SprutCAM X

USER MANUAL

SprutCAM Tech Ltd.
9, Aiolou and Panagioti Diomidous,
3020, Limassol, Cyprus

SprutCAM Tech Ltd.
9, Aiolou and Panagioti Diomidous,
3020, Limassol, Cyprus

SprutCAM Tech Limited

SprutCAM X User Manual

Generated on 27/02/2024

SprutCAM Tech Ltd.
9, Aiolou and Panagioti Diomidous,
3020, Limassol, Cyprus

Contents

SprutCAM X 17 user manual

1 Introduction to SprutCAM X

Thank you for choosing **SprutCAM X ®** our cutting-edge CAM system. This next-gen platform directly interprets your imported CAD model data, including NURBS representations, without the need for any initial approximations or triangulation. It then auto-generates an efficient toolpath for machining your model. This toolpath can be visually simulated to identify any issues with your chosen machining parameters. Once approved by the user, the system produces an NC program from a comprehensive list of available Posts. Custom Posts can also be created to meet specific needs.

SprutCAM Xis versatile enough to handle models of varying complexities, whether 2D or 3D. It supports program generation for 2-axis and lathe setups, 3-axis, and even up to 4 or 5-axis milling machines. The software also facilitates 4-axis and 5-axis machining for turn-milling centers and supports 2-axis, 2-axis with taper, and full 4-axis EDM. Users can choose from a wide array of machining methods and strategies, and even set the desired level of accuracy. Best of all, the system operates smoothly on any standard PC without demanding excessive computational power.

The most important features of the system can be distinguished in the following way:

1. Ease of Use – The system is very easy to use and is logically well organized into four main modes of working which can easily be selected from the main window by clicking on the relevant tab: 3Dmodel (import and preparation of geometric model); 2D Geometry (for 2D drawing); Machining (to generate machining processes); and Simulation (a photo realistic view of all machining, including tool and stock).
2. Import of Many Formats – Advanced ability to import and transform 2D and 3D geometric models prepared in any CAD system, and then transferred into **SprutCAM X** via IGES, DXF, STL, VRML, PostScript, STEP, 3DM, or SGM file format. Within **SprutCAM X** the model can be transformed in many ways (scaled, rotated, transposed etc.), and any or all parts constituting it can be machined in any desired sequence, while gaps and overlaps between these parts are properly processed.
3. 2D Drafting – The built-in 2D parametric drafting tools allows the creation of objects in any plane, and these can be referenced to the coordinates of the 3D model. In addition, the 3D model can be projected onto a plane. Patterns and text for engraving or pocketing can also be created within the system. All these objects can be used to define the part, fixtures, workpiece or job assignment of a cycle.
4. **Very Sophisticated Machining cycles** – The machining process can be set up easily and is made up of a sequence of available operations which are chosen by the user from a long list; i.e. roughing, finishing, rest milling, hole drilling, engraving etc. Within each operation the user chooses the parameters that should be applied; i.e. waterline, plunge or drive cutting modes, depth of cut, step-over distance, scallop height, cutting tool type and dimensions, cutting speeds, conventional or climb cutting etc. Any of these parameters can be revisited and modified without upsetting the whole operation, and if so desired, the system can set these parameters by default. The resulting machine process is very accurate and efficient with minimum loss of time, as all unnecessary tool movements can be eliminated. It is suitable for both traditional as well as high speed cutting of any material.
5. Actual workpiece state considering – The first machining cycle uses the start workpiece, to generate the toolpath. After that, the workpiece form is updated. So the next cycle uses the updated workpiece to generate its own toolpath.
6. Photo-realistic simulation with collision detection – The user can see exactly how the part is going to be machined, either in a step by step mode or variable speed continuous mode, as if a videotape is being played back. He can choose the color of the various tools used, the stock material and the intended final shape, for better understanding of the operation. Should it be desirable to change any part or parameter of the cycle, it is easy to go back to the machining operation and modify it, and then return to the simulation. Toolpaths followed by each tool can also be seen in different colors.
7. Postprocessor – Once the machining cycle is accepted, the program can automatically generate an NC program to suit the user's machine or CNC system. Besides the long list of available Posts and willingness of SprutCAM Tech to develop Posts to the user's requirements

and requests, it is also possible for the user to generate new posts or modify existing ones by using the inbuilt < Postprocessors generator >.

With the powerful **SprutCAM X** system the user can confidently undertake fast machining of very accurate parts, even if it would be very complex 3D-models, or simple engraving or pocketing, by using of any material. Typically, it can be used to machine punches, spark erosion electrodes, plastic molding, machine parts, decorative elements, nameplates etc. Because it is truly Windows based and very easy and self-explanatory, you will be up and running with minutes of installing the software, and we encourage you to follow the supplied tutorials which teach you and demonstrate how easy it is to use the system.

See also:

Base configurations

System requirements

Standard package

System installation and launch

System files

Technical support

1.1 System requirements

The minimum and recommended PC configurations for SprutCAM X:

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®/Quadro® 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

Requirements	**Minimum** (small and medium projects)	**Recommended** (large projects)
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 specification of the computer, the faster the calculations will be performed.

*SprutCAM X is not supported on Apple Macintosh® – based machines. Some customers have shown success in running SprutCAM X 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 SprutCAM X.

** 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 SprutCAM X which are running on the PC. SprutCAM Tech 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 SprutCAM X.

See also:

Introduction to SprutCAM X

1.2 Configurations and options

1.2.1 SprutCAM X configurations

1.2.1.1 Express

The most simple configuration, only single Z level machining.

Available operations: Pocketing, Hole machining, 2D contouring, Flat land, Auxiliary.

1.2.1.2 Cutting

Contains the operations for the 2D milling and cutting. Designed especially for the 2-axis milling machines or cutters.

Available operations: 2D contouring, Hole machining, Jet cutting 2D, Jet Cutting 4D, Auxiliary.

1.2.1.3 **2.5x Mill**

Parts production based on drawings. Drawing can be created with embedded SprutCAM X drawing editor or imported. This configuration does not work with faces.

In combination with "Lathe" option can be used for mill-turn machines programming.

Available operations: 2D contouring, Flat land, Face Milling, Hole machining, Engraving, Pocketing, 2.5D pocketing, 2.5D flat land, 2.5D wall machining, 2.5D contouring, Chamfering, Jet Cutting 2D.

1.2.1.4 **3x Mill Entry**

Configuration for parts production using a 3-axis milling machine and 3D model of the part. Best for production parts with simple curved surfaces.

In combination with "Lathe" option can be used for mill-turn machines programming.

Available operations:

All operations form 2.5x mill: 2D contouring, Flat land, Face Milling, Hole machining, Engraving, Pocketing, 2.5D pocketing, 2.5D flat land, 2.5D wall machining, 2.5D contouring, Chamfering, Jet Cutting 2D.

And: Roughing waterline, Roughing Plane, 3D contour, Finishing waterline, Finishing Plane, FBM.

1.2.1.5 **3x Mill Advanced**

Ultimate solution for 3-axis machine programming. Including production of parts with complex curved surfaces, for ex. mold & die.

Available operations:

All operations form 3x mill entry: 2D contouring, Flat land, Face Milling, Hole machining, Engraving, Pocketing, 2.5D pocketing, 2.5D flat land, 2.5D wall machining, 2.5D contouring, Chamfering, Jet Cutting 2D, Roughing waterline, Roughing Plane, 3D contour, Finishing waterline, Finishing Plane, FBM.

And: Morph, Scallop, 3D Helical, Optimized Plane, Complex, 5D surfacing (in 3D mode), Pencil, Corners cleanup.

1.2.1.6 Rotary

Rotary milling programming for 4-axis milling machine.

Available operations:

All operations form 3x mill advanced: 2D contouring, Flat land, Face Milling, Hole machining, Engraving, Pocketing, 2.5D pocketing, 2.5D flat land, 2.5D wall machining, 2.5D contouring, Chamfering, Jet Cutting 2D, Roughing waterline, Roughing Plane, 3D contour, Finishing waterline, Finishing Plane, FBM, Jet cutting, Morph, Scallop, 3D Helical, Optimized Plane, Complex, 4D surface (5D surfacing in 3D mode), Pencil, Corners cleanup.

And: Rotary roughing, Rotary finishing, Rotary Waterline, 4D contour, 4D Morph.

1.2.1.7 5x Mill

SprutCAM X for milling machines at its full power. Capable of programming 5-axis machines with TCPM.

Available operations:

All operations form 3x mill advanced: 2D contouring, Flat land, Face Milling, Hole machining, Engraving, Pocketing, 2.5D pocketing, 2.5D flat land, 2.5D wall machining, 2.5D contouring, Chamfering, Jet Cutting 2D, Roughing waterline, Roughing Plane, 3D contour, Finishing waterline, Finishing Plane, FBM, Jet cutting, Morph, Scallop, 3D Helical, Optimized Plane, Complex, 5D surfacing (in 3D mode), Pencil, Corners cleanup.

Plus:

All operations form Rotary: Rotary roughing, Rotary finishing, Rotary Waterline, 4D contour, 4D surface, 4D Morph.

Plus 5-axis toolpaths: 5D surfacing, 5D contouring, 5D by meshes, 5D cutting.

1.2.1.8 Lathe

Contains the full kit of the lathe operations. Designed for the lathes and turning centers.

1.2.1.9 WireEDM

Contains the operations for 2-axis, 2-axis with taper and full 4-axis wire EDM cutting. Designed especially for the wire EDM machines.

1.2.2 **Options**

1.2.2.1 Turn

Turning operations. Available for all configurations starting from 2.5x mill.

Configurations and machine match table:

Machine	**Configuration**
Lathe Z X	Turn ZXCY
Mill-turn X Z C (Y)	2.5x Mill + Turn ZXCY 3x Mill Entry+ Turn ZXCY 3x Mill Advanced + Turn ZXCY Rotary + Turn ZXCY 5x Mill + TurnXZCY
Mill-turn X Z C Y B	5x Mill + Turn ZXCYB

Multichannel machining is a separate option and not included in "Turn" by default.

1.2.2.2 3+2 **(5-axis index)** machining

This option enables LCS usage and comprehensive approach and retract, as well as links handling for multi-axis machines.

This option is included and turned off into all milling configurations starting from 2.5x mill by default. Users can turn it on when needed.

1.2.2.3 Adaptive SC

SprutCAM X's module for high speed milling.

Full reference.

1.2.2.4 Multichannel

The option enables multichannel machining with user defined synchronization points. Robots are supported. Good for dual channel. Harder to use for 3+ channel machining.

1.2.2.5 G-code based

Option adds support for G-code based operations: "G-code based", "G-code based lathe".

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 use of Wire EDM machining is **not supported**.

Full reference

1.2.2.6 Cladding

The optional module of SprutCAM X which includes operations for additive manufacturing.

Full Reference.

1.2.2.7 Disk tool

This group of operations is designed for sawing and machining materials such as wood, stone and similar with a disk tool.

Full reference

1.2.2.8 Knife cutting

Special operations for knife cutting 2D and 6D.

Full reference.

1.2.2.9 Teamcenter integration

This integration module is designed to obtain the required in creating NC code data for CNC machines from PLM system to Teamcenter and for transferring the NC code, list of processing operation and a statement of the tool to the PLM-system after tool path calculation.

Full reference

1.2.2.10 Robot+

Allows to add feature that make it possible to control 6-axis (articulated) robots when you have only configuration of SprutCAM X for usual machines (milling etc.).

Full reference

1.2.2.11 Welding

It implements the functionality of automatic weld seam geometry calculation.

Full Reference

1.2.2.12 Painting

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).

Full reference

1.2.2.13 ModuleWorks

This module useful for create tool paths for 3, 4, and 5 axis machines, and industrial robots (last only for SprutCAM X : Robots + 5D MW).

Full reference

1.2.2.14 Heat treatment

The following operations are possible

- Laser heat treatment;
- Gas-plasma heat treatment;
- Another kind of heat treatment processes.

See also:

Introduction to SprutCAM X

Features matrix

1.3 Standard package

The boxed package of SprutCAM X ® includes:

1. CD with SprutCAM X ® system.
2. Documentation (PDF file).
3. Electronic key to prevent unauthorized copying (optional – depends on configuration and protection method).
4. License agreement.
5. Package box.

The e-package of SprutCAM X ® includes:

- Executable file for install SprutCAM X ® system.

See also:

Introduction to SprutCAM X

1.4 Program installation and launch

SprutCAM X 17 uses Internet for the installation. The **SprutCAM_X_Setup.exe** setup program scans through the SprutCAM X web server for available modules and downloads them. There is an option to install only required modules skipping the others(they are not downloaded). Successive installer runs use the previously downloaded modules to speed up the installation process.

There is of course a "disk" installer based on the same technology which uses local modules for installation.

SprutCAM X installation on a computer with Internet connection.

1. Download installable file **SprutCAM_X_Setup.exe** from the WEB site.
2. In Windows run installation program (**SprutCAM_X_Setup.exe**)). This can be done, for example, from the Start –> Run menu. In the < Run > dialogue type <Path>:\SprutCAM_X_Setup.exe and click < Ok > or hit < Enter > on the keyboard.
3. The Install program will guide you through various dialogue boxes which will require some input during the installation process, including:
    - End user license confirmation;
    - Choose default, minimal or custom installation (can exclude example project files, importable files, and other auxiliary files);
    - Folder, where the program shall be installed to (by default – "C:\Program Files\SprutCAM Tech\SprutCAM X 17");
    - Wait while installer download all modules;
    - Main menu folder, where the shortcuts for the executable system files will be placed (by default – "SprutCAM X 17");

   To go to the next installation step click the < Next > button on the window.

   After the installation is complete, a window reporting that installation has been successfully completed will appear. Close the window by clicking the Done button. If the user has SprutCAM with the electronic key protection 'option', then before running the system the user will need to insert the key into a spare USB port. To run the program use Start –> Programs –>SprutCAM X 17 –> SprutCAM X. Further, we recommend that a shortcut is created for SprutCAM X on the Desktop or quick launch panel.

SprutCAM X installation from a CD

1. Insert the CD into the CD drive;
2. In Windows run installation program from the CD (**SprutCAM_X_Setup.exe**)). This can be done, for example, from the Start –> Run menu. In the < Run > dialogue type <Drive_letter>:\SprutCAM_X_Setup.exe (e.g. if your CD drive is D, then type D:\SprutCAM_X_Setup.exe) and click < Ok > or hit < Enter > on the keyboard.
3. The Install program will guide you through various dialogue boxes which will require some input during the installation process, including:
    - End user license confirmation;

- Choose default, minimal or custom installation (can exclude example project files, importable files, and other auxiliary files);
- Folder, where the program shall be installed to (by default – "C:\Program Files\SprutCAM Tech\SprutCAM X 17;
- Wait while installer verifies all modules;
- Main menu folder, where the shortcuts for the executable system files will be placed (by default – "SprutCAM X 17");

To go to the next installation step click the < Next > button on the window.

After the installation is complete, a window reporting that installation has been successfully completed will appear. Close the window by clicking the Done button. If the user has SprutCAM X with the electronic key protection 'option', then before running the system the user will need to insert the key into a spare USB port. To run the program use Start –> Programs –> SprutCAM X 17 –> SprutCAM. Further, we recommend that a shortcut is created for SprutCAM X on the Desktop or quick launch panel.

Note : The SprutCAM X installation must be performed with the administrator rights. This version of SprutCAM X requires the administrator rights while the running for addons setup also. If it is impossible to have these rights permanently then it is necessary to create the link to run SprutCAM X with administrator rights or consult in the support center.

Note : In some cases, when installing SprutCAM X with electronic key the user may need to additionally install the electronic key driver. To do so, after the installation process select in Program menu (Start –> Programs –> SprutCAM Tech –> SprutCAM X 17 –> Install Key Driver) Key drivers setup.

Note : If when starting the program it reports: *Electronic key not found!* then check that the key is properly connected to the computer's USB port. If this message appeared during the first run of the program, then the user will probably need to install the electronic key driver (see above).

Electronic key not found!

then check that the key is properly connected to the computer's USB port. If this message appeared during the first run of the program, then the user will probably need to install the electronic key driver (see above).

See also:

Introduction to SprutCAM X

1.5 System files

Main executable files of a system:

- SC.exe – executable system module.
- INP.exe – executable < Postprocessors generator > module.
- SCUpdater.exe – executable program updater module.

CAD Files of this type can be imported into SprutCAM X:

- *.igs, *.iges – file format of Initial Graphics Exchange Specification;
- *.dxf – file of Drawing Exchange Format by Autodesk ™ ;
- *.stl – file format native to the stereo lithography CAD software;
- *.vrl – file format of Virtual Reality Modeling Language;

- *.ps – file in PostScript format;
- *.eps – file in Encapsulated PostScript format;
- *.x_t, *x_b – file of Parasolid ™ kernel file format;
- *.sldasm, *.sldprt - file of native SolidWorks ™ project format.
- *.asm, *.par, *.psm, *.pwd - file of the native SolidEdge ™ project format.
- *.3dm – file in native Rhinoceros ™ project format;
- *.step, *.stp – STEP, modern graphics exchange specification.

- *.stcp – system project files, contain all information about the project: geometrical model, machining operations toolpath etc. Previously used the extension "***.stc**". It will also be supported.
- *.stcx – XML format project files, project is saved to stcp format too.
- *.sto – files, contain information about operations: structure operations in the project, parameters of operations etc.
- *.dxf ; *.eps ; *.ps – files, contain information about 2D geometry.
- *.stl – files, contain information about simulation result.
- *.xml – files, contain information about toolpath for external simulator.
- *.sppx – postprocessors tuning files for the different CNC systems (used by SprutCAM X and the < Postprocessors generator >).
- *.spp, *.inp – postprocessor tuning files of previous versions of SprutCAM X. The old format is supported by the new version of SprutCAM X too. < Postprocessors Generator > can reform the files into the *.sppx format.
- *.stfc – zip-container file that can contain any other files of the system inside (postprocessors, machine schemes, projects etc.). Content can be encrypted and digitally signed.
- *.snci – interpreters tuning files for the different CNC systems (used by SprutCAM X in G-code based simulation mode and in G-code based operations).
- *.dsk , *.cfg – files containing data about the system screen settings. Generate automatically if they are lacking.

Other files and folders, created during the installation, are required for the proper functioning of the program. Their modification or deletion can prevent the proper functioning of the program. Substitution of files is needed only when upgrading the version, and should be performed in strict accordance with the update information supplied.

See also:

Introduction to SprutCAM X

1.6 Technical support

Technical support of the software is carried out by the dealer or one of the representatives of SprutCAM Tech Ltd.

Find your nearest SprutCAM X dealer: sprutcam.com/find-reseller/

If you have any questions or comments regarding SprutCAM X ® refer to

- WEB-page: www.sprutcam.com
- WEB Forum: forum.sprutcam.com
- Technical support service: support@sprutcam.com

Postal address:

1.6.1 SprutCAM Tech Ltd.

9, Aiolou and Panagioti
Diomidous
3020 Limassol
Cyprus

For frequent notification about updates of the current version, and release of new versions of **SprutCAM X ®**, we recommend registering you on the site of our company.

See also:

Introduction to SprutCAM X

2 Brief and to the point

No content in this page. See child topics.

2.1 Ideology of SprutCAM X

SprutCAM X® 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 any part of the full item or the full 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 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, in general, 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 many parameters before. Modification of the order of the machining operations and editing their parameters is possible at any stage of the machining process designing.

SprutCAM X® 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 NC program that removes the workpiece material from outside of the part.

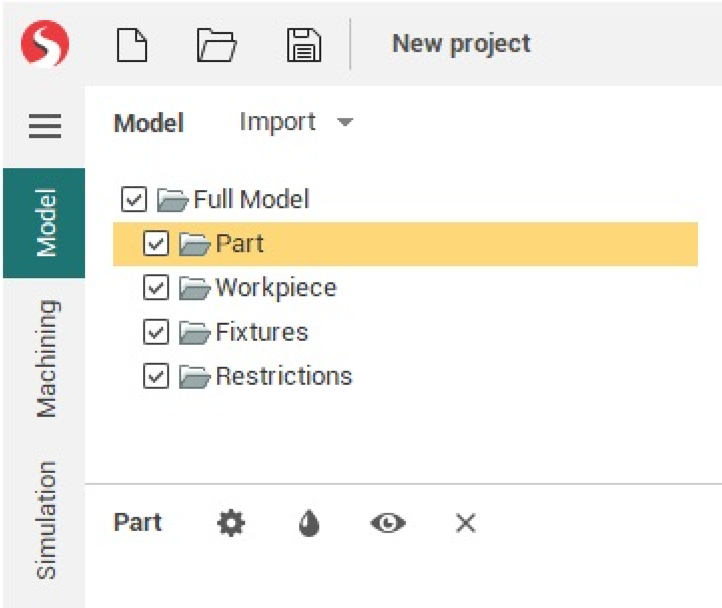
The interface of SprutCAM X® allows the customer to alter any parameter in any order that is required by them. The "active graphics" in many of the dialogues are automatically updated when a customer selects different operations or parameters. This allows to significantly reducing the time needed to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

SprutCAM Tech 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.

2.2 Fast familiarization with the system

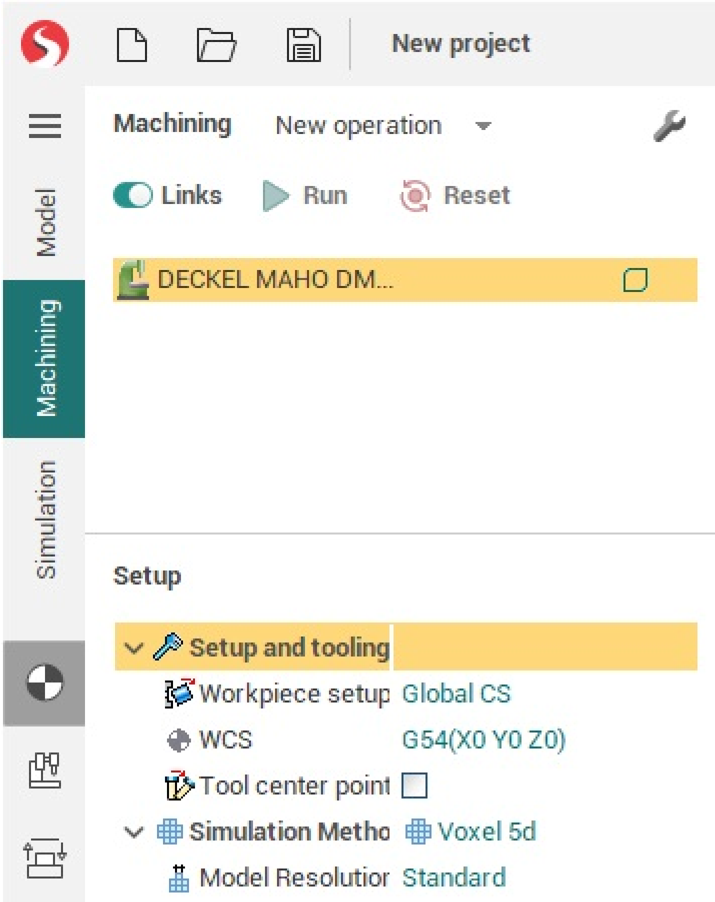
Do the following actions to compute a toolpath and to generate an NC-program in a short term. The typical action sequence allows you to get to know SprutCAM X quickly.

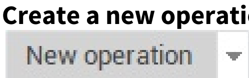
- **Set the** working mode **to <** Model **>** by selection of the same name tab sheet on the main panel.



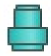
- Import a model **from the geometrical data exchange file**. To do so, just click the 

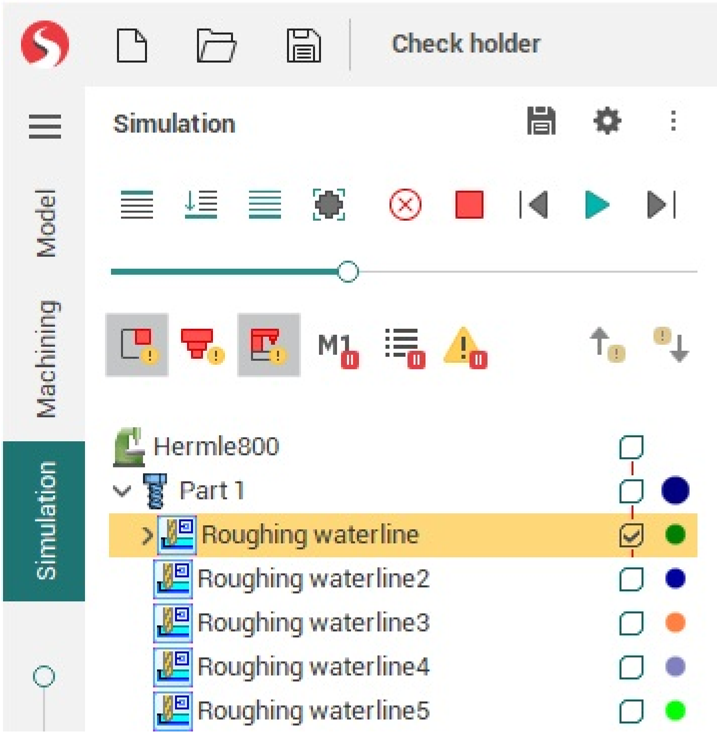
 button and select the file you want in the window. The model import will be processed in the active folder. Therefore, before the import starts, please activate the < Full Model/Part > folder for the part model import, < Full Model/Workpiece > for the workpiece and < Full Model/ Fixtures > for fixtures etc. It is a good idea to make all modifications to the geometrical model before the machining sequence definition because the default parameters value is chosen according to the part and workpiece at the operation creation moment.
- Set the working mode to <Machining> by selection of the corresponding tab sheet on the main panel:

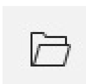


- **Check** the machine type and its parameters ; edit them if it is necessary. The available operation set depends on the machine type. For example, lathe operations only are available for the lathe machine, milling operations only – for the milling machine but the both operation types can be applied for the turning and milling center.
- Form the < Part > the < Workpiece > and < Fixtures > **for the root node of the operations list** if the models 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  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 parameters.** When creating an operation, the program sets < Part > and < Fixtures > as references to models of the same name of 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 machining. < Job assignment > is the full part as default. SprutCAM X 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 turning on < Machining result visibility >  on < Visibility panel >.
- **Check the obtained toolpath block by block** in the < 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.

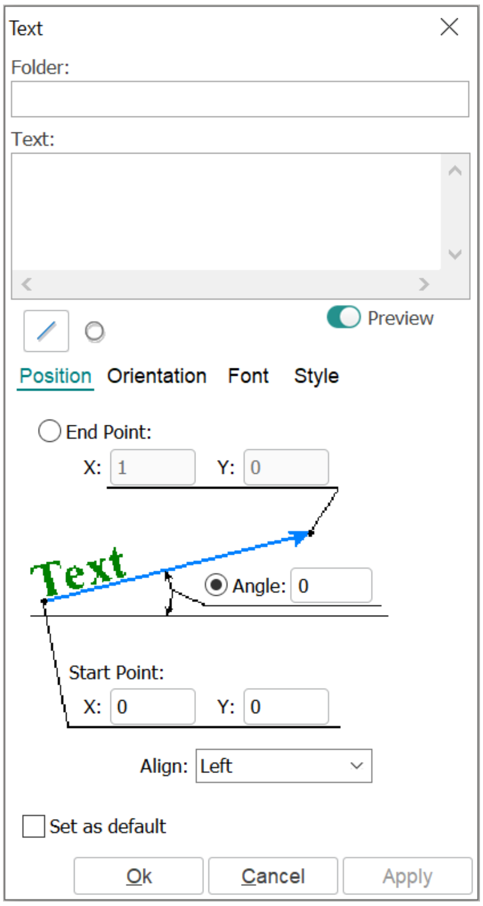
2.3 What's new in SprutCAM X 17

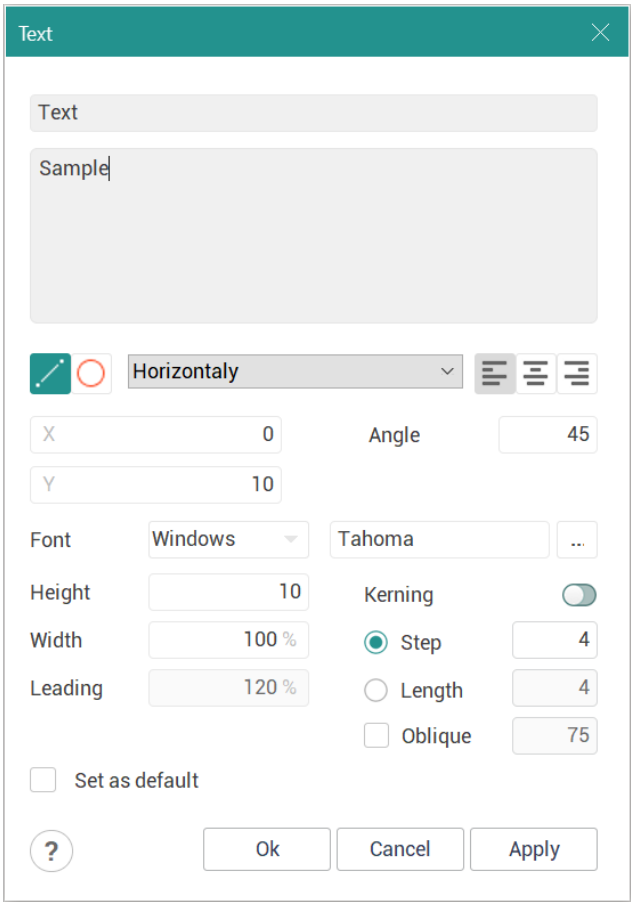
2.3.1 General improvements

2.3.1.1 Application start time reduced

2.3.1.2 Interface - new popup dialogs design.

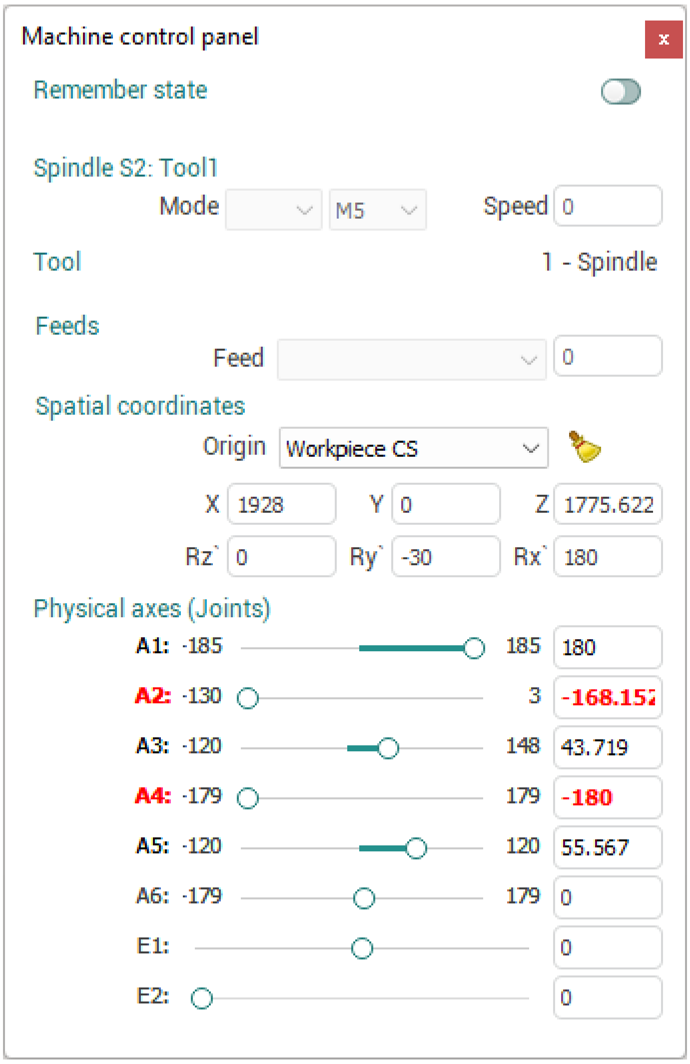
All window design is updated to conform to modern user interface standards and provide straightforward user experience. More examples are shown here.

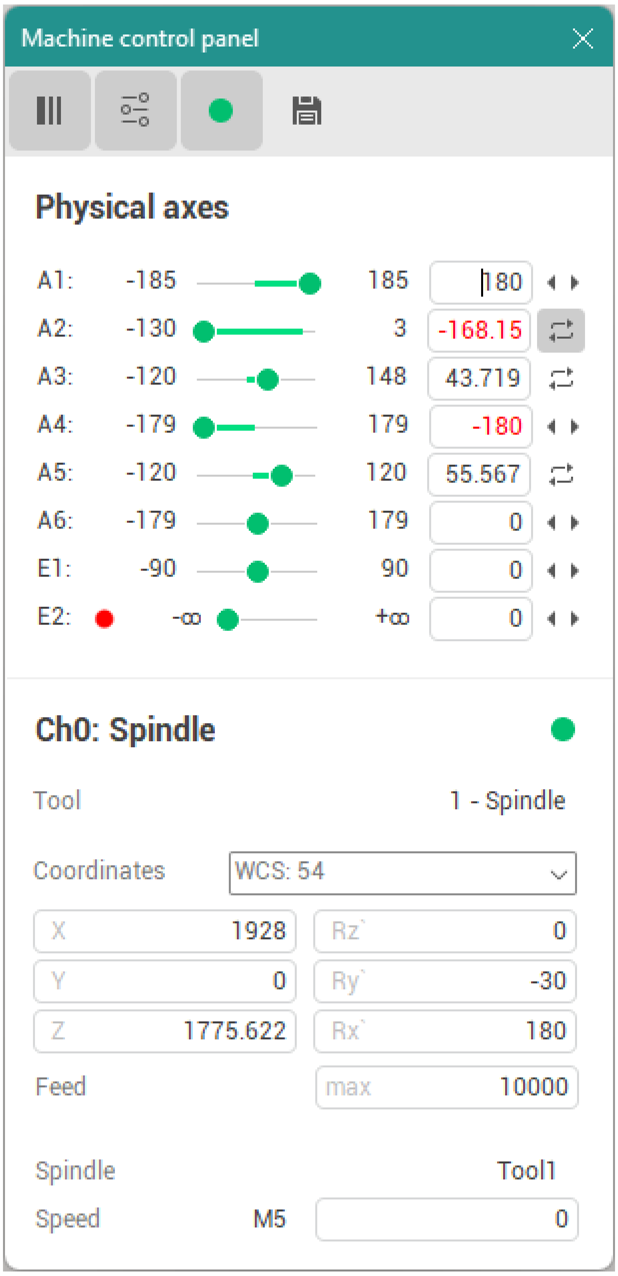




Machine control panel

Axes brakes and robot flips could now be watched using the window.

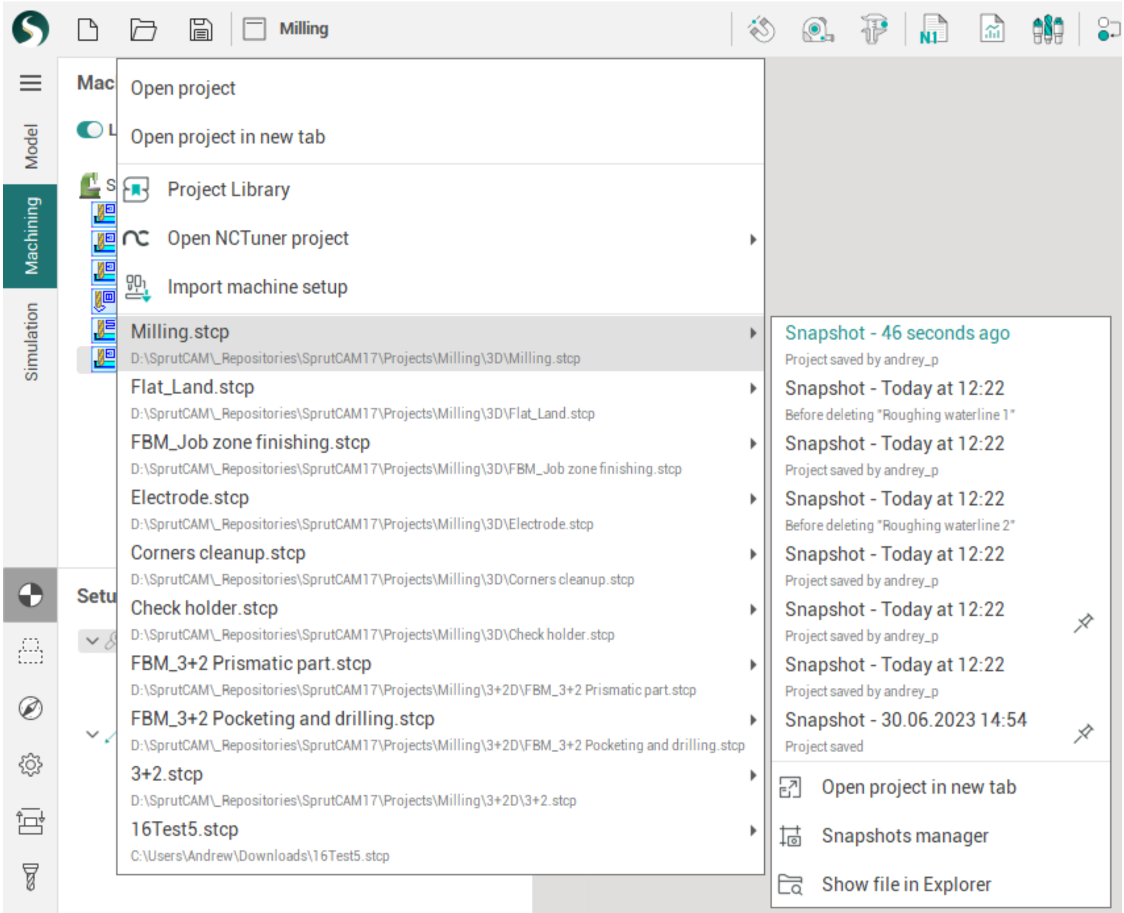

Machine control panel
Remember state
Spindle S2: Tool1
Mode
M5
Speed
0
Tool
1 - Spindle
Feeds
Feed
0
Spatial coordinates
Origin
Workpiece CS
X
1928
Y
0
Z
1775.622
Rz`
0
Ry`
-30
Rx`
180
Physical axes (Joints)
A1: -185
185
180
A2: -130
3
-168.152
A3: -120
148
43.719
A4: -179
179
-180
A5: -120
120
55.567
A6: -179
179
0
E1:
0
E2:
0



2.3.1.3 Project Snapshots

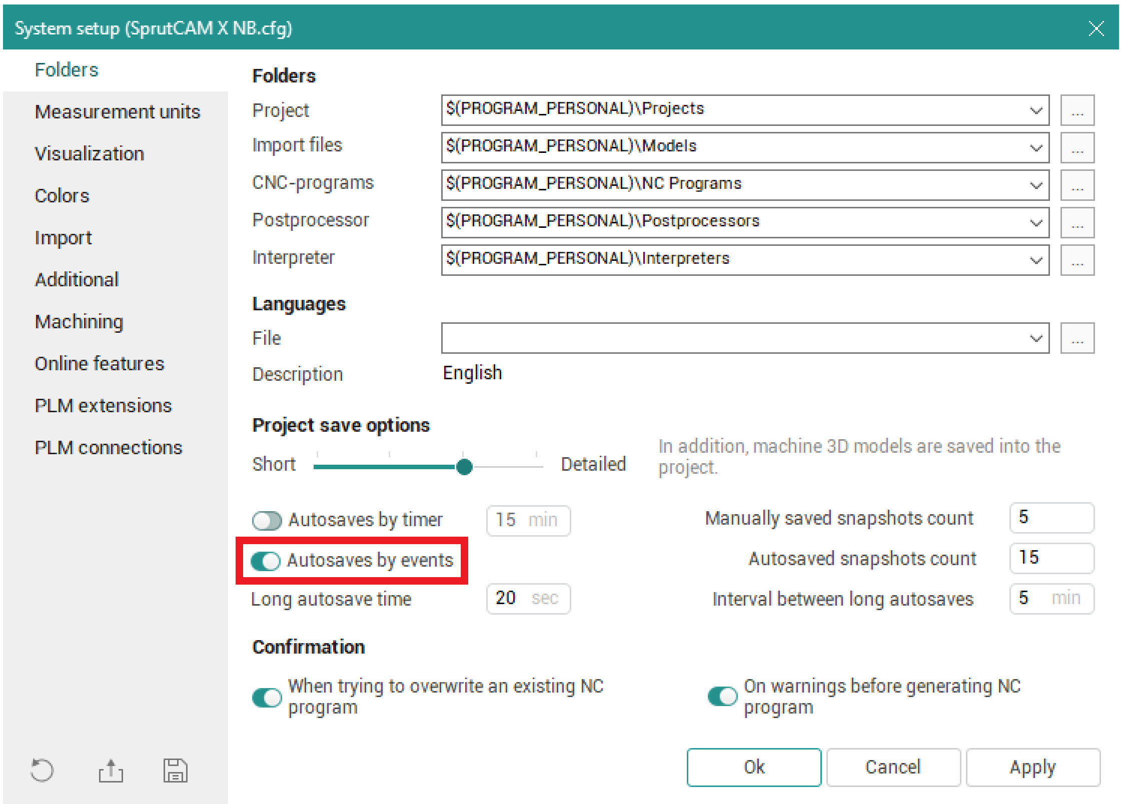
Now you can load snapshots (previously saved/autosaved versions of the project) in one click. The dropdown list of recent projects now contains a submenu where you can see the list of snapshots. You can click any item in list to load it quickly. The current snapshot is highlighted in green.

The pin button prevents automatic deletion of snapshot. The list shows only 10 snapshots. There is a separate Project snapshots manager window if you need more.



Snapshots by events

You can use Autosaves by events. In this case the system will save project during some events: before calculation run, after calculation run, before deleting operation.

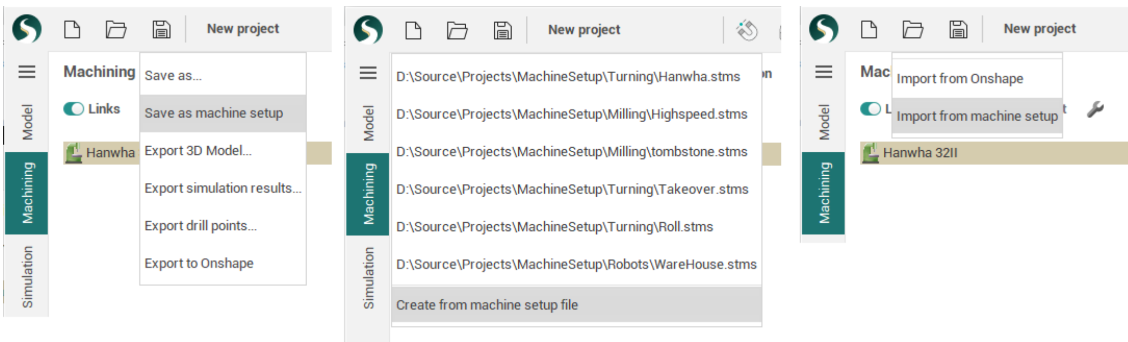


2.3.1.4 New project extension - ".stcp"

The new project extension (**".stcp"**) is used for the new version of the system. The old "**.stc**" extension will also be supported.

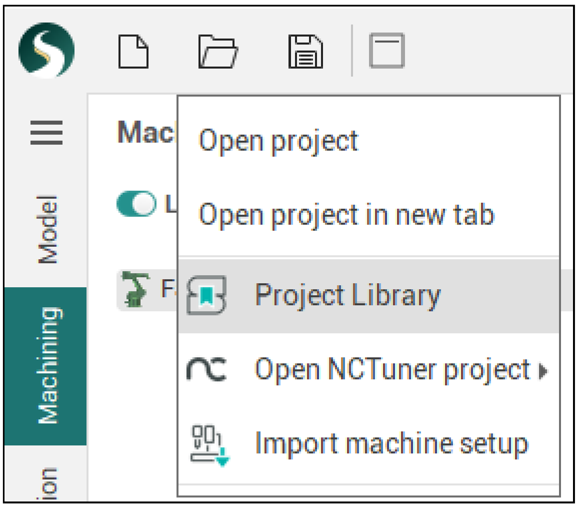
2.3.1.5 Machine setup file to create new projects quicker

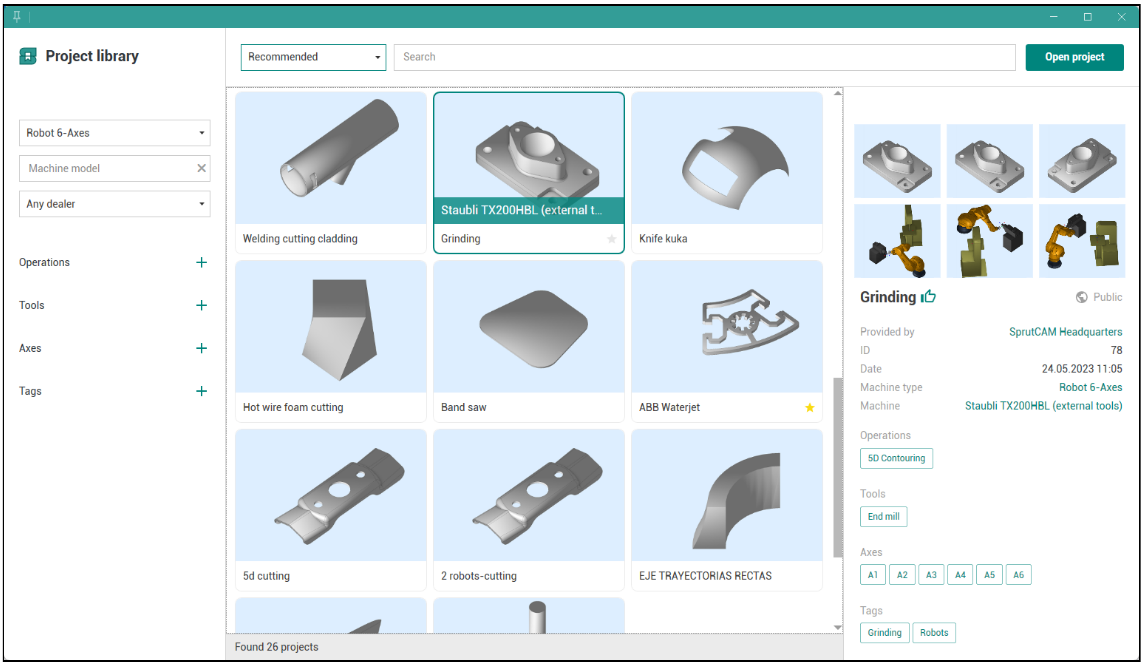
Now you can save the project as a new kind of file - machine setup file (*.stms). After that you can quickly create new projects using this file as a template. The following data are saved in the file: machine, stages, part, fixtures (including position), tools, approaches/returns, workpiece coordinate systems list, types of tool blocks, and placement in the turret. A new dropdown menu appears under the new project button where you can choose one of the recently used machine setup files. Also you can import machine setup file into the current project.



2.3.1.6 Project library

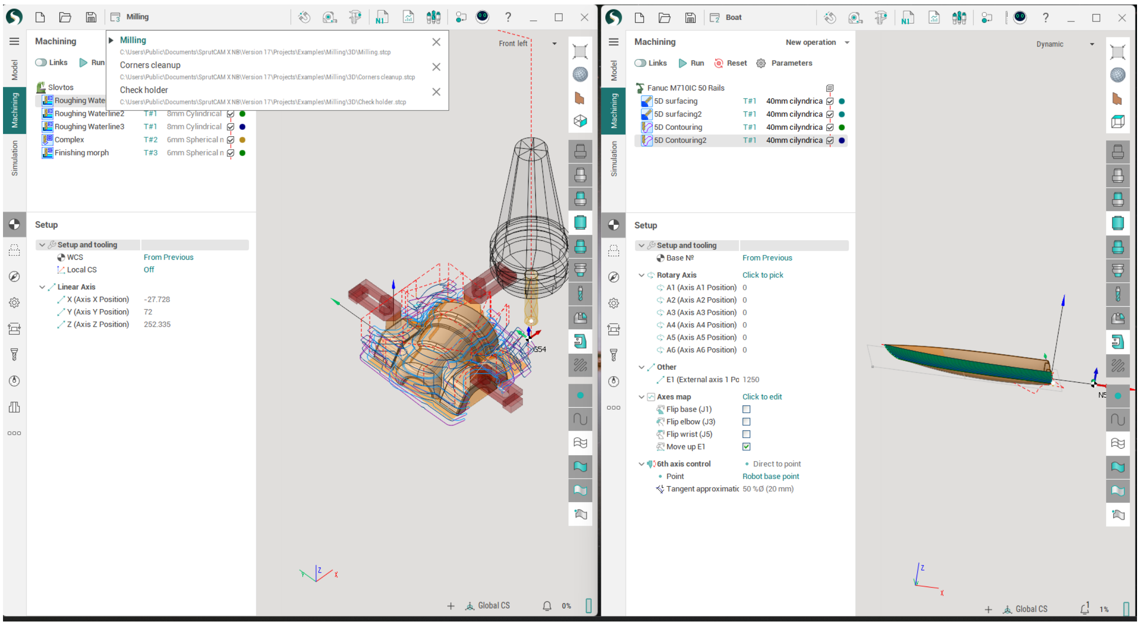
The Project Library allows to find and open example projects from our online library. Project library can be started directly form the SprutCAM X "Open" menu. More information available at this page: Project Library.





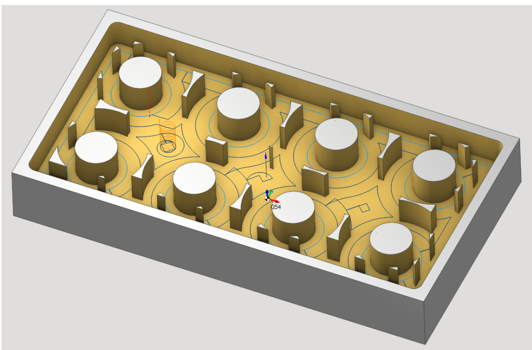
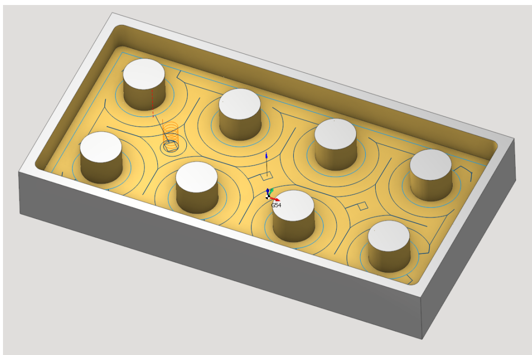
2.3.1.7 Multiproject workflow

Create or load two or more project at the same time. You can work with multiple SprutCAM X projects either using tabs to switch between them our pop-out and work with separate SprutCAM X windows.



2.3.2 Technology updates

2.3.2.1 Improvements in roughing waterline

Version 16	**Version 17**
	

If the rough step is greater than 50% of the tool diameter, then the special tool path is generated to remove the islands. Calculation time reduced up to 40%. Simulation highlights uncorrect plunges and long link feed cutting.

