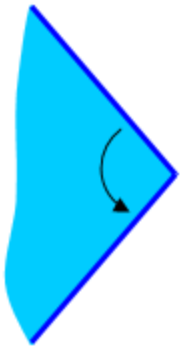
- 

When taper is turned on the taper angle value will be output into each NC frame (for example `G01 X30 Y45 A5`). Turning the taper feature on enables additional parameters.

- Taper application can be one of the following values:
- All passes – taper will be applied for all passes of the contours.
- Apply after pass – taper will be applied after the pass number set in the `Pass #` field. Taper will be disabled for the N passes, passes starting with N+1 will have taper enabled.
- Cancel after pass – taper will be canceled after the pass number set in the `Pass #` field. Taper will be enabled for N passes, passes starting with N+1 will have taper disabled.
- Corners rolling – Modern EDM NC-controllers support automatic rolling of sharp corners in the wire path. CAM system can use this feature of NC-controller. Corners rolling panel contains properties that are used to setup corners rolling feature of the NC. When the feature is enabled the output of contour coordinates into the NC-program are the same (the contour is not changed). However, in the NC-frames where the corners rolling is required additional words defining the rolling radii are output. Rolling radii can be defined separately for the top and bottom contours. For example, `G01 X95.24 Y53.09 R1.5 R5.3` – the first R word defines rolling radius for the bottom contour, the second R word defines rolling radius for the top contour 5.3.

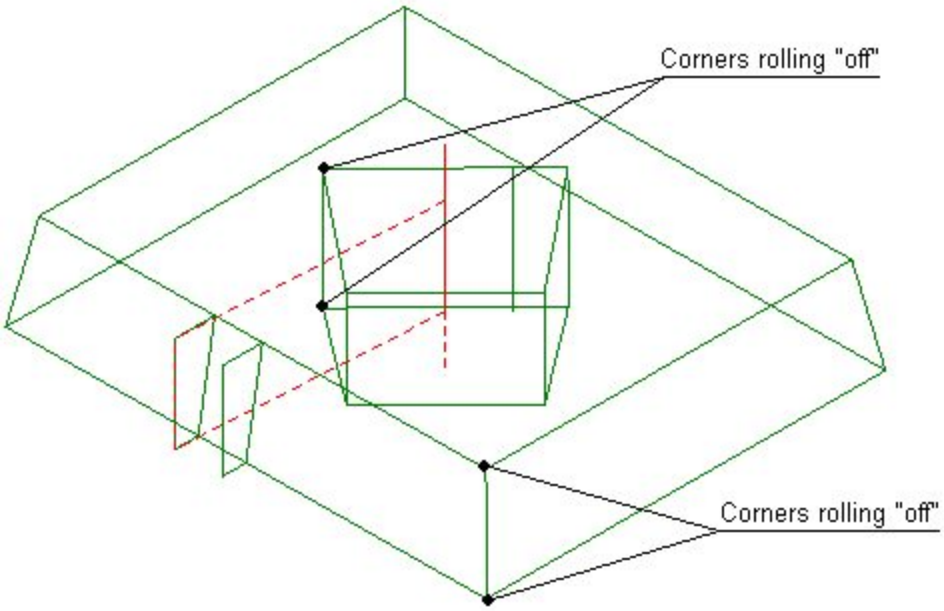
Remark: CAM system simulation supports visualization of the wire path modified by the NC-controller corners rolling.

CAM system supports various modes of corners rolling which can be set separately for inner(Inner corners rolling) and outer corners(Outer corners rolling). Whether the corner is inner or outer is determined by the angle value inside the part. Inner corner has the angle of 180° or more, outer has the angle less than 180°.

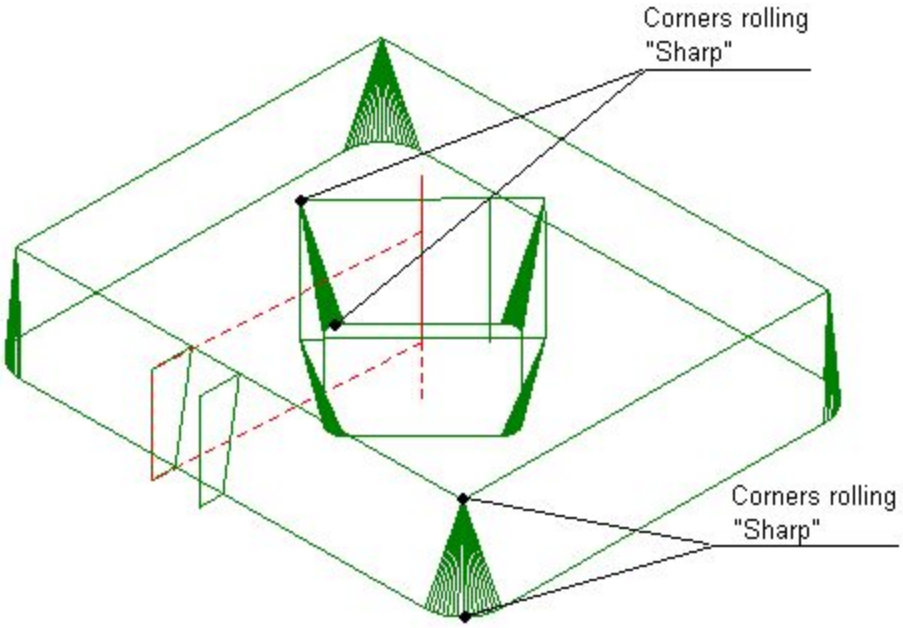
Inner corner	Outer corner
	

The following modes of rolling are supported:

- Off – in this mode corners rolling feature is disabled, rolling radii are not output into the NC-program.



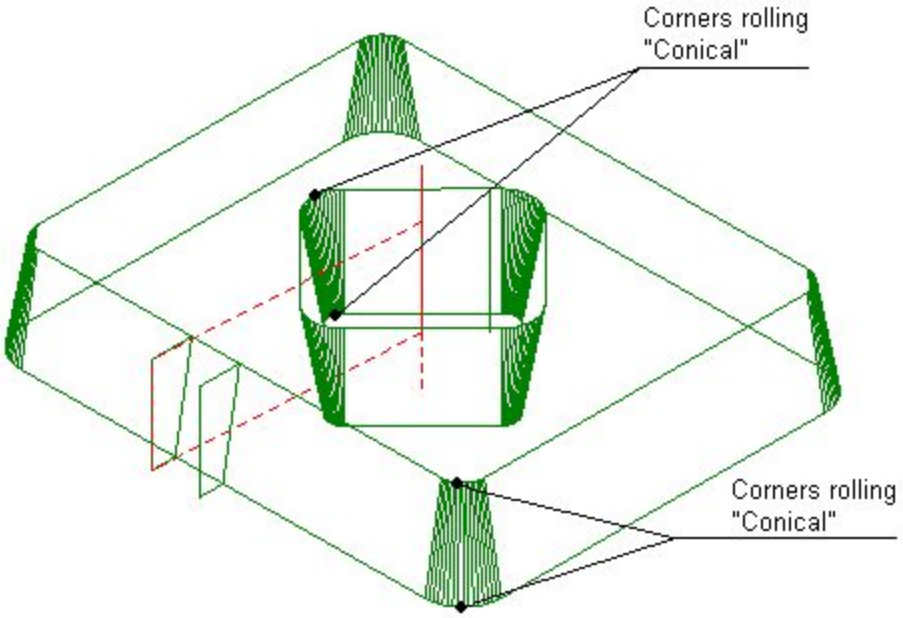
- Sharp – in this mode only the bottom contour corners are rolled, top level corners are not rolled. Therefore only the Bottom radius input field is enabled.



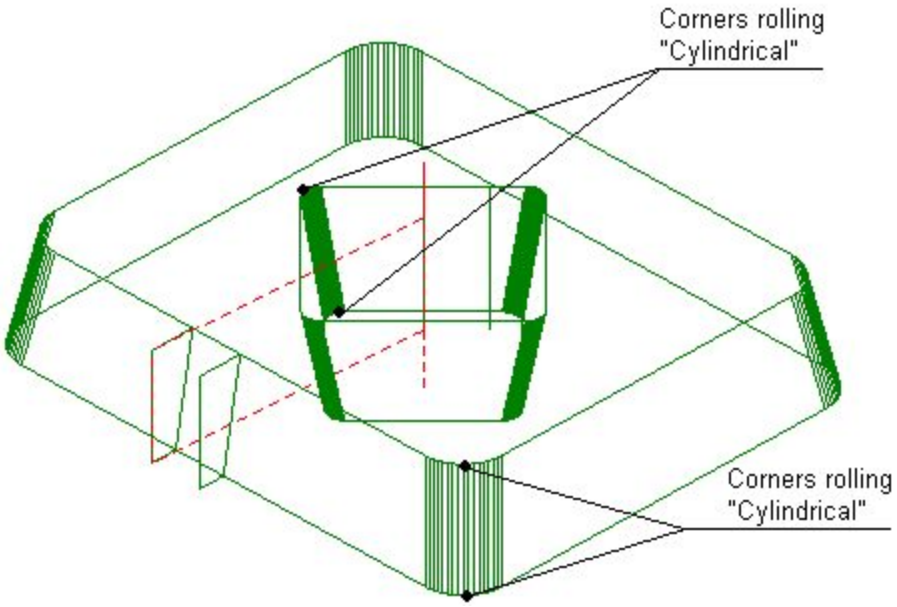
- Conical – in this mode the bottom contour corner rolling radius is set in the appropriate input field. Top contour rolling radius is defined a sum of bottom radius and a value depending on the taper angle and difference of top and bottom contour levels:

$$Rtop = Rbottom \pm h \cdot tg\ \alpha$$

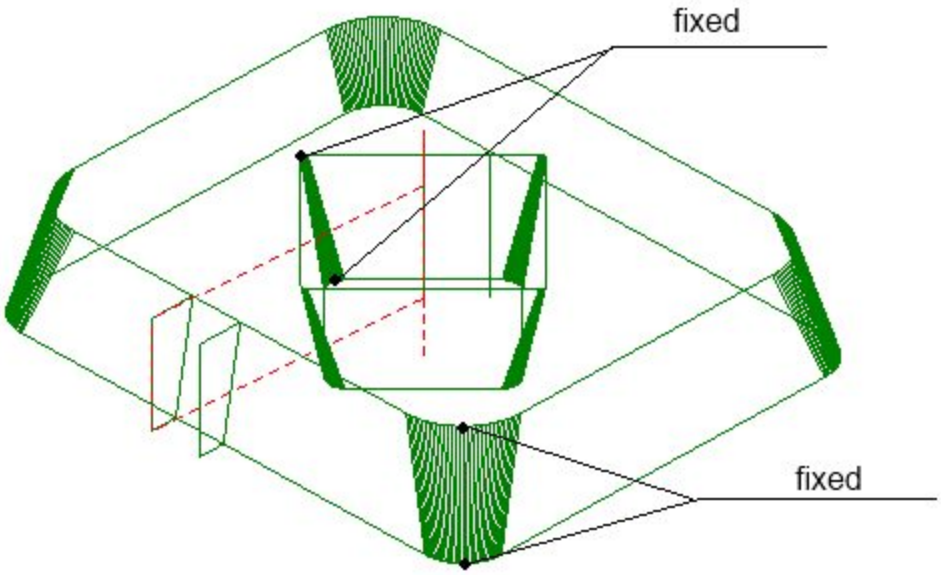
Thereby, a conical surface is machined on the part in the place of corner rolling.



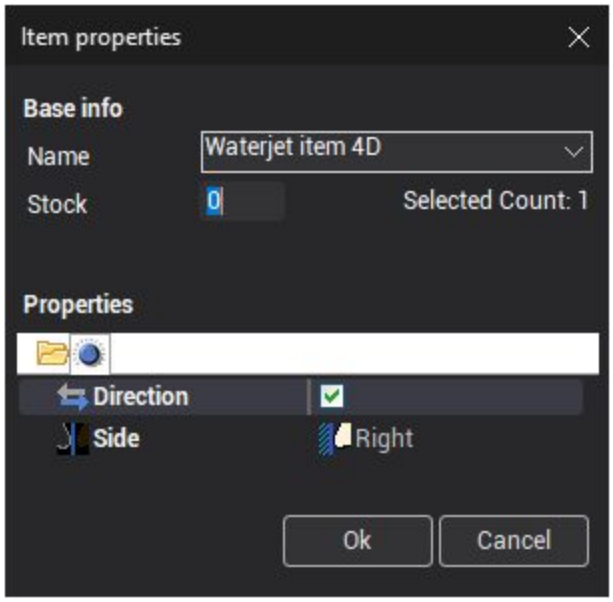
- Cylindrical – corner rolling radius is always equal for the top and bottom levels and is input in the Bottom radius field. Thereby, a cylindrical surface is machined in the place of corner rolling.



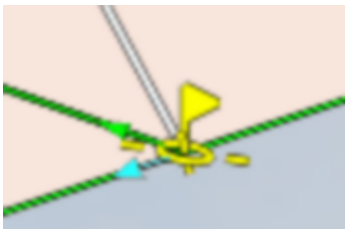
- Fixed – radii of rolling at top and bottom levels are defined independently in the respective fields and can be set to arbitrary positive values.



Use the second tab of the properties dialog to specify direction and side of the machining for each job assignment item.



In the graphic window direction is shown with a sky-blue arrow, machining side is shown with a lime arrow.



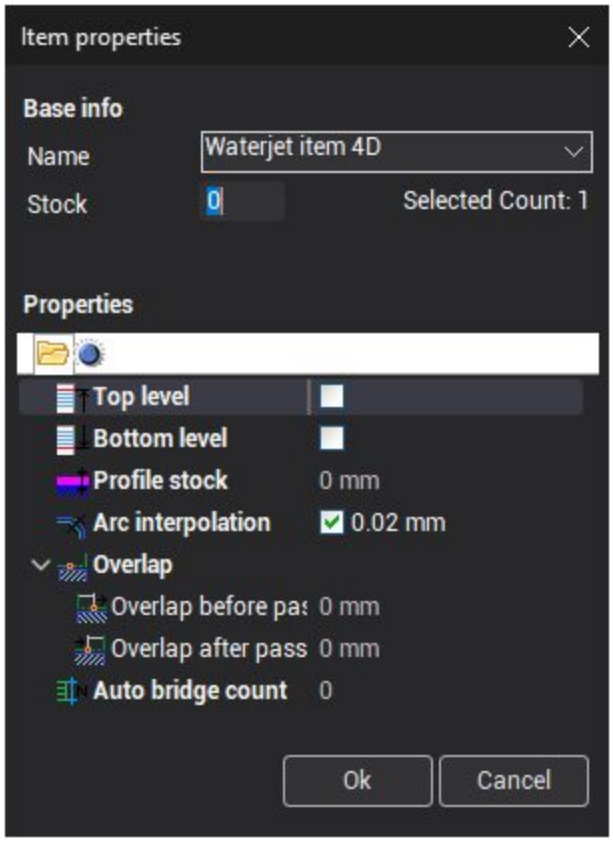
This parameters can be modified either via the properties dialog or by clicking in the arrow in the graphic window.

4D job assignment item properties

Each element of the job assignment has a set of properties.

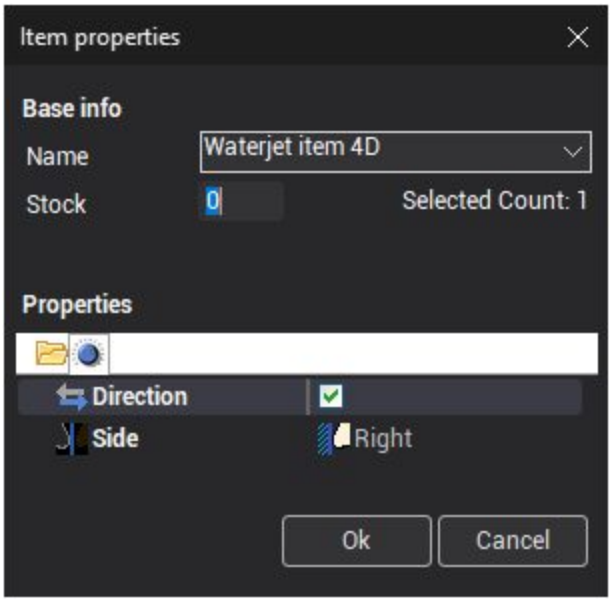
To view or edit the properties of 4D job assignment item select the item and click the Properties button, or double-click the item.

This is the item properties dialog:



All properties are the same as the 2D job assignment item properties.

Use the second tab of the dialog to set direction and side of machining for each job assignment item.



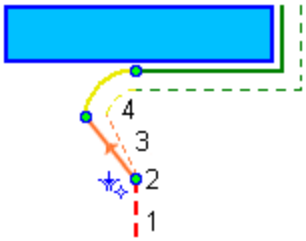
Direction and Side properties are the same as the 2D job assignment item properties.

- Swap chains – swaps the top and bottom levels.
- Inverse bottom chain – inverses the direction of the bottom level contour.
- Inverse top chain – inverses the direction of the top level contour.

Approaches/returns

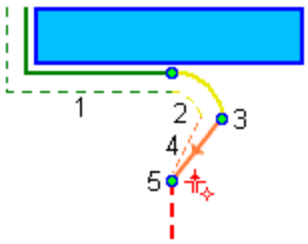
Lead-in and lead-out are the parts of the tool path, defined at the start and the end of each contour tool path. These are used for the correct machining at the start and end of a contour. These moves are used to enable various interpolation functions such as compensation, taper, multi axial interpolation, etc. To enable these features, it's needed to include one or two additional moves.

Approach to start point has these steps:



1. Approach at rapid feed to wire load point.
2. Wire loading, setup mode of cut and mode of correction and interpolation.
3. First part of lead-in – linear move from wire load point. On this step enabled modes are turned on.
4. Second part of lead-in – move to the start point of machining contour. It is necessary for composite lead-in, for example, "line and arc" lead-in.

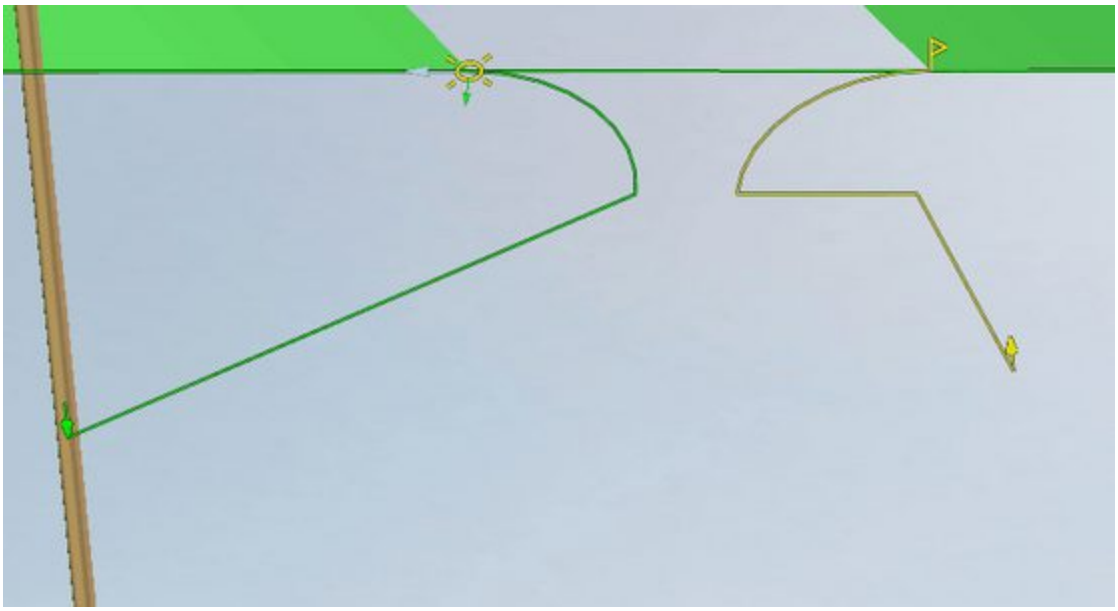
Retract from end point does the sequence in reverse:



1. Move to end point of machining contour.
2. First step of lead-out – non-linear move. It is necessary for composite lead-out, for example, for "line and arc" lead-out.
3. Turn off correction and interpolation.
4. Second step of lead-out – linear move.
5. Wire break point.

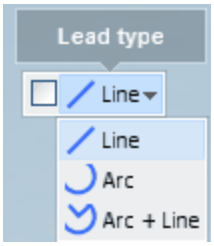
The lead-in/lead-out parameters are defined in the graphics window.

Approach\returns markers are available for each element of the job assignment and becomes available after the calculation of the operation. After changing the parameters it is necessary to recalculate the operation. The approach is green, return is yellow.



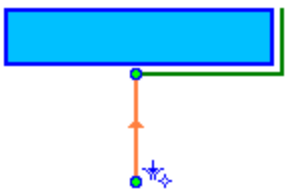
Approaches\returns markers are fully interactive. It is possible to move them and specify dimensions. Dimensions can be set as relative to other elements, as well as relative to the coordinate system.

It is possible to select lead mode in the pop-up panel:

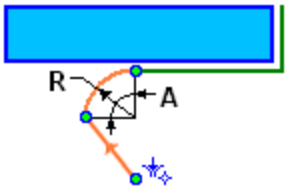


`Lead-type` – this panel is used to setup the lead type. There are several available lead types in the drop-down list:

- Line – linear motion from wire load point to start point of machining contour. The length of the linear move is determined by the position of the wire load point.



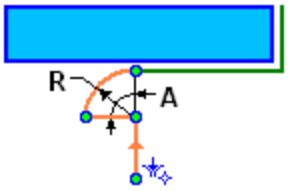
- Arc – lead-in has a linear motion from the wire load point to an arc start point. The arc move is tangent to the start of the machining contour.

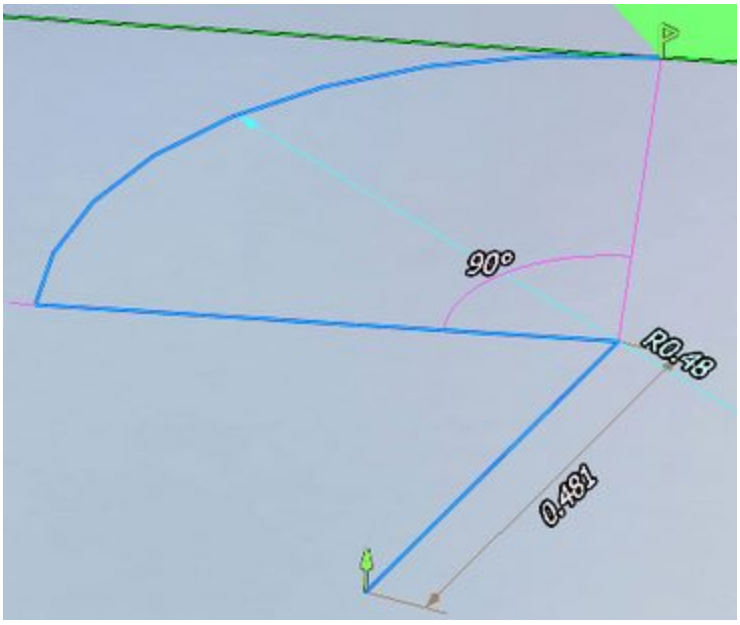


R, A - R — radius; A — approach arc angle

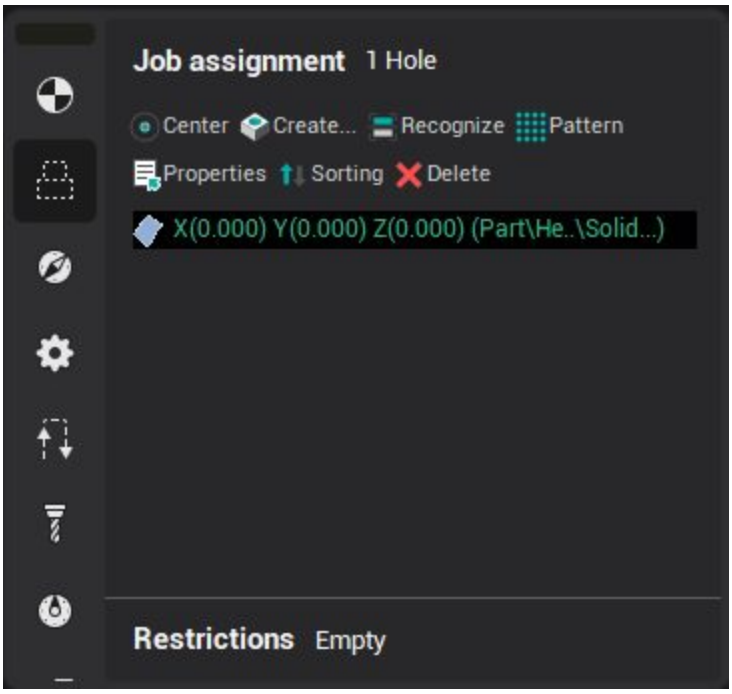


- Line and arc – the first linear motion moves from the wire load point to the arc center point, then to the arc start point. Then the arc moves tangent to the start point on the contour.





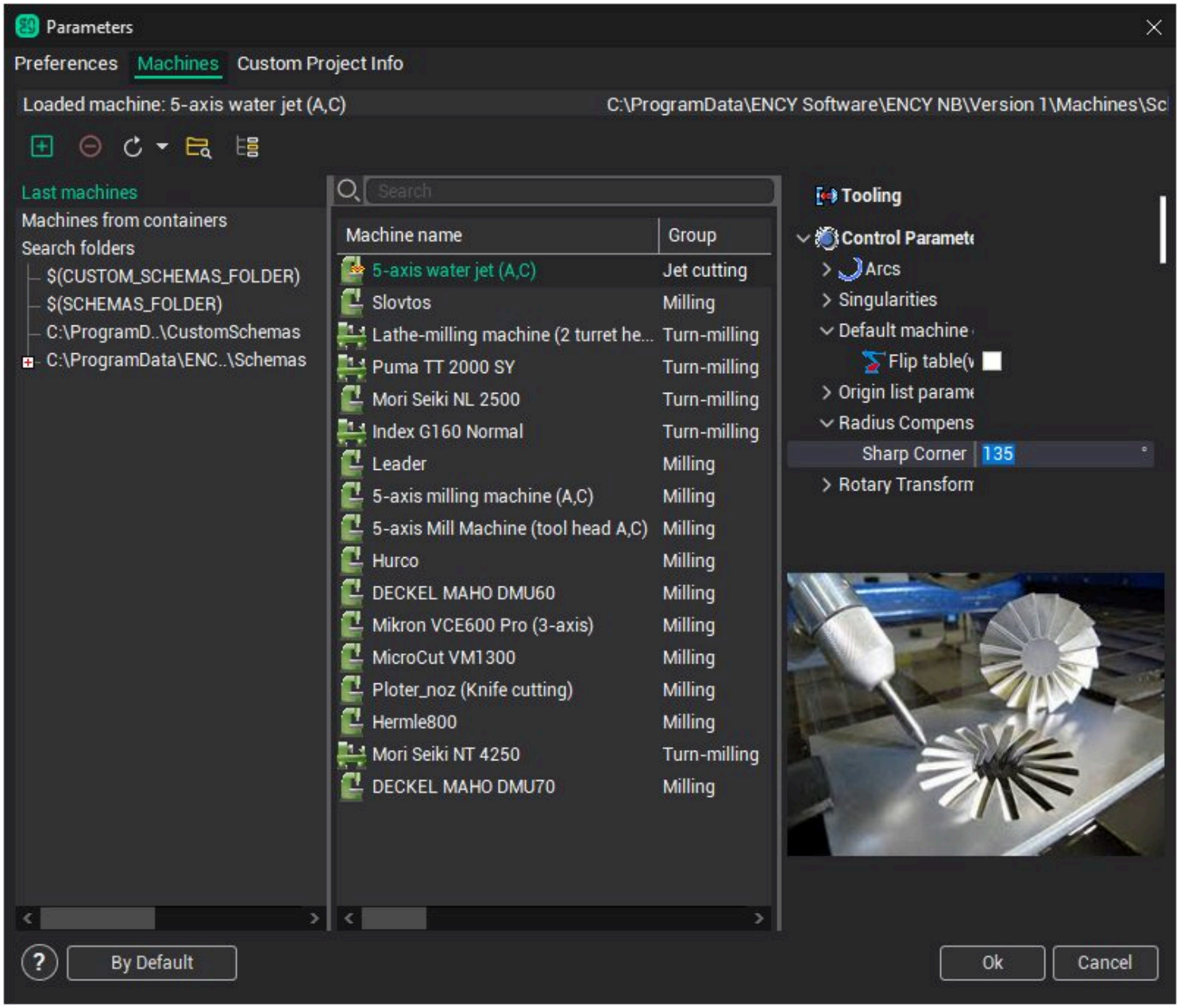
Attention: All wire load or wire break points that are used when an operation is calculated, can be viewed on the Technology panel Holes. Also, you can export this list of points to use in another CAM system project or other application. The export command is accessed from the main menu of CAM system or from the context menu of hole list `Export selected in DXF...`.



Wire radius compensation options

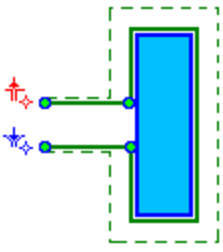
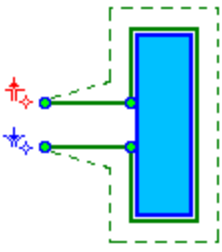
CAM system can calculate, view and simulate wire motion using compensation for the wire radius. When compensation is used, there are commands to turn compensation on and turn off included in the CLData. These are usually `G41`, `G42`, `G40` codes with a compensation number. CAM system will draw the path of the wire motion and can simulate the machining with compensation of the wire radius.

Different NC machines can use different methods for applying / canceling compensation. CAM system have several options which can be used to 'tune' CAM system's wire radius compensation so that it matches those used by the machine control. These options are available in the `Machine:... Parameters` window on the Machines tab. There is a node called Control parameters –> Radius compensation a property editor, the properties are used for tuning the CAM system simulation of radius compensation.

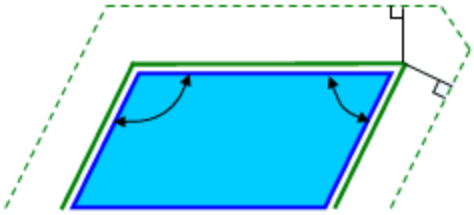


Use these properties:

- Normal approach – used for tuning the motion on approach and retract.

When **Normal approach** is on. Start and end machining point stay on normal to contour.	When **Normal approach** is off. Start and end machining points are shifted by radius compensation value.
	

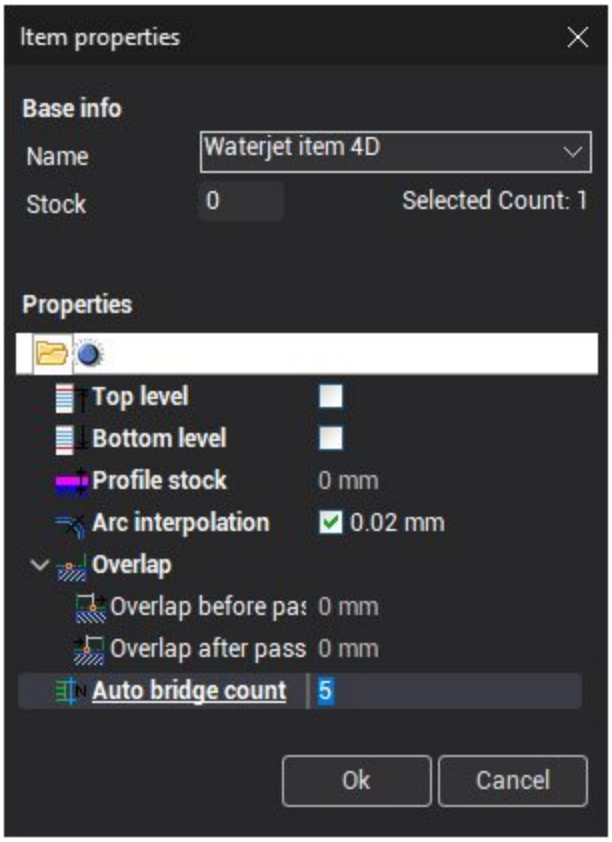
- Sharp corner – this value defines the method of rounding a corner. If the angle between the moves is greater than this value then the motion will be extended to intersect. Otherwise, if the angle is less, then each motion will extended by the value of the radius compensation and connected by a linear move. In the drawing below are shown an example where the "left" corner is greater than the sharp corner value, but the one on the "right" is less.



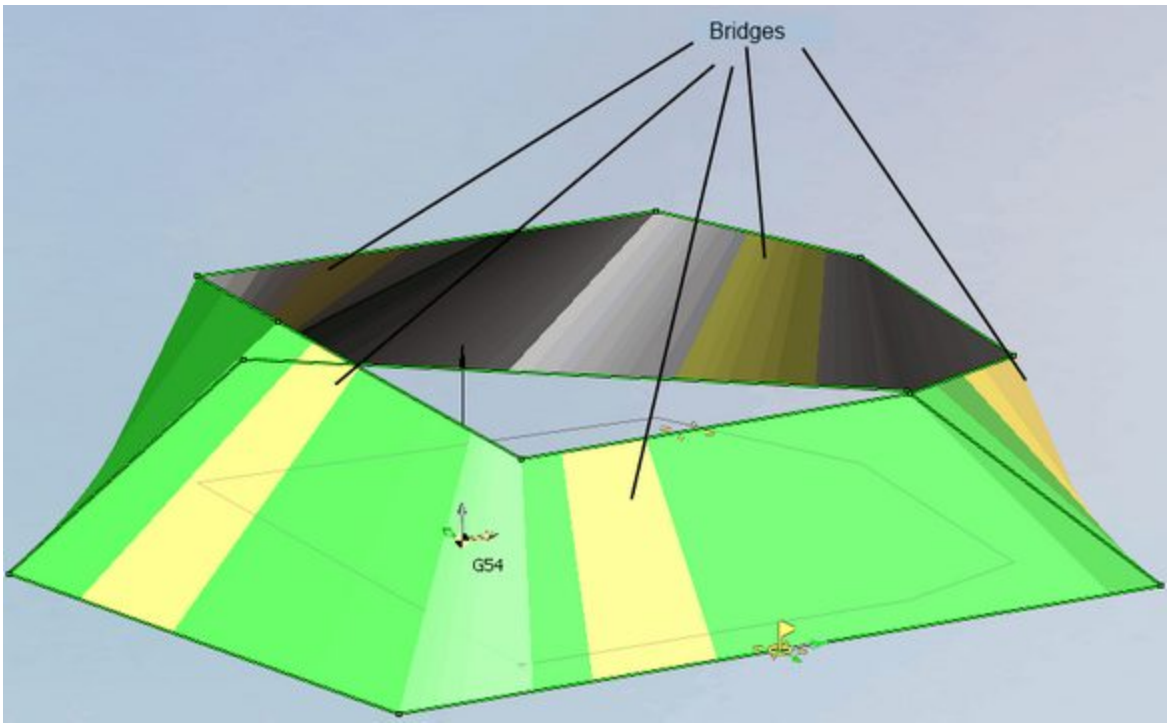
Bridges

Bridges are parts of contour that should be cut after the contour itself.

Bridges can be either disabled or set automatically. Use the Auto bridge count property to change the number of bridges.

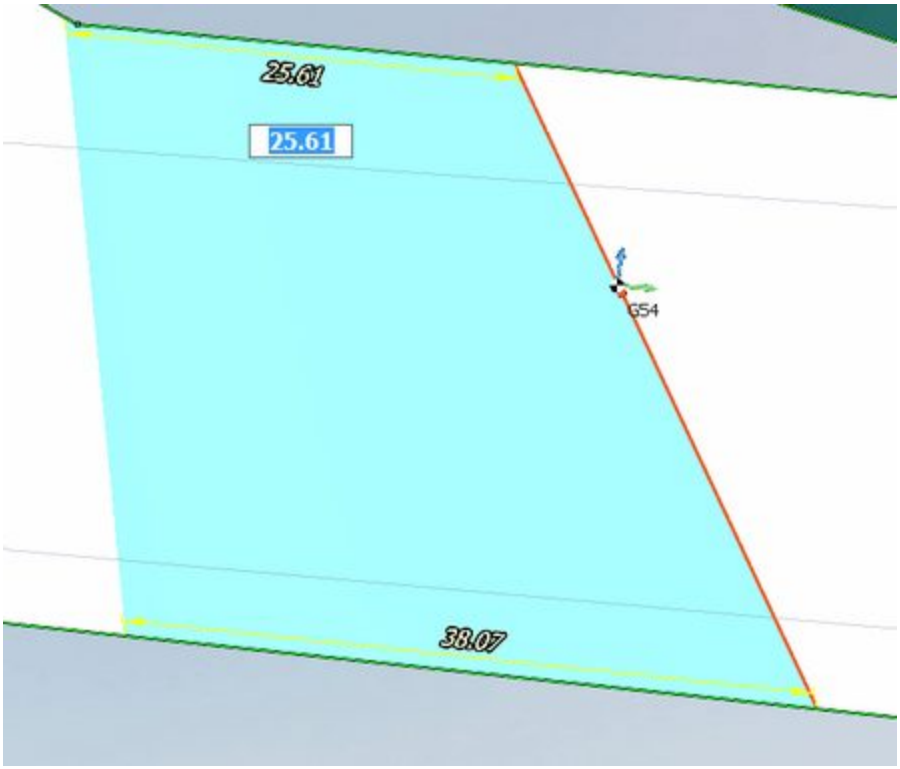


Auto bridges are placed at equal distance from each other. Change the parameters of bridges in the graphics window.



Click a bridge and drag it to a desired position.

The size of bridge can be set by the dimension line, either by dragging the edge of the bridges on the required distance. The edges of the bridge are synchronization lines.



Synchronization lines can be copied and deleted in the graphic window.

To copy select the item and holding the [Ctrl] and left mouse button drag the item to desired position.

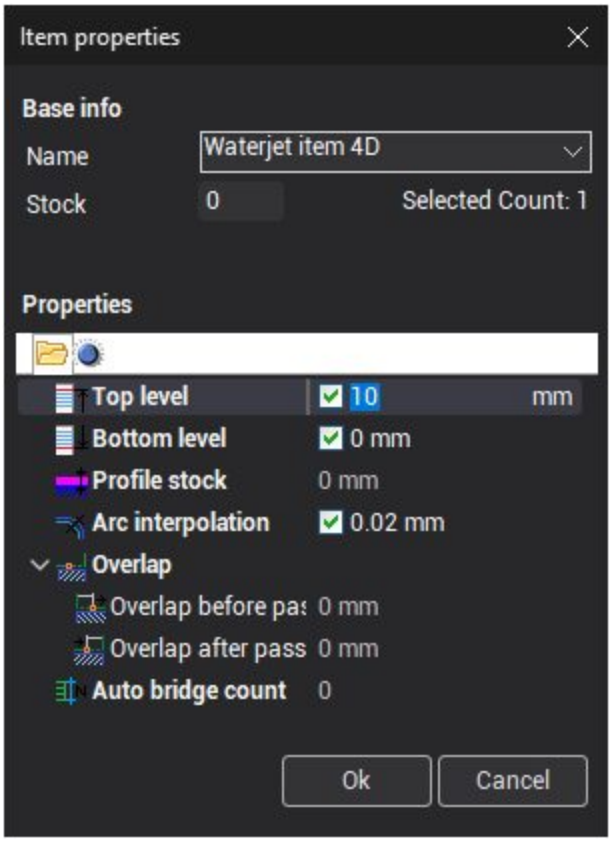
To delete an item select it by clicking it and press the [Del] button. It is also possible to use button on the pop-up panel:



It is possible to select several bridges with several parameters.

Multiselect feature

Use the floating actionbar to select items that have common attributes. Select any item and activate the activate the multiselect option.

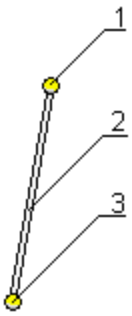


Pop-up panel will show properties of the selected item. Use checkboxes to filter selection based on the values of properties of selected element. If a property is checked only items with equal value of that property are selected. For example to check all items that have stock of 0mm select one such item and check the Profile stock property.

Synchronization lines

In the Wire EDM 2d Contouring operation synchronization lines are shown as two points connected by a line. Synchronization lines can be used to define taper angle.

In the Wire EDM 4d Contouring operation in addition to moving synchronization lines line points positioning can be changed. To move the line position the mouse pointer at the middle of the line(2), and to move the points position the mouse at the point itself (1 or 3).



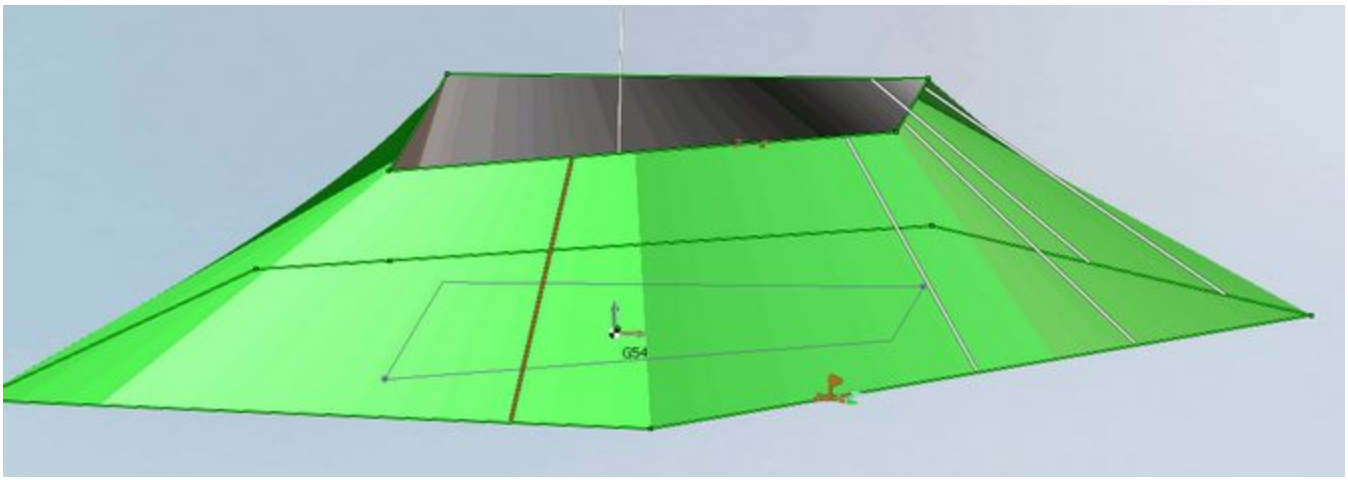
Synchronization lines can be copied and deleted in the graphic window.

To copy select the item and holding the [Ctrl] and left mouse button drag the item to desired position.

To delete an item select it by clicking it and press the [Del] button.

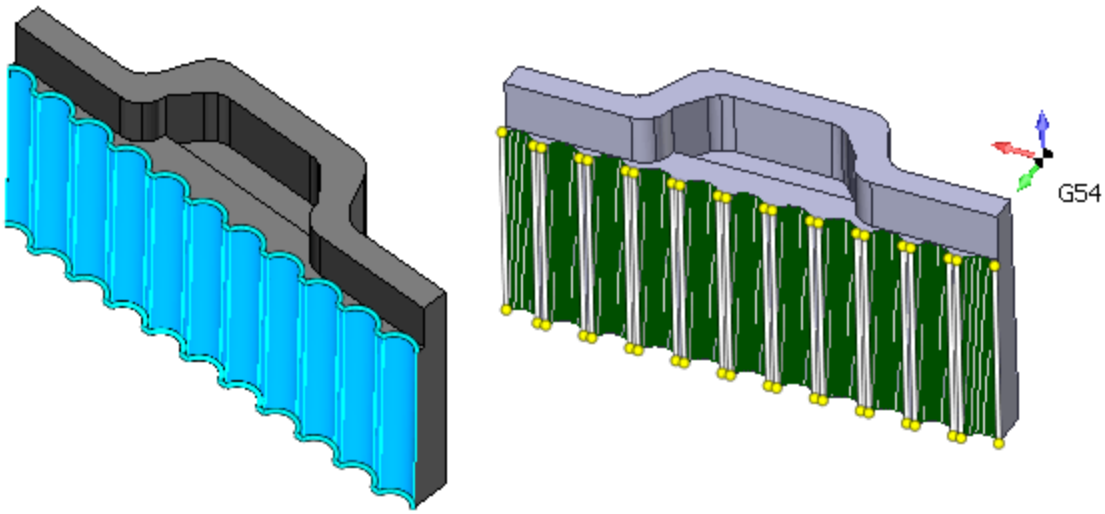
To create a new synchronization line position the mouse pointer over a fracture in the job assignment element and drag the created line to desired position.




G54

Wire EDM feature

Wire EDM feature is a chain of ruled surfaces with top and bottom edges lying in horizontal planes. CAM system automatically recognizes top and bottom curves of the element, also it places synchronization lines in appropriate parts of contours.

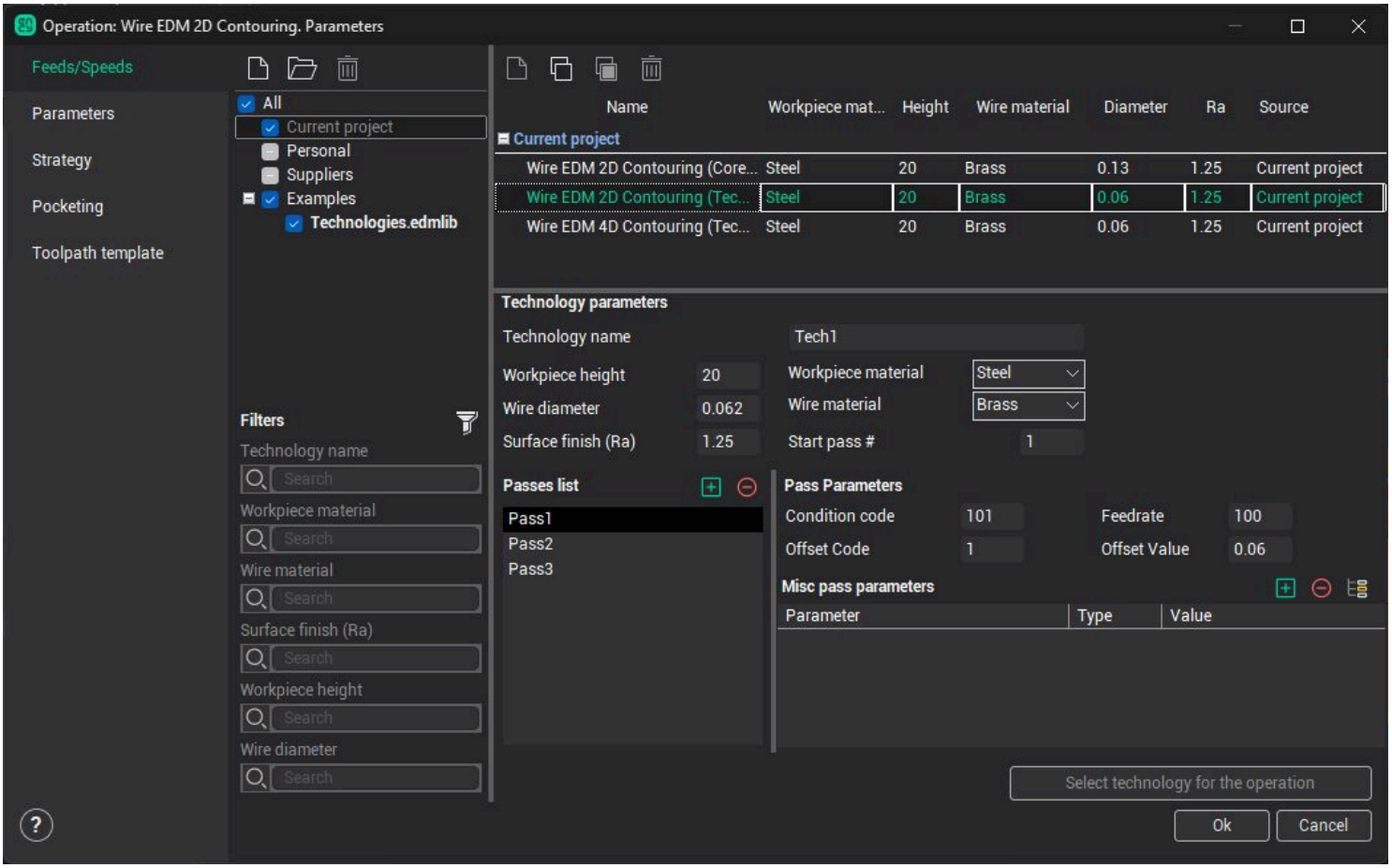


For a convenient and rapid creation of new Wire EDM operation to machine a 3D model select any of faces belonging t constructive elements you want to machine and create a new operation by selecting it in the drop-down list of the New button. CAM system auto detects constructive elements you marked, adds them to the job assignment and setups properties of the operation according to parameters of constructive elements.

To add a constructive element to existing operation select a surface belonging to the constructive element in the graphic window and press one of buttons Add cap or Add hole. CAM system will setup operation parameters according to the constructive element properties if the job assignment was empty.

Machining conditions of EDM and Jet cutting 4D operations

The Feeds/Speeds page of the operation parameters window can be used to specify machining conditions of the Wire EDM operation. It also allows you to work with condition libraries.



The left panel of the Feeds/speeds page shows the list of storages for edm libraries. The storage is either a library file (*.edmlib) or the Current project.

The storages grouped to folders. There is a checkbox next to each item. It toggles the visibility of this storage in the right panel.

The top toolbar contains the following buttons.

- New library> – creates a new library. It will shows save file dialog where you should point a path and name for new *.edmlib file. You should place your libraries outside Examples or Suppliers folder, otherwise they will be read only.
- Open library – add existing library to the list. It will show standard file open dialog where you should select one *.edmlib file. All *.edmlib files placed inside standard folder ($(COMMON_APPDATA)\Libraries\WireEDM) will be added to the list automatically.

-  Remove library – removes selected library from the list. It will not be deleted from the hard drive physically.

Present days NC-controls can to support various assignments of machining conditions, but many of them is using following algorithm. At the same time many parameters is exists and they defines specific machining conditions. Definite power characteristics (frequency, current strength, generator operating regime, etc), wire feed speed and wire offset can be assigned in subject to height and material of workpiece, diameter and material of wire and surface roughness. Usually equipment producers puts in NC-controls already defined table of process parameters or gives means to infill this table. Every set of parameters is named by definite code. Then in corresponding registers easily puts this codes when G-code is builds. NC-control compares codes with specific process parameters automatically.

A specific set of machining conditions (power, frequency, current strength, generator operating regime, offsets, etc.) associated with defined workpiece and equipment parameters (workpiece material, height, wire material and diameter) we call Machining technology.

Right top part of the window displays machining technologies list for storages selected on the left panel. Right bottom part contains parameters of selected machining technology. Each library contains the list of machining technologies. The project contains the list of operations, but each operation contains single machining technology. So both of technologies from libraries and operations can be shown in one list. The current operation in the technology list is shown in green.

To apply selected technology for the current opeartion there is as big button at the right bottom corner of the window. It will copy all content from the selected technology into the technology of current operation.



You can use buttons on the top middle toolbar of the window to manipulate technologies.

-  New technology – adds new technology to the list. It can be unuvailable if read-only storages selected in the list (Current project or library from Examples folder). Create your own library in custom folder to be possible to edit it.
-  Copy technology to clipboard – copies technology to clipboard to be possible to paste it to any other library.
-  Paste technology from clipboard - allows to paste thechnology from clipboard to selected library. The library should not be read only.
-  Delete selected technology – deletes active technology from the list.

Each machining technology contains following properties:

- Technology name – unique text identifier of process technology.
- Workpiece height – height of processed workpiece.

- Wire diameter – diameter of wire for selected technology.
- Surface finish (Ra) – roughness of surface, that will be provided by the selected technology.
- Workpiece material – material of workpiece, for which selected technology is assigned.
- Wire material – material of wire, whereby machining is does.
- Start pass # - the number of the pass from which machining should start. Default value is 1. However, cases can appears, when necessary to do machining by selected technology, but not from the very outset. For example, this necessity can arise in case of cutting separation of one part section to some operations. In that case in the strategy of first operation the number of passes is defined less then technology has. In the next operation the starting pass is defined different from 1. Closest sequence of passes with its numbers is shown on the Passes display list info panel on the Strategy tab of operation parameters window.
- Pass list / Pass Parameters – list of passes with process parameters for every pass.

The Pass term is one pass, that wire is does along the contour. You can use  buttons to add or remove passes. Following list of parameters is assigned for each pass.

- Condition code – is a value, that is specific for every machine and it is corresponding to number of register in NC-control. Its code defines process conditions. Usually registers C, E, S is used. For more information about the codes of process conditions see manual of used machine.

- Offset Code – is a number of wire offset register (number of compensation radius). For more information see manual of used machine.

- Offset Value – is a wire offset value for selected offset code. The value takes into account wire radius, overburning value and special stock for every pass. The value is send to postprocessor and can be used for initialization of registers, which is responsible for wire offset. Usually this registers is H and D. The offset value is used for compensation modeling with general stock jointly.

- Feedrate – is a rate of wire feeding. The value is measured in mm per minute or inch per minute subject to system settings. Many of wire EDM machines is not use feedrate, but the value is available if its will be needed for specific NC-control. This value is used also by system for cutting time calculation.

- Misc pass parameters – is an array of additional parameters of the pass. Every parameter is presented by line like Parameter – Type – Value. There Parameter is text description of it. Type is a type of parameter, it can to be Integer or Float. Value is a numerical value of the parameter. This parameters array and other parameters of the pass is sent to the postprocessor with PPFUN WEDMConditions(56) command and can be used for specific purposes in each specific case. Input of names and values is performed by mouse click on corresponding cell.

-  Add parameter – adds a new line to the table of additional parameters of selected pass.

-  Remove parameter – removes active line from additional parameters table.

-  – copies additional parameters table of selected pass to all passes of active technology.

It is possible to import technologies from external sources and export them outside the CAM system. To do this just select desired techologies in the list and the click Export selected technologies to XML popup menu item on right mouse button. XML is structured textual format that can be changed by any text editor or processed by any other program. After that you can import this XML to the system again using corresponding popup menu.



Machining technology library

Technology 1

- Workpiece material: steel.
- Workpiece height: 20.
- Wire material: brass.
- Wire diameter: 0.25.
- Surface finish (Ra): 1.25.

Passes

Name	Condition code	Offset code	Offset	Feedrate	Additional parameter 1	Additional parameter 2	...	Additional parameter N
Pass 1	101	1	0.75	8	0	0		0
Pass 2	102	2	0.5	10	0.5	0.3		0.7
Pass 3	103	3	0.125	3	1	0.6		1.4
...								

Technology 2

- Workpiece material: steel.
- Workpiece height: 40.
- Wire material: brass.
- Wire diameter: 0.5.
- Surface finish (Ra): 0.8.

Passes

Name	Condition code	Offset code	Offset	Feedrate	Additional parameter 1	Additional parameter 2	...	Additional parameter N
Pass 1	101	1	1.25	5	2.3	0		0
Pass 2	102	2	0.75	7	7.8	0		0
Pass 3	103	3	0.5	2	0	0.6		0
...								

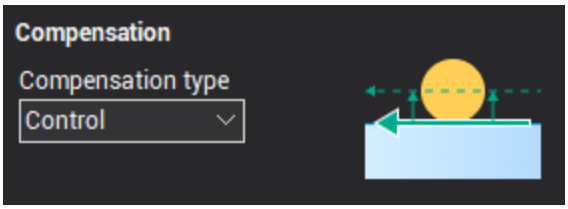
...

Strategy of EDM and Jet cutting 4D operations

Alteration of the many available parameters for the machining strategy are made in the Parameters –> Strategy window. This window is accessed by clicking the Parameters button which is located in the Machining mode. On the Strategy tab there are many panels with input fields and explanatory images. The composition of these panels are determined by the type of current operation.

Wire EDM machining operation of contours includes the following set of parameters:

- Compensation type – determines the way in which the offset of the wire is performed on a given contour.



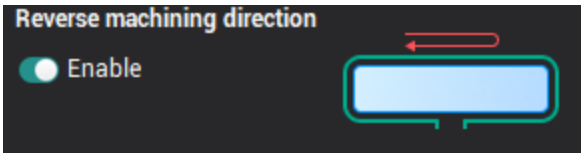
The following compensation types are available:

- Computer – the system itself calculates the corrected wire toolpath and the codes to enable compensation are not output in the G-code. In the registers responsible for compensation, the values of the offset are not added.
- Control – the system outputs into the G-code the codes to enable compensation, and does not offset the wire. The registers responsible for the value of compensation, record the values of the offset for each pass. Compensation is calculated by the CNC control.
- Both – the system outputs into the G-code the toolpath with provision for offsets already, but into G-code are outputted codes enable of compensation also. Registers that are responsible for the value of the compensation aren't filled.
- Reverse Both – correction is calculated similarly in the Both regime, but the direction of the compensation changes to the opposite.
- Off – wire offset values entered for the contour are not produced. Codes for compensation into the G-code are not output. Registers that are responsible for the amount of the compensation system are not used.

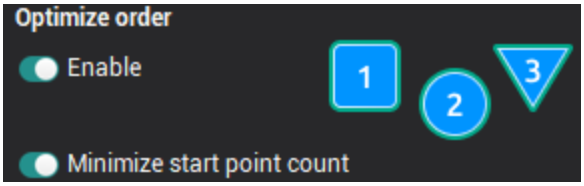
The value of compensation for each pass is defined as an Offset value, in the `Feeds/Speeds` tab, plus the stock value of the operation. For the compensation types Computer, Both and Reverse Both the value is used to construct an equidistant path, and for the Control type the value is entered into the register with a number equal to the Offset code, specified for the corresponding pass in the `Feeds/Speeds` tab.

The direction of compensation can be set for each contour individually within the Job assignment of an operation.

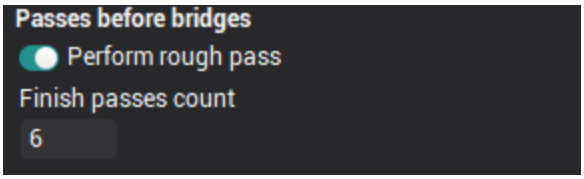
- Reverse machining direction – if you enable this option, the system will choose the direction for the pass, which provides the smallest length of the toolpath. If the option is disabled, then the direction of the pass will always correspond to that specified in the Job assignment for the contour.



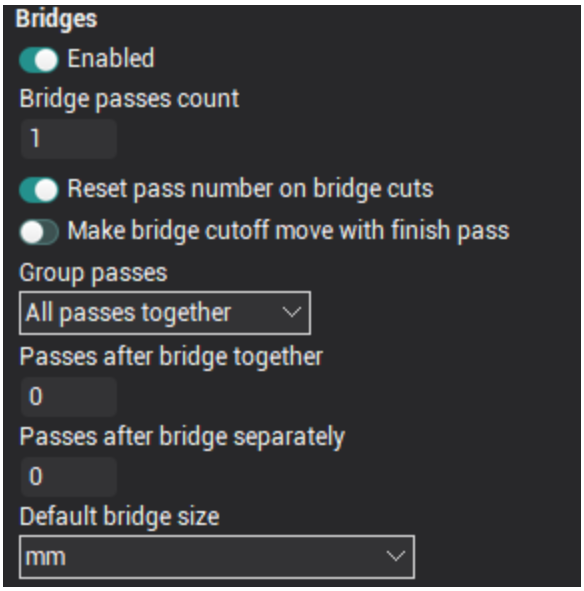
- Optimize order – this strategy determines the order of contours passes when the job assignment has more than one contour. The length of transitions between the contours will be minimal if the Optimize order option is enabled. If this option is disabled, then the order of the passes would be consistent with the order of the contours in the Job assignment.



- Passes before bridges this panel determine the number of passes that will be performed for each contour of the job assignment, to trim bridges. If the formation of the bridges is disabled, then these parameters determine the total number of passes for each contour. If the option Perform rough pass is included, then one rough pass for each contour will be executed, as well as the number of passes as defined in the Finish pass count field. When you turn this option off, rough and finish cuts to trim the bridges are not made. Approximate sequence of passes, depending on the set parameters displayed on Passes display list.



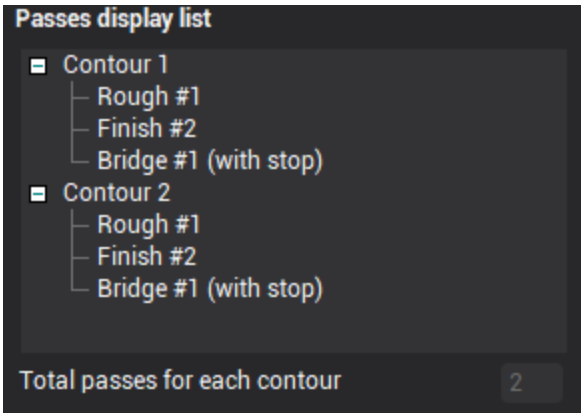
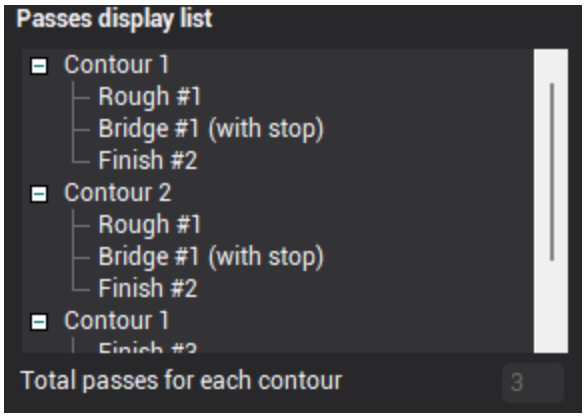
- Bridges. In some cases, for example, if the job assignment is a series of closed contours, passage of the full contour details may lead to an undesirable deposition of parts of the workpiece. The system provides a set of parameters that allow to keep the special sections without machining on the workpiece, these are called bridges. When the wire approaches such zones, the system can be add a Stop command position, to allow additional steps to fix certain parts of the workpiece, then the bridges can be automatically trimmed. Location of the bridges can be specified for each contour individually in the Job assignment section. In the Bridges section it is possible to configure the number of passes for cutting bridges, the number of passes for cleaning the contour after clipping of the bridges, and the parameters determining the sequence of these passes. If the Enabled is not selected then no bridges are cut and no clean cut after cutting bridges is available, accordingly, all fields on the panel are unavailable.



The Bridge pass count field sets the number of passes which will be performed for each bridge cutting on each contour. W hen the option Reset pass number on bridge cuts is enabled, then the count rates for the passes that define the cutting conditions for the bridge cutting moves are reset to the start value, ie from the value that is set on the `Feeds/Speeds` tab in the Start pass # field, otherwise, the count rates of the bridge cut passes will continue. For example: if the last contour pass prior to the bridge cutting was #2, the first bridge cutting pass would be #3 and the next #4 etc.

If the option `Make bridge cutoff move with finish pass` is enabled, then the bridge(s) will be cut on the final pass followed by the lead out move, then, if the bridge pass count is greater than one, the subsequent bridge cuts will be performed. If `Make bridge cutoff move with finish pass` is disabled, on the last contour pass, the lead out move will be performed leaving the bridge, and then the bridge cutting move will be preformed.

The fields Passes after bridge together and Passes after bridge separately together determine the number of finishing passes after bridge trimming that will be performed along the length of each contour as a 'clean up' pass. The difference between these two options is only affected if there is more than 1 contour feature. Example sequences for both types of final passes are shown in the pictures below:

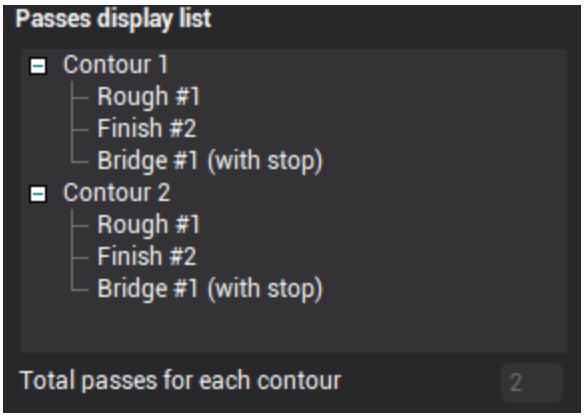
The sequence of execution of "together" passages:	The sequence of execution of "separate" passes:
	

In the drop-down Group passes menu You can choose the way of grouping different types of passes for when working with several contours.The following options are available:

- All passes together – all roughing passes, bridge cutting passes and finish passes are performed for each contour, only when completed is the next contour started.
- Bridges and finish together – all rough passes for all contours are performed, then all bridge cutting and finish passes are performed together for each contour.
- `Rough, bridges and finish separately` – first, all contours rough passes are run without bridge cutting, then the bridge cutting is performed for all contours without finishing, and finally, the finish cuts for all contours are performed.

The approximate sequence of the passes, depending on the selected parameters, are displayed in the Passes display list.

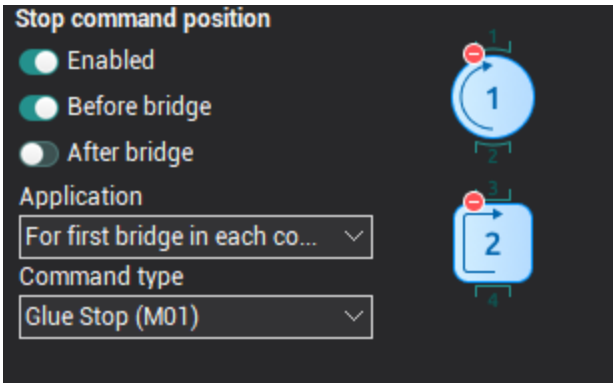
- Passes display list. On the Strategy tab for the Wire EDM operation there are many available options for contour machining that control the manner and the order of processing for the contours of the job assignment. To better understand the impact of a particular parameter on the machining sequence, there is an information panel available called Passes display list. When you change a value or parameter which influences the strategy process, this also changes the contents of this information panel. The main area of the panel takes the form of two-level tree type display. In this tree, the top-level displays the contours, and the lower level displays the types of and number of passes that defines the cutting conditions. In the bottom of the panel is a box that display the total number of passes which are made for each contour using the current settings.



Note: The information panel Passes display list only displays information and all of the fields are read-only. Alteration of the information displayed can only be made using the parameter options that are available in the main window. The information shown in the panel is approximate and may not correspond to the exact sequence of machining since its formation does not take into account the actual geometry of the contours that are in the job assignment of operations. By default, the list always contains two abstract contours.

Note: When specifying the number of passes in the strategy, these should be closely monitored so that the number of passes in the field Total passes for each contour coincides with the number of passes defined for the cutting conditions on the [Feeds/Speeds](#)tab. If there is a discrepancy in the number of passes, then the machined contours may have material remaining upon completion.

- Stop command position. The options located on this panel allow control over the output of the stop commands in the G-code for the bridge cutting passes. The stop commands are only output into the G-code when the Enabled option is selected. The next two parameters define the time of output for the stop command. The Before bridges option enables the stop command which will be output after the bridge approach move, prior to its cutting. The After bridge option enables the stop command which will be output after the cutting of the bridge but before the lead out move from the endpoint to the wire cut point. These parameters operate independently, ie they can be set simultaneously.



The drop-down list Application determines how the bridges should use the technological stops. The options include:

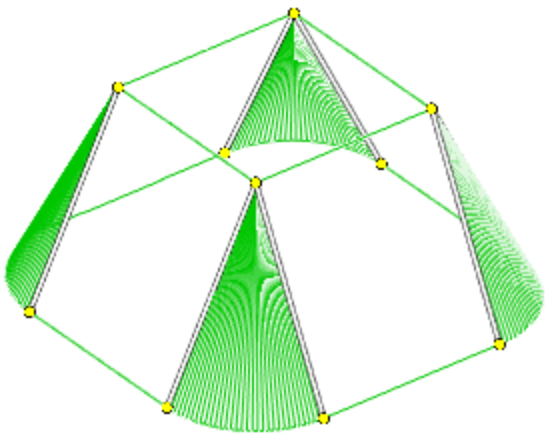
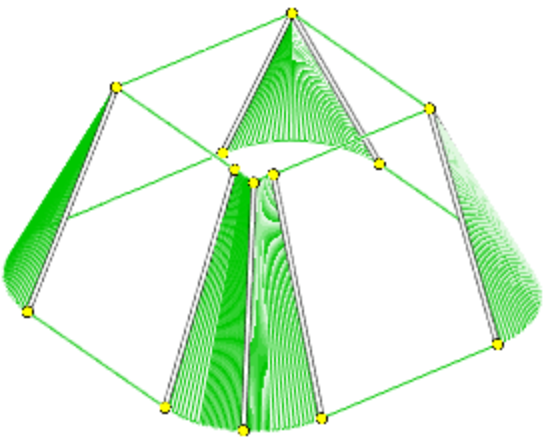
- For every bridge – stop command will be output for the bridge cutting move for every bridge specified in the operation.
- For first bridge in each contour – stop command will be output only for the first bridge of each contour.
- For first bridge in operation – stop command output only when cutting the first bridge of the operation.

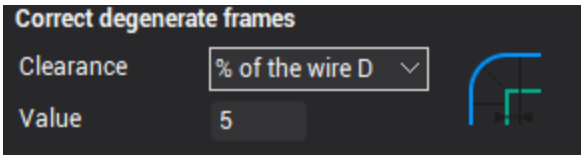
Command type this parameter defines a specific type of output stop command, and can take one of two values:

- `Glue stop (M01)` – the "optional" or "additional" stop allows, in contrast to the usual stop command (M00), when the switch on the CNC control panel is selected, it allows the operator to decide whether the process should stop. Typically, this command corresponds to the auxiliary code `M01`.

- `Stop (M00)` – this command causes an unconditional interruption of the G-code execution. Usually, it corresponds to the auxiliary code `M00`.

- Correct degenerate frames this feature is available in the Wire EDM 4D Contouring operation. Sometimes parts of a given contour can equate to a near zero or zero length. In the example below, the arcs in the left hand figure on the lower contour cause the geometry on the upper contour to degenerate to zero. Often a CNC control cannot handle such cases because they lack the geometric information required in the degenerated frame to be able to calculate for example an offset path, therefore, these cases should be avoided. This is achieved either by manually specifying correspondences in the job assignment, or, by using the correcting function for degenerate frames. In the latter case, the system automatically detects on a contour any very small lengths and "extends" them by an optional amount, as shown in the figure below right.

Contour with degenerate frames:	Contour with a corrected frame:
	



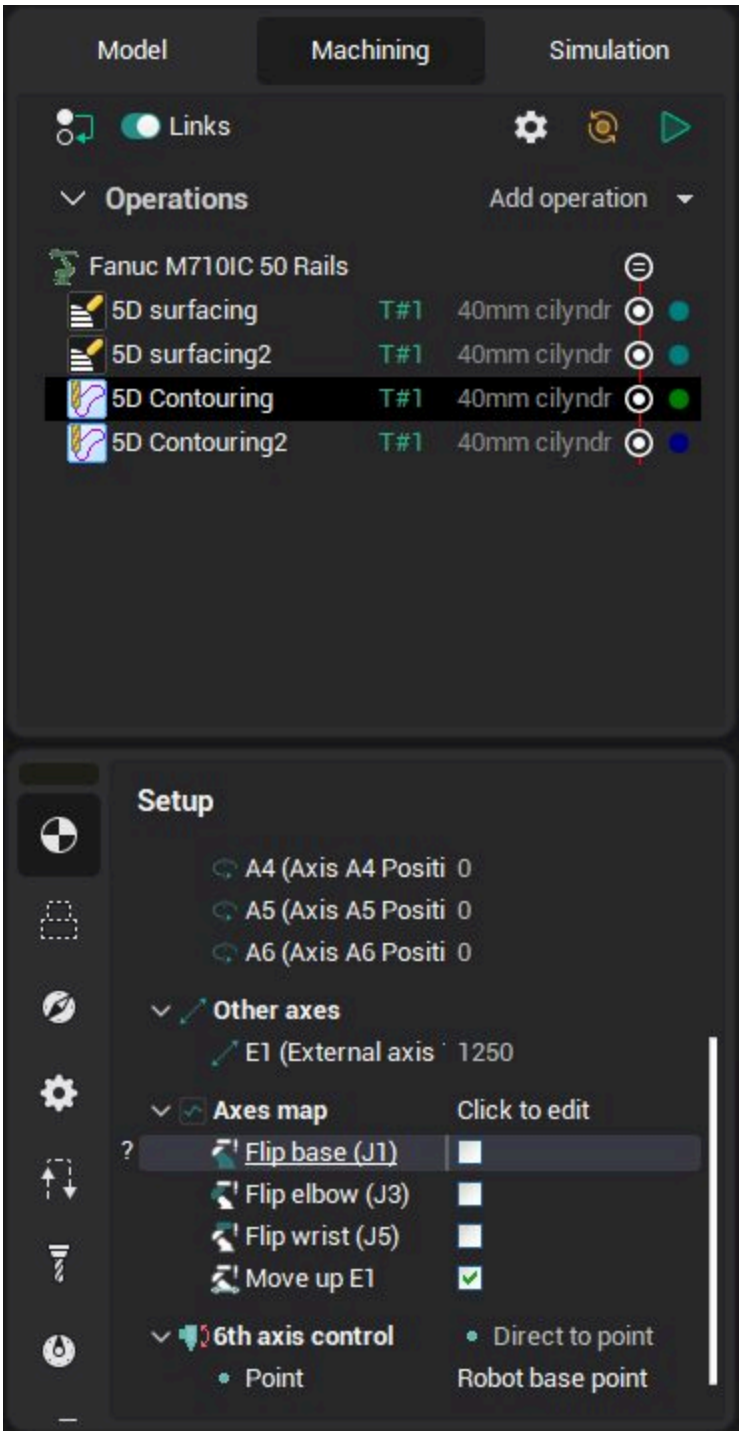
In the Correct degenerate frames section you can enable or disable the function and enter the amount on which to extend the degenerated elements. In the Clearance dropdown the following items are available:

- Off – when you select this item, the correction degenerated frames is disabled.
- Distance – this enables the function. The fixed amount of the required extension is entered into the Value field. The value relates to the currently selected units (millimeters or inches).
- `% of the wire D` – this enables the function. The amount of the required extension is entered into the Value field as a percentage of the current wire diameter. The value relates to the currently selected units (millimeters or inches).

Machining on industrial robots

CAM system can be used to program industrial robots (articulated) for cutting, milling, painting, welding and other applications. The programming process of an industrial robot is basically the same as one of a conventional milling machine, except the robot usually has 6 degrees of freedom (versus 5 degrees of freedom required to position a cutter relative to a workpiece) plus optionally additional degrees of freedom of various types of workpiece positioners (like rotary tables) and robot positioners (like rails). Therefore, when used with a robot CAM system operations offer additional set of parameters to control those excessive degrees of freedom (DOFs).

In addition to excessive degrees of freedom a robot can reach the desired position of the tool relative to the workpiece in several different states. The state to be used in the operation can be specified in the Operation Setup with the flip base, flip elbow, flip wrist checkboxes as well.



In this chapter the following robot programming features are covered:

Setting the coordinate system of the tool and the workpiece

Programming of the robot's 6th axis

Programming of rails position

Programming the rotary table

Avoiding out-of-reach zones and singularities

Programming robot's transitions (obsolete method)

The feature is available in the following CAM system configurations:

- Robots
- Master
- Pro

Setting the coordinate system of the tool and the workpiece

When programming a robot it is necessary to define the tool and the workpiece coordinate systems (the Tool Frame and the User Frame).

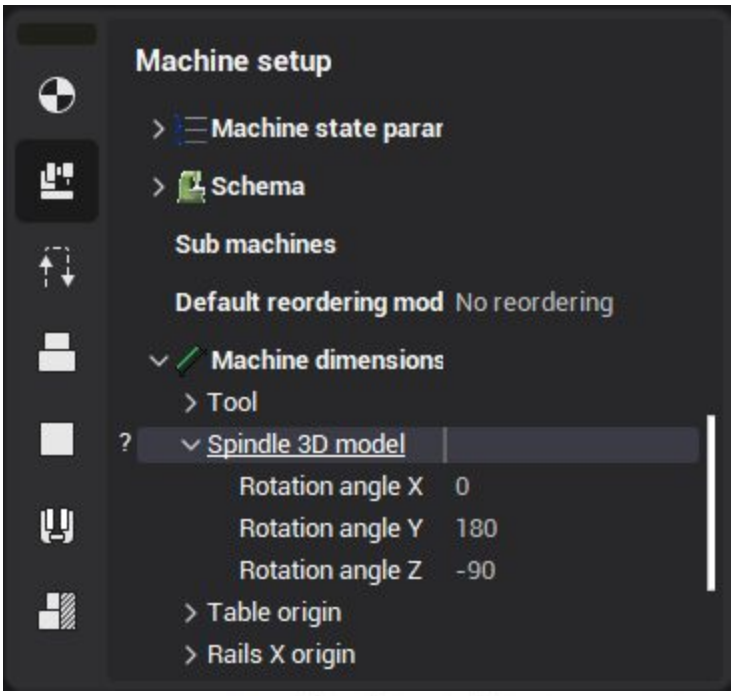
Defining the tool coordinate system

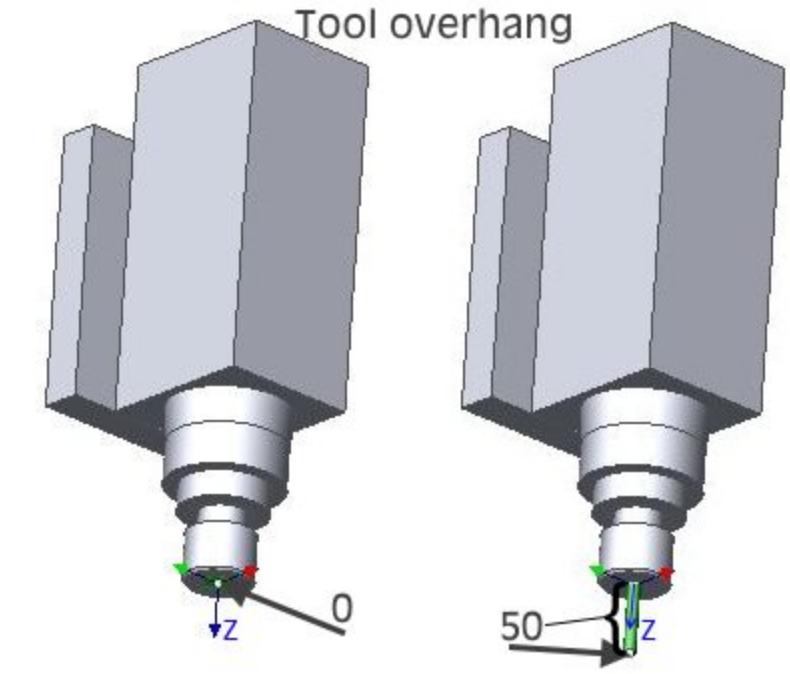
The tool coordinate system is defined by specifying the origin and the rotation angles of the tool CS relative to the base robot flange coordinate system, and the tool overhang.

1. Set up the position and the orientation of the tool head for the project

- As the first step perform the calibration of the tool coordinate system on the real robot following the instructions from the robot's manual. Ensure the positive Z axis of the tool CS looks down the direction of the tool overhang after the calibration.
- In CAM system at the Technology tab select the robot node in the job tree and at the Machine setup tab in the inspector enter the values you've just obtained after the Tool CS calibration into the `Machine dimensions.Tool` fields (see the first picture below). The values of the rotation angles A, B, C (W, P, R) usually need to be updated only when the tool head configuration is changed (the tool head is either modified or replaced).
- If a tool was used when calibrating the robot (see the second picture), enter the tool length into the Tool overhang field, otherwise leave this field as 0.
- The `Spindle 3D model` parameters define the additional transformation of the tool head 3d model relative to the tool coordinate system (for visualization and simulation). They do not affect the orientation of the tool and the resulting toolpaths in any way.
- If you want the entered values were used as the default values for the current robot in all new projects, you have to edit the robot's.xml file. For example:

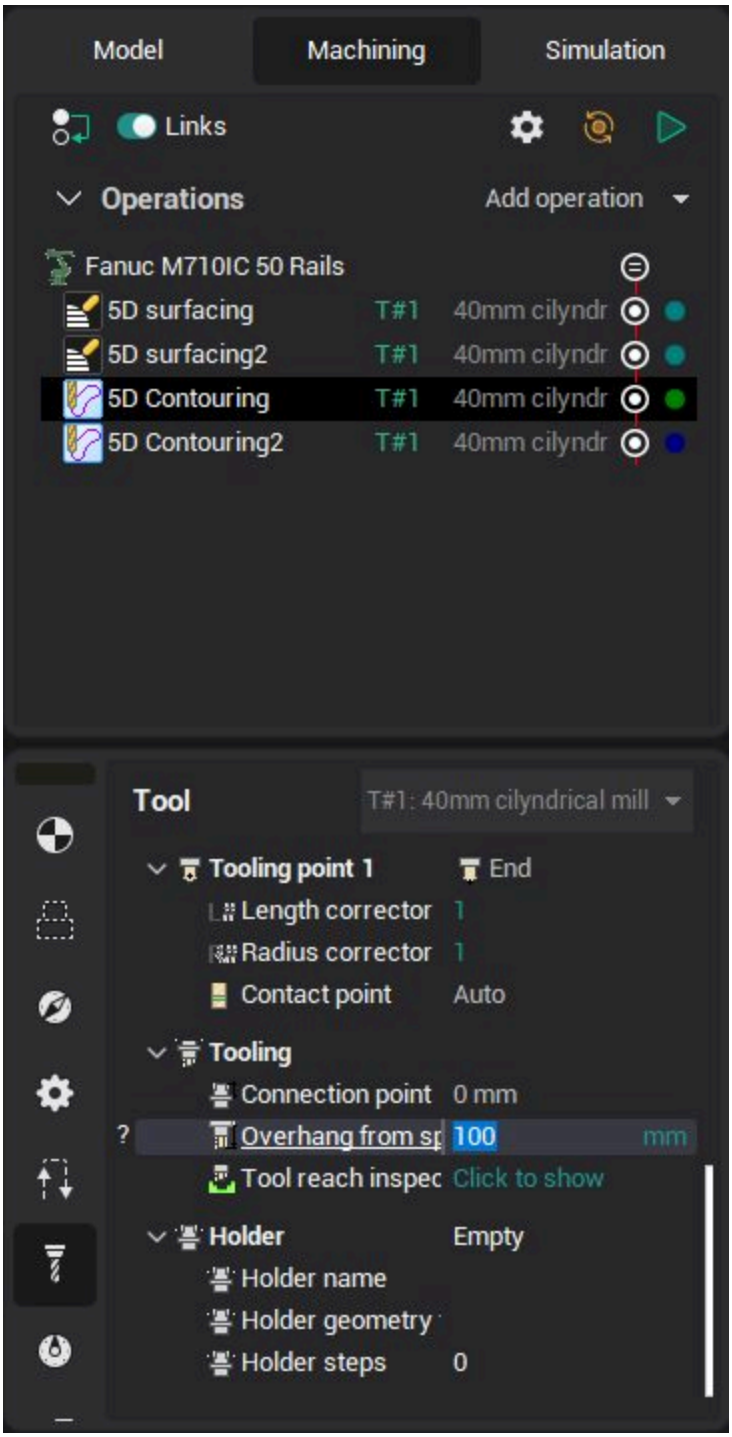
`X DefaultValue="134.83"/`).





2. Set the tool overhang for an operation

For every new tool you'll be using in the operations of the project you have to specify the tool overhang. To do so activate the Tool tab in the operation parameters inspector and click on the Overhang parameter (As shown in the picture below). The tool overhang (the lime dimension) and the three coordinates of the tool center point (TCP) in the tool flange coordinate system (X-red, Y-green, Z-blue) will be shown in the graphic view. You can edit the dimensions by clicking on them and entering the values with the keyboard or by the mouse wheel scrolling. All the four dimensions are interrelated, so when one of them is edited, the others are recalculated in such a way that only the tool overhang is changed.

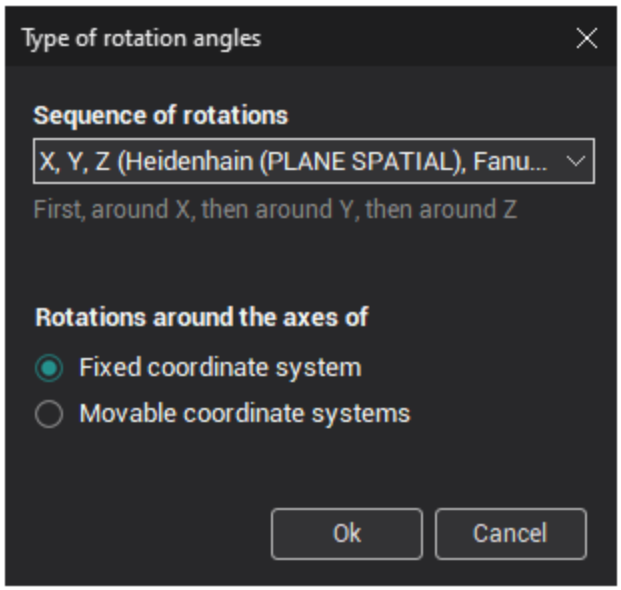


Different ways of defining coordinate systems

Robots from different manufacturers may have different ways of defining the coordinate system rotation angles. For example, the Fanuc robots have the order of rotations first around the X axis, then around Y, then around Z, while the KUKA robots have the order of rotations first around the Z axis, then around Y, then around X.

In CAM system the correct order of rotations for the specific robot is used by default, but if you want to use a non-standard rotation scheme to define a coordinate system you can always do so by selecting the rotation scheme in the Type of rotation angles window which is available in the Definition of new

coordinate system, Spatial transformations and Workpiece setup dialogs by pressing the corresponding button.



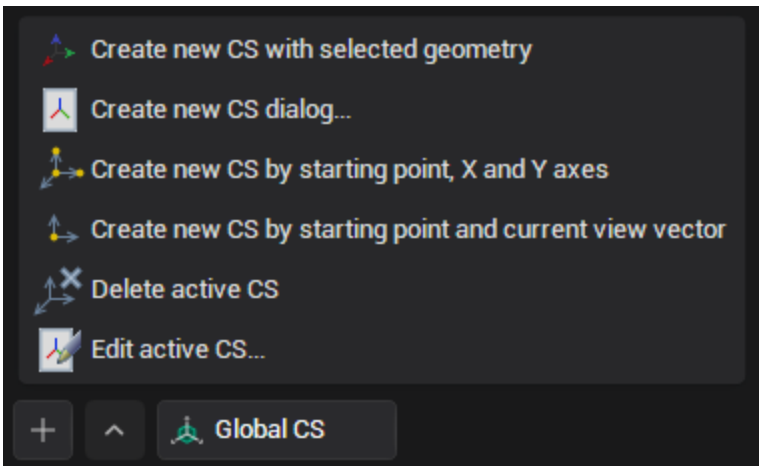
Defining the user coordinate system of the workpiece

Here is the recommended way.

1. On the real robot define the new user coordinate system by specifying three points on the workpiece (most robots support this way of defining a coordinate system). The first point is the CS origin, the second and the third points specify the directions of the X and Y axes. As the result you will get the id (the number) of the coordinate system and the coordinate system parameters XYZ ABC (WPR, RxRyRz, q1q2q3q4).

2. In CAM system create a new coordinate system using exactly the same approach you used on the actual robot:

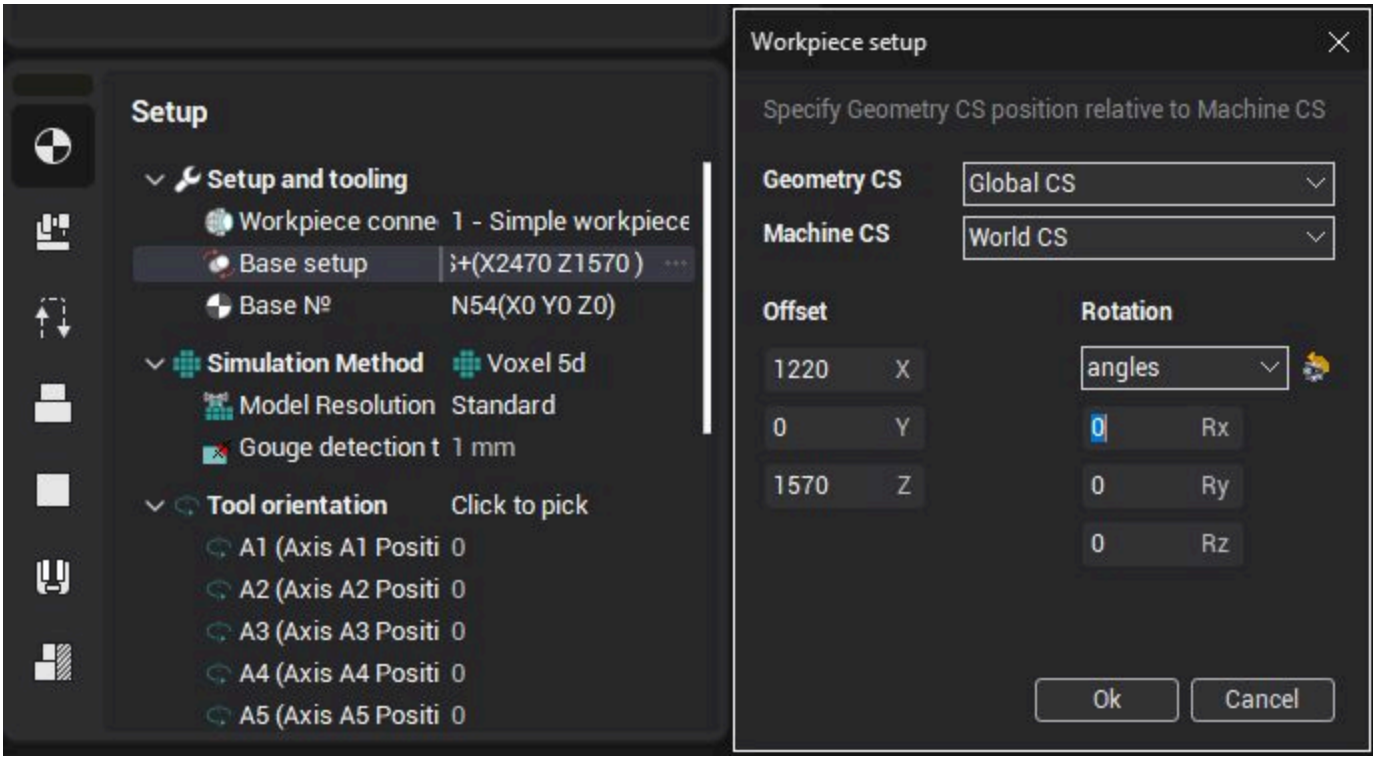
- Create a new coordinate system by selecting the Creation of CS by setting starting point CS and direction of X and Y axes command and click the three points on the model of the workpiece you have just been using when defining the user coordinate system on the real robot.

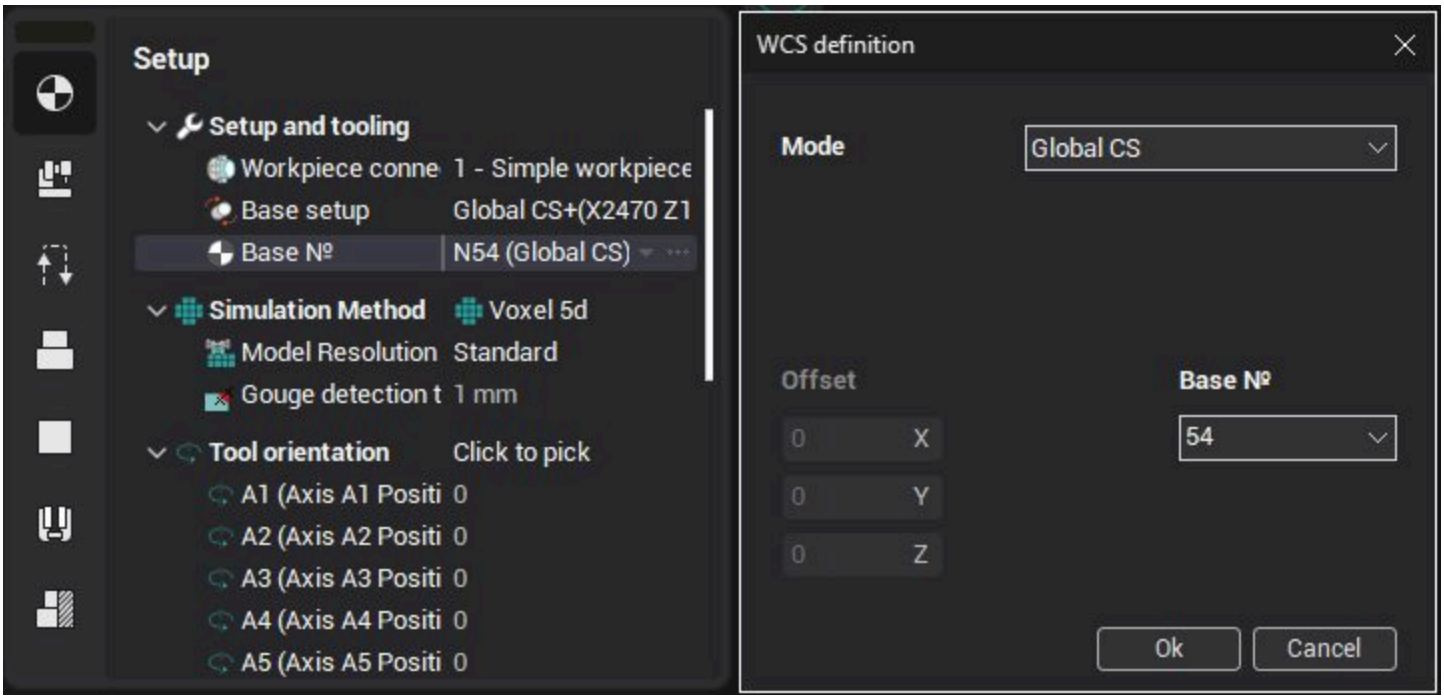


- In the Workpiece setup dialog select the newly created coordinate system as the Geometrical CS. In the boxes for the Translation and the Rotation enter the values XYZ ABC (WPR, RxRyRz, q1, q2, q3, q4) obtained on the robot.
- In the Workpiece CS dialog select the newly created coordinate system for the `Mode (way of assignment):`. Enter the number of the coordinate system that will be used in the program into the LCS number box. (Default is 54, because it is the standard coordinate system of most machines).

After that the position of the workpiece and the fixtures in CAM system have to reflect the position of the real workpiece relative to the real robot.

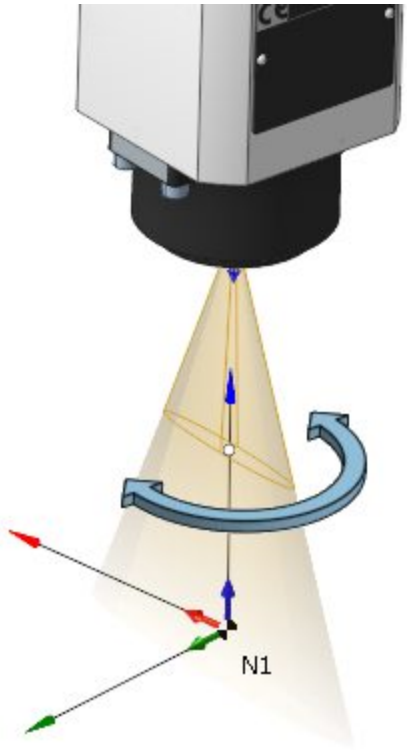
It is recommended to set up the user workpiece coordinate system in the root of the job tree (the very first item with the robot icon and caption in the tree) rather than in operations.





3. The position of the user coordinate system can be saved in the robot.xml file as the default value for the new projects. It will simplify the further use of CAM system as newly imported models of parts will be placed in more predictable positions relative to the robot.

Programming the robot's 6th axis

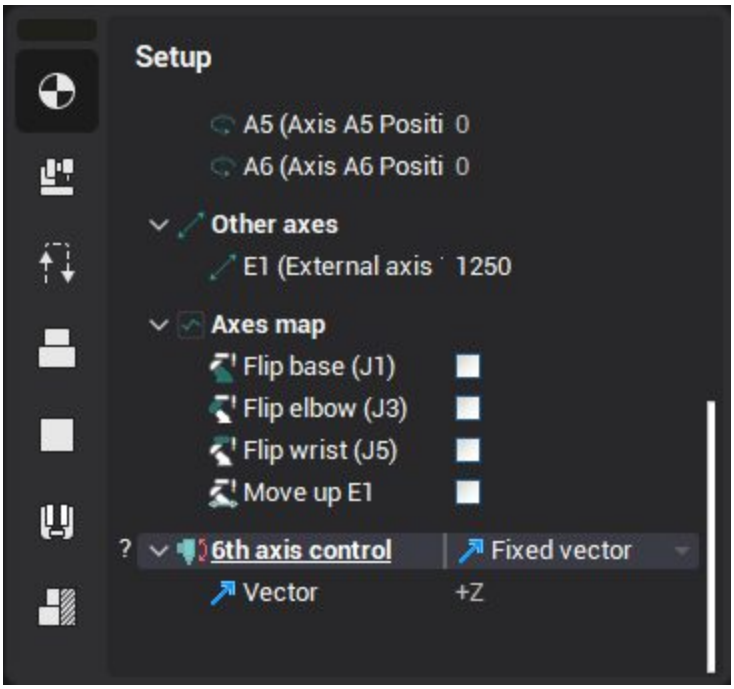


To position a rotating tool relative to the workpiece five degrees of freedom is enough, yet most of the robots have 6 degrees of freedom. The last 6th DOF is represented by an additional joint at the tool flange and is used to extend the robot's flexibility and the reach zone (by fixing the tool position and orientation and by changing the angle of the 6th joint the other joints of the robot are moving, and this helps avoid various types of kinematic and mechanical collisions when machining).

In CAM system there are two ways of programming the 6th axis:

- automatic,
- manual.

You can control it on the operation's properties inspector.

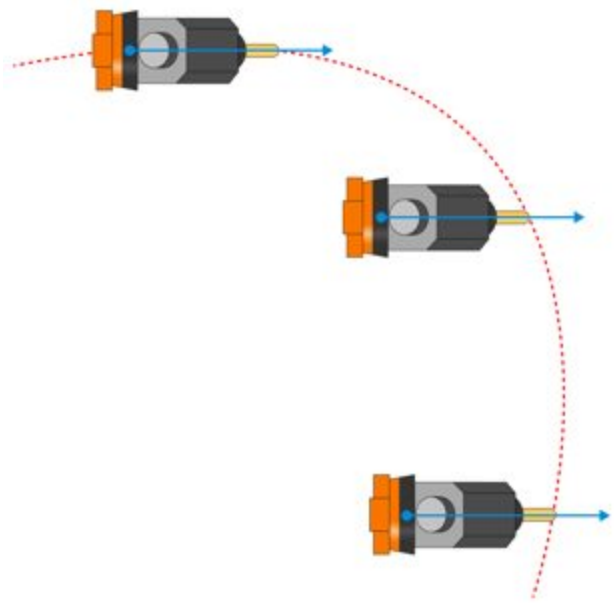


The automatic way is represented by the few modes of the 6th axis control property.

- Fixed vector,
- Direct to point
- Toolpath.

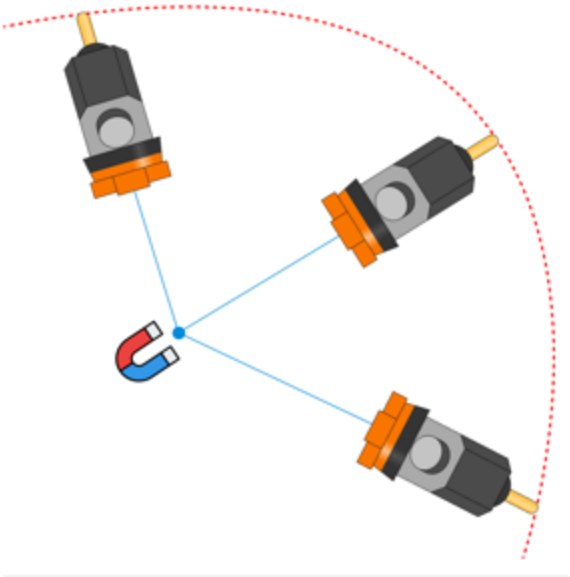
Fixed vector 6th axis control mode

In this mode you define the axis (the 3d vector) along which the Z axis of the robot's tool flange (the tool flange vector) is aligned.



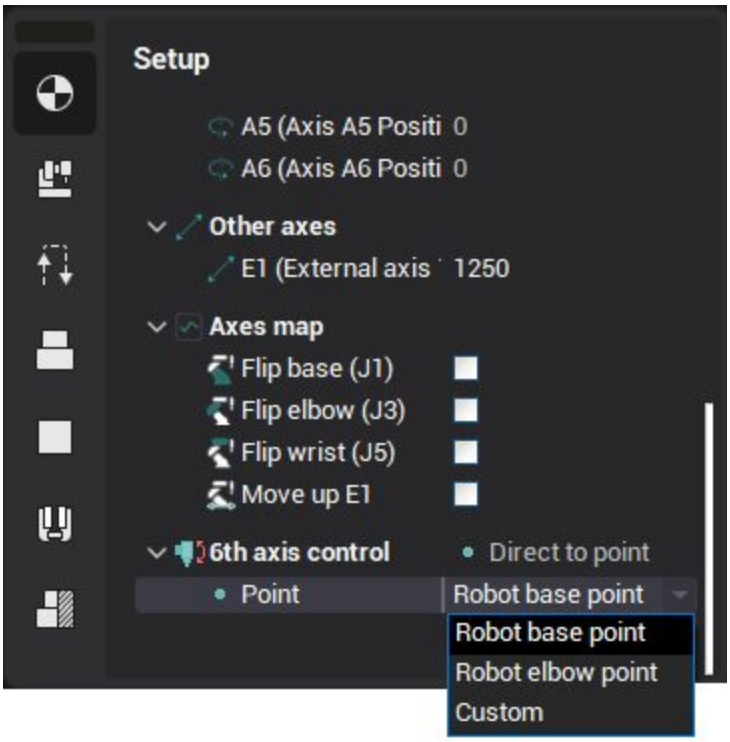
Direct to point 6th axis control mode

In this mode you specify a 3d point to which the robot's tool flange vector is directed during the machining.



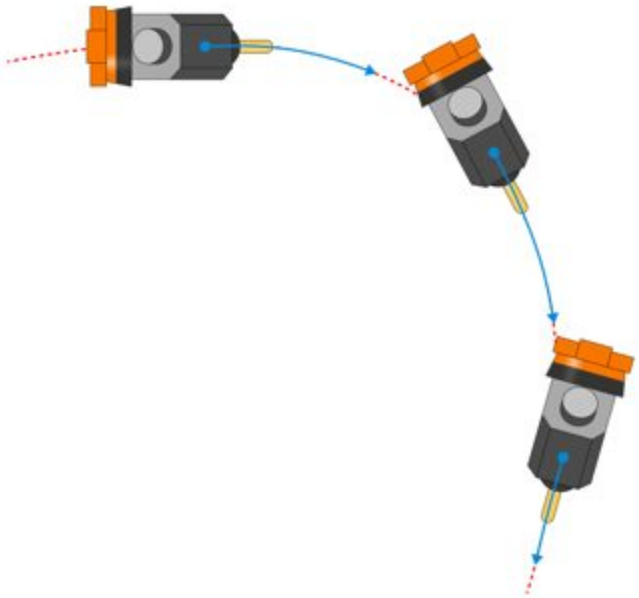
The point can be either

- the robot base point, or
- the robot elbow point,
- or a custom point.

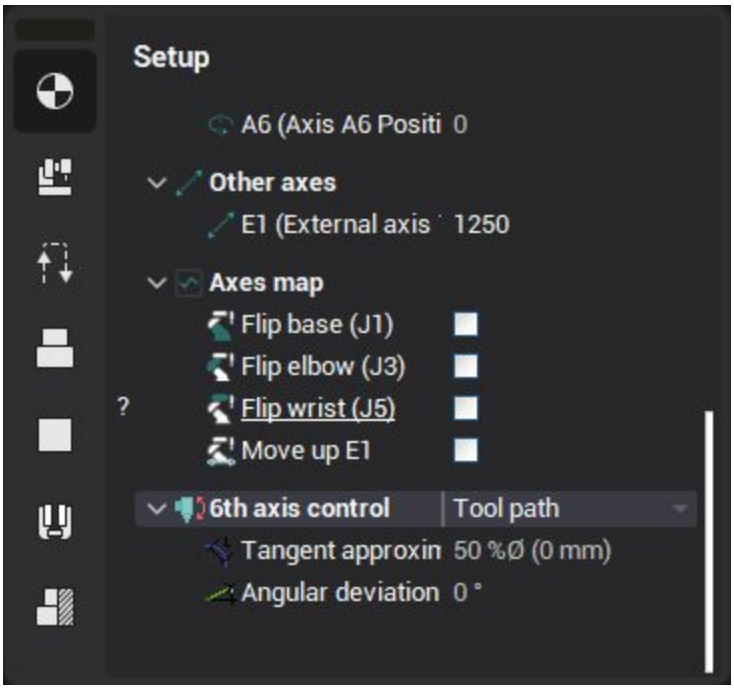


Toolpath 6th axis control mode

In this mode one axis of TCP is aligned with the toolpath tangent direction.

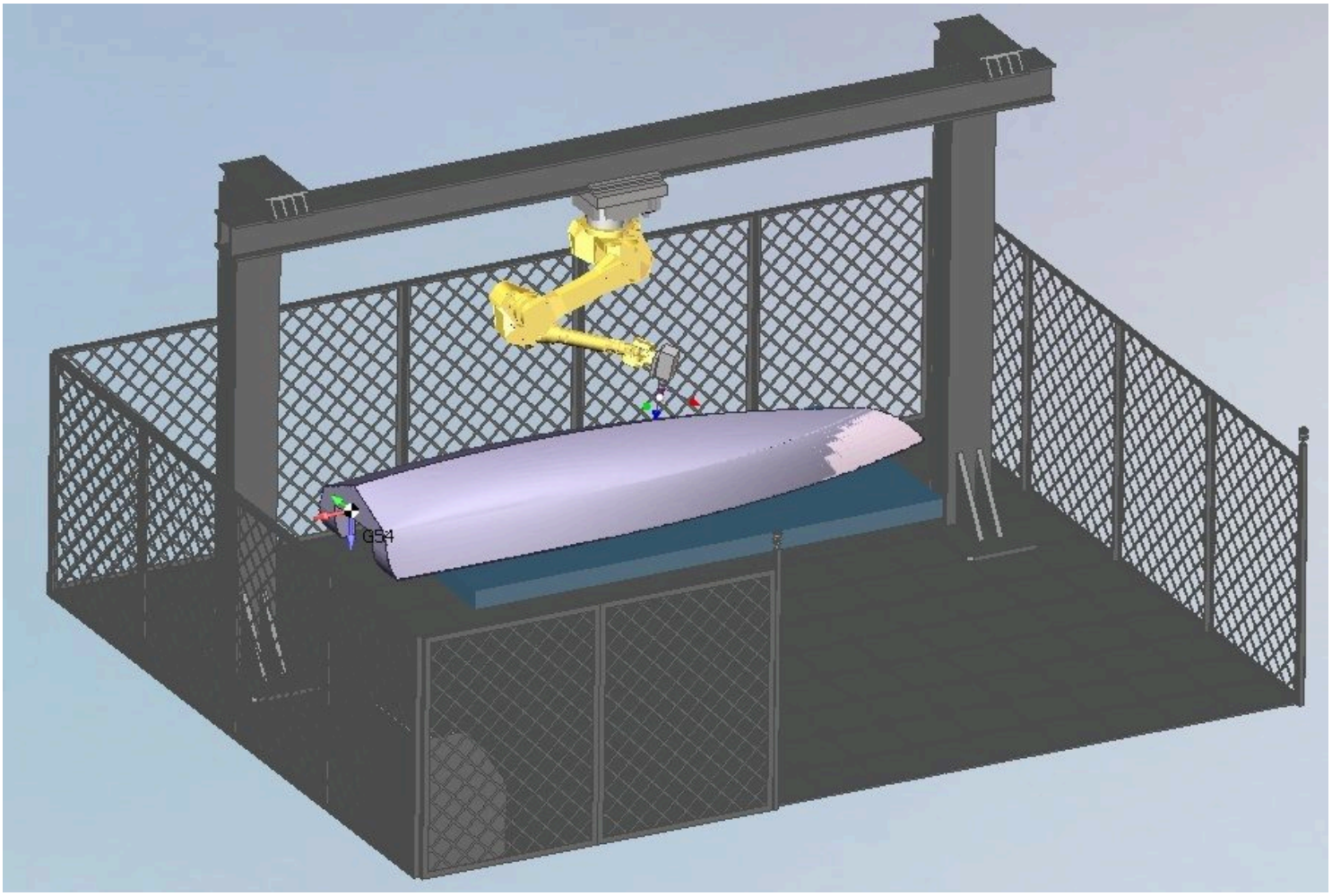


Optionally you can also define the Tangent approximation tolerance and the constant angular deviation relative to the tangent of toolpath at each point.



The manual way of programming the 6th axis is by using the Robot Axes Map (also known as the Robot Extra Axes Optimizer). It is possible to combine both approaches: use the automatic law for the most of the toolpath and apply additional corrections to the 6th axis control in the Robot Axes Map.

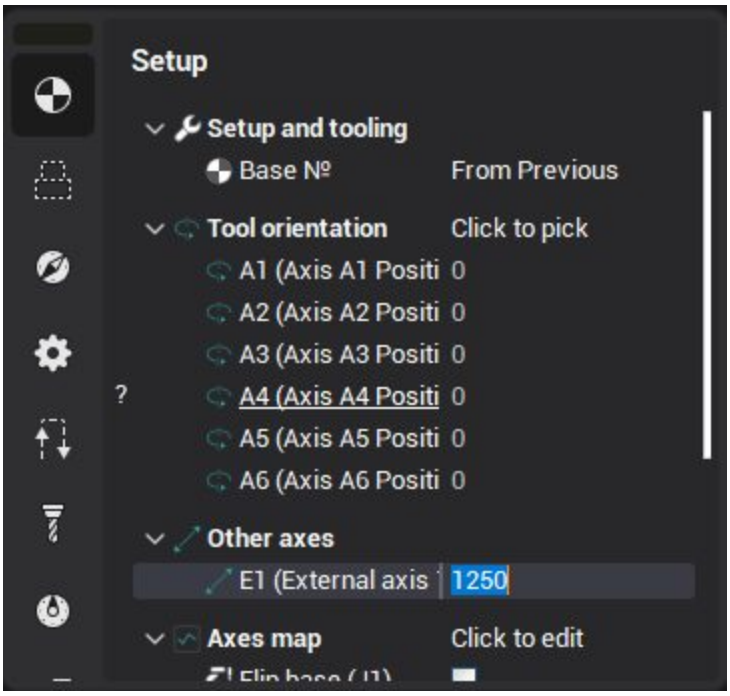
Programming the rails position



Often to increase the reach zone of an industrial robot, the robot is mounted on rails. CAM system allows simultaneous control of up to 3 rail positioners.

If the kinematic scheme of a robotic cell contains a rail or multiple rails, the following parameters become available for each rail in the operation parameters.

1. The initial rail position for the operation (E1 (External axis 1 position) in the picture).
2. The Move Up rail axis mode check box (Move up E1 check box in the picture)



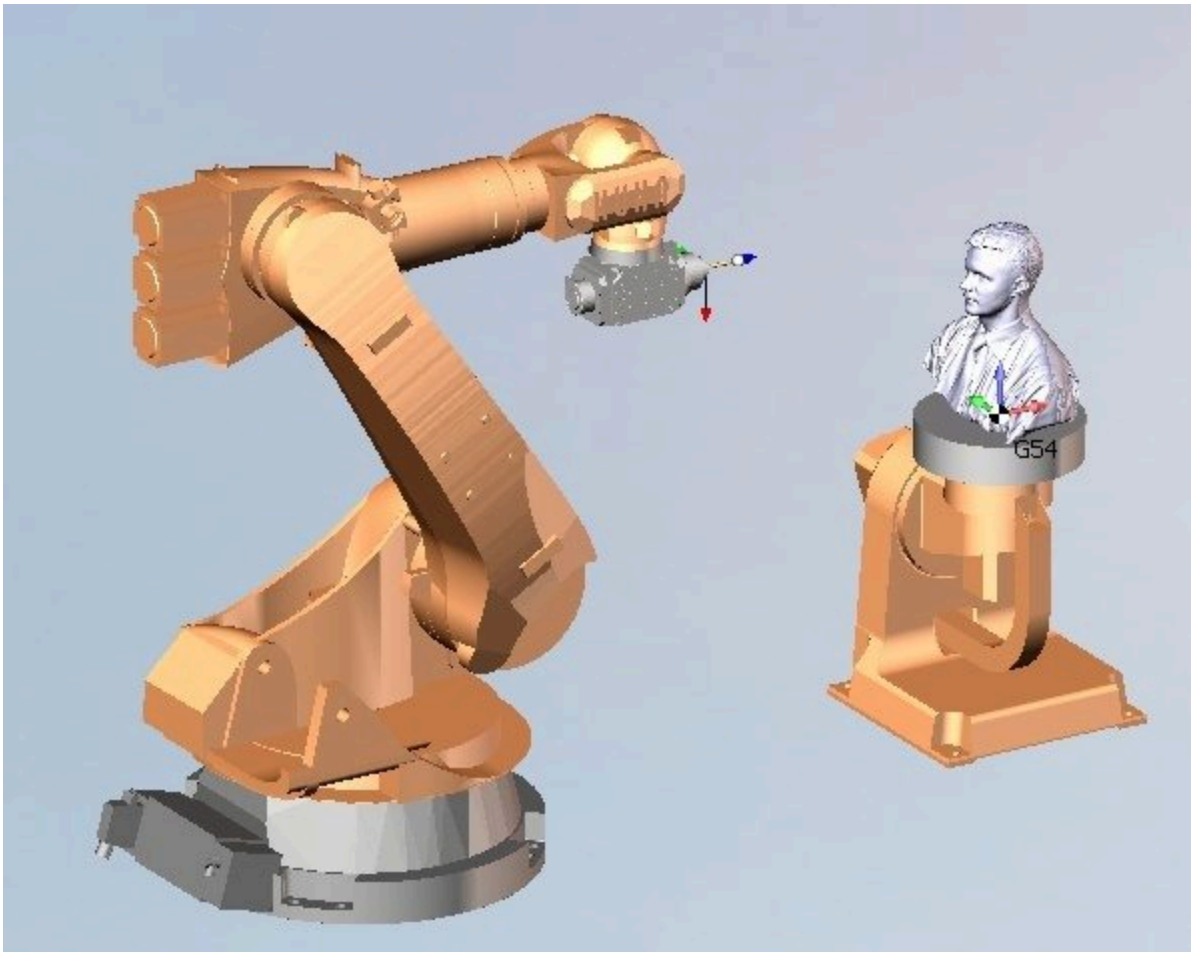
Move Up the rail axis

If this mode is enabled, the robot is automatically moved on the rail during machining in such a way that the current tool center programming point appears under the robot base zero point (or as close to it as possible).

Attention!

Having the tool center point just under the robot base zero point is not always optimal. This position may lead to singularities of the first joint. In this case it is necessary to adjust the rail position in the Robot Axes Map (adding an additional shift).

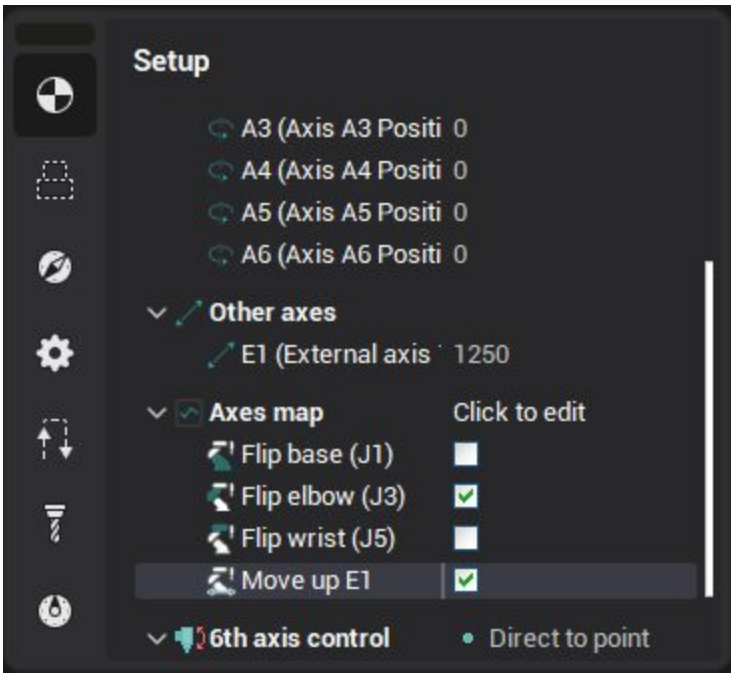
Programming the rotary table



A robotic cell can be equipped with a rotary or a tilting rotary table which have one and two additional rotary axes respectively.

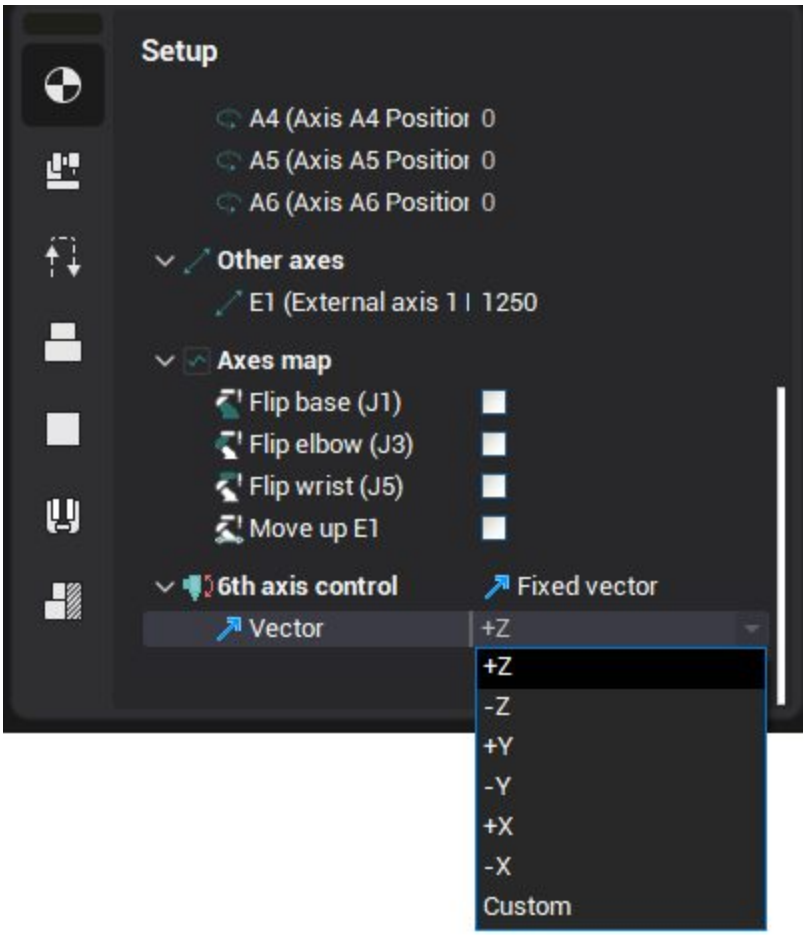
If the kinematic scheme of a robotic cell contains a rotary table, the following parameters become available in the operation setup.

1. The Rotary axes initial positions for the operation.
2. The Rotate rotary axis options.
3. The Flip table option.
4. Additional parameters for the rotary table control.

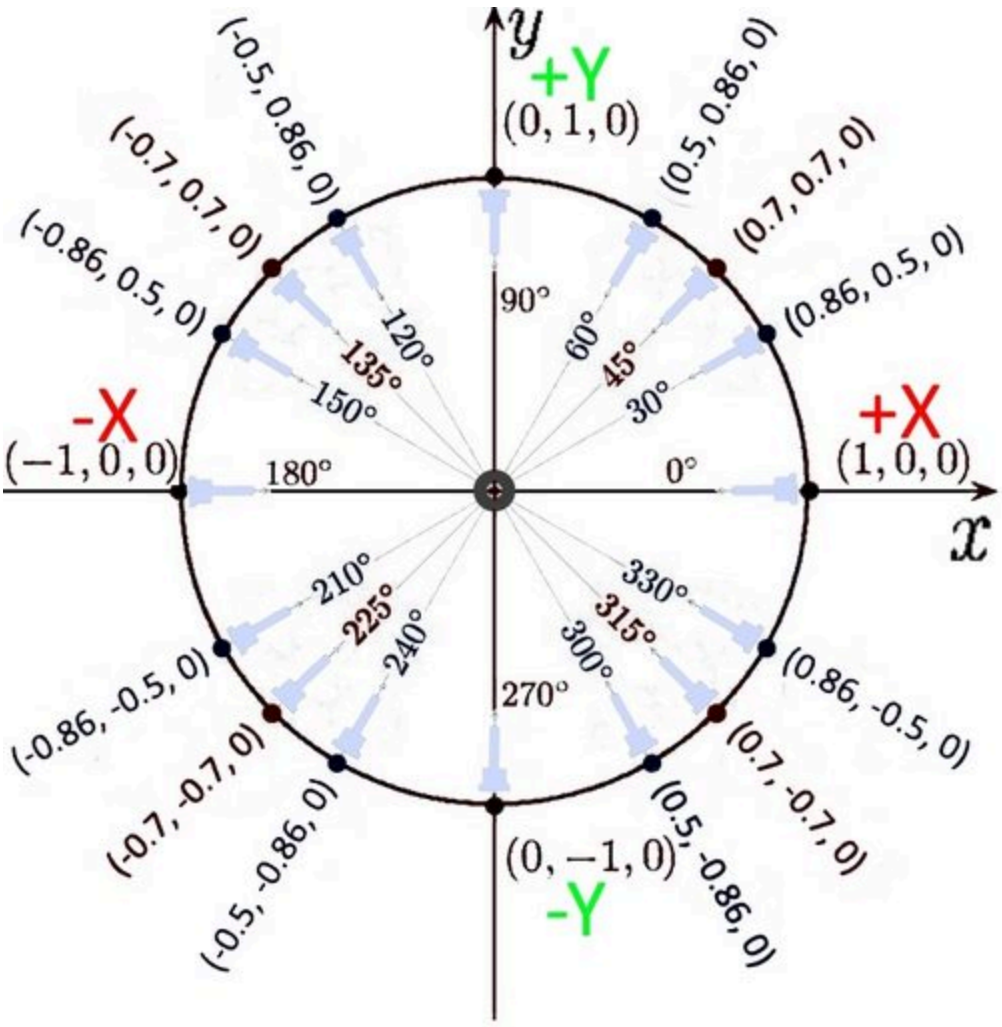


Rotate table parameters

The automatic table rotation feature allows to rotate table during the machining in such a way that the tool always stays on one side of the table. By using the Rotary table vector you can set on which side of the table the tool will be positioned (the direction vector of the tool axis to the center of the table in the global coordinate system of the robot).



In the diagram below the examples of a vectors for different angles are shown.



Robot axes map



Application area:

The Robot Axes Map allows to define manual and fine-tune automatic control laws for the excessive degrees of freedom of a robot (the 6th axis, the rails axes, the rotary table axes). The feature is available at the Operation Setup by pressing the ellipsis button next to the Robot axes map parameter.

The following axes/parameters can be controlled through axes map:

- Robot external axes (rotary table or rail). More precisely, these are the robot axes which affect the position of the tool or workpiece of the operation, and are not one of the joints (A1-A5 axes)
- Robot A6 axis
- Lead/Lean angle
- "C axis" of the 5-axis machine (see here for more information)
- Arbitrary machine axis - with the special flag. See "*Arbitrary machine parameter control*" section for more info

The principle of operation

Robot configuration

First you have to choose the robot configuration/state to be used in the operation. It's the same configuration/state that you can edit in the Operation Setup under the Robot Axes Map parameter. The robot configuration is defined by the "Flips" - the alternative positions of robot's joints (base, elbow, wrist) which allow the same position of the tool relative to the workpiece - and the modes of positioning of excessive axes such as rails and rotary tables (Move Up E1, Rotate E2).

Optimized axis

In the Optimized Axis combobox you select the axis that you define the control law for at the moment.

Collision map

Collision map is the visual representation of collision zones in the toolpath. Along the X axis is the position of the tool on the toolpath from the beginning measured by length. Along the Y axis is the value of the optimized axis. The following types of collisions are detected.

1. Out of reach zones (maroon) are zones which the robot can not reach.
2. Out of limits zones (purple) are zones which the robot can reach but some of the robot's axes go out of its' defined limits.
3. Singularity zones (yellow) are zones where the robot's joints move at extreme speeds.
4. Collision zones (orange) are zones where the robot parts collide with each other or with the workpiece.

The map resolution can be set in the Resolution box as the amount of steps along the Y axis to be calculated.

To build or refresh the map hit the Build map button.

Check part and workpiece

Under the 'Collisions' checkbox there are 2 checkboxes for specifying whether part and/or workpiece are accounted for in the collision detection. They are enabled only if the parent 'Collisions' checkbox is set to 'True'. It is advisable to switch them depending on the type of machining - cutting, additive machining, or welding.



Axis control spline

The axis control law is defined as a spline curve. By default there is no spline. It means the optimized axis value either stays the same (in manual mode) or is controlled automatically (in the automatic mode) for the entire toolpath (the horizontal gray dashed line).

The goal is to create the nicest curve possible that passes through the collision-free white zone from left to right.

To create a new spline double-click on points in the empty space the spline has to pass through. Two points is enough. After the spline is created you can change it's shape by dragging the spline control points with the mouse, by deleting the control points (by right clicking on a point) or adding new points (just pull the spline with the mouse).

Use the Clear button to erase the spline.

Use the Build automatically button to build the collision avoiding spline automatically.

Verification

In this area the status of the current toolpath is displayed. If there are no collisions in the toolpath the green Ok is displayed, if there are any collisions, the number of collisions is displayed in red. The status is updated every time you change the axis control spline.

At the same time in the graphic view the collision zones in the actual toolpath curve are also marked with thick strokes of the corresponding collision type color.

You can either click on the empty space in the axes map or move the mouse while holding the left mouse button to position the tool to the corresponding position of the toolpath.

Regenerate toolpath

To apply the axis control law to the toolpath click the Regenerate toolpath button.

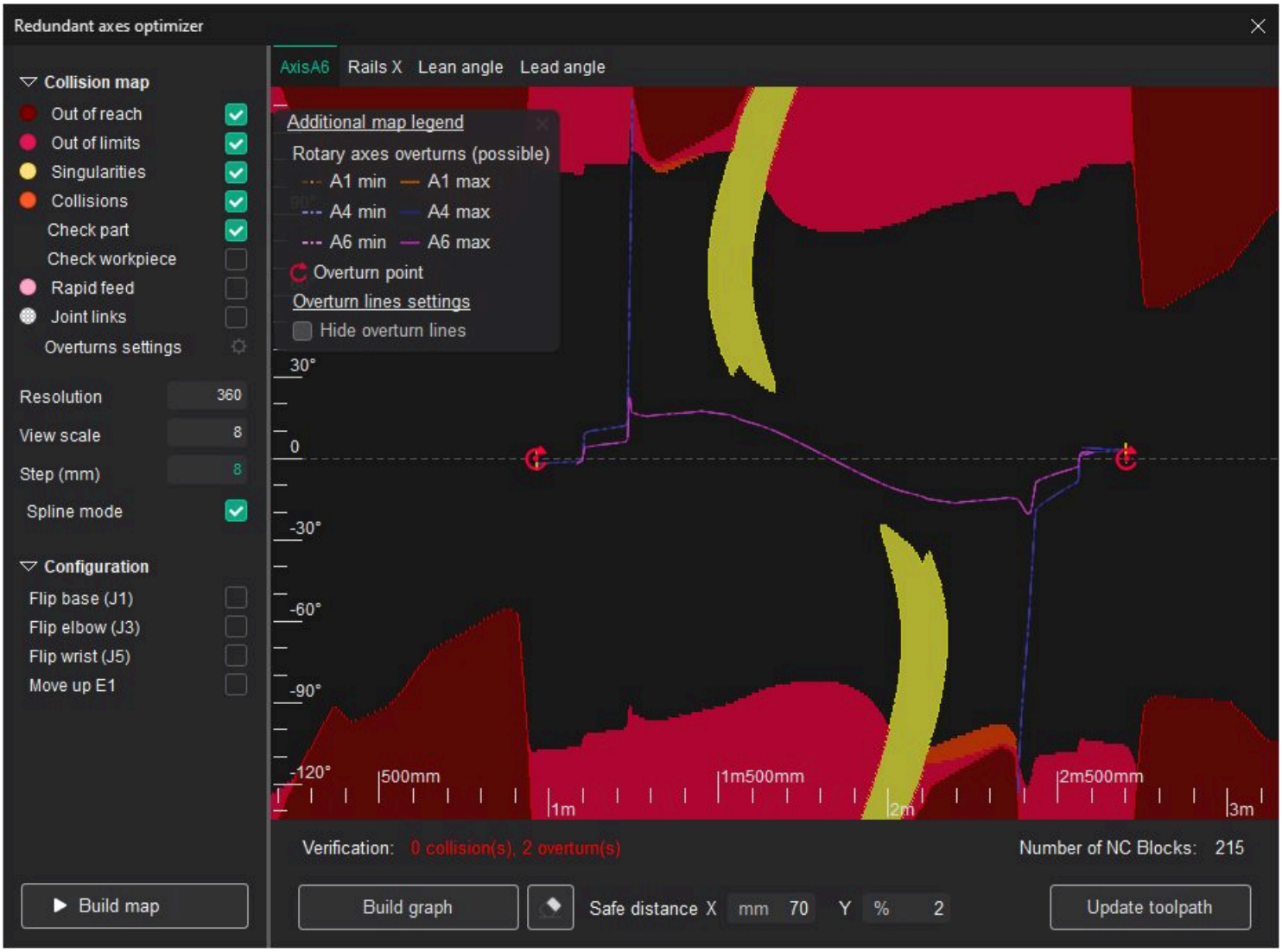
Rotary axes overturns detection and avoidance

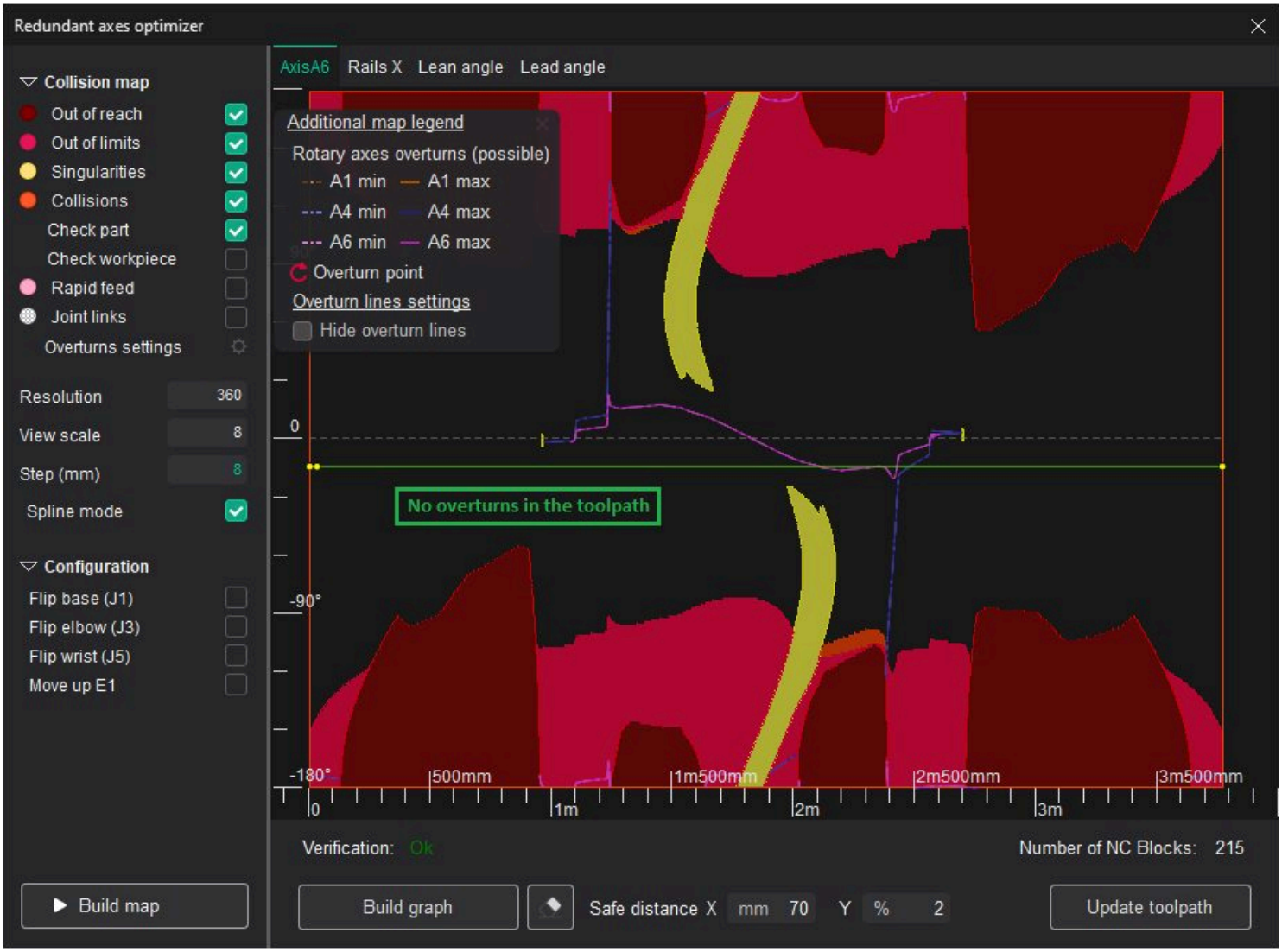
It is possible to use the redundant axis map to check for the potential problems in the toolpath caused by the rotary axes overturns. Overturns happen if the rotary axis reaches one of its limits and in order to continue machining it needs to do one full rotation (360°) forward or backward. In previous versions no information about the overturns was available to user because, despite the overturn, the axis always stays within its limits.

The light dashed and dark solid lines show the possible overturn locations in the toolpath in case the spline intersects with them. Different rotary axes, which can have overturns, have different line colors. The intersection doesn't always correspond to overturn; the true overturn locations are additionally highlighted on the spline as bold red points with the "overturn" sign. Also if there are overturns in the toolpath, their count is displayed in the "Verification" status bar. There can be 2 types of overturns:

- the overturn happens after the axis minimum is reached (shown with the dashed lines)
- after the axis maximum is reached (solid lines)

> Toggle the Show map legend check box to view the overturn lines color/style info and other additional map notations


Redundant axes optimizer
Collision map
Out of reach
Out of limits
Singularities
Collisions
Check part
Check workpiece
Rapid feed
Joint links
Overturns settings
Resolution 360
View scale 8
Step (mm) 8
Spline mode
Configuration
Flip base (J1)
Flip elbow (J3)
Flip wrist (J5)
Move up E1
Build map
AxisA6
Rails X
Lean angle
Lead angle
Additional map legend
Rotary axes overturns (possible)
A1 min
A1 max
A4 min
A4 max
A6 min
A6 max
Overturn point
Overturn lines settings
Hide overturn lines
30°
0
-30°
-60°
-90°
-120°
500mm
1m
1m500mm
2m
2m500mm
3m
Verification: 0 collision(s), 2 overturn(s)
Number of NC Blocks: 215
Build graph
Safe distance X mm 70 Y % 2
Update toolpath



Screenshot 1. The toolpath contains 2 overturns

Screenshot 2. The spline was moved lower, so there are no intersections with the overturn lines. As a result, there are no overturns in the toolpath now.

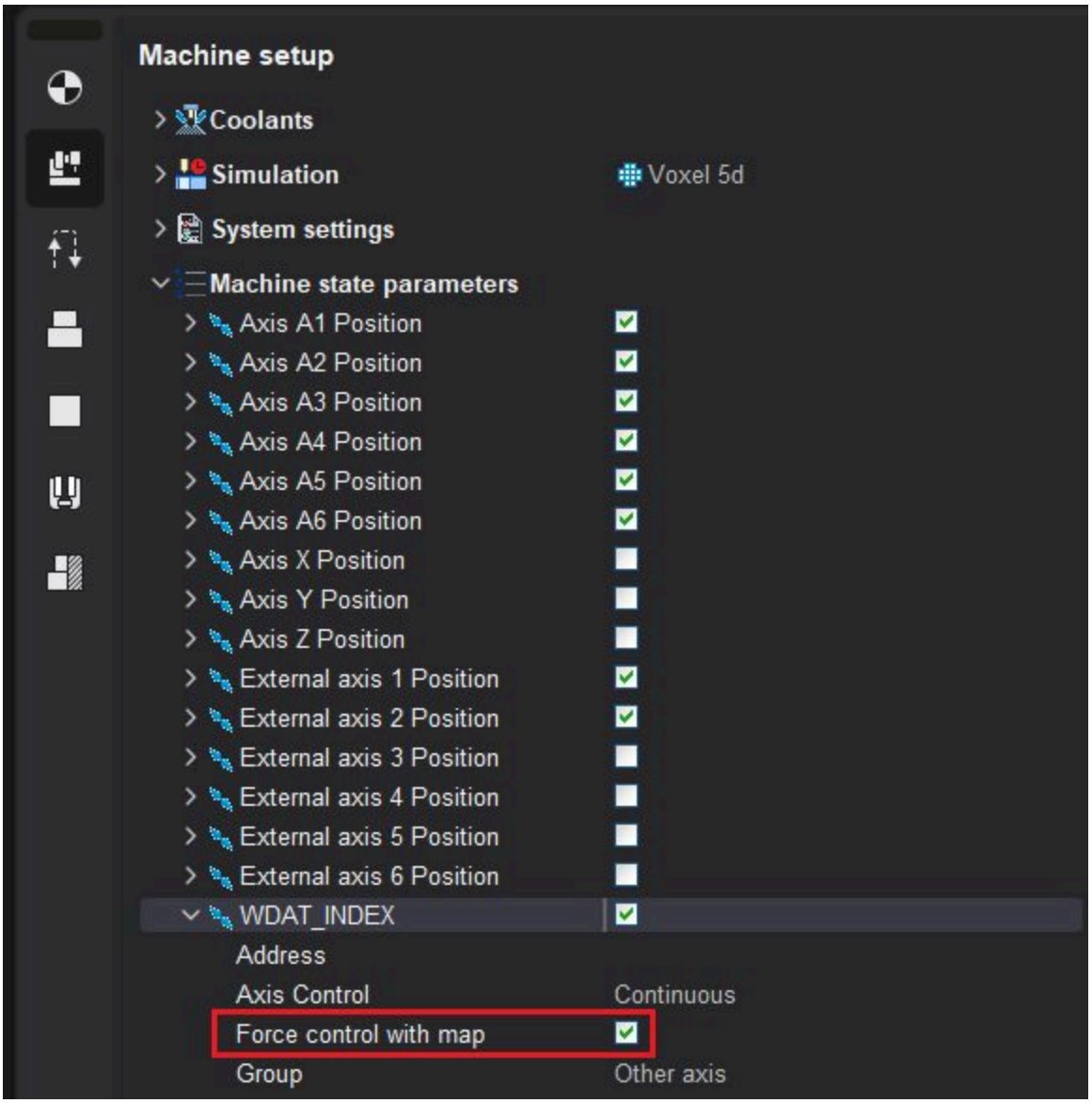
If the operation's toolpath contains overturns, you can try to avoid them by moving the spline so it doesn't intersect the given lines or the intersections are "fake" (the rotary axis value did not reach its limit yet in this point, but its value is equal to the axis min/max modulo 360°).

On the screenshot above, the spline was changed to a straight line, which doesn't intersect any of the blue/purple lines, which allowed to get rid of the overturns in the toolpath. Also changing the robot configuration (the "Flip elbow", "Flip wrist" parameters) might also help to avoid the overturns.

Arbitrary machine parameter control

If you need to control the changes of an arbitrary parameter (defined as an axis in the machine schema), you can enable the Force control with map flag in the corresponding machine state parameter.

The flag is available for each machine state parameter in the Machine Setup → Machine state parameters section of the inspector. After this the axis will appear in the axes map window, and, as usual, you can define parameter value in each toolpath point using spline.



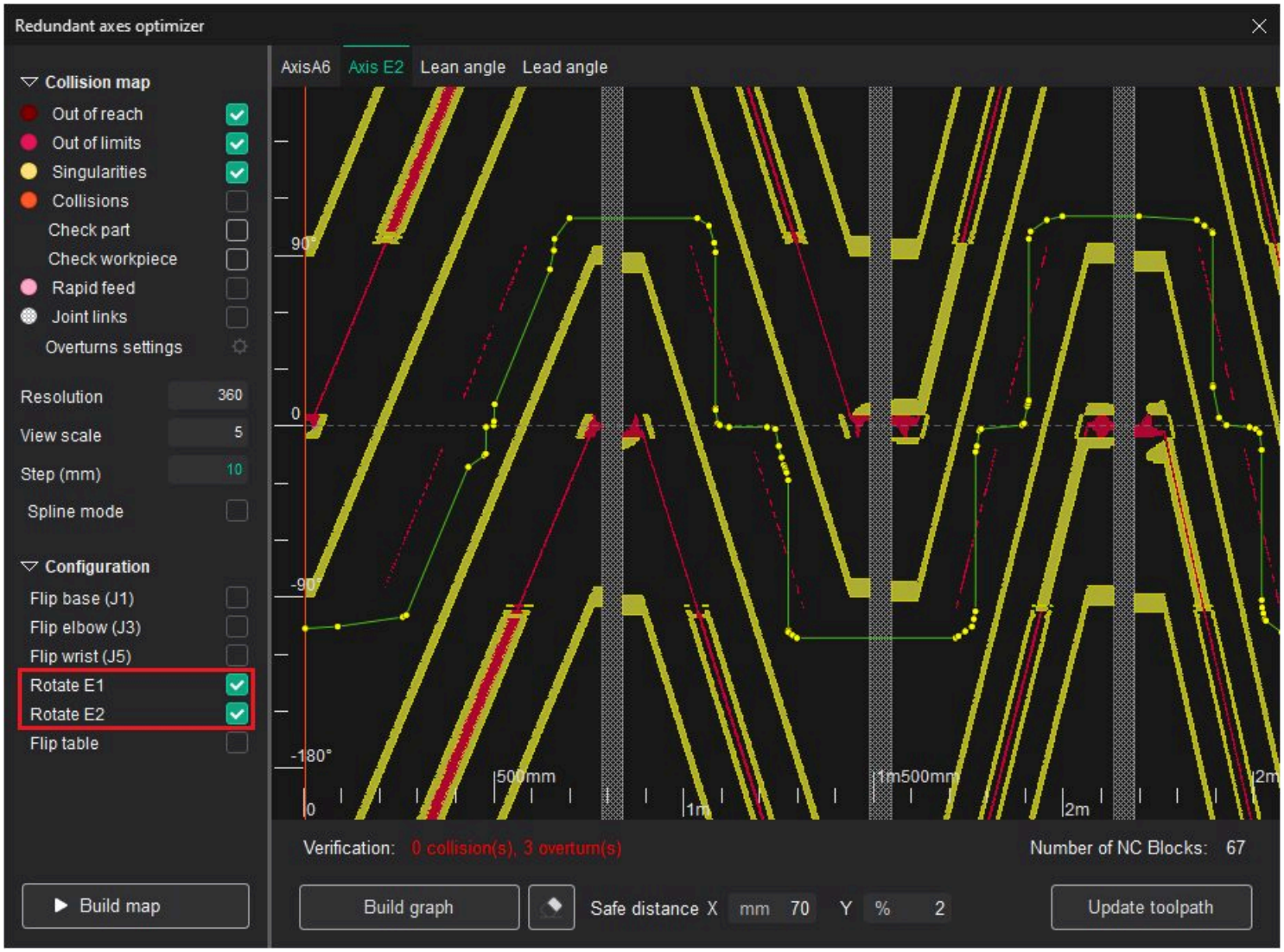
You can also set this flag directly in the xml-file of the machine in the Machine state parameter, which corresponds to the given axis.

```
                    Enabling axis control with the map in xml
<SCType ID="WDATPOS" Caption="WDAT_INDEX" type="TMachineStateParameter">
        ...
        <ControlWithMap DefaultValue="True"/>
</SCType>
```

Singularity avoidance for the 2-axis rotary table of the robot

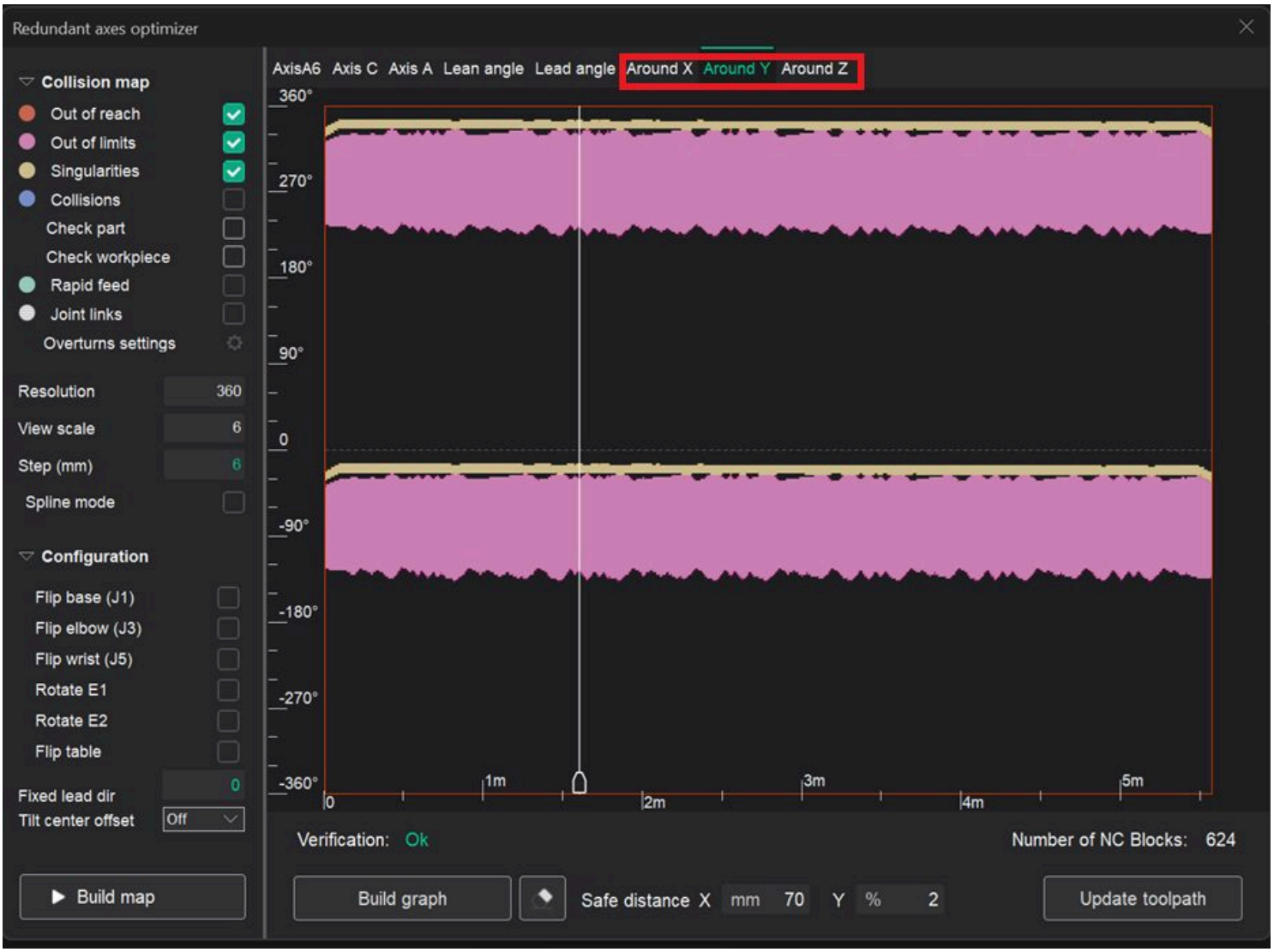
Previously, If you needed to define the trajectory of a 2-axis rotary table of the robot, and both of the rotary table flips are enabled (commonly named as Rotate E1 and Rotate E2), the resulting trajectory could contain singularity zones, causing very sharp changes of the rotary table state. That's why the special mode is enabled in the axes map in this case, similar to the Axes map for 5-axis machines. In this mode you need to define the trajectory only for one of the rotary table axes, the value for the other will be computed automatically to minimize the deviation from the tool normal defined by the operation.

The white vertical zones in the map correspond to the singularity zones. Using spline you can define the E2 axis trajectory there without abrupt changes.

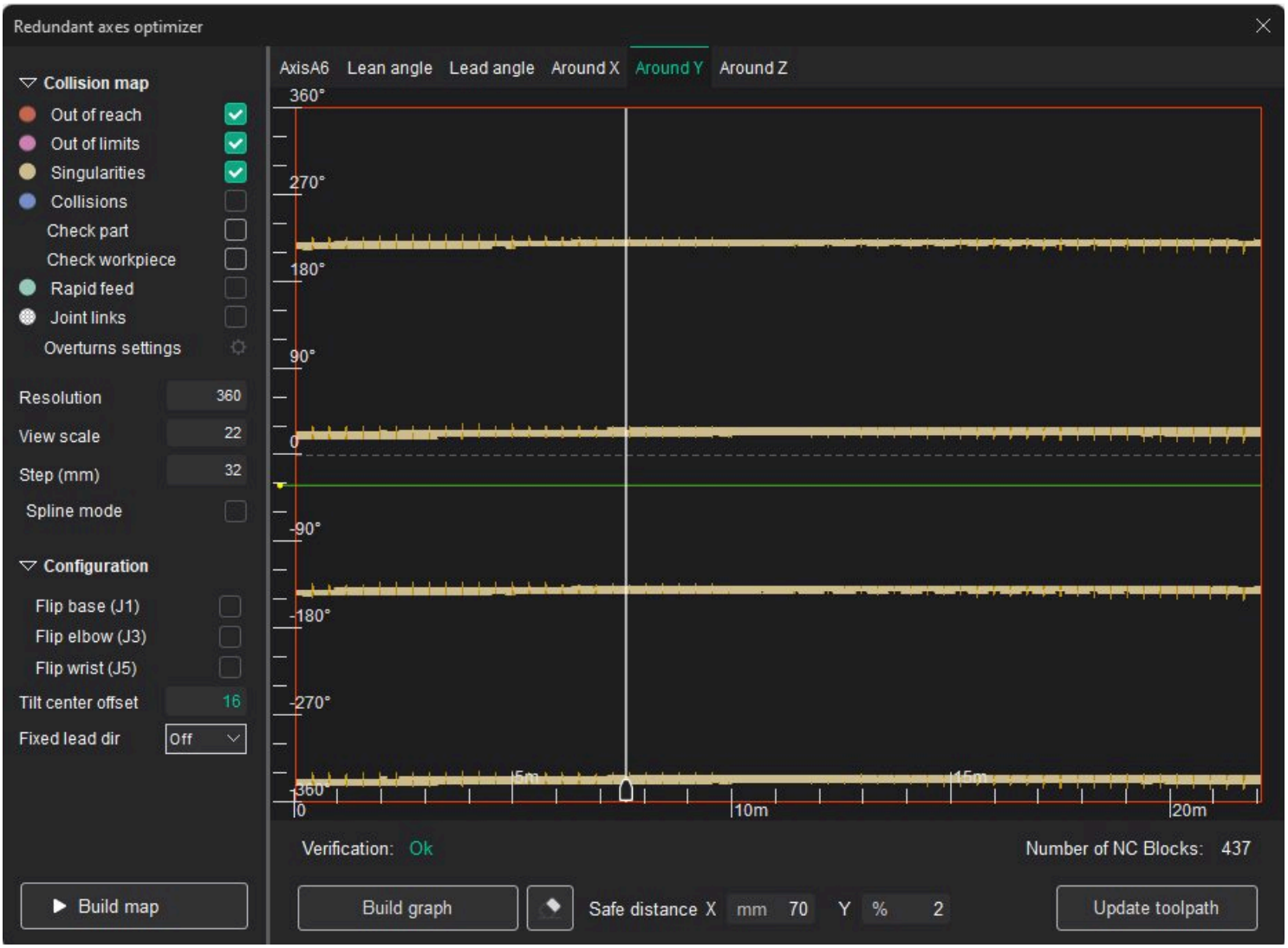


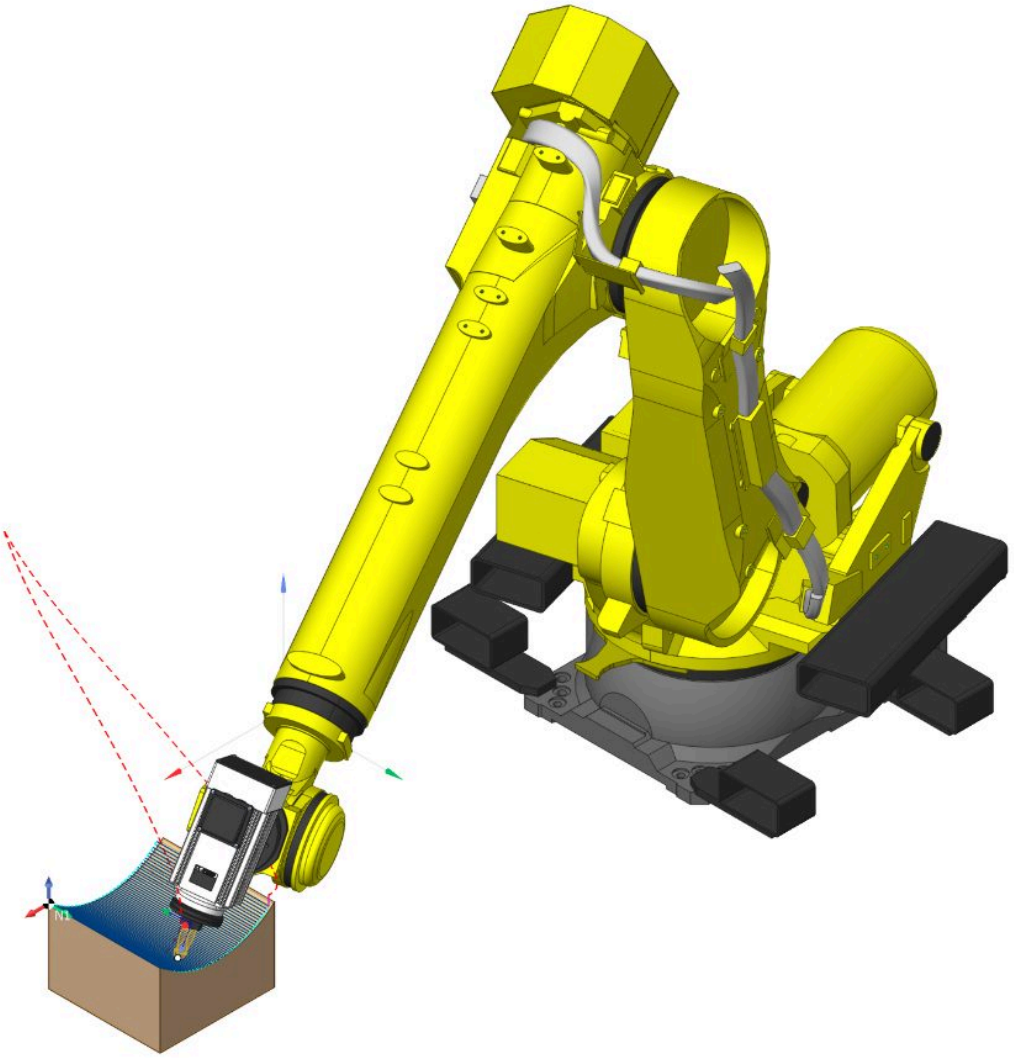
Rotate toolpath by LocalCS in Robot Map feature.

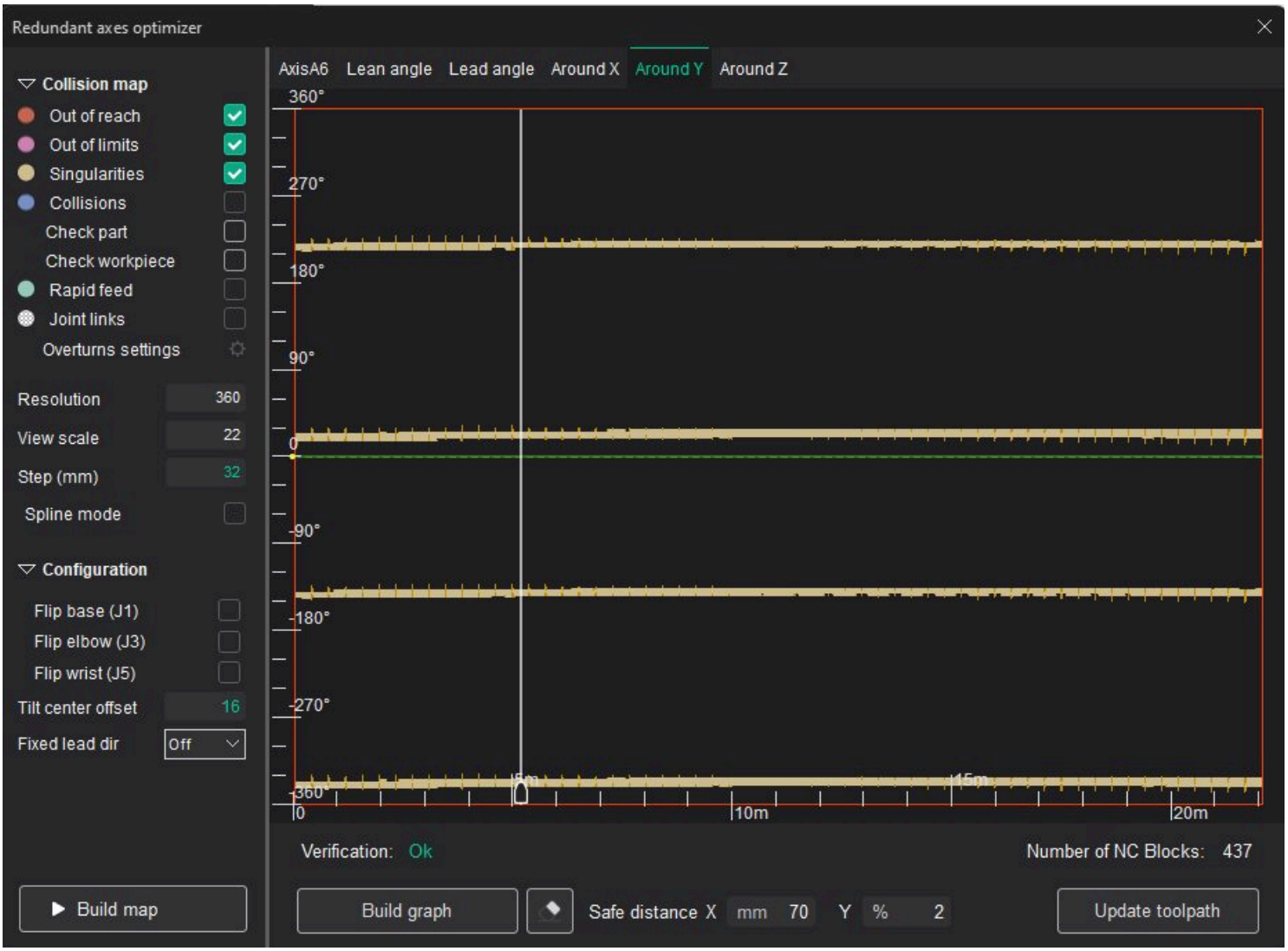
It allows you to construct rotation maps relative to a fixed coordinate system.

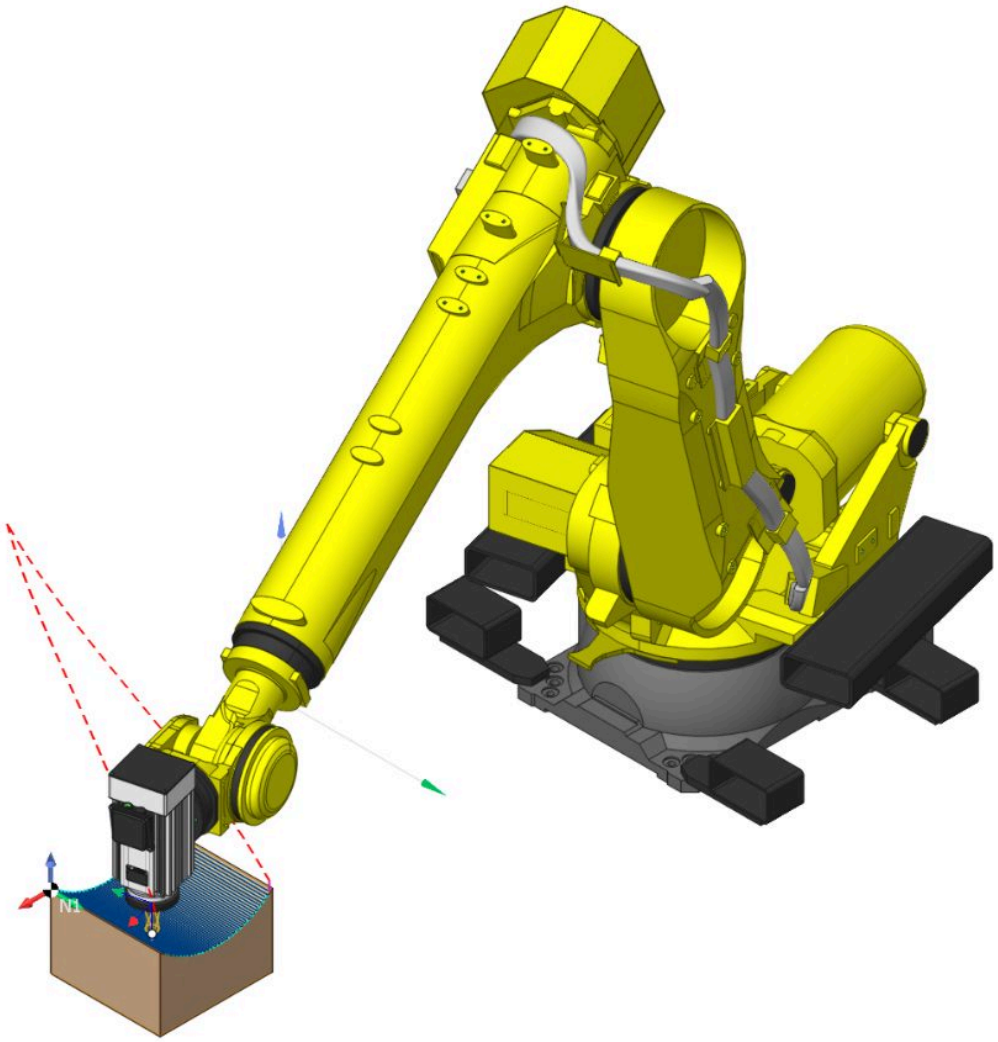


Below is an example of rotation of the axis Y.


Redundant axes optimizer
Collision map
Out of reach
Out of limits
Singularities
Collisions
Check part
Check workpiece
Rapid feed
Joint links
Overturns settings
Resolution 360
View scale 22
Step (mm) 32
Spline mode
Configuration
Flip base (J1)
Flip elbow (J3)
Flip wrist (J5)
Tilt center offset 16
Fixed lead dir Off
Build map
AxisA6
Lean angle
Lead angle
Around X
Around Y
Around Z
360°
270°
180°
90°
0
-90°
-180°
-270°
-360°
0
5m
10m
15m
20m
Verification: Ok
Number of NC Blocks: 437
Build graph
Safe distance X mm 70 Y % 2
Update toolpath



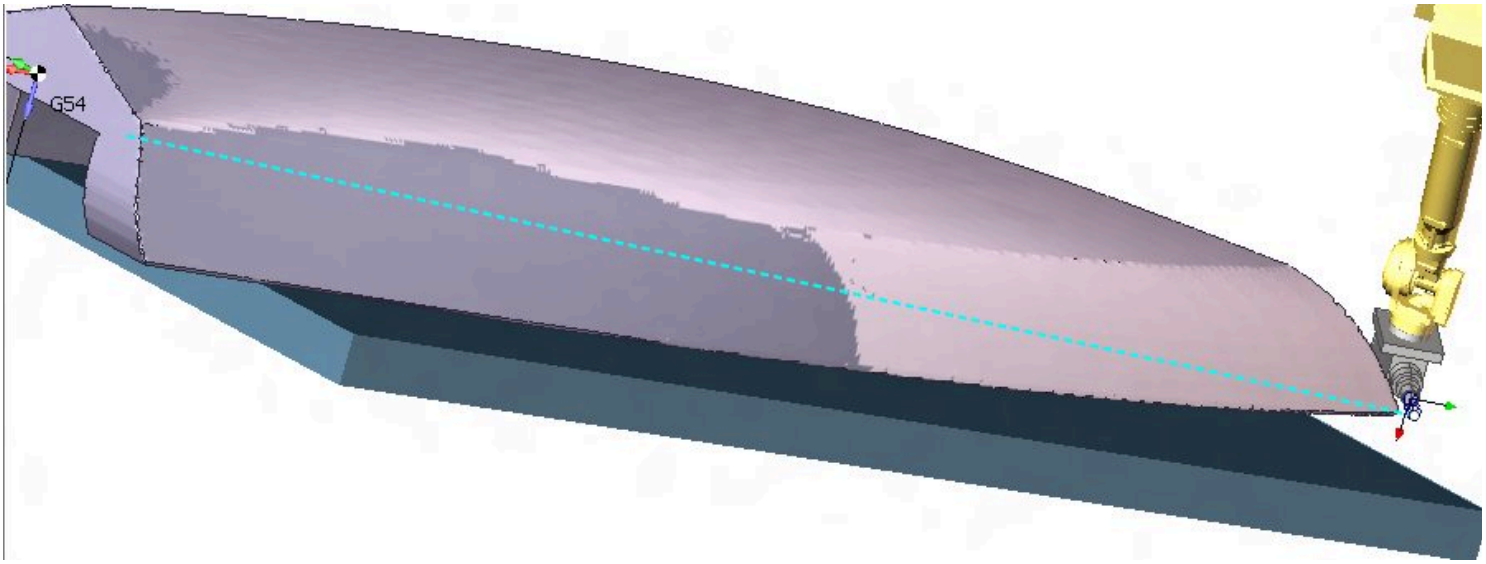

Redundant axes optimizer
Collision map
Out of reach
Out of limits
Singularities
Collisions
Check part
Check workpiece
Rapid feed
Joint links
Overturns settings
Resolution 360
View scale 22
Step (mm) 32
Spline mode
Configuration
Flip base (J1)
Flip elbow (J3)
Flip wrist (J5)
Tilt center offset 16
Fixed lead dir Off
AxisA6
Lean angle
Lead angle
Around X
Around Y
Around Z
360°
270°
180°
90°
0
-90°
-180°
-270°
-360°
0
10m
20m
Verification: Ok
Number of NC Blocks: 437
Build map
Build graph
Safe distance X mm 70 Y % 2
Update toolpath


N1

Programming robot's transitions (obsolete method)

The programming of the robot's transitions between operations is done by specifying the Approach and Return points in the Operation Setup. It can be simply done through the Motions editing toolbar in Simulation mode.

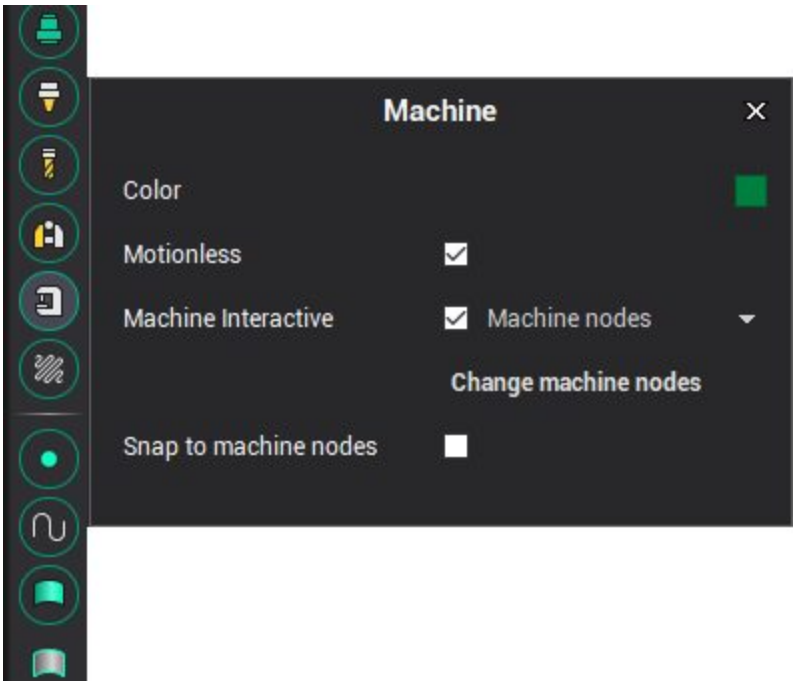
Let's take a look at the following example.



After the operations had been calculated, the simulation has revealed that the transition between two operations gouges the part. To modify the transition the list of intermediate points the robot will pass through should be formed.

The best way of doing it is by using the Machine control panel.

But first, make sure the Machine Interactive mode is enabled.



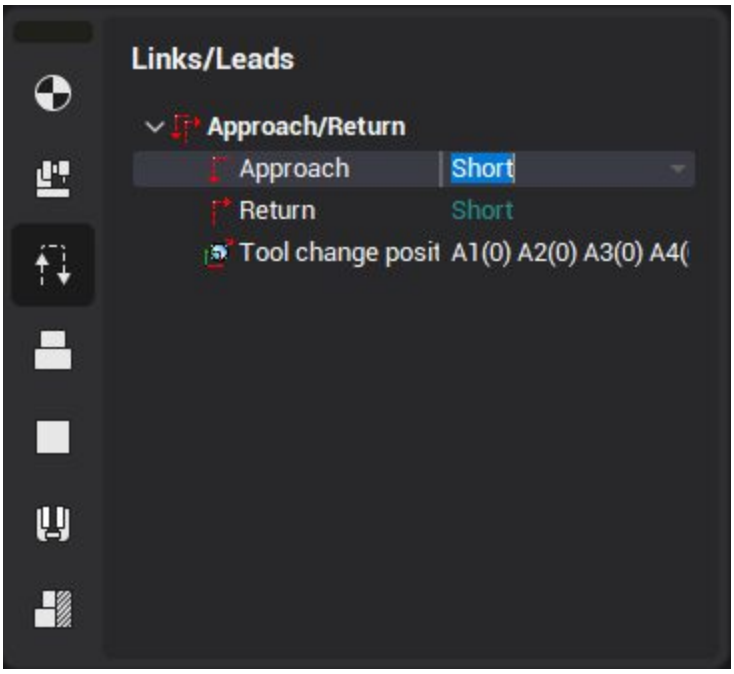
At the Simulation tab select the last command of the first operation (5D Contouring) as shown in the picture below. To define the first control point drag the tool to the desired position with the mouse.



Then hit the States panel visibility button in the Machine control panel to save the position into the memory.

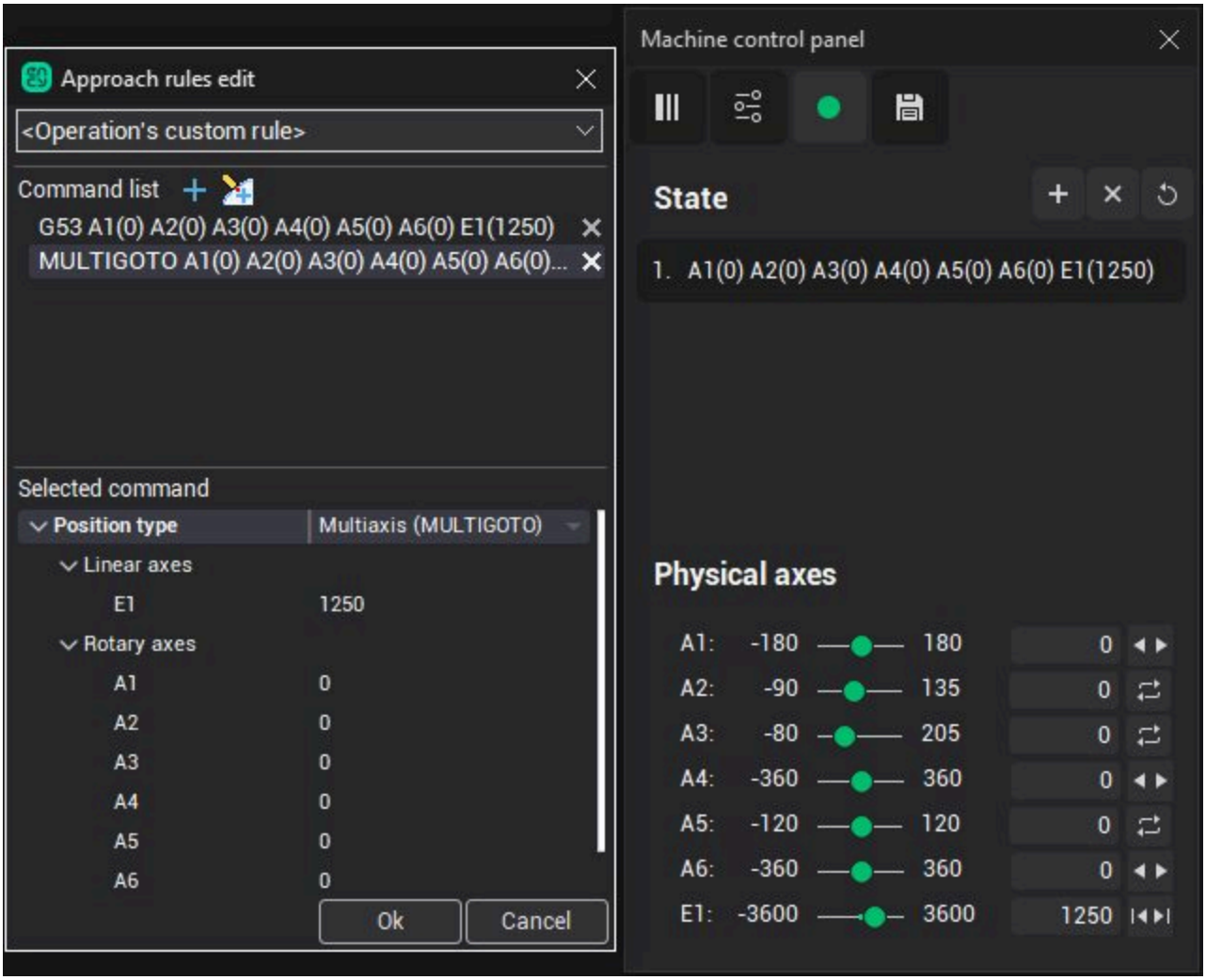
Then repeat the steps to remember the rest of the intermediate transition points.

After that switch to the Technology tab, activate the first (5D Contour) operation in the job tree, and select `Custom...` from the Return parameter drop-down at the Operation Setup as shown in the picture below.



Ensure the Machine control panel is open. Select the first remembered control point in the Machine control panel. The robot should be positioned to the remembered state. Then press the Add current state

button in the `Approach/return points list edit` window to add the current robot position as the return intermediate point as shown in the picture below.



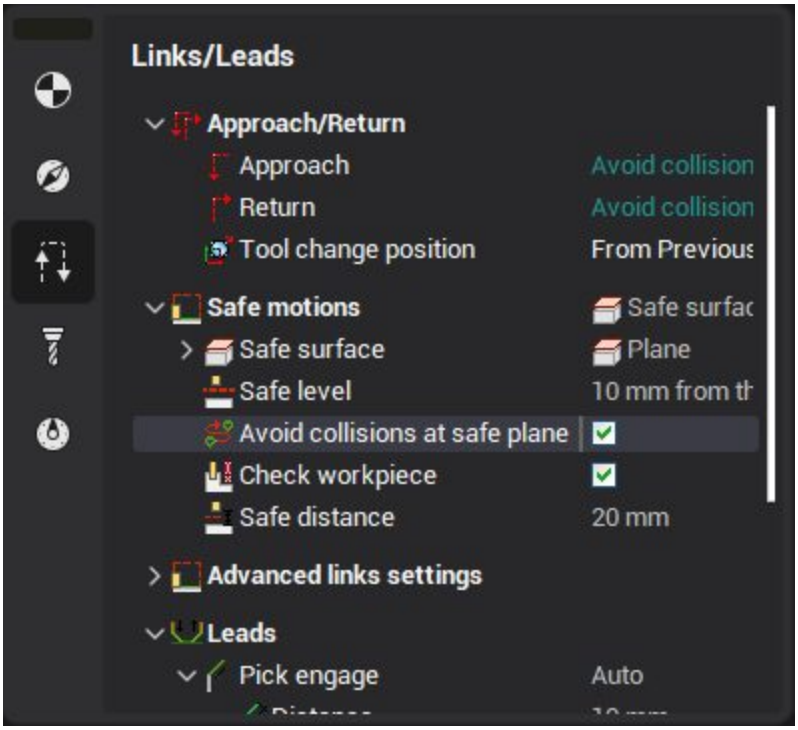
If exact values of axes are not important then control points can be added directly without formation of the list.

Similarly the approach sequence of the second operation (5d Contouring2) can be defined.

The planning of the links

CAM system uses the collision avoidance algorithm to plan the rapid tool path between and inside the operations. The algorithm requires the reliable description of the robotic cell, because it is based on the collision checking between the machine nodes.

Switch on the Avoid collisions at safe plane to activate the algorithm inside the operation. The parameter Links safe distance defines the minimum allowed distance to the collision. This value must be minimal and enough. If it's too small then the tool path is not safe enough. If the safe distance is too long, then the link can not be built or the calculation time will be unacceptable.



Supported operations

This option is available currently only for robots in the following operations (and their descendant operations, if not intentionally disabled):

- Morph
- Rotary finishing
- 5D Surfacing
- 5D contour and 6D contour
- Welding 5D and Welding 6D
- 5D by meshes

Link start and end points

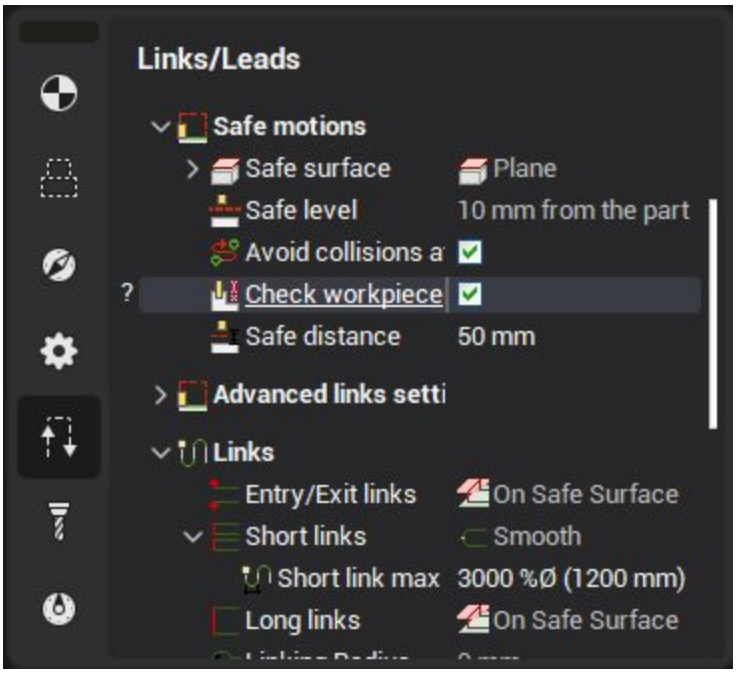
It's important to assure that the start and end point of a link is located out of collision, else the incorrect tool path will be generated. It can be done with the extra axes optimizer. The grayed zones of the map are

corresponding to the links. The restrictions are not shown in these zones. So the spline have to avoid all collisions.



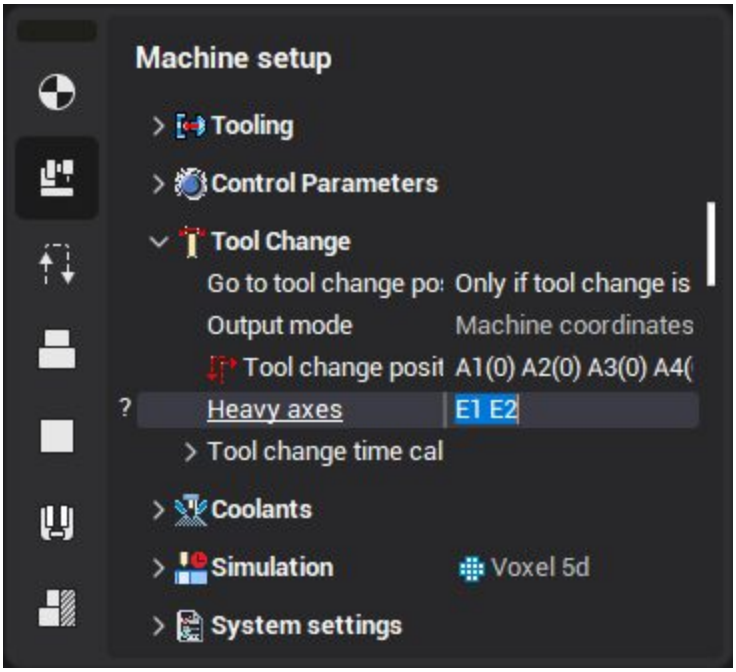
Workpiece collisions

By default, the algorithm only accounts for the part when checking collisions. To enable the collision avoidance between tool (or machine) and the workpiece, use the Check workpiece option.



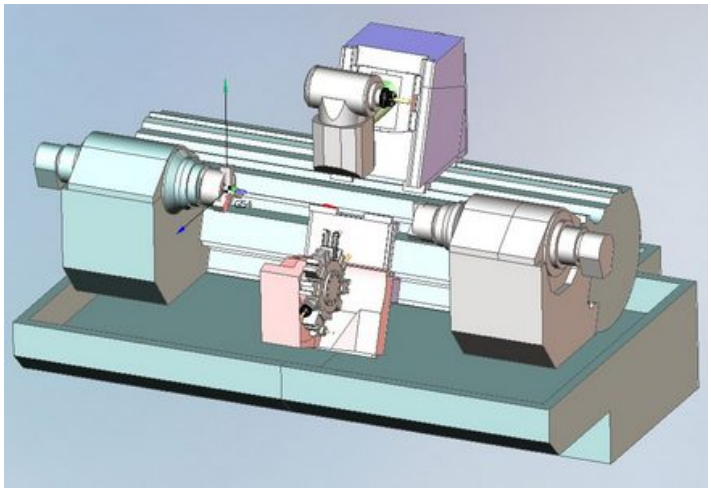
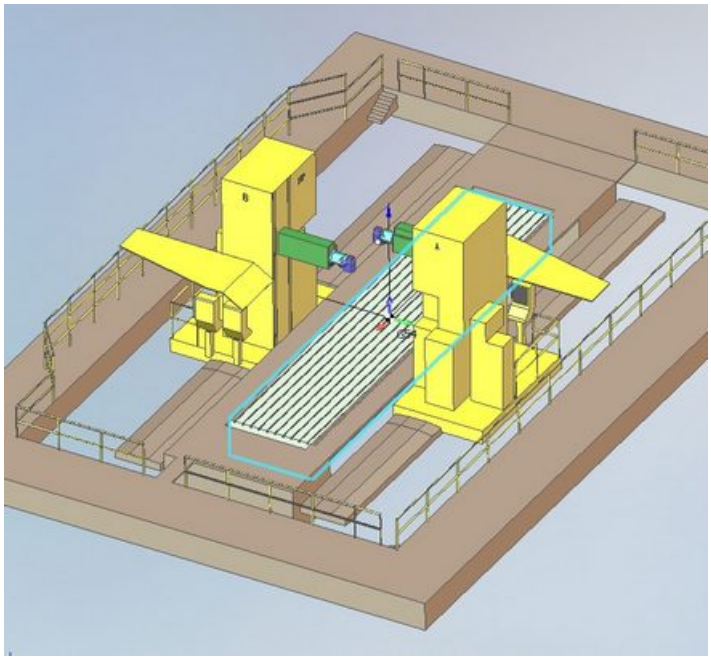
Exclude axes movement

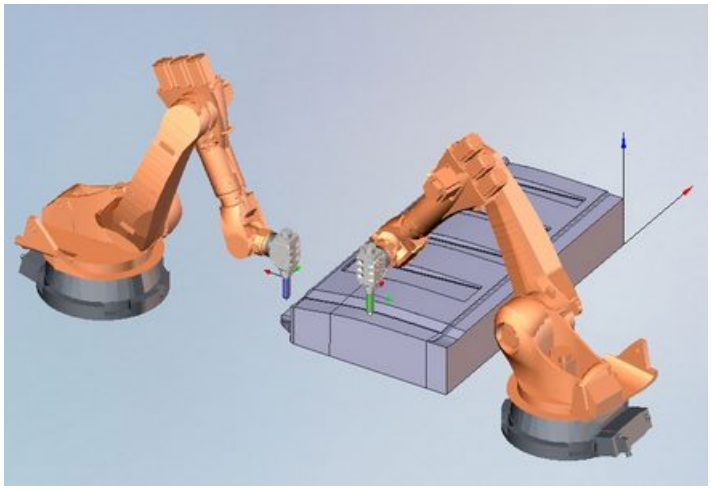
The collision avoidance algorithm uses all the axes of the machine to search the way from start to the end point. If you want to exclude or minimize the motions by some axes, then these axes must be enumerated in the list of heavy axes of machine. If the link requires the motion by the heavy axes, then it's performed in 3 stages: The first is the motion to the tool change position, after that the motion of heavy axes and then the motion to the end point.



Multi Task Machining

Multitask machines (MTM) allow to work by few tools simultaneously. MTM can be very different. For example, a multi-task turn-milling center has few spindles or/and turret heads inside one housing. Sometimes a multi-task machine is two or more identical machines that work under one workpiece.

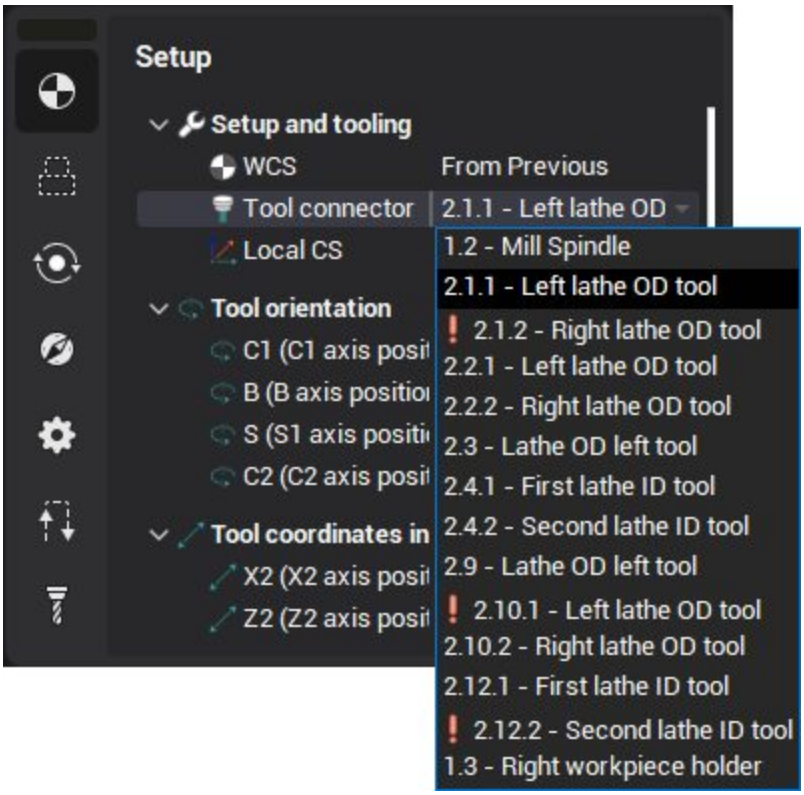
multi-task turn-milling center	
multi-task milling machine	

two robots, working together	

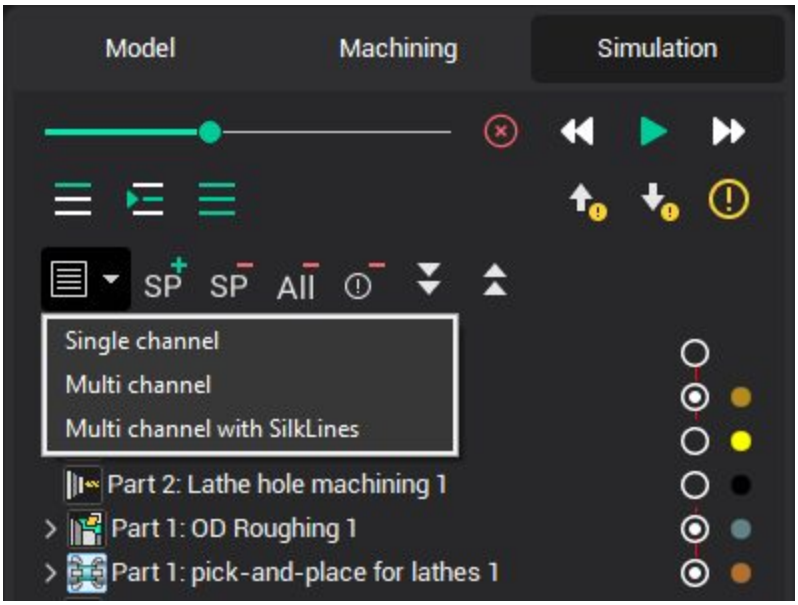
With multitask machining every tool is controlled by it's own program. This program is called as channel of control. So the multitask machine has more than one channel. Sometime the task of program of multitask machine can be presented as a creation of the separate unrelated projects for every channel. This simplified approach is unacceptable if we must consider the results of one channel in another one, or if the channels work together in one place.

CAM system gives the possibility to program multichannel machines. To activate these features it's necessary to load the kinematic scheme of the multichannel machine. The standard package has the schema for the multichannel turn-milling center Index G160 and schema for double FPT milling machine. It's enough to test CAM system. Contact your dealer if you need the schema of your own equipment.

On the "Technology" page the programming process of the multichannel machine is the same like the process of programming single channel machine. User have to understand that the definition of the operation tool holder defines the channel of machining. User have to think about equal machining time for every channel.



On the "Simulation" page for the multitask machines there is the button to choose the simulation mode:



In the single channel mode the simulation is performed in series operation by operation. It this mode it is comfortable to analyze the tool path of a separate operation without care about other channels.

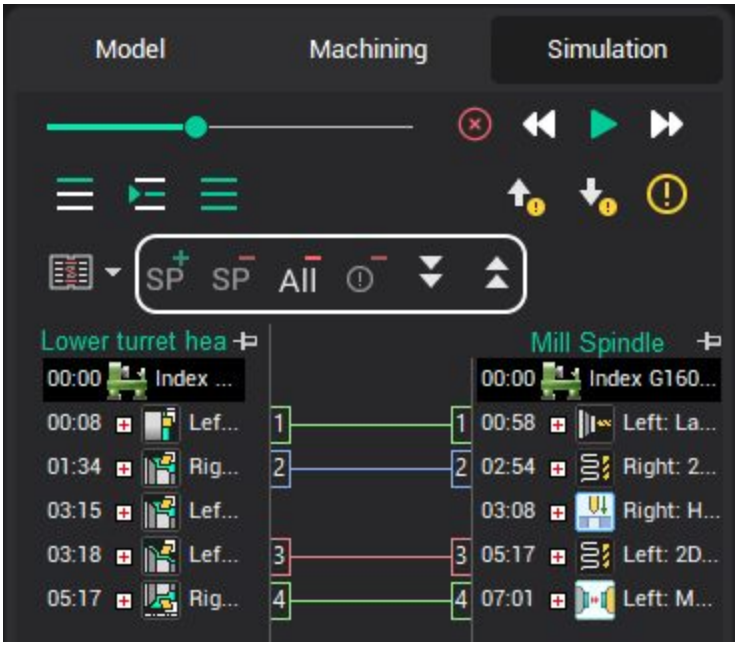
In multichannel mode the simulation is executed simultaneously in every channel like it will be executed on the real equipment. It this mode it's comfortable to check for the collisions with the taking care about motions in all channels. Detected collisions can be eliminated by the adding of sync points.

Multichannel mode with Silk lines is comfortable to add, remove and edit the sync points.

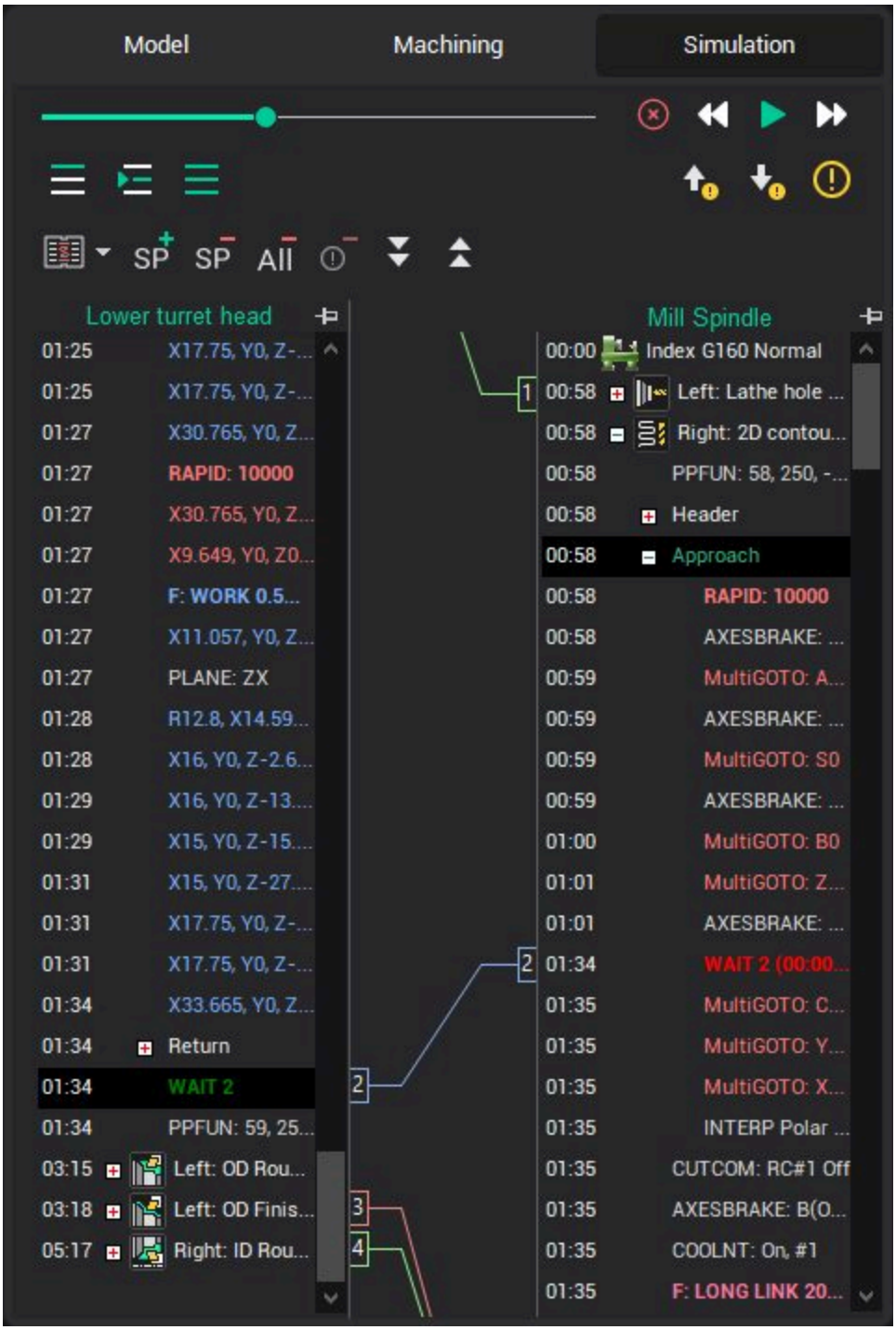
Sync points

The sync point suspends the execution of one channel until another channel will arrive the certain block. In many CNC the sync points are coded by M-codes. So if the M500 means the sync point, it must be written in more than 1 channel. The channel that arrives M500 first, will suspends the execution and wait until another channel arrives the same sync point. After that the both channels start to execute the next commands together.

The next buttons are used to insert or remove the sync points in CAM system.



To add the sync point, choose the commands that have to start together in every channel and press button "Add sync point". The "WAIT" will be inserted before the selected commands. In the channel, where the estimated time till the point is less, the WAIT will be written by red and the waiting time will be written in braces.



The list of sync points is the attribute of a project, not the attribute of operation. It gives the possibility to restore the sync points after the tool path recalculation. The restoring is based on the information about machine position. So the point will be restored correctly after the feed rate changing or after the approximation tolerance changing or after another changes, that don't affect the tool path too much. if the tool path was changed too much, then we recommend to reinsert the sync points.

To remove the sync point it's necessary to choose the "WAIT" command in any channel and click button "Delete sync point".

If some operation was deleted, disabled or moved after the insertion of sync points, then the situation is possible when a sync point can't be restored in every channel. These points are marked as "invalid". All

invalid sync points can be removed by button "Delete invalid sync points".

To remove all sync points use the button "Delete all sync points".

See example of MTM using on YouTube:

https://www.youtube.com/watch?v=52WkyXPuxEU

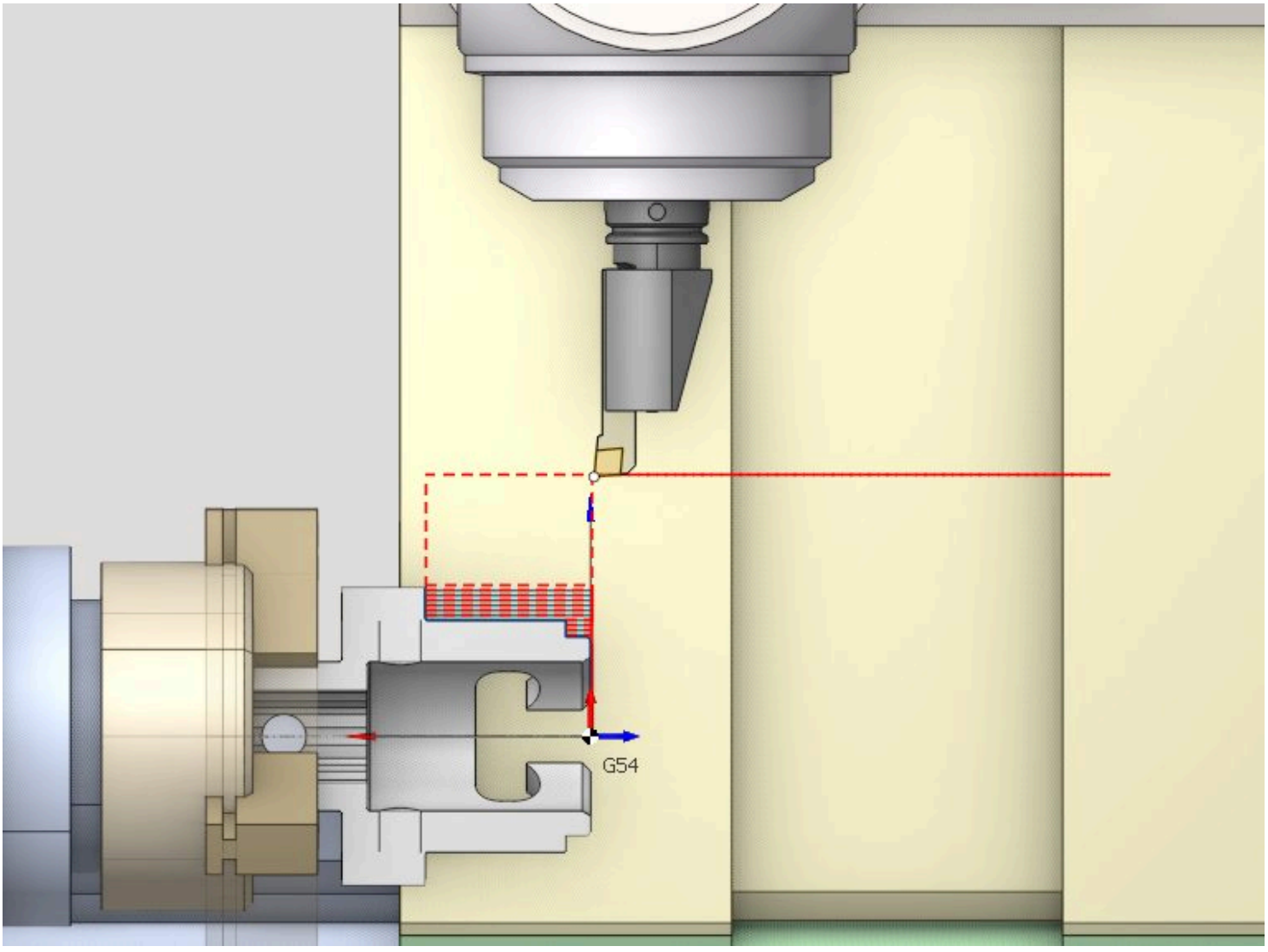
Submachine definition in the machine schemas

Submachine is a list of parameters for the `tool holder, workpiece holder` pair. This is especially topical for complex machines with multiple spindles, multiple places for the workpiece, etc.

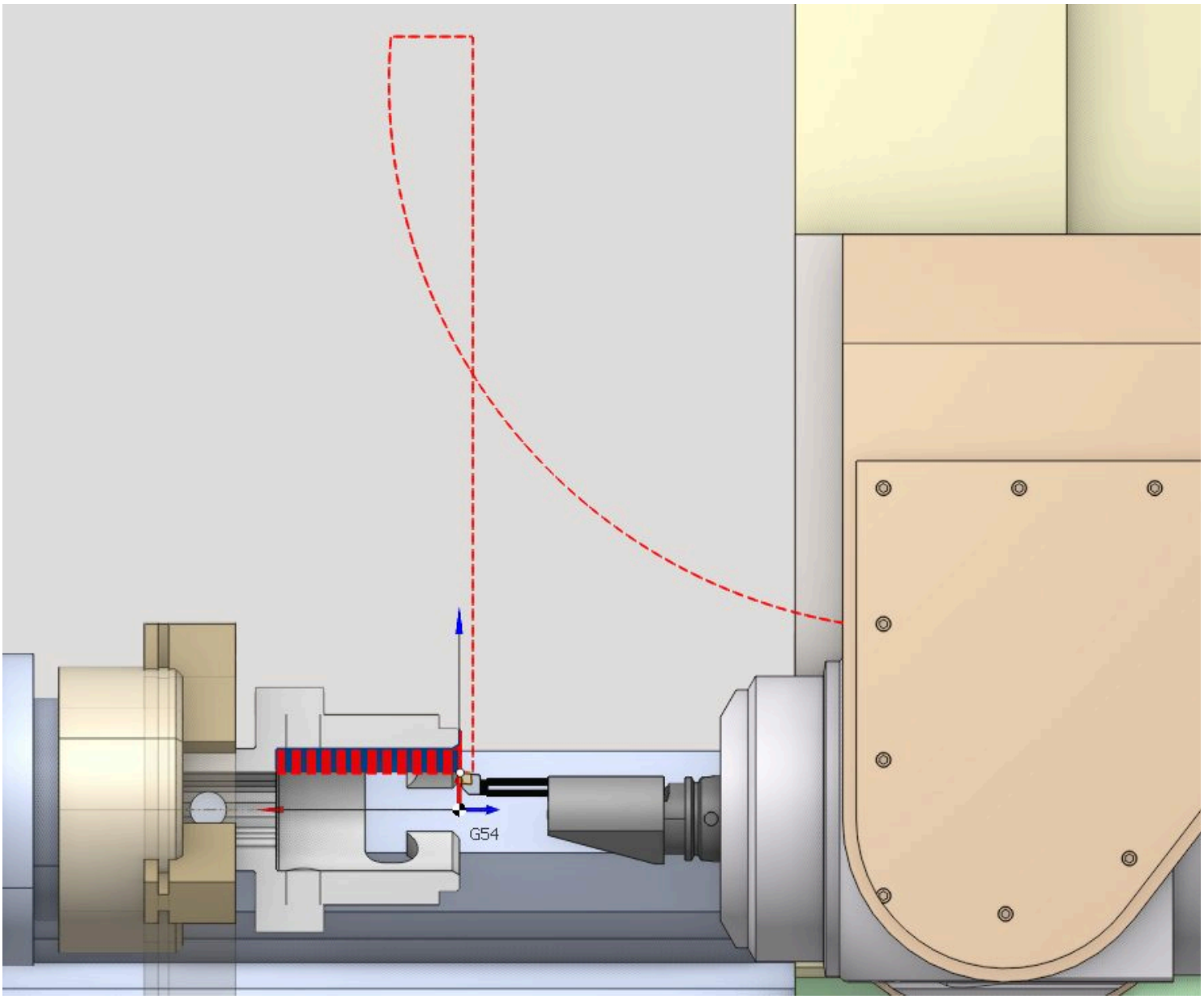
```
                    Submachine definition example for the
                <SubMachinesList Expanded="false">
                                <SCType ID="MainMill" Caption="Main spindle - Mill
head" Type="SubMachine">
                                        <ToolNode DefaultValue="AxisS"/>
                                        <WrkNode DefaultValue="AxisC"/>
                                        <Channel DefaultValue="1"/>
                                        <XAxisID DefaultValue="AxisX"/>
                                        <YAxisID DefaultValue="AxisY"/>
                                        <ZAxisID DefaultValue="AxisZ"/>
                                        <ToolAxisID DefaultValue=""/>
                                        <OriginG54BaseNode DefaultValue="Base"/>
                                        <ApproachRule DefaultValue="BC; Y Z(10);
X"/>
                                        <ReturnRule DefaultValue="Z(10); X; Z"/>
                                        <DetailedApproachRules>
                                                <TurnRadial DefaultValue="BS;Z;X"/>
                                                <TurnAxial
DefaultValue="BS;Z(10);X;"/>
                                                <MillRadial DefaultValue="BC;Z;X"/>
                                                <MillAxial DefaultValue="BC;Z;X"/>
                                                <MillLCS
DefaultValue="BC;Z;X;LCS;XY;Z"/>
                                                <MillTCPM
DefaultValue="BC;Z;X;LCS;XY;Z"/>
                                        </DetailedApproachRules>
                                        <DetailedReturnRules>
                                                <TurnRadial DefaultValue="X;Z"/>
                                                <TurnAxial
DefaultValue="Z(10);X;Z"/>
                                                <MillRadial DefaultValue="X;Z"/>
                                                <MillAxial
DefaultValue="Z(10);X;Z"/>
                                                <MillLCS DefaultValue="LCS;X;Z"/>
                                                <MillTCPM DefaultValue="LCS;X;Z"/>
                                        </DetailedReturnRules>
                                </SCType>
```

- ToolNode is a common parent node for all tool holders that belong to the sub-machine

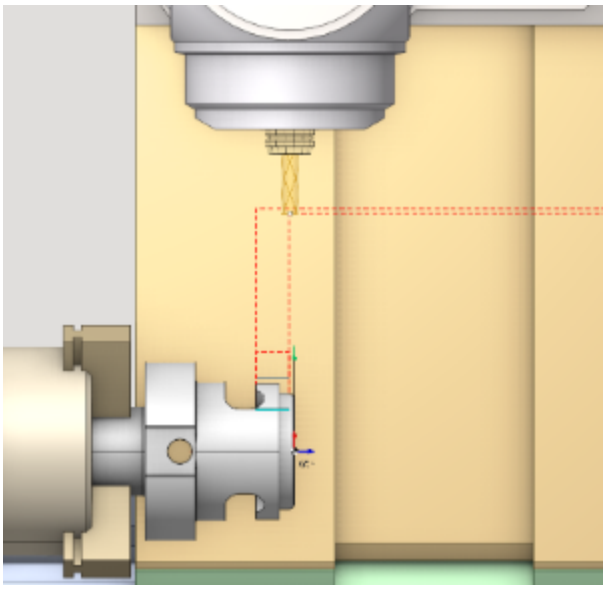
- WrkNode is a common parent node for all workpiece holders that belong to the sub-machine
- XAxisID, YAxisID, ZAxisID - the axes that are responsible for the motion along X,Y,Z.
- ToolAxisID - the ID of the turret axis if it exists.
- OriginG54BaseNode - the ID of the axis where the workpiece coordinate system (WCS) is attached
- OriginG54 - additional transformation for the WCS. For example it allows to overturn the Z-axis in the counter spindle
- ApproachRule, ReturnRule - the default rules that will be used to build the approaches or returns for the submachine in general.
- DetailedApproachRules, DetailedReturnRules - the default rules for this submachine in different tool configurations. Tool approaches and returns must consider the specifics of the tool and workpiece positions. Hence, the system allows for setting approach and return schemas individually for each tool position on the submachine. Approach rules are defined in the section of the corresponding submachine between the DetailedApproachRules tags. These rules are specified individually for each tool type and position within the appropriate tags. The following tags are available:
  - TurnRadial – a lathe cutting tool with a radial setup.



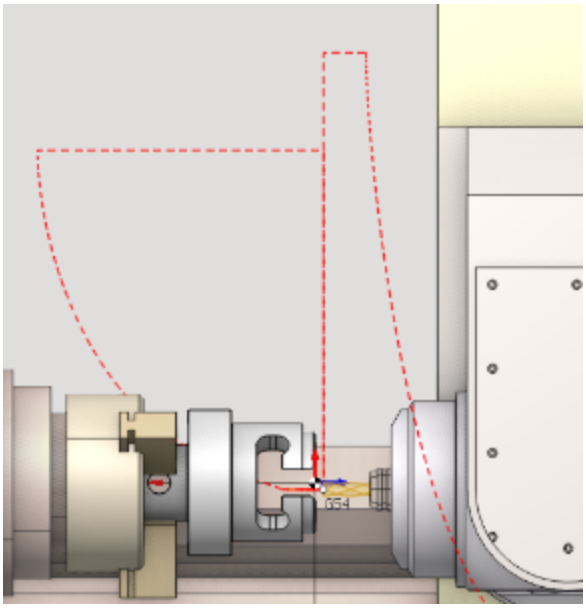
- TurnAxial – a lathe tool (e.g., a non-rotating drill) with an axial setup.



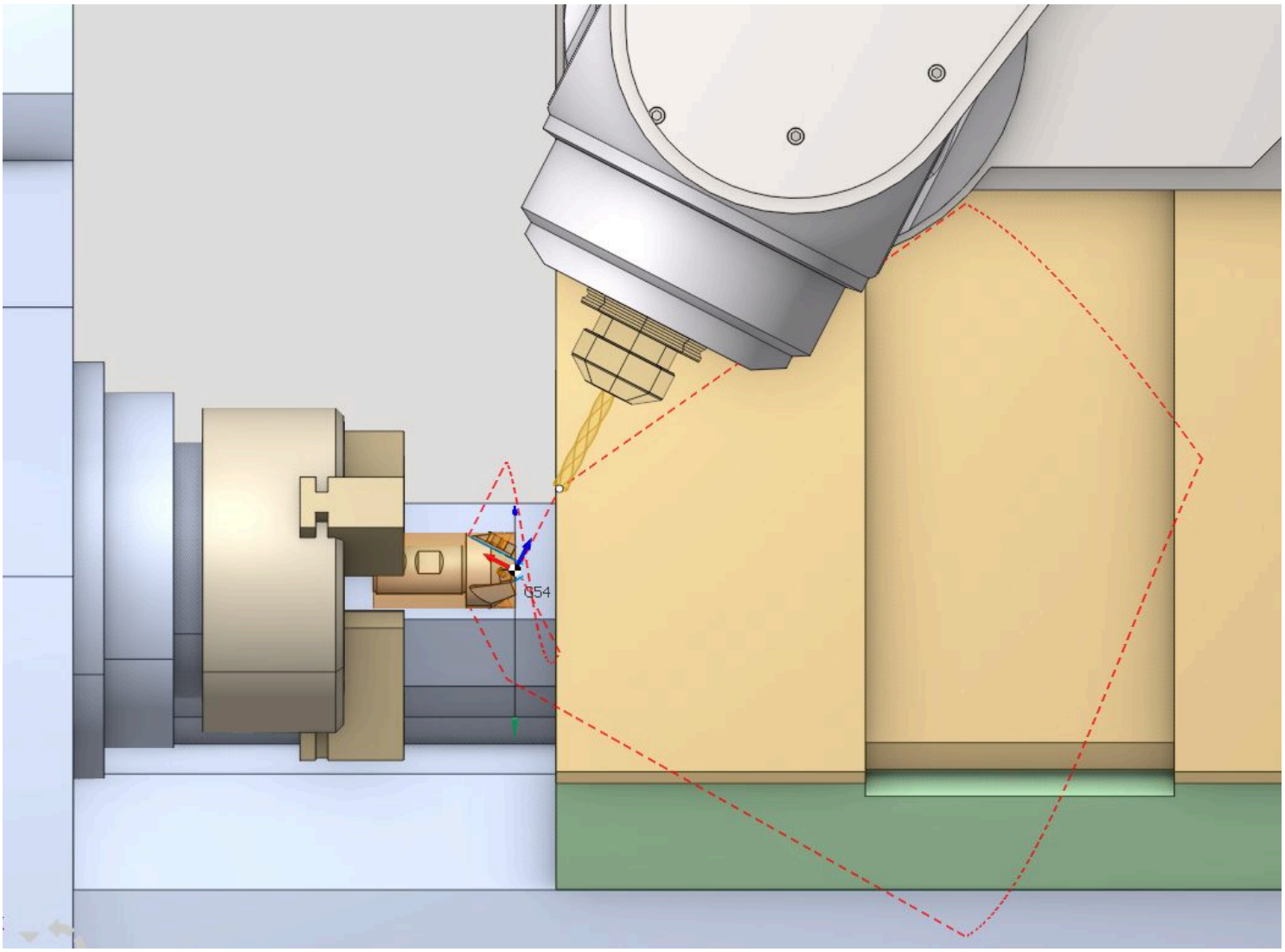
- MillRadial – a milling tool (including a rotating drill) with a radial orientation (the tool axis is perpendicular to the workpiece axis).



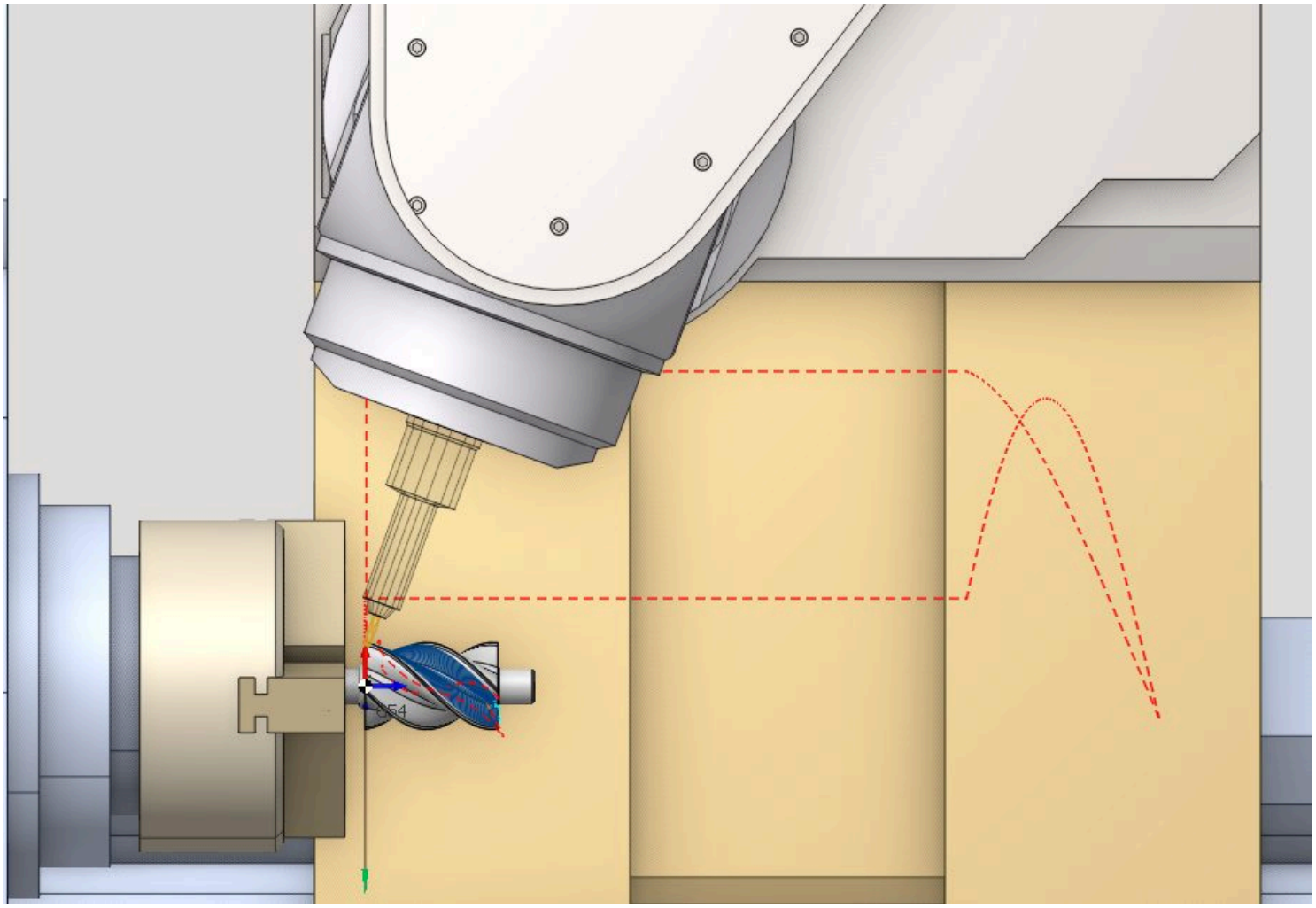
- MillAxial - a milling tool (also a rotating drill) featuring an axial arrangement (the tool's axis is parallel to the workpiece axis).



- MillLCS - a milling tool (also a rotating drill) applied in four- or five-axis indexing machining operations.



- MillTCPM - a milling tool (also a rotating drill) applied in continuous four- or five-axis machining processes with TCPM support.



DefaultValue - the approach/return rule for the specific tool setup, applied by default. It appears in the Links tab for each submachine after the relevant workpiece and tool connectors are defined. The format of rules is the same as in the general Approach and return rules.

When the approach/retract rules for submachines or a specific tool setup are not specified, the general machine default rules will be applied.

If no submachine is declared in the schema, then any tool holder can be used with any workpiece holder. It's done for the compatibility with the schemas developed for the previous versions of CAM system.

For example, for the MTM turn-milling machine with two turrets and two spindles the following submachines can be defined:

1. **`Upper turret, main spindle`**
2. **`Upper turret, counter spindle`** if Upper turret is able to work with the counter spindle
3. **`Lower turret, main spindle`** if lower turret is able to work with the main spindle
4. **`Lower turret, counter spindle`**
5. **`Main spindle, counter spindle`** for the takeover from the main spindle to the counter spindle
6. **`Counter spindle, main spindle`** for the takeover from the counter spindle to the main spindle

CAM system does not allow to choose the tool holder in the operation if it can not be used with the current workpiece holder (there is no submachine for given pair, but there are other submachines).

The examples of the submachines declaration are available in the schemas "*Index G160*" and "*Hanwha 32*" that are included into the standard package.

Swiss lathes programming

Machine requirements

Swiss type lathe is designed for the fully automatic complete machining of the turn-milling parts. It has two spindles and two or more channels. CAM system doesn't support the swiss lathes with 3 or more channels. The most popular swiss lathes (Hanwha, NEXTTURN, Citizen Cincom) has the common structure. They have 2 channel, main and counter spindles, groups of tool to work with main or with counter spindle. This structure is described in the *..ENCY\Supplements\SwissTemplate.xml* that must be ancestor for all user swiss lathes schemas. Example of HANWA 32 machine based on *SwissTemplate.xml* is also included into the machines list of the standard package.

Swiss lathe project template

The project creation workflow has 2 stages: consecutive operation planning and channels synchronization.

Consecutive operation process template is shown below:

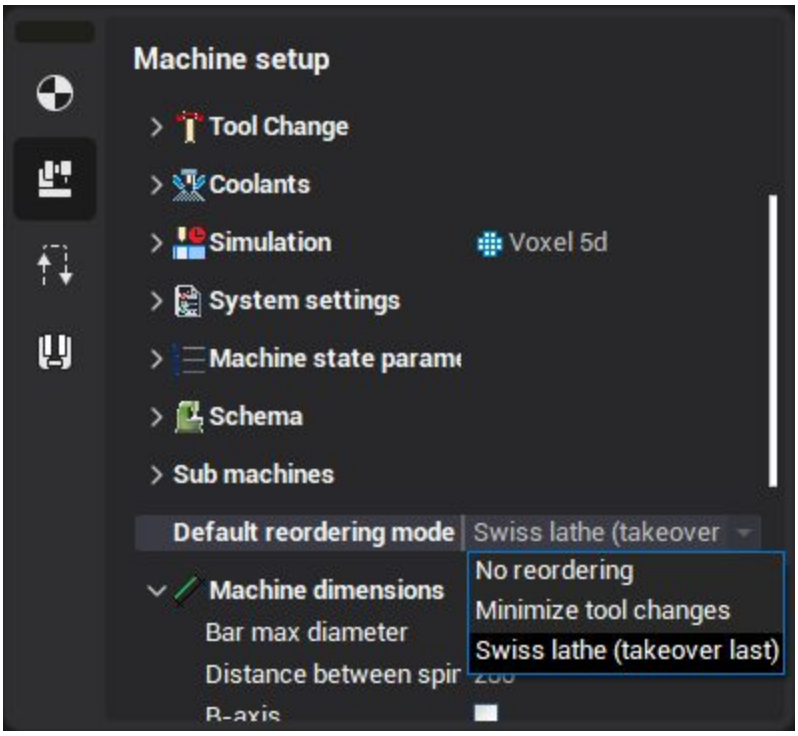
- Part 1 (main spindle)
  - Bar feeding
  - Operations to machine in the main spindle
  - Part-off
  - Takeover (synchronized with part-off)
- Part 2 (counter spindle)
  - Operations to machine in the counter spindle

Video below shows how to create the simple swiss lathe project.

Reordering and channels synchronization

Operations list must be reordered before the dividing onto the channels. It can be done in the sequencing mode. Reordering is done automatically for Swiss lathes and don't need any manual actions from user.

Default operations reordering mode is assigned in the swiss template. If Swiss lathes reordering mode is activated then the operations of the second part are placed in the beginning of the reordered list.

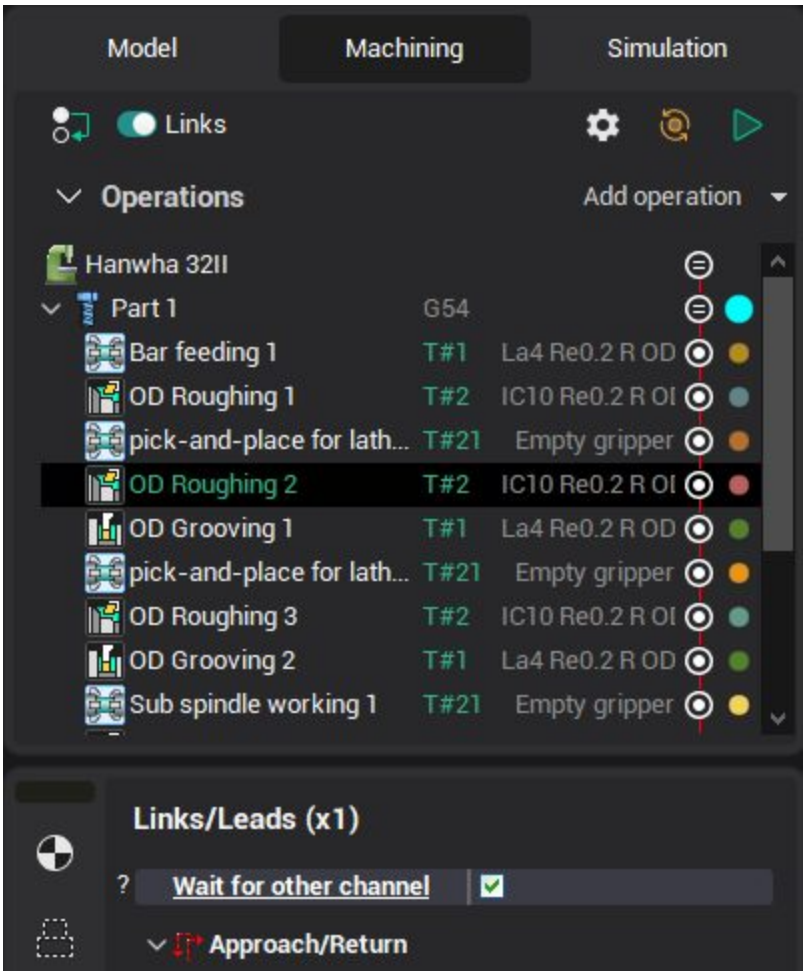

Machine setup
Tool Change
Coolants
Simulation
Voxel 5d
System settings
Machine state parame
Schema
Sub machines
Default reordering mode
Swiss lathe (takeover
No reordering
Minimize tool changes
Swiss lathe (takeover last)
Machine dimensions
Bar max diameter
Distance between spir
B-axis

Automatic insertion of wait labels

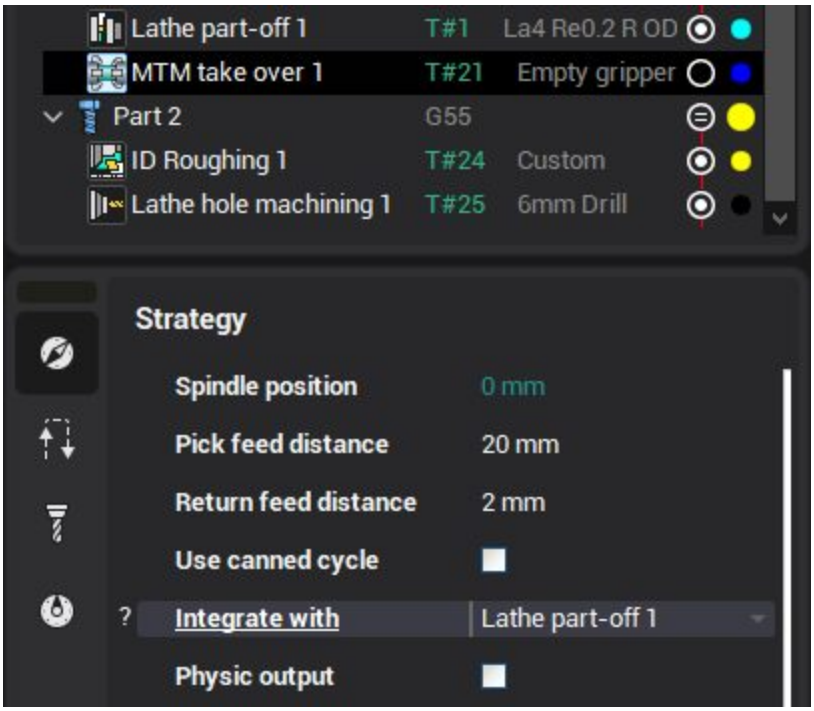
Before the insertion of wait labels, the operations must be reordered in the sequencing mode if the project contains more than one part.

Any operation can have the parameter wait other channel on the Links/Leads page. This parameter is visible only if the previous operation in the reordered sequence works in the other channel.

If the Wait other channel is enabled, the Wait CLData command is automatically inserted. So, the current operation will not start until the previous operation is finished.

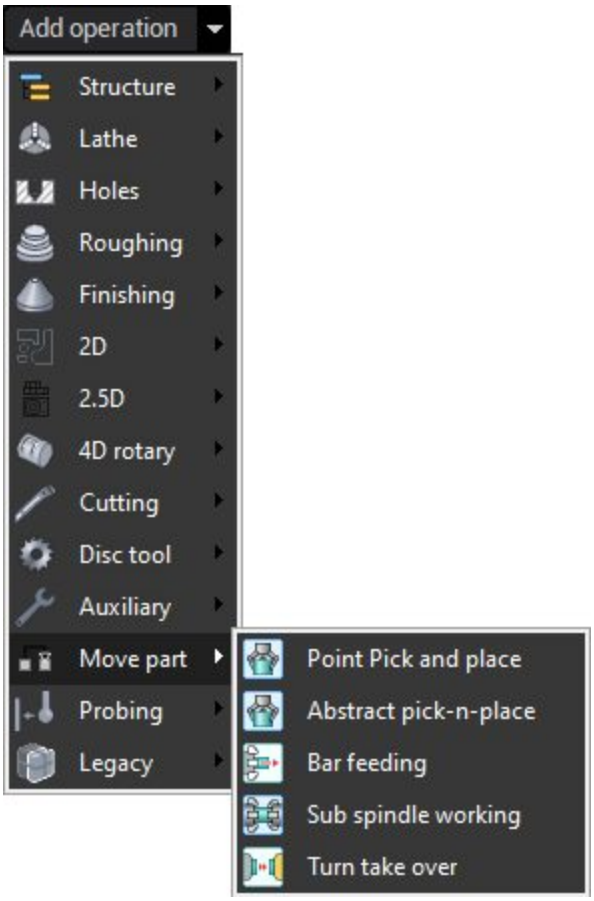


Another method for the automatic insert of the wait labels is using the Turn take over operation. It can be synchronized with the part-off operation.


Lathe part-off 1 T#1 La4 Re0.2 R OD
MTM take over 1 T#21 Empty gripper
Part 2 G55
ID Roughing 1 T#24 Custom
Lathe hole machining 1 T#25 6mm Drill
Strategy
Spindle position 0 mm
Pick feed distance 20 mm
Return feed distance 2 mm
Use canned cycle
Integrate with Lathe part-off 1
Physic output

Move part operations

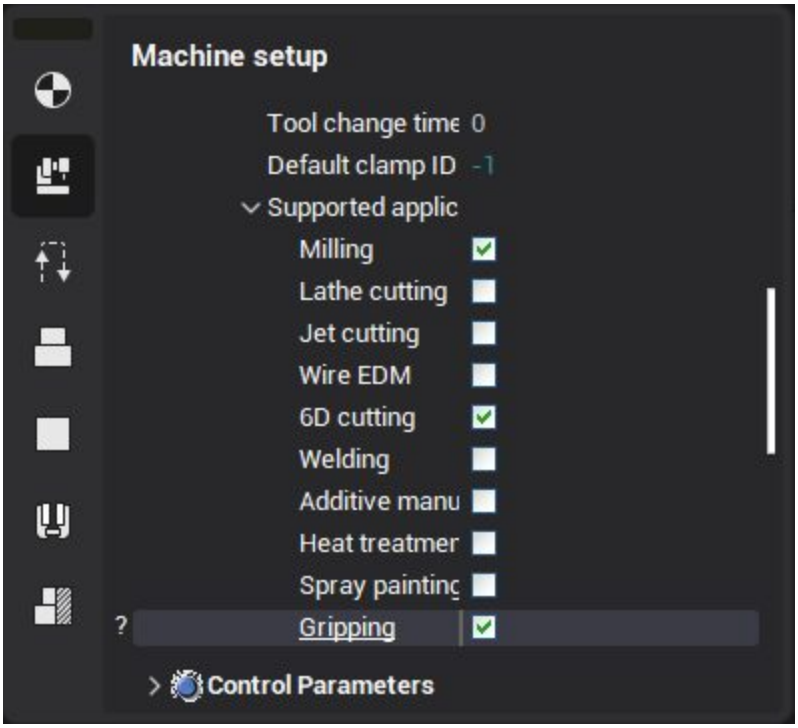
There is the separate group of the operations which can move the part using robot or swiss-type/lathe machine. It comprises Pick and Place, MTM Takeover, Bar feeding and Sub spindle working operations.



Machine requirements for part moving operations

The part moving operations can be available on any kind of machine-tool: milling center, lathe with subspindle, industrial robot. The main machine-tool requirement is the existing of the special tool holder that is marked as gripper. If gripper is absent in the current machine schema, then part move operation will not be available.

You can switch on this option on the machine setup page or in the file of machine schema description.



The fragment of the machine schema about the tool holder definiton is shown below. Note that the Gripper is enabled inside the SupportedToolTypes.

```
<SCType ID="MillSpindle" Caption="Mill Spindle" Type="TToolHolderNode">
   <VisualProperties>
       <Metallic DefaultValue="True"/>
   </VisualProperties>
   <XAxisID DefaultValue="AxisX"/>
   <YAxisID DefaultValue="AxisY"/>
   <ZAxisID DefaultValue="AxisZ"/>
   <SupportedToolTypes>
      <MillTool DefaultValue="true"/>
      <LatheCutter DefaultValue="true"/>
      <Gripper DefaultValue="true"/>
   </SupportedToolTypes>
</SCType>
```

Adaptation of the turn milling lathes with subspindle

Pick and place operation can be used to move the workpiece between the main spindle and subspindle on the turn-milling machines. To make it possible the subspindle must be declared as tool holder with the gripping application. If you have got the machine schema of turn milling machine with subspindle designed for the CAM system and earlier, then you need to modify the machine description. Below the differences between the old (left) and adapted (right) schemas is shown.



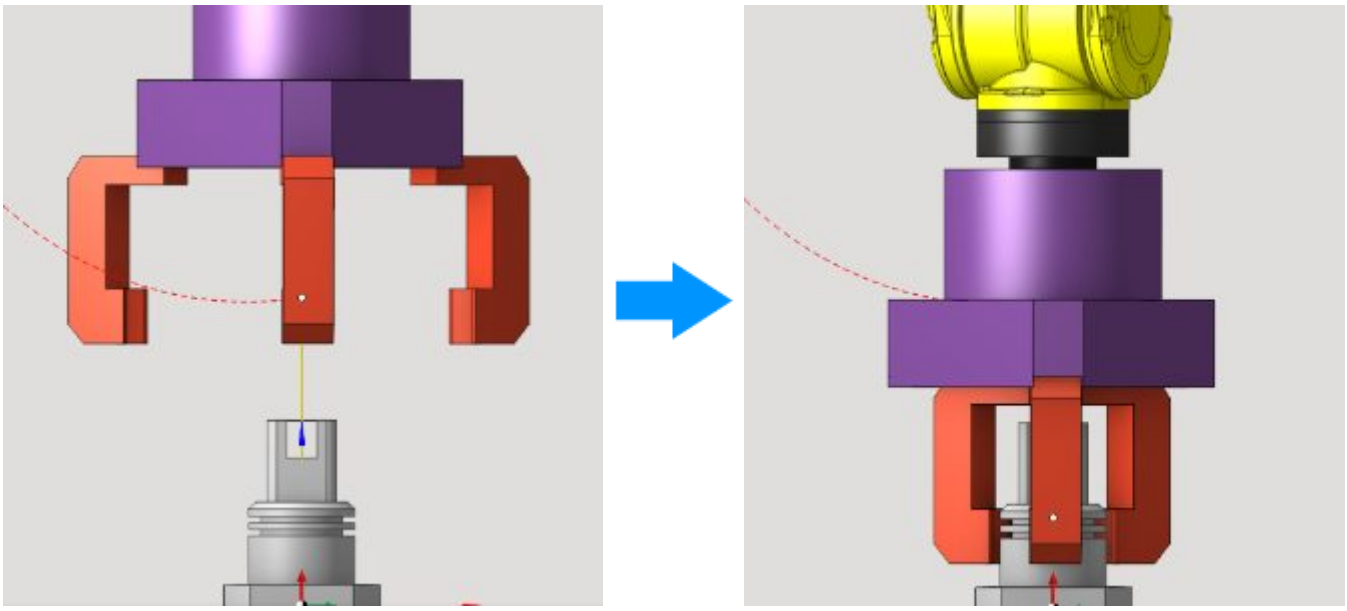
```
<SCType ID="AxisB" Caption="Tailstock (B)" type="TMachineAxis">
    <ImageFile DefaultValue="Images\AxisB.osd"/>
    <ParameterName DefaultValue="AxisBPos"/>
    <AxisType DefaultValue="Linear"/>
    <Matrix>
        <SCType ID="T1" type="TTranslateZ" DefaultValue="[MachineDimensions.SD]"/>
    </Matrix>
    <SCType ID="RightSpindle" Caption="Right spindle" type="TMachineNode">
        <Matrix>
            <SCType ID="T1" type="TTranslateZ" DefaultValue="-[MachineDimensions.NC2]"/>
        </Matrix>
        <SCType ID="AxisC2" Caption="C2 axis" type="TMachineAxis">
            <ParameterName DefaultValue="AxisC2Pos"/>
            <AxisType DefaultValue="Rotary"/>
            <SCType ID="RightWorkpieceHolder" Caption="Right workpiece holder" type="TWorkpieceHolderNode">
                <HolderType DefaultValue="RightLatheSpindle"/>
                <SpindleParamID DefaultValue="AxisC2Pos"/>
                <InverseRotationDirection DefaultValue="True"/>
            </SCType>
            <SCType ID="RightChuck" Caption="Right chuck" type="TMachineNode">
                <Matrix>
                    <SCType ID="T2" type="TRotateX" DefaultValue="180"/>
                </Matrix>
                <ImageFile DefaultValue="Images\Chuck1.osd"/>
                <SCType ID="RightJaw1Axis" Caption="Right jaw 1 axis" type="TPumaTT2000JawAxis">
                    <ParameterName DefaultValue="RightChuckState"/>
                </SCType>
```

```
<SCType ID="AxisB" Caption="Tailstock (B)" type="TMachineAxis">
    <ImageFile DefaultValue="Images\AxisB.osd"/>
    <ParameterName DefaultValue="AxisBPos"/>
    <AxisType DefaultValue="Linear"/>
    <Matrix>
        <SCType ID="T1" type="TTranslateZ" DefaultValue="[MachineDimensions.SD]"/>
    </Matrix>
    <SCType ID="RightSpindle" Caption="Right spindle" type="TMachineNode">
        <Matrix>
            <SCType ID="T1" type="TTranslateZ" DefaultValue="-[MachineDimensions.NC2]"/>
        </Matrix>
        <SCType ID="AxisC2" Caption="C2 axis" type="TMachineAxis">
            <ParameterName DefaultValue="AxisC2Pos"/>
            <AxisType DefaultValue="Rotary"/>
            <SCType ID="RightWorkpieceHolder" Caption="Right workpiece holder" type="TToolHolderNode">
                <HolderType DefaultValue="RightLatheSpindle"/>
                <SpindleParamID DefaultValue="AxisC2Pos"/>
                <InverseRotationDirection DefaultValue="True"/>
                <Channel DefaultValue="0"/>
                <XAxisID DefaultValue=""/>
                <YAxisID DefaultValue=""/>
                <ZAxisID DefaultValue="AxisB"/>
                <SupportedToolTypes>
                    <MillTool DefaultValue="true"/>
                    <Gripper DefaultValue="true"/>
                </SupportedToolTypes>
            </SCType>
```

1. In the subspindle definition TWorkpieceHolderNode is replaced by TToolHolderNode.
2. XAxisID, YAxisID, ZAxisID - the names of the axes that is responsible for the motion of the tool along the correspondent axis are added.
3. Gripper is added into the SupportedToolTypes

Clamp devices control

Move part operations often need control over the device which is used to transfer the part from the initial to final position, for example to push apart the jaws before picking the part and to close them after to fixate the part. This clamp feature allows to automatically insert CLData commands for changing the state of the clamp device during the process of transferring it from one workpiece holder to another.



- New CLData command
- Automatic clamp control
- Clamp control parameters
- Adding clamp device to the project
- List of operations that support automatic clamp control

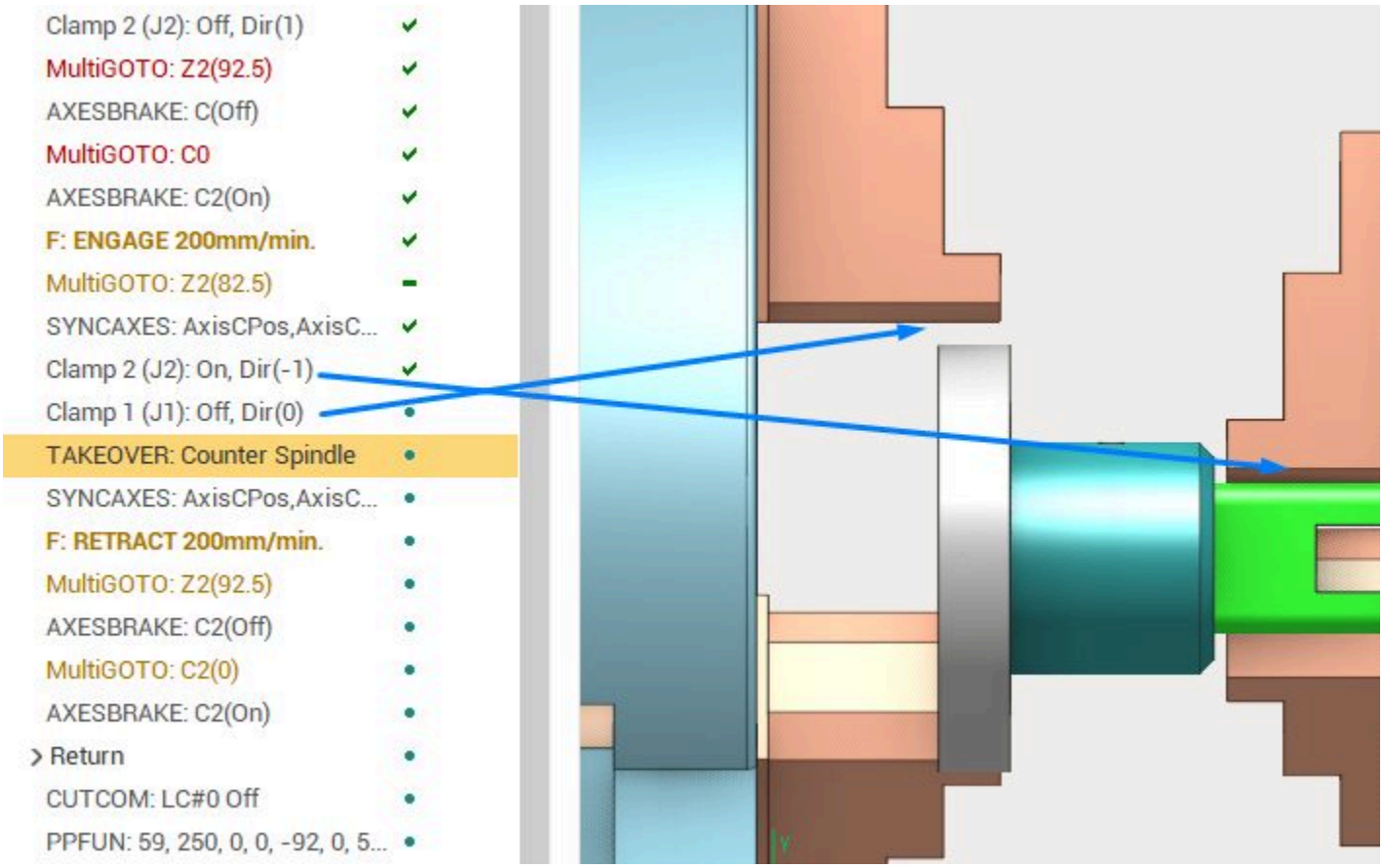
New CLData command

New special CLData command is used to indicate the change of the clamp device state. This command is taken into account during the project simulation and is also outputted to the postprocessor as the special "M" command. The Clamp CLData command has the following format:

Clamp *Clamp ID*: On/Off, Dir(*Direction*)

- Clamp ID is the unique number, used to identify the specific clamp device. See the section about creating clamp device to see how the clamp id can be assigned.
- *On/Off* is the flag indicating whether to grip or release the part, respectively.
- Direction is the integer (+1/-1/0) indicating the direction of clamp movement during the clamping/unclamping process. Let us consider the clamping process. If the jaws are pushed inside to fix the part (axis value for unclamped position is greater than the axis value for clamped position), then the direction is "-1"; if the jaws are pushed outside to fix the part (the clamp is inside the part), then the direction is "+1". For the unclamping, the direction is inverted. "0" direction indicates the

same clamp axis values for the clamped/unclamped state (usually it's an error in the clamp device parameters).



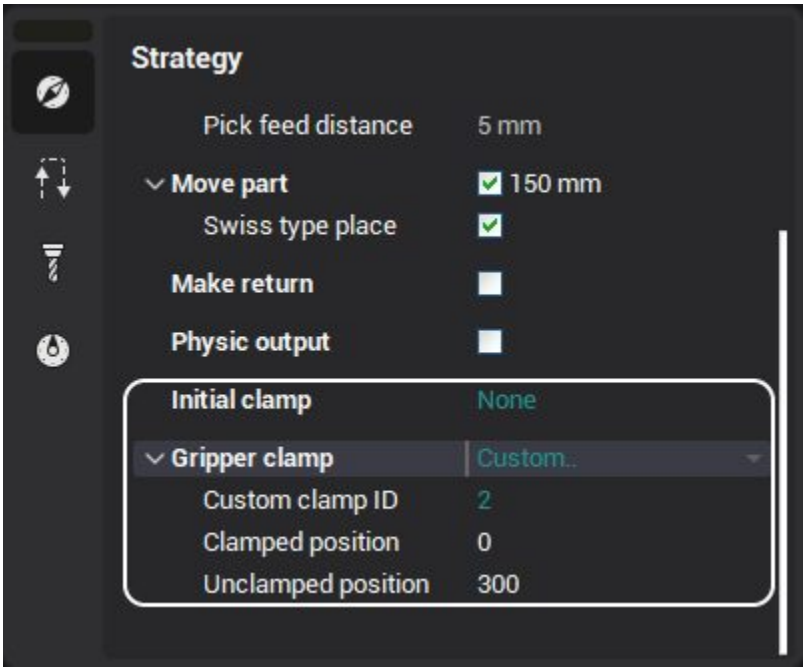
Automatic clamp control

The operations of the Move part operations group, such as Pick-and-place, can output the Clamp CLData commands automatically when the part transfer from one workpiece holder to another occurs. This transfer is done by the Takeover CLData command. If enabled by the operation parameters, the Clamp commands are generated automatically to simulate a typical part transfer process (example shown on the screenshot above), which ensures that the part is constantly held by some device:

1. Clamping the target clamp (Clamp 2: On)
2. Unclamping the initial clamp (Clamp 1: Off)
3. Takeover from the initial to target clamp (Takeover)

Clamp control parameters

The Move part operations have the following parameters which affect the output of the clamp control commands.



The Move part operations in general can operate with up to 3 workpiece holders which correspond to 3 stages of the pick and place process.

1. The part is located on the initial workpiece holder
2. The part is moved using the gripper (which is also a workpiece holder)
3. The part is placed on the final workpiece holder

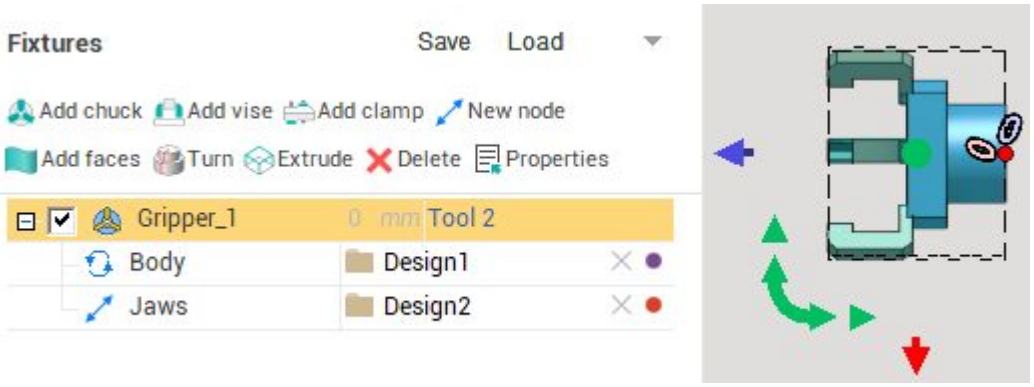
The clamp parameters have 3 parameter groups which correspond to the stages above. The workpiece holders of different stages can coincide, also some of the operations in this group have the simpler process for moving the part. This is also reflected in the clamp parameters. Let us consider the parameters of a single stage.

- Clamp device combo box allows to select the clamp which is used for the given move part stage. See the section below on how to make the clamp device available for selection in this list. This parameter can also be None, meaning that no command is outputted. The Custom enumeration item allows to specify the clamp ID of the device explicitly even if is not present in the machine schema or among the fixtures.
- Clamped position is the axis value which corresponds to the Clamped state of the device in the given move part stage.
- Unclamped position is the axis value which corresponds to the Unclamped state of the device in the given move part stage.

See the distributive projects with the "Move part" operations for an example on how to define the clamp parameters.

Adding clamp device to the project

The project has the list of clamp devices where each device is identified with the integer ID. This list is formed automatically and is used for selection of the clamp device for each move part stage. Currently, there are 3 types of clamp devices.

1. Fixture clamps 

In the "Fixtures" tab of the Machine/Stage/Part parameters you can create the clamp device fixture using the Add chuck, Add vise or Add clamp button, or load the clamp model from the .mcp file. The fixture parameters can be used to modify the clamp model, for example, to set the minimum and maximum value for the fixture clamp "axis" node. See the Fixtures documentation for more info on creating fixtures and their parameters.

2. Special axis, created in the Machine Maker. For example, you can create a gripper or tack welding tool with the special "clamp" flag in the Machine Maker as a machine axis (not fixture), and this axis will be used change the gripper state. Also the automatic clamping/unclamping of such device becomes available using the Move part operations. For further information on how to create such clamp device see the "CAM system Machine Maker" documentation.

3. Parametric jaw axes. Many lathe machines have the spindle or counter spindle jaws which are specifically defined in the machine schema file. They are rotated alongside with the spindle and are taken into account during the simulation and collision detection. Those automatically recognized jaw axes are also added to the machine clamp list. The clamp ID for them is assigned automatically. See the *Turn-Milling/Takeover.stcp* distributive project for an example of such machine schema and the usage of clamps in the MTM take over operation.

List of operations that support automatic clamp control

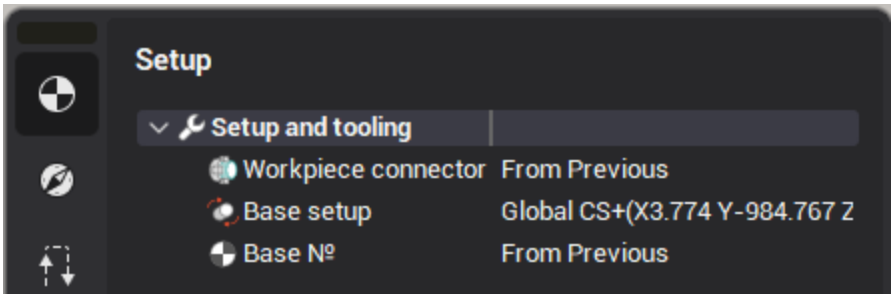
- Pick-and-place
- MTM take over
- Bar feeding

Pick-and-place

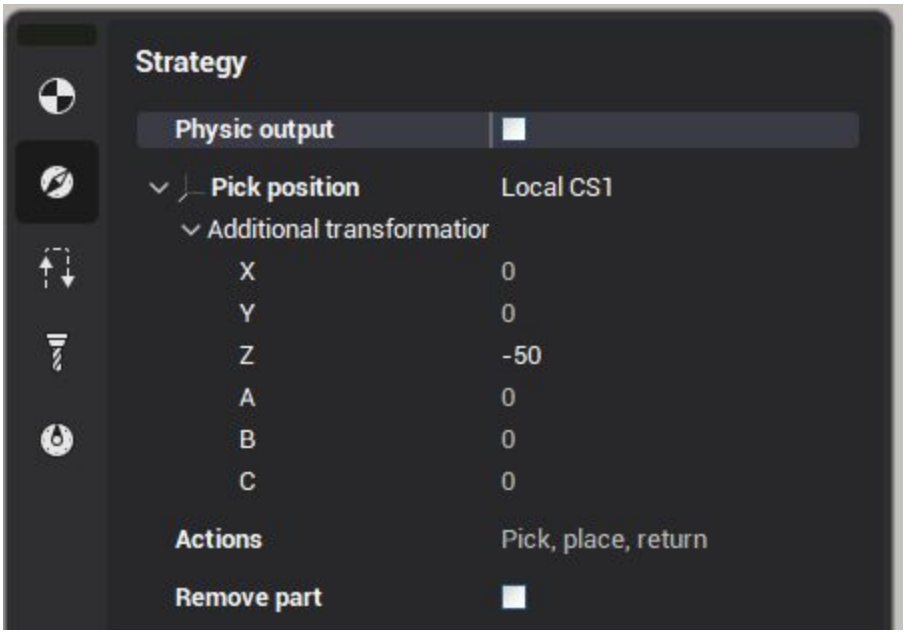
Tool path and parameters

Pick and place operation is designed to control the gripper tool to move the workpiece inside the job zone of a machine.

The workpiece is moved from the place, that was defined in the previous operation to the new place, that is defined by Workpiece connector and workpiece setup. All movements of the gripper is generated in the defined workpiece coordinate system.

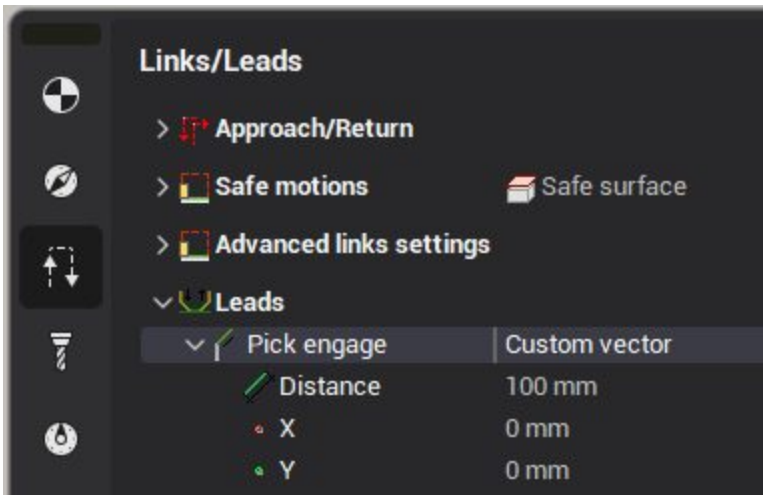


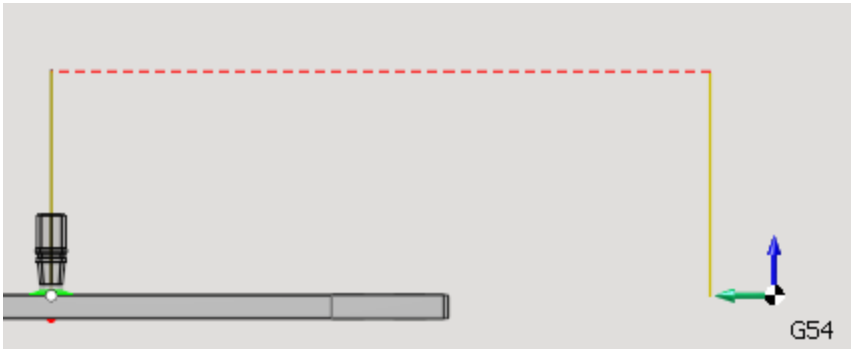
Tool path of pick and place operation has 3 main sections: pick, place, return.



Pick section contains the movement of a gripper from the initial position of the tool (usually tool change point) to the pick position of the part. Pick position is defined in geometry coordinate system with additional offsets. Place section has the movements of gripper with the workpiece from the initial position to the new one. It can be executed via the safe surface or with the enabled collision avoidance option. Return section is the movements of empty gripper from the place position to the final one (usually tool change point). Actions parameter defines the sections that must be generated. If the option remove part is enabled then the workpiece disappears after the placing.

Adding Engage/Retract





Using these parameters you can define the length and the direction for the engage/retract movements to the pick and/or place positions. Let's consider the "Pick engage". This parameter defines the position from which the tool engage (using the special "engage" feed) is made to pick the part. There are three types of the engage:None - Engage is disabled Auto - Engage is performed along the tool axis Custom vector - Engage is performed along the custom direction

The Distance parameter defines the length of the tool movement along the selected direction (the movement is done using the engage feed).

On the screenshot above there is an example of Pick and Place trajectory where the vertical segments correspond to the engage/retract (they have the same length in this example). The red color indicates rapid feed, the olive color - engage/retract feed.

"Place to next stage" operation

This is the special kind of the pick and place operation, which, like the "Turn take over", takes the position for placing the part from the next operation (usually it is the Setup stage or Part group). The rest of the parameters are the same as in the general pick-and-place operation.

The main use of this operation is for the robots and milling machines. For the turn or mill-turn machining it is recommended to use the specialized Turn take over operation instead.

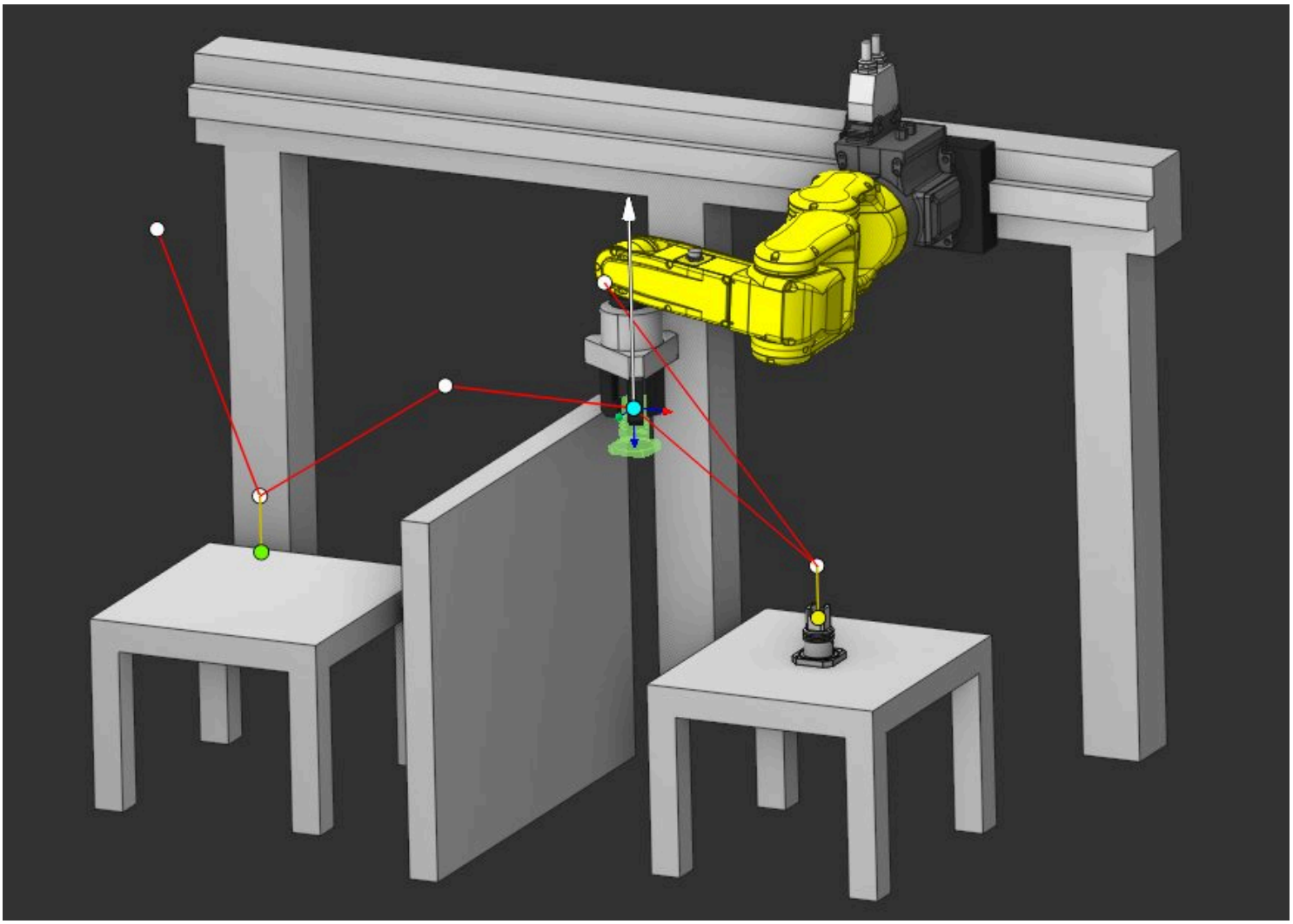
Creating a Pick and Place project tutorial

Video below demonstrates how to make the assembling projects. Details

Example project(s)

The following sample projects contain various examples of Pick and Place:
"*Milling/WoodWorking/FrameAssembly.stcp*" and the projects from the "*Robots/Pick and place*" folder.

Point Pick-and-Place

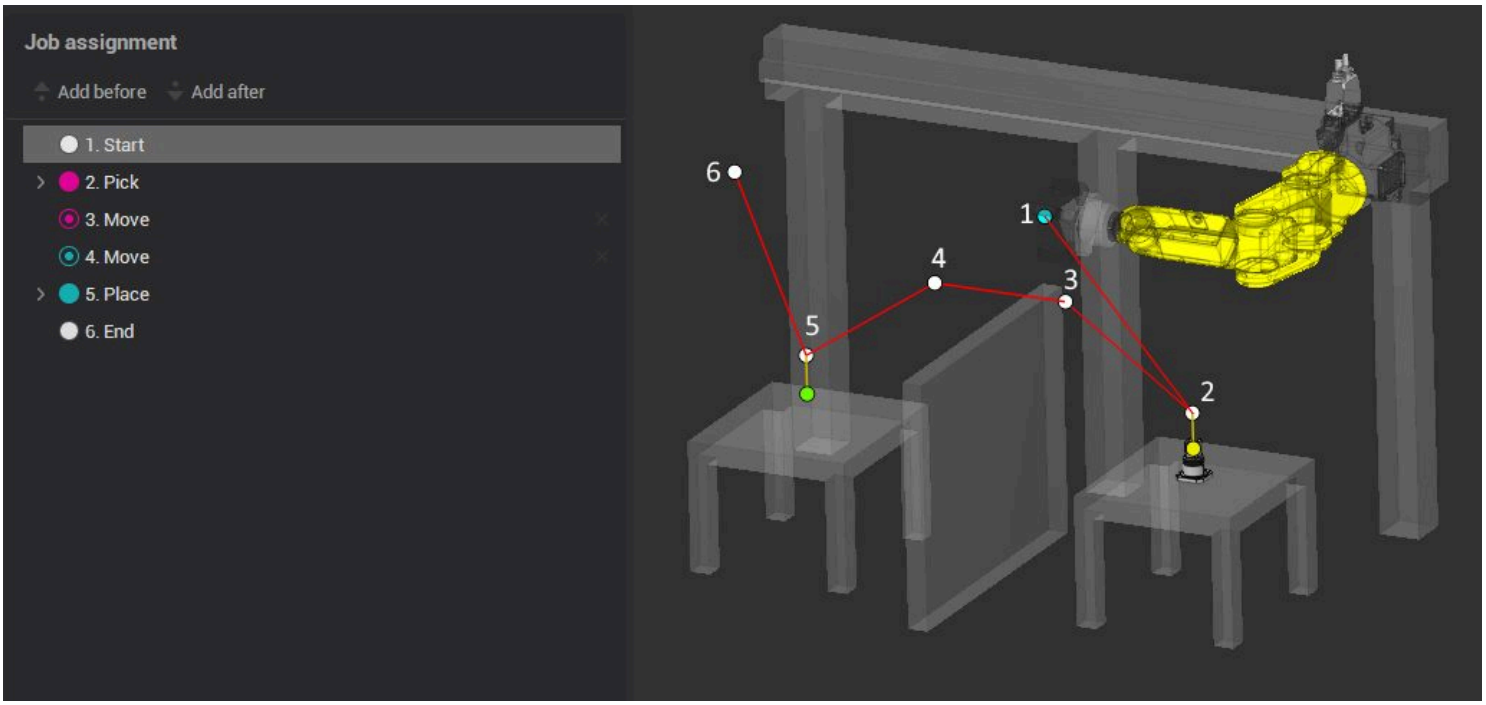


It's based on the Pick-and-place operation.

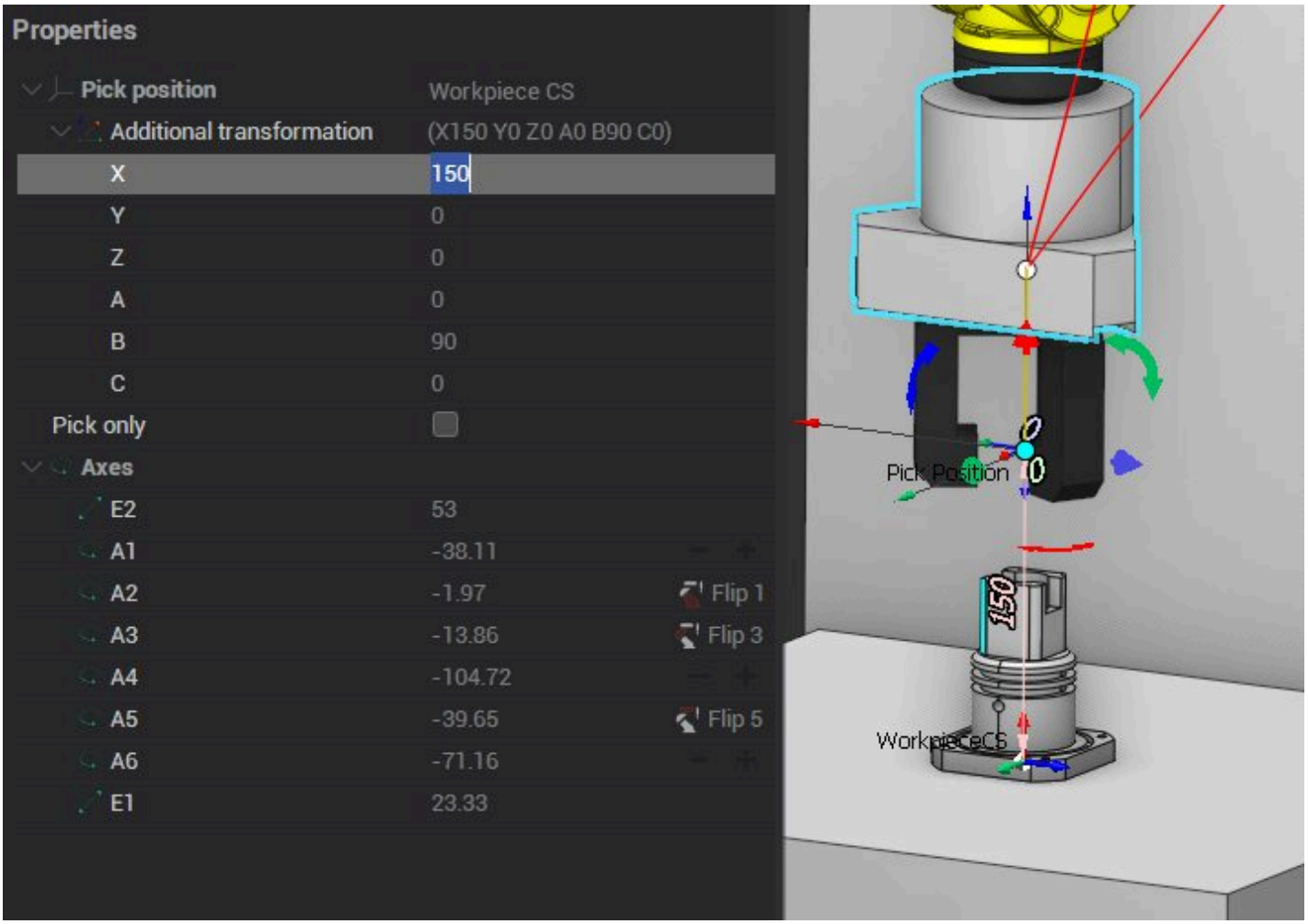
The job assignment is built on nodal points. At points, a position is set for moving the machine. By adding and removing points, you can set the desired movement of the part.

A job assignment consists of several types of points:

1. Start point
2. Pick point
3. Move point (relative to Pick point)
4. Move point (relative to Place point)
5. Place point
6. End point These points can be used to construct a chain of trajectories.



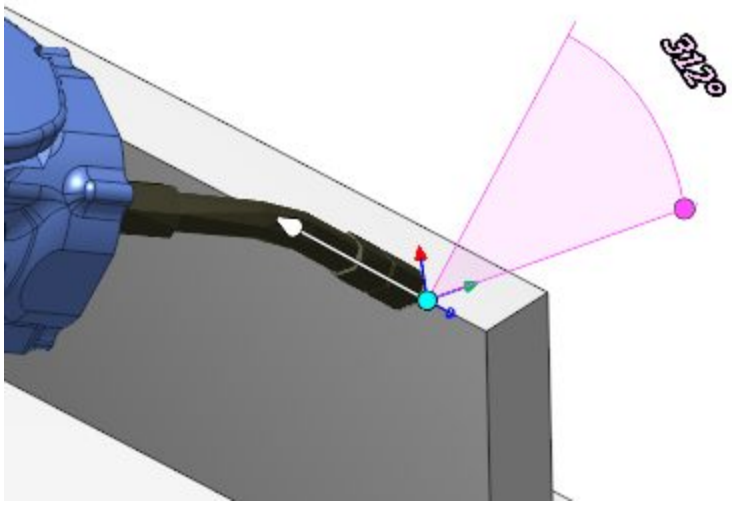
1. ◯ Start point It is read-only. This is the end point of the previous operation
2. ● Pick point The grip position of the part. Properties:



Pick position is defined in geometry coordinate system with additional transformations. if Pick only is selected, the trajectory stops at this point. Axes - machine axis at the current point. When

changing axes, two buttons appear: 

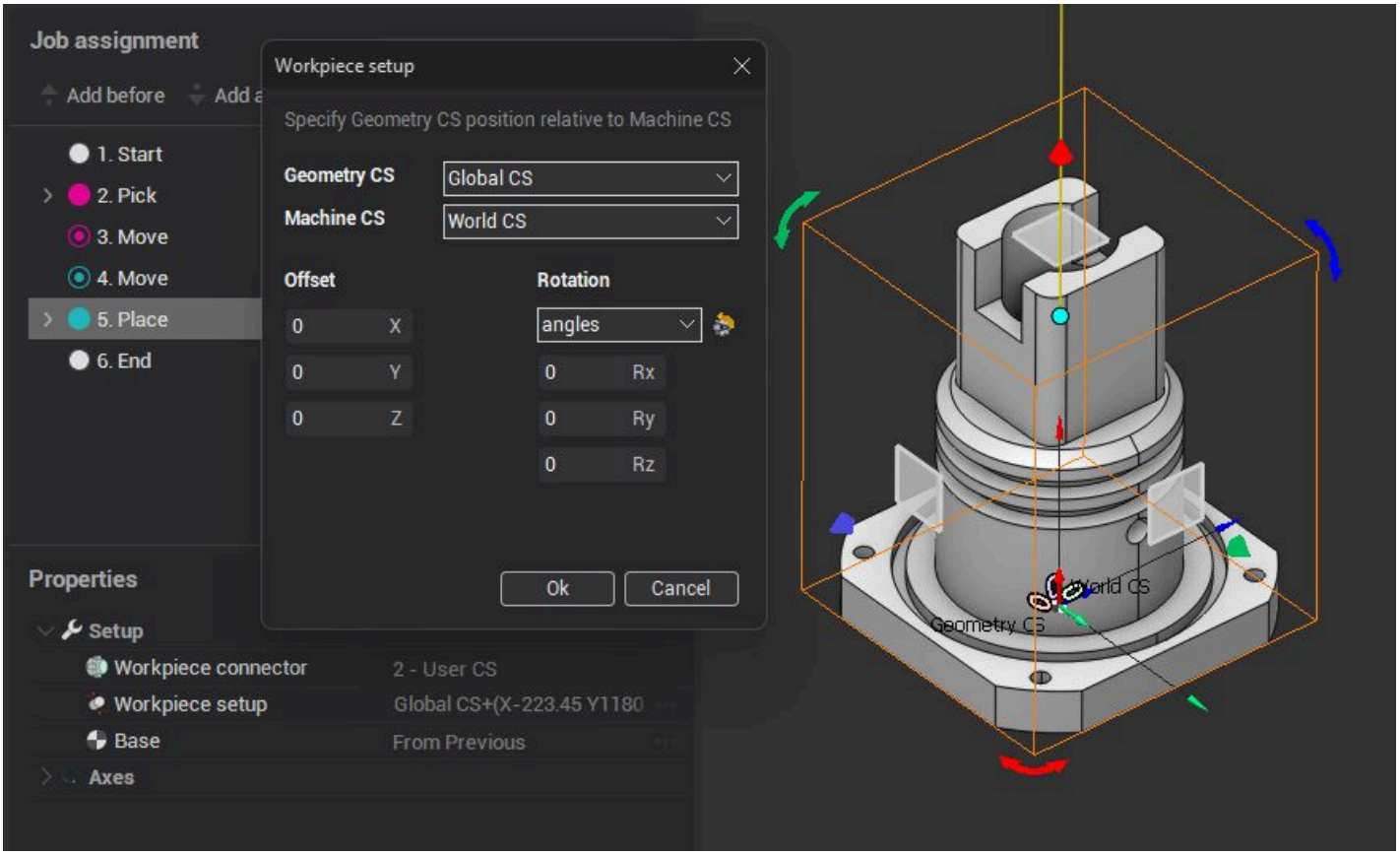
- moves the point to the tip of the machine - returns previous values - 360/+360 Flip - controls flips in the robot

3. Move point (relative to Pick point) The page has buttons "Add position before" and Add position after" to add intermediate points. By default, points are added relative to the selected one. if you change the initial position of the part, then by default this point will also move. Properties: Fix vX - enable 6 axis edit mode:  Motion type - is set by what type to move to the point:

- MultiGoto - Multi coordinate movement
- PhysicGoto - Physical machine axes movement
- Avoid collisions - Collision avoidance movement

4. Move point (relative to Place point) Differs from the previous point in that the point moves with the end position of the part.
5. Place point Position where the part should be placed. The position where the part is to be placed is defined by the workpiece connector and workpiece setup. All movements of the gripper is

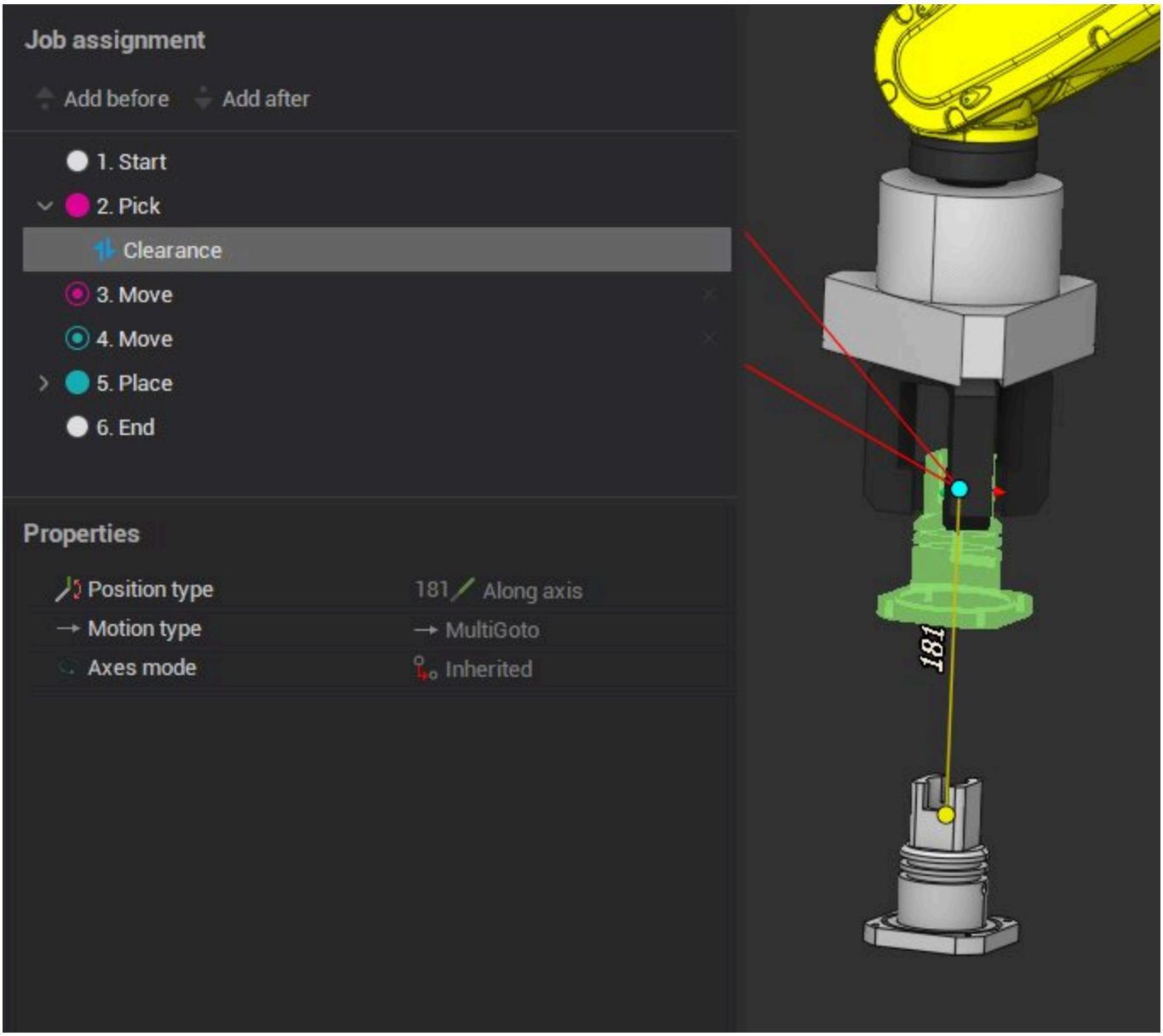
generated in the defined workpiece coordinate system.



6. ◯ End point This is a return to the tool change point. If it is off, then the trajectory ends at a Place point.

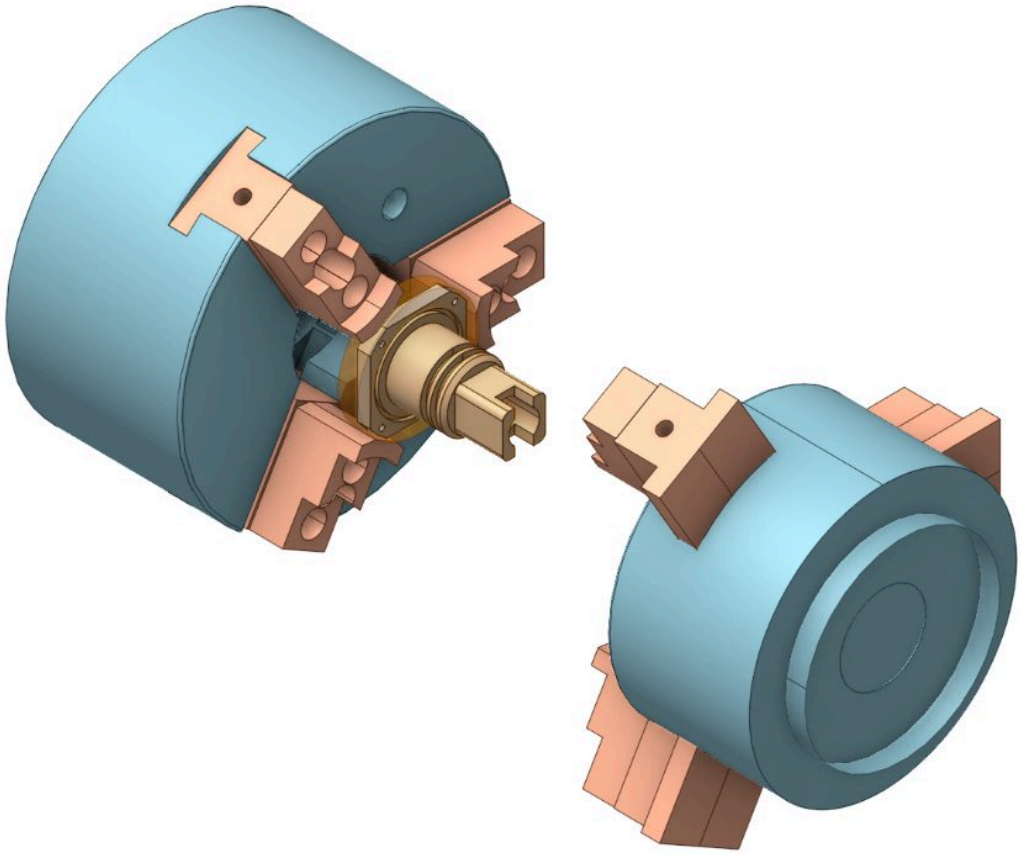
Clearance point

Using these points you can define the length and the direction for the engage/retract movements to the pick and/or place positions.



How to drag points in interactive can be found in [Point Welding operation]

Turn take over



Application area:

Turn take over is the special kind of pick and place operation designed to use in the lathe or swiss-lathe project template. The main difference of the Turn take over operation from the simple pick and place (Details) is that it takes Workpiece connector and Setup CS automatically from the next part of the project. Also the set of operation parameters is adapted specifically for the lathe or swiss-lathe type of machining.

Setup:

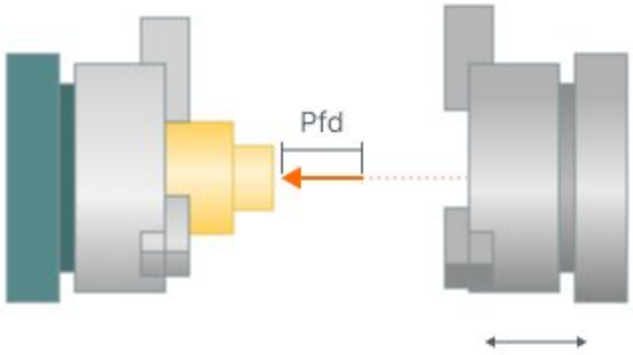
The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. See mor

Strategy:

Pick feed distance.

The length of the movement which is done using the engage feed before picking the part.

▼ Click here to expand..

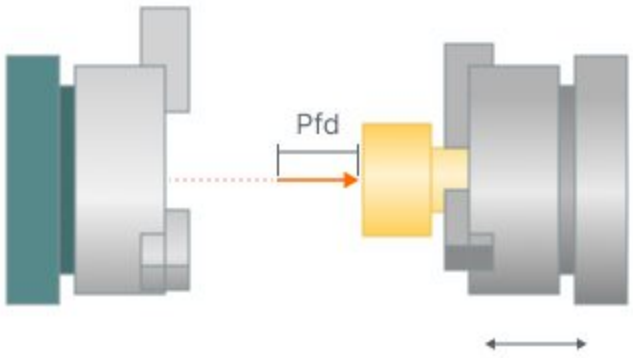


Pfd - Pick feed distance

Return feed distance.

Length of the spindle movement on "Retract" feed after picking the part.

▼ Click here to expand..



Pfd - Return feed distance

Use canned cycle.

Output the operation as a cycle.

Integrate with

Takeover operation will be synchronized with the selected Lathe part-off operation

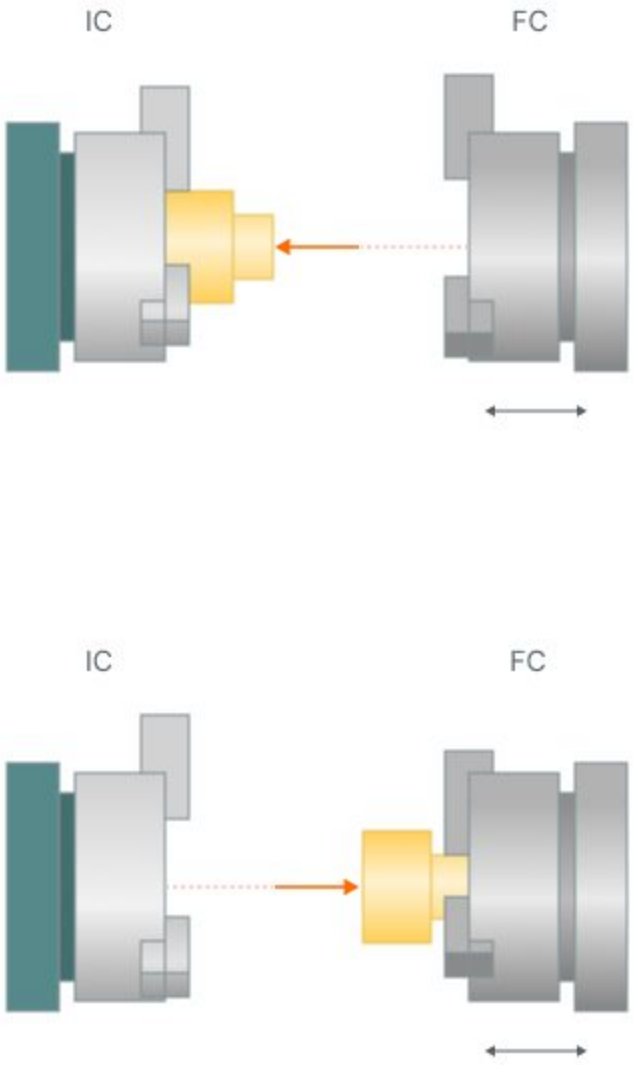
Physic output.

"Multigoto" or "Goto" commands and coordinates are generated to move the machine axes

Initial clamp(IC)/Final clamp(FC)

The clamping device that initially holds the part. Special CLData commands for clamping/unclamping the device are output automatically

▼ Click here to expand..

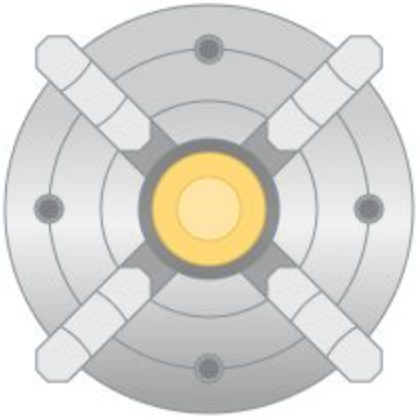


IC- Initial clamp FC - Final clamp

Clamped position. Clamp axis position which corresponds to the Clamped state of the device



Unclamped position. Clamp axis position which corresponds to the (fully) Unclamped state of the device



Phase synchronization in Turn take over

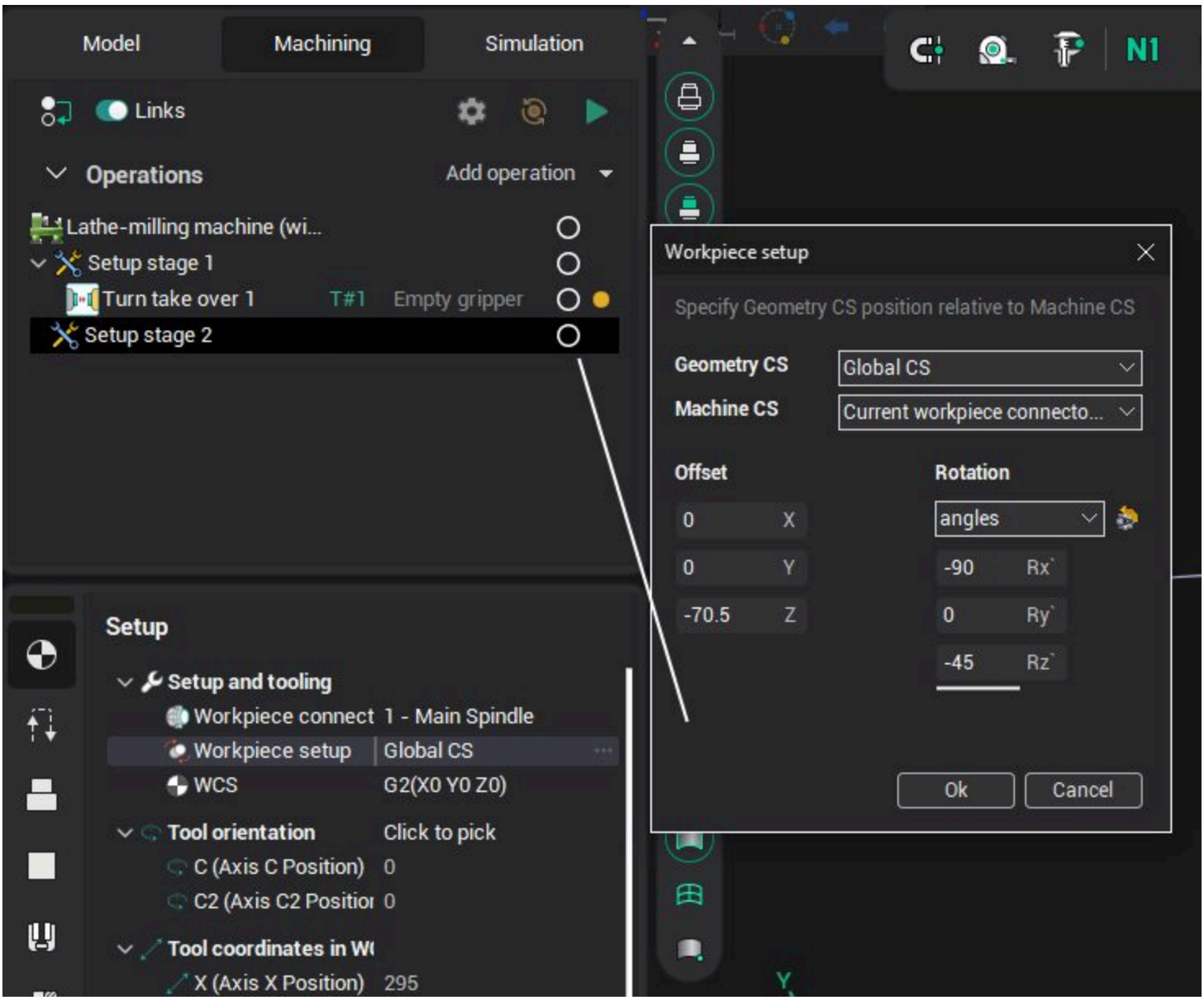
Application area:

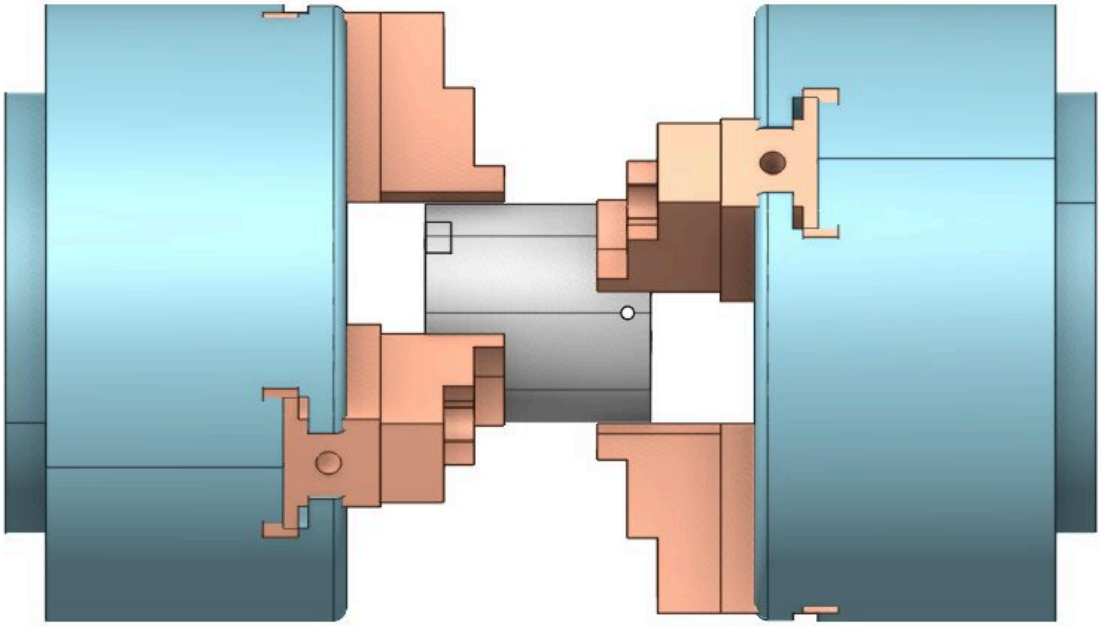
It is necessary to perform interception of parts with different phases by synchronizing the spindles.

The peculiarities of working:

To establish different phases (angles) of the spindle, it is necessary to indicate a phase different from the initial one in the subsequent position of the part (installation/part).

For example, in the figure below the angle is 45 degrees.





```
%
Opereh01
(Turn take over 1)
T101
G99
(Empty gripper)
G97 S200 M03
G28
M111 (sub chuck open)
G4 P0.5
G00 Z10.
Z20.
G199 R-315. (Sync spindles rotation)
M14 (Air on)
G4 G01 G98 F200. P0.5
M110 (sub chuck close)
G4 P0.5
M15 (Air off)
M11 (main chuck open)
G4 P0.5
G198 (Cancel spindles sync)
M05
G00 Z10.
G28
M30
%
```

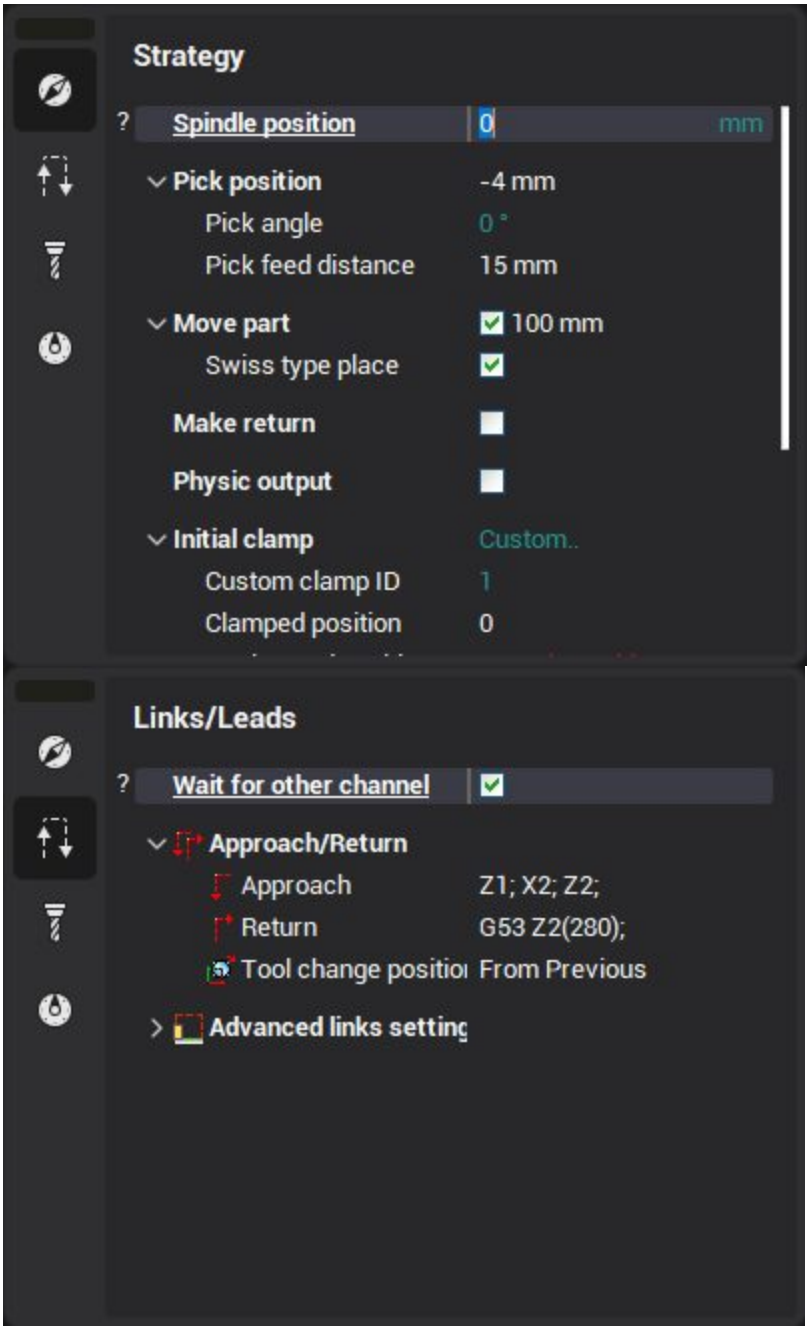
Sub spindle working

Operation overview

Sub spindle working operation is an adaptation of common pick-and-place operation for the turn-milling machines with sub spindle. It makes possible: to synchronize the main and counter spindles. to take the part by the counter spindle for the further machining in both spindles to move the part in the main(counter) spindle

So the tool path of this operation has all the same sections that common pick-and-place has.

Sub spindle working parameters



Spindle position

Initial main spindle position in the physical coordinates for this operation (if the machine is lathe/swiss-lathe with 2 spindles). It is outputted in the return section of the previous operation.

Pick position

The position of the sub-spindle where it will hold the part.

Pick feed distance

The length of the movement which is done using the engage feed before picking the part.

Do place

If the parameter is enabled, then you can define the new position of the workpiece in the active spindle (distance from the WCS to the spindle base point). If the parameter is disabled, then the position of the workpiece in the active spindle will not be changed.

Swiss type place

If the parameter is enabled, then the active spindle is moved without the workpiece, otherwise the sub-spindle is moved with the fixated workpiece.

Make return

If the parameter is enabled, then the main and opposite spindles will be unsynchronized and moved to the home position. Disable this parameter if you want the spindle and sub spindle to remain synchronized and connected. All further operations are performed on both spindles if there is no return move.

Return feed distance

The length of the movement which is done using the retract feed after picking the part.

Clamp parameters

Define the clamp devices used by the main or opposite spindle. For more information about the clamp control parameters, see the Clamp devices control article.

Wait for other channel

If this parameter is enabled, then the Wait CL-data command will be inserted before the approach of the Sub spindle working operation. This means that this operation will start the machining only after the operation from the other channel completes its machining.

This parameter is visible only if there are at least 2 channels in the machine schema and the previous operation is from another channel.

Approach/Return parameters

The common (for the majority of the CAM system operations) group of parameters, defining the approach and return sections of the operation. See the Approach and return rules documentation for more info.

Spindles synchronization

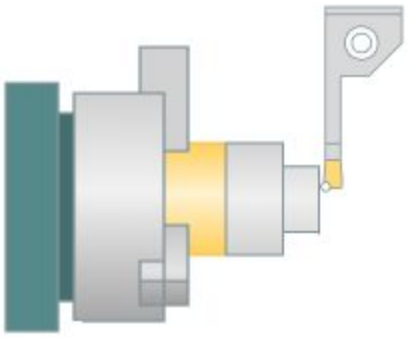
The Sub spindle working operation automatically generates New CL-data command for the spindles synchronization.



Example project(s)

See the *Turn-Milling/Hanwha* sample project for an example of lathe part machining with the spindles synchronization.

Bar feeding



Application area:

The *bar feeding* operation is essential in modern machining for automating the production process. Its main purpose is to continuously and automatically feed long metal bars into a machine tool — most commonly a CNC turning centre or lathe.

Setup:

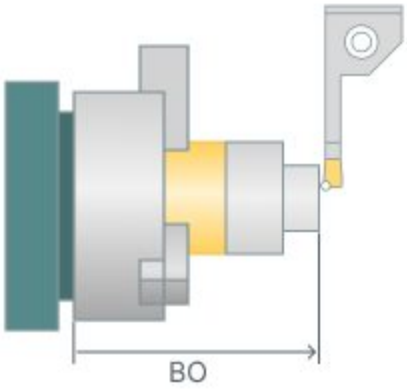
The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. See mor

Strategy:

Bar overhang.

The axial distanse between the spindle base point and the tool tip point.

▼ Click here to expand..



BO - Bar overhang

Tool touch position.

The touch position of the tool tip point in the workpiece coordinate system (G54).

▼ Click here to expand..

X,Y,Z. - The X,Y,Z axis of the specified coordinate system is the Rotary Axis.

Start with target.

▼ Click here to expand..

Enable. When the part was displaced, the coordinate system remained in place.

Disable. When the part was displaced, the coordinate system moved with it.

WCS Use canned cycle.

▼ Click here to expand..

Enable. The bar transferring process will be formalized as a cycle for the further analysis in the postprocessor

Disable. The bar transferring process will be output as a sequence of elementary commands

Physic output.

▼ Click here to expand..

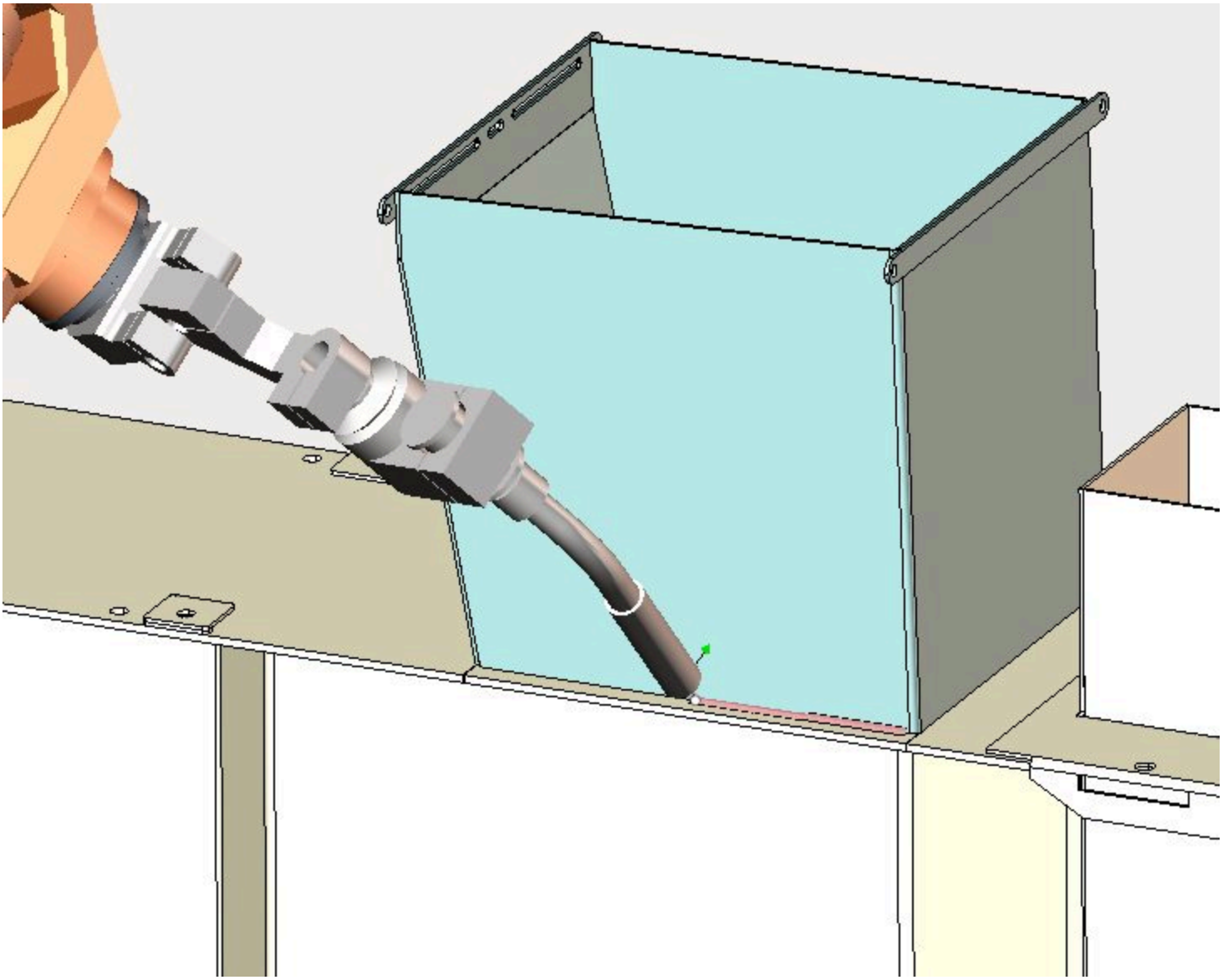
Enable. Physicgoto command and physic coordinates are used to move the machine axes

Disable. Multigoto or Goto commands and coordinates are generated to move the machine axes

Initial clamp

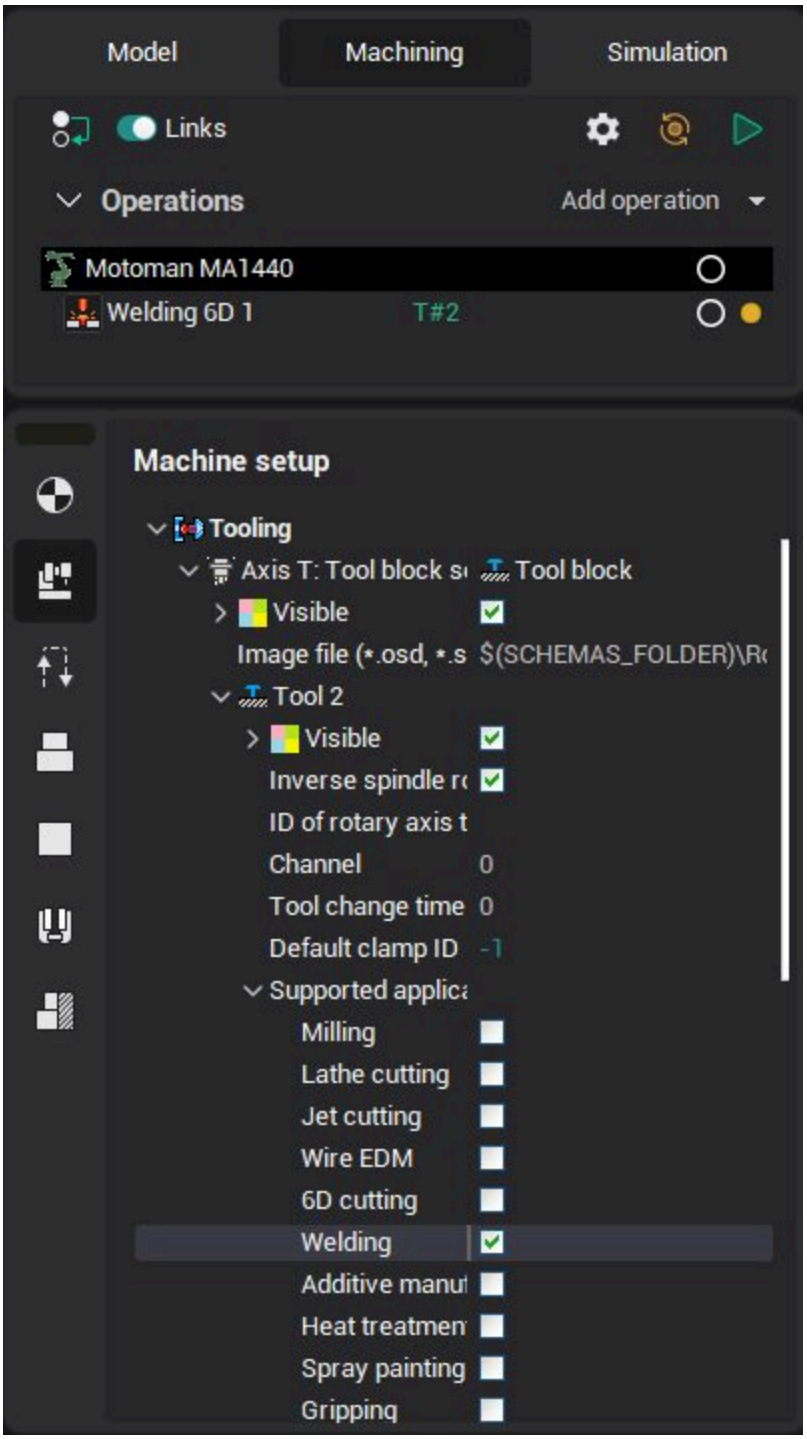
The clamping device that initially holds the part. Special CLData commands for clamping/unclamping the device are output automatically. Parameter group works similarly to the Turn take over operations. Details

Welding



To work with welding equipment is currently designed a universal welding operation 5D. It implements the functionality of automatic weld seam geometry calculation without reference to the particular type of welding equipment, ie it does not generate the specific commands to the laser, electric arc, gas burners, ultrasonic emitters etc.

In order for the welding operation has become available for the creation should be chosen machine or a robot that supports this type of machining. To ensure support of welding in machine settings need to be set for the tool holder corresponding checkbox as shown below.



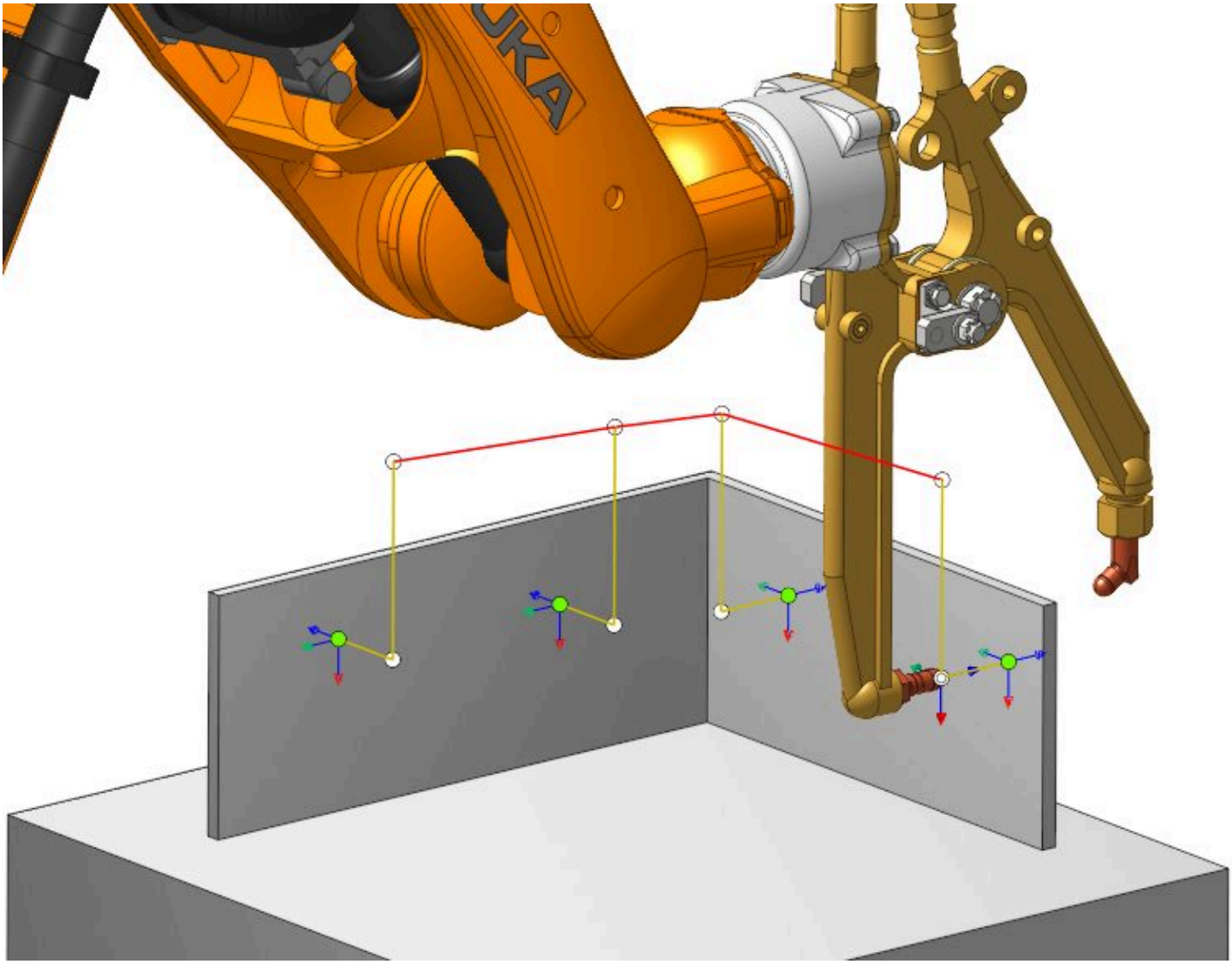
Or you can write to the machine scheme for the tool holder lines similar to the following.

```
<SCType ID="WeldingToolHolder" Caption="Welding tool holder" Type="TToolHolderNode">
<SupportedToolTypes>
<Welder DefaultValue="true"/>
</SupportedToolTypes>
</SCType>
```

Setting up of operation to work with specific type of equipment can be made by writing in the postprocessor appropriate commands to control the equipment, or, if this is not enough, by the addition

of a special operation on the basis of the Welding 5D operation, adapted to control specific equipment.

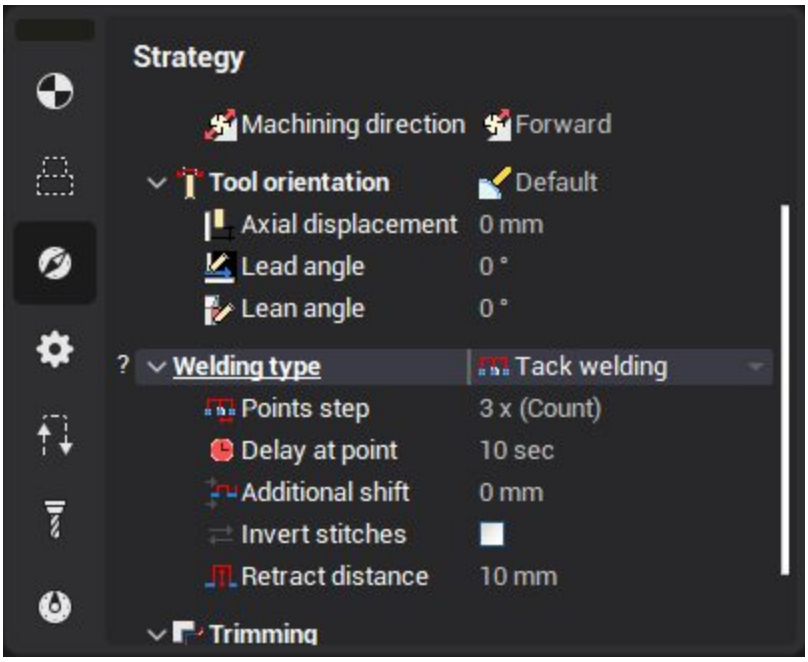
Point welding



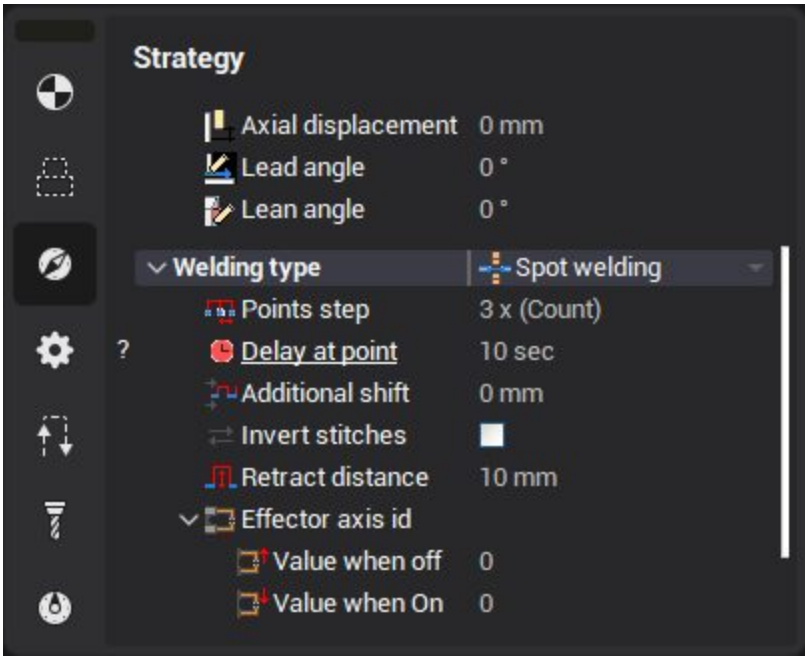
Operation can be used to Tack weld and Spot weld.

Job assignment

In the strategy tab, you need to select the type of welding: Tack weld

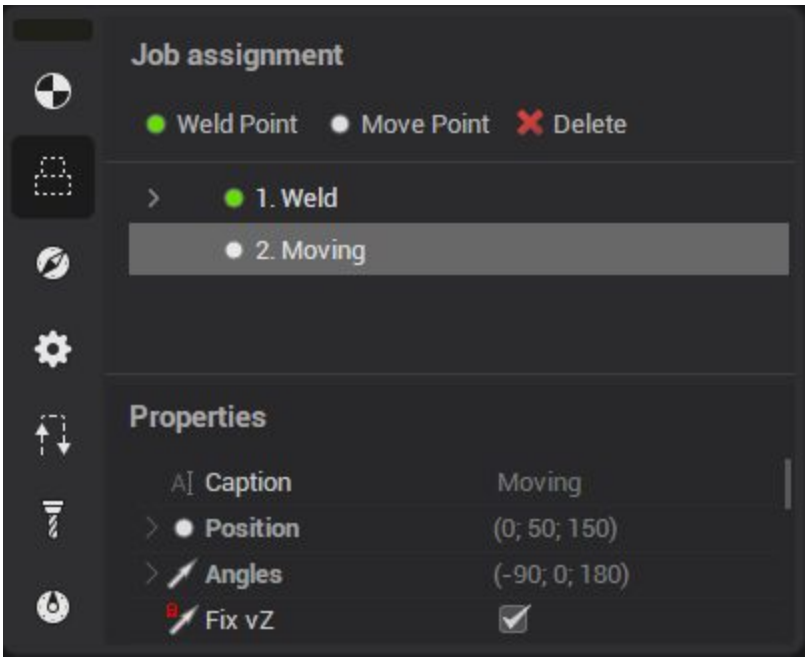


Spot weld



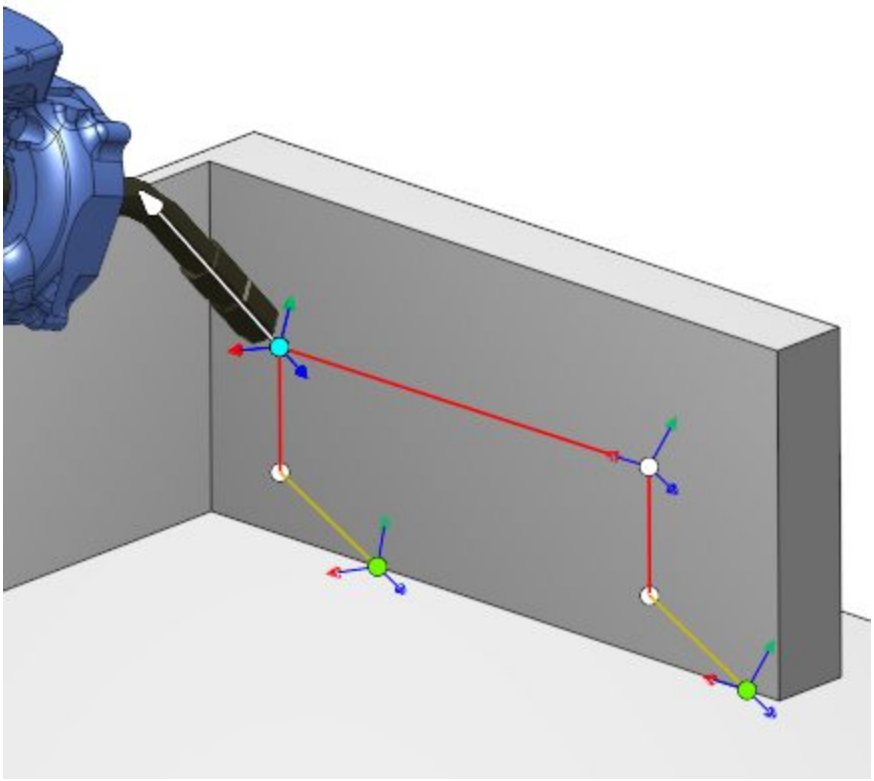
For both types the job assignment is the same.

The job assignment consists of two types of points:



Weld Point ○ Move Point

These points can be used to construct a chain of trajectories:

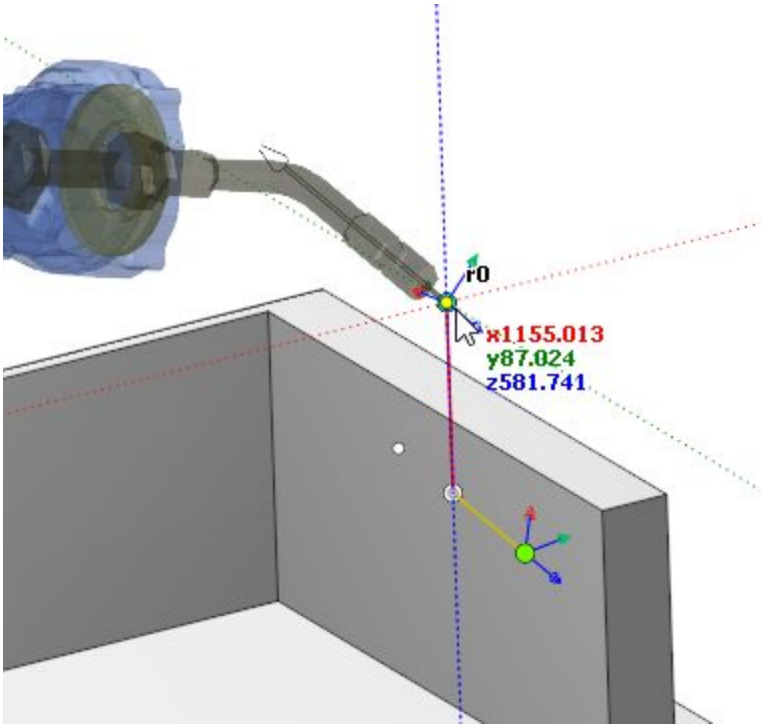


Drag points

These points can be moved by dragging.

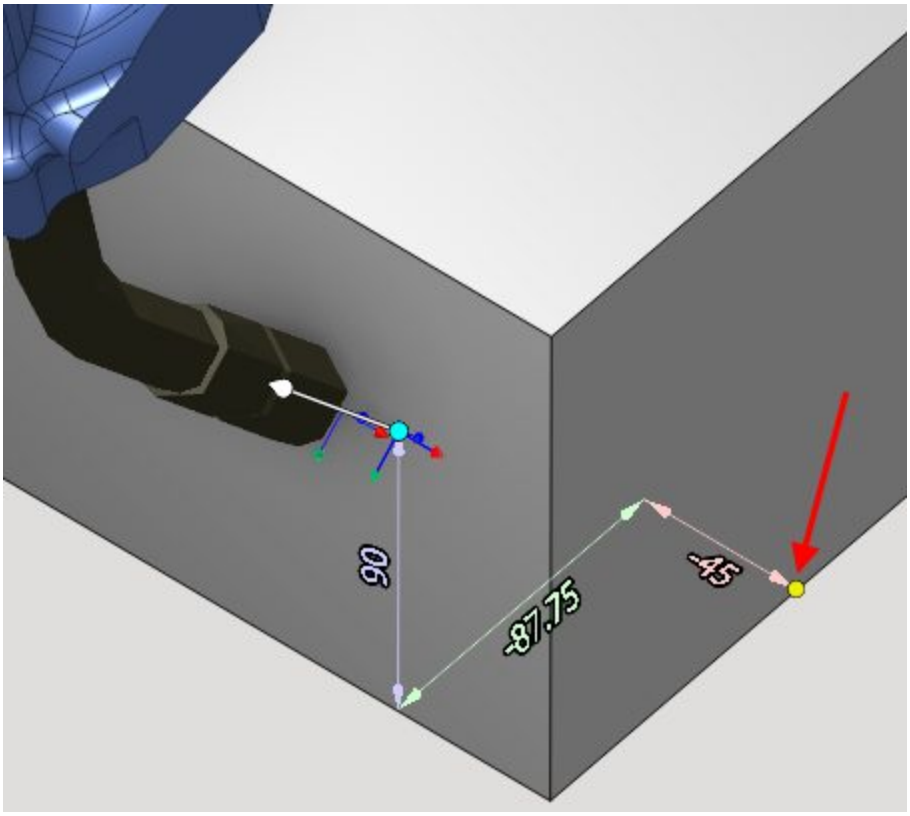
They attach to faces , curves , splines , edges , vertices . Tool axis rotates automatically when attached.

Move points can be easily moved using Smart snap :

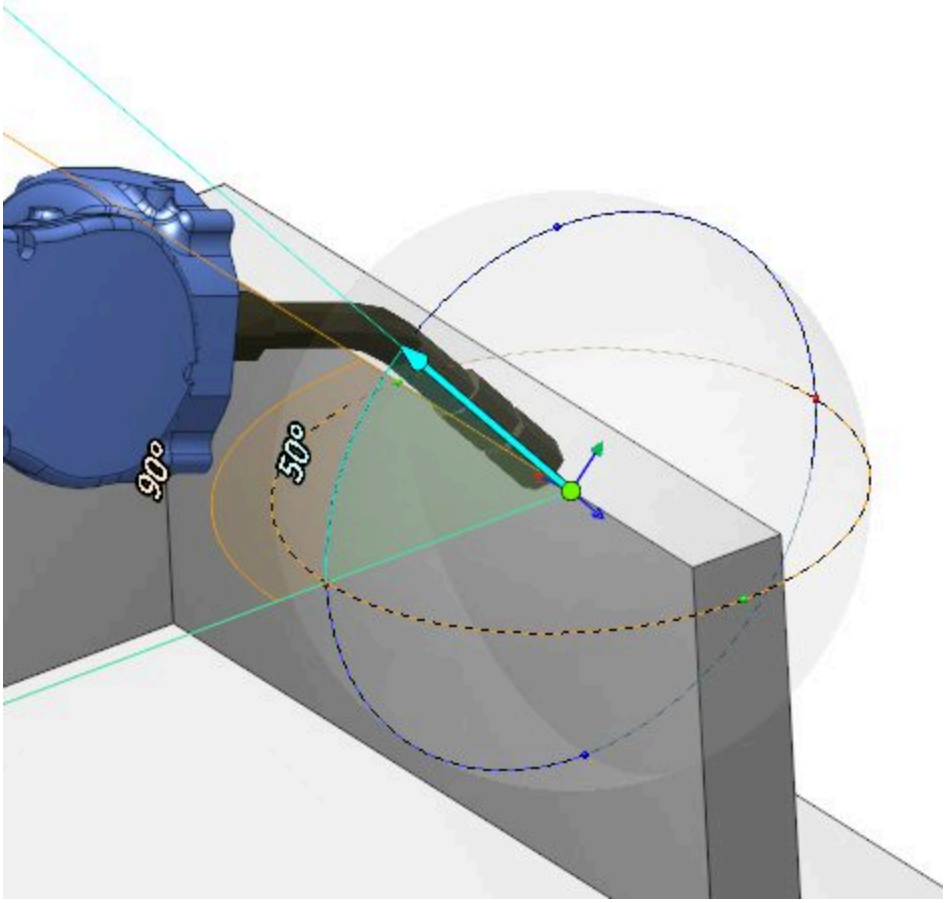


Position of the point can also be specified as an offset from the auxiliary yellow point.

To do this, first select the yellow point:



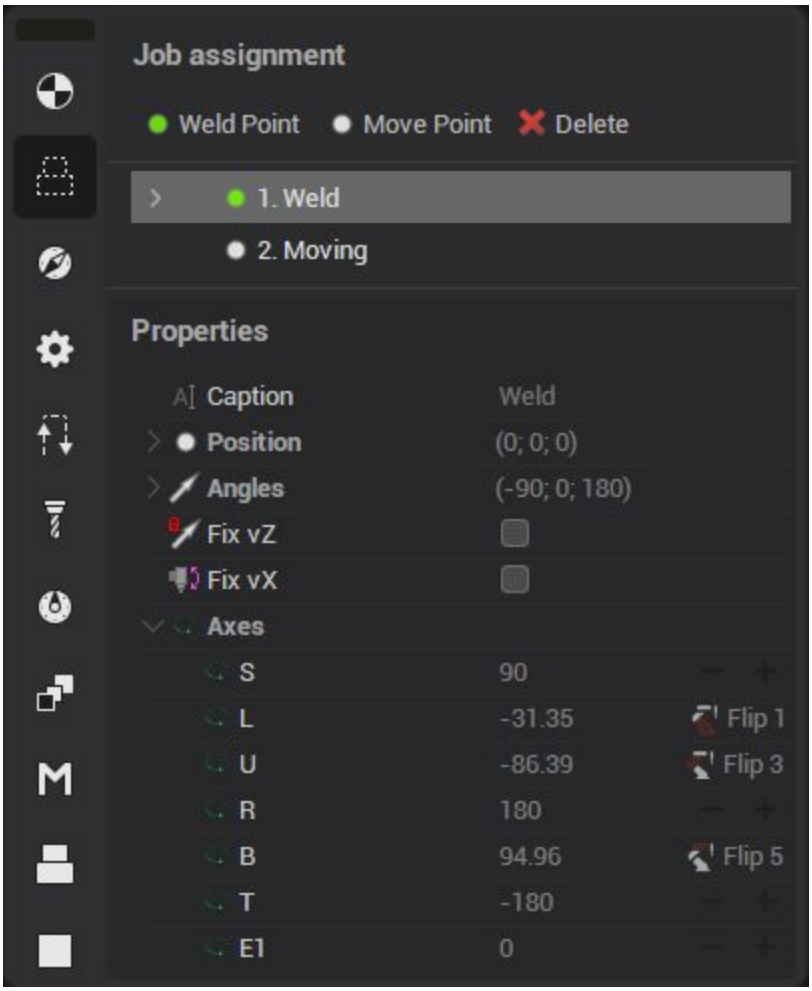
You can also rotate the axis vector by dragging the visual vector:



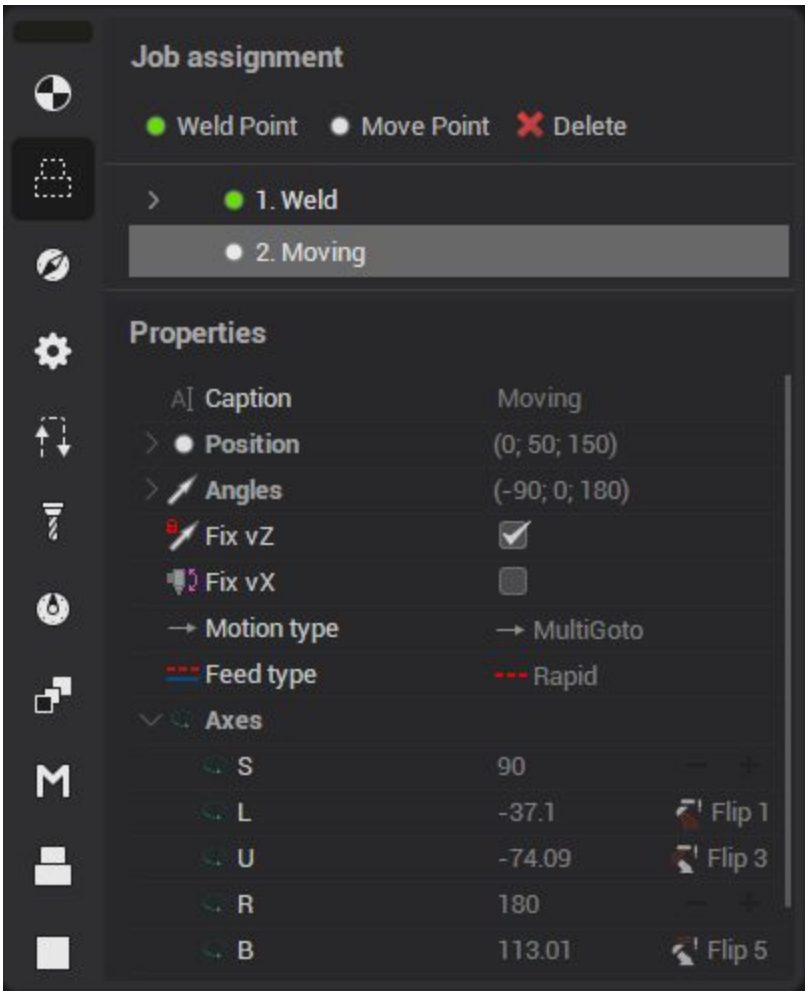
Point parameters

Points contains the following parameters:

- Weld Point


Job assignment
Weld Point
Move Point
Delete
1. Weld
2. Moving
Properties
Caption
Weld
Position
(0; 0; 0)
Angles
(-90; 0; 180)
Fix vZ
Fix vX
Axes
S
90
L
-31.35
Flip 1
U
-86.39
Flip 3
R
180
B
94.96
Flip 5
T
-180
E1
0

- Move Point



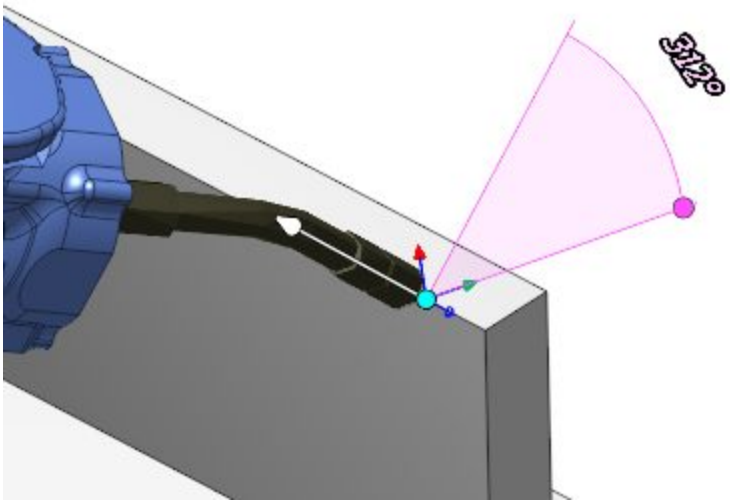
Caption - point name

Position - point coordinates.

Angles - tool axis inclination angle at point

Fix vZ - if enabled point vector does not change when dragged

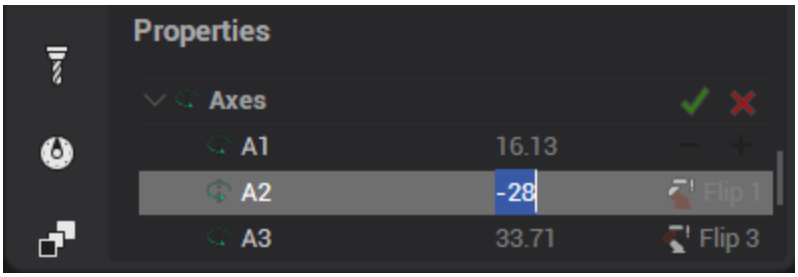
Fix vX - enable 6 axis edit mode:



→ Motion type - is set by what type to move to the point:→ MultiGoto - Multi coordinate movement ⇝ PhysicGoto - Physical machine axes movement ↷ Avoid collisions - Collision avoidance movement

Axes - machine axis at the current point.

When changing axes, two buttons appear:

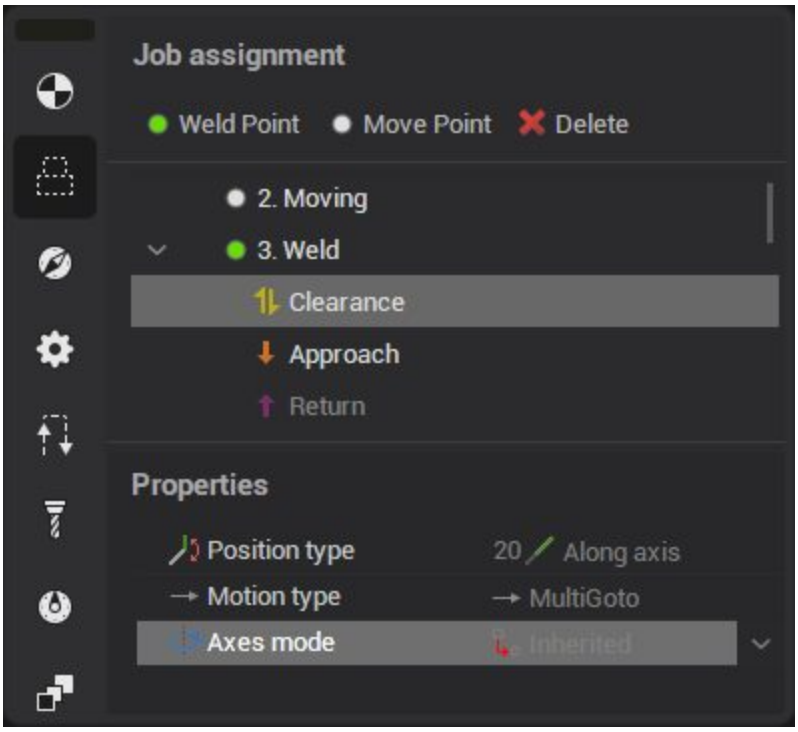


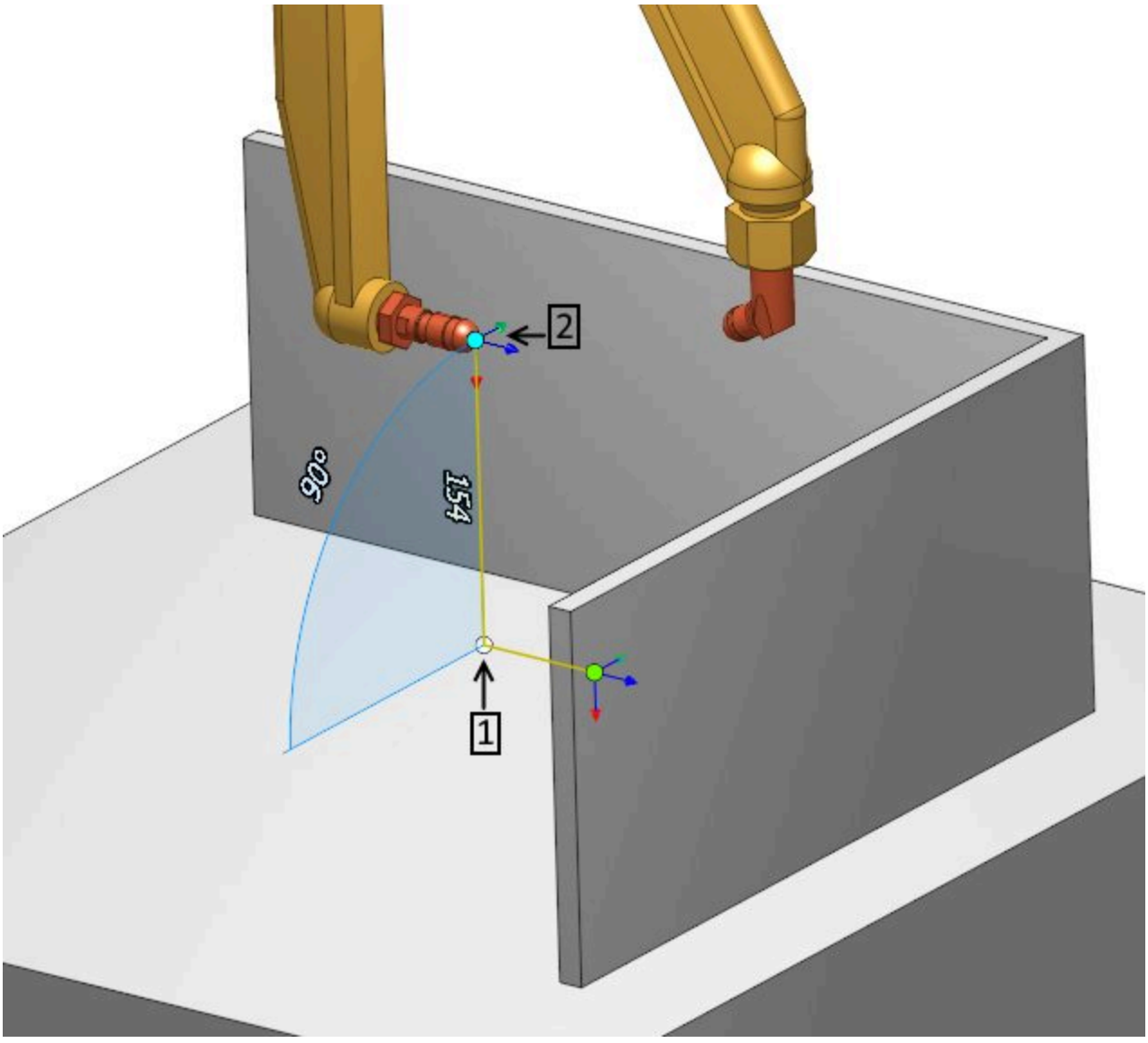
- ✔ - moves the point to the tip of the machine
- ✖ - returns previous values
- − + - 360/+360

Flip - controls flips in the robot

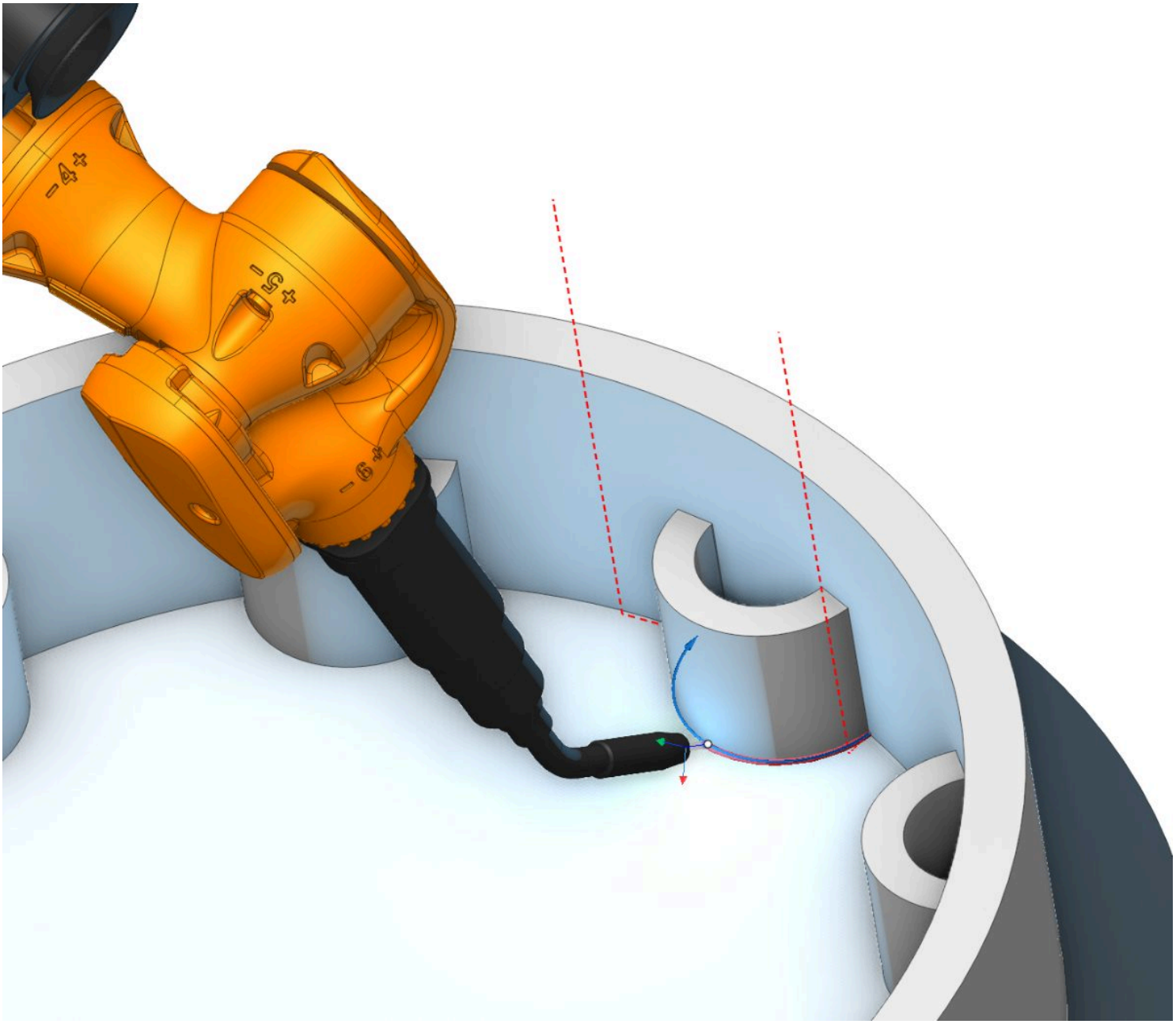
Auxiliary Point

- Weld point also contains additional auxiliary points:Clearence Engage Retract


Job assignment
Weld Point
Move Point
Delete
2. Moving
3. Weld
Clearance
Approach
Return
Properties
Position type
20
Along axis
Motion type
MultiGoto
Axes mode
Inherited


2
90°
154
1

Welding 5D and Welding 6D



Application area:

The Welding 5D and Welding 6D operations calculate the geometry of a weld seam automatically, without reference to a particular type of welding equipment. The operations do not generate equipment-specific commands to the laser, electric arc, gas burner, ultrasonic emitter, and so on; they produce the toolpath and the tool orientation along the seam.

The operations are used in the following cases:

- Continuous and intermittent welding of seams defined along edges between joined parts.
- Robotic and multi-axis welding, where the tool orientation must be maintained relative to the adjacent walls of the joint.

- Processes that require a specific stitch pattern (stitch, tack, or spot), configured through the Welding type parameter of Welding 6D.

Switch to the Simulation mode to visualize material deposition at the point where the welding head tip contacts the workpiece. The simulated layer thickness is determined by the tool's working length property; if this property is not defined or its value exceeds the tool diameter, the layer thickness equals the tool diameter.

The Effector On/Off command turns welding on and off. The command is set when the simulation type (Details) is additive or paint, according to the selected Effective Feeds. For additive operations and welding, the command is set to automatic. Details

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Weld seam. To define the welding path, it is sufficient to specify the edge between the welded parts. The system then automatically calculates the weld head orientation angles for each point along the generated curve. The calculation ensures that the weld head is positioned as close as possible to the centre of the gap between the adjacent walls of the welded components, while simultaneously preventing any collision with the walls.

5d Curve. The system generates the 5-axis tool path as the projection of the arbitrary curves on the part surface. In this case the tool contact point will move in touch with the surface. The passes can be shifted from the curve by the stock or by the levels. The tool axis is located perpendicular to the surface. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Tool contact point.

Tool Contact Point (TCP) is a specific location on a cutting tool that defines the primary interaction point between the tool and the workpiece surface or geometric entities during machining operations.

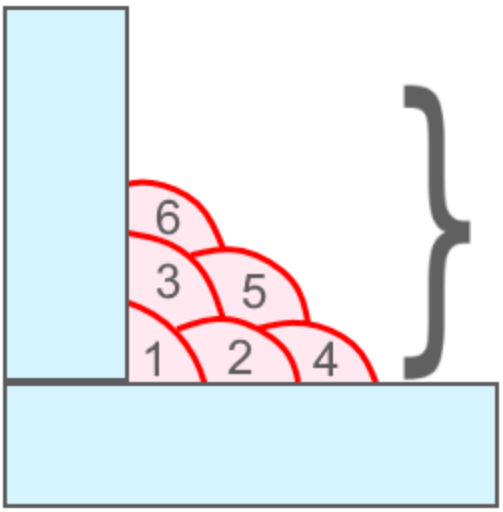
The parameters of this group are the same as in the 5D Surfacing operation. Details.

Multipass.

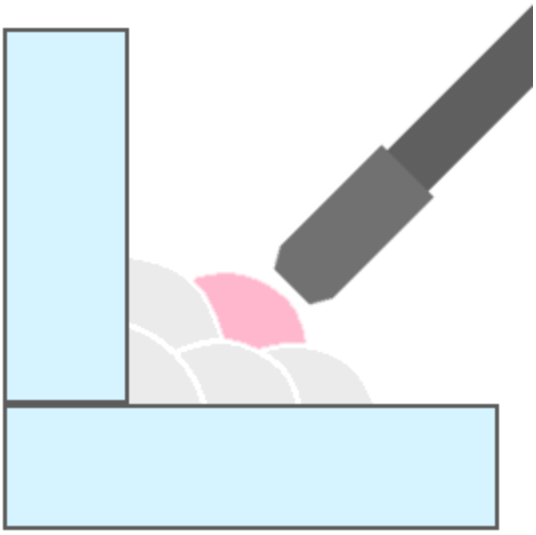
Enables welding of the seam in several passes.

▼ Click here to expand..

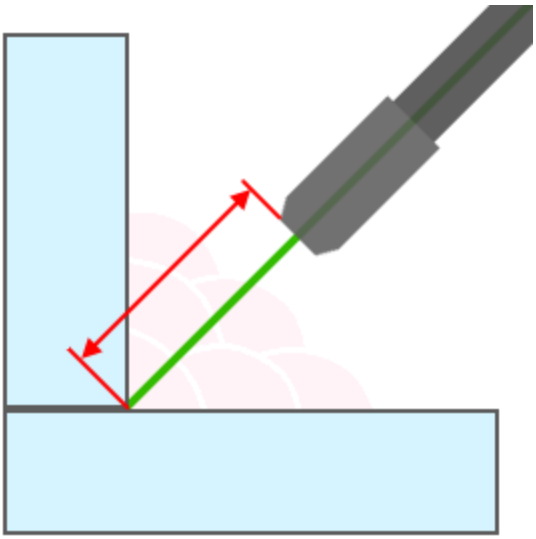
Pass count. The number of passes for all weld seams in the operation. It's possible to define this parameter separately for every weld seam on the job assignment page.



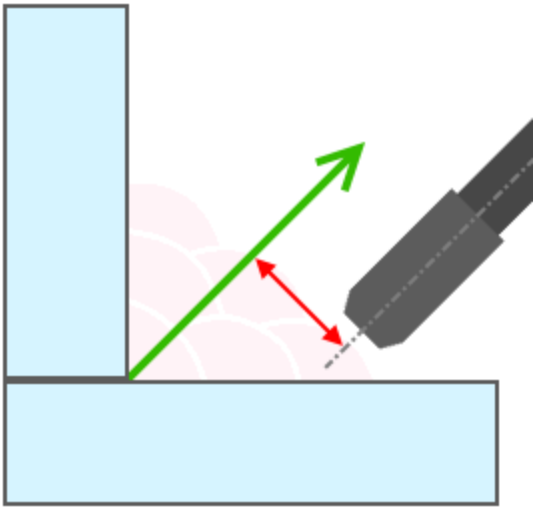
Pass. The parameters of the separate pass



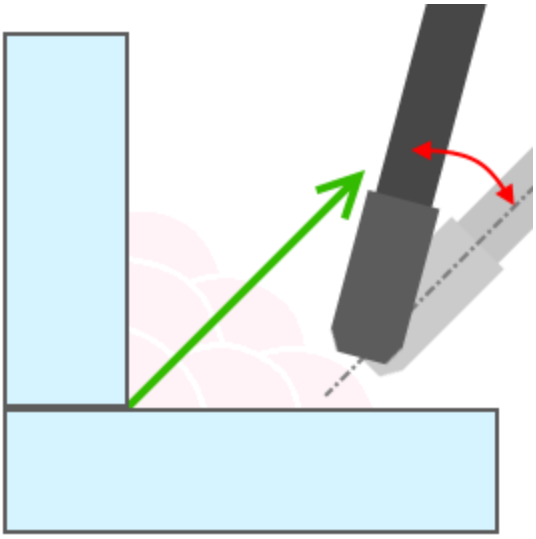
Axial offset. The distance from the main edge to the pass along the tool axis.



Side offset. Offset to side from edge to pass.



Lean offset. The additional inclination of the tool axis from the main direction.



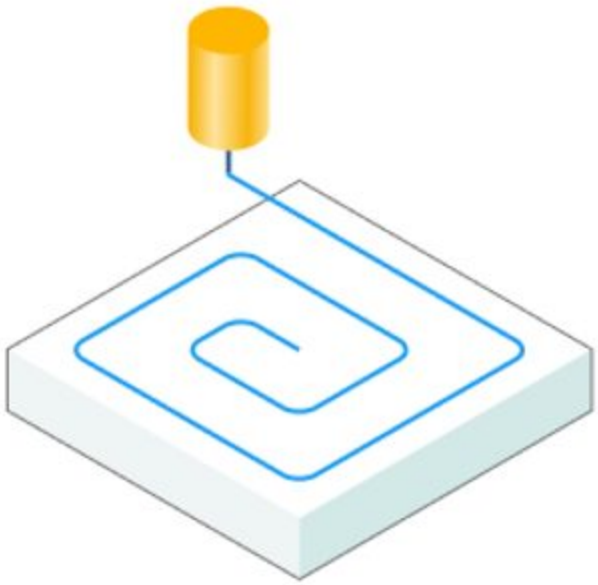
Offset return. Defines the return offset between passes.

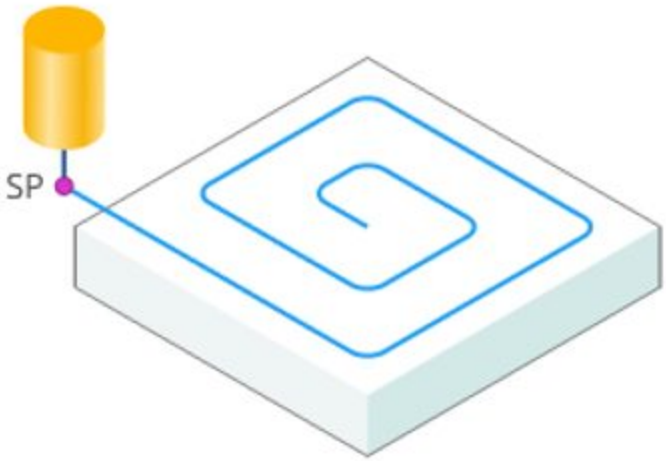
Sorting.