2.3.2.2 Redundant axes optimizer improvements

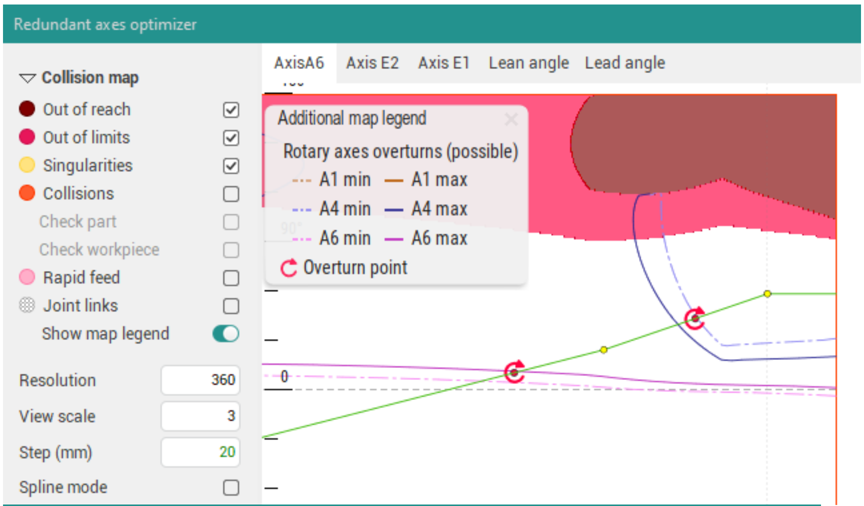
Added display of periodic rotary axes overturns

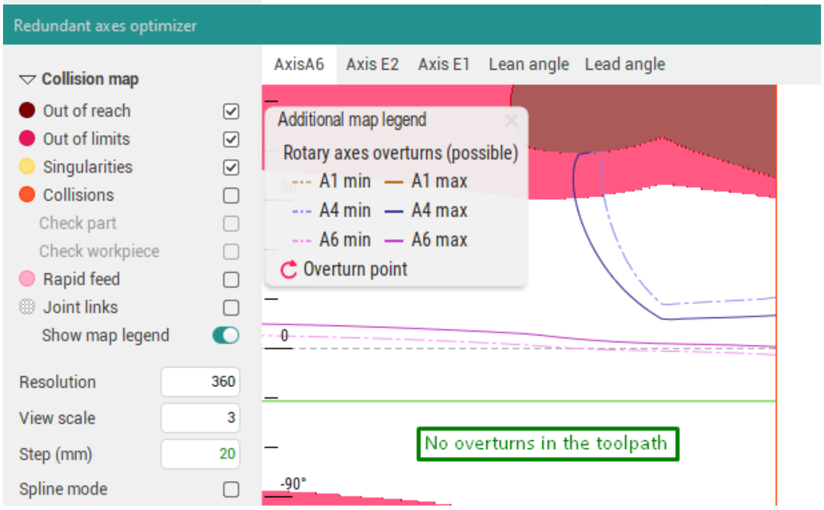
Now it is possible to use the redundant axes map to see potential problems in the toolpath, which are connected with the rotary axes overturns. Overturns happen if the rotary axis reaches one of its limits and in order to continue machining it needs to do one full rotation (360°). In previous versions no information about the overturns was available to user because, despite the overturn, the axis always stays within its limits.

The blue and purple lines show the possible overturn locations in the toolpath in case the spline intersects with them. However the intersection doesn't always correspond to overturn; the true overturn locations are additionally highlighted as red bold points with the "overturn" sign. Also if there are overturns in the toolpath, their count is displayed in the "Verification" status bar.

The light dashed/dark solid lines show the points where the given rotary axis might reach its minimum/maximum respectively.

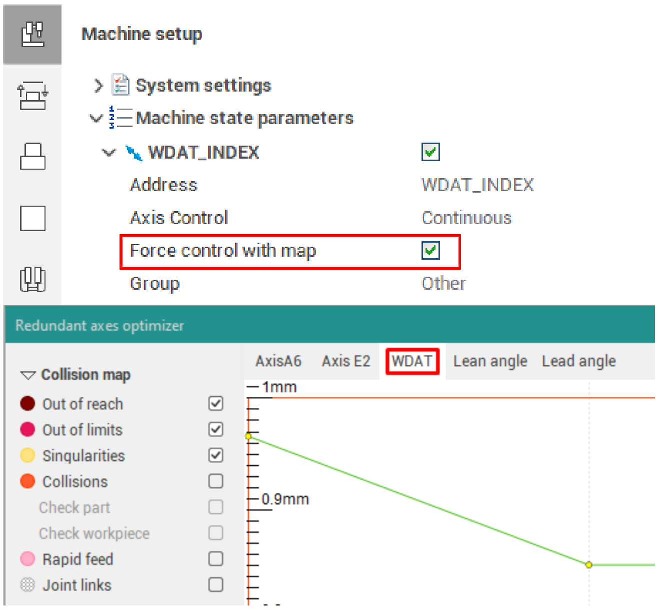
If you noticed the overturns in the operation's toolpath, you can try to **avoid** them by moving the spline so it **doesn't intersect** the possible overturn lines or the intersections are "fake" (the rotary axis did not reach its limit yet in this point). Also changing the robot configuration (the "Flip elbow", "Flip wrist" parameters) might also help to avoid the overturns. An example of spline editing to avoid the overturns is shown below on the screenshots.





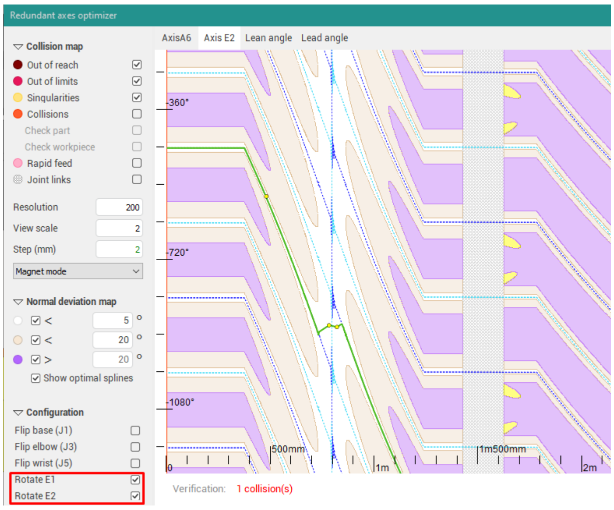
Arbitrary machine parameter control using map (optimizer)

If you need to control the changes of an arbitrary machine axis using the map, you can enable the "**Force control with map**" flag in the corresponding machine state parameter. 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. Also see the documentation on how to set this flag in the machine xml-file.



Singularity avoidance for the 2-axis rotary table of the robot

A special mode is activated for robots with the 2-axis rotary table and both enabled table flips. This mode is identical to the Axes map for 5-axis machines and allows you to get the rotary table trajectory without sharp changes and also as close to the geometrical toolpath as possible. More info about this feature is available in the documentation.

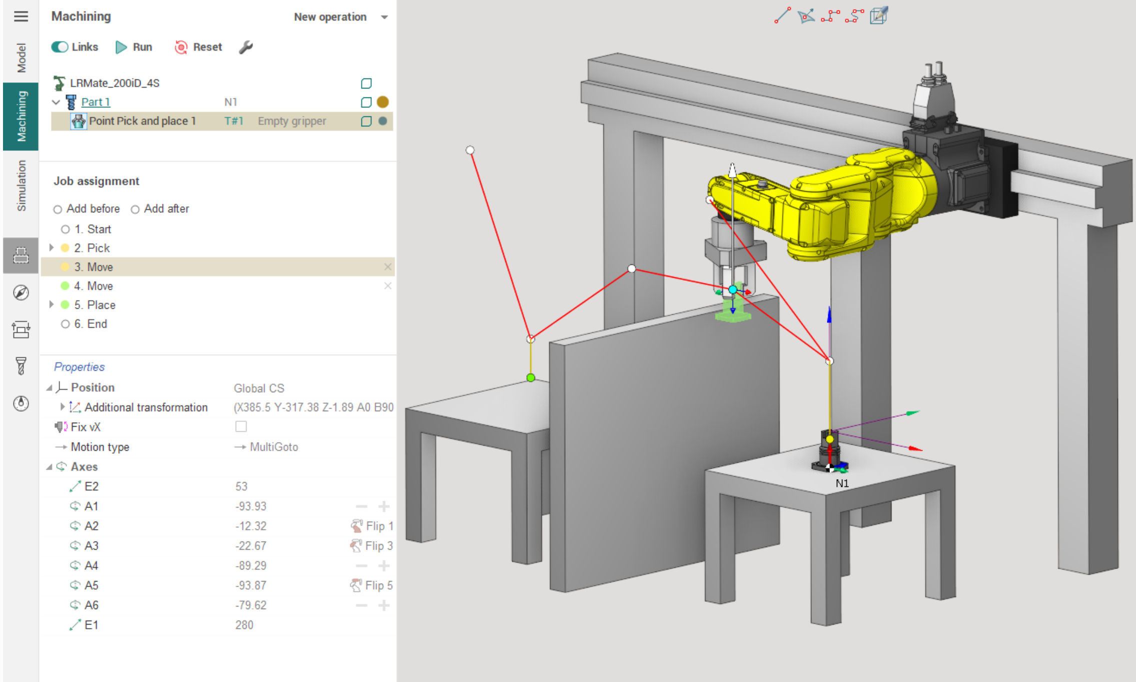


2.3.2.3 Point Pick and Place operation added

It's based on the old "Pick and Place" operation.

The operation has a new working task based 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. See more here.

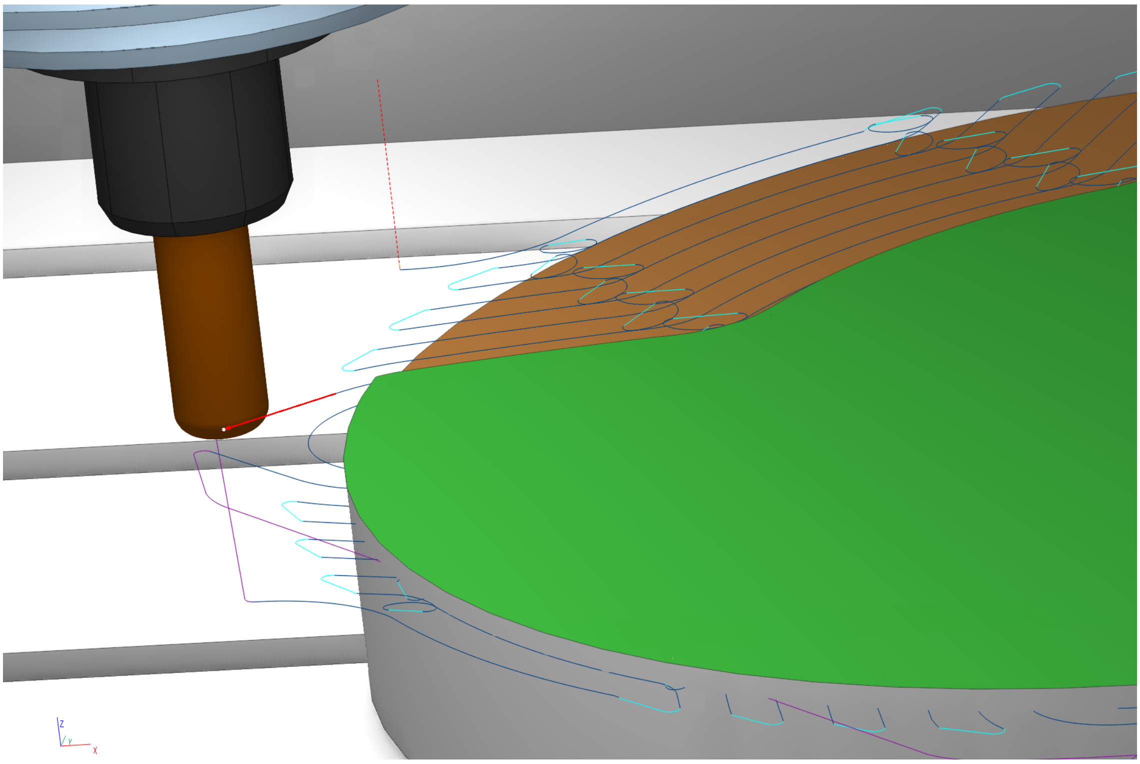
Simple example for pick and place.stcp



2.3.2.4 Links in basic milling operations improved

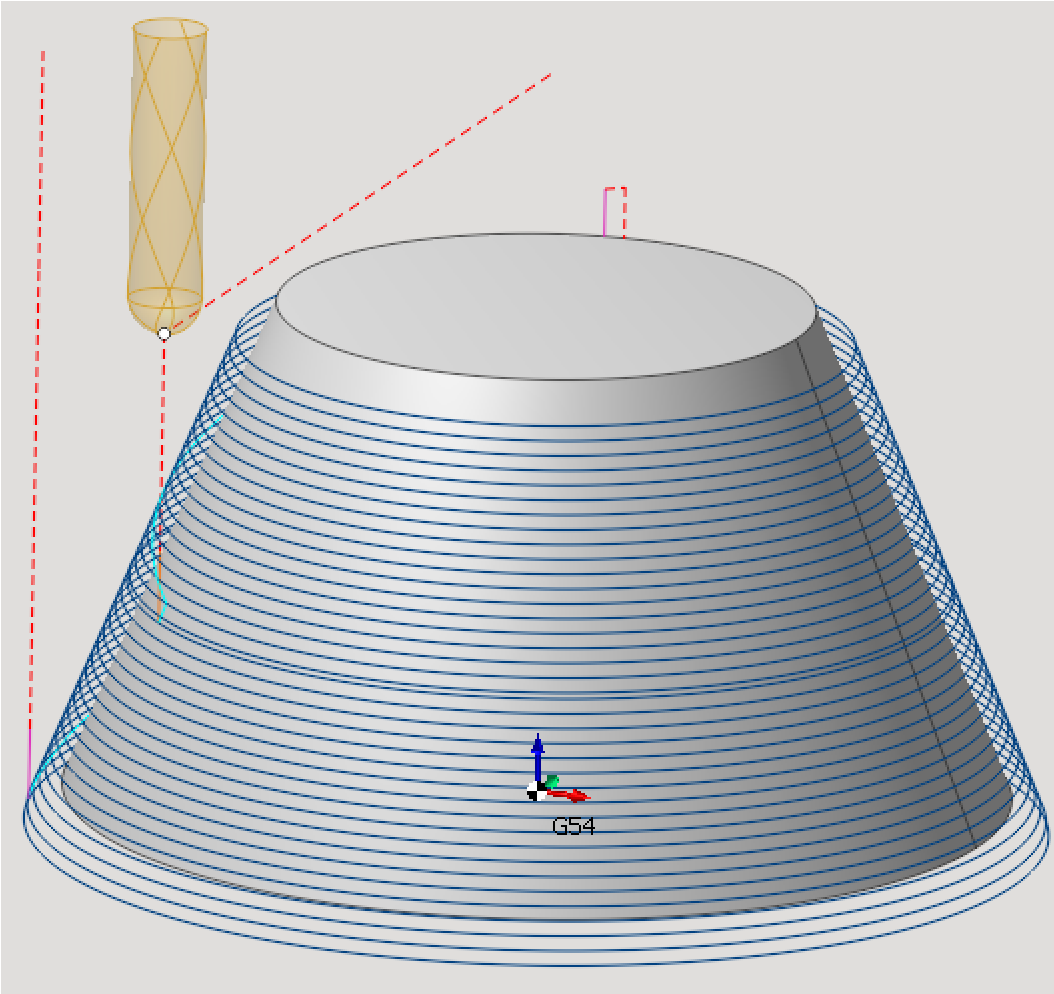
The main changes in links affected “Safe distance” and “Rounding radius” parameters. We have made a new algorithm for providing safe distance in links. It has significantly reduced excess work passes, especially, with big safe distances. Also it has reduced amount of safe motions due to replacing them with long or short links.

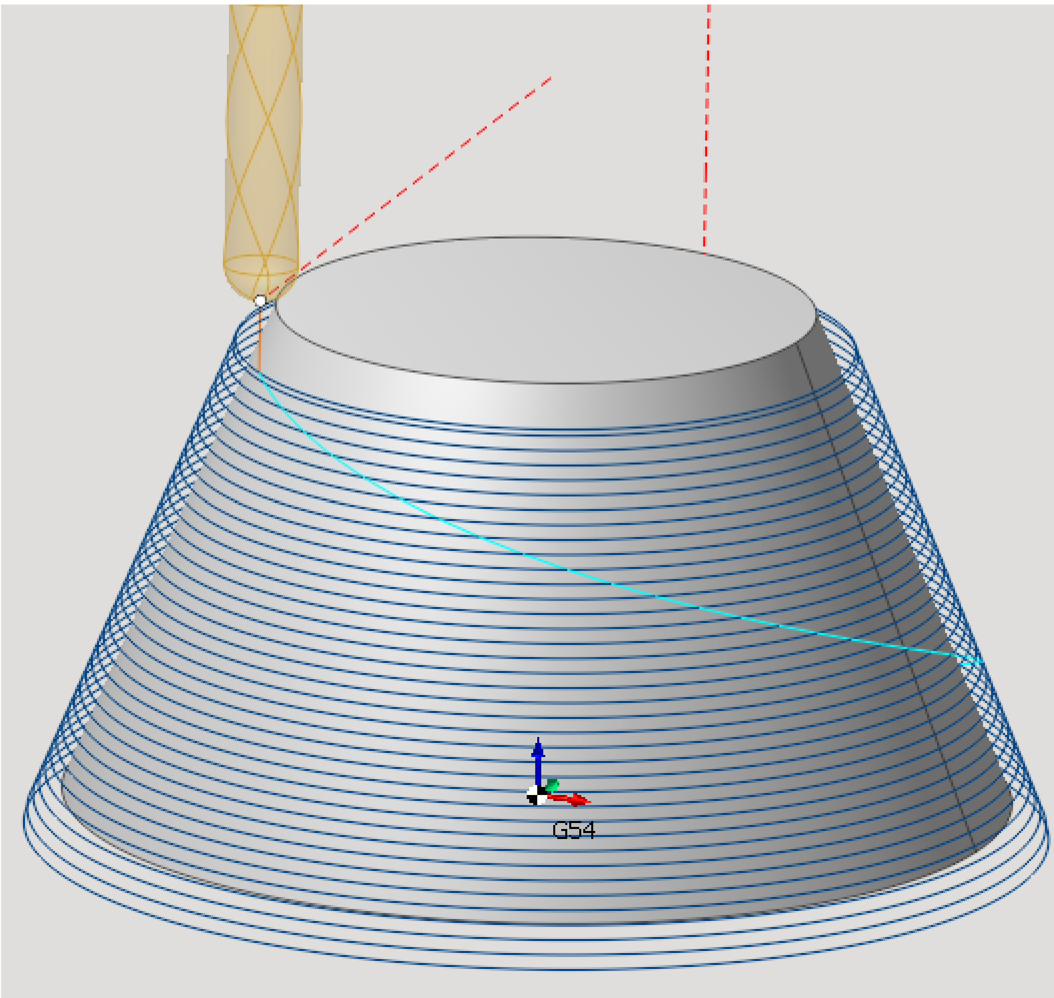
This changes affected 5 operations (“Pocketing”, “2.5D pocketing”, “2.5D flat land”, “Roughing waterline” and “Flat land”) and 3 strategies in them (“Equidistant”, “HPC”, “Deep HPC”). In “Adaptive SC” strategy only changes with reducing safe motions has been applied.



2.3.2.5 Scallop operation toolpath enhanced

The scallop operation toolpath has been changed. There is you can see two example images before and after trajectory improvement.

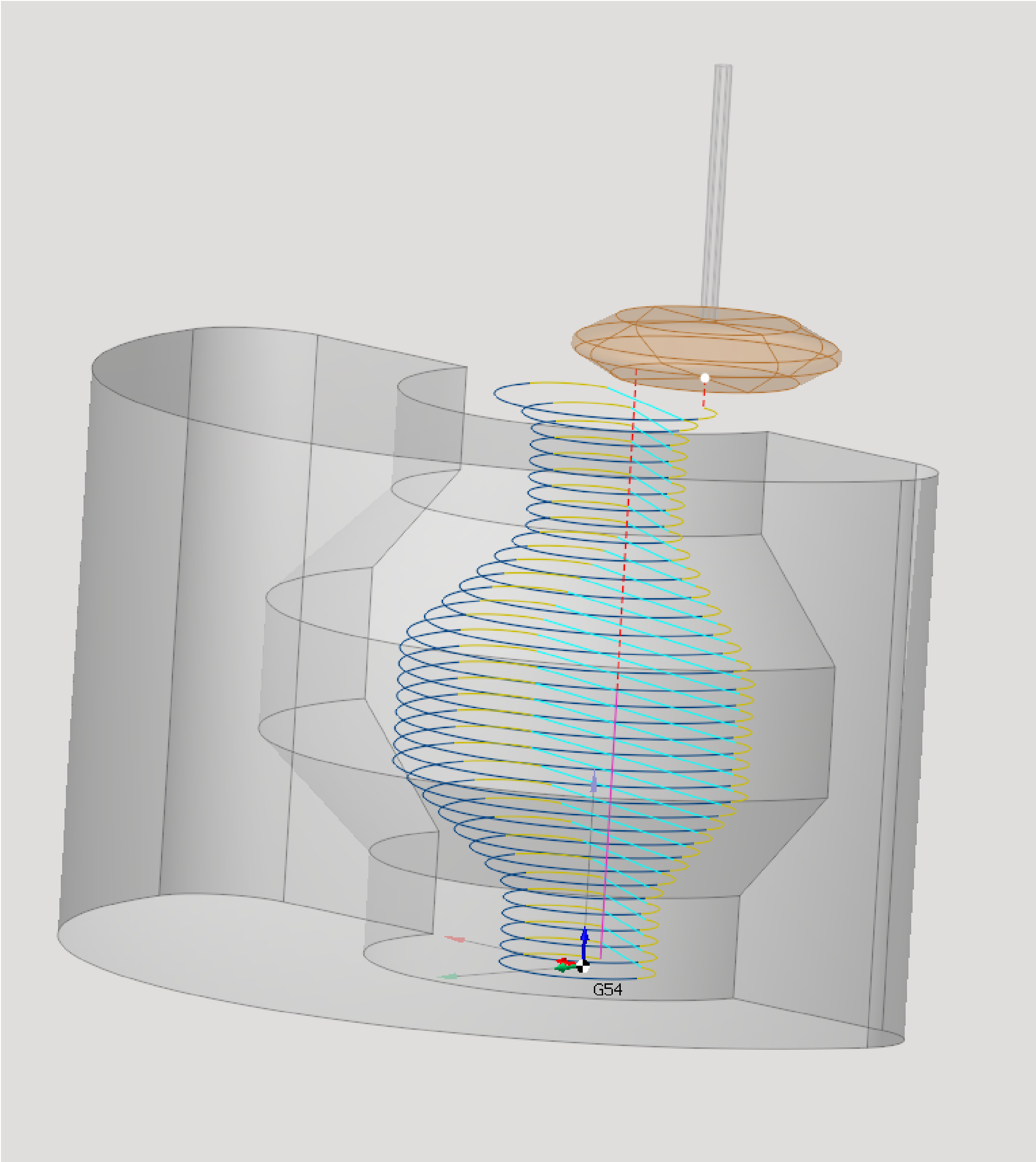

G54

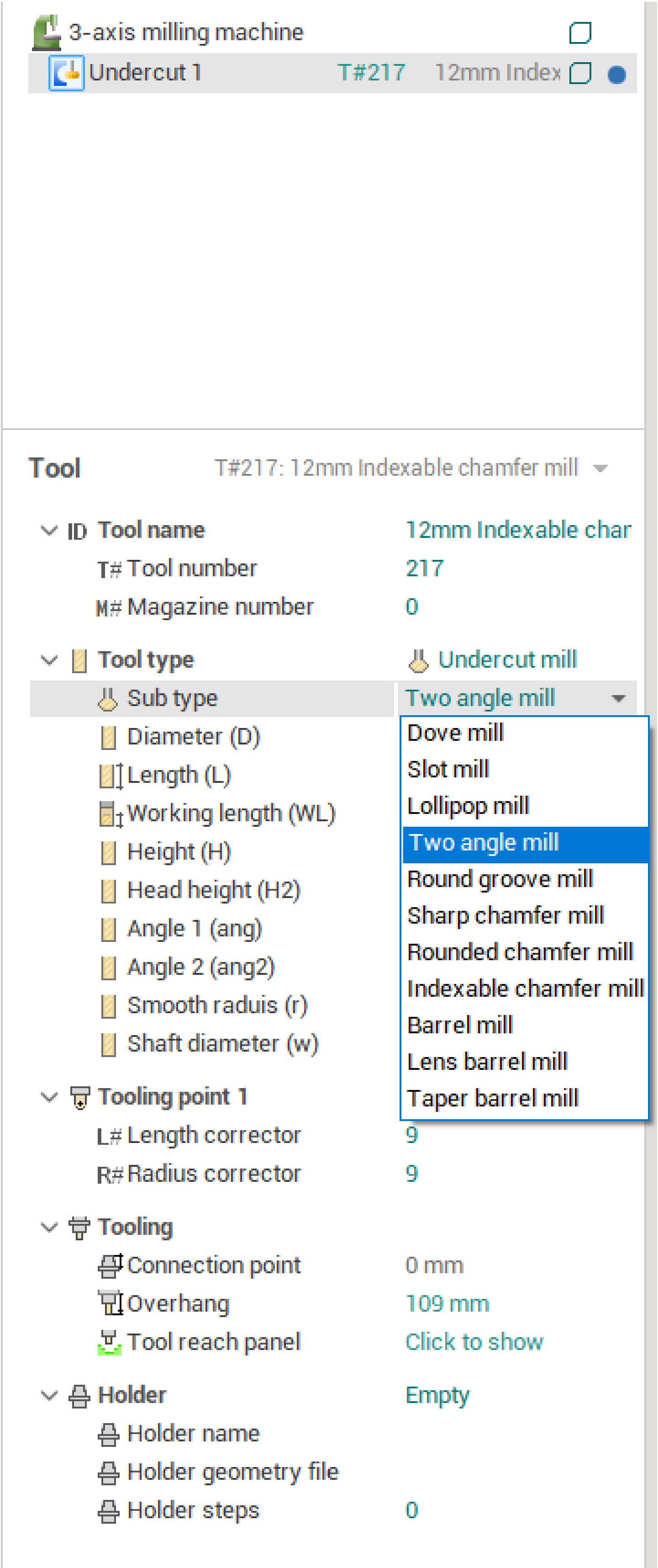


2.3.2.6 New undercut waterline operation added

Added a new "Undercut waterline" operation.

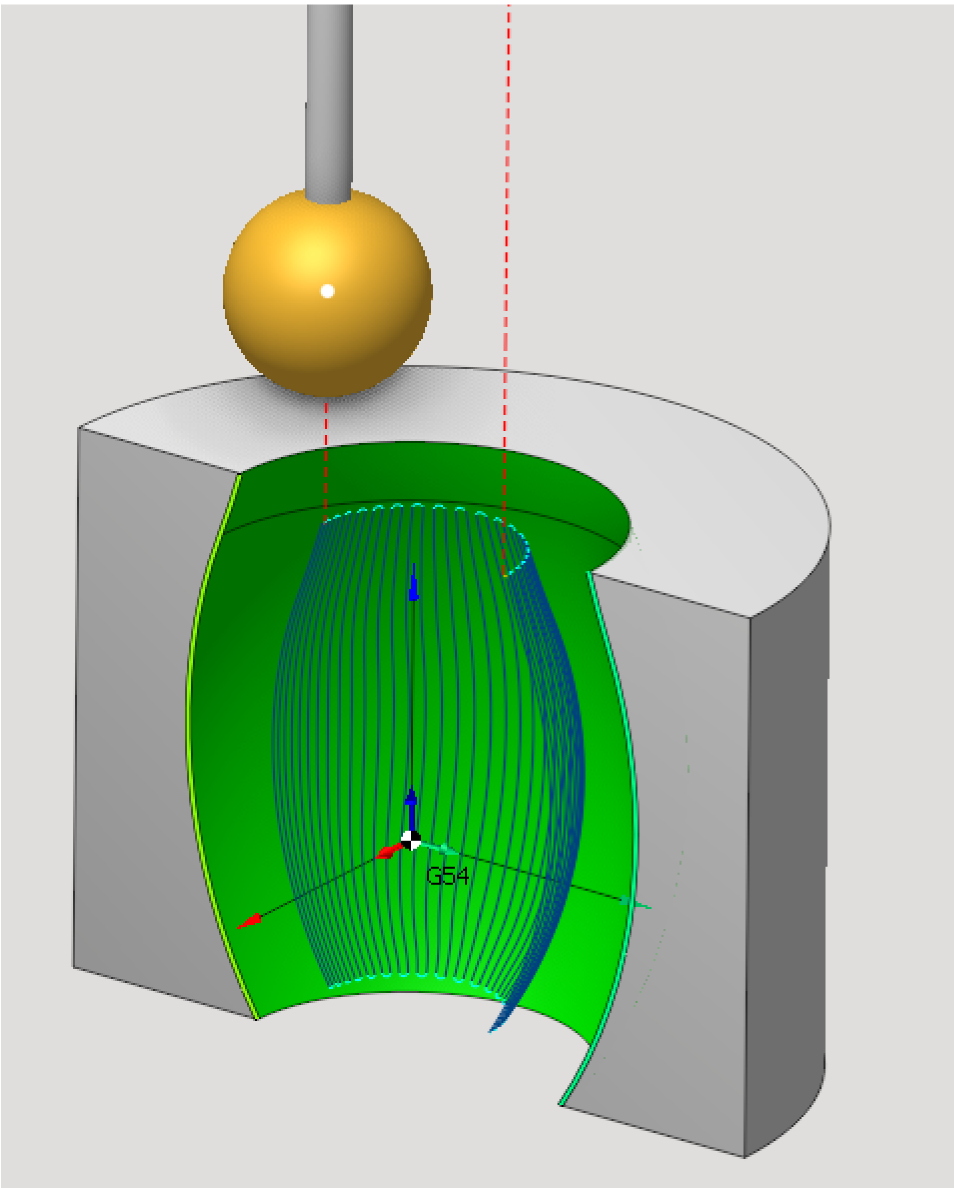
The operation allows you to create finishing and roughing undercut toolpath for supported undercut tools.


G54

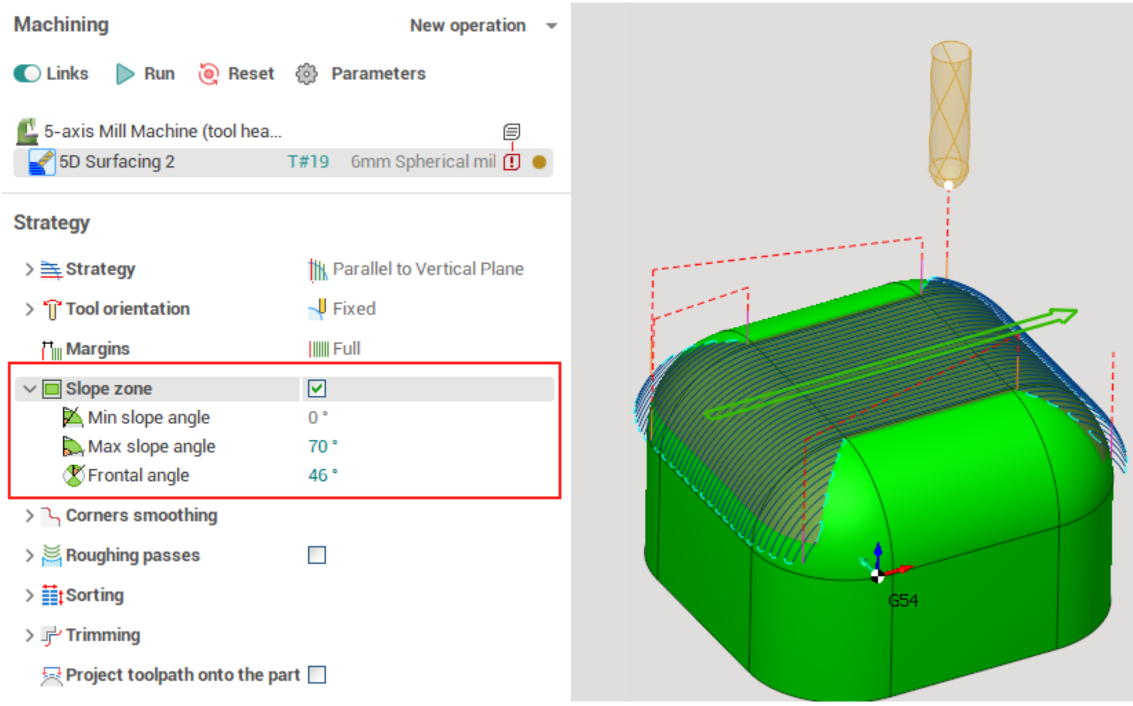

3-axis milling machine
Undercut 1
T#217
12mm Index
Tool
T#217: 12mm Indexable chamfer mill
ID Tool name
12mm Indexable char
T# Tool number
217
M# Magazine number
0
Tool type
Undercut mill
Sub type
Two angle mill
Diameter (D)
Length (L)
Working length (WL)
Height (H)
Head height (H2)
Angle 1 (ang)
Angle 2 (ang2)
Smooth raduis (r)
Shaft diameter (w)
Dove mill
Slot mill
Lollipop mill
Two angle mill
Round groove mill
Sharp chamfer mill
Rounded chamfer mill
Indexable chamfer mill
Barrel mill
Lens barrel mill
Taper barrel mill
Tooling point 1
L# Length corrector
9
R# Radius corrector
9
Tooling
Connection point
0 mm
Overhang
109 mm
Tool reach panel
Click to show
Holder
Empty
Holder name
Holder geometry file
Holder steps
0

2.3.2.7 Undercut tools support in 5D Surfacing operation.

The tool only works for fixed axis.

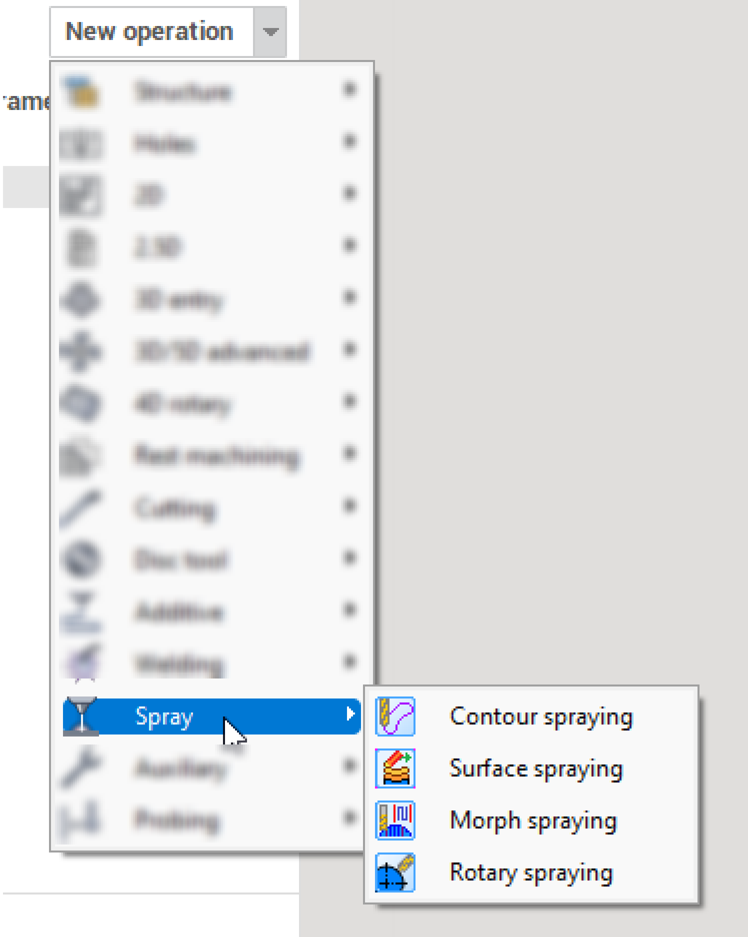


2.3.2.8 Added slope zone in 5D Surfacing operation



2.3.2.9 New group of operations added - Spray

For spraying, several new operations were created on the basis of existing ones in order to optimize the set of purchased options. They are all placed in the same Spray group and are licensed jointly. The following operations are currently available: Contour spraying, Surface spraying, Morph spraying, Rotary spraying.



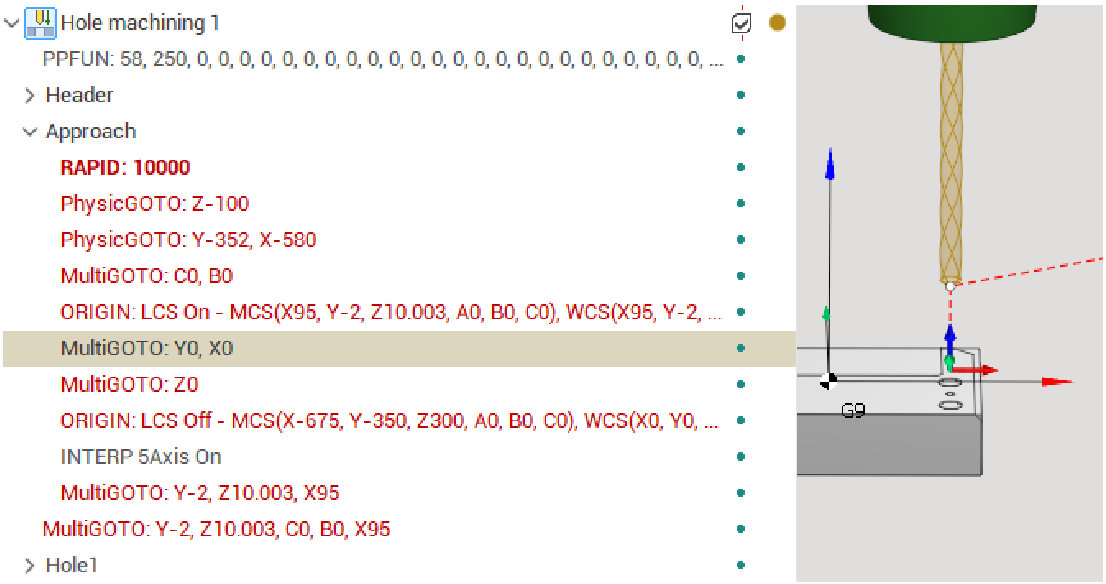
2.3.2.10 Approach/return for the TCPM enabled operations using Local CS

This feature allows to overcome some problems with the 5-axis tool center point management (TCPM), if the machine kinematics are different from the real machine. More safe approach or return can be done if the specific LCS is temporarily switched on for some tool movements. Example of such command:

G53 Z(-100); G53 Y(-352) X(-580); BC; **SLCS(** XY; Z **)**

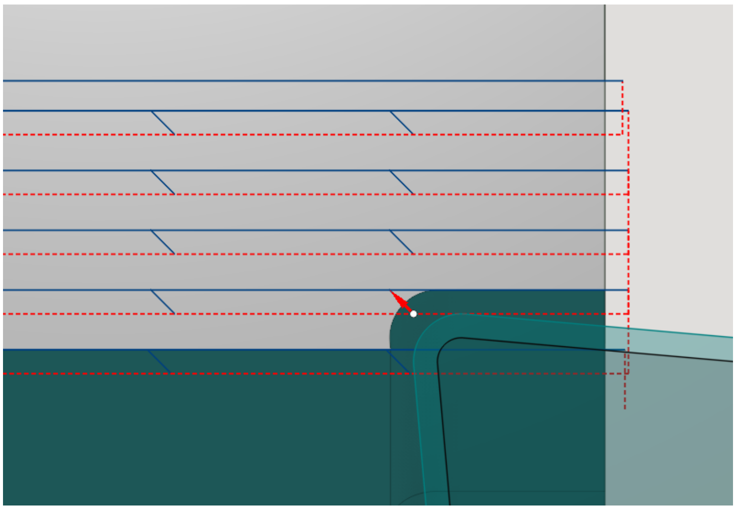
The commands inside the **SLCS** block are performed in the LCS of the first toolpath point (if it is inside the approach section) or in the LCS of the last toolpath point (inside the return section). The TCPM mode is activated in the end of the approach or beginning of the return respectively.

Below is the example of the resulting operation toolpath. See also the Approach/return rules documentation article.



2.3.2.11 Added new parameter for chip breaking in Roughing lathe cycle

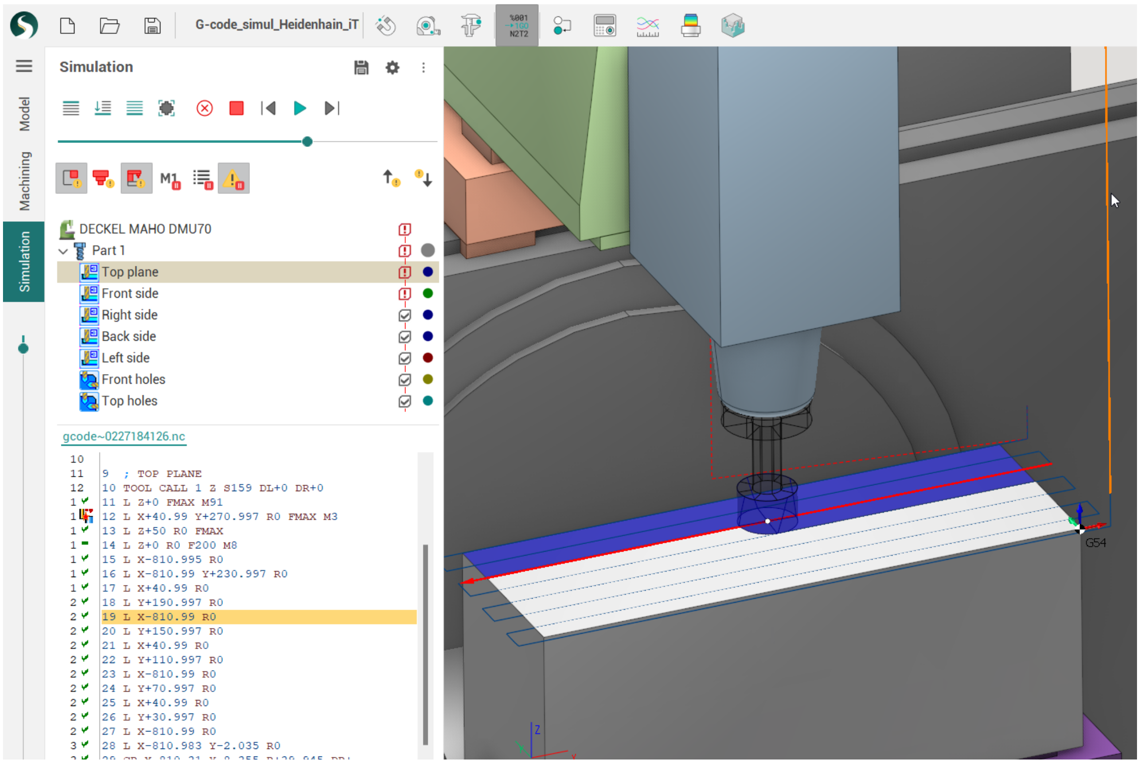
The chip breaking is a new feature in Roughing lathe cycle that helps to break chips into smaller pieces for improved efficiency and safety.



2.3.3 Postprocessing and G-code simulation

2.3.3.1 .NET postprocessors for G-code simulation

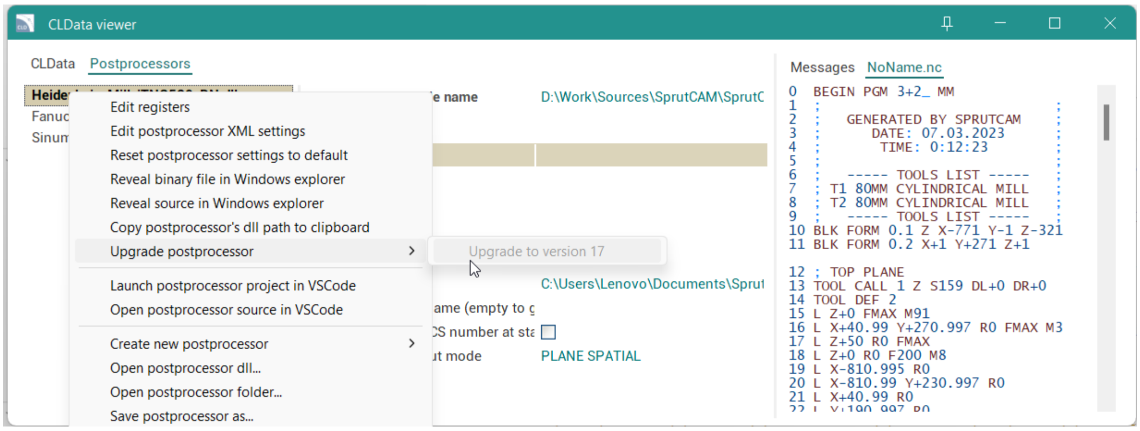
.NET based .dll postprocessors now can be used for simulation based on G-code on a par with .sppx postprocessors.



2.3.3.2 Upgrade to .NET 6.0 version

Now you can use more recent 6.0 version of .NET to create your G-code interpreters and postprocessors.

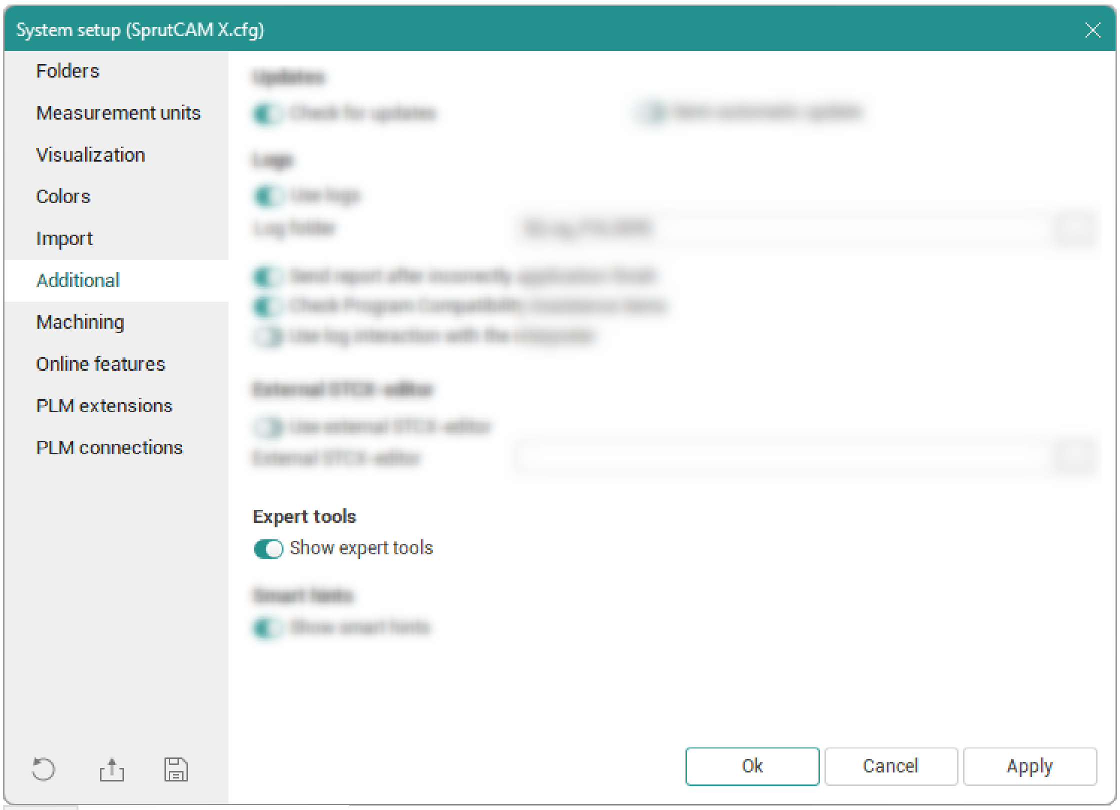
Due to the change in the folder structure in the SprutCAM X installation directory, minor changes in the postprocessor source files may be required in order to compile them in the 17th version. For this reason there is an **Upgrade postprocessor** context menu item in the CLData Viewer to be possible to automatically upgrade the source codes of postprocessors.



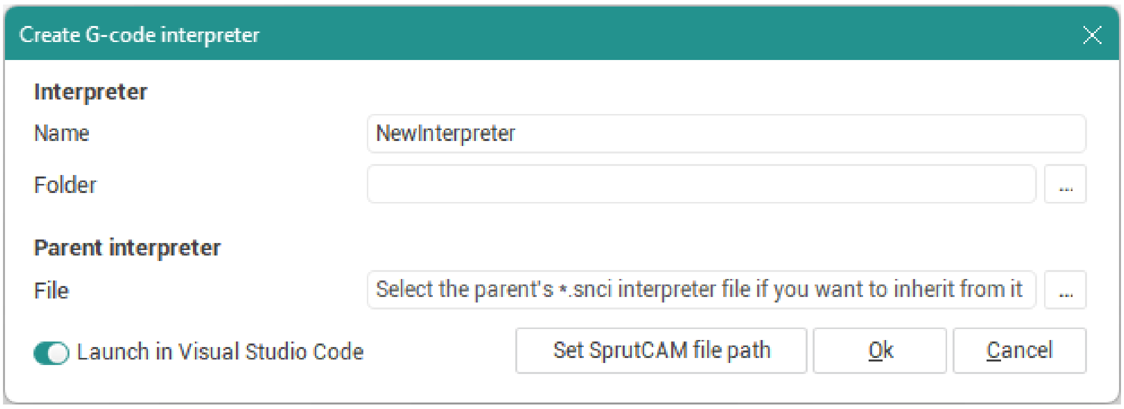
2.3.3.3 Tool for creating new interpreters

A new tool included to the official distribution to help you create your own .NET based (C#) G-code interpreters.

It is located in the utilities menu and is displayed when the **Show Expert Tools** option is enabled in the system settings.


System setup (SprutCAM X.cfg)
Folders
Measurement units
Visualization
Colors
Import
Additional
Machining
Online features
PLM extensions
PLM connections
Expert tools
Show expert tools
Ok
Cancel
Apply

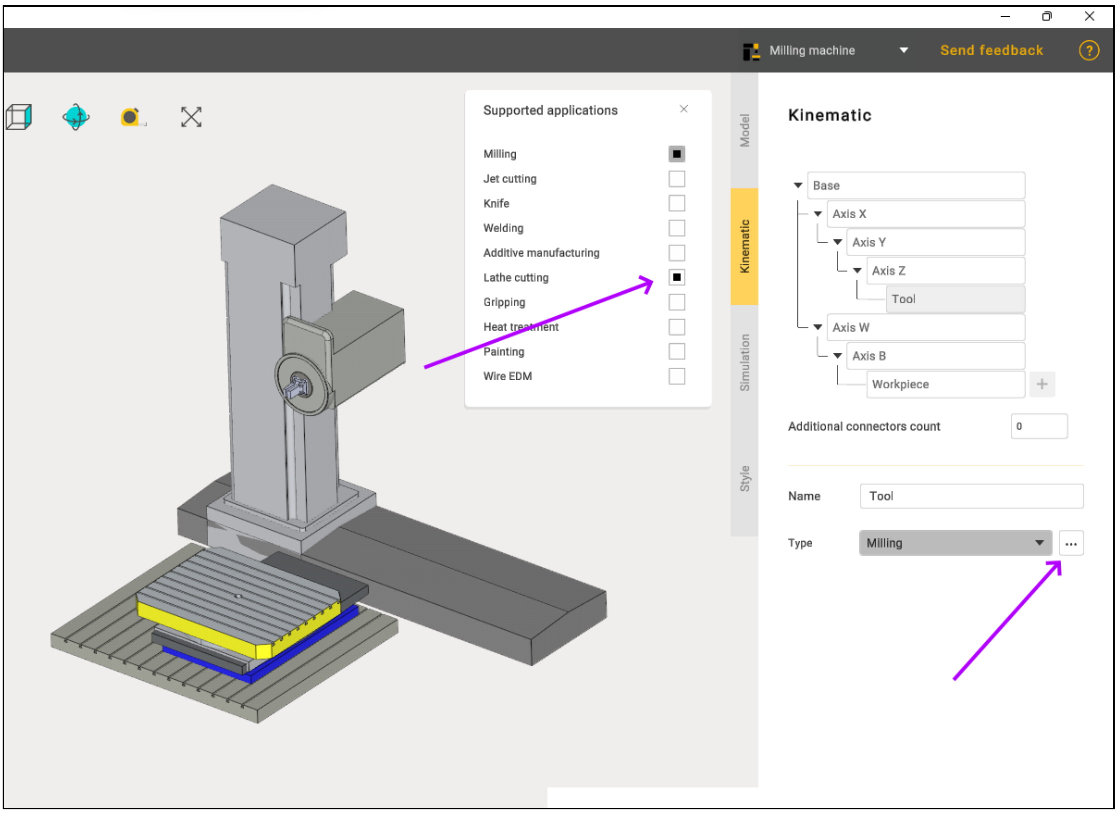

Machining
NC
CLD
Create interpreter



2.3.4 MachineMaker improvements

There are a lot of improvements in MachineMaker:

- Support of optional machine equipment for Milling machines
- Support of machine equipment in Robot Cells
- Machine validation
- Mill machine templates
- Geometry measuring
- Smart input fields
- Undo transformations
- Multiaxis machines support
- It is possible to define tool type, so you can create Lathe machines now
- Interactive 3D model simplifier
- See more...