Controls the sequence of toolpath passes during surface machining.

▼ Click here to expand..

Start Point. Activating the flag permits the input of coordinates marking the starting point of working passes in surface machining.

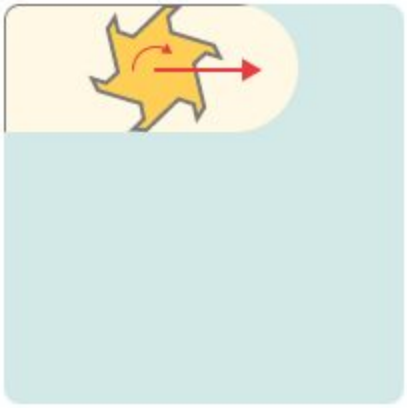




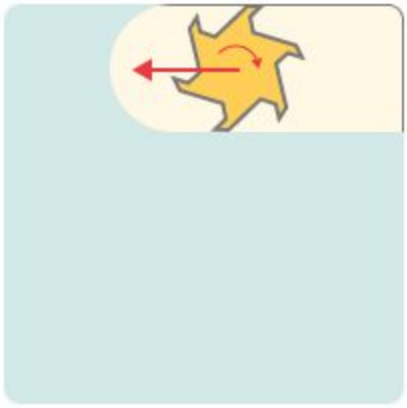
Idling minimization. The order of the profiles machining is sorted to minimize the idle motions. Parameters of this feature are the same as in the 2D Contouring operation. Details.

Machining direction. Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process.

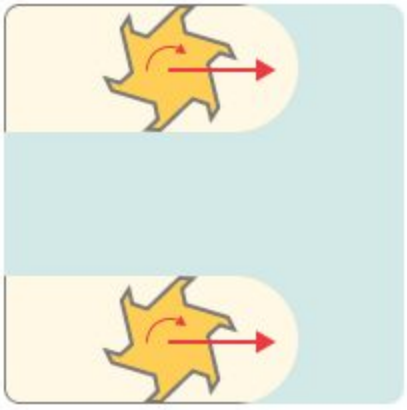
Climb. In machining refers to a cutting strategy where the tool moves in the same direction as the rotation of the workpiece. It typically results in less tool wear and a smoother finish compared to "Conventional" cutting, where the tool moves against the rotation of the workpiece. Climb cutting is often preferred when precision and surface finish are critical.



Conventional. Cutting in machining refers to a strategy where the tool moves against the rotation of the workpiece. This cutting method is used in machining operations, but it can result in more tool wear and a rougher finish compared to "Climb" cutting, where the tool moves in the same direction as the workpiece rotation. Conventional cutting is sometimes chosen when other factors, such as tool life or chip evacuation, are more important than surface finish.



Both. Refers to the practice of using both "Climb" and "Conventional" cutting strategies during a machining operation. This approach can be employed to optimize tool performance, surface finish, and other machining factors depending on the specific requirements of different parts of the workpiece or different stages of the machining process. It allows for greater flexibility and control in achieving desired results in machining.



Tool orientation.

Controls tool axis orientation. The principal parameters of this function are the same as in the 5D Surfacing operation. Details.

Welding type.

Selects the welding process type.

▼ Click here to expand..

Seam welding. Continuous welding along the curve.



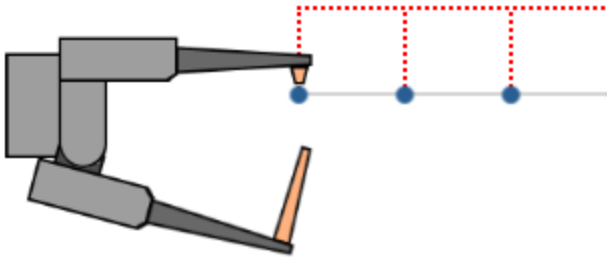
Stitch welding. Intermittent welding by stitches.



Tack welding. Small temporary welds that hold the parts together.



Spot welding. Welding at discrete points.



Here are the key parameters, the set of which may vary depending on the chosen Welding type.

Stitch length. Length of a single stitch.



Start stitch length. Length of the first stitch.



End stitch length. Length of the last stitch.

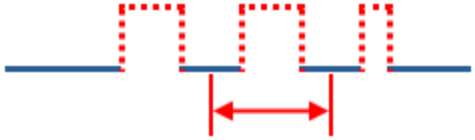


Spacing size. Defines how the spacing between stitches is set.

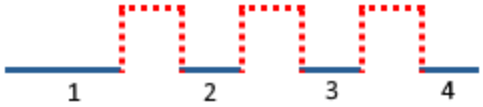
ISO spacing length. The size of the spacing itself is set.



AWS center to center. Sets the distance between the centers of the stitches.



Even (stitch count). The number of stitches is set; the spacings are distributed evenly over the remaining length.



Spacing length, Center to center distance, Stitches count. The numeric value of the spacing, entered in the field below Spacing size. The meaning of the value depends on the selected Spacing size mode: for ISO spacing length it is the length of the gap between adjacent stitches; for AWS center to center it is the distance between the centers of adjacent stitches; for Even (stitch count) it is the number of stitches.

Adjustment. Alignment of the stitches along the contour.

from start.



from center.

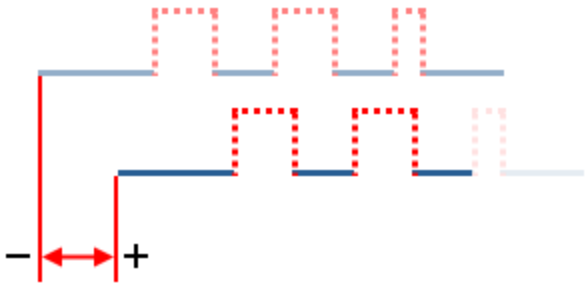


from end.

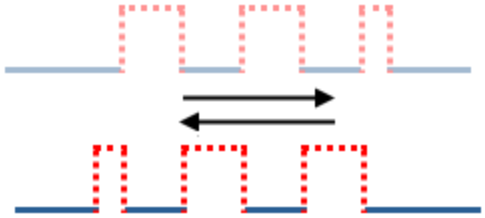


Auto resize spacing. The spacing between stitches is adjusted automatically so that whole stitches fit the contour length without a remainder, keeping the stitch length as specified.

Additional shift. Additional displacement of the stitches along the contour.



Invert stitches. Inverts the distribution of the stitches along the contour.



Retract distance. Distance of axial retraction from the contour at the transition between stitches.



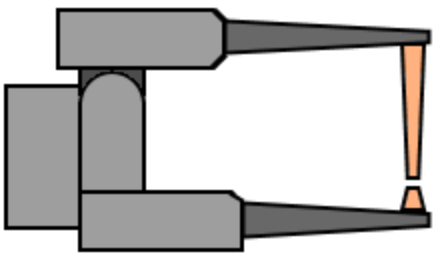
Points step. Step between points.



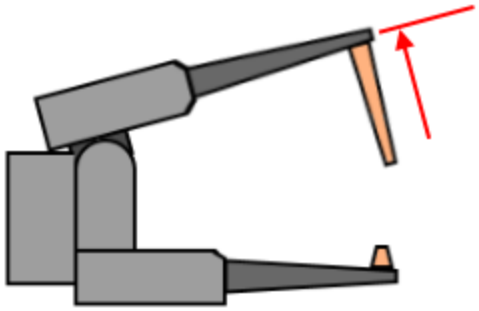
Delay at point. Dwell time at each point, in seconds.



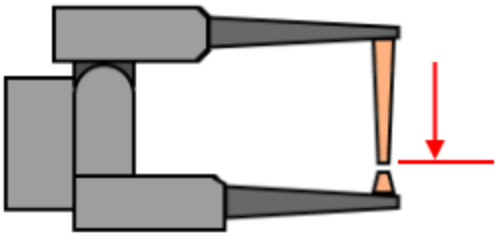
Effector axis id. ID of the machine axis that is the axis of the working body — the welding clamp.



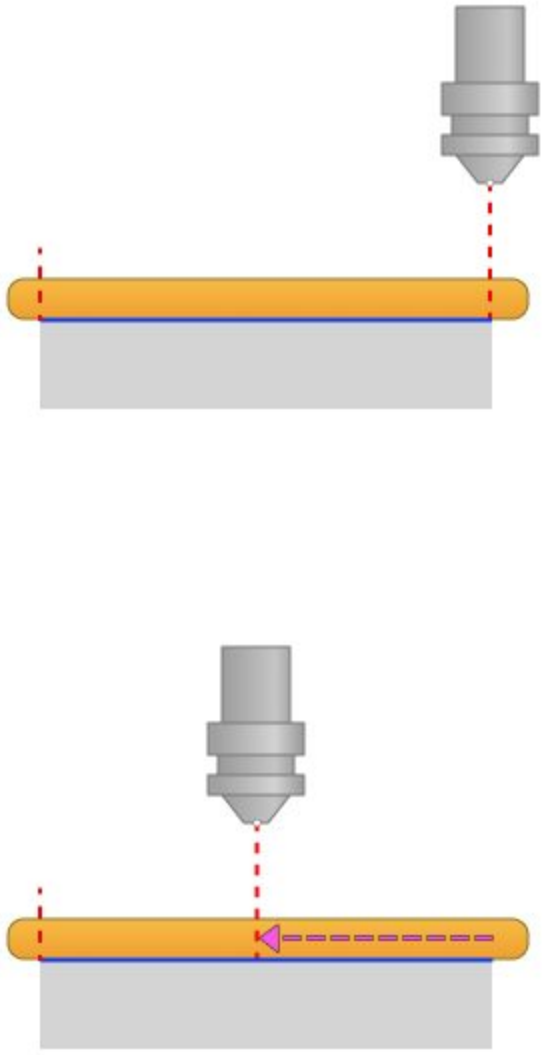
Value when off. Axis value in the off state.



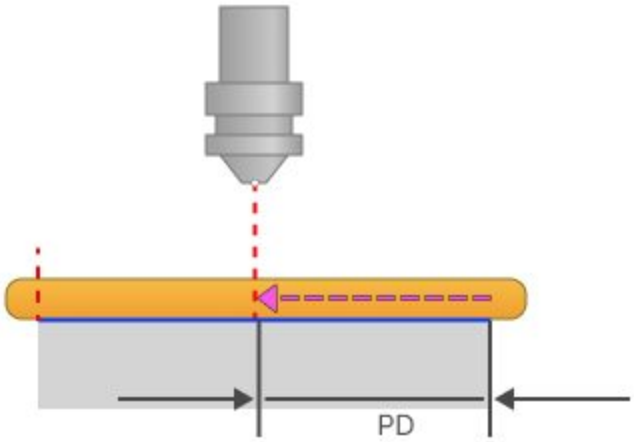
Value when On. Axis value in the on state.



Pullback. When enabled, the tool travels back along the finished weld by a configurable distance before it lifts off, so residual energy stays over already-welded material and does not mark the clean surface — useful for laser welding, where the beam may not switch off instantly.

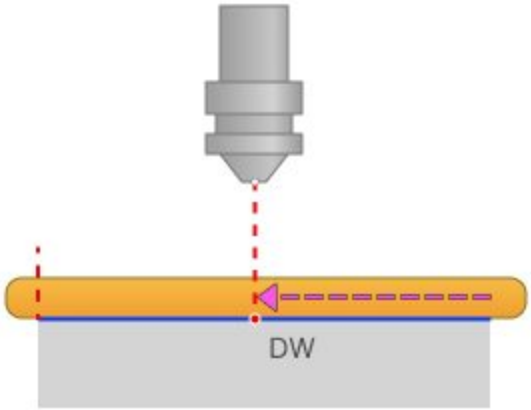


Pullback distance. The distance the tool travels back along the finished weld, measured from the weld end point in the direction opposite to the welding direction.



PD - Pullback distance

Dwell time. The dwell (hold) time at the pullback point before the tool lifts off, allowing the energy source to switch off fully while the tool is positioned over already-welded material.



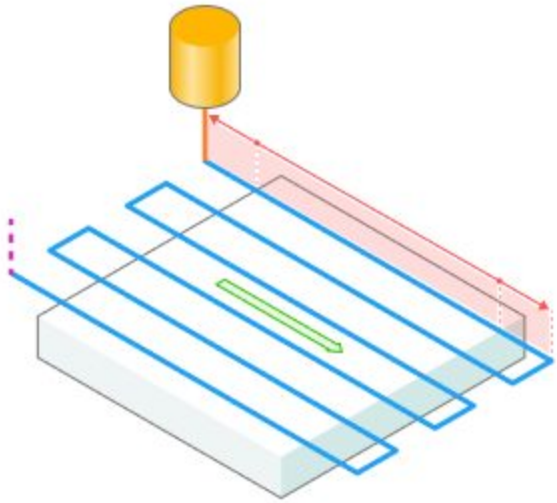
DW - Dwell time

Trimming.

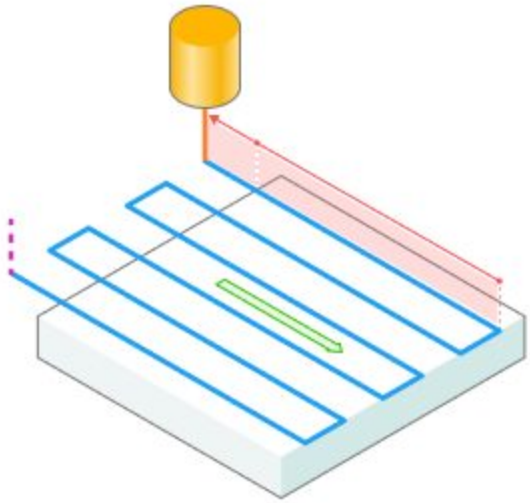
Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath.

▼ Click here to expand..

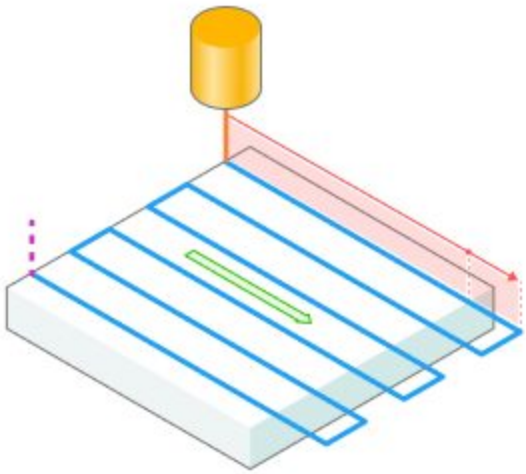
Extend (+)/Trim (-) passes. With this setting, users can extend or trim the initial and final segments of the toolpath.



Start. Entering a positive value, the system extends the start of the working passes of the toolpath by the specified amount. Entering a negative value shortens the beginning of the passes.



End. Entering a positive value, the system extends the end of the working passes of the toolpath by the specified amount. Entering a negative value shortens the end of the passes.



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Additive manufacturing

Additive manufacturing differs from the cutting process because the material is added to the workpiece instead of removing in the point of tool action. Cladding - a particular case of additive manufacturing which consist of melting the additional layer of metal or alloy on the surface of the workpiece. It allows, for example, to build on the surface of the workpiece the layer of material having specific characteristics: high hardness, strength, wear resistance, anti-friction properties, corrosion and heat resistance, etc. It allows also to restore the geometric dimensions of costly parts and tools, to repair blades, dies, molds, gears, shafts, etc.

Key features of cladding process are:

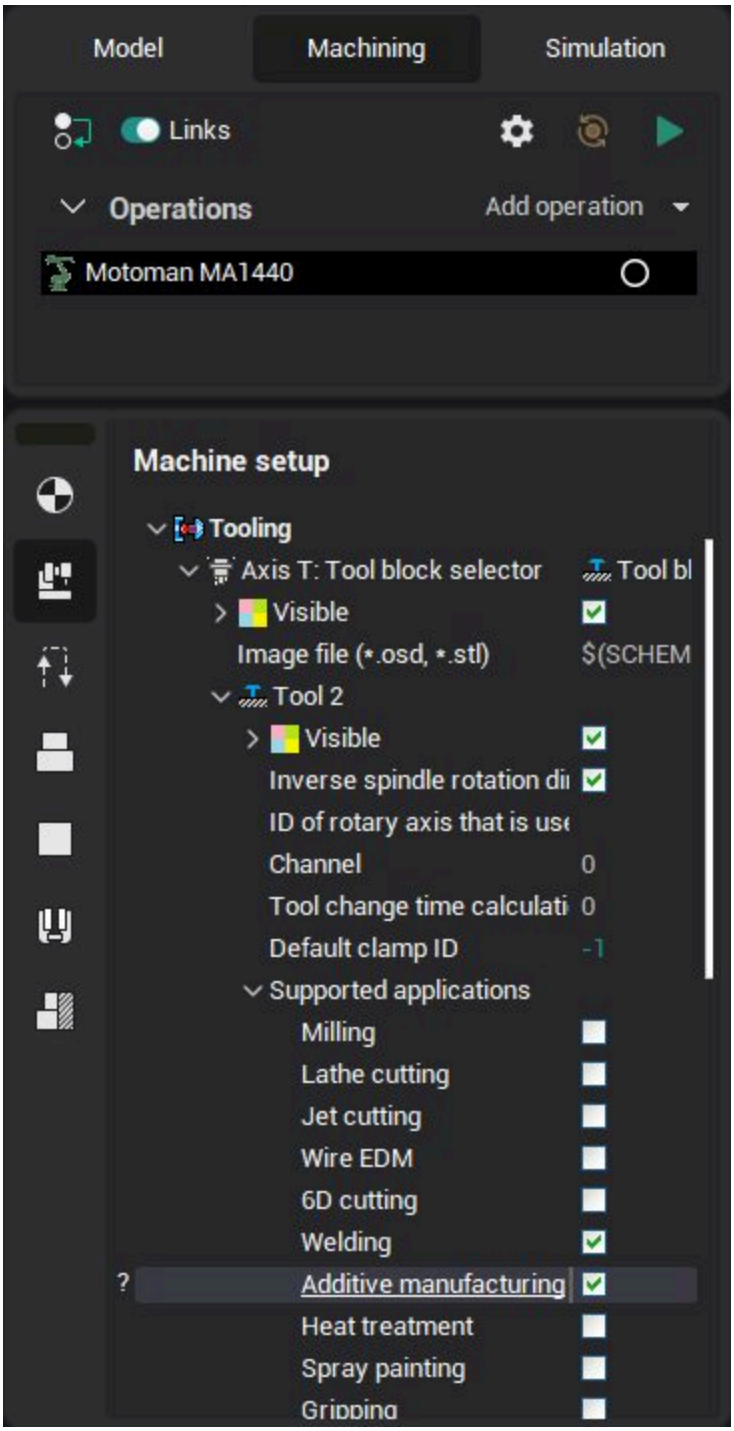
- minimal penetration into the base metal;
- minimal mixing of the added layer with the base metal;
- minimum of residual stresses and deformations in action zone;
- the size of stocks for the following machining can be reduced significantly.

There are different kinds of cladding.

- Manual arc welding with coated electrodes.
- Submerged arc welding by wires and ribbons.
- Arc welding by flux cored wire.
- Electroslag welding.
- Plasma surfacing.
- Laser cladding.
- Electron beam welding
- Induction cladding

Additive operations implemented in the CAM are universal, not tied to a particular cladding technology and type of used equipment. They only implement a geometry of the process, generate toolpath, which successively, layer by layer passes over specified surfaces, and reproduces them from the bottom upwards. Setting up of operation to work with specific type of equipment can be made by writing in the postprocessor appropriate commands to control the equipment, or, if this is not enough, by the addition of a special operation on the basis of the universal additive operations, adapted to control specific equipment.

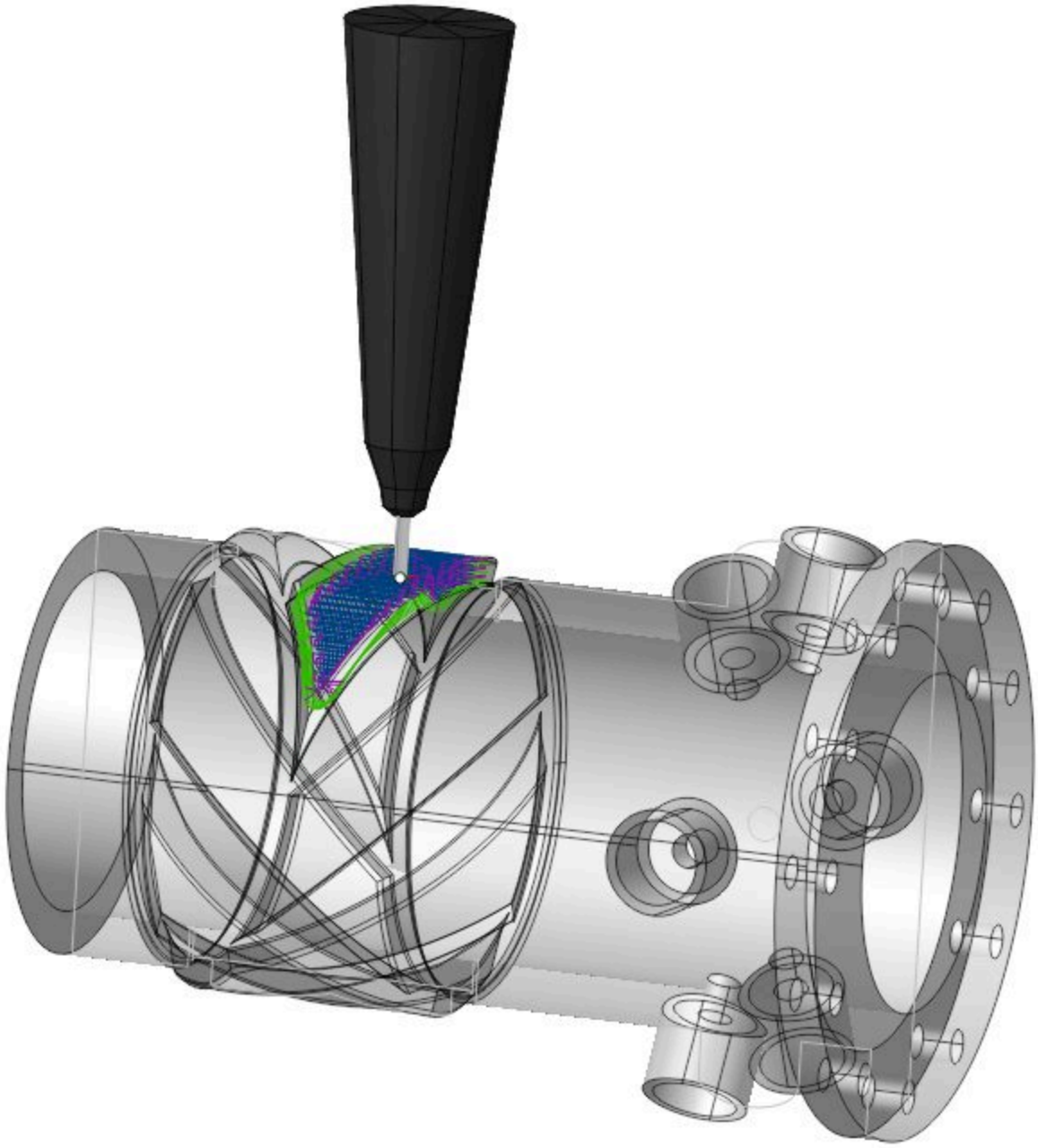
In order for the additive operations has become available for the creation should be chosen machine or a robot that supports this type of machining. To ensure support of additive operations it is need to set for the tool holder Additive manufacturing checkbox in machine settings as shown below.



Or you can write to the machine scheme for the tool holder lines similar to the following.

```
<SCType ID="AdditiveToolHolder" Caption="Additive tool
holder" Type="TToolHolderNode">
<SupportedToolTypes>
<AdditiveTool DefaultValue="true"/>
</SupportedToolTypes>
</SCType>
```

Area cladding



Application Area:

This operation deposits material onto an area enclosed by curves in the part. The user interface and job assignment of the operation is similar to the Pocketing operation.

The operation proceeds in several stages. Initially, select the base surface, on which cladding will be performed. To do this, click on "Base surface" button and then select desired surface on the screen. It can be plane, cylinder or body of revolution. The system automatically fills needed options, but if desired

you can adjust the properties of the base surface, and then click "Yes." Now, to specify the local area, select the curves on the screen or edges of 3D model along the perimeter of desired area and add them by clicking "Add boss" button. Similarly pockets (holes) may be add into the previously created bosses. To limit the upper and lower levels you should select on the screen any geometrical element, lying at the desired level, and then press the "Top level" or "Bottom level" button respectively. The upper and lower levels you can set also by numerical values in the Properties inspector window.

Job Assignment:

Base Surface. This is the surface on which the trajectory is constructed according to the curve projected onto it. Details

Add Boss. Select the closed curve inside which to deposit the material. Details

Add Pocket. Select the closed curve outside of which the material should be deposited. This task only functions if a Boss is also specified in the Job Assignment. Details

Add Inversion. Indicates a closed area, inside which the machining rules will be reversed, i.e. processed areas will become unprocessed and vice versa. Details

Top Level. Specifying the top level of material deposition by model elements. Details

Bottom Level. Specifying the bottom level of material deposition by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list. You can also modify the task type. When changing the task type to Ridge or Ditch, specify the Stock to define the element's thickness or width. Details

Delete. Removes an item from the list.

Strategy:

Machining Strategy.

This parameter allows the user to achieve a required toolpath:

▼ Click here to expand..
Type of Strategy:

Parallel. Fill the area within the Job Assignment using linear tool passes at a specified Angle to the chosen axis.

Offset. The tool paths are the lines equidistant to the curves of the Job Assignment.

Step. It is the distance between the tool passes measured along the Base Surface.

Parallel Passes Stock. The system extends the processing area outward from the selected curve (with a positive parameter value) or inward (with a negative parameter value).

Smooth Radius. The system rounds the internal corners in the processing area with the specified radius.

Gap to Prevent Overlapping. When processing a closed contour, leave a gap at the junction of the start and end of the contour to prevent layer overlap. The gap should be approximately equal to the diameter of the melting spot.

Angle. It is t he angle between the X-axis in the selected coordinate system and the tool path direction in the Parallel strategy.

Number of Fill Repetitions. The count of a dditional tool passes on each layer.

Angular Step. The angle of the additional pass relative to the preceding pass.

Turn Angle Between Layers. The angle for tool passes on the current layer in relation to the preceding layer.

Offset Pass Count. The tool passes count along the Job Assignment contour. If the count is more than one, the system generates equidistant paths with the defined Offset Pass Step. Allows to enhance the quality of the outer surface.

Offset Pass Step. The distance between equidistant passes in the toolpath along the Job Assignment contour.

Passes Order. Defines the order of passes at level from outside to inside or conversely. If Zigzag is selected then the order will change from level to level.

Parallel Passes Sorting. Controls the order of work passes during multi-area cladding.

By Regions. When filling each layer, the system forms each area separately from the others.

Sequentally. When the work pass covers multiple areas, cladding happens in several areas during a single pass.

Machining direction. Specifies the direction of tool paths.

Forward. With the Parallel strategy, tool passes follow the Angle. With the Offset strategy, the path proceeds counterclockwise.

Backward. With the Parallel strategy, tool passes oppose the Angle. With the Offset strategy, the path proceeds clockwise.

Zigzag. Tool passes alternate direction with each string.

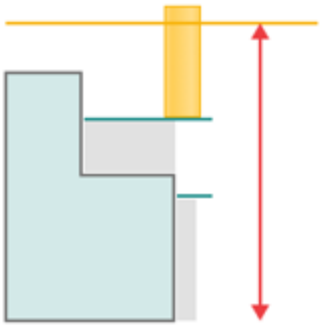
Fill Thin Zones. With the flag enabled, the system attempts to fill gaps left unfilled by the given step size.

Machining levels:

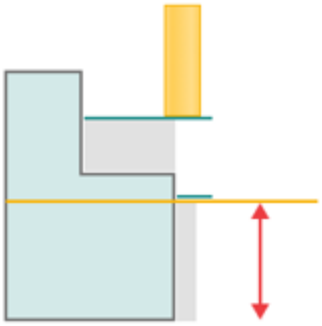
It defines the range along tool axis for the machining.

▼ Click here to expand..

Top Level. The level at which the final pass is made.



Bottom Level. Drag the plane with the mouse or drag the plane centre circle and use 'Smart snap' to align with a feature or, click on the dimension in the graph view to set the desired bottom level for machining. If the text in the field is green in colour, the level is calculated automatically as the top of the part, wokpiece and fixtures. To return to the automatic mode, simply clear the field and press 'Enter'.



Depth Step. Material cladded in one pass along the tool axis between the Top and Bottom levels.

Initial depth. The depth of first level.

Side Angle. The slope angle of the deposited surface walls.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Transformations.

A dialogue for transforming the toolpath. The parameters of Transformations are the same as in the 2D Contouring operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

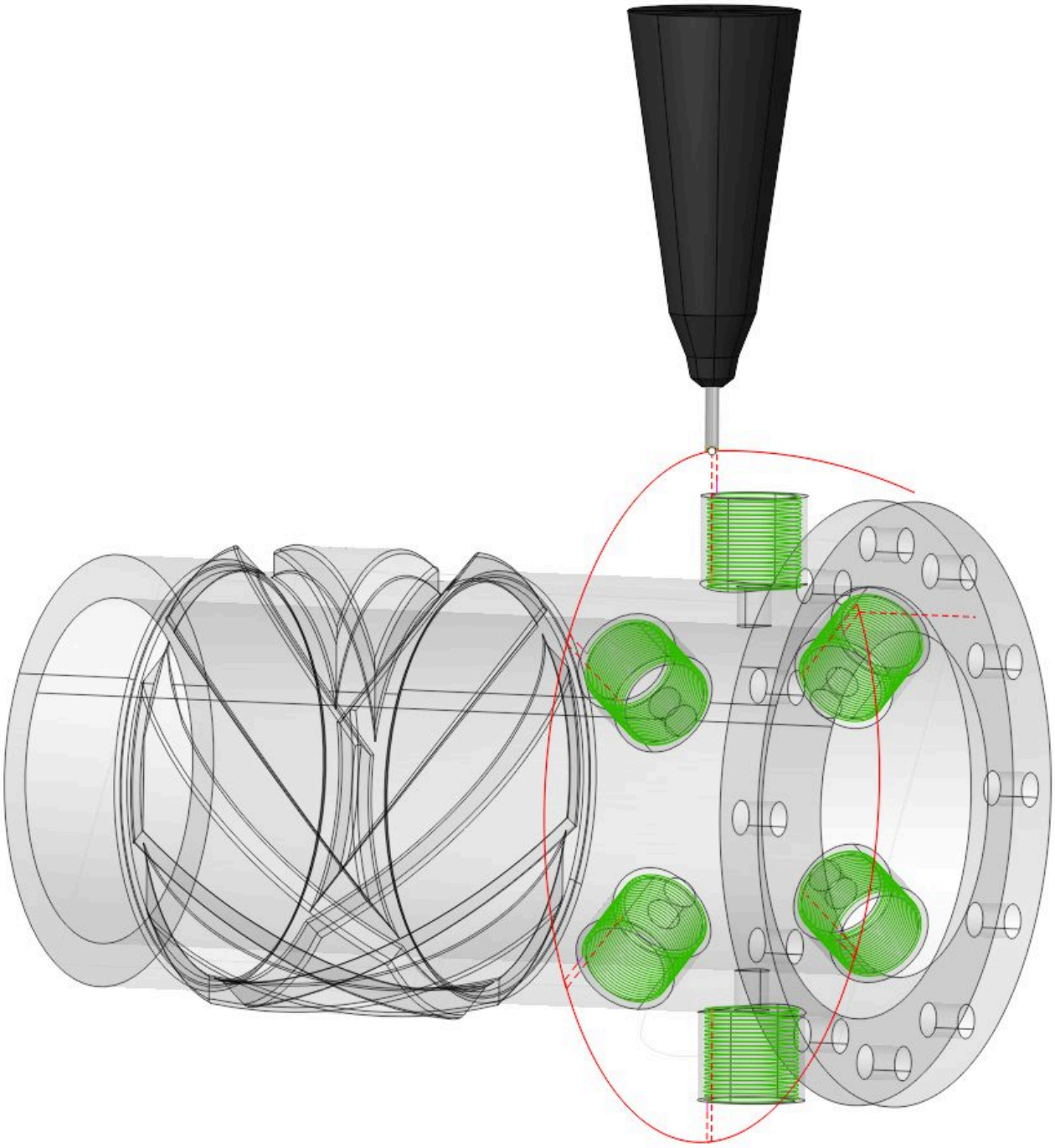
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Curve cladding



Application Area:

Additive operations that generates toolpath along curves defined inside job assignment from the bottom to top. It is useful for thin-walled models. Source curves can be placed on a plane, cylinder or body of revolution. And when the "Project toolpath onto the part" option is enabled, cladding in general can be made on the surface of an arbitrary shape. It can generate layer by layer like toolpath or helix spiral.

The operation proceeds in several stages. Initially, specify a base surface on which cladding will be performed. To do this, click on "Base surface" button and then select desired surface on the screen. It

can be plane, cylinder or body of revolution. The system automatically fills needed options, but if desired you can adjust the properties of the base surface, and then click "Yes." Now, to specify the local geometry, select the curves on the screen or edges of 3D model along the perimeter of desired area and add them by clicking "Curve" button. To limit the upper and lower levels you should select on the screen any geometrical element, lying at the desired level, and then press the "Top level" or "Bottom level" button respectively. The upper and lower levels you can set also by numerical values in the Properties inspector window.

Job Assignment:

Base Surface. This is the surface on which the trajectory is constructed according to the curve projected onto it. Details.

Curve. Set the toolpath along the selected curve . Details.

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Selecting the pocket bottom causes the system to automatically use the pocket boundary as the curve and determine the Top and Bottom levels according to the pocket levels. Details.

Top Level. Specifying the top level of material deposition by model elements. Details.

Bottom Level. Specifying the bottom level of material deposition by model elements. Details.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Strategy:

Machining Levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Area Cladding operation. Details.

Helical Machining.

Machining of a contour using a helical motion.

▼ Click here to expand..
Initial Level. With the flag set, the first pass occurs along the base surface without helical ascent. Otherwise, the passes proceed along the helical path starting from the initial point.

Smooth by radius. When choosing an Inintial Level, a smooth transition is created in the area where the slope of the spiral intersects with a flat area.

Roughing Passes.

Toolpath is offset in plane by the 'Roughing passes' thickness amount. Additional passes are added to the contour by the value specified in the parameters with the specified number of steps. This parameter group works similarly to the 2D Contouring operation. Details.

Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. This parameter group works similarly to the 2D Contouring operation. Details.

Sorting.

Controls the sequence of toolpath passes during the surface machining. This parameter group works similarly to the 2D Contouring operation. Details.

Transformations.

A dialogue for transforming the toolpath. This parameter group works similarly to the 2D Contouring operation. Details. When the Project toolpaht onto the part option is active, the initial toolpath formed on the base surface is projected onto the 3D model that is specified as the part of the operation.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

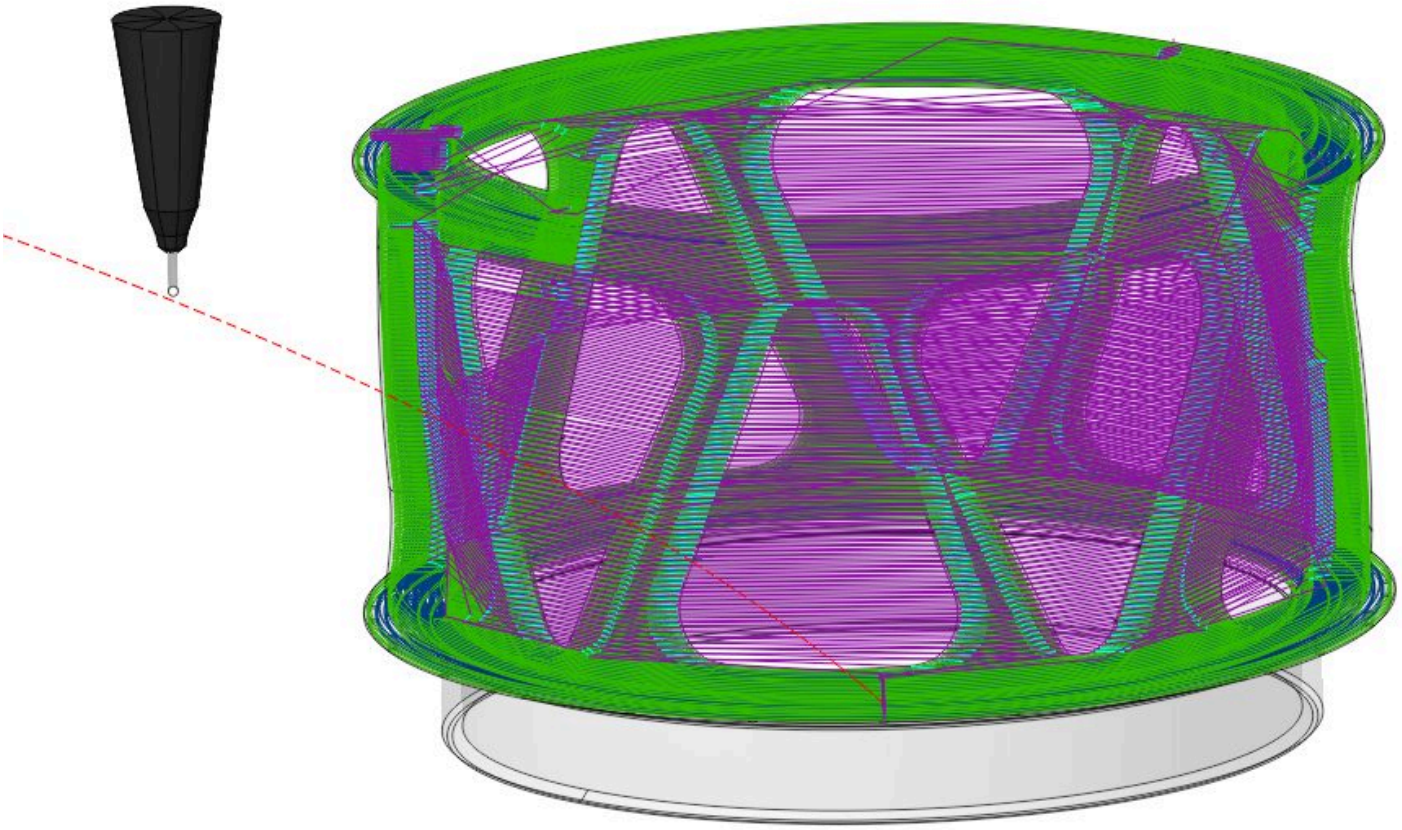
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Cladding 3D



Application Area:

This is an additive manufacturing operation that uses a 3D model as input. It is similar to Roughing Waterline operation except that it works from the bottom to top. It intersects source model layer by layer and generates toolpath to fill calculated intersection area for each level. Operation has Parallel and Offset strategies to fill the area. By default the whole part is the model to machine. If you want to grow local geometry only then you need to add desired faces to the job assignment. System automatically fills top and bottom levels by defined geometry, but if you want to limit the upper and lower levels manually then you should select on the screen any geometrical element, lying at the desired level, and then press the "Top level" or "Bottom level" button respectively. The upper and lower levels you can set also by numerical values in the Properties inspector window.

Job Assignment:

Faces. Select various surfaces of the part that constrain the material fill area. The system will calculate the trajectory based on the chosen surfaces.

Pocket. Select the pocket bottom. The system will create the adjacent pocket walls considering the stock specified in the item Properties. Details

Job Zone. Select the curves or edges of the part that define the material fill area. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level of material deposition by model elements. Details

Bottom Level. Specifying the bottom level of material deposition by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Strategy:

Machining Strategy.

This parameter group allows the user to achieve a required toolpath. It works similarly to the Area Cladding operation. Details.

Machining levels:

It defines the range along tool axis for the machining. This parameter group works similarly to the Area Cladding operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. The parameters of Trimming are the same as in the Waterline Roughing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

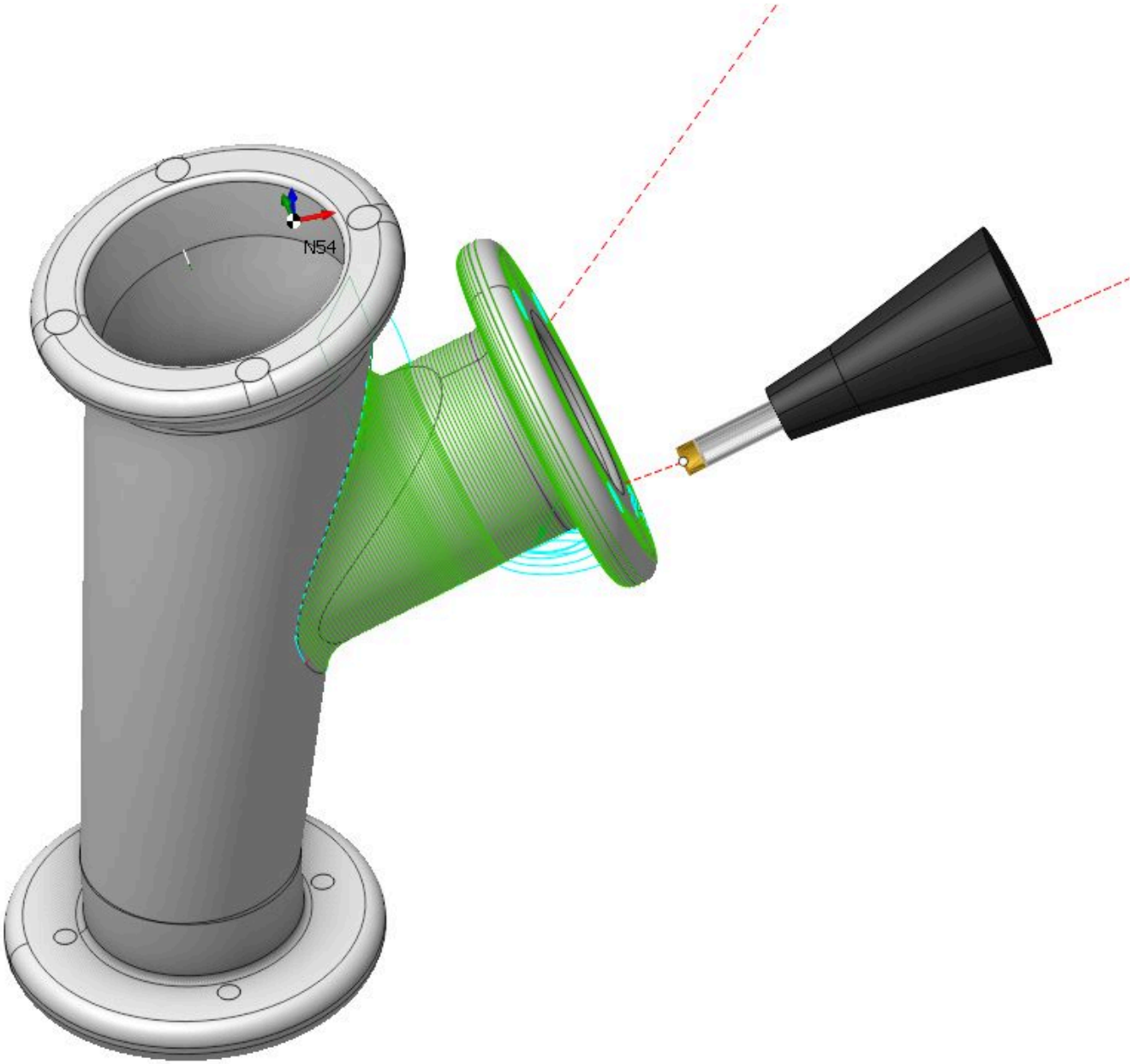
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Cladding 5D operation



Application Area:

Operation "Cladding 5D" allows to deposit material layer on the surface of a part using strategies from 5D Surfacing operation operation, because this operation is based on it. Generally, this operation uses it as a foundation but employs some new strategies for toolpath constructing and tool orientation planning. It generates intricate spatial forms using 3D models as a basis.

Job Assignment:

Machining Surfaces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

First Curve. For the Parallel to Curve strategy, this feature specifies curves for parallel passes computations. Additionally, in the Morphing Between Two Curves and Spiral Between Two curves strategies, it specifies the First Curve . You can select one or multiple, not necessarily connected, curves or edges.

Second Curve. The system uses it in the Morphing Between Two Curves and Spiral Between Two curves strategies. It specifies the Second Curve. You can select one or multiple, not necessarily connected, curves or edges.

Tilt Curve. This is the curve that the tool axis vector aims at when setting Tool Orientation specification Through Curve.

First Surface. This feature serves as the First Surface in a Morph Between Two Surfaces and Spiral Between Two Surfaces strategies, facilitating the morphing process to create toolpath lines.

Second Surface. This feature serves as the Second Surface in a Morph Between Two Surfaces and Spiral Between Two Surfaces strategies, facilitating the morphing process to create toolpath lines.

Surfaces for projection. These are the surfaces intended for machining onto which the toolpath transfers when enabling the Project Toolpath onto the Part option in the Strategy tab.

Job Zone. Use the job zone to trim the passes outside the specified 2d containment areas. The plane of the containment area is defined by the initial tool orientation. Details.

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones geometry from curves and edges to specify the workpiece areas that s hould avoid machining in the current operation. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

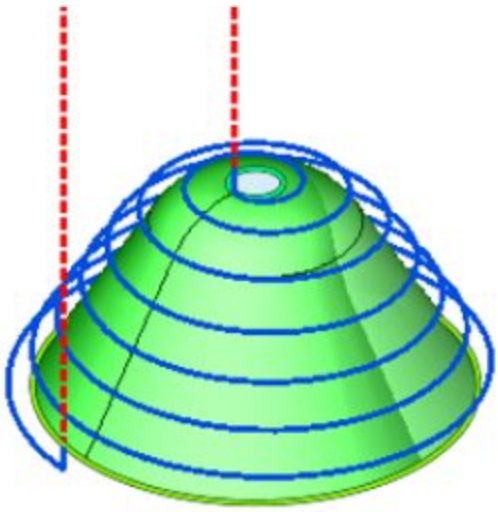
Strategy:

Strategy:

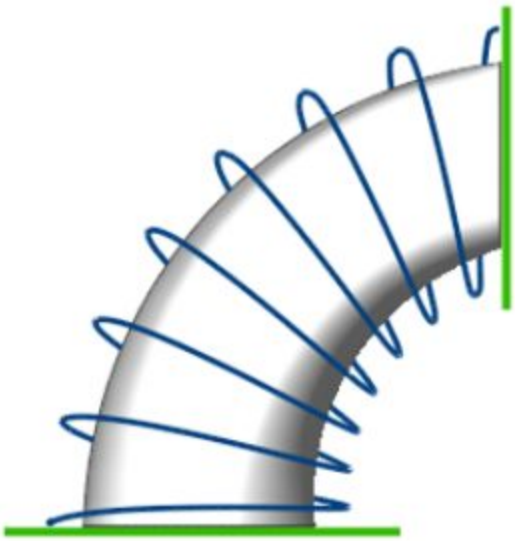
This parameter allows the user to achieve a required toolpath:

▼ Click here to expand..
Type of Strategy:

▼ Click here to expand..
Spiral Between Two curves. First, preliminary passages are generated like the Morph strategy in the 5D Surfacing operation. Details. Then each pass is converted into a spiral by gradual ascent to the next pass as it approaches the end.



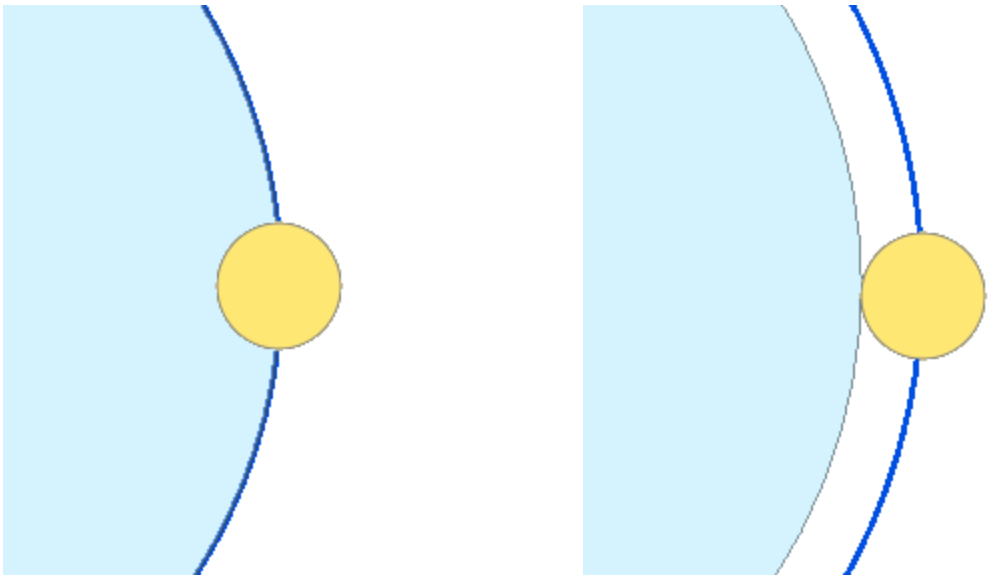
Spiral Between Two Surfaces. The tool paths take the form of a spiral, starting on the First Surface, ending on the Second Surface, and wrapping around the contour of the Machining Surfaces.



Other strategies are the same as in the 5D Surfacing operation. Details.
Step. The thickness of material cladded in one pass.

Adaptive Step. This parameter adjusts the distances between tool passes according to the curvature radius of the surface to maintain uniform scallop height after surface machining. Details.

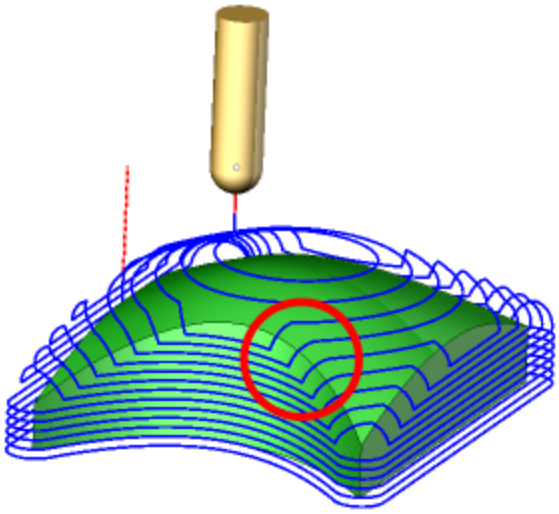
Offset Job Surfaces. When the flag is active, the system generates the toolpath with considering the tool radius.



Calculate Based on Tool Center. Flagging switches the mode that involves first offsetting the surfaces for machining by the tool profile for 3-axis machining or along the surface normal by the tool radius for 5-axis machining, and then calculating toolpath curves on these offset surfaces. In another mode, the system initially generates tool path as curves on the machined surfaces. Then, it positions and orient the tool relative to the calculated curves to keep the tool's contact point with the machined surface constant.

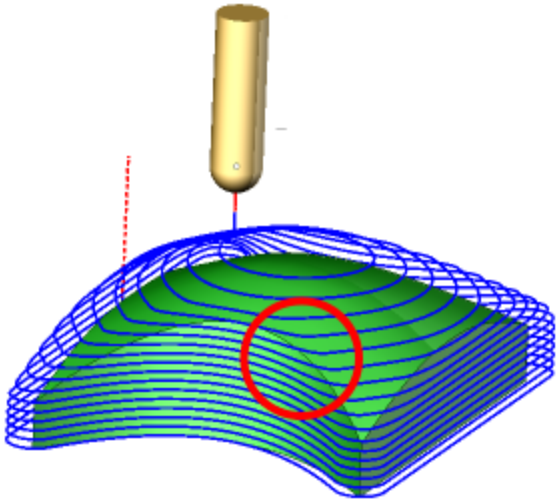
Comparison of the toolpath based on the center and the tool contact point:

Calculation based on tool contact point:



In this mode the work passes are generated by calculating curves on the machining surfaces as the first step and then positioning and orienting the tool relative to the calculated curves in such a way that the point of contact of the tool with a machining surface stays the same. It is desirable, for example, when smooth surfaces are machining, or doing flank milling.

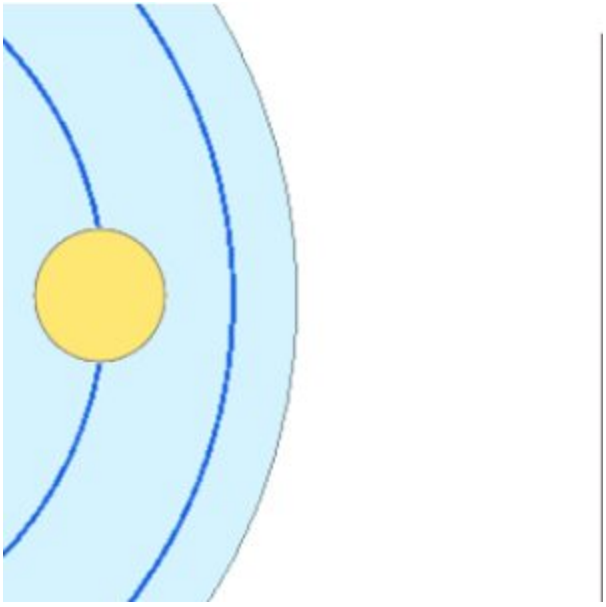
Calculation based on tool center:



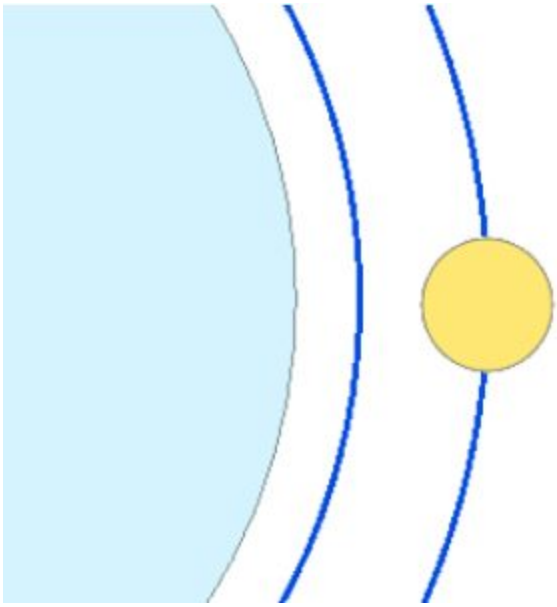
In this mode the work passes are generated in such a way that at first the machining surfaces are being offset either by the tool shape itself for 3 axis machining or by the tool radius along the surface normal for 5 axis machining and only then the sections of the offset surfaces are calculated. For example, for the Parallel to plane strategies it means that the generated work passes all lie on parallel planes. Another advantage of this mode is that not only the surfaces themselves but the edges between machining surfaces are taken into account.

Cladding Side. This parameter controls the positioning of toolpath lines relative to the geometry of the Job Assignment. It operates when the Offset job surfaces parameter is active.

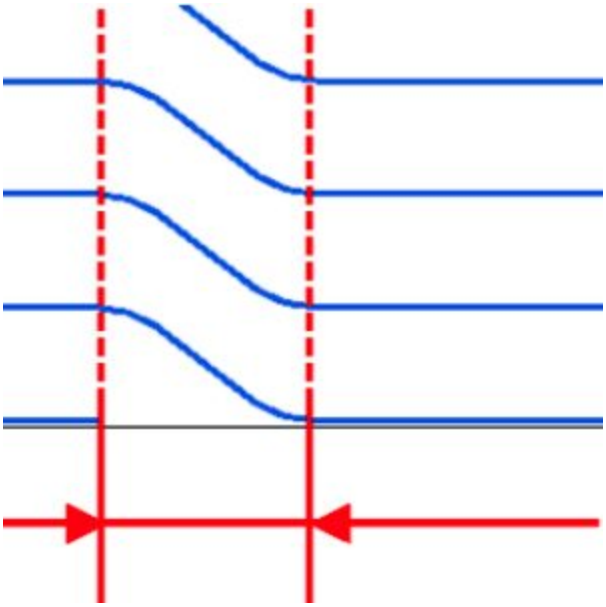
Inside. The toolpath is built inside the Job Assignment.



Outside. The toolpath is built outside of the Job Assignment.



Gap to Prevent Overlapping. When processing a closed contour, leave a gap at the junction of the start and end of the contour to prevent layer overlap. The gap should be approximately equal to the diameter of the melting spot. It allows to avoid the passage of the tool in the same place several times, as a result of which the height of layer can exceed the specified size. For this, the toolpath is shortened at the end of each curve by the specified value.



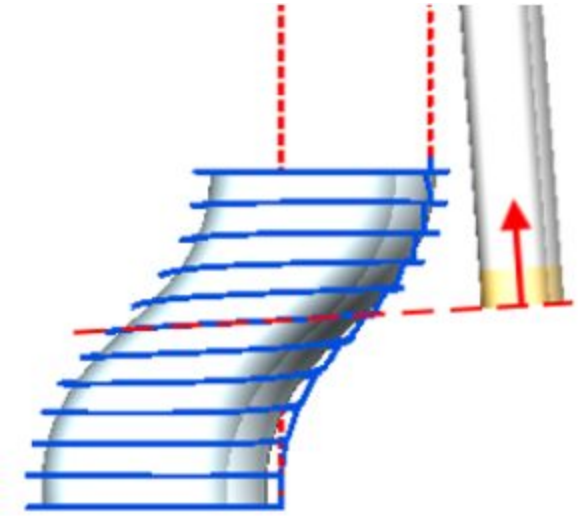
Other properties of the Strategy are the same. as the Strategy parameters in the 5D Surfacing operation. Details.

Tool Orientation:

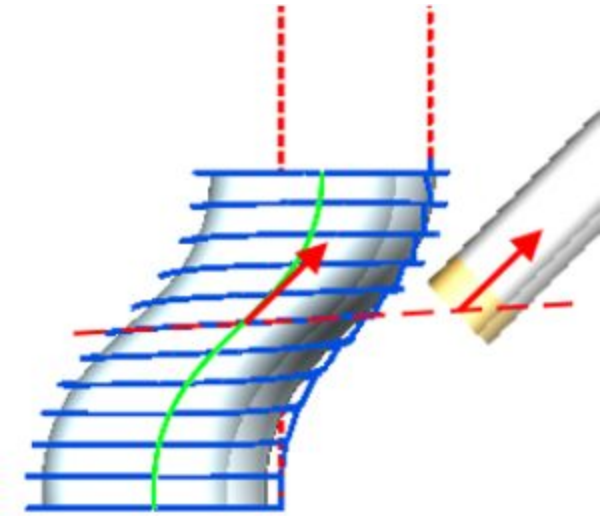
Controls tool axis orientation.

▼ Click here to expand..

Perpendicular to Pass Plane. The tool axis is perpendicular to the middle plane of the pass.



Along curve. The tool axis is set along the tangent to the Tilt Curve in point where it intersects the middle plane of the pass.



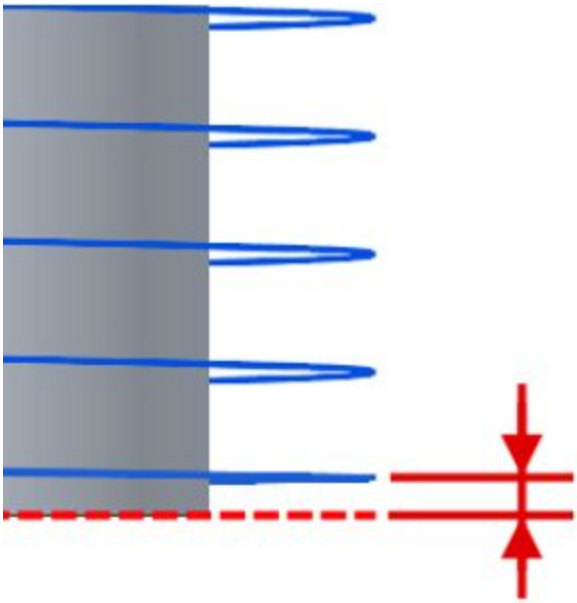
Other Tool Orientation strategies and their parameters are the same as the Tool Orientation strategies in the 5D Surfacing operation. Details.

Additional passes.

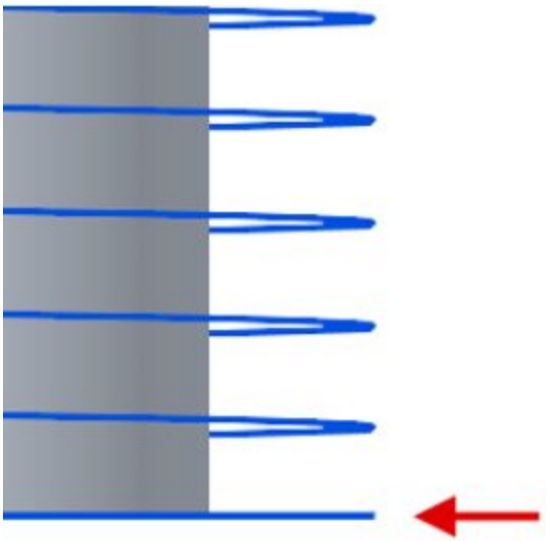
Allows to set specific machining conditions for initial and last layers: the height of cladding and the feed of the layer (on the Speeds page)

▼ Click here to expand..

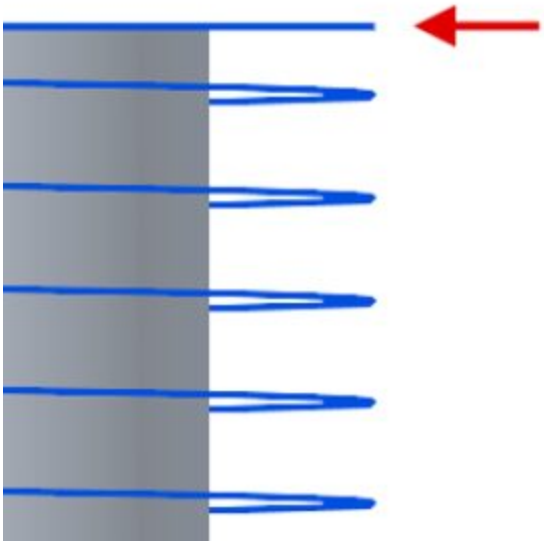
First Step. The first layer will be executed with the specified step. It may be necessary, for example, to provide better adhesion to the substrate.



Start planar pass. Before the ascending spiral, a complete pass will be made along the curve in the initial plane.



Final planar pass. The final segment of the spiral trajectory executes as a planar curve on the final plane.



Edge Side

Enable the Calculate Based on Tool Center parameter to activate the option. This option selects one of the two sides adjacent to the selected curve in the Job Assignment. The system offsets the machined surfaces along the normal to the chosen side during the tool center path calculation.

▼ Click here to expand..

Alternative Side for First Curve. Use the tool center calculation by offsetting the machined surfaces normal to the alternate side of the First Curve.

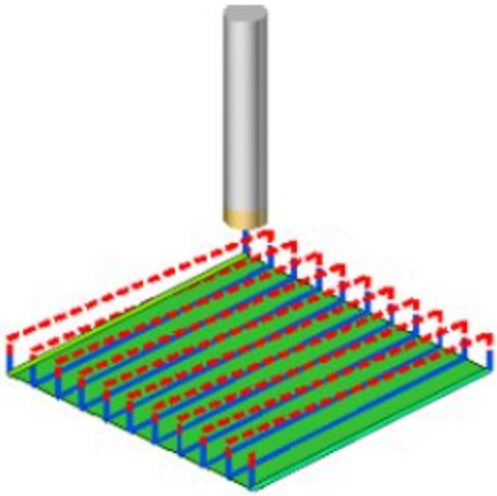
Alternative Side for Second Curve. Use the tool center calculation by offsetting the machined surfaces normal to the alternate side of the First Curve.

Direction Along Curve.

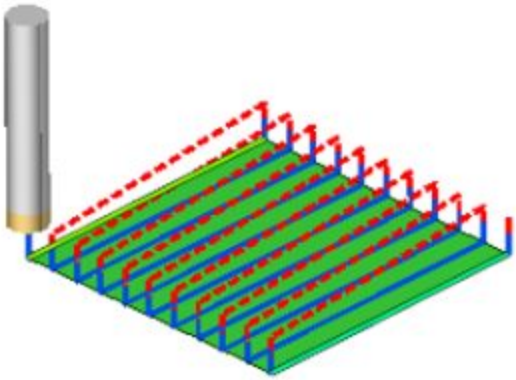
Specifies the tool movement direction along work passes.

▼ Click here to expand..

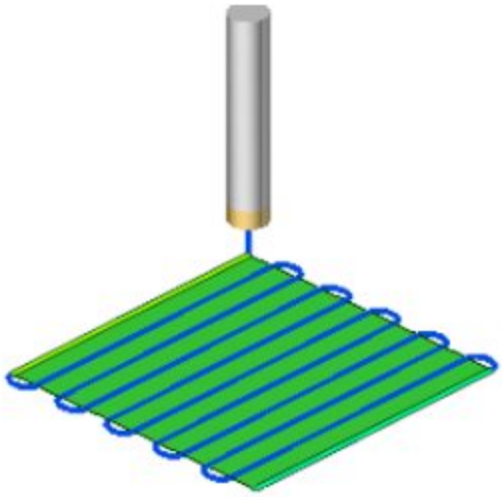
Forward. Cladding only in forward direction is enabled.



Backward. Cladding only in backward direction is enabled.



Both. Cladding in both directions is allowed.



Sorting:

Controls the sequence of toolpath passes during surface machining. This parameter group works similarly to the 5D Surfacing operation. Details.

Limit Rotation Angles:

When the flag is set, it specifies the limit values of the tool's angular position relative to the specified axis. This parameter group works similarly to the 5D Surfacing operation. Details.

Job zone.

Enables additional modification of the working area initially selected on the Job assignment tab. This parameter group works similarly to the 5D Surfacing operation. Details.

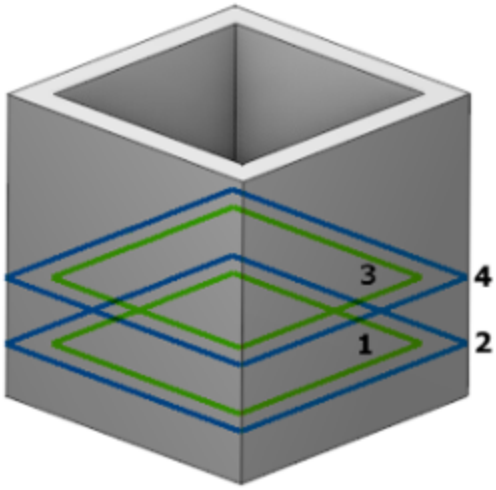
Corners smoothing.

When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model. This parameter group works similarly to the 5D Surfacing operation. Details.

Solid Model.

This option automatically sorts the model with internal and external walls. The model must be solid!

▼ Click here to expand..
Set the flag to execute the passes for the inner and outer wall formation of the model in a strict order.



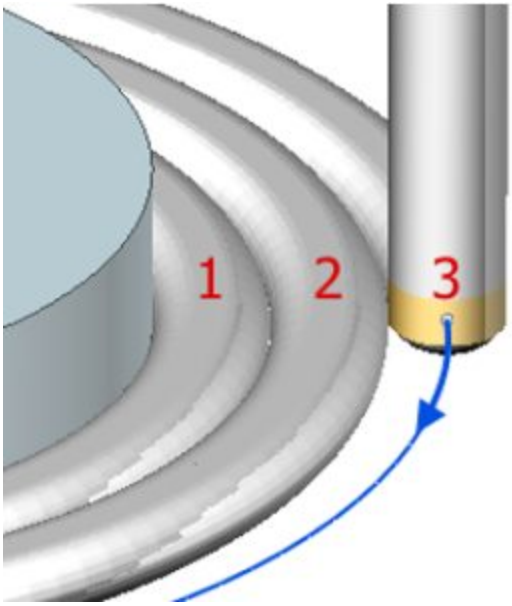
Roughing Passes.

Toolpath is offset in plane by the Roughing passes thickness amount. Additional passes are added to the contour by the value specified in the parameters with the specified number of steps.

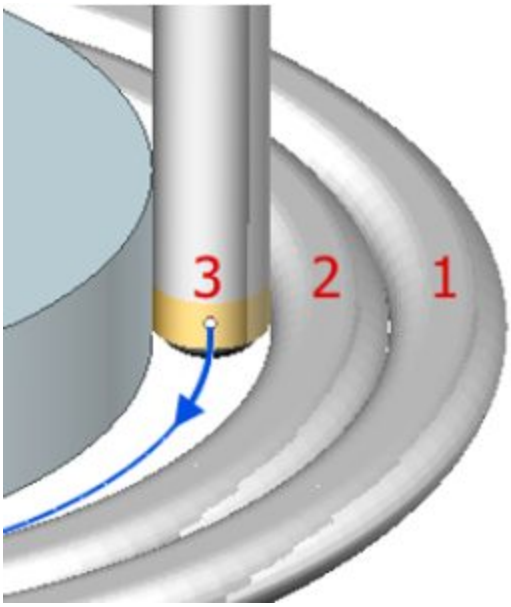
▼ Click here to expand..
Constant Step in XY Plane. Shape the trajectory to maintain equal step projections on the XY plane for both the X and Y axes. Use this option when forming tool passes angled to the XY plane.

Passes order. The choice of the order of cladding - from the outside in or inside out.

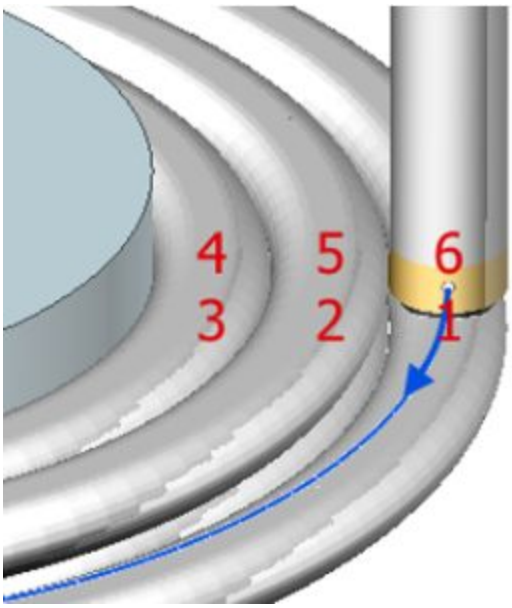
Inside to Outside. Conduct cladding passes starting from the inside and moving outward.



Outside to Inside. Conduct cladding passes starting from the outside and moving inwards.



Zigzag. When this option is active, each subsequent layer reverses the direction of the roughing passes relative to the previous layer. It can help to get the toolpath without breaks, i.e. without having to switch off the feed of the material and transition on the rapid feed.



Other properties of the Roughing Passes are the same. as the Roughing Passes parameters in the 5D Surfacing operation. Details.

Project Toolpath onto the Part:

By setting the flag, the system generates tool paths for processing the surfaces identified as Surfaces for Projection in the Job Assignment by mapping the path derived from the Machining Surface onto the Surfaces for Projection. This parameter group works similarly to the 5D Surfacing operation. Details.

Trimming:

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. This parameter group works similarly to the 5D Surfacing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details.

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

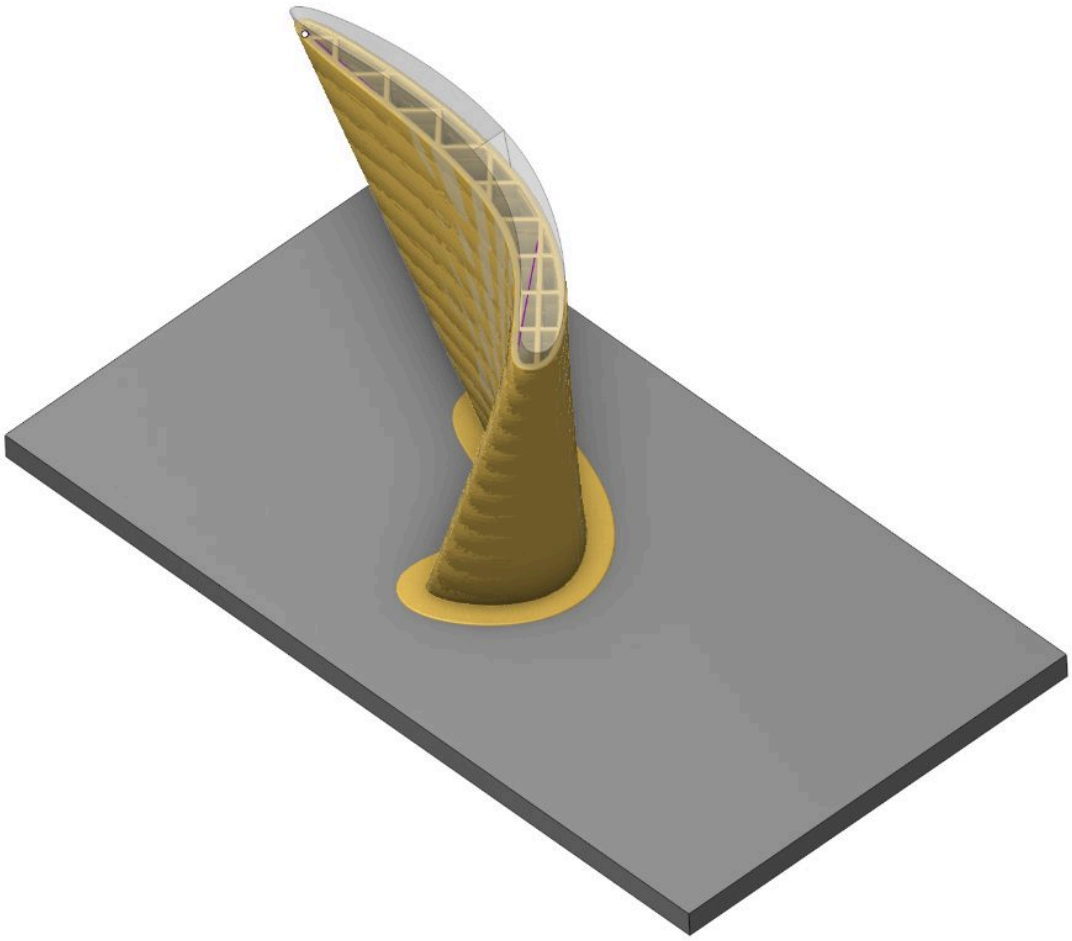
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Planar slicing



Application area:

The new operation allows for 3D printing. The trajectory is built based on Ultimaker Cura's calculations.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list

Strategy:

Auto tool parameterization.

Automatic change of tool parameters (Diameter, Working Length) based on Cura parameters

General.

List of parameters for presetting

▼ Click here to expand..
Manufacturer. Brands of printers

Printers. List of printers of the selected brand

Extruders. List of extruders of the selected printer

Material brand. Brands of materials

Materials. List of materials of the material brand

Variants. The variant, in case of the extruder, hold settings that have to do with nozzle size or specific settings that relate to this

Profiles. Intent profiles contain settings that modify the quality by giving them an intent. These profiles depend on the material, variant and quality/resolution.

Resolutions. Quality profiles hold the resolution of the print by setting the layer height.

Show custom parameters. Customizing print settings. Custom mode offers more settings for experienced users. Can be basic. advanced, expert, all.

Other parameters required to use Cura functionality are described in the help. See more

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

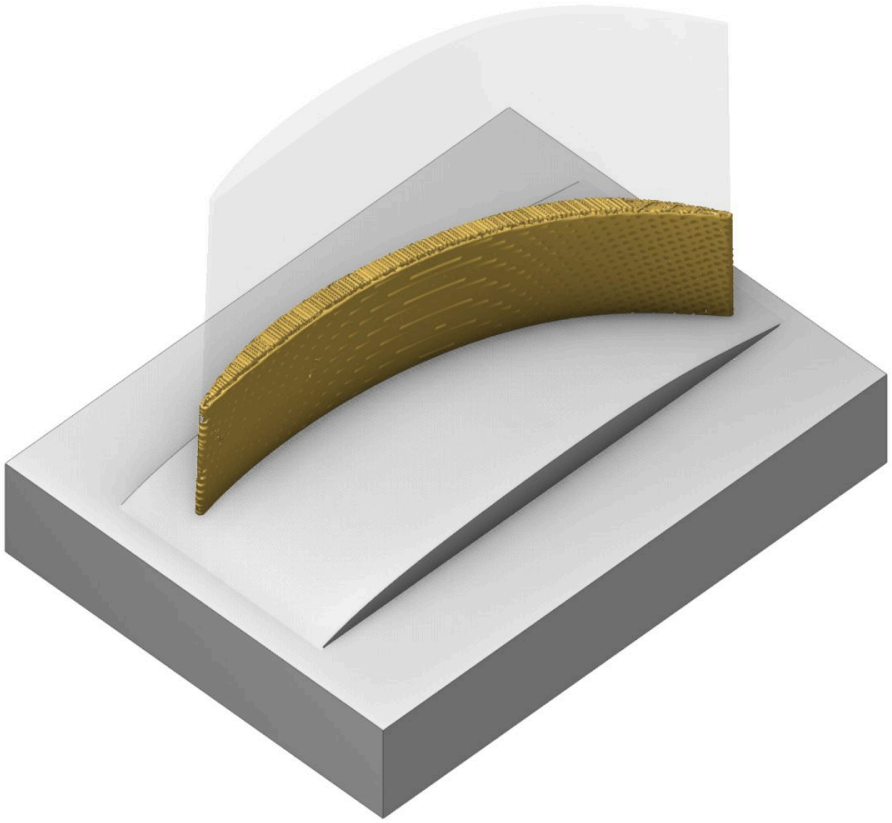
Installation:

The operation is supplied as the Extension for Planar slicing operation based on Cura extension. To install it:

1. On the Home page, open the Extension Store tab and go to the Store page.
2. Find Extension for Planar slicing operation based on Cura and click Install.
3. The operation appears in the Additive group of the new operation window. If it does not, restart the CAM system.

You can also install the extension from the web storefront apps.encycam.com. Details

Non-planar slicing



Application area:

This operation allows you to print models on arbitrary substrates. It uses a non-planar slicer engine and generates a non-planar additive printing path. But also it's possible to generate a classic planar printing path with some extra parameters.

Setup:

The Setup tab is used to configure the primary parameters of theconfigure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Non-planar substrate. Substrate for printing (for "Offset" strategy only)

Feature. Desired model, that should be printed

Properties. Disabled

Delete. Removes an item from the list

Strategy:

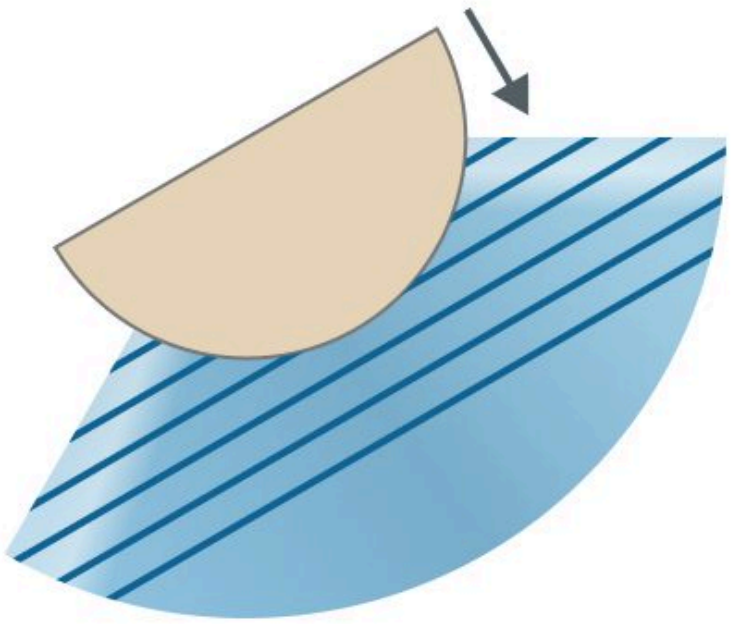
Slicing strategy.

Defines the slicing strategy.

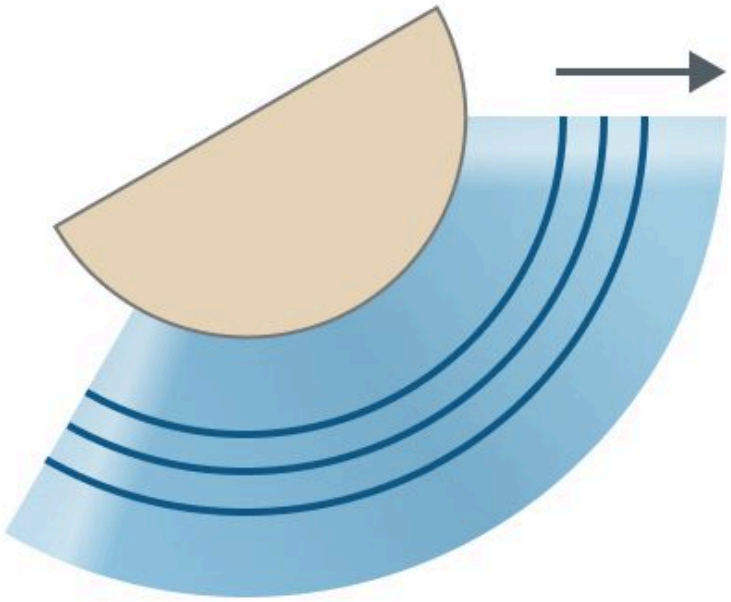
▼ Click here to expand..

Strategy types

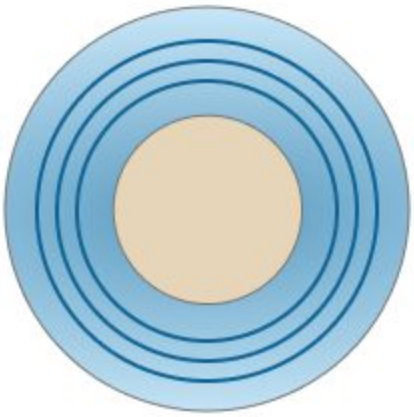
Planar. This strategy performs classic planar slicing, but supports different slicing directions and allows changing the toolpath normals.



Offset. This strategy performs slicing by offset method from the substrate, which allows to maintain a constant layer height between each layer.

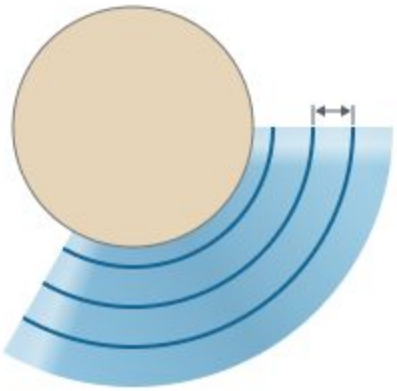


Radial. This strategy performs slicing by cylindrical surfaces around a specified rotary axis, which allows printing around the rotary axis with a constant radial step between layers.

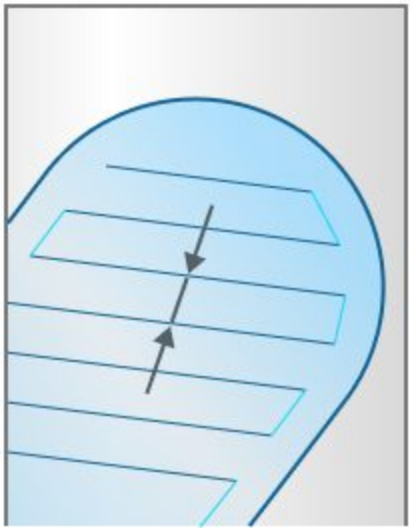


Common parameters

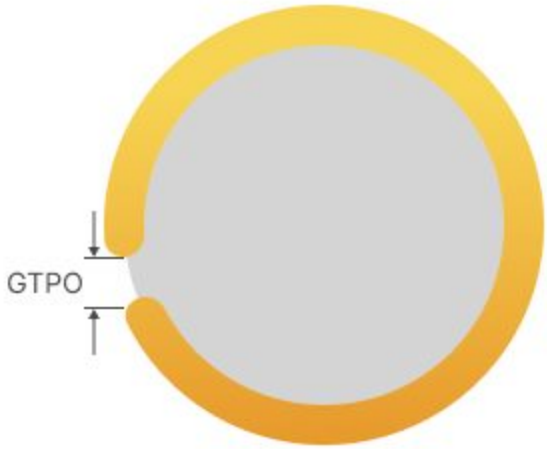
Layer height. Height of each layer in mm. By default corresponds to the working length (WL) of the tool.



Line width. Width of a single line. By default corresponds to the diameter of the tool



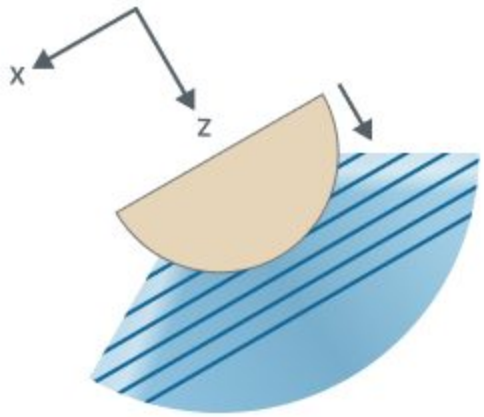
Gap to prevent overlapping. Leaves a gap at the start/end junction of closed contours to prevent material overlap.



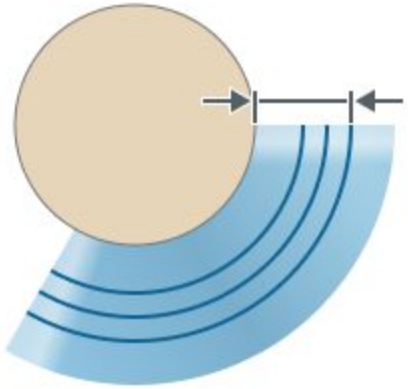
GTPO - Gap to prevent overlapping.

Planar slicer parameters

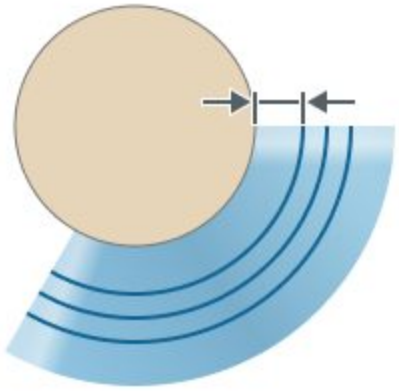
Slicing direction. Specifies the slicing direction for the Planar slicing strategy — a global coordinate axis (X / Y / Z) or a custom direction vector.



Top level. Upper bound of the planar slicing range, measured along the slicing direction — the topmost layer is placed at this level. In Auto mode it is taken from the upper extent of the feature.

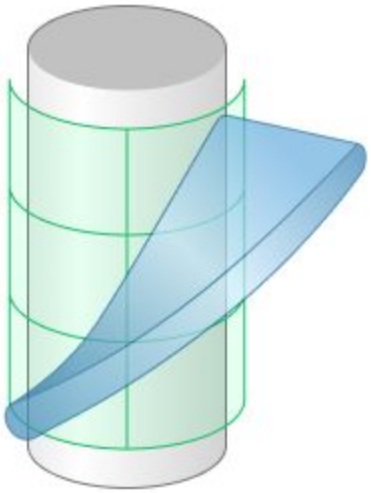


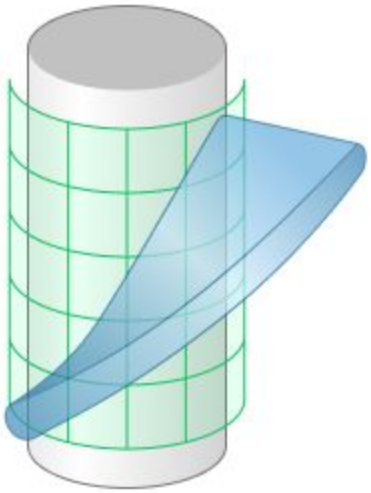
Bottom level. Lower bound of the planar slicing range, measured along the slicing direction — the first layer is placed at this level. In Auto mode it is taken from the lower extent of the feature.

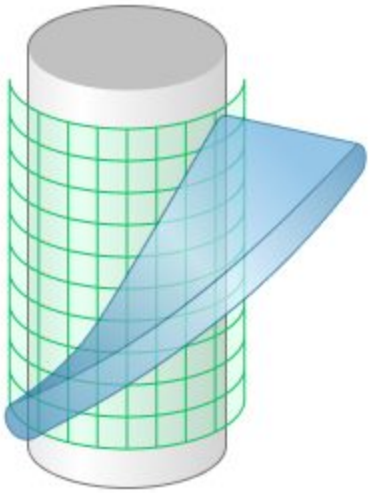


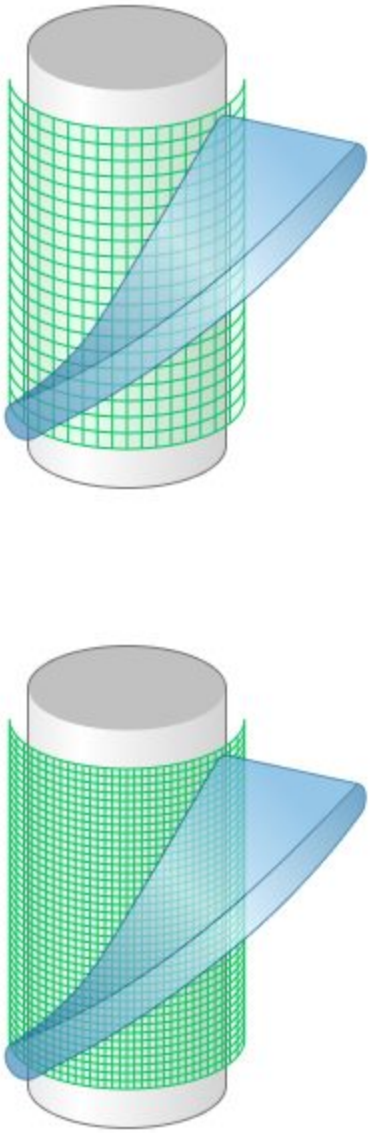
Offset slicer parameters

Surface resolution level. Represents the resolution level of the offset surfaces that will be constructed to split the model. Lower values produce faster result, but in lower resolution. Higher values produce slower result, but in higher resolution.

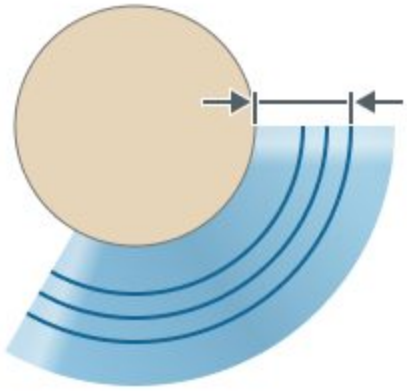




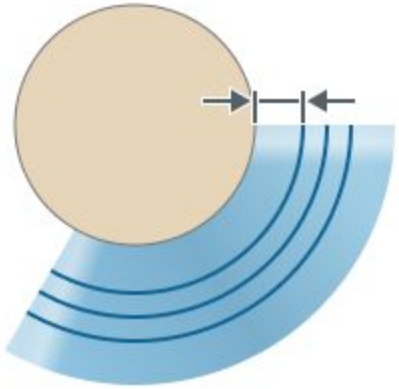




Top offset. The offset value from the substrate, which represents the finishing boundary of the printing process. However, it may stop earlier depending on the layer height parameter. In Auto mode it is taken as the maximum distance from the feature to the substrate.

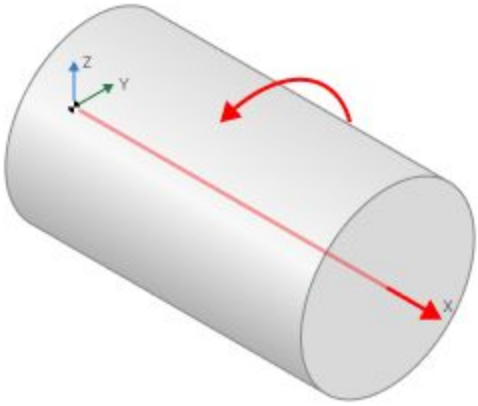


Bottom offset. The offset value from the substrate, where the printing process will start. In Auto mode it is taken as the minimum distance from the feature to the substrate.

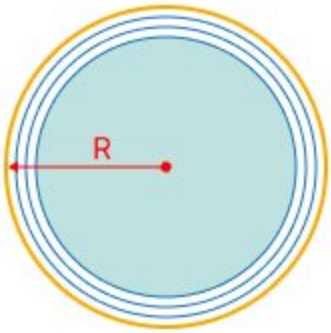


Radial slicer parameters

Rotary axis. Direction of the rotary axis around which the part is wrapped and sliced. Choose a global coordinate axis (X / Y / Z) or Custom to specify an arbitrary direction vector.

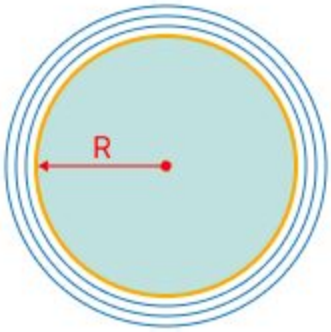


Top radius. Outer cylindrical limit of the slicing zone, measured radially from the rotary axis — layers above this radius are skipped. In Auto mode it is taken as the maximum distance from the feature to the rotary axis.



R - Top radius / level

Bottom radius. Inner cylindrical limit of the slicing zone, measured radially from the rotary axis — slicing starts at this radius. In Auto mode it is taken as the minimum distance from the feature to the rotary axis.

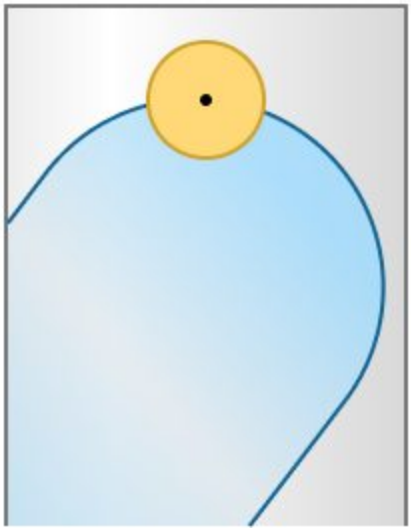


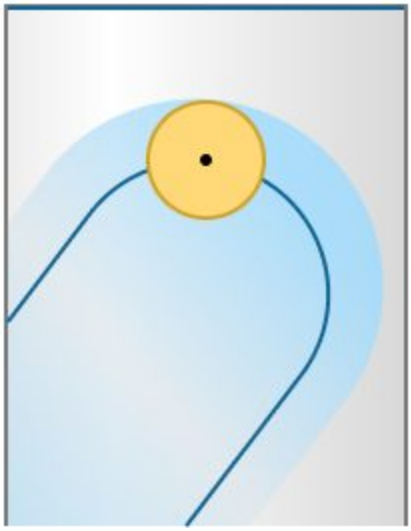
R - Bottom radius / level

Print inner volume.

Offsets the tool path into the model by half of the line width value

▼ Click here to expand..





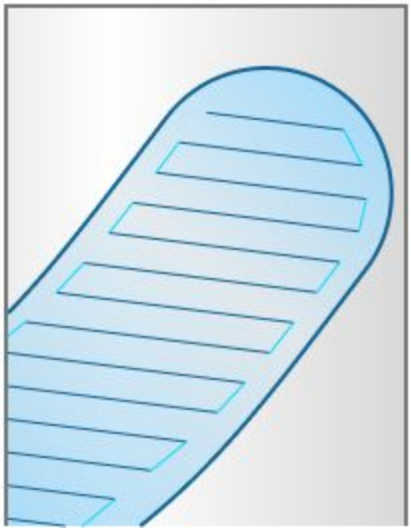
Infill.

A group of parameters to generate an infill for the model.

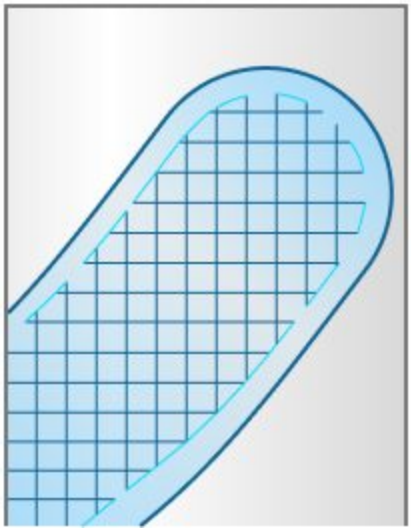
▼ Click here to expand..

Infill pattern. Defines the infill pattern.

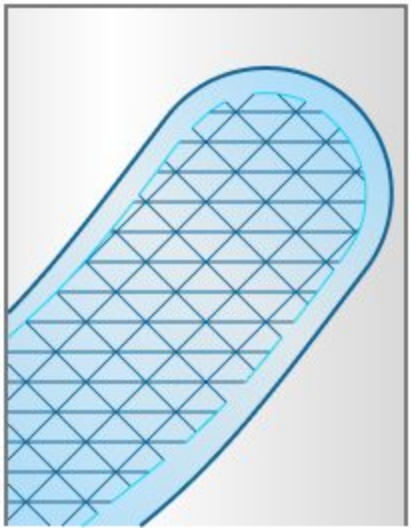
Lines. This infill pattern is created by intersecting non-planar layers with simple vertical planes



Grid. This infill pattern is created by intersecting non-planar layers with vertical planes in such a way that it forms a grid



Triangles. This infill pattern is created by intersecting non-planar layers with vertical planes in such a way that it forms equilateral triangles

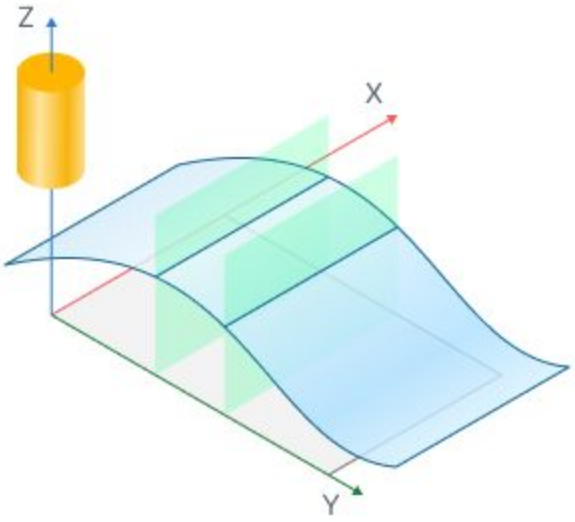


Zig Zag. This infill pattern divides each non-planar layer into regions and fills each region with a continuous zigzag toolpath. A separate zigzag toolpath is generated for each region.



Infill orientation. Determines the orientation of the infill in 3D space.

Direction. Direction along which the cutting planes are oriented. These planes intersect each layer to produce the infill lines.



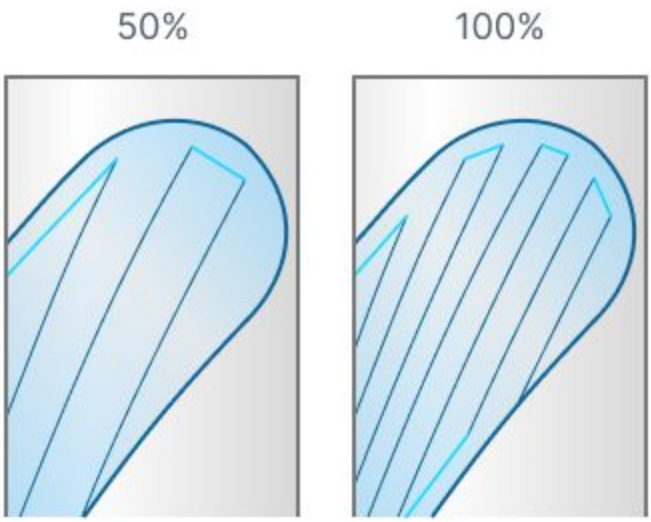
Tool orientation vector. Cutting planes are aligned along the tool orientation vector. Default for the offset strategy. Tool orientation can be set on a Setup page. Details

Slicing direction vector. Cutting planes are aligned along the slicing direction. Default for the planar strategy.

Across rotary axis. Cutting planes are aligned across the rotary axis, so infill lines wrap around the cylinder. Default for the radial strategy.

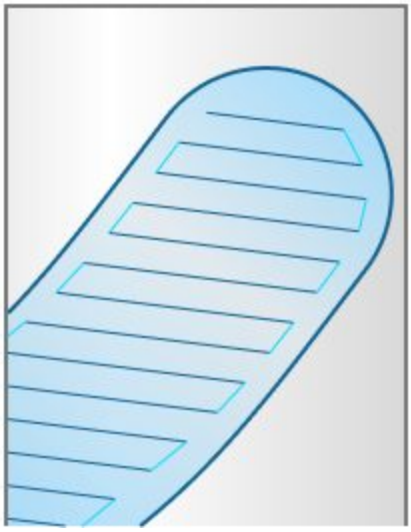
Custom. Allows you to manually set the normal vector and rotation angle for the infill.

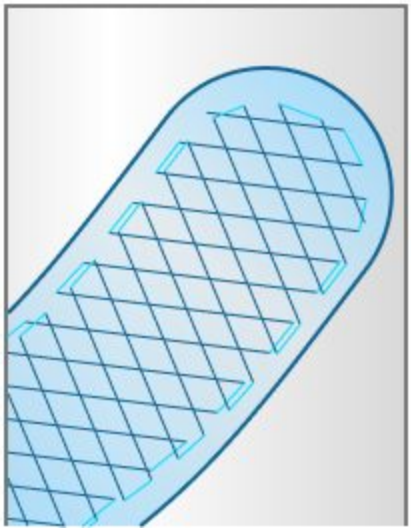
Infill density. Adjusts the density of the infill as a percentage.



Alternate lines direction. Alternates the direction of lines on each layer.

By angle. Rotation angle applied to each subsequent infill layer.





> Important note
>
> Note: In general, an angle of 0 degrees means that lines will be parallel to the X-axis. But if "Normal vector" of "Infill orientation" parameter is set to "Custom" and "Rotation angle" is not zero, in this case lines will be additionally rotated by the same rotation angle.
>
> Example: 45° means layer 1 = 0°, layer 2 = 45°, layer 3 = 90°, layer 4 = 135° and so on.

By sequence of angles. Custom angle sequence for infill direction, repeated cyclically per layer. Can be set as list of angles separated with any symbol and with parentheses around it.

> Important note

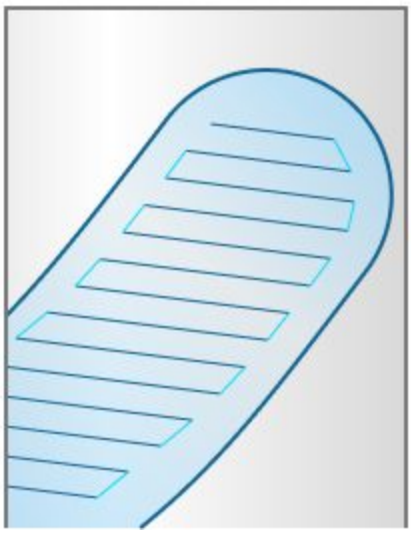
> Example: (0,90,45) means layer 1 = 0°, layer 2 = 90°, layer 3 = 45°, layer 4 = 0° and so on.

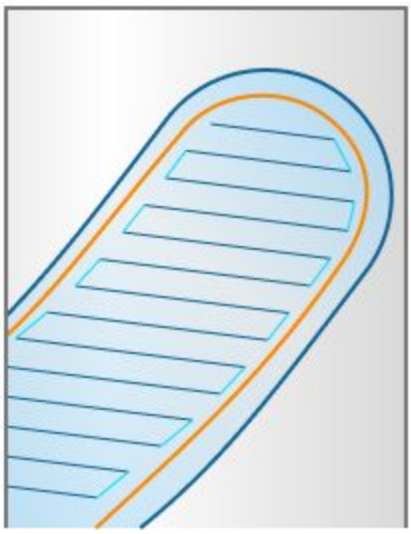
Mirror infill. *Available only for the Radial strategy.* When enabled, the infill is computed for one half-cylinder and mirrored across the rotary axis — both halves share an identical, symmetric pattern. When disabled, the infill is generated continuously around the full cylinder, allowing spiral/helical patterns with alternating layer angles.

Double walls.

Adds an additional inner wall.

▼ Click here to expand..





Sorting.

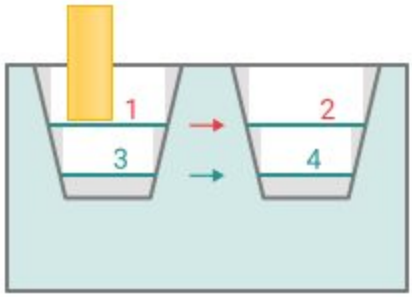
Toolpath sorting and linking parameters.

▼ Click here to expand..

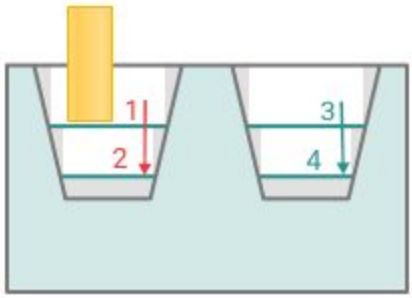
Sort by.

Allows to set printing order of layers and features.

Layers. Allows to print all features layer by layer.



Features. Allows to print all features one by one (*Note: To use this option correctly, you must configure your features as separate elements on the Job Assignment page*).



Split by layers.

This parameter allows you to split your features into a desired number of layers. It is available only when the Sort by parameter is set to Features option.

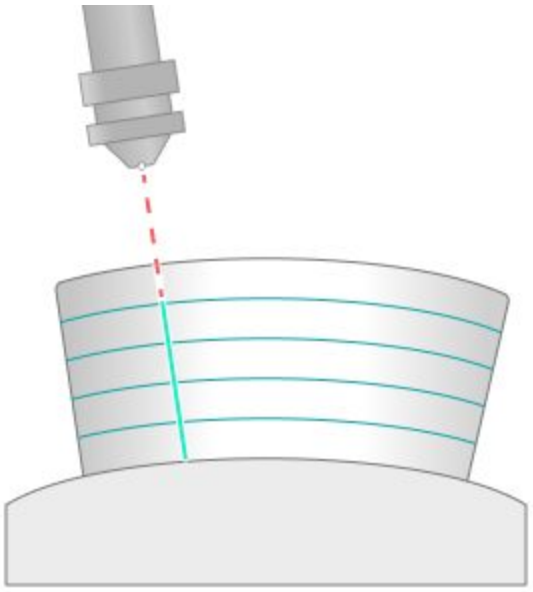
Note: This parameter is useful when printing feature by feature is not possible due to collisions with previously printed features. It also helps to achieve more even heat distribution across the printed elements and to reduce residual stress.

Start points.

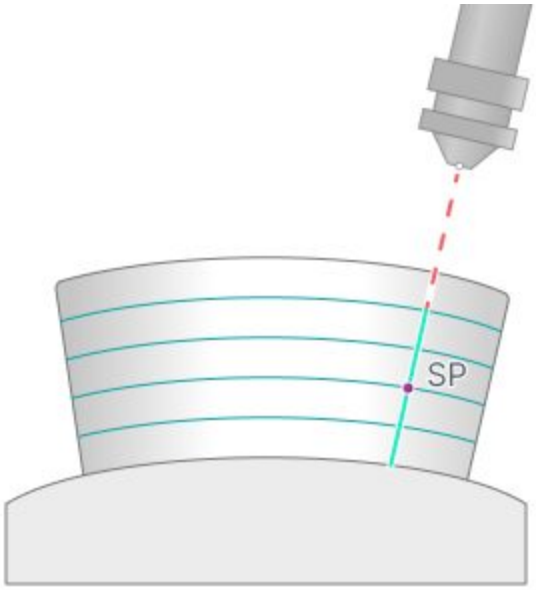
Allows to set custom rules for deciding starting point on print layers.

▼ Click here to expand..

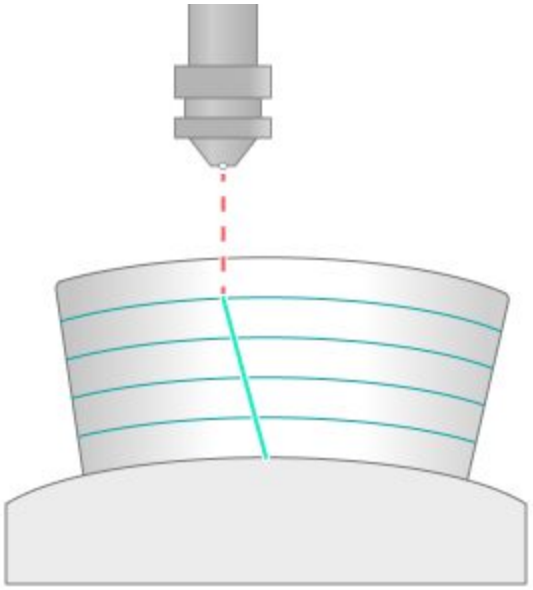
Auto. The start point is chosen automatically



Custom for first layers. Allows to manually choose the start point for the first layer of features. Start points for subsequent layers are determined by the closest distance to the start point of the previous layer. You can set them on the Job assignment page.



Custom for all layers. Allows to set start points for all layers by setting a curve. In this case, the start points are calculated based on the nearest distance to a specified curve. You can set them on the Job assignment page.

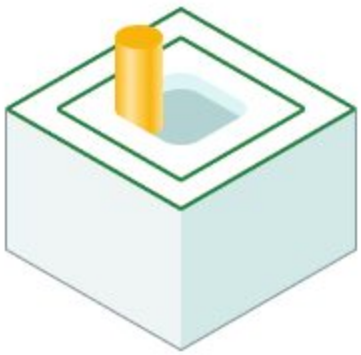


Printing order.

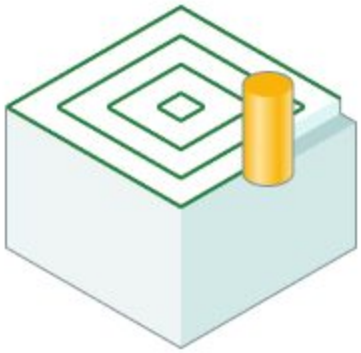
Determines the order of walls and infill when printing.

▼ Click here to expand..

Inside to outside. Infill will be printed first, after that inner walls and outer wall.



Outside to inside. Outer wall will be printed first, after that inner walls and infill.



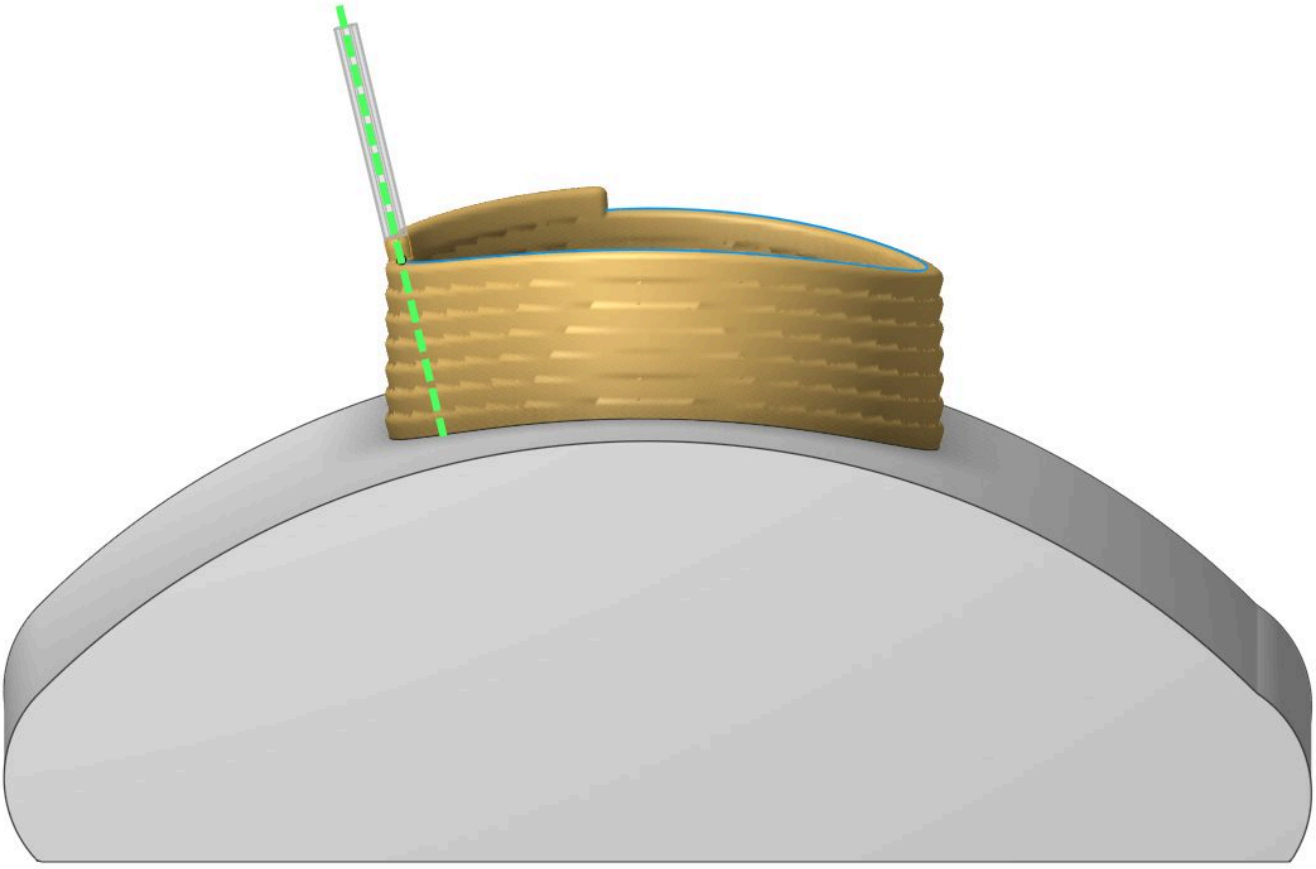
Alternate directions. The printing order will alternate layer by layer, starting from "Inside to outside" order.

Tool orientation.

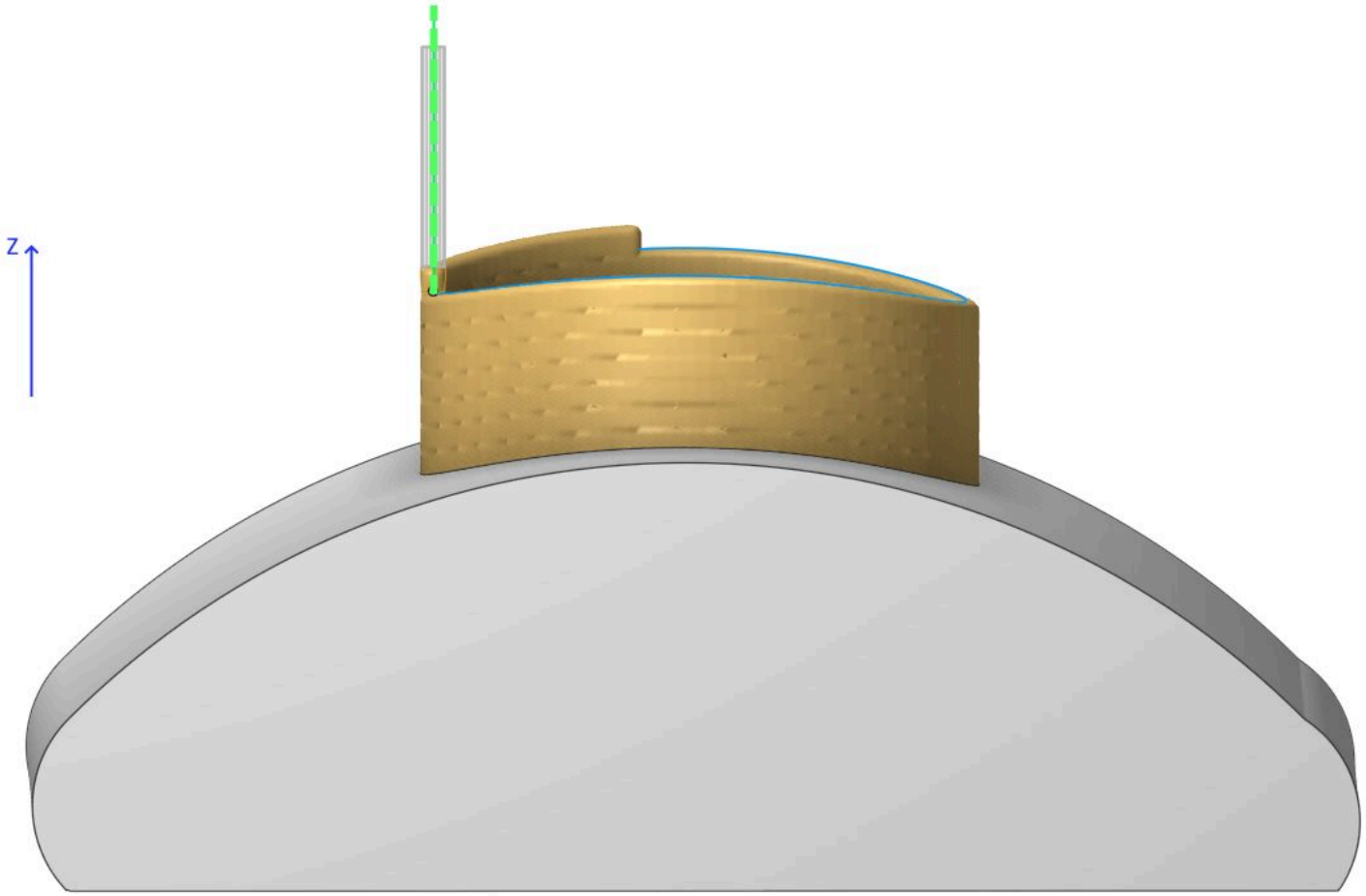
This parameter allows you to set different tool orientation (normal) without changing the original toolpath. There are five options avalaible for users: "Normal to surface", "Fixed", "To rotary axis", "Through point" and "Through curve".

▼ Click here to expand..

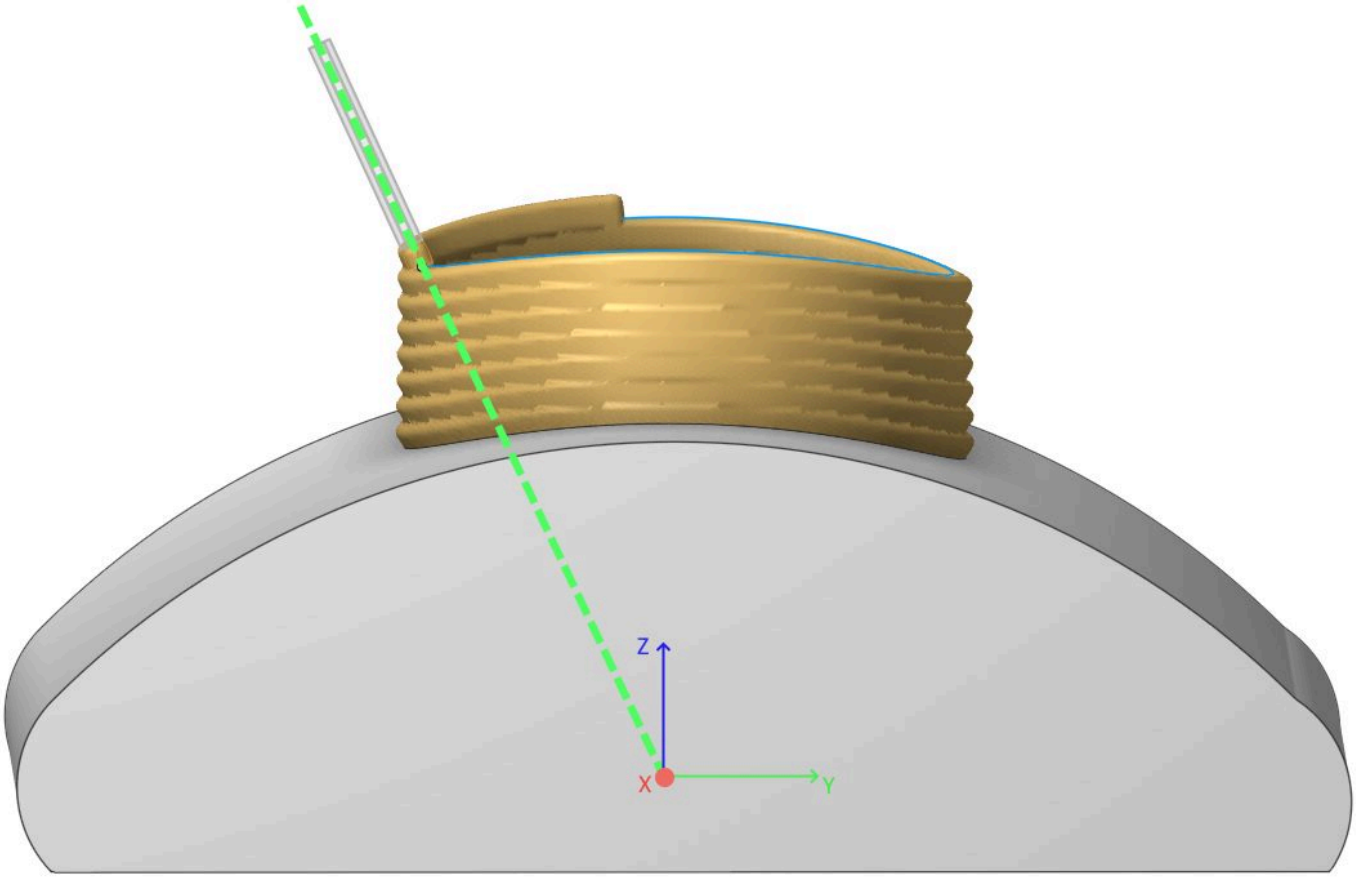
Normal to surface.



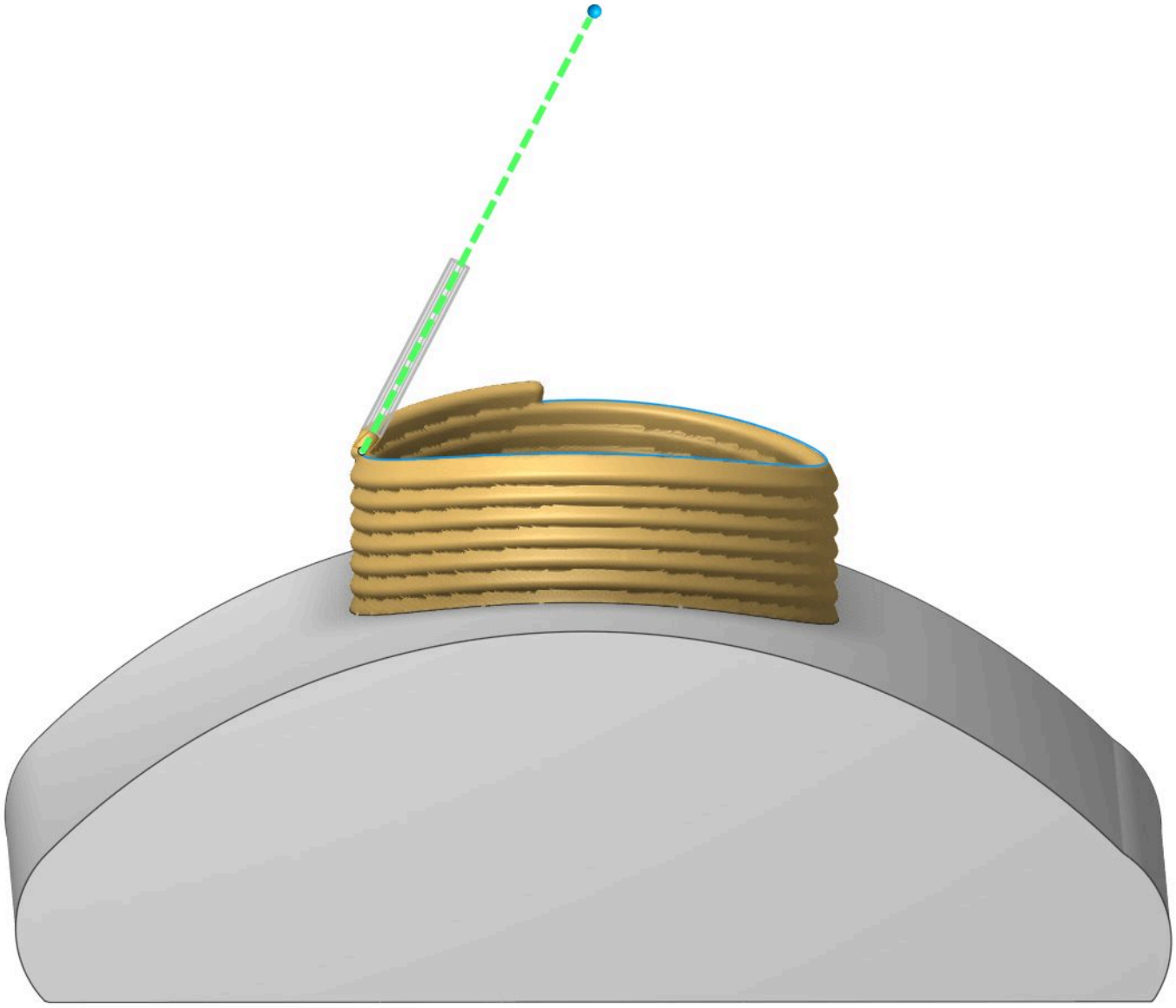
Fixed



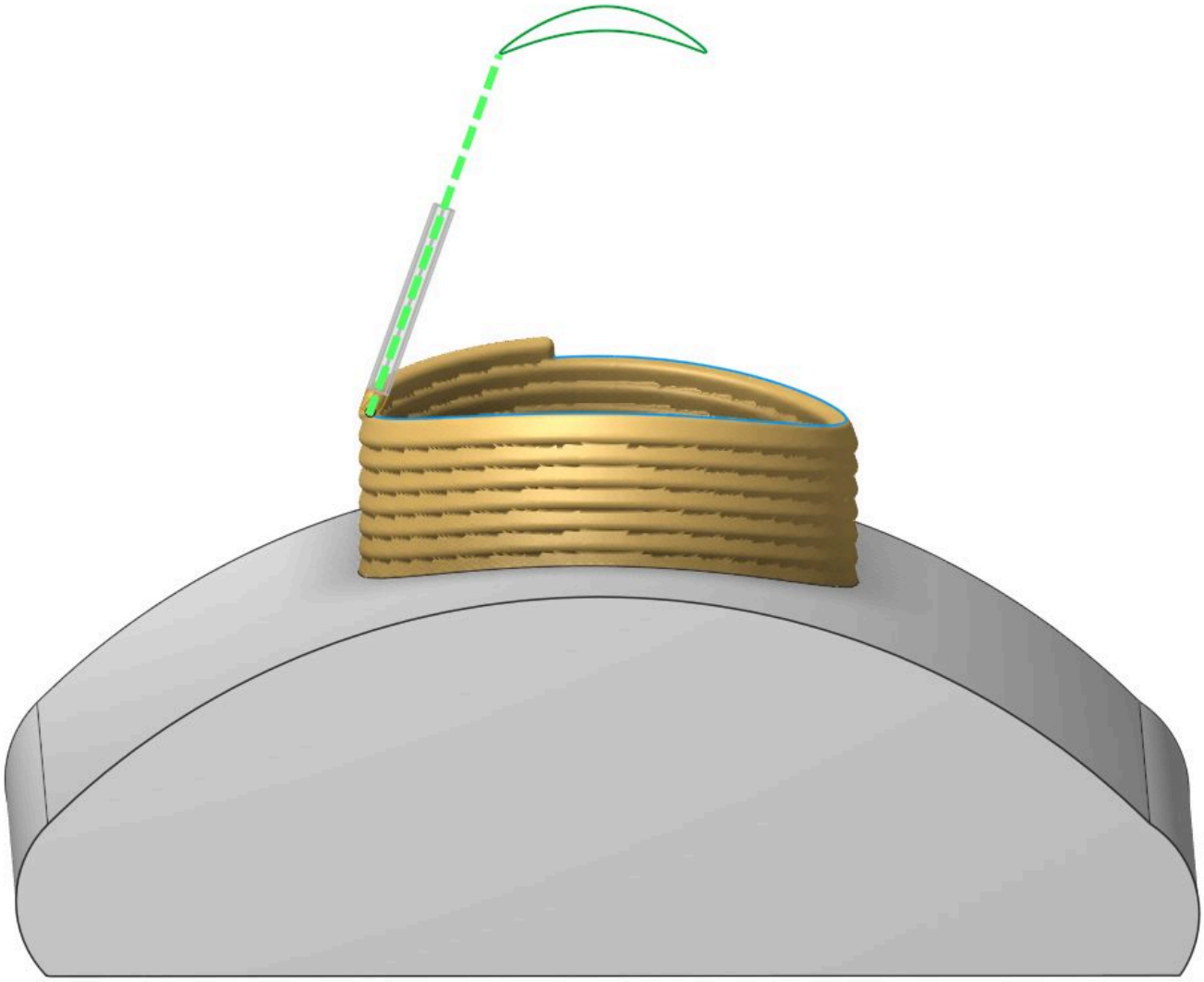
To rotary axis.



Through point.



Through curve.



More information about the Tool orientation parameter can be found in the description of the 5D Surfacing operation. Details.

Limit rotation angles.

When the flag is set, it specifies the limit values of the tool's angular position relative to the specified axis. This parameter group works similarly to the 5D Surfacing operation. Details.

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can

configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Distances. Distances used in Links rules Details.

Feeds/Speeds:

Using this dialogue the user can define the rapid feed value and the feed values for different areas of the toolpath. Also it's possible to apply "Corner control" parameters on this page. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

> Important note
>
> The Multiply scheme parameter in this operation behaves differently compared to other operations.
>
> Normally, this parameter duplicates the entire toolpath of the operation - including the slicing results and the links/leads between layers.
>
> However, in this operation, it only duplicates the slicing result before the linking stage.
>
> This allows to multiply features and apply settings like Links/Leads or Split by Layers, which would not be possible using the standard multiplying method.

Part:

A Part in additive manufacturing is a desired geometry to be manufactured by the printing process. Details

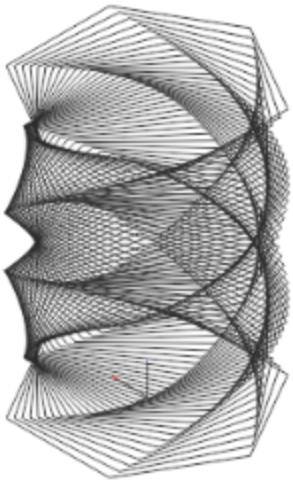
Workpiece:

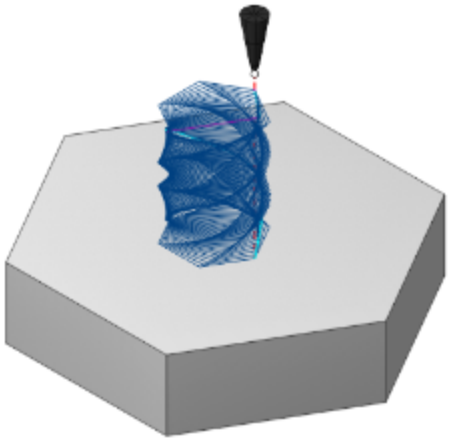
A Workpiece in additive manufacturing is t he pre-existing material or substrate that serves as the foundation for printing (can be empty). Details

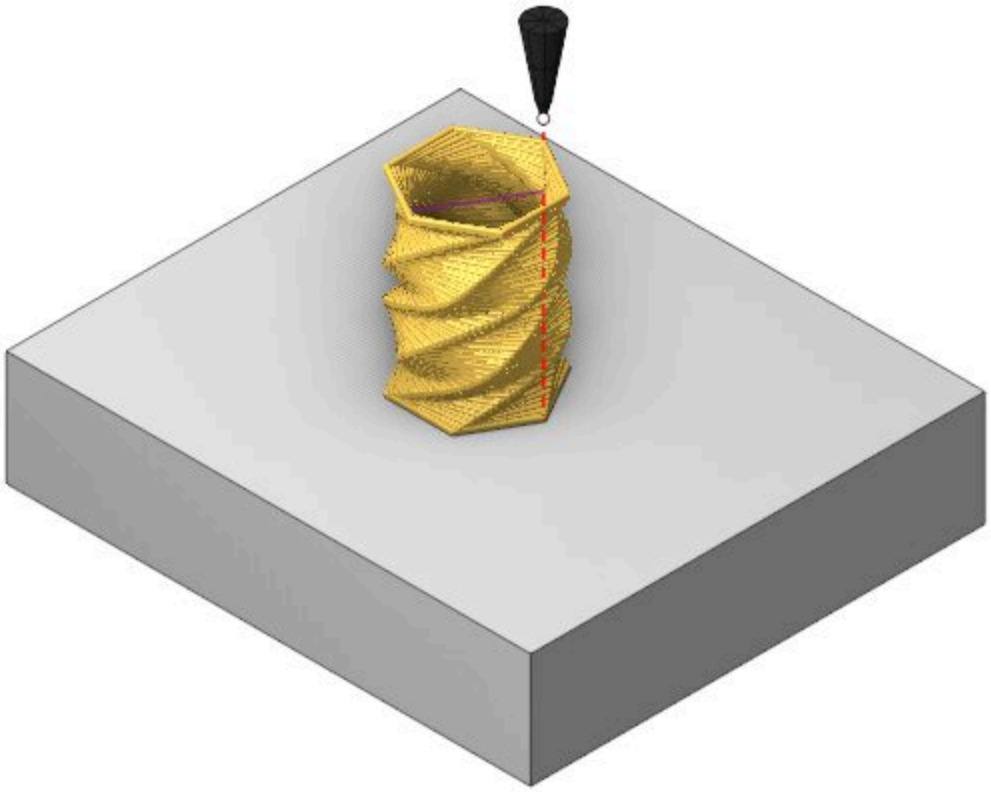
Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Direct cladding







Application area:

The operation lets you do surfacing along curves you've set up beforehand. You can prepare the curves using any tool — for example, a third-party CAD program. Basically, the operation turns a bunch of curves into a path for the tool to follow. It also automatically sorts and divides the work area into regions. You can choose between layer-by-layer and spiral processing modes.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Curve. Set the toolpath along the selected curve. Details.

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list

Strategy:

Printing strategy.

The choice of strategy affects the availability of parameters in the Sorting group.

▼ Click here to expand..
Planar. Allows you to use layer-based printing options

Helical. Allows you to use spiral printing options

Sorting.

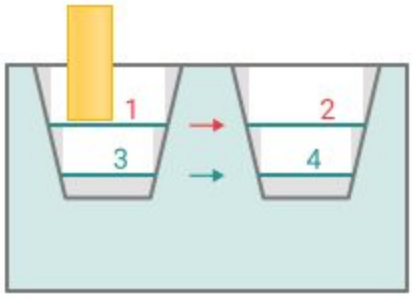
Toolpath sorting and linking parameters.

▼ Click here to expand..

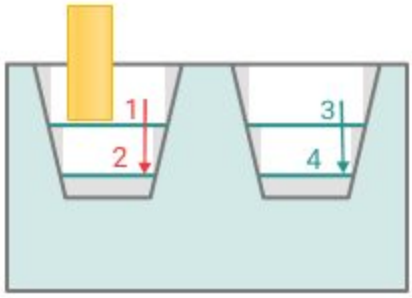
Sort by.

Allows to set printing order of layers and features.

Layers. Allows to print all features layer by layer.



Features. Allows to print all features one by one.



Split by layers. This parameter allows you to split your features into a desired number of layers. It is available only when the Sort by parameter is set to Features option.

Allow reverse direction. Changes the processing direction of a single curve.

Start point. Allows to set custom rules for deciding starting point on print layers.

Do not change. The starting point is selected automatically.

Nearest. Selects the nearest point to the instrument

Manually. The user selects the starting point interactively

Optimize order. If disabled: the curves are selected from the add list.

If enabled: the curves are selected by the shortest distance

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Installation:

The operation is supplied as the DirectCladding operation extensions extension. To install it:

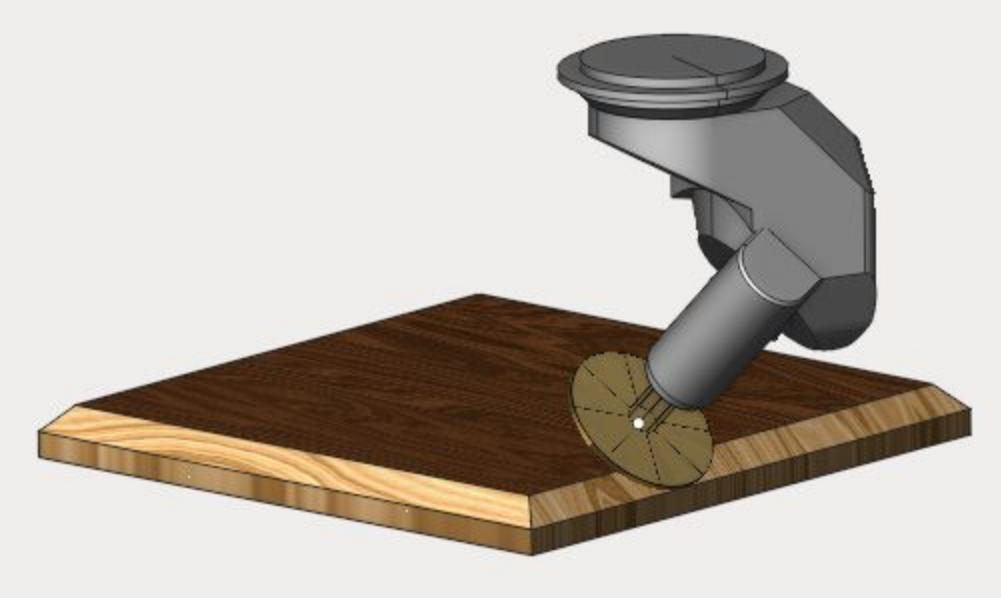
1. On the Home page, open the Extension Store tab and go to the Store page.
2. Find DirectCladding operation extensions and click Install.
3. The operation appears in the Additive group of the new operation window. If it does not, restart the CAM system.

You can also install the extension from the web storefront apps.encycam.com. Details

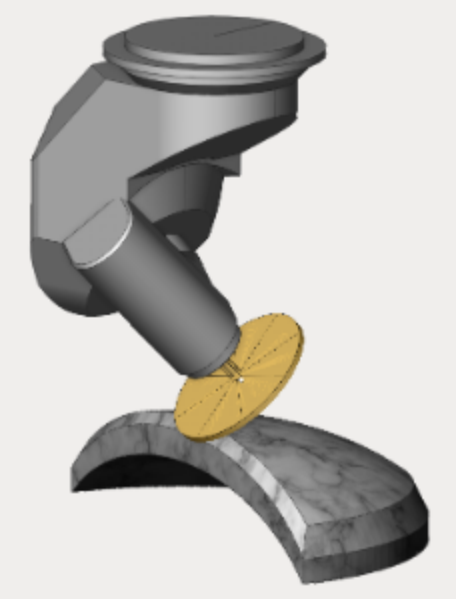
Disc tool machining

This group of operations is designed for sawing and machining materials such as wood, stone and similar with a disk tool.

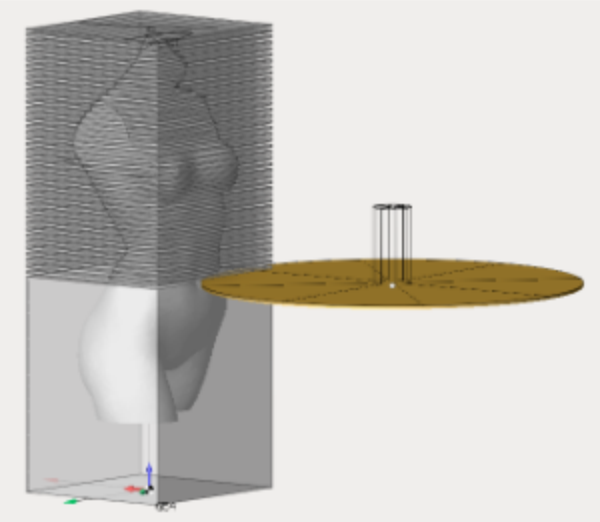
The Disc cutting 2D operation is primarily intended for sawing sheet materials, machining flat furniture facades, plates.



The Disc cutting 6D operation is designed for sawing both flat and volumetric parts. As a job assignment in this operation, both plane and spatial curves and edges of faces can be indicated.



The Disc Roughing operation is designed to prepare a stone body by applying cuts with a disk tool in the area where material needs to be removed and then manually remove the thinned layers of material by spalling, followed usually by finishing operation with the appropriate tool.



The disk tool operation group, like some other operation groups, is available under special license.

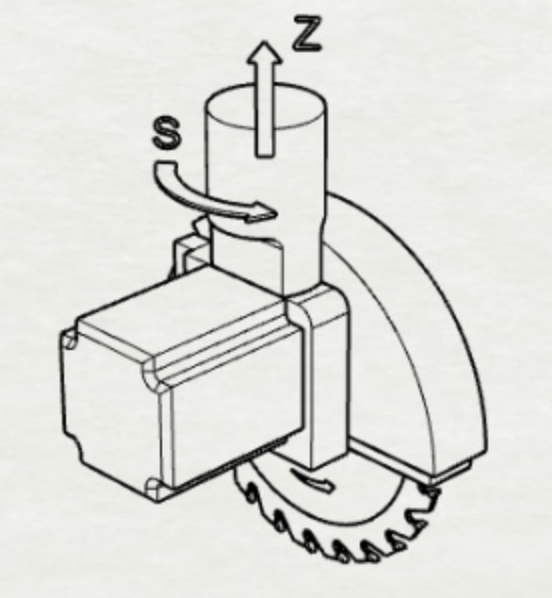
In order to use these operations the possibility of working with this group of operations must be specified in the machine schema. Thus, if the disc tool operation group is missing in the list of available operations, then the reason is either an incorrect licence, or the incorrect machine schema which cannot work with this group of operations.

Disc tool

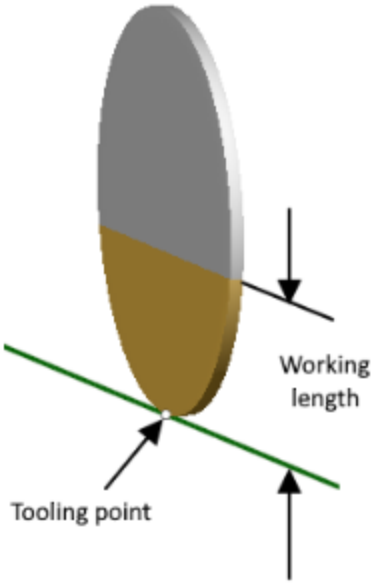
Two tool types are available for disc tool operations depending on the type of mounting.

Saw Blade.

This type assumes having an axis S (for tool rotation around Z axis). The direction of Z axis is along the tool rotation plane.

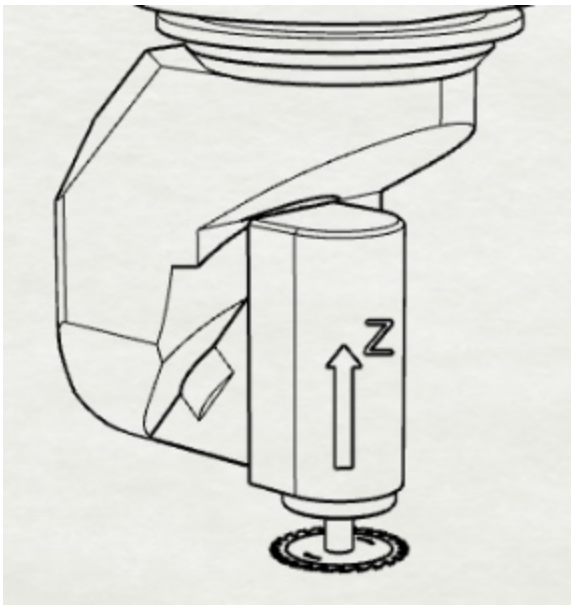


The tooling point of a saw blade is located on the edge of the saw unlike a milling tool, where this point is located on the tool end on its rotation axis. Saw blade type tools have also 'Working length' parameter which defines the height of saw's working area. The rest of the tool is assumed to be covered by the hood. This parameter directly affects the toolpath which is generated so the non-working area of the tool is not used for the workpiece material removal.

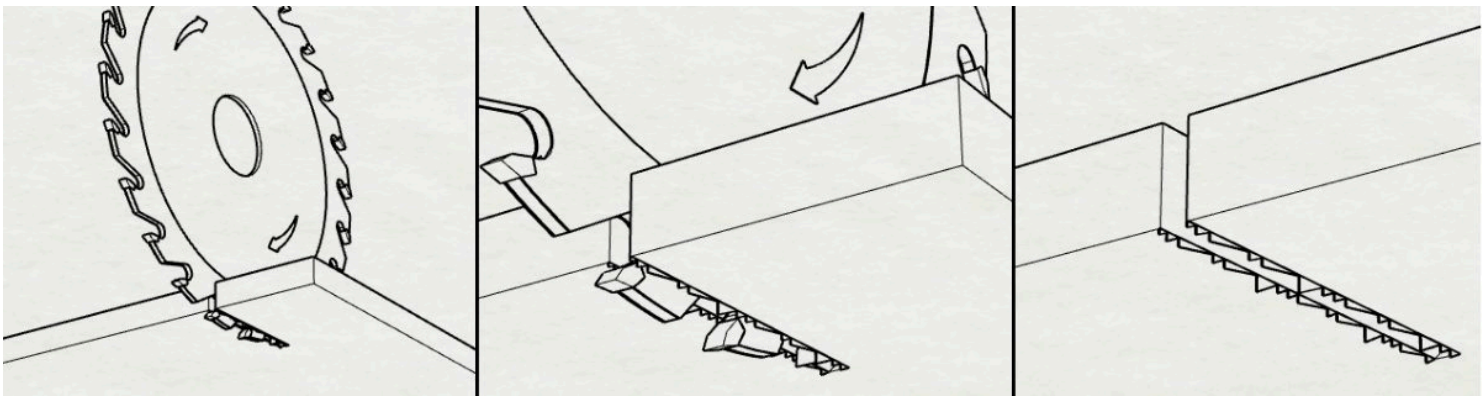


Milling Tool.

Tools of this type are usual cutters with big diameter and low height. They are put in a spindle, axis S is not needed because its functionality is provided by other rotary axes of the machine. Tool's Z axis is directed along the rotation plane normal.



There is an important aspect the dist tool sawing operation. If it is sawing of a wooden panel or a marble slab in one pass (through-and-through), then on the edge of the panel in the area where saw blade teeth get out from the material (in the image below it's bottom edge) spalls and burrs can be formed (only spalls in case of marble).



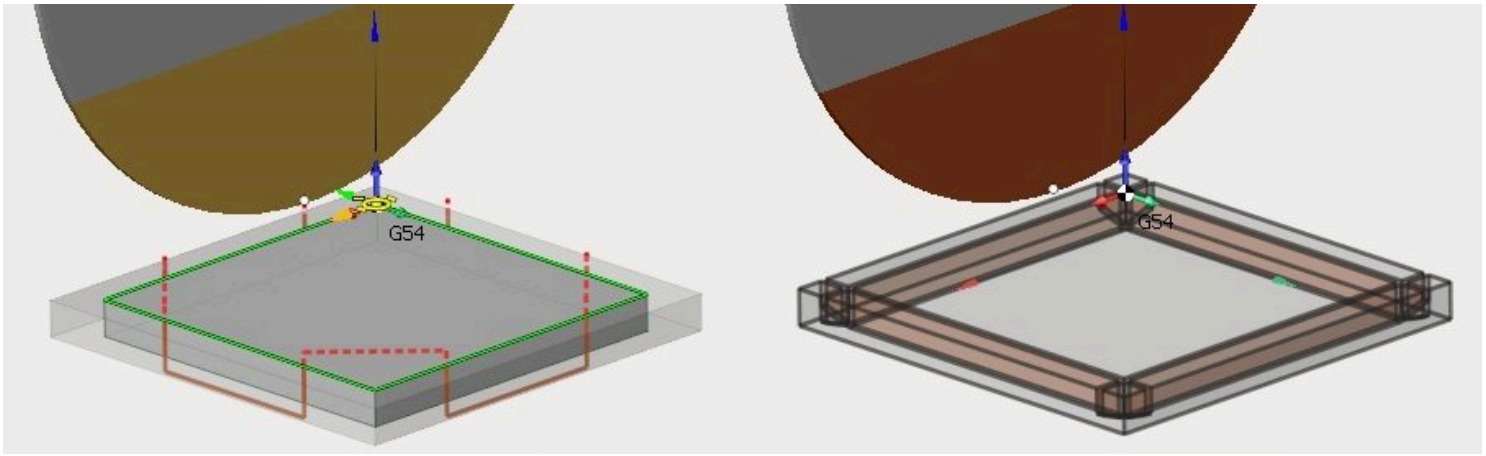
To avoid this situation (lets consider two pass variant) first saw pass is done without getting out of material on the panel's bottom level (as such ensuring that top edge contains no defects because panel's top level will be the zone where saw's teeth enter the material), while the second pass is done with exit on bottom level but moving the tool in opposite direction (or same direction rotating the S axis for Saw Blade type tool). This allows for defectless bottom edge because, using this strategy, panel's bottom level in the second pass will be the zone of entering material for saw's teeth, while the top edge was formed in the first pass.

Next aspect to pay attention to is the concept of inner and outer corner overlap.

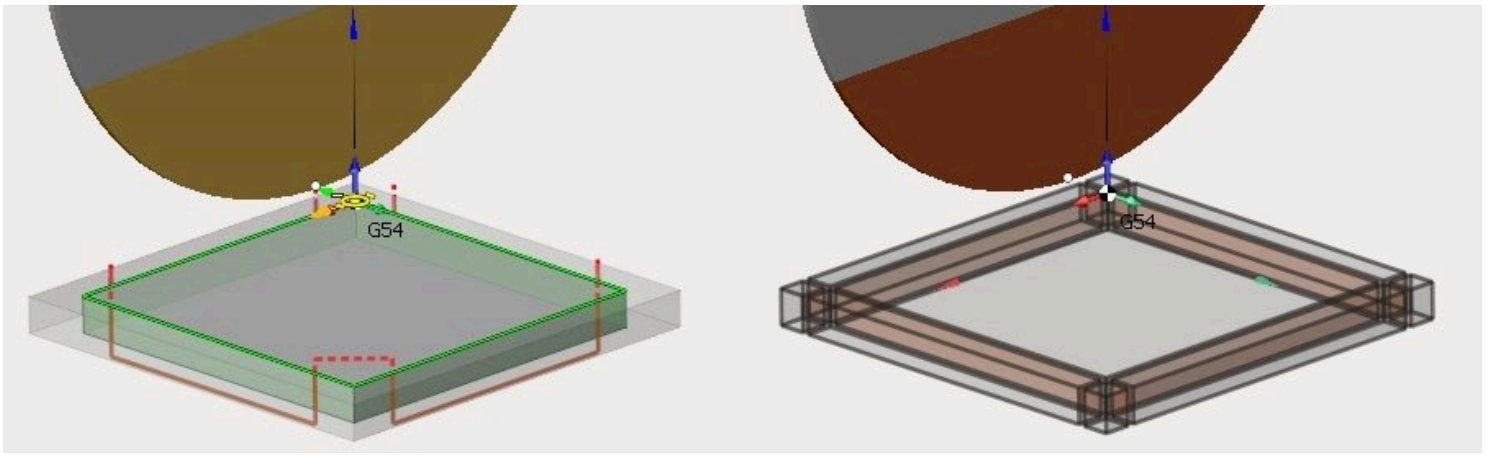
Outer corner overlap allows to increase the length of the contour along the faces for the specified amount. It can be necessary for removing workpiece chips along the faces of contour.

Lets consider an example. Suppose its needed to saw rectangular panel from the workpiece with stock.

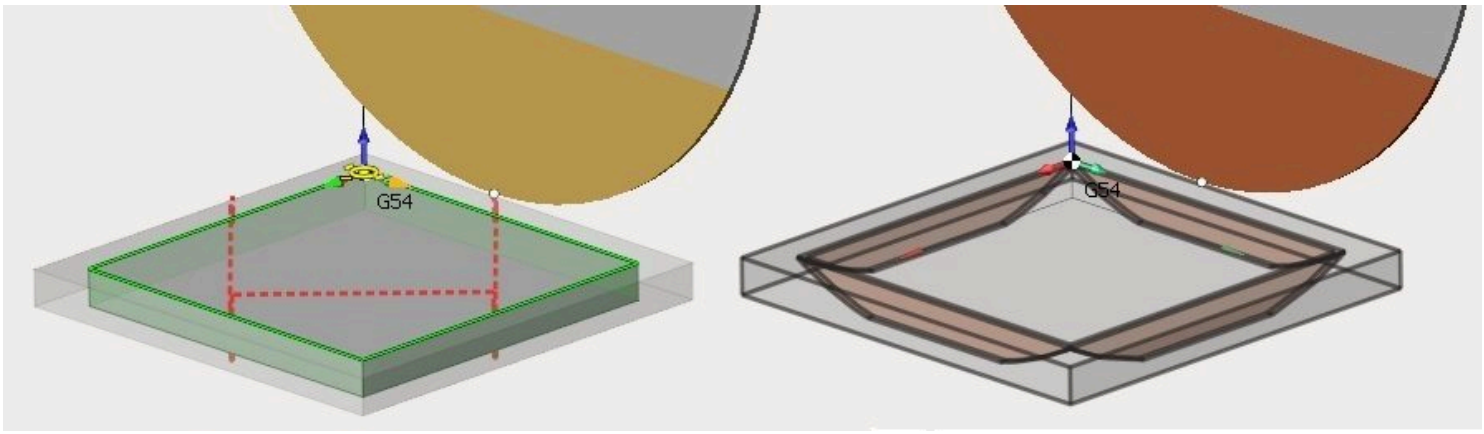
If the "Outer corner overlap" is not used, then material chips will be removed from part only after all faces are traversed by the tool, because the trajectory is formed according to the specified contour, not accounting for the workpiece stock.



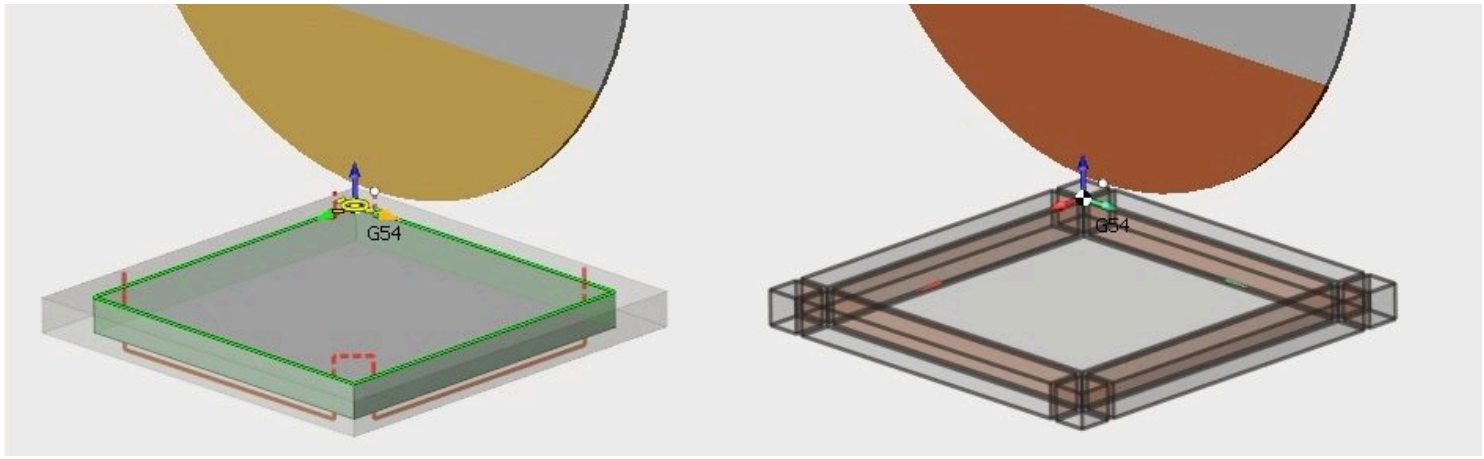
If overlap is used then material chips can be cut after a face is finished.



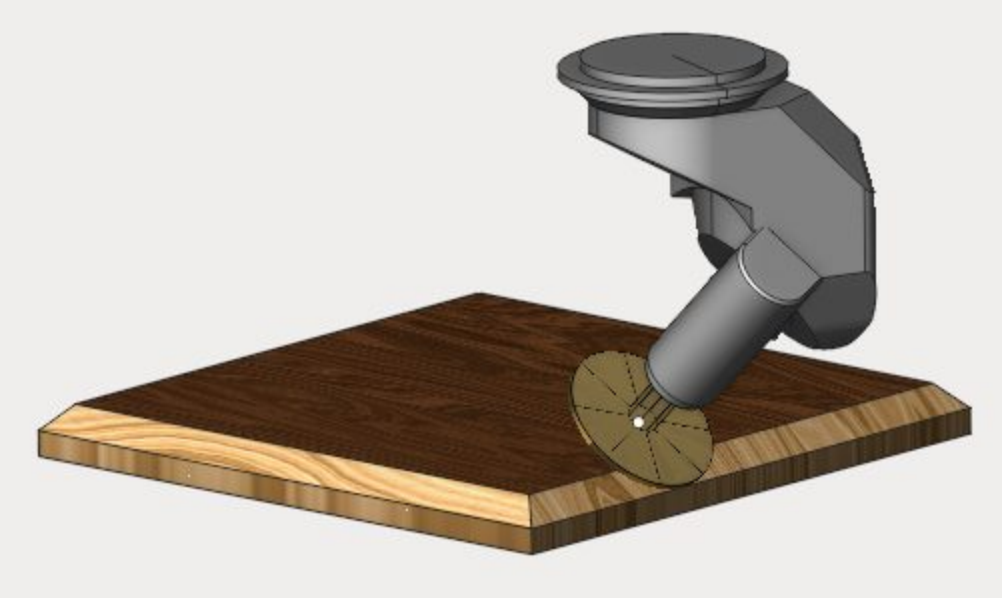
Another objective is during inner contour machining. Without changing overlap parameters toolpath trajectory will be limited by the saw profile touching the contour corner on workpiece top level.



Using overlap full contour of the part can be achieved on both top and bottom workpiece levels.



Disc cutting 2D



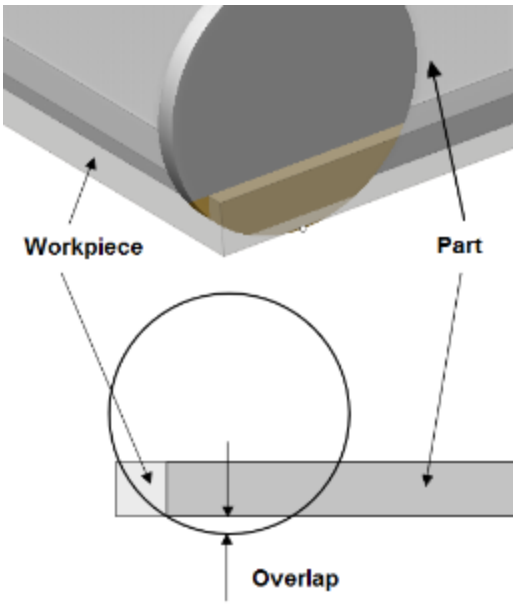
The operation is designed for the sawing of sheet materials. The most parameters of the disc cutting 2D are the same that 2D contouring operation has. The job assignment can be defined by the planar curves (profiles) or edges of the 3D model.

The job assignment is defined exactly the same way as it can be done in 2D contouring. Select the curve or the edge in the graphical view and press the Curve button. Firstly in the job assignment 3D model contours or edges are specified, along which the disc trajectory will be generated. To do so select a contour or an edge and then press the "Curve" button. Features for machining can be added the same way by selecting a surface on the 3D model and pressing the "Pocket" button, after that the selected surface borders will be added automatically as a job assignment. To limit top and bottom level select any geometrical item lying on required level and press the "Top level" or "Bottom level" button.

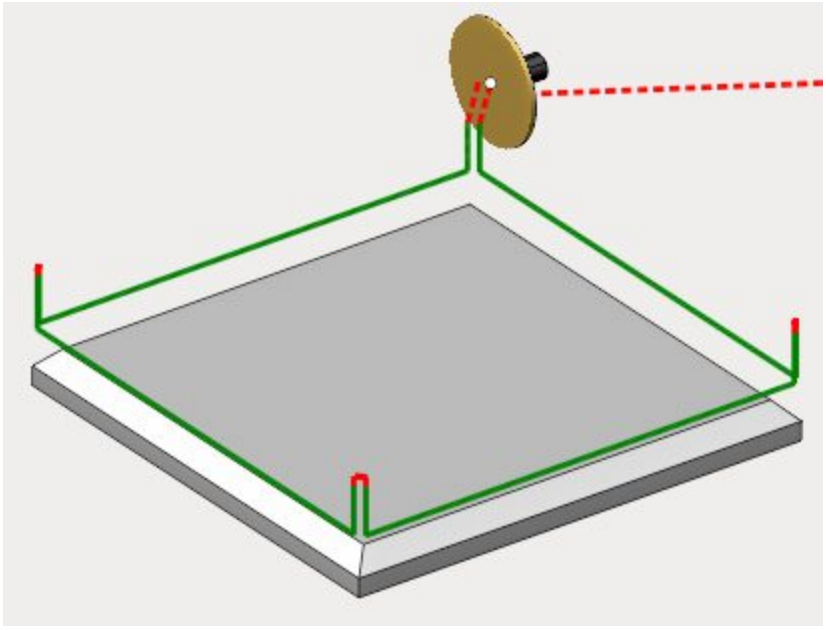
With the "Job assignment" tab open and given curves, you can also interactively select the direction of movement of the tool and its location relative to the curves (left/right) by left-clicking in the graphics window on the arrows on the curves.

Besides that the following parameters are available in the inspector

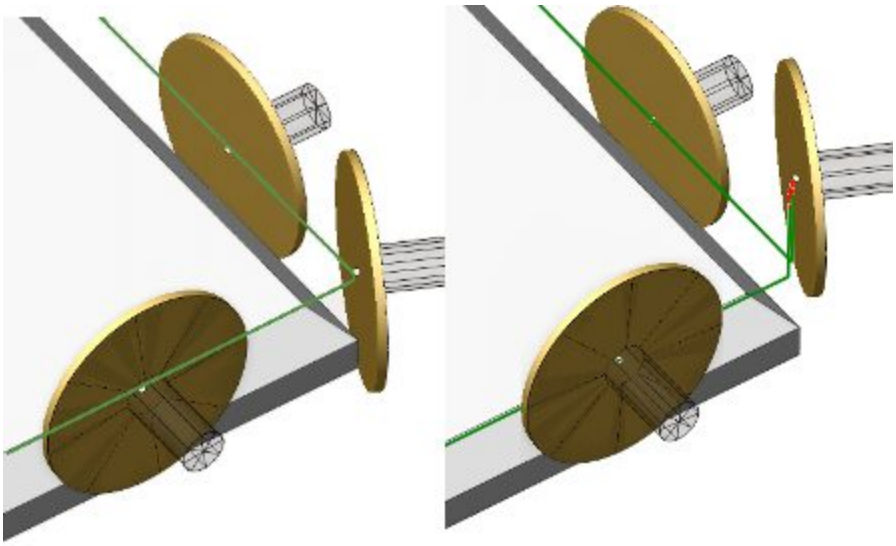
- Overlap. The parameter defines exceeding of the workpiece bottom level by the saw blade.



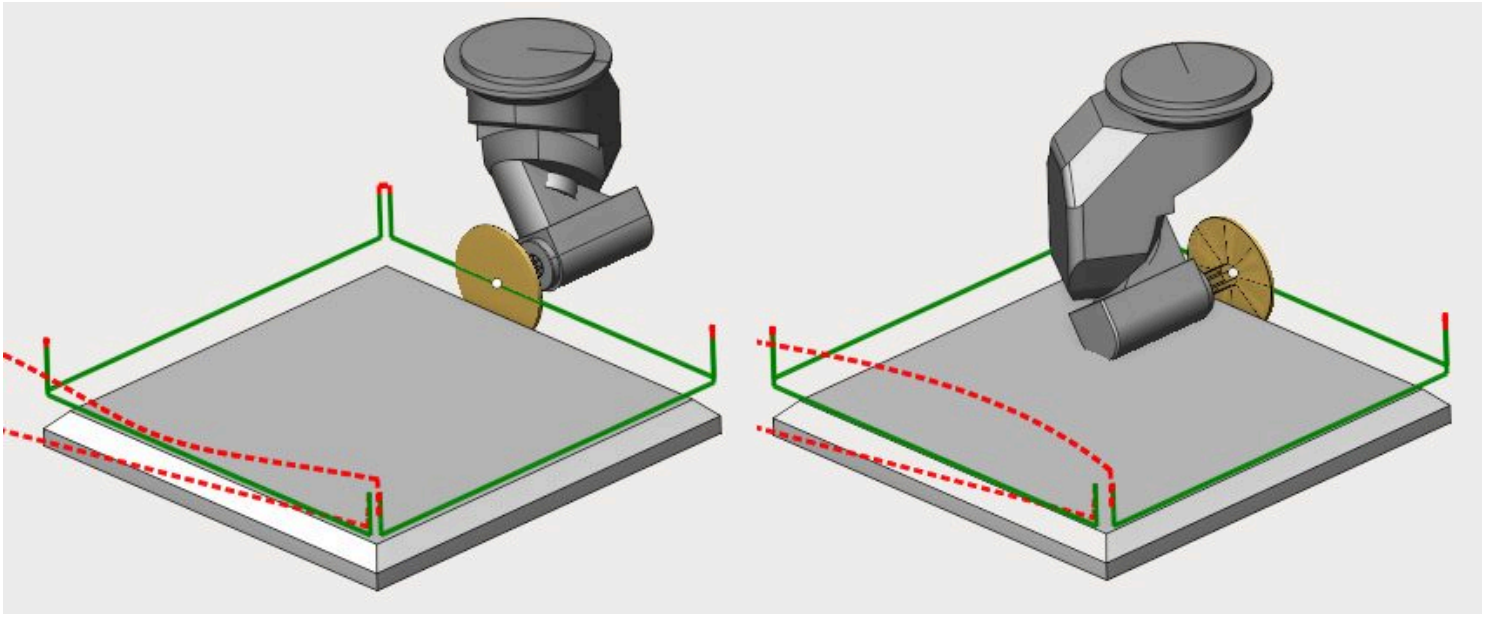
- Overlaps in corners. The parameter allows to specify the height on which the tool overturn will be done when changing side of the machined corner.



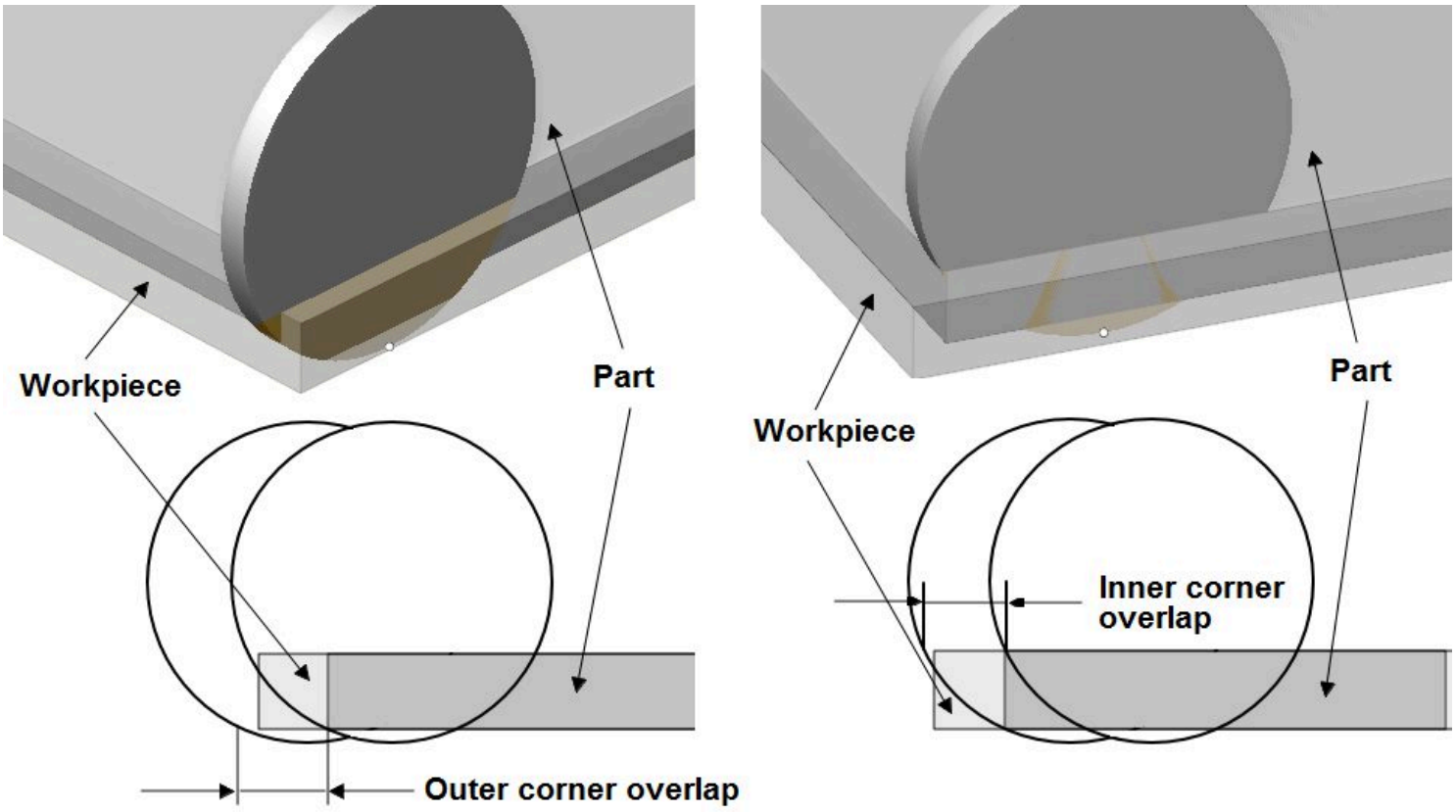
- Sharp corners. The parameter defines the angle limit for considering the corner sharp and during machining of which tool return is needed. On the image below is the passing the right angle (in the first case the sharp corner value is less than 90 degress, in the second one - greater).



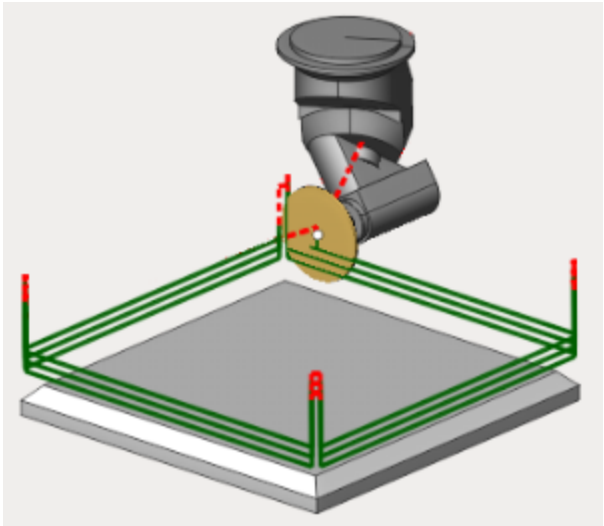
- Sawing mode. It defines the sawing type - climb, conventional, back or front. Climb/conventional - ensures the respective cutting mode. When changing the contour direction, the saw will overturn and start to machine the part with the opposite side. Back/front side - the saw always machines the part using the specified side, regardless of the direction of the contour.



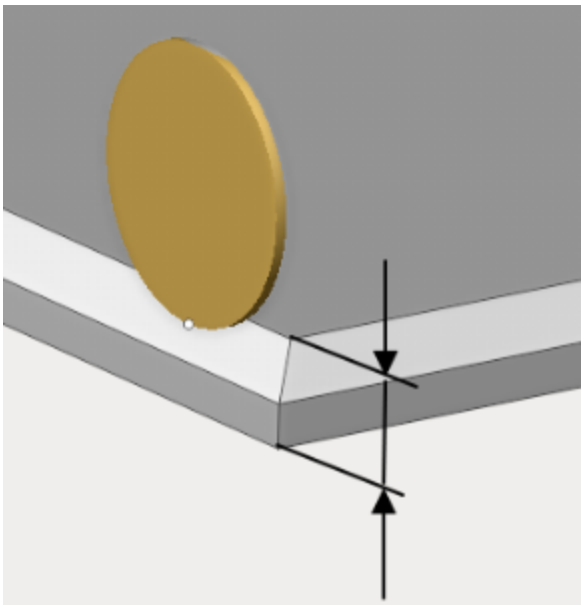
- Overlaps. Parameters specifying overlap value for outer and inner corners respectively.



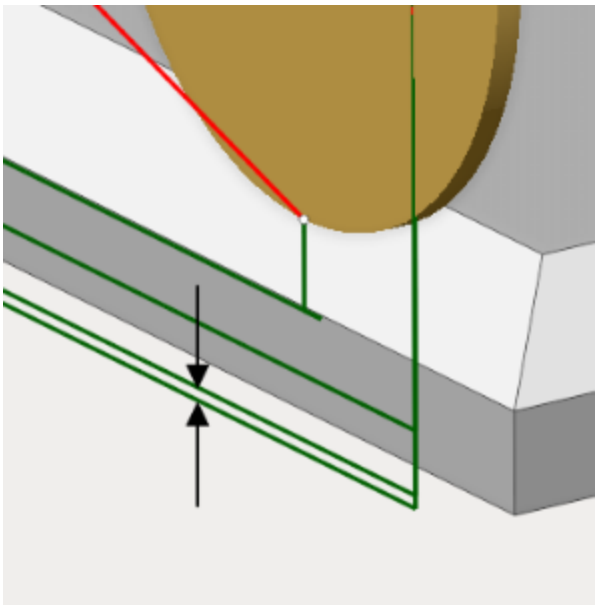
- Layers (depth). Defines the number of tool passes. Can be specified as a number of passes or automatically calculated from top and bottom machining level. Can also be specified in mm or as a tool diameter percent.



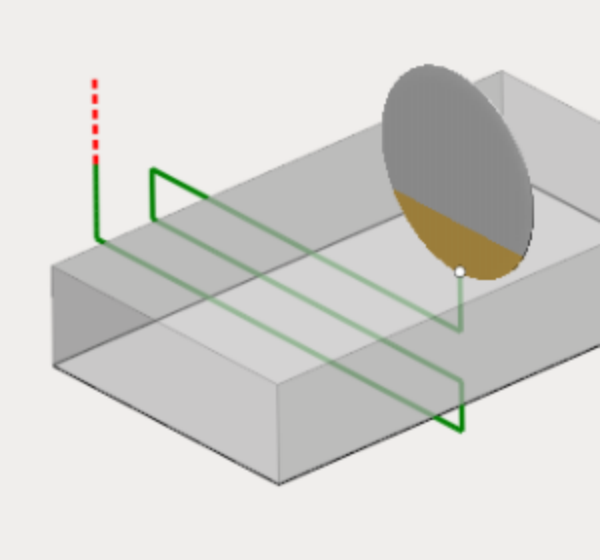
- Top and bottom part level - define the height of the part to machine.



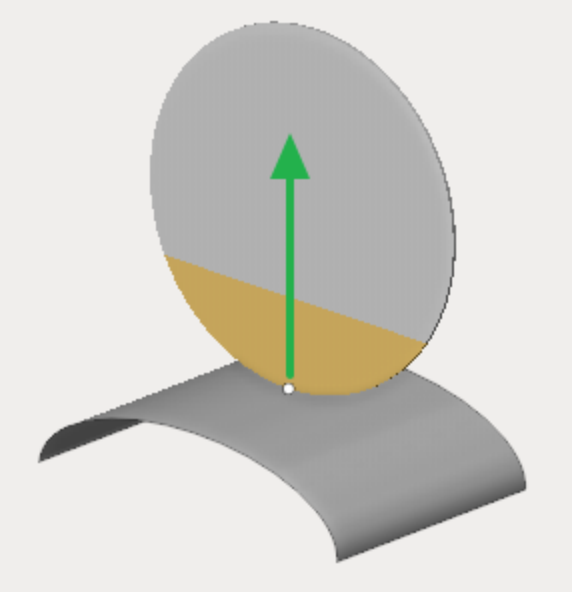
- Z cleanup. The parameter defines the stock value before last pass.