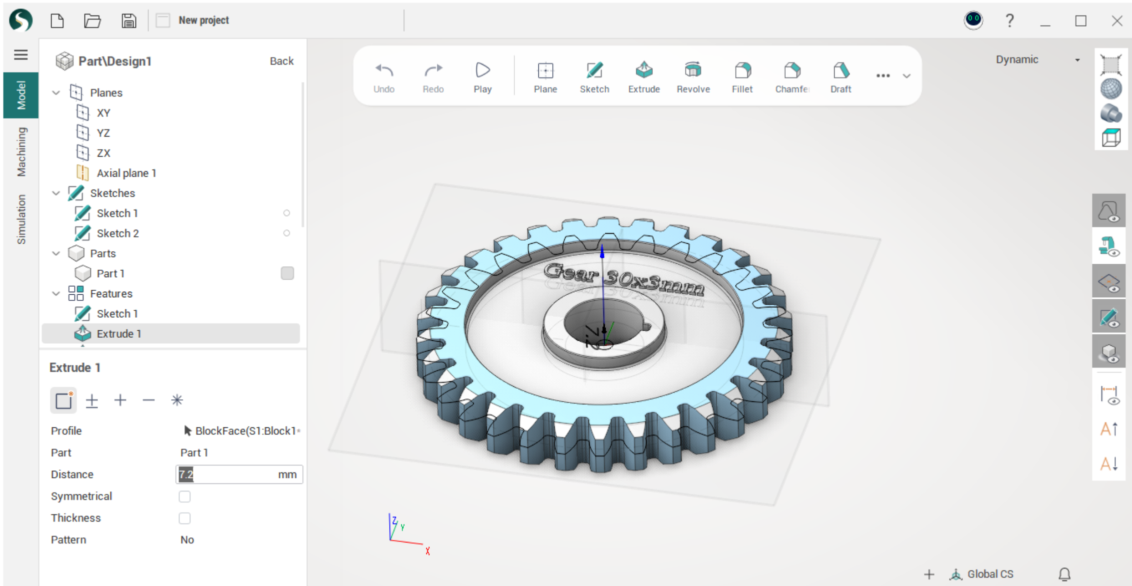
2.3.5 CAD enhancements

2.3.5.1 Design module enhancements

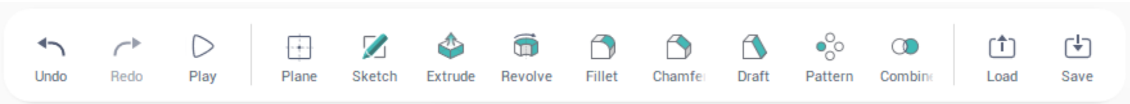
Visual overhaul and better user experience

The user interface has undergone a complete makeover. New icons, ui components and gizmos make for a whole new look and feel.

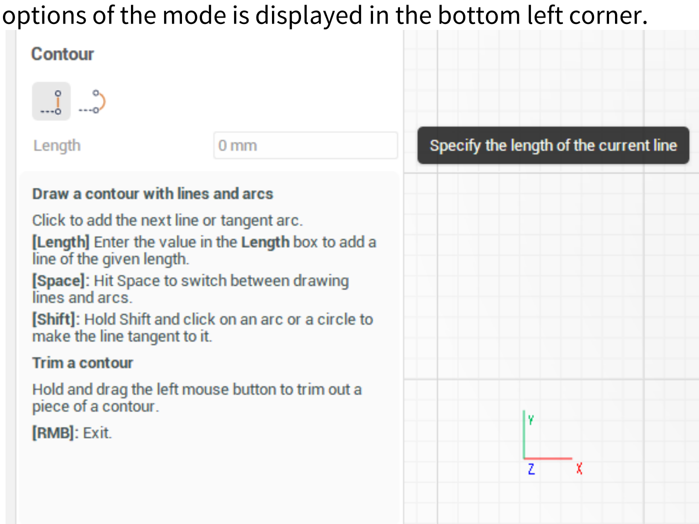
- **Support of High DPI screens**. Everything looks and works great on all display sizes and screen resolutions.



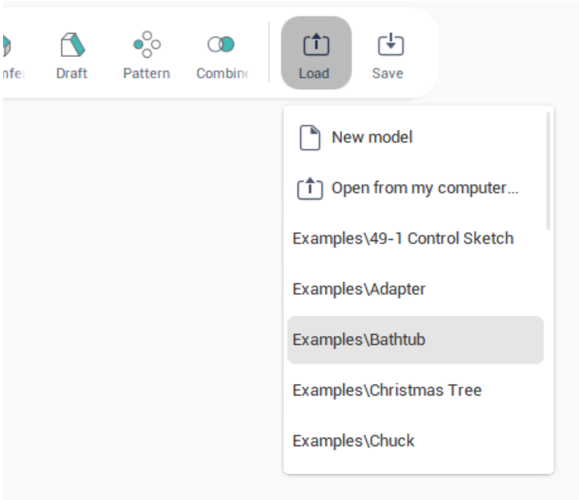
- The new **function bar** provides at a glance access to the complete CAD functionality.



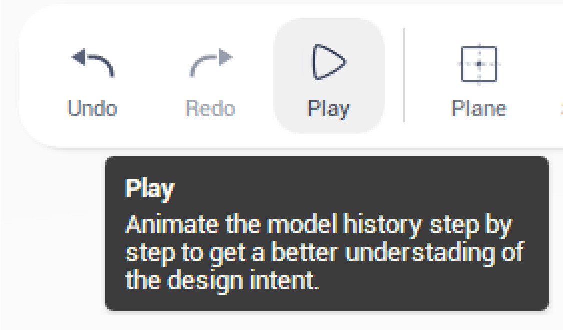
- **Tooltips and user promts**. Hover the mouse over a parameter, if you don't know what it's doing. When sketching or creating a modeling operation, a user prompt with all available options of the mode is displayed in the bottom left corner.



- **Quick access to design examples.** Quickly become familiar with the CAD functionality by discovering the provided design examples.

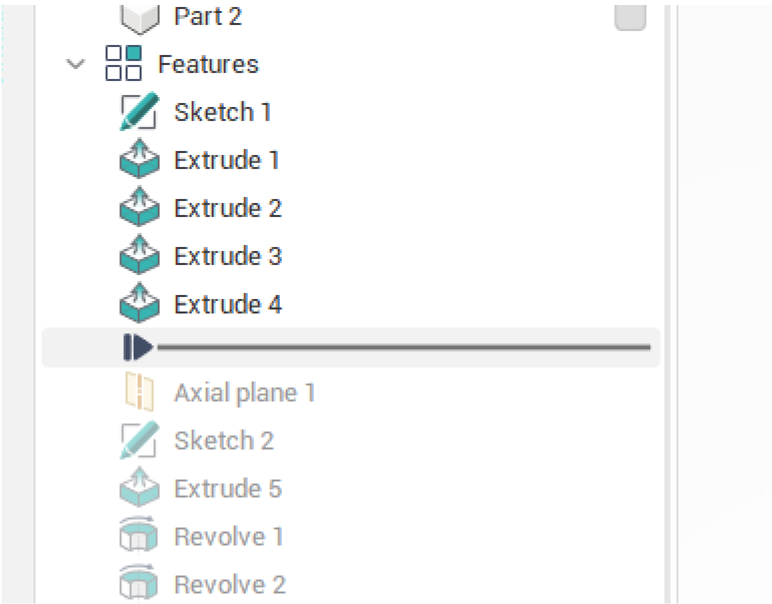


- **Play mode.** Animate the model history step by step to get familiar with the design intent.



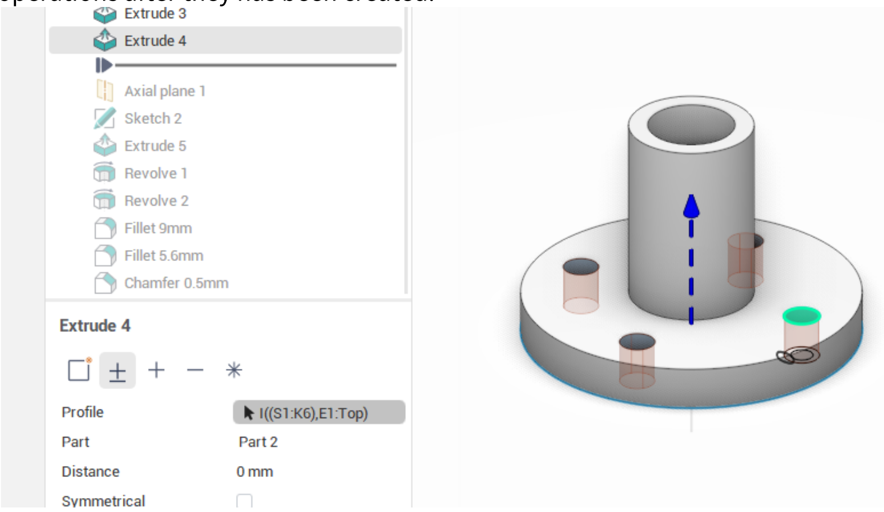
Full-fledged work with the model history

- Drag the **Rollback bar** with the mouse to get back in time in the model history and make drasting changes to the model not possible otherwise. Quickly move the rollback bar across the tree simply by double clicking on a modeling operation in the model tree.



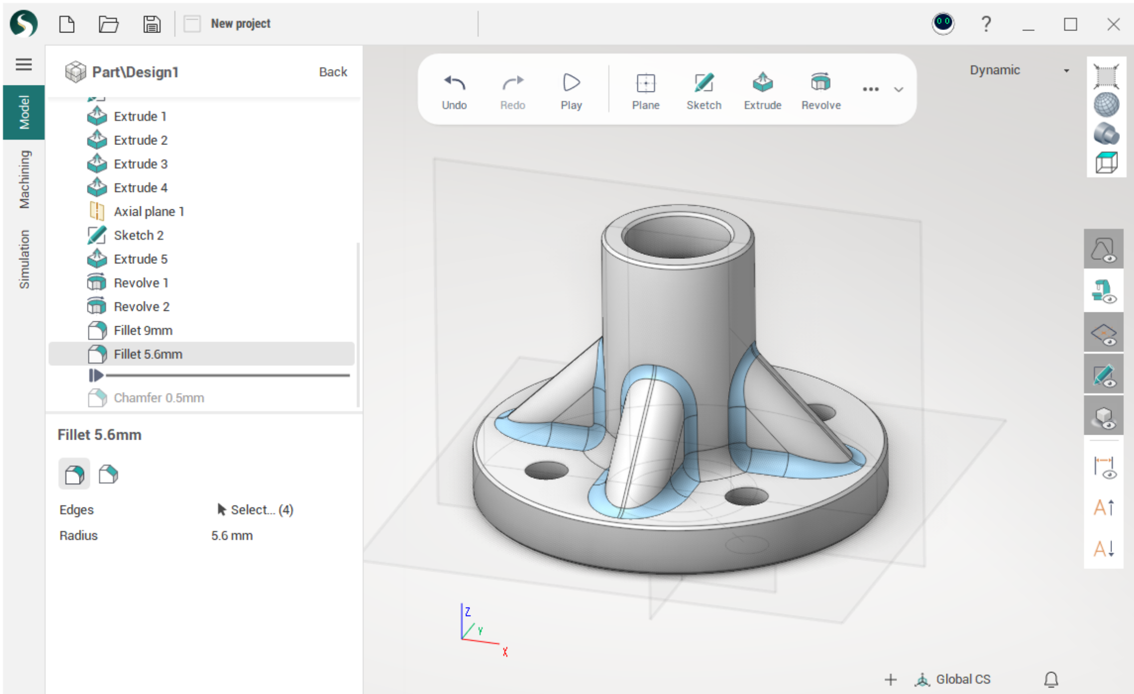
- **Reorder modeling operations** with drag and drop.

- **Edit the job assignment** of modeling operations. Reassign profiles, faces or edges of operations after they has been created.

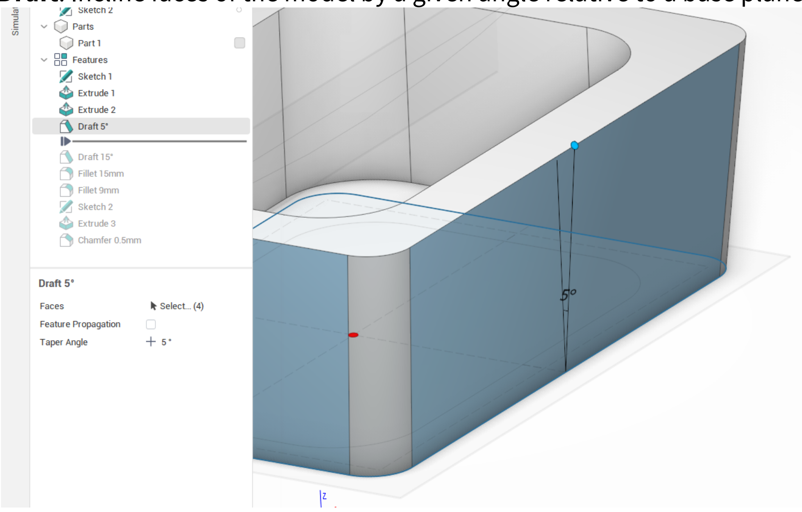


New 3d modeling operations

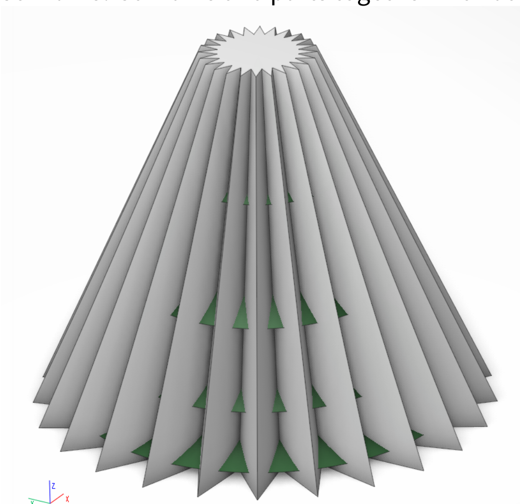
- **Fillets and Chamfers.** Round sharp corners of the model with the given radius and create chamfers.

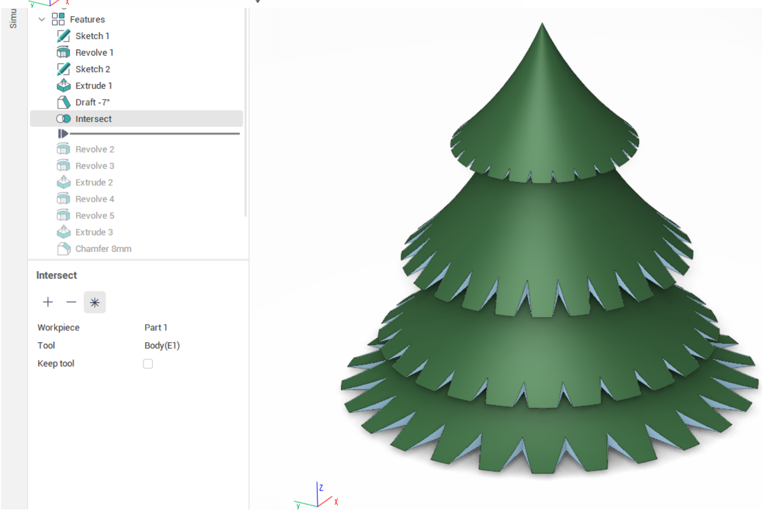


- **Draft**. Incline faces of the model by a given angle relative to a base plane.

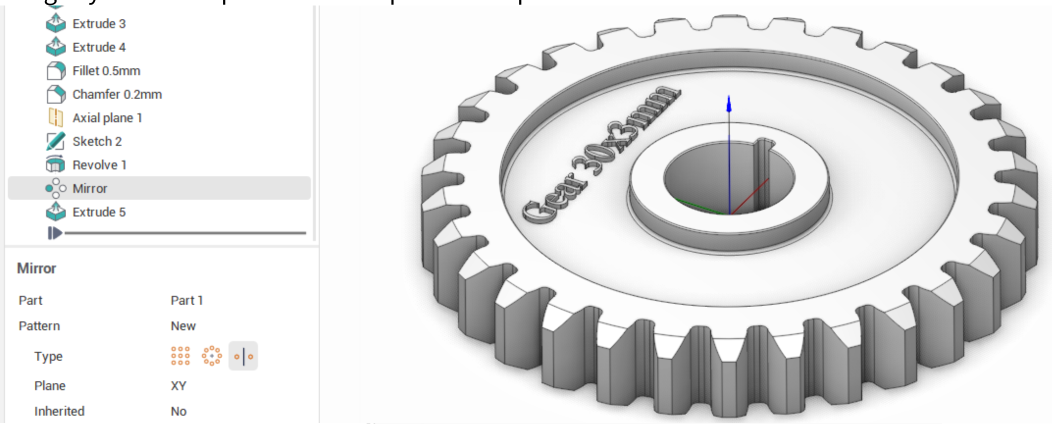


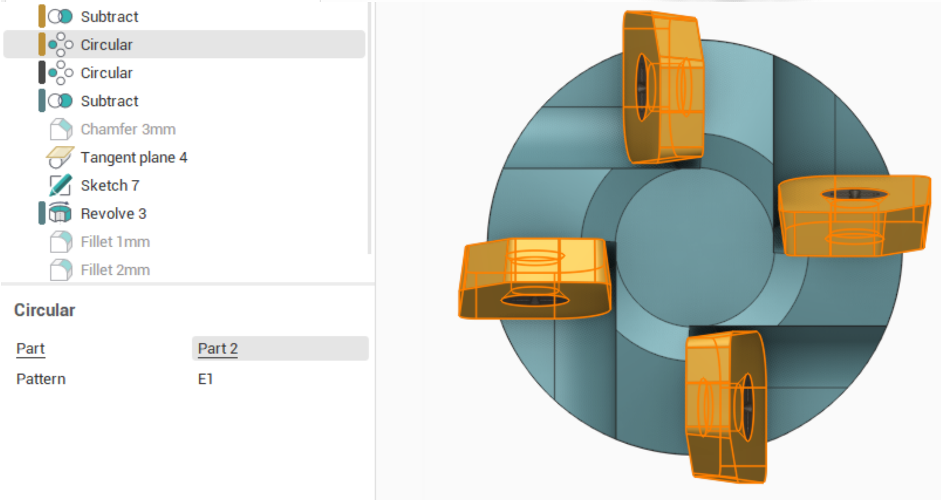
- **Combine.** Combine two parts together with boolean addition, subtraction or intersection





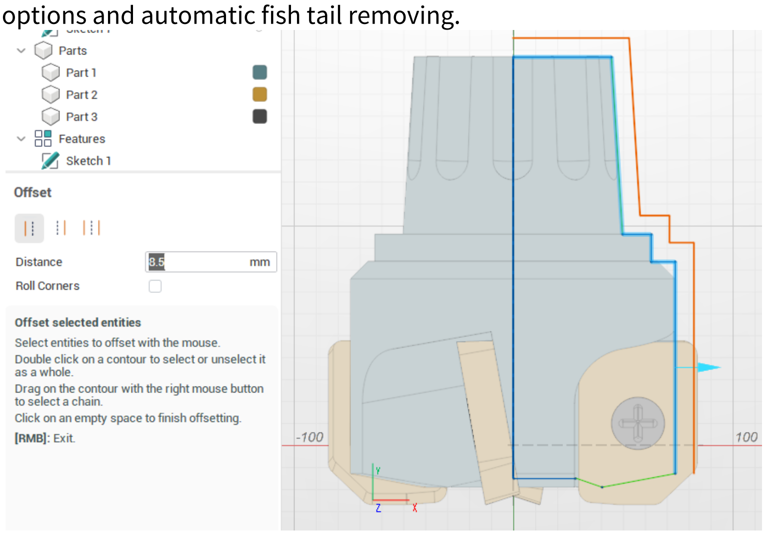
- Design symmetrical parts or create patterns of parts with **Pattern.**



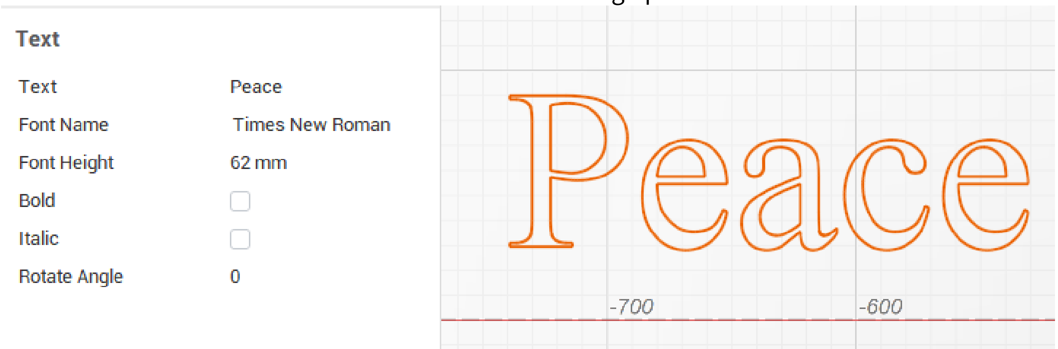


New sketching tools

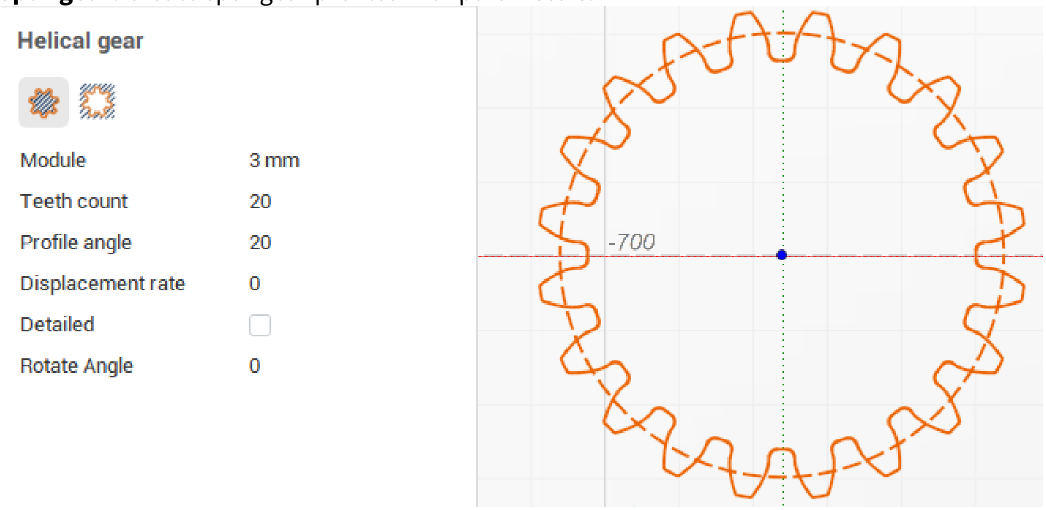
- **Offset.** Contour offsetting is now available as a separate sketching tool with more advanced options and automatic fish tail removing.



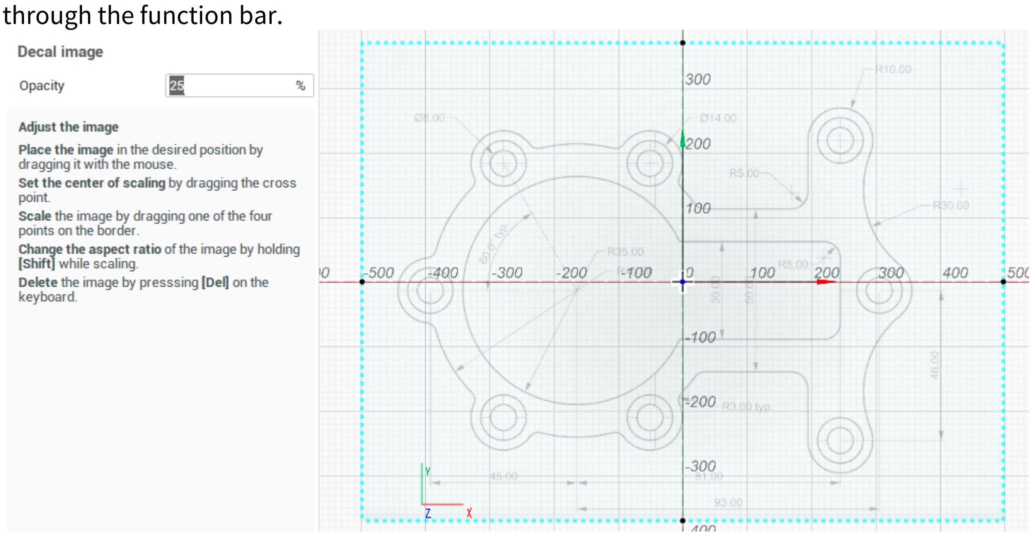
- **Text.** Create contours from text to use them in modeling operations.

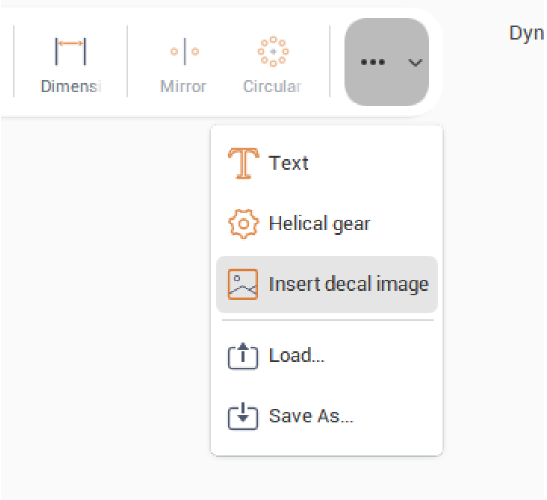


- **Spur gear.** Create spur gear profiles with parameters.



- **Decal image.** Based on user feedback, the decal image insertion is now more easily available through the function bar.





Better stability and performance

Many stability issues and small bugs has been fixed. Model regeneration times have been significantly reduces in many scenarious.

2.3.5.2 New CAD import capabilities

New CADs Addons:

Internal Importer	**Version**
SolidCAM	2021 SP5
SolidWorks	Up to 2023 SP0

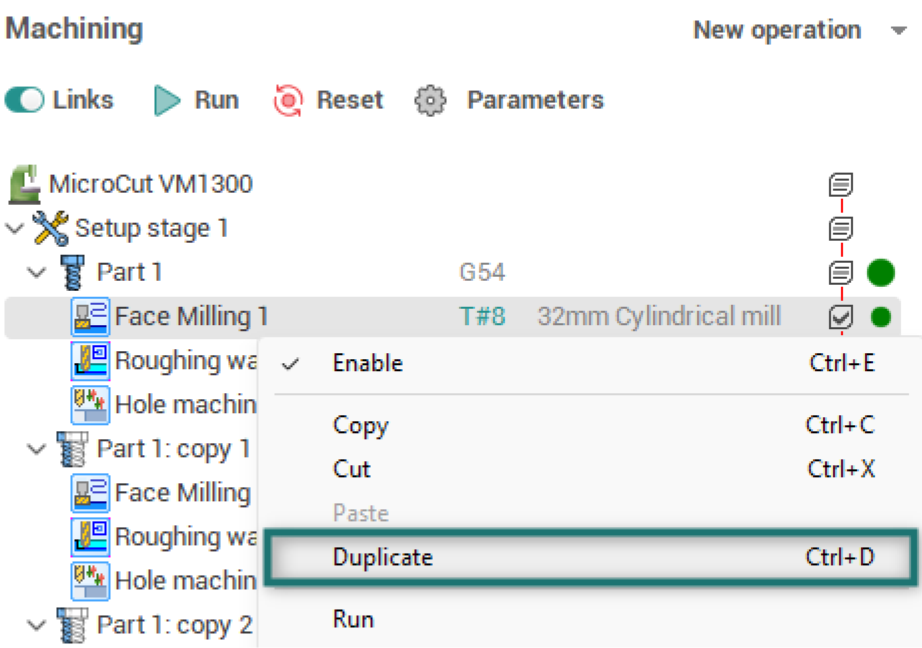
2.3.6 Minor changes

2.3.6.1 Main application executable file changed

The main executable file of the application has been changed. Now it is **sc.exe** instead of **SprutCAM.exe**. This is not just a file name change, the application type has changed. Now the start file is a .NET application. This opens up new opportunities for SprutCAM X extension developers. It becomes possible to write your own plugins and operations for SprutCAM X using the freely available C# language and development tools such as Visual Studio Code.

2.3.6.2 New item 'Duplicate' in the context menu of the list of technological operations

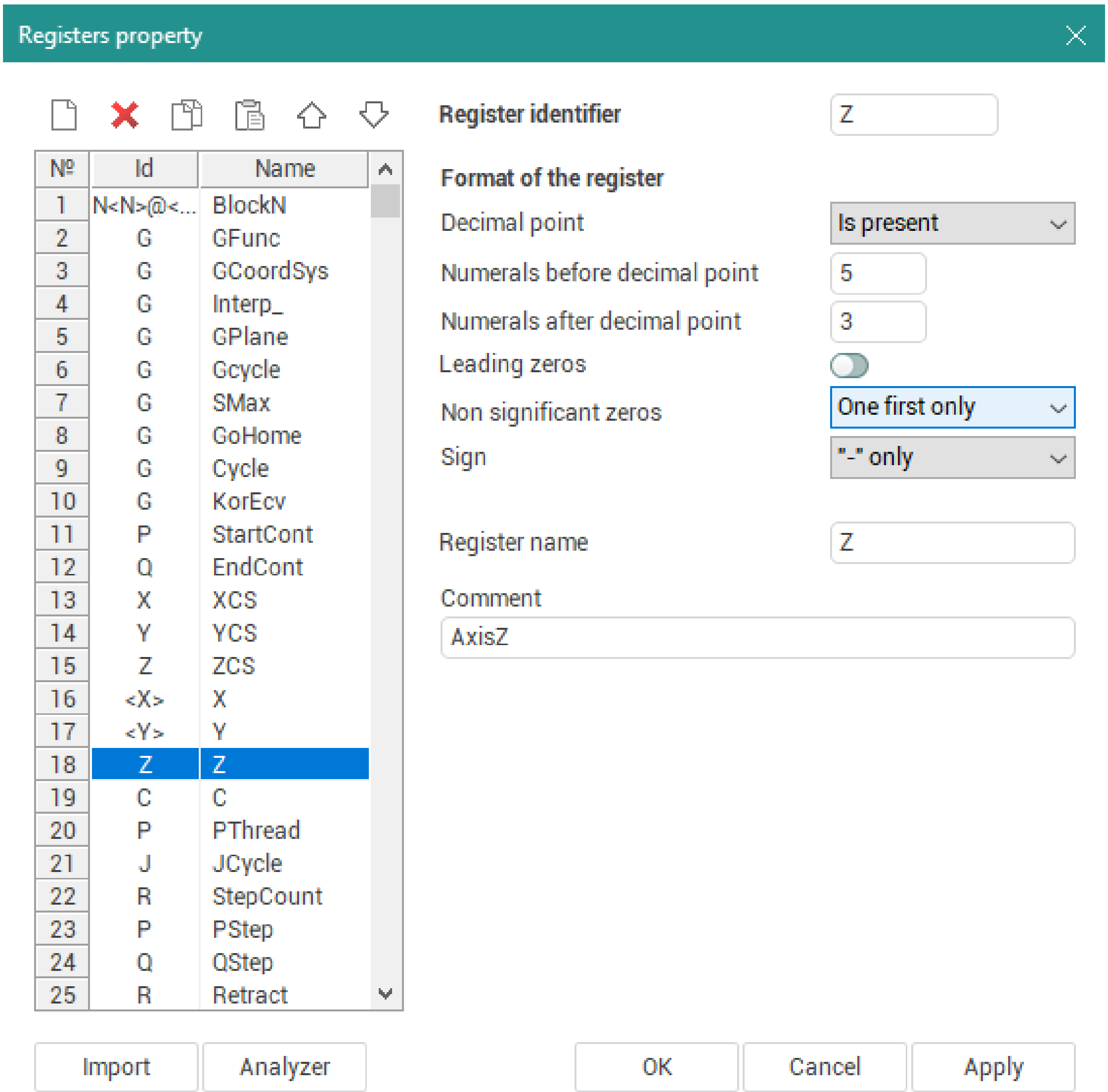
A new menu item - "Duplicate" has been added to the context menu of the list of technological operations (in the <Technology> working mode), which allows you to copy and paste the selected operation with one click.



2.3.7 Post Processor generator

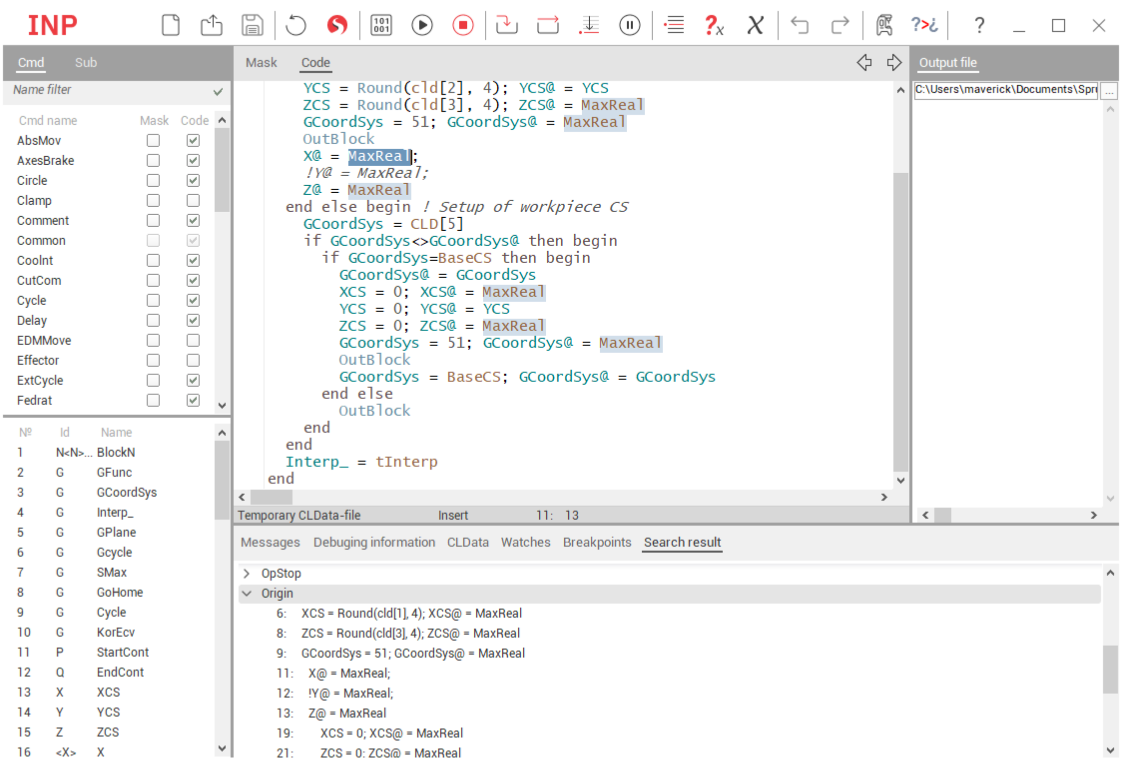
2.3.7.1 New Trailing zeros output option in Registers

In addition to the previously existing Yes/No options, a new one is: add .0 to integers.



2.3.7.2 Search in all commands and subprograms

After double-clicking on a word in the editor, an additional tab with search results will open with the ability to navigate through these results.



2.3.7.3 2D arrays and records

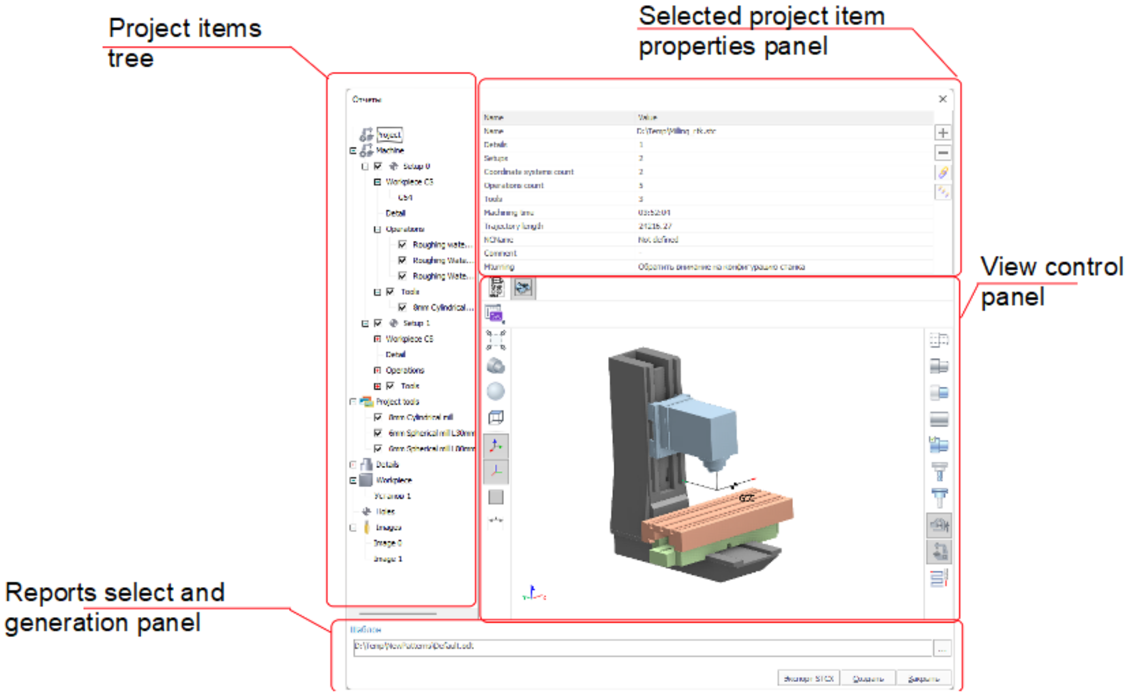
Read more about two-dimensional arrays and records, their declaration and using in documentation.

2.3.8 Report generation

Report generation window was changed.

Now it contain 4 parts:

- Project tree nodes at the left part of window;
- Parameters panel at the right top part;
- Graphical window at right bottom part;
- Report selection panel at the bottom part.

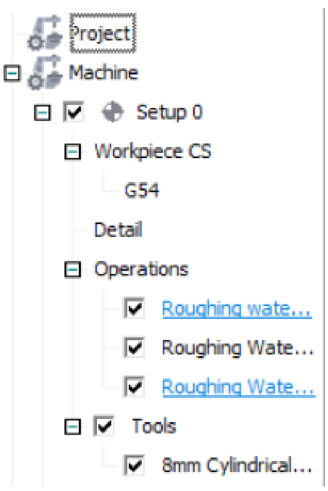


2.3.8.1 New features

- Selection items output in report;
- Creation customer parameters for each project tree node;
- Tuning images for each project tree node;
- Helper for pattern creation commands;
- Output operations by setups.

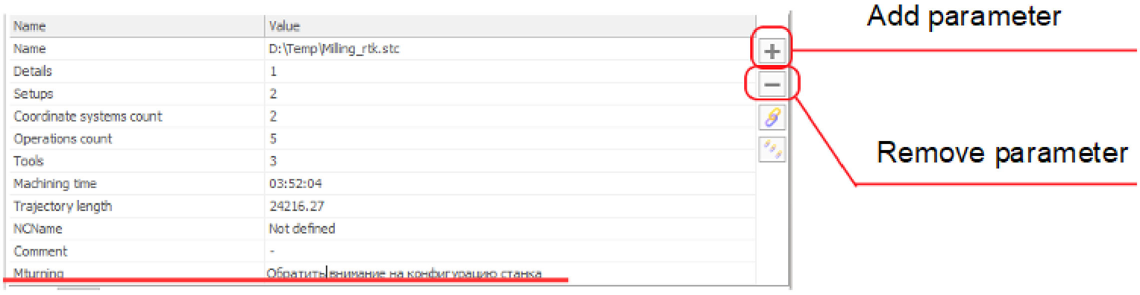
2.3.8.2 Selection items for output in the report

The main nodes of the project tree can be select for output in the project by checkboxes.



2.3.8.3 Adding customer parameters for each project tree node

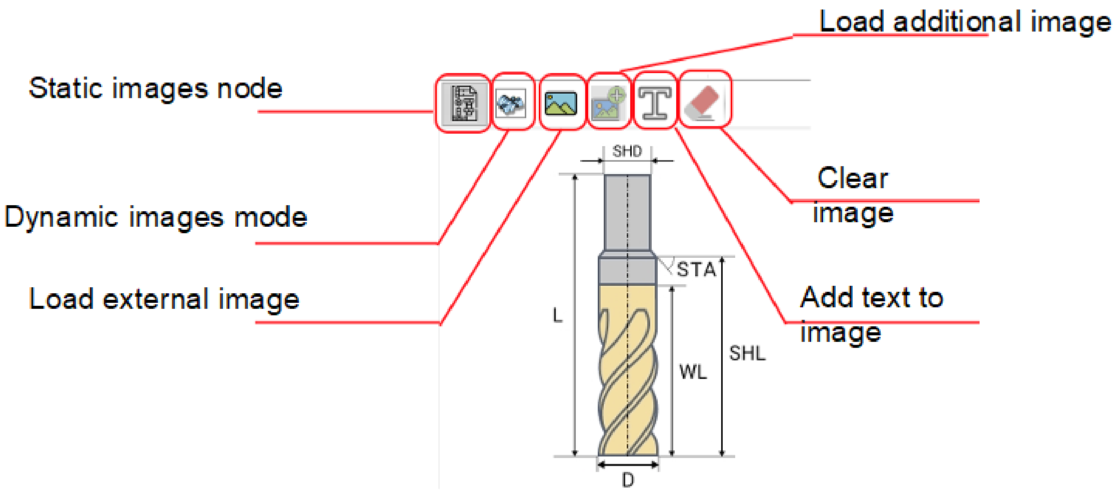
For each node it is possible add a custom parameters and output they in the result report.



2.3.8.4 Tuning images for the each project tree node

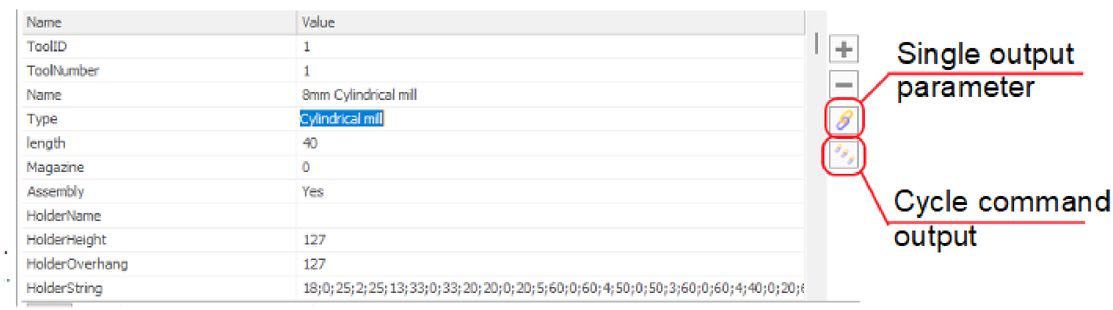
Images can be defined:
1) Dynamically with the graphic window;
2) By the static image or screenshot of graphic window;
3) By the external image.

On the static images can be added a additional images and text.



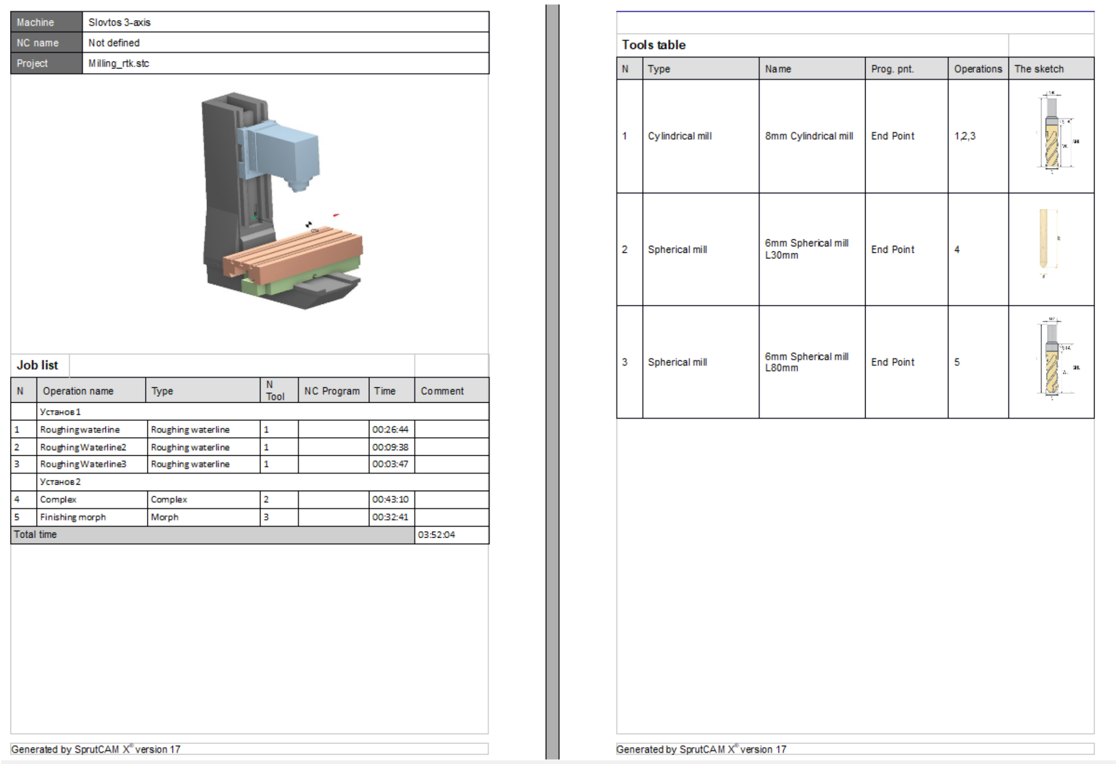
2.3.8.5 Helper for pattern creation commands

For creation the parameter output command it is enough to select the parameter in the list and press the correspond button.



2.3.8.6 Output operations by setups

Example of report.



Machine	Slovtos 3-axis
NC name	Not defined
Project	Milling_rtk.stc

Job list

N	Operation name	Type	N Tool	NC Program	Time	Comment
	Установ 1					
1	Roughingwaterline	Roughing waterline	1		00:26:44	
2	RoughingWaterline2	Roughing waterline	1		00:09:38	
3	RoughingWaterline3	Roughing waterline	1		00:03:47	
	Установ 2					
4	Complex	Complex	2		00:43:10	
5	Finishing morph	Morph	3		00:32:41	
Total time						03:52:04

Generated by SprutCAM X® version 17

Tools table

N	Type	Name	Prog. pnt.	Operations	The sketch
1	Cylindrical mill	8mm Cylindrical mill	End Point	1,2,3	
2	Spherical mill	6mm Spherical mill L30mm	End Point	4	
3	Spherical mill	6mm Spherical mill L80mm	End Point	5	

Generated by SprutCAM X® version 17

2.3.9 List of updated dialogs

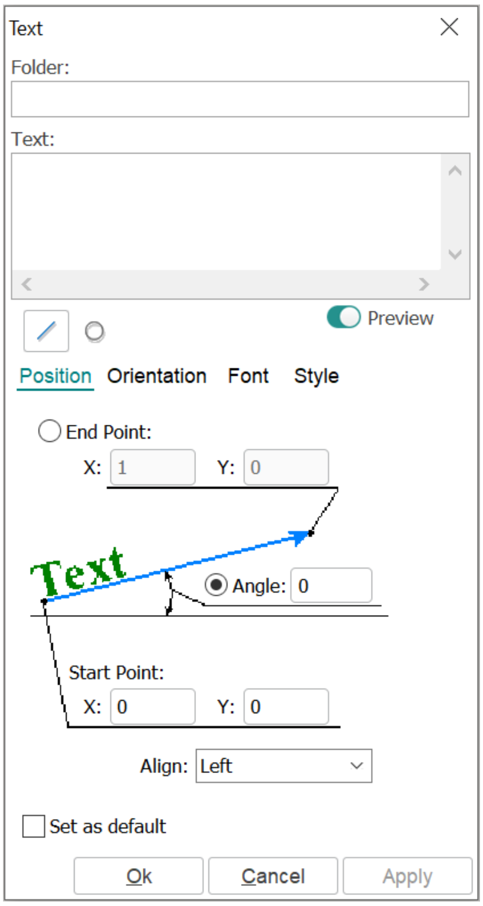
2.3.9.1 New popup dialogs design.