- Sorting. When machining by layers after performing the first pass along the first contour, a transition is made to the first pass along the next contour. The transition to the second pass is made after all contours have been processed. When machining by cavities, at first all passes along the first contour are performed, then all passes along the second contour, and so on.
- Idle moves minimization. When it's enabled, the movement along the given features is performed in such a way that the total length of idle moves is minimal. Otherwise, the transitions forming on idle moves is carried out according to the order defined in the "Job assignment" tab.
- Zigzag. The tool drops down layer by layer and performs horizontal movements with alternating directions.

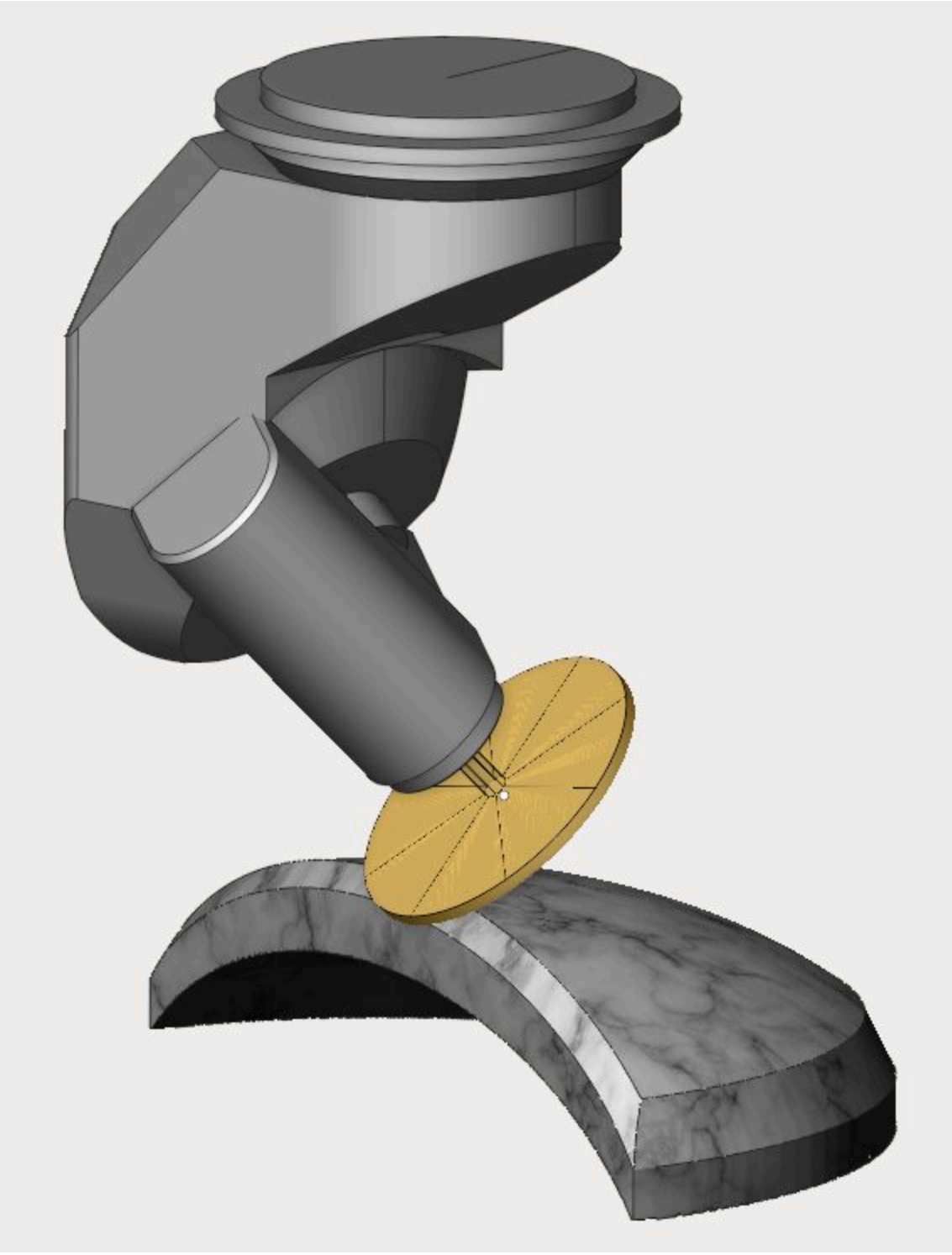


- Tool axis orientation. The parameter is used to determine the orientation of the tool axis on any part of the path. The direction of the axis can be fixed, normal to 4D and 5D surfaces, as well as normal to the base surface. It should be noted that, unlike standard milling cutter, the axis of the Saw blade type tool does not coincide with its rotation axis.



Disc cutting 6D

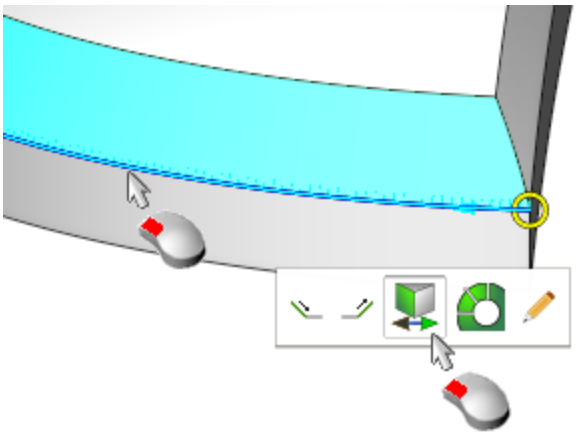
Operation "Disc cutting 6D" is designed for sawing both flat and volumetric parts.



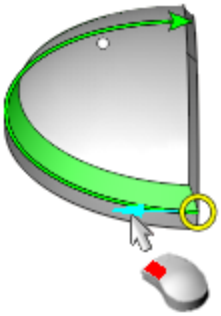
The interface for setting parameters and job zones is similar to the "5D Contour" operation.

In the job assignment, the contours or edges on the 3D model are defined along which the trajectory of the disk will be generated. To set job assignment select contours or edges in the graphics window and add them to the list by pressing the corresponding button that determines the type of object selected -

"Edge / Curve", "5D Curve", "Tilt Curve". Since the edge can belong to two adjacent faces, after its (edge's) definitions in the job assignment, it should be specified which one of the faces will be machined. It can be done by clicking the left mouse button on an edge in the graphics window, changing the face to which the edge belongs.

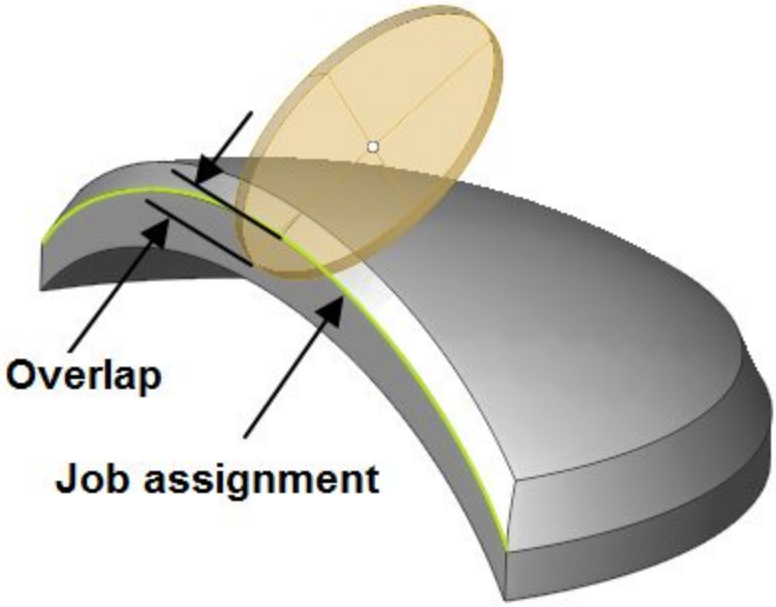


When the "Job assignment" tab is open and the feature was added, you can also interactively select the direction of the tool movement along the curve by left-clicking the arrow in the graphics window.

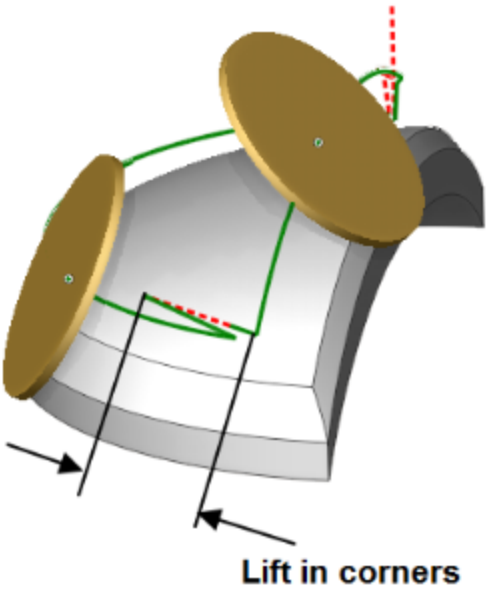


The following operation parameters are available in the inspector ("Strategy" tab).

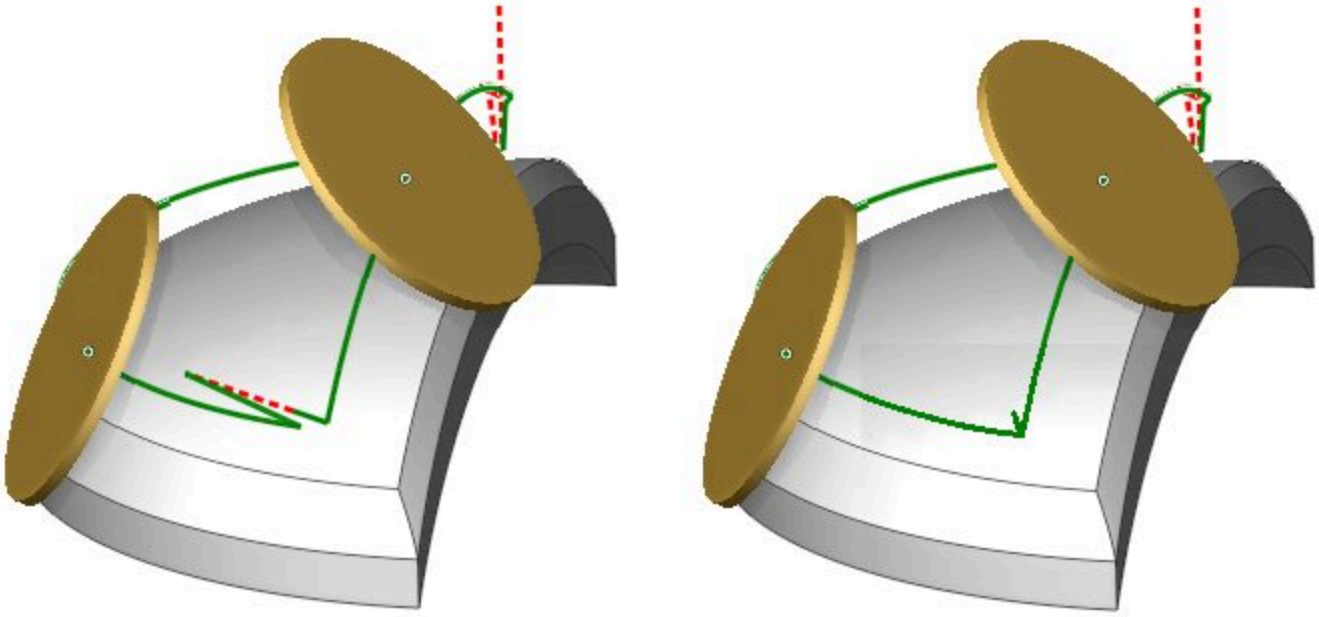
- Overlap (deepening). The parameter allows the saw in the end to move past the level defined in the job assignment (by an edge/curve).



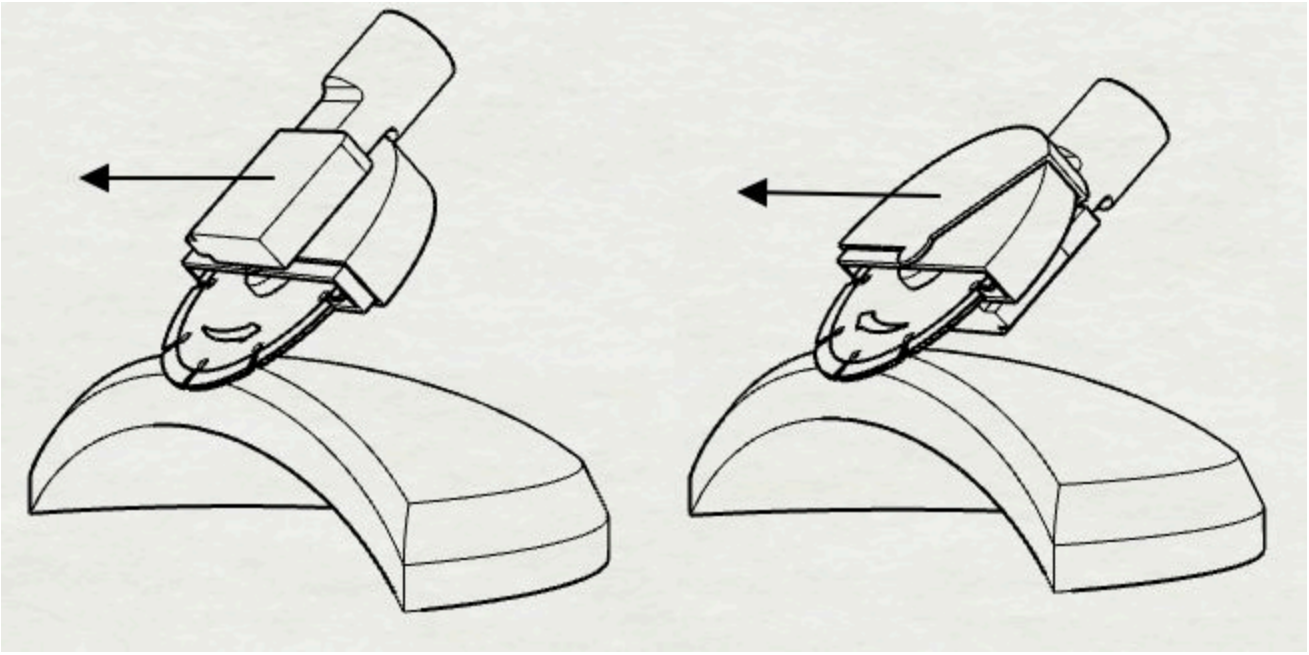
- Lift in corners. The parameter allows to specify the height where the tool overturn will be performed when changing sides of the machined corner.



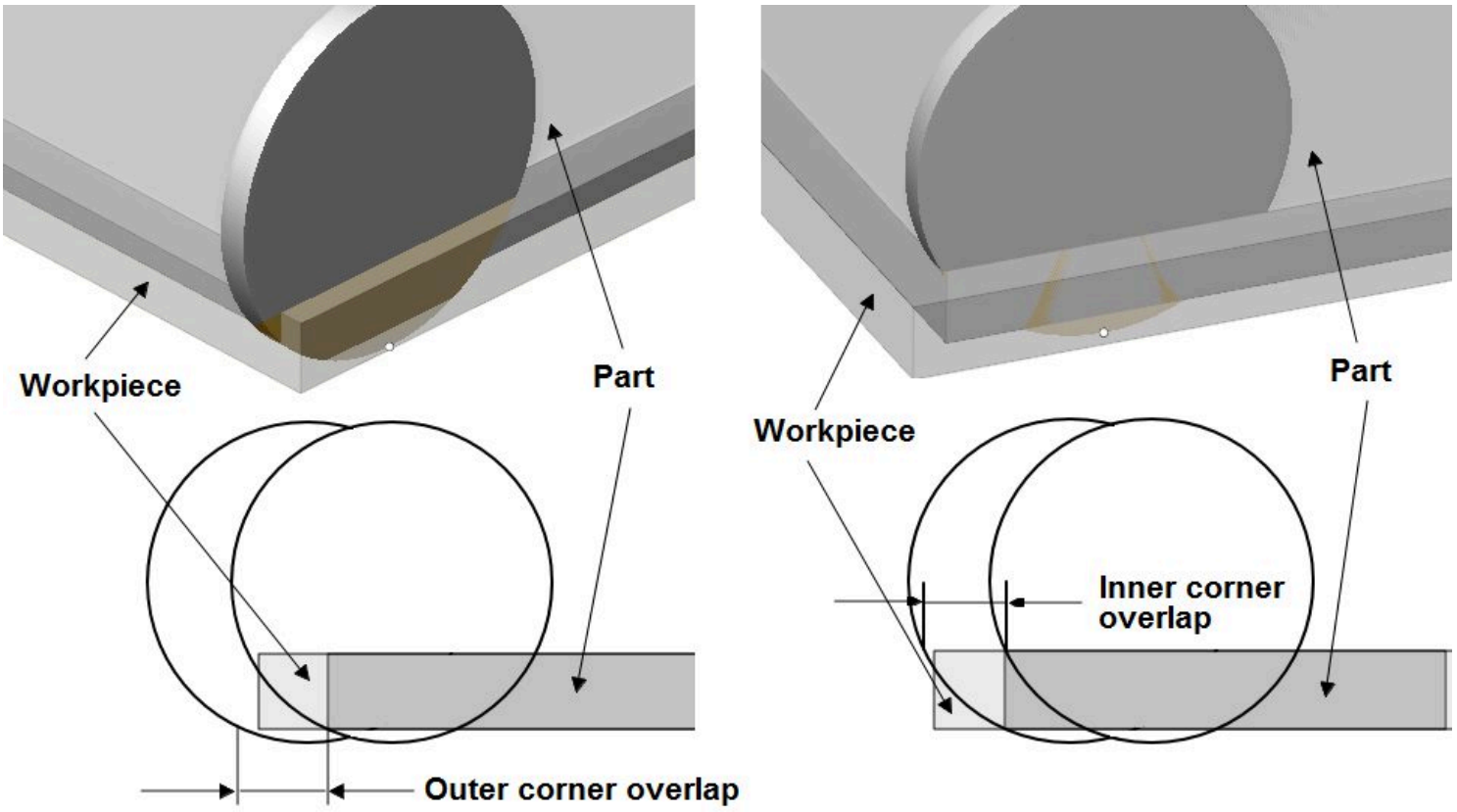
- Sharp corner defines the minimum value of the angle to consider a corner sharp, i.e. when tool overturn is necessary. On the example below two different trajectories around a corner are shown (in the first case sharp corner value is less than 90 degrees, in the second - more than 90).



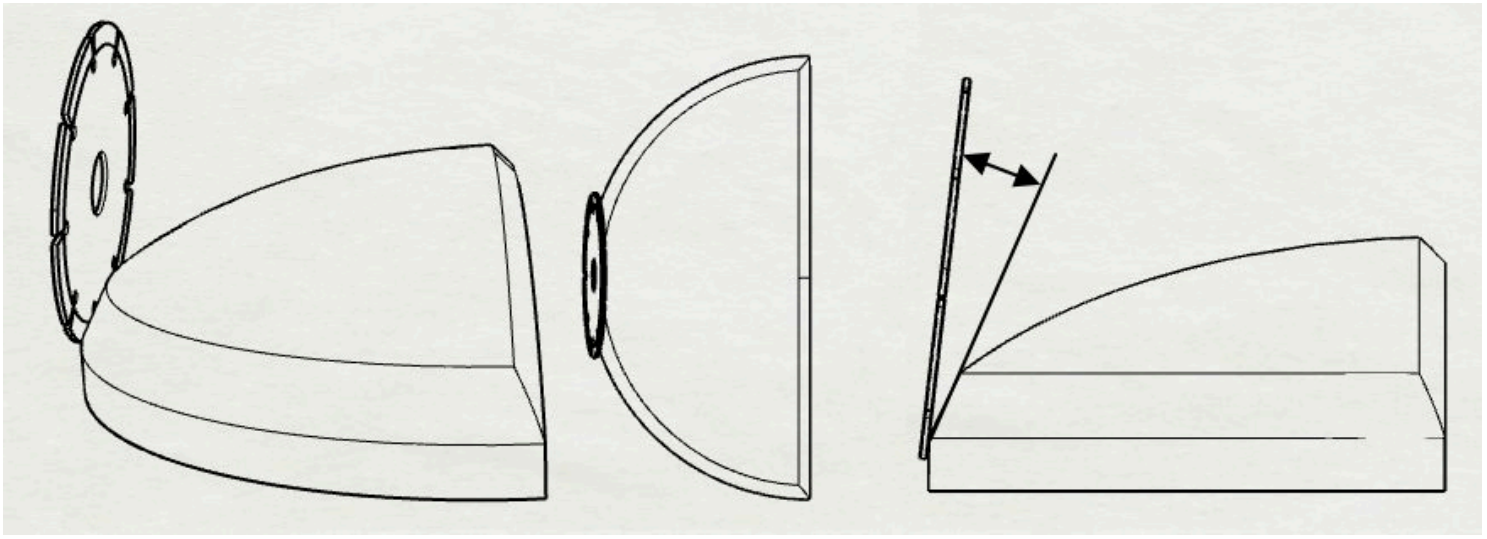
- Sawing mode. It defines the sawing type - climb, conventional, back or front. Climb/conventional - ensures the respective cutting mode. When changing the contour direction, the saw will overturn and start to machine the part with the opposite side. Back/front side - the saw always machines the part using the specified side, regardless of the direction of the contour.



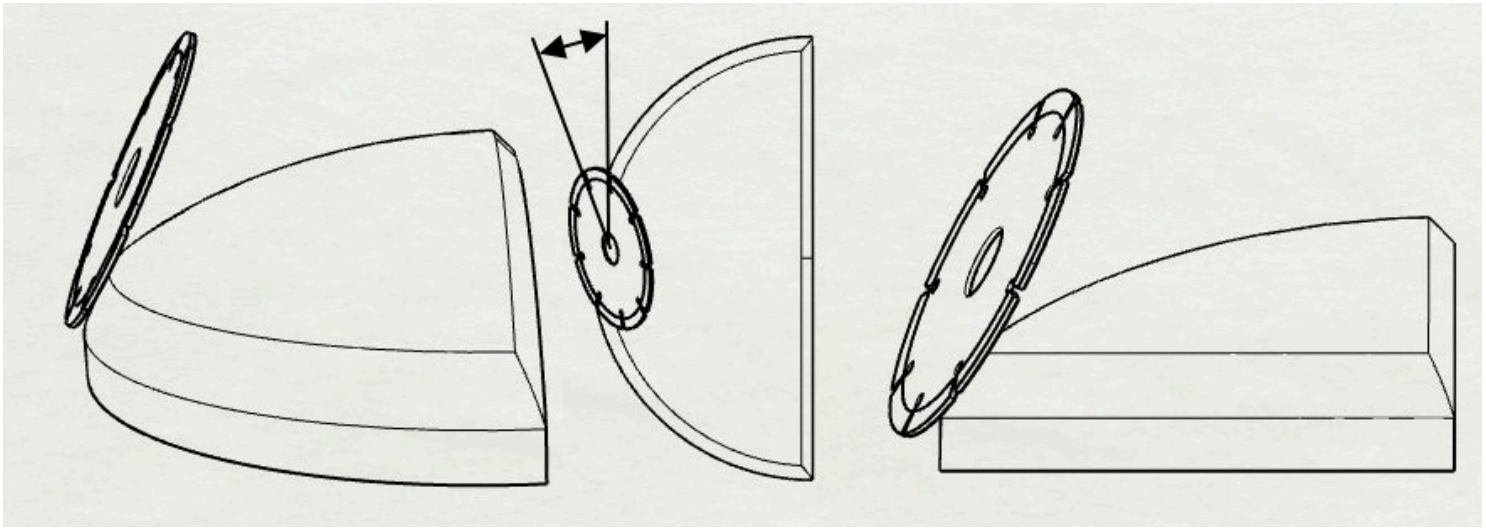
- Overlaps - parameters that determine the distance of overtravel for the outer and inner corners (outer and inner corner overlap respectively).



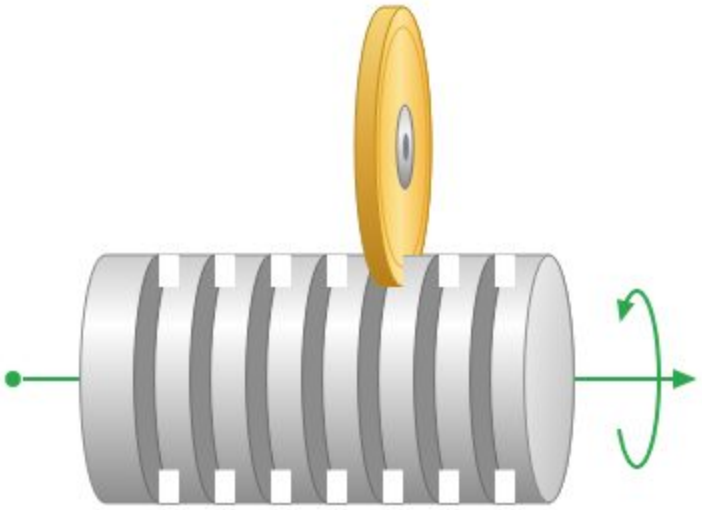
- Lean angle. The parameter defines the deviation angle of the saw's side surface from the machined face.



- Lead angle. The parameter defines an additonal angle between the surface of the saw and the tangent to the work contour at each trajectory point.



Disc roughing



Application area:

The operation prepares stone for finishing by making cuts with a saw blade. The thin webs of material between the cuts are then removed manually by spalling, followed by finishing with an appropriate tool.

The primary tool is a saw blade. The operation also supports end mills and undercut tools. The descriptions and illustrations below refer to a saw blade.

Setup:

The Setup tab defines the part position on the machine, the coordinate system, and the main project parameters. Details

Job assignment:

The model to be machined is defined in the operation’s Part. The job assignment defines machining levels and restrictions.

Top level. Sets the upper machining limit using a selected geometric element. Details

Bottom level. Sets the lower machining limit using a selected geometric element. Details

Restrictions. Exclude specified areas from machining. Details

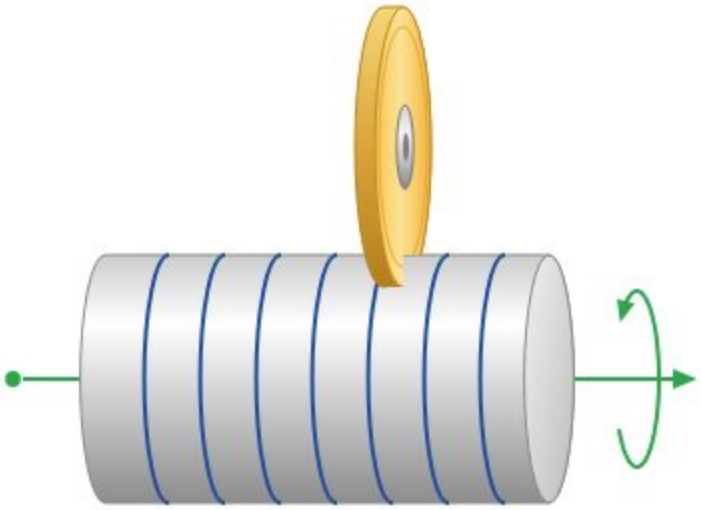
Strategy:

Toolpath:

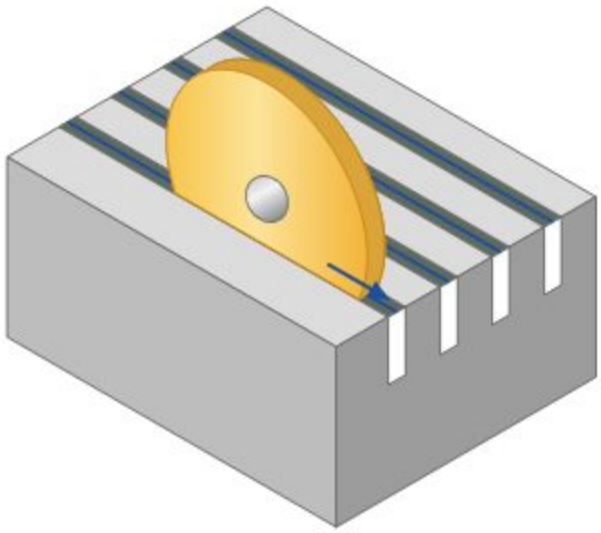
Defines the arrangement of the cuts.

▼ Click here to expand..

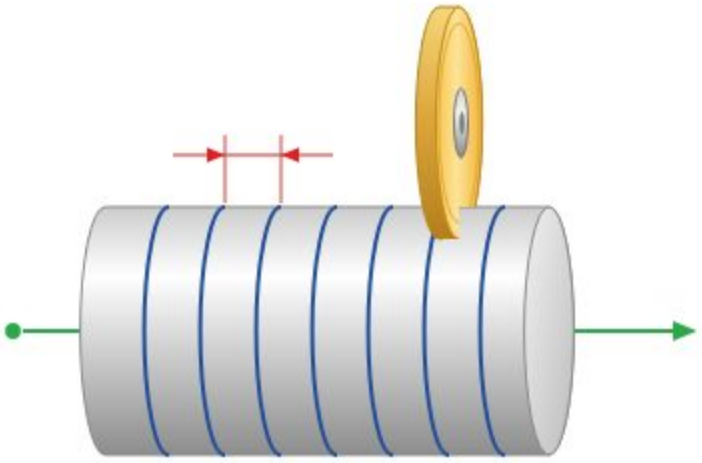
Rotary. Cuts are generated around the rotary axis and distributed along it.



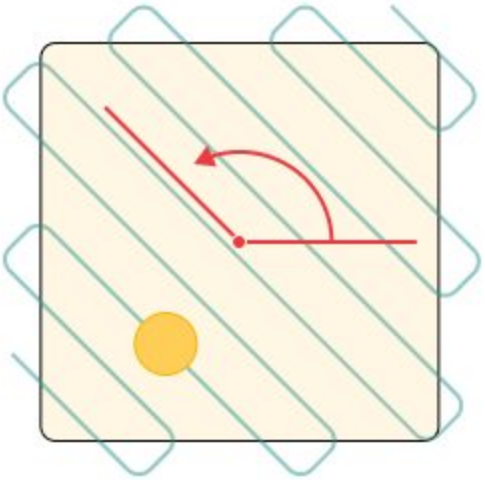
3d. Parallel cuts are generated in a plane.



Step. The distance between adjacent cuts (strings). Enter a distance or a percentage of the disc diameter; the default is 20%. If the axial zone is not an exact multiple of the step, the step is reduced to distribute the cuts evenly.

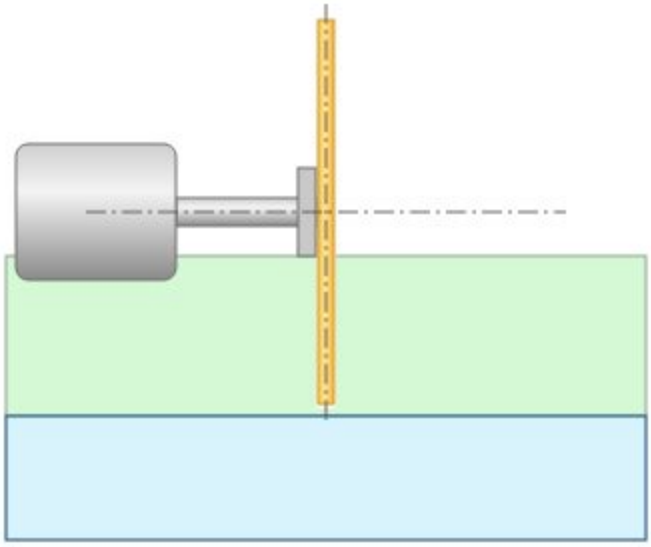


Angle. Sets the cut direction in the plane, measured from the X axis. Available only for 3d.

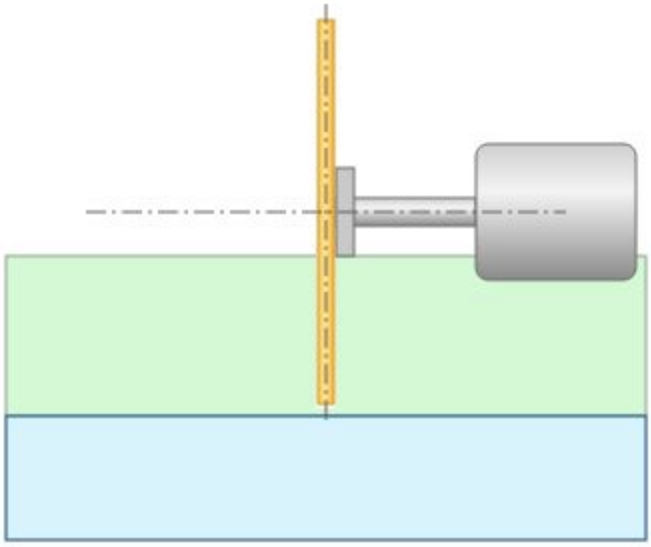


Change side. Reverses the tool axis and shifts the toolpath point by the disc thickness while preserving the cut position.

▼ Click here to expand...
OFF. The original tool position.



ON. The tool is positioned on the opposite side of the cut.

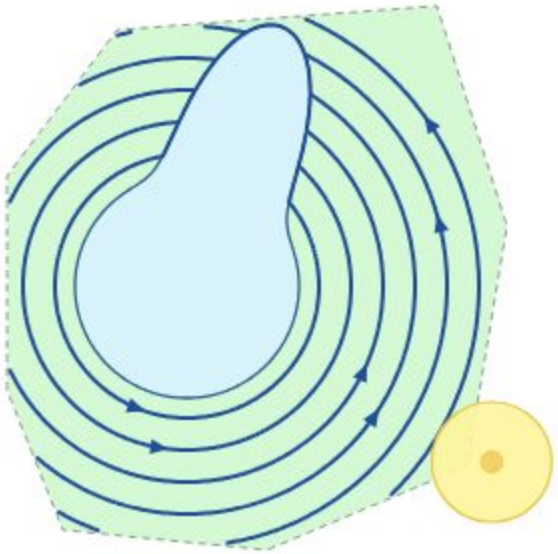


Roughing:

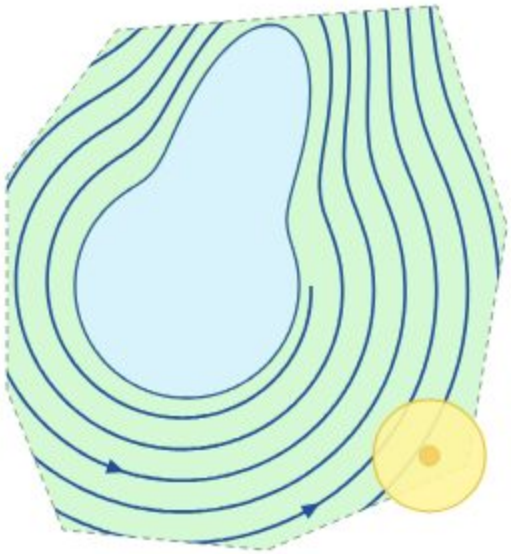
Defines how material is removed within each cut.

▼ Click here to expand..

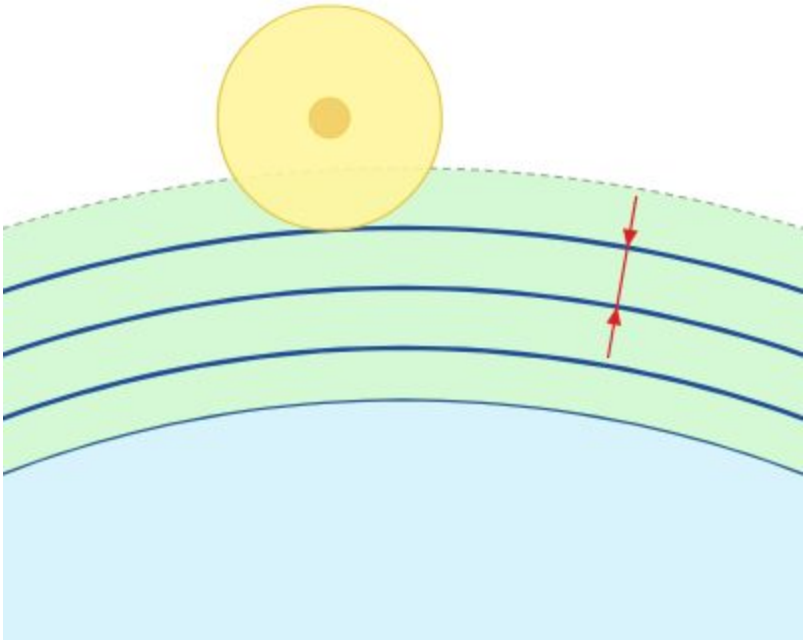
By layers. Machining uses separate layers at the specified depth of cut. The shape of the turns relative to the part and workpiece depends on Workpiece.



Adaptive. Material is removed along a spiral while limiting the width of cut. Rounding radii control the smoothness of roughing passes, links, and toolpath corners.



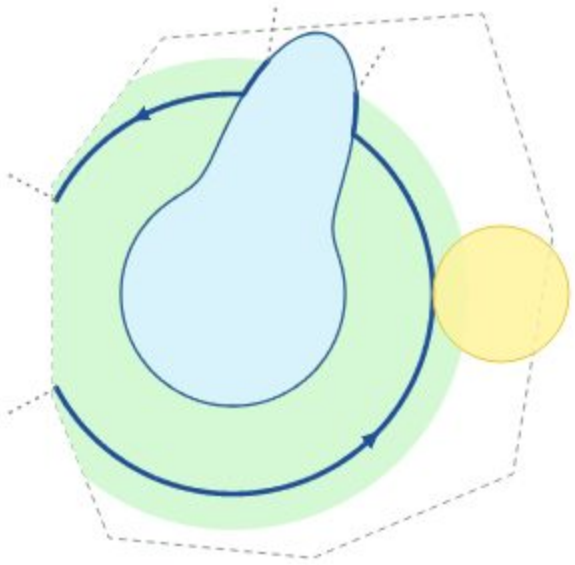
Depth of cut. The distance between adjacent layers within a cut. Enter a distance or a percentage of the disc diameter; the default is 100%.



Workpiece. A parameter within Roughing on the Strategy tab. It is available only when Roughing is set to By layers and determines how the turns of each layer are generated relative to the workpiece. With Adaptive, the parameter is hidden and the workpiece is always taken into account.

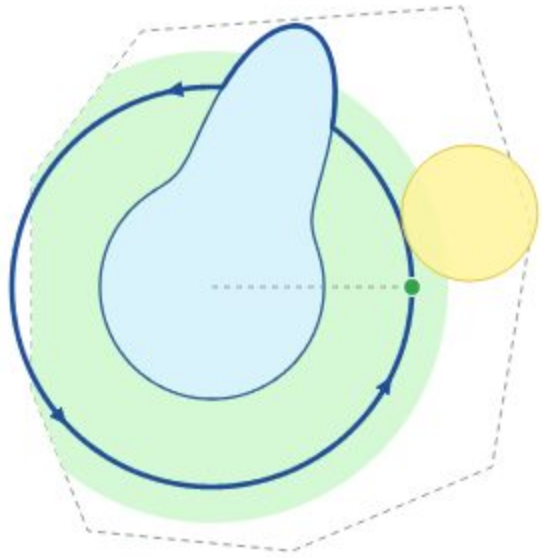
▼ Click here to expand...

- Use workpiece. The default mode. A turn is generated only where workpiece material is present. Where the part rises above the layer, the turn is interrupted. The resulting segments are connected by links with approaches and retracts.

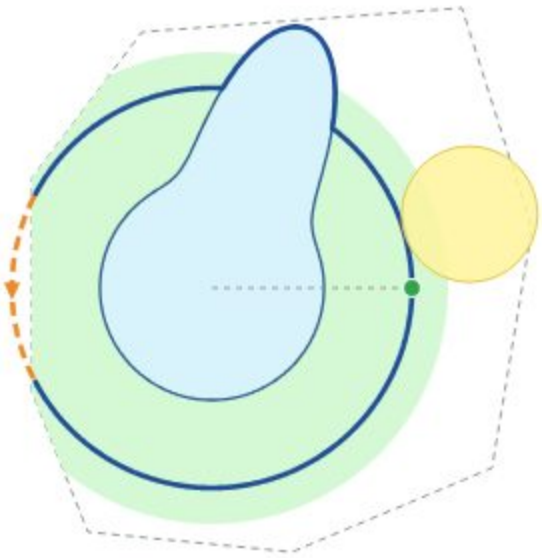


- Ignore workpiece. The workpiece and the tool cover are not taken into account when generating turns. Each turn is a continuous pass along its entire length: it follows the layer where the part is lower and the part surface where the part is higher. There are no breaks or retracts within a turn. The

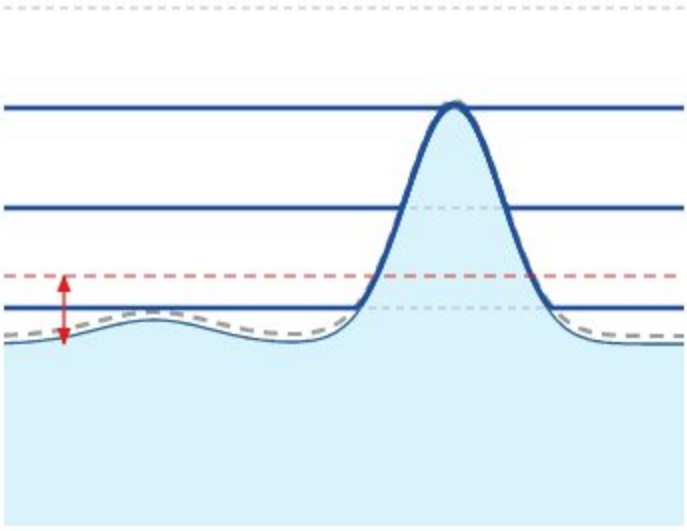
layers are limited only by the top machining level, rather than the top of the workpiece, so some turns may pass through empty space.



- Link feed outside. The turns are continuous, as in Ignore workpiece mode. Where no workpiece material is present, the tool moves at the short link feed; within the material, it moves at the working feed. Only the feed changes at the material boundary, without a retract. In the illustration, blue lines indicate the working feed and the orange dashed line indicates the short link feed.

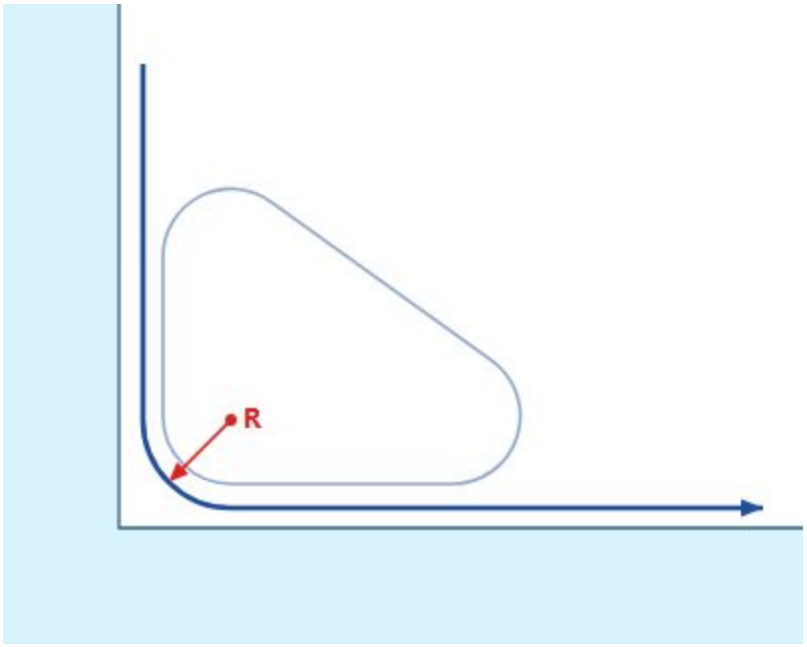


Minimum cut depth. A parameter within Roughing, available only when Workpiece is set to Ignore workpiece or Link feed outside. The default value is 0.1 mm. The cut depth of a turn is the distance from the top of its layer to the lowest point of the part along that turn. Turns with a cut depth below the specified value are not generated. This removes lower turns that follow the part surface almost entirely and remove very little material. A value of 0 allows all turns to be generated.

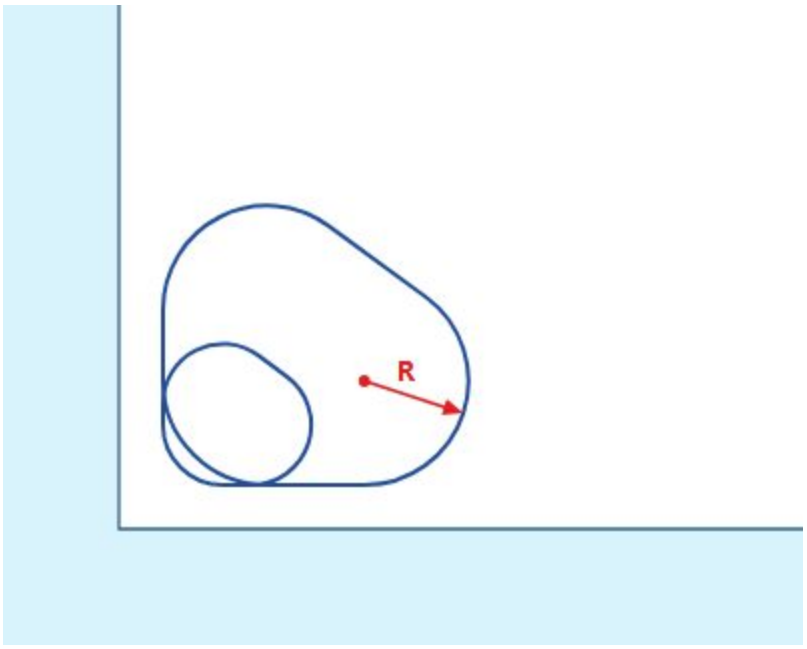


The following parameters are available only with Adaptive roughing.

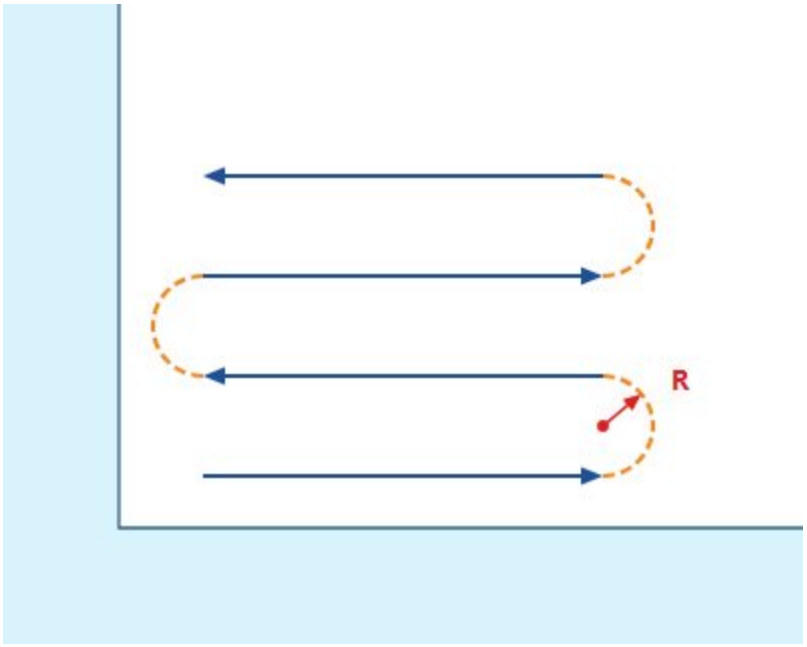
Corners smoothing. The radius used to smooth toolpath corners near the part surface.



Rough rounding radius. The rounding radius for roughing passes.



Linking radius. The radius of arcs connecting the passes. It must be at least the safe distance; the inspector marks an invalid value as an error.

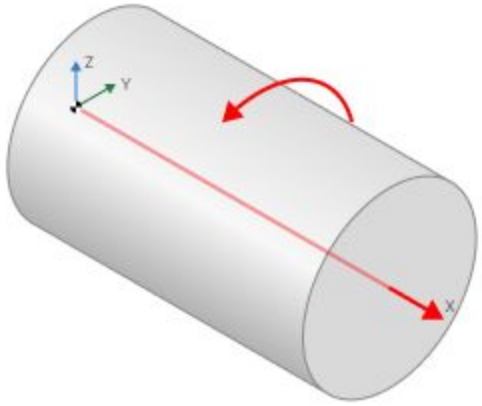


Job zone:

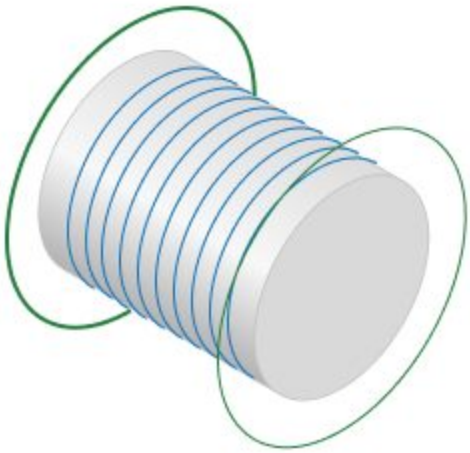
Limits machining along the rotary axis and within an angular range. Available only for Rotary machining.

▼ Click here to expand..

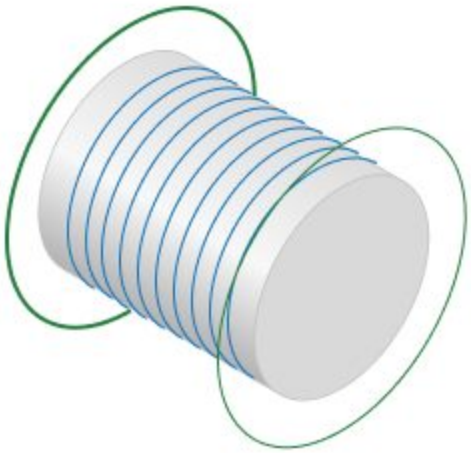
Rotary axis. Set the axis in the inspector or with the mouse. Orientation follows X, Y, or Z of the base coordinate system, or a custom direction; Origin defines the axis position. Base CS is Global CS or WCS.



Min. axial position. The lower axial limit: Auto or a value measured from the rotary axis origin.

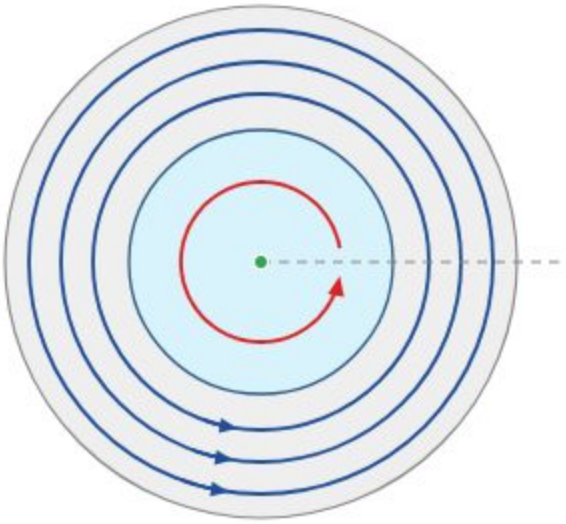


Max. axial position. The upper axial limit: Auto or a value measured from the rotary axis origin.

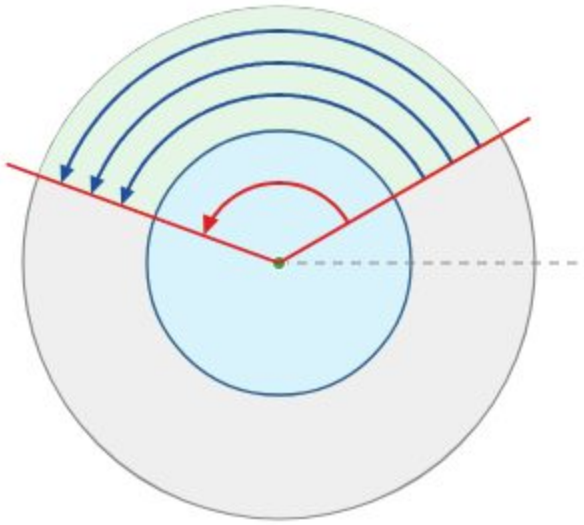


Angular domain. Selects a full revolution or a sector between the start and finish angles. Adjust the limits by dragging or entering a value beside the dimension line.

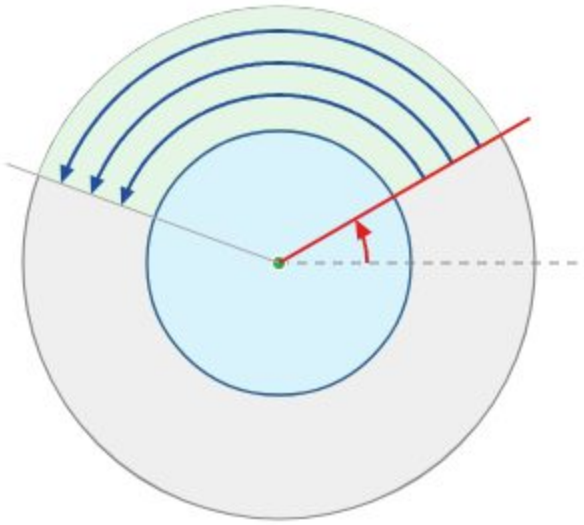
Full circle. Turns extend around the full circle.



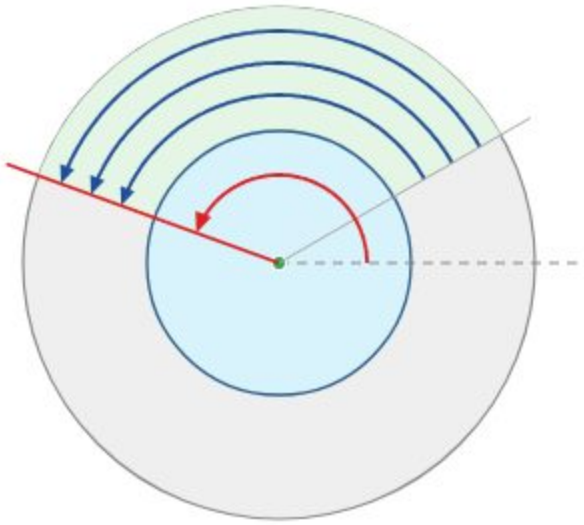
Sector. Turns are limited to the specified angular range.



Start angle. The start of the sector, measured from the zero-angle direction.



Finish angle. The end of the sector, measured from the zero-angle direction.

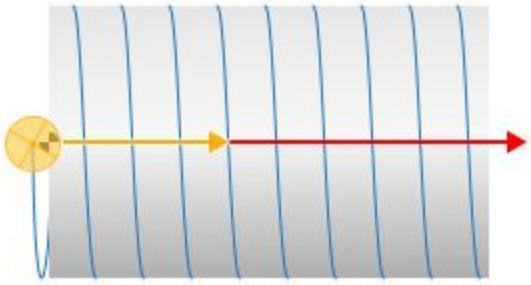


Axial direction:

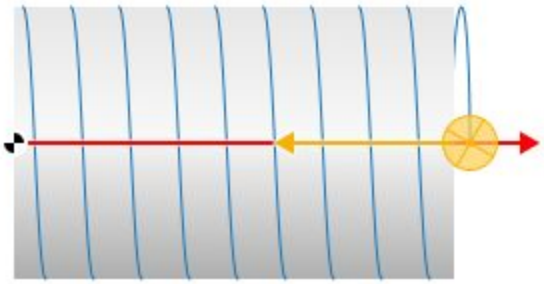
Defines the order of cuts along the axis. Available only for Rotary machining.

▼ Click here to expand..

Forward. From the minimum to the maximum axial position, in the rotary axis direction.



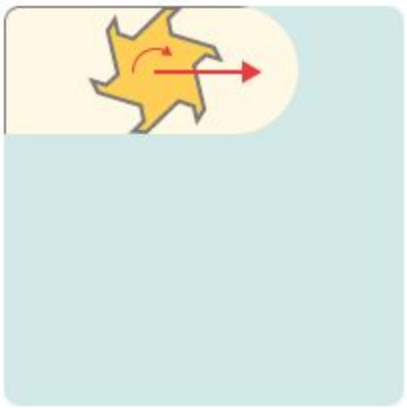
Backward. From the maximum to the minimum axial position.



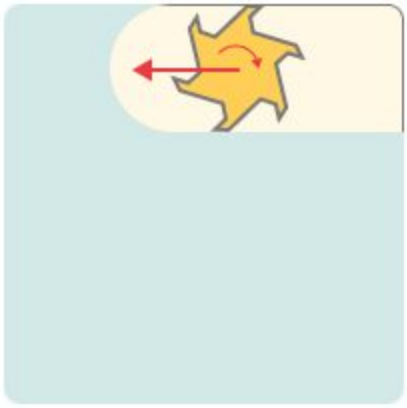
Milling type:

▼ Click here to expand..

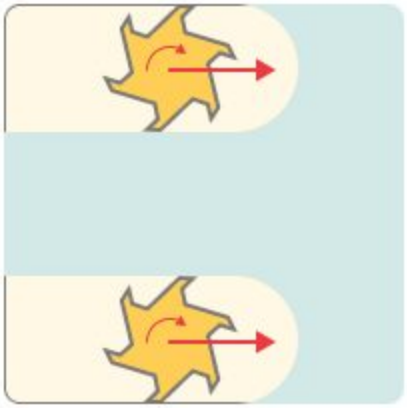
Climb. The feed follows the cutting edge motion at the contact zone.



Conventional. The feed opposes the cutting edge motion at the contact zone.



Both. Allows both machining directions.

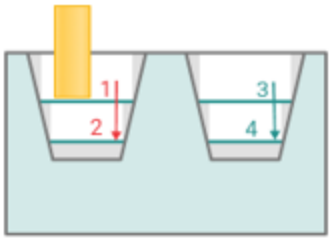


Sorting:

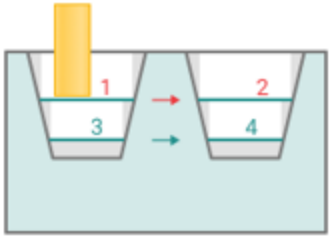
Toolpath uniting mode defines the order in which strings and layers are machined.

▼ Click here to expand..

By layers. Each cut is machined to its full depth before the next cut is started.



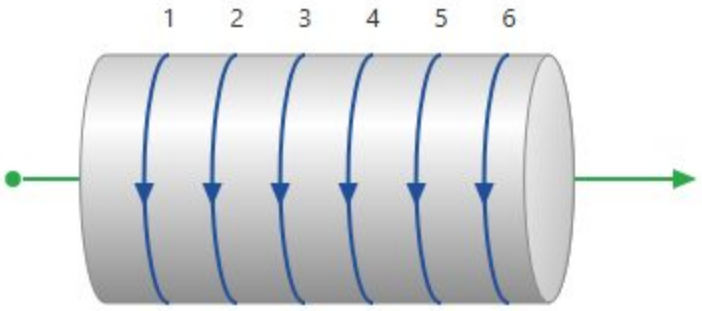
By strings. One layer is machined across all cuts before the next layer is started. Within a layer, machining proceeds region by region, completing one region before moving to the next.



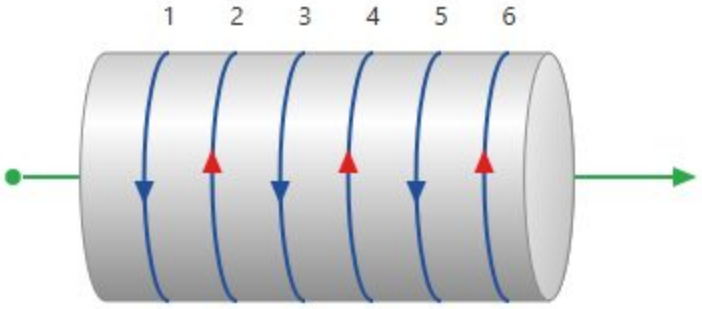
With Link feed outside, the layer is not split into regions: the turn remains continuous.

Reverse odd strings. Reverses the machining direction of every second cut: the 2nd, 4th, 6th, and so on.

OFF. Adjacent strings follow the same direction.



ON. Adjacent strings alternate direction.

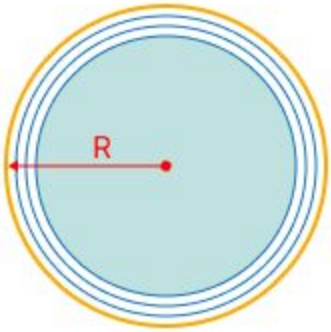


Machining levels:

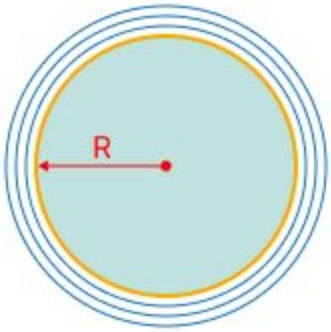
For Rotary machining, levels are distances from the rotary axis, not Z heights.

▼ Click here to expand..

Top level. The upper limit of the machining layers. Available only with By layers roughing.



Bottom level. The lower machining limit; no turns are generated below it.



Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

This tab defines tool approach and return, safe motions, and links between working passes.

▼ Click here to expand..
Consists of:

Approach/Return. Moves from the tool change position to machining and back. Details

Safe motions. The safety surface and safe level for tool movements. Details

Advanced links settings. Rotary axis motion settings for links. Details

Links. Rules for entry and exit links and links between levels and strings. Details

Distances. Distances used by linking rules to switch feeds. Details

Feeds/Speeds:

Sets the spindle speed and feeds for toolpath segments. First pass feed and Return feed are unavailable in this operation. Short link feed is used outside the material with Link feed outside. Details

Transformations:

Coordinate transformations applied to the calculated toolpath. Details

Part:

The part model defines the geometry to machine and is checked for gouges. Details

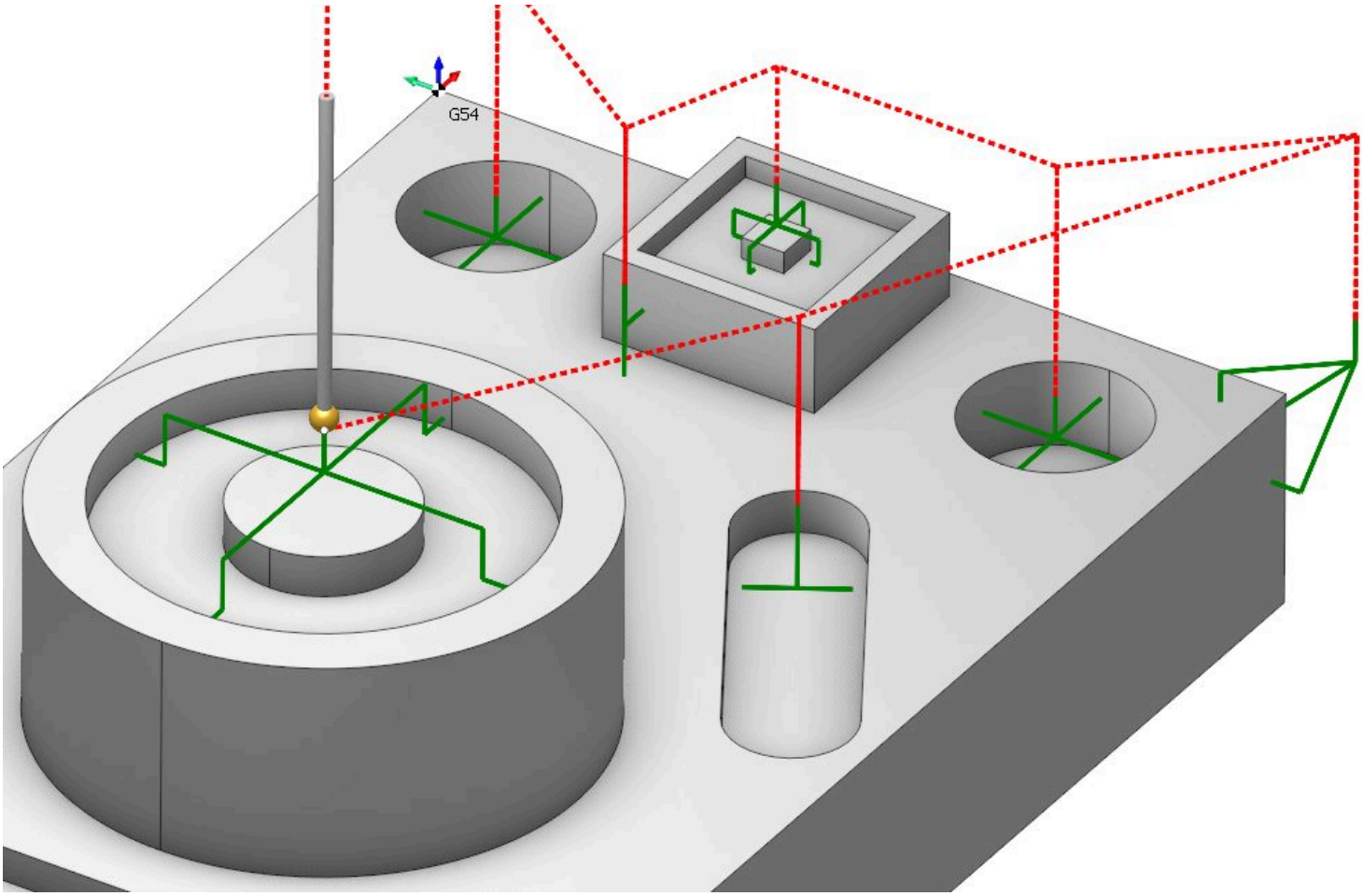
Workpiece:

The model of the initial material to machine. Details

Fixtures:

Clamping devices and other objects to account for during machining. Details

Probing



Added a new group of operations - Probing. Measuring of parts and tools using special measuring equipment and canned cycles.

Measuring cycles can be used on lathe machines, milling machines and robots. Measuring cycles allow you to determine the integrity of the tool, dimensions of parts, angles and machining elements of part. Due to this, you can get more correct machining and avoid errors.

Usually you need special devices for measuring, such as probes, CMM (coordinate measuring machine).

The main purpose of these cycles are listed below:

- control of the dimensions of critical surfaces of parts with output to the report;
- measurement of a part to compensate for inaccuracies in the location of the part and the geometric dimensions of real parts;
- measurement to detect inaccuracies in tool sizes;
- tool breakage control.

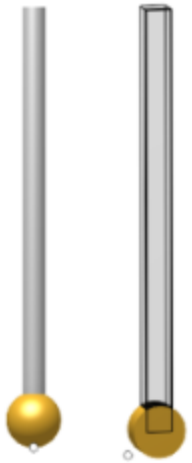
The system has special operations for creating a toolpath based on measuring cycles:

- Mill part probing;

- Mill tool probing;
- Turn part probing;
- Turn tool probing.

```
These operations are practically the same. It only overrides the default values for
some parameters depending on the purpose of the operation (for example, a tool).
```

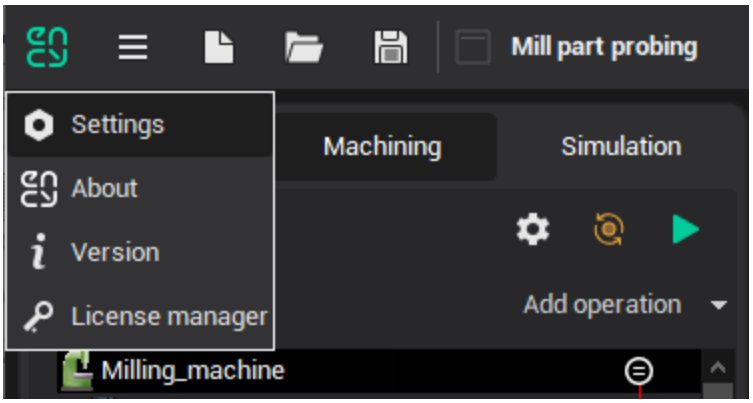
New group of Probing tools added: one tool for the axial case (tooling point at the axis) another for the turn case (tooling points can be at side).

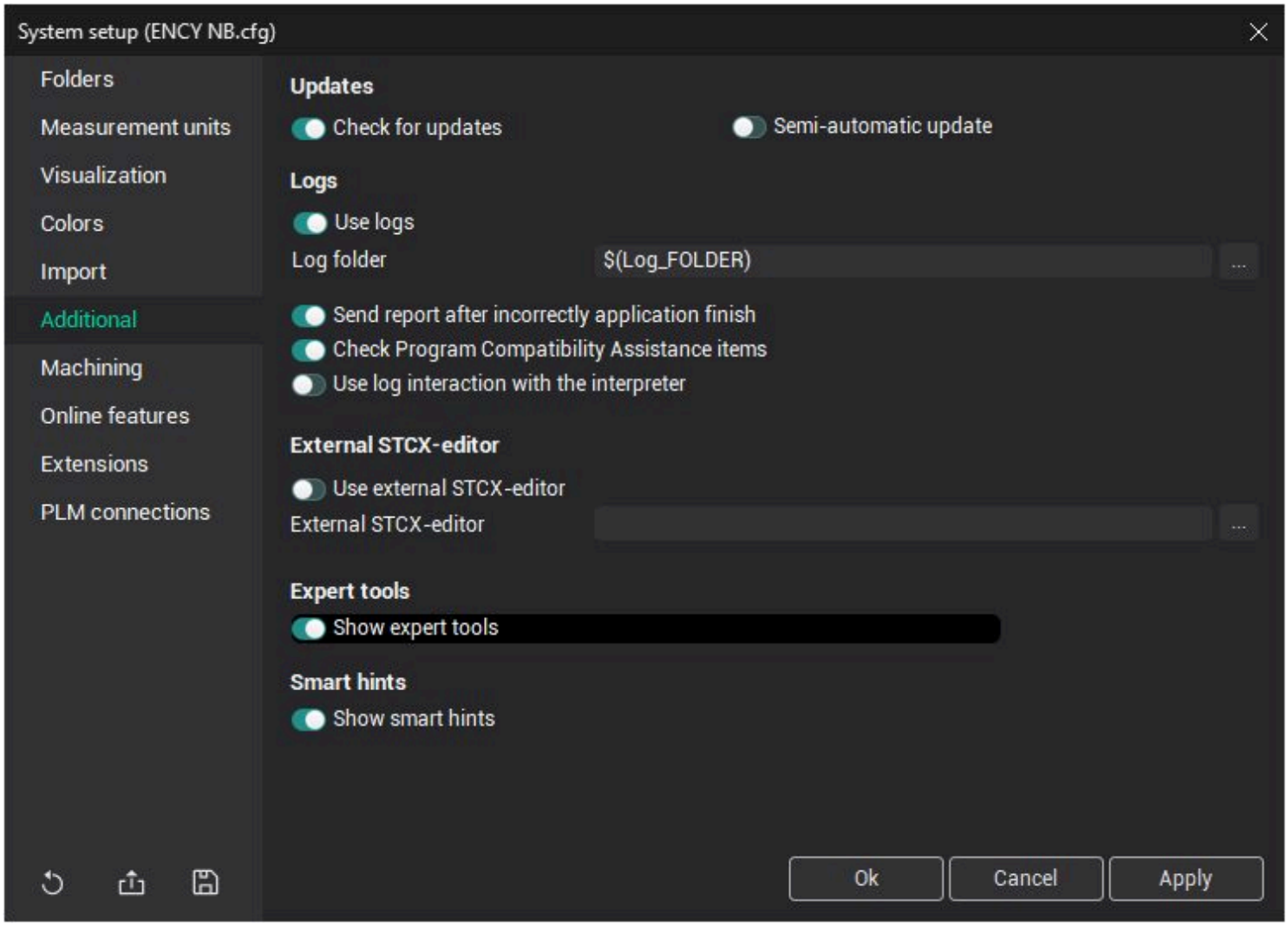


Creating own probing cycles (templates)

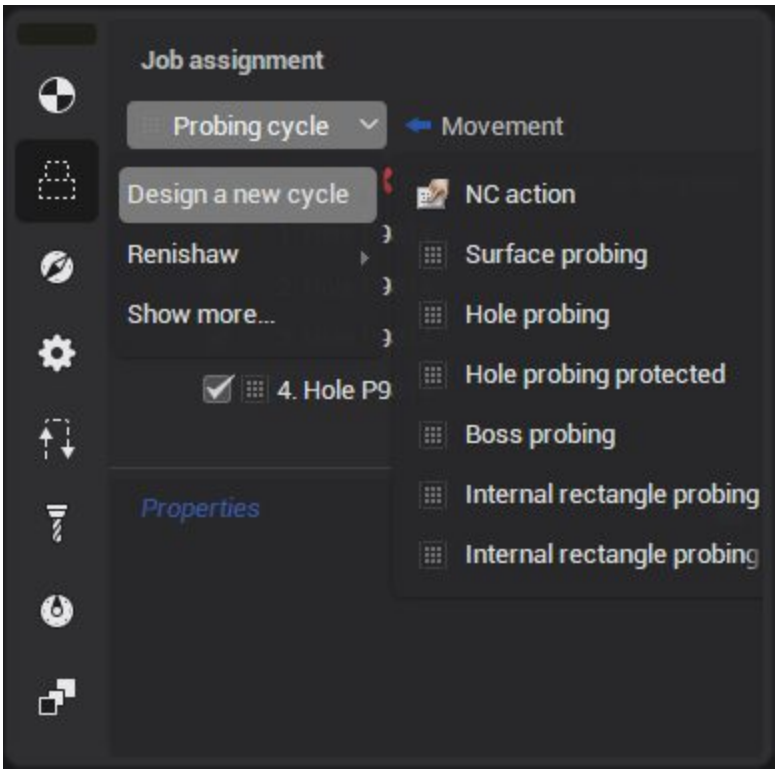
You can create your own probing cycles with individual properties for future use. New probing cycles may be united in library which is a separate file with the scpbl extension. This library can be shared with other users.

At first you need to activate experts rules. To do this, you need to open Settings. Then switch Additional tab and in the options that appear, enable Show expert tools and click on Ok. After that empty cycles will be available.



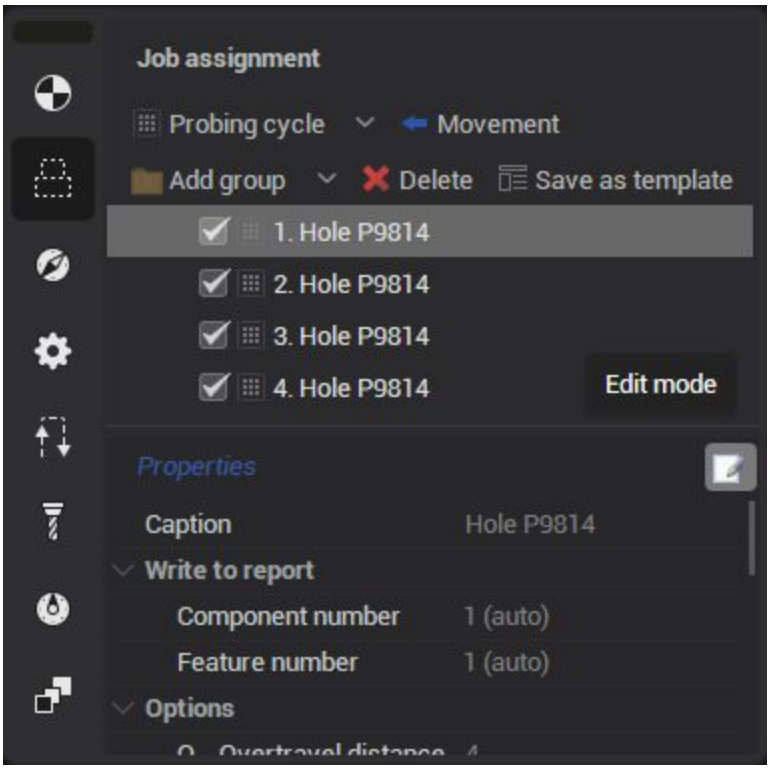


You can add one of empty cycles. Click Probing cycles in Job assignment tab of selected cycle. You find list of available cycles in Design a new cycle sub menu. There are 15 cycles and 1 additional element NC action. More information about this cycles is here.



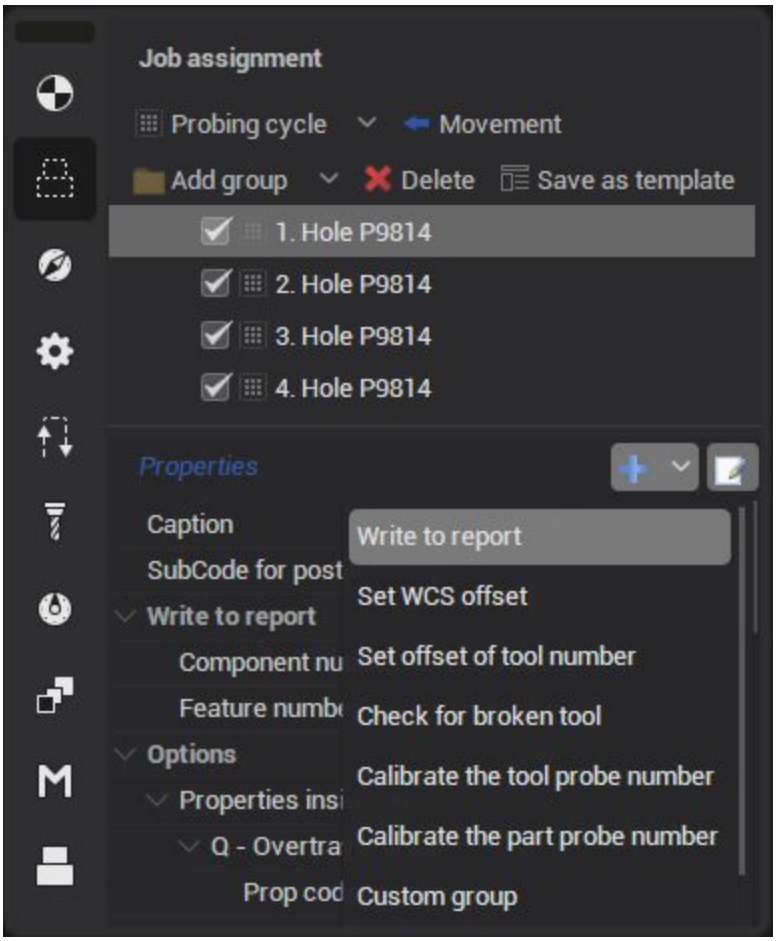
A fter adding the required cycle it is not necessary to set its geometric parameters, since these parameters will need to be set when using this cycle in the future. The main thing is to specify its properties that will be stored in the created template based on this cycle.

To add properties click on the Edit button with the pencil icon. After that the SubCode for postprocessor field and Add custom property button with the plus icon will be appeared. The SubCode for postprocessor field is t he unique cycle code. Used in the postprocessor to uniquely identify the cycle and parse its parameters. See also about it in postprocessor documentation "Probing cycle WProbing".



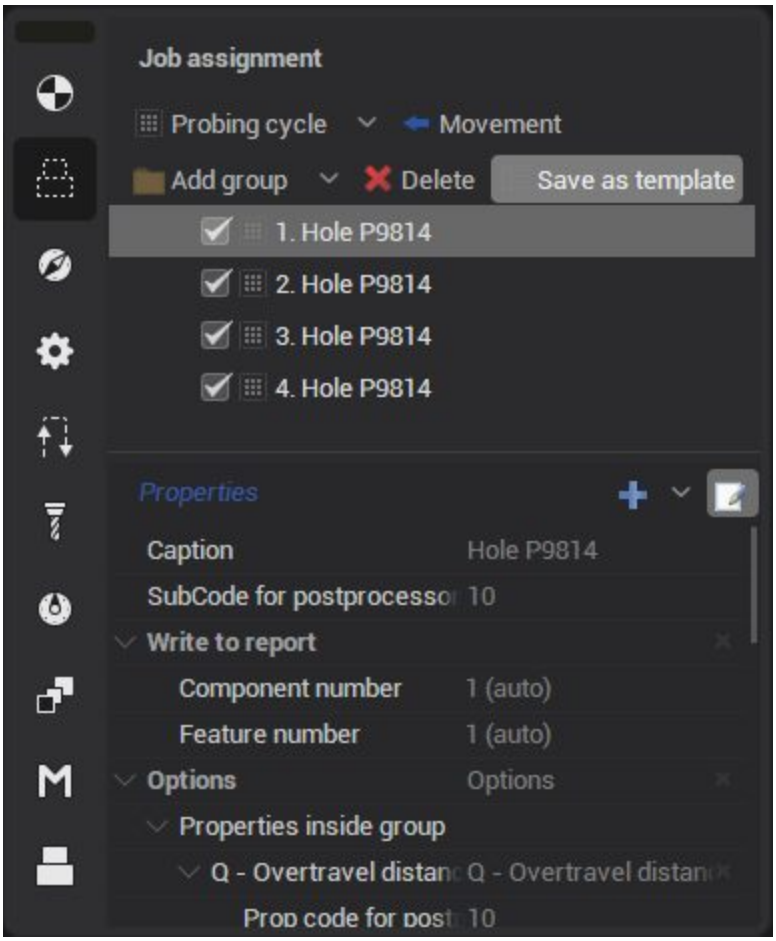
Click on Add custom property to add property from dropdown list. There are 8 properties with own parameters in list:

- Write to report - entering measured values in a report line. There are two parameters: Component number and Feature number. These parameters are auto-incrementing but you can also enter values manually;
- Set WCS offset - set workpiece CS to zero. It can be set in different ways (globally shift all CS, shift one of G54-G59, shift via parameters or via local CS). You can choose one of four options: Global (all G54-G59), One WCS (one of G54-G59), Parametrical offset (G10 L2 P1 XYZ), Local CS offset (G52 XYZ). The CS number parameter available for One WCS (one of G54-G59) and Parametrical offset (G10 L2 P1 XYZ). Here you can specify the required CS;
- Set offset of tool number - set tool zero. You need the number of the tool and corrector in which to write the offsets;
- Check for broken tool - check the deviation of the tool dimensions from the reference ones and generate an error if it is greater than the limits. You also need the number of the tool and corrector where to get the dimensions for comparison;
- Calibrate the tool probe number - calibrate the tool probe. You need the number of the probe and its corrector;
- Calibrate the part probe number - c alibrate the part probe. You need the number of the probe and its corrector;
- Custom group - creates an element to group custom properties;
- Custom property - custom property to add an additional parameter. You can set: Prop code for postprocessor, Property type (double, integer, Boolean. string) and Value.

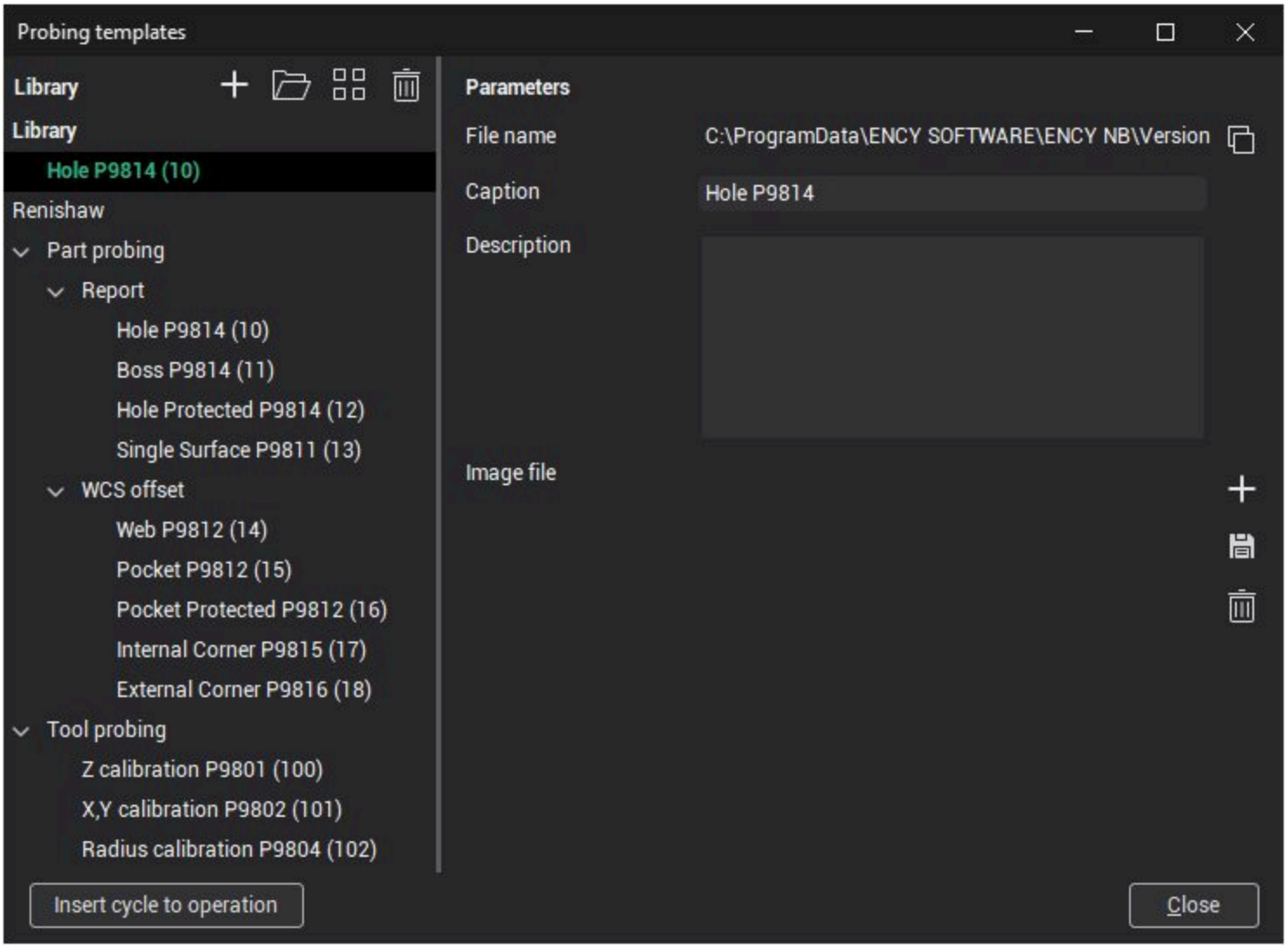


Access codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Additional parameters".

A fter you have prepared the cycle template, you need to add it to the library. Click on Save as template.



The Probing templates window will open. A new template will be added to this window. If custom libraries were created earlier, then the template will be located in this library, otherwise a new library will be created. Changes in this window are saved when the window is closed. More information about working with The Probing templates window here.


Probing templates
Library
Library
Hole P9814 (10)
Renishaw
Part probing
Report
Hole P9814 (10)
Boss P9814 (11)
Hole Protected P9814 (12)
Single Surface P9811 (13)
WCS offset
Web P9812 (14)
Pocket P9812 (15)
Pocket Protected P9812 (16)
Internal Corner P9815 (17)
External Corner P9816 (18)
Tool probing
Z calibration P9801 (100)
X,Y calibration P9802 (101)
Radius calibration P9804 (102)
Parameters
File name
C:\ProgramData\ENCY SOFTWARE\ENCY NB\Version
Caption
Hole P9814
Description
Image file
Insert cycle to operation
Close

Types of measuring cycles

```
There are several probing cycles with geometry:
```

- Surface probing;
- Hole probing;
- Hole protected probing;
- Boss probing;
- Internal rectangle probing;
- Internal rectangle probing protected;
- External rectangle probing;
- Web probing;
- Web probing (three points);
- Groove probing;
- Groove probing protected;
- Double wall internal corner probing;
- Double wall external corner probing;
- Triple wall internal corner probing;
- Triple wall external corner probing.

```
There are also additional elements of working with cycles:
```

- NC action;
- Movement;
- Elements to group: Component, Feature, Group.

Cycles with geometry

All cycles with geometry have some common parameters:

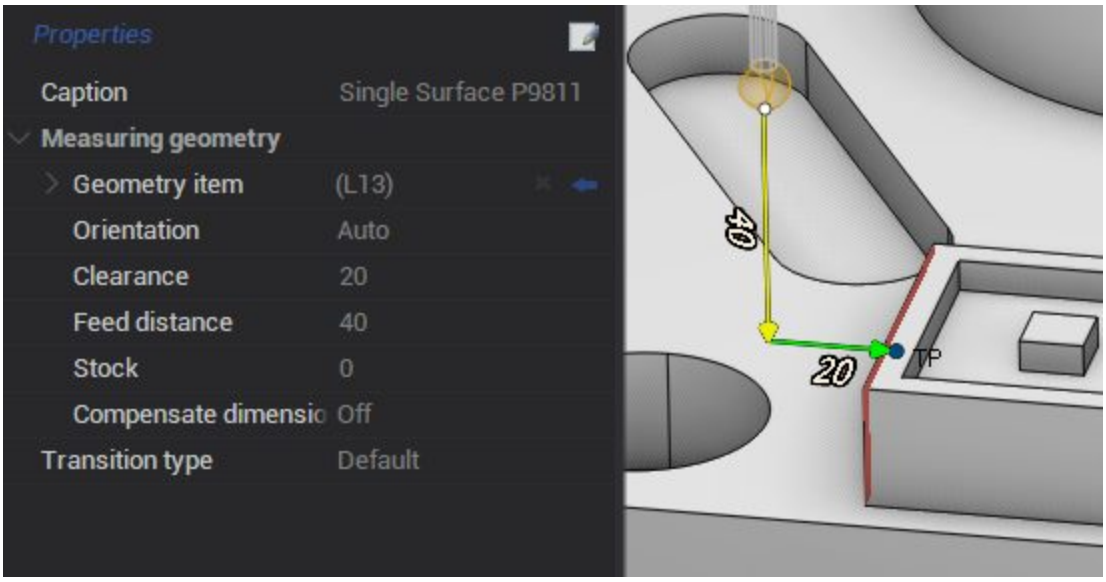
- Orientation - setting start orientation of tool in cycle. After adding cycle the orientation is set by Auto. You can switch on Manual and set orientation manually;
- Feed distance - distance of approach to start position of cycle and return from last position of cycle;
- Stock - additional shift of the target point along the target vector;
- Transition type - setting an individual transition for current cycle. Transition values are described here;
- Compensate dimensions - toolpath shift with tool compensation. This parameter is used in cycles where tool compensation is possible

You can set name of selected cycle in Caption field f or every cycles.

Notes: there is an additional Cycle variant parameter for turn probing operations. Cycle variant has two values: Spindle on and Spindle off. If Spindle on is selected then the spindle rotate before each approach to the touch point. Notes: there is no Cycle variant in surface probing. Hole, hole protected and boss probing has own Cycle variant.

Surface probing

Probing cycle using one surface surface.



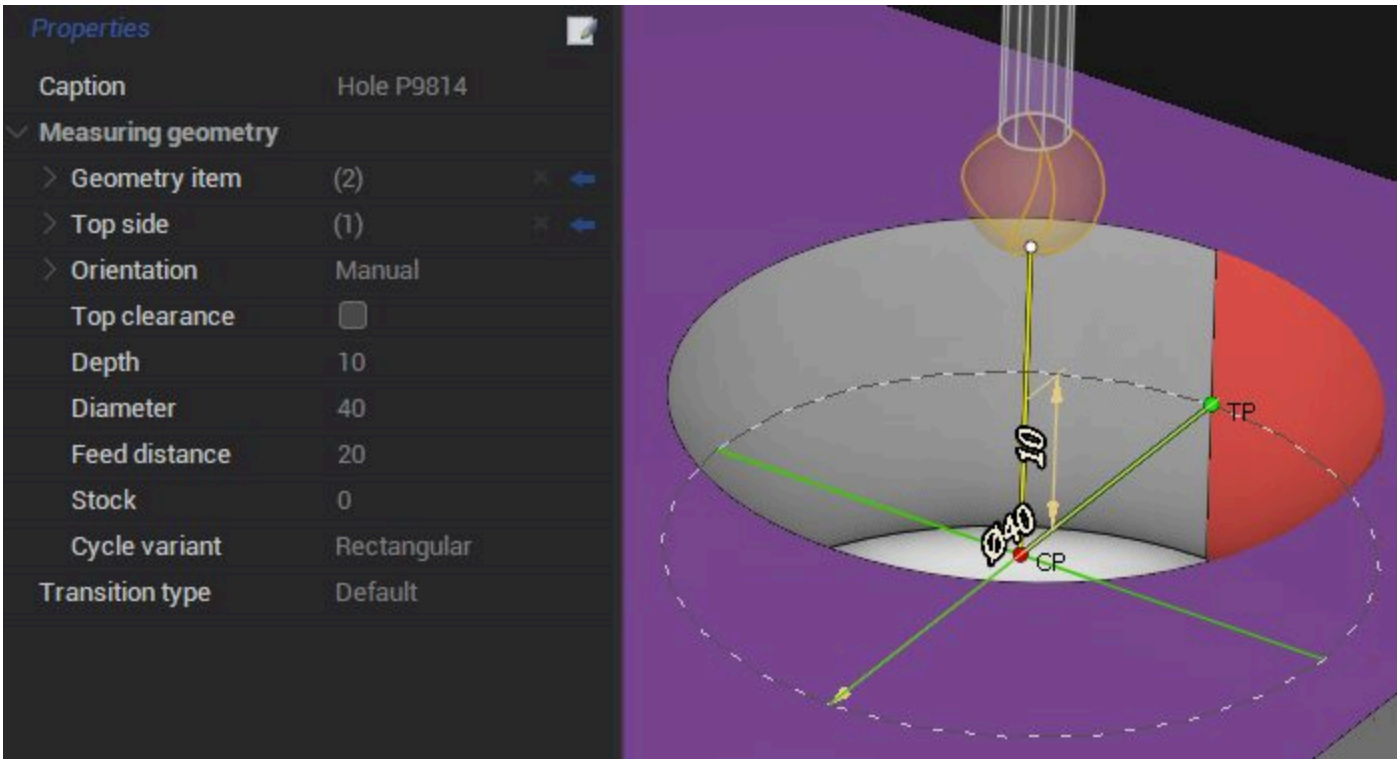
Parameters of surface probing:

- Geometry item - measured surface. You can edit position and vector of the point manually. You need to expand Geometry item and set values in the corresponding field s: Target point, Target vector;
- Clearance - distance between approach point and touch point.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Surface probing parameters".

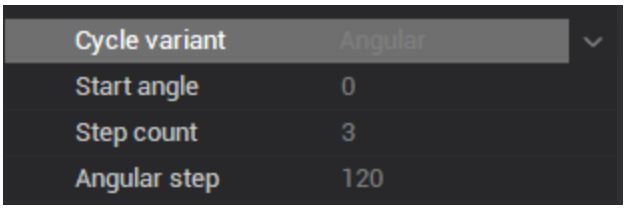
Hole probing

Probing cycle designed for measuring bore.



Parameters of hole probing:

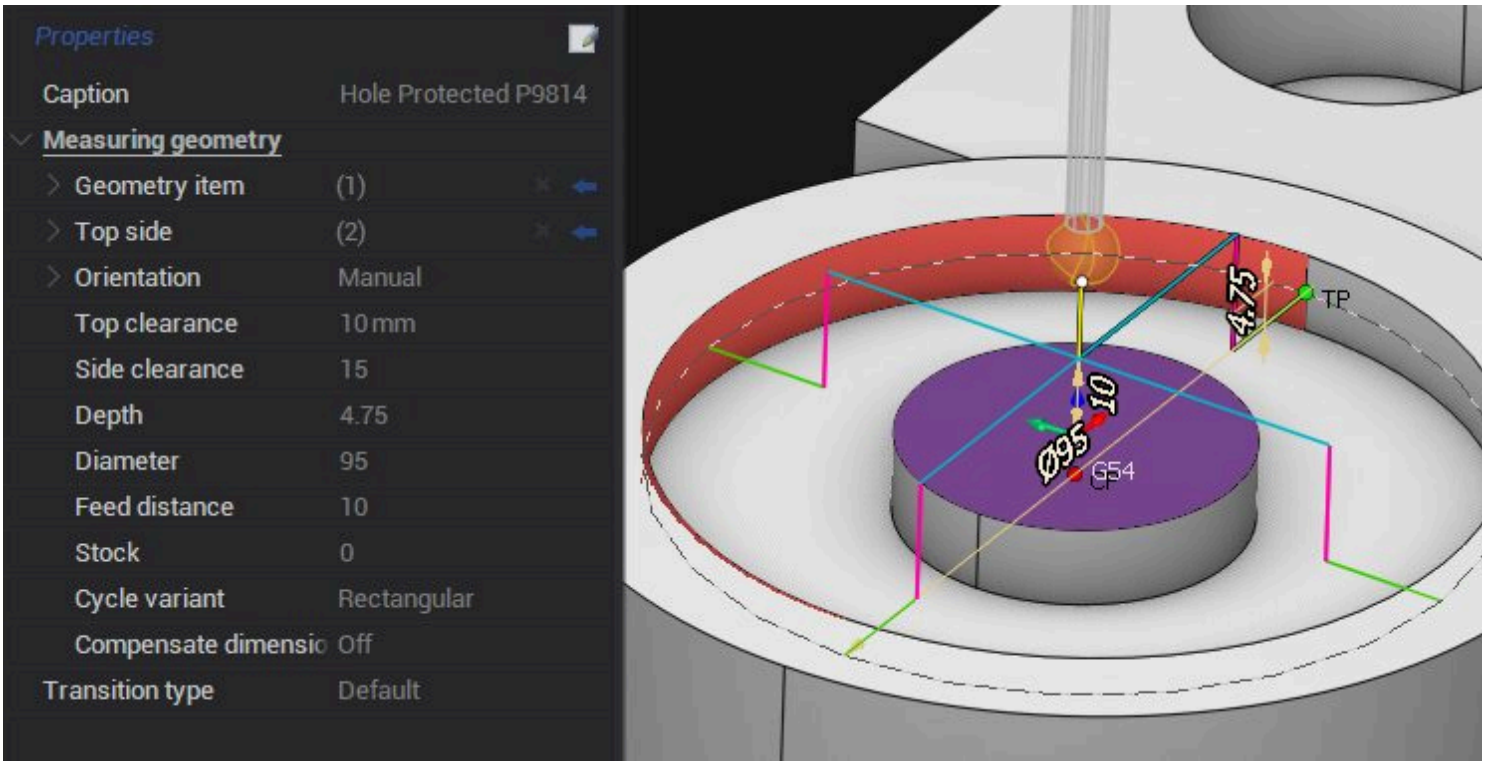
- Geometry item - measured surface of hole. Y ou can edit position, vector and center of the point manually. Y ou need to expand Geometry item and set values in the corresponding fields: Target point, Target vector, Center point;
- Top side - determination of the top level from which we count the distance to the starting point of the cycle;
- Top clearance - i f the parameter is checked then you set the distance from Top side to the start point of the cycle. If the parameter is unchecked then the start point is located at the Center point;
- Depth - distance from Top side to Target point in the direction of the hole;
- Diameter - measuring hole diameter;
- Cycle variant - the parameter has two values: Rectangular and Angular.
    - Rectangular mode creates 4 touch points with an angle of 90 degrees. Bypassing the touch points is implemented as follows: the first touch point is specified in the Target point, the second touch point is the opposite from the first. The third touch point is a point rotated 90 degrees from the first. The fourth point is the opposite of the third point. Notes: The spindle turns before approaching the third point in turn probing operations;
    - Angular has own parameters. Start angle allow to rotate Target point to specified angle. Angular step is distance between touch points. Step count allows you to set count of touch points. Bypassing the touch points is implemented as follows: the first touch point is specified in the Target point, then the points are bypassed along the specified Angular step. Notes: T he spindle turns before approaching each touch point.



A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Hole probing parameters".

Hole probing protected

Probing cycle designed to measure bore inside which there is an obstacle.



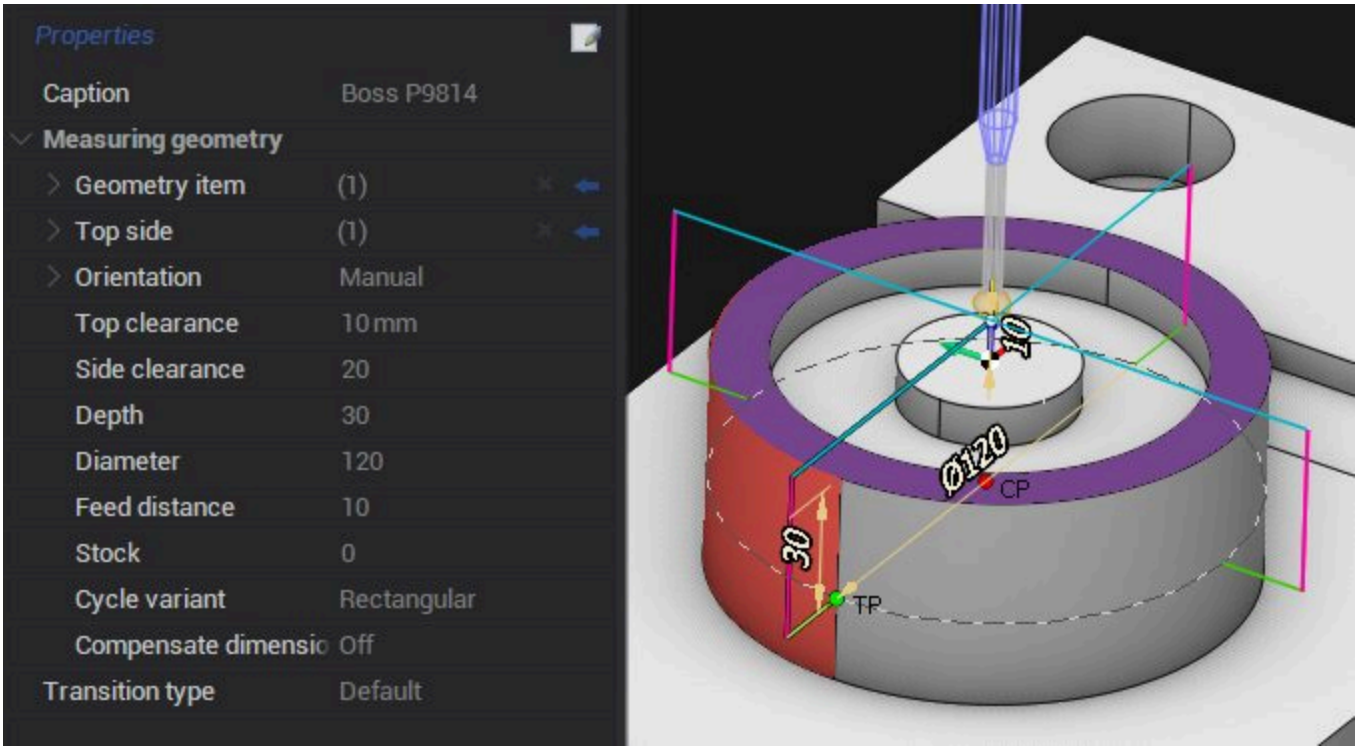
Parameters of hole probing protected:

- Geometry item - measured surface of hole with obstacle. Y ou can edit position, vector and center of the point manually. Y ou need to expand Geometry item and set values in the corresponding fields: Target point, Target vector, Center point;
- Top side - determination of the top level from which we count the distance to the starting point of the cycle;
- Top clearance - distance from Top side to the start point of the cycle;
- Side clearance - distance between approach point and touch point;
- Depth - distance from Top side to Target point in the direction of the hole;
- Diameter - measuring hole diameter;
- Cycle variant - exactly the same as cycle variant in hole probing.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Hole probing protected parameters".

Boss probing

Probing cycle for measuring cylindrical bosses.

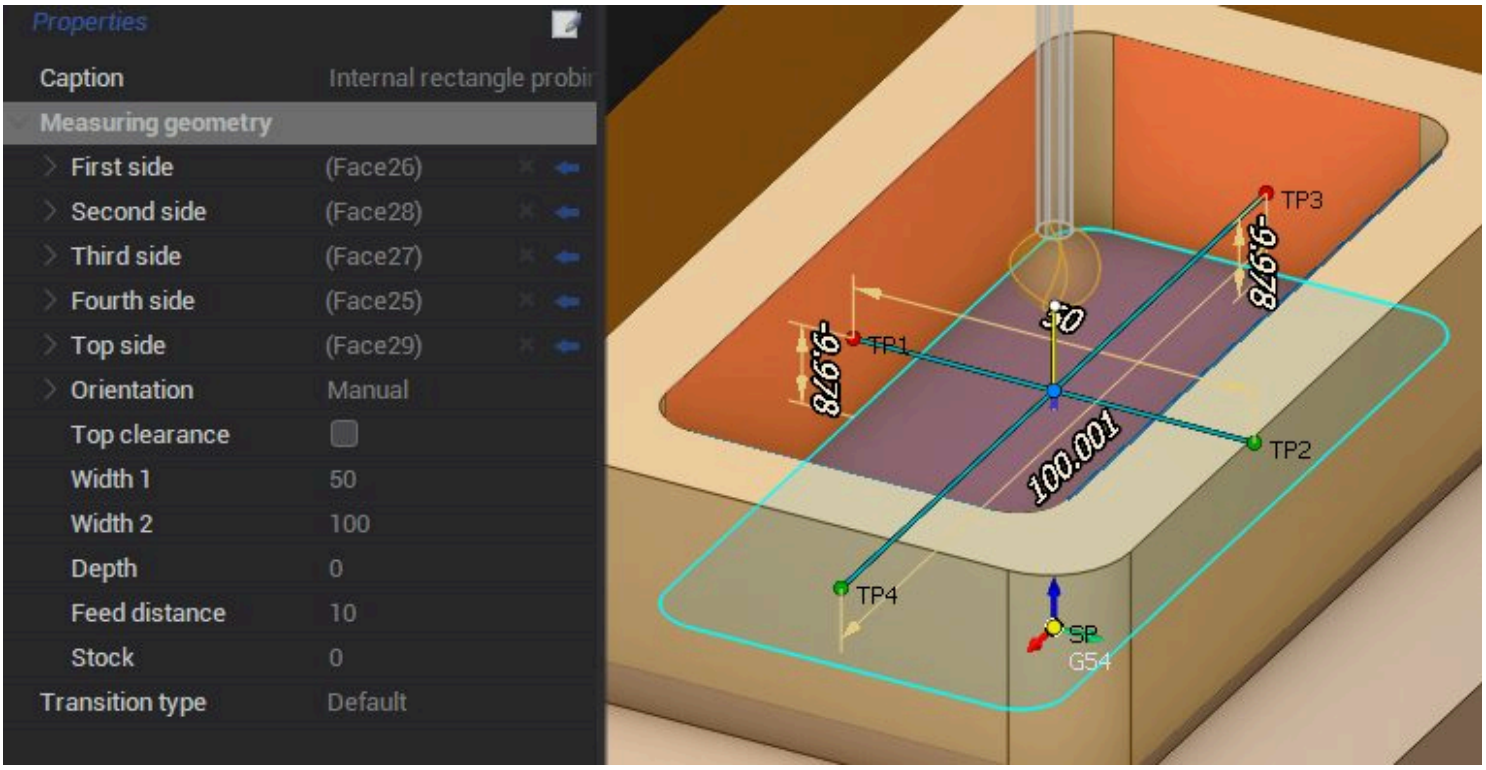


Parameters of boss probing are the same as in hole probing protected. T he only difference is that Geometry item is the boss, external surface.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Boss probing parameters".

Internal rectangle probing

Probing cycle for measuring grooves and other recesses on four sides.



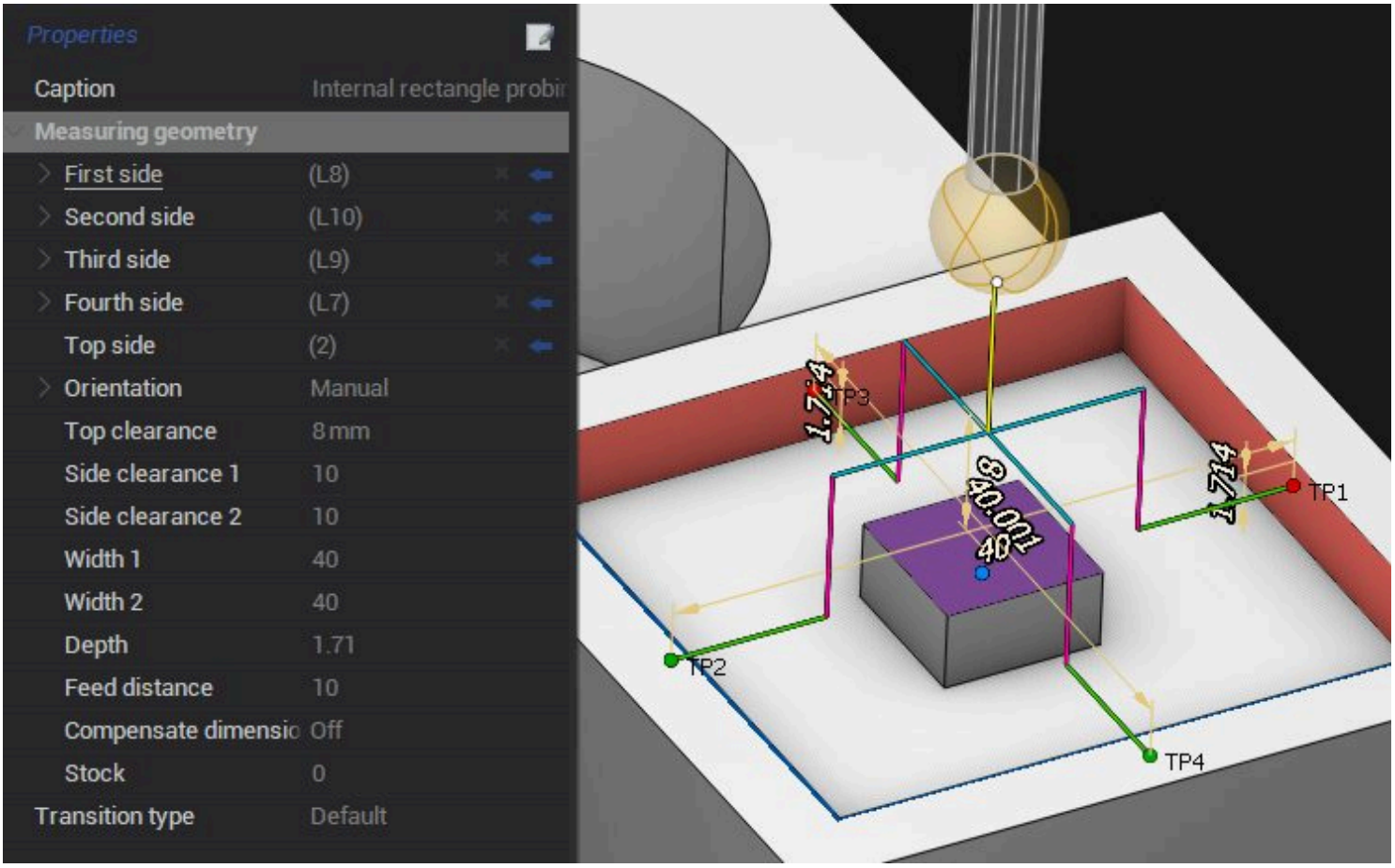
Parameters of internal rectangle probing:

- First side - the first touch point. Y ou can edit position and vector of the point manually. Y ou need to expand First side and set values in the corresponding fields: Target point 1, Target vector 1;
- Second side - the second touch point opposite the first point. Y ou can edit position and vector of the point manually. Y ou need to expand Second side and set values in the corresponding fields: Target point 2, Target vector 2;
- Third side - the third touch point. Y ou can edit position and vector of the point manually. Y ou need to expand Third side and set values in the corresponding fields: Target point 1, Target vector 1;
- Fourth side - the fourth touch point opposite the third point. Y ou can edit position and vector of the point manually. Y ou need to expand Fourth side and set values in the corresponding fields: Target point 2, Target vector 2;
- Top side - determination of the top level from which we count the distance to the starting point of the cycle;
- Top clearance - i f the parameter is checked then you set the distance from Top side to the start point of the cycle. If the parameter is unchecked then the start point is located at the intersection of four points;
- Width 1 - distance between First side and Second side;
- Width 2 - distance between Third side and Fourth side;
- Depth - distance from Top side to points position.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Internal rectangle probing parameters".

Internal rectangle probing protected

Probing cycle for measuring grooves and other recesses on four sides inside which there is an obstacle.



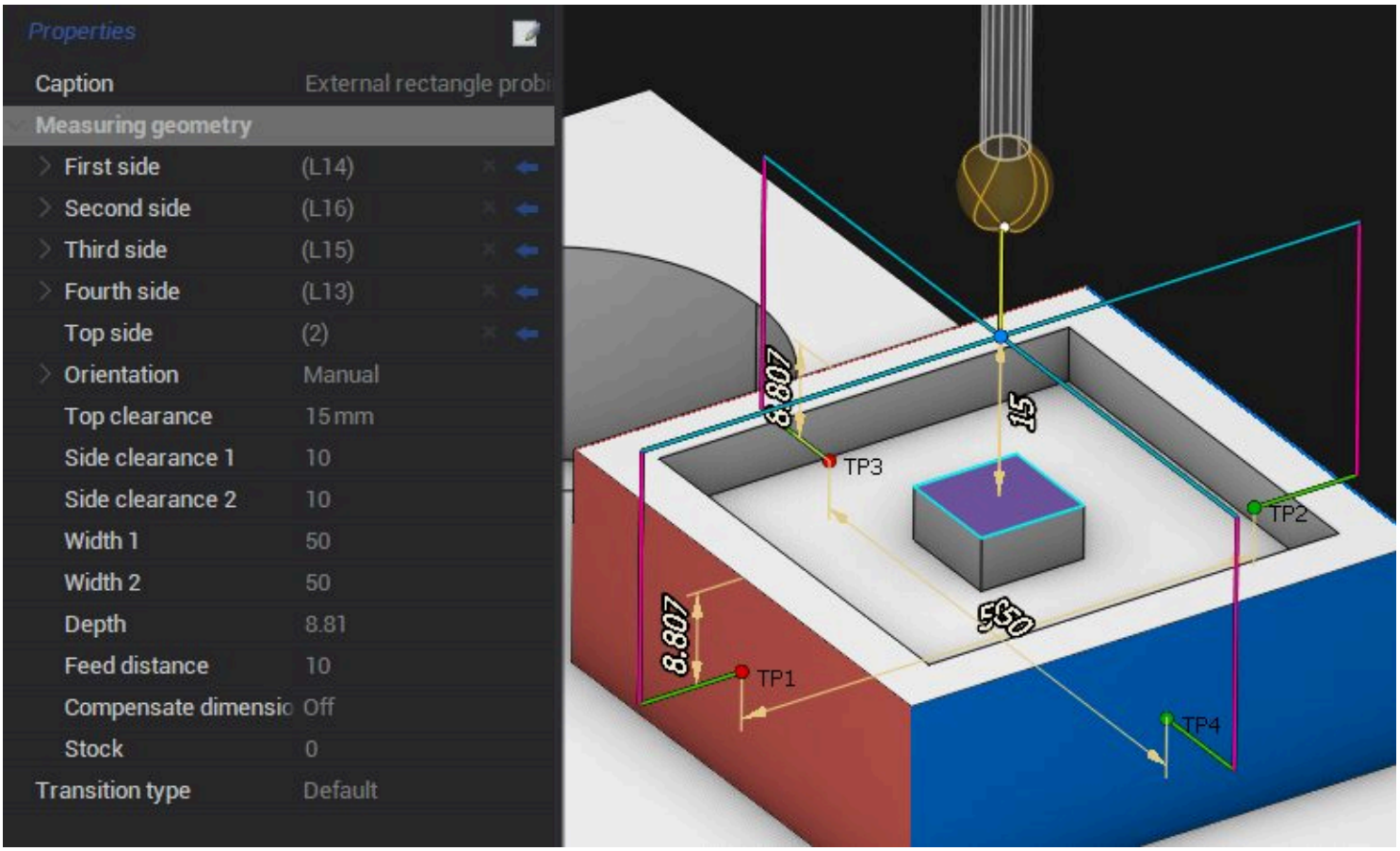
Parameters of this probing are the same as in internal rectangle probing. But it has two own parameters to avoid obstacles (Side clearance 1, Side clearance 2) and one parameter is changed (Top clearance):

- Side clearance 1 - distance between approach point and First side and distance between approach point and Second side;
- Side clearance 2 - distance between approach point and Third side and distance between approach point and Fourth side;
- Top clearance - distance from Top side to the start point of the cycle.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Internal rectangle probing protected parameters".

External rectangle probing

Probing cycle for measuring projections or dimensions on four sides.

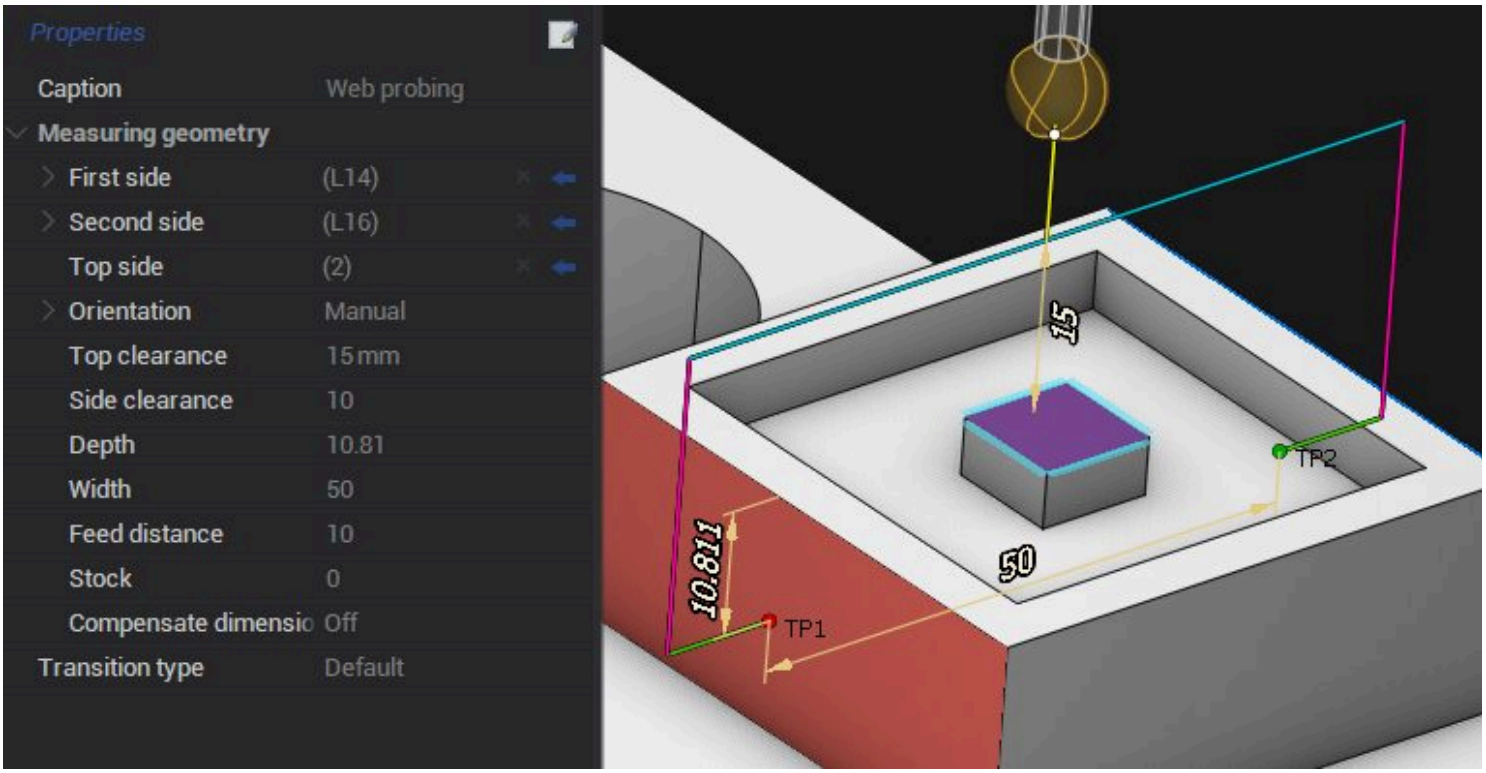


Parameters of extarnal rectangle probing are the same as in internal rectangle probing protected. T he only difference is that touch sides are external surfaces.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - External rectangle probing parameters".

Web probing

Probing cycle for measuring projections or dimensions on two sides.



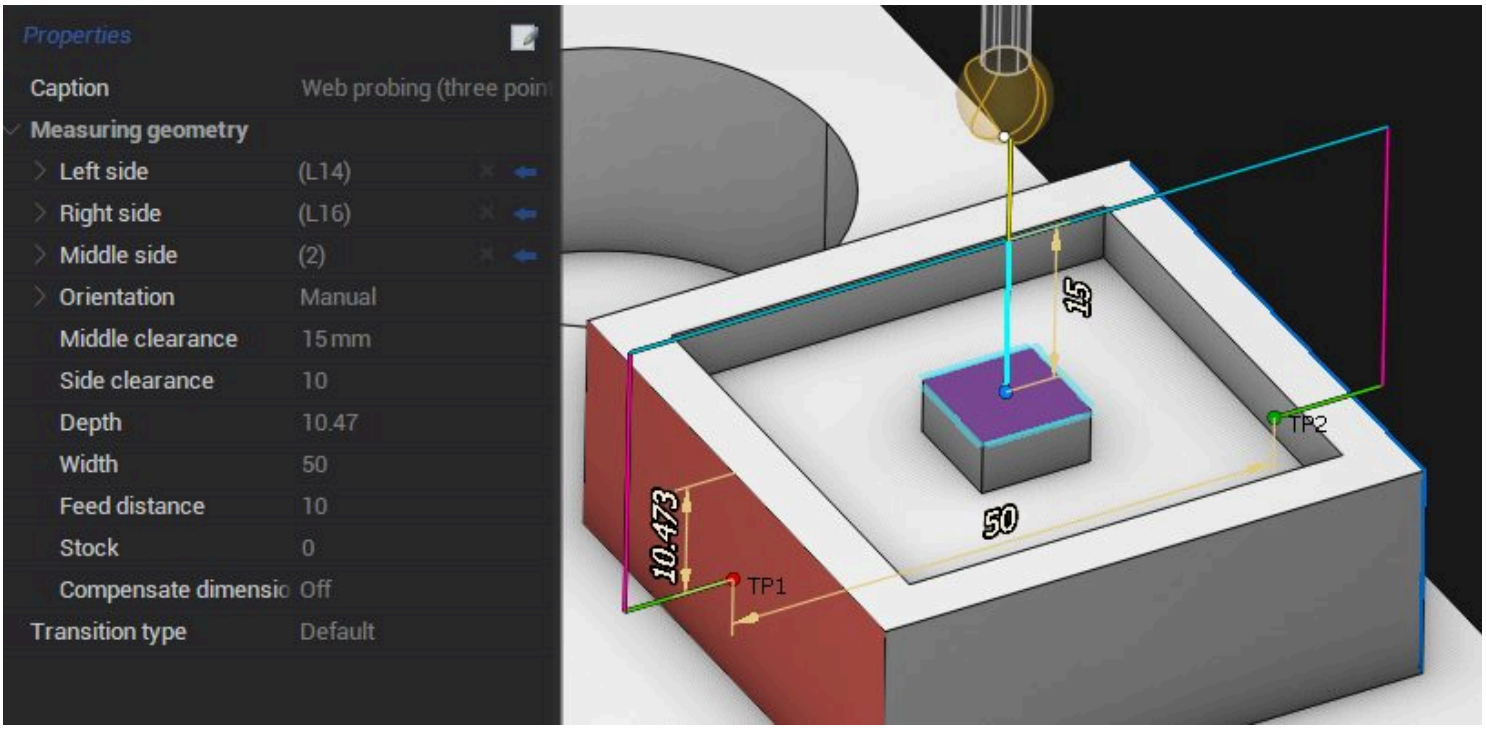
Parameters of web probing:

- First side - the first touch point. Y ou can edit position and vector of the point manually. Y ou need to expand First side and set values in the corresponding fields: Target point 1, Target vector 1;
- Second side - the second touch point opposite the first point. Y ou can edit position and vector of the point manually. Y ou need to expand Second side and set values in the corresponding fields: Target point 2, Target vector 2;
- Top side - determination of the top level from which we count the distance to the starting point of the cycle;
- Top clearance - distance from Top side to the start point of the cycle;
- Side clearance - distance between approach point and First side and distance between approach point and Second side;
- Depth - distance from Top side to points position;
- Width - distance between First side and Second side.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Web probing parameters".

Web probing (three points)

Probing cycle for measuring projections or dimensions on three sides. T he third side is between the first and second.



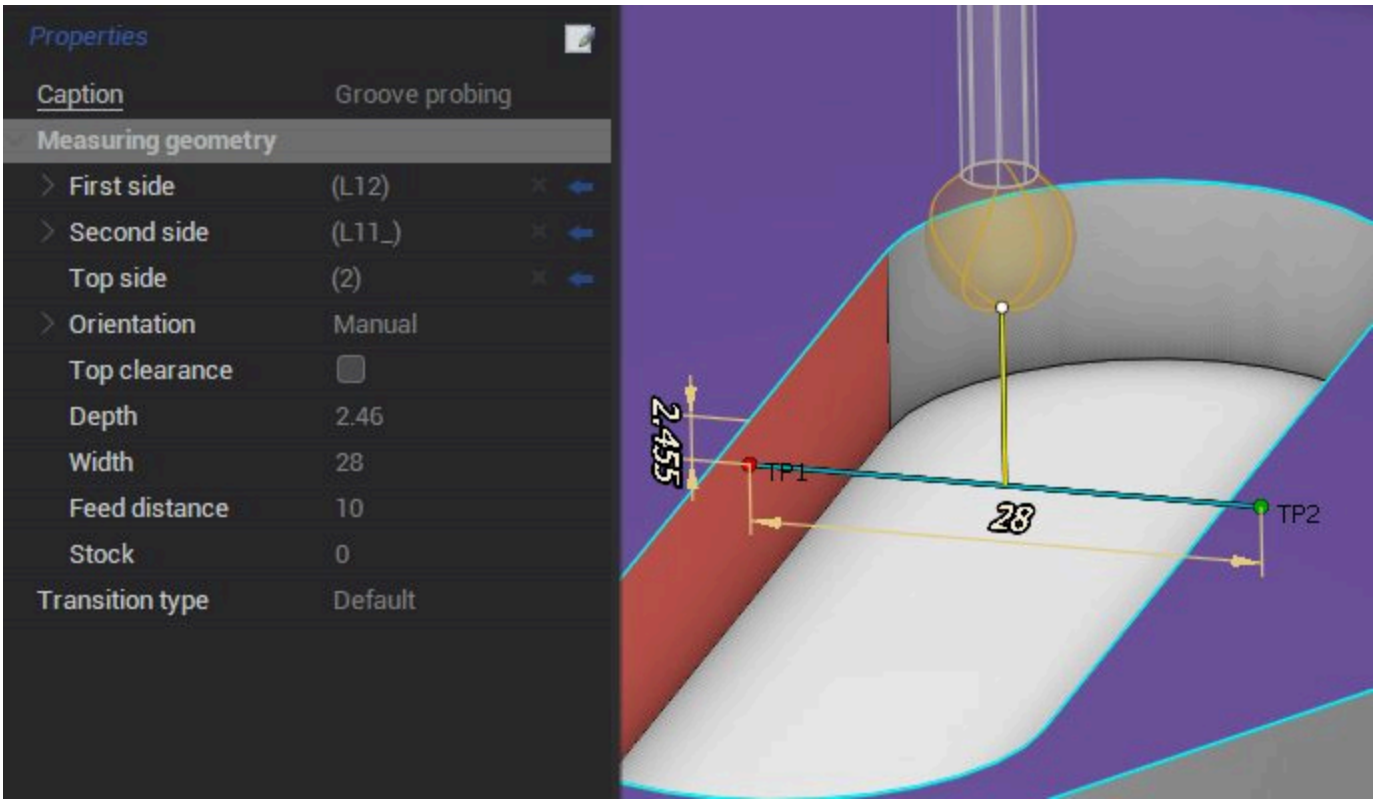
Parameters of web probing (three points):

- Left side - the first touch point. Y ou can edit position and vector of the point manually. Y ou need to expand Left side and set values in the corresponding fields: Target point 1, Target vector 1;
- Right side - the second touch point opposite the first point. Y ou can edit position and vector of the point manually. Y ou need to expand Right side and set values in the corresponding fields: Target point 2, Target vector 2;
- Middle side - the third touch point between the first point and the second point. Y ou can edit position and vector of the point manually. Y ou need to expand Middle side and set values in the corresponding fields: Target point 3, Target vector 3;
- Middle clearance - distance between approach point and Middle side;
- Side clearance - distance between approach point and Left side and distance between approach point and Right side;
- Depth - distance from Middle side to the first and the second touch points;
- Width - distance between Left side and Right side.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Web probing (three points) parameters".

Groove probing

Probing cycle for measuring grooves and other recesses on two sides.



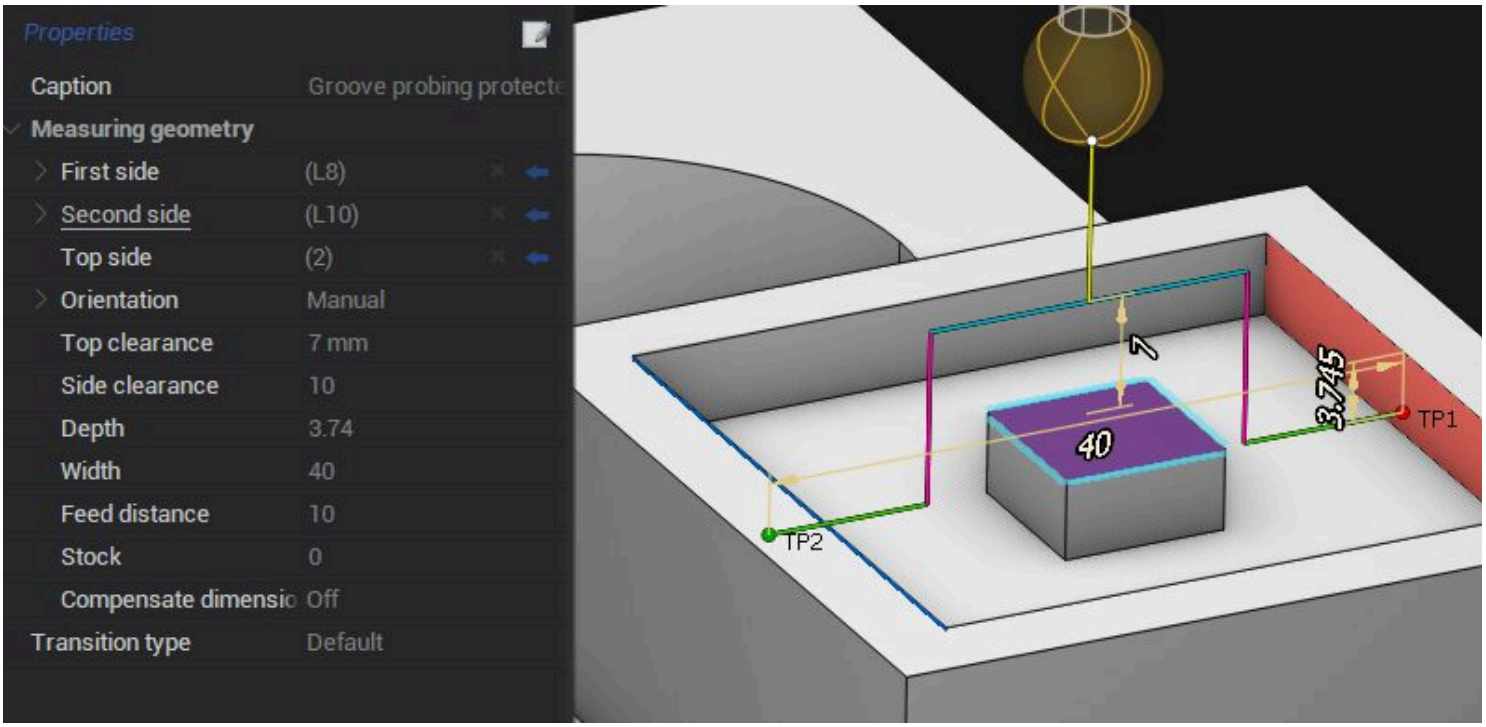
Parameters of groove probing:

- First side - the first touch point. Y ou can edit position and vector of the point manually. Y ou need to expand First side and set values in the corresponding fields: Target point 1, Target vector 1;
- Second side - the second touch point opposite the first point. Y ou can edit position and vector of the point manually. Y ou need to expand Second side and set values in the corresponding fields: Target point 2, Target vector 2;
- Top side - determination of the top level from which we count the distance to the starting point of the cycle;
- Top clearance - i f the parameter is checked then you set the distance from Top side to the start point of the cycle. If the parameter is unchecked then the start point is located at the intersection of four points;
- Depth - distance from Top side to points position;
- Width - distance between First side and Second side.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Groove probing parameters".

Groove probing protected

Probing cycle for measuring grooves and other recesses on two sides inside which there is an obstacle.



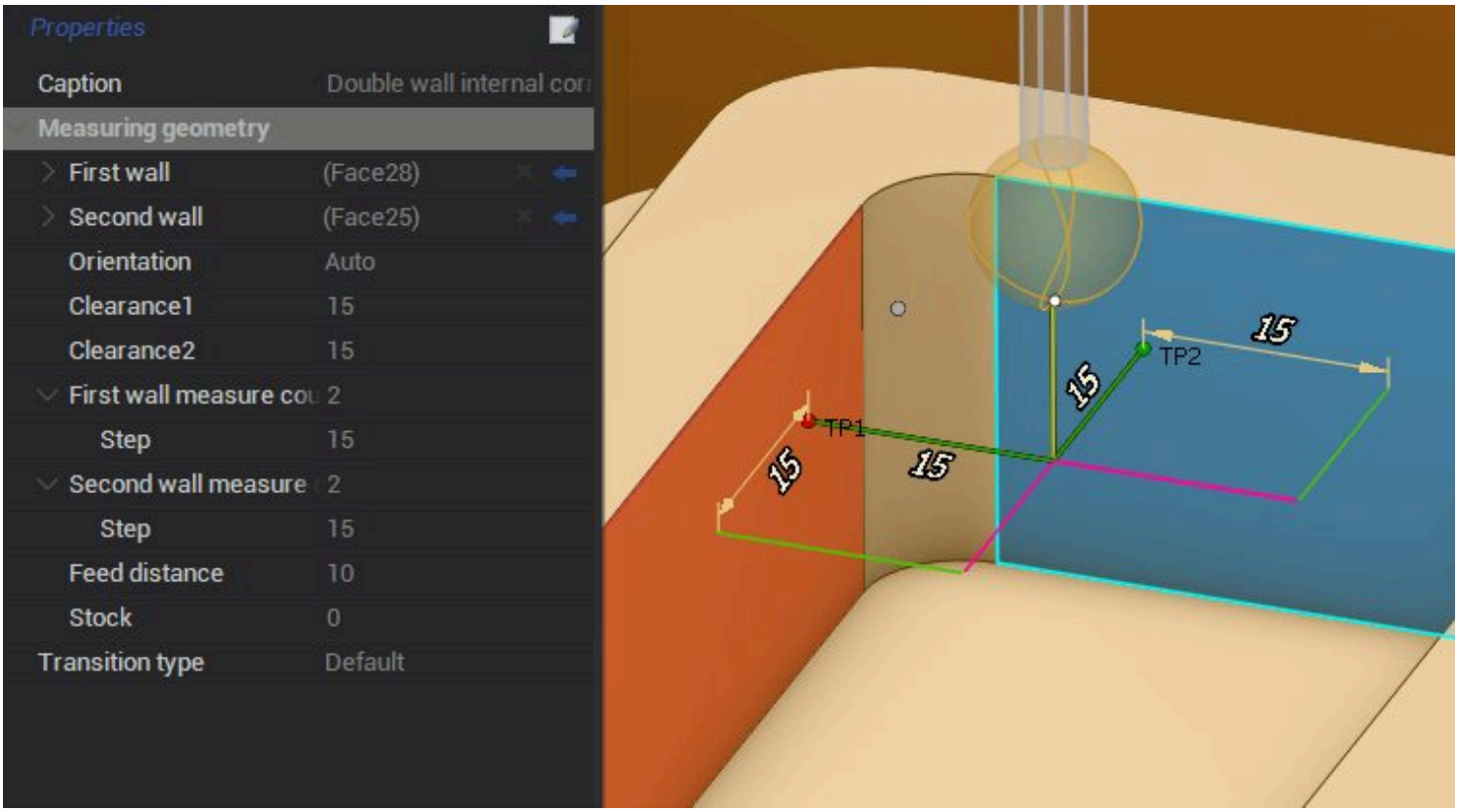
Parameters of this probing are the same as in groove probing. But it has one own parameters to avoid obstacles (Side clearance) and one parameter is changed (Top clearance):

- Side clearance - distance between approach point and First side and distance between approach point and Second side;
- Top clearance - distance from Top side to the start point of the cycle.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Groove probing protected parameters".

Double wall internal corner probing

Probing cycle for measuring the internal angle between two surfaces.



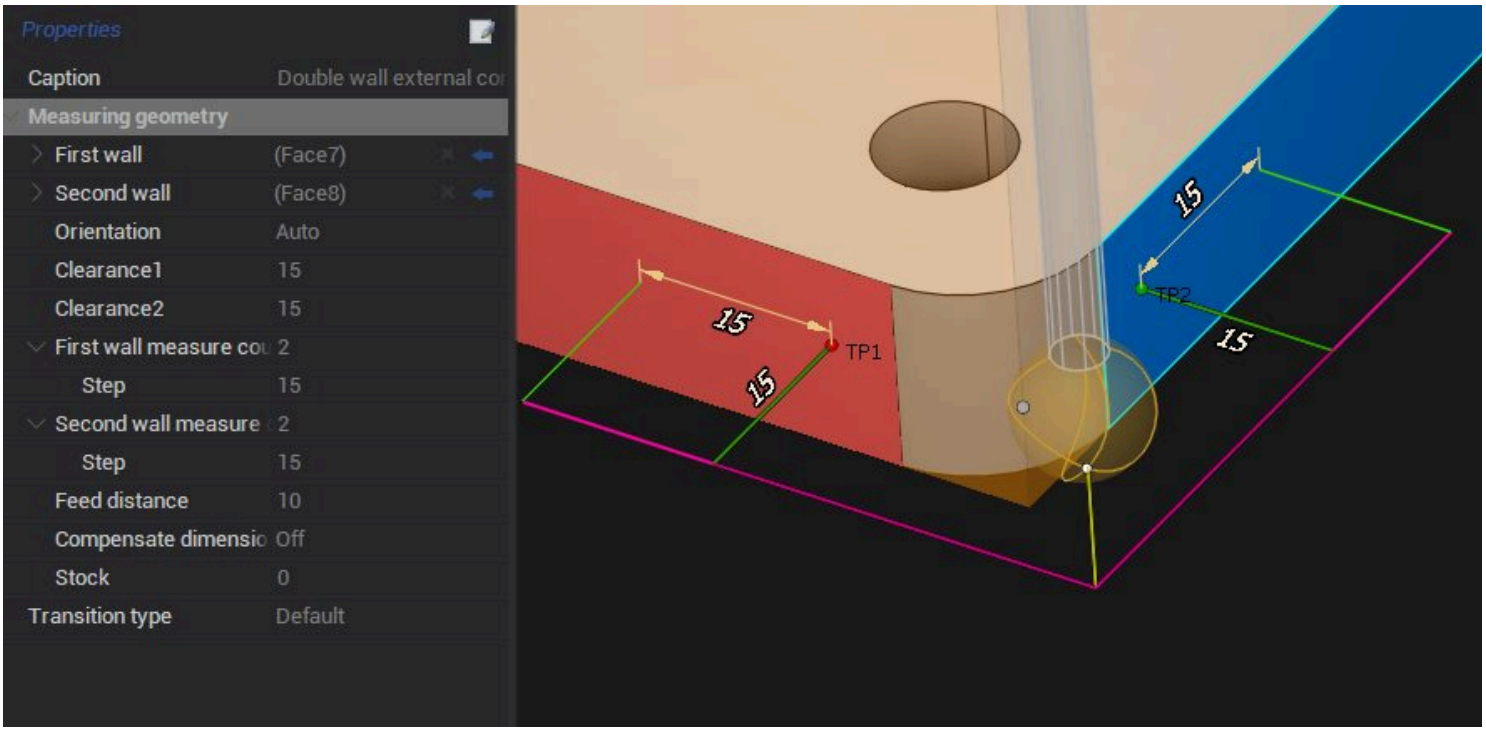
Parameters of double wall internal corner:

- First wall - the touch point of the first wall. Y ou can edit position and vector of the point manually. Y ou need to expand First wall and set values in the corresponding fields: Target point, Target vector;
- Second wall - the touch point of the second wall. Y ou can edit position and vector of the point manually. Y ou need to expand Second wall and set values in the corresponding fields: Target point, Target vector;
- Clearance1 - distance between approach point and First wall;
- Clearance2 - distance between approach point and Second wall;
- First wall measure count - count of touch points on the first wall. If count is more than one then you can set Step. Step is distance between touch points on the first wall;
- Second wall measure count - count of touch points on the second wall. If count is more than one then you can set Step. Step is distance between touch points on the second wall.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Double wall internal corner probing parameters".

Double wall external corner probing

Probing cycle for measuring the external angle between two surfaces.

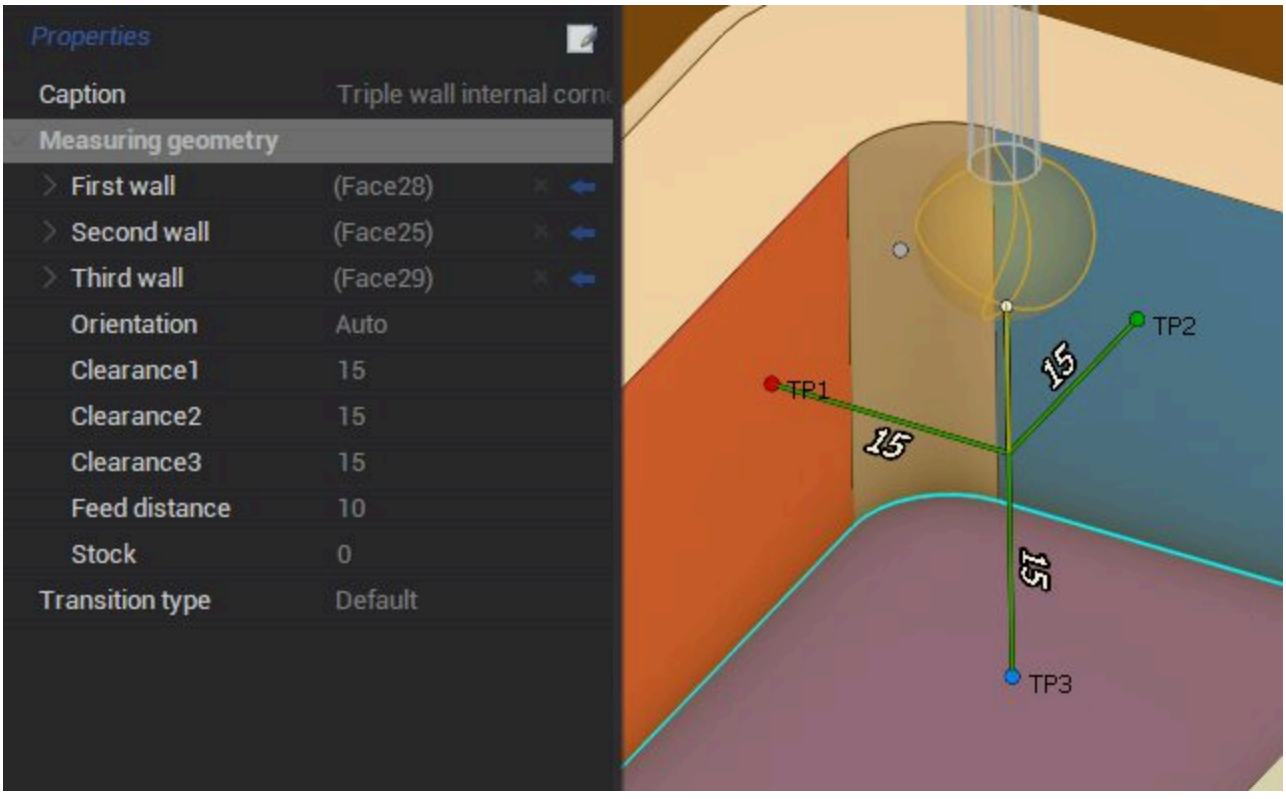


Parameters of this probing are the same as in double wall internal corner. T he only difference is that touch walls are external surfaces.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Double wall external corner probing parameters".

Triple wall internal corner probing

Probing cycle for measuring the internal angle between three surfaces.



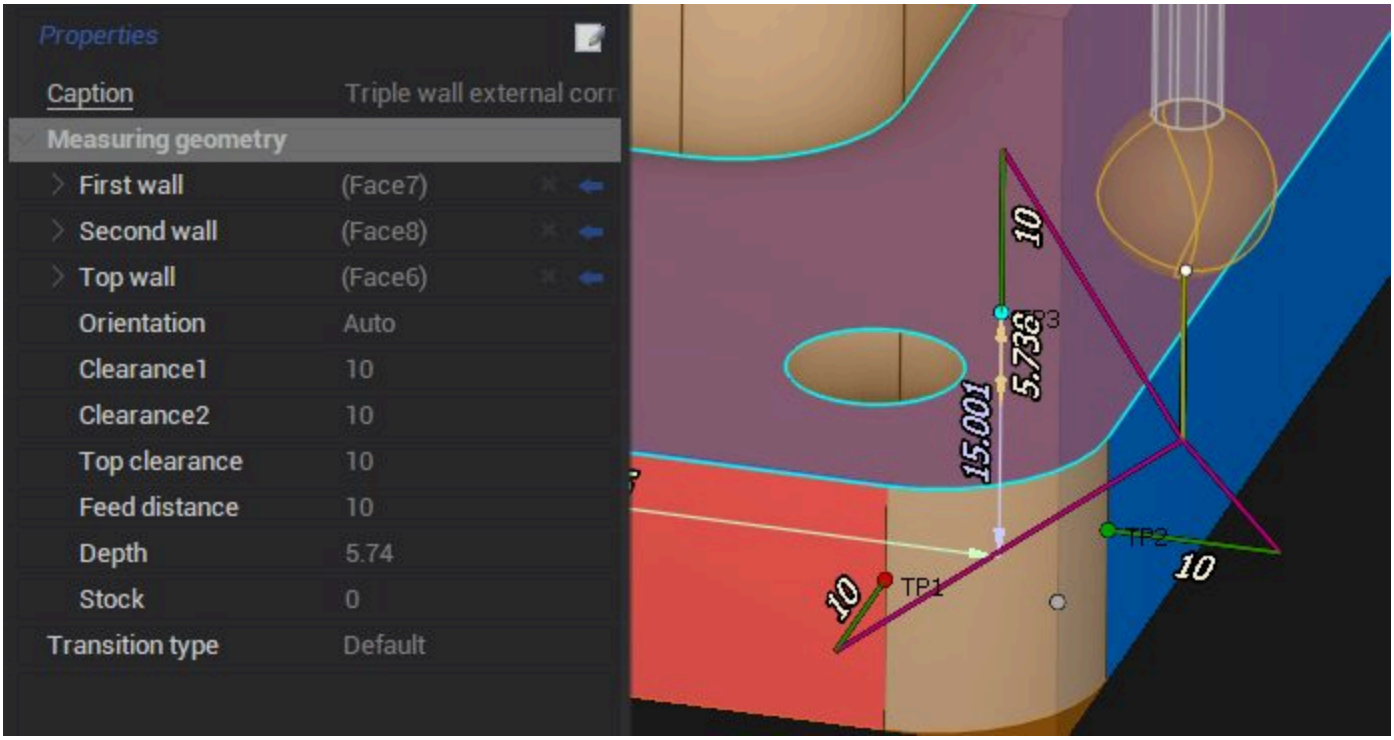
Parameters of triple wall internal corner:

- First wall - the touch point of the first wall. Y ou can edit position and vector of the point manually. Y ou need to expand First wall and set values in the corresponding fields: Target point, Target vector;
- Second wall - the touch point of the second wall. Y ou can edit position and vector of the point manually. Y ou need to expand Second wall and set values in the corresponding fields: Target point, Target vector;
- Third wall - the touch point of the third wall. Y ou can edit position and vector of the point manually. Y ou need to expand Third wall and set values in the corresponding fields: Target point, Target vector;
- Clearance1 - distance between approach point and First wall;
- Clearance2 - distance between approach point and Second wall;
- Clearance3 - distance between approach point and Third wall.

A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Triple wall internal corner probing parameters".

Triple wall external corner probing

Probing cycle for measuring the external angle between three surfaces.



Parameters of triple wall external corner:

- First wall - the touch point of the first wall. Y ou can edit position and vector of the point manually. Y ou need to expand First wall and set values in the corresponding fields: Target point, Target vector;

- Second wall - the touch point of the second wall. Y ou can edit position and vector of the point manually. Y ou need to expand Second wall and set values in the corresponding fields: Target point, Target vector;
- Top wall - the touch point of the top wall. Y ou can edit position and vector of the point manually. Y ou need to expand Top wall and set values in the corresponding fields: Target point, Target vector;
- Clearance1 - distance between approach point and First wall;
- Clearance2 - distance between approach point and Second wall;
- Top clearance - distance between approach point and Top wall;
- Depth - distance from Top wall to the touch point of the first and the second wall.

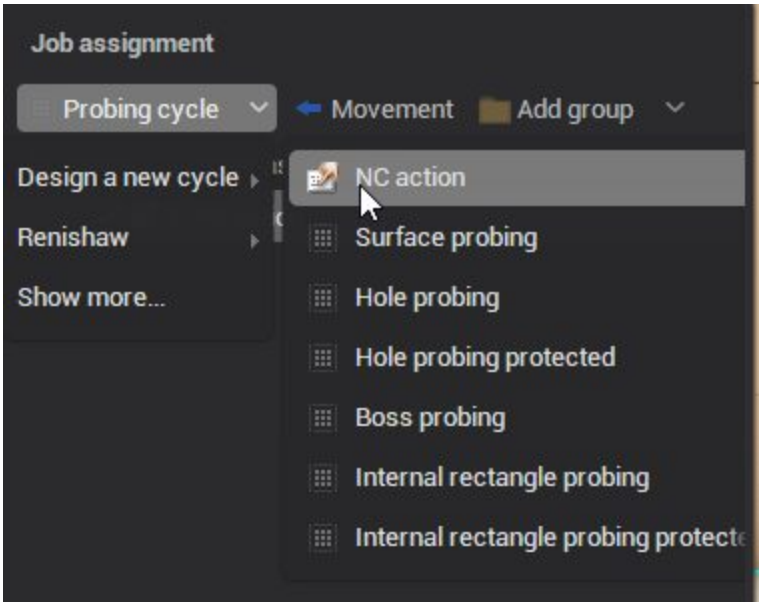
A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Triple wall external corner probing parameters".

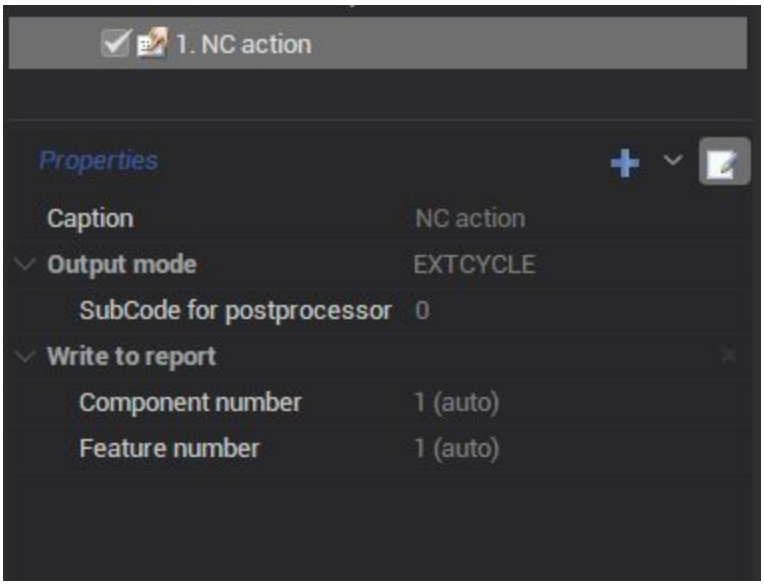
Additional elements

NC action

The certain subroutine can be called after several successive measuring cycles. It performs additional actions or calculations based on the measurements just taken. These can be: calculation of average values, deviations, intersection points between several surfaces, recording these values in some rack variables.

It does not generate any movements. However, it can contain parameters that need to be either directly displayed in the NC using the Insert command or as a cycle with an array of numeric parameters. Through the mechanism of custom properties, you can add any parameters there. Based on these parameters in the postprocessor, it will be possible to form the necessary calls to subroutines and additional actions. Notes: In order to add custom properties, the Show expert tools mode must be enabled. NC action is located in Probing cycle - Design a new cycle.

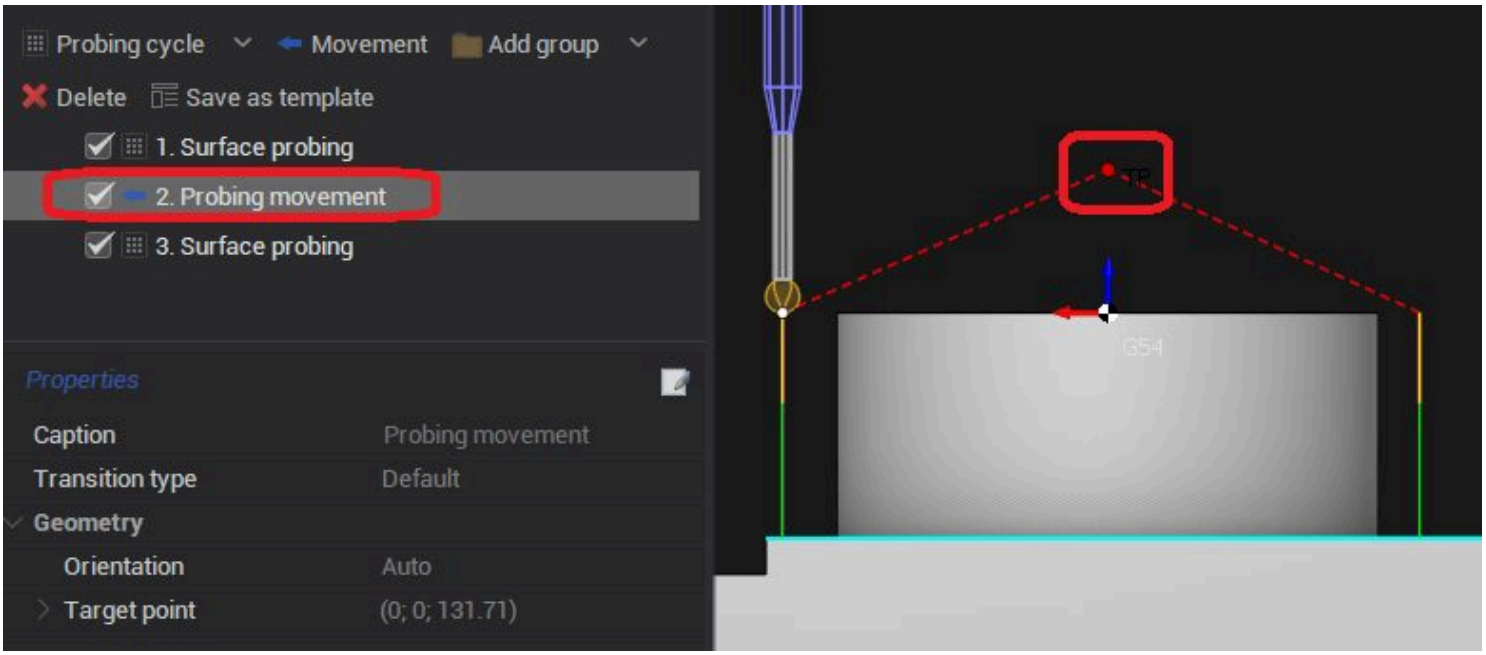




A ccess codes for parameters in CLData are described in postprocessor documentation "Probing cycle WProbing - Additional parameters".

Movement

Element for additional movement, for example to avoid obstacles. It is necessary to set the Target point through which an additional movement will be created.



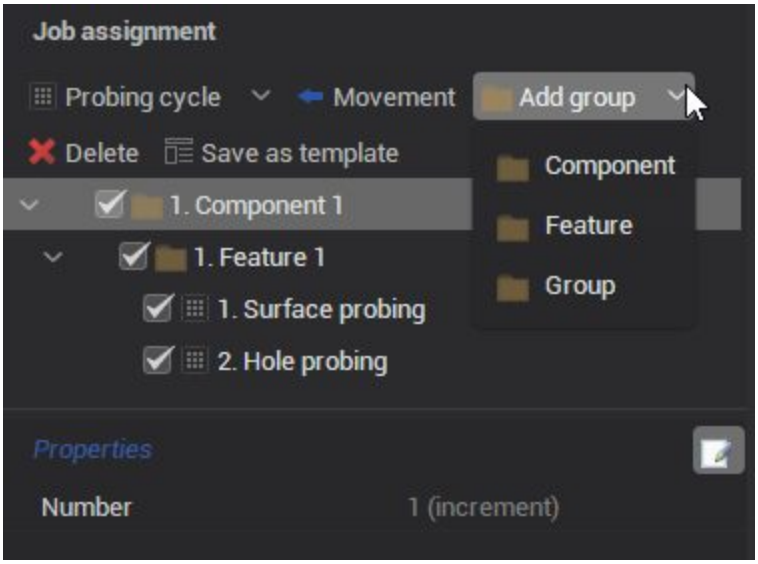
Elements to group

Complex measuring cycles for the measurement of complex parts are usually performed not in isolation but in successive measurements connected with each other. For example, several measurements are performed and then the total result is calculated and written to the report under certain numbers which correspond to the hierarchy level of the measured element.

There are three types of group:

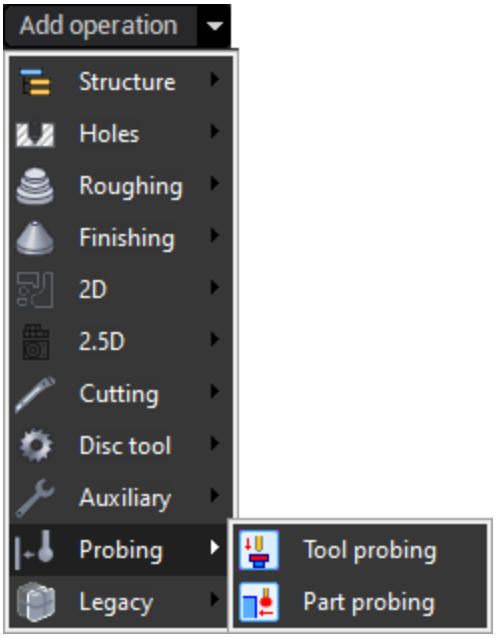
- Group - general grouping;
- Component - part to be measured;
- Feature - individual element or surface currently being measured.

The level below goes directly measuring cycles. In the properties of the groups, you can set the number manually but by default it is automatically incremented relative to the previous value within the group of the corresponding type. The type and number of the group determines its name.



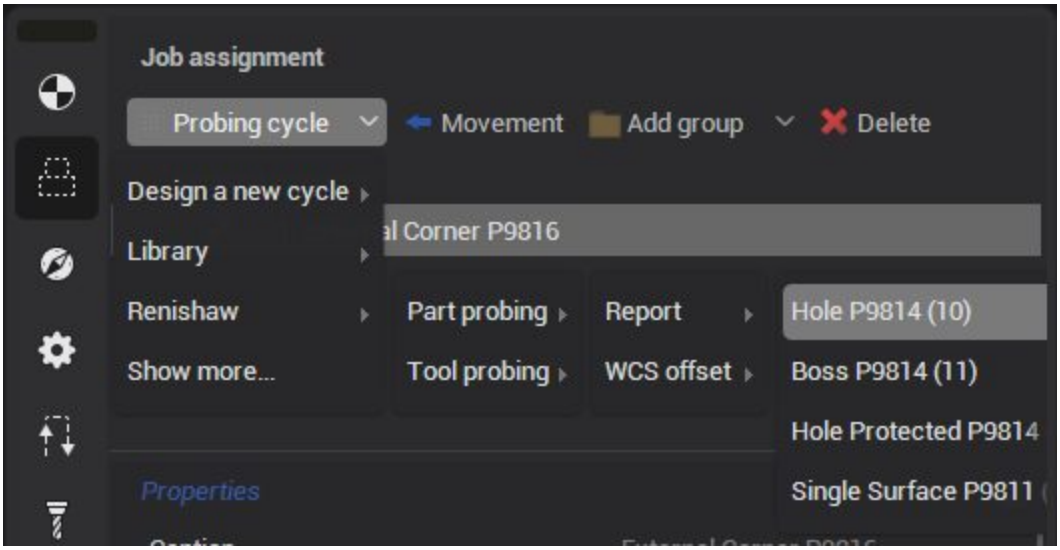
Use of prepared measuring cycles

Y ou need to create one of the available operations to start working with measuring cycles. Measuring operations are in a group Probing. Mill part probing and Mill tool probing operations are available for mill machines and robots. Turn part probing and Turn tool probing operations are available for lathe machines. All four operations can be used when choosing lathe-mill machine.



Part probing

A dd an part probing operation from the list and switch to the Job assignment tab and add one of the existing cycles. Press to Probing cycles button to show context menu. Show more allows you to open Probing templates window in which there are libraries with cycles ready for work with own properties. Also these libraries located in context menu of Probing cycles button.

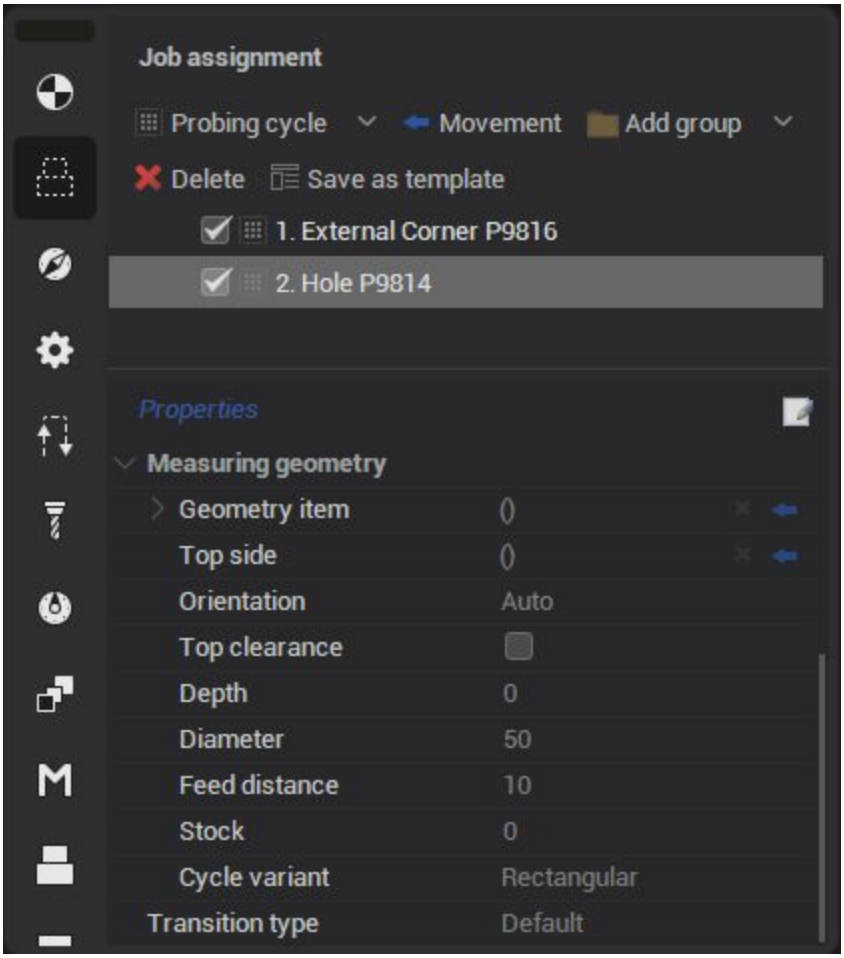


The example of working with cycles will be presented based on Hole probing.

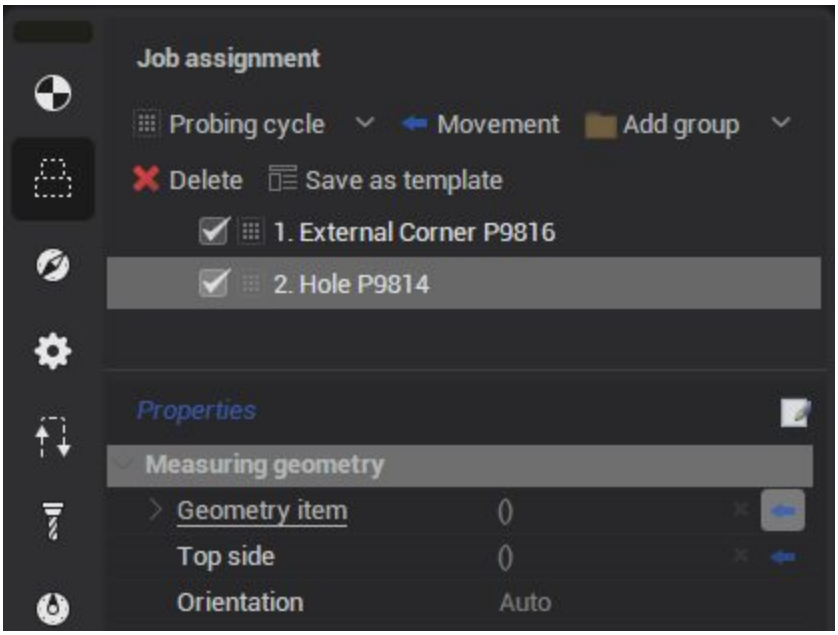
Add new probing cycle Hole P9814 from default library Renishaw, group Report.

Properties of the added cycle will appear at the bottom of the Job assignment.

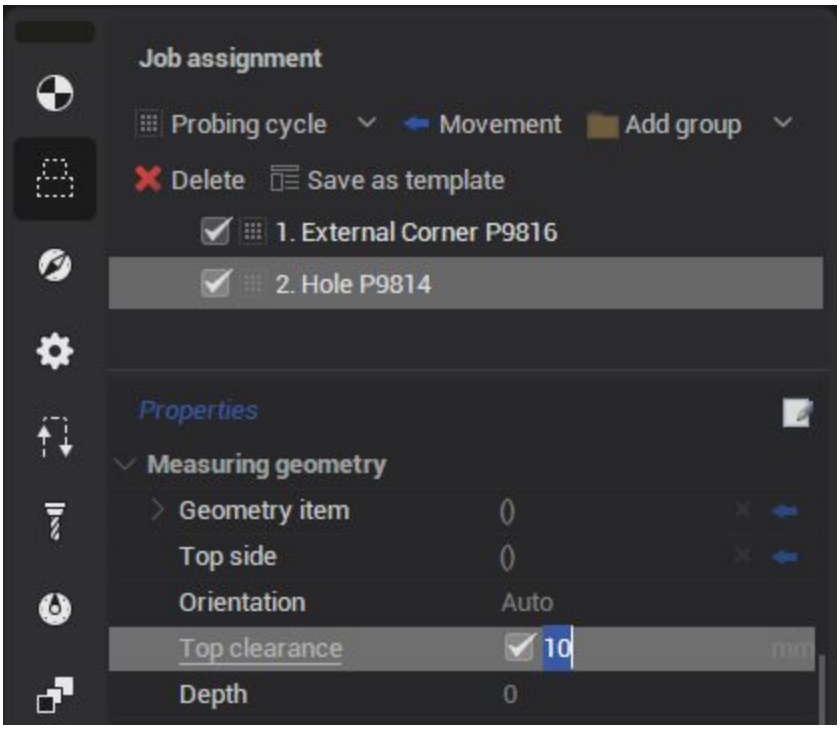
Use Measuring geometry to set parameters for cycle calculation.



C lick on the arrow in the right part of the field and select a surface in the graphics window t o add Geometry item or Top side. The selected surface is automatically added to the current field.

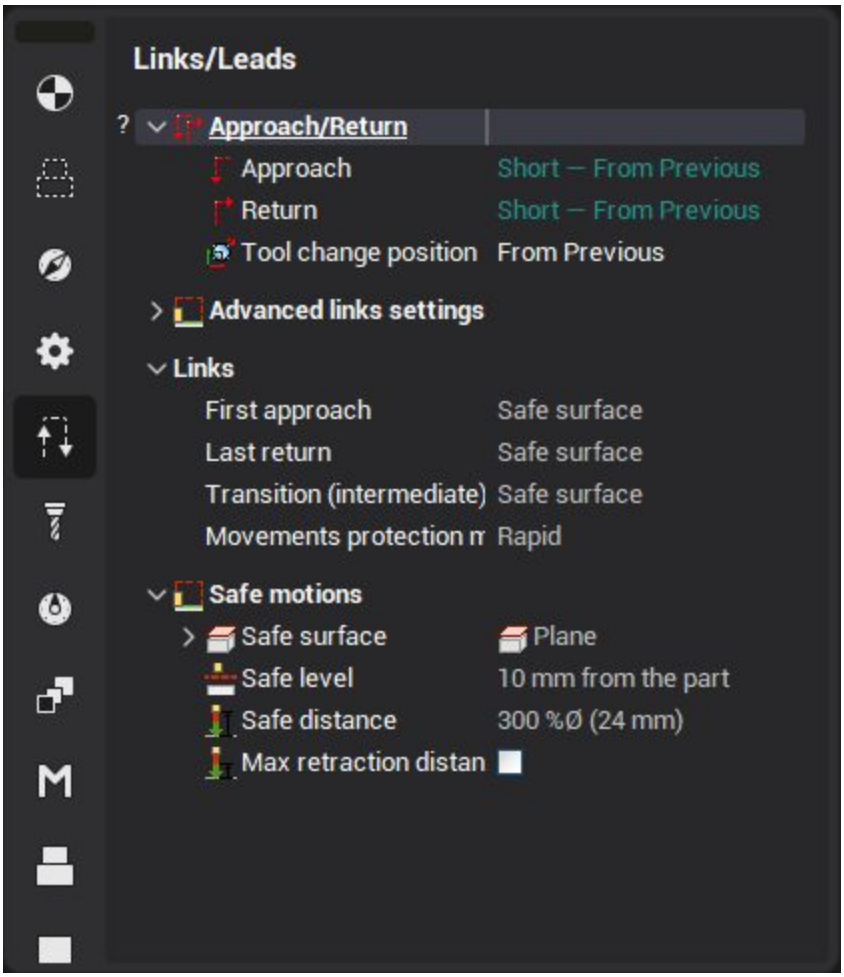


Other parameters are entered manually in the corresponding field. Some cycle parameters can be edited both in the inspector and in the graphics window.



Transitions

Go to the tab Links to set the parameters of transitions between probing cycles.



In addition to standard parameters in this tab, new parameters have been added for measuring operations:

- First approach - creating an approach to the beginning of the first cycle of the operation;
- Last return - creating a return from the last cycle of the operation;
- Transition (intermediate) - creating transitions between operation cycles;
- Safe distance - setting the distance for the approach and return of the cycle. There are 2 kinds of value settings: mm and percentage of tool diameter.

There are 3 common types of value for transitions listed above:

- Short - transition is created directly;
- Safe distance - transition is created by retreating to the distance specified in the Safe distance field;
- Safe surface - transition is created by retreating to the safe surface.

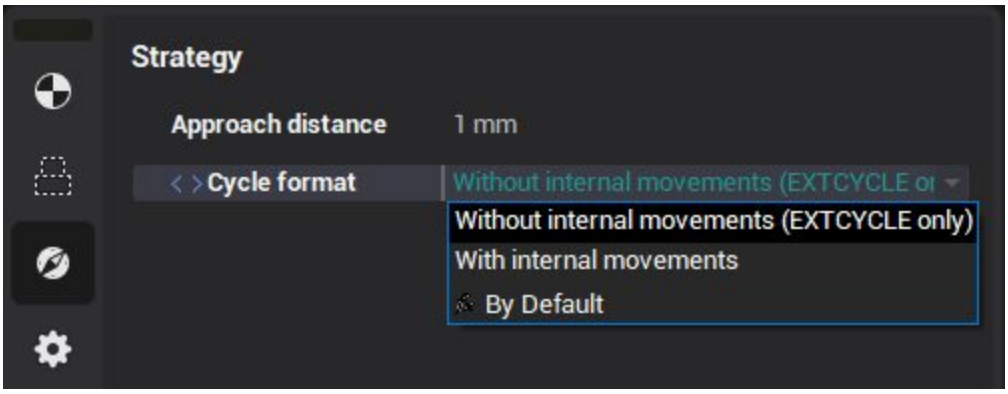
There is additional value for Transition (intermediate): Orthogonal - i f the start and end points of the transition are located at the same level in Z then go directly. If the start point is lower then at first go to the level of the end point then go directly. If the end point is lower then go directly to a point raised above the end point to the level of the starting point then go directly.

You can also choose which feed to move (Movements protection mode):

- Rapid - moving at rapid feed;
- Non protected - moving at long link feed;
- Protected - moving at short link feed.

Cycle format

You can choose one of the options for outputting cycles in CLData. To do this, go to the Strategy tab.

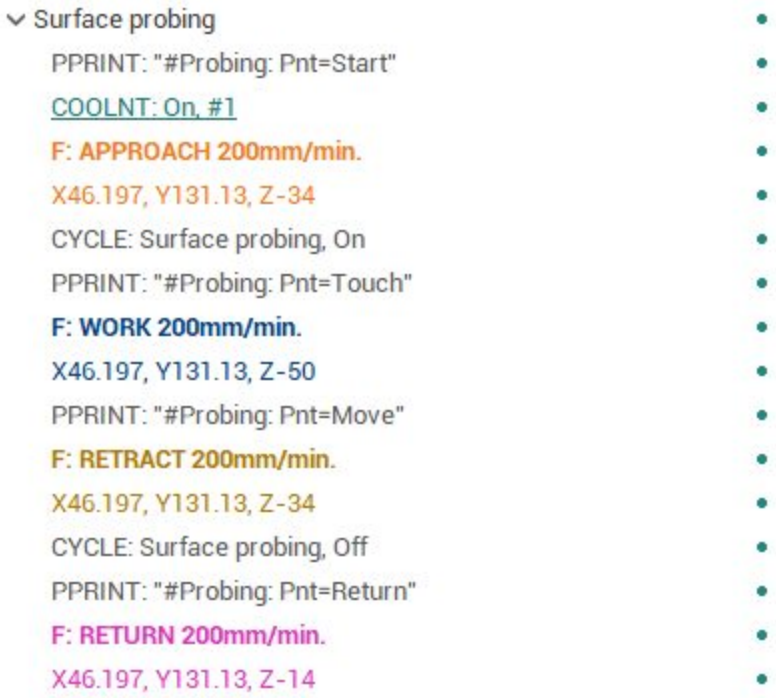


Cycle format has two values:

- Without internal movements (EXTCYCLE only) - cycle movements are output in a separate EXTCYCLE command with callsub.



- With internal movements - cycle movements are output without a separate EXTCYCLE command.

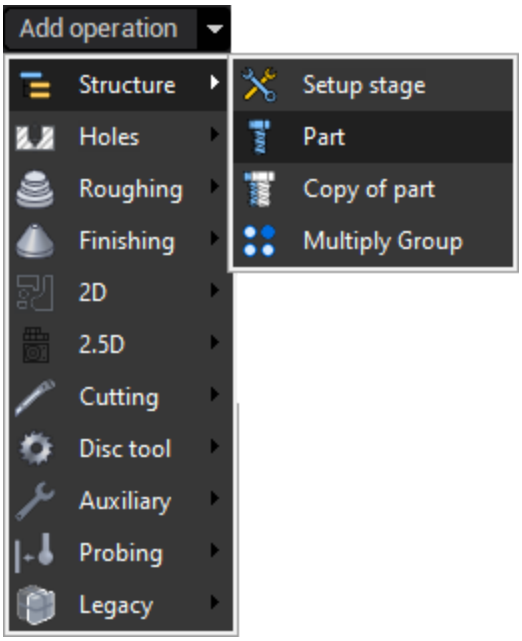


Tool probing

Setting parameters and transitions of cycles for the tool probing is exactly the same as for the part probing.

The probe can be not only a tool, but also a part which fix in a special place on the table. About such a probe, the tool is usually measured for its initial calibration or to check its integrity.

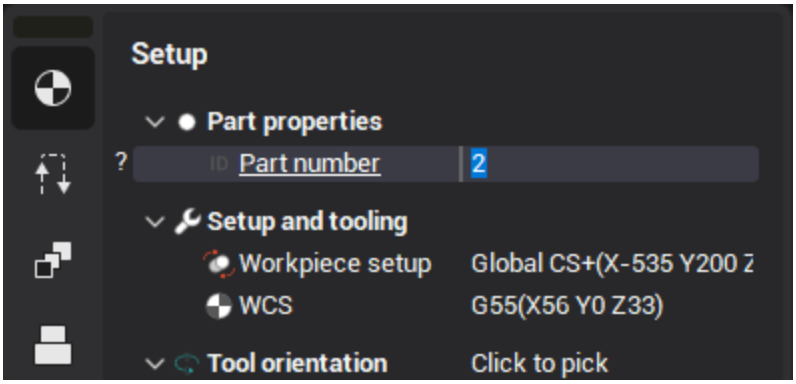
You need to add the Part group f or correct work.



It is necessary to create separate measuring operations for each tool within this group.



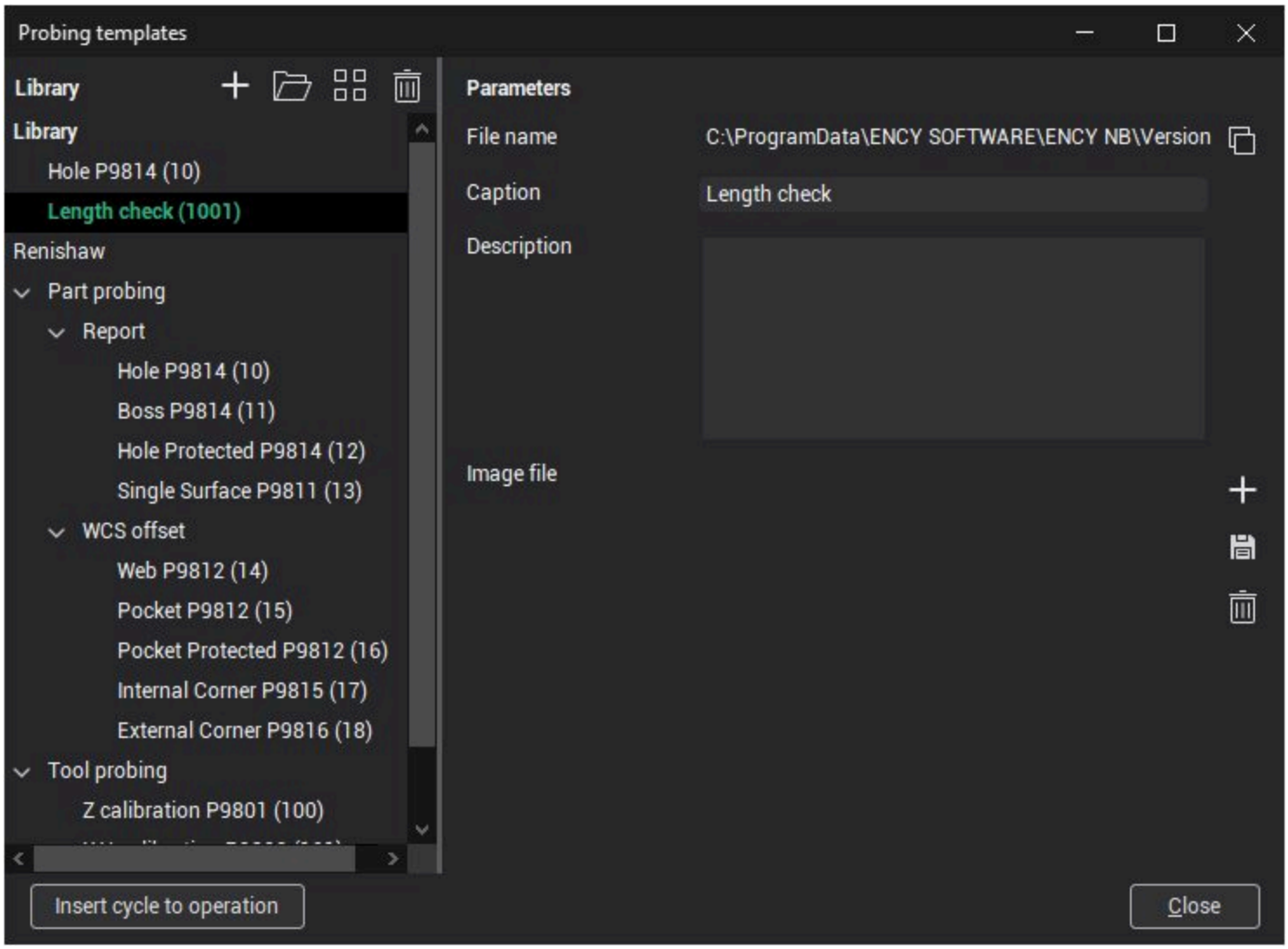
You need to switch to the Setup tab and set Part number. Part number can be used as probe ID.



The example of creating and working with tool probing operation can be found in the distribution project "Tool probing" in "Probing" folder.

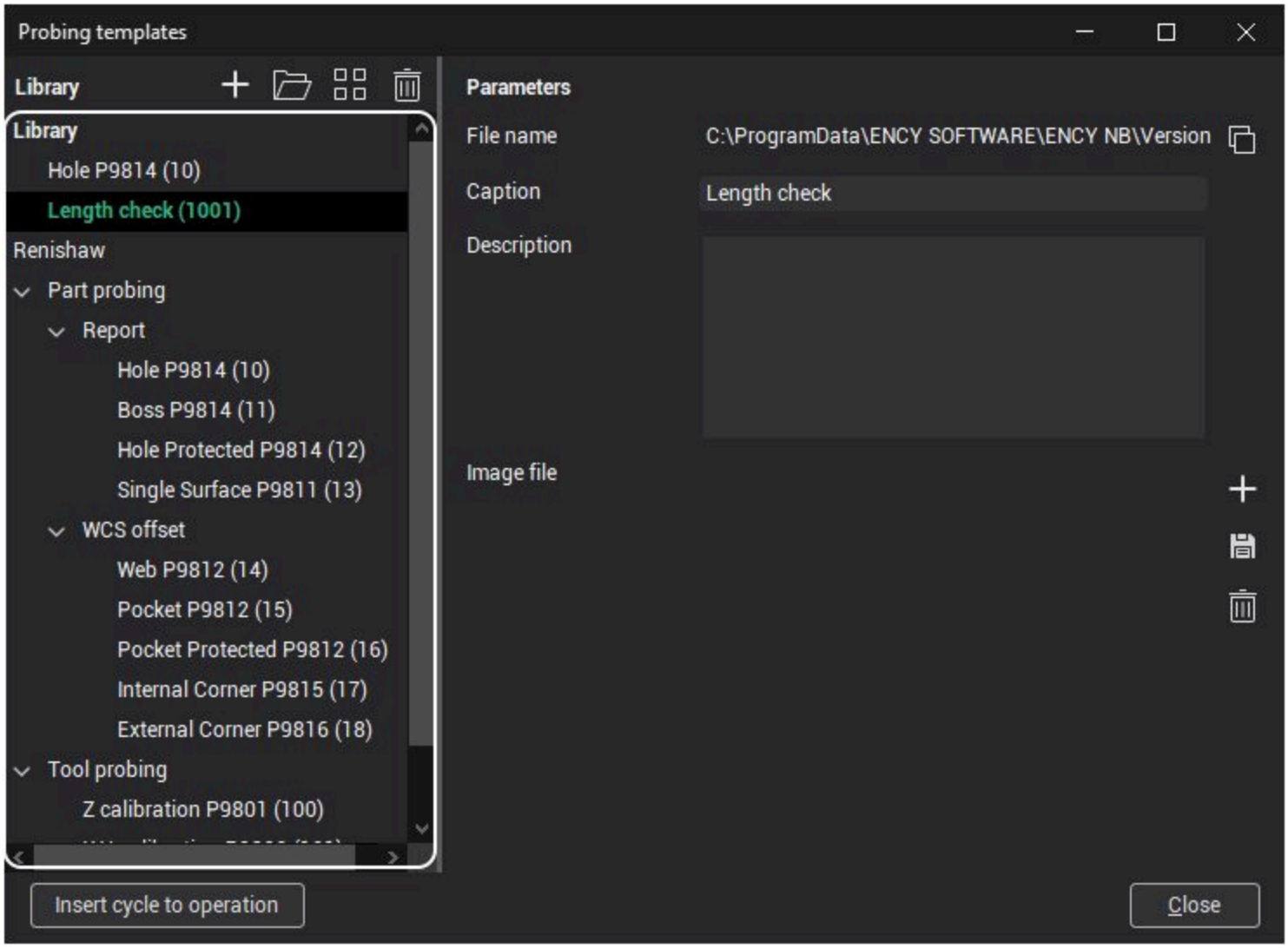
Probing templates window

T he window displays libraries with cycles ready for calculation. Here you can add a description to the cycles, specify the display name of the cycle in the window, distribute them into groups and add them to the required library. Each library is saved locally and stored by default in the default folder of libraries ($(Libraries_FOLDER)) in subfolder " ProbingCycles ".



Library tree

On the left part of the window there is a tree with a list of libraries and cycle templates. A ll previously created libraries and libraries from the Examples and Suppliers folders are displayed here. if the Suppliers folder does not exist, then you need to create it in the following path: default folder of libraries ($(Libraries_FOLDER)) in subfolder " ProbingCycles ".



Libraries are sorted as follows: first, created user libraries are located, followed by libraries from Examples and Suppliers folders.

Elements are moved using drag'n'drop. You can move like this:

- Libraries are moved between other libraries on the same level;
- Groups are moved between other groups at the same level, from one library to another, and also from one group to another;
- Cycle templates are moved between other groups at the same level, from one library to another, and also from one group to another;
- It is forbidden to move items to libraries from Examples and Suppliers folders.

The window can also display the SubCode for postprocessor of each cycle template, which will be displayed in brackets after the name. To do this, you must enable the Show expert tools mode. How to do this is described here.

When you select an element in the tree, its parameters will be displayed in the right part of the window, which can be edited.

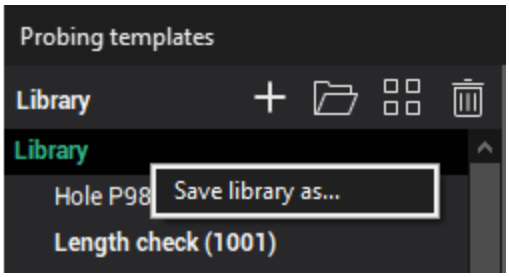
Control elements

The tree element control panel is in the upper left part of the window. There are 4 controls on the panel:

1.  Open library is button to add existing local libraries. A fter clicking on the button, a dialog window will open with the choice of the location of existing library;
2.  Create new library is button to create new library on computer and add new library to window. After clicking on the button, a dialog window will open with the choice of the location of the new library;
3.  New group is button to create group for sorting and combining cycle templates. A group can only be created inside a library or inside another group;
4.  Remove is button to remove selected element from the window.

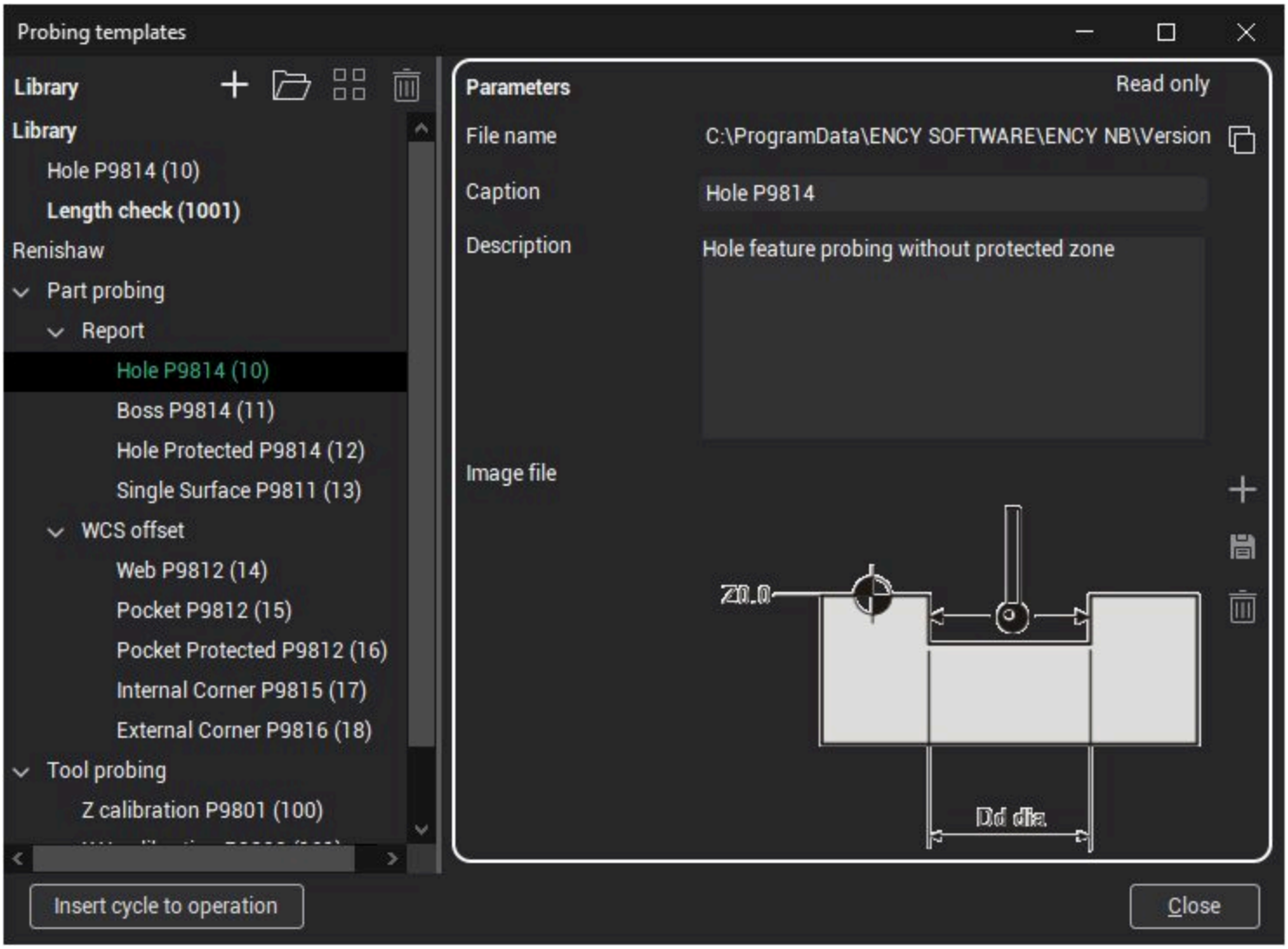
You can also make a copy of the library and save it to another location on your computer. To do this, select the required library and click the right mouse button. In the context menu that appears, click Save library as. A fter clicking on the button, a dialog box will open with the choice of the location of the copy library. After that the copy will be saved on the computer and added to the window.

Note: New group is disabled for libraries from Examples and Suppliers folders.



Parameters panel

Parameters panel is on the right side of the window.



There is two general parameters for all elements:

- File name - location path of the library on the computer and file name of library;
- Caption - name of element in probing templates window.

Other parameters are not available for library and group elements including image control buttons.

Two additional parameters are available for cycle template elements:

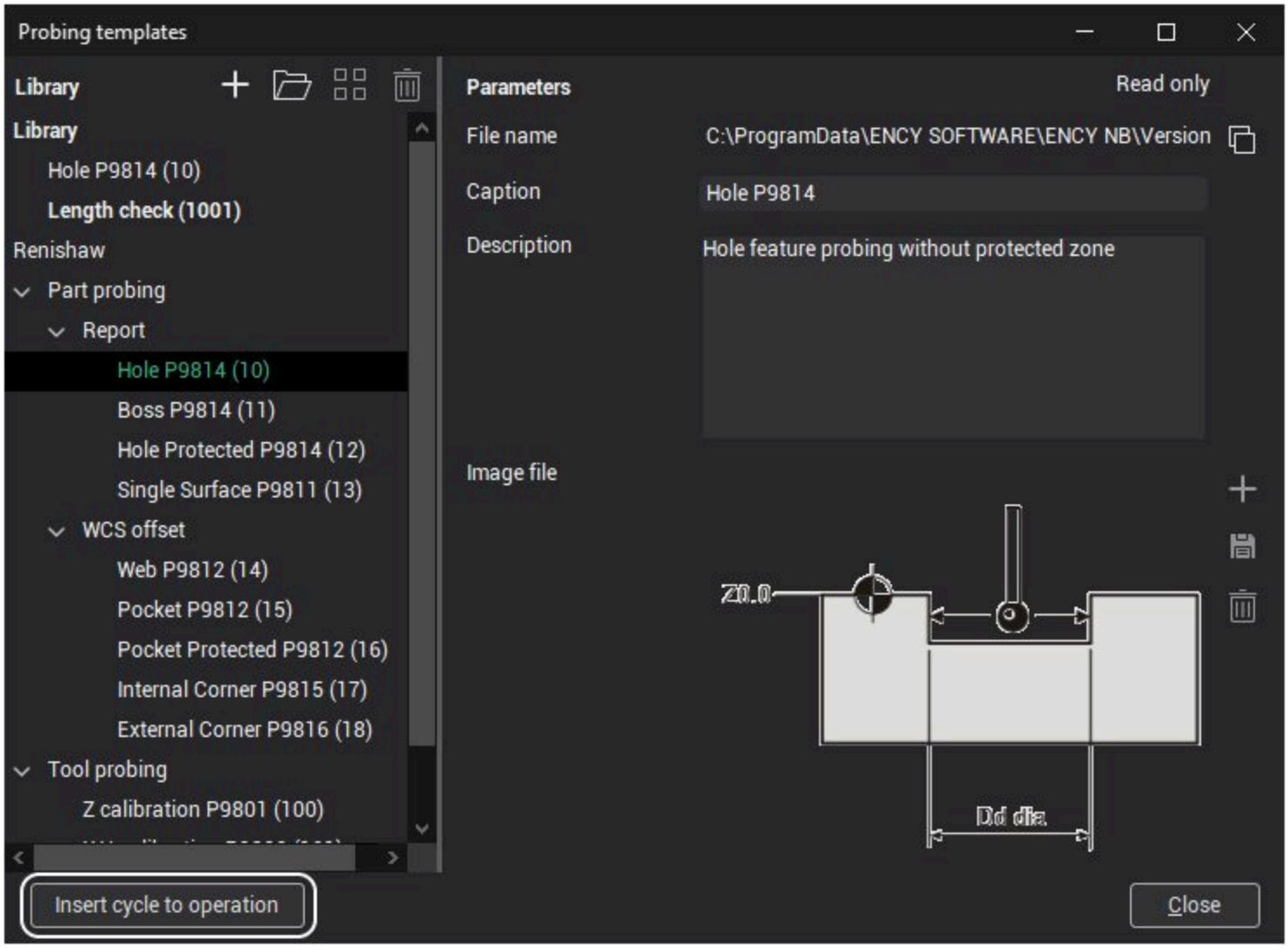
- Description - the purpose of a certain cycle is indicated, as well as any other additional information;
- Image file - the image that displays current cycle.

Note: editing elements from libraries located in Examples and Suppliers folders is prohibited. This is indicated by the Read only inscription located in the upper right part of the window.

Saving and using cycles

All the changes made (creating libraries, changing the position of elements, editing parameters, etc.) save when the window is closed.

In order to use a cycle template, you need to select the required template and click on the button Insert cycle to operation. After that the selected template will be inserted into the Job assignment of the probing operation.



Spray painting

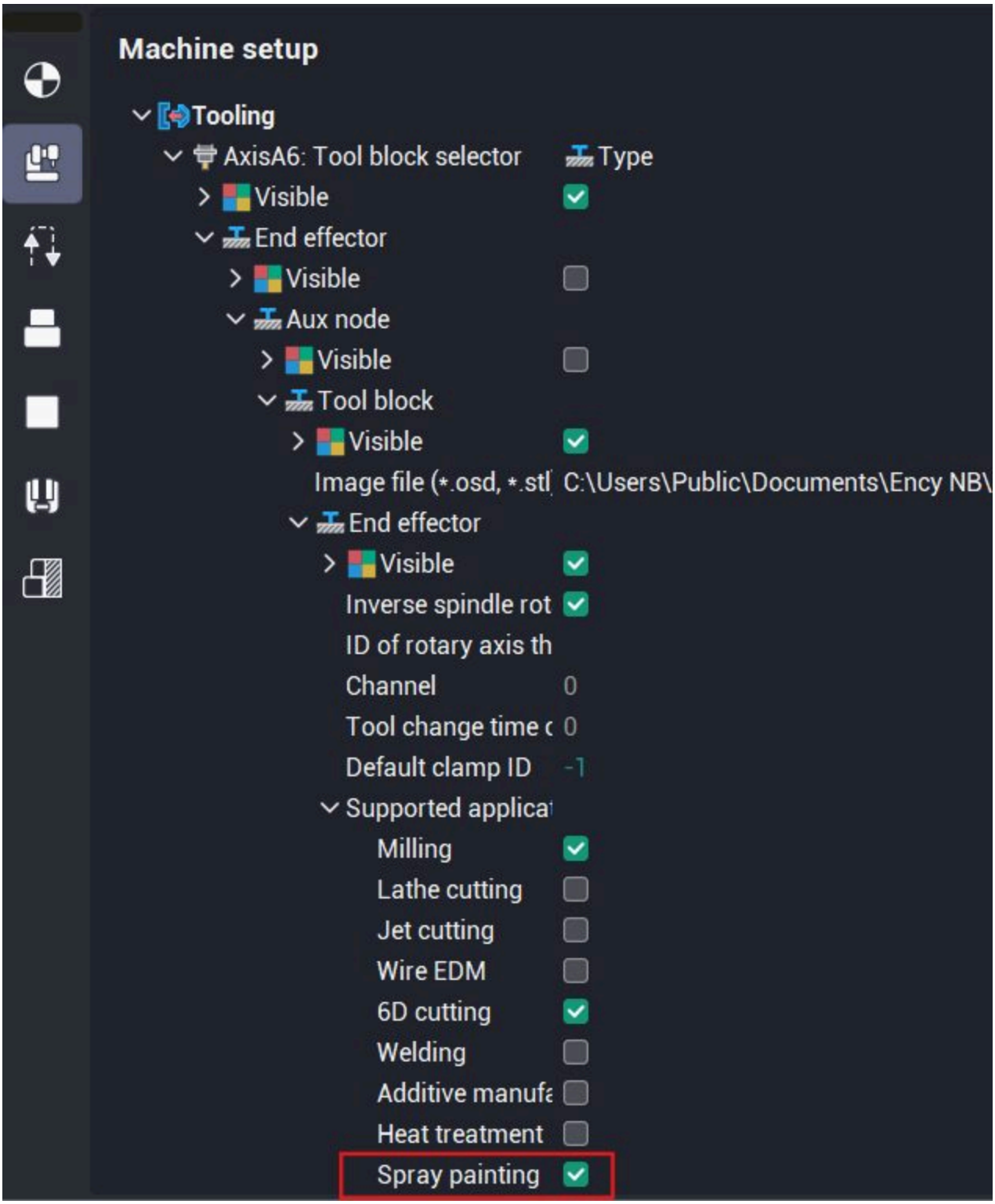


Spray painting is a painting technique in which a device sprays coating material (paint, ink, varnish, etc.) through the air onto a surface. In order to be able to process hard-to-reach places, such machining is often performed on equipment that has a large number of degrees of freedom, such as industrial robots.

The system supports programming and simulation of such machining.

Machine schema

To be possible to use spray painting first of all you need to choose appropriate machine schema. If the machine is configured correctly, then the list of supported applications of a tool connector should include "Spray painting" as shown on the picture below. This will limit the list of applicable technological operations and affect the choice of default values for some parameters (for example, simulation type).



Technological operations

There are 4 operations in the system, designed exclusively for spray painting. These operations located in "Spray" group of operations and they available only if you have machine schema with "Spray painting" supported application.

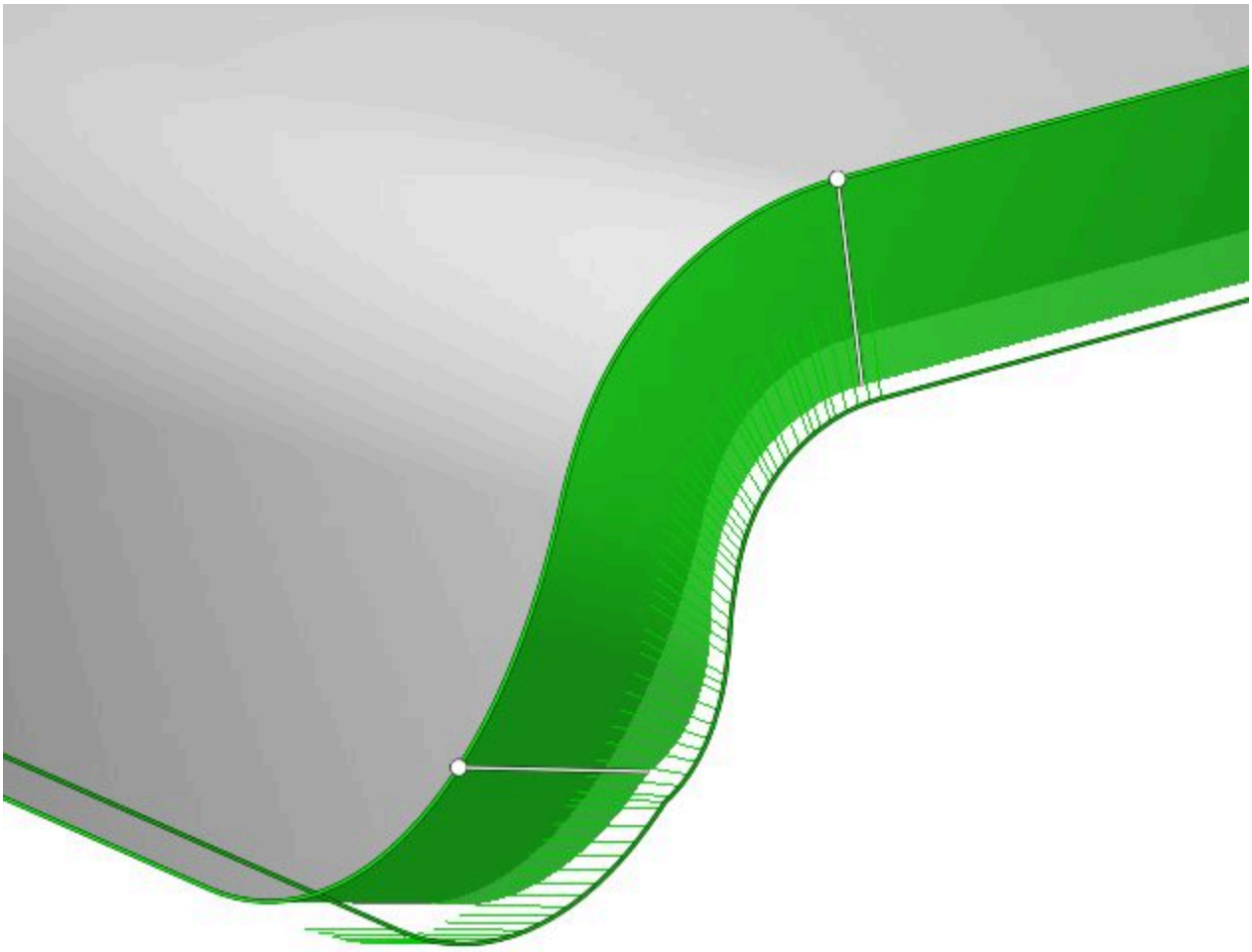
1. Contour spraying (based on a 6D Contouring operation);
2. Surface spraying (based on a Cladding 5D operation operation);

3. Morph spraying (base on a Morph operation);
4. Rotary spraying (based on a Rotary finishing operation).

> When you create any spraying operation, the system will automatically choose the spray tool for this operation on the "Tool" tab and sets the "Painting" simulation as a "Simulation type" on the Parameters tab by default.
>
> You can find information about spray tools in the section below.

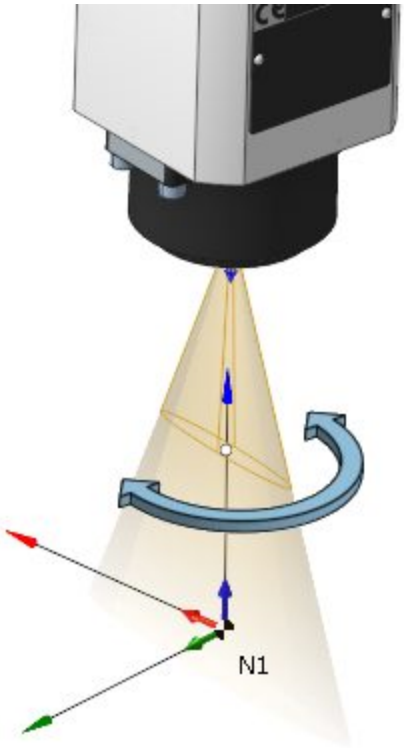
You can choose the most suitable operation from these ones, depending on the specific machining case.

For example, if you need more flexible control of a tool position in each point of the toolpath, we recommend you to use "Contour spraying" with Custom tool vectors feature.

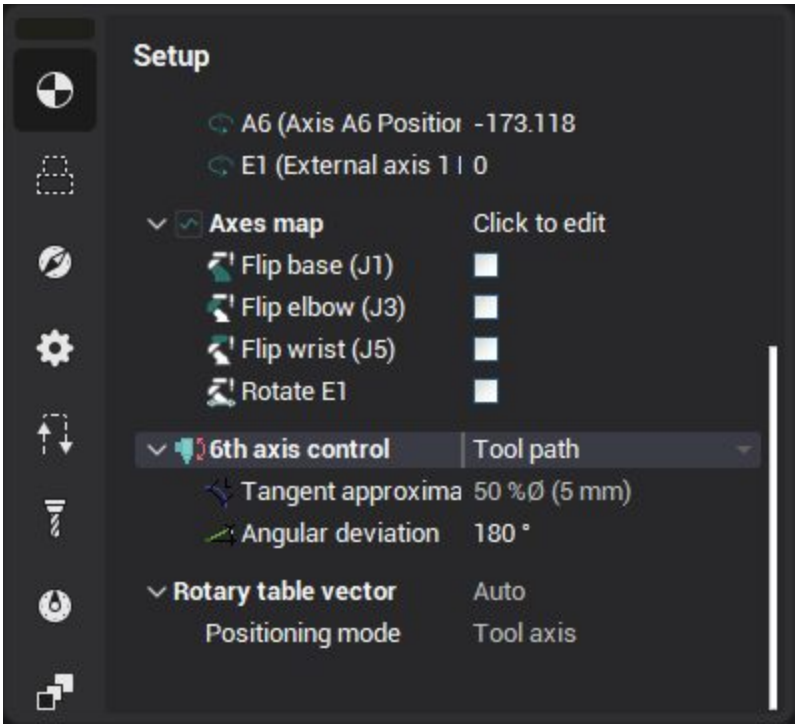


When using a tool that has a non-circular shape of the fan (for example, elliptic fan spray gun), it becomes important to maintain the specified orientation of the tool relative to the direction of the

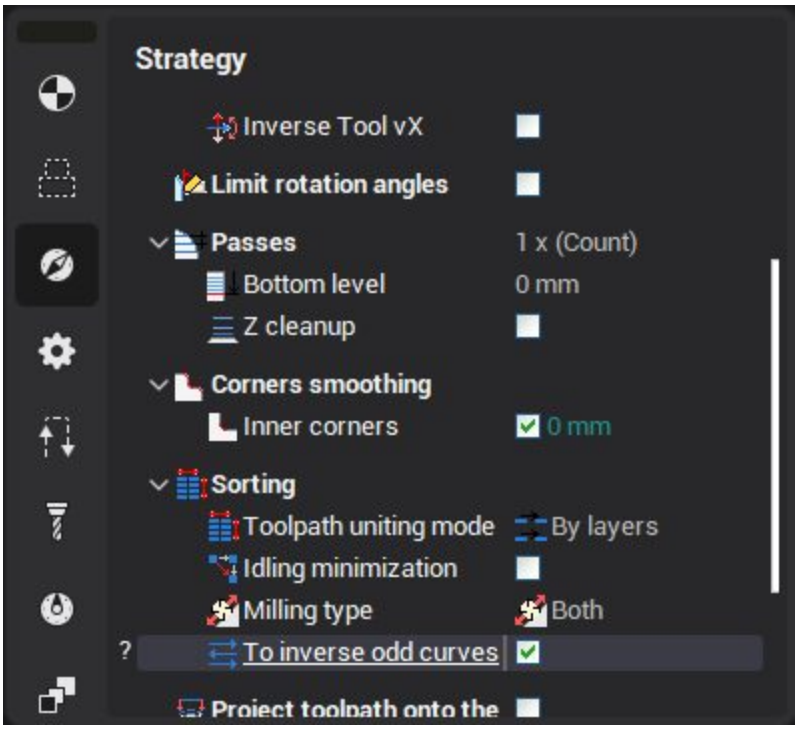
toolpath. This applies to equipment that has a large number of degrees of freedom, such as industrial robots. It is able to additionally rotate the tool around its axis.



In order for the tool orientation to be preserved relative to the toolpath, you need to activate the Toolpath mode of the 6th axis control property.

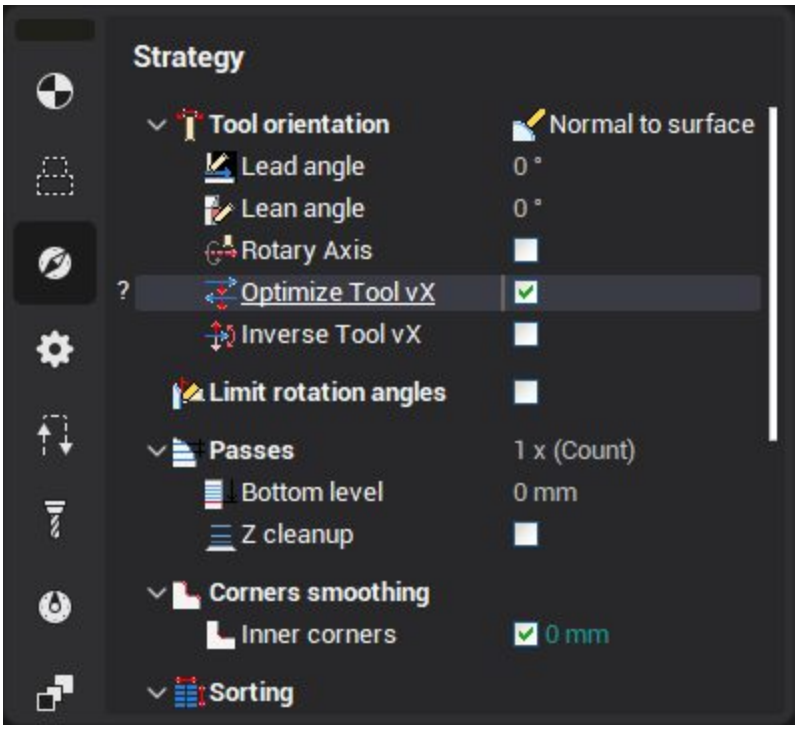


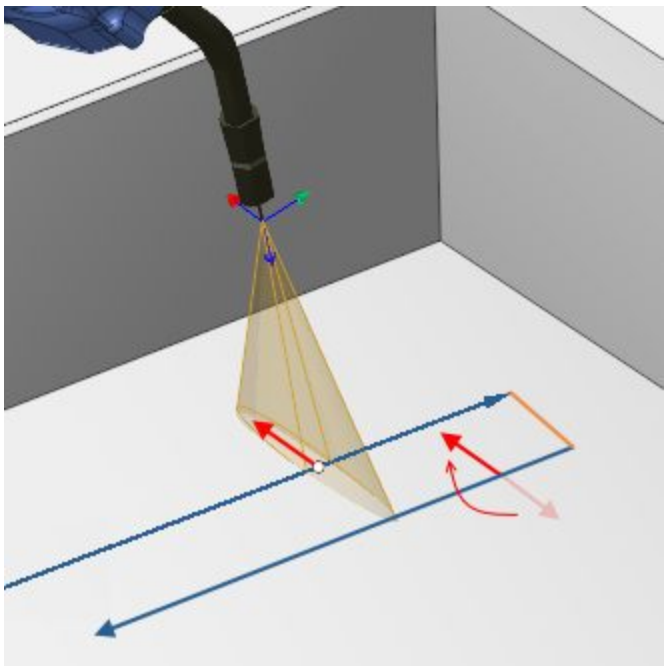
Especially for spraying purposes, a new option has been added to the " Contour 6D" and "Contour spraying" operations - To inverse odd curves. It allows you to automatically invert the direction of odd passes without changing the tool orientation angle relative to the path tangent manually.



Also a new option has been added to the "Contour 6D" and "Contour spraying" operations - Optimize Tool vX.

It allows you to automatically invert the direction of the tool's X vector in the same direction for all passes to minimize rotation.

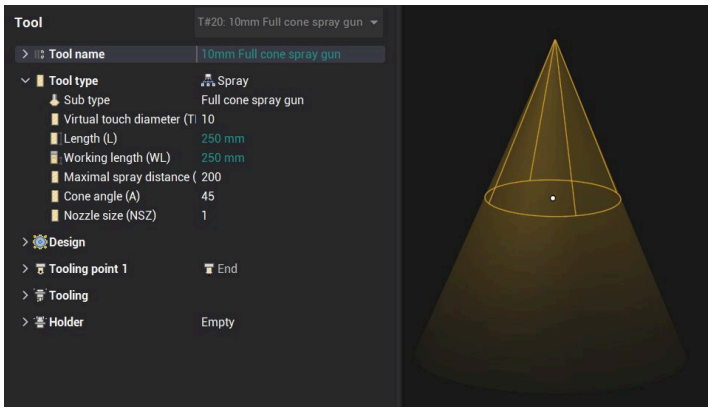
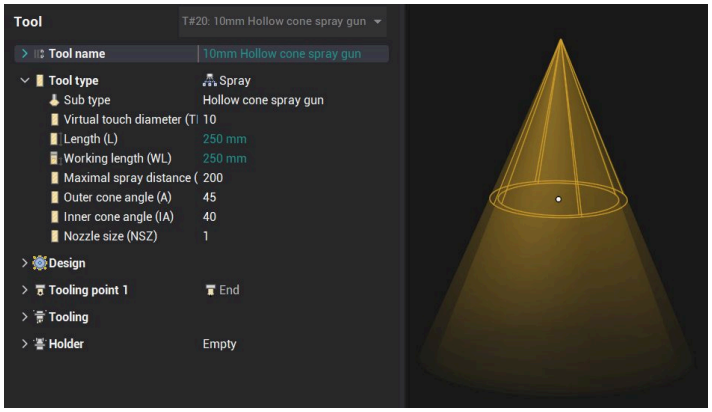
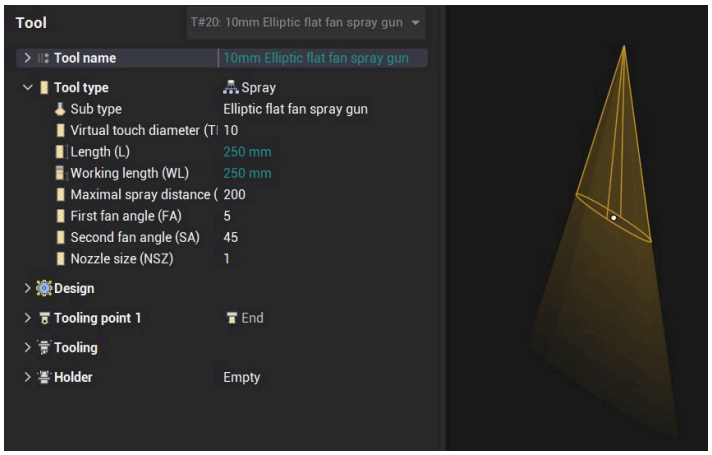


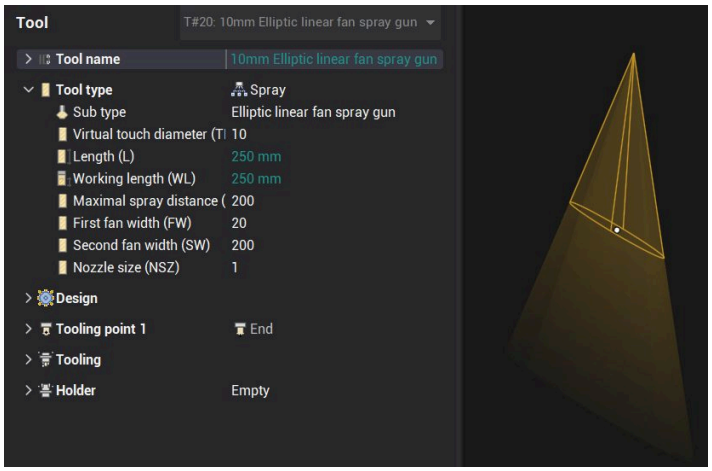
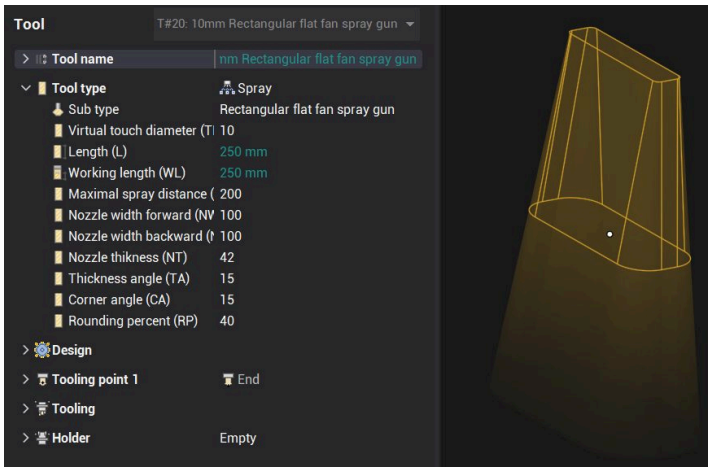


Spray tools

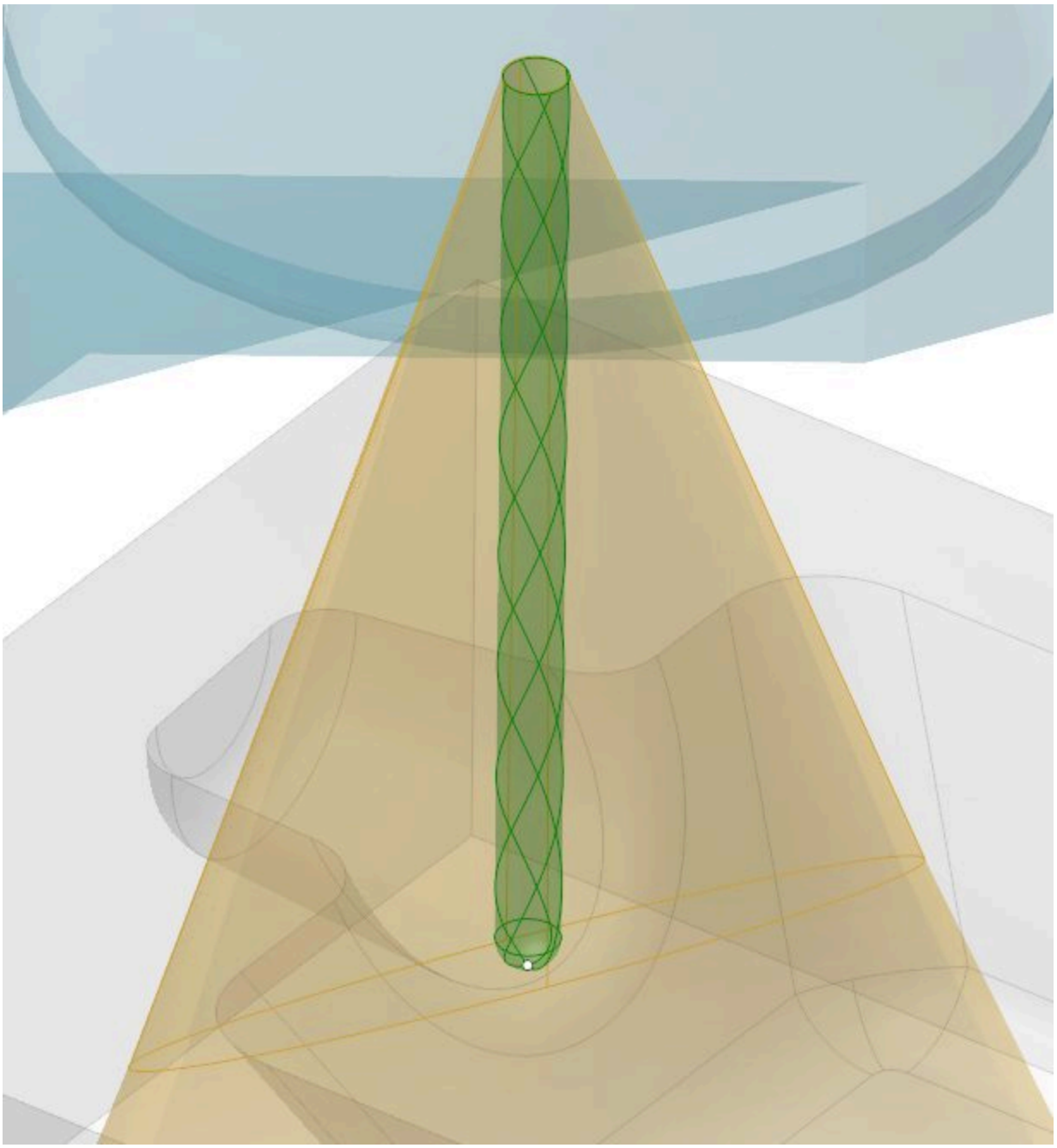
There is a special group of spray tool types in the system. It contains a few spray guns with a different shape.

You can read about this types in the table below.

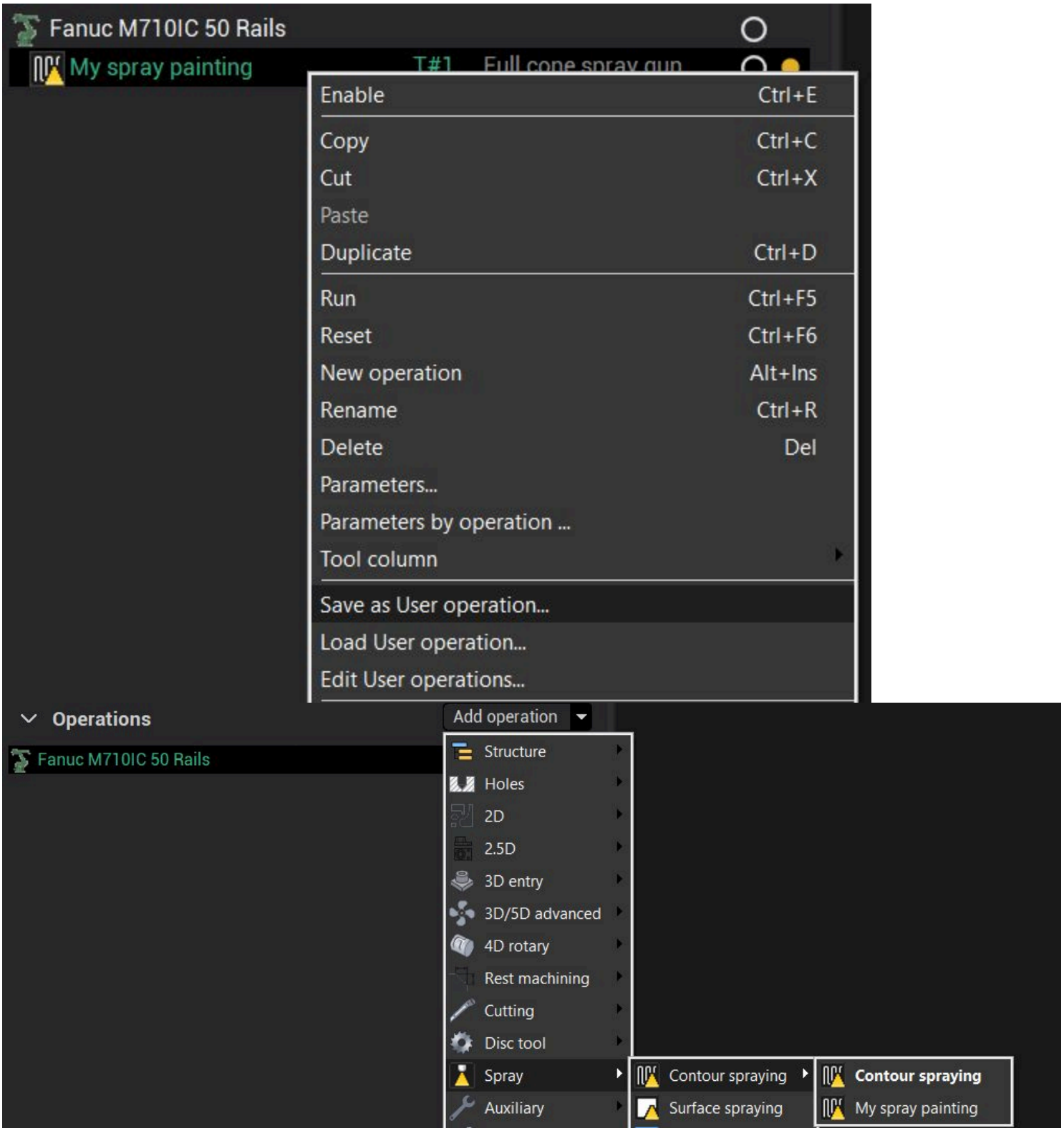
Tool type name	Picture
Full cone spray gun. It paints approximately evenly over the entire area of the circle.	
Hollow cone spray gun. It does not paint in the center, only a narrow strip along the edge works.	
Elliptic flat fan spray gun. The elliptical shape of the fan is given by two separate angles of inclination along and across the main direction. Use the Mounting angle property to rotate the main direction.	

Tool type name	Picture
Elliptic linear fan spray gun. The elliptical shape of the fan is given by two separate linear widths of the area at working length along and across the main direction. Use the Mounting angle property to rotate the main direction.	
Rectangular flat fan spray gun. It's a versatile tool. By changing its properties, you can flexibly adjust the shape of the torch. Allows you to simulate the case of several nozzles lined up in a row. Use the Mounting angle property to rotate the main direction.	

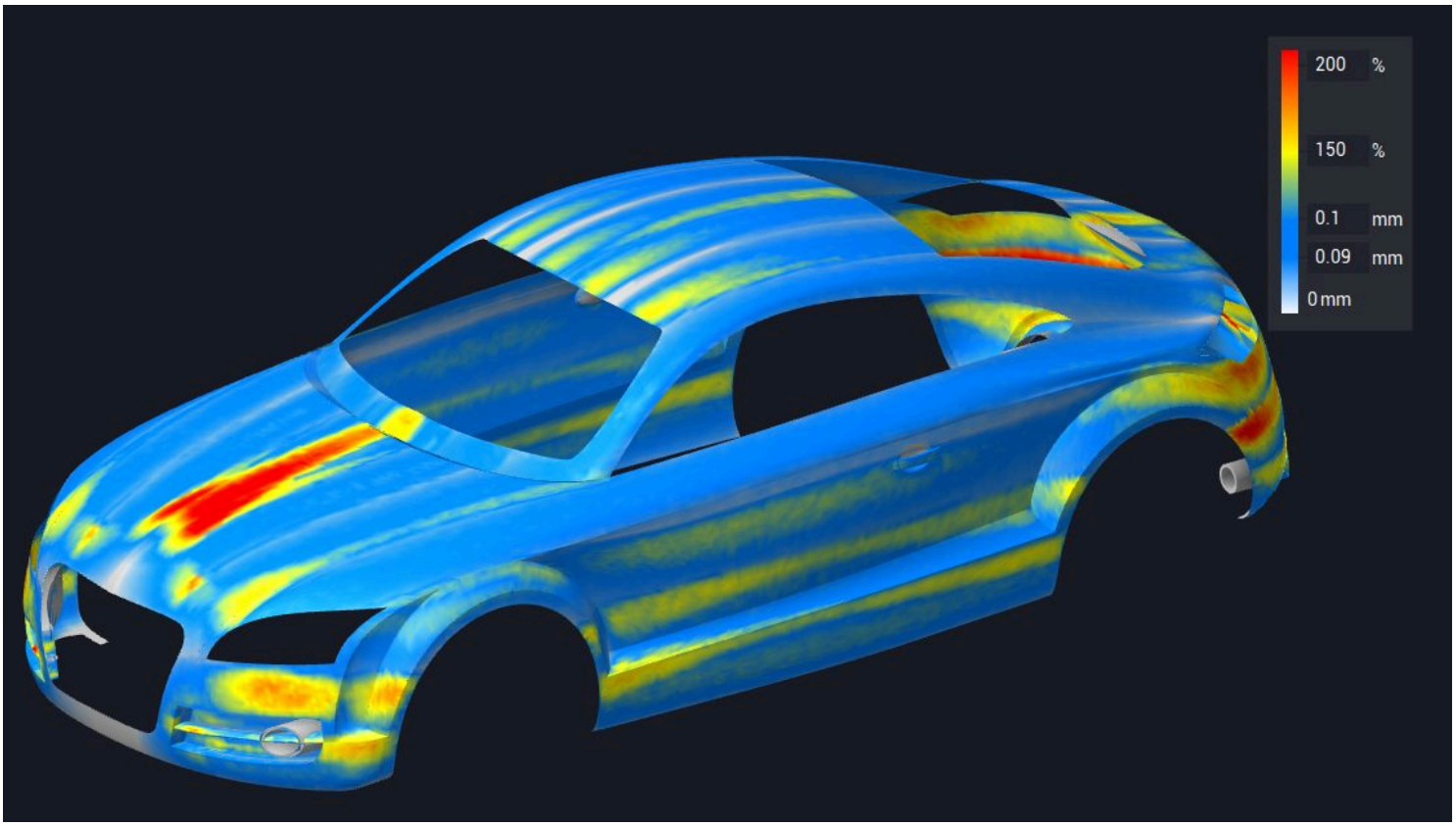
All spray tools contain the Virtual touch diameter property. It is a diameter of virtual spherical tool, like shown on the picture below. It is used in toolpath calculation to calculate tool-to-part contact. This makes it possible to use an existing universal strategies for toolpath calculation.



After a single setting of all the necessary parameters of the operation for specific purposes, in this case for spray painting, then you can easily save this operation as a template that can be easily used many times later in the same or other projects. To do this, select the Save as user operation item in the context menu of the operation. See the User operations section for more information about this feature.



After the operation toolpath is ready, you can go to the Simulation tab to check the result of painting. See Painting simulation section for more details.


200 %
150 %
0.1 mm
0.09 mm
0 mm

Legacy operations

Application area:

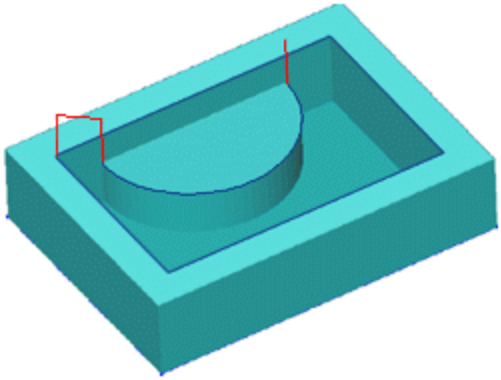
This section provides reference information for legacy machining operations, including 2.5D chamfering, cover machining, pocketing, wall machining, finishing plane, optimized plane, complex machining, combined waterline-drive machining, drive finishing and roughing, and sawing.

Use these topics when reviewing or modifying a project that contains one of these operations. Each topic describes the operation's purpose, geometry requirements, and machining parameters, helping to explain how its toolpath is generated.

The following operations are also documented in this group:

- Finishing plane. Generates surface finishing passes in parallel planes. Details
- Optimized plane. Combines two perpendicular plane strategies, with each strategy machining areas selected by their surface slope. Details
- Complex. Combines plane and waterline finishing for flat and steep areas. Details

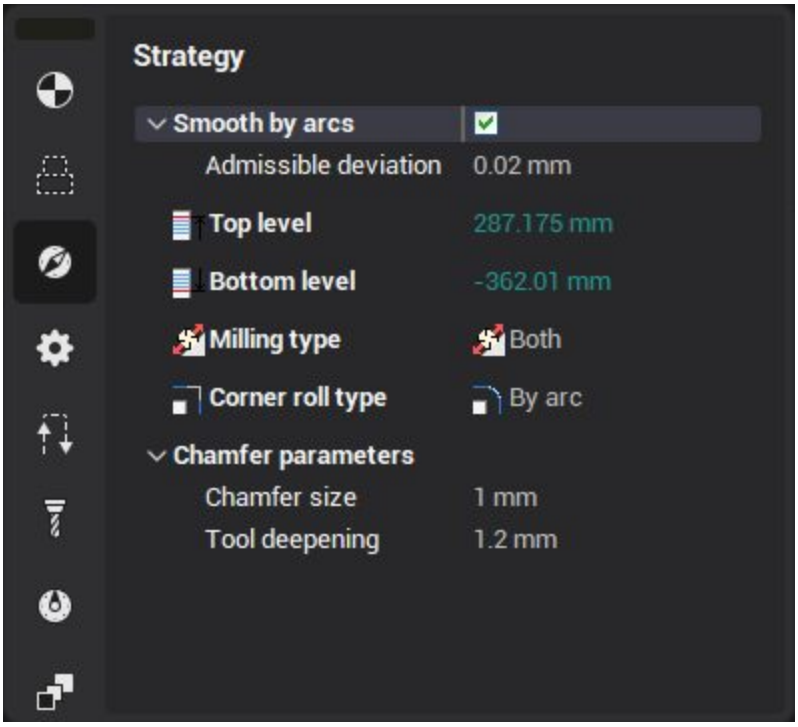
2.5D chamfer machining operation



This operation is designed for chamfering or rounding of horizontal edges of a model.

The model creation method for this operation is identical to the model creation method for the other 2.5D machining operations.

Having created the model the user opens the chamfer parameters window by pressing the operation Parameters button.

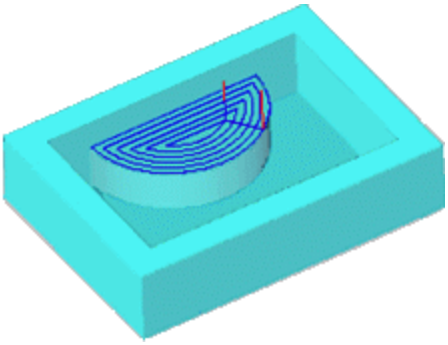


The default for all elements of the model is for no chamfer to be produced, which means that in the chamfer machining operation these curves will be ignored. For a chamfer to be machined it is necessary that the required curve has a chamfer value assigned in the Chamfer type dialogue. The size of the chamfer is assigned by two values, the height of the chamfer – is the distance from the top part of the element to the end point of the tool, and the width of the chamfer.

The same should be done for all elements that need to be machined in the operation. The sequence of actions can be performed either on one curve or on a group of curves, if the chamfer parameters for them are the same.

For the operation the user can also assign milling type, corner roll type, curves approximation and step-over type.

2.5D cover machining operation



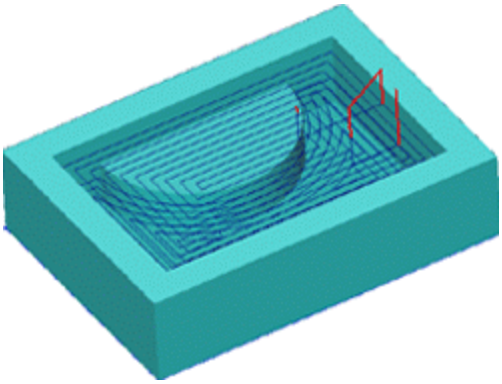
This operation is designed for the machining of horizontal areas of a model – these are known as "covers".

The method for defining the model for machining of horizontal areas is identical to the model definition method for other 2.5D machining operations.

The operation performs removal of workpiece material, which is located above the horizontal areas of the model being machined and outside any restricted areas. The workpiece, machining areas and the restricted areas are assigned by projections of closed curves.

Depending on the defined strategy, the material of a layer can be removed using spiral strokes, starting at the center moving outwards, from the outside inwards, or by parallel passes. In the operation the user can also define milling type, corner rolling and corner smoothing.

2.5D pocketing operation



The operation is used for machining of pockets and islands, and for preliminary material removal.

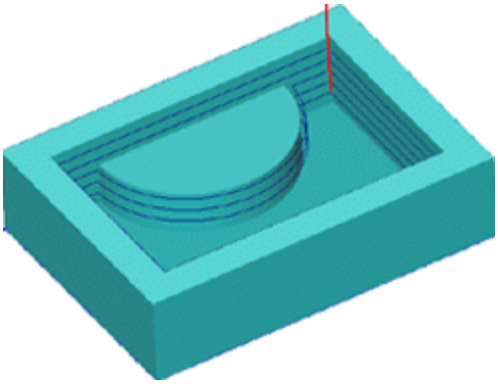
The model being machined is formed from the visual model that has been created from a set of flat curves and points. The visual model is created by successive addition of curves or groups of curves into a model. Any curve can define a ridge or a ditch of a defined thickness by means of the additional stock; closed curves can be added as a ledge or a cavity. Additional stock can be assigned for every curve or groups of curves.

In the operation the system performs removal of the entire workpiece material, which is located outside the model being machined and the restricted areas. The workpiece, machining areas and the restricted areas are defined by projections of closed curves.

Material is removed layer by layer, using the defined step between layers. Depending on the defined strategy, the material of a layer can be removed using spiral strokes, starting at the center moving outwards, from the outside inwards, or by parallel passes. Plunge to the next machining depth can be performed either by one of plunge methods (axial, spiral, zigzag), or through

With drill points, the system will first search for a hole of an appropriate diameter and depth in the holes list for the operation, and then in the open holes list of the machining process. If no appropriate hole is found, then coordinates for the hole center will be created automatically by the system. The coordinates of the center of the new hole, if possible will be round numbers (integers).

2.5D wall machining operation



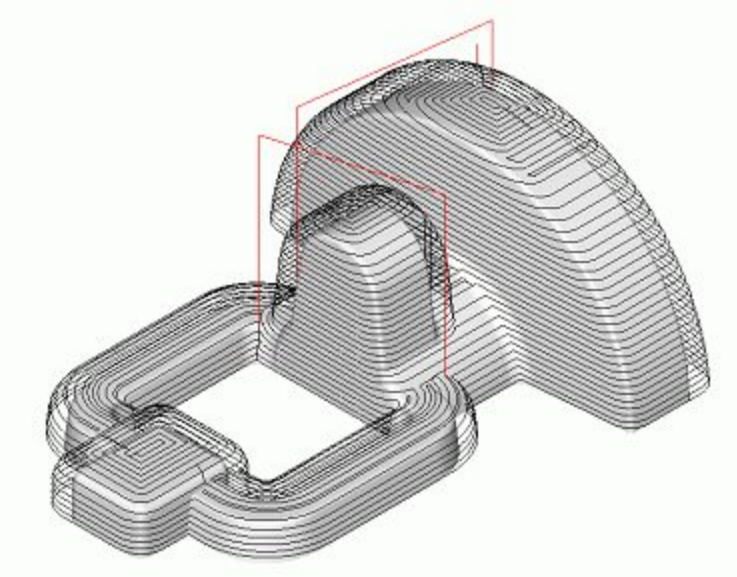
This operation is designed for machining the vertical walls of a models.

The method of creation of the model being machined is described in detail in chapters 2.5D pocketing operation.

The operation performs removal of workpiece material, which is located along walls of the model being machined and outside any restricted areas. The workpiece, machining areas and the restricted areas are assigned by projections of closed curves.

The material is removed in layers, using the defined step between layers. In the operation strategy the user can define the milling type, the corner roll type and corner smoothing if required.

Combined operation (waterline-drive)



By using the combined operation, an equally good finish can be achieved on both flat and steep areas. A proportionally even scallop height can be obtained even when using a fixed step. The combined strategy provides easier conditions for the cutter; this makes it possible to use longer tools with a smaller diameter. The operation gives good quality finish machining irrespective of the complexity of a models surface angle, and also minimizes the machining time.

For every geometrical object or a group of objects, an additional stock can be defined, which during machining will be added to the main stock for the operation.

If a workpiece and a restricting model are not defined, then the system performs machining of the entire available surface of the model being machined. Otherwise only those surface areas will be machined, which lie within the workpiece and outside the restricting model.

The workpiece can be assigned as a cube, cylinder, a mould with stock or prismatic form, as residual material after machining by previous operations, and also as a free-form geometrical model, consisting of solid bodies, surfaces, meshes and prisms whose bases are projections of closed curves. In the restricting model, solid bodies, surfaces and meshes which are required to be controlled during machining, and also machining areas and restricted areas, defined by projections of closed curves can be defined.

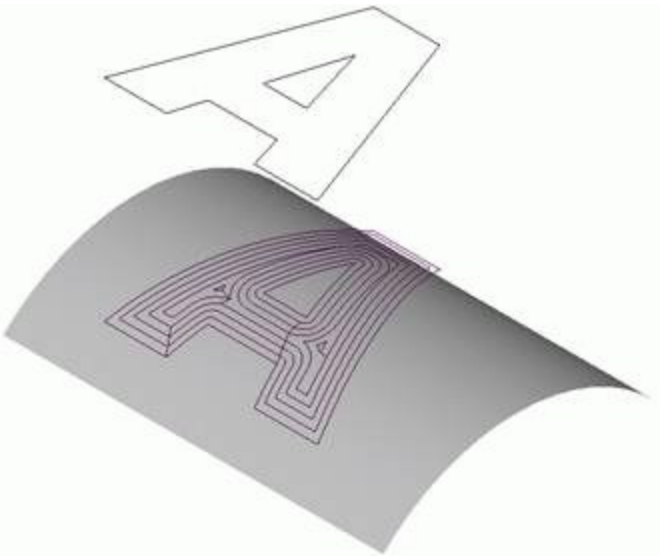
The toolpath for surface machining of a volume model is created in two stages. First, the horizontal tool paths are constructed (similar to the waterline operation), and then, by using the rules of the drive operation, toolpaths are created along a leading curve (leading curves in this case are the borders of the unmachined areas). Thus, models surface areas close to vertical are machined as a waterline operation, and flat – as drive finish. This allows the user to obtain proportionally good machining for models of virtually of any shape. The step between passes is assigned separately for the vertical plane and for horizontal plane, and also can be calculated from the defined height for the scallop.

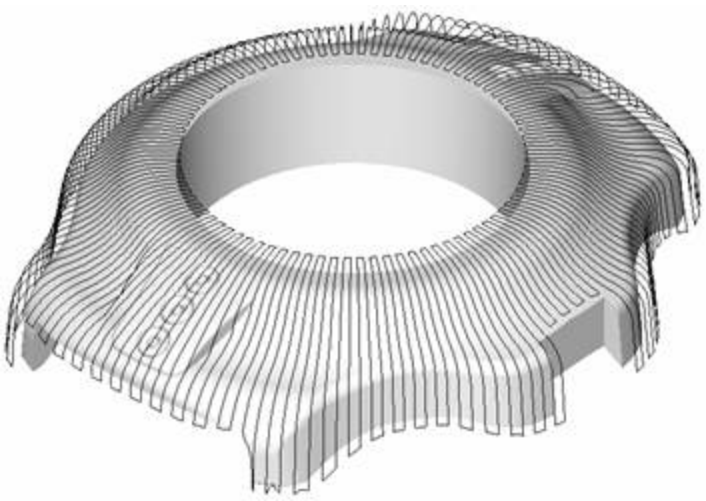
When using a local coordinate system or a rotary axis, the position of the model being machined will not change, the tool rotation axis is parallel to the Z axis of the local coordinate system, the horizontal passes are located parallel to the XY plane of the current coordinate system, and then any unmachined areas will be milled according to the rules of the drive operation.

It is also possible to restrict machining from entering areas of the restricting model and areas of edge rounding in the resulting toolpath.

Joining of the work passes into a single toolpath can be performed going downwards or upwards. Transition between neighboring work passes can be performed on the surface, using retract and approach moves or via the safe plane.

Drive finishing operation





The drive finishing operation is best used when machining separate areas of a model with complex prelate curvilinear surfaces. It is recommended for remachining areas of a model of a specific shape, for machining of some models with slightly changing surface geometry, and also for milling of inscriptions and drawings on the model surface. When using the drive finishing operation for machining of flat areas of a models surface, it is recommended to use the outer edges as the leading curves and the along curve strategy. When machining steep areas use the across curve strategy with the same leading curves.

For every geometrical object or a group of objects, an additional stock can be defined, which during machining will be added to the main stock for the operation.

If a workpiece and a restricting model are not defined, then the system performs machining of the entire available surface of the model being machined. Otherwise only those surface areas will be machined, which lie within the workpiece and outside the restricting model.

The workpiece can be assigned as a cube, cylinder, a mould with stock or prismatic form, as residual material after machining by previous operations, and also as a free-form geometrical model, consisting of

solid bodies, surfaces, meshes and prisms whose bases are projections of closed curves. In the restricting model, solid bodies, surfaces and meshes which are required to be controlled during machining, and also machining areas and restricted areas, defined by projections of closed curves can be defined.

As with the plane operation, machining of the surface of a volume model is performed using separate paths. Depending on the operation parameters, the paths lie either in the vertical plane (across leading curves) or in vertical mathematical 'cylinders', the shape and location of which are defined by the drive curves (along leading curves). The step between neighboring work passes can be either permanent or calculated regarding the defined height of the scallop.

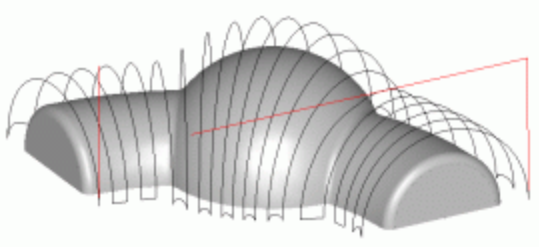
When using a local coordinate system or a rotary axis, the position of the model being machined will not change, the tool rotation axis is parallel to the Z axis of the local coordinate system, and all work passes are located in planes or mathematical cylinders, perpendicular to the horizontal plane of the local coordinate system.

The areas of the surface model being machined can be limited depending on the slope angle of the normal to the Z-axis. If for example, the user needs to machine flat areas that have a slope angle to the surface normal of less than 30 degrees, then it is advised to set the values for the minimum and maximum slope angles to 0 and 30 degrees accordingly.

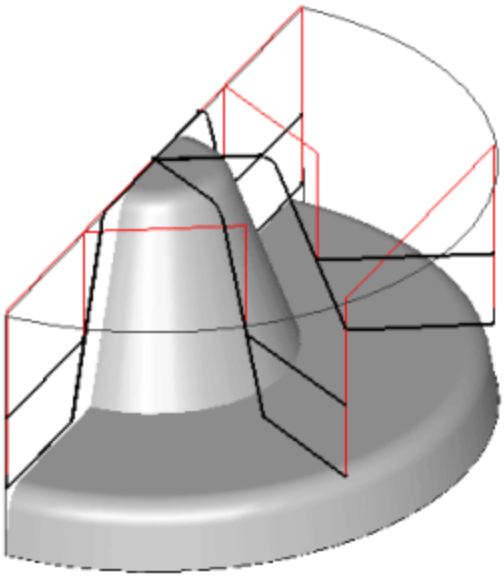
In order to machine only areas that have a small deviation from the normal to the plane of the work pass, it is advised to limit the frontal angle. For example, if one needs to perform machining of surface areas that are nearly perpendicular to the surface of a work pass, then it is advised to set a smaller value for the frontal angle (e.g. 5 degrees).

It is also possible to restrict machining from entering areas of the restricting model and areas of edge rounding in the resulting toolpath.

Joining of the work passes into a single toolpath can be performed going downwards or upwards. Transition between neighboring work passes can be performed on the surface, using retract and approach moves or via the safe plane.



Drive roughing operation



In some cases a model after machining with drive curve roughing can be very close to the required finished model, however, due to the unevenness of the volume of the material being removed it is not always possible to reach the optimum machining time. The drive roughing operation is recommended for use when a model's periphery (outer edge) is lower than the center and the outer workpiece contour is similar to the model contour.

A model being machined using the drive roughing operation is assigned by a set of solid bodies, surfaces and mesh objects. For every geometrical object or a group of objects, an additional stock, which during machining will be added to the main stock of the operation, can be defined.

The workpiece can be assigned as a cube, cylinder, a mould with stock or prismatic form, as residual material after machining by previous operations, and also as a free-form geometrical model, consisting of solid bodies, surfaces, meshes and prisms whose bases are projections of closed curves. In the restricting model, solid bodies, surfaces and meshes which are required to be controlled during machining, and also machining areas and restricted areas, defined by projections of closed curves can be defined.

The operation performs removal of the entire material of the workpiece, which lies outside of the model being machined and outside the restricting model. As in the plane operation, separate paths are used to perform surface machining of the volume model. Depending on the operation parameters, the work passes lie either in vertical planes (across leading curves) or in vertical mathematical cylinders, the shape and location of which are defined by the leading curves (along leading curves). The step-over between the toolpaths of neighboring work passes can be either fixed or calculated according to the defined height of the scallop. To limit the pressure on the tool, the depth of cut (Z axis) can be limited. That is, if the thickness of the workpiece material being removed exceeds the user defined depth, then the material will be removed in several passes.

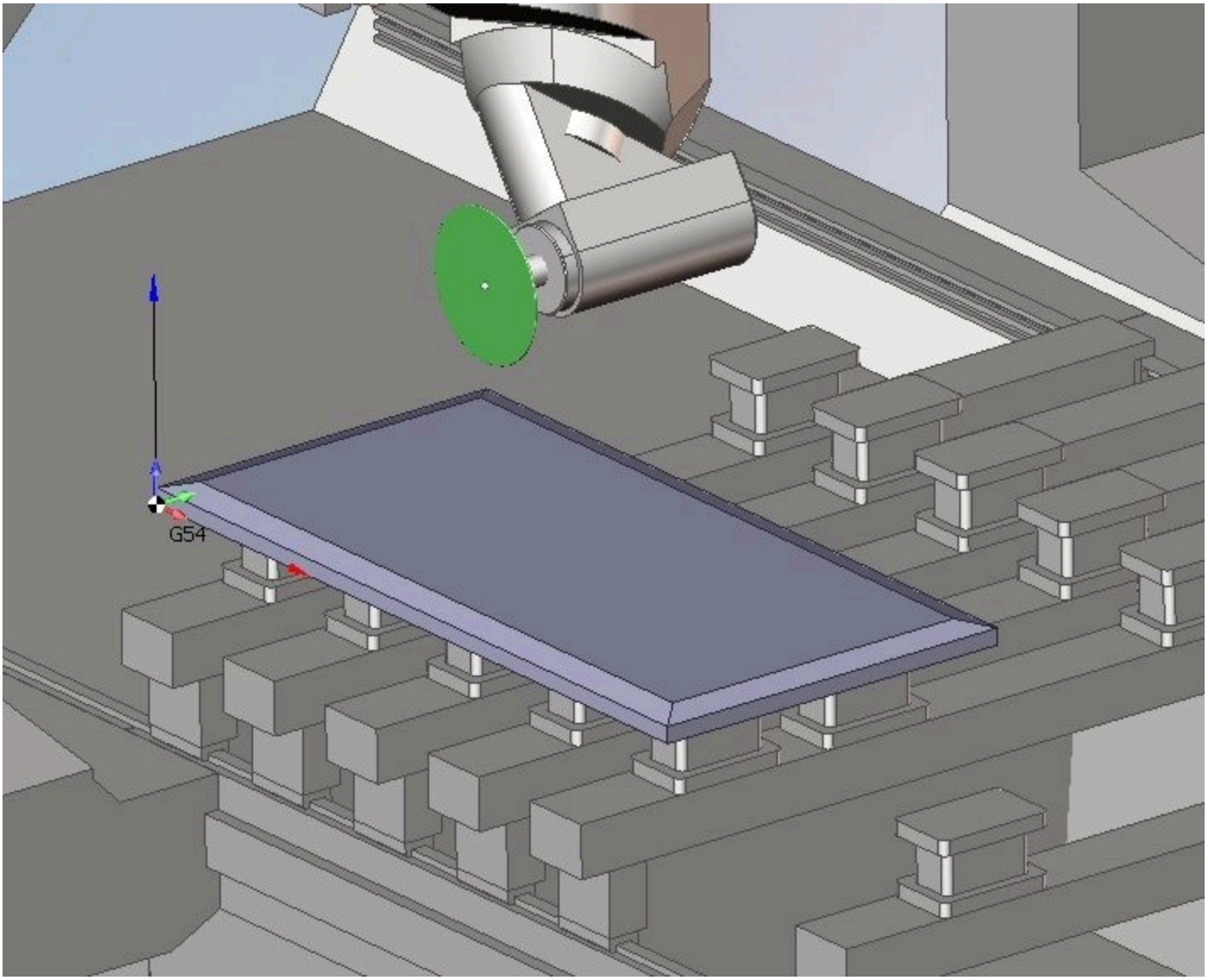
When using a local coordinate system or a rotary head, the position of the model being machined will not change, the tool rotation axis is parallel to the Z axis of the local coordinate system, and all work passes are located in planes or mathematical cylinders, perpendicular to the horizontal plane of the local system.

If during machining, the tool must not cut any material that is over a user-defined angle, then the downward movement of the tool can be limited. The available types of limitation are: machining upwards only with maximum cutting angle without rest milling of the shadow areas, with a maximum cutting angle with rest milling of shadowed areas, and without downwards movement control.

Transition between work passes can be performed via the shortest distance, with the addition of approach and retract moves, or via the safe plane. If material removal is performed is divided into depths, then the system first removes the entire material at the first depth before starting on the next one.

There is can to be used quick calculation method also.

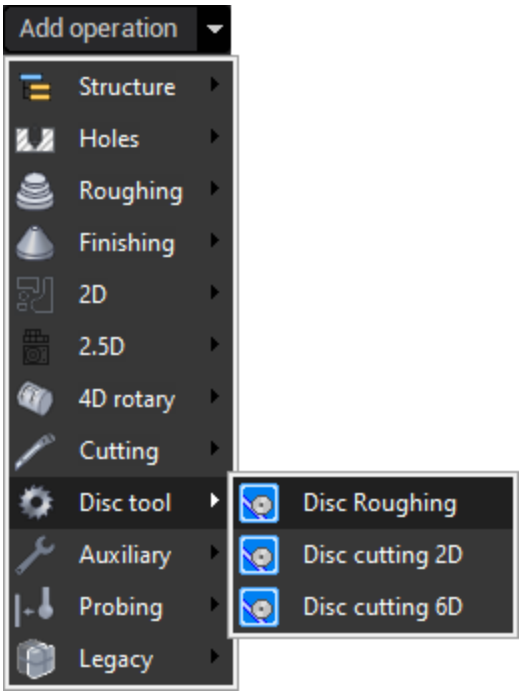
Saw operation



The operation is designed for the circular saw cutting on the 5-axis milling machines or robots with spindle. The sawing operation is based on the "5D contour" operation. Only few changes are made for the correct approach and retract or the disc tool. Source data for programming is a solid. Machining of the curves taken for example from.dxf are not supported.

Way of use:

1. Create sawing operation.

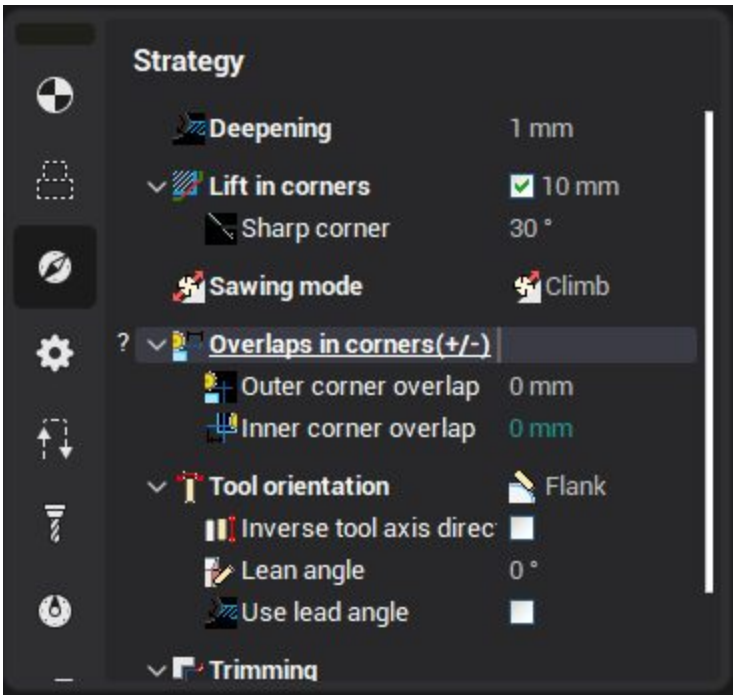


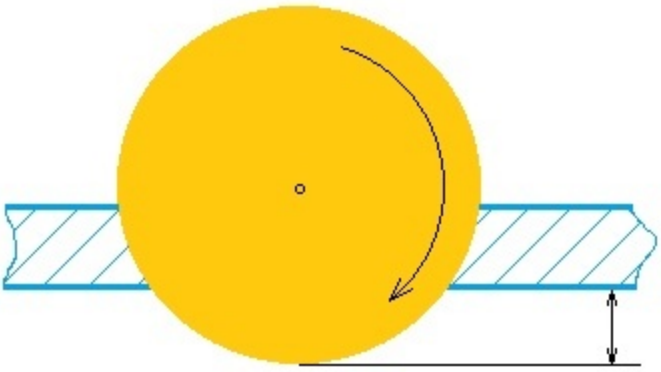
2. Define the required tool size and speeds, the same way as it can be done for all other operations.

3. To define the job assignment it's necessary to choose the edge of a solid and press "Edge/Curve on surface". Selected edge will be added into the job list. One of the adjacent surfaces will be highlighted by green color. This face will be formed as the result of sawing. If you need to make another adjacent face, you have to click on edge once more, after that the panel with two buttons appears. Press "change the side" button.



4. Unlike "5D contour" there is the additional parameter in sawing operation to define the overlap.

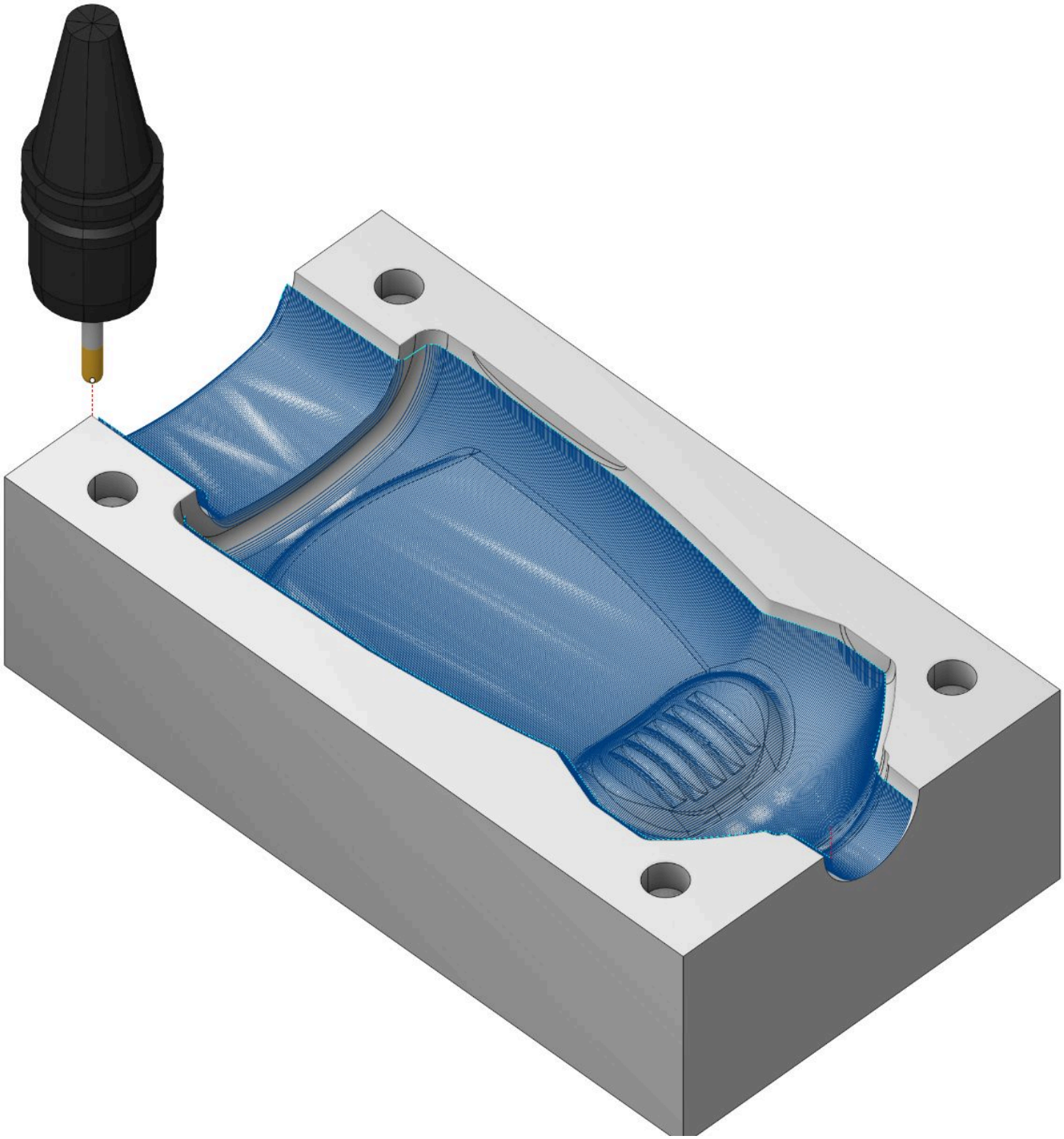




See example of operation on YouTube:

http://www.youtube.com/watch?v=op5vH1P-V3s

Finishing plane



Application area:

The plane finishing operation is designed for the machining of smooth areas of a model's surfaces, and also for areas close to vertical, whose (steep) trajectories are along the toolpath. For further remachining of other areas, it is better to use the waterline operation (Details) or another plane operation with a toolpath, which is perpendicular to that of the first operation.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

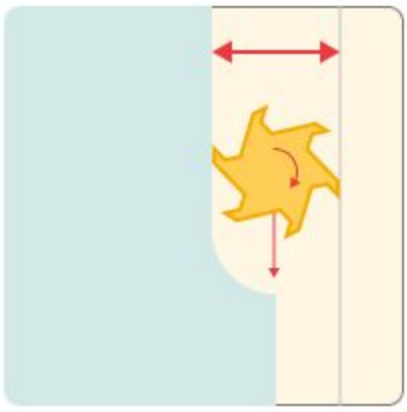
Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

Step.

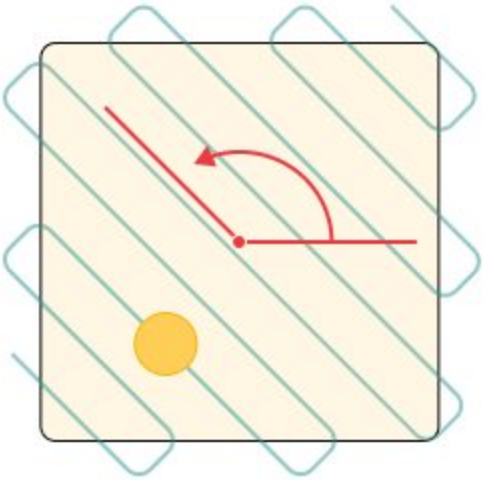
The maximum allowed width of cut.

▼ Click here to expand..



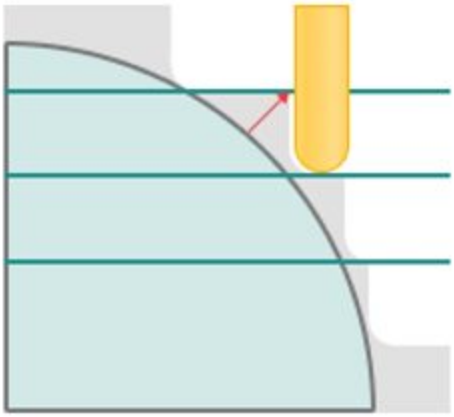
S - Step

Angle. Assigns the cut angle direction for the plane operations.



Scallop. If disabled: the distance between passes is calculated using the Step parameter only.

If enabled: the step of machining is calculated automatically based on the maximum given height of scallop. Where necessary the tool passes will be generated more closely to each other to achieve the desired scallop height.



Machining levels.

It defines the range along tool axis for the machining. This parameter group works similarly to the Waterline roughing operation. Details

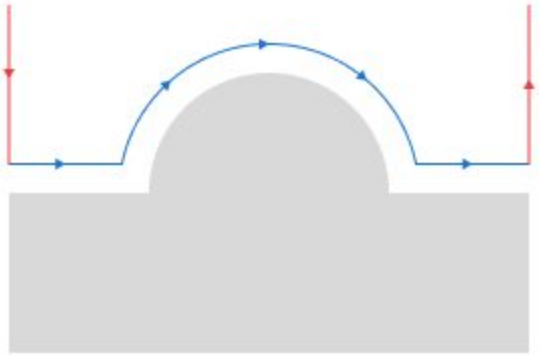
Milling type

Can be assigned in almost all operations, except for the curve machining operations. This allows the user to control the required milling type (climb or conventional) during the toolpath calculation process. This parameter group works similarly to the Waterline roughing operation. Details

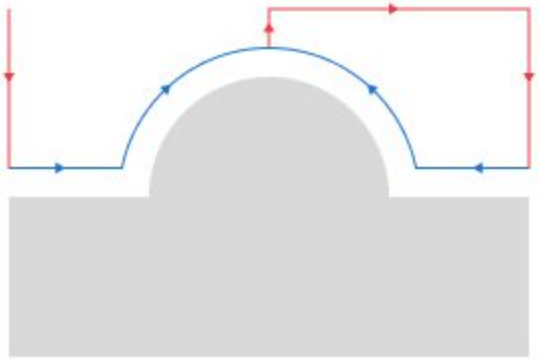
Sorting.

Controls the sequence of toolpath passes during the surface machining.

▼ Click here to expand..
Upwards only. If disabled: the tool is allowed to move both in up and down directions.



If enabled: the tool is allowed to move only upwards. Where necessary the tool passes are split and relinked.

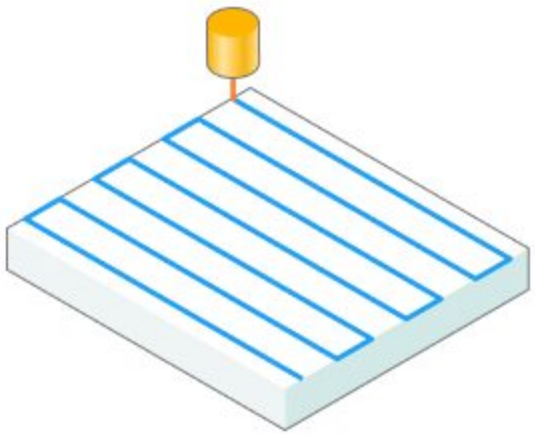


Job zone.

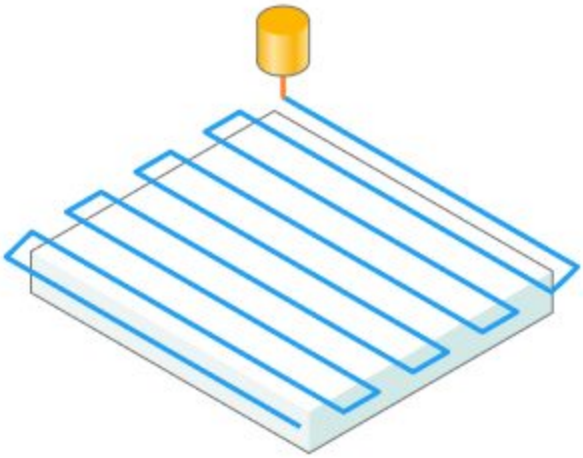
Enables additional modification of the working area initially selected on the Job assignment tab.

▼ Click here to expand..
Surface Roll Type. Surface rolling method.

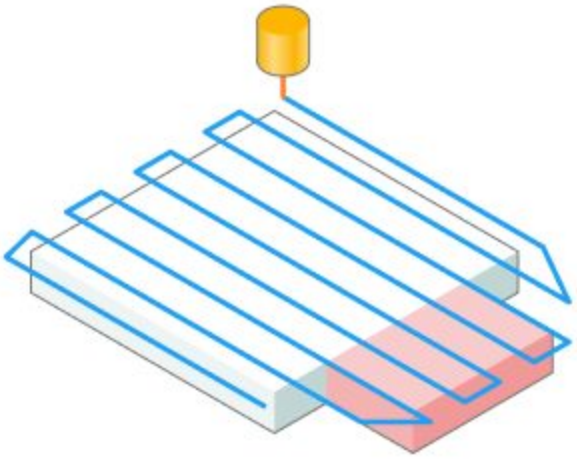
Surface only. It machining only on selected surfaces.



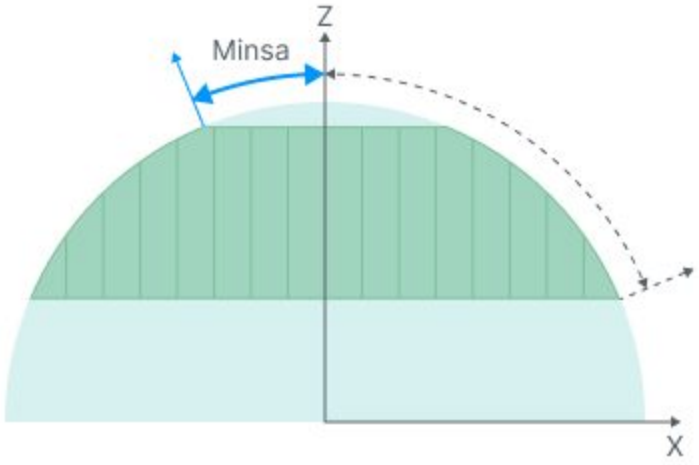
With edges. It machining on selected planes and with edges. In the With edges mode, the resulting toolpath consists of the machining areas of the form-creating surfaces and the areas of edges rolling. This mode could be used for instance, to make edges round when machining a model with a positive stock.



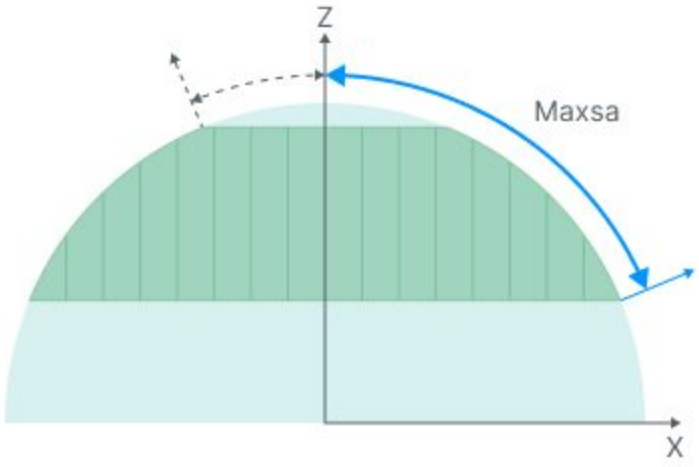
With restriction model. It machining on selected plane,restriction and with edges. When using the With restricting model mode, all machining and rolling areas of the model and the restricting model will be included in the resulting toolpath.



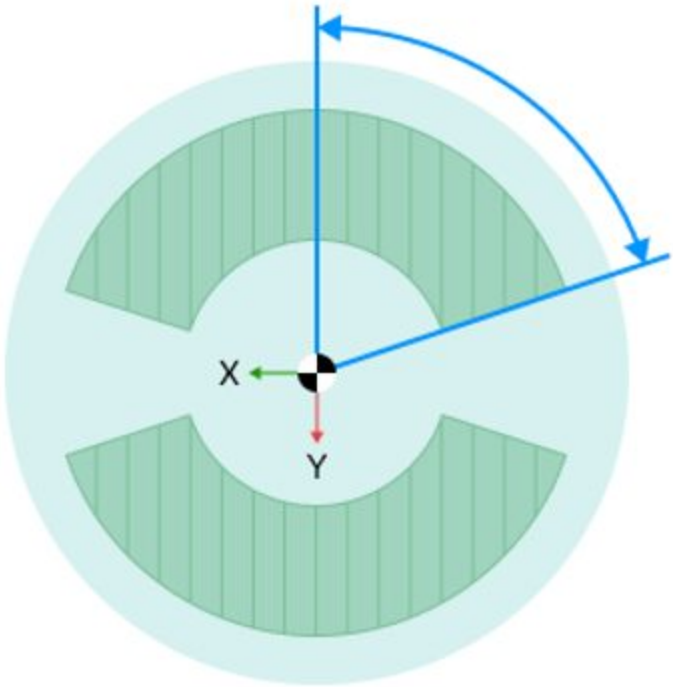
Min slope angle. In the system it is possible to perform selective machining of the model surface areas, depending on the angle between the normal to the surface and the vertical axis Z. The range being machined can be defined by the minimum and maximum slope angles.



Max slope angle. In the system it is possible to perform selective machining of the model surface areas, depending on the angle between the normal to the surface and the vertical axis Z. The range being machined can be defined by the minimum and maximum slope angles.



Frontal angle. The frontal angle is the angle between projections onto the horizontal plane of the tool movement direction and the normal to the surface at the cutting point. Details



Corners smoothing.

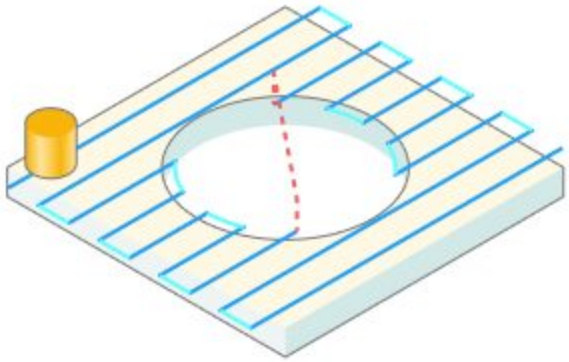
When there is a sudden change of the tool direction, the milling control performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the toolpath has very few or no breaks. For this reason, in the system there is the toolpath smoothing function using the defined radius for machining inner or outer corners of the model.

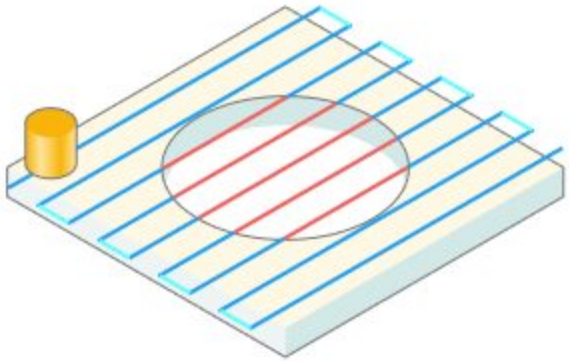
Trimming.

Allows for the skipping of surfaces not intended for machining and allows for the extension of the toolpath. Some parameter group works similarly to the Waterline roughing operation. Details

▼ Click here to expand..

Gap capping. When enabled, gaps along cut with the size less than the given value are connected with lines.





Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details.

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

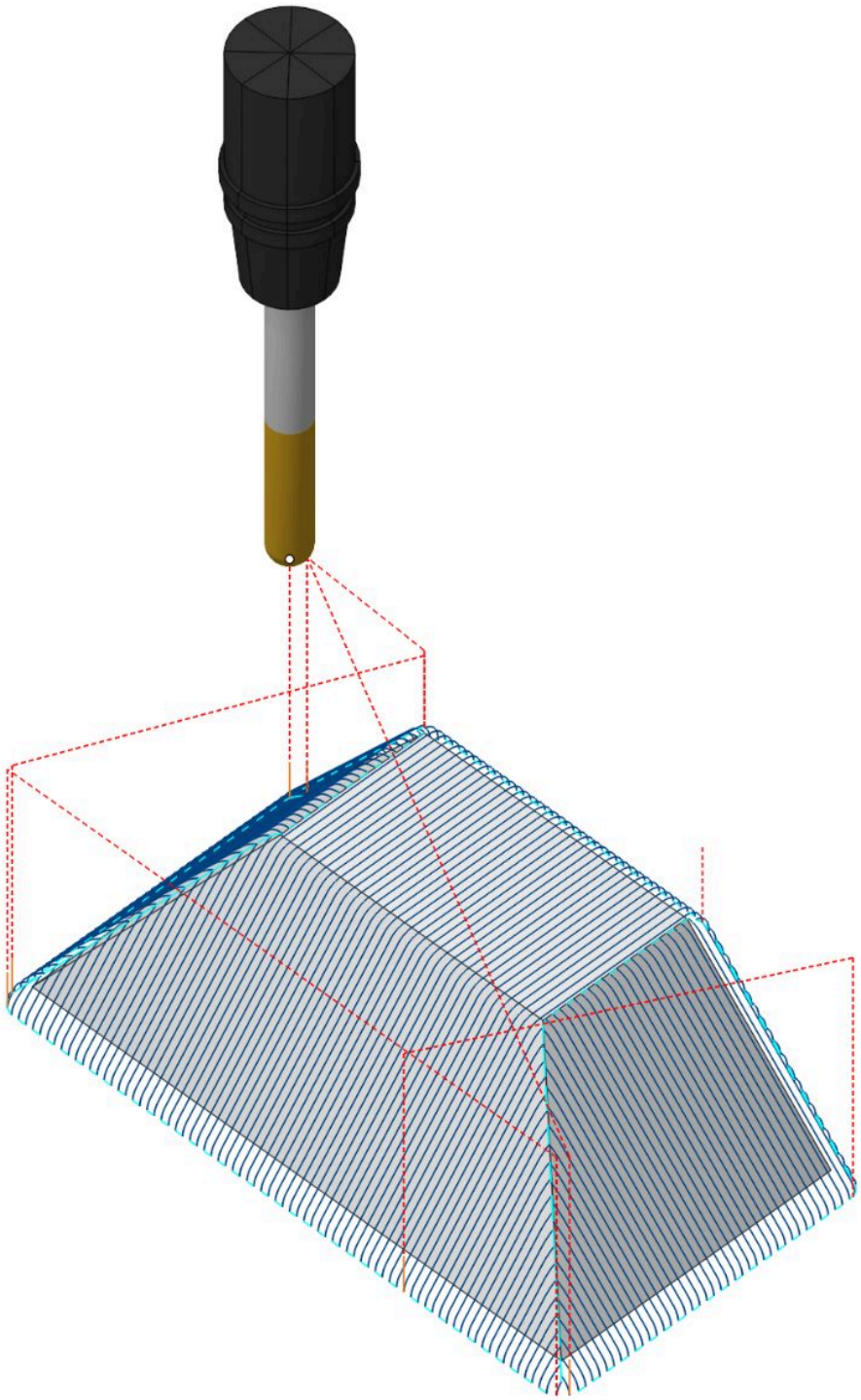
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Optimized plane



Application area:

The optimized plane toolpath consists of two finishing plane passes lying in mutually perpendicular planes. Each pass machines only those areas where the frontal angle of the surface slope measured

relative to the cutting direction is 45 degrees or less. This ensures that no surface is machined twice. And thanks to that a consistent scallop height across entire part is achieved. This makes optimized plane a good choice for high quality finishing machining of complex parts.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

Strategy:

This parameter group works similarly to the Plane finishing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details

Distances. Distances required to switch feeds Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

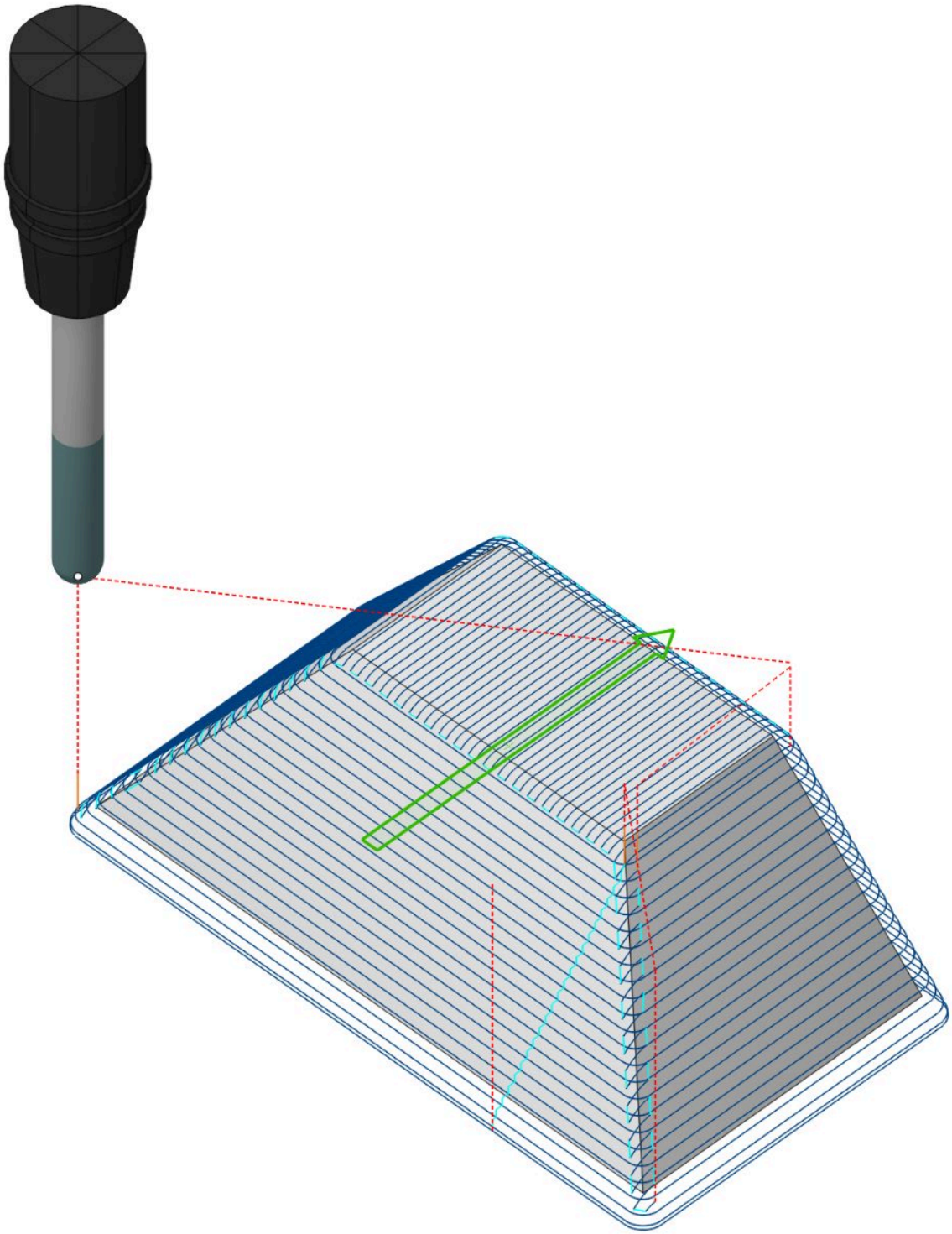
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Complex



Application area:

The complex operation consists of two toolpaths: plane and waterline. The plane toolpath machines only areas with the slope angle less than the Split slope angle while the waterline toolpath machines areas

where the surface slope angle is greater than the Split slope angle. By default the split slope angle is set to 45 degrees. This strategy ensures that shallow areas are machined with the plane toolpath, and steep areas are machined with the waterline toolpath. The benefits of such an approach are optimal cutting conditions, consistent scallop height and reduced machining time.

Setup:

The Setup tab is used to configure the primary parameters of the project. This can involve the positioning of the part on the equipment, the coordinate system of the part, and more. Details

Job assignment:

Faces. Select various surfaces of the part as the working task. The system will calculate the trajectory based on the chosen surfaces. Details

Pocket. A typical prismatic or turn-milling part consists of many simple shapes.To simplify this task, we can recognize the parts elements in a 3d model and automatically convert them to basic job assignment items, such as job zones and machining levels. Details

Job Zone. Job zones are used to define the part areas that have to be machined by roughing and finishing milling operations. Details

Restrict Zone. In addition to Job Zones in system you can use Restrict Zones to specify the workpiece areas that have not to be machined in the current operation. Details

Top Level. Specifying the top level by model elements. Details

Bottom Level. Specifying the bottom level by model elements. Details

Properties. Displays the properties of an element. It is possible to add the stock. You can also call this menu by double clicking on an item in the list.

Delete. Removes an item from the list.

Restrictions. It allows you to restrict areas that should not be machined. Details

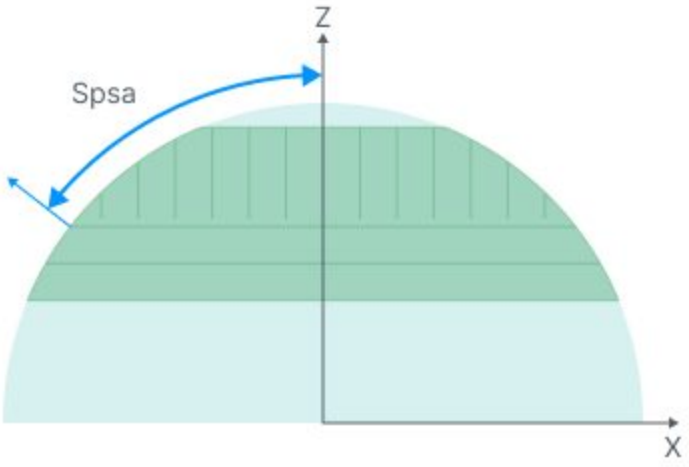
Strategy:

This parameter group works similarly to the Plane finishing operation. Details

Job zone

Enables additional modification of the working area initially selected on the Job assignment tab.

▼ Click here to expand..
Split slope angle. Angle between vertical and horizontal passages.



Spsa - Split slope angle

Some parameter group works similarly to the Plane finishing operation. Details

Parameters:

This tab contains the main parameters for controlling the part, workpiece, fixture and other items, which allow you to change the trajectory. Details

Links/Leads:

In the Links/Leads tab, you define the parameters for rapid movements. These movements include tool approach from the tool change position, engage to the start of the working stroke, retraction after the final cutting motion, transitions between working passes, and return to the tool change point. You can configure the sequence of movements along the coordinates, the trajectory of these motions, and the magnitude of displacements.

▼ Click here to expand..
Consists of:

Approach/Return. This group of parameters manages the tool movements from the tool change position to the start of the engage toolpathes, and back. Details.

Safe motions. Defines the type of Safety Surface and the distance from the workpiece to it. This surface facilitates transitions between the processed surfaces or tool paths, and the first stage of approach or return occurs toward it. Details.

Advanced links settings. This group of parameters specifies the features of trajectory formation for primarily five-axis operations, considering optimal movements of rotational axes. Details.

Links. Defines the rules governing Entry/Exit links and links between work moves. Details.

Leads. Allows you to add additional engage and retract to the working moves using their feed. Details

Distances. Details.

Feeds/Speeds:

Using this dialogue the user can define the spindle rotation speed; the rapid feed value and the feed values for different areas of the toolpath. Spindle rotation speed can be defined as either the rotations per minute or the cutting speed. The defining value will be underlined. The second value will be recalculated relative to the defining value, with regard to the tool diameter. Details

Transformations:

Parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool. Details

Part:

A Part is a group of geometrical elements that defines the space to check for gouges. Details

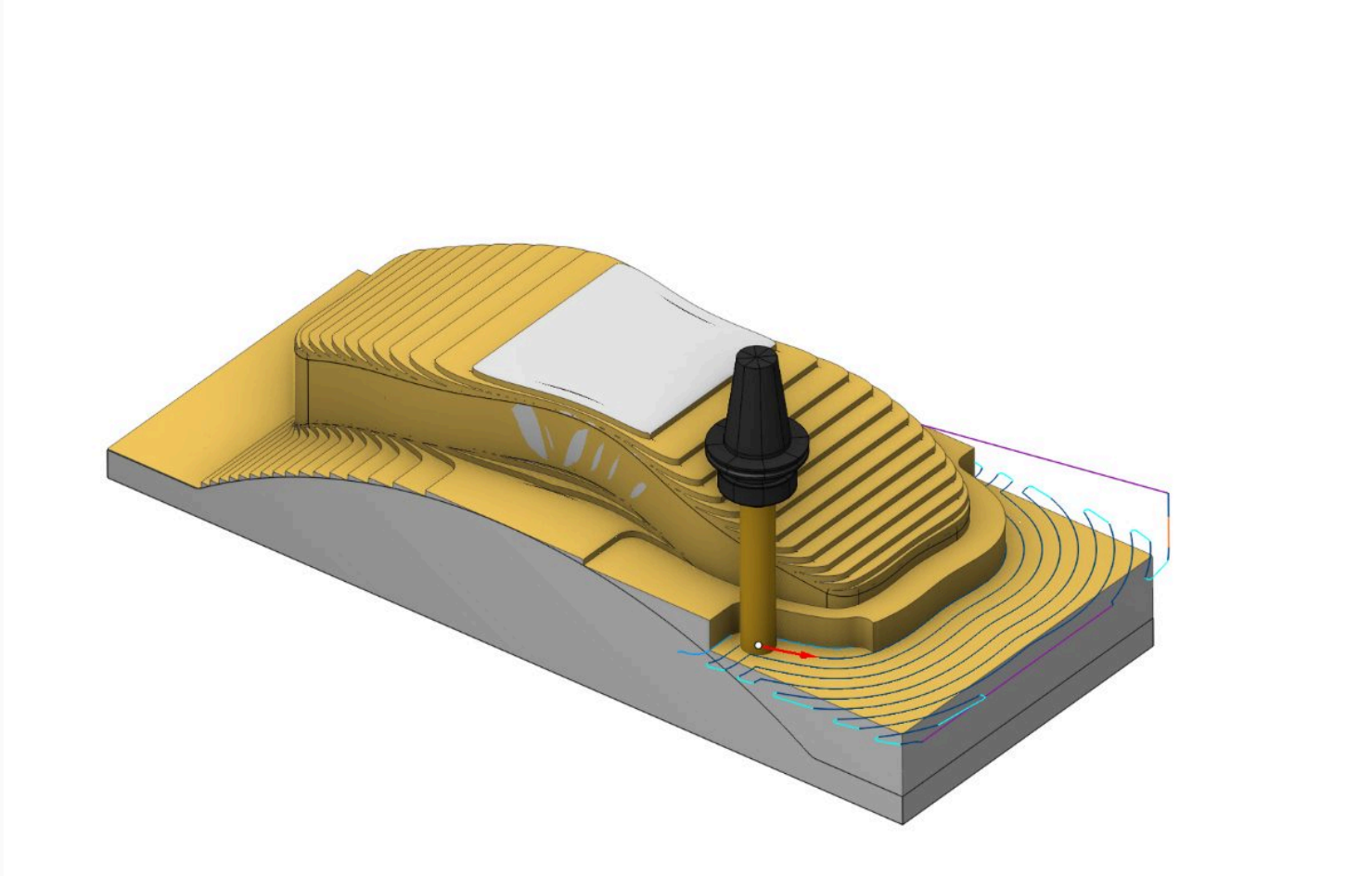
Workpiece:

A workpiece model of an operation defines the material to be machined. Details

Fixtures:

As the Fixtures the fixing aids such as chucks, grips, clamps, etc., and the restriction areas of any other nature are usually specified. Details

Simulation



Application area:

CAM systems offer different modeling modes, each changing the modeling method. These modes come with their own advantages and disadvantages. Voxel 5D is the default mode, offering quick modeling for rapid prototyping. However, Solid (Details) modeling, while taking longer, provides greater precision. For example, when working on intricate details, such as small holes in a large part, Solid modeling is the preferred choice due to its high accuracy. Ultimately, the selection of modeling mode depends on the specific project requirements and the desired level of detail.

Click the Simulation bookmark to switch on the simulation mode. The mode allows:

- to control by eyes the cutting process;
- to see the machining quality and to discover the possible defects;
- to compare the machined part with the source model;
- to discover and to mark the problem tool path fragments using the different criterions;
- to edit the calculated tool path to bring to the requirements of the user;
- to optimize the feed rates.

While simulating mill operations true solids are used to represent a tool and a workpiece. Thus, the quality of a resulting workpiece does not rely neither on the tool or the view orientation. The used method ideally suit for multi-axis simulation.

While simulating turn operations solids of revolution are used to represent a tool and a workpiece irrespective to the form of the source models. It is so to gain an effect of a revolving workpiece on a machine. If a mill operation follows a turn one, the true solid model is used for its simulation again.

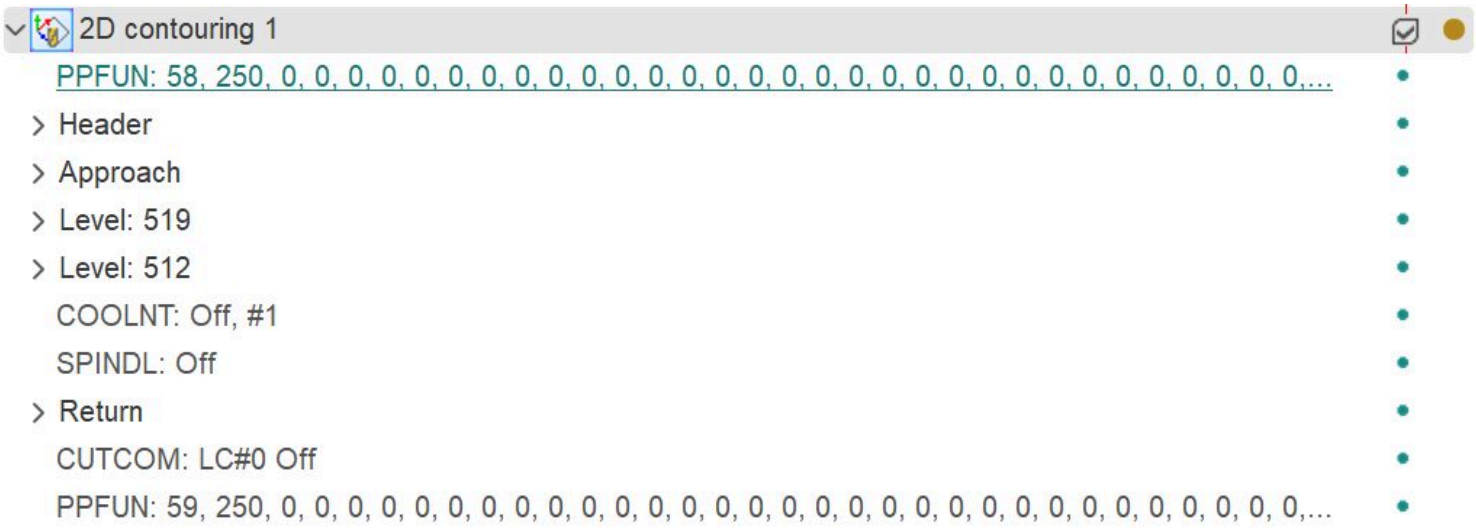
Simulation type (Located on the Parameters tab for each operation): Details

Tool path motion (CLData/ G-code format).

The simulation mode gives the access to the tool path (CLDATA) of each operation. If the operation is calculated then the calculated result is represented by the sequence of the CLDATA commands. The CLDATA commands are united into the hierarchical structure that is organized corresponding to the features of the concrete operation type.

▼ Click here to expand..
The data can be presented in either ClData or G-code format.

CLData. ClData is a data format that allows for the representation of both geometric and topological information, facilitating the creation of accurate and efficient machining programs. Details



G-code. G-code is a standardized numerical control programming language that is used to control the movements and operations of CNC machines. Details

```
%


( GENERATED BY SprutCAM )
( DATE: 25.06.2024 )
( TIME: 12:30:22 )

( TOOLS LIST )
( T39 CYLINDRICAL_MILL D80 )

G00G21G40G49G69G80G90G17
G53Z0.
G53B0.C0.
( 2D CONTOURING 1 )
G53Z0.
G53X0.Y0.
T39M6 (80MM CYLINDRICAL MILL)
G1
S40M3
G00B0.C0.
Y454.
G68.2X0.Y0.Z0.I0.J0.K360.
G53.1
G43H39X222.Y454.Z539.
G01G94Z519.F200M8
Y204.
Y-204.
G02X182.Y-244.I-40.J0.
G01X-182.
G02X-222.Y-204.I0.J40.
G01Y204.
Y454.
G00Z539.
```

Controlling simulation process (buttons).

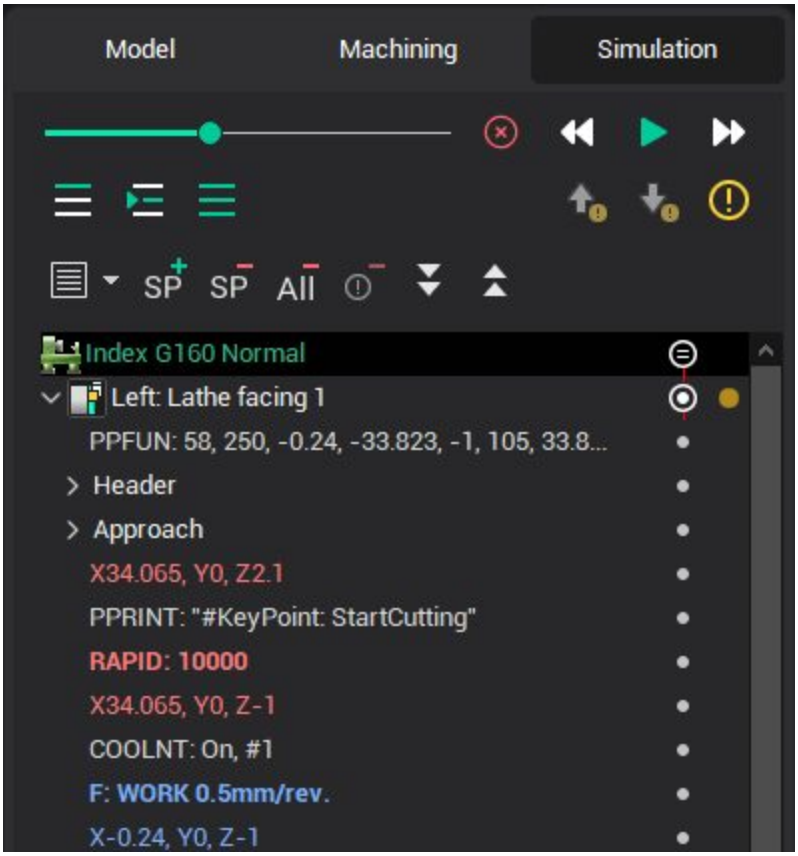
Machining simulation mode allows the user to obtain an image of the model being machined during the machining process. This allows the user to visually check the machining quality, analyze the presence of any rest material and over-cuts. Details

Tool path motion

This section describes tool path structure, CLDATA commands, selecting commands from the graphical view, and tool path editing and spatial transformations.

The structure of the tool path

The simulation mode gives the access to the tool path (CLDATA) of each operation. If the operation is calculated then the calculated result is represented by the sequence of the CLDATA commands. The CLDATA commands are united into the hierarchical structure that is organized corresponding to the features of the concrete operation type.



For example, the tool path of the plane rough operation consists of the levels; every level consists of the strings; and the string consists of the elementary CLDATA commands like GOTO etc. Therefore, the complicated tool path can be examined by logical part, analyzed and edited if necessary. For example, the tool path of the plane rough operation consists of the levels; every level consists of the strings; and the string consists of the elementary CLDATA commands like GOTO etc. Therefore, the complicated tool path can be examined by logical part, analyzed and edited if necessary.

The list of the basic CL-data commands

The CLDATA command that is generated by CAM system is listed in the table. The full list of the CLDATA commands see in the part 3 of this manual Postprocessors generator.

Command	Description
AXESBRAKE	Machine axes brakes control
CIRCLE	Motion by circle arc
COMMENT	Commentaries
COOLNT	The coolant Switch on/off
CUTCOM	The tool radius and tool length compensation
DELAY	Delay of time
EDMMOVE	EDM movement
EXTCYCLE	Extended cycle (drilling cycle, lathe cycle, milling cycle)
FEDRAT	Feedrate
FROM	Original point of the toolpath
GOHOME	Return to the home position
GOTO	The linear motion
INSERT	Insertion of a string to the NC code
INTERPOLATION	Different kind of Interpolation mode switch on/off
LOADTL	The tool loading
MULTIARC	Multiaxis circle movement
MULTIGOTO	Multi coordinate movement
OPSTOP	Optional stop of the NC code execution
ORIGIN	Coordinate system definition

Command	Description
PhysicGOTO	Physical machine axes movement
PLANE	The work plane (XY/YZ/ZX)
PPFUN	Postprocessor function
RAPID	Rapid feedrate
SELWORKPIECE	Active workpiece holder selection
SINGLETHREAD	Singlethread command
SPINDL	The spindle switch on/off
STOP	Stop of the NC code execution
TAKEOVER	Workpiece takeover into other spindle
WAIT	Waiting for synchronization point

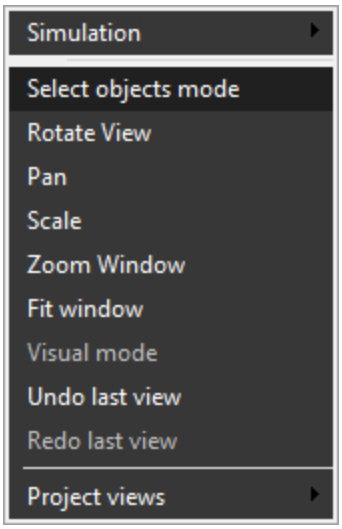
The selection of the CL-data commands from the graphical view

All CLDATA commands can be divided in two groups: the commands that move the tool and commands that do not. To the first group belong commands like GOTO, CIRCLE, MULTIGOTO, MULTIARC, PHYSICGOTO, GOHOME. These and other motion commands define the tool toolpath curve.

If a tool path is visible (the  button on the visibility panel is checked), then the active (i.e. selected) tree node with all its sub nodes is displayed in the view. Activating another node in the tree changes the view to display the appropriate toolpath fragment.

One can walk through the tree directly in the graphical view.

Enable the "Select objects mode" option in drop-down menu of the view port before the selection of the required tool path fragment.



The fragments inside or the selected node is highlighted then the mouse is moved. It is necessary to click the left mouse button on the highlighted object to select it. The tool path outside of the selected node is shown transparently and cannot be selected. To move to the parent of the selected node one can simply double click in the view.

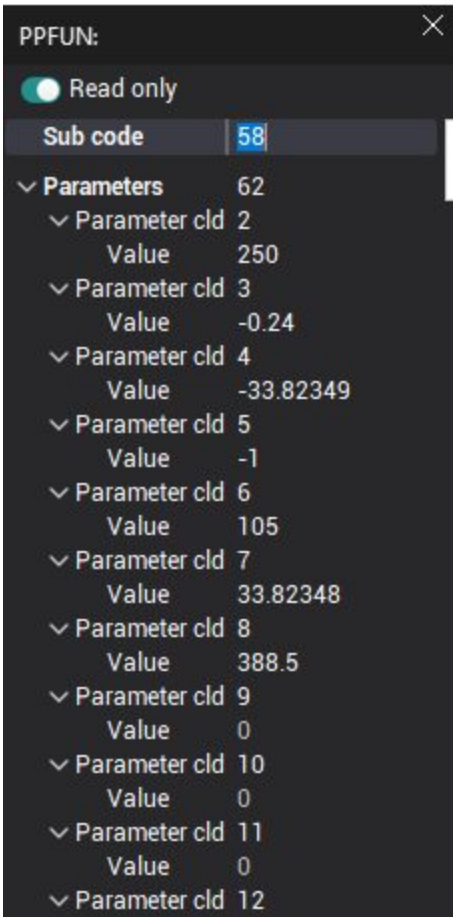
Tool path editing

There is the possibility to edit the sequence of the CLDATA commands. The tool path re-calculation resets all changes.

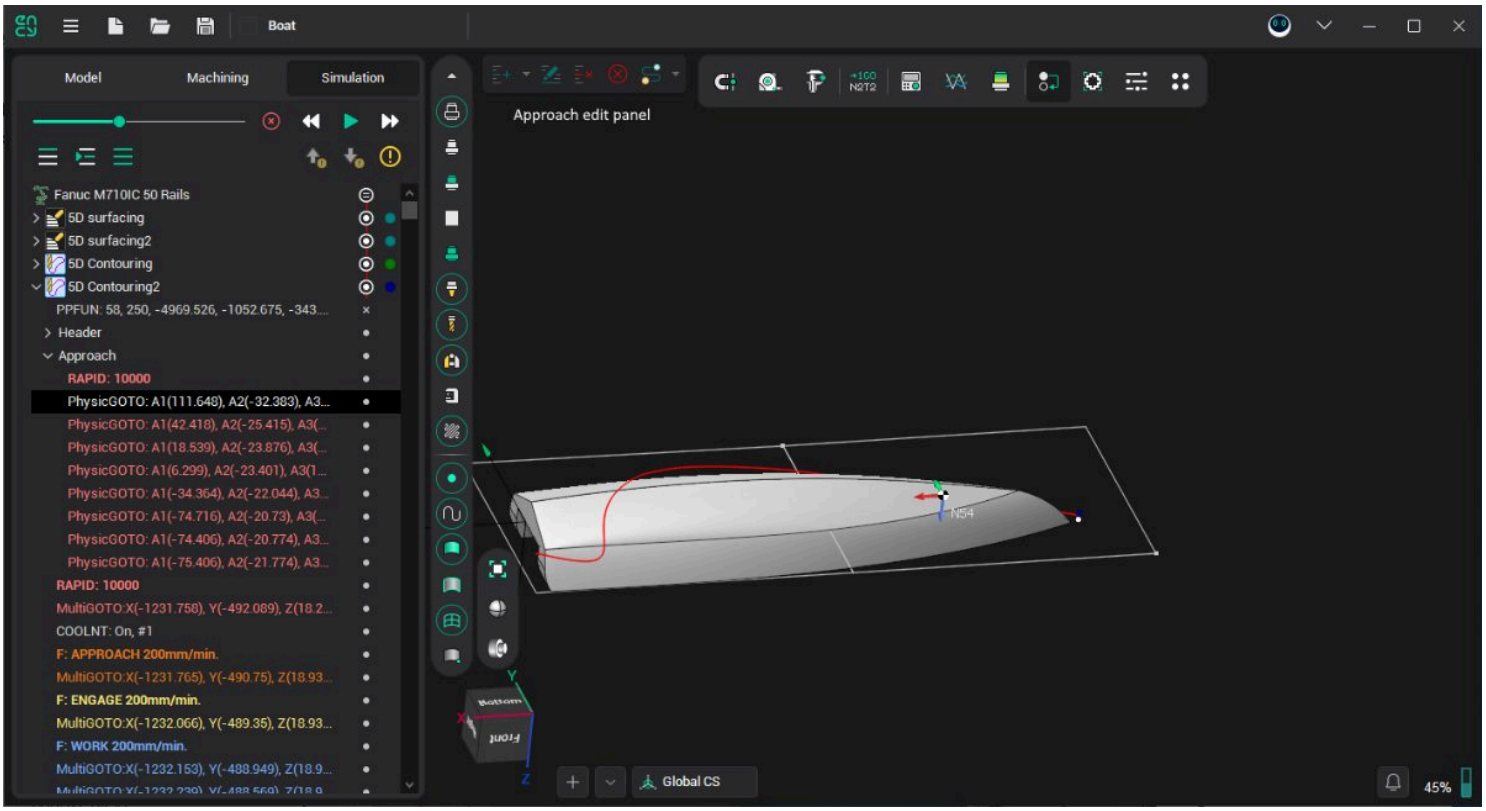
The selected node of the tool path can be deleted, copied, or cut into the clipboard by the context menu or by the standard keys:

- [Del] – deletes the current node of the tool path tree;
- [Ctrl+X] – cuts the current node of the tool path tree into the clipboard;
- [Ctrl+C] – copy the current node of the tool path tree into the clipboard;
- [Ctrl+V] – inserts the command from the clipboard before the current node.

Double click on the node or select Edit item from the context menu to edit the parameters of the selected CLDATA command. The opened dialog is not modal i.e. there is no need to close the dialog box to edit other node. The dialog caption is a type of the edited node. The list of the parameters depends on the node type. To start editing parameters you must turn off read only mode.



Approach/Return editing

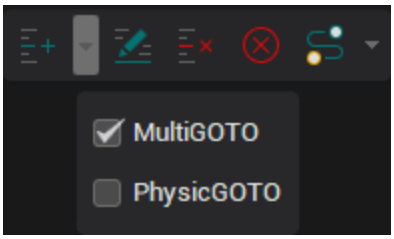


If the user selected the 'Approach' or 'Return' section inside tool path tree, additional panel becomes visible (see screenshot). On this panel the following buttons are available for more convenient editing of approach or return section (buttons are listed from left to right):



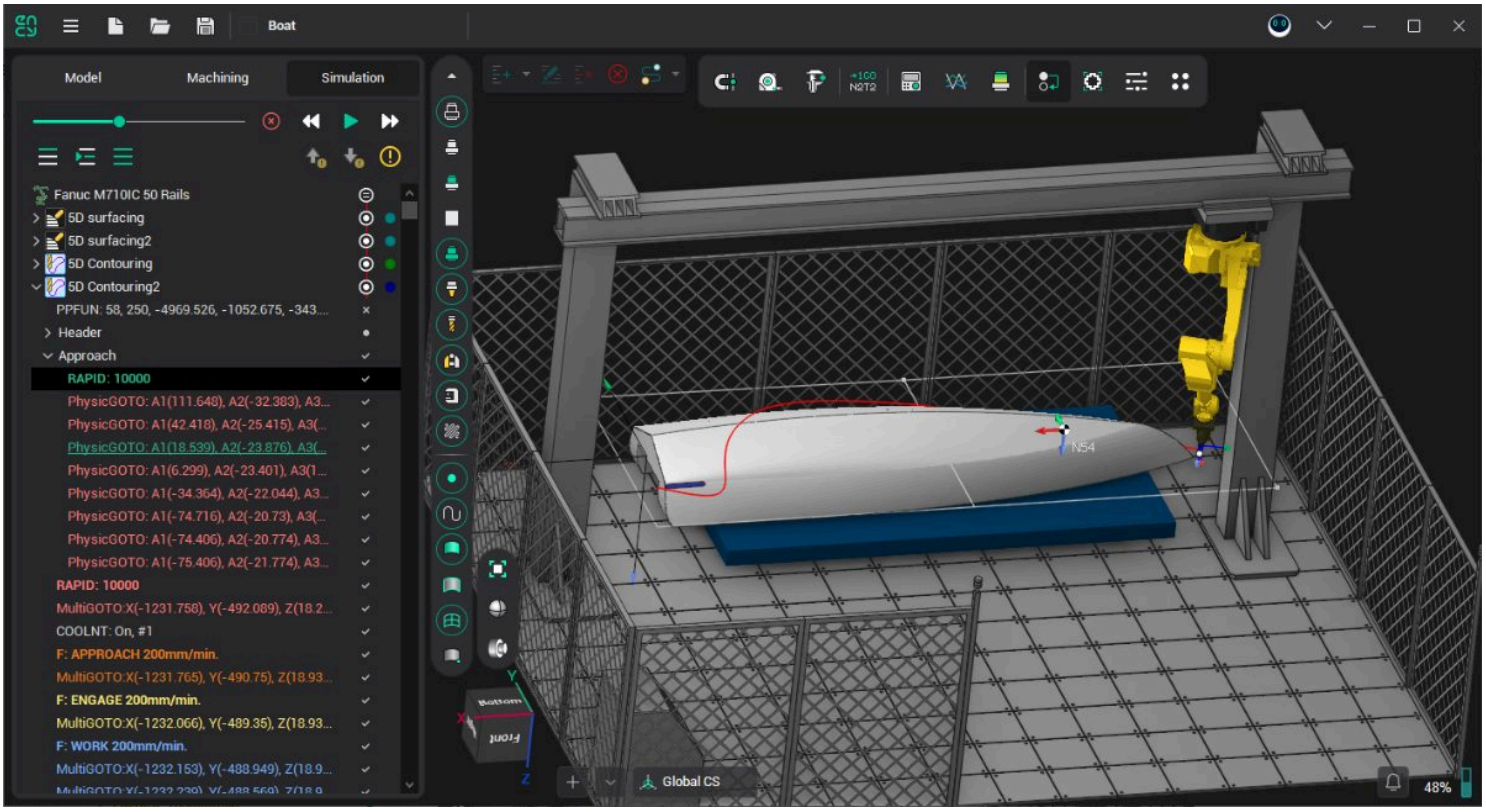
- Insert current state – this command inserts current machine state as a node of tool path tree. The state is inserted before current selected node of the tree.

Dropdown menu can be used to change type of inserted node (PhysicGOTO or MultiGOTO).



- Edit current state – press this button to start editing interactively current selected state of tool path tree. The coordinates of selected node are synchronized with current machine state. To end interactive editing process press this button again.
- Delete current state – deletes the current node of the tool path tree.
- Clear approach/return – clears all commands inside approach or return section except last (which corresponds to the end state of approach/return).

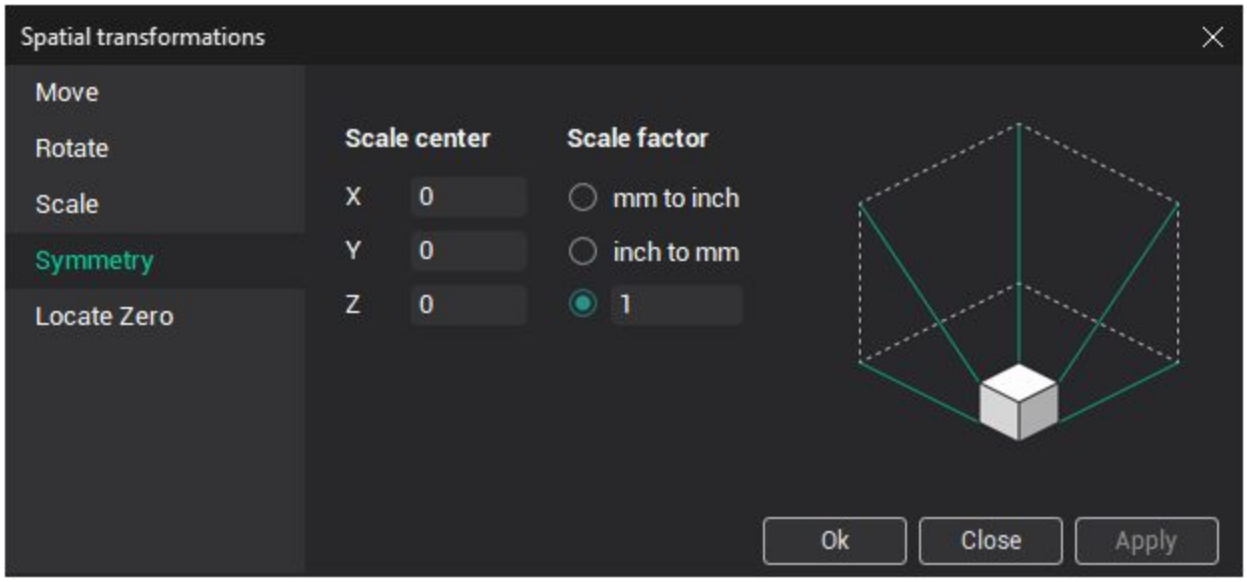
- Calculate with Motion Planner - build trajectory of approach or return automatically using one of the algorithms for motion planning with excluding collisions between nodes of machine. Several parameters of the motion planner, such as time limit for calculation of path, can be specified in the drop-down menu.



On this screenshot whole approach trajectory was generated automatically using motion planner. If the generated trajectory is not satisfactory interactive editing can be used to fix the approach/return path.

Tool path spatial transformations

Click the right mouse button on the operation and select the Transform tool path item from the context menu. The opened dialog matches to the model transformation dialog.



The type of the spatial transformation is selected by the bookmark:

- Move bookmark allows to define the linear move the tool path by three axis;
- Rotate bookmark allows to rotate the tool path around Z axis;
- Symmetry bookmark allows to make the symmetry transformation relative any vertical plane;
- Scale bookmark allows to perform evenly and unevenly scaling of tool path;
- Locate Zero bookmark allows to move the tool path using the dimensions of toolpath.

Controlling simulation process

Machining simulation mode allows the user to obtain an image of the model being machined during the machining process. This allows the user to visually check the machining quality, analyze the presence of any rest material and over-cuts.

When the simulation window is opened, the system automatically creates the workpiece model for machining. If the window is reopened, the model being machined will not change. This means that the simulation results will be saved, and machining simulation can be continued.

The initial workpiece for simulation is always one specified in the root node of the machining tree. A continuous simulation of machining will modify the form of that workpiece.

However, pressing one of the  buttons will load the workpiece from the nearest operation. So the result may differ from that achieved after a continuous simulation from the very beginning if one or more operations override the default workpiece – the machining result of a previous operation (i.e. use a different workpiece).

Notice: Often projects developed in earlier versions of the system do not contain information to reconstruct the workpiece transmission chain or the workpiece for entire operation sequence can not be determined. In those cases simulation may look strange. To settle the problem we advise you to activate the first operation of machining sequence, press the  *button to load the initial workpiece than run smooth machining simulation.*

To drop the simulation results, one should press the  button. When pressed, the workpiece will be reinitialized according to the parameters defined in the root node of the machining tree.

Tool motion controlling

The simulation process can be started by one of the buttons in the next list. The last three buttons start the fast simulation i.e. the screen will be updated once - at the end of the formation of the model. Other buttons start the step-by-step simulation, i.e. the graphical view will be updated over and over again depends on the tool movement method.

- – simulates the node before current ones. The screen updating depends on the tool movement method;
- – stops the simulation process;
- – runs the simulation of the current node. The process stops after the next node arrival;
- – runs the step-by-step simulation from the current node to the end of the list. The screen updating depends on the tool movement method;
- – runs the fast simulation of the selected operation (tool path node). All CLDATA commands inside the selected node are run. The simulation stops when the next operation (node) attained. Screen performs once upon completion of the simulation;
- – runs the fast simulation till the current operation. Work off all technological commands from the beginning of technological process and up to the selected node path or operation. Screen performs once upon completion of the simulation;
- – runs the fast simulation all operations at once. Screen performs once upon completion of the simulation.

Note: If you need to simulate only some part of machining (some operation) you can go to this operation (or some command of this operation at CLDATA view)and press button to prepare workpiece for this operation. Then you can use Smooth or `By-block` simulate.

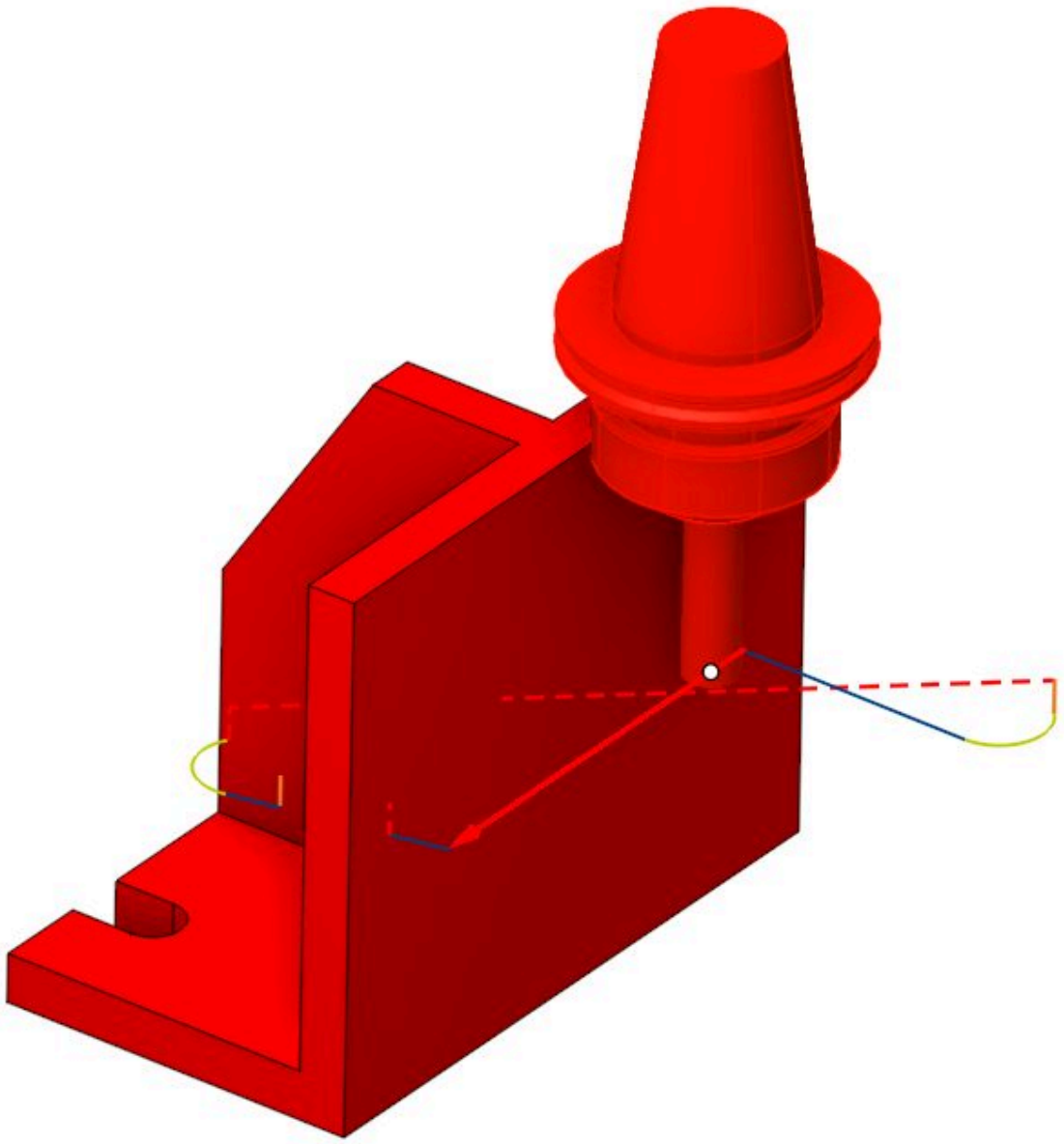
One of two modes can be selected while machining simulation: Smooth or By block.

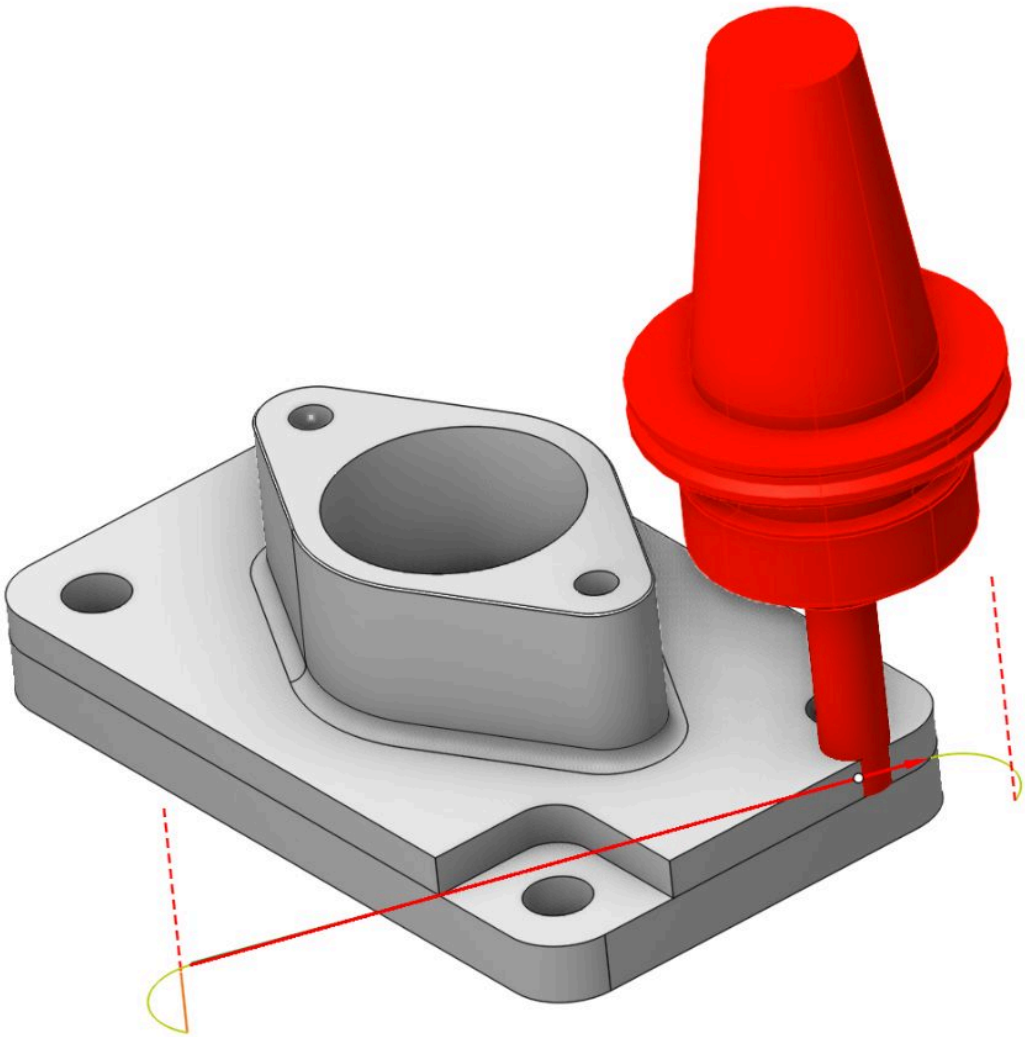


- When the Smooth method is selected (checked option), the tool will move smoothly with constant speed. In this mode, user can control the visual speed of the tool by altering the tool speed slide. The far left position is the slowest movement speed, the far right – the fastest.
- If By block is selected (unchecked option), then the tool will move block-by-block, with drawing at intermediate points only.

Changing the method and movement speed is possible during the simulation process.

Tool path errors detected by simulation





Application area:

The section covers error identification in tool paths during simulation, and provides filter controls to selectively display or suppress specific error types for focused analysis.

In the CAM system, on the Technology tab, operations have status indicators reflecting their current state. Details

On the Simulation tab, each CLDATA row has its own status, enabling quick identification of any issues. The status of the node is indicated by an icon:

-  - CLDATA commands are disabled. It is not output to the NC program and not simulated.

-  - CLDATA commands are not simulated.

-  - CLDATA command is simulated without errors.

- CLDATA command is simulated with error (various icons may appear here; they will be described in detail below.);

-  - CLDATA command is idle. The stock is not removed by this tool motion command.

During operation, various checks are performed. If an error occurs, the CLDATA row is marked with the corresponding icon.

The following types of errors are checked:

-  - Stock removal by rapid feed is forbidden. If the stock is removed and the feed rate is rapid, then the command is marked as an error.
-  - Toolholder collision. If the toolholder touches the workpiece when the CLDATA command is marked as an error;
-  - Impossible to calculate the compensation. Tool radius compensation is performed while doing the simulation. The compensation value is defined in the operation parameter window on the tool bookmark. If compensation cannot be made for the CLDATA command, then the motion is marked as an error;
-  - Tool plunge cuttings with an angle exceeding the maximum value specified in the tool parameters are marked as errors.
-  - Collision of machine nodes.
-  - Gouging the part is detected.

-  - Axis travel over the limits.
-  - Inappropriate tool and spindle rotation direction error.
-  - Tool overload (appears if option ❗ Mark overloads is enabled in *the Adaptive federate* section on *the Feeds/Speeds* parameter panel for roughing waterline operation). Details

Collision Check Level Panel.

This panel contains buttons similar to those used for Background Simulation. Details

There are several important considerations when working with this panel:

- By default, on the Simulation tab, the collision check level matches the check level of Background Simulation. That is, when you change the Background Simulation check level in the system settings and click Apply, the collision check level settings on the Simulation tab are automatically updated (Only if the verification level on the Simulation tab was higher than that of Background Simulation).
- On the Simulation tab, you can set a check level that differs from the Background Simulation settings — but not lower than the level defined in Background Simulation. Decreasing the check level is not allowed.

Example. *If the Machine check level is selected in Background Simulation, it cannot be lowered below this value on the Simulation tab. However, you can increase the check criterion by moving the slider to the Hidden Nodes position. This will help detect additional errors, if any are present.*

- When you increase the check level on the Simulation tab after performing a background calculation, the operation status is reset to the value . Similarly, changing the Simulation accuracy will also reset the operation status.
- If you run a step-by-step simulation and lower the check level below that of Background Simulation, the operation statuses will change depending on the current check level.

Example. *If Holder is set as the check level in Background Simulation and a step-by-step simulation is then performed with the Hidden Nodes check level, lowering the check level to Machine or Dressing will cause the operation statuses to update. The user will see only those errors that correspond to the selected check level.*

Part gouge. Allows displaying errors where gouging of the part has been detected.

Additionally, the system includes dedicated auxiliary buttons that allow users to stop full simulation under different conditions:

-  - Allows stopping at CLDATA rows where the M1 stop command is specified. The M1 command can be inserted using the Macro of Toolpath Template functionality. Details For better visualization of the technological process, an Auxiliary operation can be created, and the M1 command can also be set in the Macro of Toolpath Template section. Details

-  - Allows stopping after each operation.

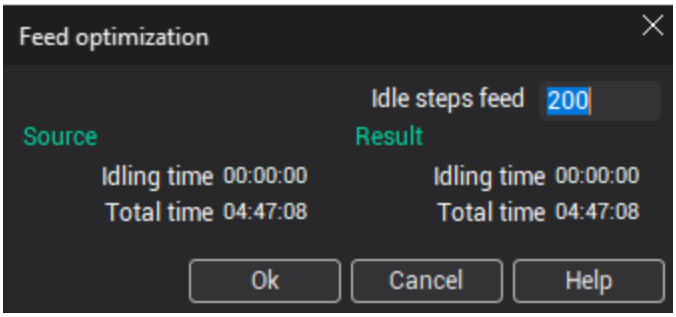
-  - The button defines the action, then the error is detected. If the button is down, then the simulation process is break after the error detecting. Else, the node is marked by the red exclamation mark and the simulation process goes on. After the simulation, it is possible to find the marked nodes by shortcuts or from the context menu:

-  - Go To Next error – move the selection to the next error, marked node;

-  - Go To Previous error – move the selection to the previous error marked node.

Feed rates optimization

After the simulation, it is possible to change the feed rate of nodes that is marked as the idle node. Click right mouse button on the required operation and select the Optimize feed from the context menu. The next dialog is opened:

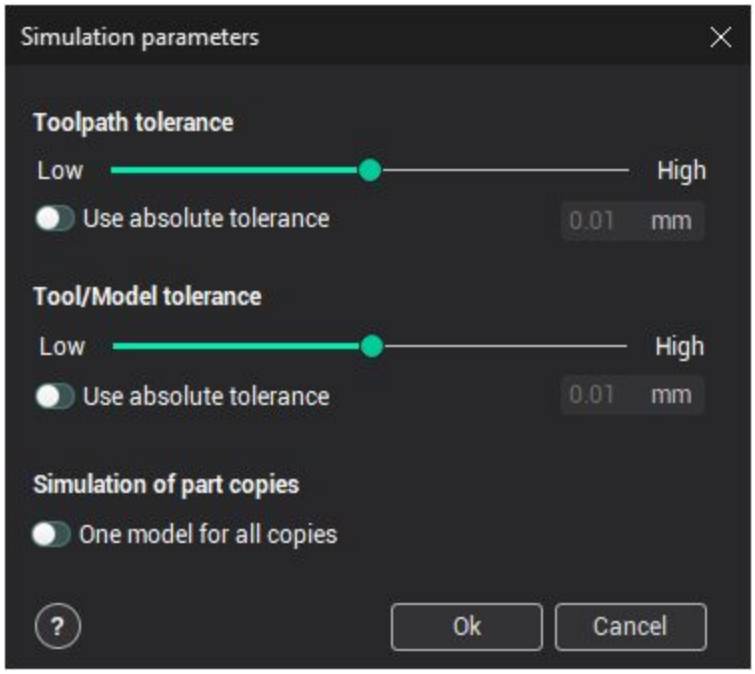


On the left hand, side of window the total machining time and the time idle motion is displayed. On the right hand side, the obtained with the defined feed rate value the total and idle time is displayed. After OK the new FEDRATE commands is inserted into the CLDATA sequence to change the feed rate before the idle motion.

Note: In an operation does not have the idle motions then the optimization does not give any effect. If the idle steps feed value is smaller then the real feed value then the optimization does not have the sense because of the machining time increases.

Assigning workpiece parameters

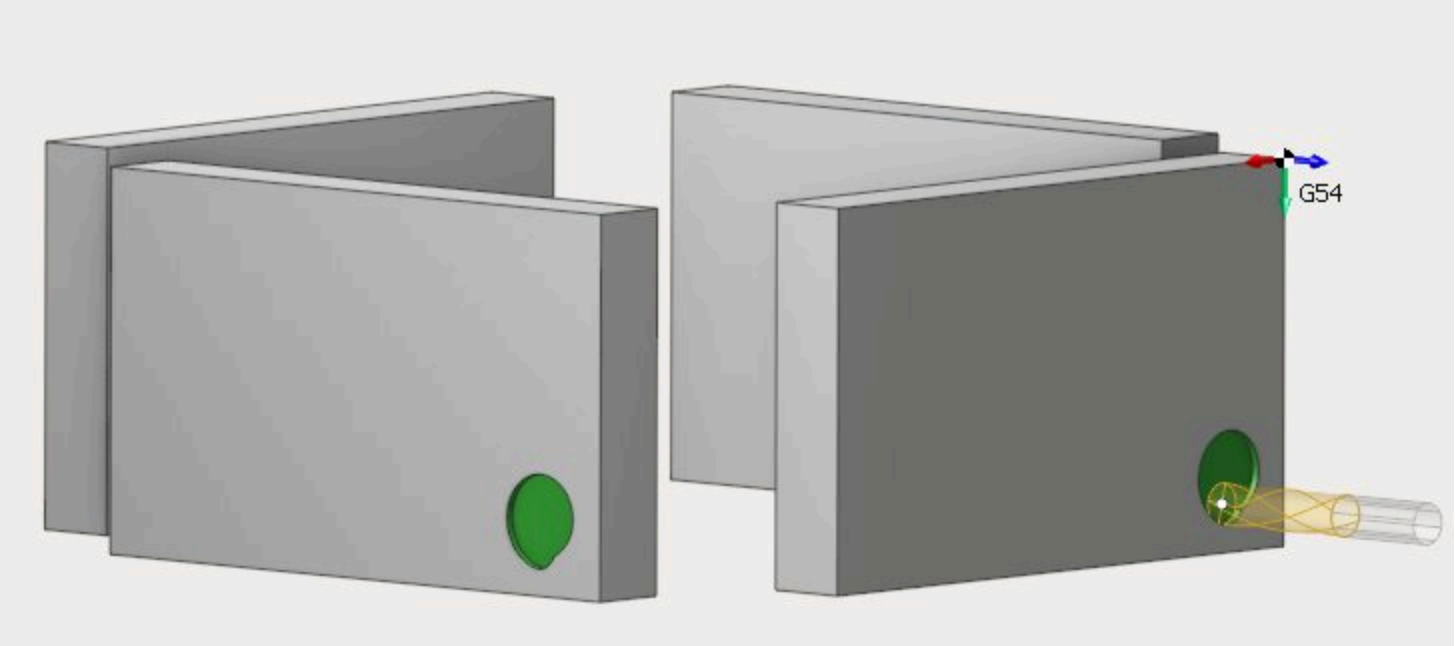
The simulation settings editing window opens when the  button in the machining simulation window is pressed.



The tolerance of simulation may be specified either using the slider that ranges from Low to High or in absolute units, if the Use absolute tolerance box is checked. The chosen tolerance affects the tool, the initial workpiece and the toolpath. The lower the value is the faster simulation runs but lower the quality of a resulting model is. By default the relative value specified with the slider is used. In this case the actual absolute tolerance values for the tool, the workpiece and the toolpath are chosen automatically according to their extents. This guarantees a good result for most workpieces and toolpaths. However one can to set the common exact tolerance value to guarantee reliable result in hard cases.

The used tolerance value can be modified dynamically while simulating a toolpath. So most interesting fragments can be simulated with higher tolerances.

The One model for all copies parameter is really useful when you work on a project with many copies of a part. This parameter allows to use less RAM and as a result to speed up the simulation process.



Notice:

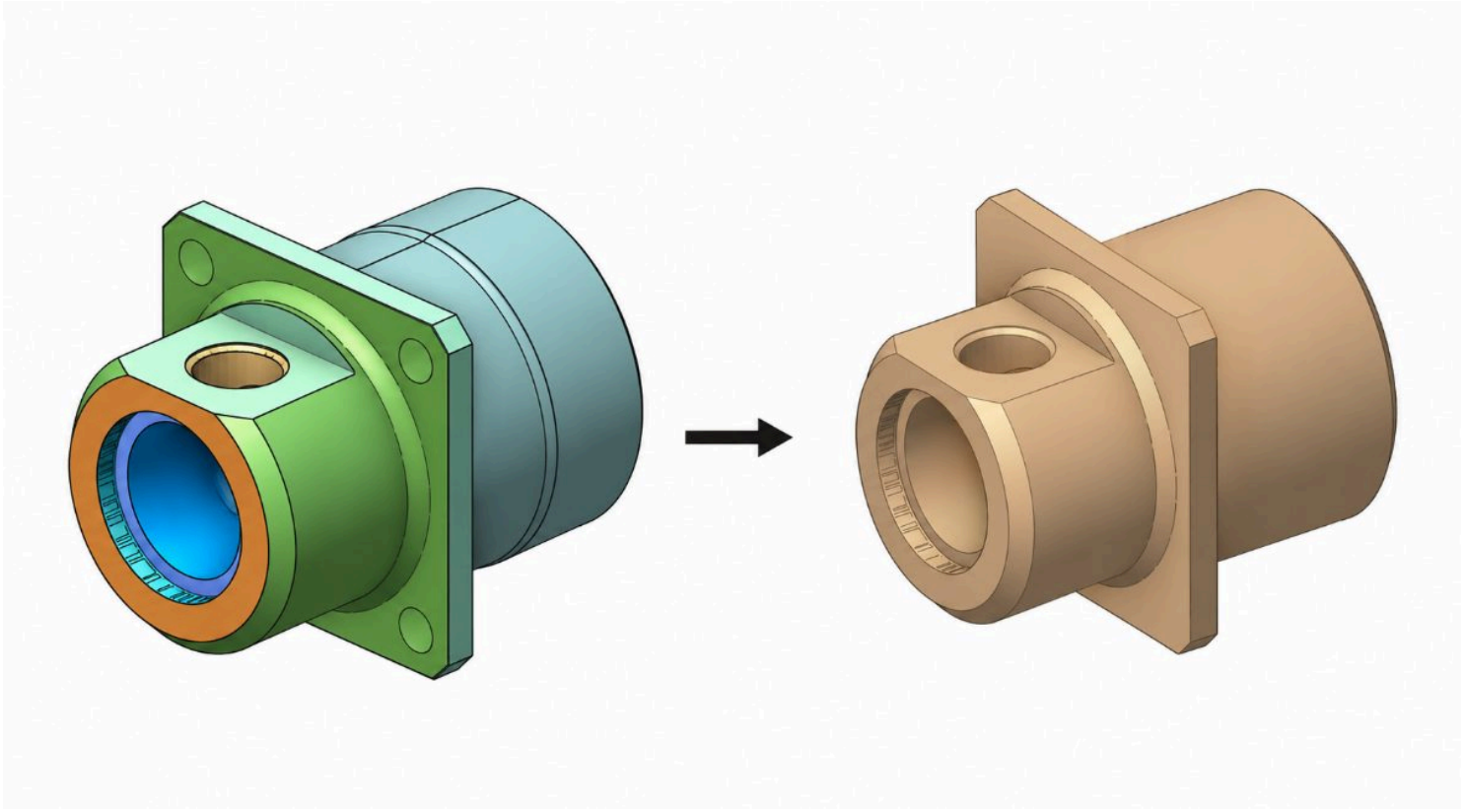
- *We do not recommend you to use high tolerance for simulation on low-end computers*
- *If you made any changes in this window, you should reset workpiece parameters on simulation page, by clicking the reset button* ⊗

Turbo simulation mode

While simulating a 3d Milling project in CAM system you can use the turbo simulation mode. This mode is turned on by checking the Turbo mode button at the Simulation panel. In turbo mode simulation speed increases dramatically, especially in the Wire display mode. So you can easily backplot huge toolpaths even on low-end PCs.

By now Turbo mode supports only 3 axis milling. There is no support for 3+2 and 5 axis milling as well as turning. The turbo mode resolution is controlled by the `Tool/Model tolerance` slider in the Simulation Tolerance dialog. The dialog appears after pressing the Tolerance button at the bottom of the Simulation panel. The model resolution varies in 125-250-500-100-2000 ranges. The default resolution is 500x500 dexels.

Export simulation result as model



Application area:

The simulation result shown on the Simulation page can be saved to a file and later reused as a workpiece in another project, or, for the STL format, in third-party software. This carries the machined state of a part from one project into the next, so machining continues from the already-processed stock.

Typical scenarios:

- Splitting a long machining job into several projects and passing the intermediate stock between them.
- Reusing the exact machined model as the workpiece for a follow-up setup without loss of detail.
- Handing the simulated stock to external CAM/CAD software through a neutral mesh format.

Saving the simulation result.

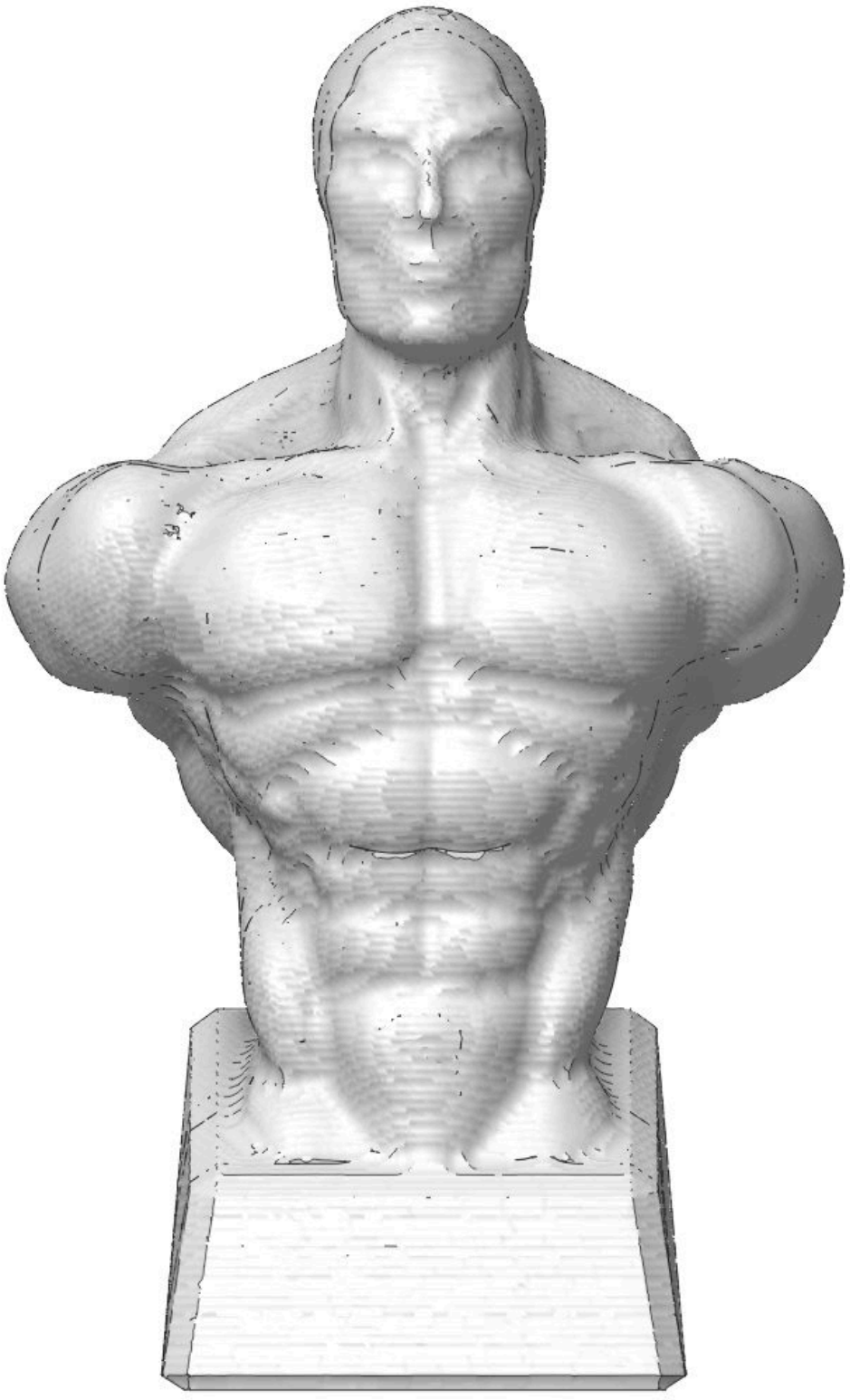
Two entry points are available. Both open the Save As dialog.

Context menu on the Simulation page. Right-click in the simulation window and choose Save simulation result.

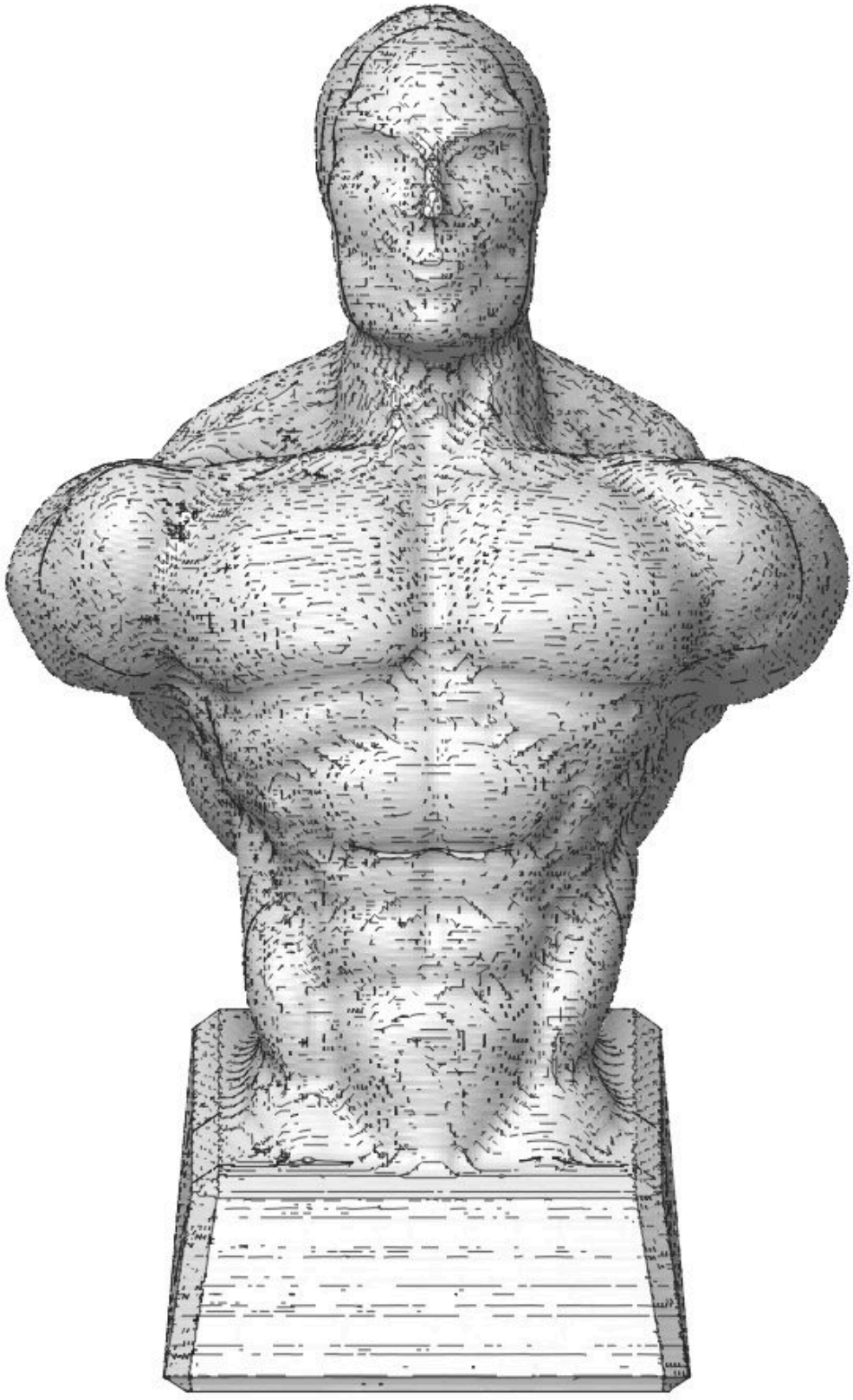
Main menu. Open the Save menu (disk icon in the top-left toolbar) and, under Export, choose Simulation result.

In the Save As dialog, enter a file name and select the format in the Save as type drop-down:

- Simulation result, precise (*.camsr). Selected by default.



- Simulation result, STL with losses (*.stl).



File formats.

Simulation result, precise (*.camsr)

The native simulation format. It stores the exact simulation model as computed, with no conversion.

- No loss of precision. The model is preserved as-is; when imported back as a workpiece it is used directly, without any approximation step.
- Recommended whenever the file is reused inside the CAM system. This is the default option.

Simulation result, STL with losses (*.stl)

The standard STL triangle mesh.

- Universal. Opens in most CAM/CAD applications and any tool that reads STL.
- Lossy. The precise model is converted to a triangle mesh on export; on re-import it is loaded as a mesh and re-processed, so the reused stock is an approximation of the original rather than the exact model.

Reusing a saved result as a workpiece.

A *.camsr file can be loaded back into another project as the workpiece:

1. Go to the Workpiece tab on the Machining page.
2. Click the Sim result button.
3. Select the *.camsr file.

The exact machined model is added as the workpiece, and machining can continue from it — no re-voxelization, no detail loss.

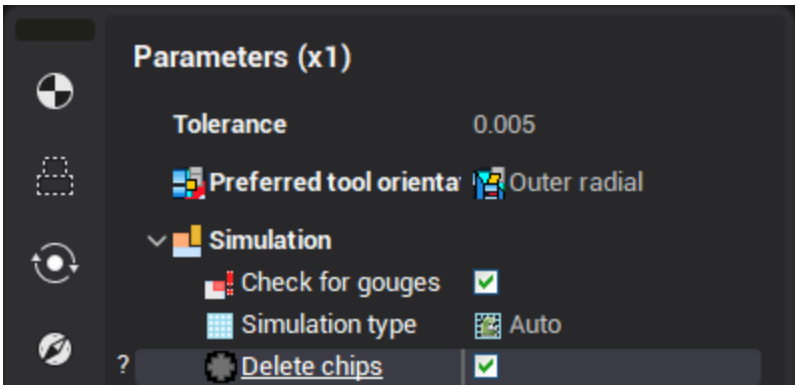
> Note
>
> The *.camsr file records the measurement units it was saved in. If you import a result whose units differ from the current project's, the import is canceled, because automatic unit conversion (mm ↔ inch) is not supported yet. Save and reuse results within the same unit system.

Delete chips function

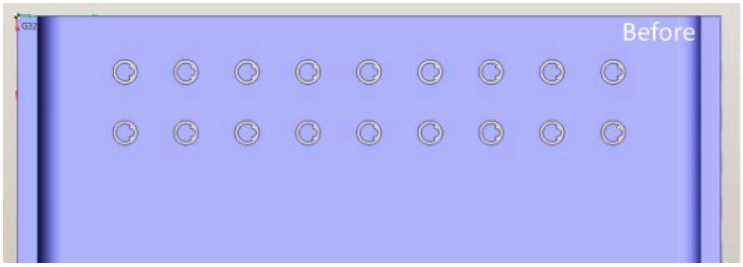
The function deletes chunks of the workpiece (or simply, chips) that do not contain the part geometry. (So, presence of the part is crucial for the correctness of the result).

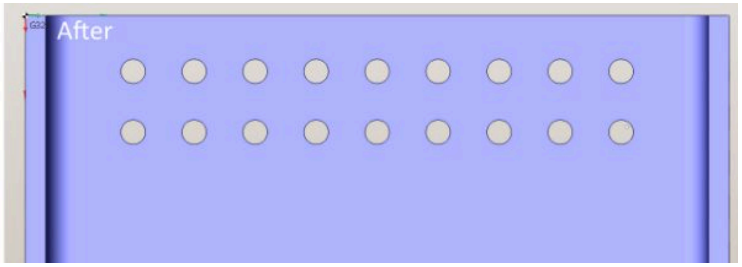
The function can be run in two ways:

1. By pressing the button on the "Simulation" page
2. By enabling the Delete chips option in the parameters of operation on the "Parameters" tab. In this case, the chips will be deleted automatically at the end of the operation simulation process.



Example:





Simulation of machining by NC program

The mode of machining simulation by NC program (NC) allows you to take into account the peculiarities of postprocessor implementation when simulating machining. In this mode, the system automatically generates the NC for each operation during its calculation. Simulation control is performed in the same way as in the normal simulation control.