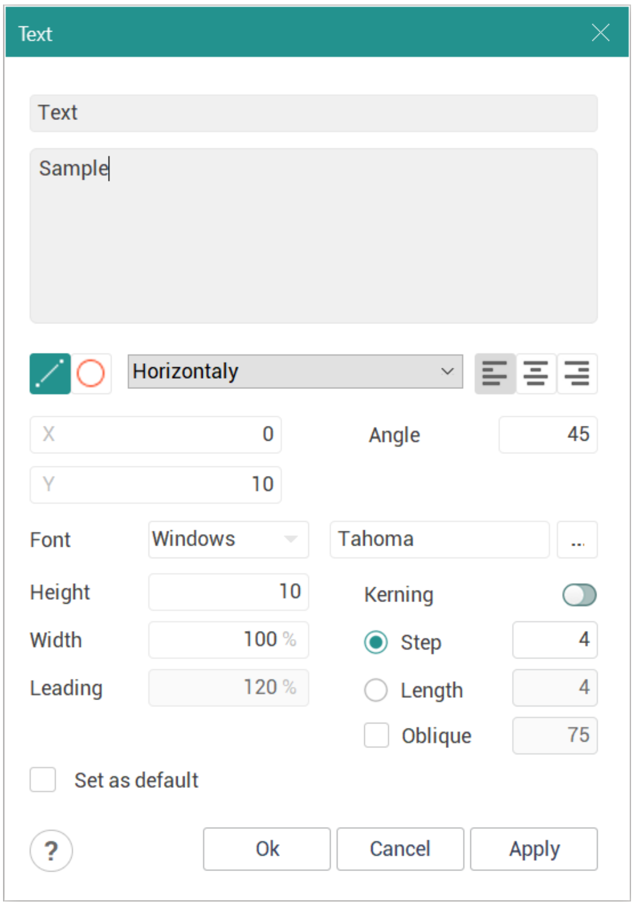
All window design is updated to conform to modern user interface standards and provide straightforward user experience. Take a look at some examples below

2.3.9.2 Geometry model

Text

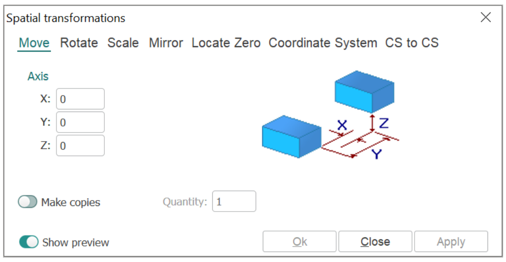
Text curve dialog options and features are placed on single page

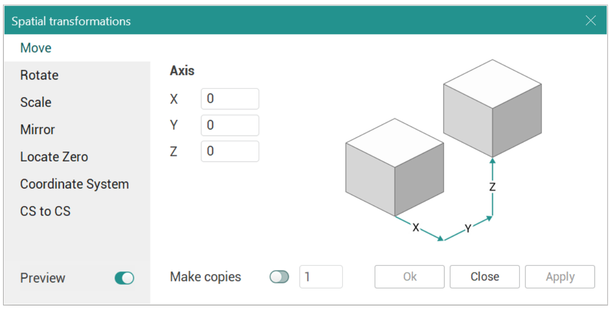


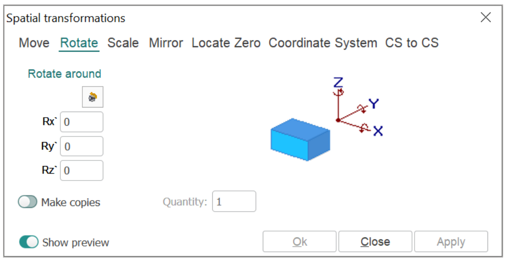


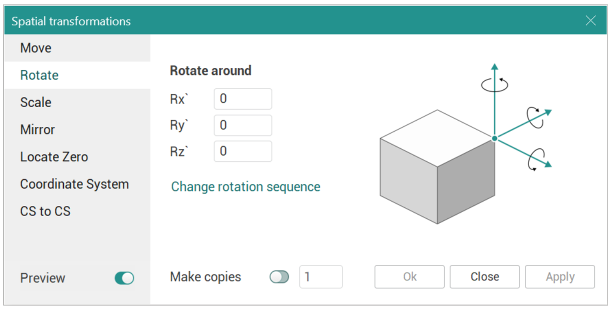
Spatial transformations

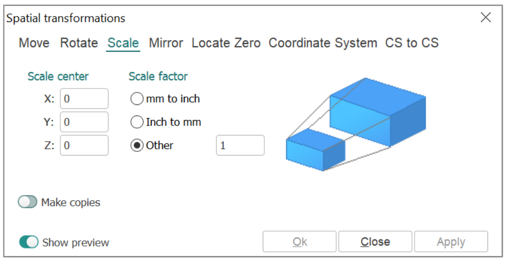
Geometry model transformations tools are rearranged for easier access, new graphics provide more accurate hints to what each option performs

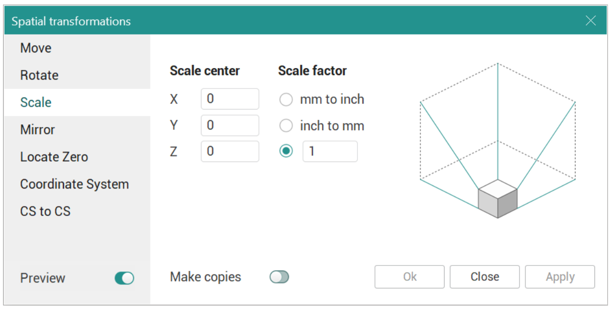

Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
Axis
X: 0
Y: 0
Z: 0
X Z Y
Make copies
Quantity: 1
Show preview
Ok Close Apply

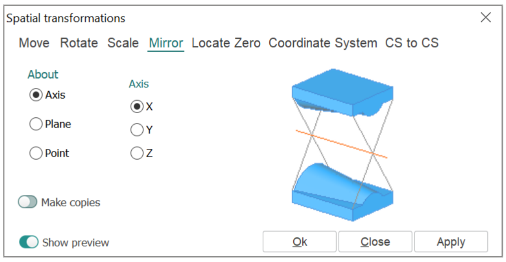

Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
Axis
X 0
Y 0
Z 0
Z X Y
Preview
Make copies 1
Ok Close Apply

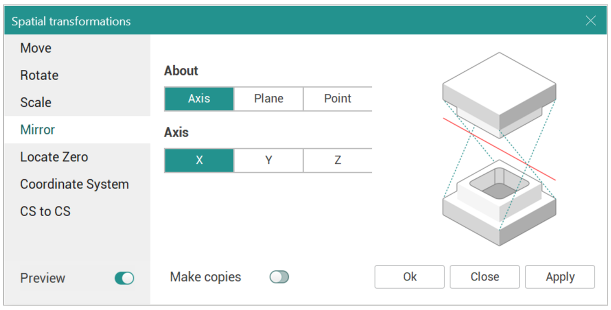

Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
Rotate around
Rx` 0
Ry` 0
Rz` 0
Z Y X
Make copies
Quantity: 1
Show preview
Ok Close Apply

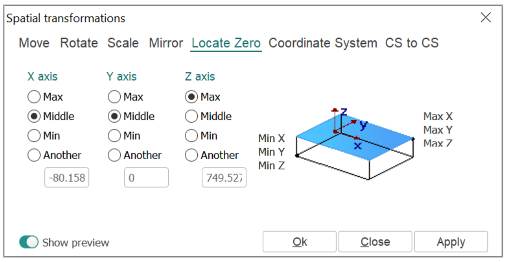

Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
Rotate around
Rx` 0
Ry` 0
Rz` 0
Change rotation sequence
Preview
Make copies 1
Ok Close Apply

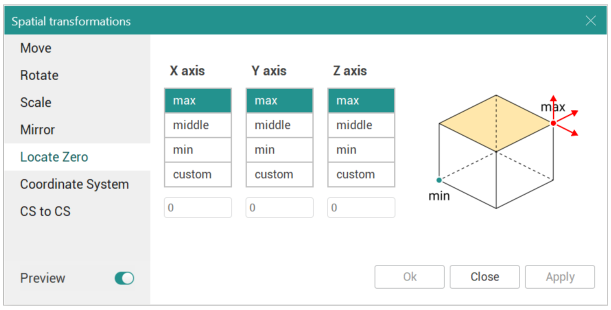

Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
Scale center
X: 0
Y: 0
Z: 0
Scale factor
mm to inch
Inch to mm
Other 1
Make copies
Show preview
Ok Close Apply

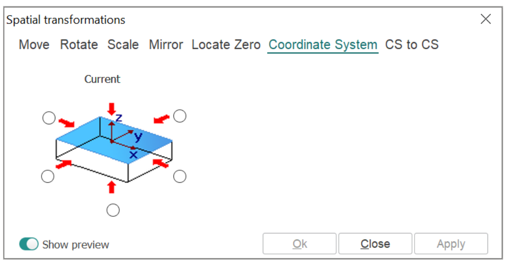

Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
Scale center
X 0
Y 0
Z 0
Scale factor
mm to inch
inch to mm
1
Preview
Make copies
Ok Close Apply

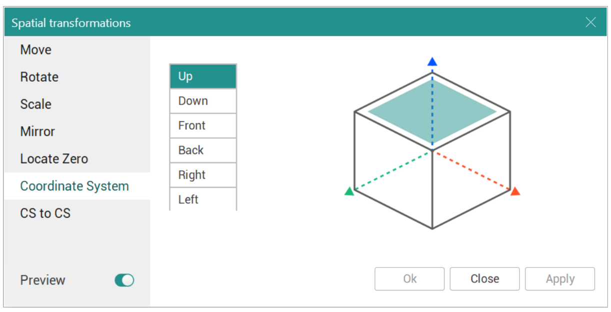

Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
About
Axis
Plane
Point
Axis
X
Y
Z
Make copies
Show preview
Ok Close Apply

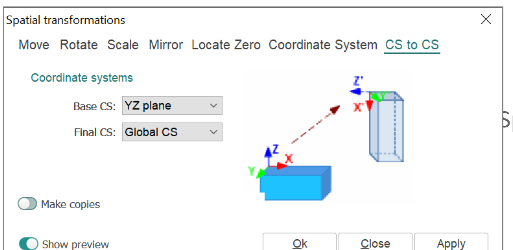

Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
About
Axis Plane Point
Axis
X Y Z
Preview
Make copies
Ok Close Apply

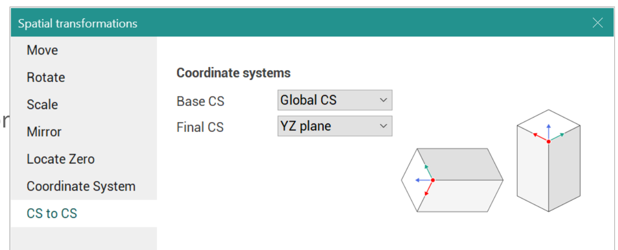

Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
X axis
Max
Middle
Min
Another
-80.158
Y axis
Max
Middle
Min
Another
0
Z axis
Max
Middle
Min
Another
749.527
Min X
Min Y
Min Z
Max X
Max Y
Max Z
Show preview
Ok Close Apply


Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
X axis
max
middle
min
custom
0
Y axis
max
middle
min
custom
0
Z axis
max
middle
min
custom
0
max
min
Preview
Ok Close Apply

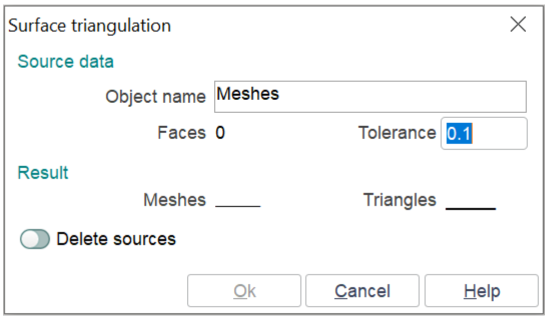

Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
Current
Show preview
Ok Close Apply

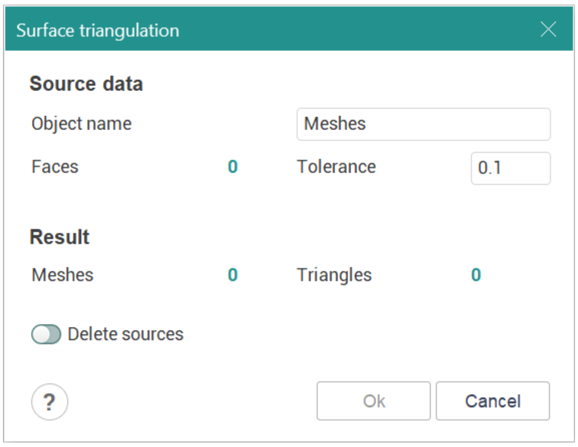

Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
Up
Down
Front
Back
Right
Left
Preview
Ok Close Apply


Spatial transformations
Move Rotate Scale Mirror Locate Zero Coordinate System CS to CS
Coordinate systems
Base CS: YZ plane
Final CS: Global CS
Make copies
Show preview
Ok Close Apply

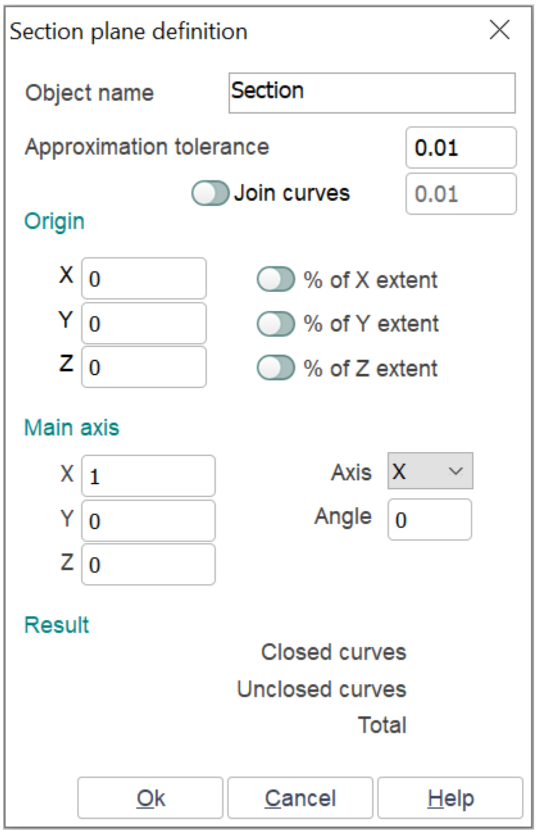

Spatial transformations
Move
Rotate
Scale
Mirror
Locate Zero
Coordinate System
CS to CS
Coordinate systems
Base CS Global CS
Final CS YZ plane

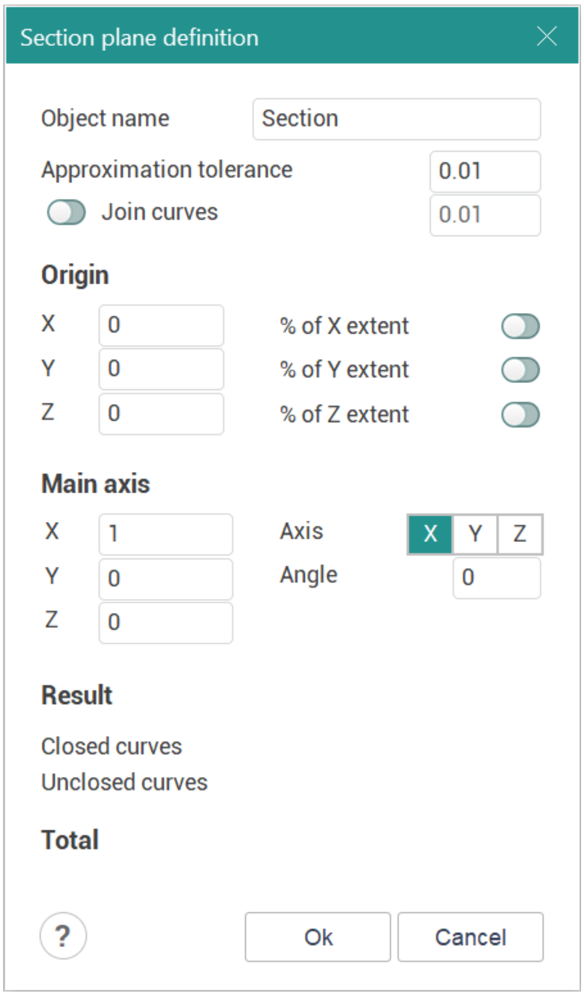
Triangulation



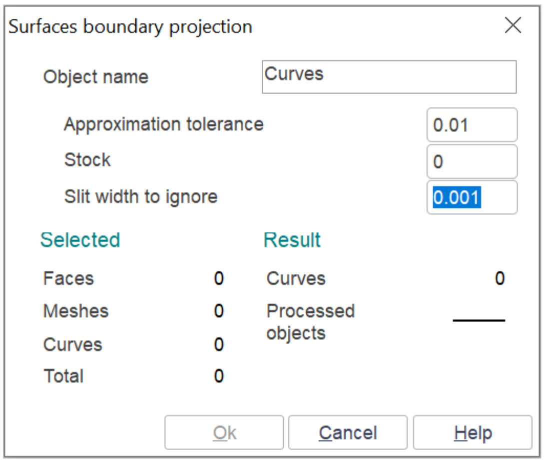


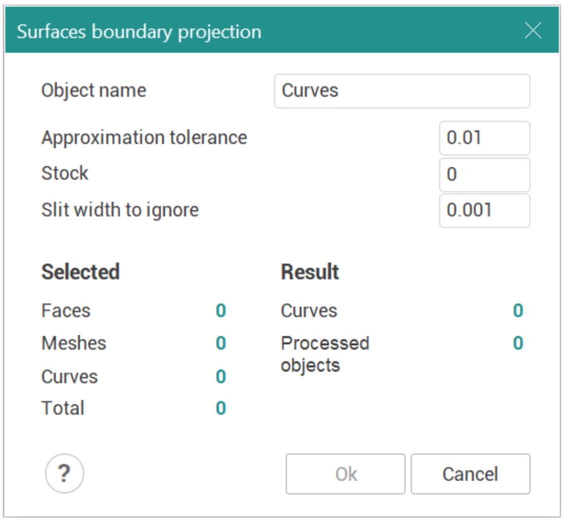
Section



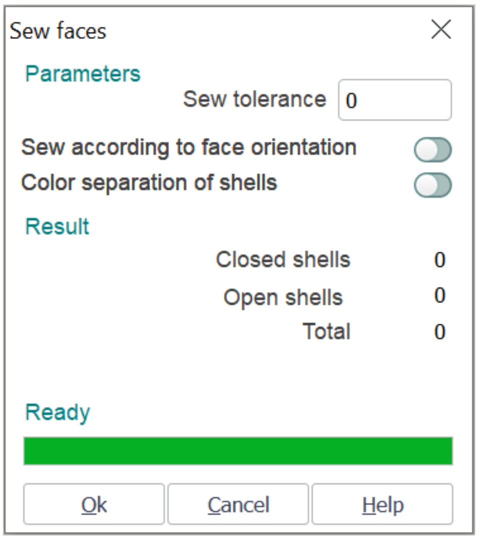


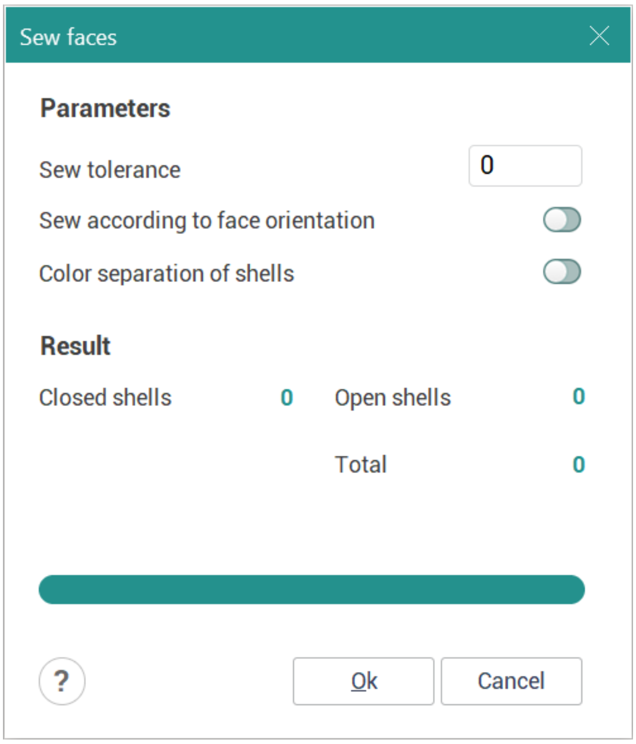
Boundary projection



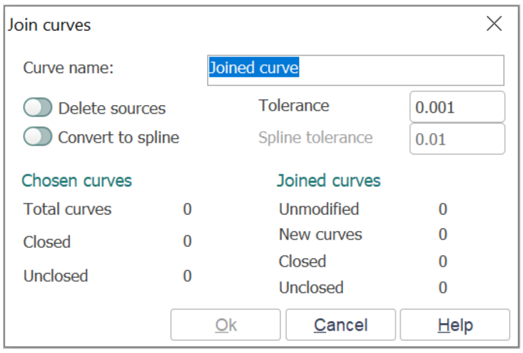


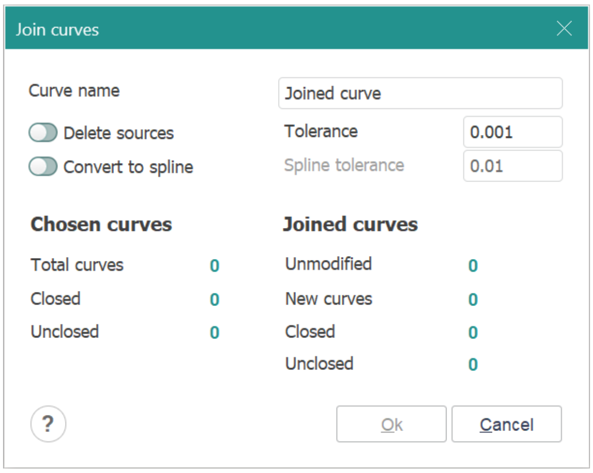
Sew faces



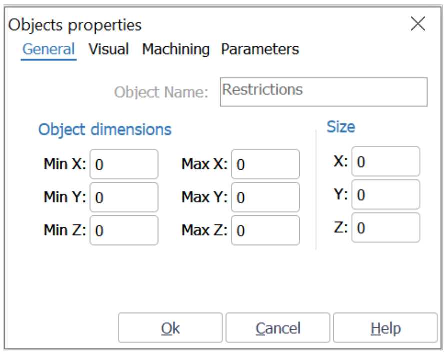


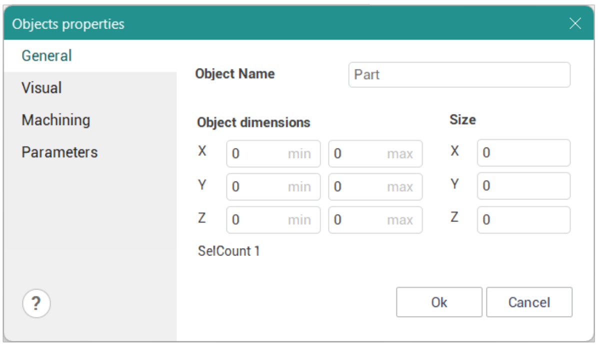
Join curves

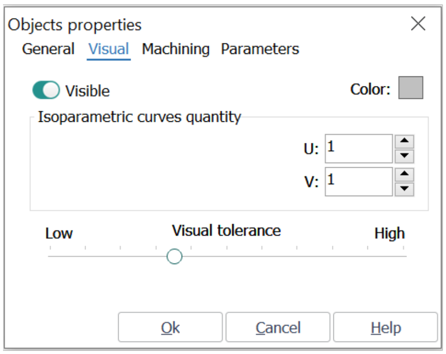

Join curves
Curve name:
Joined curve
Delete sources
Convert to spline
Tolerance
0.001
Spline tolerance
0.01
Chosen curves
Total curves 0
Closed 0
Unclosed 0
Joined curves
Unmodified 0
New curves 0
Closed 0
Unclosed 0
Ok
Cancel
Help

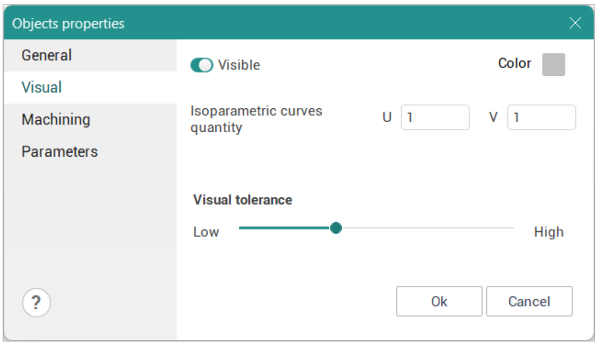

Join curves
Curve name
Joined curve
Delete sources
Convert to spline
Tolerance
0.001
Spline tolerance
0.01
Chosen curves
Total curves 0
Closed 0
Unclosed 0
Joined curves
Unmodified 0
New curves 0
Closed 0
Unclosed 0
?
Ok
Cancel

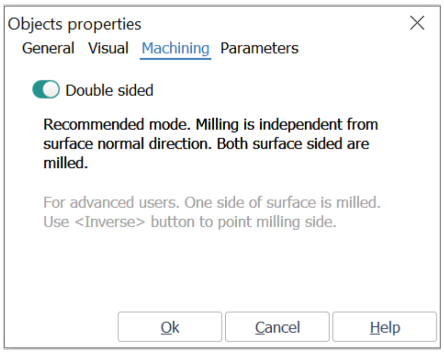
Model object properties

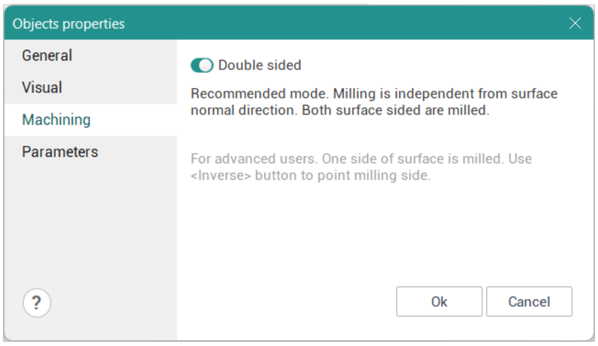


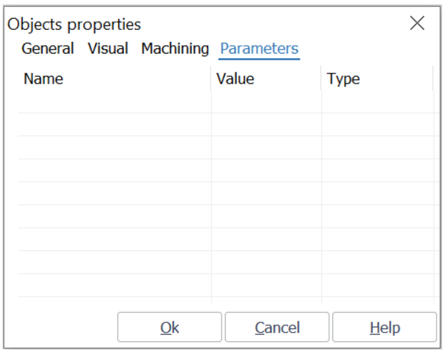


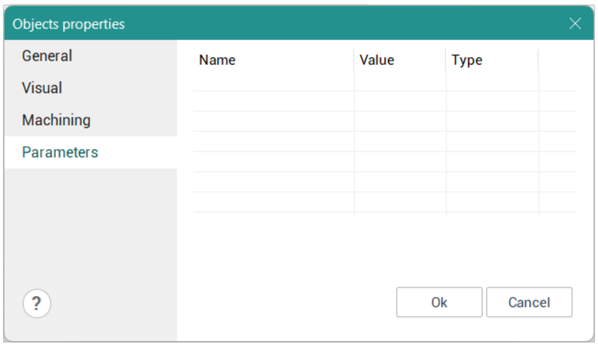






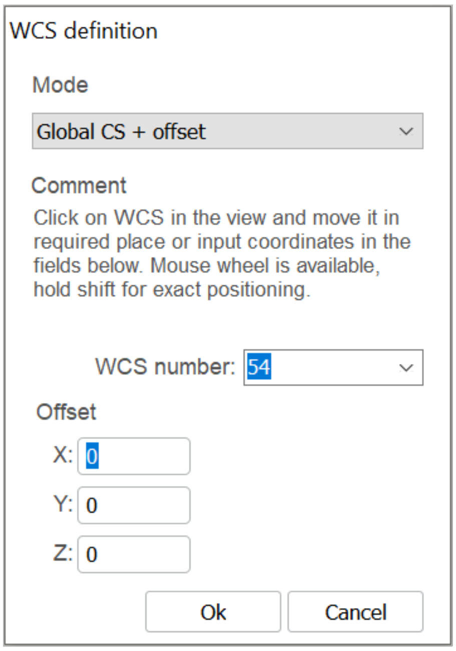


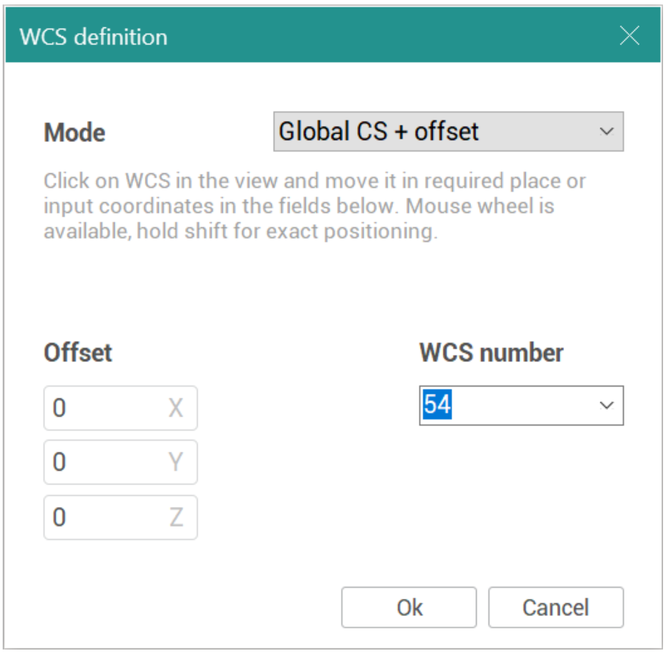




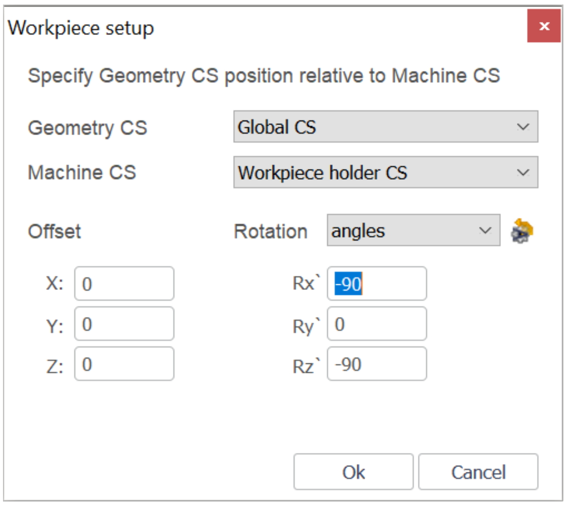
2.3.9.3 Technology

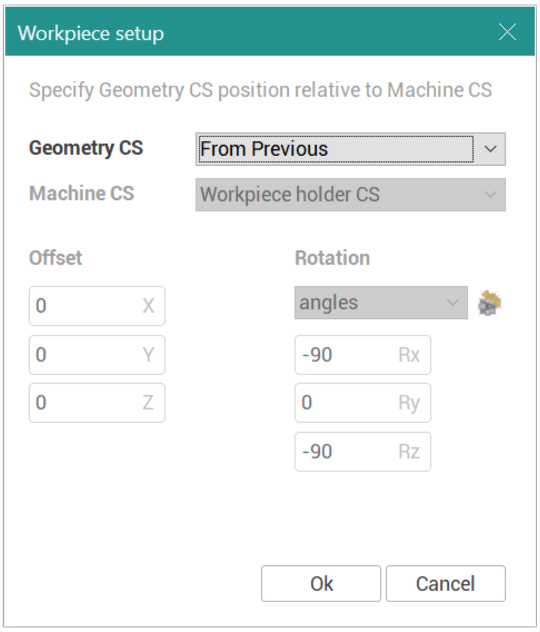
Workpiece CS definition



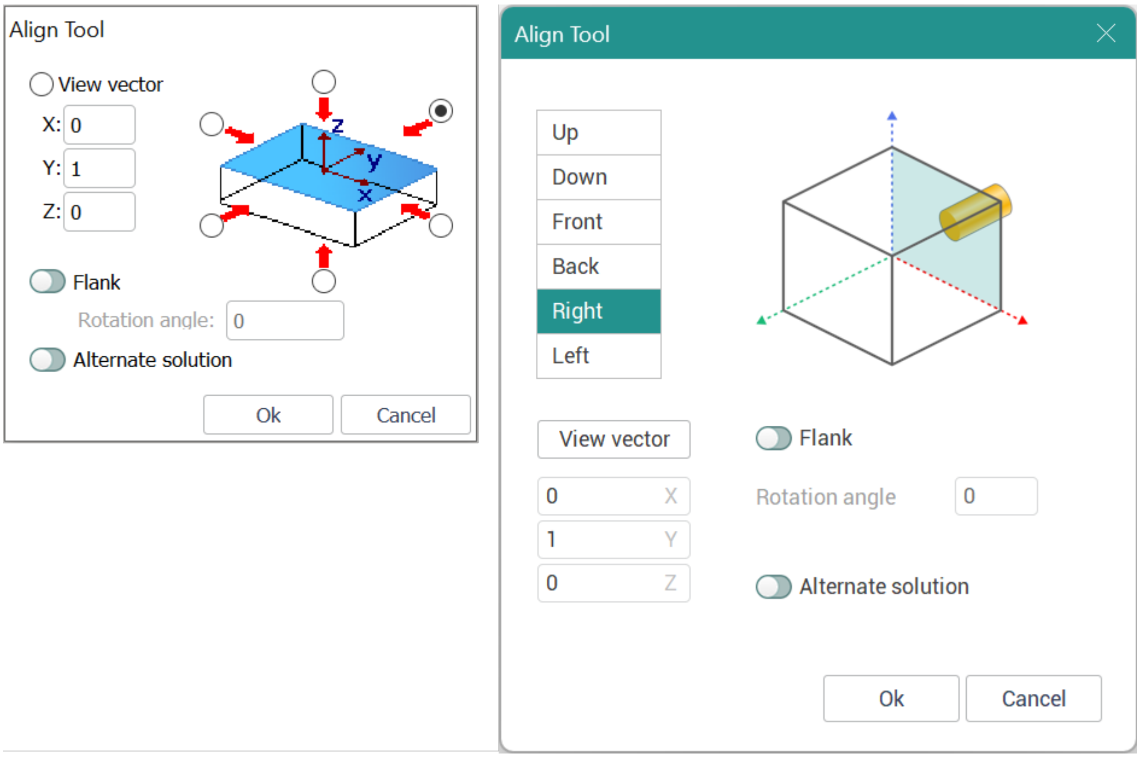


Workpiece setup

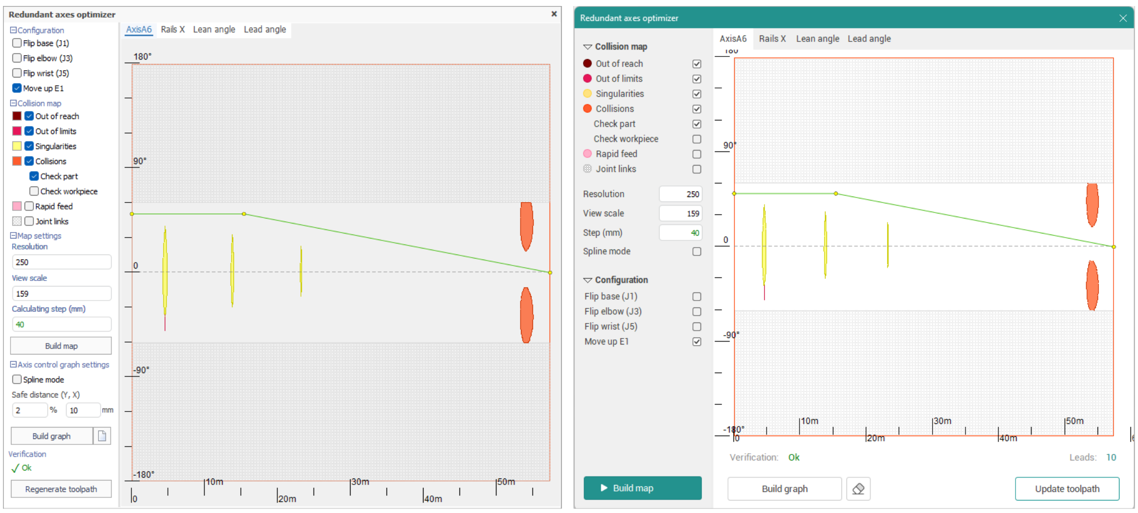




Rotary axis (Align tool)



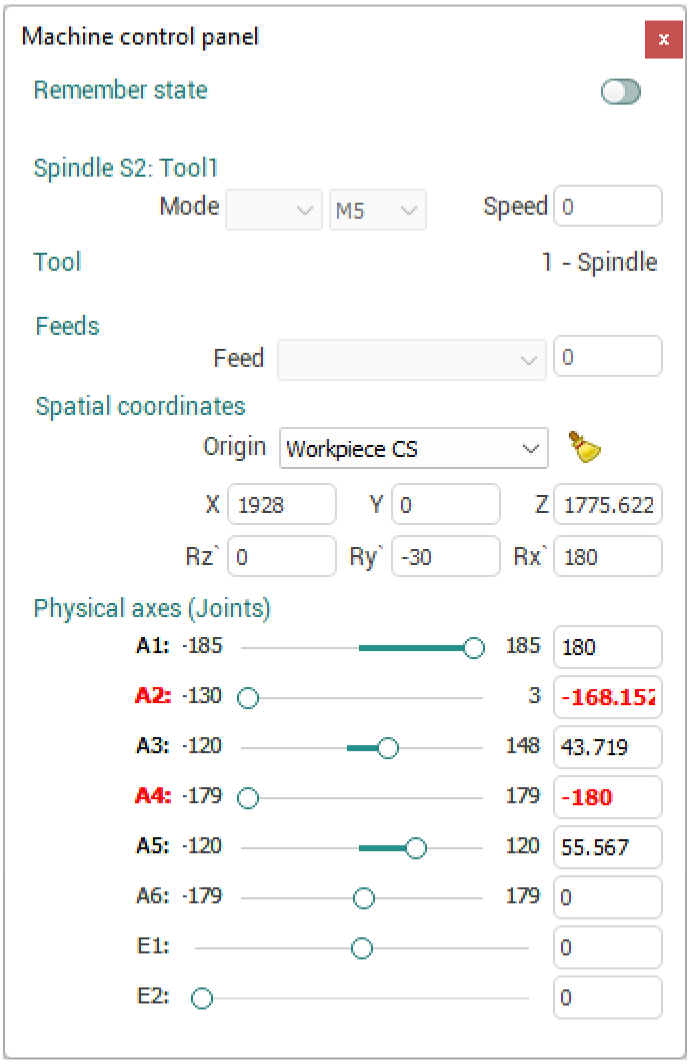
Robot axes map (Redundant axes optimizer)

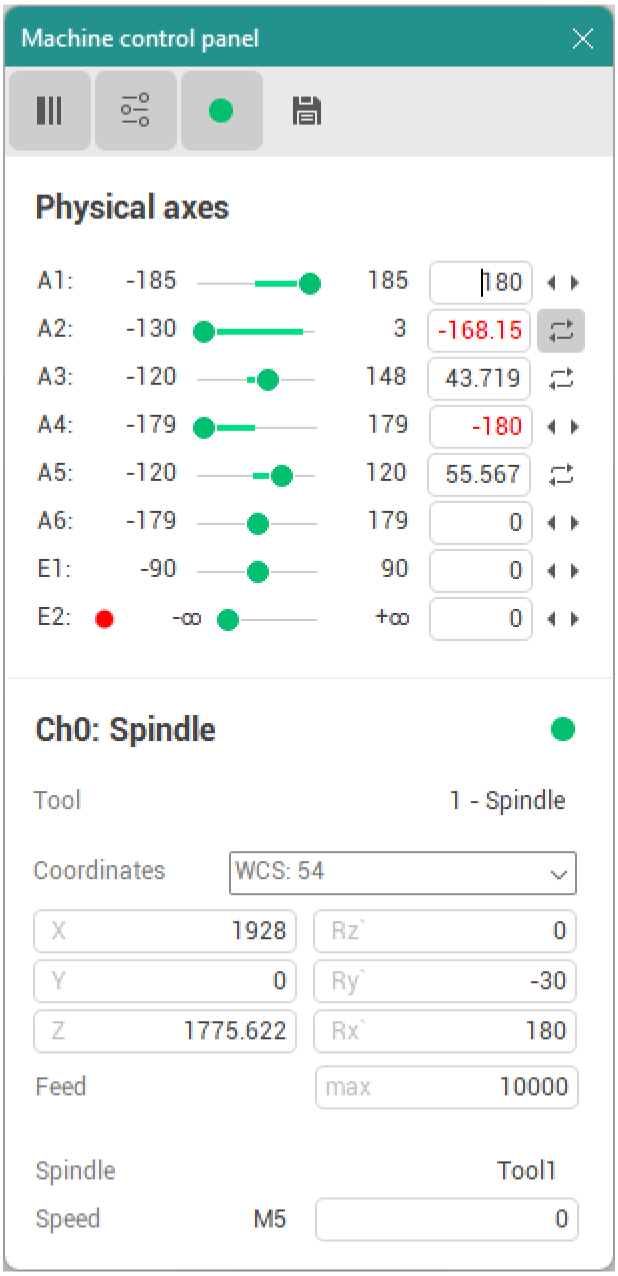


Machine control panel

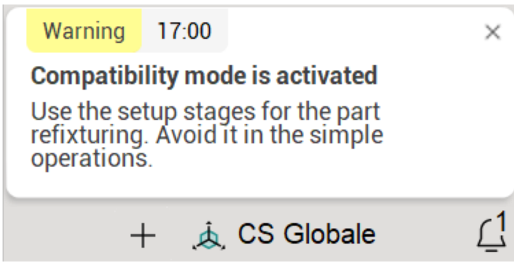
added in milestone 2

Axes brakes and robor flips could now be watched using the window.


Machine control panel
Remember state
Spindle S2: Tool1
Mode
M5
Speed
0
Tool
1 - Spindle
Feeds
Feed
0
Spatial coordinates
Origin
Workpiece CS
X 1928
Y 0
Z 1775.622
Rz` 0
Ry` -30
Rx` 180
Physical axes (Joints)
A1: -185 185 180
A2: -130 3 -168.152
A3: -120 148 43.719
A4: -179 179 -180
A5: -120 120 55.567
A6: -179 179 0
E1: 0
E2: 0



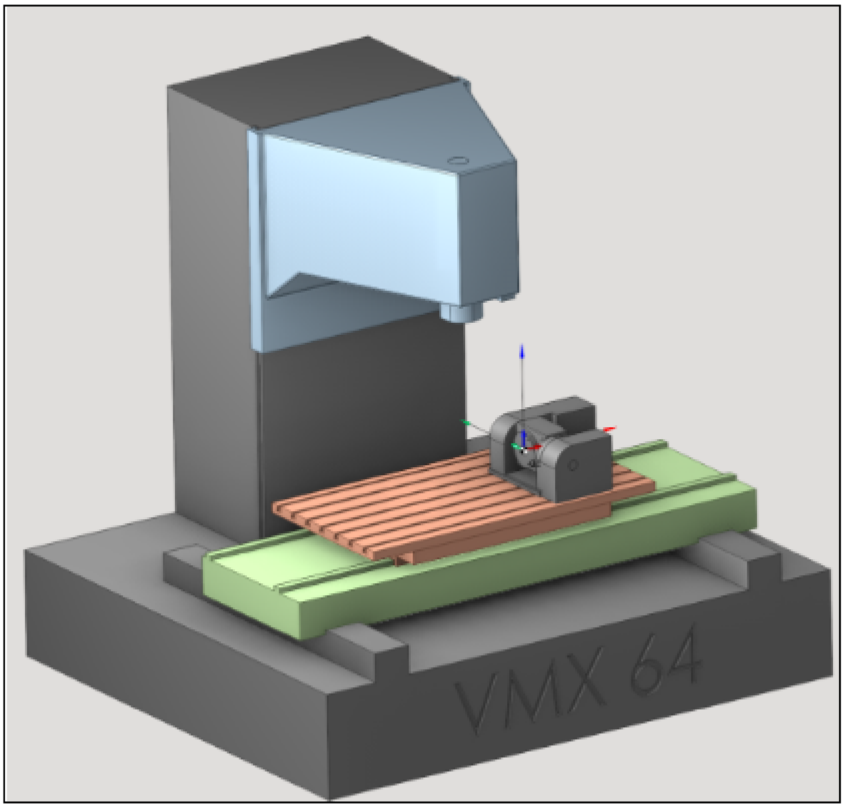
2.3.9.4 Notification popup



2.3.10 What's new in Machine Maker

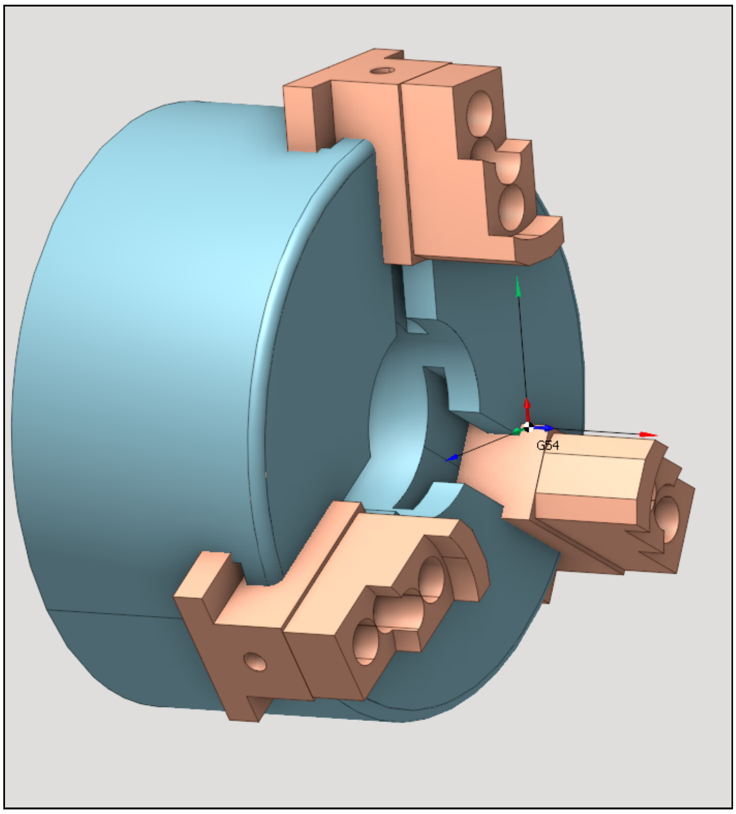
Support of optional machine equipment for Milling machines.

Now you can add optional axes (machine equipment) to your milling machines. That allows to create 3+2 machine schemas. It is possible to change position of optional axes (and even turn them off) in SprutCAM X.



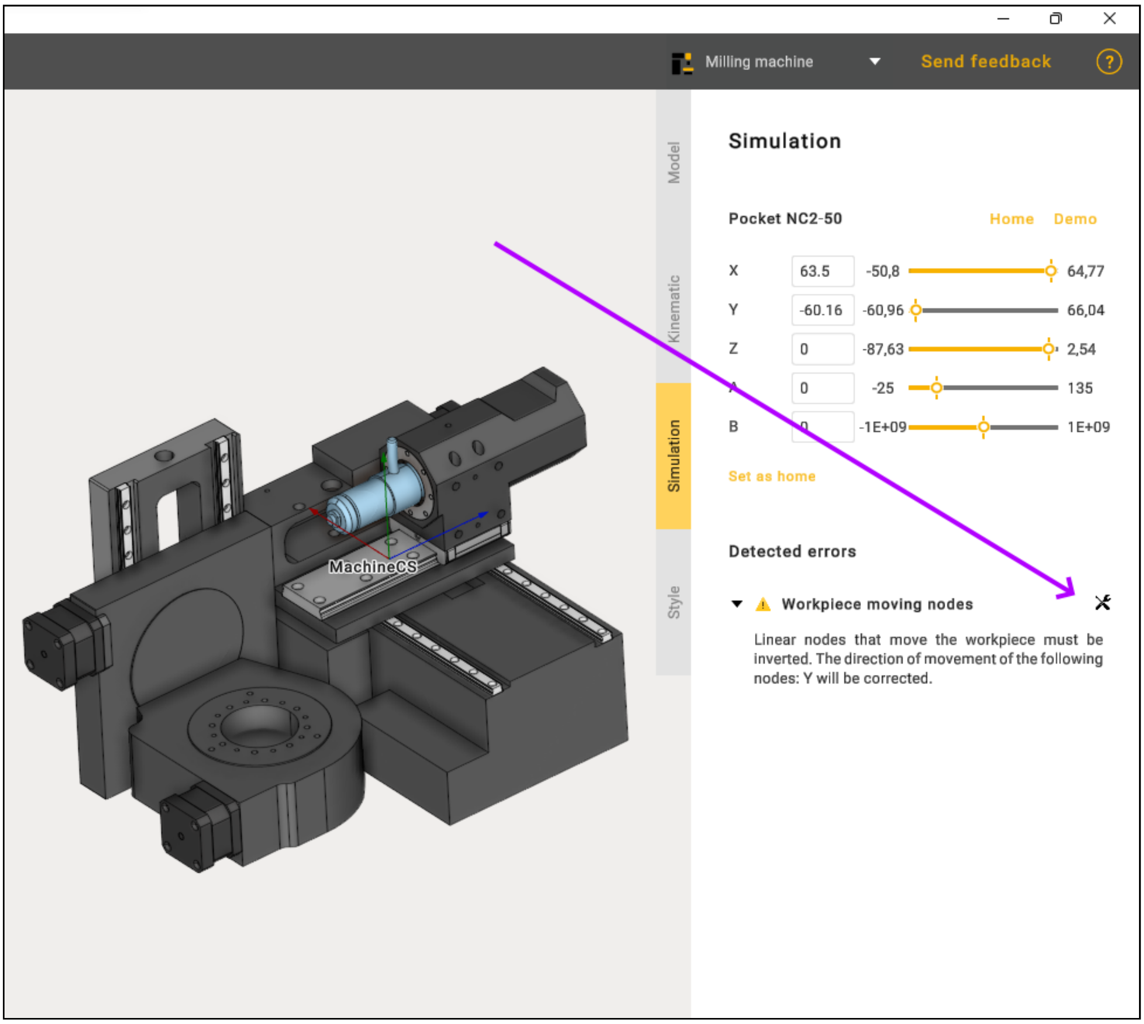
Support of machine equipment in Robot Cells.

Now you can add complex machine components like Chuck with Jaws to your Robot Cells.



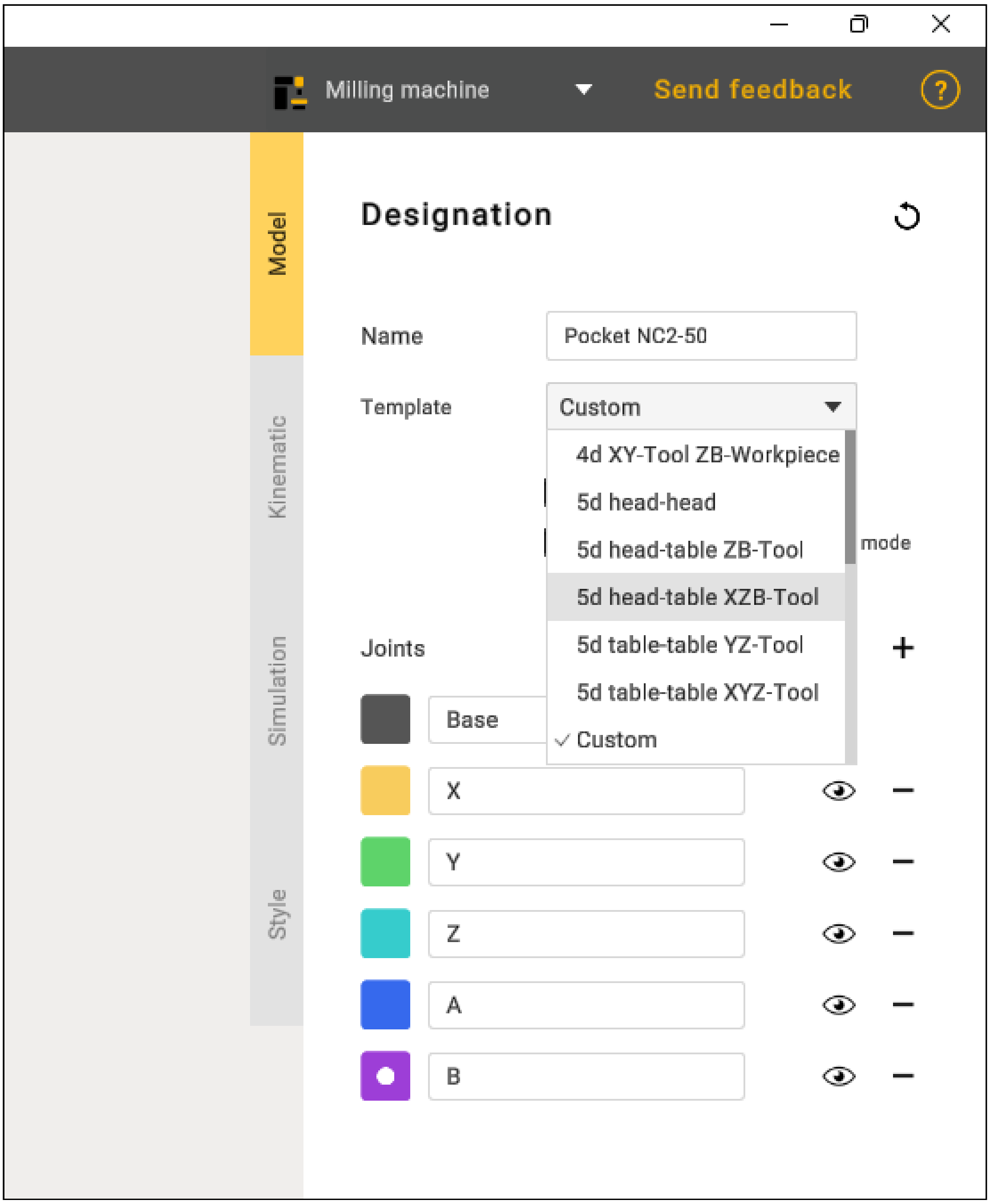
Machines validation.

MachineMaker will check you machine and warn about possible problems.

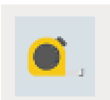


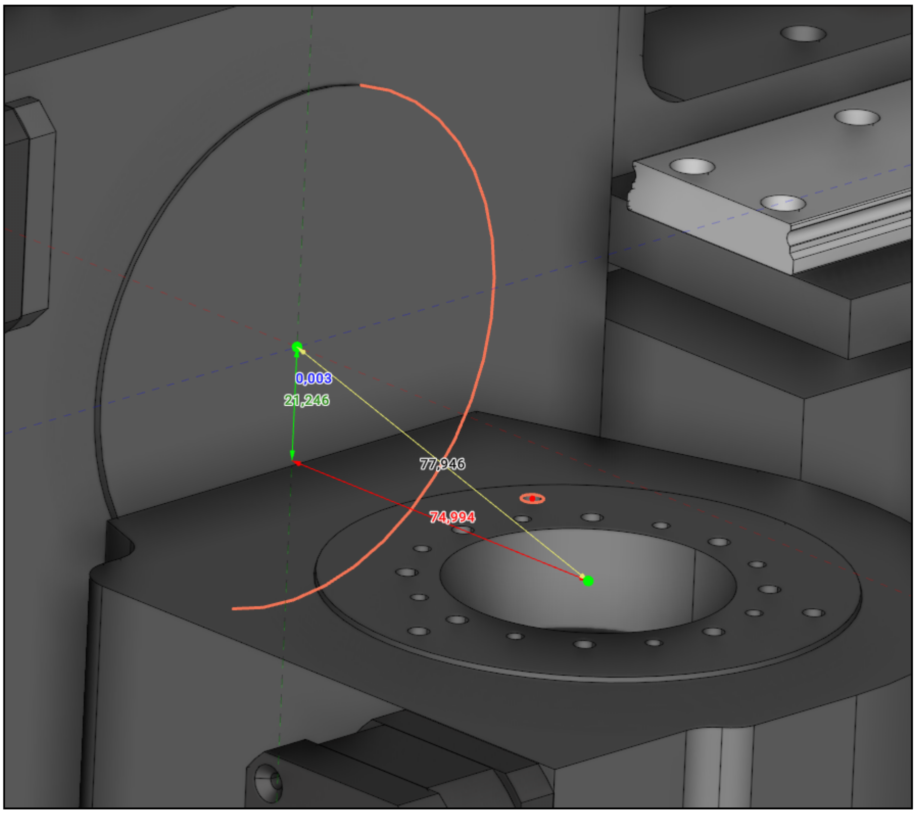
Mill machine templates.

You can select predefined kinematic schema template and create milling machine just in few minutes.



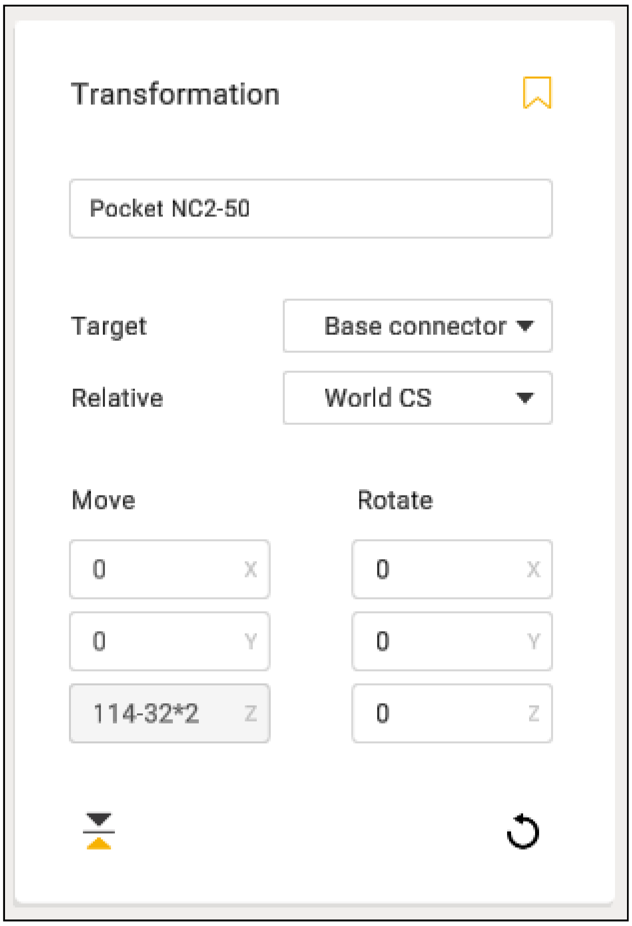
Geometry measuring.

`It is possible measure distance between any points of your geometry model.` 



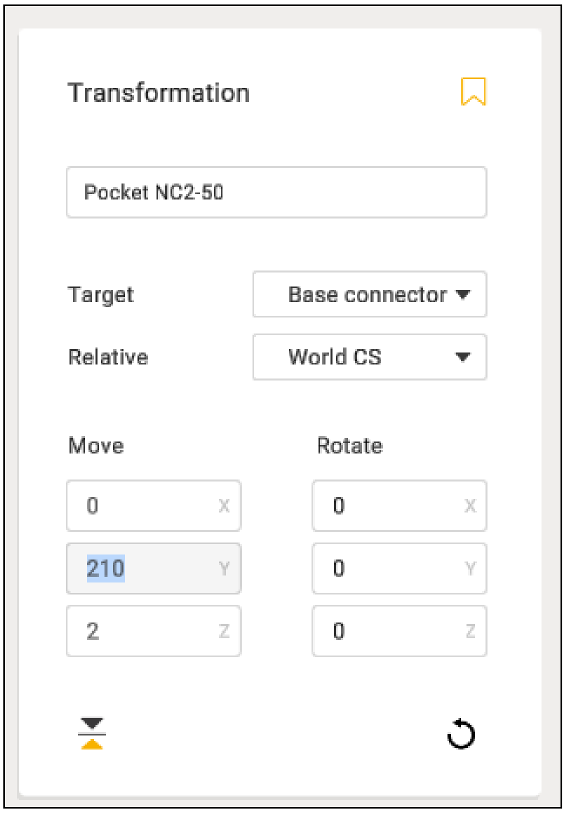
Smart input fields.

`Use any MachineMaker's input filed to calculate math expressions.`



Undo transformations.

`Input fileds supports Ctrl+Z key to undo changes.`

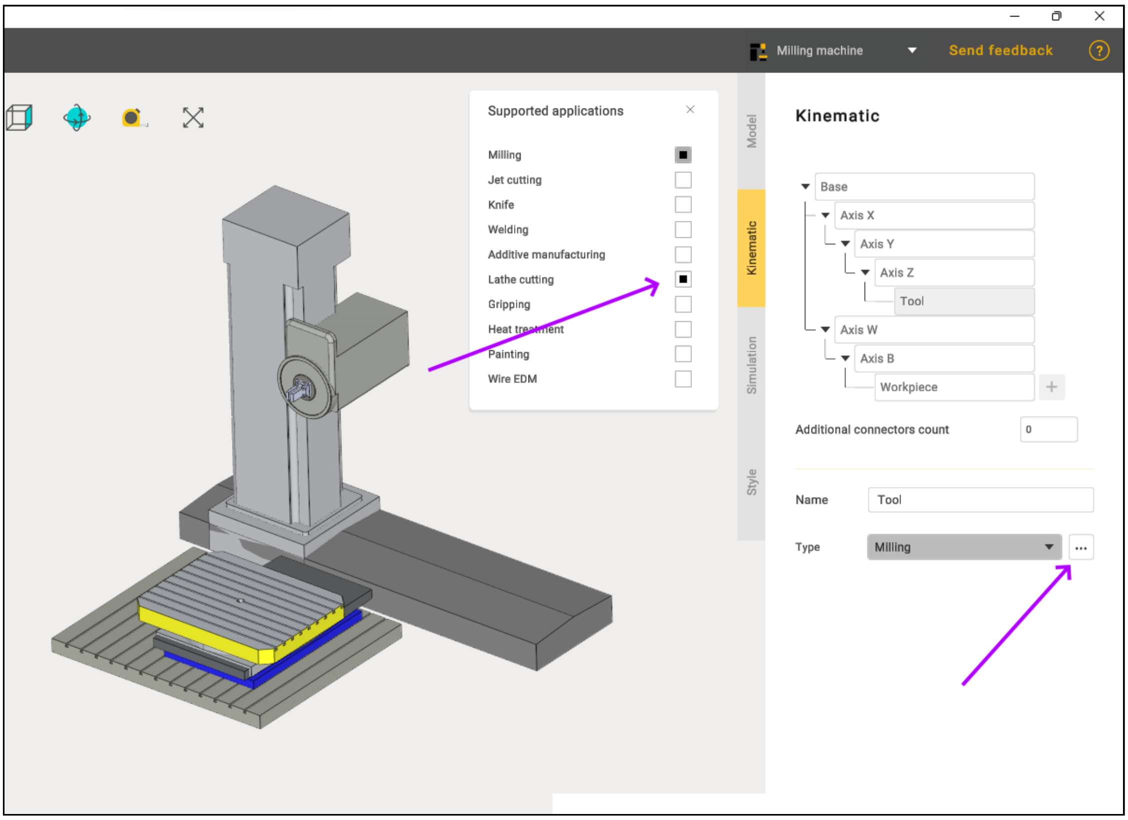


Multiaxes machines

MachineMaker is not limited to 5-axes machines anymore. It is possible to create machines with any axes count.

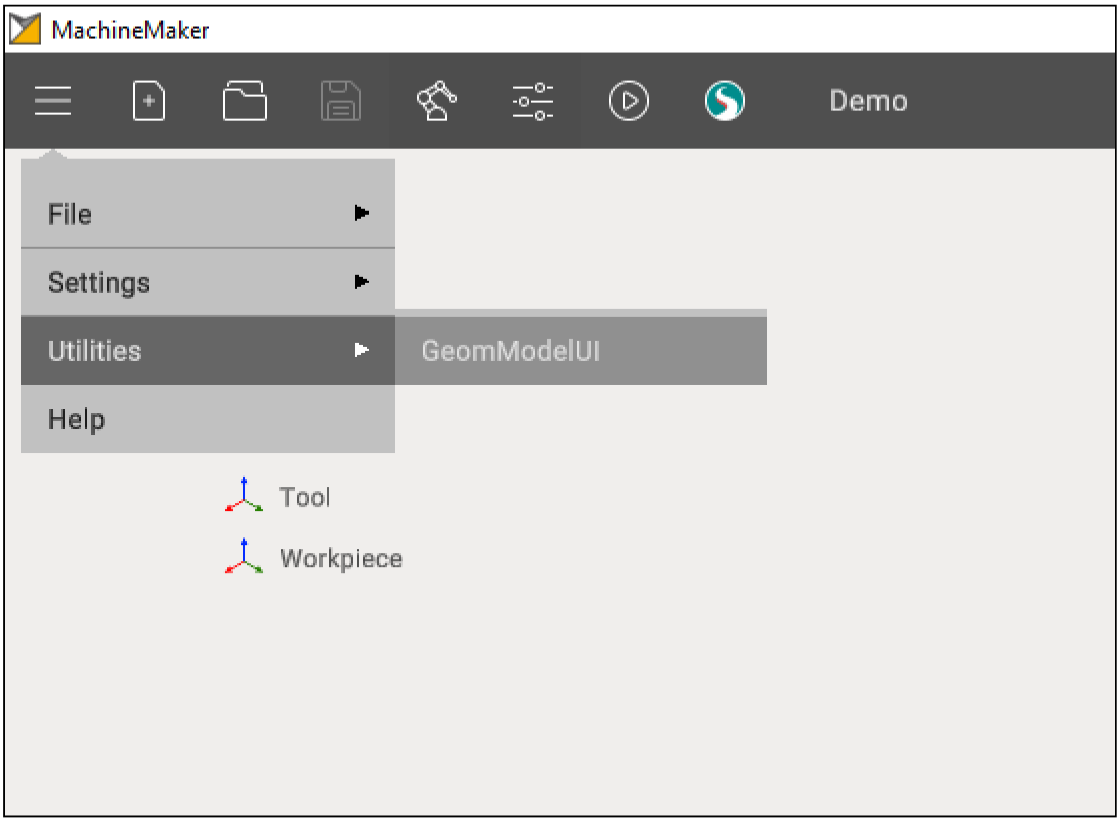
Lathe machines

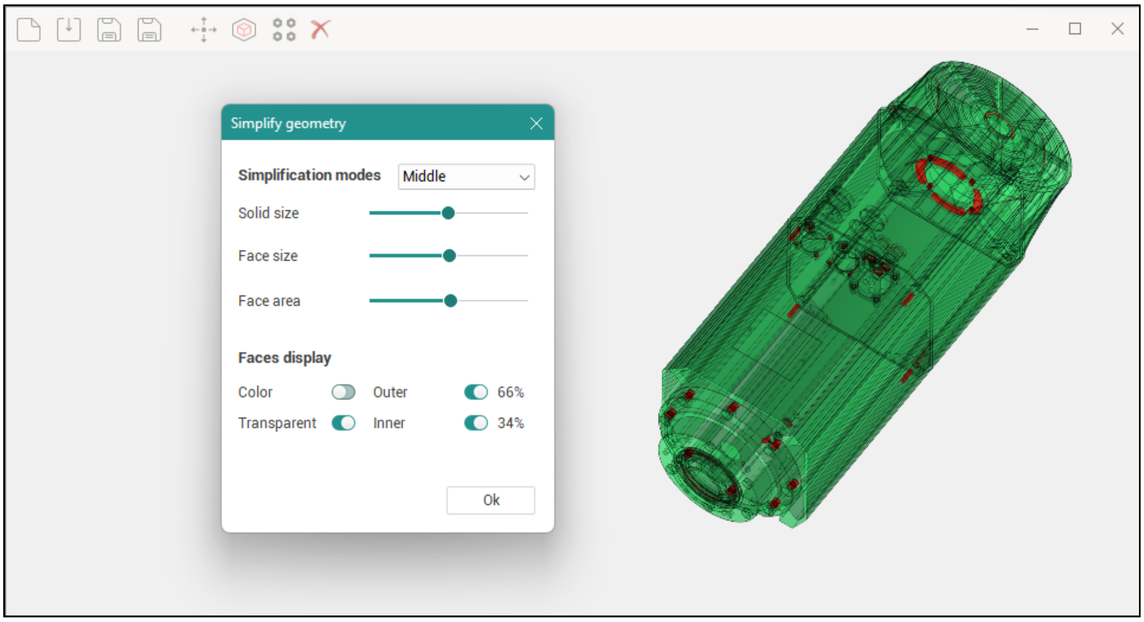
It is possible to define tool type. So you can create simple Lathe machines now.



Interactive 3D model simplifier (beta).

MachineMaker provides and amazing interactive 3D models simplifier.

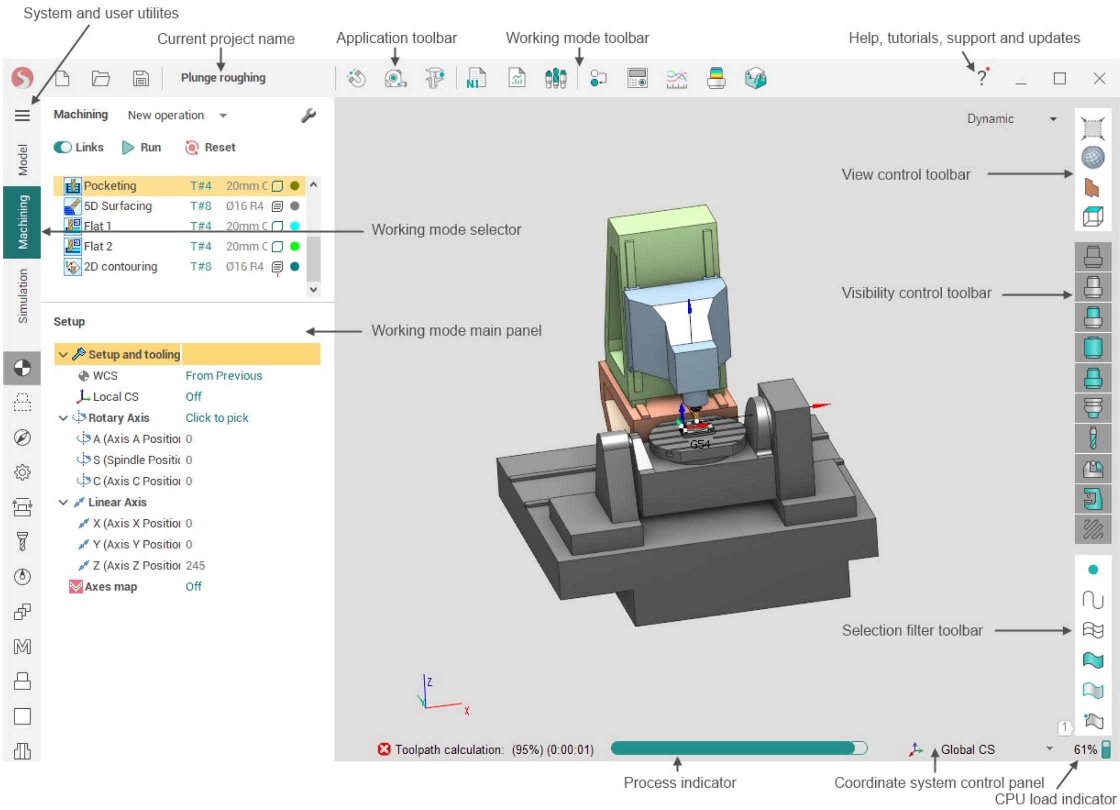

MachineMaker
Demo
File
Settings
Utilities
GeomModelUI
Help
Tool
Workpiece


Simplify geometry
Simplification modes
Middle
Solid size
Face size
Face area
Faces display
Color
Outer
66%
Transparent
Inner
34%
Ok

3 General information

3.1 System's main window

The system's main window has the following view:



Graphics window is in the center. Depending on current system 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.

Application toolbar is at the top left corner of the window. It contains buttons to manage projects and control global features like Smartsnap.

Working mode toolbar is at the center of window topbar. It contains features useful in current system mode.

Project title bar shows the name of current project.

Utilities and Help buttons are located to the right of the window topbar. **Utilities** drop-down menu provides system and user utilities such as Postprocessors generator. Use **Help** drop-down menu to access help, tutorials and get support.

Working mode panel is on the left. Depending on current system mode it shows geometry model items, technology tree and properties inspector, or tool path tree (CL-data). Click on one of tabs (<**Geometry**>, <**Technology**>, <**Simulation**>) to change system mode.

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

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

Use Selection filter toolbar to change whether objects of the appropriate type should be selected from the screen. These settings also control what objects are visible in the listview in **Geometry** mode.

Process indicator displays current calculation progress.

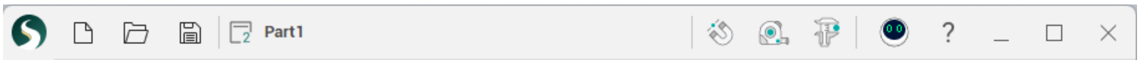
Use Geometrical coordinate system control panel to add, remove and modify coordinate systems.

The CPU indicator displays current overal load on the CPU.

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

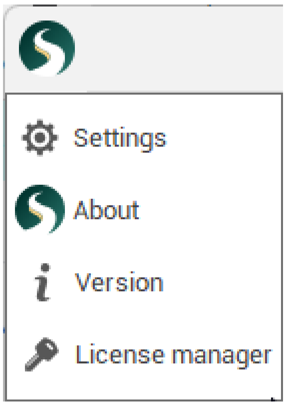
3.1.1 Application toolbar

Application toolbar placed on the top panel of the main window.



There are drop-down menus associated with some buttons.

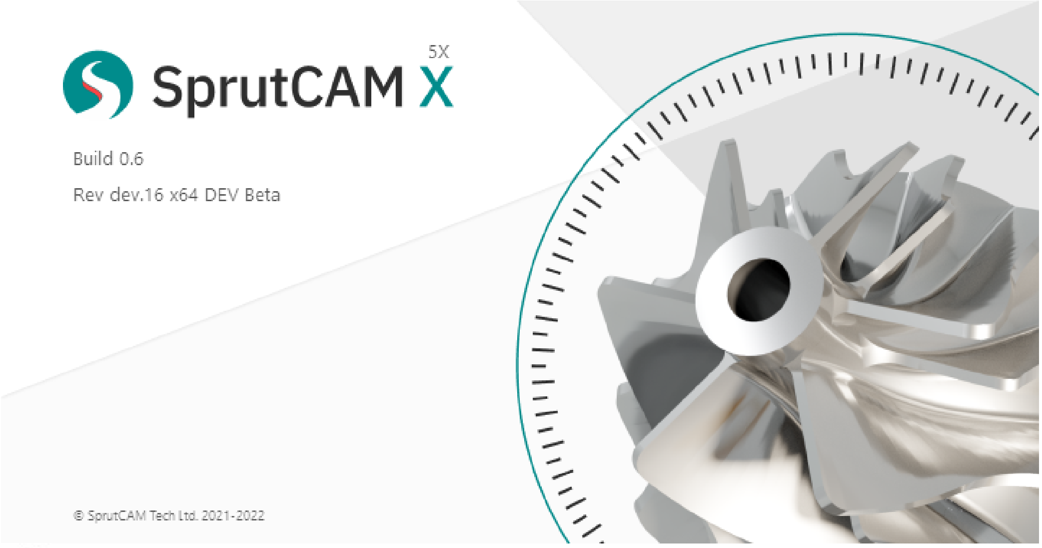
3.1.1.1 Application button menu



The main drop-down menu appears when you click on the system logo and contains the following items.

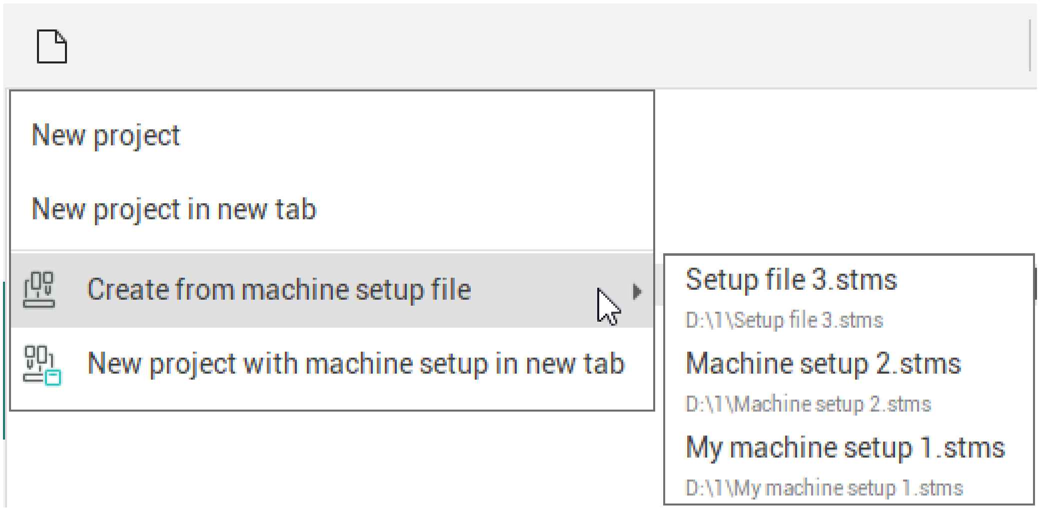
-  Settings. Opens a window for editing system settings.

- About. Splash screen of the application will appear.



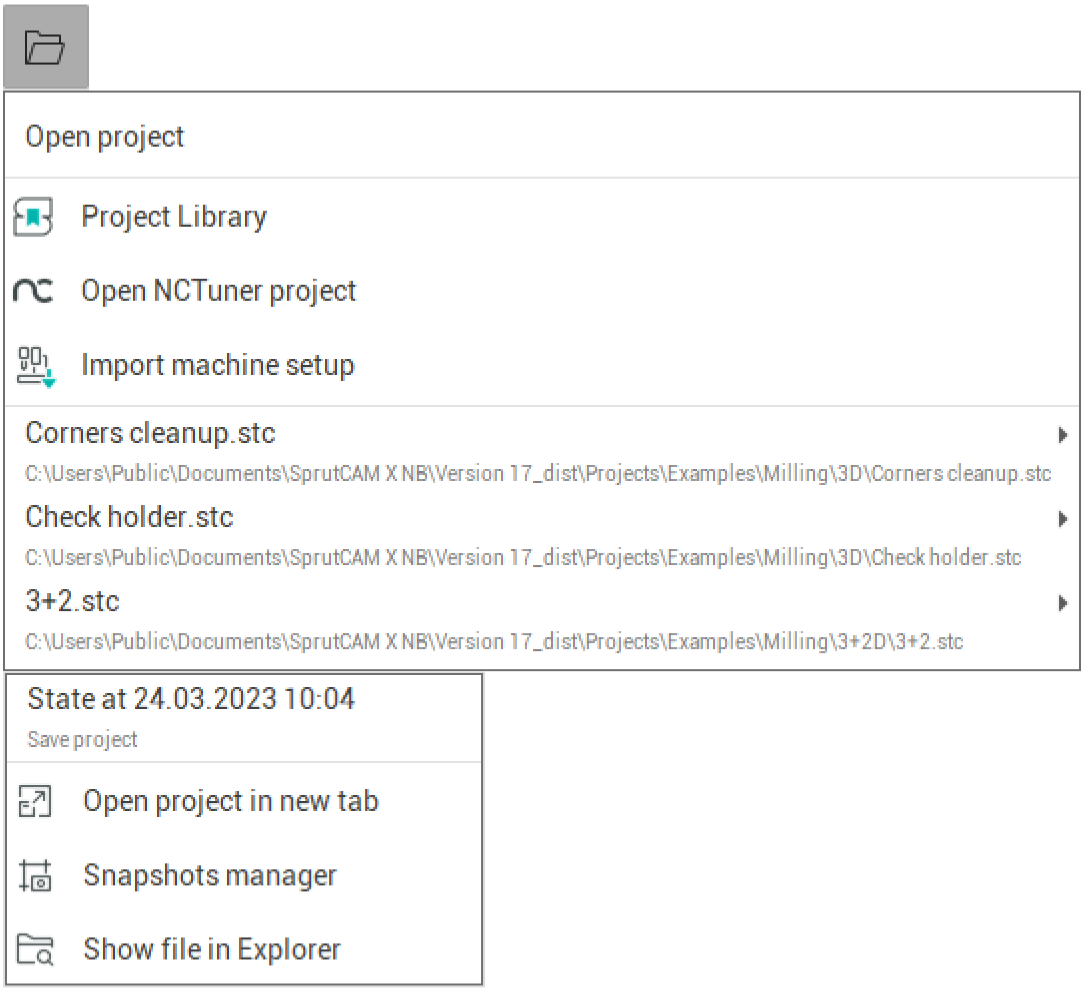
- Version. The Application version information window will appear.
- Licence manager will appear.

3.1.1.2 New project button menu



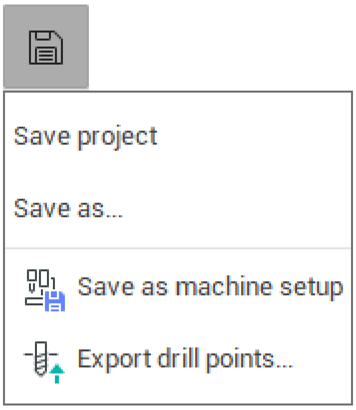
- New project – closes the current project and initiates the system state anew in the current tab.
- New project in new tab - opens a new tab and creates a new project inside it.
- Create from machine setup file – closes the current project and creates a new project from the machine setup file. Here you can choose one of the recently opened machine setup files or open the dialog to find a file on your drives.

3.1.1.3 Open project button menu



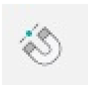
- Open project – loads an existing project from a file, a standard file selection dialog opens.
- Project library – a window that allows you to search and open sample projects from around the world.
- Open NCTuner project – loads a project created in NCTuner. If you hold the cursor on this menu item, the option to open the project in a new tab will appear.
- Import machine setup – imports data from a machine setup file into the current project.
- The following is a list of previously loaded projects. Each project in this list has its own drop-down menu with additional actions you can do with the project,
    - The first one or more items allow you to quickly load recent backup versions (snapshots) of the project.
    - Open project in new tab - opens selected project in a new window.
    - Snapshots manager - opens the window to manage all snapshots (backup versions) of the project.
    - Show file in explorer - opens Windows explorer and selects the file of this project.

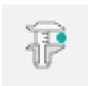
3.1.1.4 Save project button menu



- Save project – saves all changes in the current project. If the project has not been saved before, a new name will be requested.
- Save As... – saves the project under a new name. The save dialog will appear.
- 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 a current operation (if it has Holes page in properties) to DXF-file. The file with these points can be imported later to the model page of another project.

3.1.1.5 Additional tools and buttons

 Smart snap. Enable / disable snapping to objects in the graphics window.

 Geometry measuring. Activates the mode of measuring objects in the graphics window.

. Turns on the mode of measuring distances between two arbitrary points on the screen.

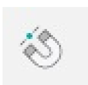
 - AI assistant button opens the AI assistant chat window.

 Help button opens the help menu.

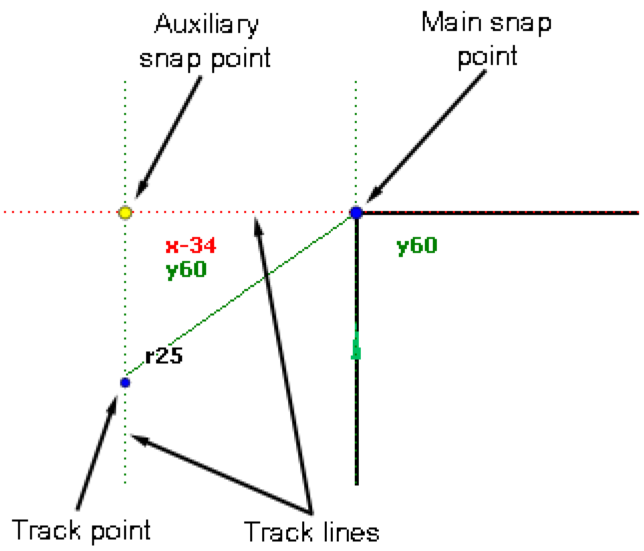
See also:

System's main window

3.1.1.6 Smart snap

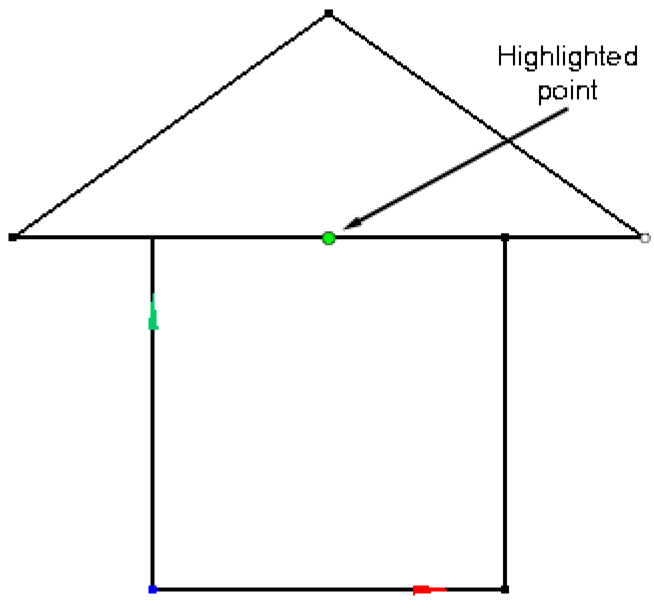
The Smart snap  button enables / disables snapping to objects in the graphics window. This is useful when creating new or editing existing geometric objects, such as points, lines, contours, coordinate systems, etc.

For example, using smart snap feature you can easily create exact well constrained drawings on the fly.



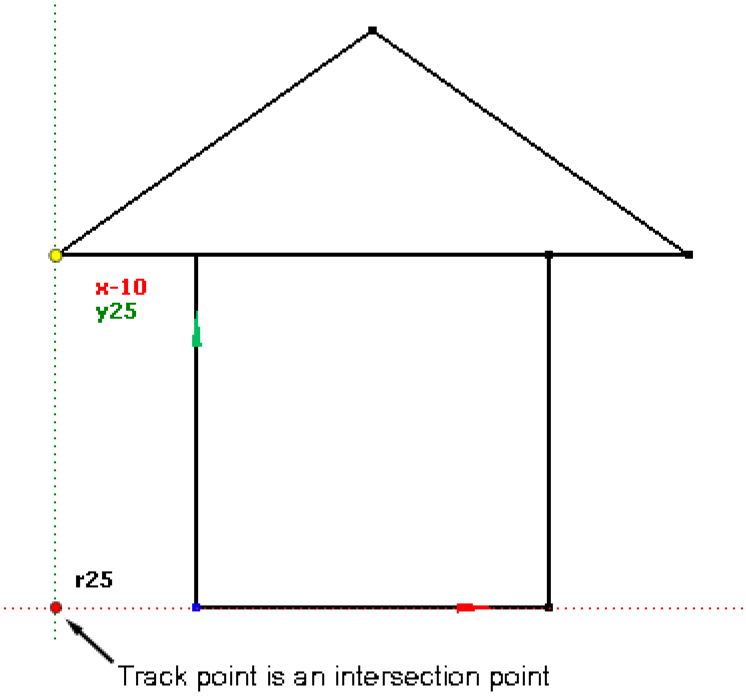
The main building blocks of the smart snap are:

- Highlighted points. When you move the cursor over geometry entities the points you can snap to are highlighted with the lime color. These are the terminate points of cuts and arcs, the distinct points, the center points of arcs and circles, the middle points of cuts, the intersection points etc.

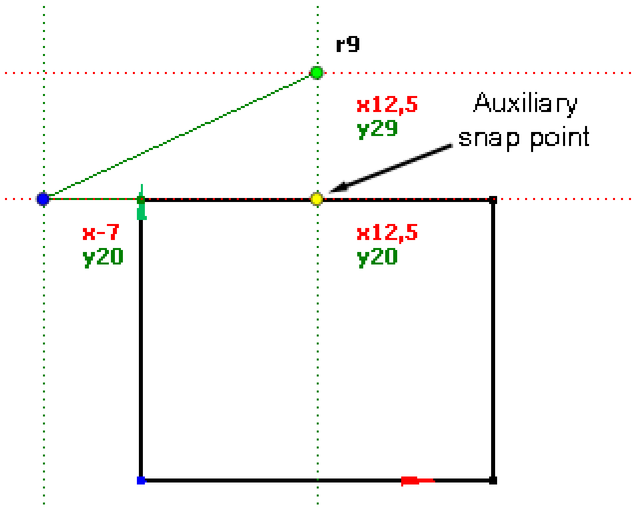


- Main snap point. When ever you click with the left or middle mouse button on a highlighted point or on a track point (see below) you create the main snap point with associated set of track lines. The main snap point is persistent. It means the point doesn't disappear when you move the cursor away from it. The main snap point is labeled with the point coordinates.
- Main track lines. These are the lines originated from the main snap point. Some of the lines are aligned with the current Local coordinate system, the others are aligned with the neighboring geometry entities (e.g. if the main snap point is the terminate point of a cut, the main track lines include the lines parallel to and perpendicular to the direction of the cut).

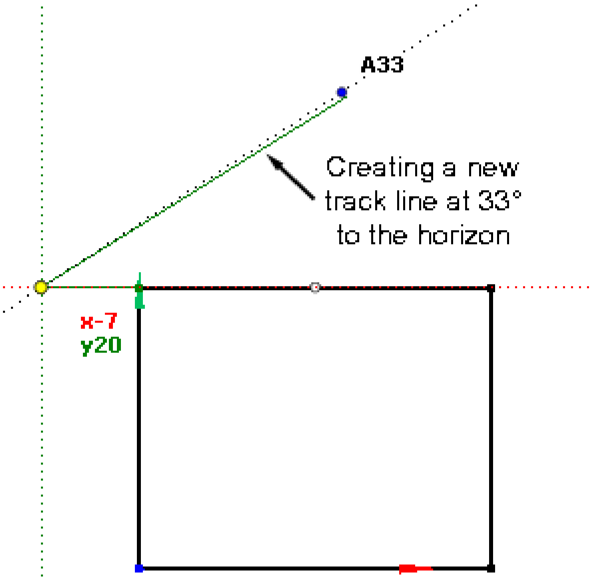
- Current track point. The track point is the point on a track line. When you move the cursor over a track line the track point is moved too. The track point is labeled with the distance from the current snap point. The track point may also be an intersection point of a geometry entity or another track line with the current track line. The point is colored red in that case.



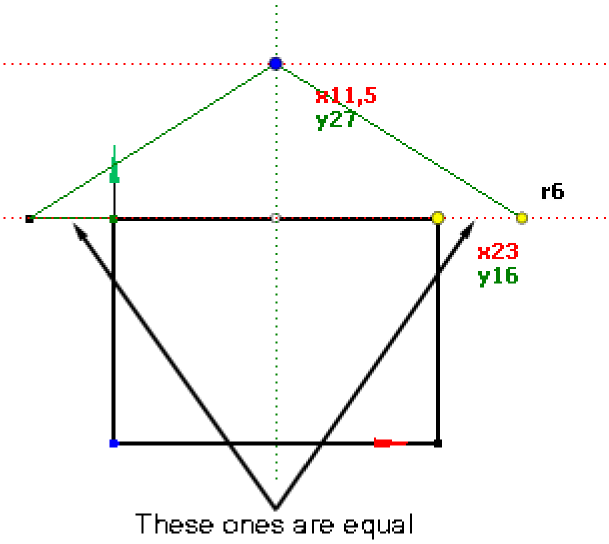
- Auxiliary snap point. When you hold the cursor over a highlighted point or a track point for some time the point is turned into the auxiliary snap point. The auxiliary snap point is the same as the main snap point except it is not persistent. It means the point disappears when you move the cursor away from it and its track lines. The auxiliary snap point has its own set of track lines. So you can construct intersection points between main an auxiliary track lines on the fly. Actually this is very useful feature.



- Dragging of the track lines with the middle mouse button. You can press the middle mouse button over a track line and holding the button drag the track line. This action will create the new track line placed at some angle relative to the source track line. When you drag the line you can see the actual angle value on the screen.



- Keeping in mind last track distances. When you make a drawing you may want to make some cuts of equal length or something like that. To resolve this task the smart snap remembers last three track distances you have accepted when created the drawing. So when you move the cursor over the track line sometimes the track point is colored yellow. It means the distance from the track point to the snap point is the same as one of the last three distances.



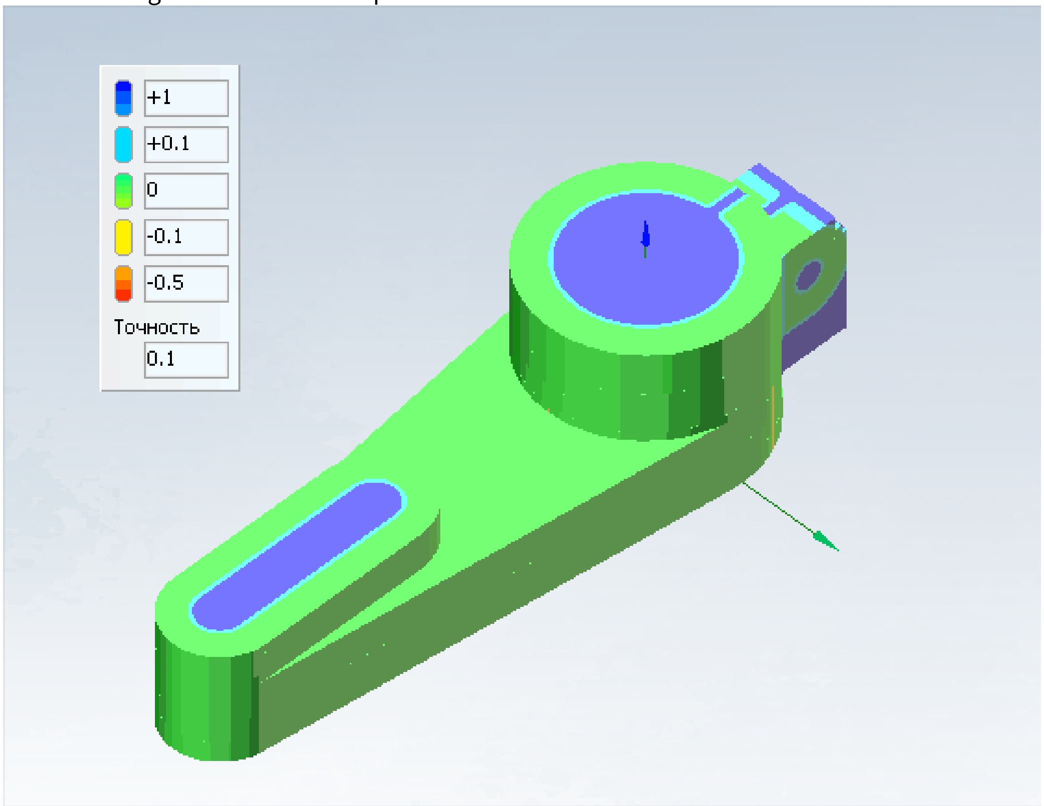
See also:

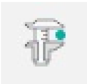
Main window

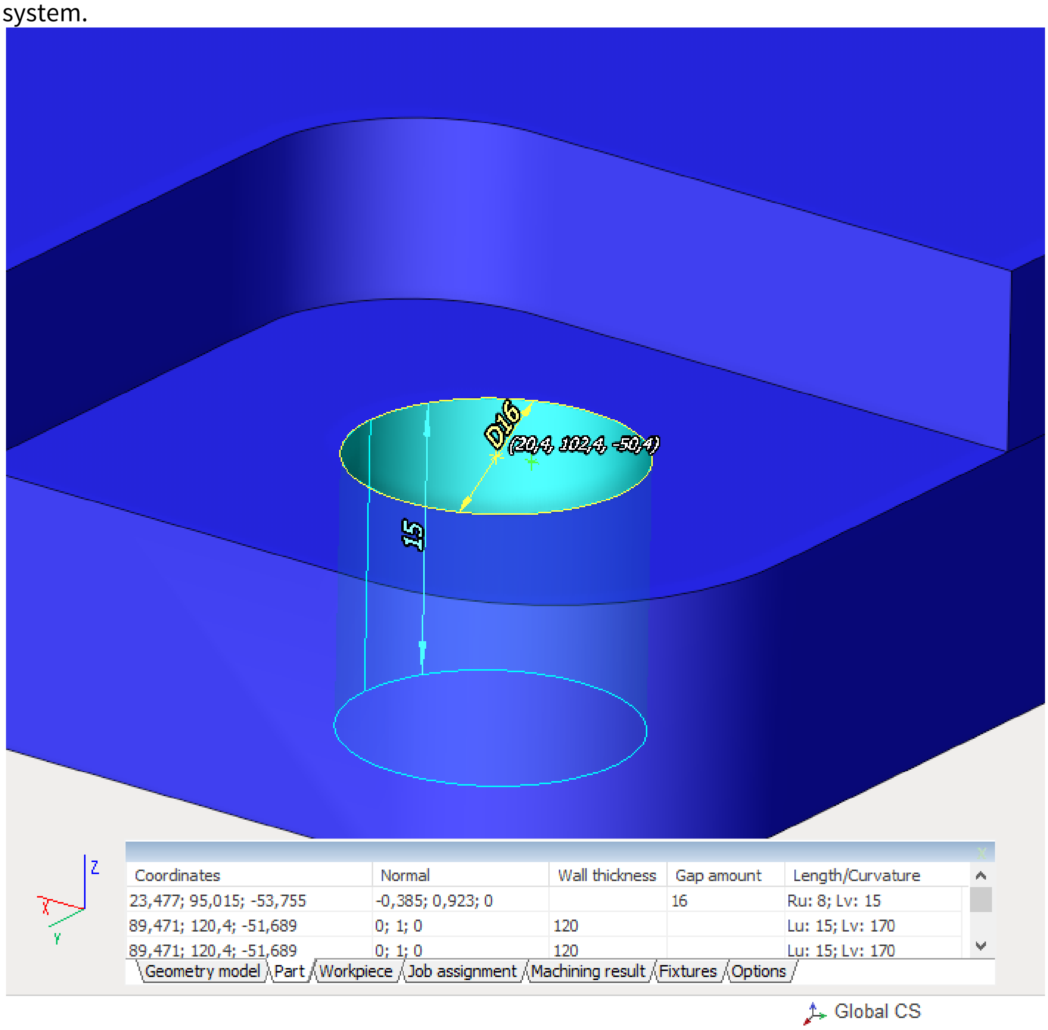
3.1.1.7 Measurement and analysis

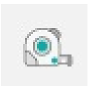
SprutCAM X system incorporates methods to conveniently measure geometrical parameters of the model and to analyze the simulated machining result.

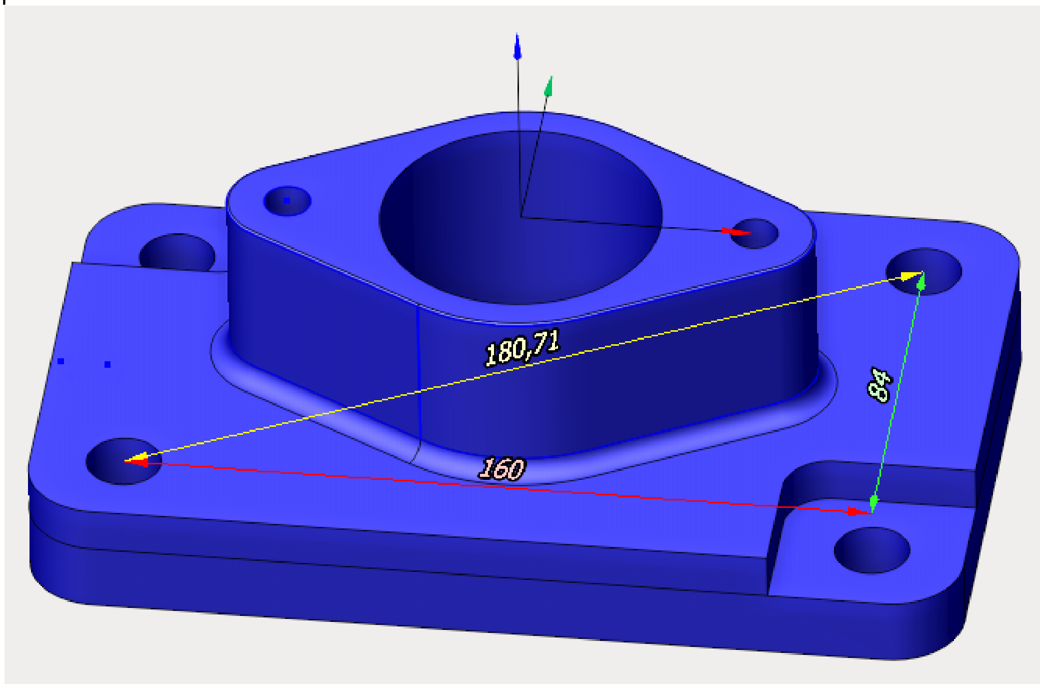
-  Machining result with part compare can be activated in Machining or Simulation mode. It allows you to display the difference between the original part and the result of machining in the form of a color scheme where each color is associated with a specific deviation range between the compared elements.



-  Geometry measuring tool allows you to find out the main geometric dimensions of the object selected in the graphics window. When the mode is activated, an additional window appears in which you can select the type of object being measured (part, workpiece, machining result, etc.) and set additional measurement parameters. Then, when you select an object on the screen, the main dimensions are displayed in the graphics window, and a row with the main parameters associated with the point selected on the screen is added to the measurement window. Some dimensions can depend on the active geometrical coordinate system.



-  Measure tool button turns on the mode of 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 the dimensions depends on the active geometric coordinate system. Select the following points in pairs to measure others.



3.1.1.8 Autosaves and Project snapshots

The program has a system of automatic backup versions of projects, which allows you to return to the previous version of each project if any unforeseen problems suddenly arise.

The key concept in this system is a **snapshot** - this is the complete state of the project at one specific point in time. Every time a project is saved, the system generates a snapshot. The most recent snapshot is always saved inside the project *.stcp file. Usually, the system stores several backup versions at once, so the remaining project snapshots are placed in a special backup file located in the history subfolder next to the project: "*<projectfolder>*__history*<projectname>*.stcp.~back". Here

- *<projectfolder>* is the folder of your current project;
- *<projectname>* is the name of your current project.

Despite the fact that a snapshot stores the full state of the project, this does not mean that the size of two snapshots will be equal to the size of two projects. When saving each next snapshot, the system saves only the data that has changed since the previous snapshot was saved. This saves disk space.

New snapshots are generated in two main cases:

- when the user explicitly saves the project (manual snapshots);
- when the system automatically saves the project itself (autosaved snapshots).

These two cases are handled separately by the system, and even if the project is saved automatically, it never replaces the state that the user saved manually. These states are stored in parallel in the list of snapshots.

You can set autosave and snapshot options in system settings window.

The autosave process is optional. The program can call automatic saves in two ways.

- By timer at specified intervals (Interval in minutes setting, if **Autosaves by timer** option is enabled).

- By individual events occurring in the system, for example, before deleting a technological operation, before calculating the toolpath, immediately after calculating the toolpath, etc. (**Autosaves by events** option in the system settings).