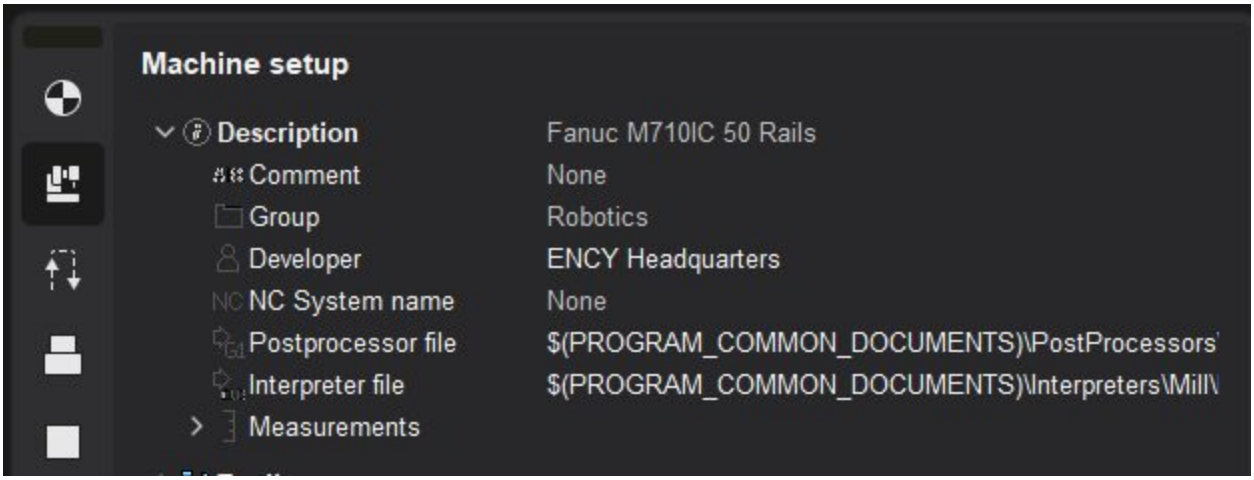
Enabling the mode of modeling by NC.

Switching on is carried out with the button 

on the toolbar, when the Modeling tab is active.

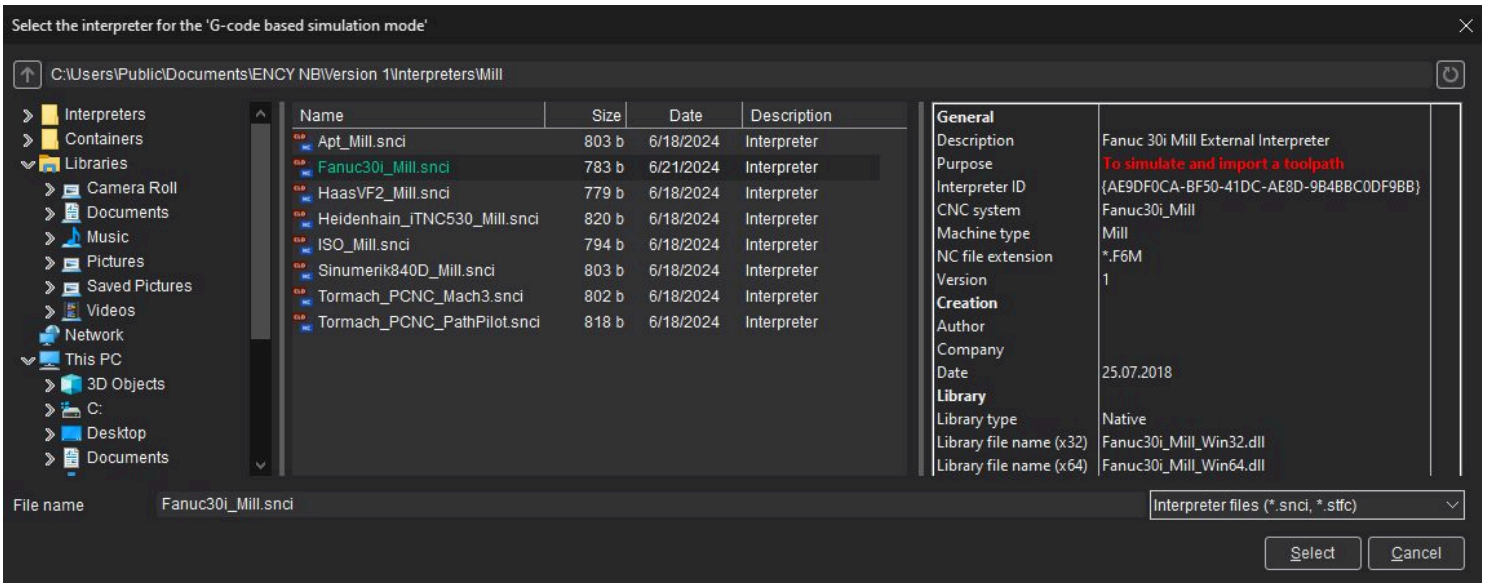
Note: The button becomes unavailable for toggling if the multitasking mode is used;

Parameters of the simulation mode by NC program. Simulation by control program text is performed on the basis of the selected postprocessor file and interpreter file defined in the parameter panel Machinery of the machine tool.

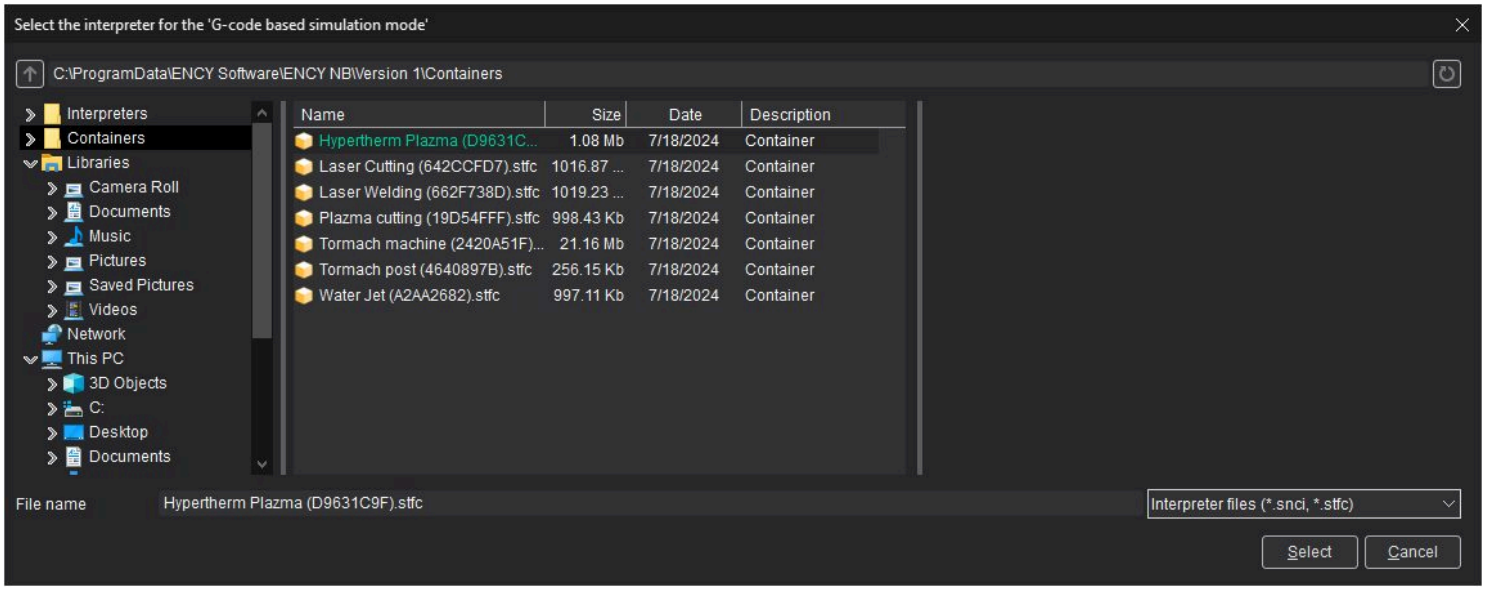


> The interpreter is a machine tool control settings file (*.snci) located in the $(PROGRAM_PERSONAL)/Interpreters directory.

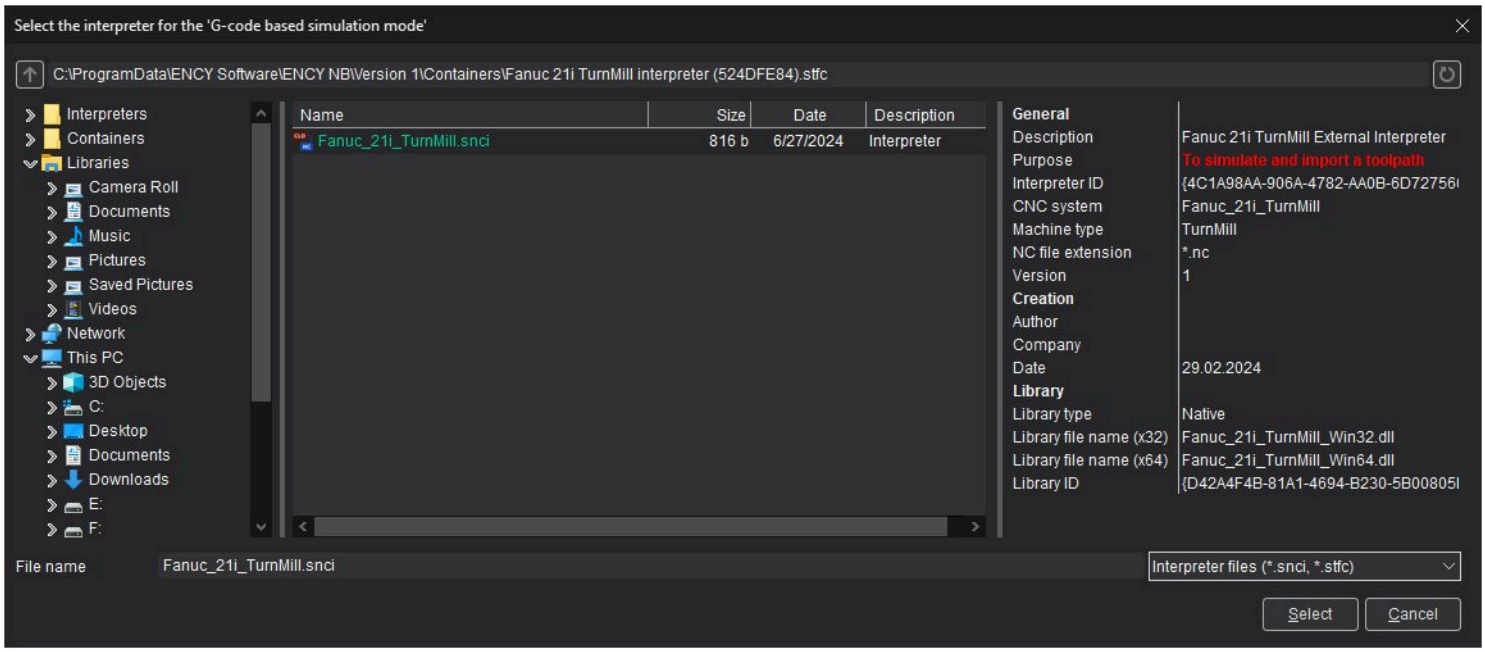
During the selection process, a preview panel of interpreter information (description, purpose, stand type, authors, and more) is available:



The ability to select an interpreter from a container is supported. To do this, get a container with interpreter. Then open the interpreter selection window and navigate to the containers folder.



Log into the resulting container as a folder, select the interpreter to model inside the container.



> Defining the default postprocessor and interpreter in the machine tool's kinematic diagram
>
> It is possible to set the postprocessor file name and the interpreter file name "by default" in the kinematic diagram of the machine. To do this, add tags SPPFile, SNCIFile with references to the corresponding files to the XML file of the machine diagram and restart the CAM system.
>
> Now, when you select the machine in the CAM system, the postprocessor and the interpreter will already be set and their values will be obtained from the kinematic diagram. If necessary, from the ENCY user interface, the postprocessor and interpreter values can be overridden for the current project.
>
> Example:
>
> ```
> SCType ID="Fanuc 30i" Caption=""Fanuc 30i" type=""Fanuc30i" Enabled="true" ... other tags... / SPPPFile DefaultValue="$(PROGRAM_PERSONAL)\Postprocessors\Mill\Fanuc (30i)_Mill.sppx"/ SNCIFile DefaultValue="$(PROGRAM_PERSONAL)\Interpreters\Mill\Fanuc (30i)_Mill.snci"/. ... other tags... / /SCType
> ```

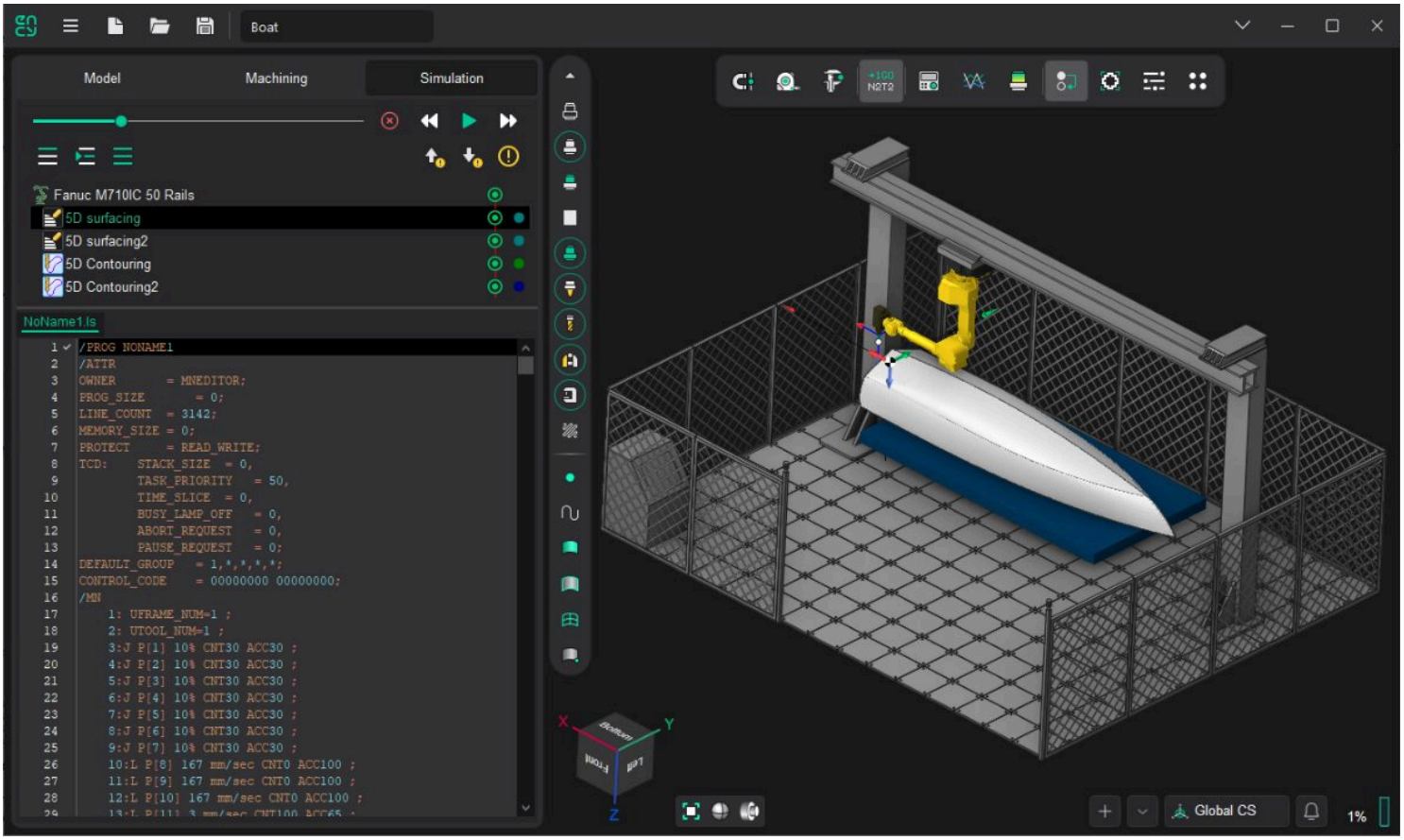
Features of modeling by NC

If the mode is enabled, aftercalculating the tool trajectory, ENCY will automatically generate a control program for the CNC machine with a pre-selected postprocessor settings file, and will convert the text of the control program into a tool trajectory. The generated toolpath will take into account the specifics of the postprocessor implementation. The text of the control program for the selected operation is displayed at the bottom of the Simulation mode page, immediately after the list of process operations.

After a short delay of the mouse pointer over the line with the control program text, the description of the associated nodes of the trajectory tree (CLData technological commands) appears in the tooltip.

Note: The tooltip can be hidden by moving the mouse pointer slightly or pressing any key, e.g. [Esc] or one of the keys for navigating through the control program text: [↓] or [↑].

To the left of the control program text there is a service field for displaying auxiliary information. In addition to the number, the status of all associated nodes in the trajectory tree is displayed for each line. The values of the displayed icons are similar to those used for the CLData process commands. In this way, you can unambiguously identify the control program frame in which erroneous trajectory tree nodes exist. Buttons  and  allow you to navigate through the control program text between errors.



Support for external interpreters.

Third-party interpreters for simulation by the control program text are supported. The interpreter settings file (*.snci) must contain a reference to the program library that is used to interpret the control program text. The Creating your own interpreter page describes the process of creating your own interpreter: settings file and application program interface (API).

G-code based program simulation

In G-code based simulation mode, two more stages are added to the calculation of the operation, in addition to the forming of the tool path:

- generating of the NC-code along the generated tool path;
- interpretation of the NC program into the secondary (restored from the NC-code) tool path.

Machining simulation uses a secondary (restored) tool path.

Generating the NC program along the primary tool path

NC-code is generated using the postprocessor selected for the machine. During post-processing, the postprocessor mode service variable WorkingMode takes the value 1.

Interpretation of the NC-code

By interpretation, we will understand the process of converting the text of a control program, its words and blocks, into technological CLDATA commands, that form the tool path. In the process of interpreting CAM system, using the interpreter selected for the machine:

- sequentially read the blocks of the NC;
- using the register information received from the interpreter, selects words (or semantic constructions) from the NC-code's blocks or offers the interpreter to select words independently;
- passes the words to the interpreter in the order they are in the block;
- the interpreter, taking into account the features of the machine, converts the words of the NC program into CLDATA technological commands, which form the restored tool path.

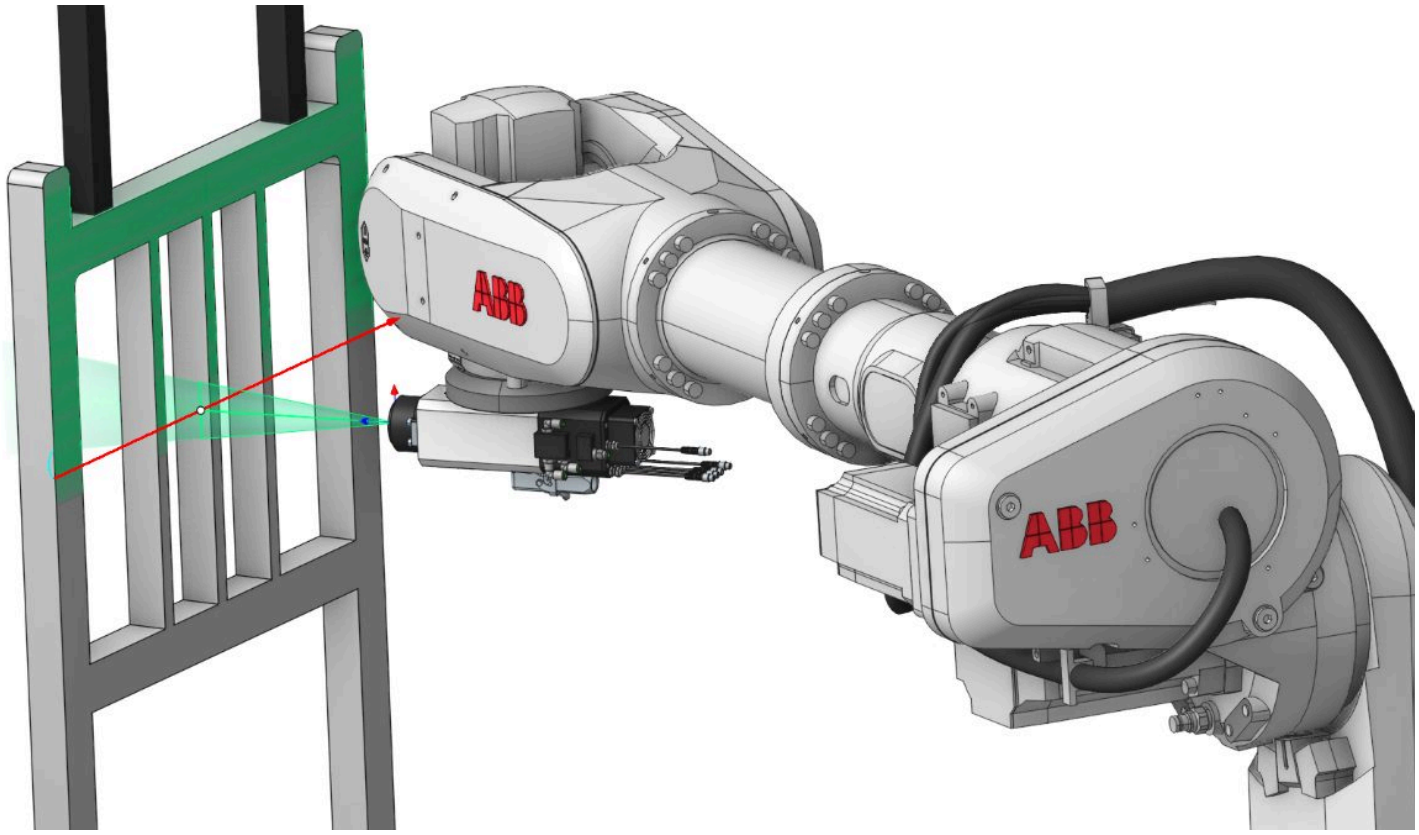
The description of the interpreter is given in the section Interpreter structure.

The interaction of CAM system with the interpreter to obtain a secondary (restored) tool path is described in detail in the section G-code based program interpretation.

Creating your own interpreter

The description of the creating your own interpreter is located at this link Creating your own interpreter.

Painting simulation



Application area:

Painting simulation allows you to see the areas of the parts that will be painted, as well as perform control of the thickness of the future paintwork (depending on the chosen type of painting).

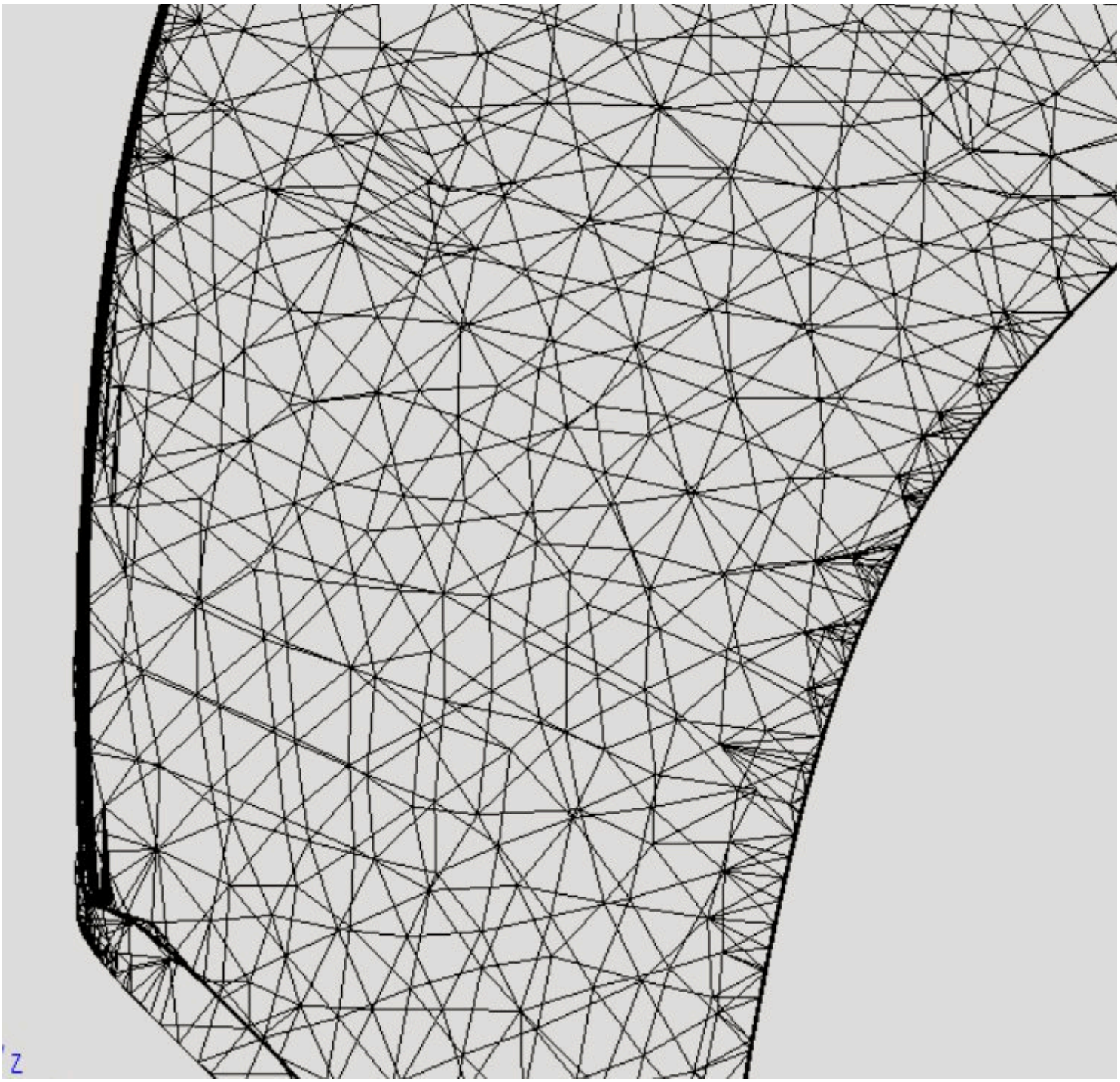
Tips for Creating a Painting Project

1. Recommendation for the model

▼ Click here to expand..

Before loading model to our CAM system we recommend you to prepare this model, by deleting some small and unnecessary elements for painting, and by simplifying your model as mush as possible.

If you load STL or PLY model, we recommend you to make an uniform mesh, using special software, just about like on the picture below.



When you simplify your model, according to these recommendations, you will get much smoother and faster painting simulation.

2. Preparing operation for painting

▼ Click here to expand..

Before creating a painting operation, you should first load the part on the "Model" page. After creating the operation, you should perform the following actions:

- set the part for the operation that should be painted;
- set the empty workpiece for the operation;
- select the Painting simulation type in the operation parameters.

3. Choose Painting type

▼ Click here to expand..

There are two types of painting simulation in the system:

- Simple. This type of painting simulation shows which areas of the model will be painted. After selecting this type, it remains only to calculate the toolpath and on the "Simulation" page to see the result of painting.
- Thickness measurement. This type shows not only the areas to be painted, but also displays the approximate thickness of the future paintwork. To use it correctly, you should perform the following steps:

1. set the work feed
2. set the paint flow rate

After that, you can perform the calculation of the operation and go to the Simulation page.

4. Change parameters of the simulation

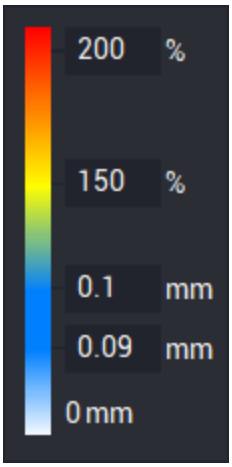
▼ Click here to expand..

For better performance or for better quality of the simulation process you can change the model resolution and tolerance of the toolpath. You can read more about this information here.

5. Control thickness of your painting result

▼ Click here to expand..

To control thickness of the painting simulation result, click on the "Verify compare" button (you can find it here), and you will see the form, like on a picture below:



This form allows you to manage parameters for visual control of paint distribution on a model.

We use a simple colormap to make the process easier. It starts with a transparent color and gradually changes to the current operation color. After that, it changes to a yellow color and then to red.

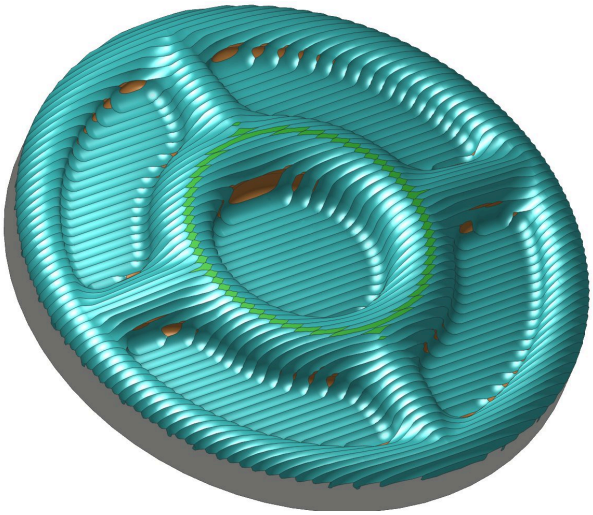
The current operation color represents the acceptable thickness values. To set it, you need to specify a range of values, which means that any value of thickness inside this range will be acceptable for you. Such areas on the model will be shown in "full" color without transparency.

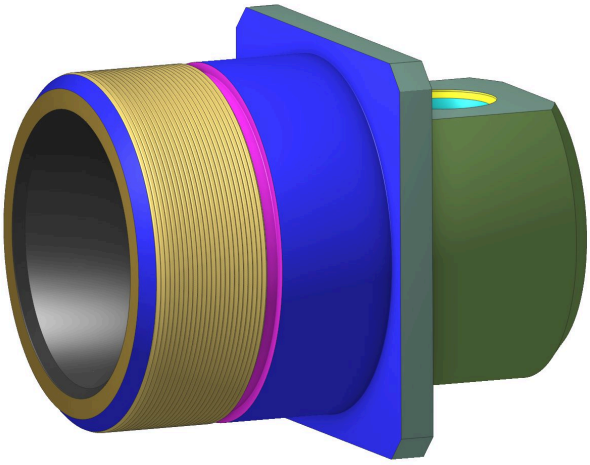
If the paint thickness is not enough in some areas of the model, they will be displayed using the same color, but with some degree of transparency. The degree of transparency depends on how far the current thickness value is from the acceptable value.

If the paint thickness is more than enough, you will see yellow and red colors on your model. These values are set as a percentage of the maximum acceptable thickness (e.g. 150% and 200%).

To save changes press Enter or move focus to another field.

Solid Simulation





Application area:

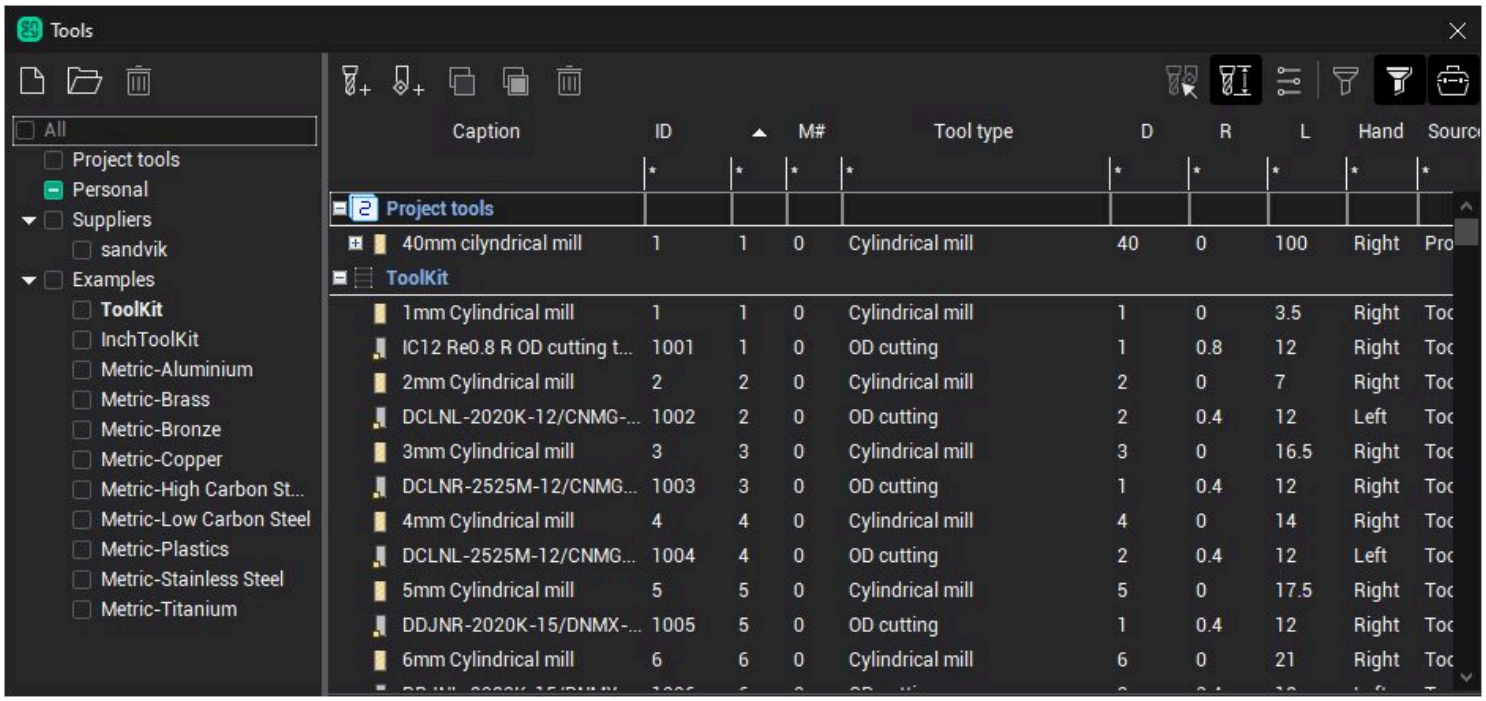
The solid simulation method represents the model of the workpiece as a polygonal solid. This allows for exact representation of complex parts with many small features and sharp corners. The method is well suited for simulating machining of prismatic parts. Also the solid method is really good for turn-milling projects, especially for simulation of thread milling.

Notice: simulation of milling for 5-axis toolpaths works with limitations in beta version

Machining tool features

This section describes the Tools window, the project tool list, shaped tool creation, the Holders (*.osd) window, the machining tools import API, and tool assemblies.

Tools window



The purpose of the Tools window is to view and edit project tools, operations tools, create and fill tool libraries.

This window can be opened in several different ways:

- when you click the  Tools button on the main toolbar in Machining mode;
- open the operation parameters window on the Tool tab;
- double clicking on the name in the tool column in the list of operations;
- clicking on the tool select button in the property inspector header.

Depending on the method of opening, the appearance may slightly differ (the availability of some buttons changes; the sizes of panels and columns are remembered for each of the window opening modes separately).

In the central part of the window is a list of tools. Each line corresponds to a separate tool. Strings can be nested in each other. At the top level there are always storage of tools (project or library), the second level are tools and at the last nested level are operations tools that use a tool with ID same as the parent tool. Each of them can be edited individually.

Nested operations tools can be dragged from one tool node to another. This allows you to quickly change the tool used by the operation.

The right part of the top toolbar is occupied by filter buttons, with which you can limit the list of visible tools.

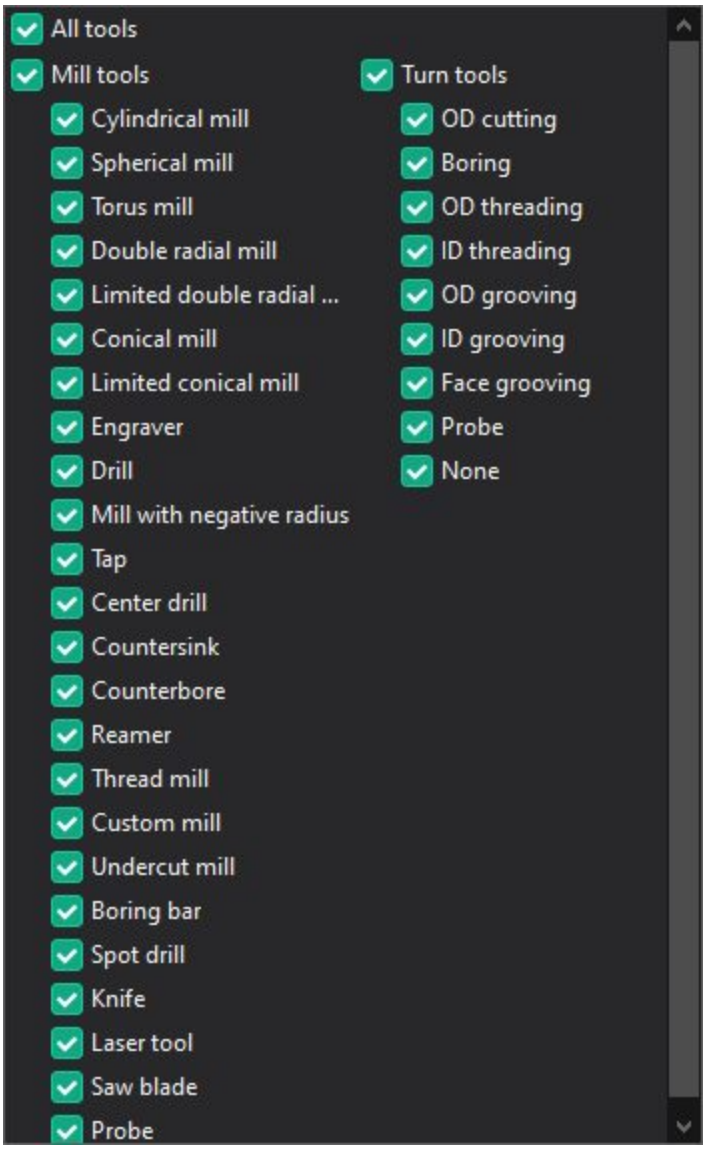
-  — show properties of the tool under cursor
-  — turns on the visibility of the columns of the main geometric properties of tools.
-  — turns on the visibility of the cutting condition columns.
-  — makes visible the tools used in project operations.
-  — makes visible tools that are not used in any of the project operations.

In addition, you can adjust the visibility of rows by imposing additional restrictions on the values of parameters that are displayed in each of the columns. To do this, use the topmost row of table filters (with *). You need to click the mouse in the desired cell and enter some value.

For string parameters, the limit is determined by matching characters. For numeric parameters, you can use additional operators to specify the constraint:

- 123 simple numerical value - the lines will be displayed in which this parameter strictly corresponds to the specified value;
- <123 - expression with the "less" operator - the lines in which this parameter is strictly less than the specified value will be displayed;
- <=123 - expression with the "less or equal" operator - the rows will be displayed in which this parameter is less than or equal to the specified value;
- >123 - expression with the "greater" operator - the lines in which this parameter is strictly greater than the specified value will be displayed;
- =123 - expression with the "greater or equal" operator - the lines in which this parameter is greater than or equal to the specified value will be displayed;
- 12..34 or 12:34 - expression with the "range" operator - the lines in which this parameter is included in the specified range of values will be displayed.

The filter by the type of tool is specified in a special way. When clicking in the filter cell in the Tool type column an additional window appears on the screen. Here you can select the desired tool types with checkmarks.



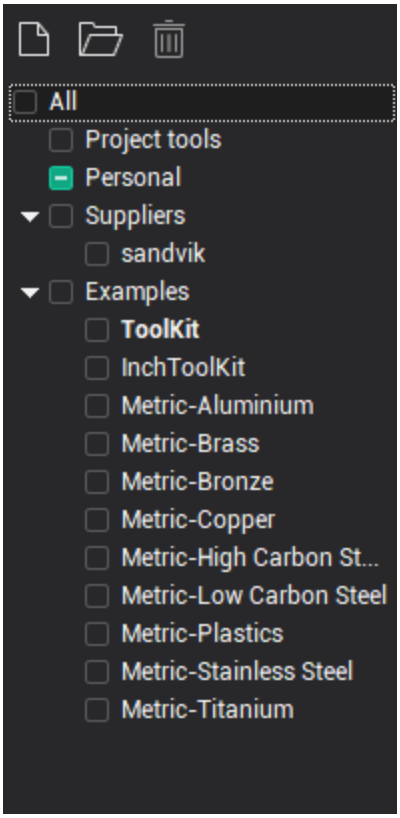
Sorting table rows is controlled by clicking on the desired column header.

The following columns are available.

- ID - ID of the tool, independent of the location of the tool on the machine.
- Caption - the name of the tool or the name of the operation depending on the type of node.
- # - tool number in G code. Usually it corresponds to the number of position in which the tool is attached to the machine. The number in this column can be displayed in red. This means that it is incorrect and conflicts with some other tool that is used in the project. Tools are considered conflicting if they have a different ID, but
    1. have the same tool number, magazine number and corrector number at the same time;
    2. or inserted into one connector of the machine (in one position of the turret).
- M# - magazine number, for the case of machine with several tool magazines.
- Tool type - indicates the type of machining tool (cylindrical mill, drill, boring tool, etc.)
- L - tool overall length.
- D - for milling tools, the diameter of the tool; for turning tools, it either does not fill up or is filled with different parameters depending on the tool subtype.

- R - key tool radius (different depending on the specific type).
- F - working feed.
- S - spindle speed.
- Direction - spindle rotation direction.
- Coolant - set of included cooling piping.

In the left part of the window there is a panel for choosing libraries. The list of tools is always displayed for libraries which are nested in the selected node and which are marked. In addition to the library, a list of tools for the current project can be selected. Favorite library is shown in bold. The Favorite Library is the one that appears first in the list and from which the system tries to automatically select tools when creating operations. Libraries from Suppliers and Examples folders are available in read-only mode.



A more detailed description of the actions that can be performed on libraries using this panel can be found in the Managing libraries chapter.

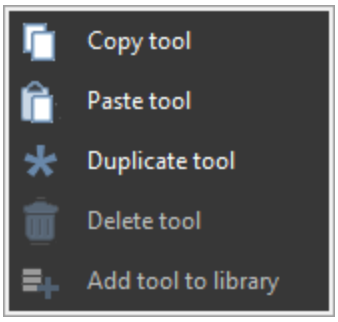
On the top toolbar focused actions on the tools.

- — Adding a new milling tool.
- — Adding a new turning tool.
- — Copy selected tools to clipboard. Multiple tools for copying can be selected at once.

-  — Paste copied tools from clipboard. Using copy-paste, you can quickly transfer tools from one library to another.
-  — Delete selected tools. Several tools can also be selected for deletion.

From the Popup menu of a tool you can additionally

- Duplicate tool - quick duplicate selected tool.
- Add tool to library - quick add selected tool to the favorite library.

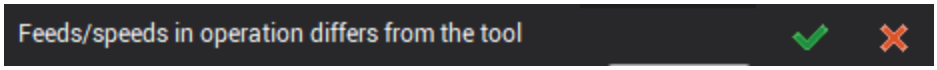


At the bottom of the tool window is a panel for viewing and editing properties. The window is built on the principle of "I'm editing what I see", i.e. editing is always performed for the tool that is in focus in the list. Edited can be any of the tools in list. Right on the panel displays a live preview of the edited tool. In the main graphics window, the model of the selected tool is also drawn in the correct position on the machine.

The list of properties depends on the type of tool. The values for some properties can be displayed in green font. This means that the property has a default value, which depends on the values of other properties. For example, the tool name contains the diameter. Then, when changing the diameter, the name will be updated automatically. To return the property to its default value, you need to delete the entire contents in the edit field using the Delete key on the keyboard.

Properties on the editing panel are divided into several tabs according to the meaning of the information.

- Geometry - the main geometric dimensions of the tool, as well as its type and name.
- Numbers - the tool ID is independent of the machine, the tool number on the machine, the magazine number, the numbers of the correctors.
- Design - Number of teeth, maximum plunging angle, units, tool durability.
- Tooling - tool overhangs are set, as well as the type and location of the adjustment points.
- Holder - You can edit the name and size of the holder, specify the CAD-model file (in stl or osd format). Here you can also call a dialog to select (and edit) the holder of a milling tool from the library.

- Feeds/Speeds. Here you can set the mode, speed and direction of rotation of the spindle, the work feed and the enabled cooling pipes. If a line with an operation is selected in the tool list, then the cutting conditions of the operation are displayed and edited; if the project or library tool is selected, then the conditions of tool. If the conditions in operation and the conditions in tool are different, then the corresponding caption appears in the header of the panel, as well as buttons that allow them to be synchronized. 

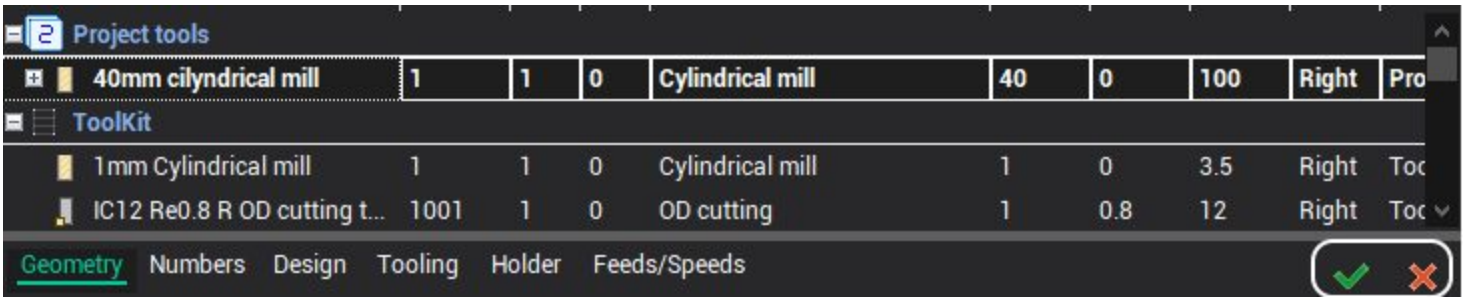
If the operation tool is being edited,

- The ✔ Apply button assigns cutting conditions from the operation to the project tool,
- The ✖ Reset button copies the cutting data from the project tool to the operation.

If a project tool is being edited,

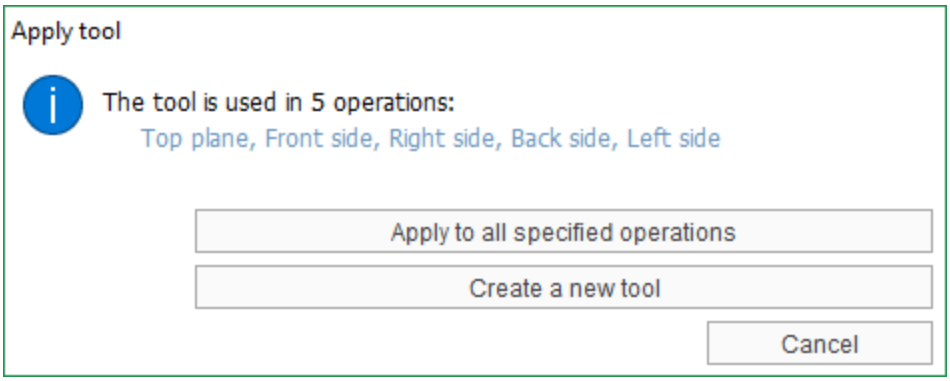
- The ✔ Apply button copies the conditions from the project tool to all operations using this tool.
- The ✖ Reset button copies the cutting conditions from the first operation tool, whose modes are different, to the project tool.

Similar buttons to apply and reset are also available for general tool parameters.



They appear at the top right of the edit panel if the operations tool and the project tool are different from each other (different tools are highlighted in bold in the list). Buttons allow you to easily synchronize project tools and operations, as well as allow you to roll back random or unwanted changes.

If there are several operations using this tool and the parameters of the tool in these operations are different, then after pressing the Apply button, an additional dialog box will be displayed.



The Apply to all specified operations button will copy the tool parameters from the current operation to the project tool, as well as into all operations that use the tool with the same ID.

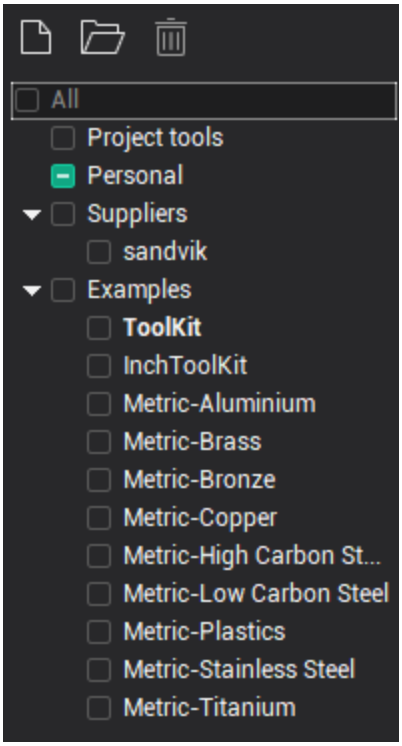
The Create a new tool button assigns a new ID to the tool of the current operation (it will be generated automatically) and creates a copy of this tool in the global list of project tools. Thus, the connection with copies of the tool in other operations is broken.

Some tools in the list may be read-only, as evidenced by the inscription on the right above the editing panel. This was done, firstly, in order to limit some undesirable work scenarios, for example, accidentally edited the project tool instead of an operation, and after closing the window did not see the changes made, secondly, to be able to roll back random changes.

If the window is opened in the tool selection mode for the current operation, the button "Select tool for the operation" becomes available at the bottom right. When pressed, the tool selected in the list will be applied to the operation.

Manage libraries

To manage tool libraries and project's tools, use the panel that is in the left part of the Tools window.



The panel displays all tool libraries connected to the system and divides them into several groups:

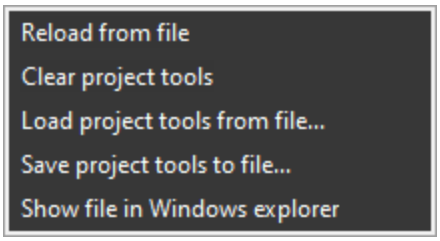
- All - a root node that makes it convenient to view tools from all sources of interest at once.
- Project tools - the list of tools for the current project.
- Suppliers - the list of libraries from the "$(CommonAppData)\Libraries\Tools\Suppliers" folder and which are installed by the distribution. Libraries from this group are available in read-only mode.
- Examples - the list of libraries from the "$(CommonAppData)\Libraries\Tools\Examples" folder and which are installed by the distribution as an example. Libraries from this group are available in read-only mode.
- Personal - the list of libraries located elsewhere in the file system. If the library is located at "$(CommonAppData)\Libraries\Tools" folder then the system will add it to this group automatically.

The list of tools is always displayed for libraries which are nested in the selected node and which are marked.

There is a concept Favorite library - this is a library that appears first in the list, just after Project tools. When creating a new operation, a tool is automatically searched for that is suitable for it. First, the search is performed in the list of project tools, then in the favorite library. Favorite library is shown in bold and can be changed from popup menu.

In the first place is always the Project tools item. This is a list of the tools of the current project, which is stored inside the stcp file, but can be saved in a separate text file (XML with *.tom file name extension).

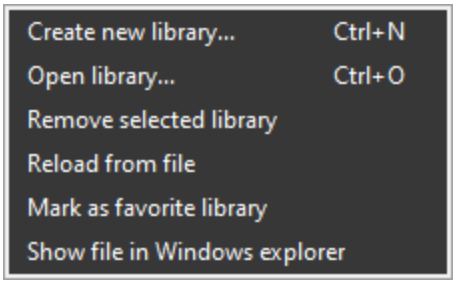
You can learn more about it in the Project tools chapter. The key difference from the library is that along with the list of tools, the state of the machine is preserved, which means the correct location of the tool in the machine schema, the placement of the turret blocks. Those it is the tools setup list. The project tool list file is not intended to store too many tools.



The following actions are available for the list of project tools from the popup menu.

- Save project tools to file... - saving to an external *.tom file. The system remembers the path to the selected file and will automatically load tools from this file when creating a new project with the same machine.
- Load project tools from file... - loading from an external *.tom file. The system remembers the path to the selected file and will automatically load tools from this file when creating a new project with the same machine.
- Reload from file - reload from a previously saved *.tom file (without opening the file open dialog).
- Clear project tools - removal of all unused tools from the project and break the connection to the external *.tom file if the project tools list was saved to the external file before.
- Show file in Windows explorer - opens the Windows explorer and selects the *.tom file if the project refers to an existing one.

Subsequent items on the list are regular tool libraries. A library is a SQLite database format file. Only tools are stored in libraries and no information about the machine is stored. They can easily hold a very large number of tools (tens of thousands).



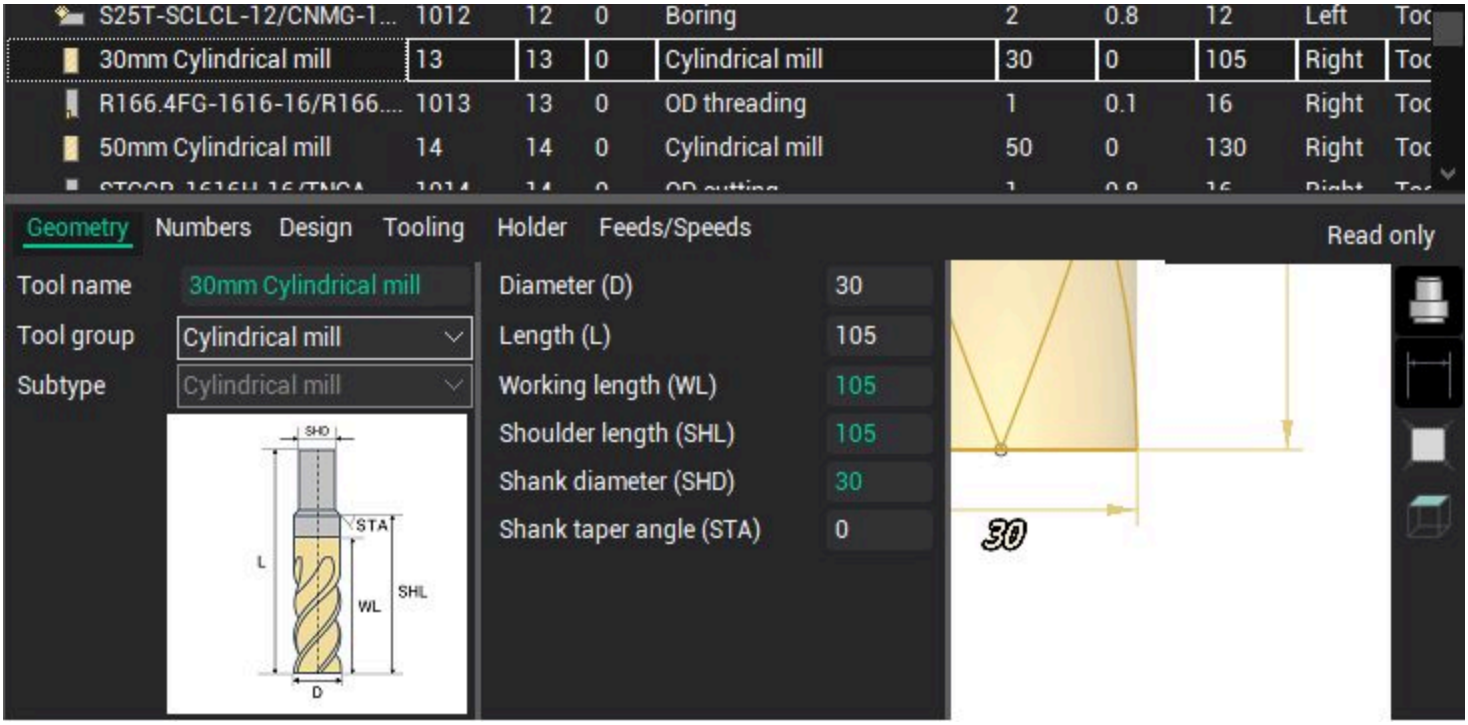
The following actions are available for the tool library from the popup menu and buttons on the top toolbar:

- Create new library... - opens a file save dialog where you can select the name and location of the new tool library *.db file.
- Open library... - opens a file open dialog where you can select an existing tool library *.db or *.csv file and system will add it to the list.

- Remove selected library - removes the library from the list.
- Save selected library - saves changes to a file. It is necessary only for special cases, since when you close the tool window, all changes in the libraries are automatically saved. If the library has old *.csv format then a message box will appear with a proposal to convert this file to a new *.db format.
- Save all libraries - saves changes to files for all connected libraries (when you close the tool window, all changes in the libraries are automatically saved).
- Reload from file - reloads all data from the file and all unsaved changes will be lost.
- Mark as favorite library - makes the selected library a favorite.
- Show file in Windows explorer - opens the Windows explorer and selects the library file.

The list of linked libraries, the state of their checkboxes and which one is favorite is saved in the system settings file (*.cfg).

Milling tool editing



When you select a milling tool in the Tools window, a panel with its properties is displayed in the lower part of the window, as shown in the figure above.

The system implements the following types of milling tools, which differ in the list of available geometric parameters:

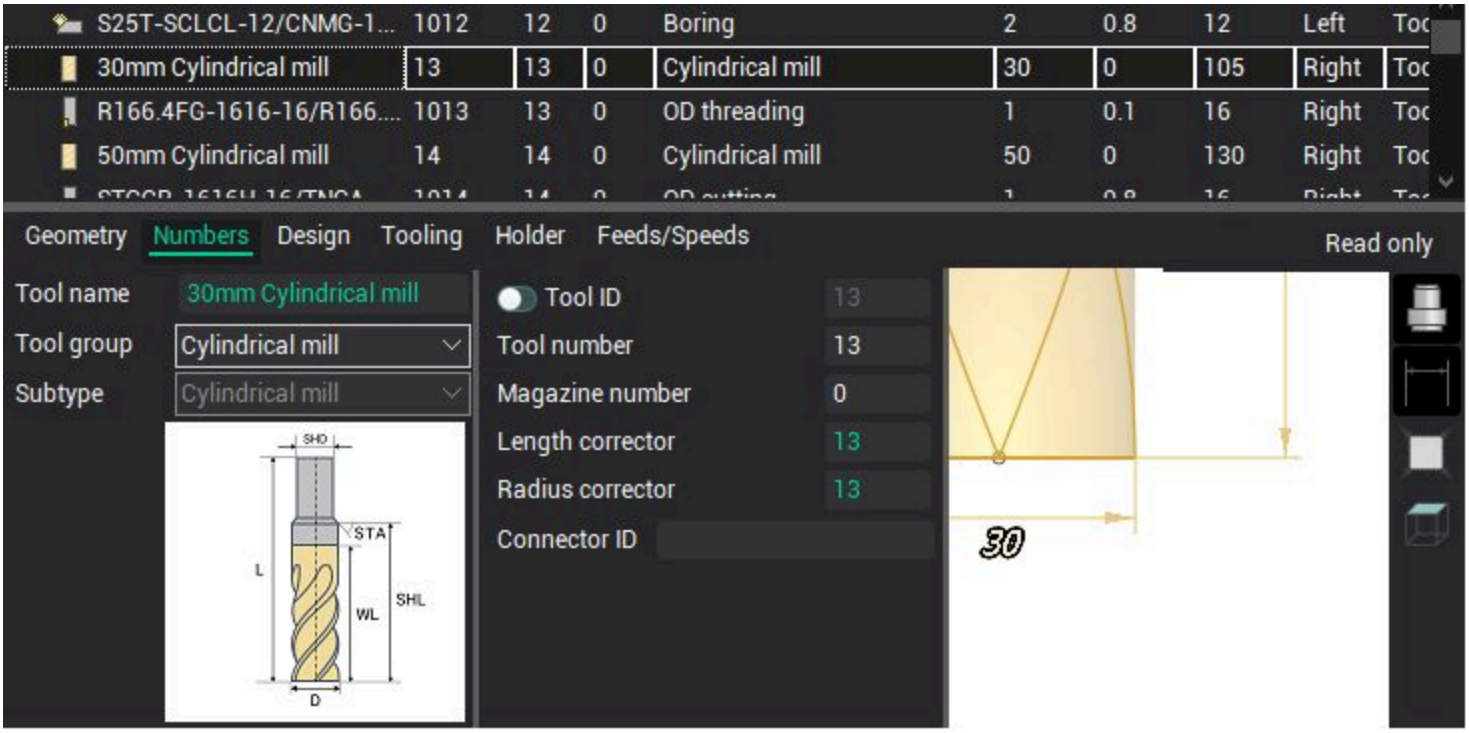
- Cylindrical mill;
- Spherical mill;
- Torus mill;
- Double radius mill;
- Limited double radius mill;
- Conical mill;
- Mill with negative radius;
- Limited conical mill;
- Engraver;
- Drill;
- Cutter;
- Tap;
- Thread mill;
- Center drill;
- Countersink;
- T-slot mill;
- Shaped mill;

- Knife;
- Saw blade;
- Probing;
- Spray;
- etc.

You need to select the group and subtype of the tool correctly to see the list of its geometric properties in the window on the Geometry tab.

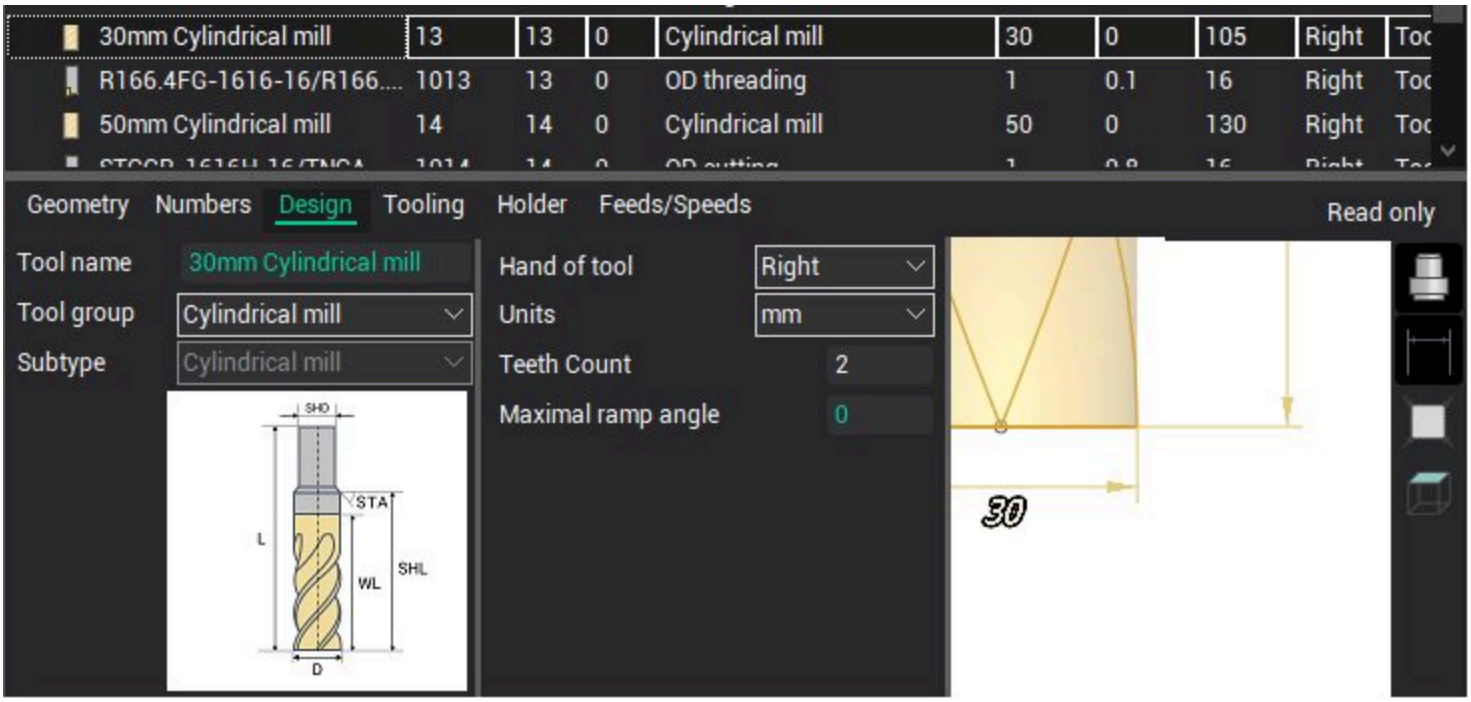
On the Numbers page, the tool identifier properties are displayed.

- ID is a unique tool identifier independent of its location on the machine.
- Tool number - the tool number on the machine. Usually corresponds to the position number in which the tool is fixed on the machine.
- The tool magazine number is used if there are more than one magazines on the machine.
- The corrector number for the length and for the radius is the record number corresponding to the tool in the tool offset table in the machine.



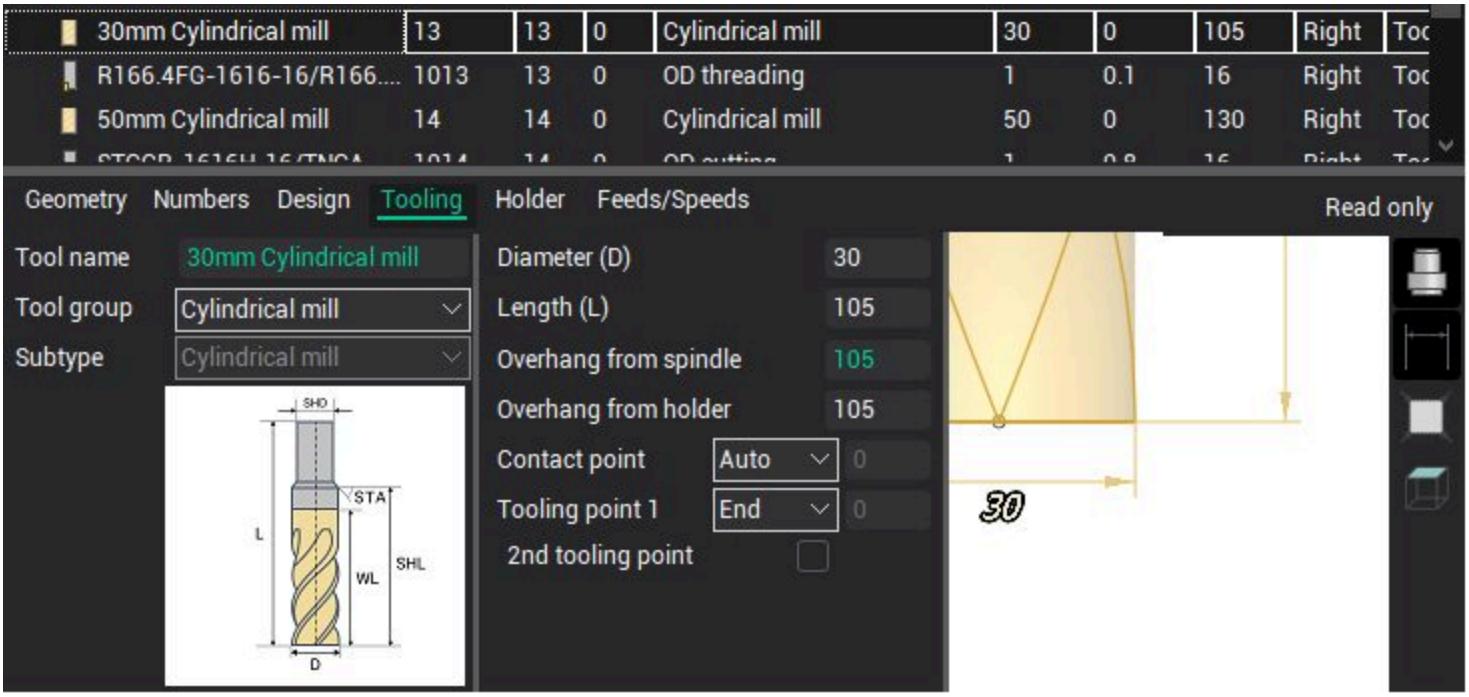
The following parameters are listed on the Design page.

- The number of teeth.
- Maximal plunge angle
- Tool units (mm or inch).
- Tool durability in minutes.



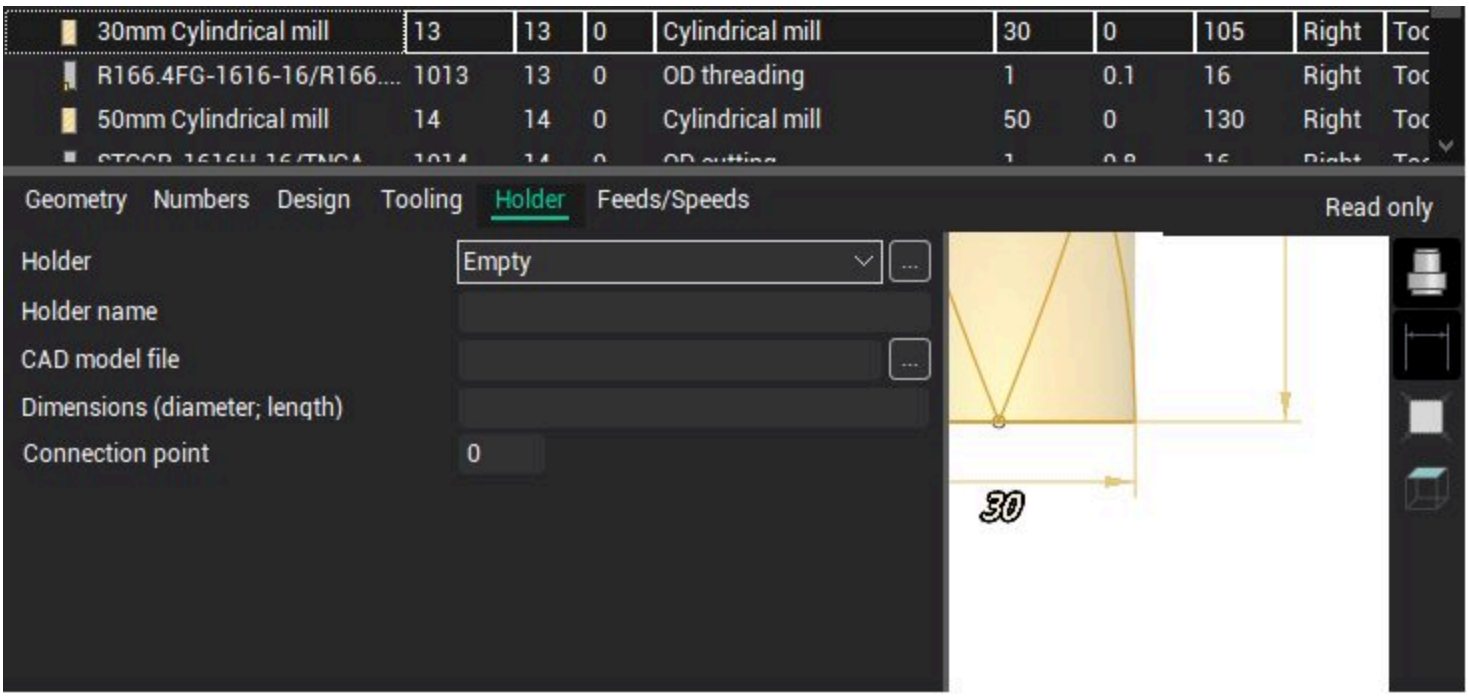
The Tooling tab contains such parameters.

- Tool overhang - distance from tool fixing point in spindle to tool tip.
- Tooling point 1 - point on the tool, the movement of which is defined by the G-code relative to zero of the part.
- Tooling point 2 - used with some types of tools to ensure that the G-code is independent of the size of the tool.
- Tool contact point - point on the tool with which it should primarily touch the geometric elements specified in the job assignment of the operation.



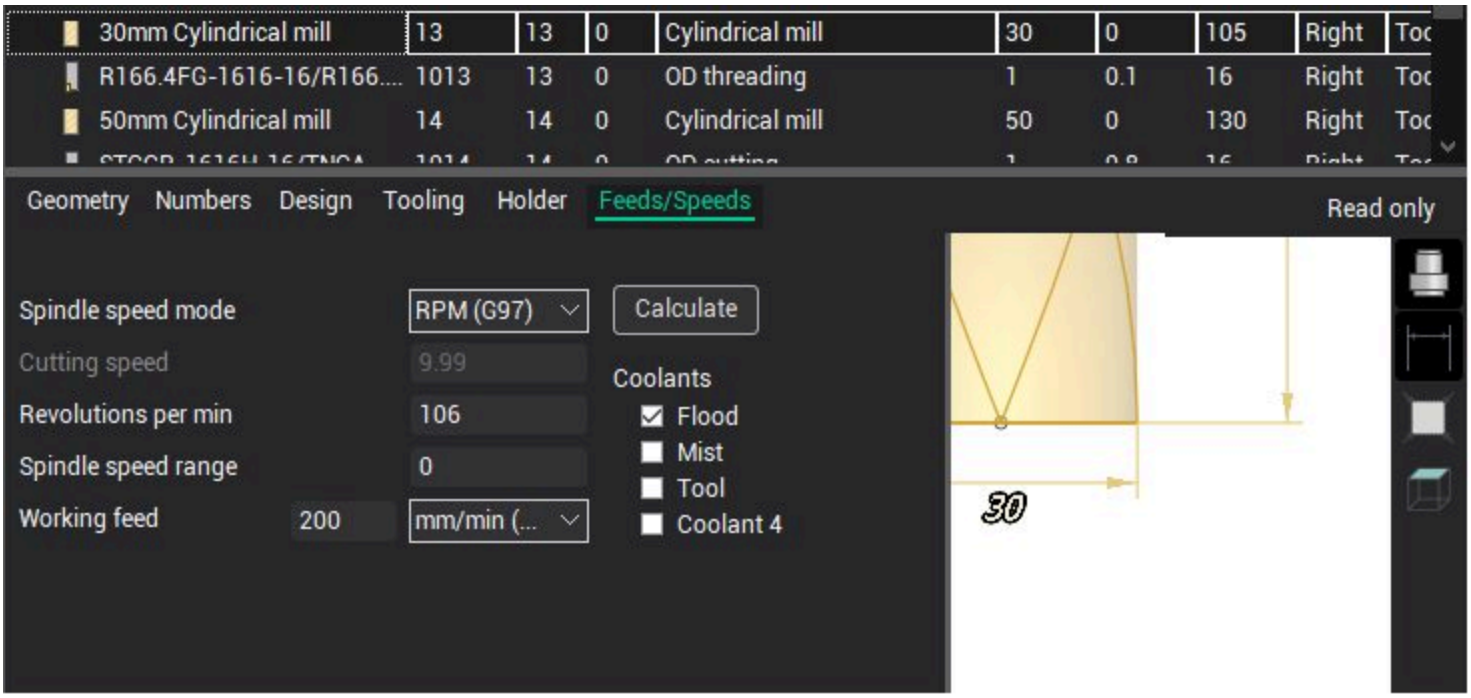
The Holder page allows you to set the parameters of the holder.

- Holder - drop-down list of holders from current library and project.
- Holder name - arbitrary text string.
- Holder dimensions - a sequence of pairs of numbers separated by a semicolon, defining the diameter and length for each of the steps.
- Connect point - distance from the highest point of the holder to the connection point of the holder with the machine spindle.
- CAD-model file. The button opens a standard file selection window. You can specify files in the format *.stl and *.osd. Sets the 3D model of the holder for visualization and control of collisions during simulation.
- Clicking on the "..." button opens a new dialog for selecting and editing the holder.

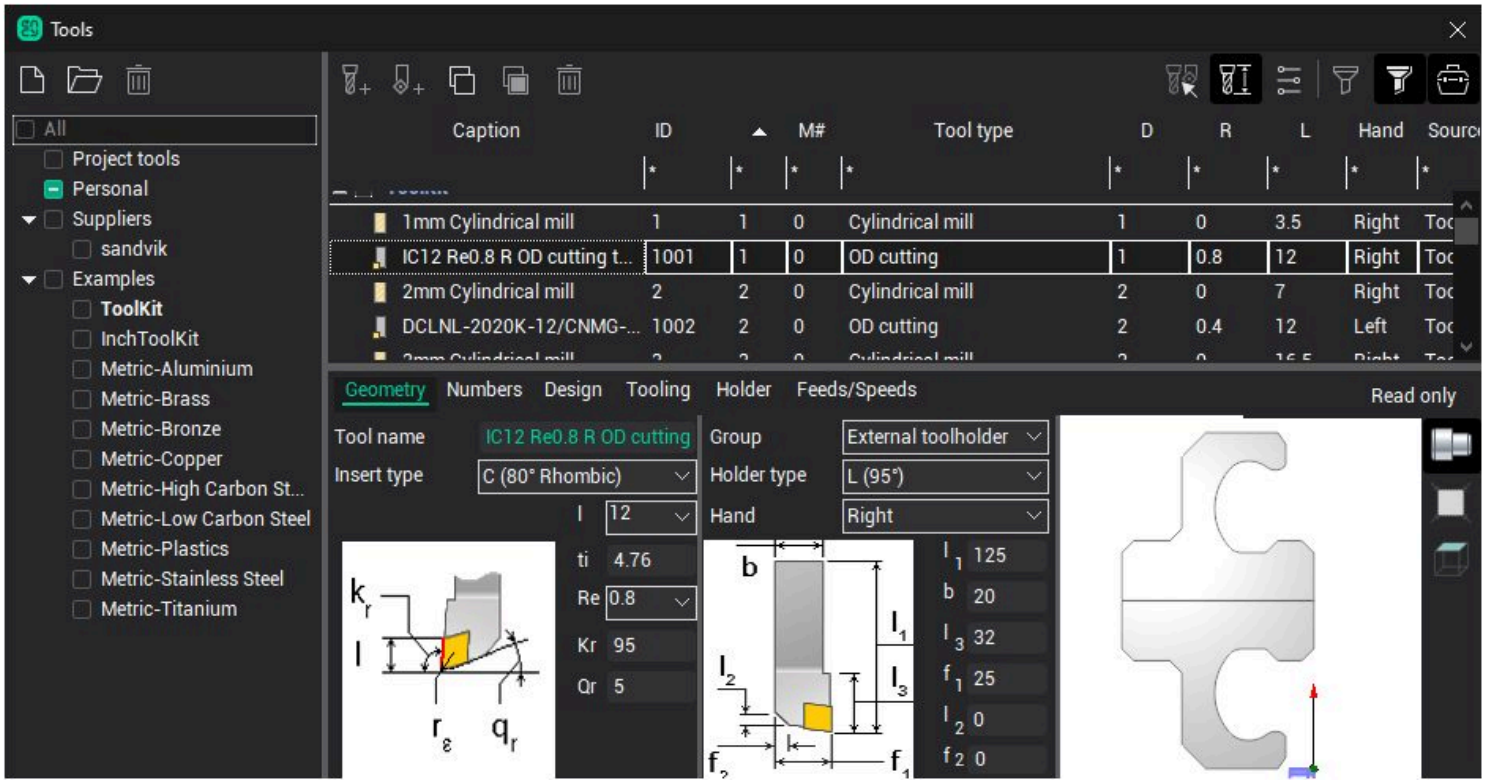


On the Feeds/Speeds tab you can edit such parameters as

- Spindle rotation mode: constant revolutions or constant surface speed.
- Cutting speed.
- Spindle revolutions.
- Spindle speed range.
- Spindle rotation direction.
- The value and dimension of the working feed.
- Enabled cooling tubes.


30mm Cylindrical mill 13 13 0 Cylindrical mill 30 0 105 Right
R166.4FG-1616-16/R166.... 1013 13 0 OD threading 1 0.1 16 Right
50mm Cylindrical mill 14 14 0 Cylindrical mill 50 0 130 Right
Geometry Numbers Design Tooling Holder Feeds/Speeds
Read only
Spindle speed mode
RPM (G97)
Calculate
Cutting speed
9.99
Revolutions per min
106
Spindle speed range
0
Working feed
200
mm/min (...
Coolants
Flood
Mist
Tool
Coolant 4
30

Turn tool editing



When you select a turning tool in the Tools window a panel with its properties is displayed at the bottom of the window, as shown in the figure above.

Properties on the panel are divided into several tabs.

The Geometry tab displays the main geometric dimensions of the insert and holder, as well as the type of insert, type of holder, tool name. The composition of properties varies depending on the selected types.

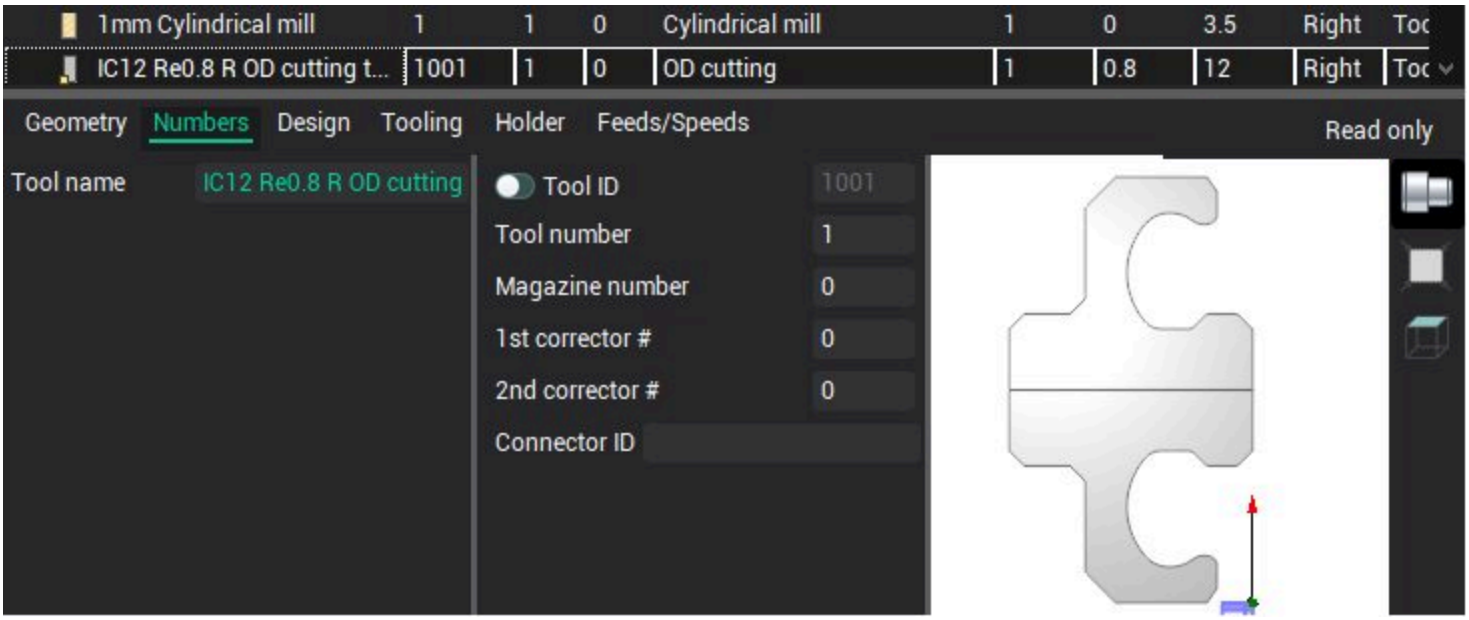
The left panel shows all the main parameters that relate to the insert: type, geometric parameters, taking into account the characteristics of each group of inserts. Inserts compatible with the holder are marked in black, and non-compatible inserts are shown in gray. When choosing an insert type that is not compatible with the current holder, a compatible holder will automatically be selected. In order to specify an insert of any type, it is necessary to select the insert type Custom. This insert is compatible with any holder.

The right panel shows all the main parameters related to the holder: type, geometric parameters with regard to the group, direction. As with inserts, incompatible types are drawn in gray.

On the Numbers page, the tool identifier properties are displayed.

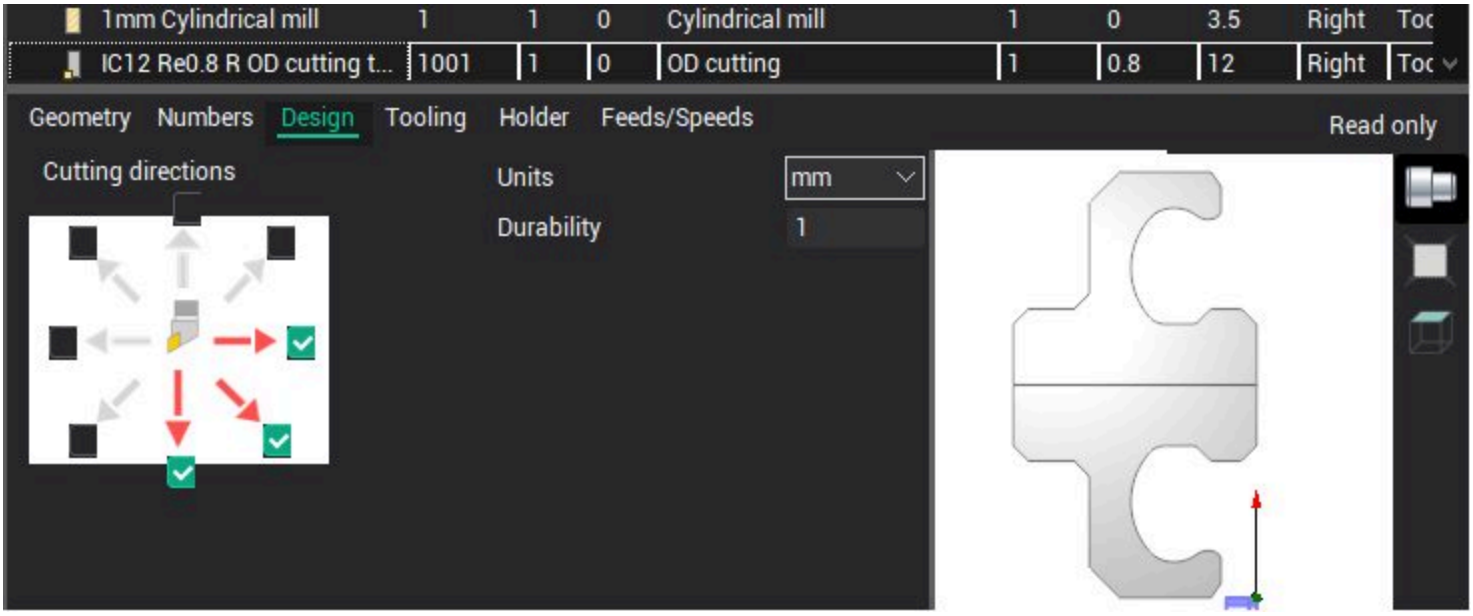
- ID - unique tool identifier independent of its location on the machine.
- Tool number - tool number on the machine. Usually corresponds to the position number in which the tool is fixed on the machine. It can change automatically on turret machines when changing the connector of the machine (the position of the turret head) in the property inspector.

- The tool magazine number is used if there are more than one magazines on the machine.
- Corrector numbers - record numbers corresponding to the tool in the table of tool offsets on the real machine.



The following parameters are listed on the Design page.

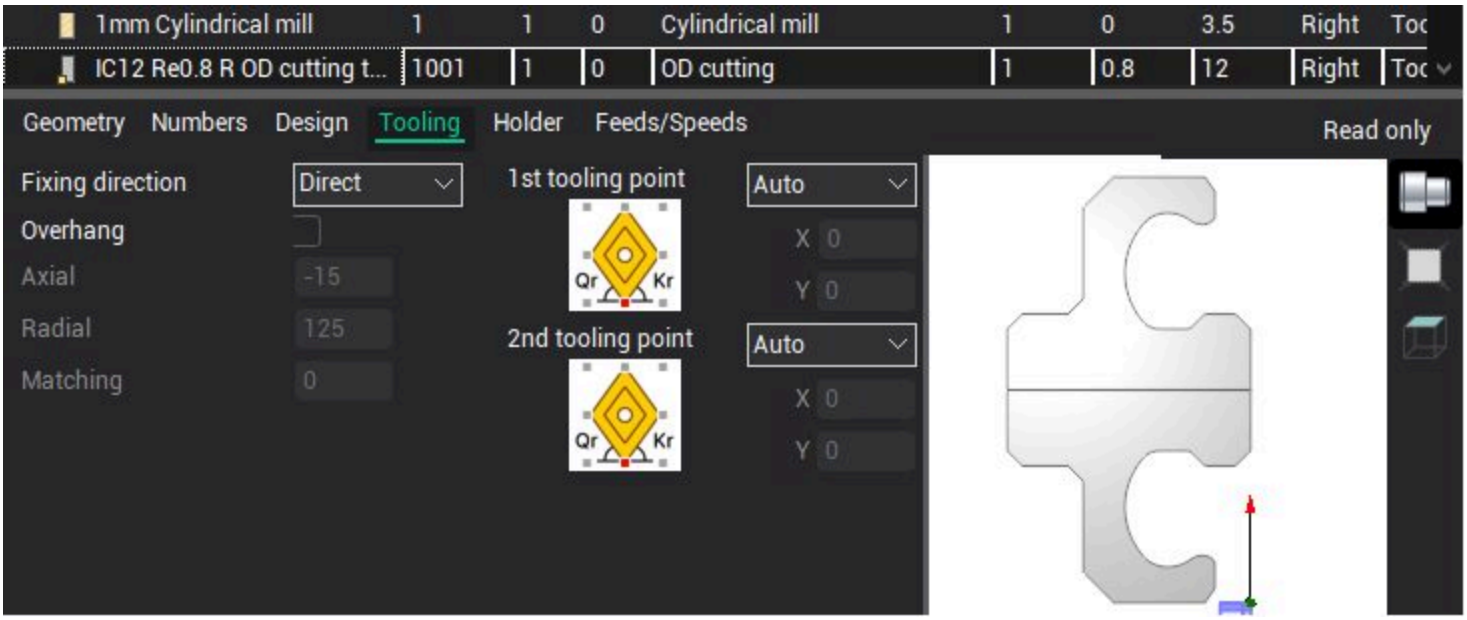
- Permissible cutting directions.
- Tool units (mm or inch).
- Tool durability in minutes.



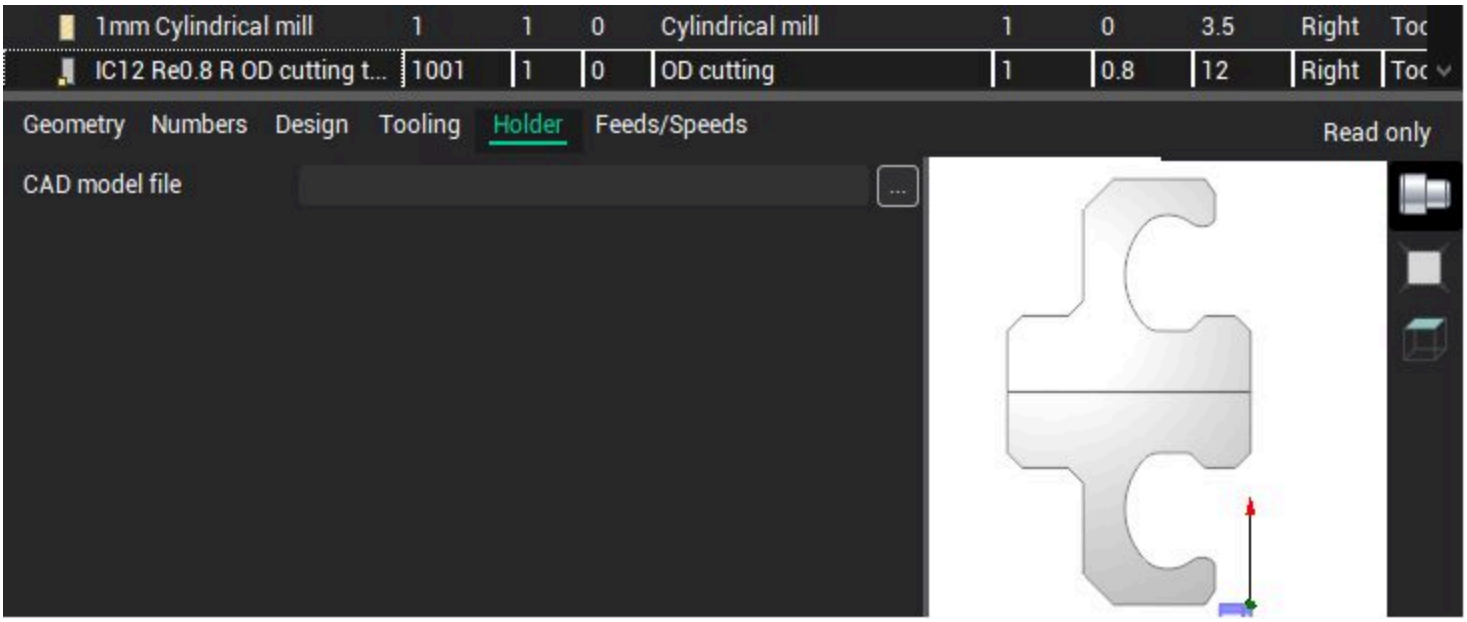
The Tooling tab contains such parameters.

- The tool fixing direction in the turret block - direct or inverted.
- Tool overhang - distance from tool fixing point in spindle to tool tip. If the tick is off, the overhang is calculated automatically according to the dimensions of the tool.

- First tooling point - point on the tool, the movement of which is defined by the G-code relative to zero of the part.
- Second tooling point - used with some types of tools to ensure that the G-code is independent of the size of the tool.



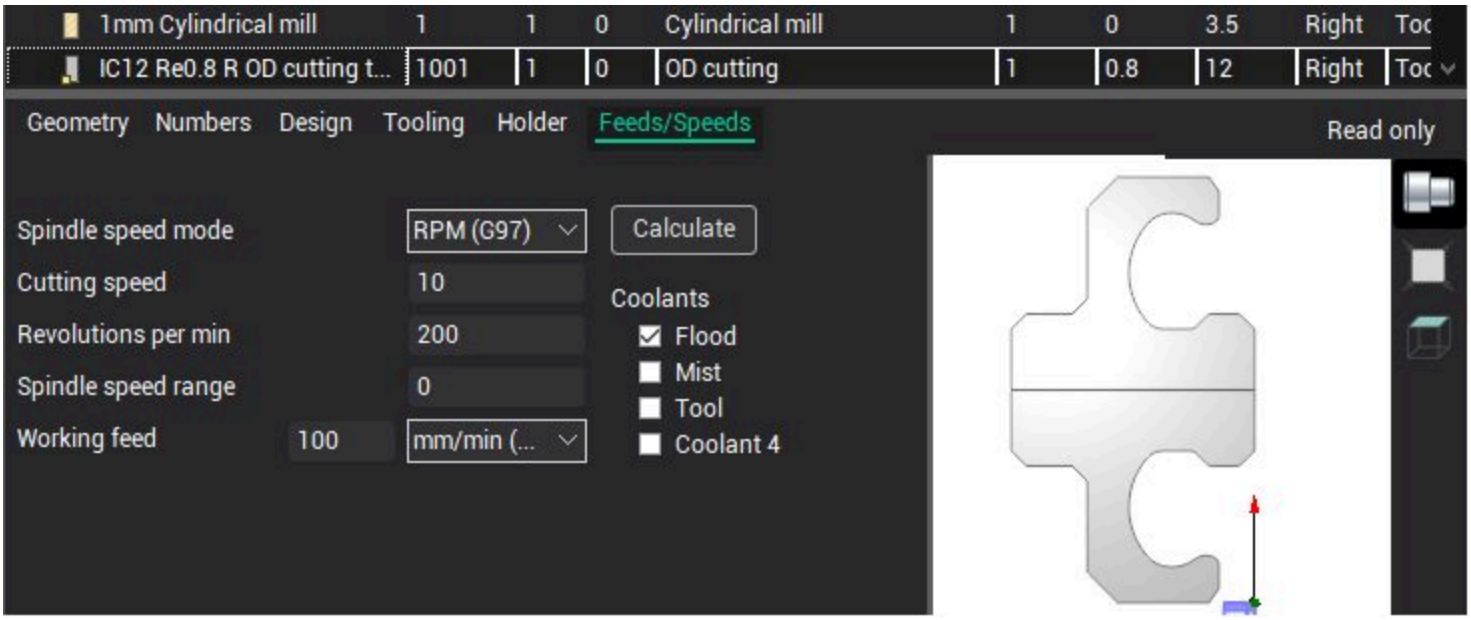
The Holder page allows you to specify the name of the CAD model file. The button opens a standard file selection window. You can specify files in the format *.stl and *.osd. Sets the 3D model of the holder for visualization and collision control during simulation.



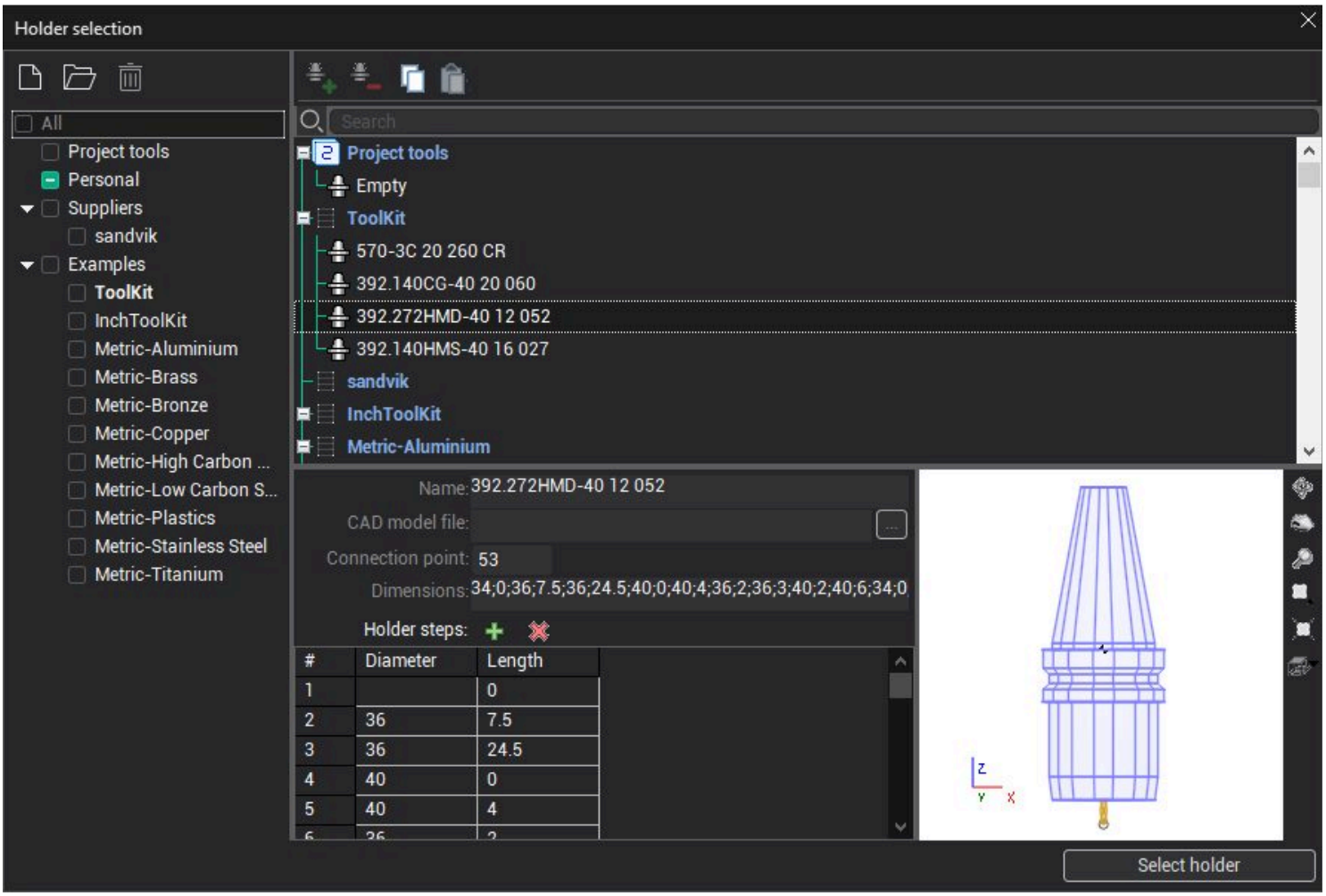
On the Feeds/Speeds tab you can edit such parameters as

- Spindle rotation mode: constant revolutions or constant surface speed.
- Cutting speed.
- Spindle revolutions.

- Spindle speed range.
- Spindle rotation direction.
- The value and dimension of the working feed.
- Enabled cooling tubes.



Mill holder selection window



The window for selecting the holder of the milling tool can be opened by clicking the button on the Holder tab in the Tools window.

In the left part of the window there is a panel for choosing libraries. The list of holders is always displayed for libraries which are nested in the selected node and which are marked. In addition to the library, a list of holders for the current project can be selected. Favorite library is shown in bold. Libraries from Suppliers and Examples folders are available in read-only mode.

In the center is a list of holders from the selected library.

At the bottom of window - the parameters of the holder, selected in the list and a preview of the selected holder.

On the top toolbar there are buttons.

- Add new holder.
- Delete selected holder.
- Copy the selected holder to the clipboard.
- Insert the holder from the clipboard.

The Name field specifies the display name of the holder, an arbitrary text string.

The CAD model file name parameter allows you to specify files in *.stl and *.osd formats. Sets the 3D model of the holder for visualization and control of collisions during simulation. The button opens a standard file selection window.

Connection point - distance from the highest point of the holder to the connection point of the holder with the machine spindle.

The size of the holder can be set either in the text field or in the table of steps. Both views are automatically synchronized with each other during the editing process.

Dimensions in the text field - a sequence of pairs of numbers, separated by a semicolon, defining the diameter and length for each of the steps.

You can use the   buttons to add and remove steps.

Project tool list

The Project tool list is a new functionality (appeared since version 12.0.1), which greatly facilitates the work with machining tools in the system. It allows

- quickly identify and correct errors when filling in the parameters of tools,
- quickly select a previously tuned and correct tool, rather than filling all its parameters again,
- easy to copy a tool from one operation to another,
- store information about feed and speed with the tool, and it is easy to apply them to the operation,
- save the set of tools used in the project together with information about in which position on the machine they are fixed (for example, in which position of the turret and with which block) into a separate file on the disk for later use in other projects,
- transfer machine settings from one project to another (placement of blocks on a turret, approach/return rules, tool change point, discreteness, etc.).

The Tool list is a kind of virtual tool magazine (more precisely, a list of tools from all magazines, if there are more than one). It is stored inside the project. The following is saved in it.

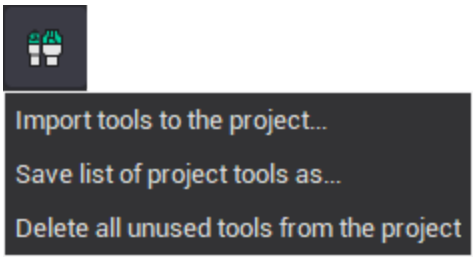
1. A snapshot of the current state of the machine (the full set of properties of the machine, the data superimposed over the scheme).
2. List of tools.

For each tool in the list is stored such data.

- ID - unique identifier of the tool in the list. It allows us to determine that two tools (for example, in two different operations) with the same ID are actually two copies of the same tool. May be string (not necessarily numeric as a number). There cannot be two tools with the same ID in the list.
- Tool number - an integer that is output to the G-code, and in fact determines to which position of the magazine the tool is attached on the real machine.
- Magazine number - an integer identifying a magazine on a real machine.
- Connector ID (attachment point) of the tool in the machine scheme inside the CAM system.
- Tool overhang - tool sizes from the point of attachment to the machine (connector) to the tooling point.
- Corrector numbers, which define the records in the table of the correctors of a real machine where the dimensions (overhangs) of the real tool are stored.
- The name of the tool.
- Tool type - cutter, boring, end mill, drill, tap, etc.
- Geometrical parameters, such as diameter, length, tip radius, etc. The parameter set differs according to the type of tool.
- Tool adapter parameters.
- Machining conditions: working feed, spindle speed, numbers of included cooling pipelines.

The too list is automatically filled by the technological operations during the normal work on the project. The algorithm for adding tools like this. When creating a new operation or changing the parameters of a tool in an existing operation, the system checks if a tool with the same ID is in the list. If there is no such tool, the system adds a copy of the operation tool to the list. At the same time, the system remembers that this is a new tool, i.e. it is not yet stored in the list on the hard disk, but is only present in memory. If after this you delete all the operations that use such a "new" (not yet saved to disk) tool, then it is automatically removed from the list. If you save the list of tools, the tool moves from the "new" state to the saved state. Saved tools are not automatically deleted or updated. They can only be deleted from the list by the user manually with the corresponding button in the tool selection window.

The Tools button is located on the top toolbar and are displayed in the Technology mode. When you click on it the Tools window opens. The button has a drop-down menu with an additional set of features.



Save tool list as... - saves the list of tools to the file on a hard drive. The file name will be requested - the standard file saving dialog will open. The list of tools is saved to an XML text file with the extension *.tom. In the system settings, the name of the *.tom file and the name of the current machine are stored. When creating a new project (or when changing the machine), if the name of the *.tom file is found in the system settings for the current machine, the list of tools from this file will be automatically loaded.

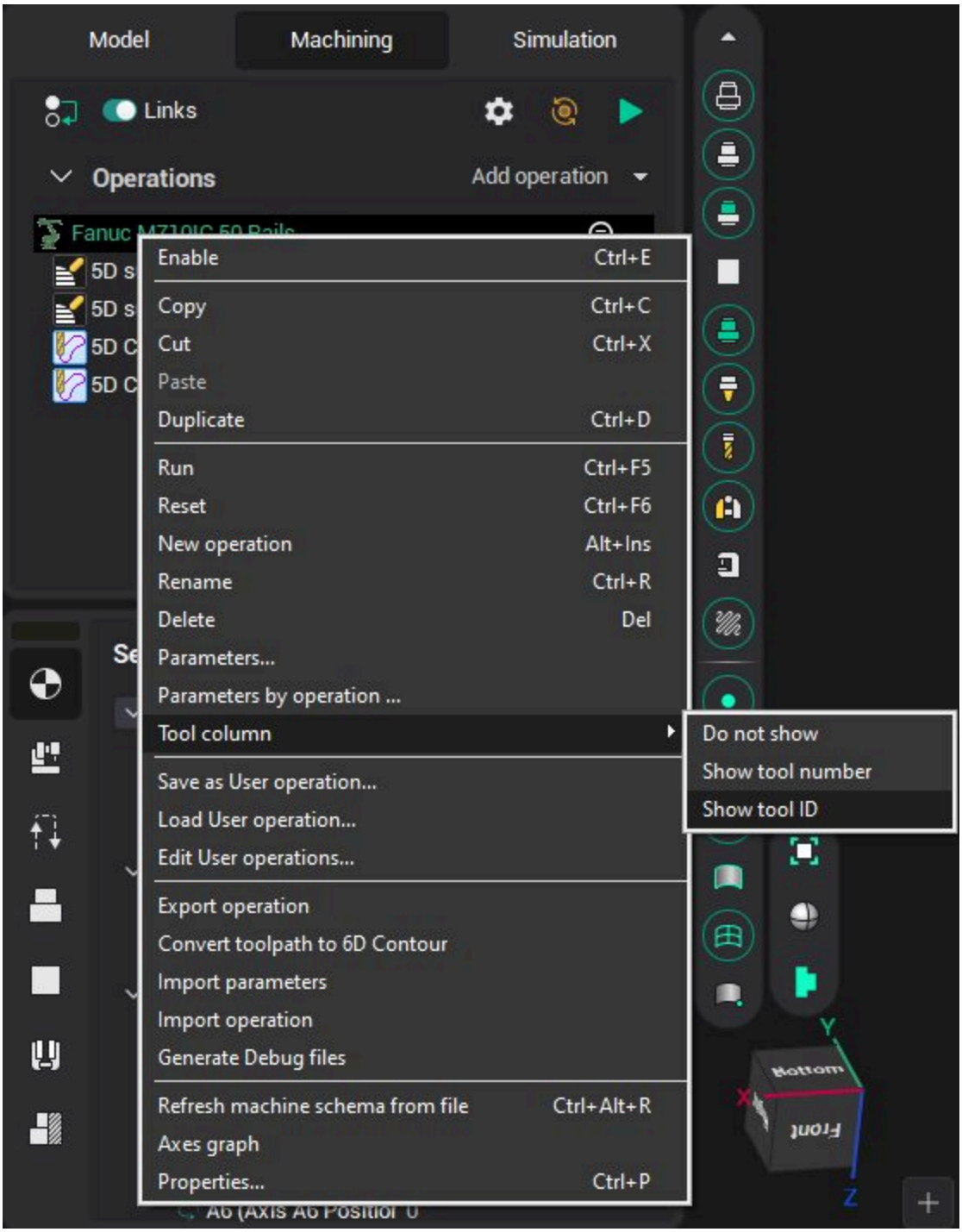
Load existing tool list... - loads a tool list from a file. A standard file selection dialog opens. The name of the loaded *.tom file is stored in the system settings. It also allows you to load tools from the tool library file (*.csv or *.db + *.properties formats) and add them to the current list. It should be noted that there is no information in the tool library about which tool connector of the machine the tool is attached to, so this information may not be filled correctly when importing into the tool list.

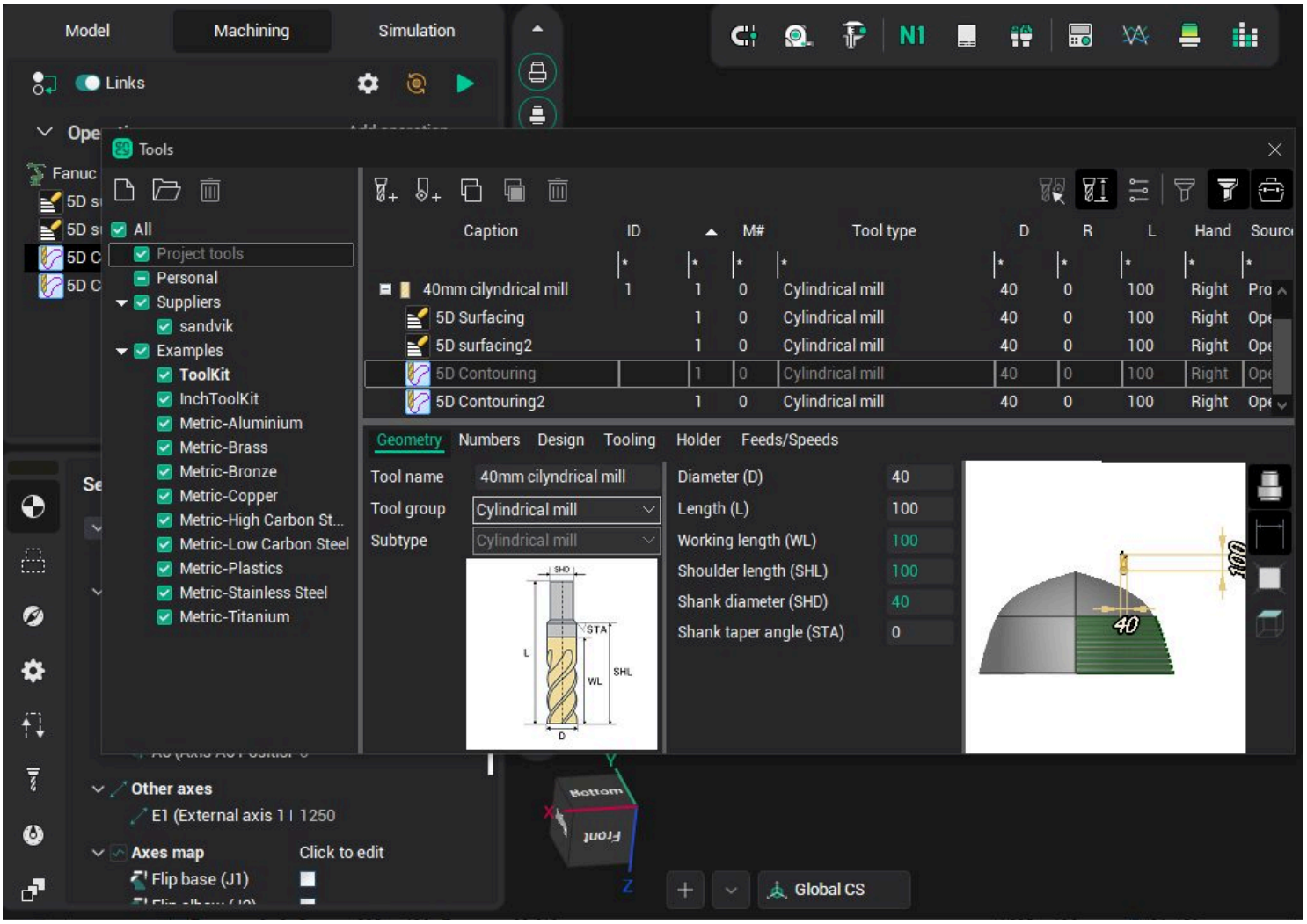
Delete all unused tools from the project - clears the tool list, deletes all unused tools and breaks the link to the file if the tools were previously saved to a file. The memorized file name for the current machine is deleted from the system settings. The new empty list of tools is automatically filled in according to the list of operations of the current project according to the algorithm described above.

The existing functionality of the tool list allows you not only to store tools in one place, but also implements a mechanism for controlling the parameters of tools within project operations. Consider how this is implemented.

The project contains many copies of each tool - one copy in each operation and one copy in the global list of tools. The indication that the two tools are copies of each other, and not different tools is the common tool ID. All these copies may differ from each other. Control of differences between different copies of tools relative to each other is carried out on the Technology tab in the tree of operations and also in the separate tool list window.

A separate tool column can be displayed in the operation tree. The visibility of the tool column is enabled in the context menu of the operations tree. In this column, for each operation of the project, the tool identifier and its name are displayed. The displayed identifier can be either the tool ID or its number.





The difference of tools from each other is depicted by using different colors and fonts.

1. The color of the tool name is used to indicate differences in the operation tool from the same tool in the global tool list:
2. gray - the operation tool does not differ from the tool in the list;
3. black - the operation tool is different from the tool in the list;
4. Font thickness is used to indicate the difference between the tool and same tools inside another operations of the current project:
5. normal font - the tool of this operation does not differ from the tools of other operations;
6. bold - the tool of this operation is different from the tools of other operations.
7. The tool identifier color in the operations tree column is used to display the correctness of tool numbers in the project:
8. red - tool number is incorrect, since there are several different tools in the project, but which have the same tool number, magazine number and corrector number;
9. gray - the tool number is correct, i.e. this set of tool number, magazine number and corrector number is used only by copies of the same tool.
10. The dashed underscore identifier of the tool in the column is used to indicate differences in machining conditions (feeds and cutting speeds):

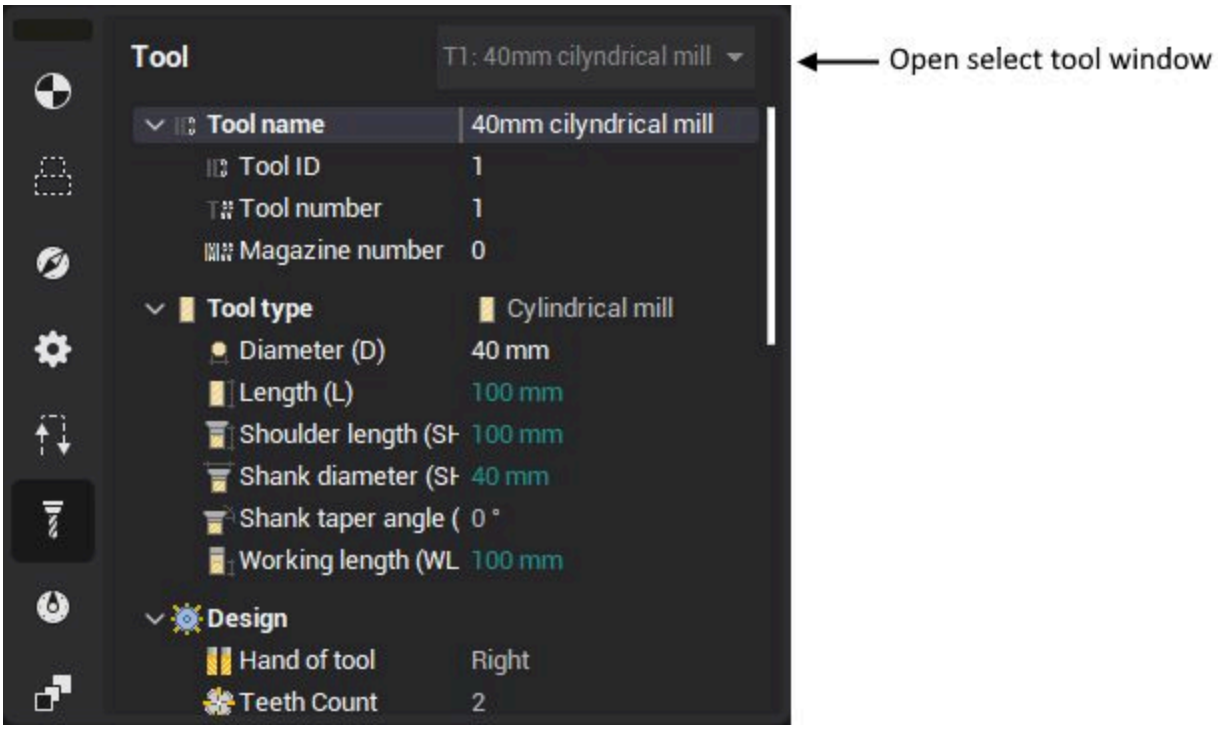
11. without underscore - the machining conditions in this operation are no different from the conditions specified in the tool in the global list of tools;
12. with underscore - the machining conditions in this operation differ from the conditions specified in the same tool in the global list.

Thus, a gray tool with a normal font is completely "correct" (not different from any copy), and the most black and bold tool is most likely the most "wrong" (different from all other copies).

Consider this on the example of the state of the tools shown above.

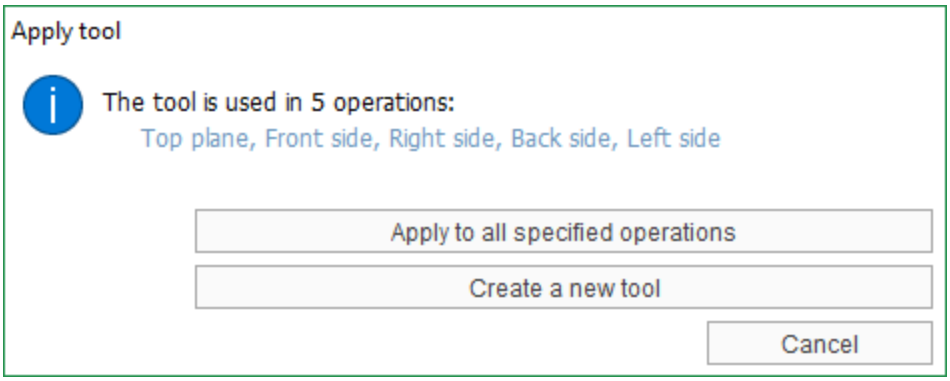
- Top plane operation tool - T#2: 81mm Cylindrical mill. Drawn in black, because its diameter (D = 81) differs from diameter of tool 2 in the list (D = 80). It is also drawn in bold, because it differs from the T#2 tools in the Front side, Right side, Back side and Left side operations, in which its diameter is 80.
- Front side, Right side, Back side and Left side operations tool - T#2: 80mm Cylindrical mill. Drawn in gray, because it does not differ from the tool in the list, but is drawn in bold because it differs from the T#2 tool in the first "Top plane" operation.
- Front holes operation tool - T#3: 20mm Drill. Drawn in a gray thin font, because it is no different from its other copies. However, the T#3 tool identifier is shown in red, since the project also has another "6mm Spherical mill" tool from the Corners cleanup operation, which has the same number 3.
- Top holes operation tool T#4: 20mm Cylindrical mill. Drawn in a gray thin font, because its parameters do not differ from other copies.
- 5D Contouring operation tool T#4: 20мм Цилиндрическая фреза. Drawn in a gray thin font, because its geometrical parameters do not differ from other copies, but its T#4 identifier is drawn with a dotted underline because the tool feed in operation differs from the feed recorded in the tool list.
- Corners cleanup operation tool - T#3: 6mm spherical mill. Drawn in thin gray font because It does not differ from the other copies. The tool identifier is red because there is another tool "T#3 20 mm Drill", which has the same tool number equal to 3.

After finding the differences between the copies of the tool, the task of synchronizing these copies arises. This task is easily solved with the help of special buttons that (if there are differences) appear in the title bar of the operation properties inspector on the Tool and Feeds/Speeds tabs. Moreover, these buttons work on each of the tabs independently. The buttons on the Tool tab synchronize only the parameters of the tool itself and do not change the machining conditions. The buttons on the Feeds/Speeds tab synchronize only the machining conditions and do not change the parameters of the tool itself.



The Reset changes button allows you to copy parameters from the tool located in the global tool list to the tool inside the operation.

The Apply changes button allows you to copy parameters from the tool of operation to the tool in global list. If there are several operations using this tool and the tool parameters in these operations are different, then an additional dialog box will be displayed.



The Apply to all specified operations button will copy the tool parameters from the current operation to the tool in the list, as well as into all operations that use the same tool.

The Create a new tool button will assign a new ID to the tool of the current operation (it will be generated automatically) and will create a copy of this tool in the global tool list. Thus, the connection with copies of the tool in other operations is broken.

Double clicking on the tool column in the operations tree, as well as clicking on a separate button with the tool name in the title bar of the properties inspector on the Tool and Feeds/Speeds tabs, opens the Tools window.

Creating shaped tools

Use shaped tool to perform form machining.

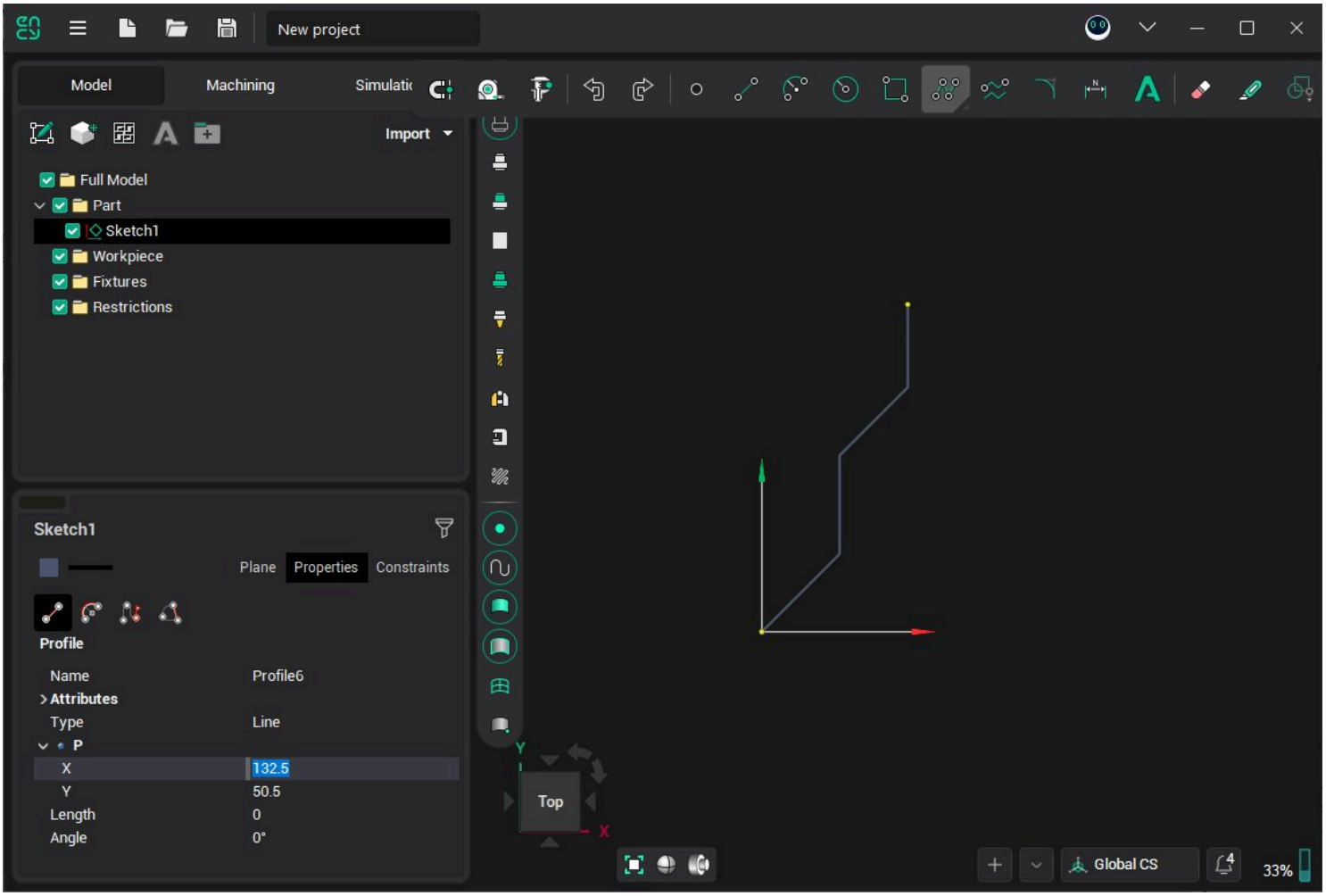
To support this type of machining CAM system implements shaped tool definition using contour (shape).

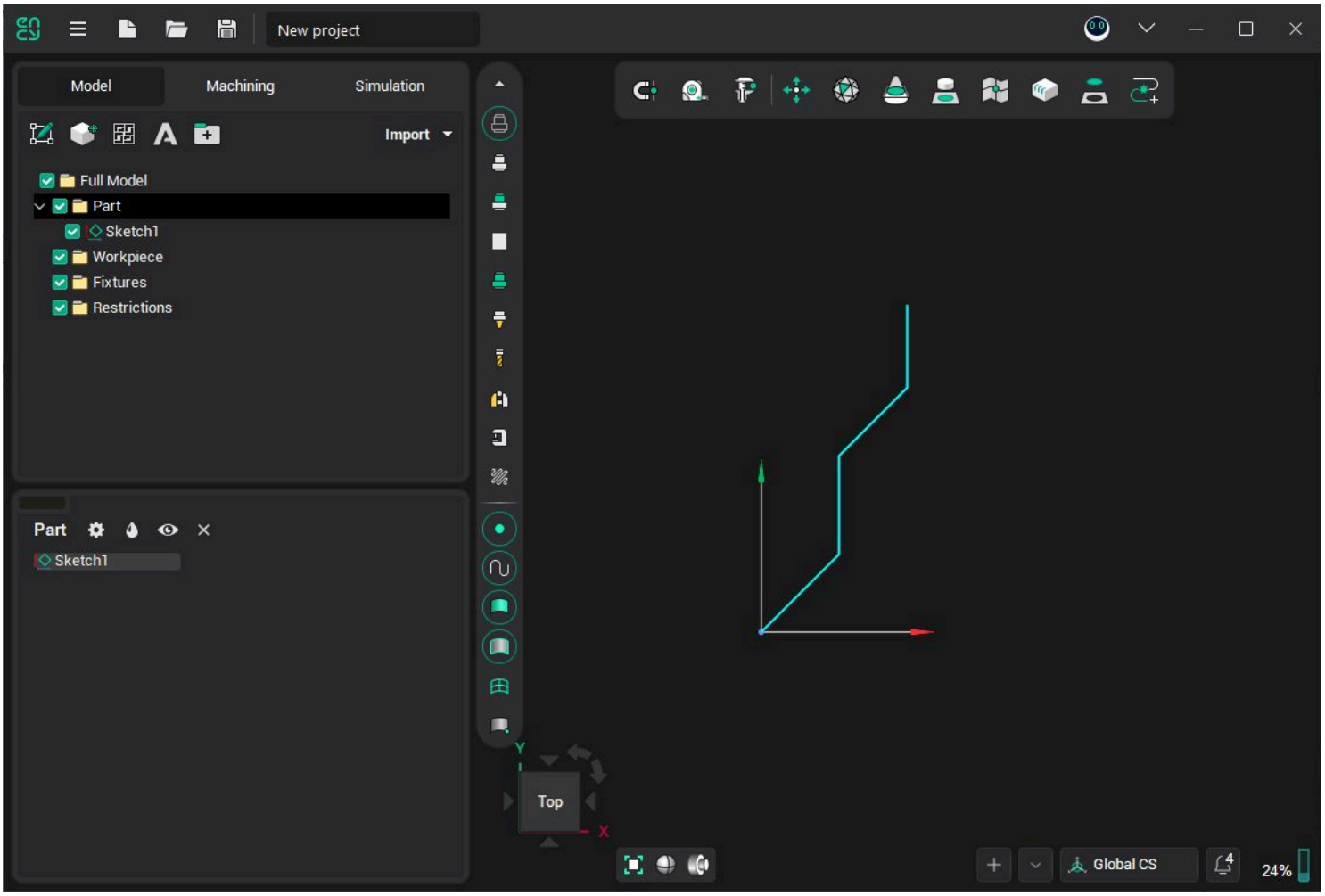
For milling tools the tool contour must be specified as generating line. It's accessible in the 2D contouring, 3D contouring and 5D contouring operations.

The curve can be get directly from the model or created using Sketch feature.

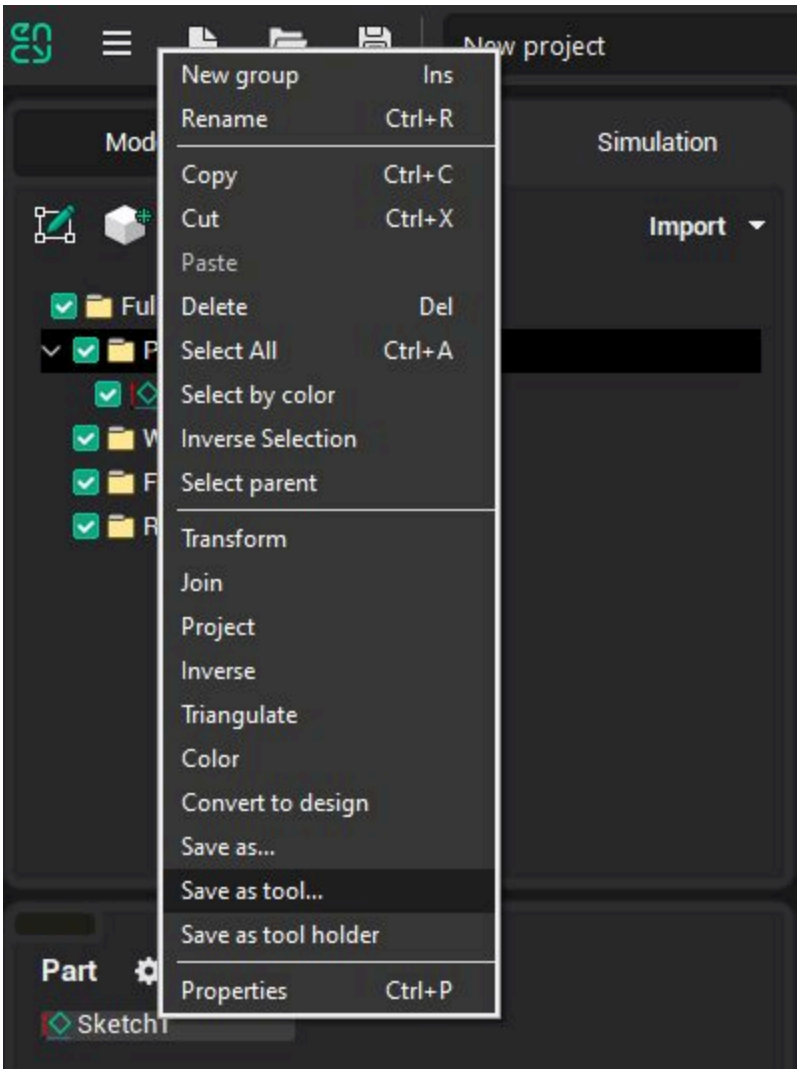
To create a shaped tool perform following steps:

Switch to Drawing mode  and draw the shape (profile) of the tool. The contour can be parametrized if necessary.



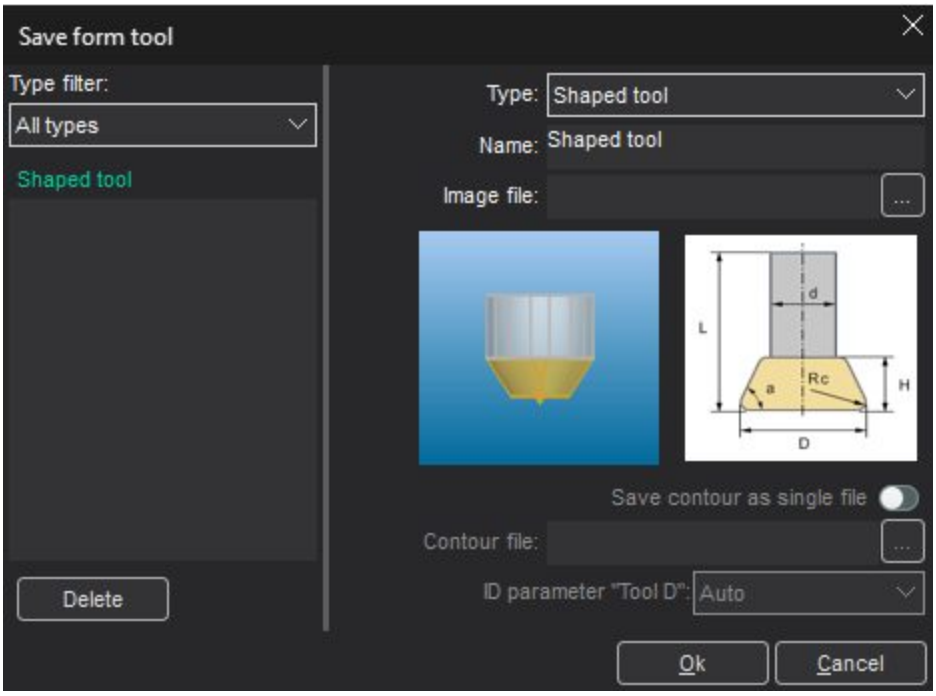


Select it in the list or in the view. Right-click on the item in the list to popup menu and click Save as tool:



All tool shape files are saved in the "C:\ProgramData\ENCY Software\ENCY\Version 1\Tools\MillTools" folder.

In the appearing dialog specify shape's type, it's name and tool scheme for which it was set (optional):

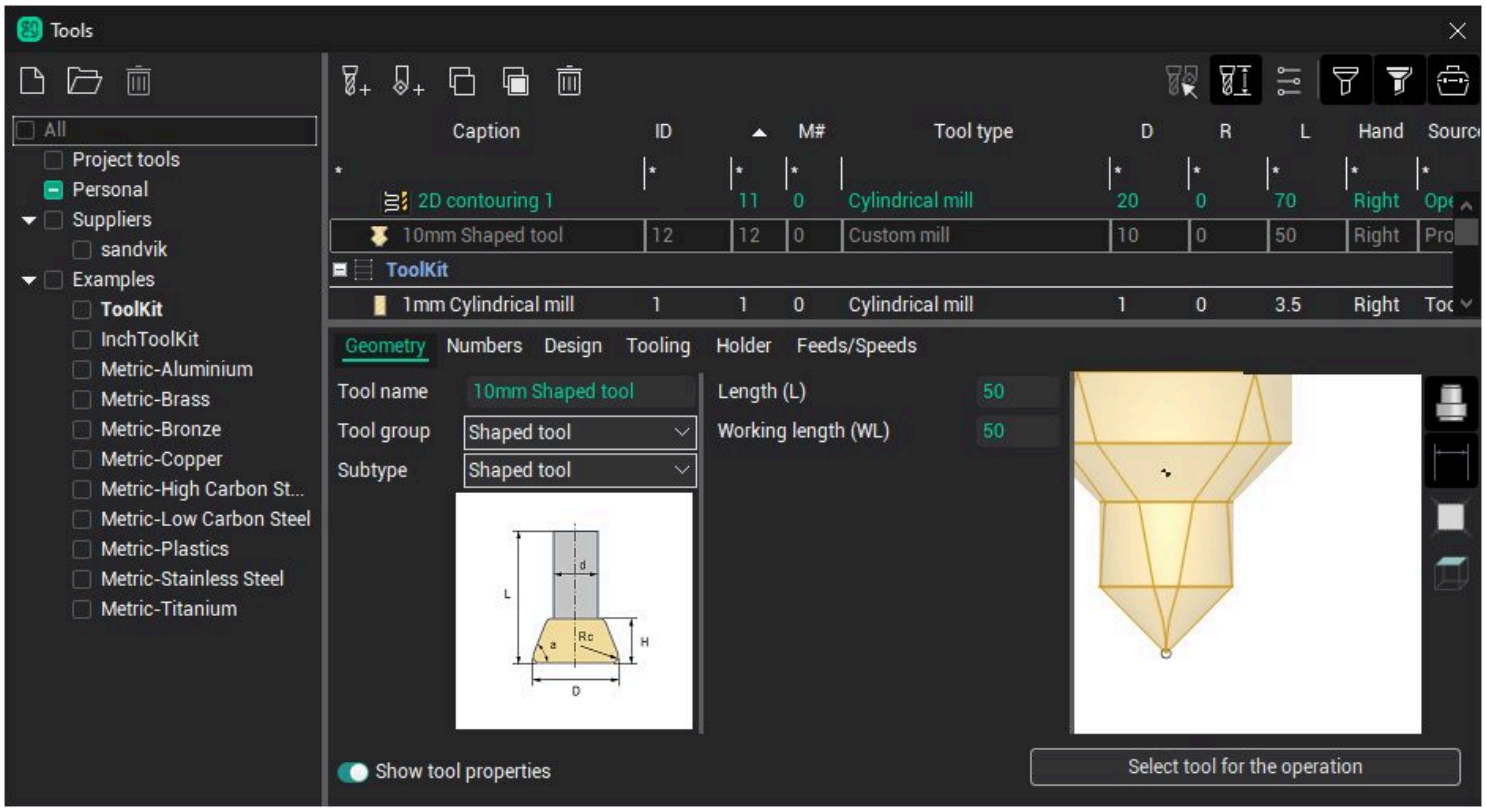


In the left part of the window there is shape type filter and the list of previously created shapes. If necessary you can rename or delete previous shapes.

Save contour as single file – check this option to save shape geometry separately from it's description.

In the next Geometrical parameters dialog specify contour file name. Later you can load the contour to edit it.

Finally, you can select newly created tool type in the Tools window:



It is important to remember that than tools library is copied contour files are not copied along. So to copy tools library to another computer you need to copy the contour files manually.

Holders (*.osd) window

The window for opening and adding 3d models (*.osd) of holders is implemented.

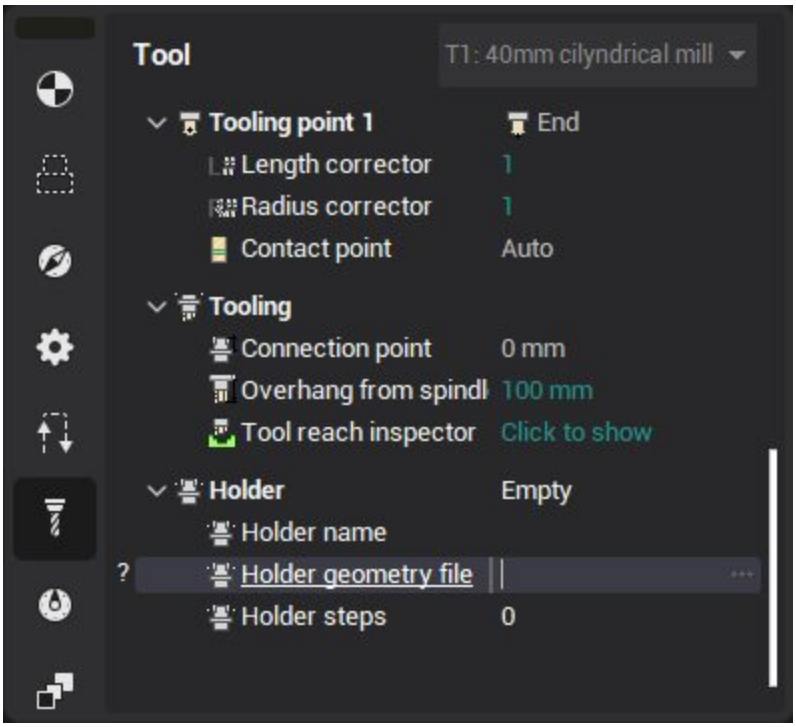
It is possible to add previously prepared holder models from the computer to this window.

Previously added holders in this window are saved to the configuration file and will be loaded automatically.

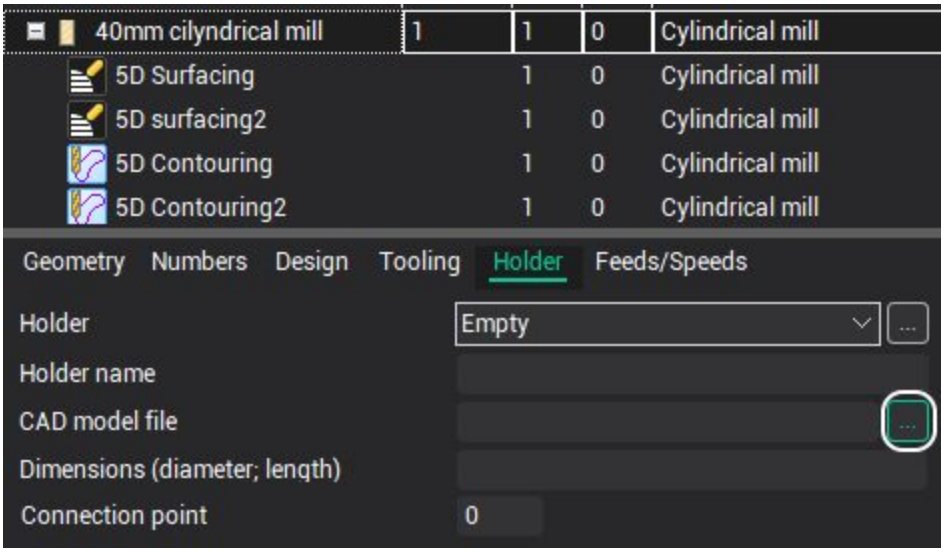


This window can be caused from the following locations:

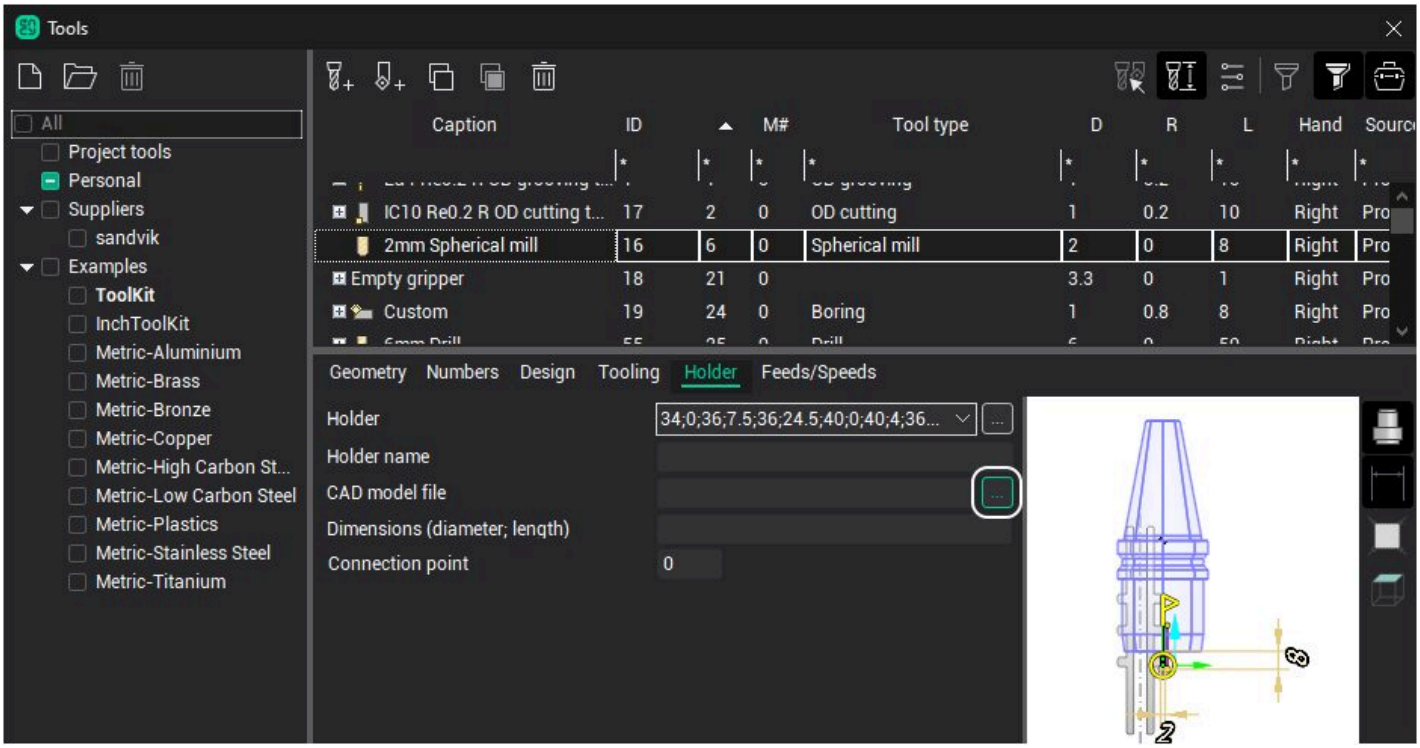
1. Inspector, Tool tab



2. Operation parameters, Holder tab



3. Holder selection window



Machining tools import API

There is an Application programming interface (API) using which you can write your own program to import machining tools into the CAM system from any external system you want (like TDM/PDM/PLM systems). It allows you to create simple program (like shown on the picture below) inside which you can for example:

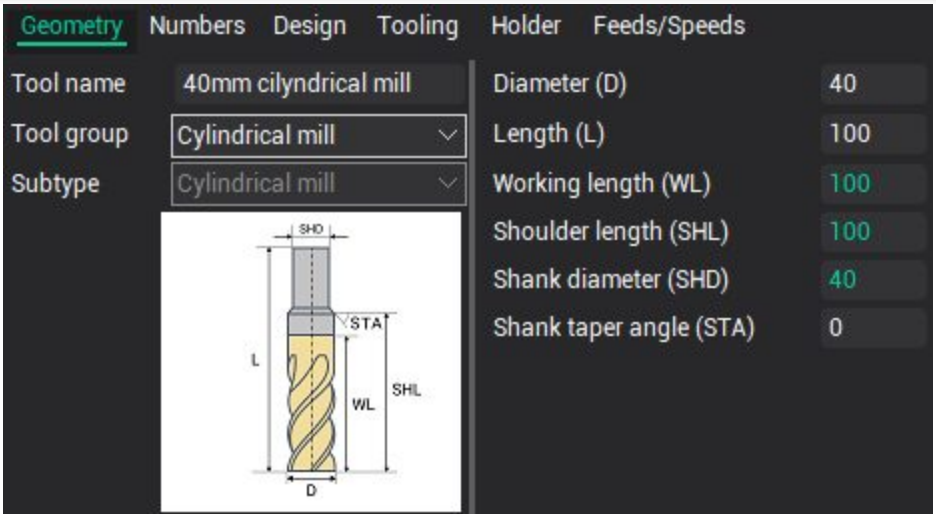
- create a new tool of desired type (turning, milling or custom type);
- specify your own geometric dimensions and parameters of this tool;
- add this tool to a storage (tools library) and save it to a file on a hard drive. Then you can use this library inside the CAM's tools window usual way.

For full information about using this API, visit this page.

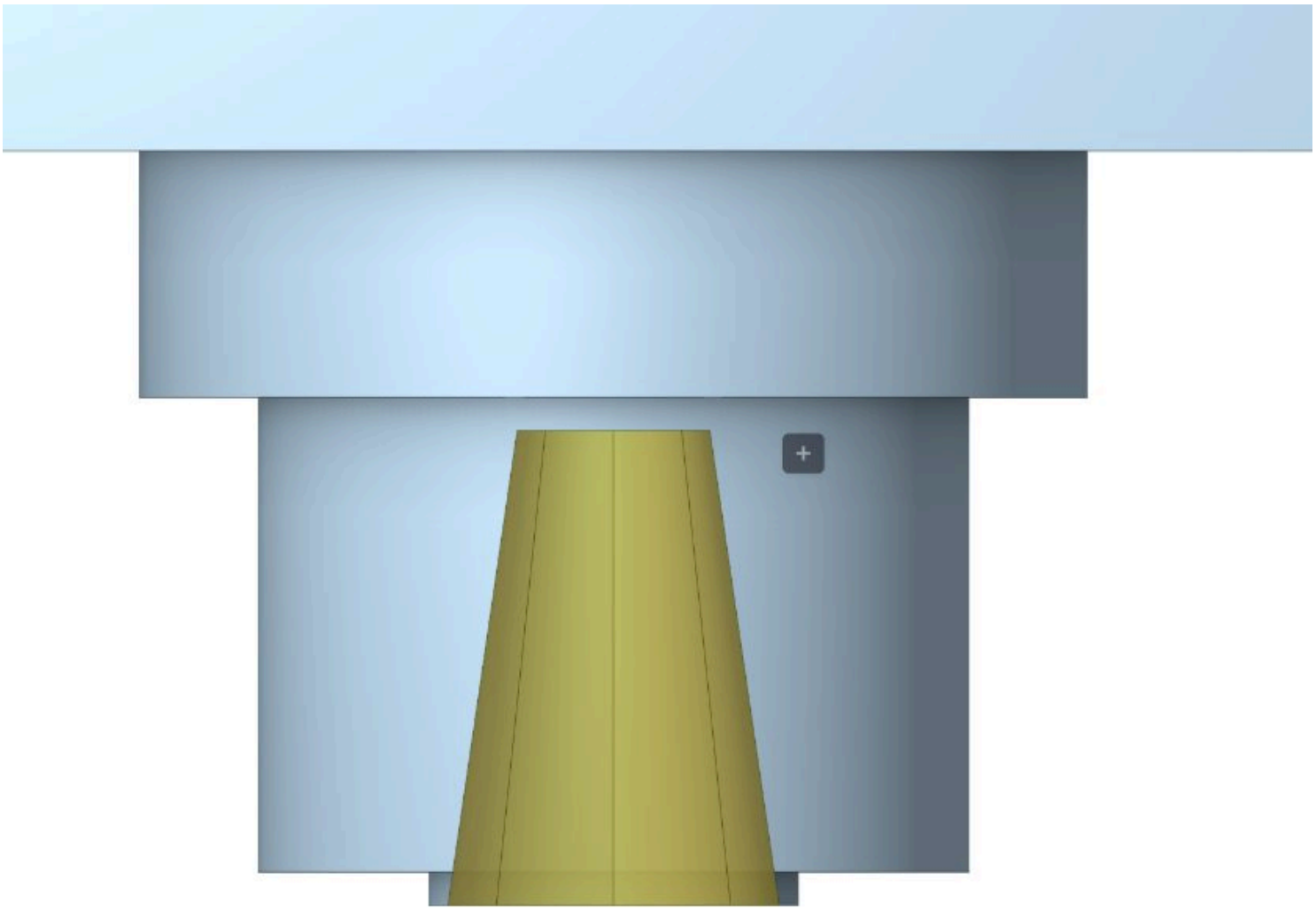
Code example

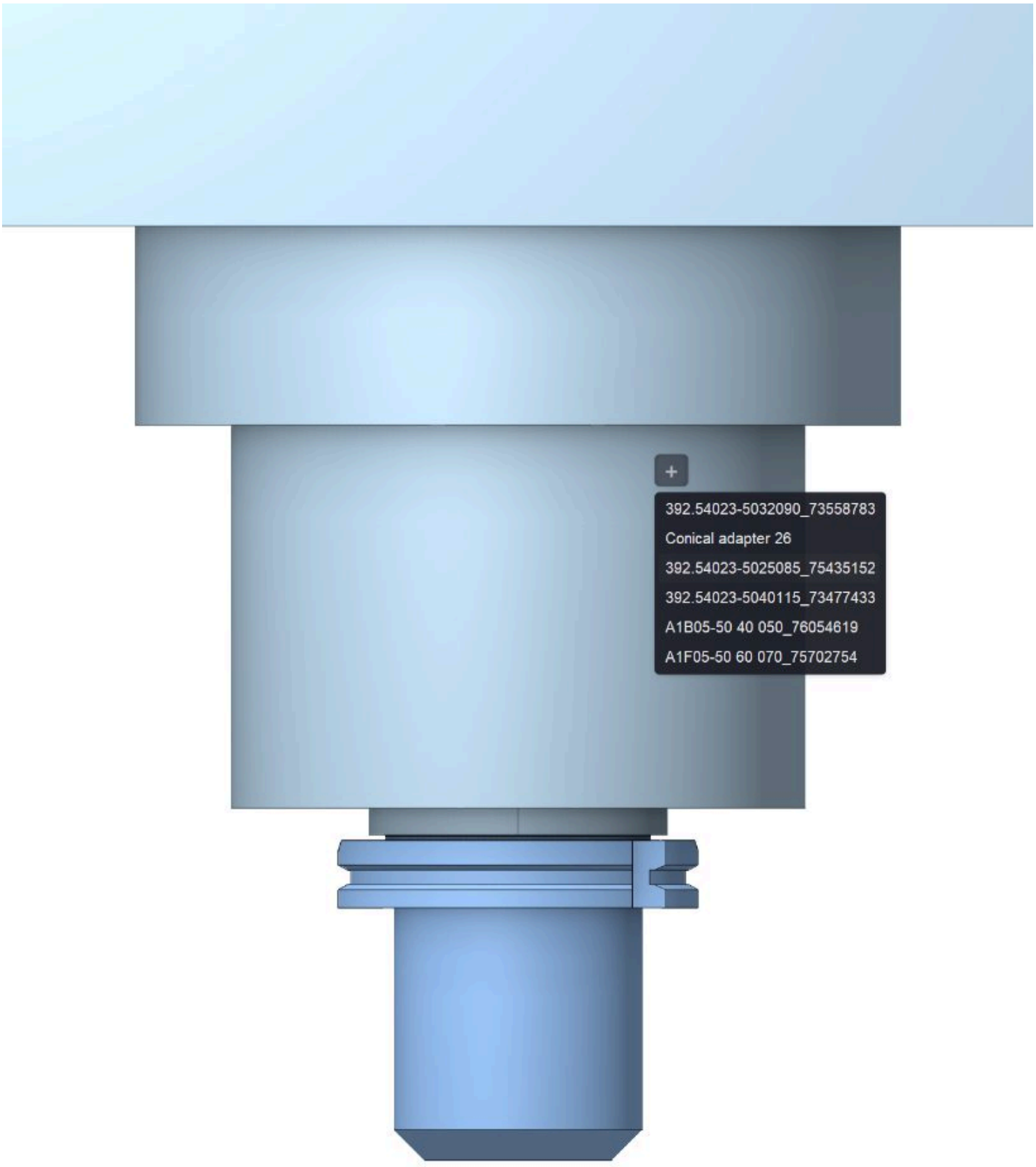
```
var importer = MTIMachiningToolsImportHelper.CreateImporter(assemblyPath);
var storage = importer.OpenExistingToolsStorage(toolStoragePath);
var tool = importer.CreateCylindricalMill();
tool.SetName("My Cylindrical Mill");
tool.CuttingDiameter = 10;
tool.OverallLength = 80;
tool.WorkingLength = 30;
tool.ShoulderLength = 50;
tool.ShankDiameter = 11;
tool.ShankTaperAngle = 60;

storage.AddToolItem(tool);
MTIMachiningToolsImportHelper.FinalizeImporter();
```



Tool assemblies




392.54023-5032090_73558783
Conical adapter 26
392.54023-5025085_75435152
392.54023-5040115_73477433
A1B05-50 40 050_76054619
A1F05-50 60 070_75702754


MH20-R025A25-08M_75501240
R216F-25A25S-045_75606597
DS20-D1800L25-04_77094836

Application area:



Tool assemblies are the CAM system mechanism for describing tooling as a set of connected components — a cutting tool mounted into one or more adapters that plug into a machine magazine. The mechanism is designed to automate tool selection, which is an essential stage of process planning and a prerequisite for the automatic generation of a machining process.

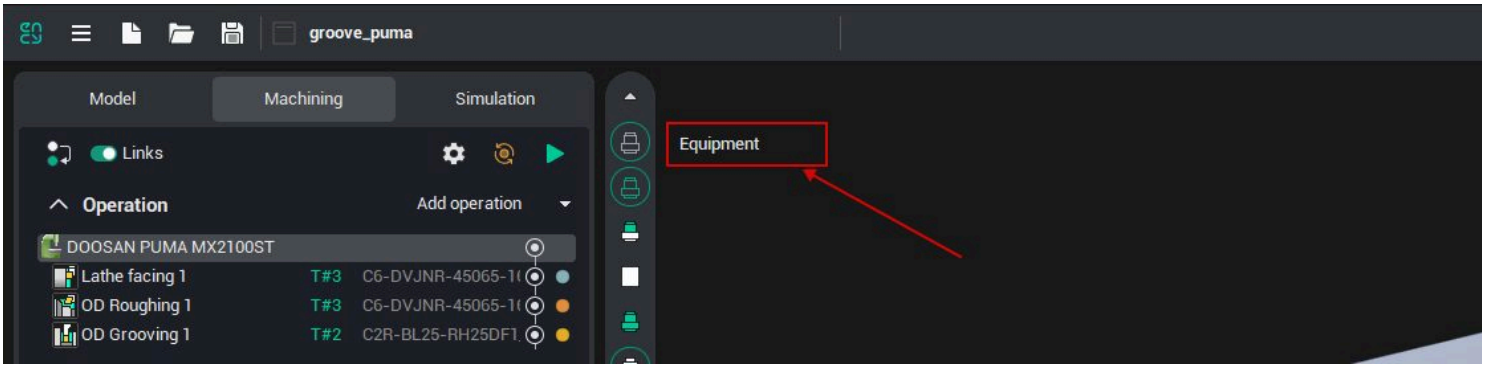
Automatic process generation is not possible without solving the automatic tool selection task. For that task to be solvable, two conditions must hold:

- The tool must be present in the user database.
- An adapter must exist that ensures correct mounting of the tool into the machine magazine.

Tool operation modes.

The CAM system supports two modes of working with tooling:

- Classic tools mode. The standard mode of working with tools.
- Advanced tools mode. The mode of working with tool assemblies. When the system is in advanced mode, the Equipment button is shown on the technology page.



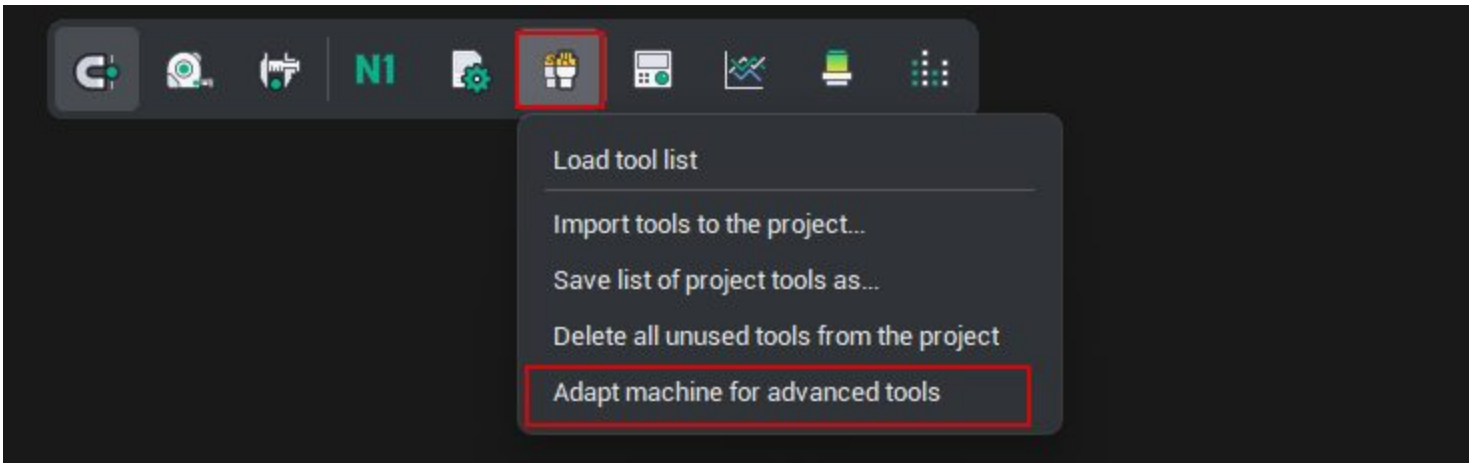
Adapting a machine for advanced tools mode.



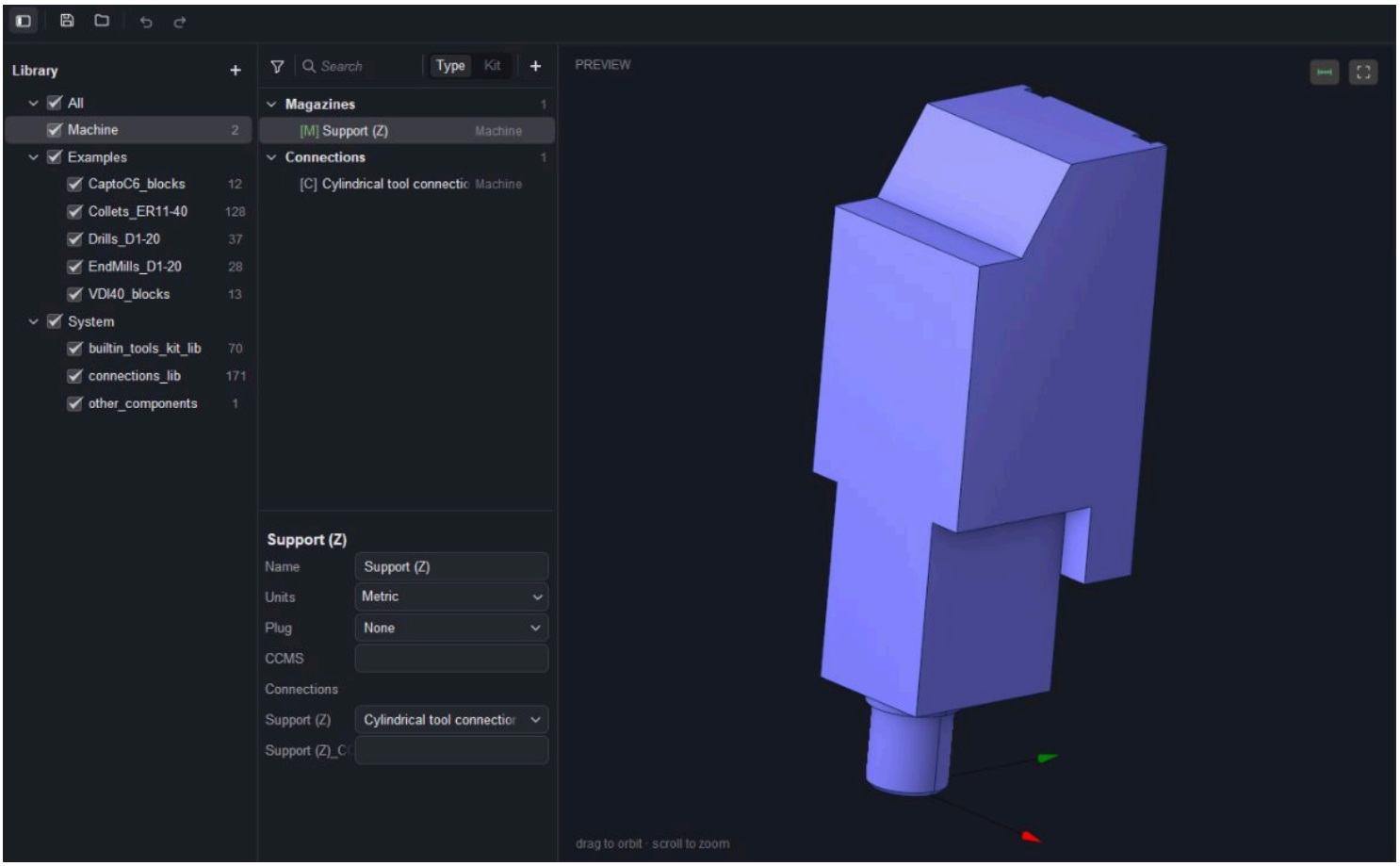
Application area:

The ability to switch to advanced mode is determined by the machine scheme in use. A machine scheme in the old (classic) format cannot work with tool assemblies directly — it must first be adapted to the new format.

Adaptation is performed by the Adapt machine for advanced tools command, available in the tools menu when the active machine scheme has not yet been adapted. During adaptation, a machine library is formed (machine kit). The machine kit file always resides next to the machine scheme file and contains a special component type — the magazine (a component that has sockets of a specified connection type and no plug).

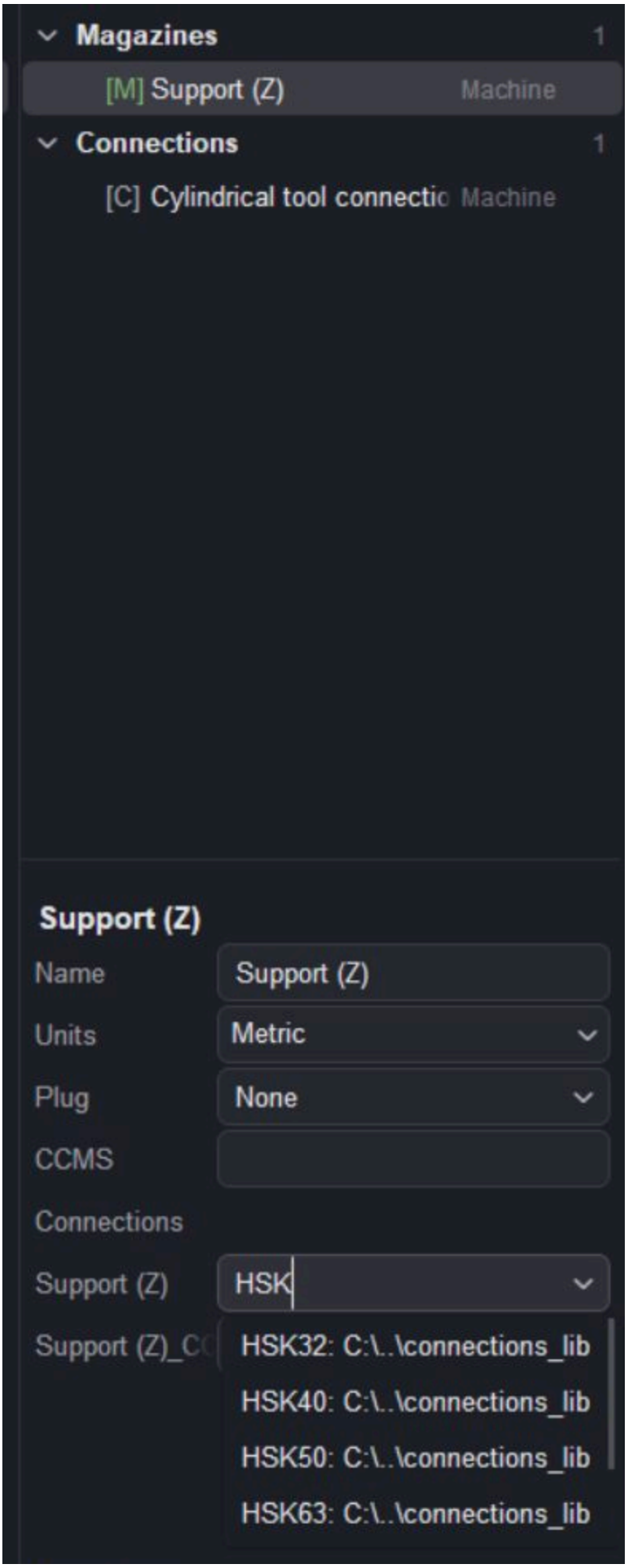


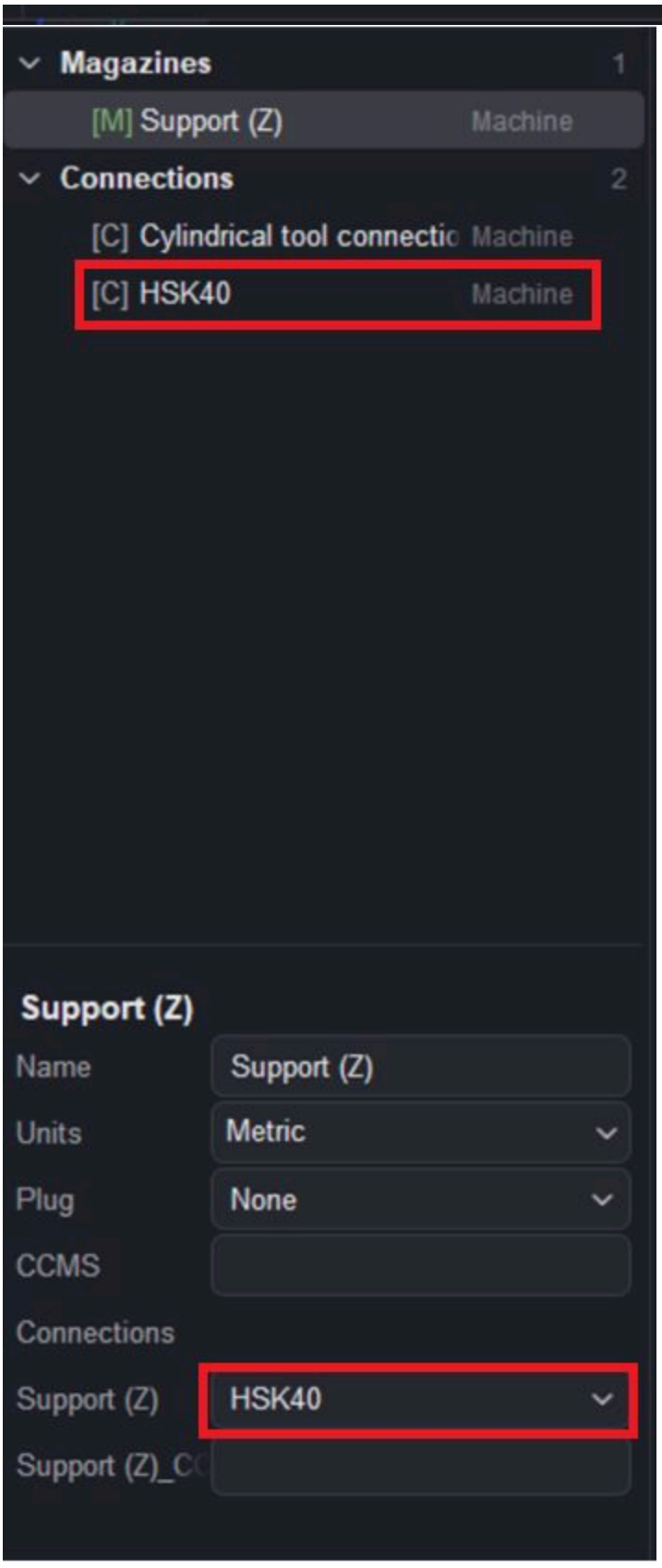
After the machine has been adapted, the Component libraries window opens. It already contains the machine nodes (magazine components) that hold the mounting point for the tool assembly. Initially, a machine node carries a standard connection — in this case a Cylindrical tool connection, shown under Connections and assigned as the node's Support (Z) connection.

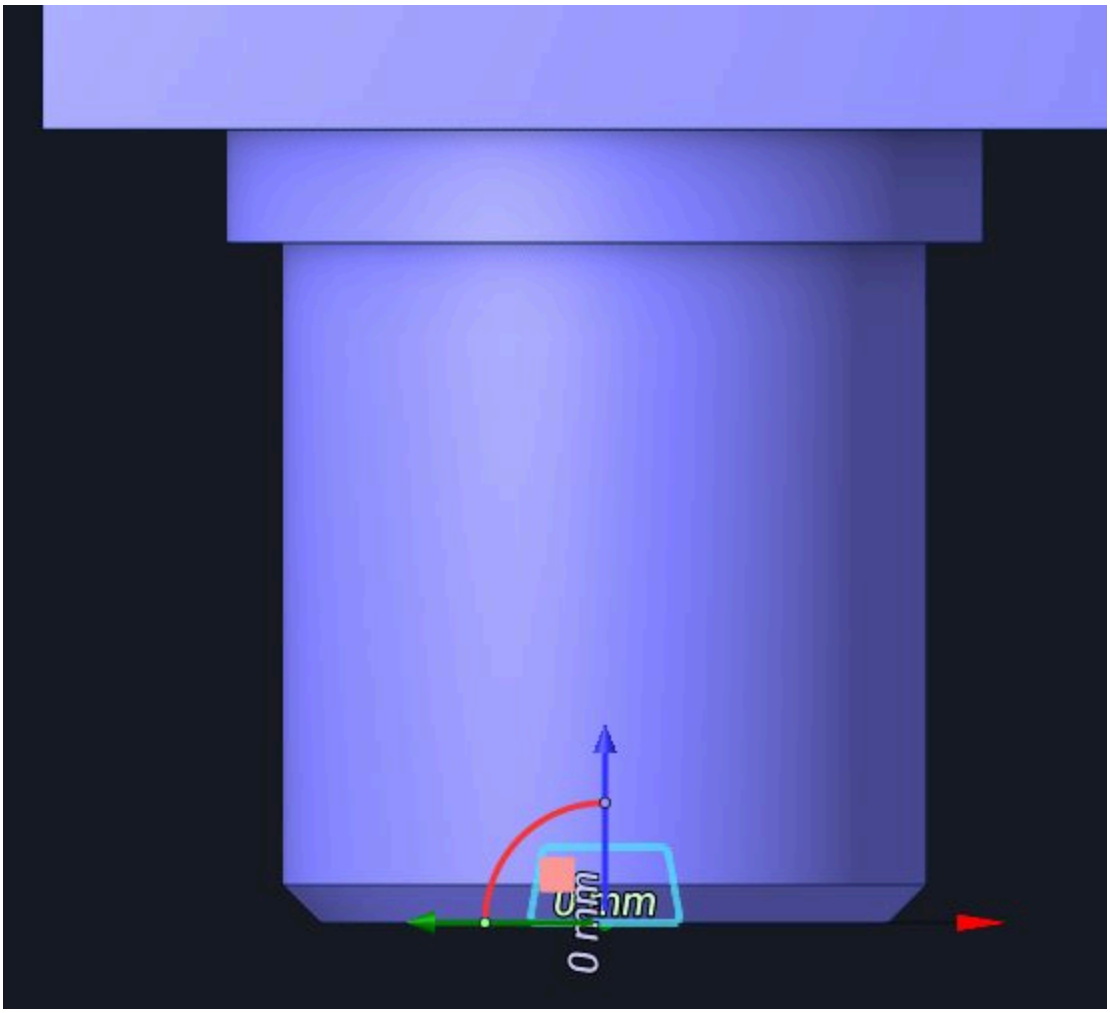


Assigning a connection. A connection does not have to be invoked from the menu via the + button. Its name can be typed directly into the connection field. The value can either be selected from the suggested options or defined as a new one. After a new name is entered in the field, a new connection type is created.