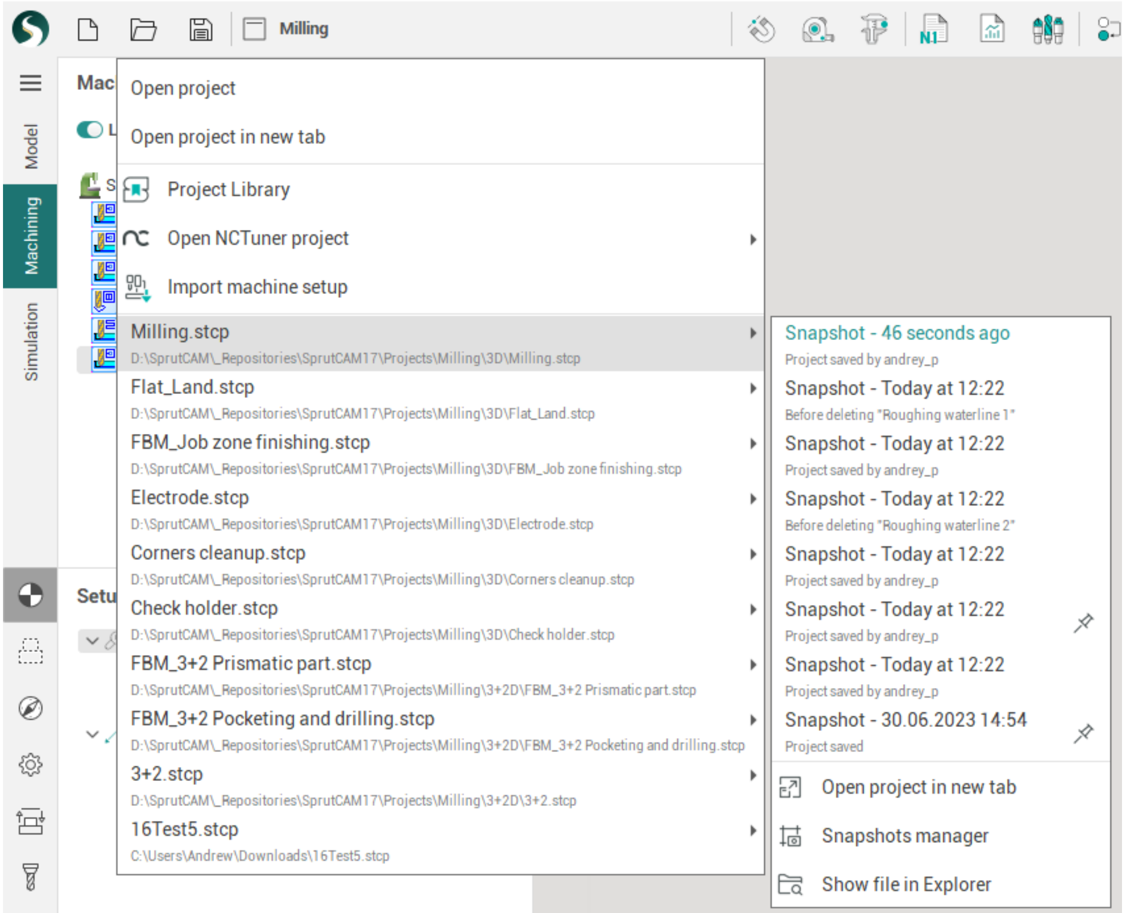
Thus, during long-term work on the project, a large number of snapshots can accumulate. In order to prevent disk overflow, the program automatically deletes snapshots if their number exceeds a certain number specified in the system settings. Because the system considers snapshots saved manually by the user to be more reliable, two different storage depths are set in the system settings.

- **Manually saved snapshots count** - the maximal number of snapshots saved by the user manually (by default it has value 5).
- **Autosaved snapshots count** - the maximal number of snapshots saved by the system automatically by timer or by events (by default it has value 15).

It is also possible to mark any snapshot as **persistent**. This will mean that the system will not delete this snapshot even if the maximum allowable number of snapshots is exceeded. Such snapshots are explicitly deleted only by the user himself. This allows you to do some kind of alternative variants of the same project.

On large projects, saving can take quite a long time. Therefore, in order for too frequent automatic saves not to interfere with the work on the project, the system performs automatic saves no earlier than after a specified period of time - **Interval between long autosaves** option (5 minutes by default). At the same time, the system considers projects large if the last save took longer than specified in the **Long autosave time** parameter (20 seconds by default). For example: if the previous snapshot was saved for more than 20 seconds then the next autosave of the snapshot will occur no earlier than after 5 minutes.

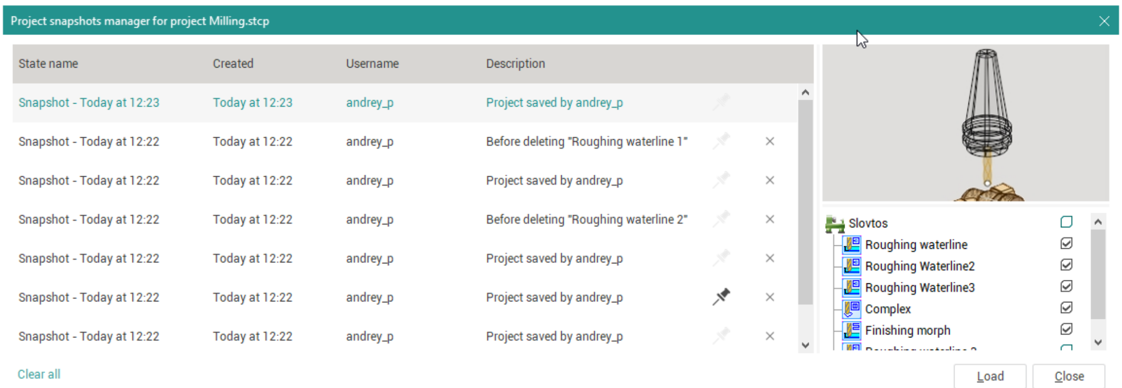
The user interface of the system allows you to quickly load any of the saved project states. The list of snapshots for each project is shown as a drop-down menu next to the project name in the list of recent projects on the main toolbar of the application. Snapshots are shown in reverse chronological order - most recent at the top. The current snapshot is highlighted in green. The Pin button on the right side allows you to mark the snapshot as persistent to prevent automatic deletion.



In order to keep the menu short, not all snapshots can be shown in it.

- For the current project it shows only no more than 10 of the most recent ones plus all persistent ones.
- For non-current project only the persistent snapshots will be shown.

If you need more snapshots, then the full list can be seen in the **Project snapshots manager** window.

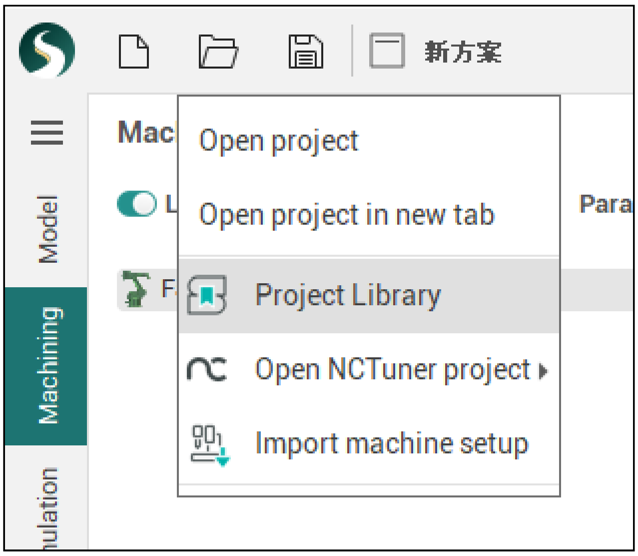


In the Project snapshots manager window you can see the full list of snapshots, edit their name or description, see a thumbnail, user (who saved snapshot) and a list of operations for this snapshot, delete or make it persistent. You can also load a snapshot from this window.

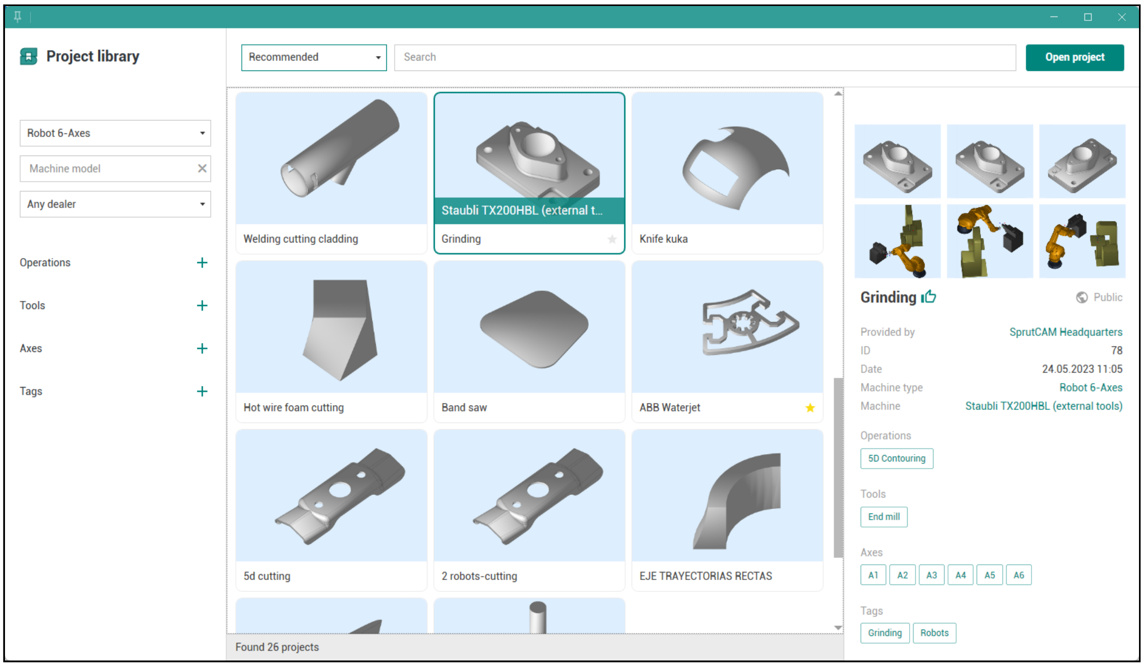
If you hover over the creation date, the full date will be displayed in the hint.

3.1.1.9 Project library

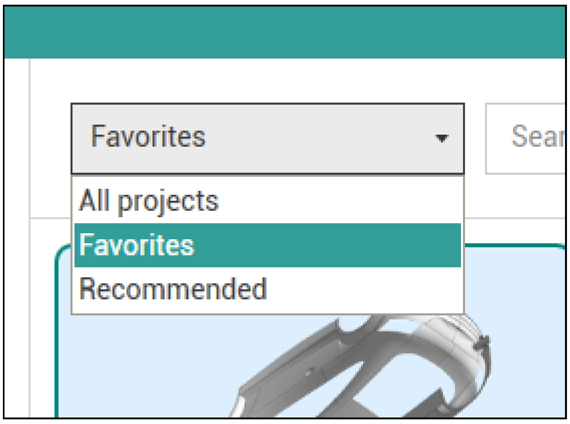
The Projects Library allows to find and open example projects from our online library. Project library can be started directly form the SprutCAM open project menu.



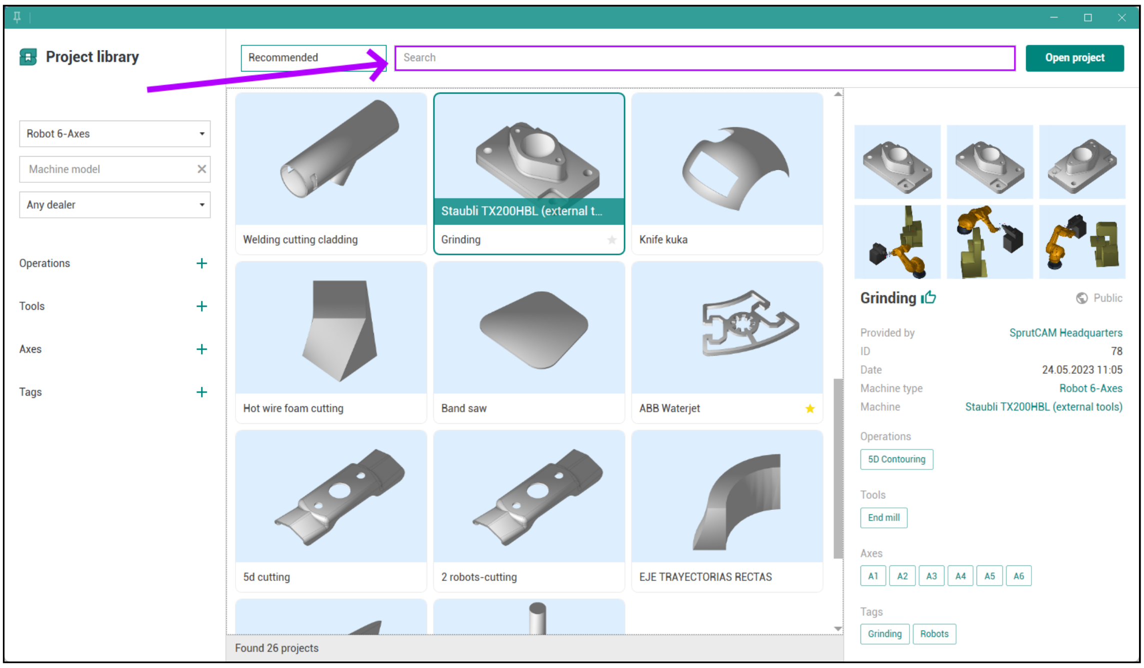
In the open main window of the project library, all available projects that can be loaded into SprutCAM appear.



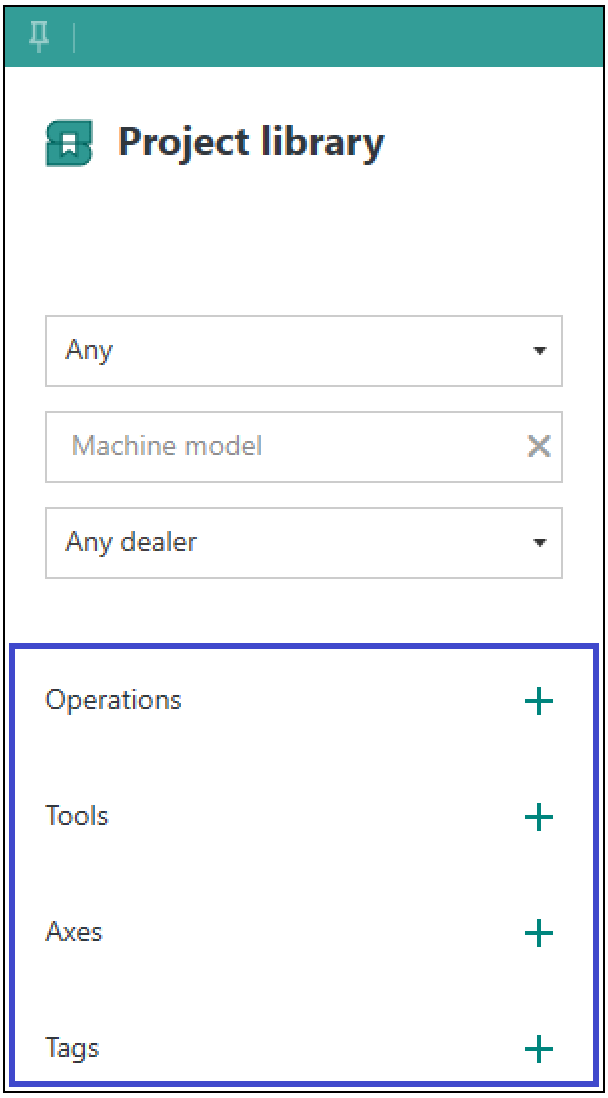
Use projects source selector to show All projects, Favorite projects or Recommended projects only.



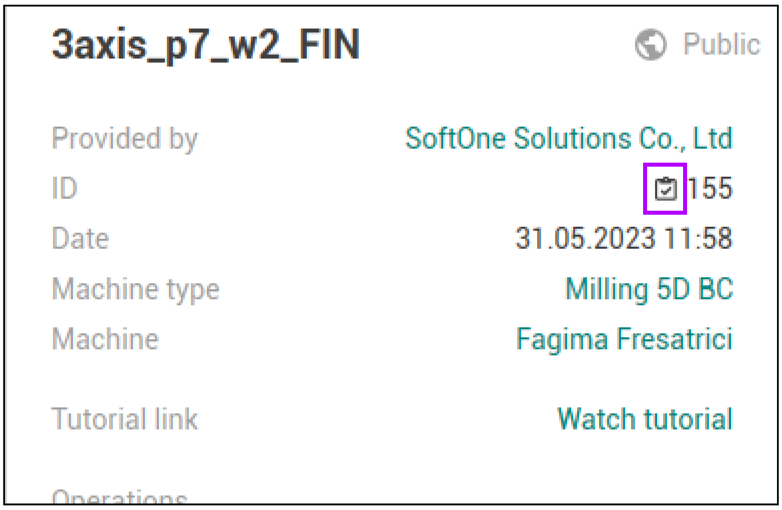
It is possible to search projects by id, name, machine name, machine type, operations, tool types and etc. It is also possible to use filters on the left side of window.

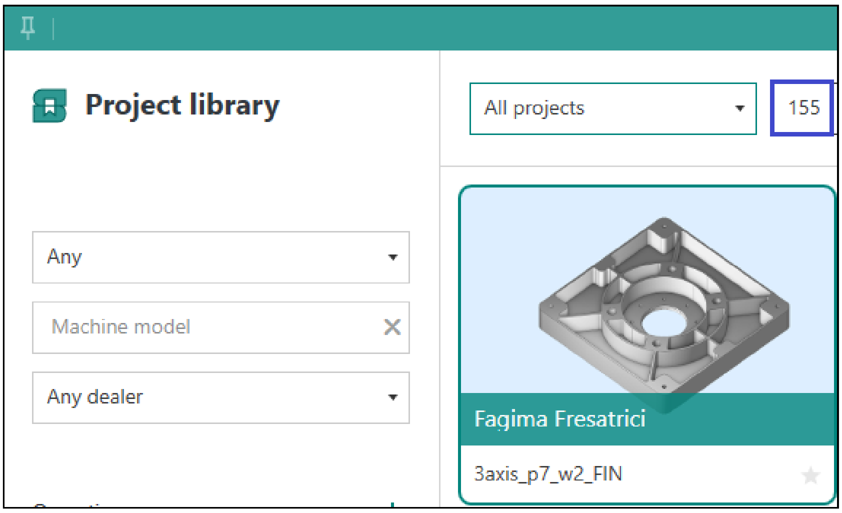


You can also search using the search menu for operations, tools, axes and tags.

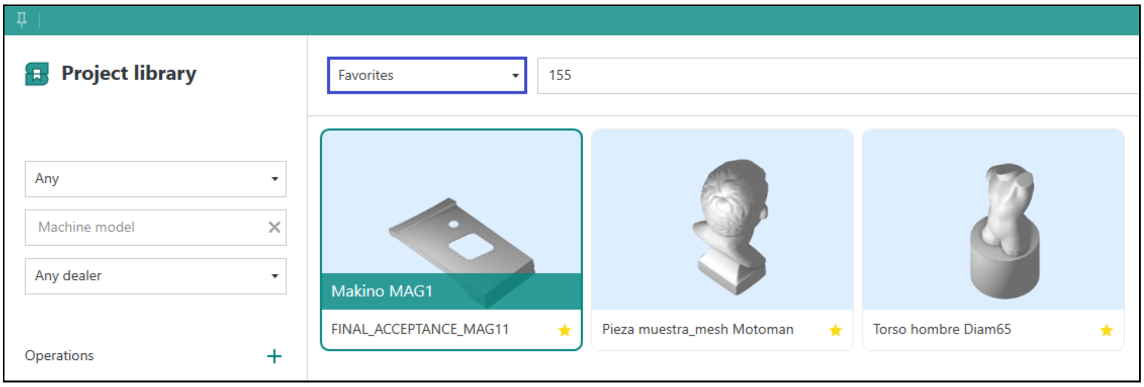


Use button to copy project id to clipboard. This id may be usefull for searching this project later.

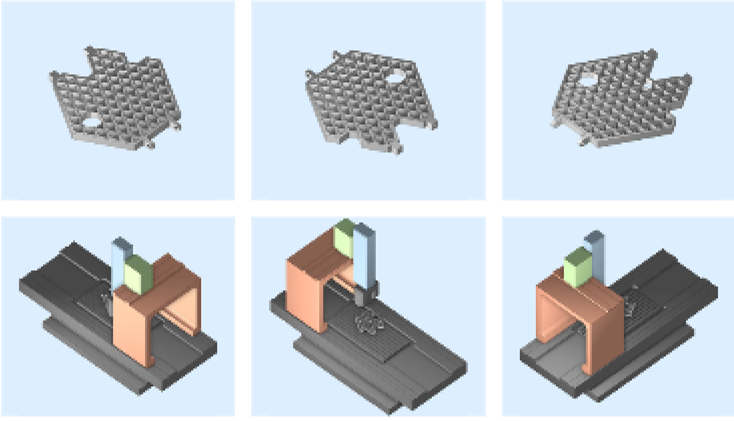




Use ☆ icon to save any project to your favorite projects list.



All green values on the project details panel are clickable. Click to the value to include it to the filters list. Click the value on the filters panel to exclude this value from the filters.



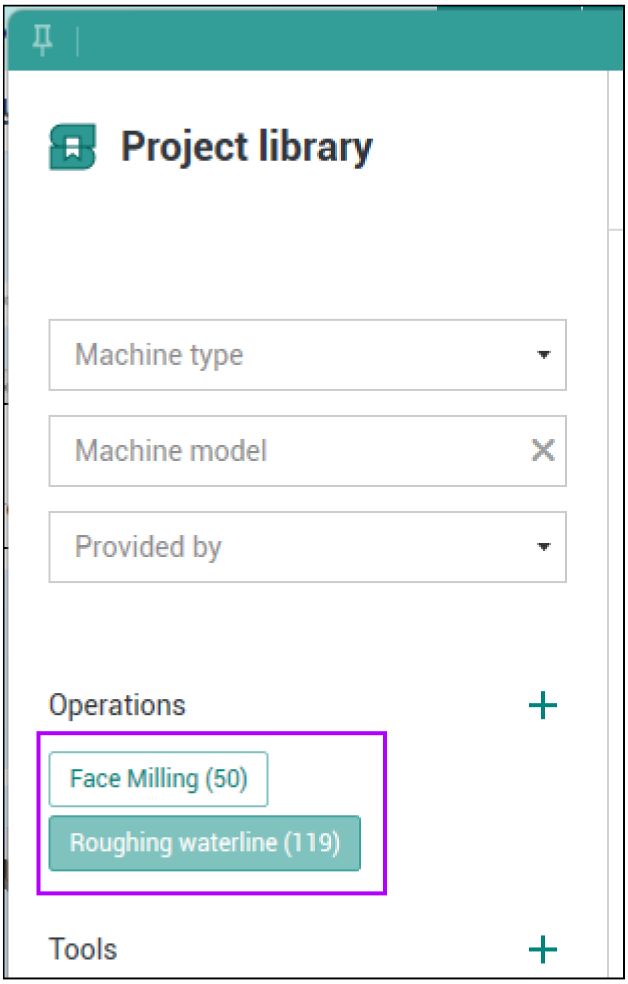
3axis_large_part_2_V2

Public

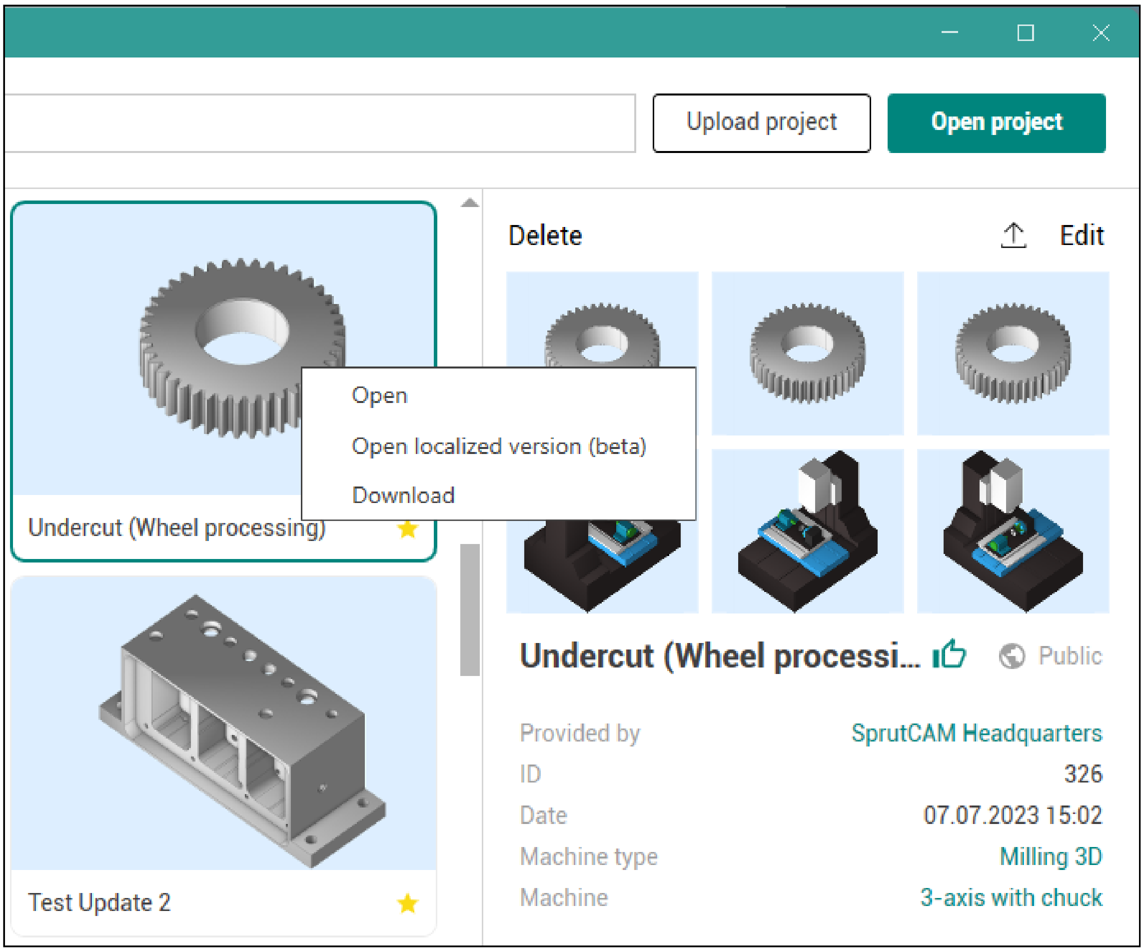
Provided by	SoftOne Solutions Co., Ltd
ID	165
Date	31.05.2023 12:01
Machine type	Milling 5D BC
Machine	SFY_5AX_2000
Tutorial link	Watch tutorial

Operations

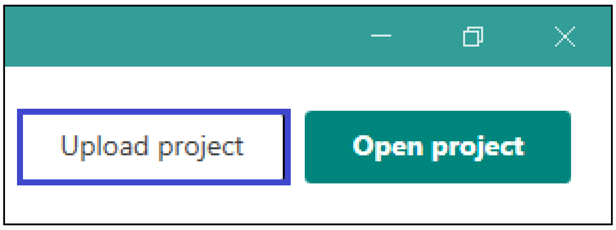
2D contouring | Face Milling | Hole machining | Pocketing | Roughing waterline



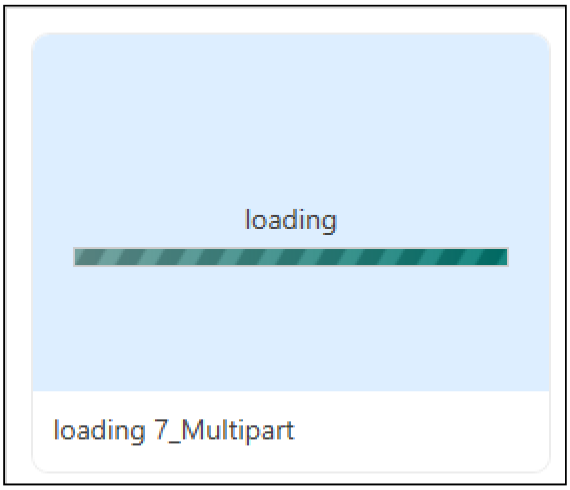
Use right mouse button to open selected project actions list. Here you can open project, open localized version of project and download project file. It is also possible to use Open project button of double click to open project in SprutCAM.

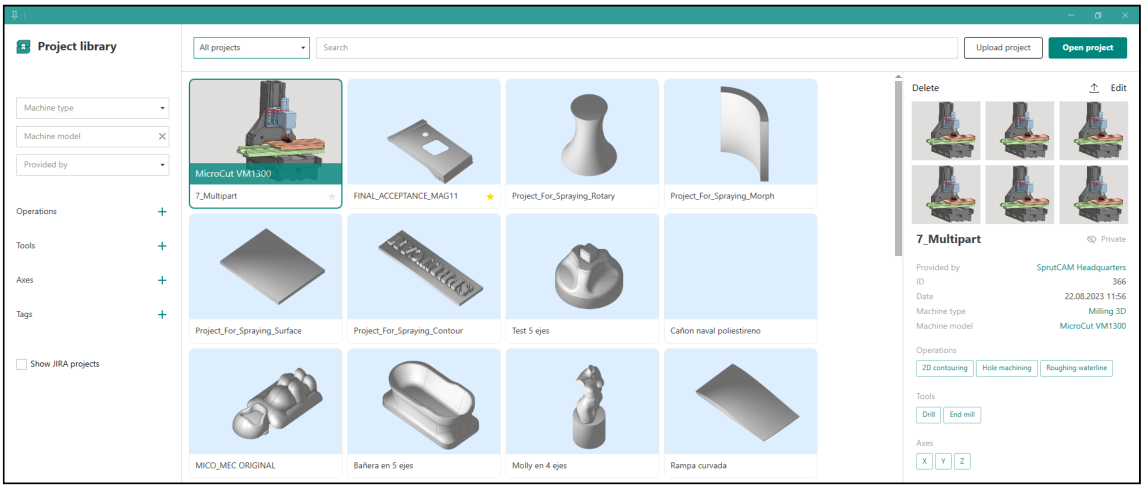


To upload your project click on the "Upload project" button.



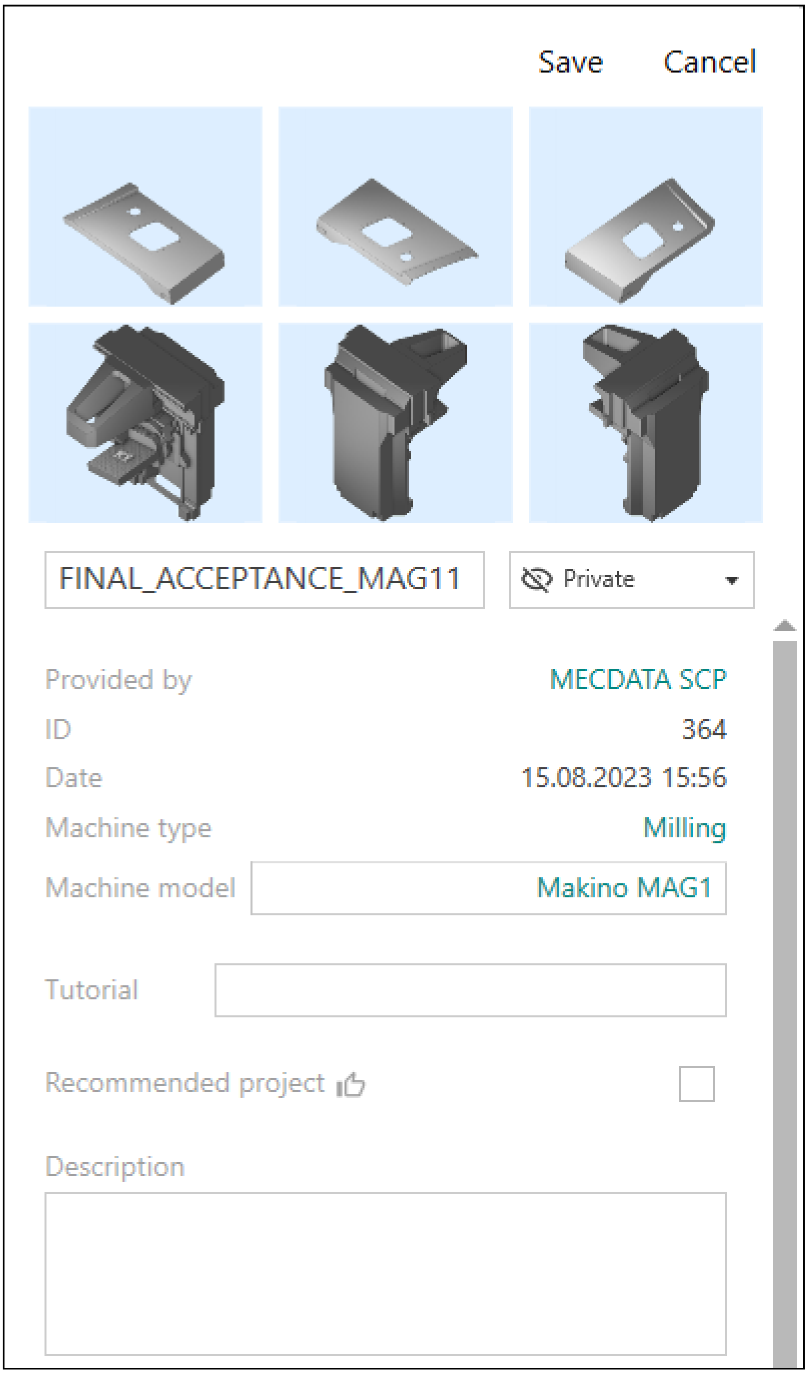
Once you have selected a project you will see that it is loaded into the project library.



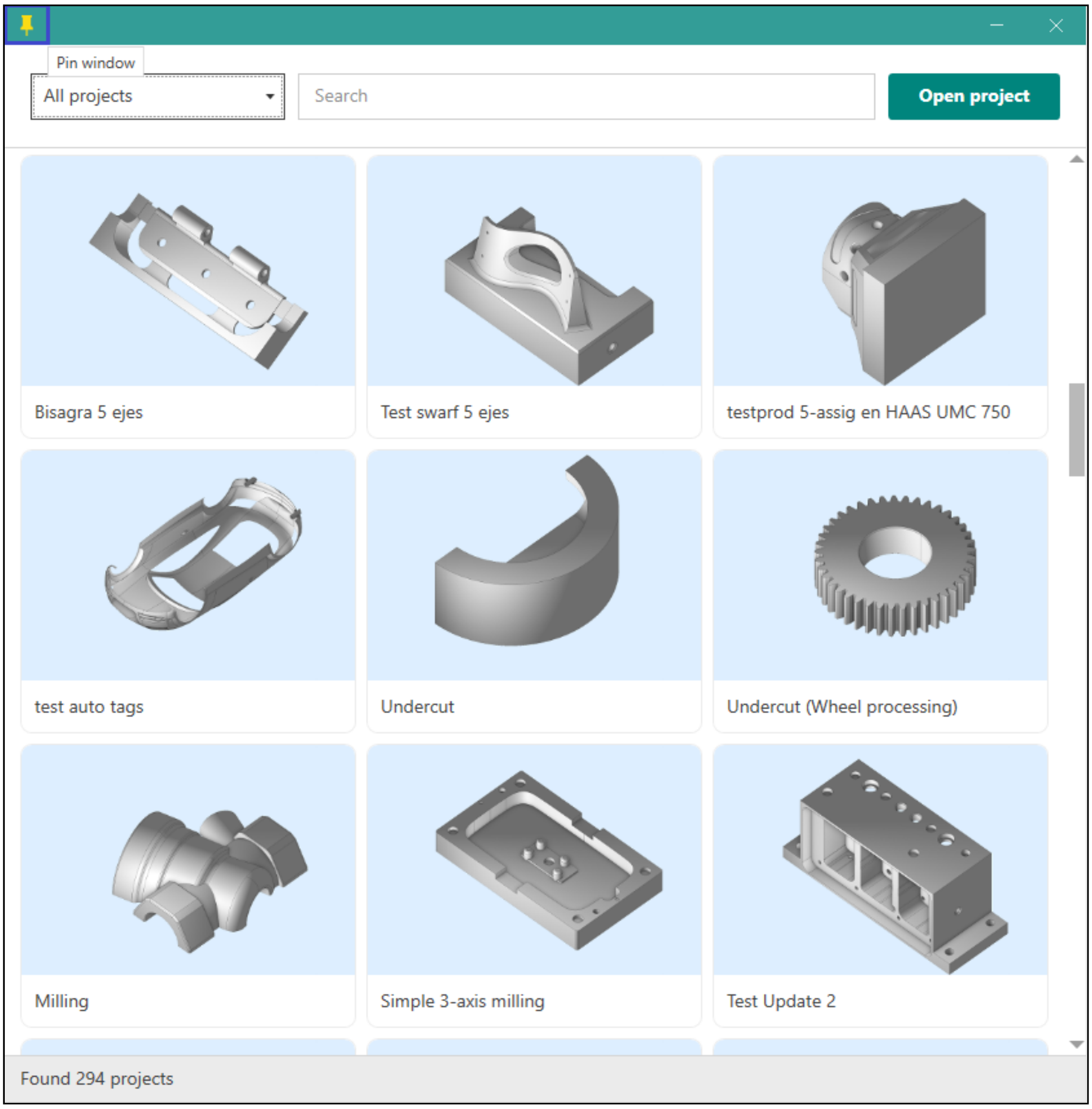


You can delete, edit your projects.

When editing your project, you can add a description to the project, change its name.



Pin window button is located at the left top corner of the window. This button switches Project library to the compact mode and force window stay on top.



3.1.1.10 Machine setup (project template)

The machine setup file is designed to quickly create a new project on the template.

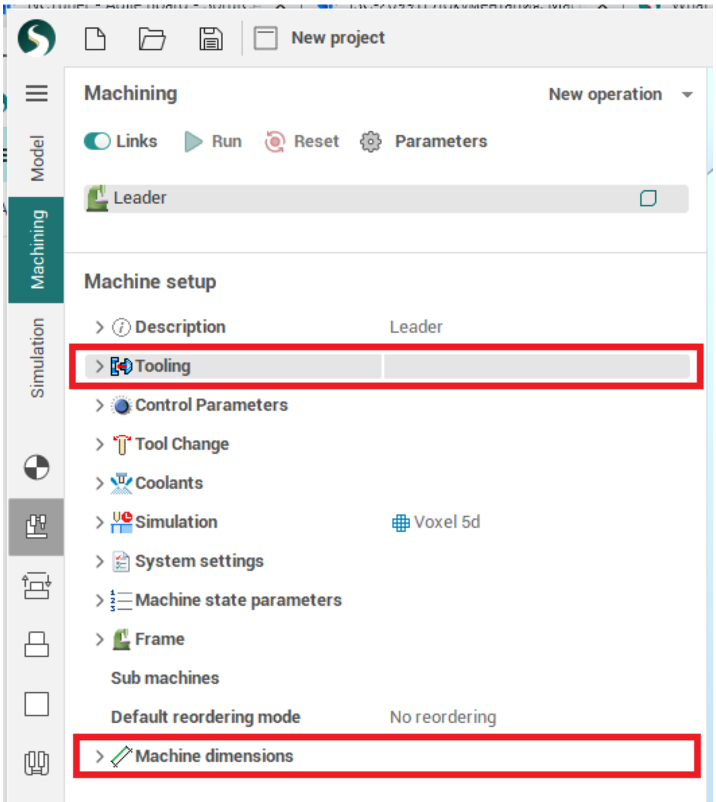
You can save the project in the format of the machine setup file (*.stms). Use the menu item "Save as machine setup" in the drop-down menu of the project saving button on the application toolbar. After that, you can quickly create new projects using this file as a template.

The following data is saved in the file:

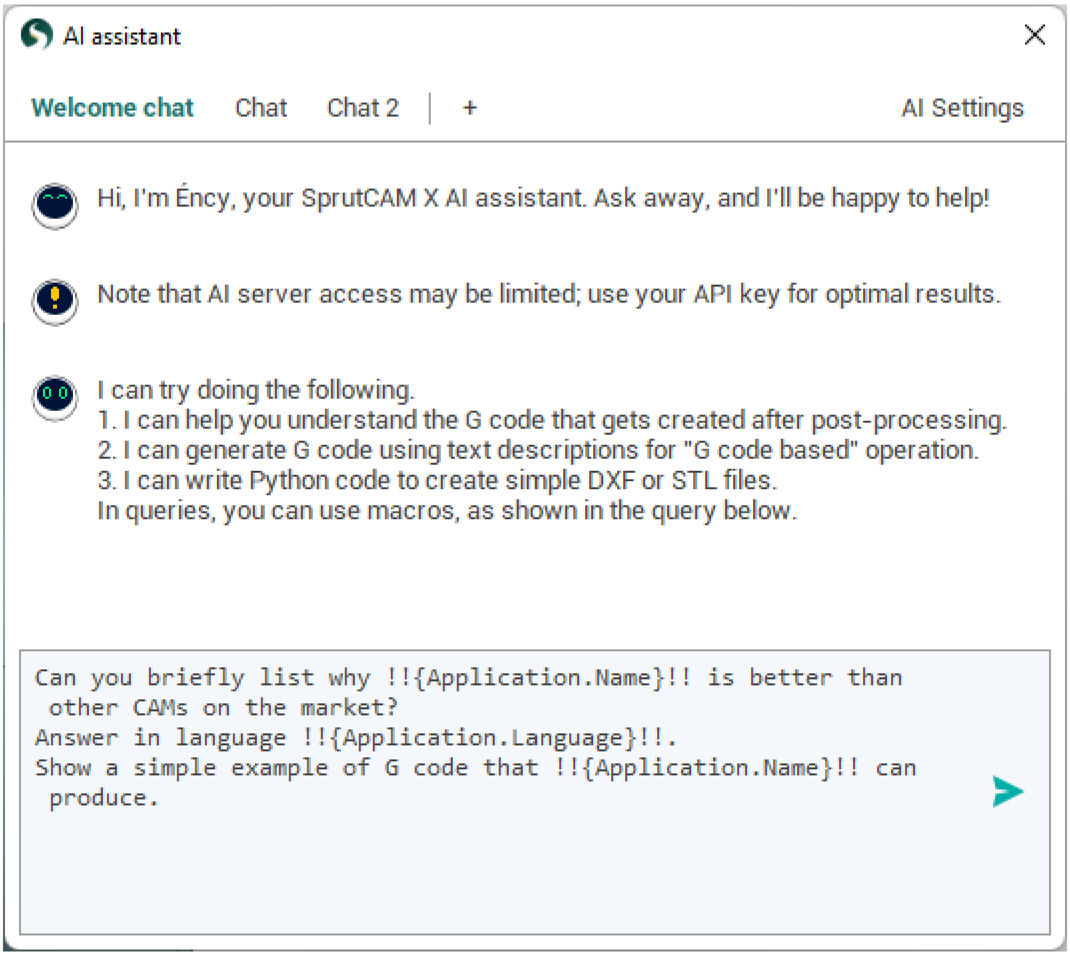
- machine
- stages, part (as a group of operations)
- fixtures (including position)
- tools
- approaches/returns
- workpiece coordinate systems list
- types of tool blocks, and placement in the turret

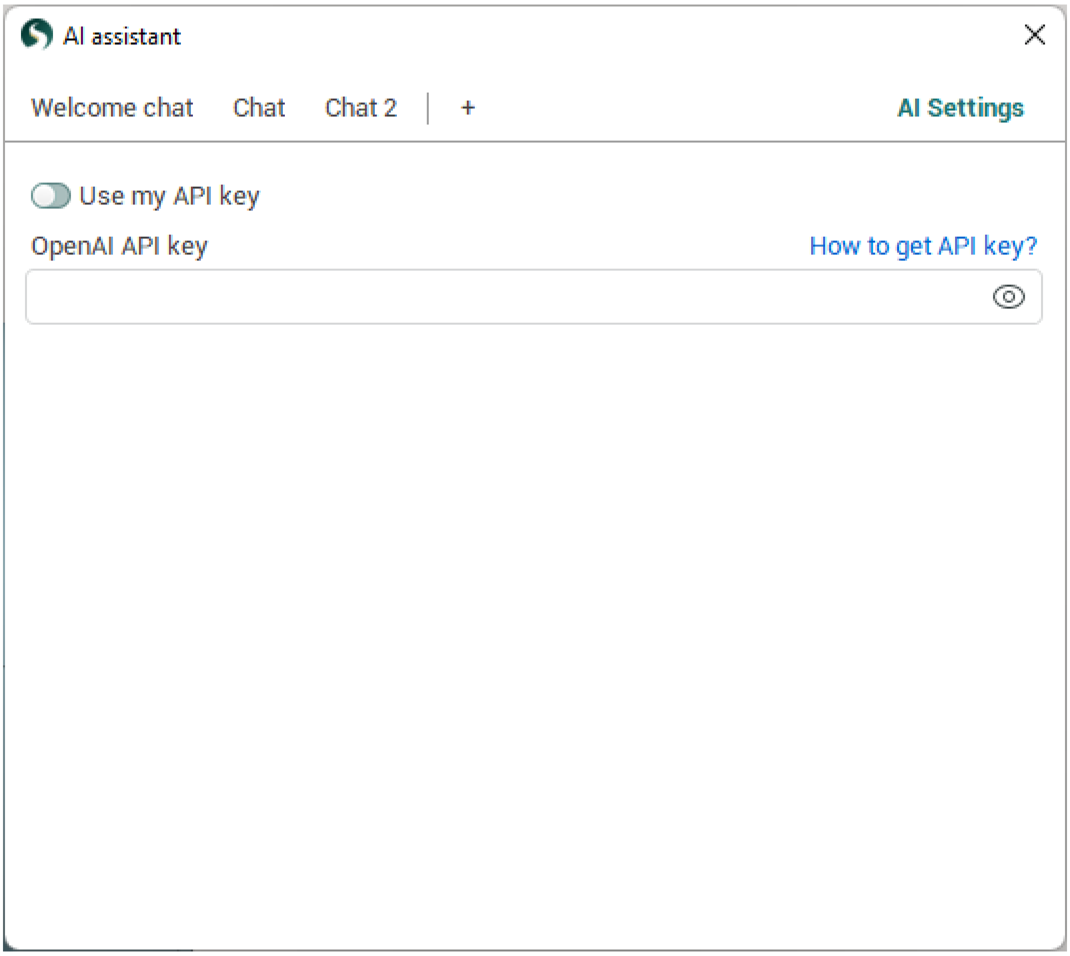
Under the new project button on the application toolbar, a new drop-down menu will appear where you can choose one of the recently used machine setup files. Or select another machine setup file, for this, use the "Create from machine setup file" menu item.

You can also import the machine setup file to the current project. To do this, use the "Import machine setup" from the drop-down menu of the opening button on the application toolbar. During import, all data from the machine setup file are added to the data of the current project, with the exception of the machine. If the project machine coincides with the machine from the machine setup file, then the machine parameters are updated from the machine setup file. If the machines are different, then, if possible, the parameters "Machine dimensions" and "Toooling" are updated.



3.1.1.11 AI assistant


AI assistant
Welcome chat
Chat
Chat 2
+
AI Settings
Hi, I'm Ency, your SprutCAM X AI assistant. Ask away, and I'll be happy to help!
Note that AI server access may be limited; use your API key for optimal results.
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.
Can you briefly list why !!{Application.Name}!! is better than other CAMs on the market?
Answer in language !!{Application.Language}!!.
Show a simple example of G code that !!{Application.Name}!! can produce.



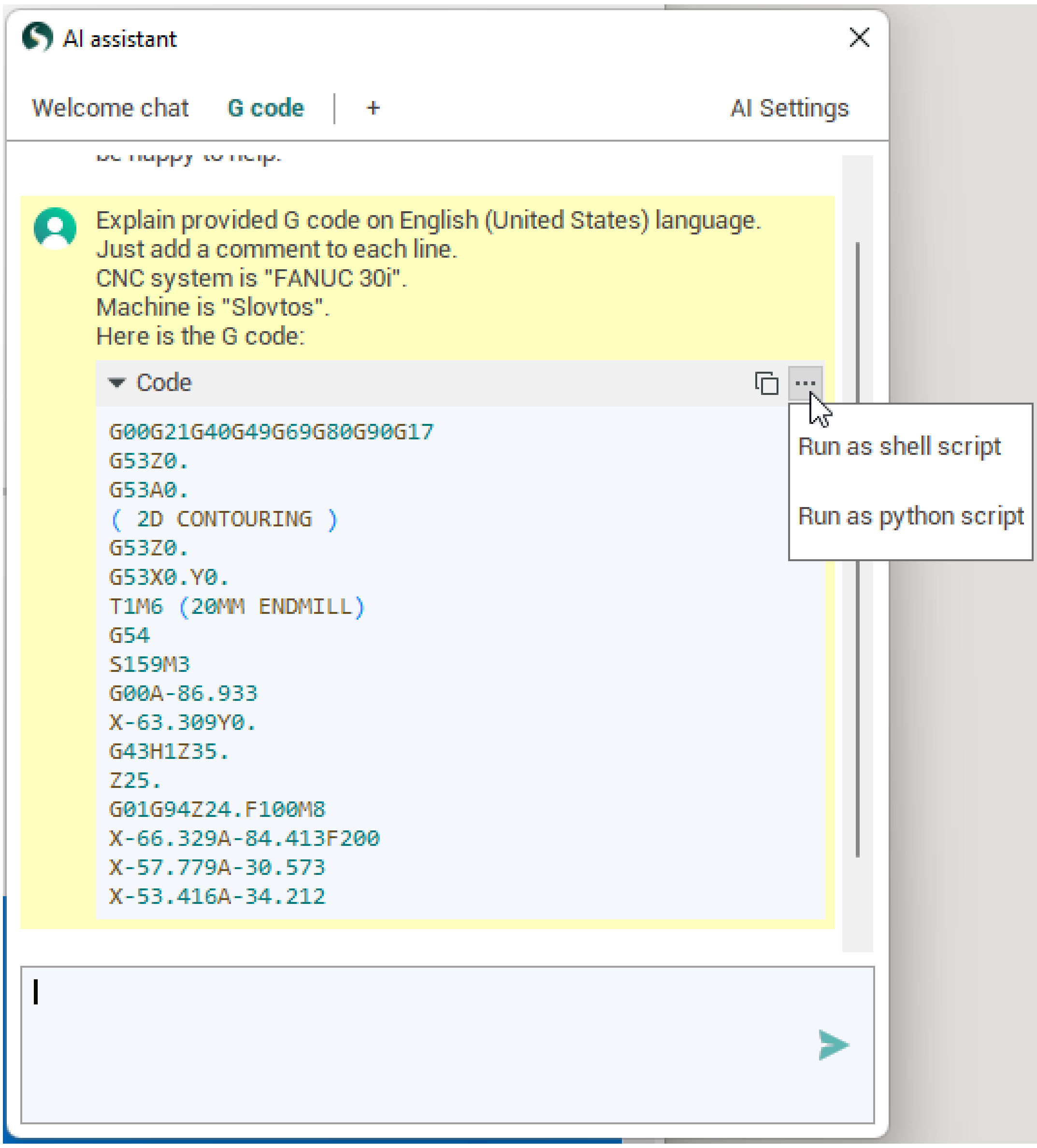
The AI virtual assistant that can help users with various aspects of CAM workflow. The AI assistant is based on ChatGPT technologies.

The AI wizard is called Éncy (pronounced like [ˈɛnsɪ]) and it greets the user with the following message: “Hi, I’m Éncy, your SprutCAM X AI assistant. Ask away, and I’ll be happy to help!”

The AI assistant is powered by the OpenAI API, which uses deep learning models to understand and generate natural language. Engineers can communicate with the AI assistant using text commands in multiple languages, and the AI assistant responds accordingly.

To create the requests, CNC-engineers can use various macros that are automatically adapted to the context of the task being performed. Use “Ctrl + Space” key combination inside the prompt edit box to see possible macros values.

You can also perform some actions on blocks of code that the assistant displays. The list of actions is opened by clicking the button with three dots in the upper right corner of the code block.



The list of available actions and macros always depends on the context from which each specific chat tab was opened. For example, if the chat was opened from the G-code generation window, then the *!!{GCodeWindow.SelectedText}!!* element will be available in the macro list. This means that when sending a request, the selected text from the window will be substituted for this macro.