If a connection is selected from the library, all its dimensions are taken into account in accordance with the standard


Magazines 1
[M] Support (Z) Machine
Connections 1
[C] Cylindrical tool connectic Machine
Support (Z)
Name Support (Z)
Units Metric
Plug None
CCMS
Connections
Support (Z) HSK
Support (Z)_C
HSK32: C:\..\connections_lib
HSK40: C:\..\connections_lib
HSK50: C:\..\connections_lib
HSK63: C:\..\connections_lib

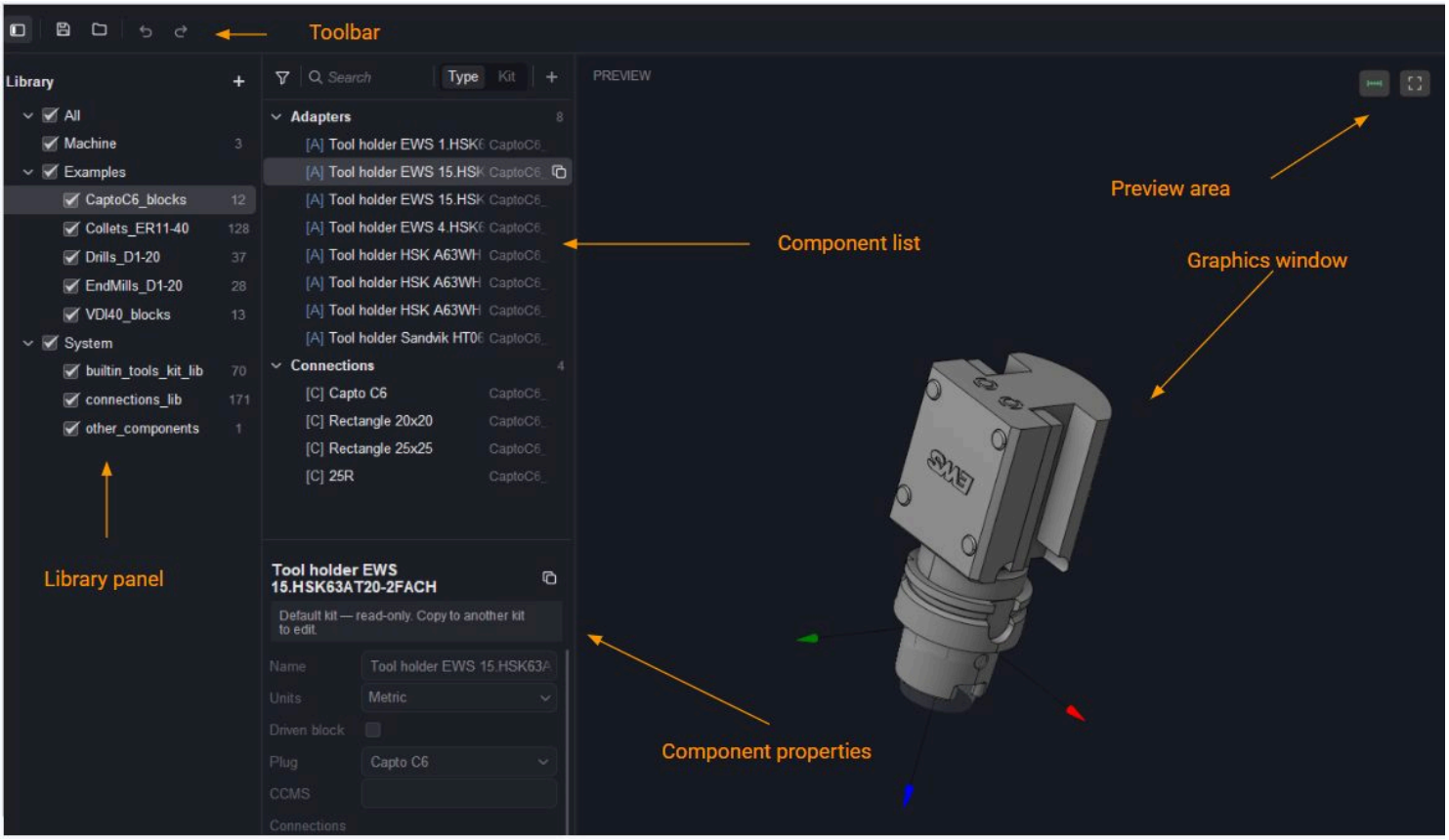

Magazines 1
[M] Support (Z) Machine
Connections 2
[C] Cylindrical tool connectio Machine
[C] HSK40 Machine
Support (Z)
Name Support (Z)
Units Metric
Plug None
CCMS
Connections
Support (Z) HSK40
Support (Z)_C



> Note
>
> When the window is closed, an *.mKIT file — the setup file — is created next to the machine. This file contains all tools bound to that machine.

Returning to classic mode is performed from the same menu with the Activate classic tools mode command.

Component libraries

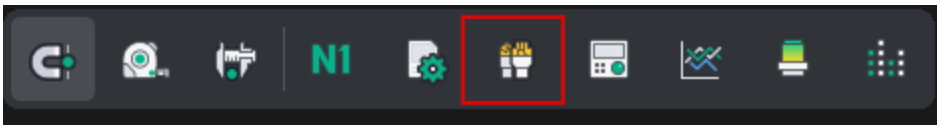


Application area:

The Component libraries window is the editor for component libraries — the storage where the building blocks of tool assemblies are kept: cutting tools, adapters, machine magazines, and the connections that join them. Each component has a visual representation, is defined by its dimensions, and carries a shank (Plug) and, for adapters and magazines, one or more seats (Sockets) of a specific connection type.

The window is used to prepare and maintain the tooling data that makes automatic tool selection and tool-assembly formation possible.

The component library window is opened by the corresponding toolbar button (in advanced mode).



Component libraries consist of the following components:

Toolbar (top-left).

- Toggle panel. Shows or hides the left Library panel.
- Save. Saves changes to the library file on disk.

- Open. Opens a library file from disk.
- Undo. Cancels the last change.
- Redo. Repeats the previously cancelled change.

Library panel.

- Add (+). Adds a new (user) library to the panel.
- Category / library checkboxes. Toggle the visibility of a category (All, Machine, Examples, System) or an individual library in the component list.

A library is stored on disk as a file. Component libraries fall into the following categories:

- All. The top-level node that shows all libraries at once — every category and its libraries are displayed together.
- Machine. Formed during adaptation of a machine scheme for work in advanced tools mode. A machine library file always resides next to the machine scheme file. Such a library contains a special component type called a magazine. A machine magazine is a component that has sockets with a specified connection type and has no plug. When a machine scheme is activated, the library located next to it is activated automatically.
- Examples. Reference libraries supplied with the software.
- System. Libraries supplied and maintained by the CAM system.

Component list (central panel).

- Filter. Opens filtering options for the component list (funnel icon).
- Search. Filters the list by the entered component name.
- Type. Switches the list grouping to view by component type (Magazines, Connections, Tools, ⋯).
- Kit. Switches the list grouping to view by kit (source library).
- Add (+). Creates a new component in the current library.

Component properties (lower panel): *(fields, not buttons)* Name, Units, Plug, CCMS, and the Connections list (Support (Z), ⋯).

Preview area.

Show dimensions. Toggles the display of component dimensions in the 3D preview.

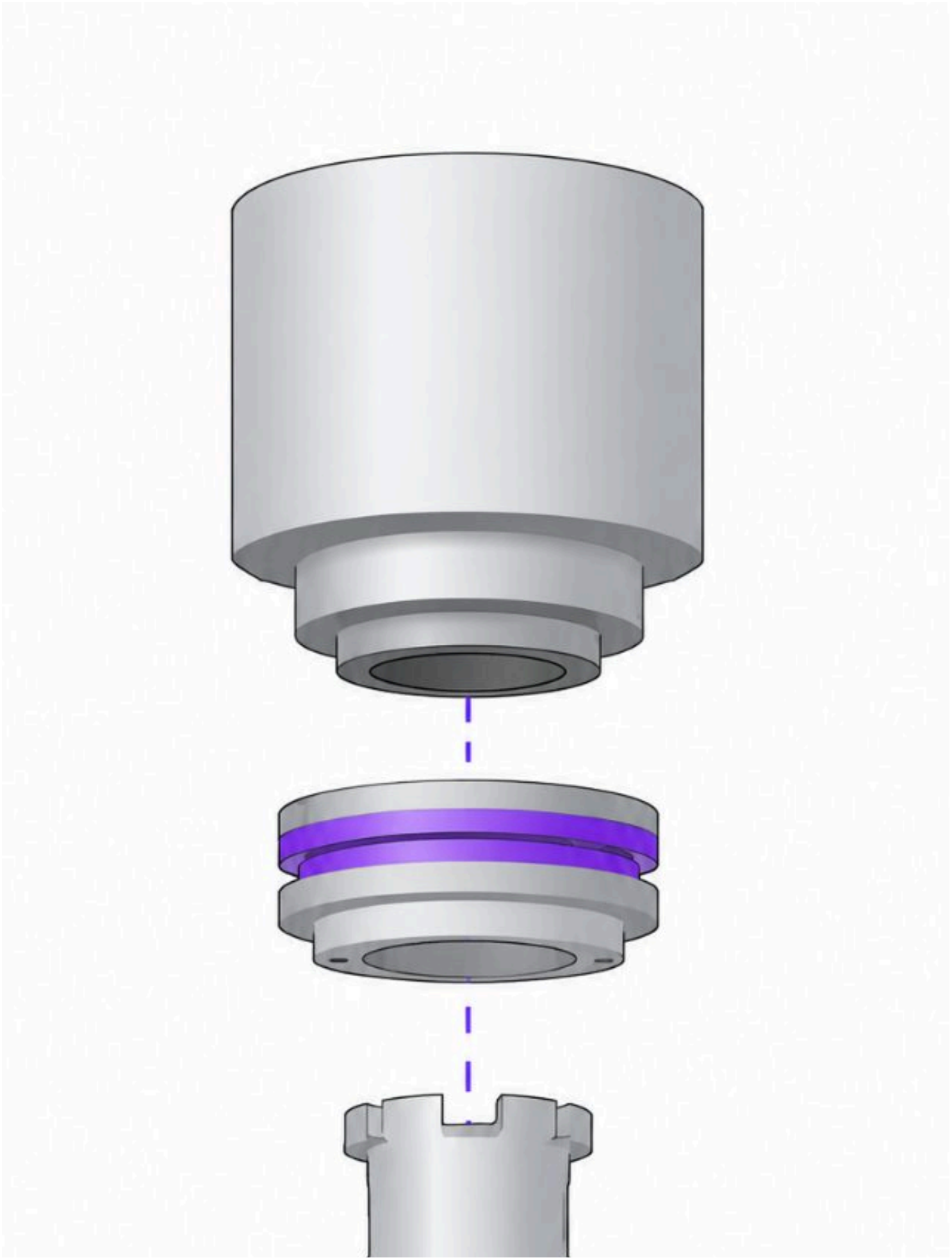
Zoom to fit. The model is rotated by dragging and zoomed with the scroll wheel (*drag to orbit · scroll to zoom*).

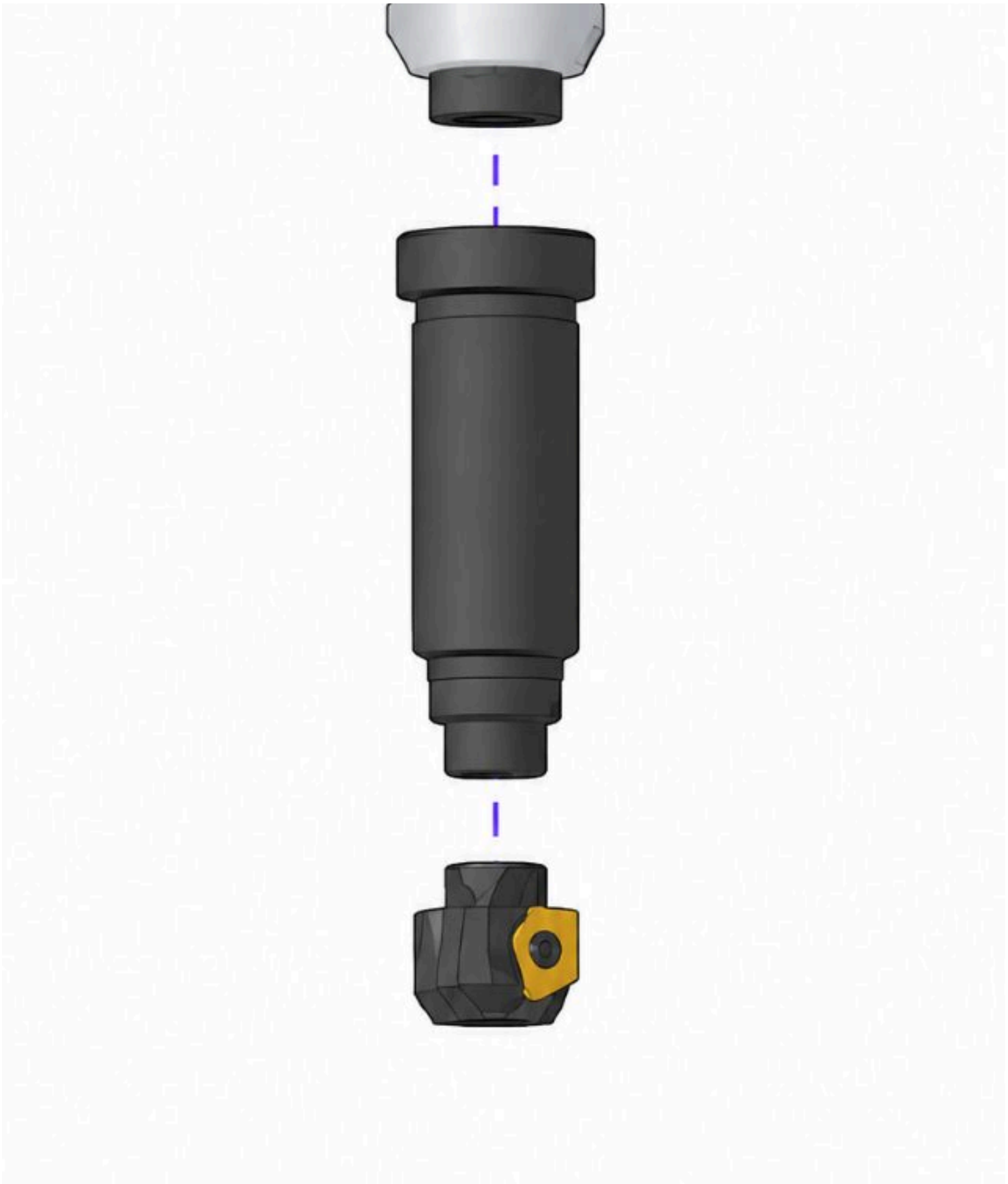
Forming libraries.

Filling a component library covers the following activities.

- Creating built-in parameterized components. The Plug and Sockets must be filled in, preferably with standard connections.
- Creating objects from 3D models. The fields used by filters must be filled in.
- Working with filters. Filter fields drive the search for suitable components.

Building a tool assembly





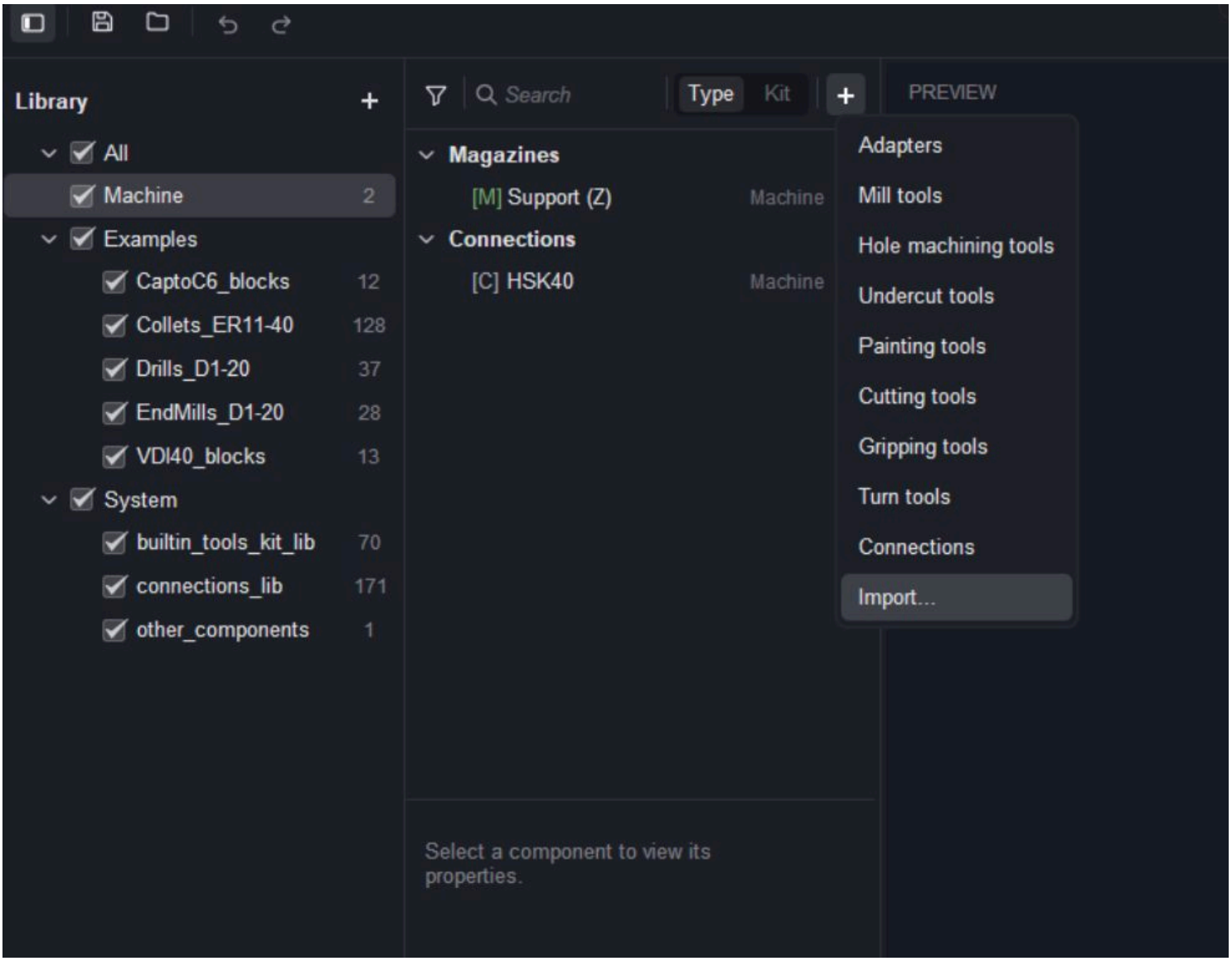
Application area:

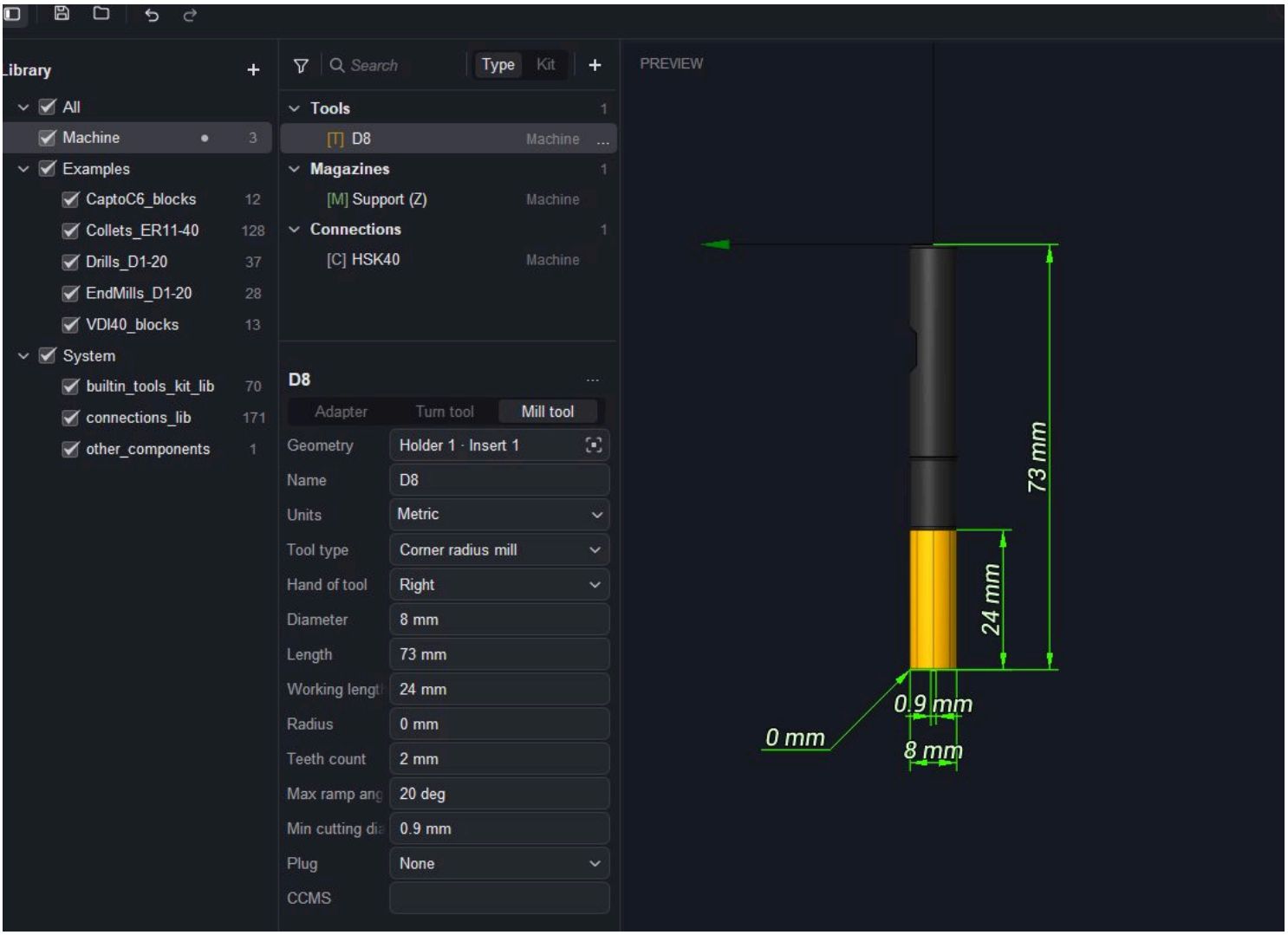
This example shows how a simple assembly is built in the Component libraries window — a milling tool joined to an adapter that plugs into the machine magazine. The assembly is created bottom-up, from the

cutting tool to the machine mounting.

Add the component and check recognition.

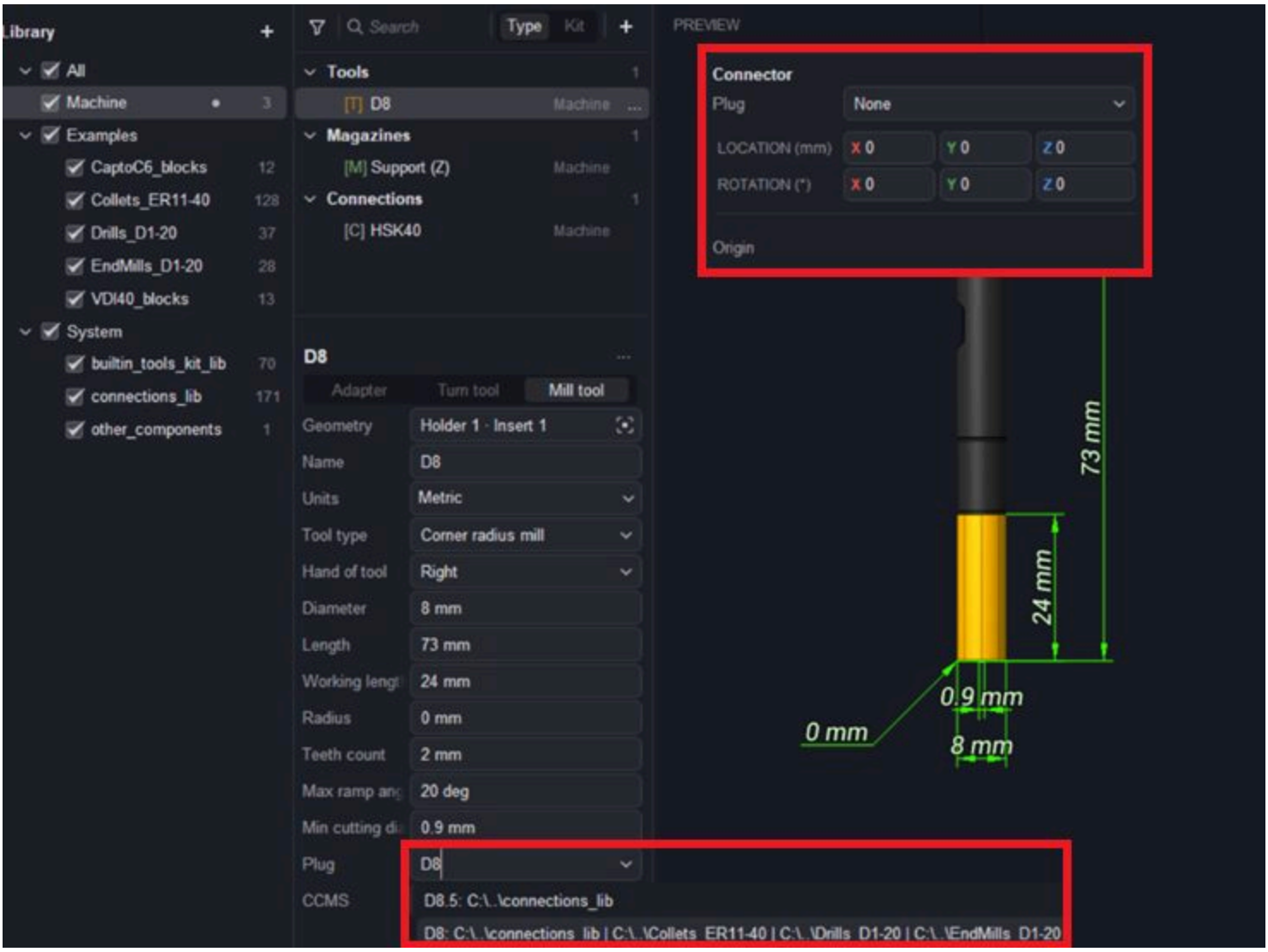
A component is added from a 3D model. The system automatically tries to recognize what has been loaded — whether it is a tool or an adapter (holder), and, for a tool, its type. The recognition result can be corrected by the user with the Adapter / Turn tool / Mill tool tabs, and the tool type is refined in the Tool type field. The component and its dimensions are shown in the Preview.


Library
All
Machine 2
Examples
CaptoC6_blocks 12
Collets_ER11-40 128
Drills_D1-20 37
EndMills_D1-20 28
VDI40_blocks 13
System
builtin_tools_kit_lib 70
connections_lib 171
other_components 1
Search
Type
Kit
PREVIEW
Magazines
[M] Support (Z) Machine
Connections
[C] HSK40 Machine
Adapters
Mill tools
Hole machining tools
Undercut tools
Painting tools
Cutting tools
Gripping tools
Turn tools
Connections
Import...
Select a component to view its properties.



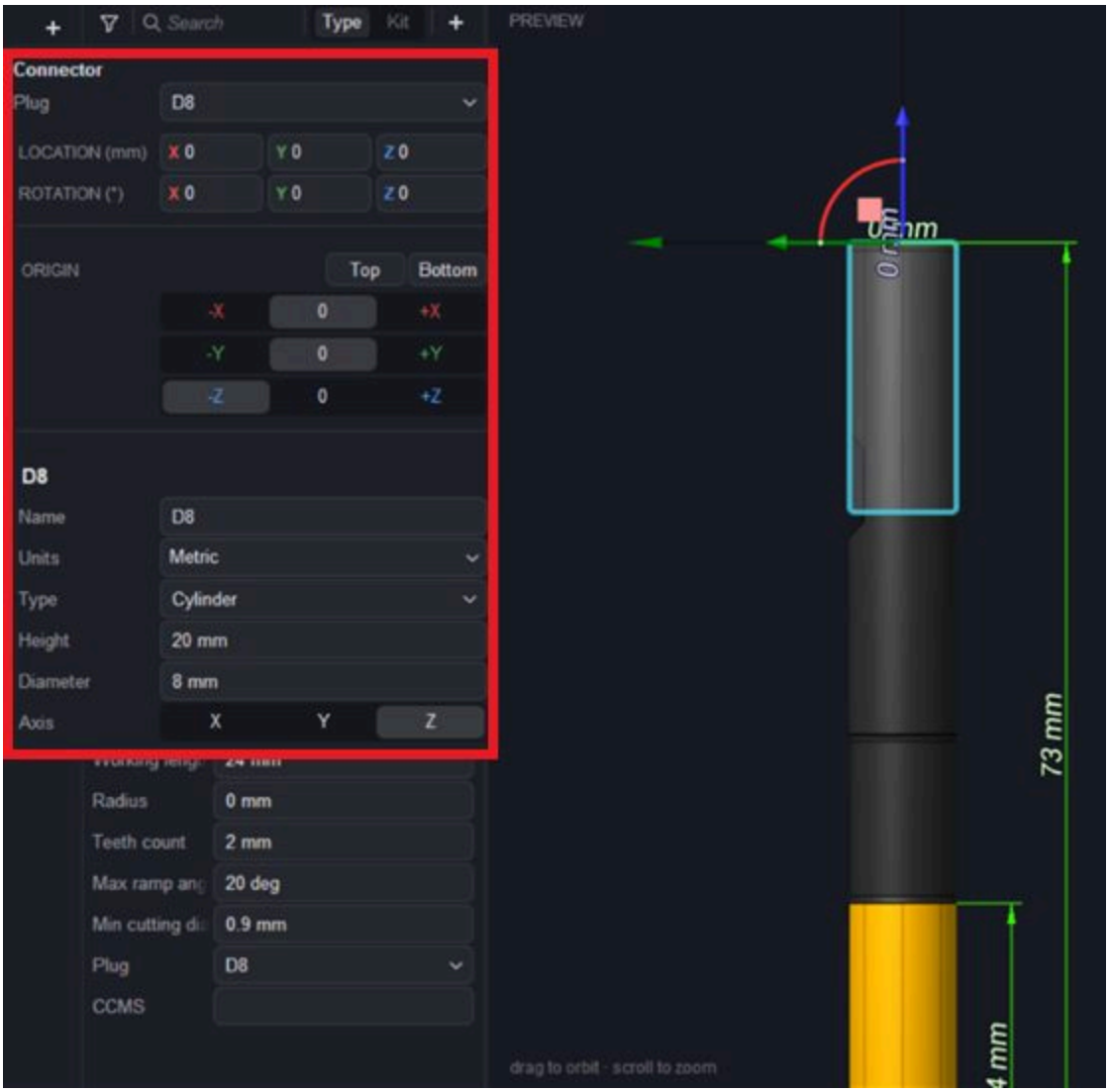
Assign the tool plug

The tool needs a shank so it can be inserted into an adapter. The connection name is entered in the Plug field — either selected from the suggested options or defined as a new one (here, D8). If the connection is selected from the library, all its dimensions are taken according to the standard.



Check and position the plug connection.

With the plug assigned, the Connector panel becomes available. It shows the connection Plug, its LOCATION and ROTATION, and the Origin controls, together with the connection's own properties: Type (Cylinder), Height (20 mm), Diameter (8 mm), and Axis (Z). The plug is highlighted on the model in the Preview. Using Origin, the connector must be positioned correctly relative to the component. This is done with the +Z / −Z controls (as well as Top / Bottom), or simply by dragging the coordinate system in the Preview.

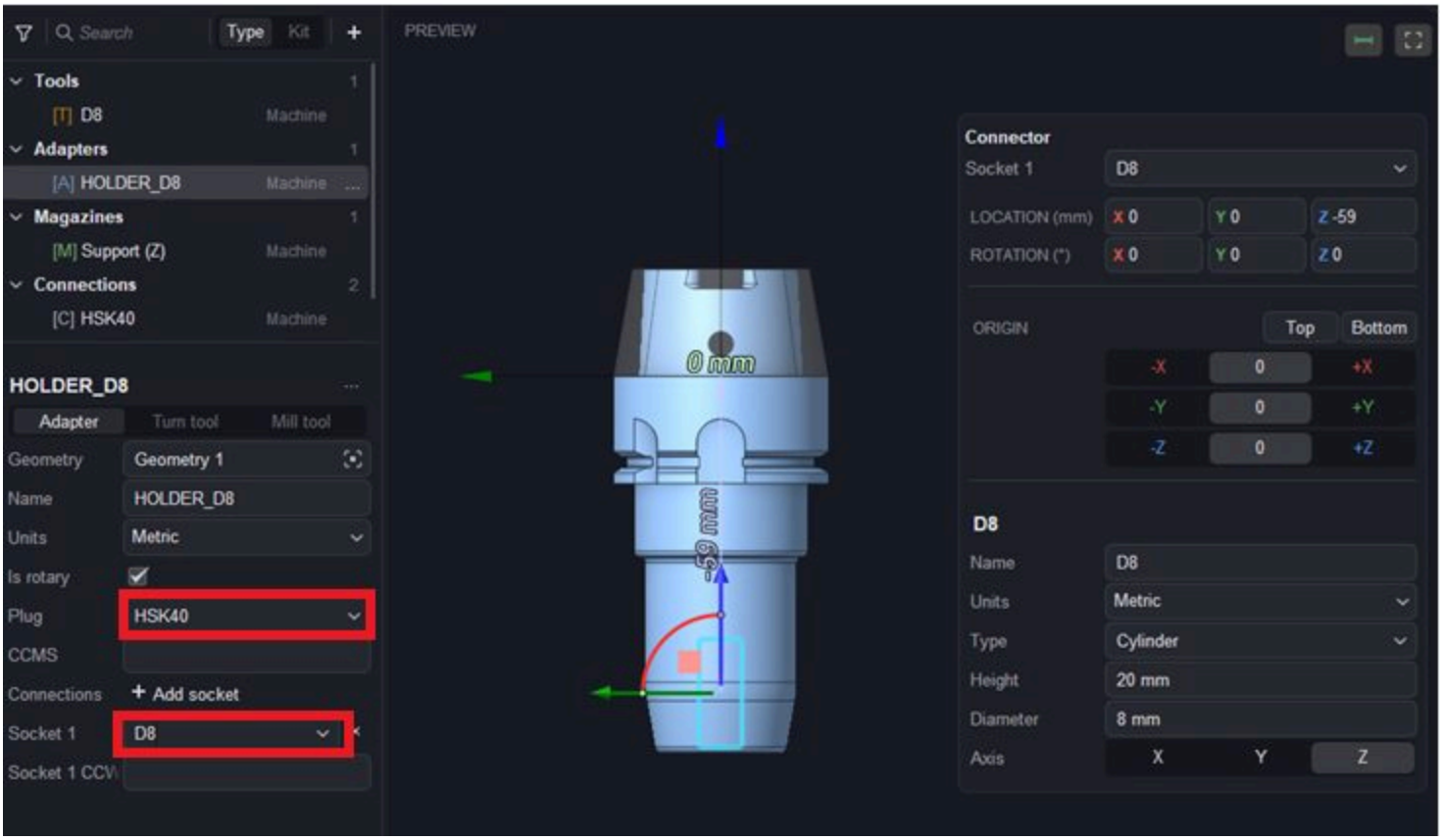


Create the adapter.

The Add (+) button is pressed again and Adapters is chosen. A new adapter is added to the Adapters group.

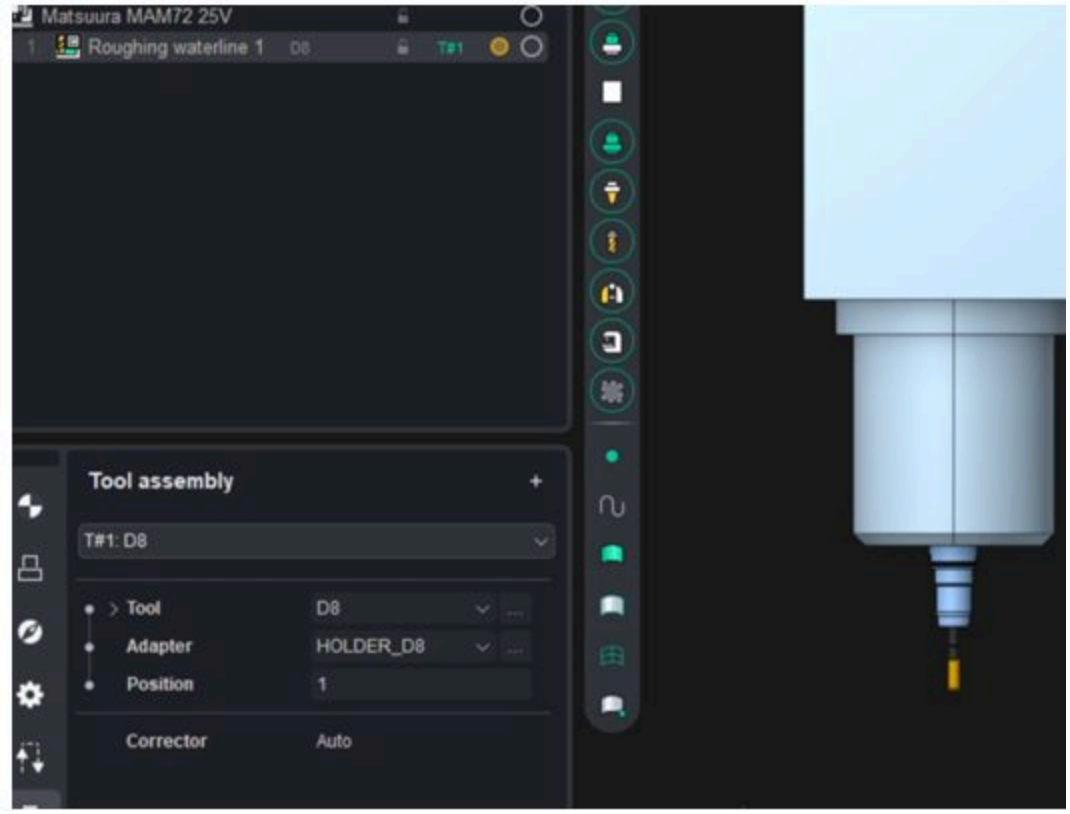
Set the adapter's plug and add a socket

The adapter needs two connections: a plug to mount it into the machine and a socket to receive the tool. In the adapter properties the Plug field is set to the connection that mounts the adapter into the machine magazine — here HSK40 — and in the Connections section the Socket 1 is set to the connection D8, the same type as the tool's plug. In this case both the Plug and the Socket are chosen from the list.



Automatic assembly in the operation.

When the components are available in the libraries, the system automatically attempts to build a tool assembly for the operation from the existing components. In the operation's Tool assembly panel the assembly is composed and shown: the Tool (D8), the Adapter (HOLDER_D8), the magazine Position, and the Corrector (Auto). The assembled tool is mounted on the machine and displayed in the graphics area.

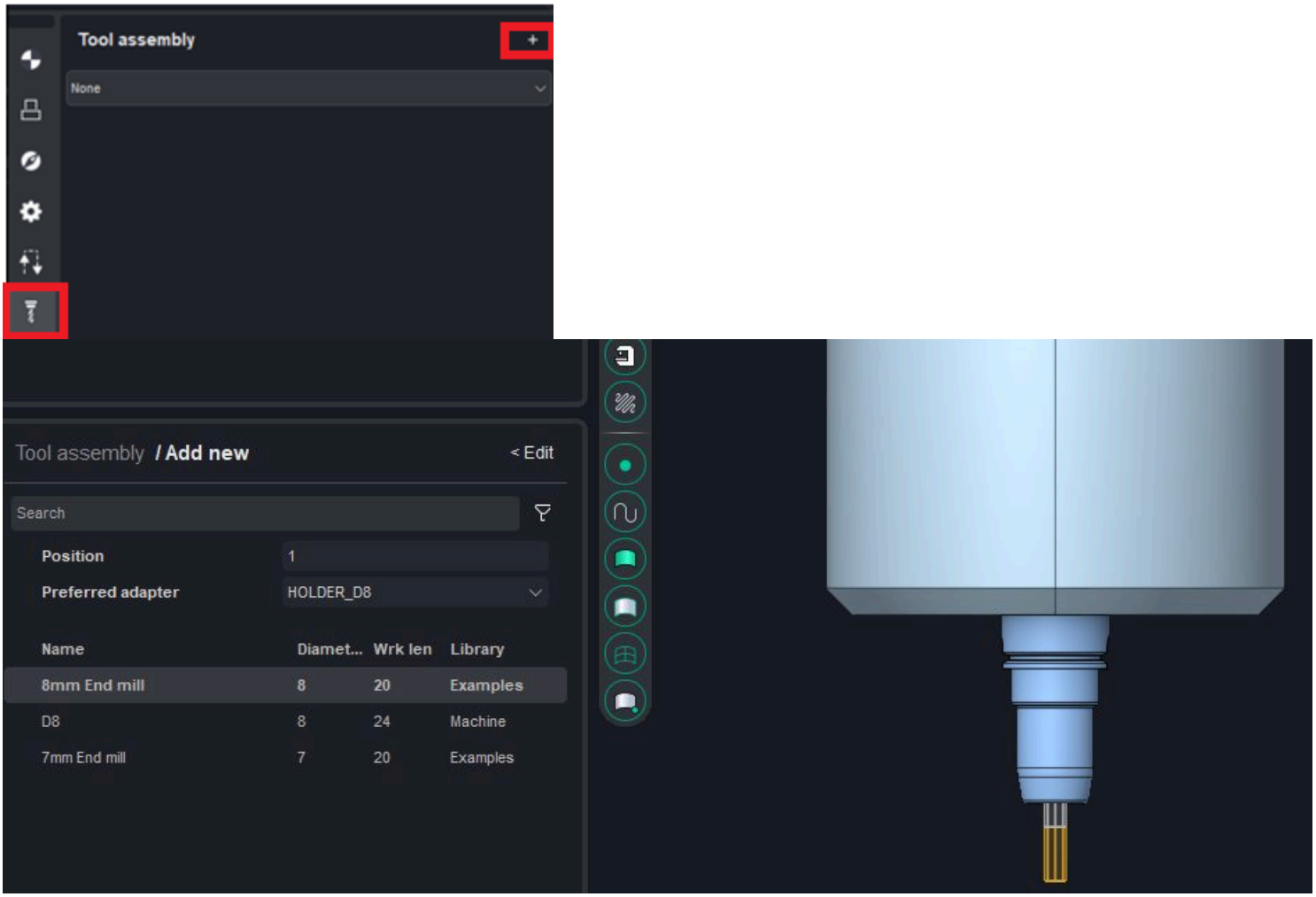


Result.

The assembly is complete and used by the operation: the D8 tool plugs into the HOLDER_D8 adapter through the shared D8 connection, the adapter plugs into the machine magazine through its HSK40 connection, and the system assembles the whole chain automatically from the components found in the active libraries.

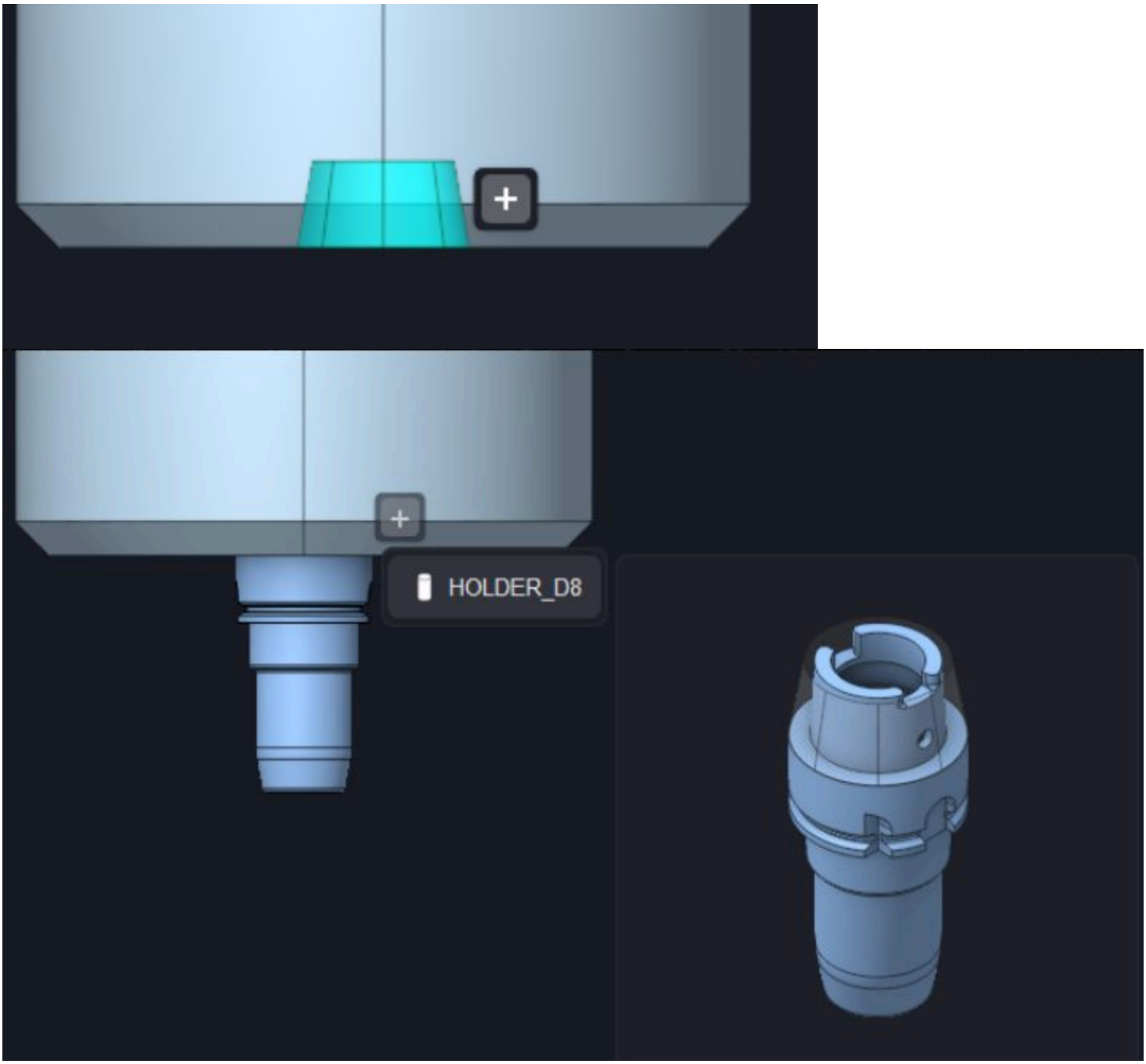
Manual assembly through the tool window.

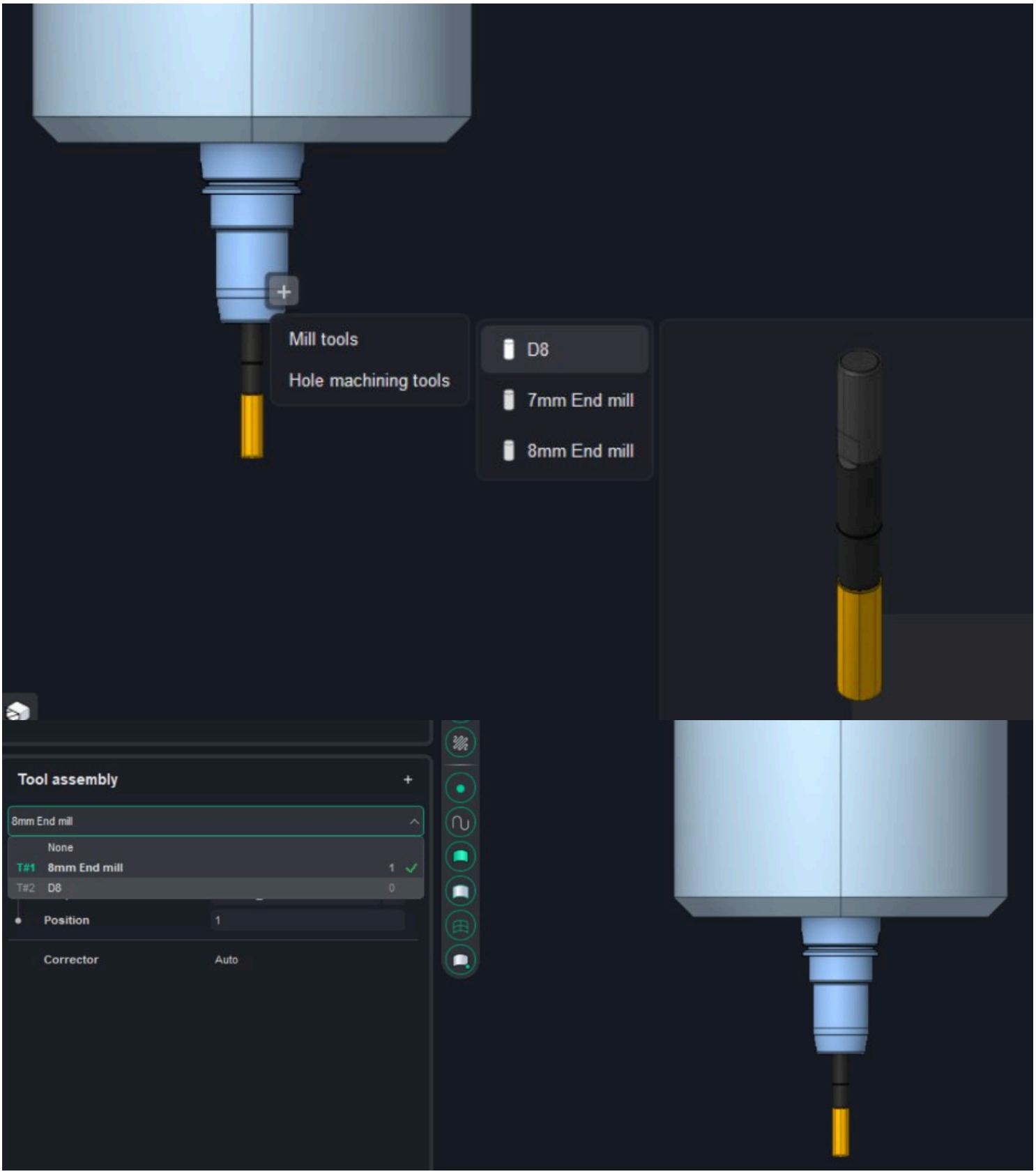
A tool assembly can also be created manually. In the operation's Tool assembly panel the + button opens the Add new panel, where the assembly is composed. For the given operation the system proposes a set of matching assemblies, each shown with its Name, Diameter, Working length, and source Library (for example, 8mm End mill — Examples, D8 — Machine, 7mm End mill — Examples). The selected assembly is mounted on the machine and displayed in the graphics area.



Building directly from the machine.

An assembly can also be created directly in the graphics area, from the machine to the tool. Free sockets are shown on the machine; clicking a free socket displays a + button, and pressing it opens the list of components that match the socket by connection type. First an adapter is added to the machine socket (here HOLDER_D8), then the + button on the adapter's free socket is pressed and a tool is chosen from the offered types (Mill tools, Hole machining tools) and models (D8, 7mm End mill, 8mm End mill). Once built, the assembly becomes available in the operation's Tool assembly list (for example, T#1 8mm End mill, T#2 D8), where it can be selected for the operation.


HOLDER_D8


Mill tools
Hole machining tools
D8
7mm End mill
8mm End mill
Tool assembly
8mm End mill
None
T#1 8mm End mill 1
T#2 D8 0
Position 1
Corrector Auto

Extension and integration possibilities

Application area:

The API allows external developers to seamlessly integrate and extend our program, opening up a world of possibilities. With the API, developers can add custom menu items, giving users even more control and flexibility. They can also read active project properties, enabling deeper analysis and manipulation of project data.

Another great capability provided by the API is the ability to explore the list of operations and tools available in our program. This allows developers to create comprehensive plugins and extensions that enhance the functionality of our program.

In addition, developers can leverage the API to run postprocessors and generate G code programmatically. This automation empowers users to streamline their workflows and easily generate machine instructions.

Furthermore, the API facilitates the import of geometrical models, making it easier than ever to incorporate external models into our program.

Please note that the API is currently in the alpha stage and is actively being developed. While we are confident in its functionality, we recommend using it with care and accuracy.

Harness the power of our program with the new API, and unlock endless possibilities for customization and integration. Happy coding!

Separate API documentation for developers you can find here.

Scripts in ENCY

ENCYcan be extended and customized by scripting.

ENCY use script engine.

There are free ways to use scripts in ENCY.

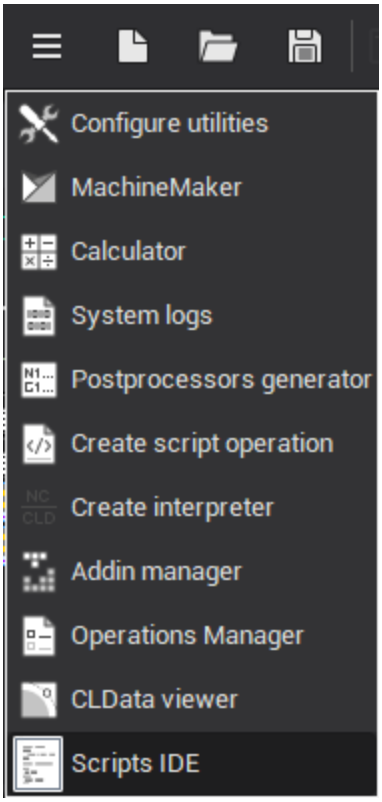
- Create scripted operations. Scripted operations use user provided programming to modify toolpath parameters or create the whole toolpath.
- Place scripts into process of operation calculation. These scripts are executed when a command is added to toolpath.
- Execute script at launch of ENCY.

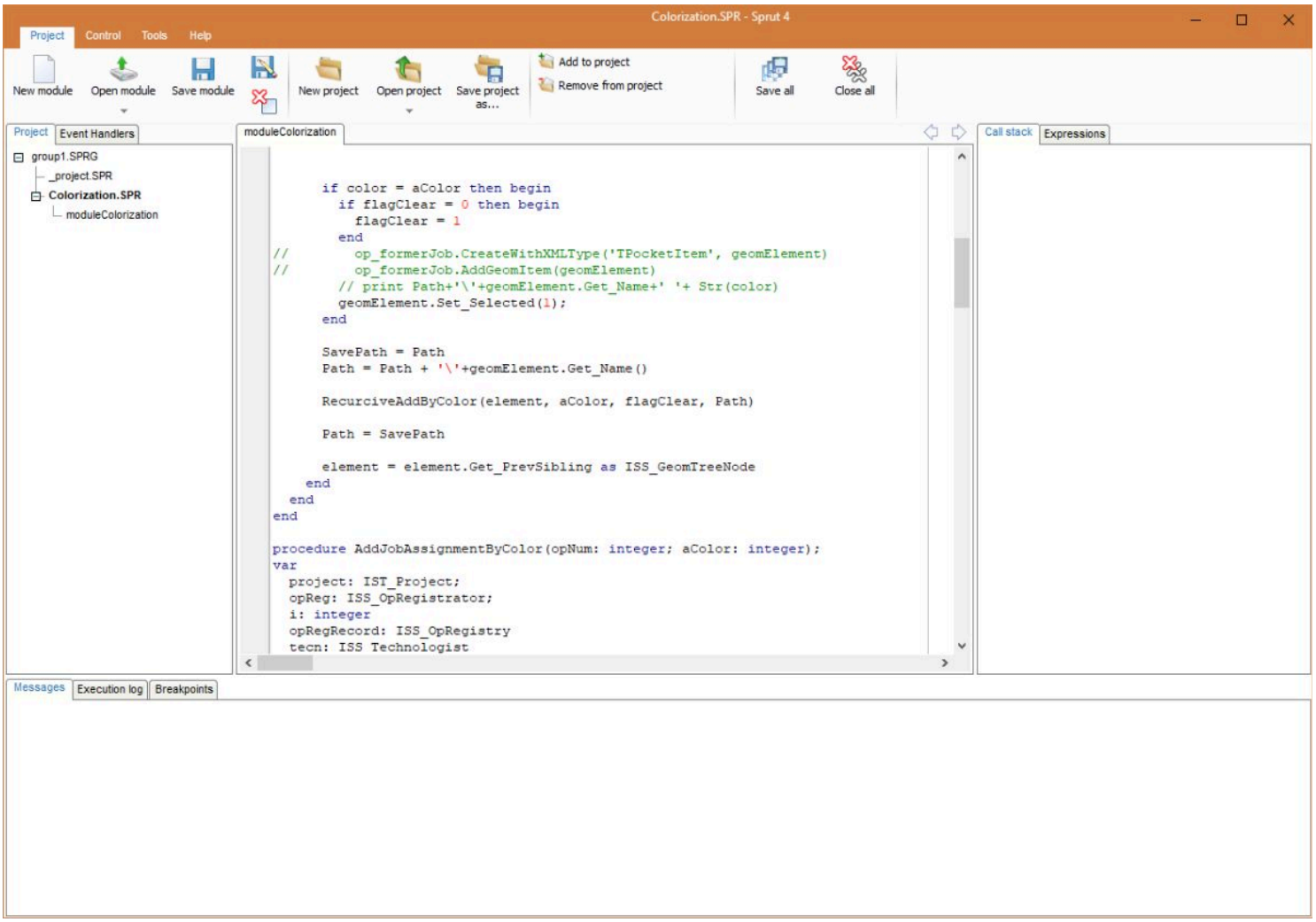
Brief Scripts in ENCY

Sprut4 is a multipurpose programming language. Sprut4 programs are compiled into byte-code and interpreted by Sprut4 virtual machine. Language description is located in Sprut4 Language documentation

Scripts IDE

Scripts IDE window can be opened from "Utilities" submenu by selecting . Scripts IDE lets you create, delete and edit script modules, assign event handlers (procedures) for some objects in the system.





On the left side of the window there are three main panels to manage scripts and procedures to handle events: "Objects" panel, "Events" panel and "Scripts" panel.

"Script units" panel shows a list of files, called script units. Each of these files may contain code written in ENCY PL language. Script units can be stored in different ways.

- As external files that are stored as usual in the file system of your computer. This method of saving allows you to use the same programming code in several projects.
- As files that are embedded into the project file *.stcp. Such file may contain programming code relating to a single project and that can be used from different locations of the project.
- As files that are stored inside the technological operation. This way should keep the programming code that applies only to a single operation. After that this operation can be saved as default parameters.

In the central part of the window there is a code editor, where you can view and edit the code of script units. To view the code of particular unit, double-click on the unit name in the list on the panel "Scripts". Each unit opens in a separate tab of code editor. Thus, you can edit simultaneously several files.

To manage units and tabs you should use the buttons on the toolbar at the top of the window.

The "Create script" button allows to create a new empty script unit. Depending on which item is selected in the "Scripts" list the new unit will be stored either inside the operation or inside the project or as an external file. After pressing the button opens a dialog box in which the need to specify a unique unit name (file name for an external file).

The "Open script" button allows you to open an external file in the Scripts IDE and add it to the list of script units of the system. After clicking the button opens a standard file selection dialog box.

The "Save script" button keeps changes of text of the active script (active tab), made in the code editor, to the file. If the script is stored inside an operation or project, the changes are only saved inside the operation (project). In this case, to write the changes to the disk, you must also save the project.

The "Delete active script" button removes active script unit physically.

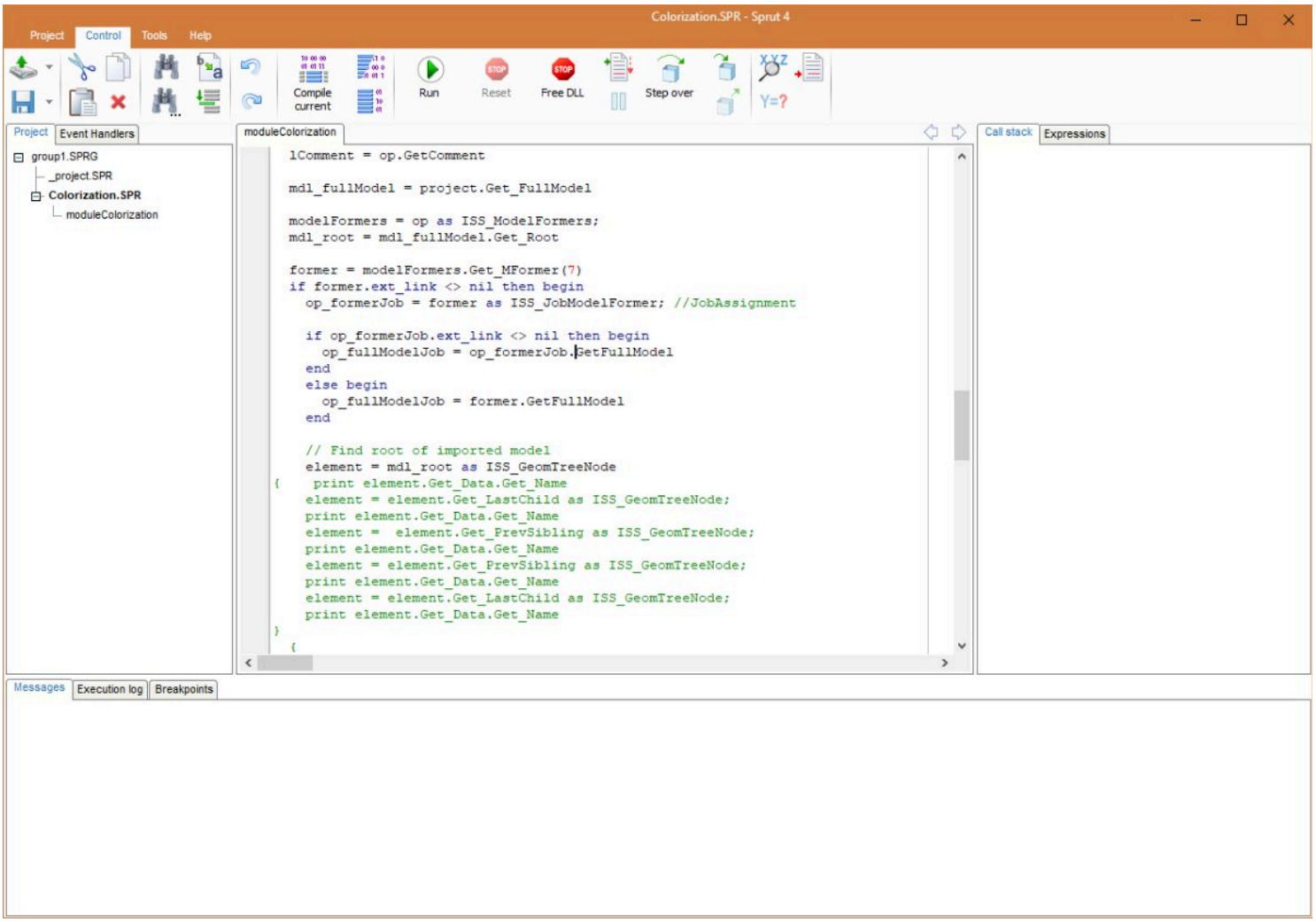
"Objects" panel holds objects that may have event handlers. List of available events for the selected object are shown in the "Events" panel below.

Event is a message that appears at various points of toolpath calculation. Events are designed to provide user an ability to affect toolpath generation in some way. Every time the event is fired for an object the assigned handler routine is called.

Key objects In ENCY that can contain any events are technological operations. Therefore, a list of objects in the script editor is very similar to the list of operations of the technological process. Operations, in turn, can have child objects that can have events.

The event handler - a ENCY PL routine (function or procedure), in which the user can describe on the programming language execution of some actions when the event occurs. In ENCY in order to assign some procedure for processing a particular event of object you should select the desired object in the object list and should choose the desired event in the event list. Then in the "Script unit" field next to the name of the event you should set (choose from list) the name of script unit in which you want to look for a handler procedure. In the "Method name" field is need to specify the name of the procedure. If the event handler procedure is not written yet, you can automatically create an empty prototype of the handler. To do this make double-clicking the mouse on the desired event in the list, or click on the button next to the name of the event. If in the "Script unit" field is the name of an existing unit then new procedure will be added to this unit, otherwise it will be added to the unit that is currently opened in the code editor.

Control menu panel



"Control" menu contains tools that help edit, check syntax and debug scripts.

"Compile" button allows you to perform code check for syntax errors.

"Run" button used to continue script execution after pause.

"Reset" button is used to reset script execution and toolpath calculation.

"Run to line" button runs script until execution reaches current line in editor.

"Pause" button pauses script execution.

"Step over" button executes current script command without stepping into invoked subprogram (procedure or function).

"Step in" button executes current script command and enters code of invoked subprogram.

"Step out" button runs code of current subprogram and pauses when execution returns to the invoking subprogram.

"Add breakpoint" button  marks a line in code where execution must be paused.

At the bottom of the Scripts IDE are "Messages", "Execution Log" and "Breakpoints" panels. Messages panel holds various system and compiler messages. Execution log panel contains system and print output. "Breakpoints" shows list of breakpoints.

Application Programming Interface

Application Programming Interface (API) - A set of predefined classes, procedures, functions, structures and constants that provided by the application for use in external programs and user programs. API defines the functionality that is provided by a program (module, library) for the user.

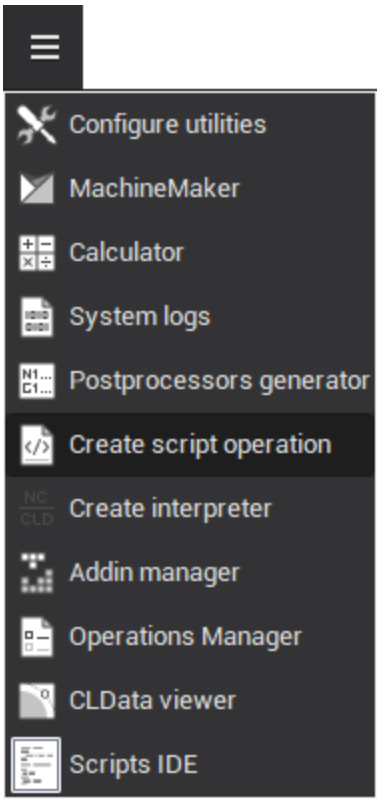
API for working with the system using ENCY PL language can be found in ENCY system API documentation

Scripted operation

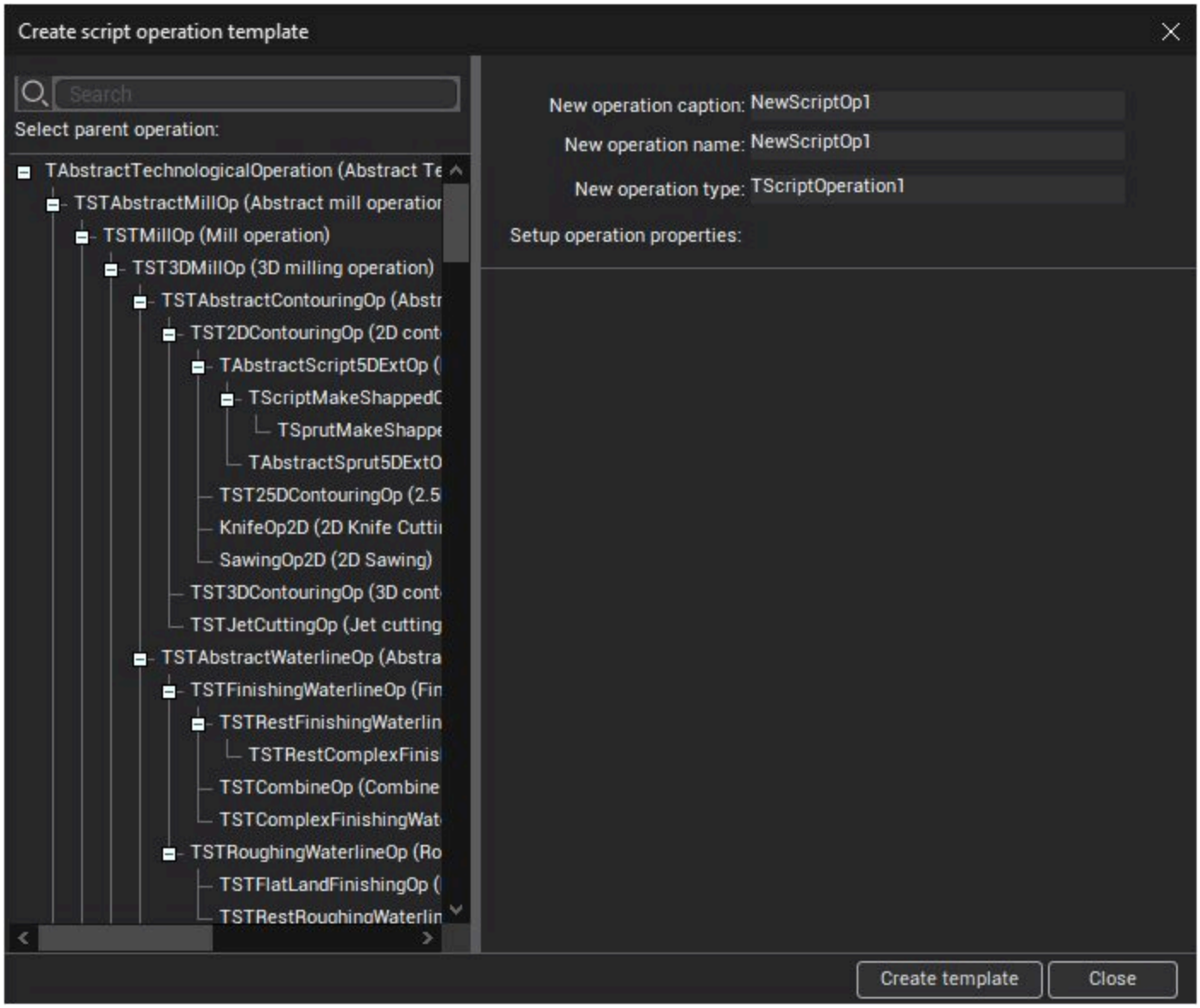
Script operation can have OnCreateMethod, ChangePropertyMethod and MakeWorkPathMethod methods defined.

- OnCreateMethod is a ENCY PL subprogram that is executed when operation is created.
- ChangePropertyMethod is executed when operation property is changed.
- MakeWorkPathMethod is executed when operation toolpath is calculated.

To create script operation use Create script operation.



First select parent operation. Any operation can be selected as parent. Parent operation defines basic properties of new script operation.



Input new operation parameters:

- Caption or title - will be displayed in the technology tree. Operation caption is value of Comment node in operation xml-descriptor.
- Name - this value will be new operations name in dialogs and menus for adding new operation to project. Name node in operation xml-descriptor.
- Type - new operation type. Id attribute of operation xml-descriptor.

Use Operation properties panel to select what inherited properties will be visible in inspector.

When all parameters are set click Create template button to create xml-unit containing xml-descriptor of new operation and template ENCY PL project.

Also you can manually create operation descriptor xml-file and include it into ENCY xml-configuration.

Operation with scripted events

Operation with scripted toolpath commands is created from a regular ENCY operation using Scripts Editor.

On Scripts Editor window's Left Panel select Events Handlers tab. Select an item in Objects tree and the Events tree will display available events for that object. Double-clicking on event will create a code template for method that will be executed when ENCY processes that object in toolpath. See Scripts IDE for more information.

Scripted ENCY launch

Supply ENCY PL -script as command line parameter for ENCY launch. ENCY will execute that script right after system initialization is complete. Script can import geometry and create operations.

PLMIntegrations

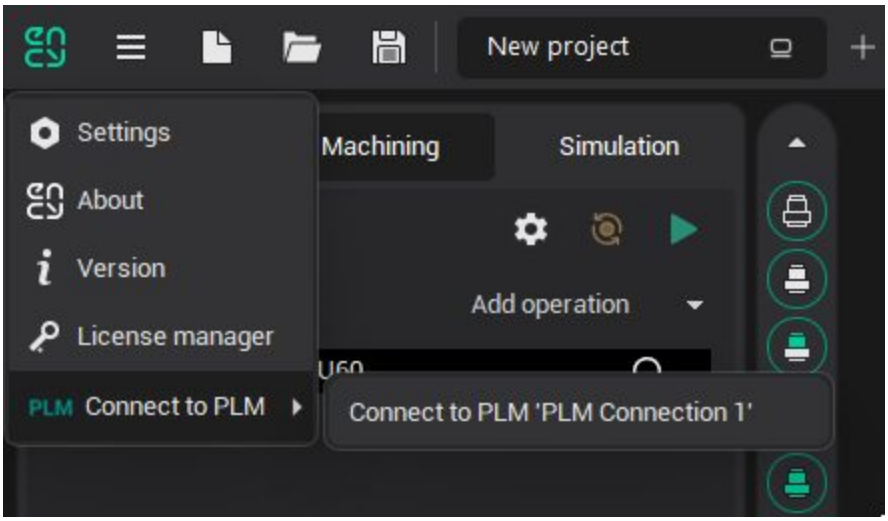
The CAM system offers the capability to integrate with PLM/PDM systems. It provides API interfaces that enable the development of custom integration modules with external systems. Comprehensive documentation is available here.

Currently, the PLM integration functionality supports the saving and loading of projects, machines, postprocessors, tools, and the import of geometry models. When CAM is connected to an external system via an integration module, additional interface buttons for interacting with the system appear in the main and auxiliary windows.

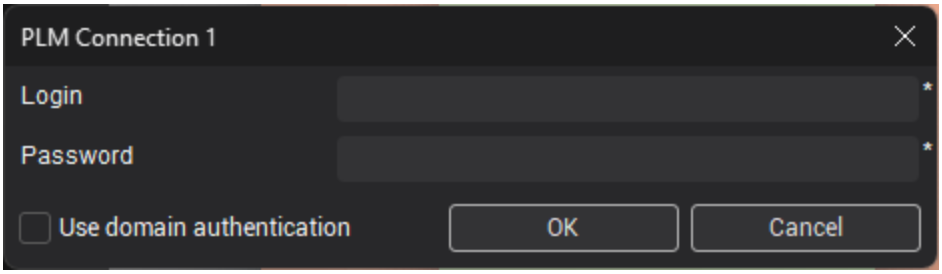
You can try out the integration using an example extension, available in the assets section of the GitHub releases page. Navigate to the latest release and download the provided extension file.

Authorizing an External System in the CAM Main Window

To authorize an external PLM system, go to the main menu and hover over the Connect to PLM option. *This option appears in the menu only if at least one PLM connection is configured.*

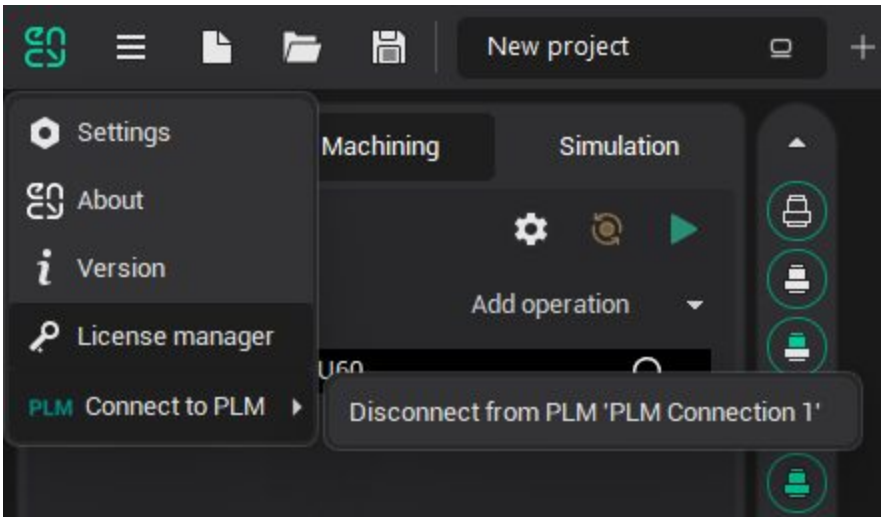


A submenu will appear, displaying all available PLM connections. Select the desired PLM connection to open the authorization window.



Enter your account credentials or select the Use domain authentication checkbox to proceed.

To disconnect from the PLM system, return to the main menu, hover over Connect to PLM, and click Disconnect from *[connection name]*.


New project
Settings
About
Version
License manager
PLM Connect to PLM
Machining
Simulation
Add operation
Disconnect from PLM 'PLM Connection 1'

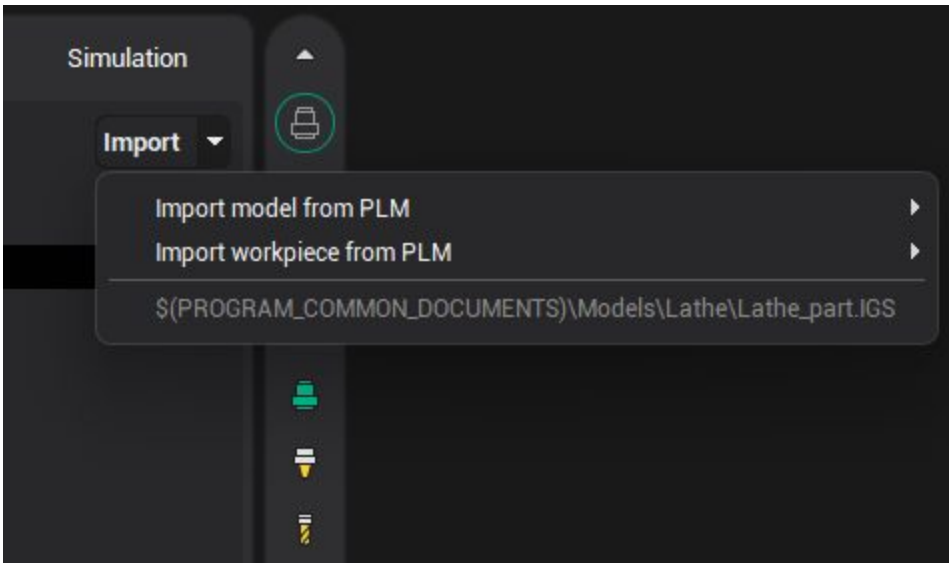
Importing a Geometry Model from an External System

Geometry models of parts and workpieces can be imported directly from a PLM system using the Import button in the main window toolbar. This button includes a small arrow that opens a drop-down menu with additional import options:

- Import model from PLM
- Import workpiece from PLM

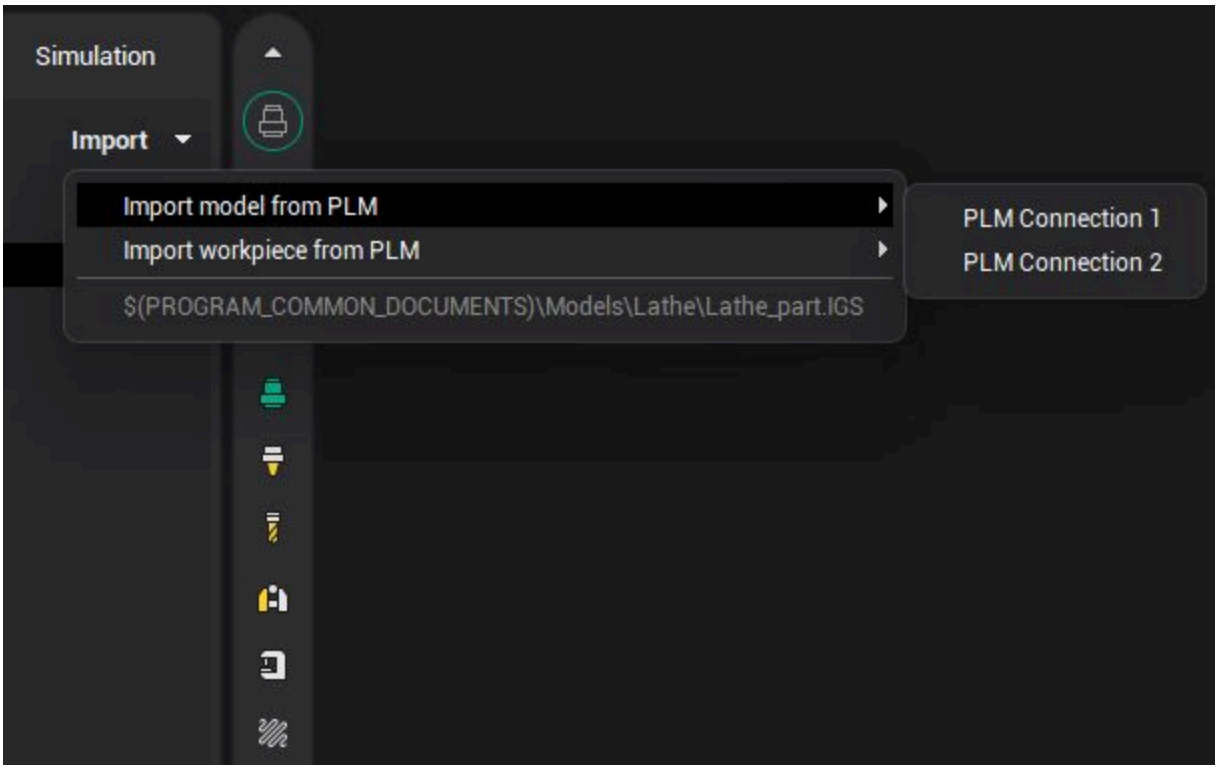
Each option includes a submenu where you can select a configured PLM connection.

Note: The PLM-related options appears only if at least one PLM connection is configured.

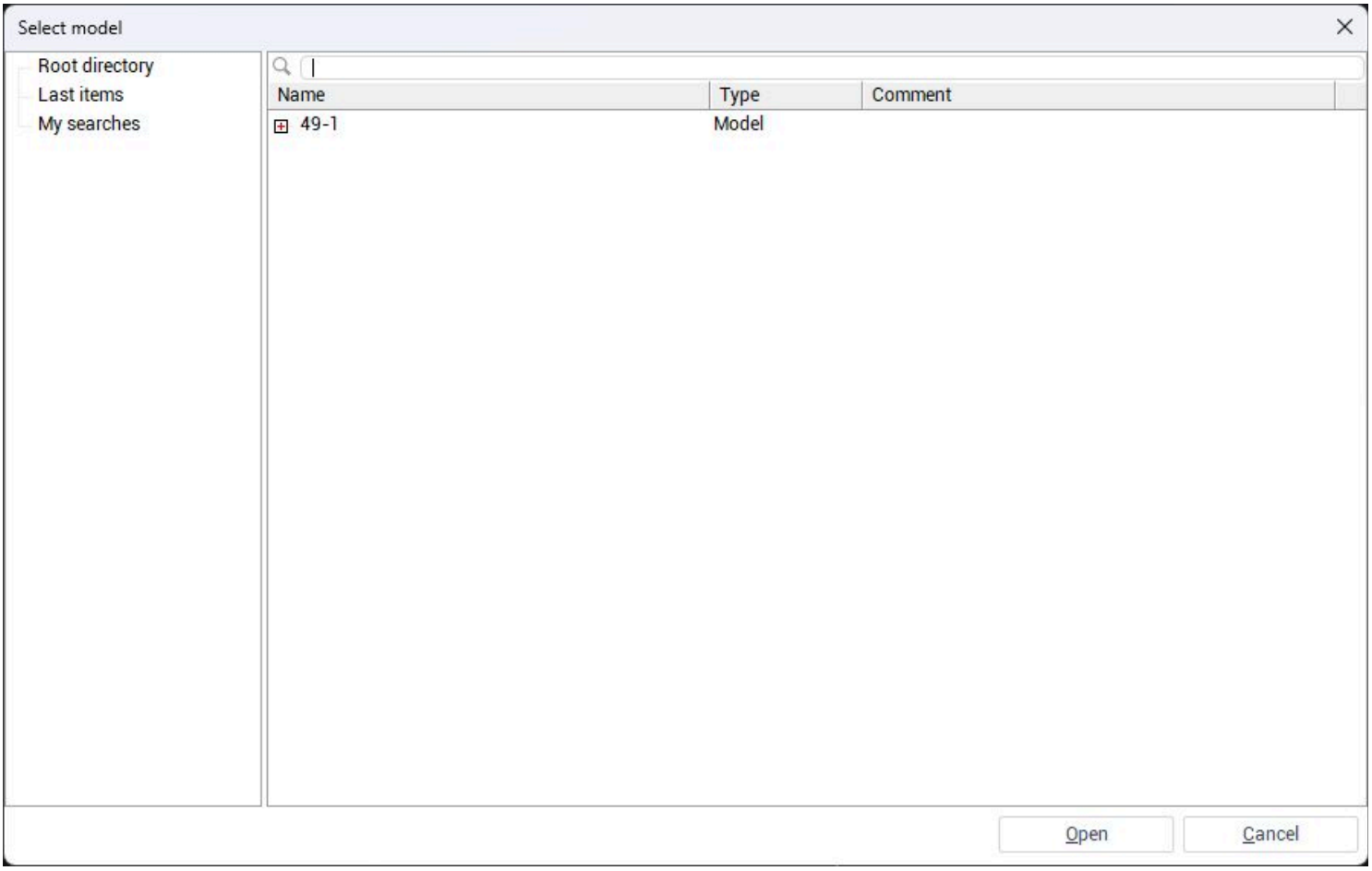


Importing a Geometry Model from an External System

Click the arrow next to the Import button, then select Import model from PLM → *[connection name]*.

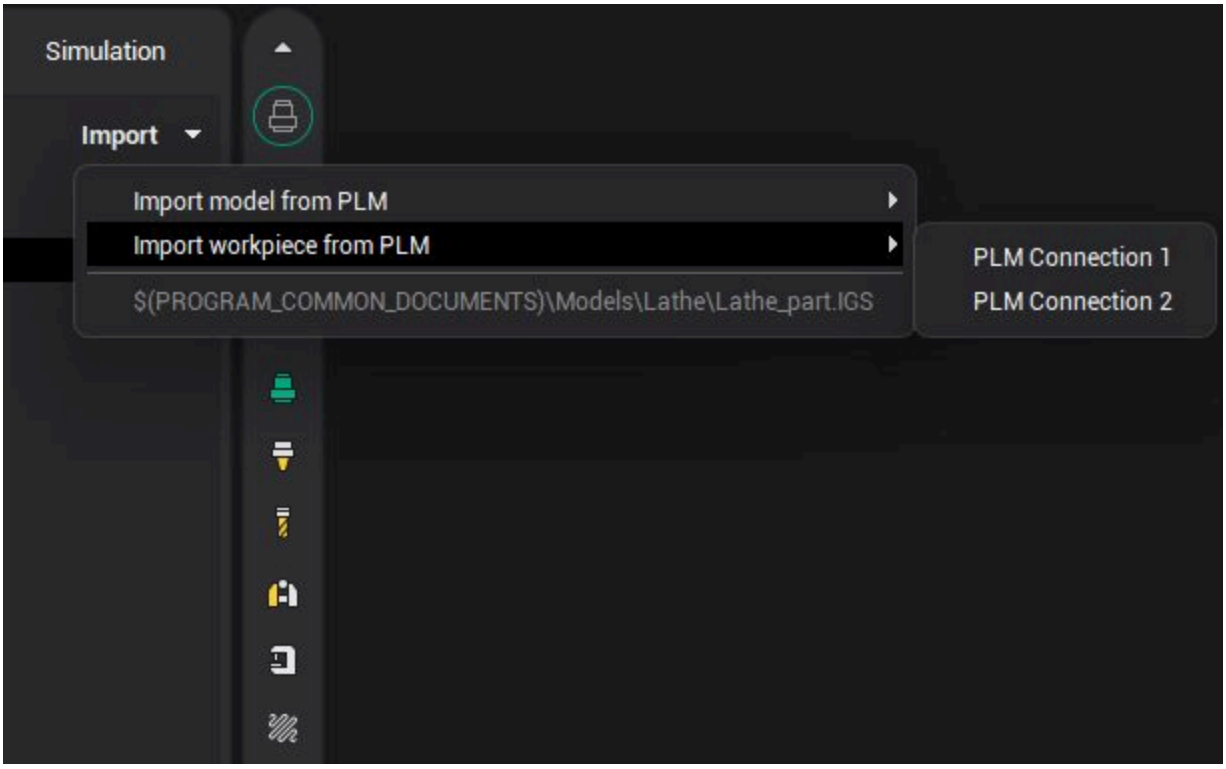


A window will open with a tree view of available geometry models in the external system. Select the desired model and click Open. The model will be imported into the current project.

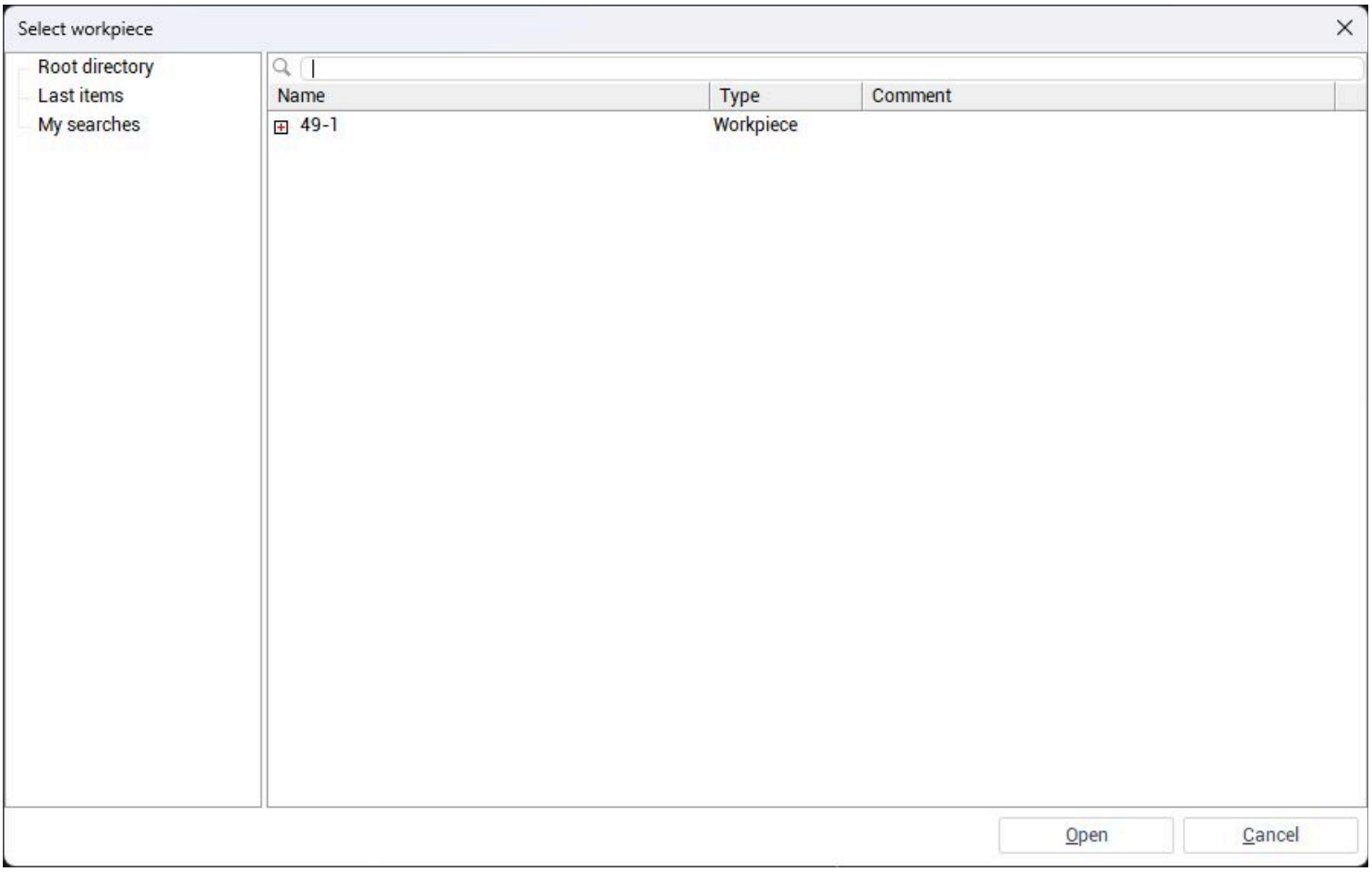


Importing a Workpiece from an External System

Click the arrow next to the Import button, then select Import workpiece from PLM → *[connection name]*.



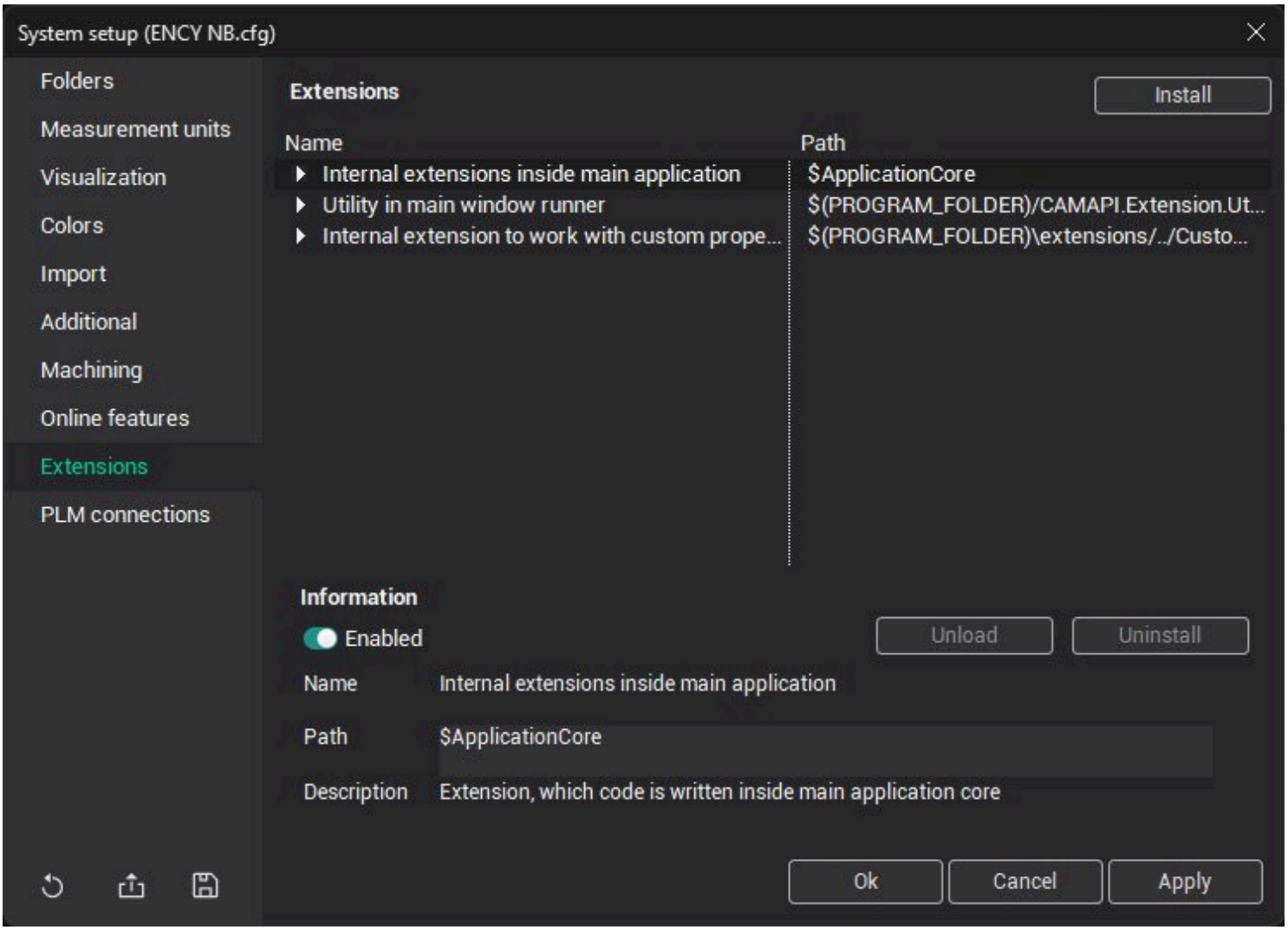
Browse the external system's object structure to find the required workpiece, then click Open. The selected workpiece will be imported into the current project.



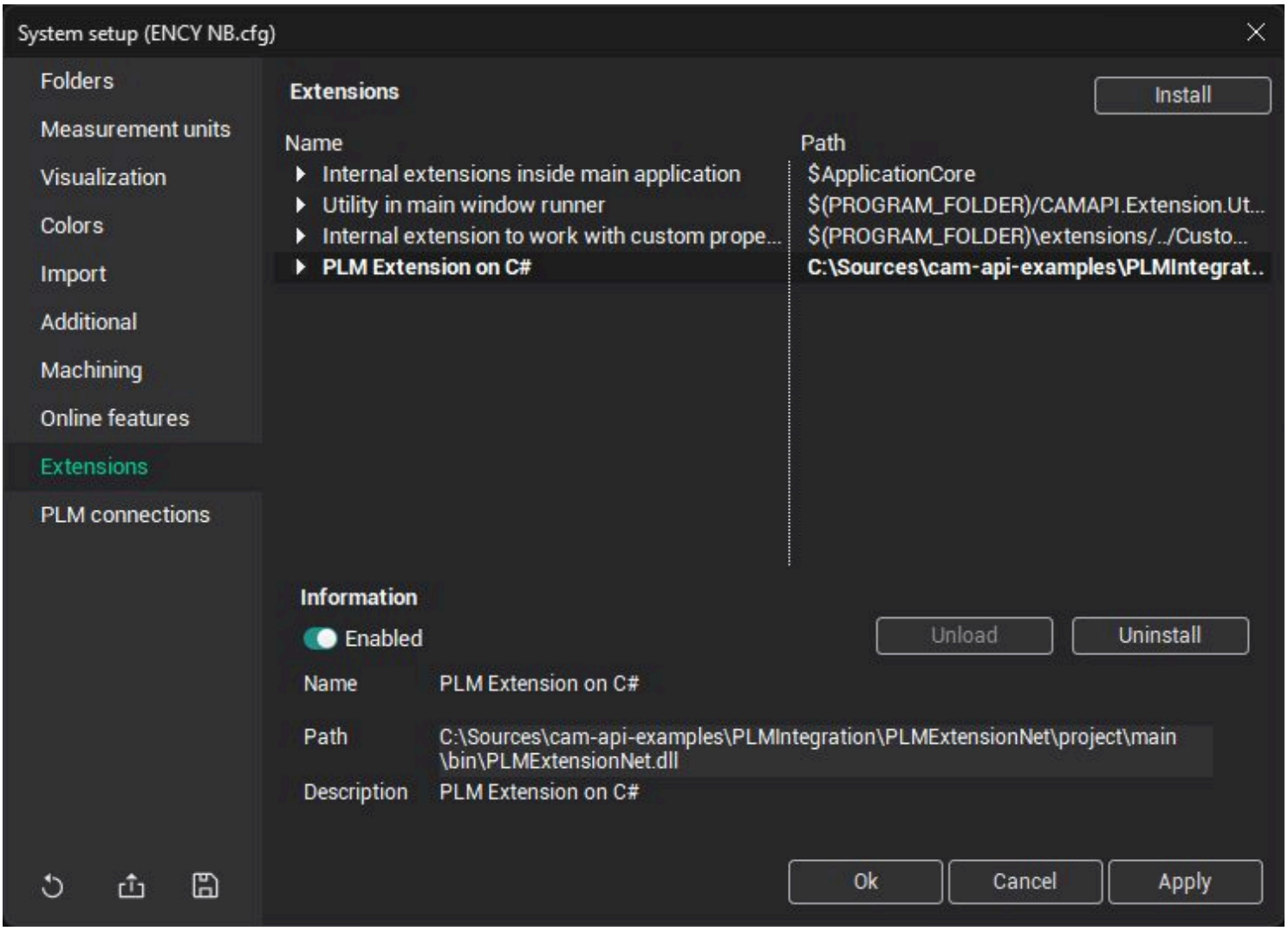
Installing and Configuring PLM Integration Modules

Setting Up the Integration Extension

To setup PLM system integration module, open System Settings window and go to Extensions tab.



Click the Install button. In opened File Explorer window choose extension's json-file or dext-file and open it. Successful installation example:



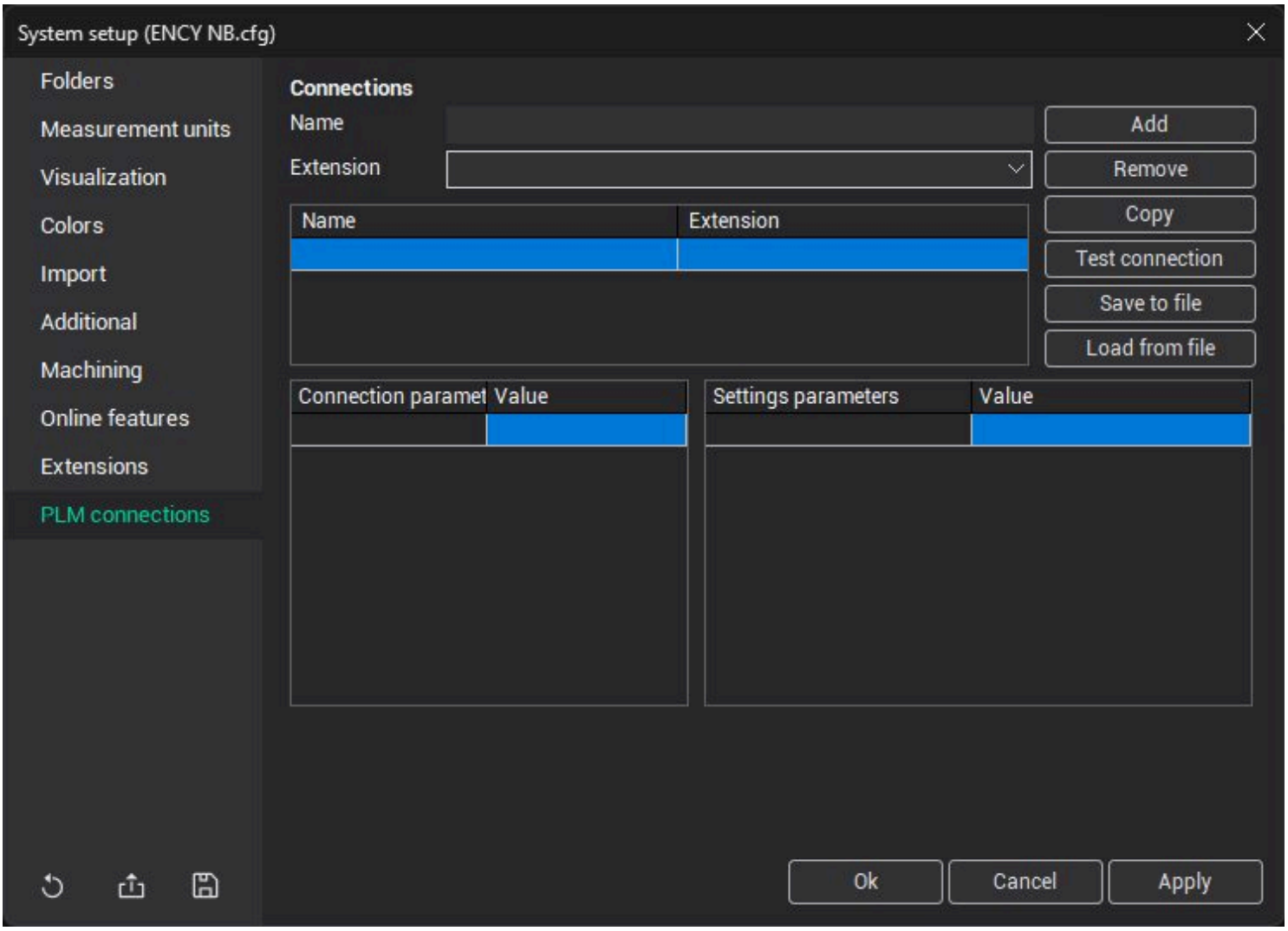
You can also disable extension or uninstall it, select it in list of extensions and switch toggle on Enabled or click the Uninstall button.

The extension can also be installed via the command-line interface (CLI, ExtensionManagerCLI.exe could be found at the program installation folder). To do so, open the Windows Command Prompt and enter the command:

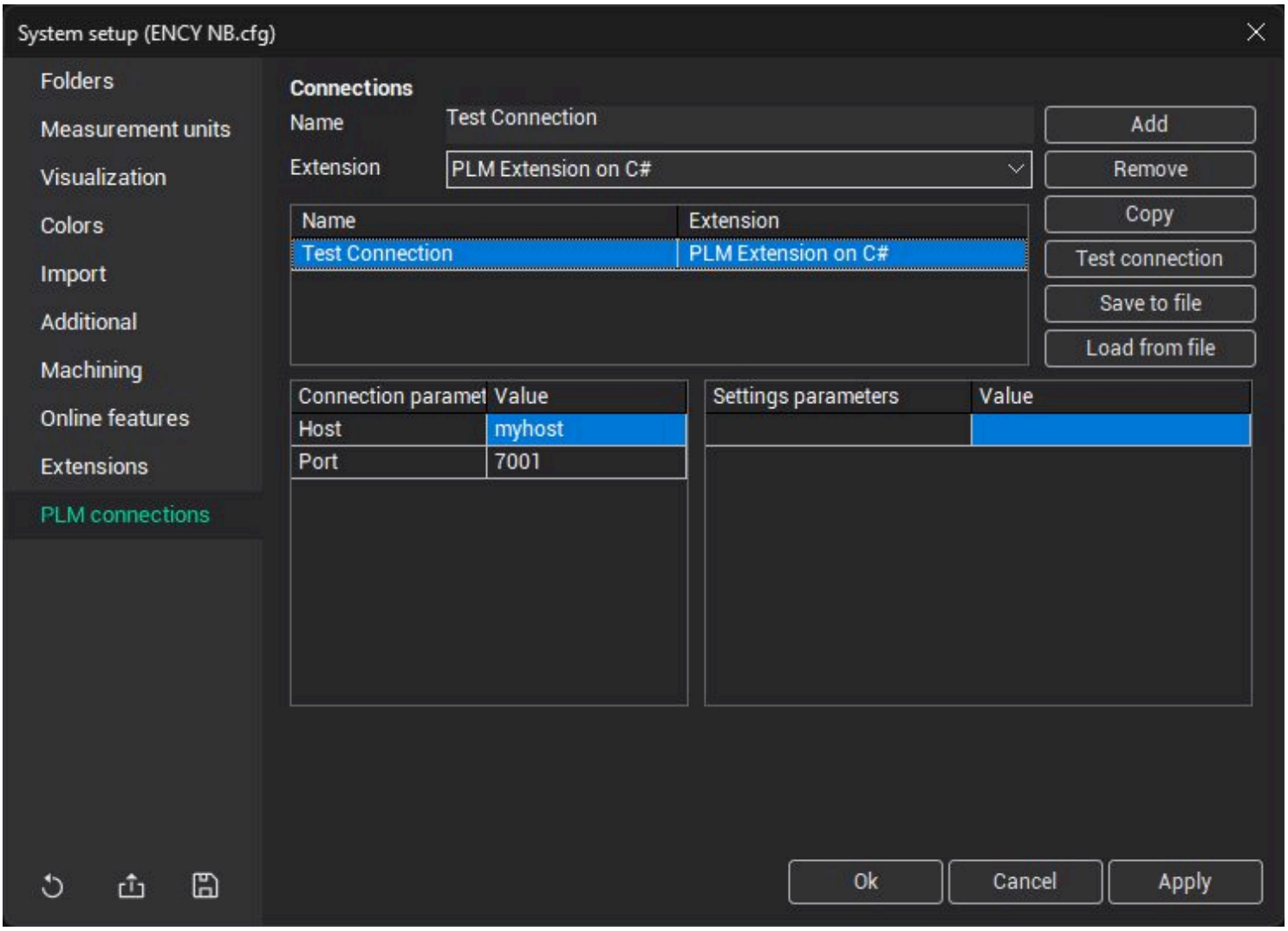
ExtensionManagerCLI.exe register-library --path "*[path to the extension]*" --storage stAllUsers

Configuring PLM Connections

To create and configure connection, open the System Settings window and go to the PLM connections tab.



Write connection name in the Name text field and choose installed PLM extension in drop-down menu. Then click Add button.



You can configure selected connection changing values of suggested connection parameters or settings parameters in tables below.

Also you can save settings to file, click the Save to file button. This file can be used to load settings, click Load from file button.

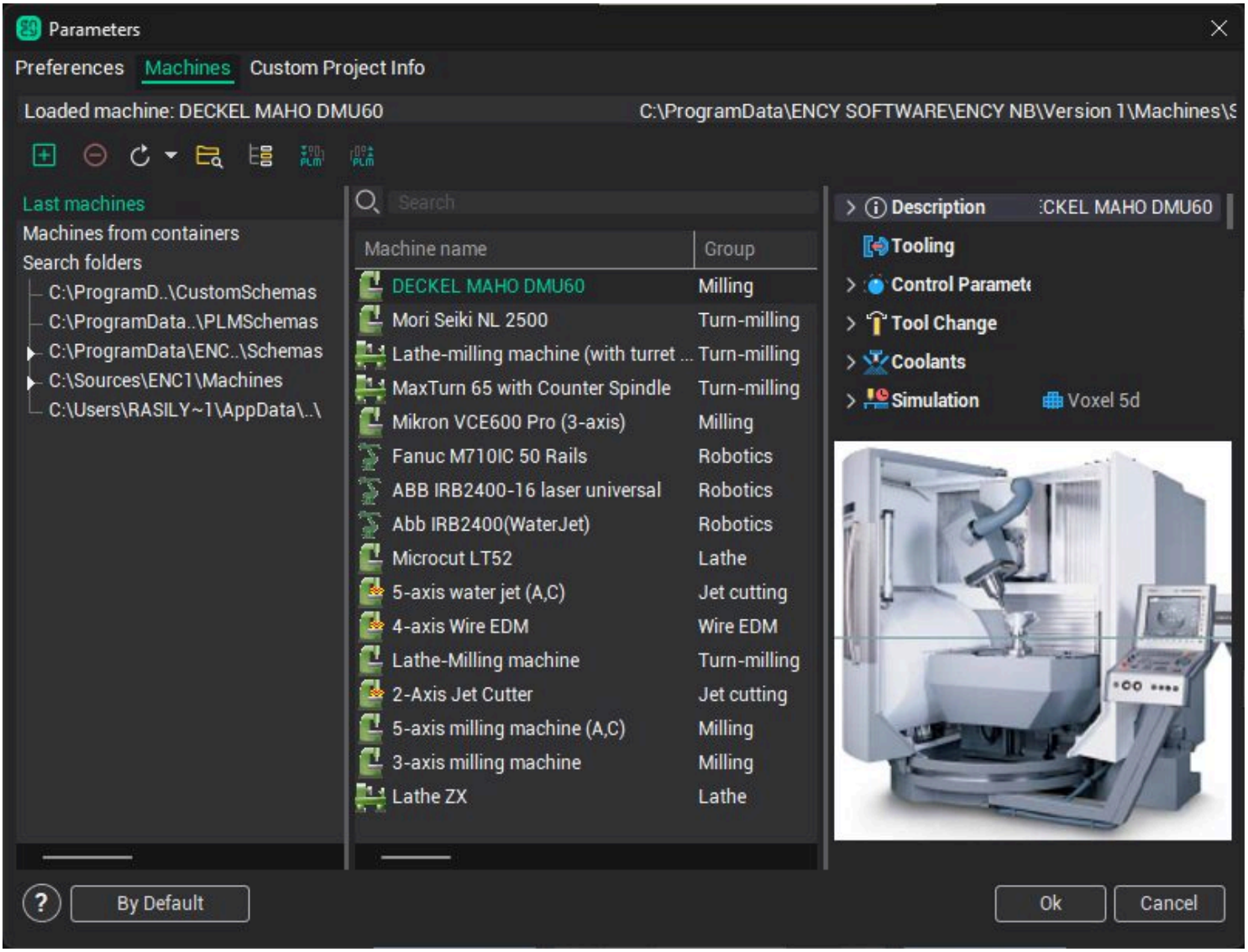
You can remove connection. Select it in list of connections and click the Remove button.

To check connection, click the Test connection button and enter your account credentials.

Click the Apply or Ok buttons to apply changes.

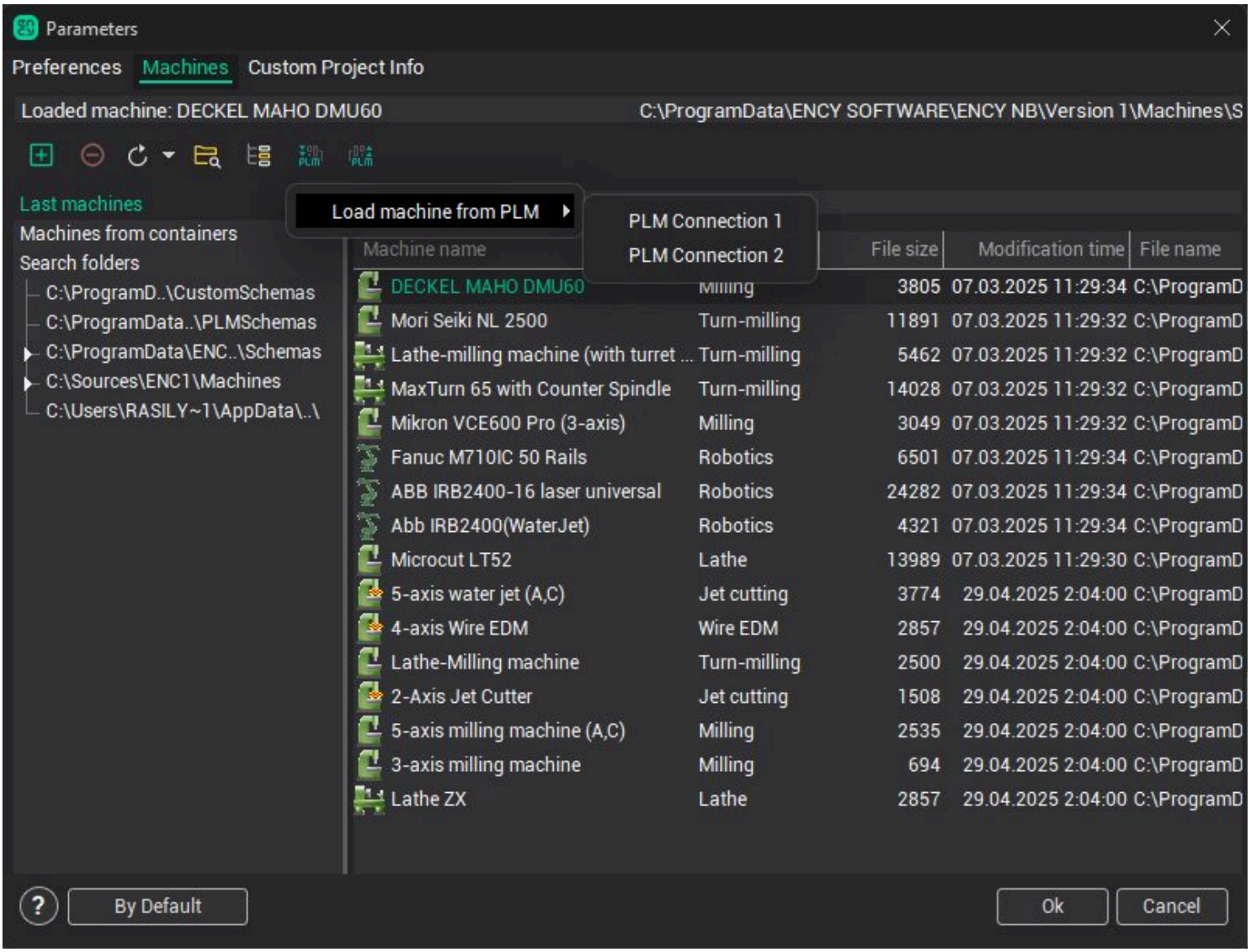
Loading and Saving Machines to an External System

To manage machines with an external system, open the Parameters window and navigate to the Machines tab. In the upper panel of this tab, you will see the buttons Load machine from PLM and Save machine to PLM, which allow interaction with the external system.

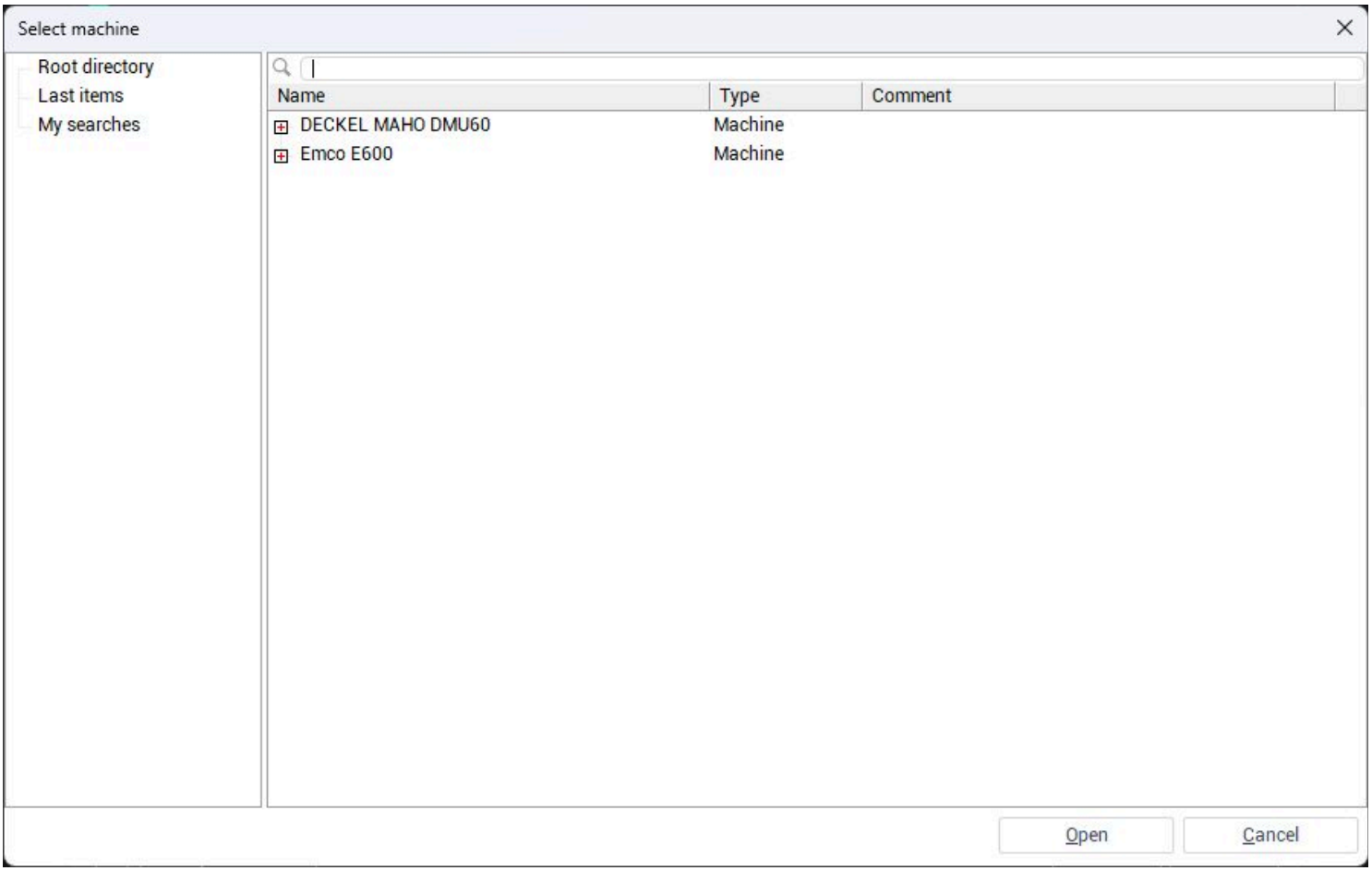


Loading a Machine from an External System

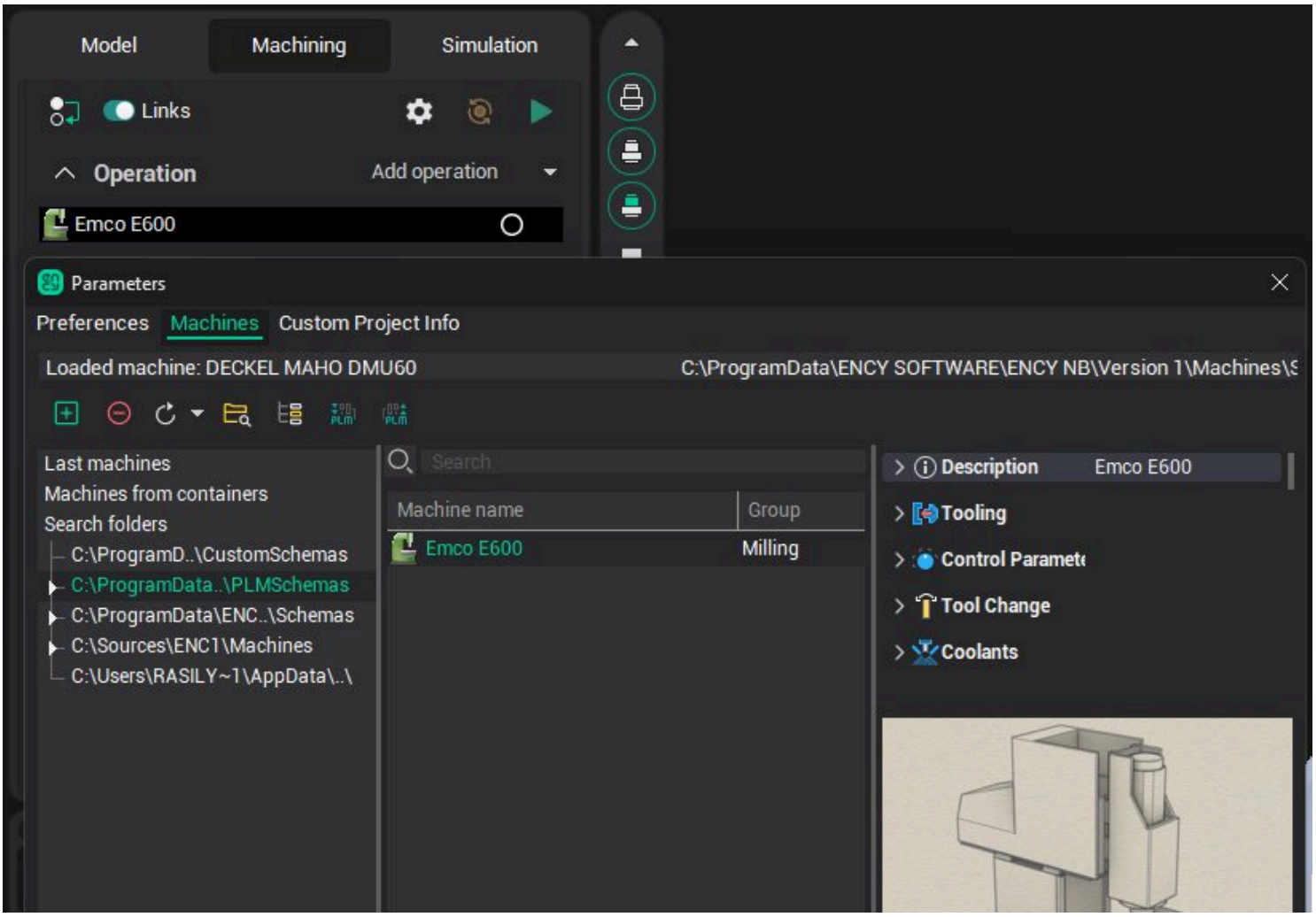
To load a machine from an external system, click the Load from PLM button. In the drop-down menu, select *[connection name]* to open a window displaying a tree of available objects from the external system.



Locate the desired CAM machine by expanding the nested objects in the tree or by using the name search. Then click the Open button.

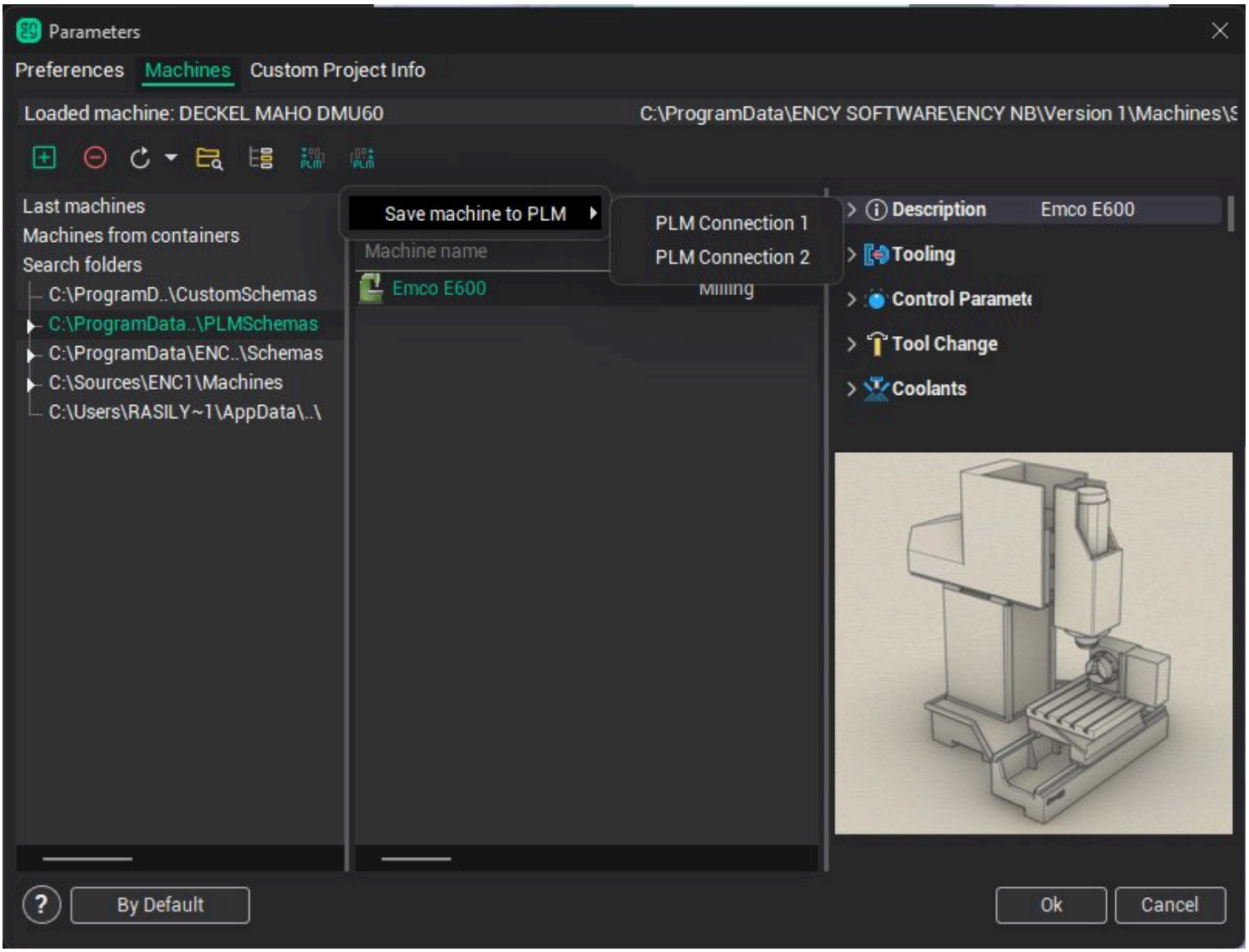


The selected machine will be loaded and set as the active machine in the main window.

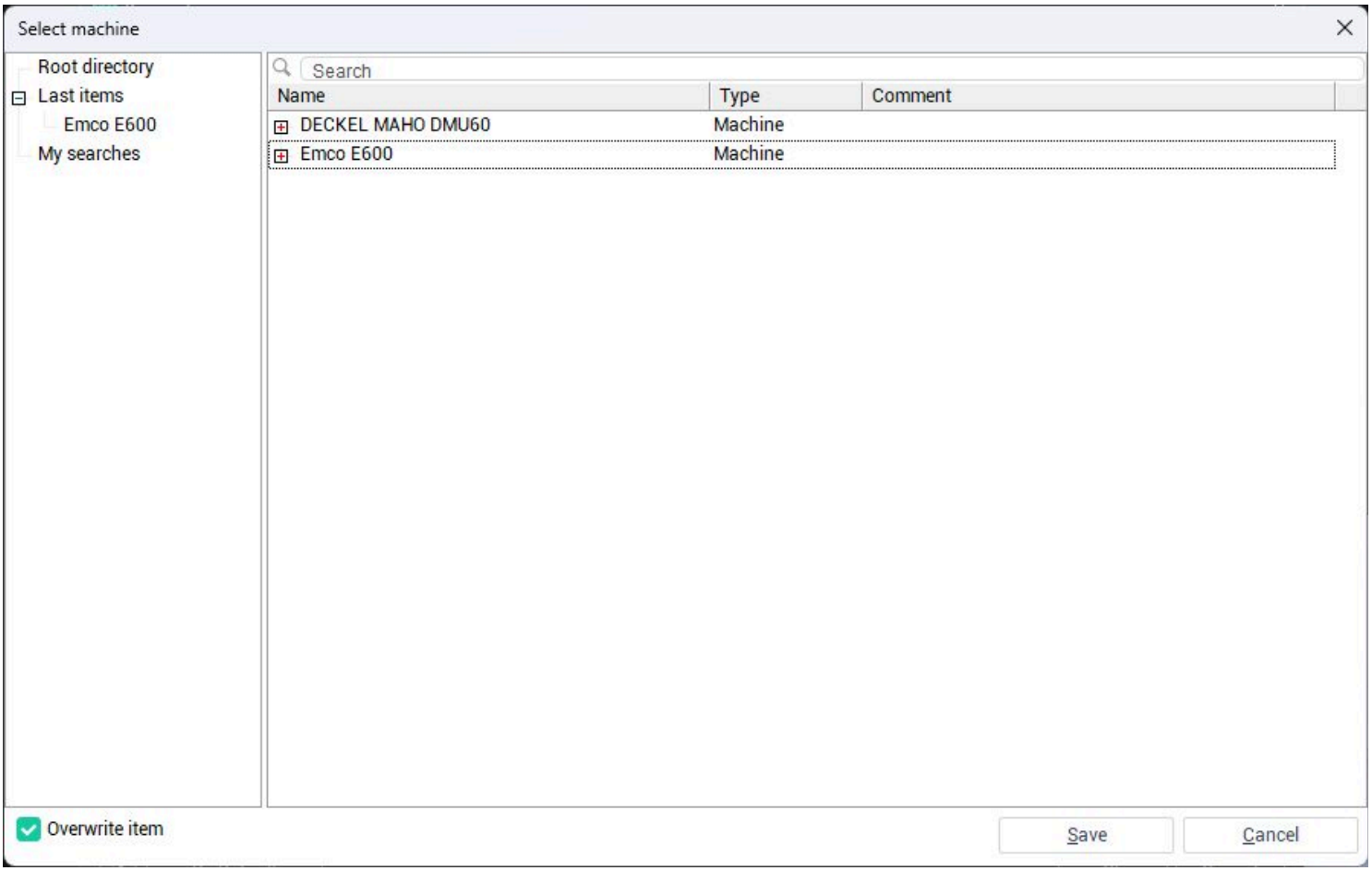


Saving a Machine to an External System

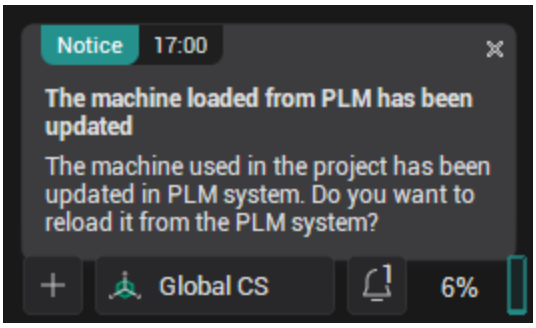
To save a machine to an external system, select the machine you want to save, then click the Save to PLM button and choose *[connection name]* from the drop-down menu to open a window displaying a tree of available objects from the external system.



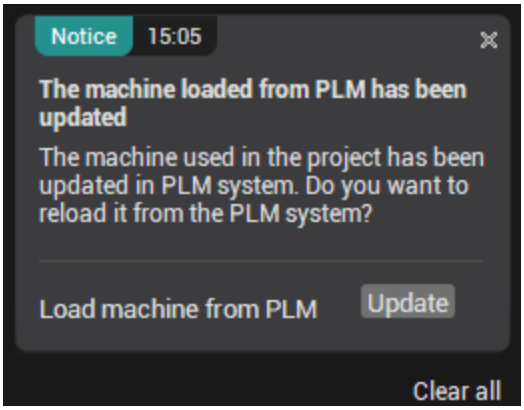
Select the destination element where the machine files should be saved. To overwrite existing files, check the Overwrite item checkbox, then click the Save button.



If a machine element has been modified in the external system, a notification will appear in the main window.



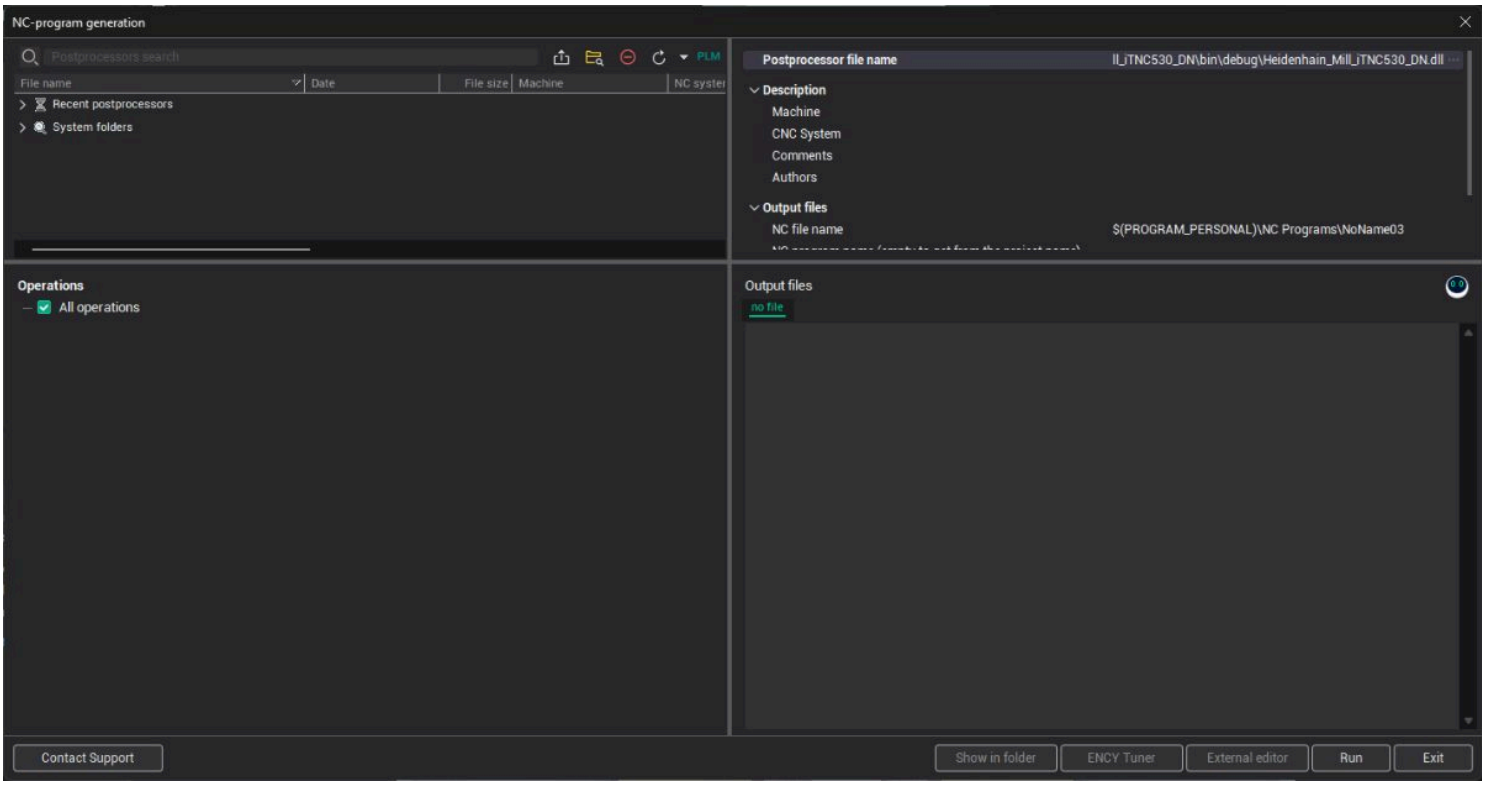
Expand the notification and click the Update button to load the updated version of the machine.



You can check for updates in the Parameters window by right-clicking the machine and selecting Check for updates in PLM from the context menu.

Loading and Saving Postprocessors to an External System

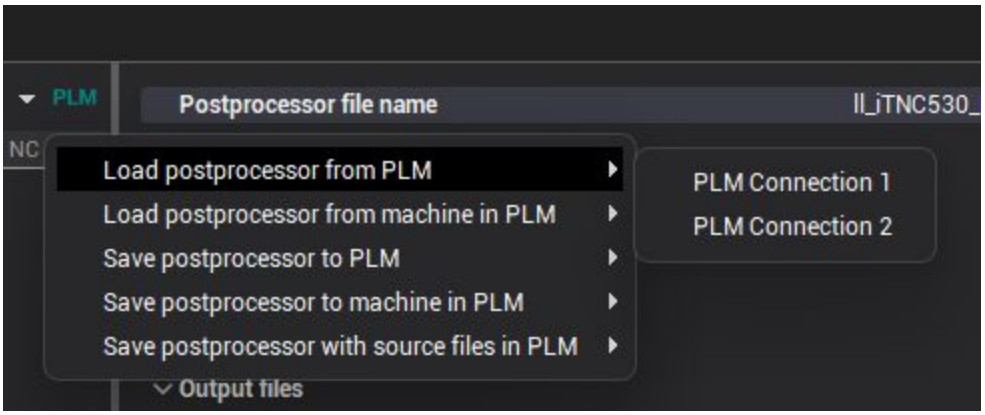
To manage postprocessors with an external system, open the NC-program generation window.



In the upper panel of this window, there is a PLM button. Clicking this button opens a drop-down menu with the following options:

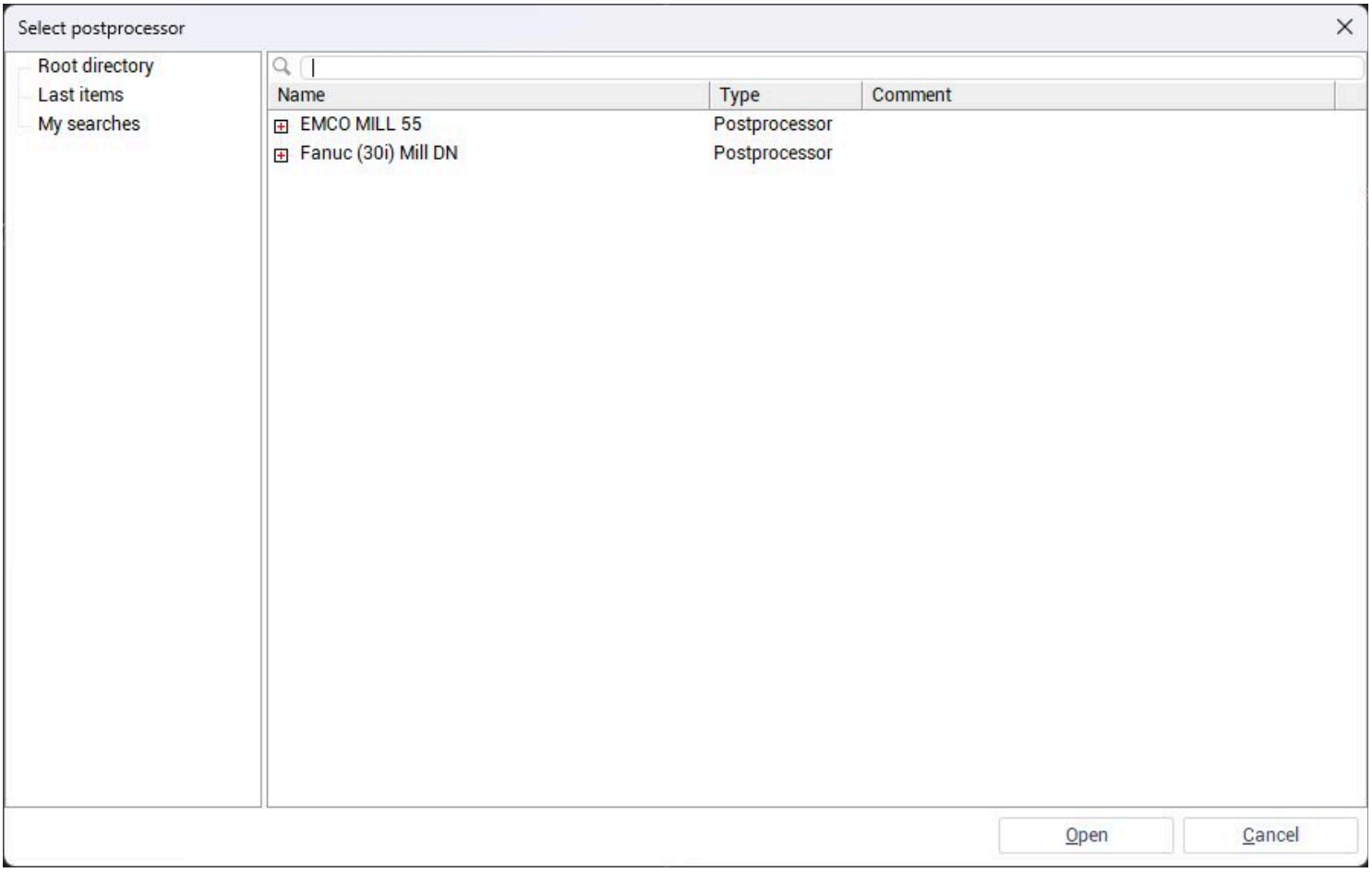
- Load postprocessor from PLM
- Load postprocessor from machine in PLM
- Save postprocessor to PLM
- Save postprocessor to machine in PLM
- Save postprocessor with source files to PLM

Each of these options includes a submenu for selecting a PLM connection *only if more than one connection is configured*.

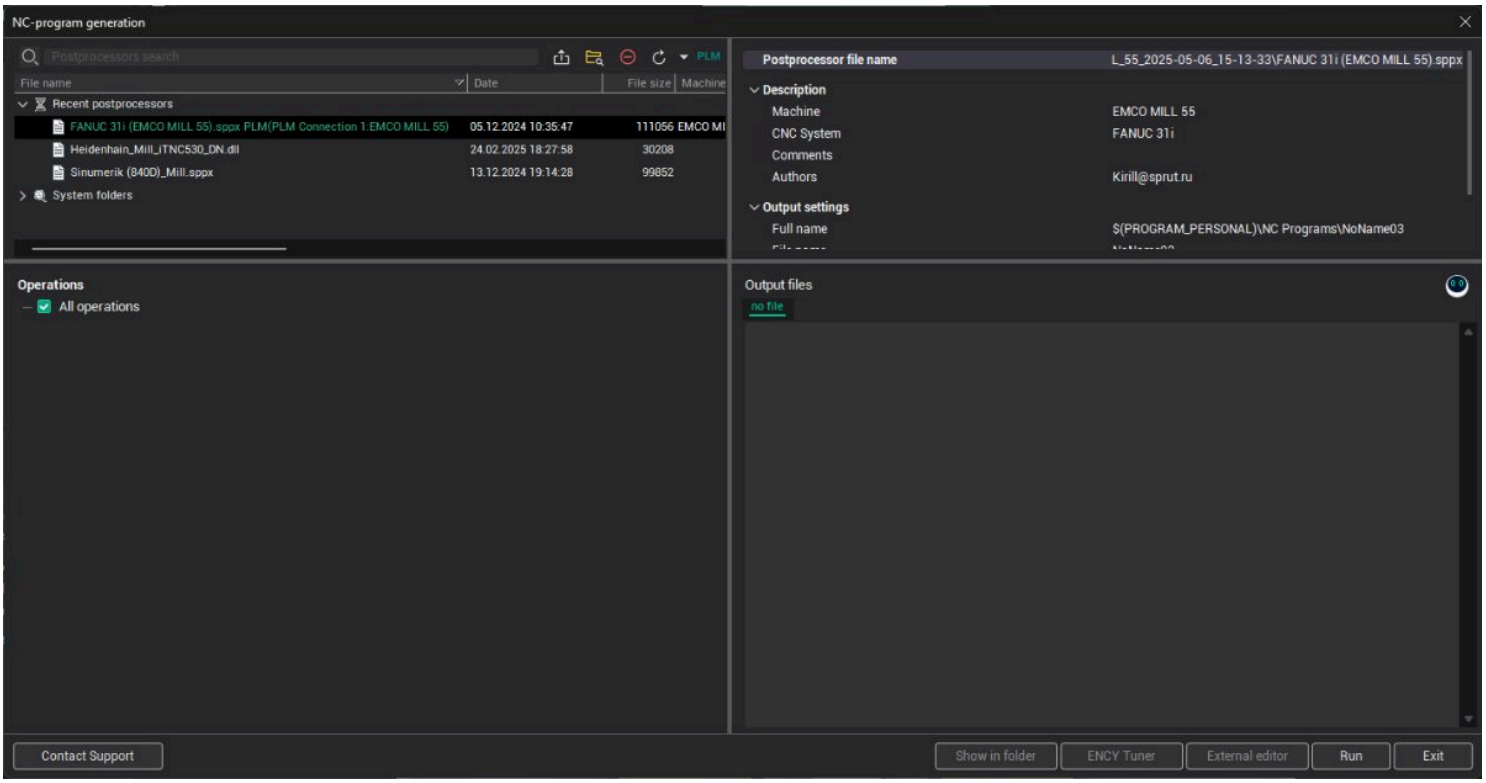


Loading a Postprocessor from an External System

To load a postprocessor from an external system, click PLM → Load postprocessor from PLM→ *[connection name]*. A window will appear displaying a tree of available objects from the external system.



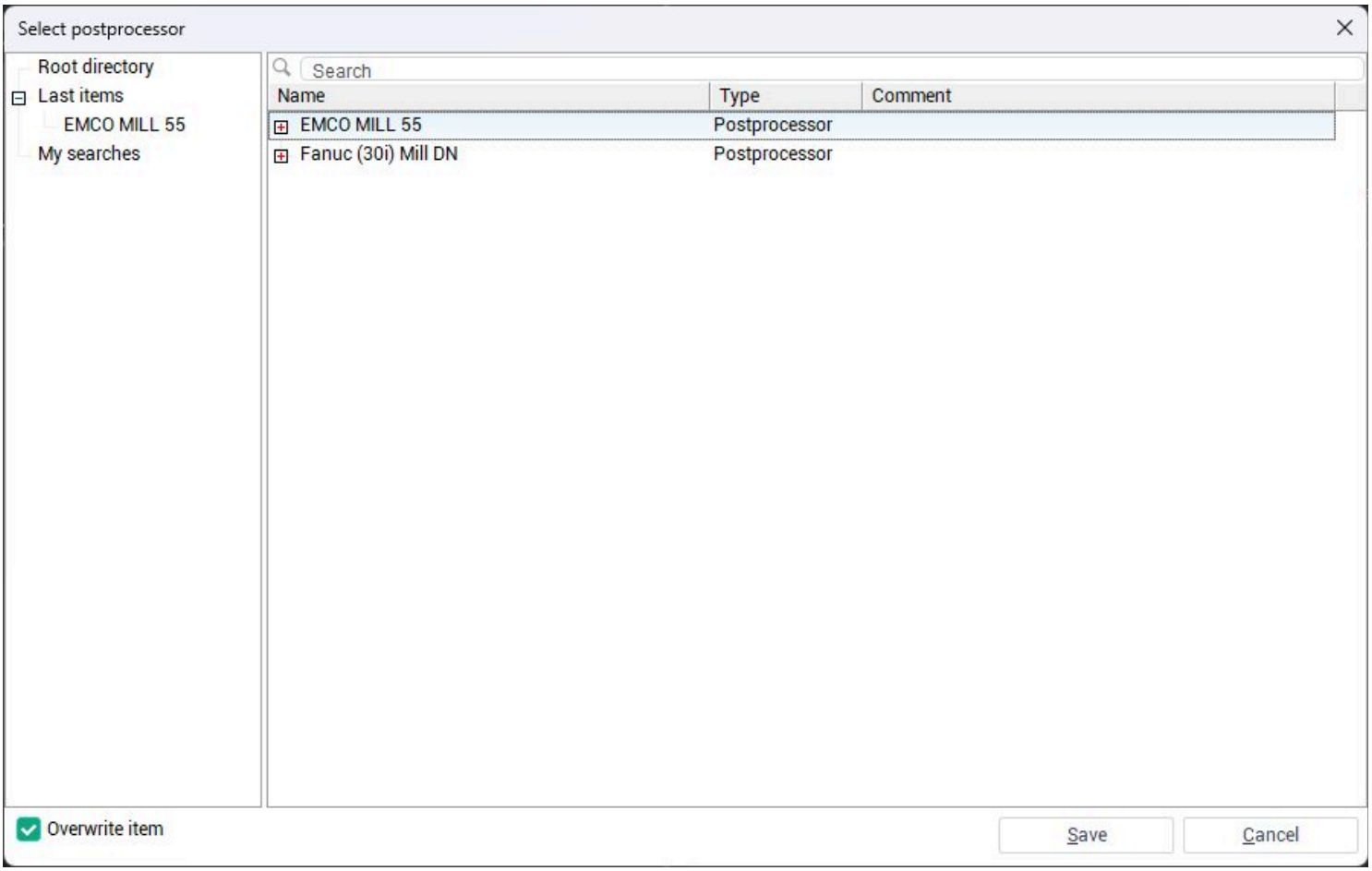
Locate the required postprocessor by navigating the object tree or using the name search, then click Open. The selected postprocessor will be loaded and displayed in the Recent postprocessors list with a name that reflects its origin from the external system.



To load a postprocessor associated with a specific machine in the external system, click PLM → Load postprocessor from machine in PLM → *[connection name]*. In the opened window, browse to the machine that contains the postprocessor, then select it and click Open to load it into the application.

Saving a Postprocessor to an External System

To save a machine to an external system, select the postprocessor you want to save, then click PLM → Save postprocessor to PLM → *[connection name]*.



A window will open displaying the object tree from the external system. Select the destination element where the postprocessor should be saved. To overwrite existing files, enable the Overwrite item checkbox, then click Save.

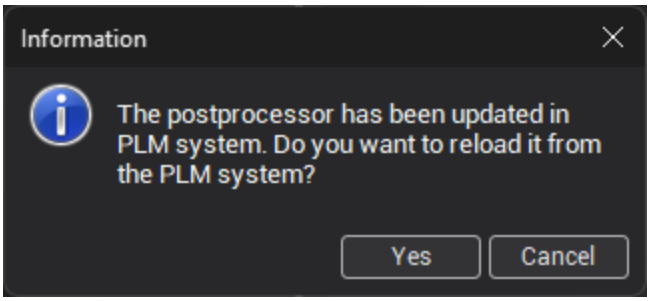
To save both the compiled postprocessor and its source files, select PLM → Save postprocessor with source files to PLM → *[connection name]*. This option is available only if the postprocessor has a .dll extension and associated source files. The source files will be saved as an archive along with the compiled file.

To associate a postprocessor with a specific machine in the external system, click PLM → Save postprocessor to machine in PLM → *[connection name]*.

Then, follow the same steps as in the standard saving process — select the destination location, enable Overwrite item if necessary, and click Save.

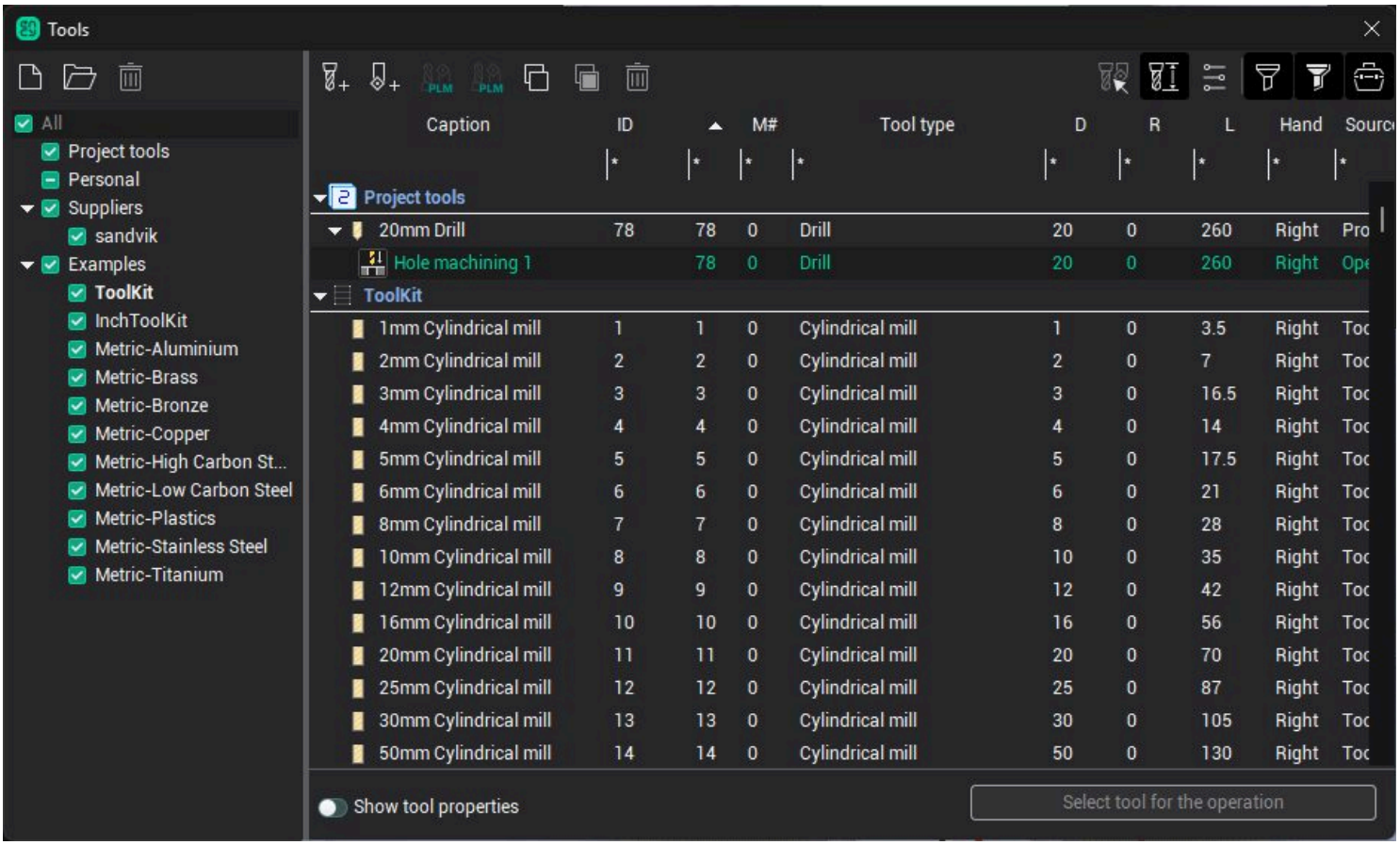
If a postprocessor has been updated in the external system since it was last loaded, an informational dialog box will appear when you attempt to run it. The dialog notifies you that a newer version is available. You can choose one of the following actions:

- Click Yes to download and apply the updated version of the postprocessor.
- Click Cancel to continue using the currently loaded version without updating.


Information
The postprocessor has been updated in PLM system. Do you want to reload it from the PLM system?
Yes
Cancel

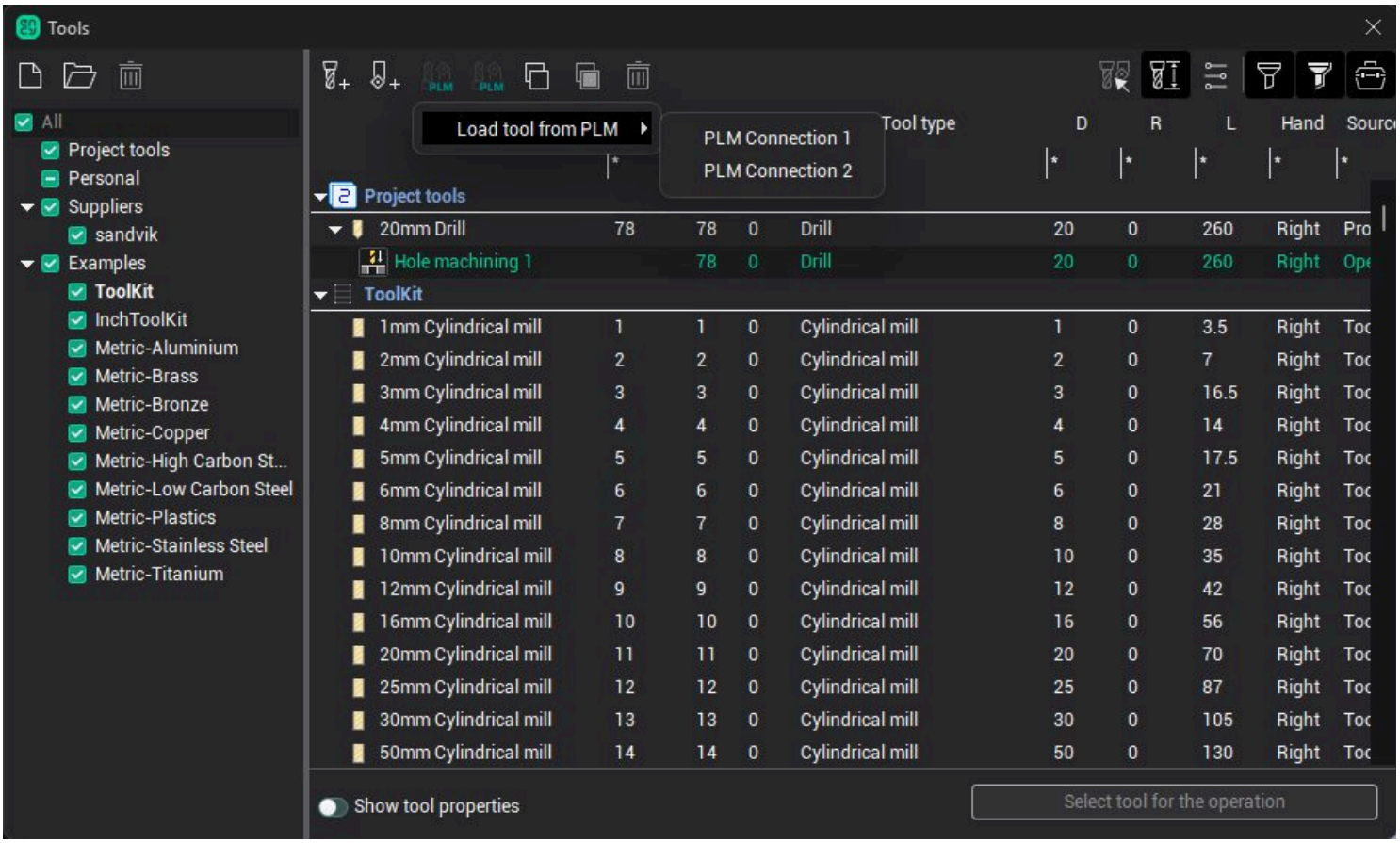
Loading and Saving Tools to an External System

To manage tools with an external system, open the Tools window. In the upper panel, you will find the Load tool from PLM and Save tool to PLM buttons.

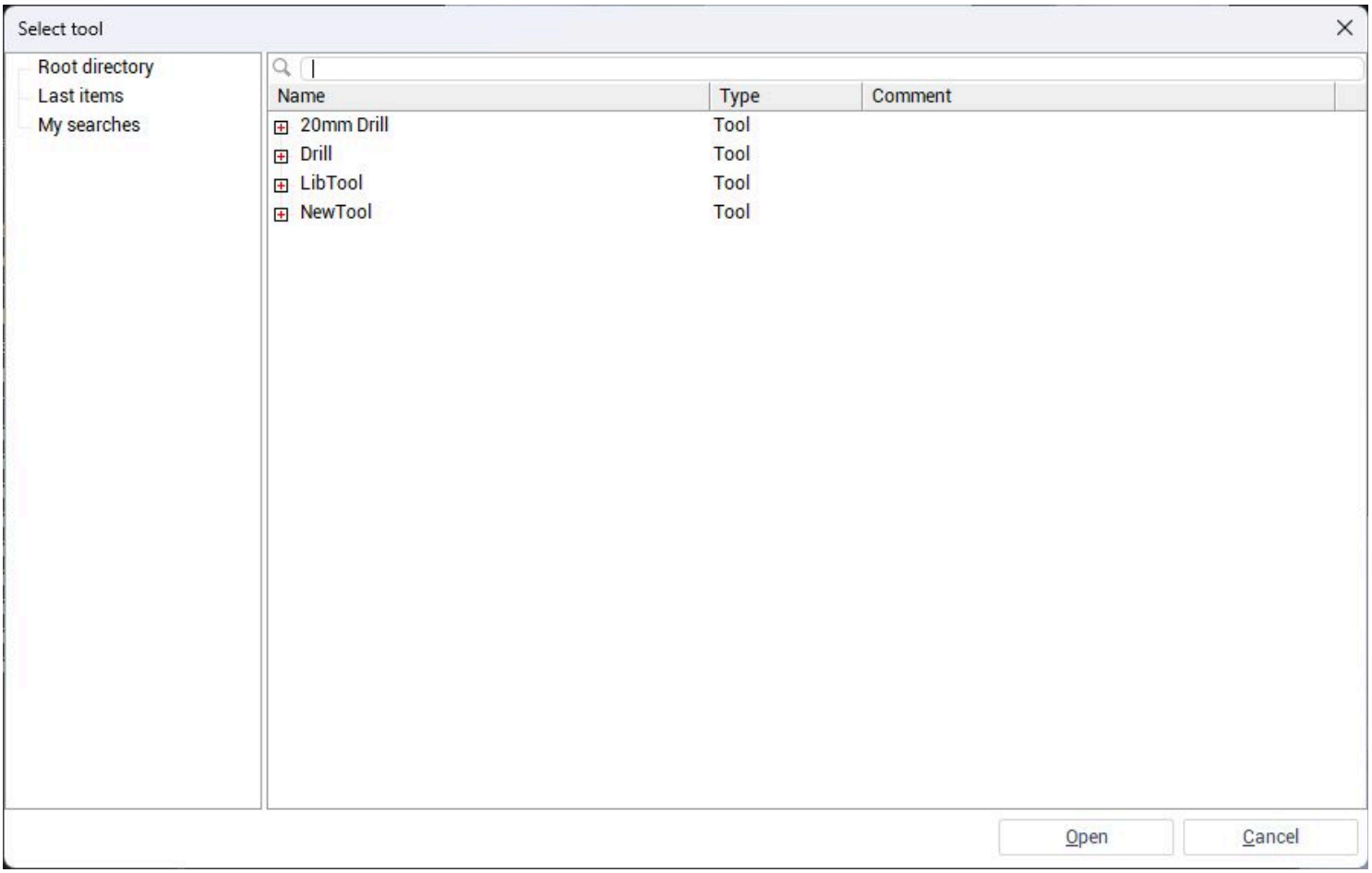


Loading Tools from an External System

Click Load tool from PLM, then choose the required connection.

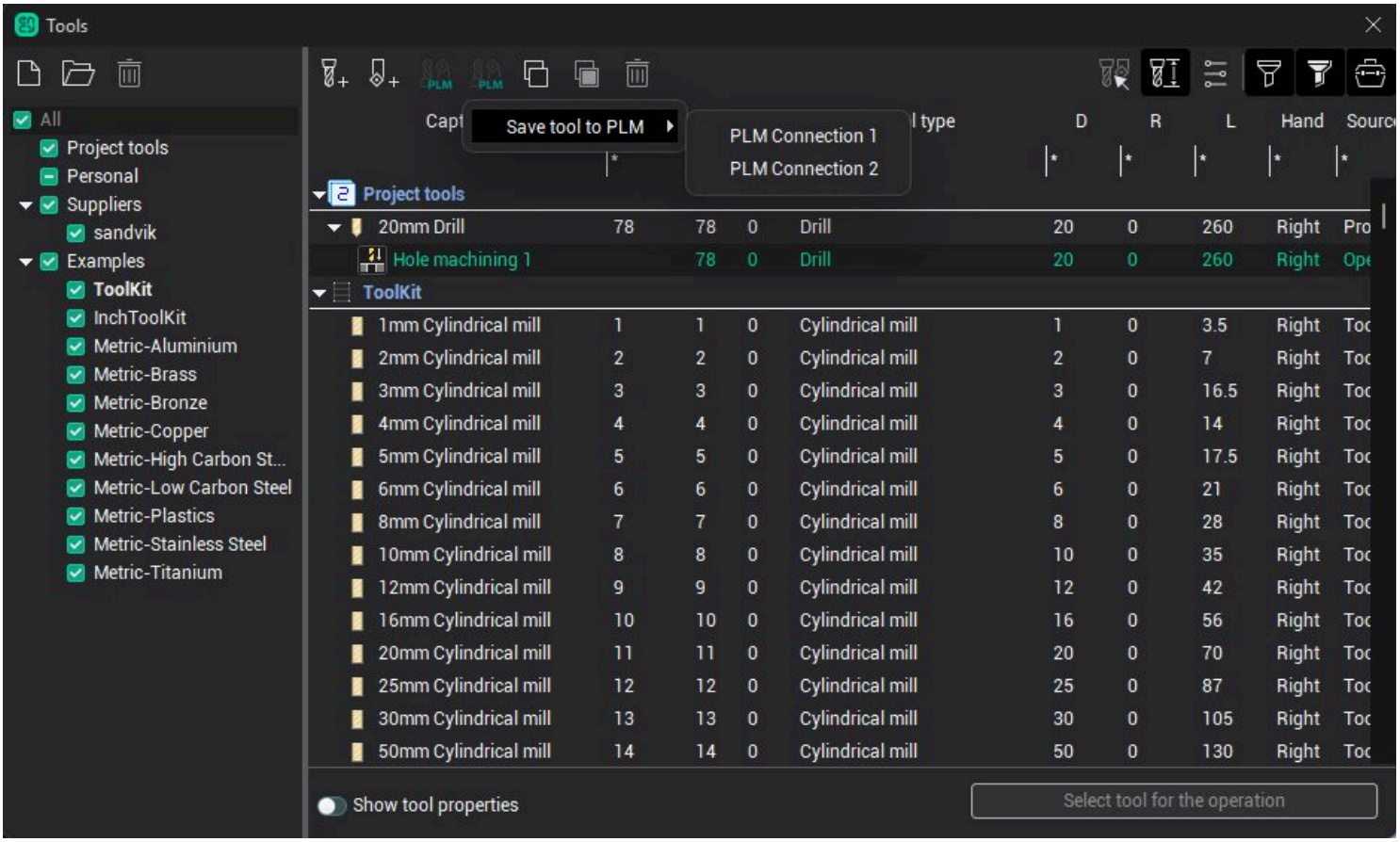


A window will appear showing a tree of available objects. Select the tool set you want to load and click Open. The tools will be imported and displayed in the Tools window.

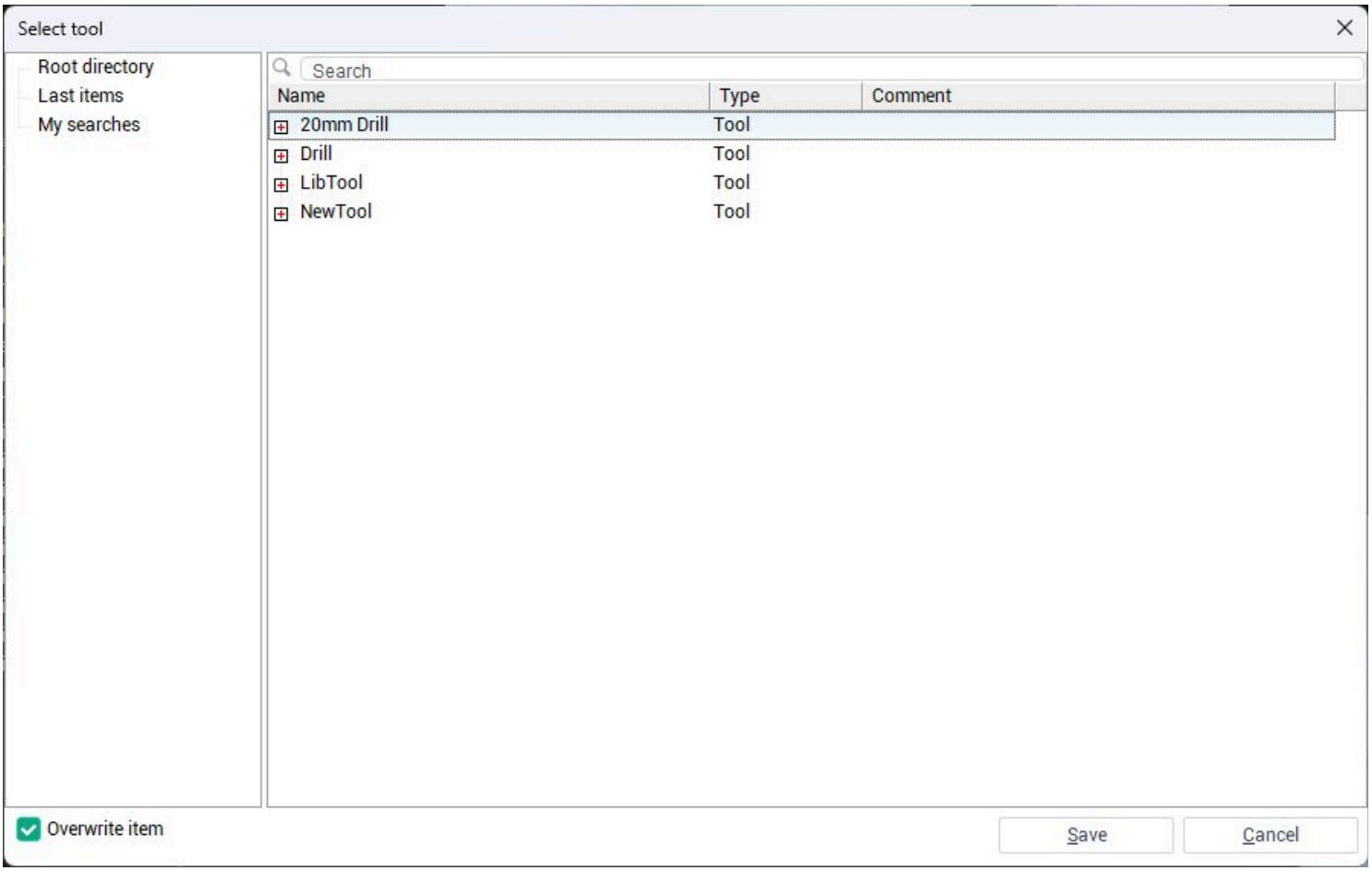


Saving Tools to an External System

Select a tool, click Save tool to PLM, select the desired connection, and choose the destination in the external system's structure.



To overwrite existing files, enable the Overwrite item checkbox. Then click Save to complete the operation.


Select tool
Root directory
Last items
My searches
Search
Name
Type
Comment
20mm Drill
Tool
Drill
Tool
LibTool
Tool
NewTool
Tool
Overwrite item
Save
Cancel

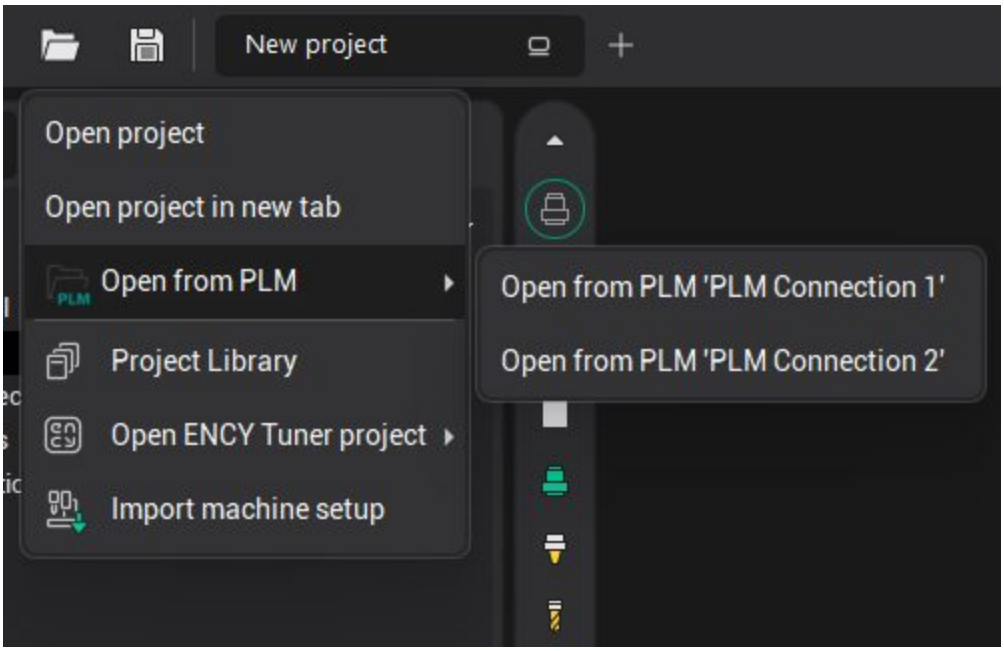
Opening and Saving Projects to an External System

To manage projects with an external system, use the Open Project and Save Project buttons in the main window toolbar. When hovering over these buttons, a drop-down menu appears with additional options, including PLM-related option. Selecting this item reveals a submenu with all available PLM connections.

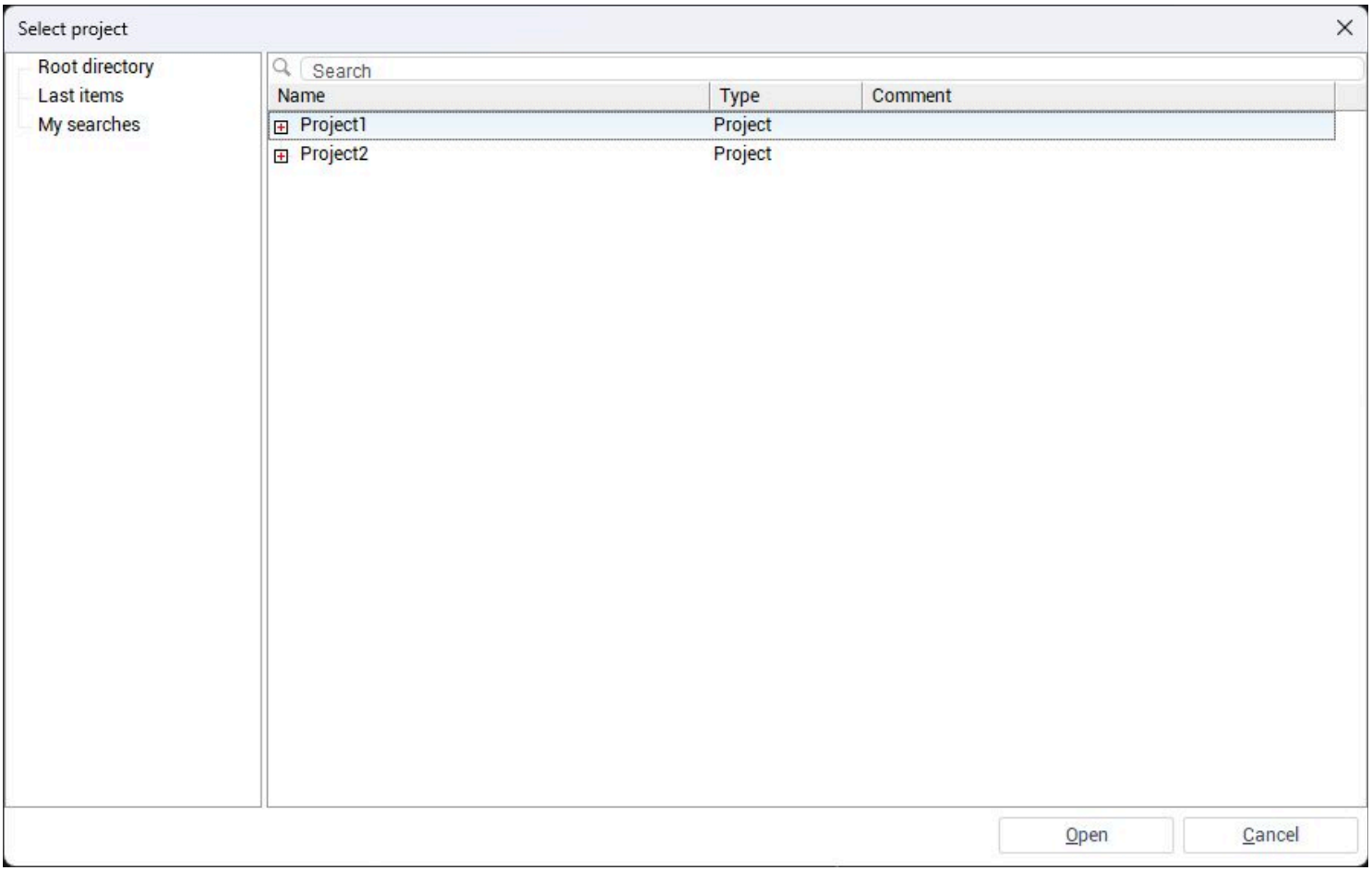
Note: The PLM-related option appears only if at least one PLM connection is configured.

Opening a Project from an External System

Hover over the Open Project button, select Open from PLM → Open from PLM *[connection name]*.

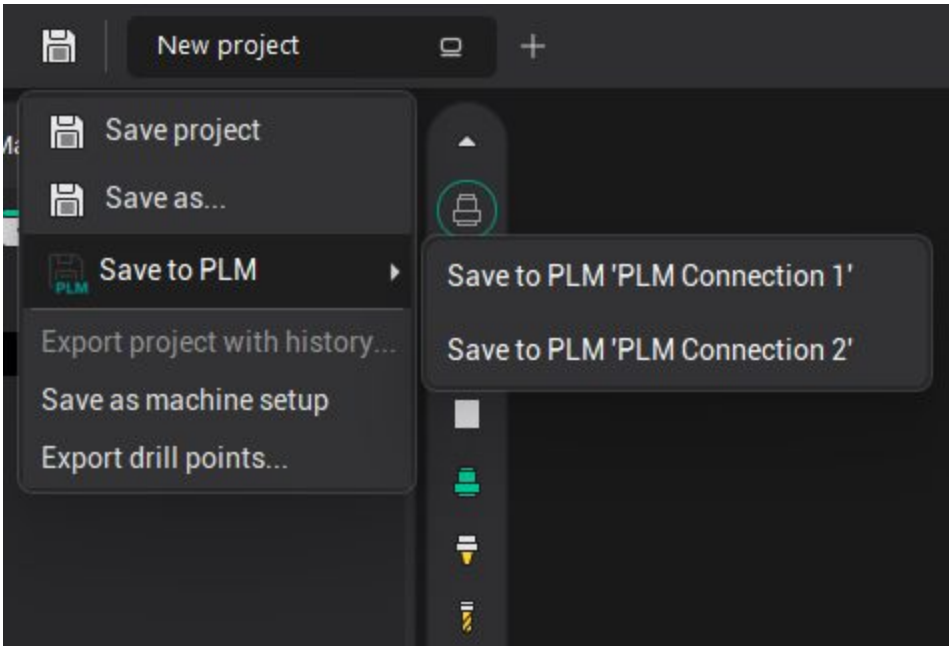


A window will open displaying the object tree from the external system. Locate and select the desired project, then click Open. The project will be opened.