Ёncy can perform the following tasks:

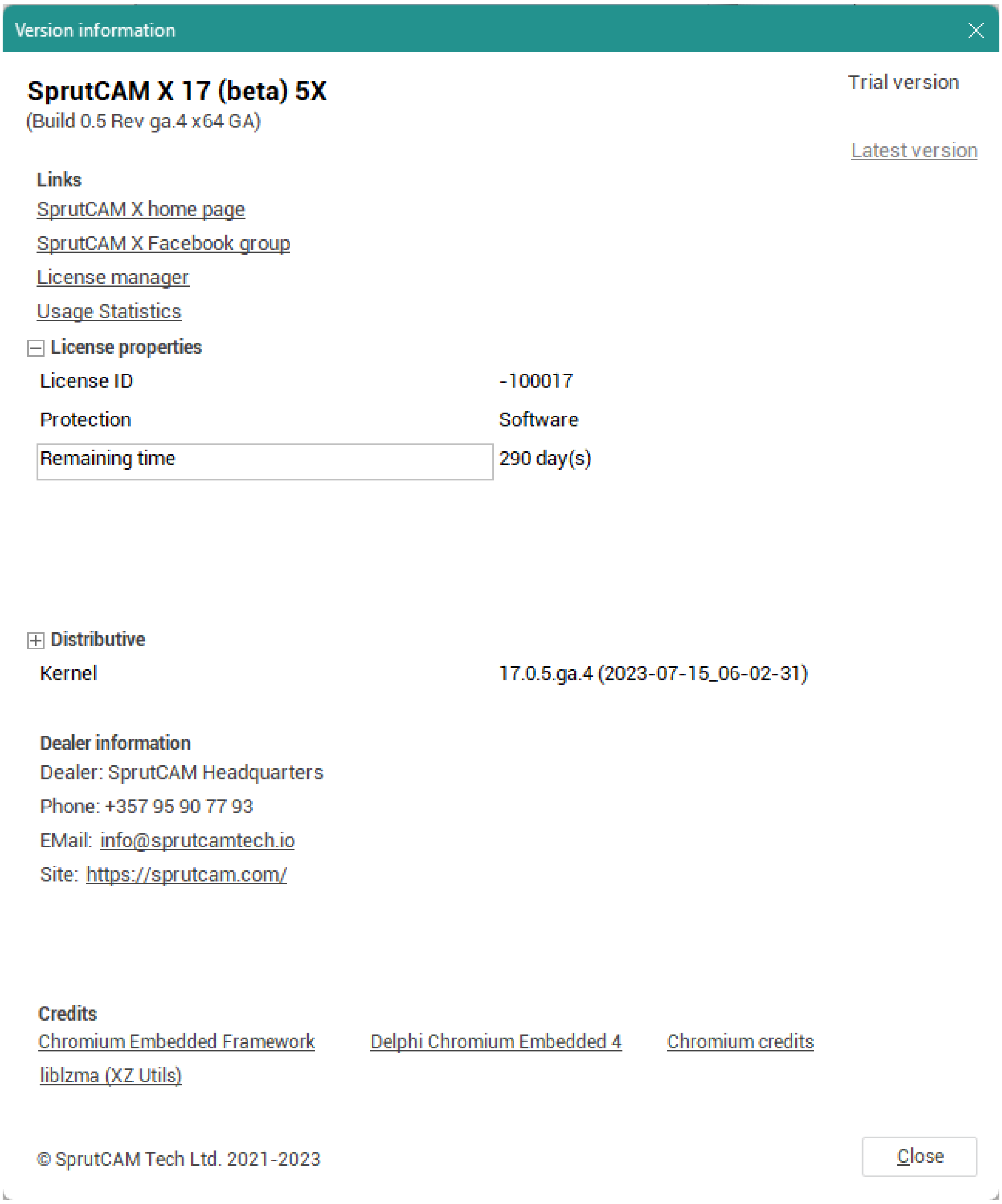
- Explain the G-code generated as a result of post-processing. You can ask the AI assistant to clarify any line of code or command, and it will give you a detailed explanation of what it does and why it is necessary. Click “AI assistant” button inside G-code generation window with selected piece of text.
- Generate a G-code using a text description of the operations. You can simply type what you want to do, such as “drill a 10 mm diameter hole at point (100, 25)” and the AI wizard will generate the corresponding G-code for you. In the window of the Job assignment of the G-code based operation, you need to click on the AI assistant button. In this case, elements will appear in the list of code block actions that allow you to quickly apply the code block generated by the assistant to the operation, calculate the operation and see the result immediately.
- Write code in Python to create .dxf or .stl files. You can use the AI assistant to create 2D or 3D models of your parts using Python code. The AI assistant guides you through the process and shows you the results in real time. Just ask it to generate a python code which will make a dxf

file the form you need, then click “Run as python script” inside actions list of generated code bock. If the script is correct, then the resulting file will be inside My documents folder, “$(ApplicationName)\AIActions” subfolder. You can import it standard way. If the script is incorrect, just copy the error message and ask the assistant to resolve the problem.

- Provide reference information for the industrial robot or CNC-machine when creating kinematic schemes in MachineMaker, a zero code application for building digital twins from SprutCAM Tech. 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, etc.
- Answer any user's question, even not related to the operation of the software. You can chat with the AI assistant about anything you want, such as CNC tips and tricks, industry news, best practices, etc. The AI assistant will try to answer your questions as best it can, or direct you to relevant resources if it doesn’t know the answer.

> *Access to OpenAI services may be limited, so we recommend that you enable the "Use my API key" option on the AI Settings tab. In this case, access to the service will be determined only by the capabilities of your OpenAI account.*

3.1.1.12 Version info window



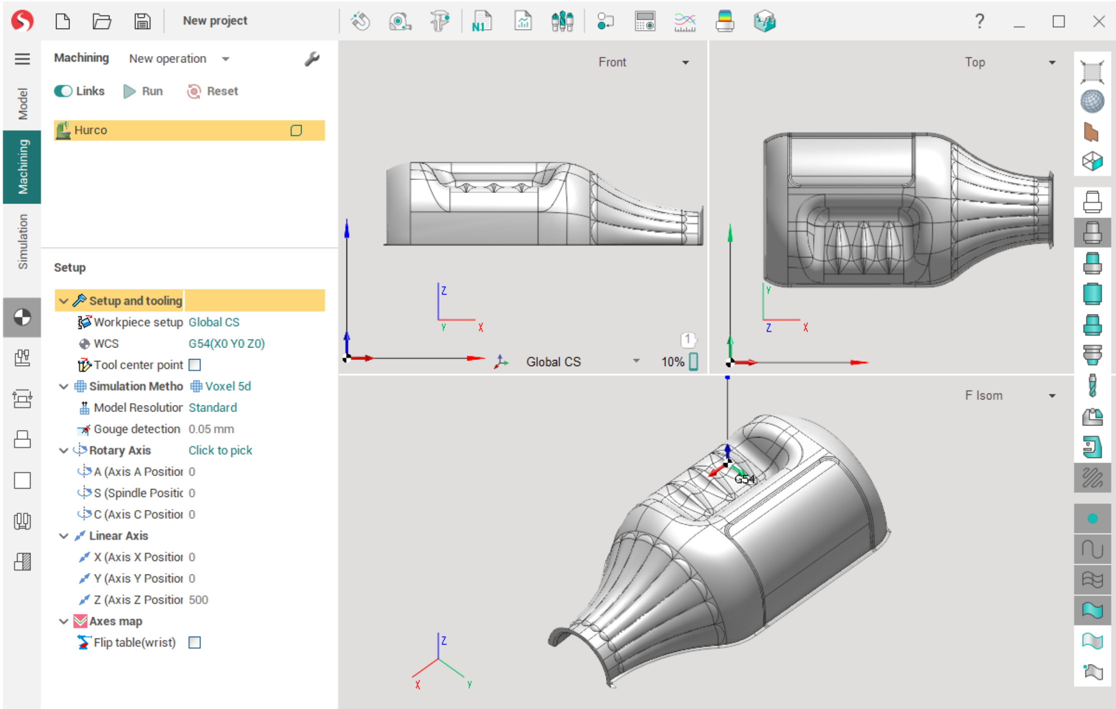
The version information window opens from the first button on the top application toolbar.

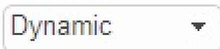
Here you can see the following.

- The application name.
- Application build identifier.
- Useful links on our web resources.
- Basic license properties.
- Version list for each of the application modules.
- Information about the dealer of software.
- Copyrights.

3.1.2 Graphic window and visualization control

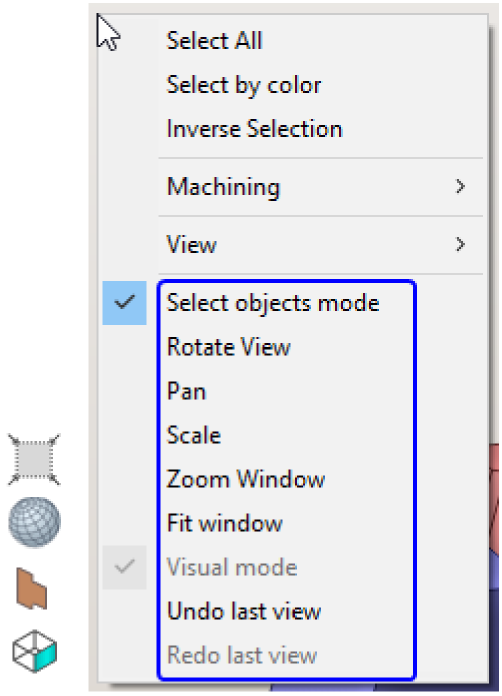
The main part of the screen takes graphics window. It can display such objects as geometric models, machine, workpiece, the toolpath of operations, etc. It is possible to group several views in one graphic window.



To add new view needed to click on 

panel on right up angle of the window and select <New view> from menu (or use **View** submenu inside popup menu of the graphics window). Active view can be closed from this menu too. Some visualization control hotkeys can be changed in <System settings> window on <Visualization> tab.

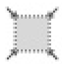
The visualization parameters can be changed on the View control toolbar on the right side of the main window or via the pop-up menu of the graphic screen:



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

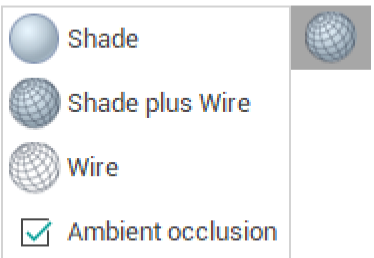
- Select objects mode. Left-click will select objects from the screen (this is the default state).
- Rotate view. Moving the mouse while pressing the left button will rotate the view.
- Pan. Moving the mouse while pressing the left button will shift the view.
- Scale. Moving the mouse while pressing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

However, all these actions can be easily performed without switching 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 to change the view using hot keys. See child topics: Window zoom, Interactive rotate, Interactive pan, Interactive zoom in-out, Zoom extents.

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

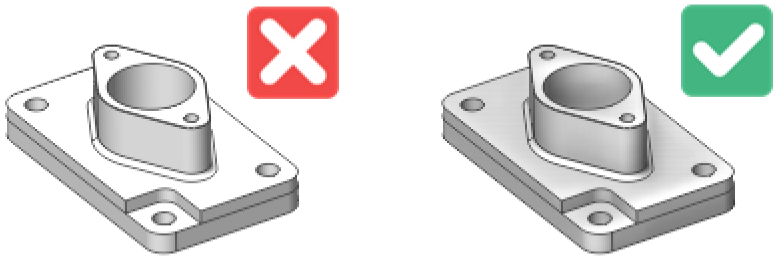
There are several standard views that are switched by these buttons  (the exact appearance of the button depends on the type of machine).

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



The button with the drop-down menu in the figure above allows you to switch the display mode of three-dimensional objects: **Shade** - shaded without edges, **Shade plus Wire** - shaded with edges and **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 example below.



The button above defines a visualization mode for revolution bodies only: 3D, 3D without a quarter, half of 3D and 2D axis section.

See also:

System's main window

Objects selection mode

Window zoom

Interactive rotate

Interactive pan

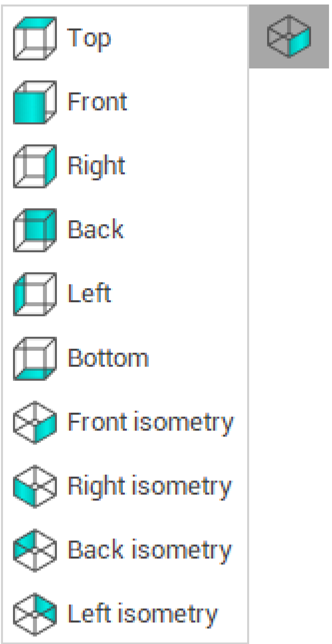
Interactive zoom in and out

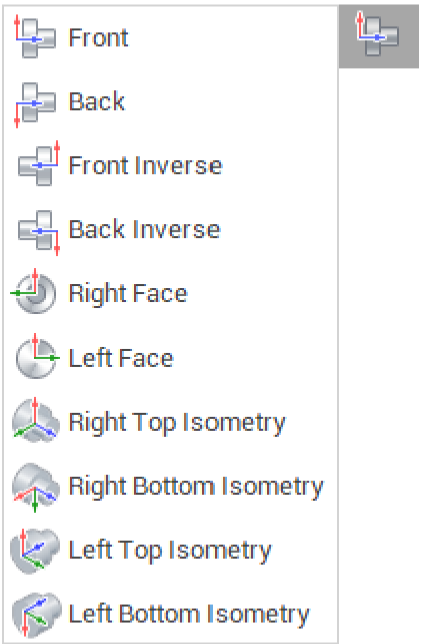
Zoom extents

Undo view

Redo view

3.1.2.1 Standard views





The standard views panel of the main window may have a different view depending on the type of machine: turning, 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 using another method, the sunken button on the panel releases automatically.

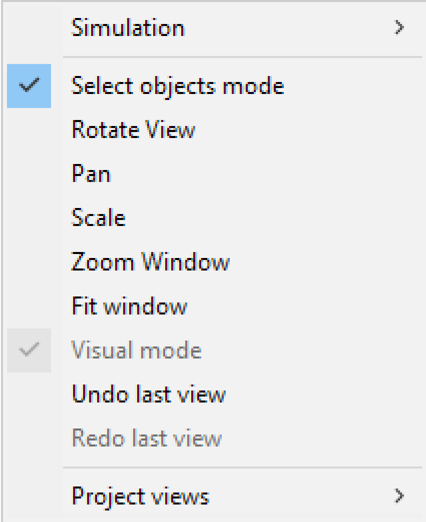
Clicking by middle mouse button (mouse wheel) sets one of the nearest standard views.

See also:

System's main window

3.1.2.2 Objects selection mode

Enable the "Select objects mode" option in drop-down menu of the view port to activate the objects selection mode in the graphical window (it is enabled by default). The mouse pointer should take its usual form  . After that objects can be selected by the mouse cursor in the graphic window. When you move the pointer on the screen, the object below it will be highlighted if it can be selected. The left mouse button click selects the highlighted object.



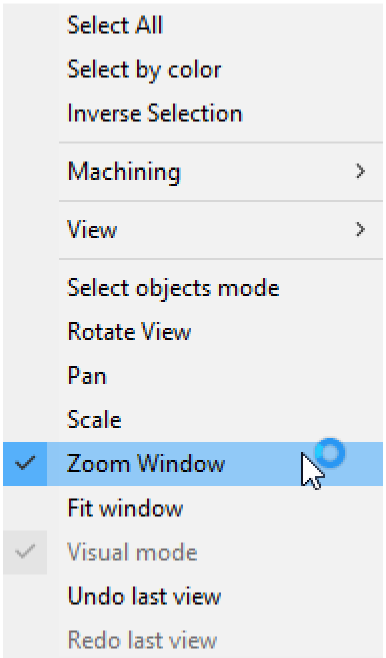
See Objects selection topic for more advanced ways to select objects from the screen.

See also:

Graphic window and visualization control

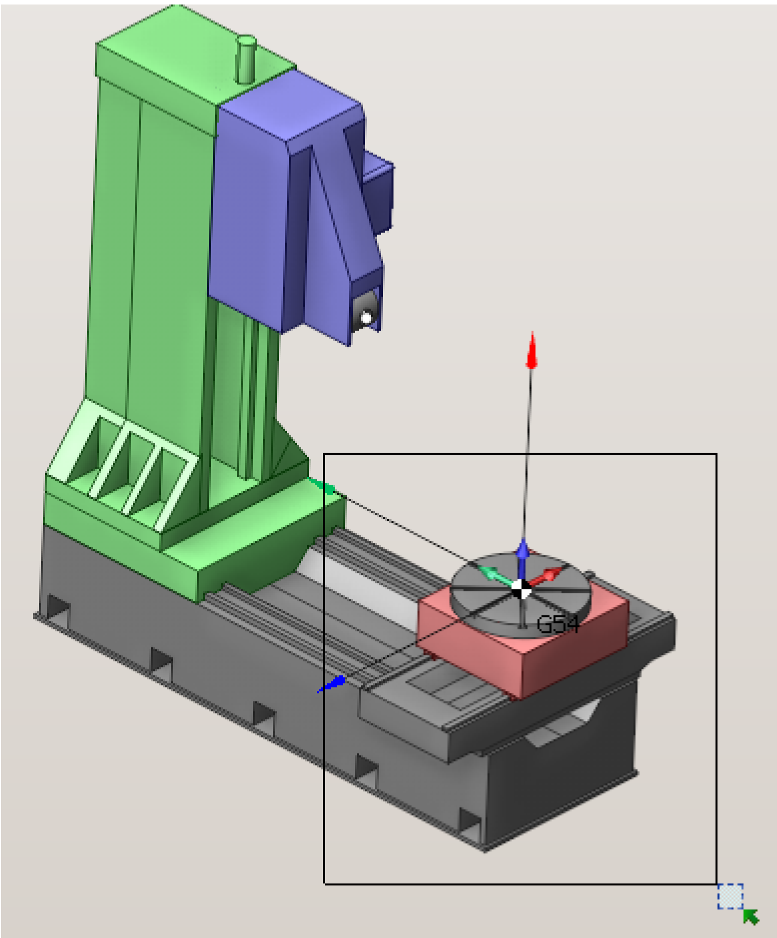
3.1.2.3 Window zoom

Zooms to display an area specified by two opposite corners of a rectangular window.



Select Zoom window popup menu item of the graphic window to set the window zoom mode.

Then specify the first window corner, hold the left mouse button down and then specify the second one. After releasing the mouse button, the area within the rectangle will be magnified to the size of the viewport and the mode will be canceled.



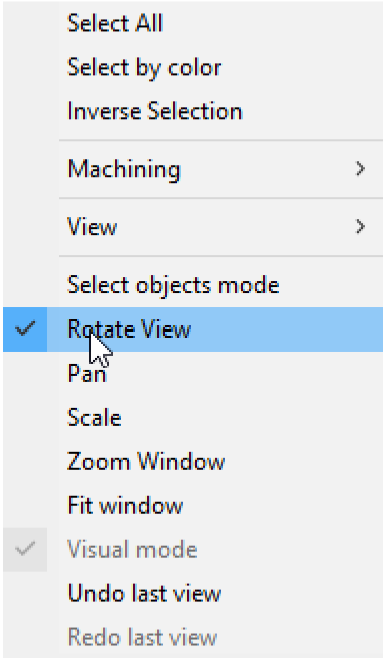
Also you can use [Alt]+right mouse button or left and right mouse buttons at one time to do the same thing without changing the mode.

See also:

Graphic window and visualization control

Interactive zoom in and out

3.1.2.4 Interactive rotate



Press and hold the right mouse button, and drag to the main visualization screen to rotate. Or use the Rotate view popup menu item of the graphic window to switch on the rotate view mode. Drag up and down to rotate around the horizontal screen axis. Drag left and right to rotate around the vertical screen axis.

Press and hold [X], [Y] or [Z] button and drag the mouse to rotate around X, Y or Z axis of the active coordinate system. Hold [**Space**] while rotating to loop the rotation and animate it.

Changing the view vector is also available via the <Standard views> button.

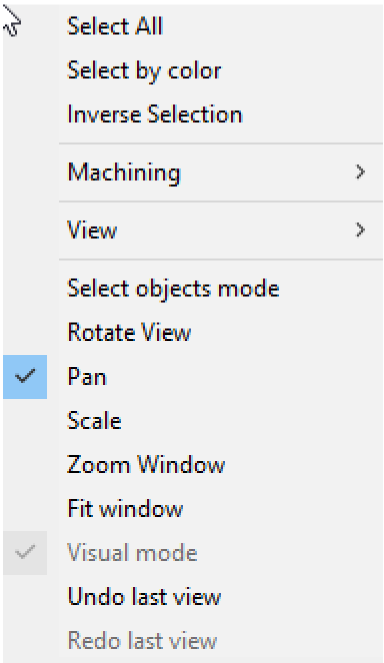
Click the middle mouse button (wheel) in the graphics window provides complete rotation of the current view vector to the nearest standard.

The program also supports various 3D mouse devices, e.g., SpaceNavigator.

See also:

Graphic window and visualization control

3.1.2.5 Interactive pan



Press and hold the middle mouse button (wheel), and drag in the main visualization screen to move graphic objects to a new location. Or use the left mouse button with the pan mode turned on, which can be enabled in the pop-up menu of the graphics window.

See also:

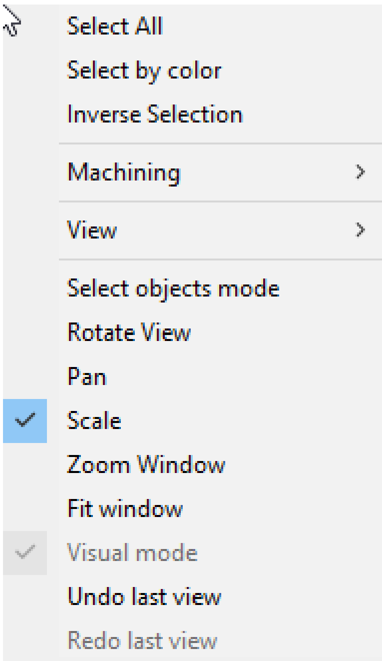
Graphic window and visualization control

3.1.2.6 Interactive zoom in and out

Rotate the **mouse wheel** to zoom.

Also you can use [Ctrl] + **right mouse button move** to zoom.

An alternative way is to activate the **Scale** mode in the **context menu** of the graphics window. Then click and hold the left mouse button while moving the pointer vertically in the graphics window to dynamically zoom in and out. Moving the pointer horizontally in this mode does not affect the image.

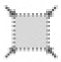


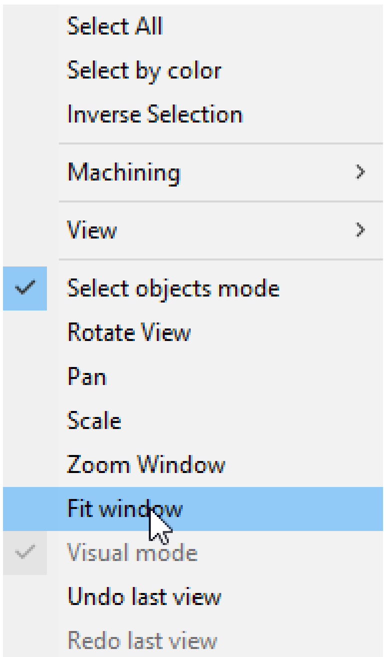
See also:

Graphic window and visualization control

Window zoom

3.1.2.7 Zoom extents

For quick scaling the view to display all visible objects use this  button or **Fit window** popup menu item of the graphic screen.



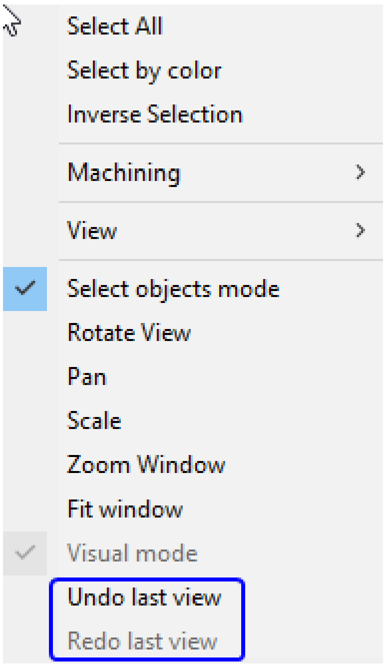
This function can be also activated by **double middle mouse clicking**, but it also activates the closest standard view.

See also:

Graphic window and visualization control

3.1.2.8 Undo view

The **Undo view** feature allows the user to restore the parameters of the active viewport (scale, visualization vector, etc.) to the previous state. Use the **popup menu** of the graphic window to activate it.



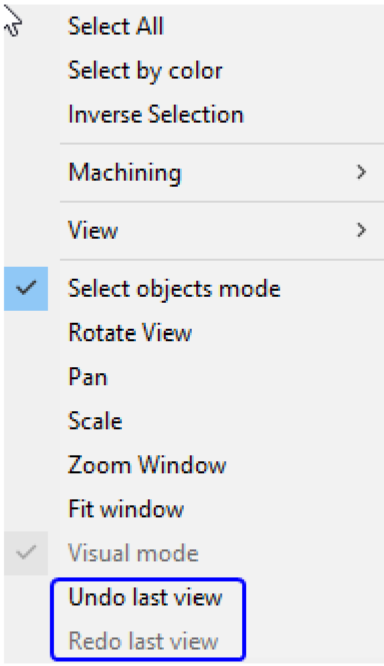
See also:

Graphic window and visualization control

Redo view

3.1.2.9 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 undo feature. Use the **popup menu** of the graphic window to activate it.

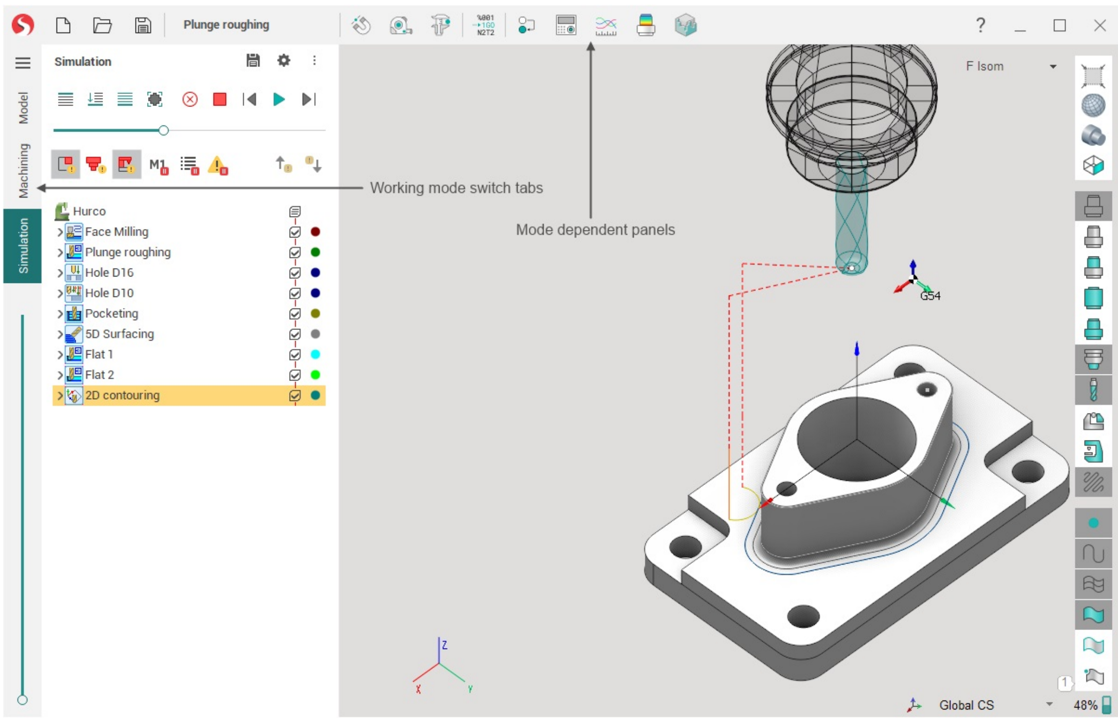


See also:

Graphic window and visualization control

Undo view

3.1.3 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 the tab, the contents of the panels in the main window change:

- **Working mode content page** in the left part of the window.
- **Working mode toolbar** in the middle of the uppermost window pane.

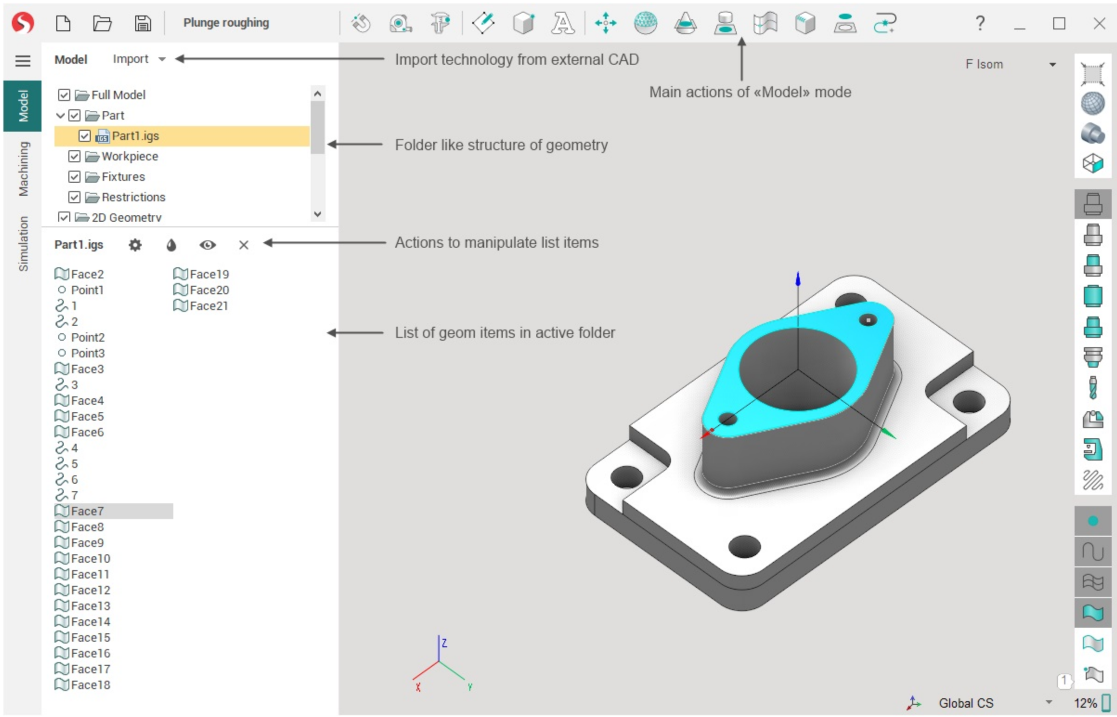
Due to this, the number of buttons displayed simultaneously on the screen is significantly reduced, the interface is simplified, the space for main work expands.

The sequence of tabs approximates the order of basic user actions when working on a project, although the relationship is not strict.

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

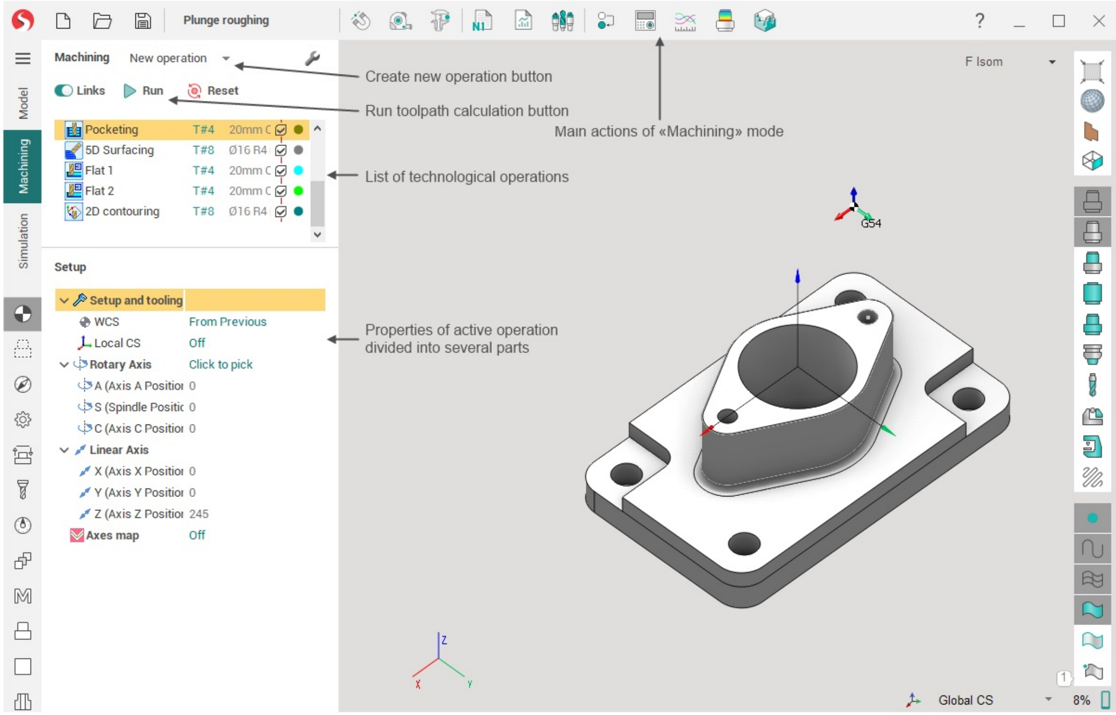
3.1.3.1 Model

In the 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 the existing ones, and manage the object's visual properties.



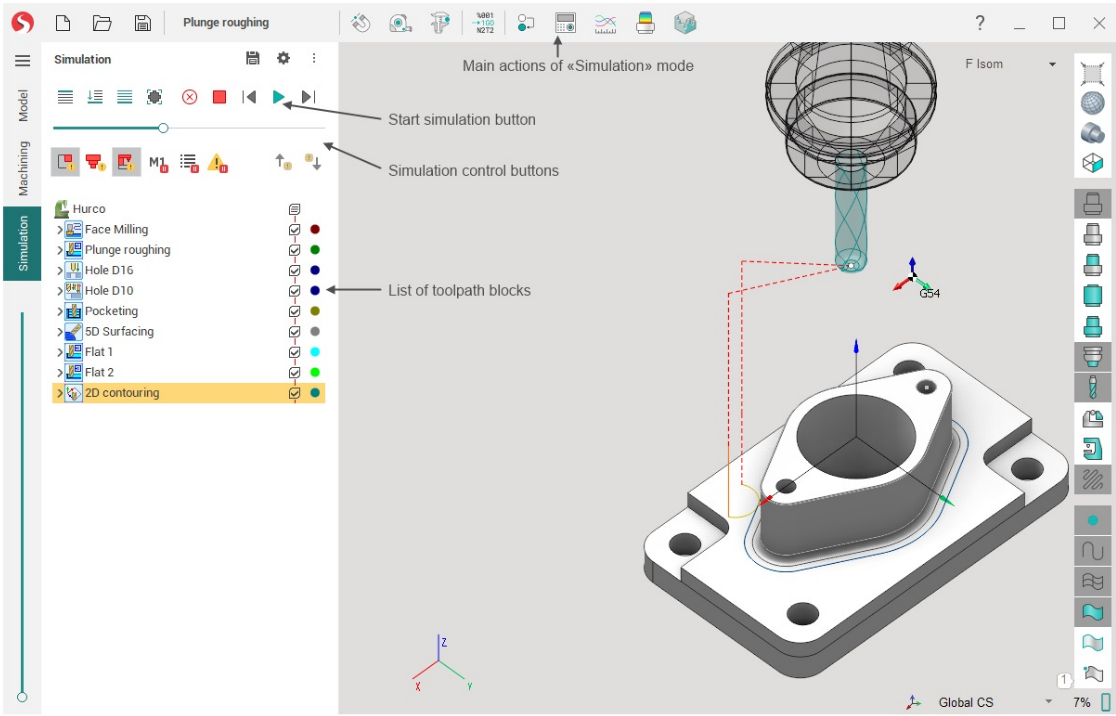
3.1.3.2 Machining

In the Machining mode the user creates the machining process of the part, choosing from the list of available technological operations. The operation determines the processing strategy and the type of toolpath. Here also, the fine-tuning of all the machining operations parameters and the calculation of the tool movement toolpath can be performed. After receiving the toolpath it can be seen on the screen, as well as the preliminary result of machining. After debugging the process, you can run the postprocessor to generate the NC program and machining report.



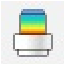
3.1.3.3 Simulation

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



There are several buttons that are common to the Machining and Simulation modes.

-  Machine control panel allows you to watch and modify the current values for all machine / robot coordinates.

-  The Graph of the axes displays the change in each of the machine axes over time in the process of working out the toolpath of the current operation as a graph. This allows, for example, to find unfavorable for the machine parts of the toolpath with bounce, which can lead to vibrations.
-  allows you to display the difference between the original part and the result of machining in the form of a color scheme where each color is associated with a specific deviation range between the compared elements.
-  panel allows to view zones of the part where tool holder does not have collisions and to determine the best angle for parts processing.

See also:

System's main window

3.1.4 Selection filter

Objects selection filter toolbar located at the bottom of the right pane of the main window.

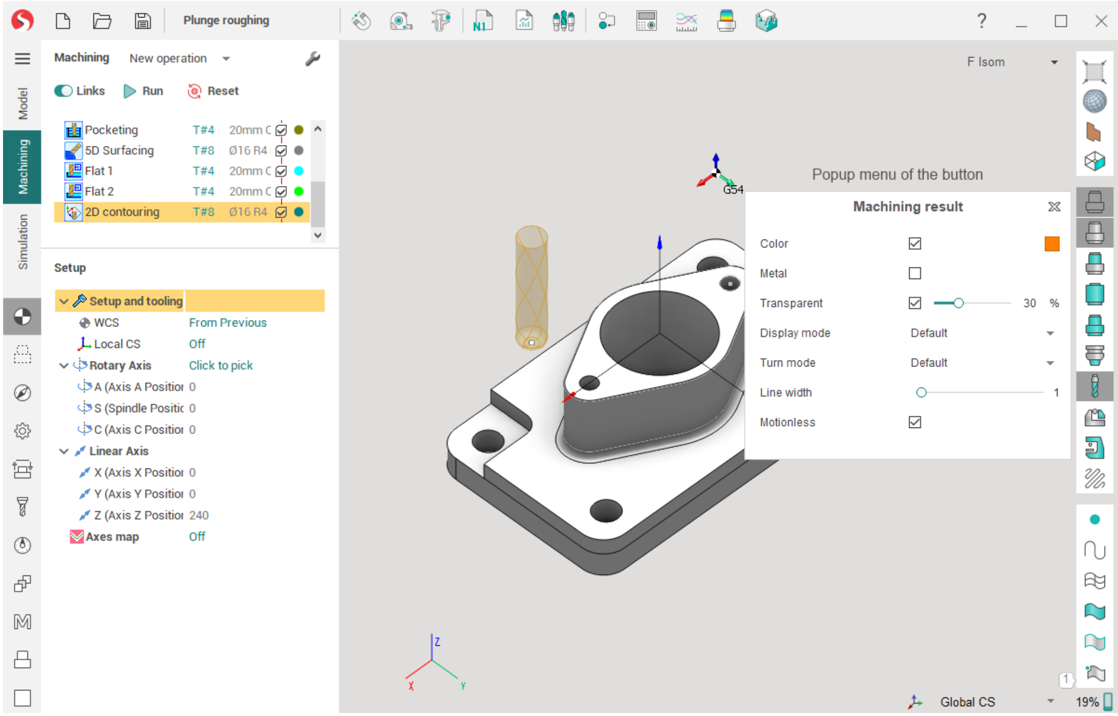
When choosing among the many geometric objects in the graphics window, it is useful to limit the selectability of objects depending on their type (point, curve, mesh, surface, e.t.c.). Filter parameters can be set up by pressing the corresponding buttons on the toolbar. By left-clicking the chosen button the corresponding filter is toggled on and off.

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

See also:

System's main window

3.1.5 Visibility panel



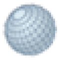
The visibility panel is aimed to manage visibility and visualization parameters of objects for different working modes individually (Model, Machining, Simulation).

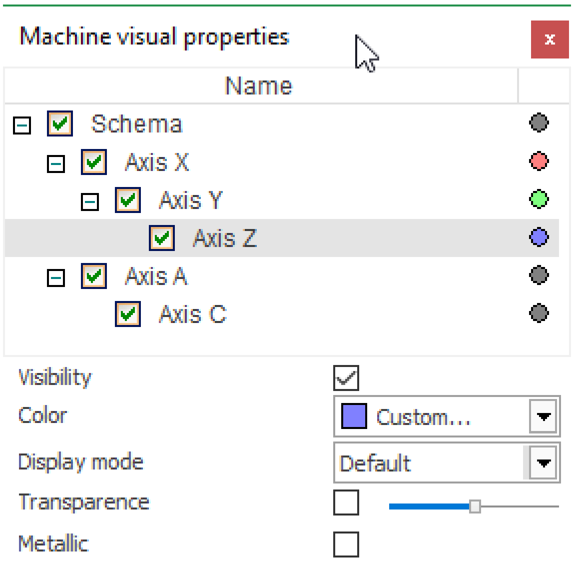
The panel contains buttons to control visibility of:

- Geometrical model
- Part
- Job assignment
- Workpiece
- Machining result
- Tool holder
- Tool
- Fixtures
- Machine
- Tool path

The button pressing by the left mouse button switches visibility on or off for the corresponding object. But the right mouse button click on the same button opens a pop-up menu to change visualization properties of the object.

The pop-up menu can contain the following items (vary depend on item type):

- Color – switches how to draw the object: by one selected color or by different colors (subject to the tool trace for simulation for example). Click to colored box to select color.
- Transparent – the object will be drawn as transparent if the option is selected but as solid otherwise. The transparence extent can be defined in the <Transparence> trackbar.
- Metallic – turns on the metallic reflection for the object faces.
- Woody – turns on the woody reflection for the **machining result only**.
- Display mode – switches the visualization mode for the object (wire, shade, shade with edges and default) The Default item means that the object drawing mode is associated with the main drawing mode which is set by the  buttons on the Visualization control toolbar.
- **Turn mode** - switches the drawing method for revolution bodies of the object (3D, 3/4, 1/2, 2D and default). The Default item means that the object drawing mode is associated with the main drawing mode which is set by the  button on the Visualization control toolbar.
- **Line width** - defines the width of the line to paint edges and lines of exact object.
- **Show points** - for toolpath only. Allows to visualize points at the end of each toolpath block.
- Machine interactive - for the **Machine** button only. It provides the ability to interactive control the machine nodes and change its visualization parameters.
- **Change machine nodes** - The **Machine visual properties** dialog will appear where you can setup visual settings for each machine node individually.



Visibility and visualization properties of objects are defined separately for each of working modes of SprutCAM X. The <Machining result> button controls the workpiece which is machined for the <Simulation> mode but for other modes the same button manages visualization parameters of the machining result of the current operation (or of the full technological process if there is not any selected operation).

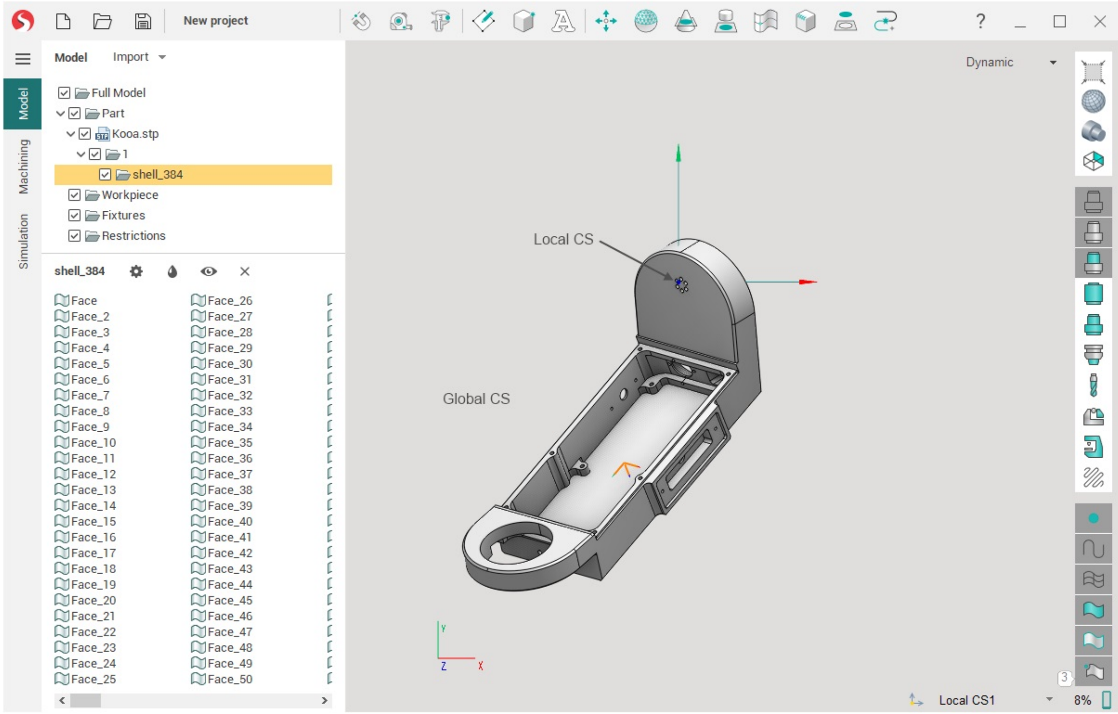
The object visualization settings are saved in the SprutCAM X configuration file and they are loaded for every execution of the application.

See also:

System's main window

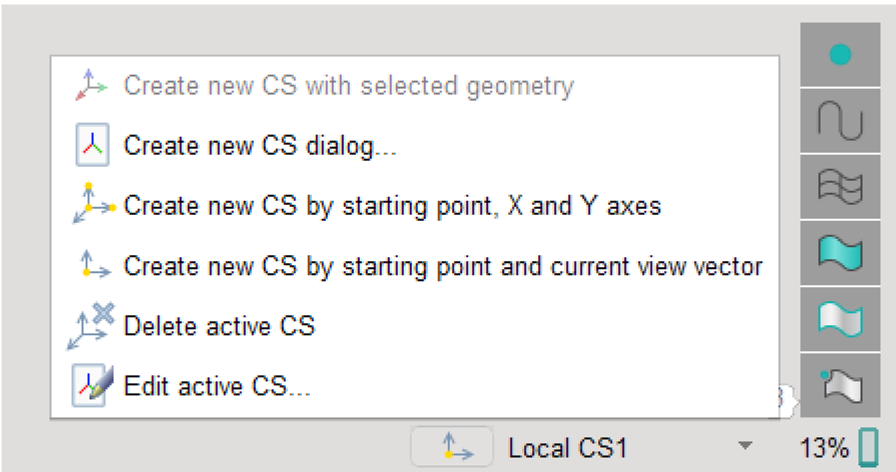
3.1.6 Geometrical coordinate systems

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



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

Coordinate systems management panel located in the lower right corner of the main window



There is a concept of an active coordinate system. Only one CS can be active at the moment, its name is displayed on the panel. Some features of the system take into account the active coordinate system, for example, the overall dimensions of geometric objects are shown in this CS; the plane for new sketch initialized by active CS, etc.

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 name of the CS has a drop-down menu in which the possible actions on coordinate systems are collected.

- Create new CS dialog... Opens a window where you can directly set the parameters of the new CS. If at the time of the click any geometric element is selected, then the window does not open, but the procedure of smart CS creation starts.
- Create new CS by starting point, X and Y axes. The new coordinate system is defined interactively by the origin point and two leading vectors X and Y.
- Create new CS by starting point and current view vector. The new coordinate system is defined interactively by the origin point and the current view vector.
- **Delete active CS**. Deletes the active coordinate system. All listed coordinate systems below this will rise by one level up. The global coordinate system cannot be deleted.
- **Edit active CS**. Opens the parameters window for the active coordinate system.

See also:

System's main window

Creation of the coordinate system by the dialogue window

Smart LCS creation

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

Creation of the coordinate system by the current view direction

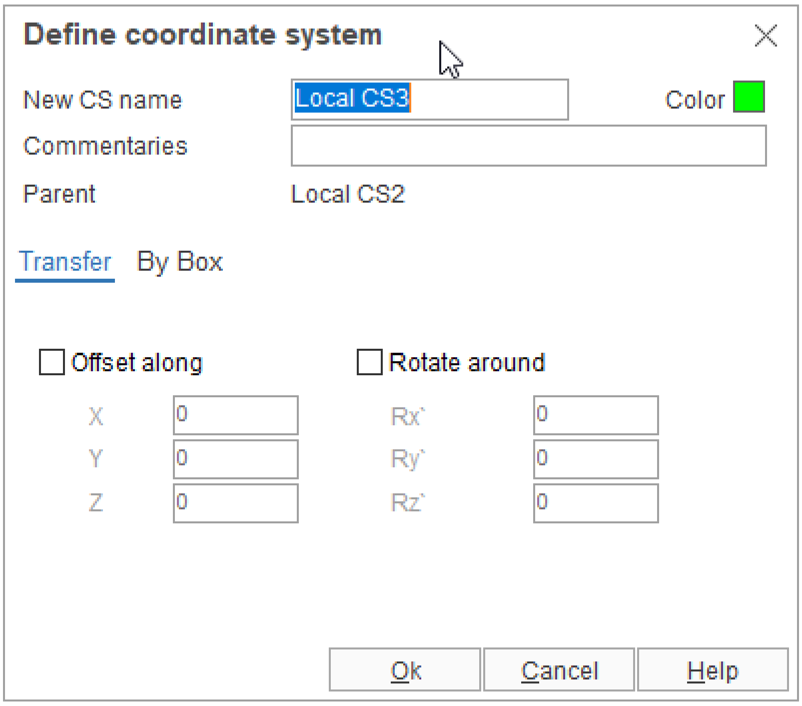
Properties changing of the existing coordinate system

Machine coordinate system G54 - G59

Operation local coordinate system

3.1.6.1 Creation of the coordinate system by the dialogue window

Window of creation new coordinate system opens by pressing the menu item on the coordinate systems panel. If at the time of the click on it any geometric element is selected, then the window does not open, but the procedure of smart CS creation starts.

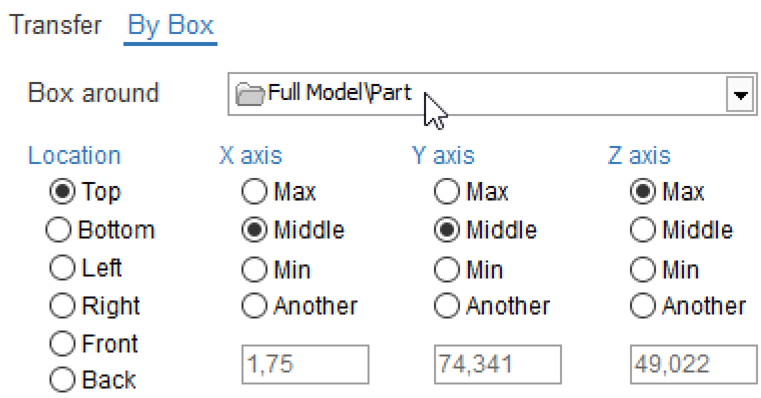


In this window is assigning <New CS name>, its <Color> and <Commentaries>.