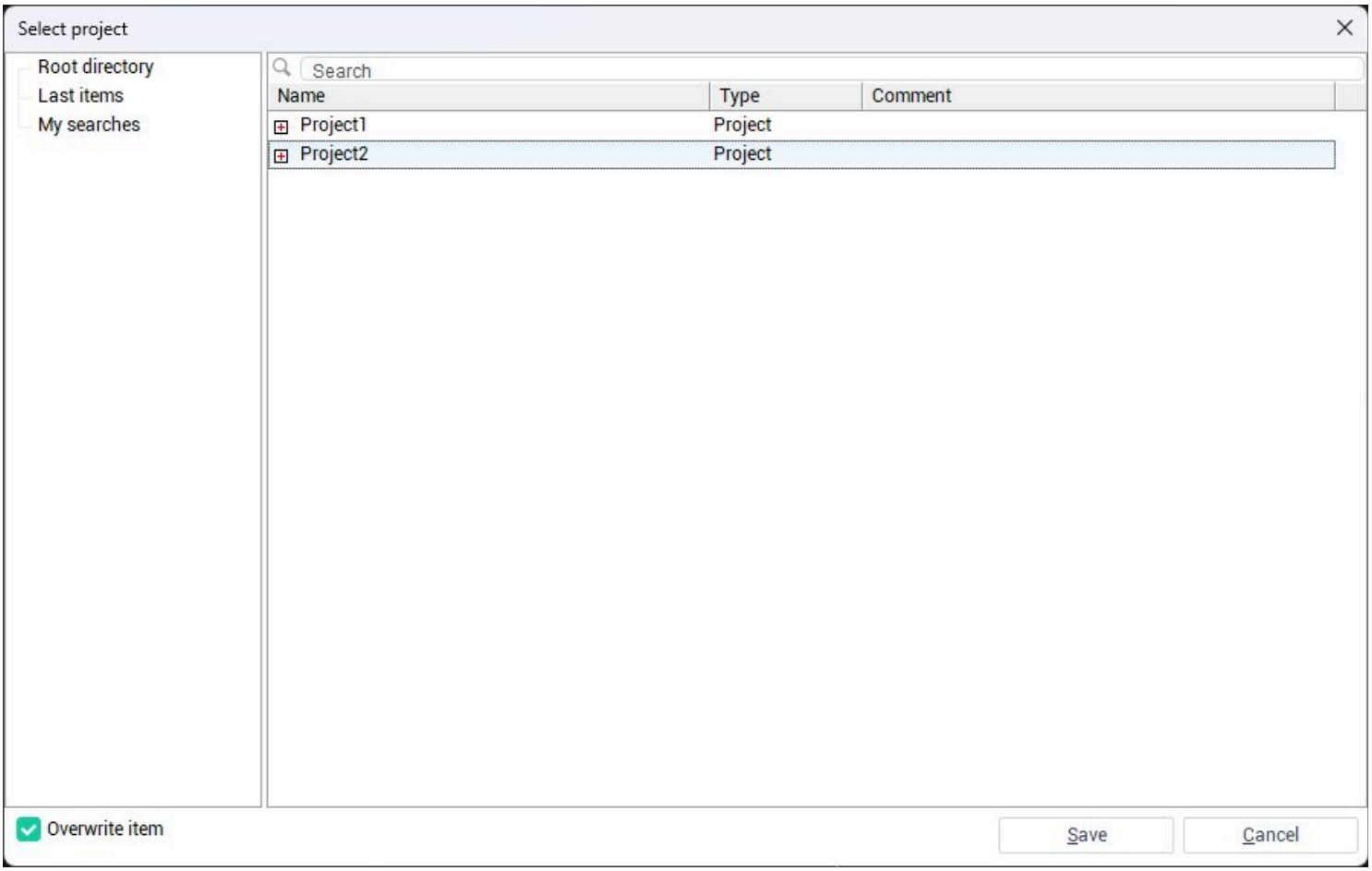


Saving a Project to an External System

Hover over the Save Project button, select Save Project to PLM → Save Project to PLM *[connection name]*.



A window will open displaying the object tree from the external system. Choose the location to save the project, optionally enable the Overwrite item checkbox, and click Save.


Select project
Root directory
Last items
My searches
Search
Name
Type
Comment
Project1
Project
Project2
Project
Overwrite item
Save
Cancel

Extensions of the CAM system

Extensions are auxiliary, disconnectable software modules that can extend the CAM system functionality and perform actions not included in the basic CAM module. Extensions can be developed by both CAM programmers and users themselves.

Extensions typically consist of several files, which can have extensions such as *.dll, *.json, *.exe, *.xml, and others. The main file through which CAM system identifies an extension is a text description file with the *.json extension.

The entire set of extension files can be packaged into a zip archive with the *.dext extension.

Additional information on developing extensions can be found in the official developer API documentation (docs.encycam.com/CAMAPI/3/en/).

Examples of extensions can be found in the corresponding source code repository for API usage examples (https://github.com/EncySoftware/cam-api-examples/tree/v3/main/docs).

CAM system extension template: project creation, build, and publication in the Store - https://github.com/EncySoftware/ency-extension-template.

AI development tools: creating and publishing extensions with an AI assistant - https://github.com/EncySoftware/ency-extension-mcp.

Installing extensions from a file

An extension that is not in the store can be installed from a file: a `.dext` or `.nupkg` package provided by the extension author.

1. On the Home page, open the Extension Store tab and go to the Manage page.
2. Click Install from file⋯ and select the file. You can also drag the file onto the tab.
3. Wait for the installation message. If it asks you to restart the application, click Restart the application.

Besides packages, Install from file⋯ accepts an extension description file `*.json`: the extension is registered in the folder where its files are located.

On the Manage page, you can also enable, disable and remove installed extensions. Details

You can also install an extension using the command line.

Installing extensions using the command line

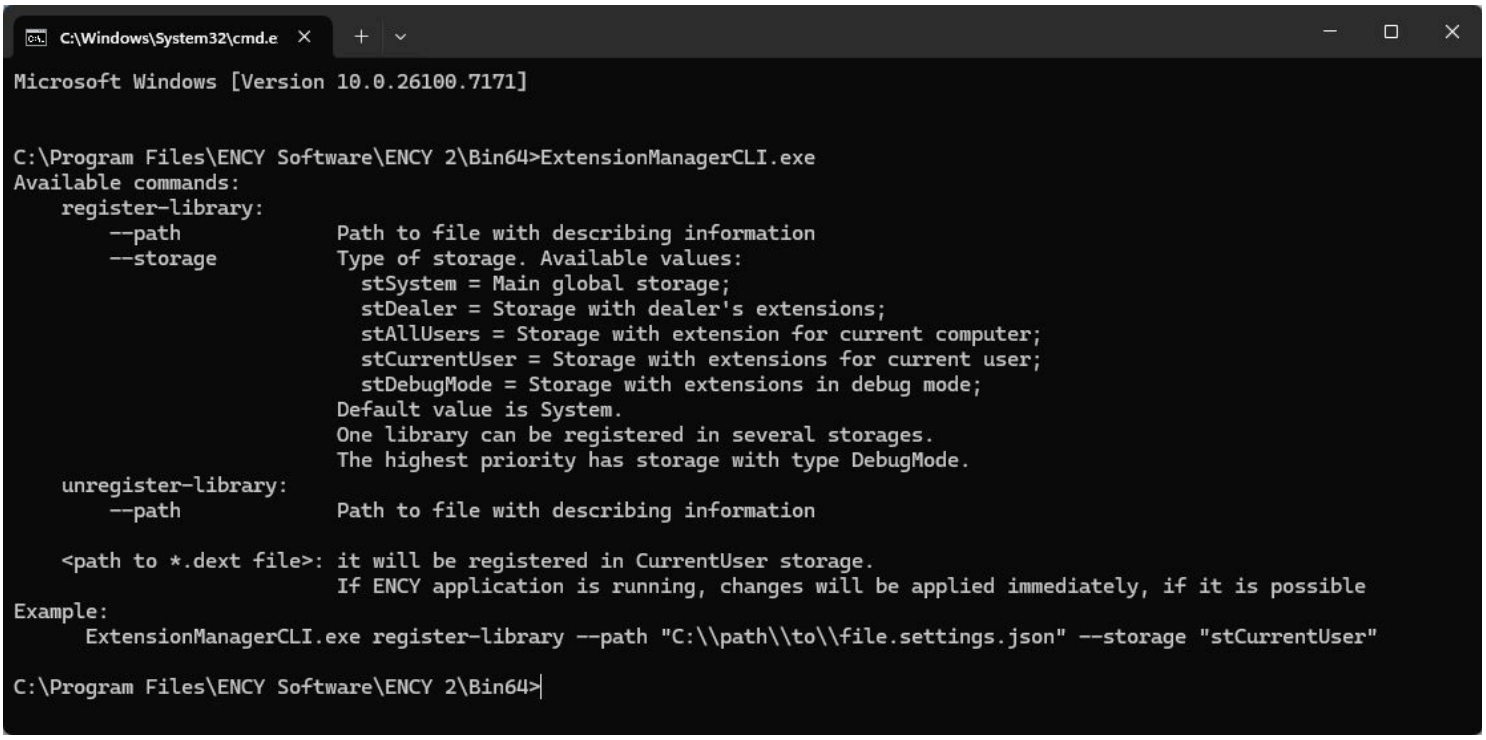
Extensions can be installed using a command line utility. This is convenient for batch installations, when you need to quickly repeat the process on multiple computers.

To install extensions using the command line, follow these steps:

- Find the ExtensionManagerCLI.exe utility in the CAM system installation folder.

Note: The ExtensionManagerCLI.exe utility is used to register and remove CAM system extensions from the command line.

- Run the command line from the CAM system installation folder. Running the utility without parameters will do nothing, it will simply display help information on valid command line arguments.



- Run the command:

ExtensionManagerCLI.exe register-library --path "ExtensionFolder\ExtensionSettings.json" --storage stAllUsers

Instead of "ExtensionFolder\ExtensionSettings.json", specify the path to the extension's JSON file.

Instead of the --path parameter, you can also specify the path to the extension's dext file.

Using the --storage parameter (possible values: stAllUsers, stCurrentUser), you can specify whether the extension will be available to all users or only the current user.

Below is an example of the command execution.



Note: Depending on the extension type, changes may only be applied after restarting the CAM system.

Macro



Application area:

A macro is intended for recording a sequence of actions performed on a project and replaying it on demand. A sequence recorded once — creating operations, assigning geometry, selecting items in the model tree, editing properties — is converted by the system into a small, runnable extension that reproduces those actions whenever required. A macro is also the fastest way to create a custom extension: a working source code is generated from the recorded macro and serves as a template for further development.

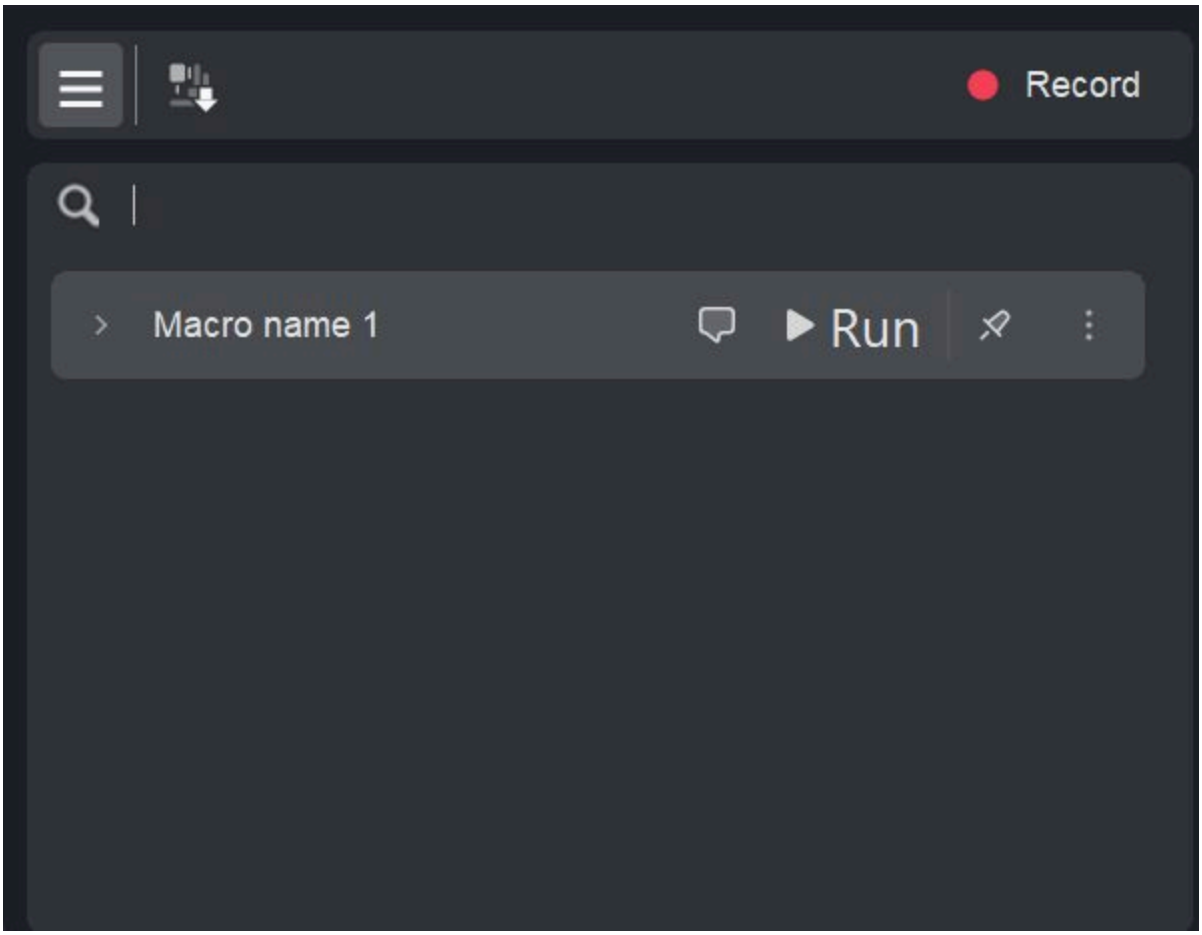
Macros are launched from the Utilities button on the main panel.

Macros are used wherever the same work is performed repeatedly:

- applying a standard sequence of operations to similar parts;
- re-creating a typical machine, workpiece, and strategy setup at the start of a project;
- assigning the same geometry — faces, zones, curves, holes — in a uniform manner;
- sharing a repeatable procedure with other members of the team.

The Macro panel:

The Macro panel contains the saved macros and the tools for recording new ones. The panel is divided into three zones (top to bottom): a toolbar with the recording controls, a Search field, and the list of saved macros.



Toolbar buttons.

List toggle (≡). Shows or hides the list of saved macros.

Project state. Determines which parts of the current project — machine setup, workpiece and stock, strategy parameters — are captured by the macro at the start of recording.

Record. Starts recording a new macro; pressing it again stops the recording. Details

Search field. It allows you to find a macro in the list.

The macro list.

Name. The macro name.

Comment icon. Displays the macro description on hover.

Run. Plays the macro. Details

Pin. Keeps the macro at the top of the list.

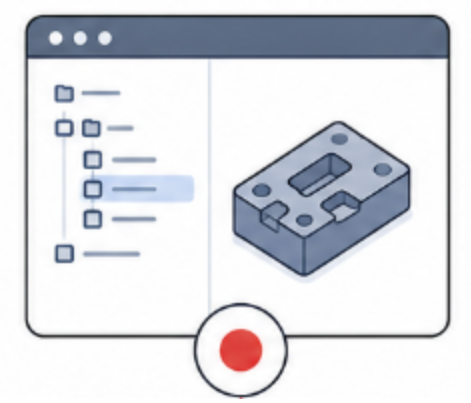
"⋯" menu. Per-macro actions:

- Open in editor. Opens the generated source code of the macro for reading or manual extension. Details
- Edit. Re-opens the macro steps with the full set of editing controls; saving the changes rebuilds the macro. Details
- Export.
- Pin to top. Pins the macro to the top of the list, above the separator line; selecting it again unpins the macro.
- Delete. Removes the macro and its files. Unavailable while any macro is running.

→. Reveals the individual steps of the macro.

> ### Macro storage
>
> Each macro is placed in its own folder inside the personal CAM system folder, under `macros`. The folder contains the generated source code and a human-readable list of the macro steps, which allows the macro to be copied together with that folder.

Recording a macro



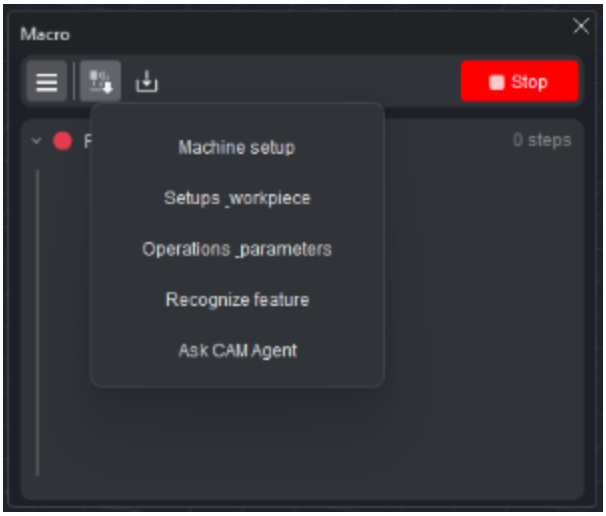
Application area:

Starts recording a new macro; pressing it again stops the recording. A recording is bound to the project active at the moment it is started. When the project is switched during recording, the recording is terminated automatically. Recording and editing an existing macro are mutually exclusive: a new recording cannot be started while a macro is open for editing, and vice versa.

Capturing the project state

A macro may begin by re-creating the current project setup, ensuring that a later run reproduces the same starting conditions. The set of data to be captured is determined before the Record button is pressed.

1. Open the Project state drop-down on the toolbar. 2. Select the necessary items: Machine setup, Setups workpiece, Operations parameters, Recognize feature, Ask CAM Agent.



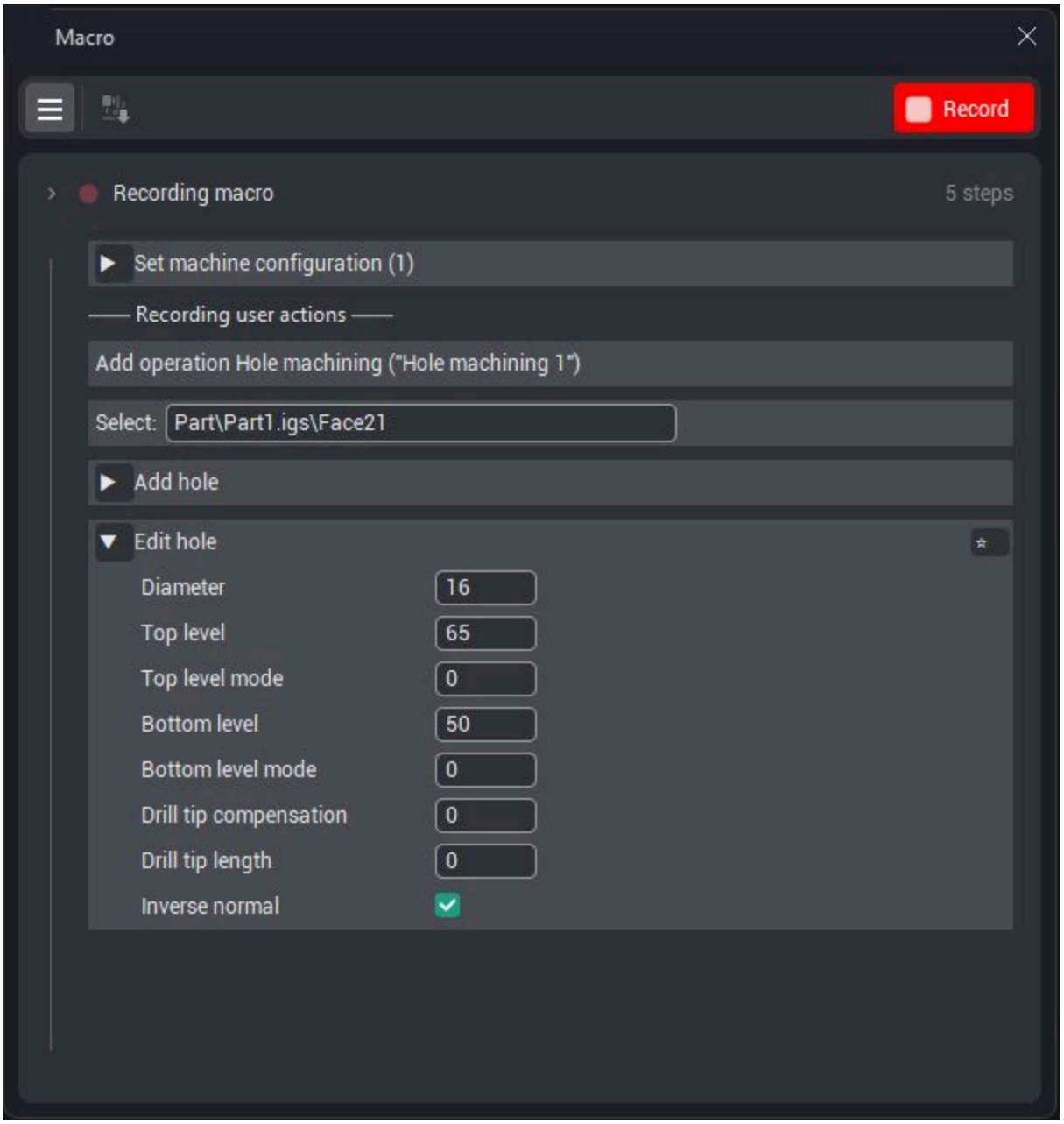
Recording the actions

1. Press Record. The button takes the form of a stop icon, the panel header displays Recording macro, and the step count is set to 0 steps. 2. Perform the required actions in the project. Each recognized

action is added to the list as a step, and the step count updates in real time. The following actions are recorded, in particular:

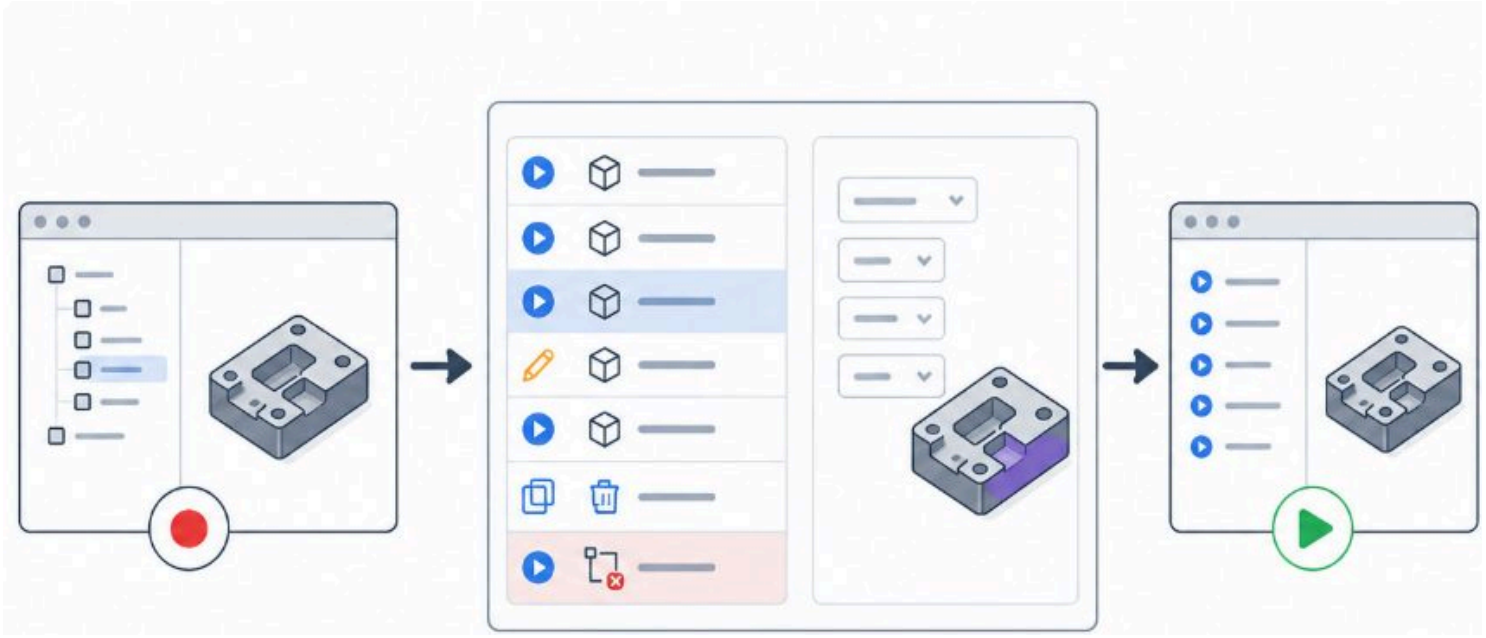
- creating an operation;
- selecting an operation in the operations tree;
- setting up the workpiece — adding and editing workpiece primitives (box, cylinder, casting, prism, and so on) and their stock;
- adding fixtures — chucks, vises, clamps, supports — and editing their stock, color, and name;
- assigning geometry to an operation — faces, job and restricted zones, curves, holes, drive faces, areas;
- selecting items in the 3D model; editing item properties.

3. Press the same button, now in the form of a Stop button, to finish the recording. The header takes the value Recording complete.



When the recording is stopped, the system fills in a suggested macro name (for example, Macro name 1) and opens the Save panel. Before saving, the recorded steps may be reviewed and refined. Details

Editing steps



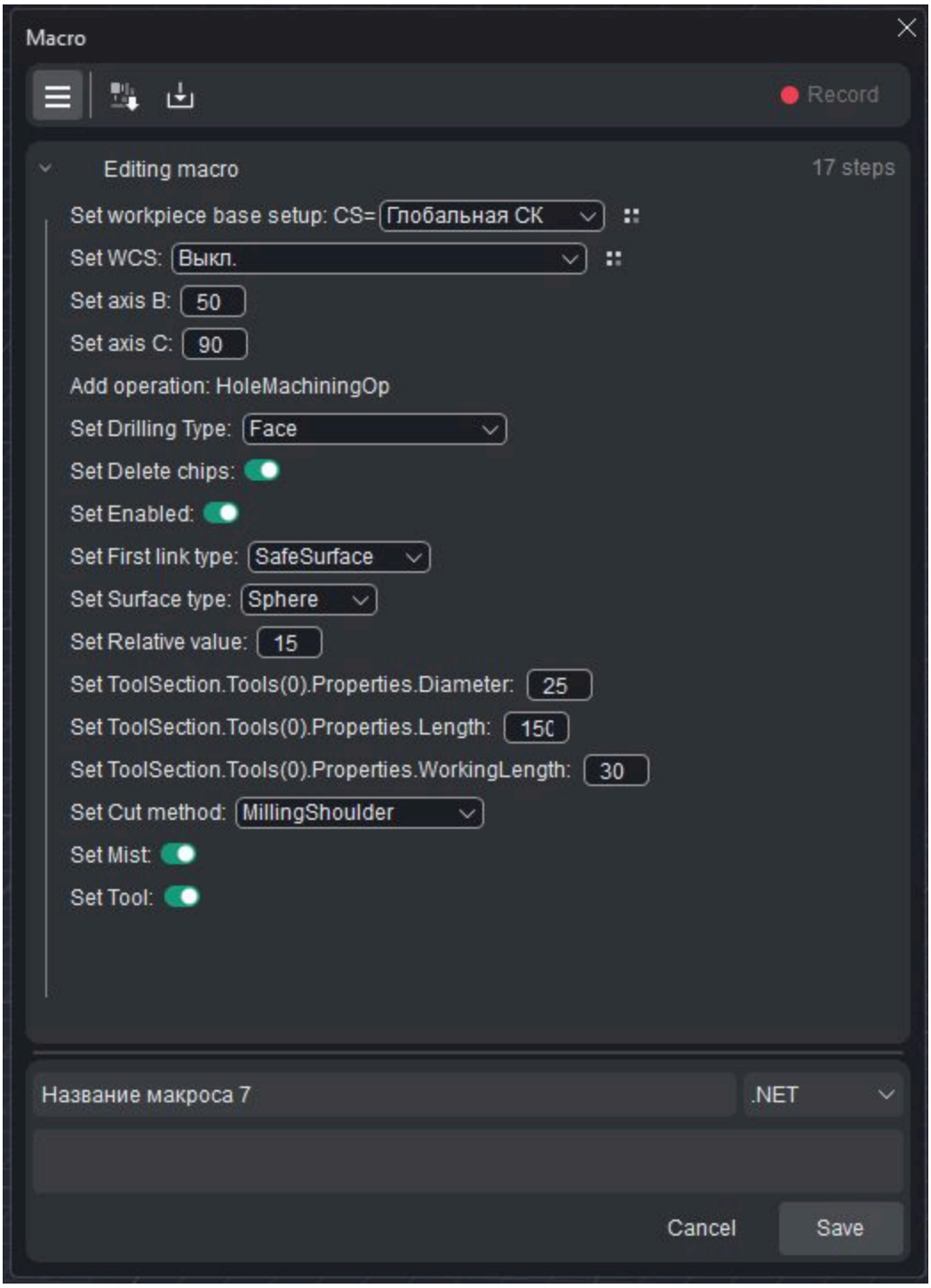
Application area:

Most step-editing operations are available during the recording of a macro or when it is opened for editing. A saved macro may also be adjusted for a single run.

The step list

Each recorded action is represented by a step row containing:

- Status dot. Located on the left of the row; also used to set a breakpoint
- Caption. Describes the purpose of the step.
- Editable parameters. Where present, one or more drop-downs or text fields.

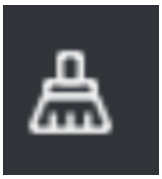


Changing a parameter value

For a drop-down, the required value is selected; for a text field, the value is typed. The new value is retained with the step.

Marking a field as an input

Recording captures the values actually used. To convert a fixed value into an input supplied on each run of the macro, its field is marked as required:

1. With the cursor over the step, press the  next to the field.
2. The field becomes an input (t*he field will be highlighted*): the macro cannot be run until a value is supplied.

This mechanism is applied to values that change from part to part — a diameter, a level, an allowance.

Re-targeting a geometry item

Steps that reference an item in the model — a face, curve, hole, and so on — display that item as a field that allows re-selection:

1. Click the field (t*he field will be highlighted*).
2. Select the new item in the 3D model.

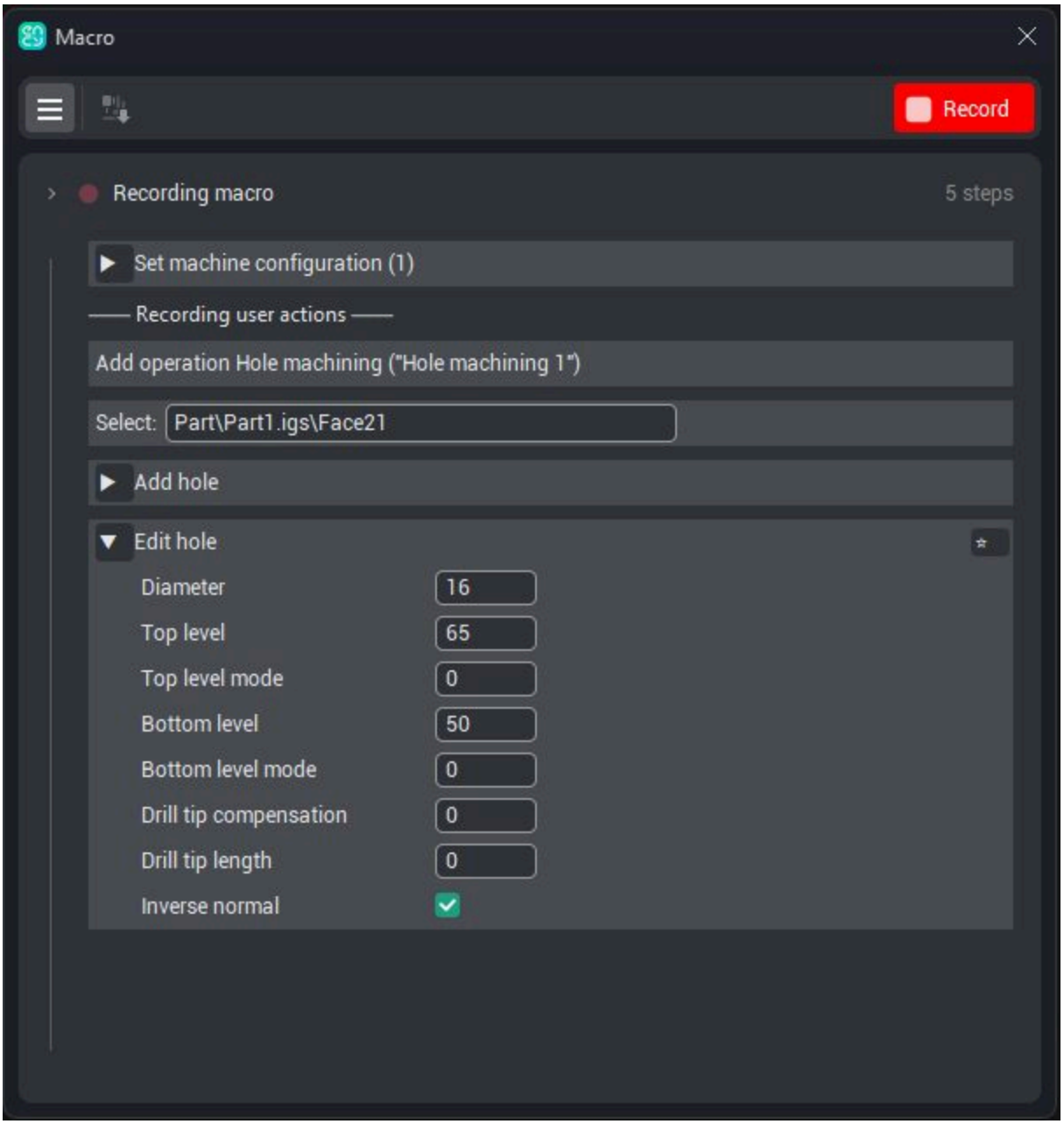
The step updates to the selected item. The mechanism operates both during editing and for a saved macro before a run, which allows a single macro to be targeted at different geometry.

Removing a step

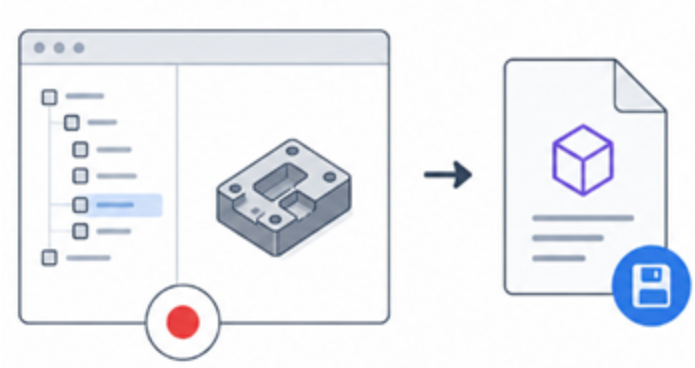
During recording or editing, with the cursor over the step, pressing the × at the end of the row removes the step from the macro.

Grouped steps

Related steps are folded under a group header that displays the group name — for example, Edit hole (5). Clicking the chevron on the header expands or collapses the group. Grouping simplifies the review of a lengthy macro.


Macro
Record
Recording macro
5 steps
Set machine configuration (1)
Recording user actions
Add operation Hole machining ("Hole machining 1")
Select:
Part\Part1.igs\Face21
Add hole
Edit hole
Diameter
16
Top level
65
Top level mode
0
Bottom level
50
Bottom level mode
0
Drill tip compensation
0
Drill tip length
0
Inverse normal

Saving a macro



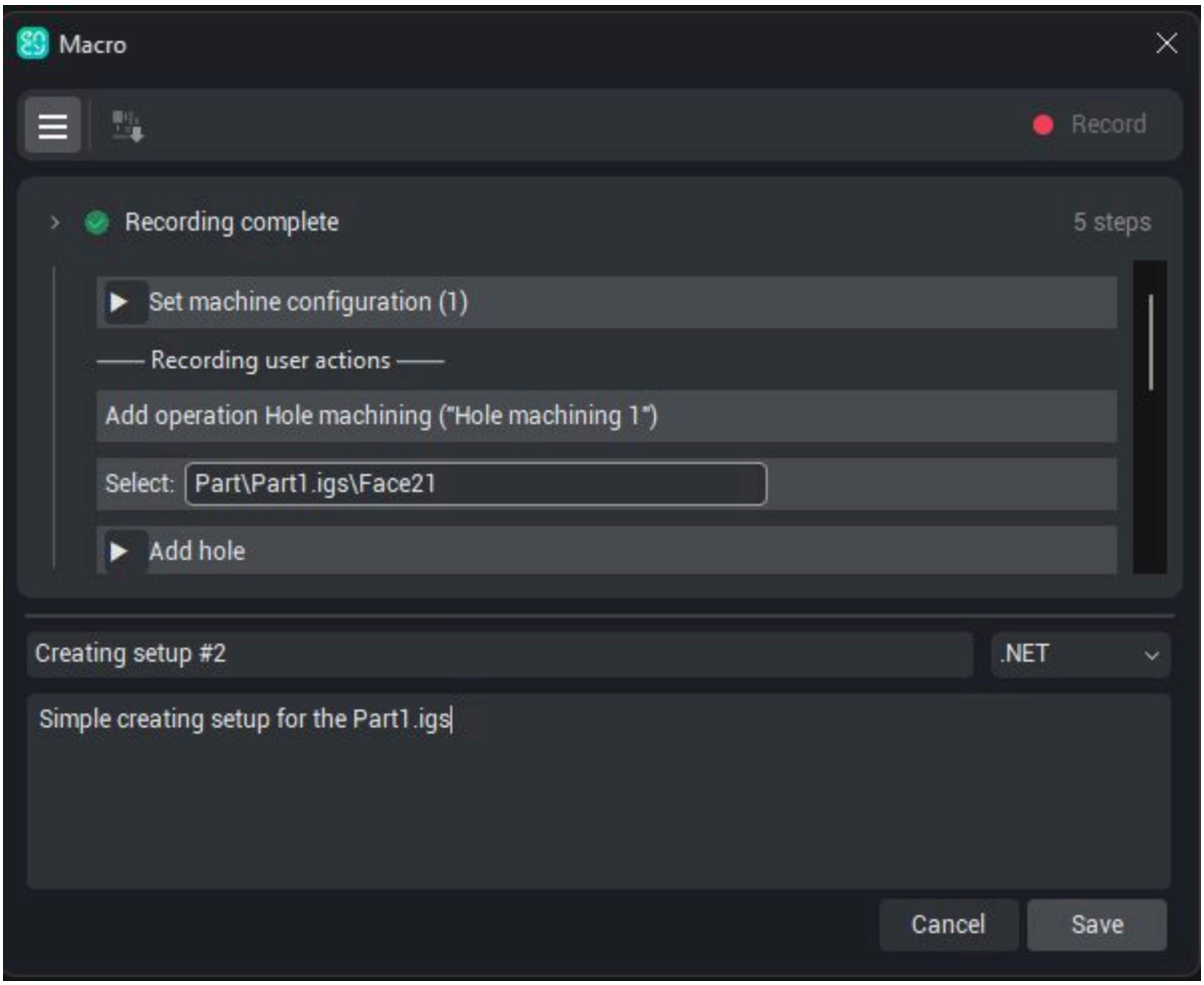
Application area:

Starting a run, supplying required inputs, monitoring progress, pausing, stopping at breakpoints, and overriding values for a single execution instance.

The Save panel

When you stop recording (or finish editing), the Save panel appears with three fields:

- Name — pre-filled with a suggestion such as *Macro name 1*. Change it to something you will recognize in the list.
- Language — the language the macro is built in: .NET or Spr.
- Comment —an optional description, displayed on hover over the comment icon in the macro row. The comment becomes the macro's description. It is shown when you hover the comment icon on the macro's row in the list, so a short note about what the macro does — and on what kind of part — pays off later.



Choosing the language

- .NET — the macro is built as a.NET extension.
- Spr — the macro is built in Spr, ENCY's built-in scripting language.

Both options produce a macro run in the same manner. The choice of language is significant mainly when the macro source is subsequently opened for development Details

The build-environment check

When the Save panel is opened and on every change of language, a check is performed to confirm that all components required to build the selected language are present. If the required components are absent:

- The message Macro build environment is not ready is displayed, and the Save button becomes unavailable.
- Clicking the message and confirming the Try to fix it? prompt initiates the setup of the environment by the system.

A re-check is performed afterwards; once the environment is ready, the Save button becomes available again.

Saving or cancelling

Save. Builds the macro and adds it to the list, ready to run. Macros are stored in the personal ENCY folder, under `macros`.

Cancel. Discards the recording without saving.

Running a macro



Application area:

This chapter explains how to play a saved macro: starting a run, supplying any required inputs, following progress, pausing, stopping at breakpoints, and overriding values for a single run.

Running a macro

- Locate the macro in the list (for a lengthy list, Search is used).
- If necessary, expand the macro and adjust the values for the current run.
- Press Run. The macro plays its steps in order against the current project.

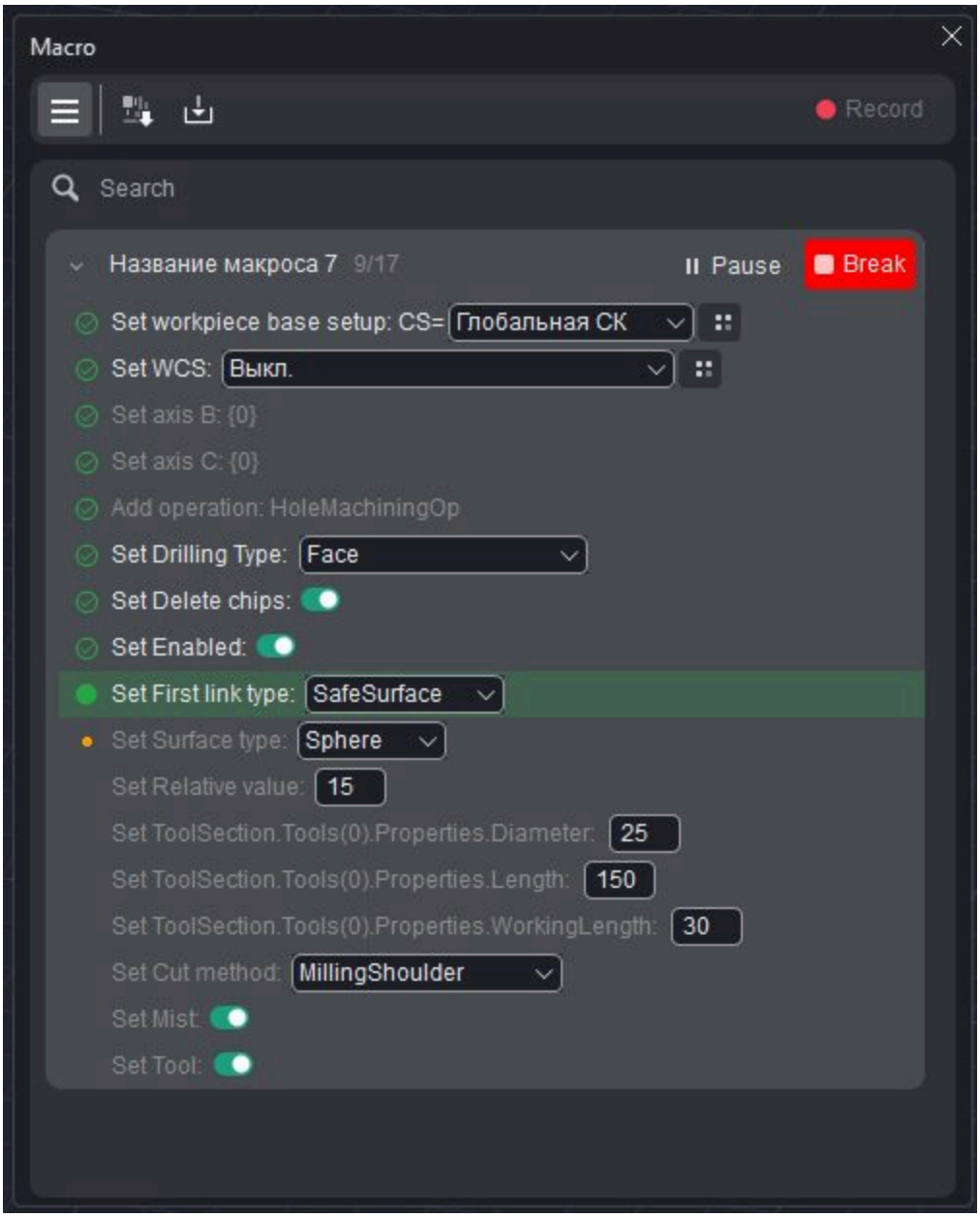
Required inputs before a run

If the macro has empty fields marked as inputs (required), the run is not performed. The macro is expanded automatically, and the first empty input field is outlined in red. Once the field is filled in — a value is typed or the geometry is re-selected — the Run button is pressed.

Following progress

During the run of a macro:

- The current step is marked with an green dot.
- Completed steps are marked with a green check; steps not yet reached are dimmed.
- A counter next to the macro name displays progress in the format current / total — for example, 9/17.



Pausing and continuing

During playback, the Run button changes function: pressing it places the run into the Pause state after the current step; in the paused state the button displays Continue to resume the run. You can also stop the macro by clicking the Break button.

Stopping at a breakpoint

To stop a macro at a particular step, a breakpoint is set on that step:

1. Click the status dot on the left of the step (clicking again clears the breakpoint). A breakpoint cannot be set on the step currently running.

2. When the breakpoint is reached, playback is paused, and the run button displays Continue.

3. After the result is inspected, pressing Continue resumes the run. Alternatively, you can Break execution.

Breakpoints are used for a step-by-step check of a macro on its first run.

Overriding values for one run

The behavior of a macro may be changed for a single run without editing the saved macro: the macro is expanded in the list, its fields are adjusted, after which Run is pressed. The supplied values apply to that run only. For a permanent change, the macro is edited.

> A macro cannot be deleted while it is running

Editing the code



Application area:

Each macro is a small extension generated by the system from the recorded steps. Recording covers the common cases, while the generated source code is a fully functional extension.

Opening it is appropriate for:

- going beyond the capabilities of recording — adding custom logic, loops, or conditions;
- studying the CAM system API by means of a concrete, runnable example;
- creating a new extension from a known-good template rather than an empty file.

.NET and Spr

The language is selected when the macro is saved and is retained by the macro. Details

.NET.The macro is generated in C# and built as a.NET extension against the CAM system API. Selected for development of the macro in C# or use of the wider .NET ecosystem.

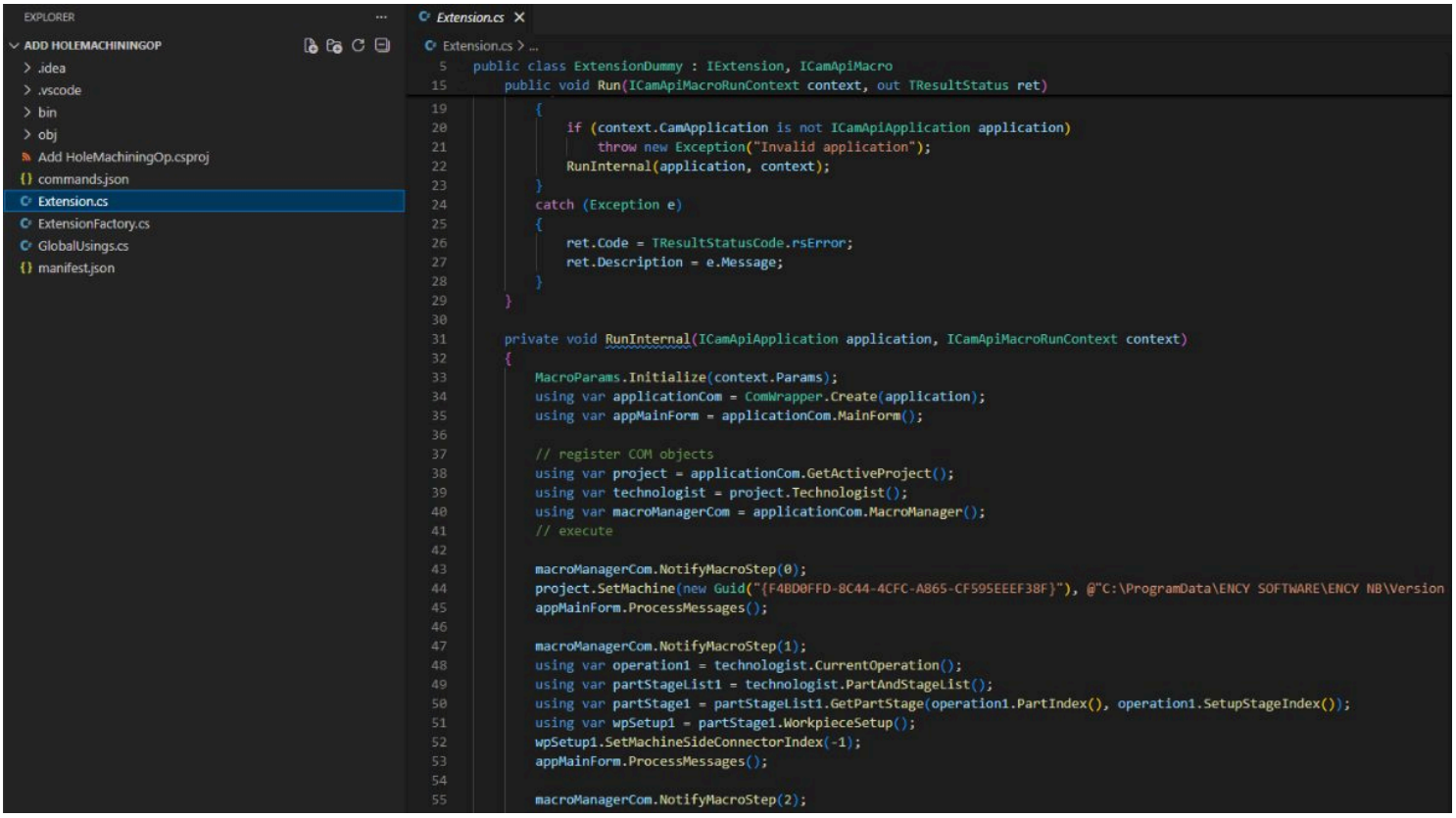
Spr. The macro is generated in Spr, the built-in scripting language of CAM system, and opens in the built-in Spr editor. Selected when work is performed in Spr.

Both options are run in the same manner from the macro list; the difference lies only in the source code being edited.

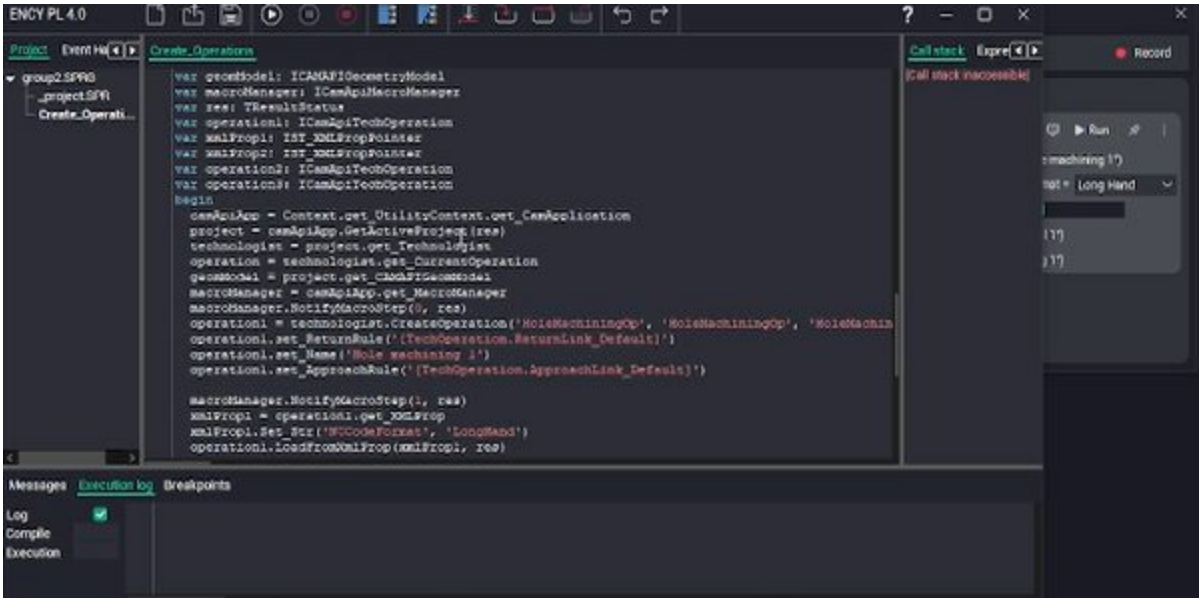
Opening a macro in an editor

1. In the macro list, open the "⋯" menu of the macro and select Open in editor.
2. The system opens the generated source code of the macro — the C# source for a .NET macro, or the `.spr` script in the Spr editor.
3. Make the changes and save.

The generated source code of a .NET macro — the C# `Extension.cs` file together with the project files (`commands.json`, `manifest.json`, and the `.csproj`):



The generated code of a Spr macro — the script opened in the Spr editor:



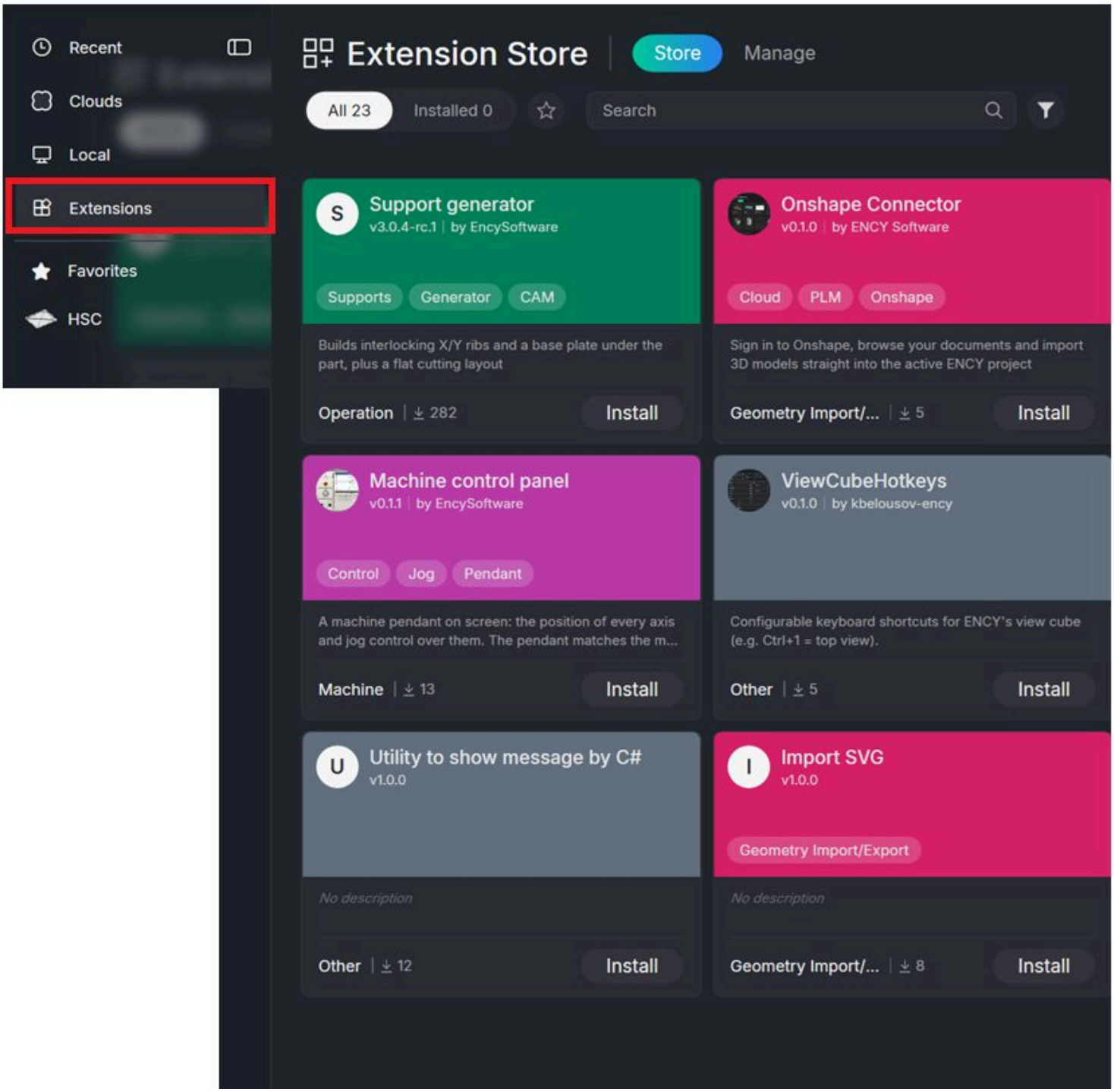
How edits take effect

- A Spr macro is run directly from its script, so saving the edited script is sufficient.
- A .NET macro is run from a compiled file, so C# edits take effect after the macro is rebuilt. The build-environment check analogous to the one described in applies. Details

Source code location

The macro source code and its human-readable step list are placed together in the macro folder, inside the personal CAM system folder under `macros`. Keeping the folder as a whole ensures that the macro and its source code remain synchronized.

Extension Store



Application area:

The Extension Store is the place where optional add-ons for the CAM system are discovered, installed, updated, and removed. It connects a web catalog with the CAM application: the web storefront lists extensions and can start an installation, while the in-application tab performs the actual download, registration, and removal on the local machine.

The store is used for the following scenarios:

- Browsing extensions by category, publisher, tag, or price.
- Installing free extensions directly into the CAM system.
- Starting a free 14-day trial for paid extensions.
- Updating installed extensions to newer versions.
- Removing extensions that are no longer needed.
- Managing extension sources and locally registered libraries.

> Note
>
> The Extension Store interface is displayed in English regardless of the language selected in the CAM system.

Ways to Work with Extensions in the CAM Ecosystem:

- Web storefront - https://apps.encycam.com/ (Browsing the catalog, viewing extension details, publishing extensions, and starting installation from a browser.)
- In-application tab - Home page (Details) → Extension Store. (Installing, updating, enabling, disabling, and removing extensions on the local machine.)
- ExtensionManagerCLI - Command-line utility next to the CAM application (Packaging and local registration of extensions by authors and administrators.)

Main components:

The Extension Store consists of two linked parts:

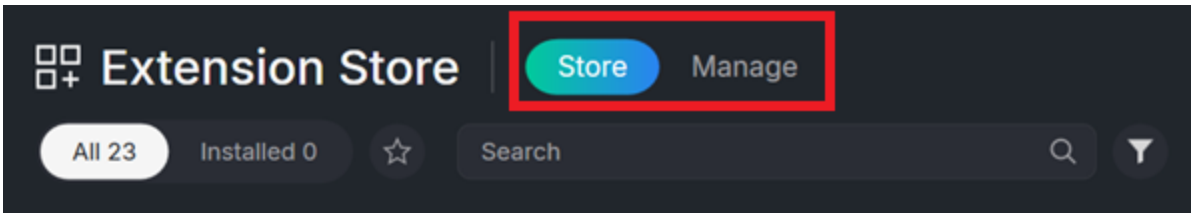
- Web storefront. The catalog is visible to any visitor. Signed-in users can add extensions to favorites, request trials, and publish extensions. Authors and administrators access moderation and configuration screens.

▼ Click here to expand..

- In-application tab. Contains two pages: Store for the catalog and Manage for installed extensions and feeds. A link in the upper part of the tab opens the web storefront in the default browser.

▼ Click here to expand..

The tab is opened from the Home page. It has two pages: Store (Browses the extension catalog and installs or updates extensions) and Manage (Lists installed extensions, enables or disables them, installs from files, and configures feeds).



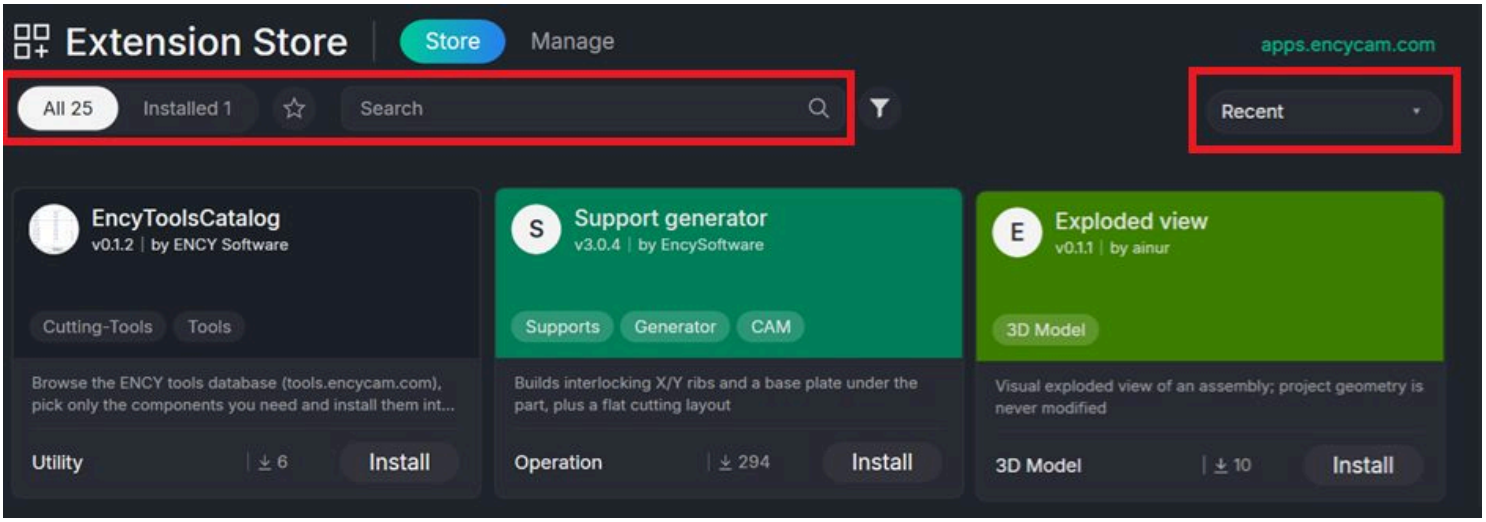
Store page toolbar.

Search field. Prompt text is *"Search"*.

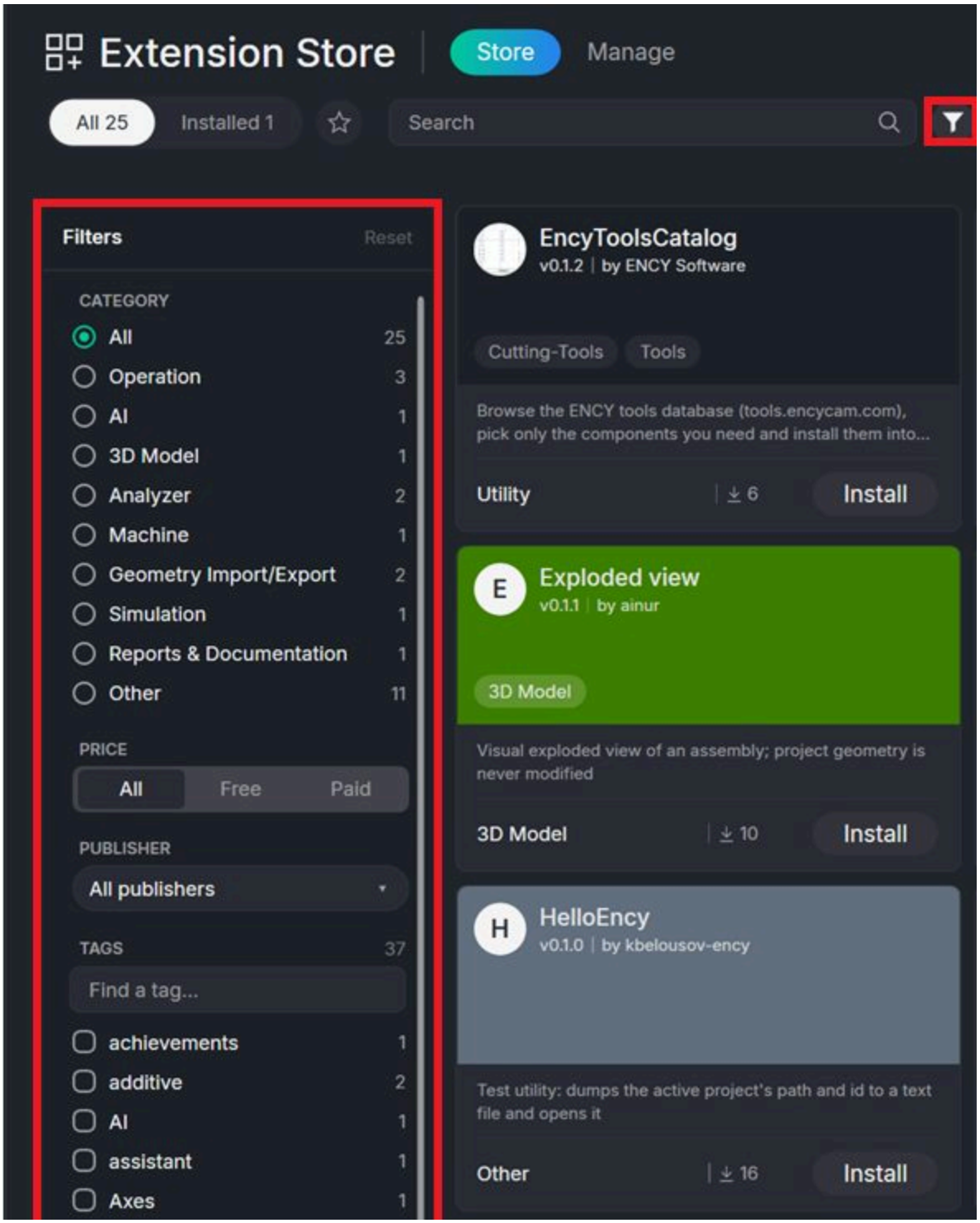
Sort options. Recent, Name, Publisher, and Type. Default order follows the feeds.

Scope switch. All shows every extension offered by the feeds. Installed shows only extensions already on the machine.

Favorites. Star filter showing only starred extensions.

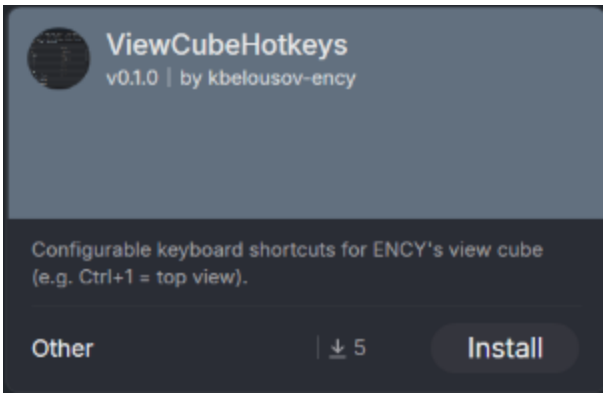


Filter button. Opens the filter panel. Sections are CATEGORY, PRICE (choices All, Free, Paid), PUBLISHER (default All publishers), and TAGS with the search prompt "Find a tag…"



Extension card in the Store page.

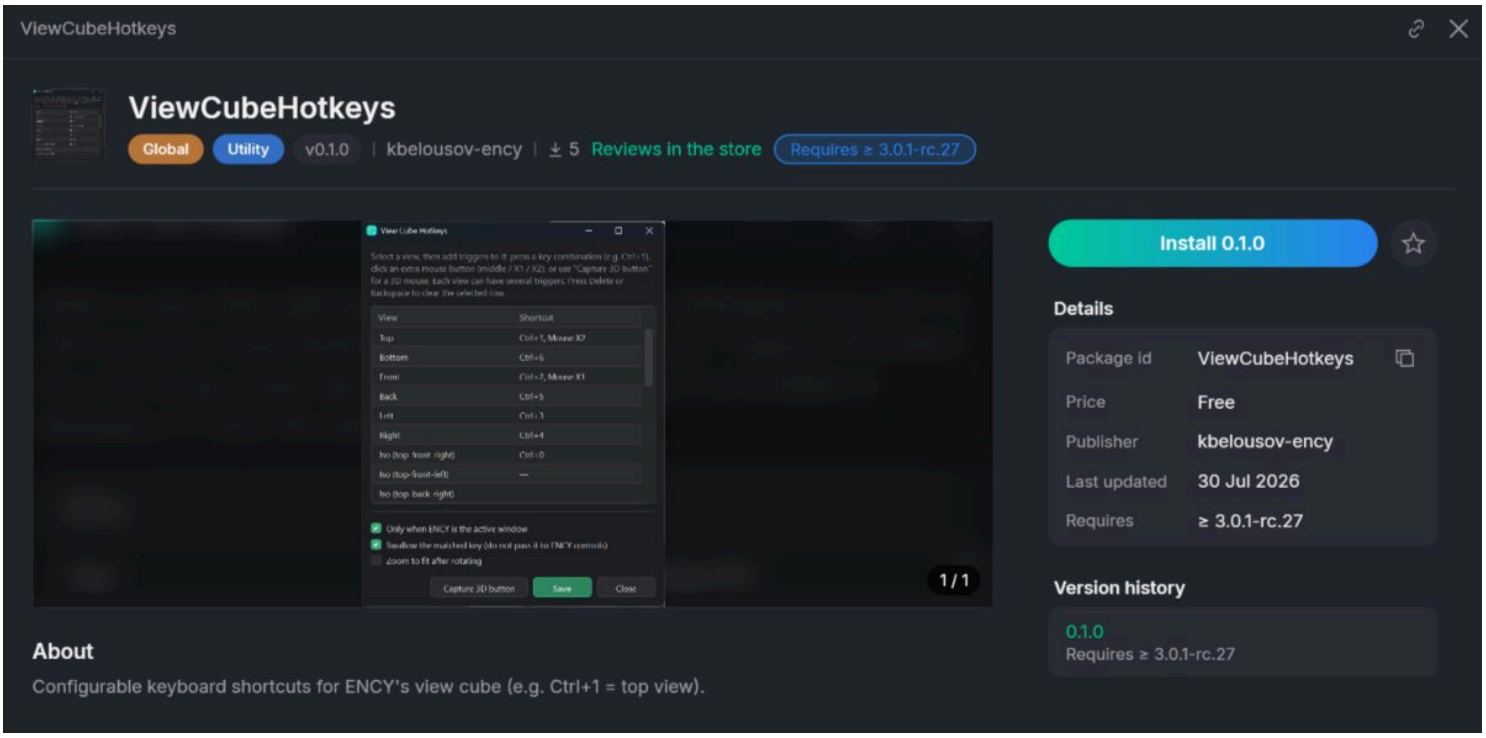
The card shows a cover, icon, name, publisher, version, description or "No description", Paid badge, extension type, download count, and a favorite star. The action button changes according to state: Install, Update, Uninstall, View, or Installed.



When you click on the card, you can read the extension description and how to use it.

Each card displays the following:

- Cover. A screenshot provided by the author, or a solid color derived from the category, or an entry-point icon when neither screenshot nor category is present.
- Name, version, and publisher.
- Description. If the author left it empty, the text *"No description"* appears in italics.
- Tags. Up to three tags are shown, followed by an indicator for additional tags. The tag that duplicates the category name is hidden.
- Paid badge. Appears when the extension requires a license.
- On moderation badge. Visible to the author for new extensions that are awaiting approval.
- Download count and category name.
- Favorite star. Visible only to signed-in users.



Installing an extension.

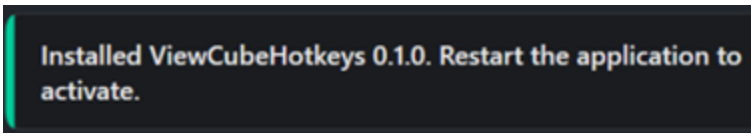
There are three starting points for installation, all of which converge to the same in-application process.

▼ Click here to expand..
From the web storefront. Press Install in CAM system (v version) on the extension page.

From the in-application catalog. Press Install on the card, or open the detail view and press Install version.

From a file. On the Manage page, press Install from file… or drag a .nupkg or .dext file onto the tab.

After installation starts, the button becomes a progress indicator and passes through the stages: "Looking in the store…", "Searching the feeds…", "Checking compatibility…", "Downloading…", "Reading package…", "Installing…", "Registering…".



Location of Installed Extensions in the CAM System Interface.

The store copies files and registers the extension, but does not start it. Where the extension appears depends on its entry point, which is defined by the extension author.

▼ Click here to expand..
Utility. Utilities palette, opened with the Utilities button on the top toolbar or the Ctrl+Shift+E shortcut.

Operation Popup. Right-click menu on an operation in the technology tree.

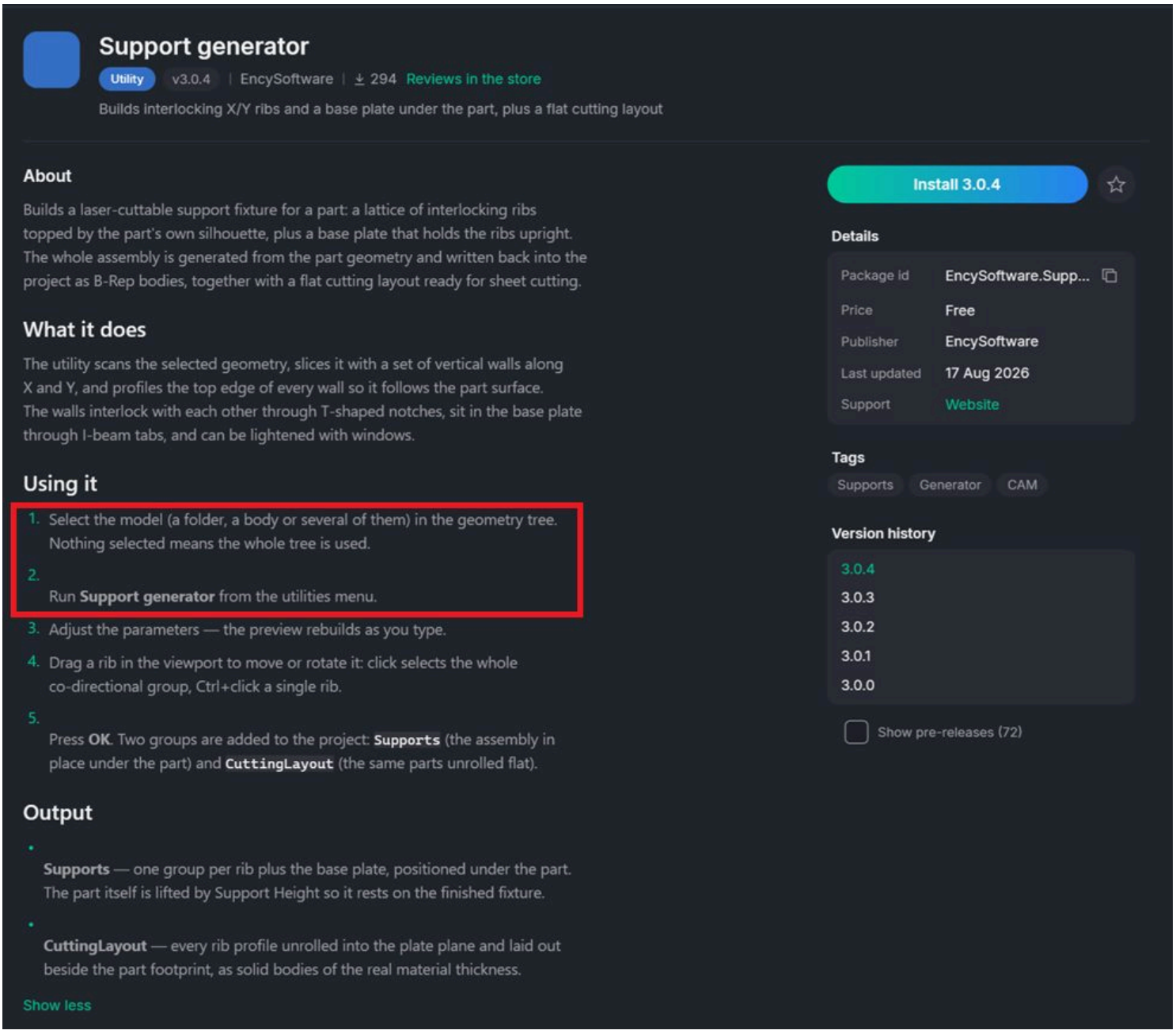
Geometry Node Popup. Right-click menu on a node in the geometric model tree.

Operation Solver. As an operation in the project. The extension computes the operation and its toolpath.

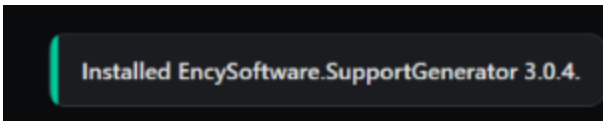
Global, CLData Converter, Utility Runner. No visible location. The code runs automatically on startup, during toolpath calculation, or when a utility is launched.

If an installed extension does not appear where expected, the library or its entry point may be disabled on the Manage page, or the CAM system may need a restart.

As an example, the Support Generator extension is installed. The card description provides instructions for invoking it.

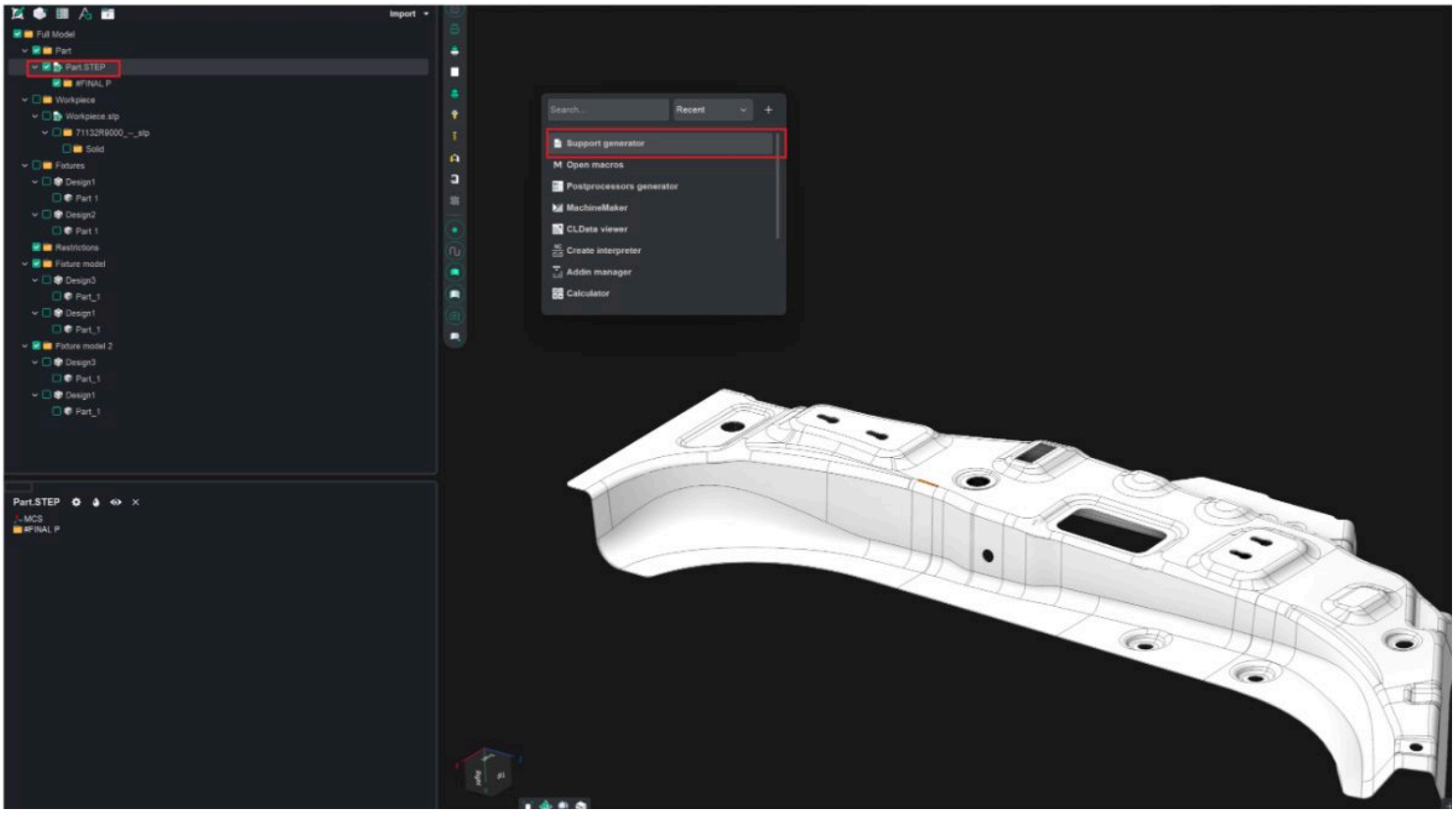


A notification will appear at the bottom of the screen indicating that the extension has been installed.

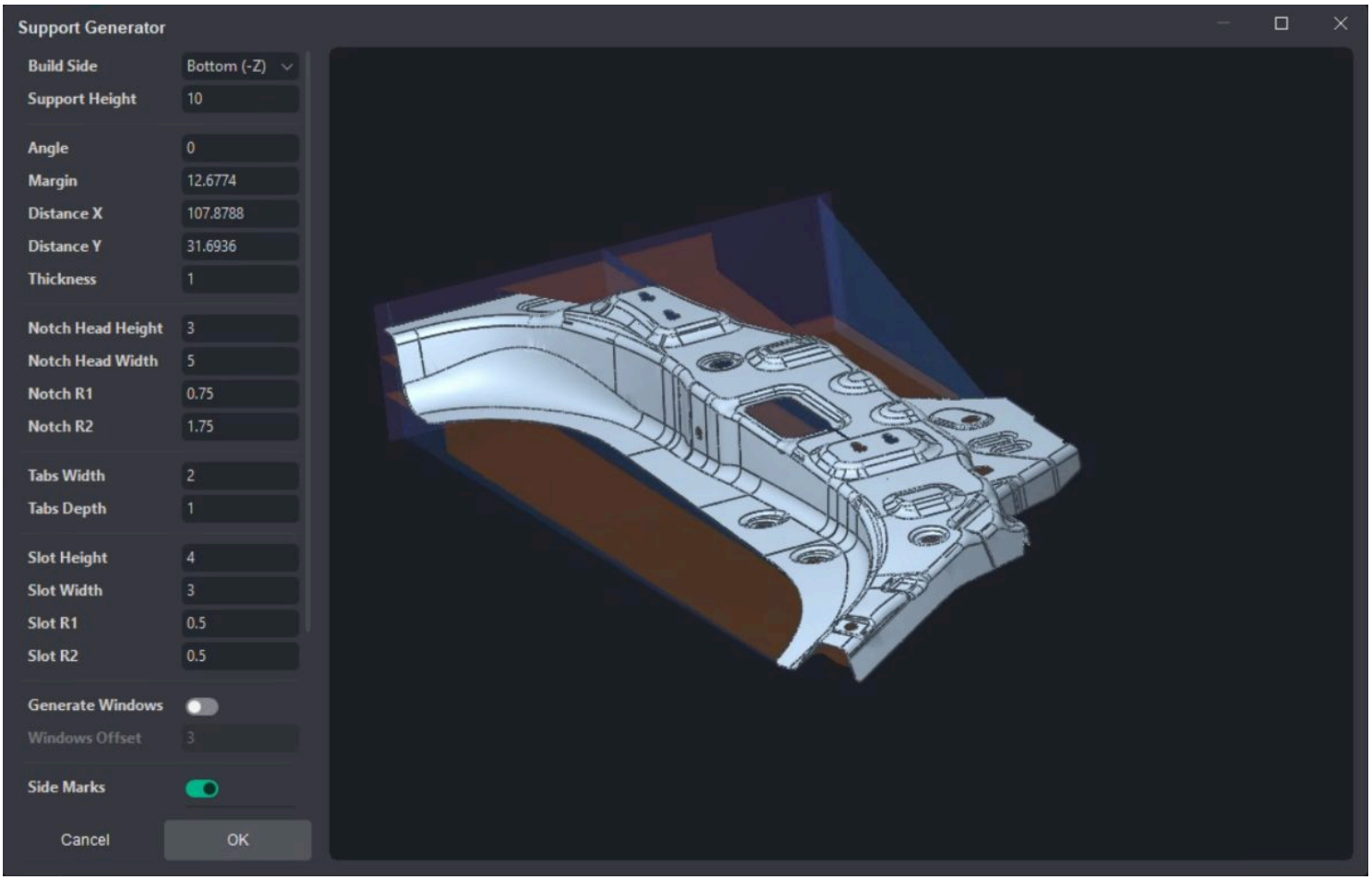


The project is opened from the Recognize Features project library.

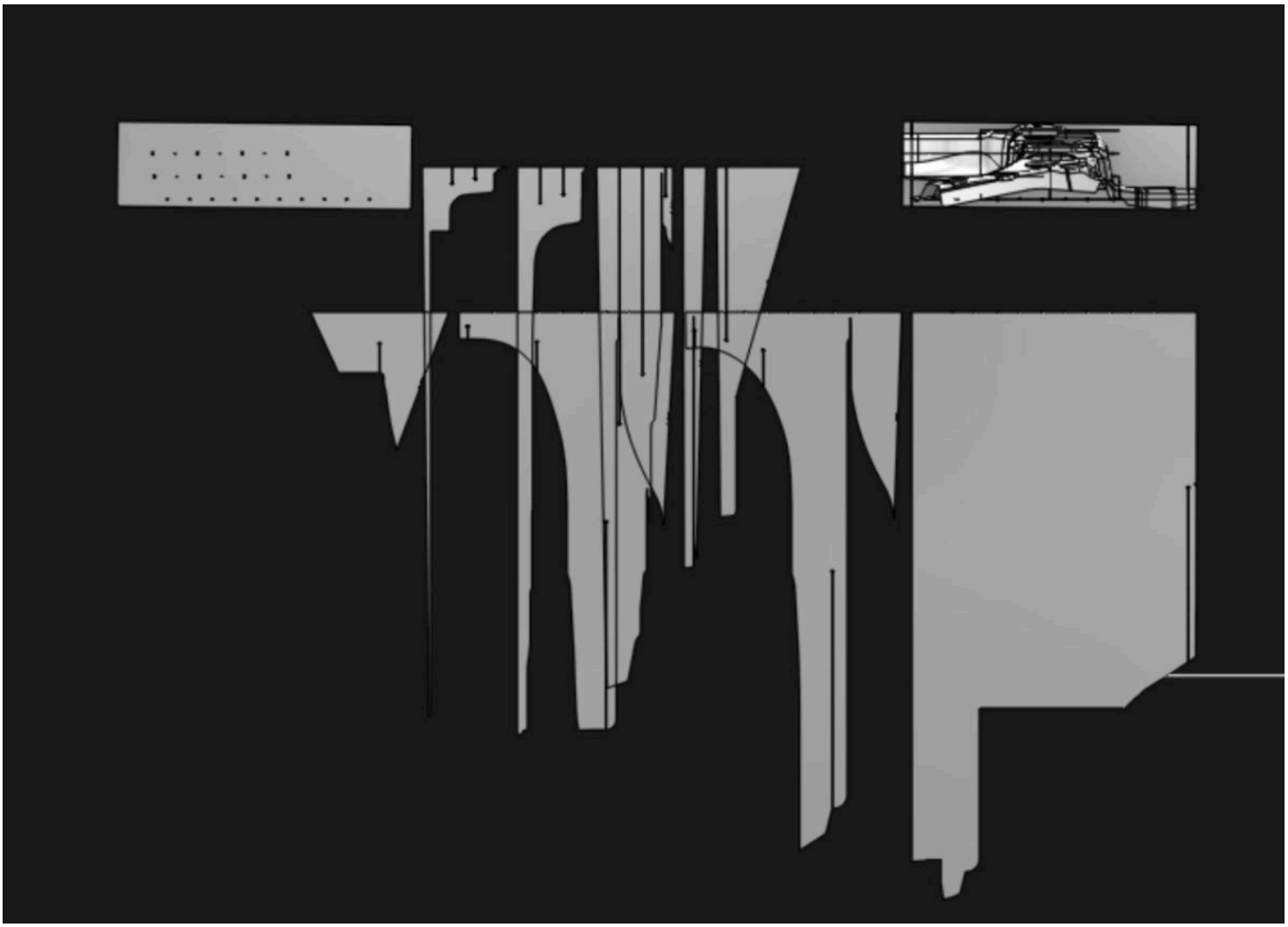
On the Model tab, select the part geometry and generate the supports.



In the dialog box that appears, set the required parameters.



Once you close the dialog box, the necessary supports for the part will be created.



Program's licensed modules

Program's licensed modules — it is features, which can be turned on in one of the existing configurations when additional license is present.

Adaptive

Module for high-speed milling.

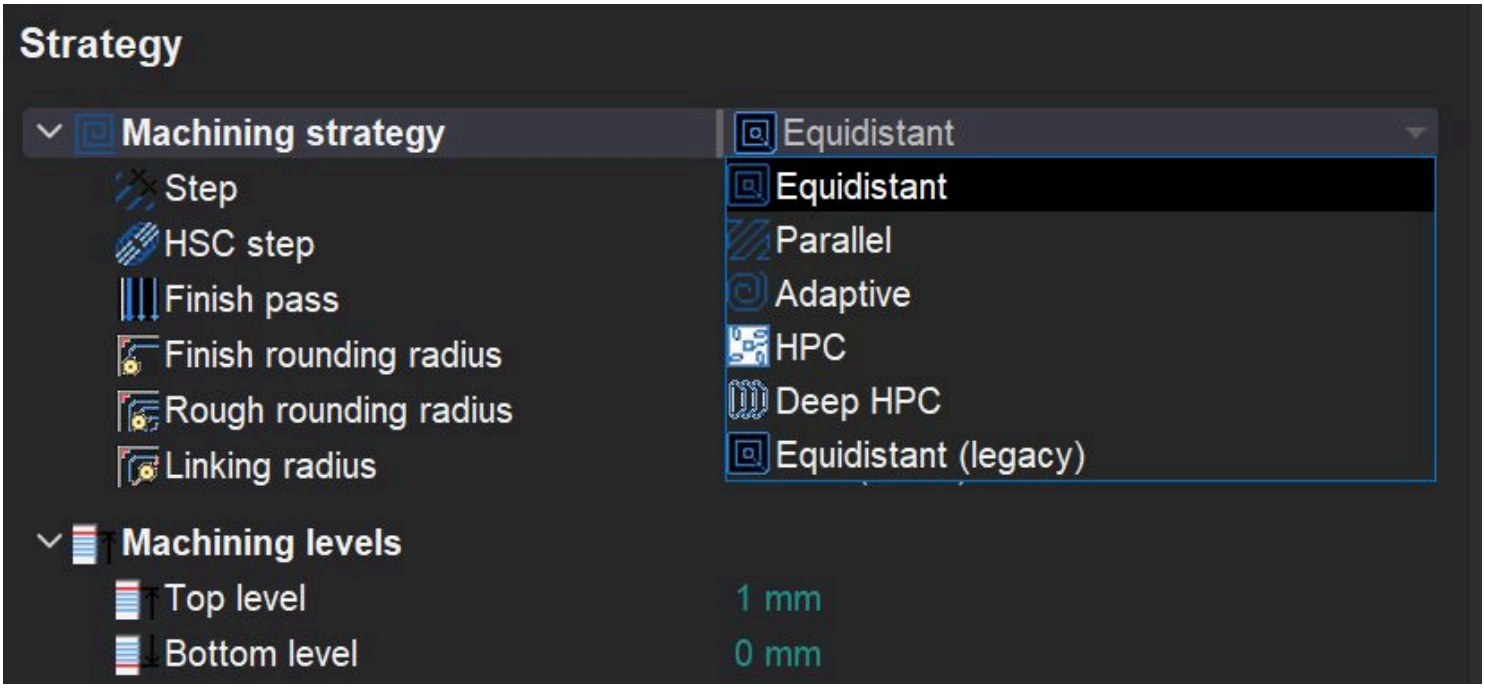
> Note:
>
> Need an additional license.

The pocketing strategies are designed for the removing of material in the open and closed pockets.

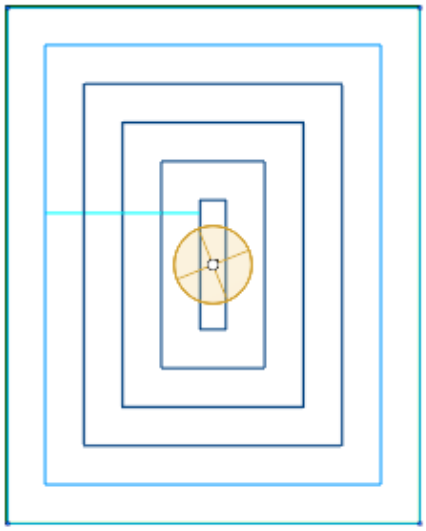
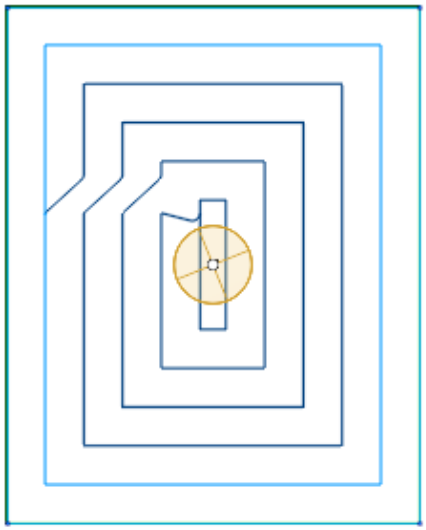
These strategies are available in the following operations:

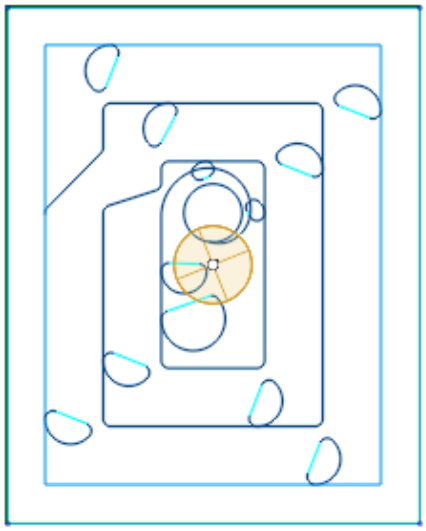
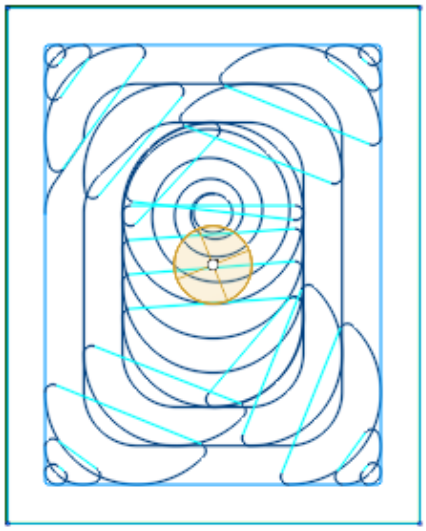
- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

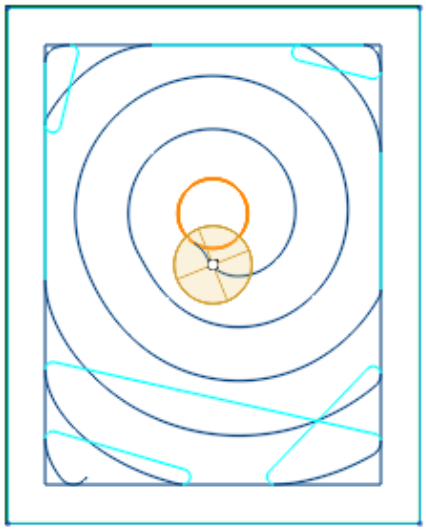
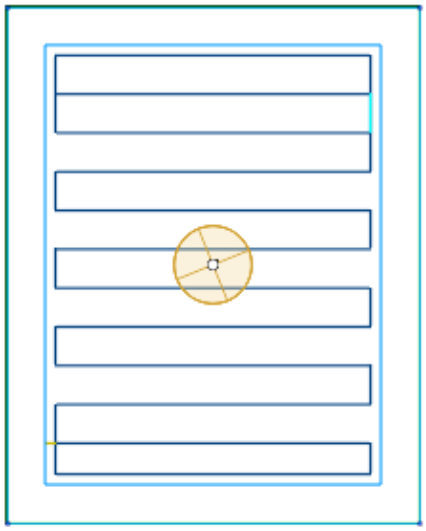
Seven strategies are available to select in the Machining strategy drop-down:



There are 6 strategies. Some items are optional and require an additional license. So many strategies are the result of long-term development. Every strategy has its own advantages and disadvantages, so no one of them can't be removed from the system.

Strategy		
Equidistant (legacy) 	Advantages Fast calculation Simple tool path	Disadvantages Residual unmachined islands are possible if the step is more than 50% Uneven tool load and chip thickness Many Z motions to/from the safe plane
Equidistant 	Advantages It's possible to define the safe distance The most of the links are performed without the climbing of the safe plane Links rounding is available	Disadvantages Residual unmachined islands is possible if the step is more than 50% Uneven tool load and chip thickness

Strategy		
HPC (high performance cutting) 	Advantages All advantages of the equidistant strategy Special arc is added to remove the residual unmachined islands	Disadvantages Uneven tool load and chip thickness The special arc's radius can be too small, that gives the uneven feed rate.
Deep HPC 	Advantages All advantages of the HPC strategy The even tool load	Disadvantages Tool path is longer than the HPC strategy Idle motions are possible Unstable calculation. Sometimes the tool load can be greater than required.

Strategy		
Adaptive 	Advantages The even tool load The perfect tool path for the open pockets	Disadvantages The length of tool path can be longer than the length of the DeepHPC strategy with the same parameters. It's actual for the big closed pockets.
Parallel 	Advantages Fast calculation Simple tool path	Disadvantages A lot of Z-motions in the complex pockets

Features of Adaptive strategy

The strategy is used to effectively remove large volumes of material with high feed rates, maximal cutting depths (up to the flute's length) and relatively shallow cutting widths (5% to 30% of the tool diameter). Such parameters are possible as the specified tool engagement angle (which is defined as the width of cut, or step) is guaranteed to never be exceeded by the strategy.

The material is removed in spiral-like fashion. There are no sharp corners in the toolpath. Smoothness of the toolpath is precisely controlled by the dedicated parameters for the roughing rounding radius, the finishing radius and the linking radius. Linking is done preferably in the working plane with an additional small Z clearance, which helps fight heat buildup. The tool engages material using the so called 'Roll-In technique' which prolongs tool life. Both climb and mixed (climb and conventional) milling is available. For

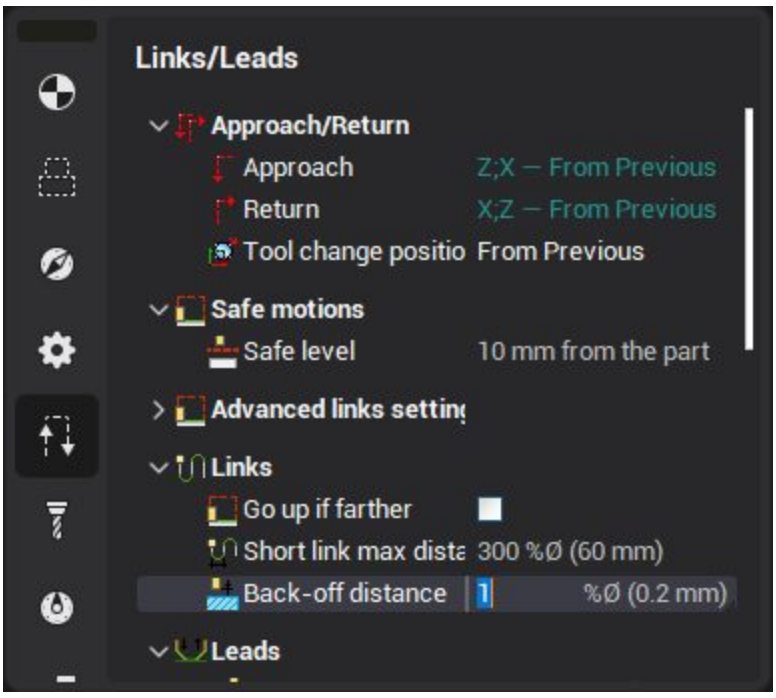
the mixed milling, the width of cut and the feed rate of conventional passes can be set separately from the climb passes.

How to choose the pocketing strategy

1. The choice number one is Adaptive This strategy is not set as default, only because it requires the additional licensing. So we strongly recommend purchasing it. All other variants must be tested only if this strategy is not available or gives the improper toolpath.
2. If Adaptive is not possible, and you need the even tool load, then try Deep HPC strategy.
3. If even tool load is not necessary and the machining step is more than 50% of the tool diameter then try HPC strategy
4. If even tool load is not necessary and the machining step is less than 50% then try Equidistant strategy.
5. Use Parallel strategy at your own discretion.
6. Use Equidistant (legacy) if all other strategies give improper toolpath.

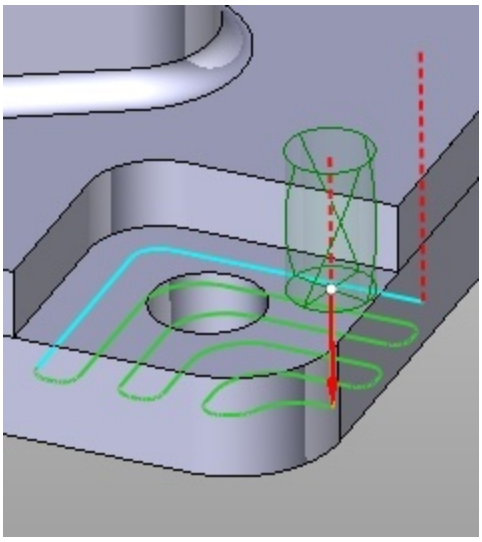
Tool path parameters

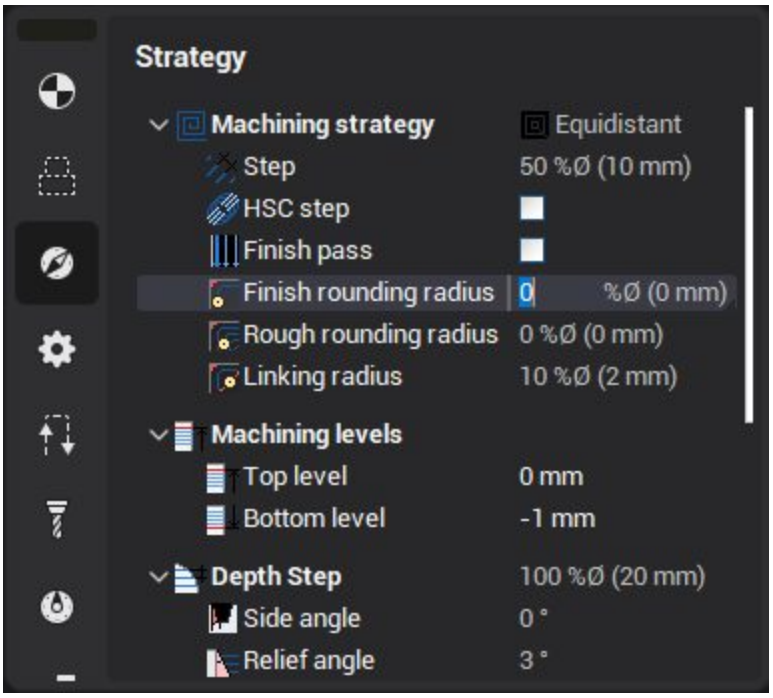
- Back-off distance parameter



The tool can be lifted above the already machined surface when it moves to the next trochoidal arc start position.

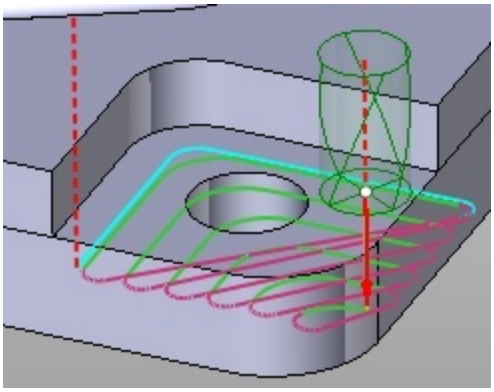
- Rounded links in zigzag mode





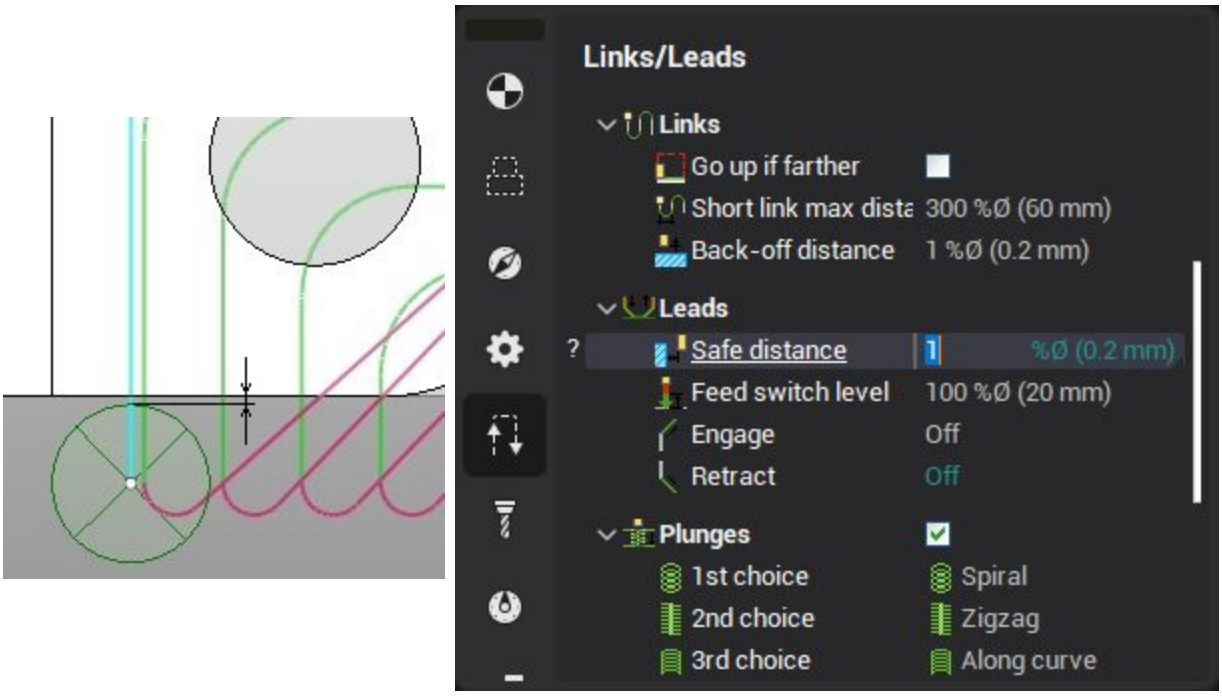
The 'Finish rounding radius', 'Rough rounding radius' and 'Linking radius' value is used for rounding of the links.

- Links on the same Z-level



In the climb and conventional mode, the tool goes directly to the next path without retraction to the safe level. If a rapid motion is performed over an already machined surface, then the "Tool back-off distance" is used. "Idle radius" is also used to make the motion smooth.

- Safe distance

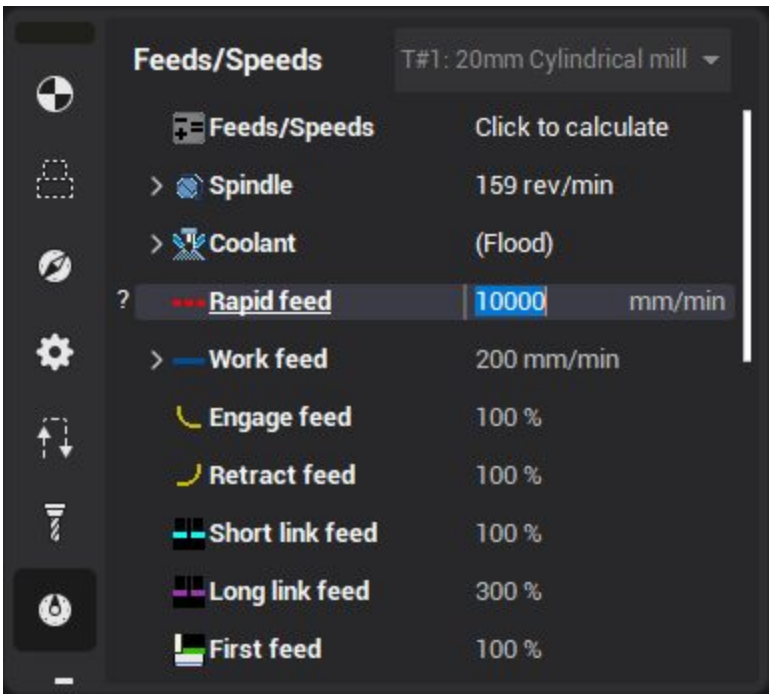


Safe distance is used to move the tool down/up from/to the safe surface.

The vertical motion is performed at this distance from the workpiece. So in version 10 there is no longer the need to enable the approaches/retractions to exclude the rapid feed collisions.

If you use a pre-drilled hole to plunge when roughing, the pre-drill tool diameter must be greater than the mill tool diameter by at least double the safe distance amount, otherwise the pre-drilled holes will not be detected.

- Rapid feed links



The link moves can be calculated using either the next feed or the return feed values. If the link length is less than the 'short link' distance, then the 'next feed' value is used, else the 'return feed' value is used. The return feed is set to 300% of the work feed by default, which is a non-cutting feed. If cutting is detected during a 'return feed' move when simulated, this move will be marked with an error.

Robot +

It allows adding features that make it possible to control 6-axis (articulated) robots when you have only the configuration for normal machines (milling etc.).

These features:

- Advanced robot
- Advanced multi axis robot control

This module is available for configuration:

By default:

- Robot

As option:

- 5x Mill*
- Master*
- Pro*

- – an additional license for support robots is needed.

Robot Mill

Including the ability to use milling operations in the 'Robot' configuration.

This module is available for configuration.

> Note:
>
> Need an additional license.

Advanced multi axis control

This module extend machines/robots machining features (6 or more axes).

There are features:

- Interactive machine
- Machine control panel

This module is available for CAM system configuration:

By default for machines:

- Cutting
- Wire EDM
- Lathe
- Machinist
- Universal
- 3x Mill
- Expert
- 5x Mill*
- Master*
- Pro*

By default for robots:

- Robot
-
  - the additional 'Robot +' license for support robots is needed.

Operations

Some of CAM system operations may be flexible added into various software configurations by getting a special license. They are grouped into modules listed later in this section.

Sawing

Sawing operation allows calculating a toolpath for cutting with a circular saw. Available as an option.

> Note:
>
> Operation must be licensed separately.

5D cutting

The option for the Cutting configuration which allow to use 5D Contouring operation to calculate curved surfaces cutting toolpath.

Note: option must me licensed separately.

Welding

This operation is available as option of CAM system.

It implements the functionality of automatic weld seam geometry calculation without reference to the particular type of welding equipment, ie it does not generate the specific commands to the laser, electric arc, gas burners, ultrasonic emitters etc. Setting up of operation to work with specific type of equipment can be made by writing in the postprocessor appropriate commands to control the equipment, or, if this is not enough, by the addition of a special operation on the basis of the Welding 5D operation, adapted to control specific equipment.

Note: option must me licensed separately.

Cladding

The optional module of CAM system which includes operations for additive manufacturing, eg Area cladding operation.

Additive operations of this module are universal, not tied to a particular cladding technology and type of used equipment. They only implement a geometry of the process, generate toolpath, which successively, layer by layer passes over specified surfaces, and reproduces them from the bottom upwards. Setting up of operation to work with specific type of equipment can be made by writing in the postprocessor appropriate commands to control the equipment, or, if this is not enough, by the addition of a special operation on the basis of the universal additive operations, adapted to control specific equipment.

Note: option must me licensed separately.

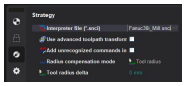
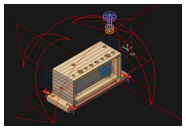
Knife

Two separate operations:

- knife 2D
- knife 6D

These operations licensed separately.

G-code based, G-code based lathe

Set NC program		Select the interpreter		Press Run		Get the toolpath
	...		...		...	

G-code based operation and G-code based lathe operation operations located in Auxiliary list. It can be also used for indexed and continuous processing at 4 and 5 coordinate machining centers. All available simulation types are supported, including additive manufacturing to simulate material layer buildup.

> Note:
>
> - G-code based operation does not support the turning tool.
> - G-code based lathe operation supports only turning tool.
> - Both operations do not support Wire EDM machining.

These operations allow you to perform:

- direct control of the machine simulation using G-codes;
- check and optimize the NC program;
- convert the text of the NC from one controller to another (for machines with identical kinematic scheme);
- debug your own interpreter during its creation.

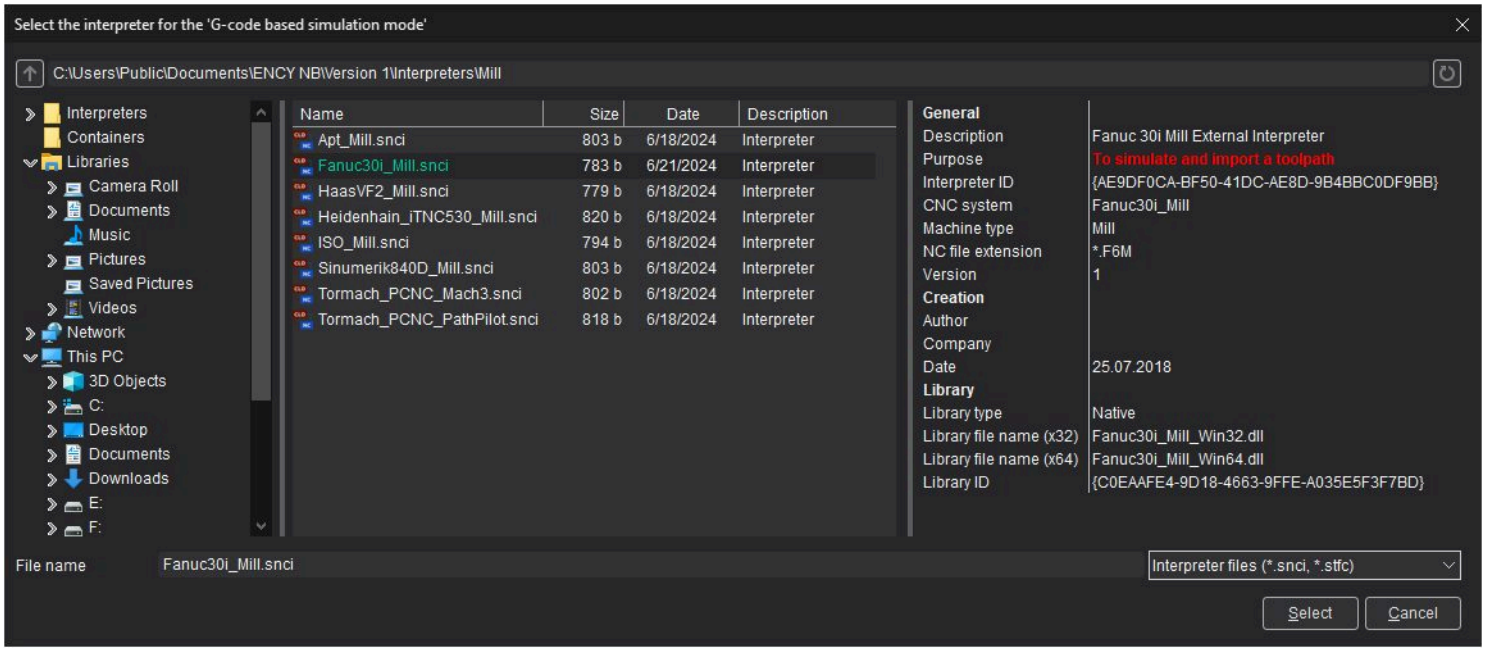
The toolpath is formed on the basis of the following operation parameters:

- Specified NC text on the Job assignment parameters panel;
- Selected interpreter on the Strategy parameters panel;
- Assigned tool on the Tool parameters panel.

The NC text can be written manually or can be loaded from an external file and edited, if necessary. The built-in text editor supports syntax highlighting of the main key structures of the CNC programming language, as well as a wide range of keyboard shortcuts for working with text.

> More detailed information about the possibilities of working with the NC text is described in the section Job assignment for G-code based operation, G-code based lathe operation

Interpreter file (*.snci) defines the format of recognition of the controller commands in blocks of the NC program. The corresponding parameter specifies the full path to the selected interpreter. The parameter value can be entered manually as well as by using the file selection dialogue, which is accessed by using the ... entered button. During the selection process, selection process, a preview of the interpreter information is available (description, purpose, CNC system, authors, etc.):



The ability to select an interpreter from the container is supported.

Use advanced toolpath transformation- this parameter converts the NC toolpath from machine to geometric. Which, in turn, makes it possible to correct the toolpath using the robot axes map. If the setting is disabled, then the final toolpath is formed (as closely as possible repeating the one specified in the NC program), without the possibility of changing it.

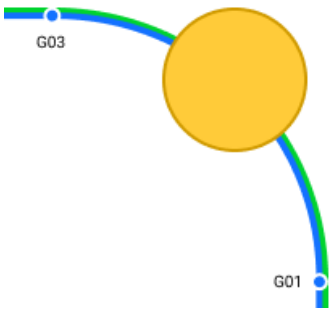
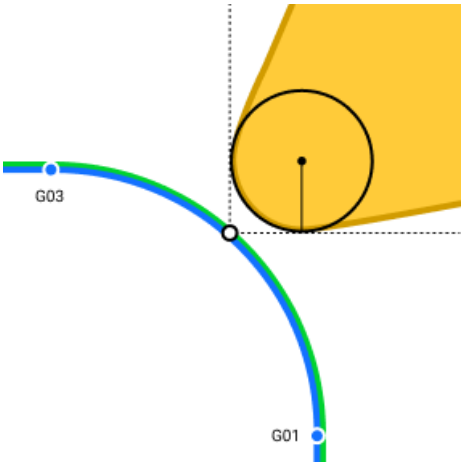
Step of physic movements dividing - available only when using Use advanced toolpath transformation (see above). In this mode, at the first stage, the tool path is converted into a geometric curve. At the next stage, machine movements are rasterized with the step specified in the current parameter to ensure maximum similarity with the original toolpath. The smaller the value of the rasterization step, the higher the accuracy of construction and, accordingly, the similarity with the original toolpath.

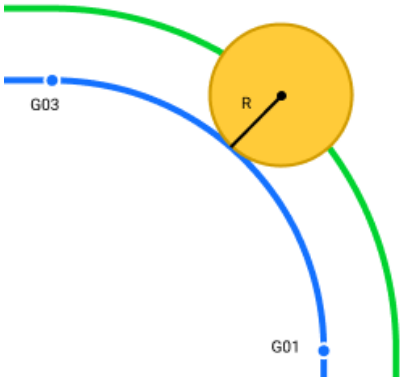
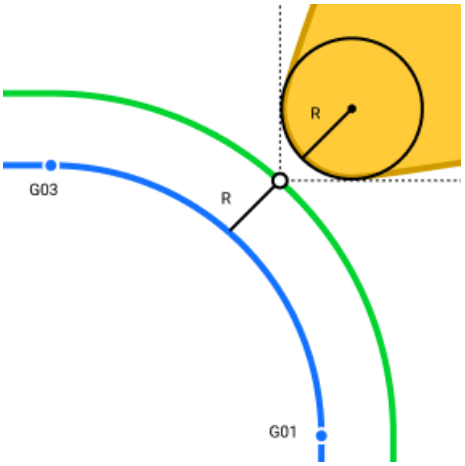
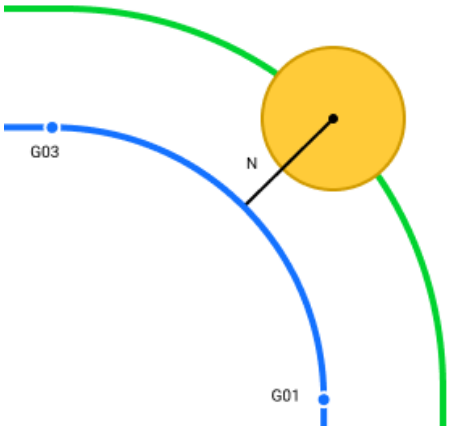
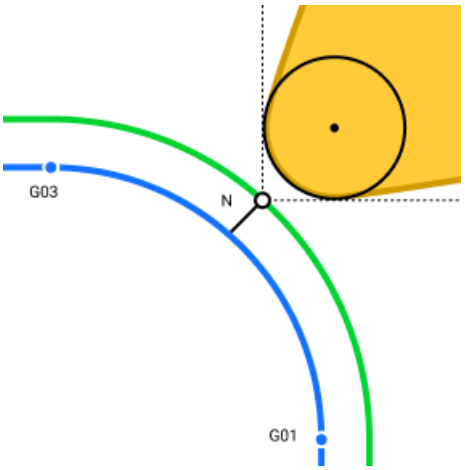
Add unrecognized commands in the trajectory - this parameter adds all commands of the control program not recognized by the interpreter to the toolpath. Unrecognized commands are added to the tool path description as a technology command parameter INSERT before the recognized commands.

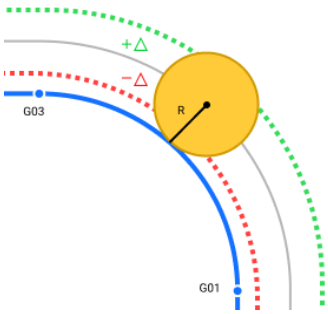
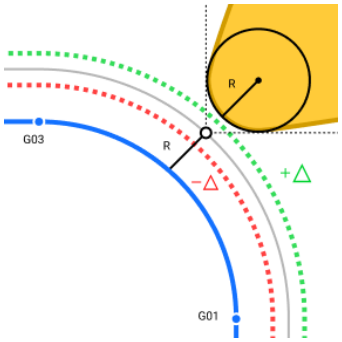
> Example of how the parameter "Add unrecognized commands in the trajectory" works

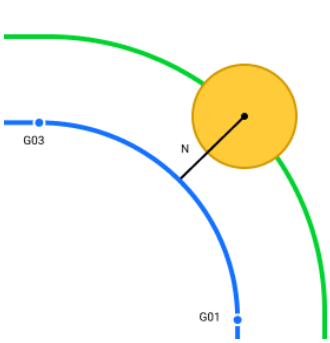
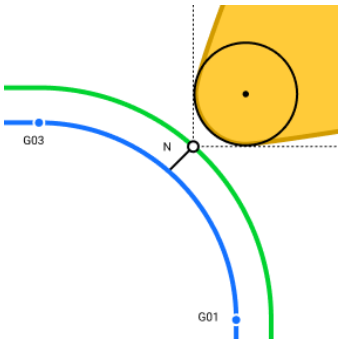
NC program	This parameter is disabled	This parameter is enabled
M107 G0 F3600 X13.885 Y55.321 Z0.15 G1 F3000 X14.618 Y55.174 E0.09324 G1 X15.622 Y55.102 E0.21879 G1 X17.918 Y55.102 E0.50516 G1 X18.665 Y55.142 E0.59846 G1 X19.403 Y55.261 E0.6917 G1 X19.926 Y55.396 E0.75907 G1 X20.67 Y55.619 E0.85594 Commands M107, E - are not recognized by the interpreter	**F: WORK 200мм/мин.** X13.885, Y55.321, Z0.15 **F: WORK 2100мм/мин.** **RAPID: 10000** **F: WORK 3000мм/мин.** X14.618, Y55.174, Z0.15 X15.622, Y55.102, Z0.15 X17.918, Y55.102, Z0.15 X18.665, Y55.142, Z0.15 X19.403, Y55.261, Z0.15 X19.926, Y55.396, Z0.15 X20.67, Y55.619, Z0.15	INSERT: "M107" INSERT: "E0.09324" **F: WORK 200мм/мин.** X13.885, Y55.321, Z0.15 **F: WORK 2100мм/мин.** **RAPID: 10000** **F: WORK 3000мм/мин.** X14.618, Y55.174, Z0.15 INSERT: "E0.21879" X15.622, Y55.102, Z0.15 INSERT: "E0.50516" X17.918, Y55.102, Z0.15 INSERT: "E0.59846" X18.665, Y55.142, Z0.15 INSERT: "E0.6917" X19.403, Y55.261, Z0.15 INSERT: "E0.75907" X19.926, Y55.396, Z0.15 INSERT: "E0.85594" X20.67, Y55.619, Z0.15

Radius compensation mode - A bility to disable or set an arbitrary compensation value on the tool radius.

Parameter value	Milling	Turning
Off	 *Radius compensation* is disabled. Commands *G40, G41, G42* are ignored	 *Cutter radius compensation* is disabled. Commands *G40, G41, G42* are ignored

Parameter value	Milling	Turning
Tool radius	 *Radius compensation* is equal to the *tool radius*	 *Cutter Radius compensation* is equal to the *tool nose radius*
Custom value	 *Radius compensation* is determined by the specified arbitrary value.	 *Cutter radius compensation* is determined by the specified arbitrary value.

Parameter	Milling	Turning	Note
Tool radius delta	 *Additional tool offset*. The total offset is calculated as the sum of the *tool radius* and	 *Additional tool offset*. The total offset is calculated as the sum of the *tool nose radius*	The setting is only available when using the Tool radius value set as the radius compensation mode (see above)

Parameter	Milling	Turning	Note
	the value of the *additional offset*.	and the value of the *additional offset*.	
Compensation value	 Arbitrary value for radius compensation. *By default*, the value is equal to the *tool radius* — 50% Ø	 Arbitrary value for cutter radius compensation. *By default*, the value is equal to the *tool nose radius* — 50% Ø	The setting is only available when using the Custom value value set as the radius compensation mode (see above)

The tool that will be processed is determined on the corresponding tab of the operation parameters window. When creating G-code based operations, it is assigned a default tool for appropriate processing (milling or turning).

Note: Currently, the tool number indicated in the NC text is not taken into account when selecting from the list of project or library tools. Due to the above feature, only the NC text in which the processing is carried out by one tool can be assigned as a job assignment for each such operation.

> G-code based operation demo video
>
> ▼ Watch demo video

Job assignment for G-code based operation, G-code based lathe operation

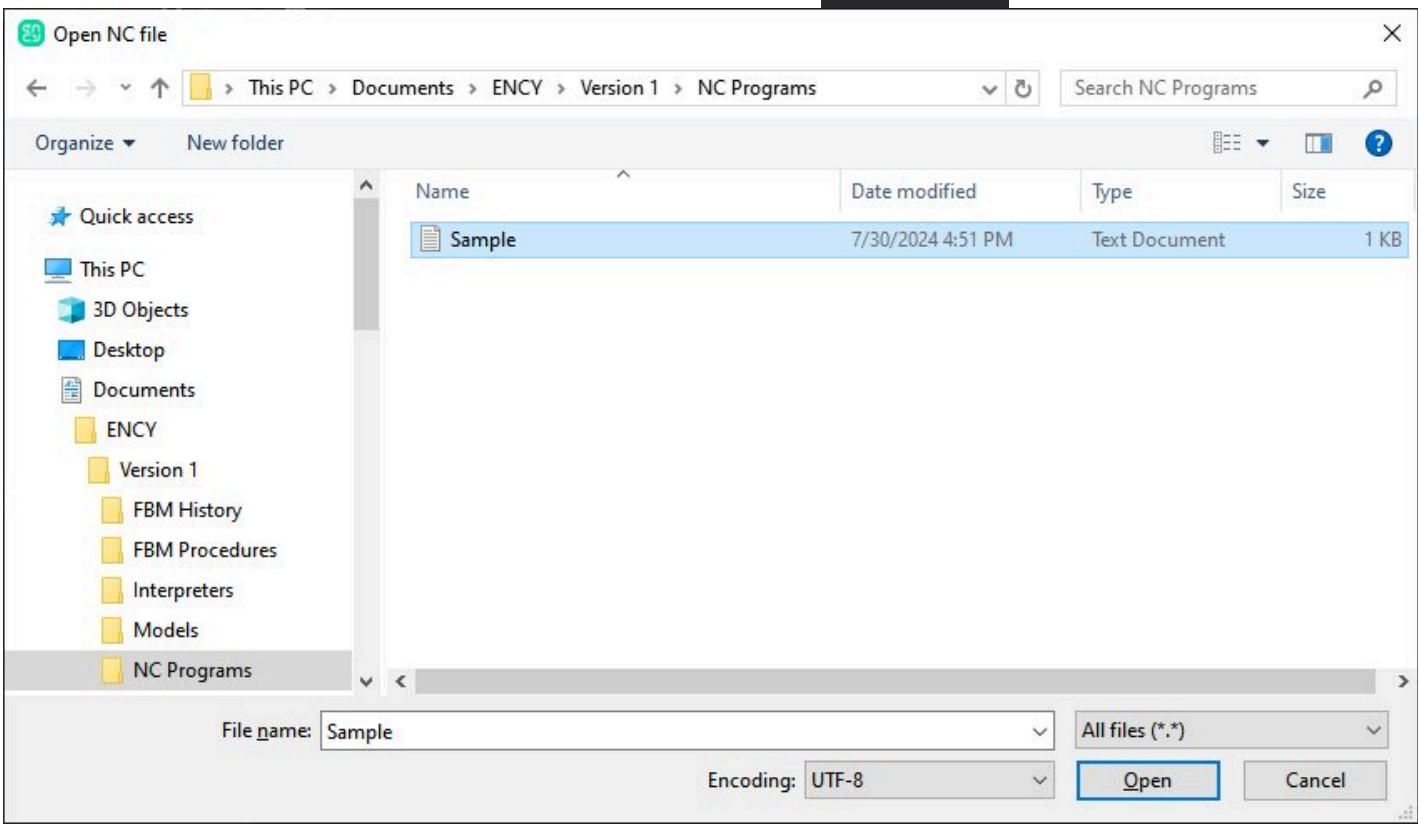
Unlike other operations, the Job assignment for these operations is the NC text. The NC text can be specified in several ways:

1. Written manually directly in the built-in text editor:

> NC example in the format of G- and M- codes (ISO 7 bit)
>
> ▼ Watch an example
>
> ```
> G0 X50 Y50 Z100
> Z0
> G1 Y0 F1000
> X100
> Y100
> X0
> Y0
> X50
> Y50
> G0 Z100
> ```

2. Loaded from an external file and edited, if necessary. File opening is done:

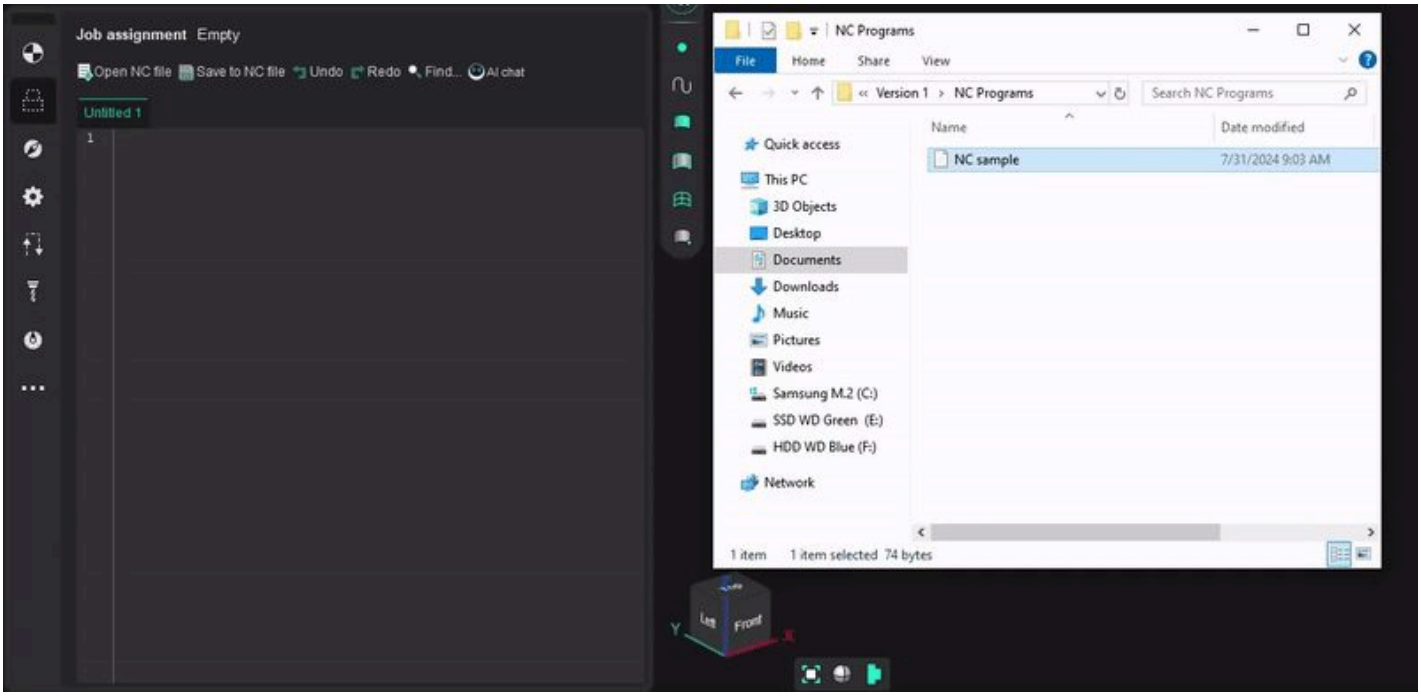
3. Using the dialog box displayed after clicking on the button Open NC file;



4. Interactively, with a simple drag and drop. To do this, drag the required file from the Windows Explorer directory into the built-in text editor window with the left mouse button pressed, and then release it. > Opening a file in interactive mode demo video > >

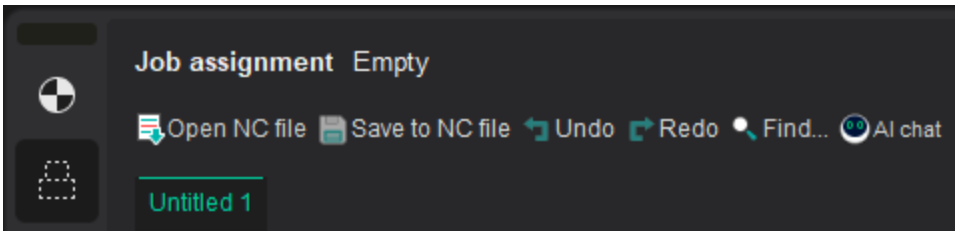
▼ Watch video

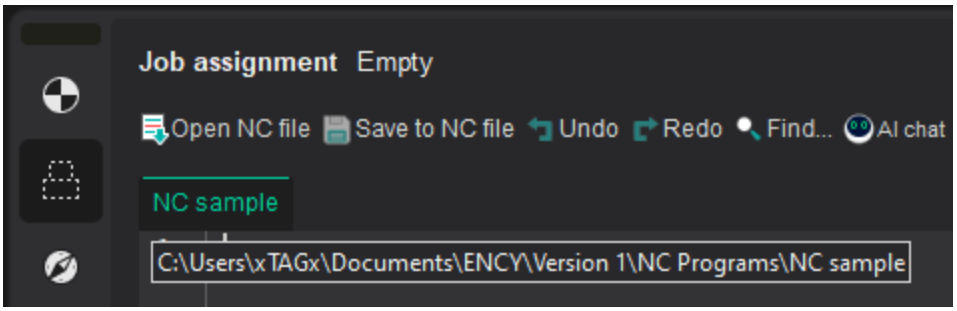
> > >



> >

By default, the title of the operation parameters panel is Job assignment Empty, and the text editor tab name is `Untitled #` for new projects or `Project_Name #` for previously saved projects. The `#` symbol indicates the tab sequence number. After saving the text of the NC program to a file using the Save to NC file button or open the external NC file in the built-in text editor, the tab name will receive the name of the file, and the tooltip displays the full path to it. For example, for the `NC sample.txt` file located in the NC Programs directory of the shared CAM system documents directory. The title of the operation parameters panel and the tab name of the built-in text editor will change their names to Job assignment and `NC sample.txt` respectively.

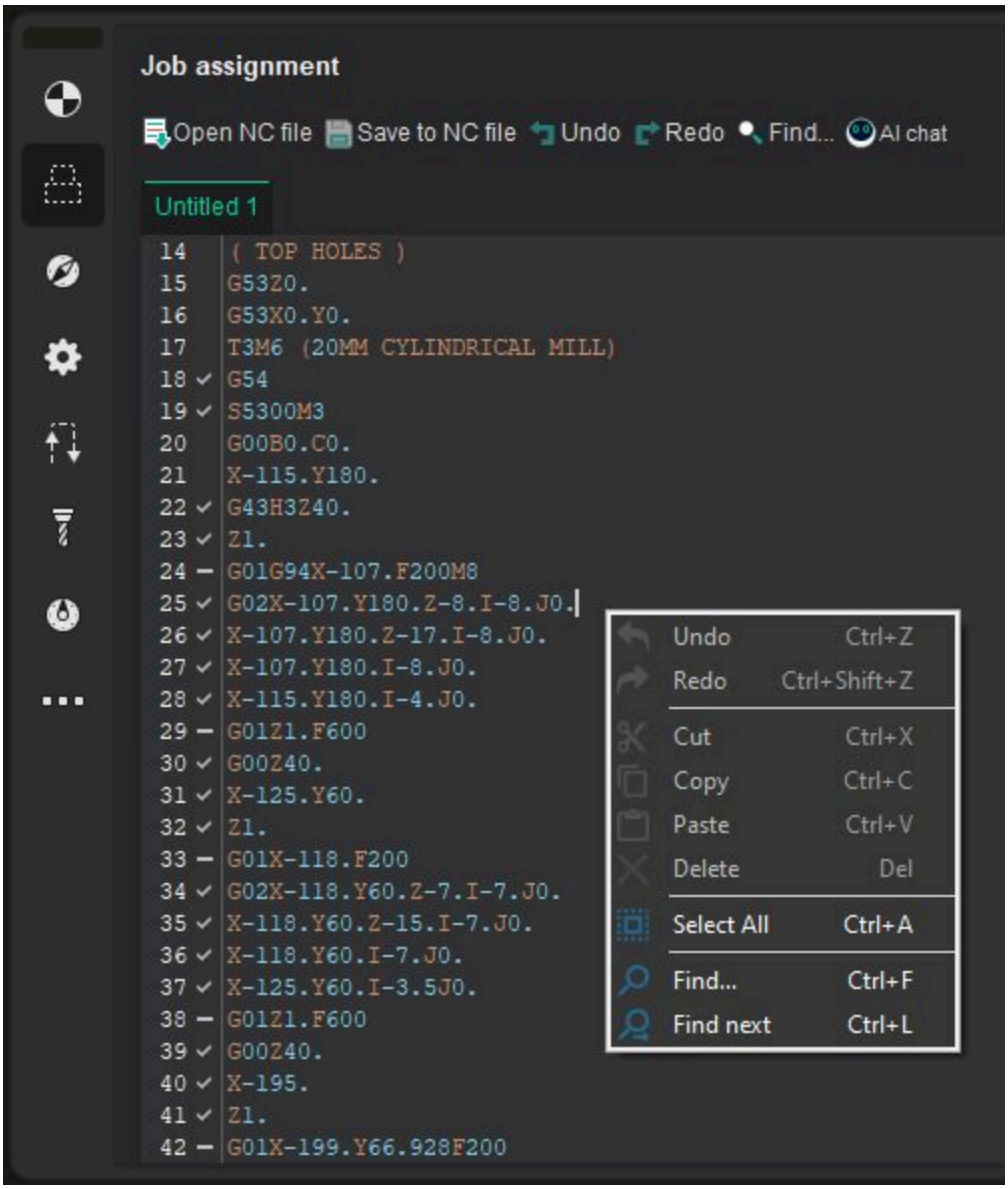




Note: *Changes made outside the built-in text editor to the open or saved file used as Job assignment for the G-code based operations are not currently tracked. To update the job assignment in this case, you need to re-open the modified file by one of the methods described above.*

The built-in text editor supports syntax highlighting of the main key structures of the CNC programming language, as well as a wide range of keyboard shortcuts for working with text to help with the text of an NC program.

The main actions for working with text are also available from the context menu, which can be called up by right-clicking in the text editor area on the Job assignment operation parameters panel.



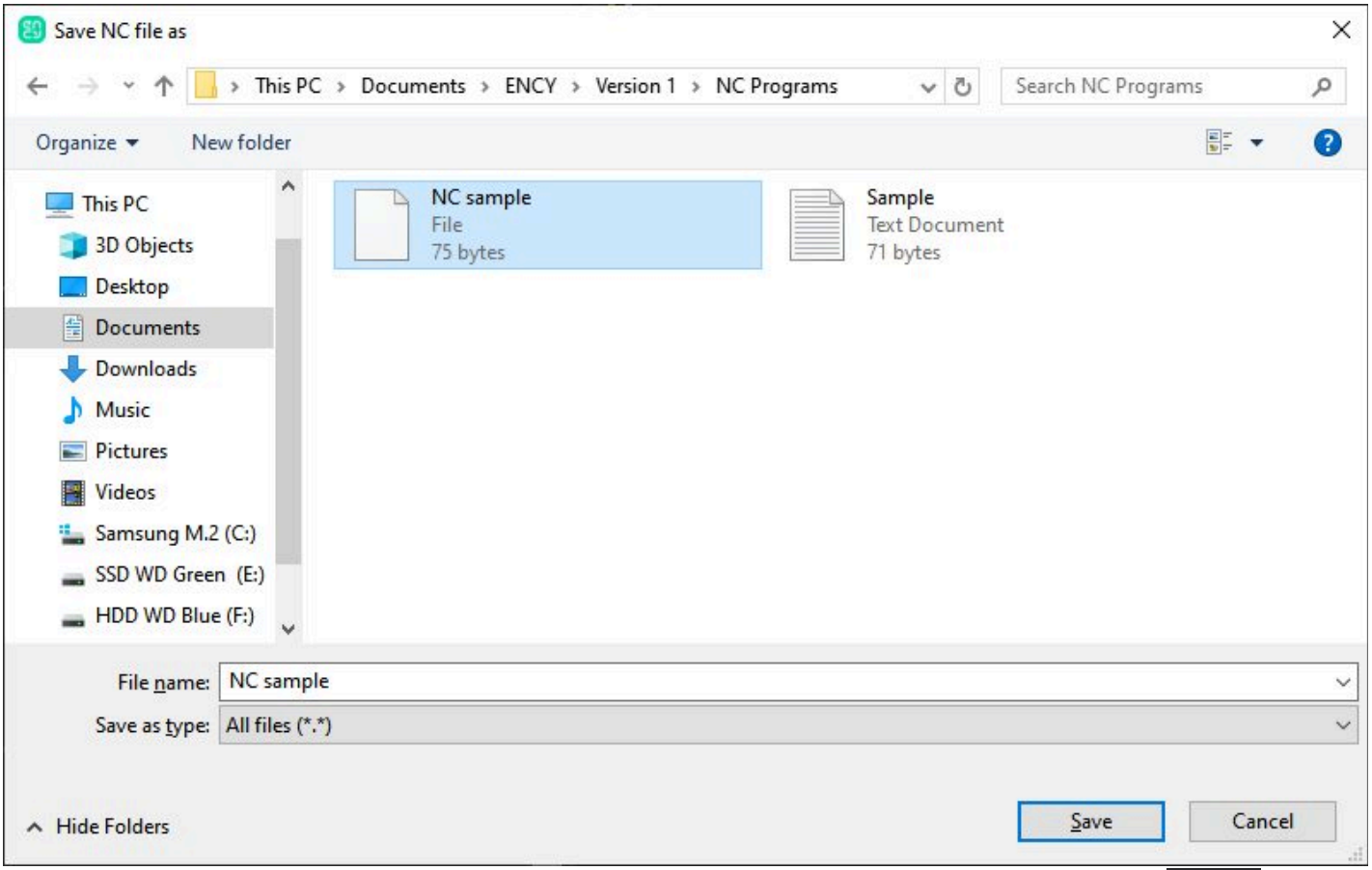
Displaying the name of the tab in bold and also the * symbol at the end of its name during editing indicates that there is a change between the last text state at the moment of opening or saving the NC file and its current representation on the screen.

Tab name	
In the absence of changes	In case of changes
Drill cycles	Drill cycles*

The * symbol disappears, and the text of the tab name takes the usual form if:

- The changes you make will be saved by clicking on the button Save to NC file. If the text of the NC program was written manually and has not been saved before, you will be asked to select the directory and file name under which it should be saved. Otherwise, you can choose to overwrite the

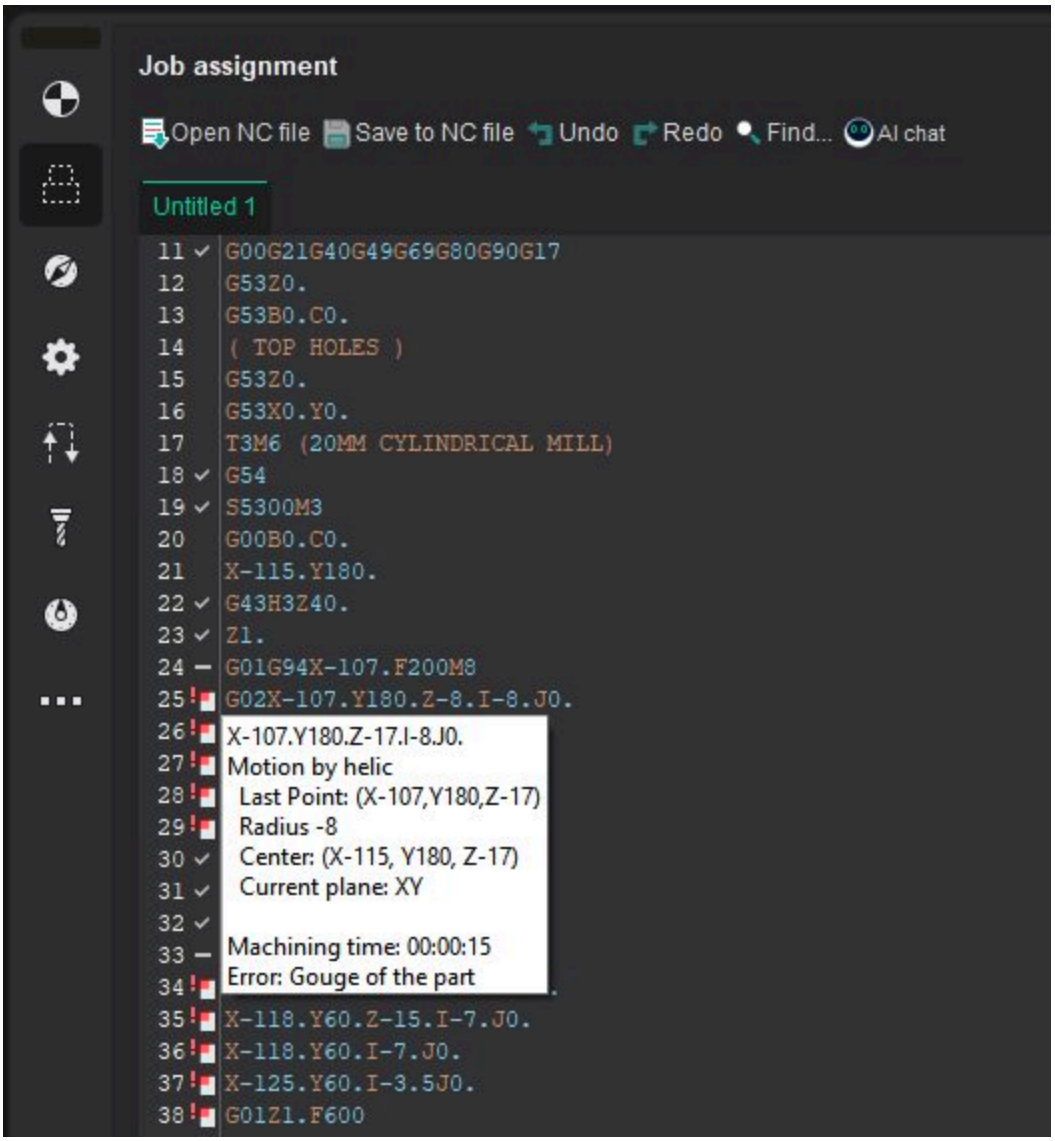
current file or save the NC text in a file with a new name.



- All changes made since the last state savings will be canceled by pressing the button Undo or using the hot key Ctrl + Z.

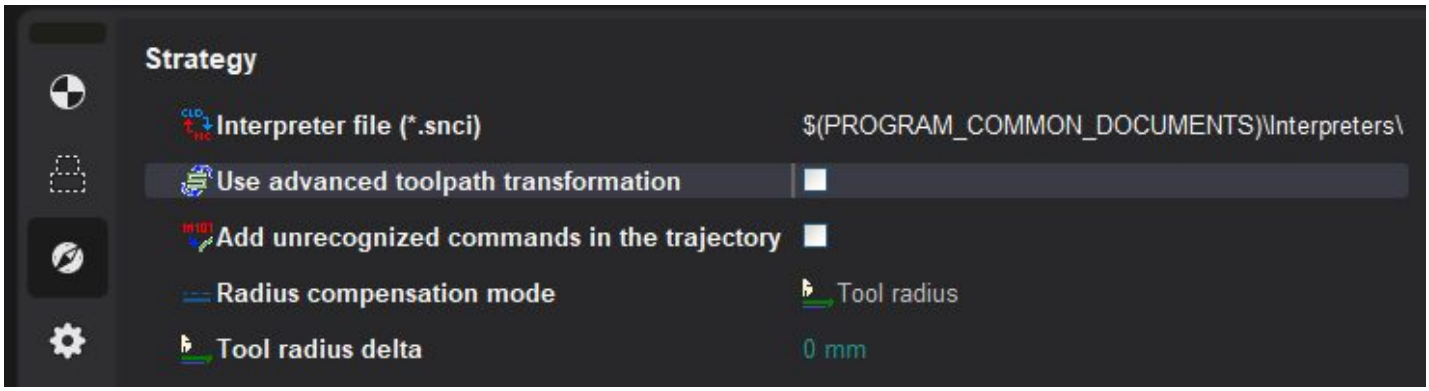
Note: *If the changes made are accidentally canceled, they can be returned by pressing the button* Redo *or using the hotkey Shift + Ctrl + Z.*

After a slight delay of the mouse pointer over the line with the NC program text, a popup hint shows a description of the associated nodes of the trajectory tree (CLData commands).



Note: The hint can be hidden by moving the mouse pointer slightly or pressing any key, such as [Esc] or one of the navigation keys for the text of the control program: [в†“] or [в†‘].

> Restrictions on displaying the popup hint
>
> Popup hints will only be displayed if the Use advanced toolpath transformation mode is disabled. Since when the mode is enabled, it is impossible to uniquely determine the correspondence of the lines with the text of the NC program and the nodes of the geometric CLData after their conversion from machine CLData.



To the left of the control program text in the built-in text editor, there is a gutter for displaying auxiliary information. If the operation has not yet been calculated, the numbering of the lines in the text entry field is displayed. Otherwise, in addition to the number, the status of all the nodes of the trajectory tree associated with it is displayed for each line. The meanings of the displayed icons are similar to those used for CLData technology commands. Thus, it is possible to unambiguously identify the block of the NC program in which there are erroneous nodes of the tree of the trajectory.

> Display of correspondence between the line with the NC text, the technological command CLData and the section of the trajectory
>
> If, in the Technology mode, click on any line of the NC program in the built-in text editor for the calculated G-code based operations, and then switch to the Simulation mode, the first suitable node of the technology command will be selected CLData and highlighted the corresponding toolpath section in the graphics window, if there is one, of course. The reverse is also true: when you left-click on a toolpath section or a technological node of the CLData command in the Simulation mode, the first matching line associated with it will be highlighted when you switch to the Job assignment options parameters panel in the Technology mode (in the case of a tie for this node CLData commands).

Keyboard shortcuts for working with NC text

The following set of keyboard shortcuts for working with text is supported:

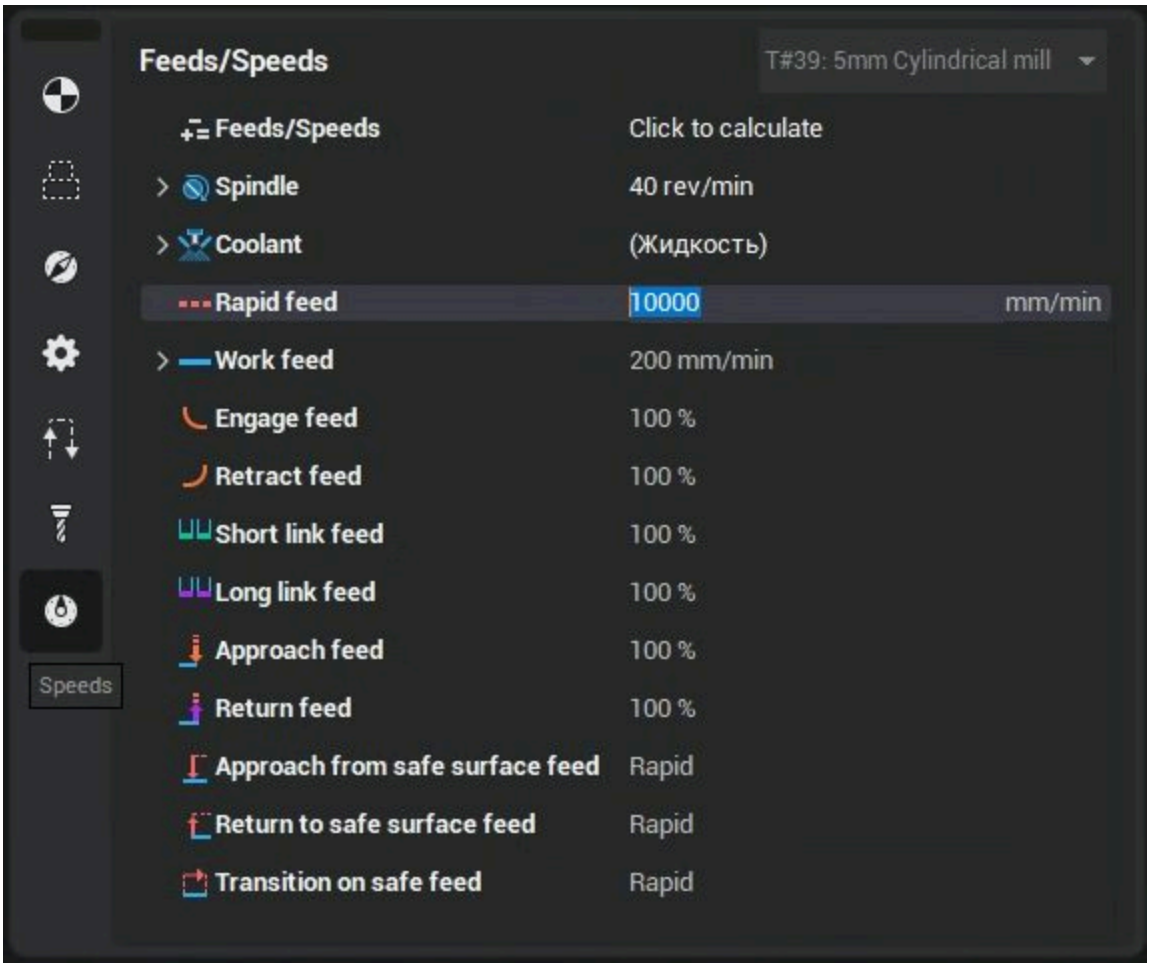
Key strokes	Perform action
←	Move cursor left one char
→	Move cursor right one char
↑	Move cursor up one line
↓	Move cursor down one line
Ctrl + ←	Move cursor left one word
Ctrl + →	Move cursor right one word
Home	Move cursor to beginning of line
End	Move cursor to end of line
Page Up	Move cursor up one page
Page Down	Move cursor down one page
Ctrl + Page Up	Move cursor to top of page
Ctrl + Page Down	Move cursor to bottom of page
Ctrl + Home	Move cursor to absolute beginning
Ctrl + End	Move cursor to absolute end
Shift + ←	Move cursor left one char with text selection
Shift + →	Move cursor right one char with text selection
Shift + ↑	Move cursor up one line with text selection
Shift + ↓	Move cursor down one line with text selection
Shift + Ctrl + ←	Move cursor left one word with text selection
Shift + Ctrl + →	Move cursor right one word with text selection

Key strokes	Perform action
Shift + Home	Move cursor to beginning of line with text selection
Shift + End	Move cursor to end of line with text selection
Shift + Page Up	Move cursor up one page with text selection
Shift + Page Down	Move cursor down one page with text selection
Shift + Ctrl + Page Up	Move cursor to top of page with text selection
Shift + Ctrl + Page Down	Move cursor to bottom of page with text selection
Shift + Ctrl + Home	Move cursor to absolute beginning with text selection
Shift + Ctrl + End	Move cursor to absolute end with text selection
Ctrl + A	Select entire contents of editor, cursor to end
Ctrl + C Ctrl + Insert	Copy selection to clipboard
Ctrl + ↑	Scroll up one line leaving cursor position unchanged
Ctrl + ↓	Scroll down one line leaving cursor position unchanged
Insert	Toggle insert/overwrite mode
Shift + Ctrl + N	Normal selection mode
Shift + Ctrl + C	Column selection mode
Shift + Ctrl + L	Line selection mode
Shift + Ctrl + B	Go to matching bracket
Backspace Shift + Backspace	Delete last char
Delete	Delete char at cursor
Ctrl + T	Delete from cursor to end of word

Key strokes	Perform action
Ctrl + Backspace	Delete from cursor to start of word
Shift + Ctrl + Y	Delete from cursor to end of line
Ctrl + Y	Delete current line
Enter Shift + Enter Ctrl + M	Break line at current position, move caret to new line
Ctrl + N	Break line at current position, leave caret
Ctrl + Z Alt + Backspace	Perform undo if available
Shift + Ctrl + Z Shift + Alt + Backspace	Perform redo if available
Ctrl + X Shift + Delete	Cut selection to clipboard
Ctrl + V Shift + Insert	Paste clipboard to current position
Shift + Ctrl + I	Indent selection
Shift + Ctrl + U	Unindent selection
Tab	Add indent
Shift + Tab	Remove indent

Rapid feed

When calculating operations based on the G-code in robots, it is necessary to set the rapid feed rate, which is specified in the "Feeds/Speed" -> "Rapid Feed" (the default value is set to 10,000 mm/min).



Note: Usually, robotic control programs do not have a specific rapid feed command, as implemented in lathes or milling machines with the G00 command. All movement commands are specified with a feed value. To ensure correct calculation and rendering of the robot's movement trajectory, please set the rapid feed value.

Operations which requires adaptation

The operations which included into this section requires individual adaptation to the particular type of equipment of a customer.

Available only by request.

Development performed on the basis of provided by the Customer technical specifications of equipment and technological parameters of the machining process.

Heat Treatment

The following operations are possible

- Laser heat treatment;
- Gas-plasma heat treatment;
- Another kind of heat treatment processes.

Note: option must me licensed separately.

Welding

The following operations are possible

- «Laser welding»;
- «Arc welding»;
- Another kind of welding.

Note: option must me licensed separately.

Cladding

The following operations are possible

- Laser cladding
- Arc weld cladding
- Another kind of additive technologies

Note: option must me licensed separately.

Jet Cutting

The following operations are possible

- «Laser cutting»
- «Hydro cutting»
- «Plasma cutting»

Note: option must me licensed separately

Teamcenter Integration Module

The description of the Teamcenter integration module is located at this link Teamcenter PLM Integration Module.

Appendix

This section contains reference materials on the operations and CAM system features matrices, the list of interpreters, getting a trial license, bug report best practices, license manager authorization troubleshooting, and additionally installed software.

Operations matrix

Configurations Operations	Wire EDM	Lathe	Cutting 2D	Cutting 5D	Cutting 5D Advanced	2.5X Mill	3X Mill	3X Mill Advanced	Rotary	5X Mill	5X Mill Advanced
Structure											
Setup stage	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Part	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Copy of part	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Multiply Group	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Lathe											
Lathe facing		✓				○	○	○	○	○	○
OD roughing		✓				○	○	○	○	○	○
ID roughing		✓				○	○	○	○	○	○
OD finishing		✓				○	○	○	○	○	○
ID finishing		✓				○	○	○	○	○	○
OD Grooving		✓				○	○	○	○	○	○
ID Grooving		✓				○	○	○	○	○	○
Face grooving		✓				○	○	○	○	○	○
OD Adaptive turning						○	○	○	○	○	○
OD Threading		✓				○	○	○	○	○	○
ID Threading		✓				○	○	○	○	○	○
Profile threading		✓				○	○	○	○	○	○
Lathe part-off		✓				○	○	○	○	○	○
Part-off with takeover		✓				○	○	○	○	○	○
Lathe hole machining		✓				○	○	○	○	○	○
Robotic turning +			○+	○+	○+					○+	○+
Holes											
Hole machining			✓	✓	✓	✓	✓	✓	✓	✓	✓
FBM Mill			✓	✓	✓	✓	✓	✓	✓	✓	✓
2D											
Pocketing						✓	✓	✓	✓	✓	✓
Engraving						✓	✓	✓	✓	✓	✓
2D contouring			✓	✓	✓	✓	✓	✓	✓	✓	✓
Basic											
Face milling						✓	✓	✓	✓	✓	✓
Roughing waterline						✓*	✓	✓	✓	✓	✓
Finishing waterline							✓	✓	✓	✓	✓
Roughing plane							✓	✓	✓	✓	✓
2.5D contouring			✓	✓	✓	✓	✓	✓	✓	✓	✓
Flat land						✓	✓	✓	✓	✓	✓
3D contouring				✓	✓		✓	✓	✓	✓	✓
Advanced											
Combo Finish							✓	✓	✓	✓	✓

Configurations Operations	Wire EDM	Lathe	Cutting 2D	Cutting 5D	Cutting 5D Advanced	2.5X Mill	3X Mill	3X Mill Advanced	Rotary	5X Mill	5X Mill Advanced
Surfacing								✓**	✓***	✓	✓
Scallop								✓	✓	✓	✓
Helical								✓	✓	✓	✓
Morph								✓	✓	✓	✓
Undercut waterline								✓	✓	✓	✓
Swarf											✓
5D roughing waterline											✓
5D by meshes										✓	✓
6D contouring				✓	✓					✓	✓
Rotary											
Rotary waterline									✓	✓	✓
Rotary roughing									✓	✓	✓
Rotary finishing									✓	✓	✓
Morph 4D***									✓	✓	✓
4D surfacing***									✓	✓	✓
4D contouring***									✓	✓	✓
Turbomachinery Milling											
Impeller Machining										○	✓
Blade Machining										○	✓
Rest machining											
Corners cleanup								✓	✓	✓	✓
Pencil								✓	✓	✓	✓
Chamfering				✓	✓	✓	✓	✓	✓	✓	✓
Legacy operations											
Finishing plane							✓	✓	✓	✓	✓
Optimized plane								✓	✓	✓	✓
Complex								✓	✓	✓	✓
Cutting											
Jet cutting			✓	✓	✓	✓	✓	✓	✓	✓	✓
Jet cutting 5D				✓	✓					✓	✓
Knife cutting 2D			○	○	○	○	○	○	○	○	○
Knife cutting 6D				○	○	○	○	○	○	○	○
Disc tool											
Disc roughing				○	○			○	○	○	○
Disc cutting 2D				○	○	○	○		○	○	○
Disc cutting 6D				○	○					○	○
Wire EDM											
Wire EDM					○	○	○		○	○	○
Additive											
Area cladding							○	○	○	○	○
Curve cladding							○	○	○	○	○

Configurations Operations	Wire EDM	Lathe	Cutting 2D	Cutting 5D	Cutting 5D Advanced	2.5X Mill	3X Mill	3X Mill Advanced	Rotary	5X Mill	5X Mill Advanced
Cladding 3D							○	○	○	○	○
Cladding 5D										○	○
Planar slicing				○	○	○	○	○	○	○	○
Non-planar slicing				○	○				○	○	○
Welding											
Welding +			○+	○+	○+					○+	○+
Spray											
Painting +				○+	○+					○+	○+
Auxiliary operations											
Group	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Auxiliary operation	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
G-code operations		○	○	○	○	○	○	○	○	○	○
Move part											
Point Pick and place			○	○	○					○	○
Bar feeding		✓				✓	✓	✓	✓	✓	✓
Sub spindle working		✓				✓	✓	✓	✓	✓	✓
Turn take over		✓				✓	✓	✓	✓	✓	✓
Place to next stage			○	○	○					○	○
Probing											
Tool probing			✓	✓	✓	✓	✓	✓	✓	✓	✓
Part probing			✓	✓	✓	✓	✓	✓	✓	✓	✓
Turn tool probing		✓				✓	✓	✓	✓	✓	✓
Turn part probing		✓				✓	✓	✓	✓	✓	✓

✓ — included in the configuration

○ — available as an additional option

- "Robot+" is needed

*limited to machining pockets with vertical walls only

**limited to generate only 3-axis toolpath

***limited to generate up to 4-axis toolpath *** *operations available depending on the milling or turning configuration

Combo Finish in 3X Mill: Plane, Waterline, and Plane + Waterline strategies only; Scallop machining is not included. The full operation is included in 3X Mill Advanced, Rotary, 5X Mill, and 5X Mill Advanced.

Cutting 5D Advanced: Includes all features of Cutting 5D and advanced 6D contouring with Feature Recognition, Support Generator, and NC cycle output. These advanced features are available in Cutting 5D, 5X Mill and 5X Mill Advanced with the Laser Cutting PRO option.

Turbomachinery PRO: Includes Impeller Machining and Blade Machining. Available as an option for 5X Mill and included in 5X Mill Advanced.

Adaptive Tilting: Automatically avoids holder collisions by tilting the tool along two axes. It is available in 3-axis operations; converting the toolpath into a 5-axis path requires a 5X Mill license.

CAM system features matrix

Configuration	Wire EDM	Lathe	Cutting	Cutting 5D	Cutting 5D Advanced	2.5x Mill	3x Mill	3x Mill Advanced	Rotary	5x Mill	5x Mill Advanced
Network license	○										
Types of machining											
3+2						✓	✓	✓	✓	✓	✓
Turn XZCY						○	○	○	○	○	○
Turn XZCYB									○	○	○
WireEDM						○	○	○	○	○	○
Features											
SOLIDWORKS Reader	○	○	○	○	○	○	○	○	○	○	○
Laser Cutting Pro				○	✓					○	○
G-code based simulation		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Adaptive						○	○	○	○	○	○
Robot+**			○	○	○					○	○
Multichannel						○*	○*	○*	○*	○**	○**
Nesting	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓
Additional operations											
G-code based operations			○	○	○	○	○	○	○	○	○
Turbomachinery Pro										○	✓
Additive Planar 3D*							○	○	○	○	○
Additive Non-planar 5D										○	○
Disc Roughing							○	○	○	○	○
Disc cutting 2D			○	○	○	○	○	○	○	○	○
Disc cutting 6D				○	○					○	○
Knife cutting 2D			○	○	○	○	○	○	○	○	○
Knife cutting 6D				○	○					○	○
Welding										○	○
Spraying / Painting										○	○
Robot Turning										○***	○***

✓ — available

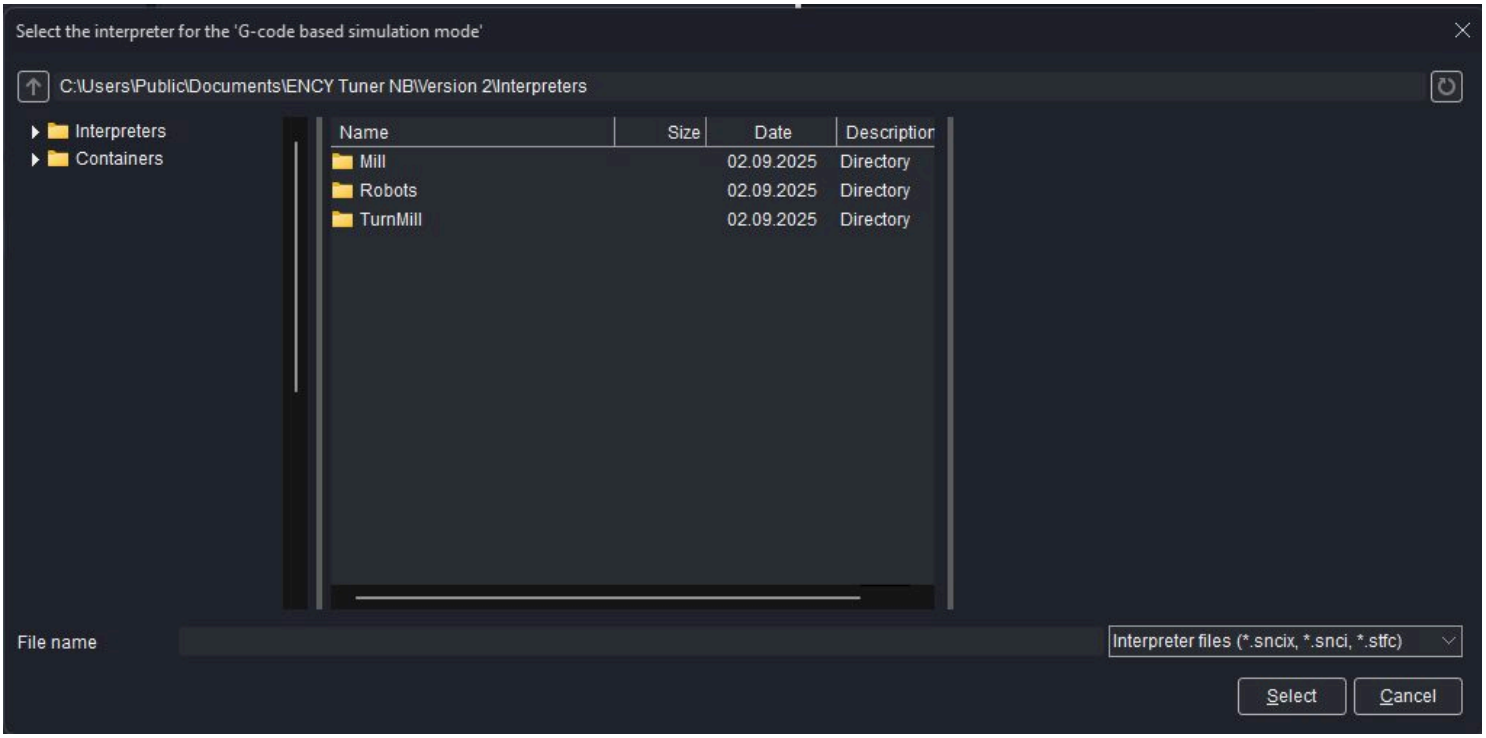
○ — option (need additional license)

○* — suggested for Mill-Turn configurations, including Milling + Turn XZCY or Turn XZCYB

○** — suggested for Mill-Turn configurations or Robotic configurations including Robot+

○*** — Suggested for Upgrade license from ENCY Robot configuration. Lock Turn XZCYB in case of migration

List of interpreters



Application Area:

Pre-built interpreters are available in the system. Some are available under a general license, while others require specific licenses. You can see the full list at the link. Details

Getting trial license

Before purchasing our software, we offer you a trial license so that you can test our product on your device and machine.

Step-by-step guide

1. First you need to go to the main page of the site encycam.com
2. Next, click the Sign in button located in the upper-right corner of the site.
3. Follow the appropriate link to fill out the registration form.
4. Fill out the registration form by providing the following information:

- First name
- Last name
- Email
- Country
- Password

5. Check the correctness of the entered data and click the Register button.
6. After successful registration, you will be automatically redirected to the main page of the site encycam.com. icon should appear in the upper right corner of the site.
7. To download the program distribution, you need to go to the Downloads → Software Products.
8. And click the Download (.exe) button.
9. Once the download is complete, run the downloaded file to begin the program installation process.
10. Complete the installation of the program by following the on-screen prompts provided.
11. After installation is complete, click the Run button in the program installer window.
12. Click the Sign In button in the License Manager window that opens.
13. Enter the email address and password you specified during registration and click the Sign In button.
14. Wait a few seconds while authorization occurs.
15. Complete the login in your browser by clicking the Open in desktop app button.
16. Close the browser tab and return to the application.
17. The trial license should appear in the License Manager list.
18. Click the Run ENCY button.

> License information
>
> To find out more information about the CAM system license
>
> 1. Click on the program icon in the upper left corner of the screen
> 2. Select the Version menu item from the drop-down list
>
> The new window will display the following information:

- License ID
- Protection
- Remaining time
- Dealer information

Bug report best practice

Before making a bug report, make sure that you are using the available latest CAM system version.

If not, update first.

If the problem occurs with the latest version, then giving is as much information as possible is highly appreciated.

More info means more probability that we will be able to fix the problem. And avoid time waste on the email correspondence.

So, bug report best practice consists of:

- Update CAM system first!
- Text description of the problem: what exactly it is, and what steps lead to its occurrence.
- Screenshots showing the problem.
- Video screencast (if screencast is large, use Google Docs, Dropbox or any other cloud storage service and provide a URL)
- Attach CAM system project and logs if possible (attached automatically when creating a ticket from the program window).

Check your email from time to time. If we need additional information, the service desk will send you a notification.

Troubleshooting with authorization in the license manager

Problem: The Sign button is unavailable in the License Manager window.

Solution:

1. Check the accessibility of the web resources webservices.encycam.com and encycam.com using command-line commands: a) *ping webservices.encycam.com* b) *ping encycam.com* If successful, the service response time will be returned. Otherwise, resolve the connectivity issue to these services with your local system administrator.
2. In the default browser, execute the request: *https://webservices.encycam.com/api/v1/status* If the licensing service is accessible, the response should be: *{"status":"OK"}* Any other response will indicate a connection error with the licensing service. Fix with your system administrator network settings according to the returned error.
3. Repeat the request in the default browser (Edge or Internet Explorer), which uses system-wide proxy/network settings instead of its own configurations, as the License Manager relies on these.
4. Run the command in the command line: *curl "https://webservices.encycam.com/api/v1/status"* If successful, the response should be: *{"status":"OK"}.* If an error code is returned, work with your system administrator to adjust network settings accordingly.
5. Check that in the network settings IPv4 protocol is active and IPv6 is disabled.

Problem: The License Manager fails to start.

Solution:

Close the License Manager and delete the following directory: *%temp%.net* Restart the License Manager and verify if the issue persists.

Problem:

Unable to identify the issue, or the solutions above did not resolve it.

Solution:

Send a message to technical support at: support@encycam.com Include a description of the problem, reproducible steps (if possible), and attach the archived file: *"%temp%\SCLicenseManagerLog.txt"*

Additional software and installation changes

The installer sets up the software components required by the CAM system and configures their integration with Windows.

Required software

Component	Purpose
HASP drivers	Enable the CAM system to use HASP hardware license keys.
Microsoft Visual C++ Redistributable	Provides runtime libraries required by application components built with Microsoft Visual C++.
Microsoft .NET 10 Runtime	Runs application components built for .NET 10.
Microsoft ASP.NET Core 10 Runtime	Runs web and server components built for ASP.NET Core 10.

Changes to Windows

During a standard installation, the installer makes the following changes:

- Windows registry. Registers supported file types and records application installation details.
- File Explorer integration. Registers the application's shell extension.
- NVIDIA graphics settings. Creates an application profile for using the high-performance GPU.
- Windows Firewall. Adds outbound rules for the main application and related components, including the help system, cloud services, start page, and project library.

Portable installation skips the system configuration steps that require administrator rights, including the registry, shell extension, NVIDIA profile, and firewall changes listed above.