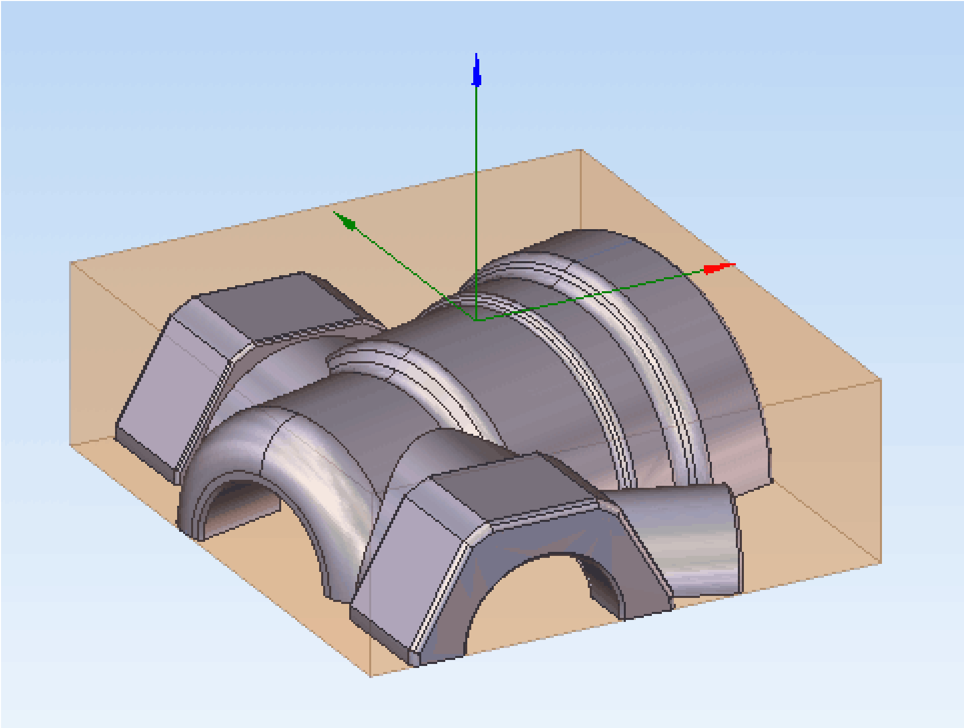
There is two methods of position defining available:

1. <Transfer> – all transformations are performed relative to the active coordinate system. A newly created system can be displaced and/or rotated relative to the parent coordinate system.

2. <By Box> – a newly created system sets on external dimensions of group, that selected in <Box around> list. Select in <Location> how coordinate system will be positioned and give original point. Coordinates of point on corresponding axes can be selected as <Max>, <Middle> and <Min> relatively to the box of group or in absolute coordinates (<Another>).



All changes will be previewed in graphic window immediately.



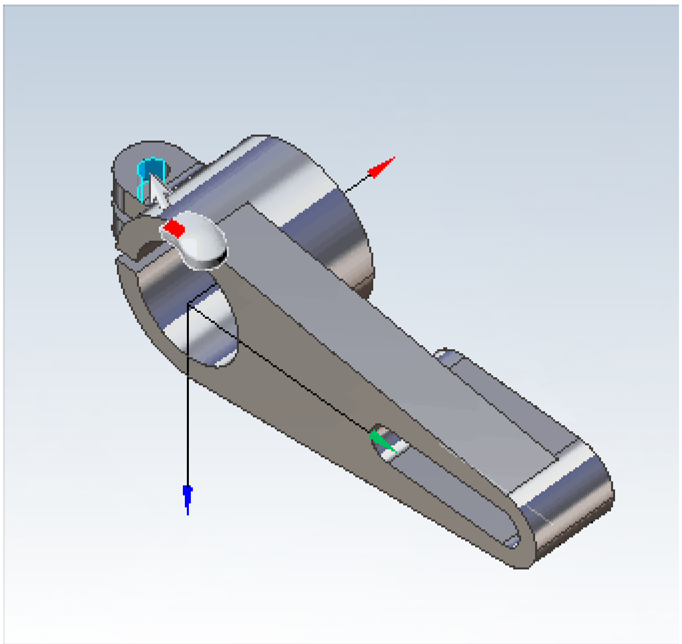
See also:

Geometrical coordinate systems

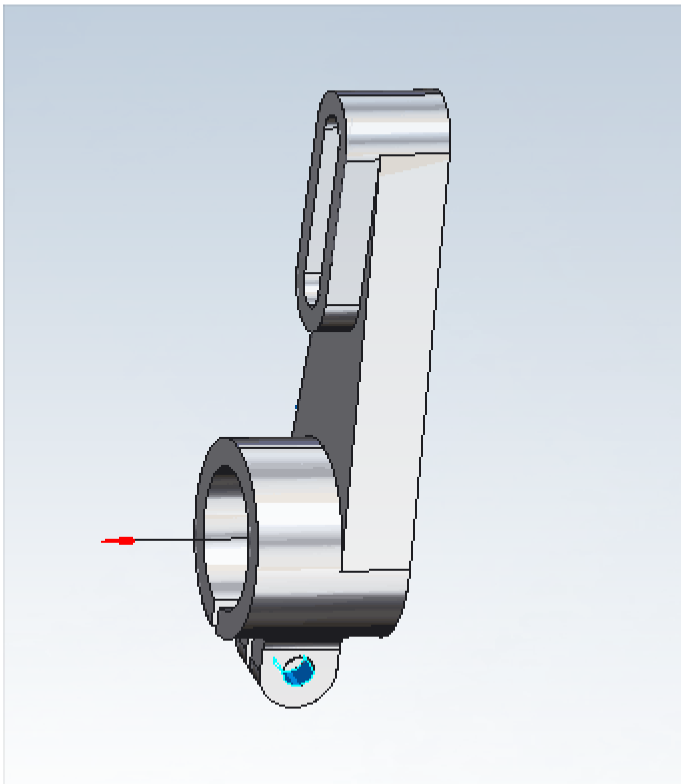
3.1.6.2 Smart LCS creation

You can easily create a Coordinate System you wish with approximately two mouse clicks. It is very straightforward.

1. At first you select in the graphic view a design feature which defines the location of the new coordinate system. You can select any entity that has an origin or an axis or both. E.g. a circular arc, a line, a revolution surface, a plane define 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, as well the X axis direction taken from the neighboring edge. A straight edge of a solid model defines the origin that is placed in the middle of the edge, the Z axis direction taken from the neighboring surface and the X axis direction taken from the edge itself.

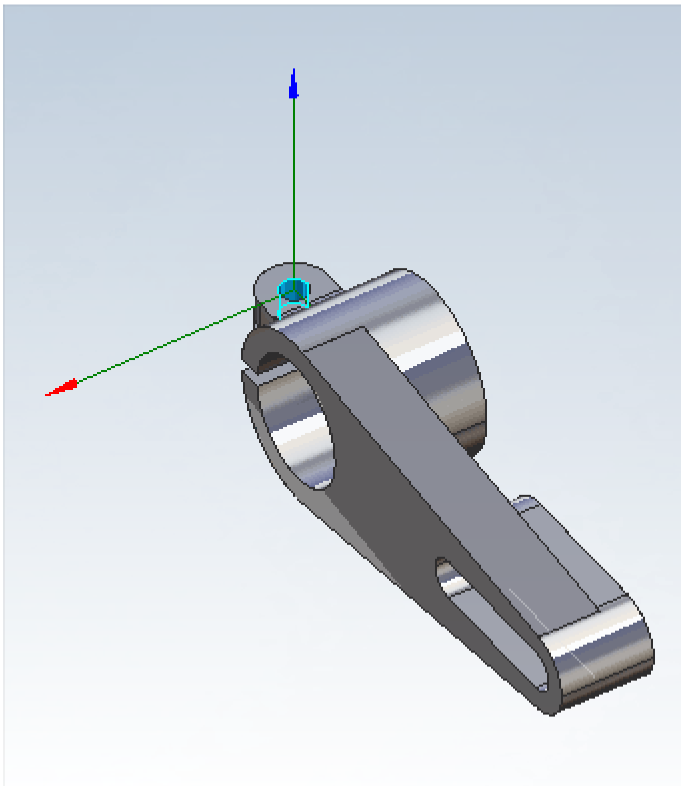


2. As no geometry entity defines a local coordinate system completely SprutCAM X should decide how to select the missing parameters e.g. the Z or X axis direction of the new LCS. And you can easily tell SprutCAM X your wishes. Just rotate the view with mouse in such a way that SprutCAM X could select the missing parameters of the new LCS based on the orientation of the view, just as you do when you orient the view using smart middle mouse click.



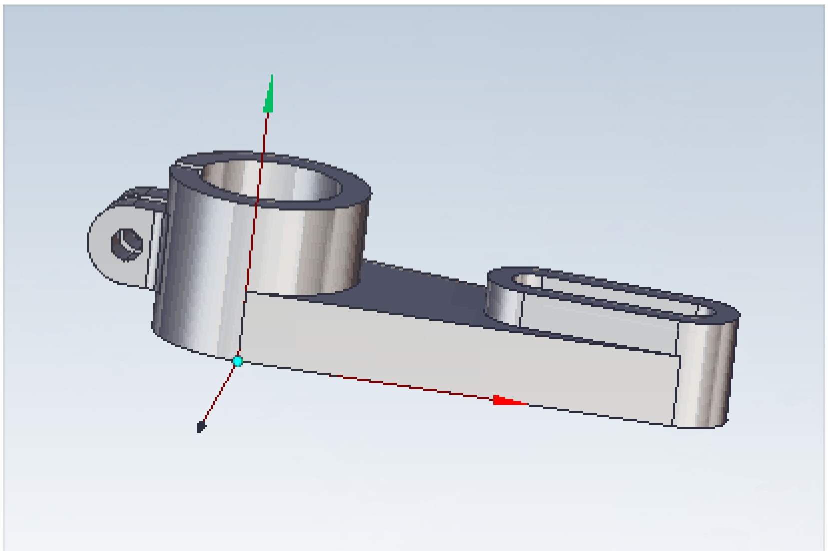
3. The last step is simple click on the

menu item on the coordinate systems panel.

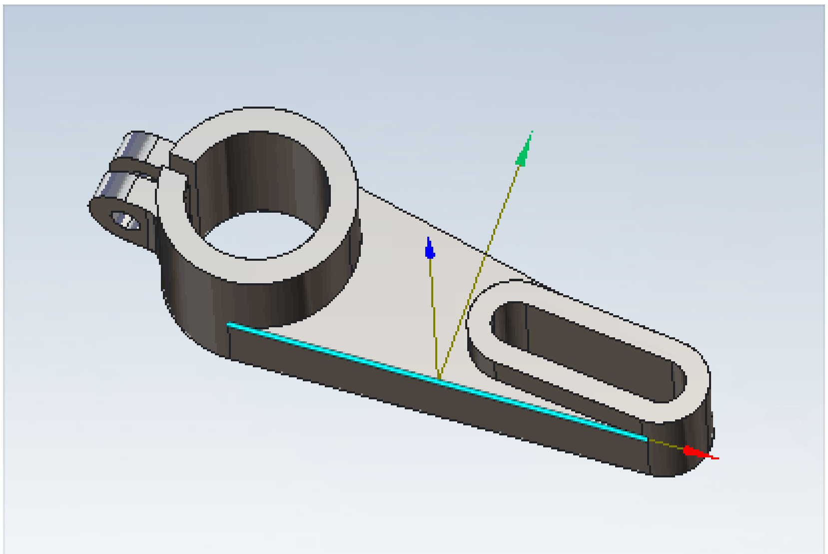


Examples of Smart LCS creation.

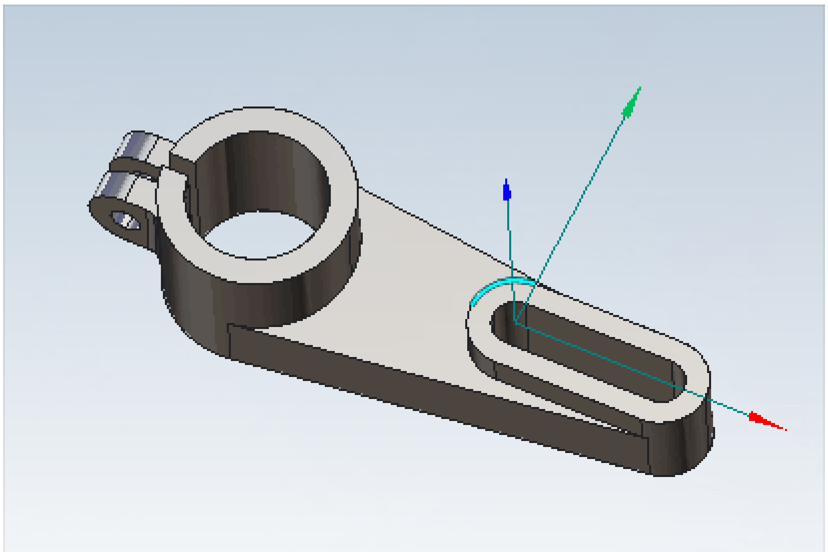
- LCS by a solid vertex:



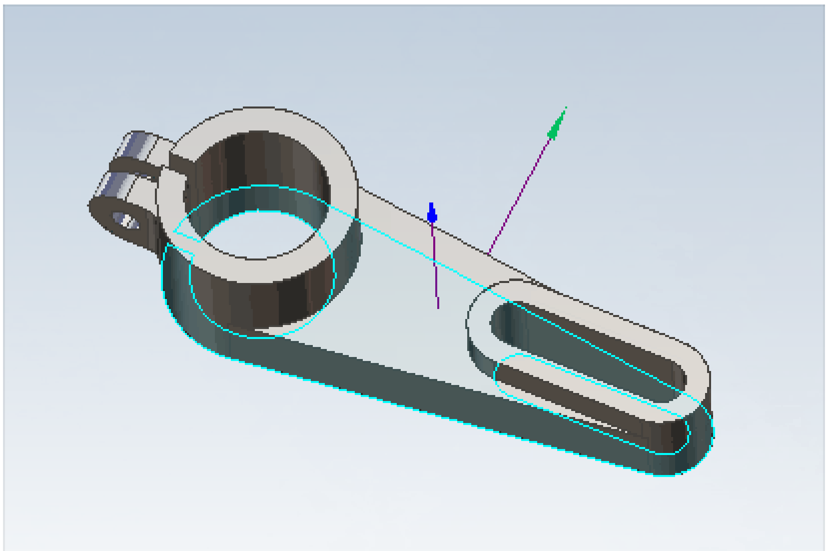
- LCS by a solid straight edge:



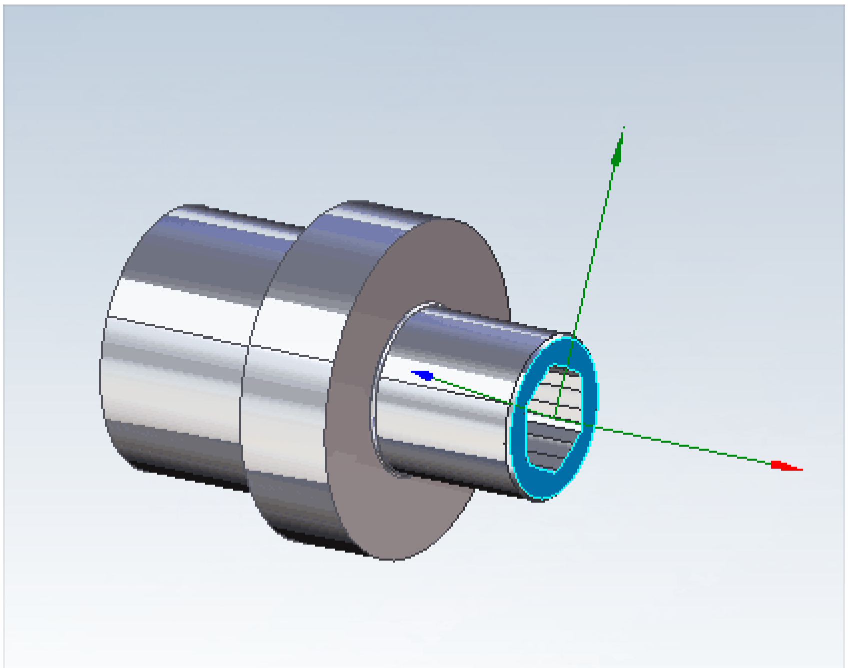
- LCS by a solid circular edge:



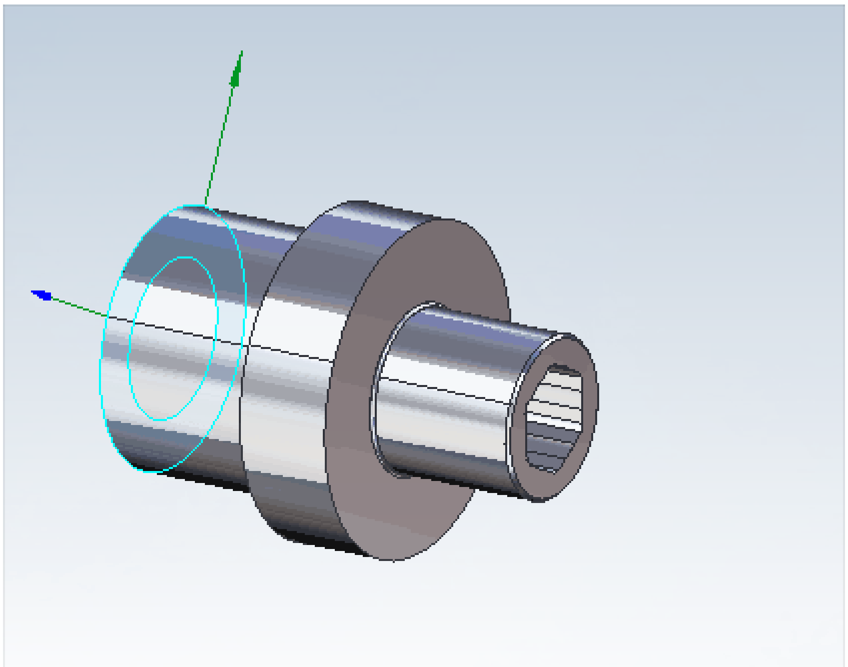
- LCS in the center of a planar face with an inversed Z axis direction:



- A XOY LCS in the center of a flat face:



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



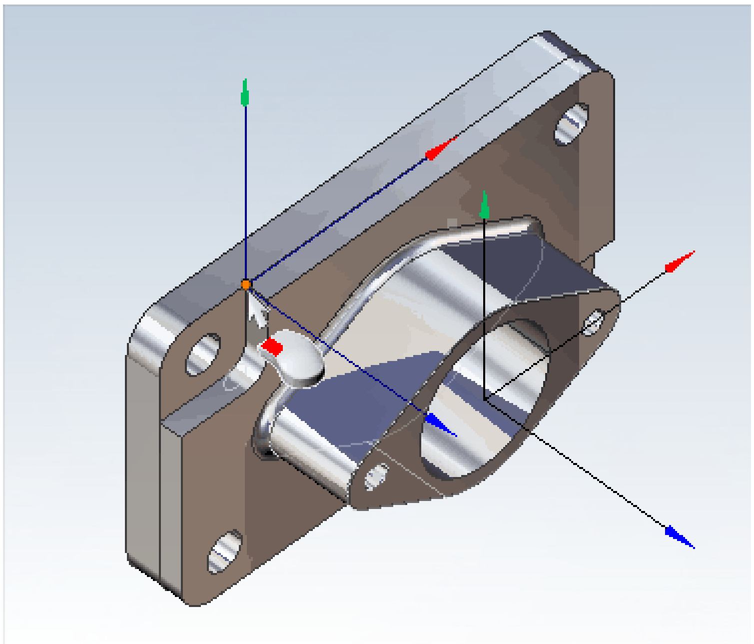
See also:

Geometrical coordinate systems

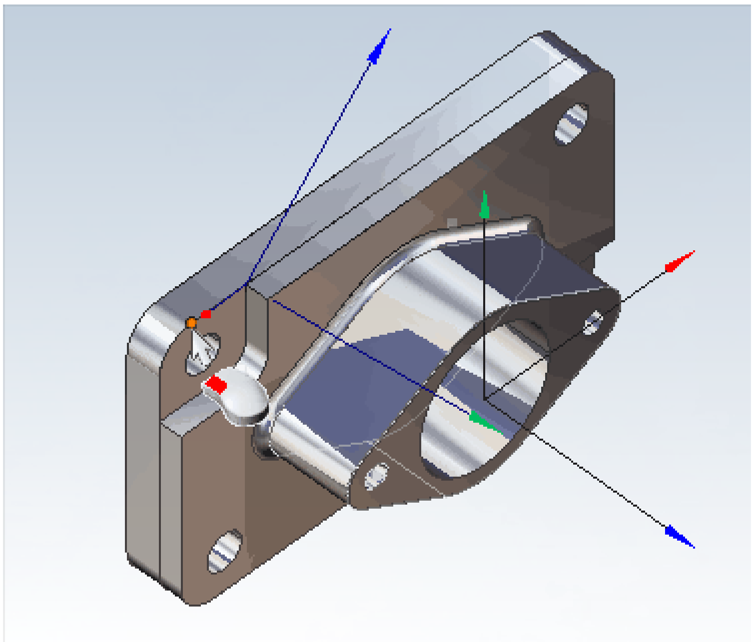
3.1.6.3 Creation of the coordinate system by the origin point and two leading vectors

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

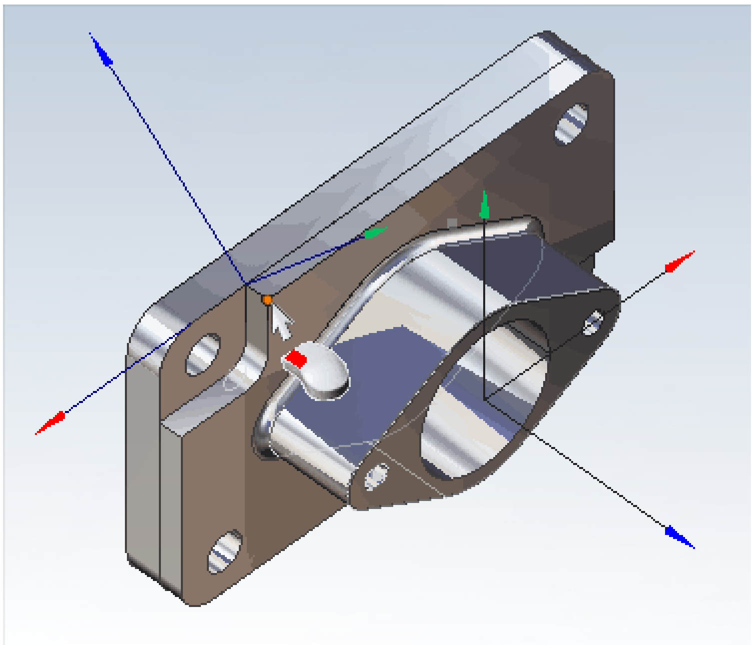
The new coordinate system is defined by the origin point and two leading vectors X and Y. When creating the coordinate system by this method, first the Zero point must be assigned by moving the cursor on the graphic window to the desired point and if it is a valid point to be used as the origin of the coordinate system, then it will be highlighted. The selection is confirmed by left clicking on it:



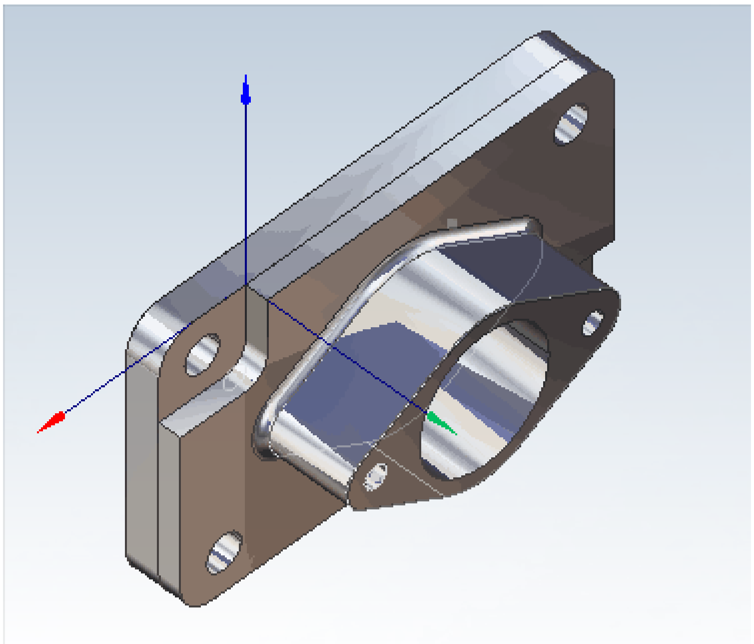
After that the direction of the X-axis must be specified by choosing a point on the screen through which the X-axis will pass, and left clicking the mouse:



And then repeating for the Y-axis:



After that the newly created coordinate system becomes active:



At a later time name of the coordinate system, its color and original point can be changed.

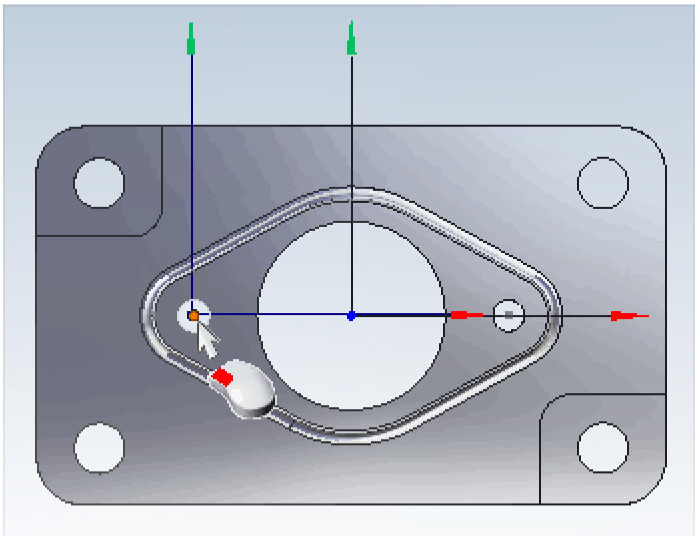
See also:

Geometrical coordinate systems

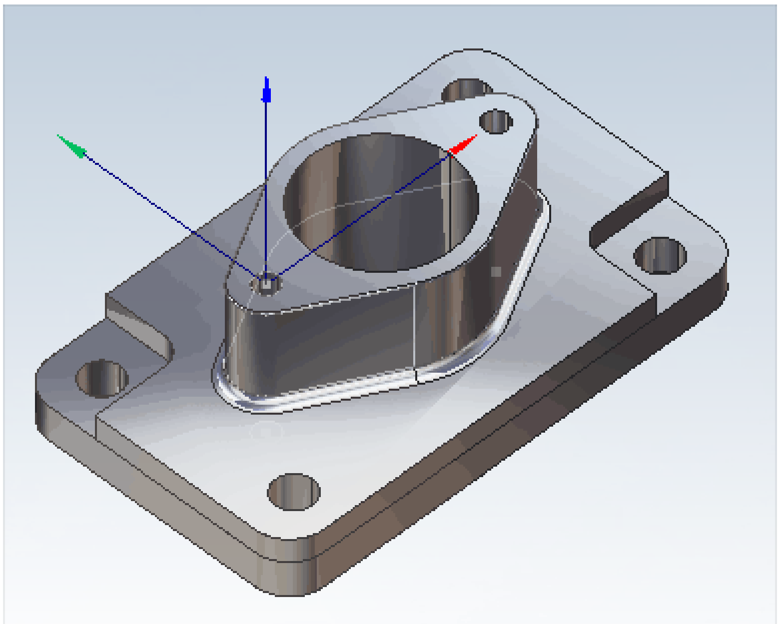
3.1.6.4 Creation of the coordinate system by the current view direction

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, first the required view must be installed, or select one of the standard view. Next an origin point must to be assigned by moving the cursor to the desired point on the graphic window. If this one is a valid point which can be used as an origin of coordinate system then it will be highlighted. The selection is confirmed by left clicking on it:



After that, the newly created coordinate system becomes active:



At a later time name of the coordinate system, its color and original point can be changed.

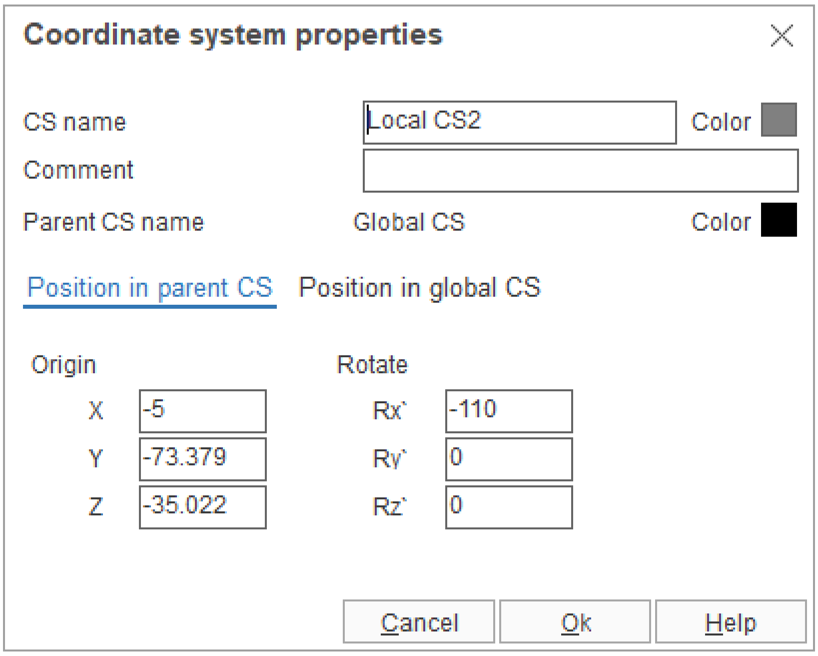
See also:

Geometrical coordinate systems

3.1.6.5 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:



The **CS n**ame can be changed in this window, as well as its <Color> and <Comment>.

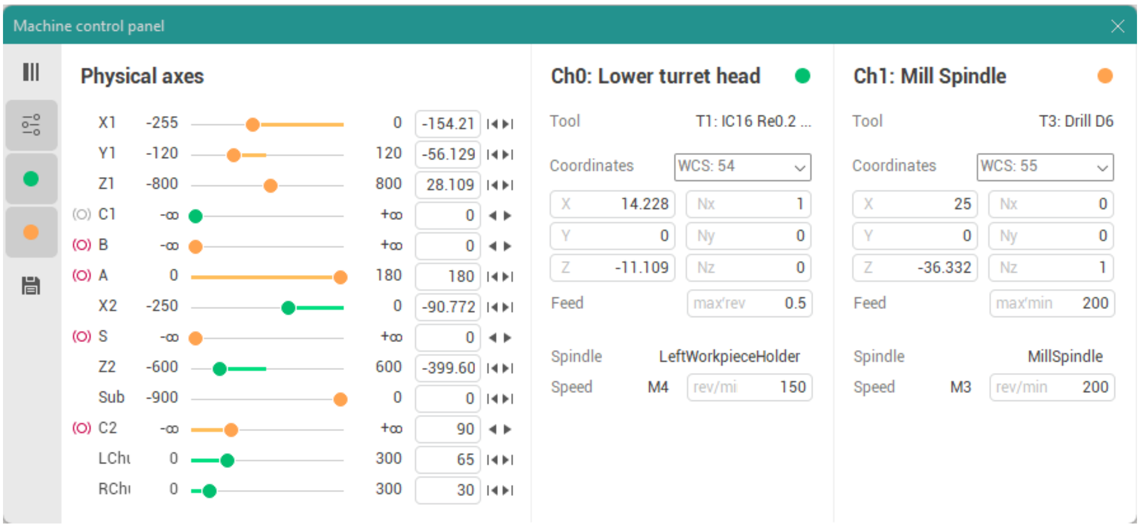
To move or rotate coordinate system, define the displacement value for the <X>, <Y>, <Z> axes or corresponding rotation angles. These values are relative to the <Parent> or <Global> coordinate system depend on the active tab.

See also:

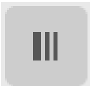
Geometrical coordinate systems

3.1.7 Machine axes control panel

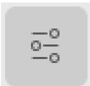
Machine axes control window allows you to watch and modify the current values for all machine / robot coordinates. To show this windows you need to click  button on the main toolbar. It is helpful when analyzing tool path in simulation mode, when constructing of manual approaches or when determining the initial position of the machine to calculate the technological operation.



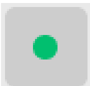
Visually, the window is divided into several areas. The number of panels and their content may vary depending on the particular machine configuation. The toolbar at the top or left side controls the visibility of these panels.



- The **Switch layout button** switches the layout of panels between vertical and horizontal.



- The **Physical axes panel** button shows/hides the panel with the list of physical machine axes.



- The **Channel panel visibility** button shows/hides the panel with the information about the exact control channel of the machine. There may be several such buttons, depending on the number of channels that a particular machine has. The color of the icon on the button indicates the unique color associated with the channel. All other information in the window will be displayed in this color if it refers specifically to this channel.

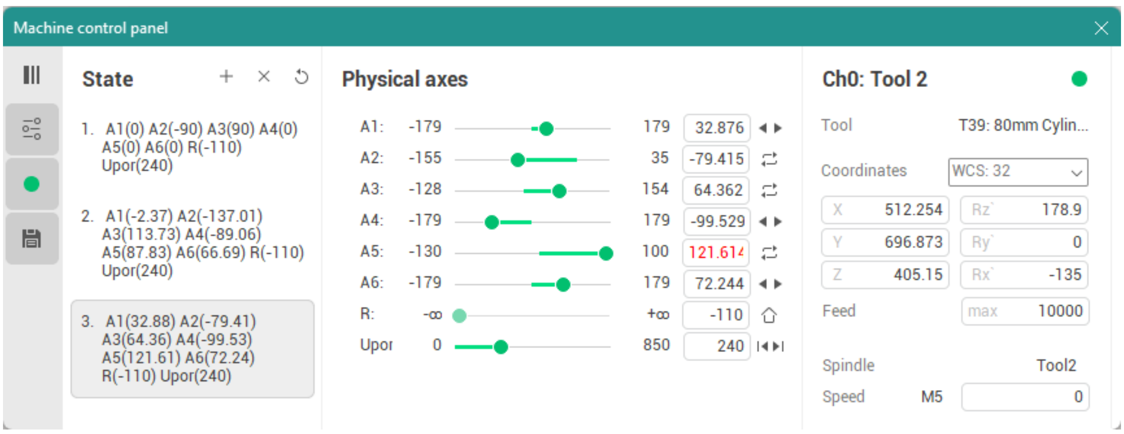
- The **States panel visibility** button shows/hides the panel which displays a list of stored states of the machine.

The **Channel panel** displays the following values.

- Color, index and name of the channel.
- Active tool numer and name of the channel.
- Active WCS number combo where you can also choose some additional coordinate systems to display tool coordinates.
- Tool coordinates and orientation angles, vector or quaternion in the specified spatial CS. Orientation angles type can vary depend on exact 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.
- Current spindle speed, rotation direction (M3, M4, M5) and rotation mode (measurement units, e.g. rev/min for RPM and m/min for CSS).

The **Physical axes** panel contains a list of linear, rotary and auxiliary machine axes that are defined in the machine schema. The following information is displayed for each axis.

- Axis name.
- Current physical value of the machine axis which does not depend on the current WCS and it depends only on how this axis is defined in the machine schema by its creator. It may be red if the axis is out of range.
- Minimal and maximal limits of the axis.
- The state of the machine axis brake if this axis has a brake.
  - - brake is off.
  - - brake is on.
- The button to quick modify the axis value. The type of the button depend on machine axis properties.
  - - returns the axis to its home position.
  - - moves the axis to its minimal limit.
  - - moves the axis to its maximal limit.
  - - subtracts one period for the rotary periodic axis if it does not exceed the limit.
  - - adds one period for the rotary periodic axis if it does not exceed the limit.
  - - switches the associated machine axes to an alternative solution if the machine can provide the same relative position of the tool and the workpiece in several ways.



The State panel displays a list of stored states of the machine and a few buttons to control the elements of the list. Buttons have the following purpose.

- - **Add new state** button adds the current machine state to the list.
- - **Delete selected state** button deletes selected machine state from the list.
- - **Delete all states** button removes all states from the list.

You can use double-click it in the list item to quick switch the machine to one of the stored states. Using the <Remember state> feature is helpful while constructing of manual transitions on machines with complex configuration of the axes (such as 6-axis articulated robots).

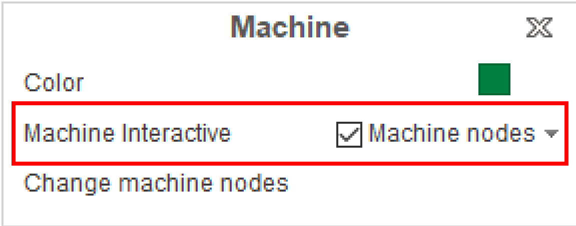
See also:

Step-over building

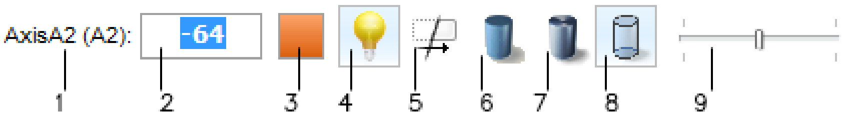
System's main window

3.1.7.1 Interactive Machine

System provides the ability to interactive machine control and change its visualization parameters. To enable or disable this feature you use special item in the pop menu of machine visibility button:



Machine components under the cursor will be highlighted. Dotted line shows the rotation axis for rotary axes. After click on any machine node visual parameters window will be shown:



1 - Node name;

2 - Current position of axis. You can type wanted value or use mouse wheel to set it. This field will be shown only for movable machine nodes;

3 - Machine node color. Click to show Select Color dialog;

4 - Machine node visibility;

5 - Switch to "Ghost mode". In this mode only invisible nodes will be shown;

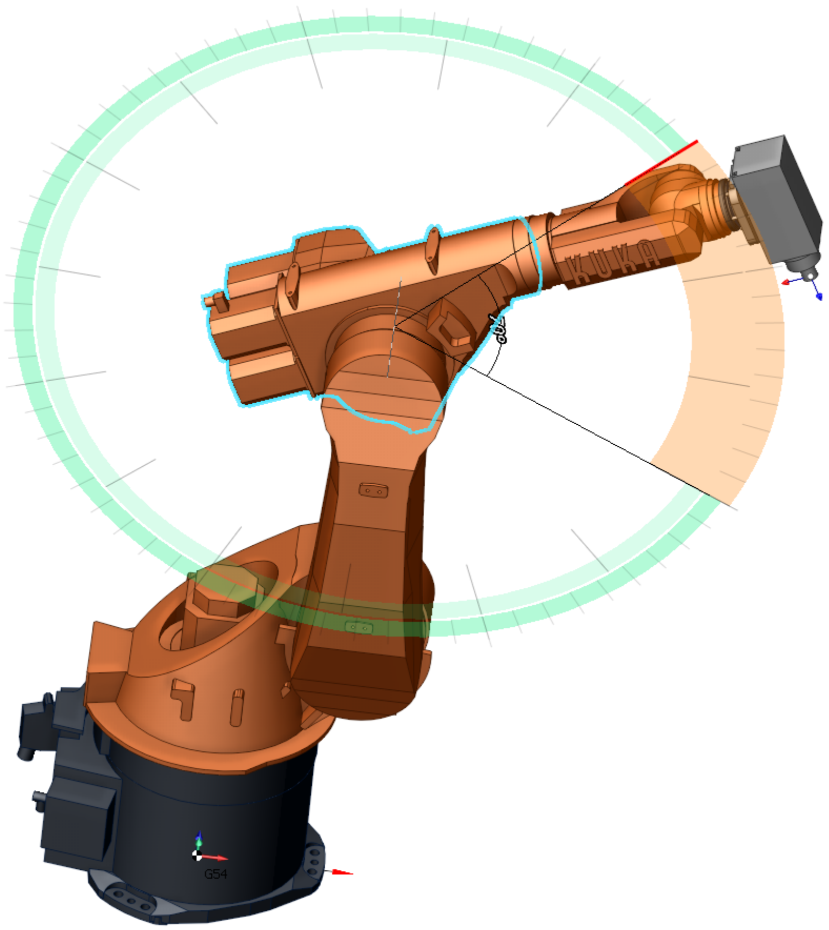
6 - Machine node painting mode (Wire, Shade, Edge-Shade);

7 - Turn on metallic mode;

8 - Machine node transparent;

9 - Machine node transparency.

It is possible to move machine nodes position using drag-and-drop. If node is not movable, nearest movable parent will be moved. For rotary nodes auxiliary circle, showing rotation plane, will be shown. The radius of the circle is equal to the distance from the axis of rotation to the point of clicking on machine node:



It is also possible to set whole machine position by setting wanted tool position. Machine will change position of nodes to provide wanted tool position. Drag-and-drop tool to use this feature. If machine has not active tool, yellow point will be shown instead. The tool moves along view plane, so set view vector along axes X,Y,Z for detailed control.

See also:

Graphic window and visualization control

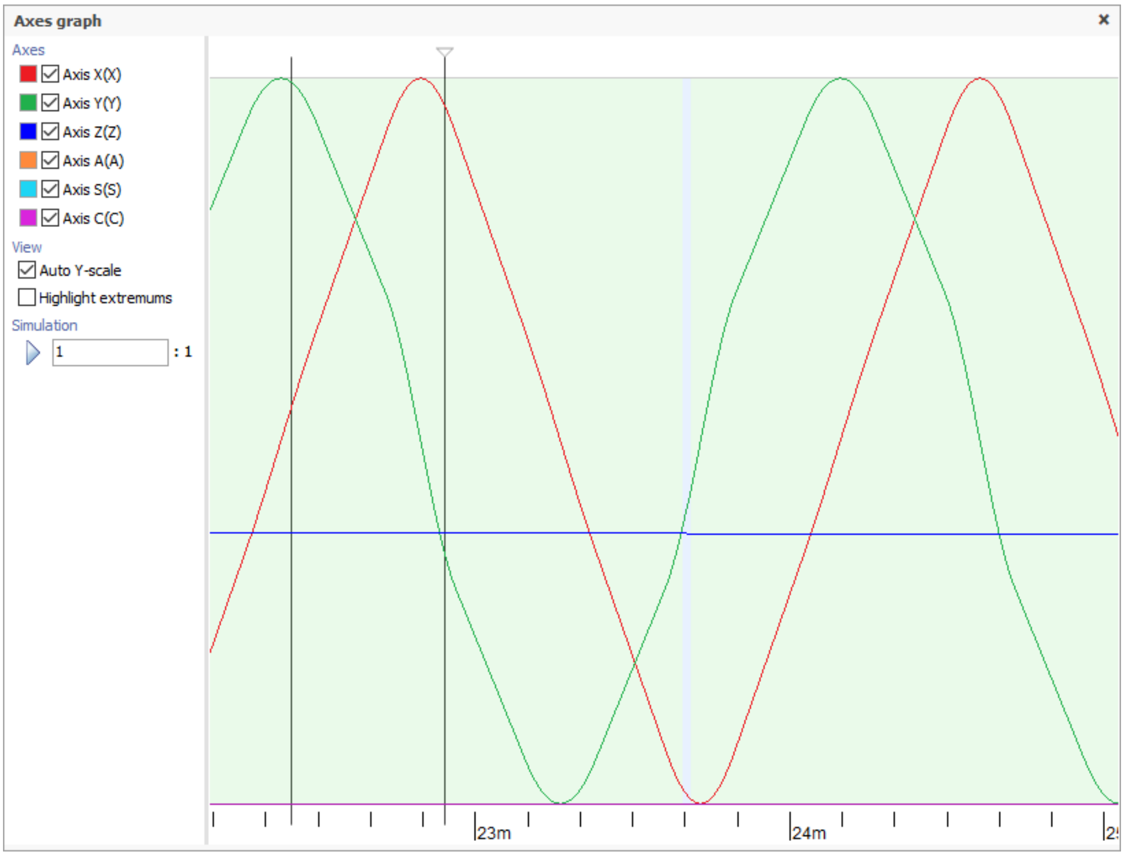
Standard views

Video example about interactive machine

3.1.8 Graph of the machine axes window

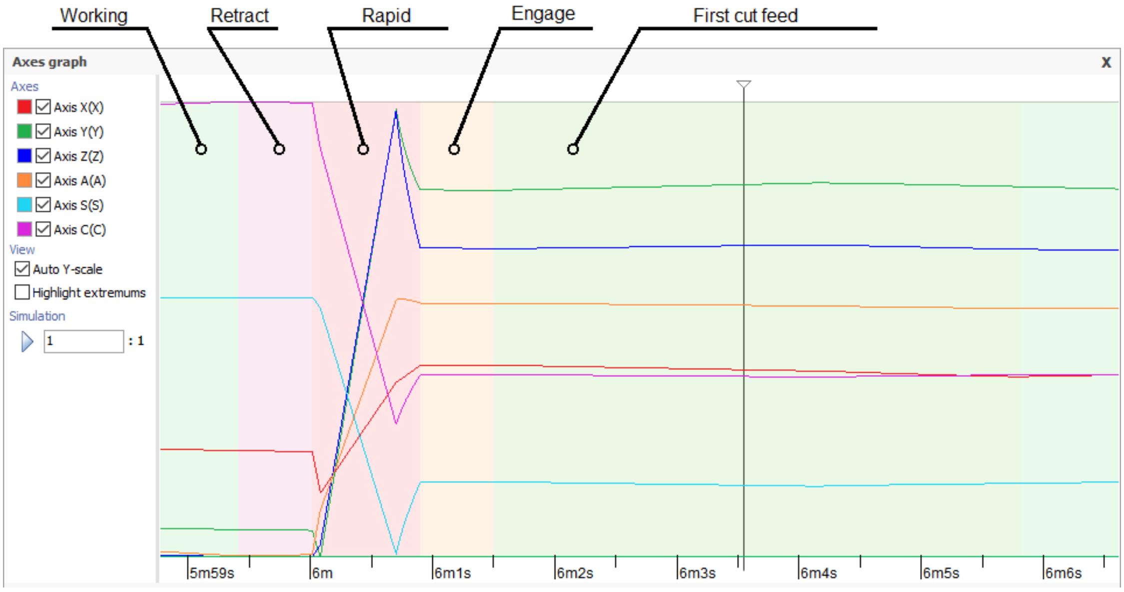
This feature allows you to view the change in the values of the machine axes over time as the tool travels along the path for the current operation as a set of graphs. In order to get a graphic, you need to select the calculated operation in the list of operations and click on the button  on the main toolbar.

The window has the following form.



- **Axes**. It is possible to disable the visibility of individual axes and assign a color to them by clicking on the colored square.
- **Auto Y-scale**. For ease of viewing, the vertical size of the graph of each axis is stretched to the same height. When enabled, the scale on the vertical axis is automatically selected by the maximum and minimum values of the axes. The scale on the horizontal axis can be changed by rotating the mouse wheel while the pointer is in the graphics area.
- **Highlight extremums**. Allows you to display in the form of points the places in which the direction of change of the axis is reversed.
- **Simulation**. The  button starts the simulation of the movement of the tool and all axes of the machine synchronously in the main graphics window and on the axes graph. The running marker will display the current position on the axis chart. By clicking the left mouse button on the graph, you can set the position of the marker at an arbitrary point in time. You can change the speed of the simulation by scrolling the mouse wheel or manually entering in the field  .

The background color below the graph indicates the type of feed on which the movement is performed: working, rapid, engage, retract, etc.



3.1.9 Holder occlusion check utility

Holder occlusion check panel allows to view zones of the part where tool holder does not have collisions and to determine the best angle for parts processing.

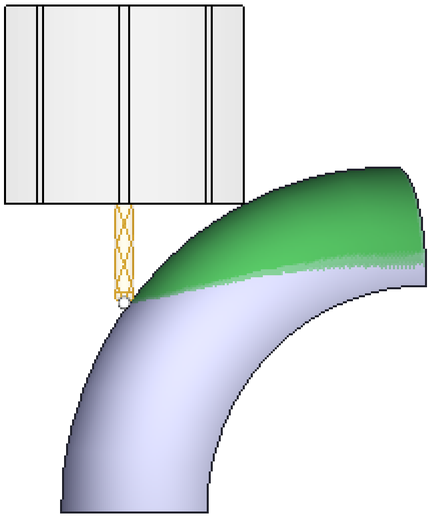
Its use is helpful in the preparation of technical process while parts making.

To show this windows you need to click  button on the <Control> tab of the main panel.

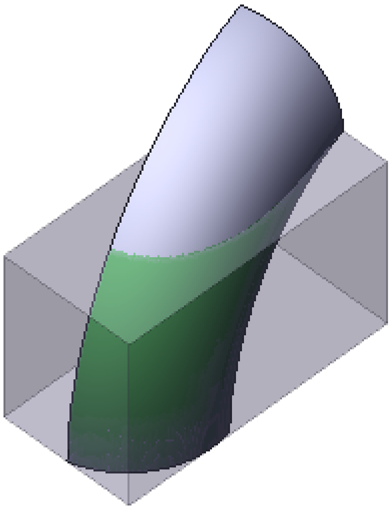
Visually, the window is divided into two areas. Their content may vary depending on the particular machine configuration.

The top panel displays the main parameters of operation that affect on safe zone size of tool holder.

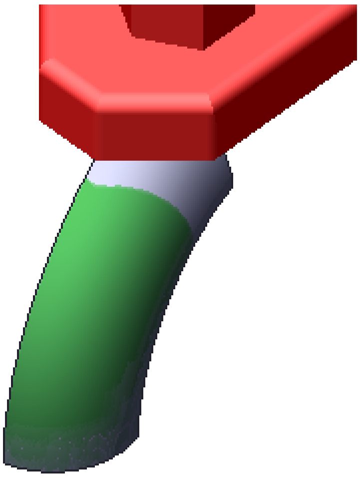
- Check Holder



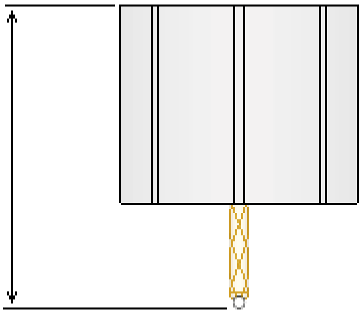
- Check Workpiece



- Check Fixtures



- Overhang



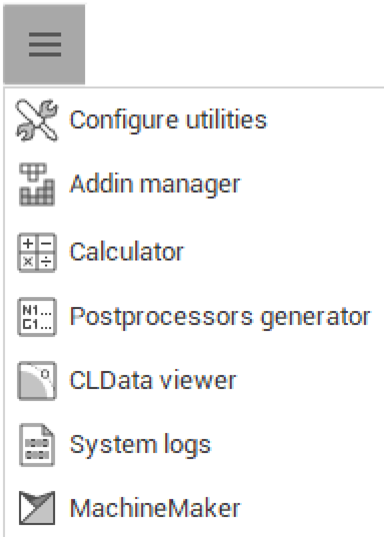
At the bottom of the window is the <Rotary Axis> panel. It displays information about the current machine axes status and tool for searching the optimum tool angle.

See also:

System's main window

3.1.10 Utilities button of main panel

The utilities menu contains items for running some additional internal tools and external user utilities. You can tune it by clicking the **Configure utilities** menu item. Some items can be hidden, depending on the **Show expert tools** option in the system settings.



Button	Description
	Configure utilities – shows the Utility manager window.
	Addin manager – opens the Addin Manager window.
	Calculator – runs an internal calculator.
	Postprocessors generator – runs the Postprocessors generator - application to generate postprocessor tuning files.
	CLData viewer – starts the CLData viewer – the application which helps to create C# based postprocessors.
	Create interpreter - shows the Create interpreter window.
	System logs - opens the window which contains a list of events that have occurred in the application since launch.
	Machine maker – starts the application to create Machine schemas.
	Scripts IDE - opens the window to write and debug Sprut script files.
	Operations manager - starts the manager that allow to enable/disable external operations, whose parameters are described inside separate xml-files and stfc-containers.

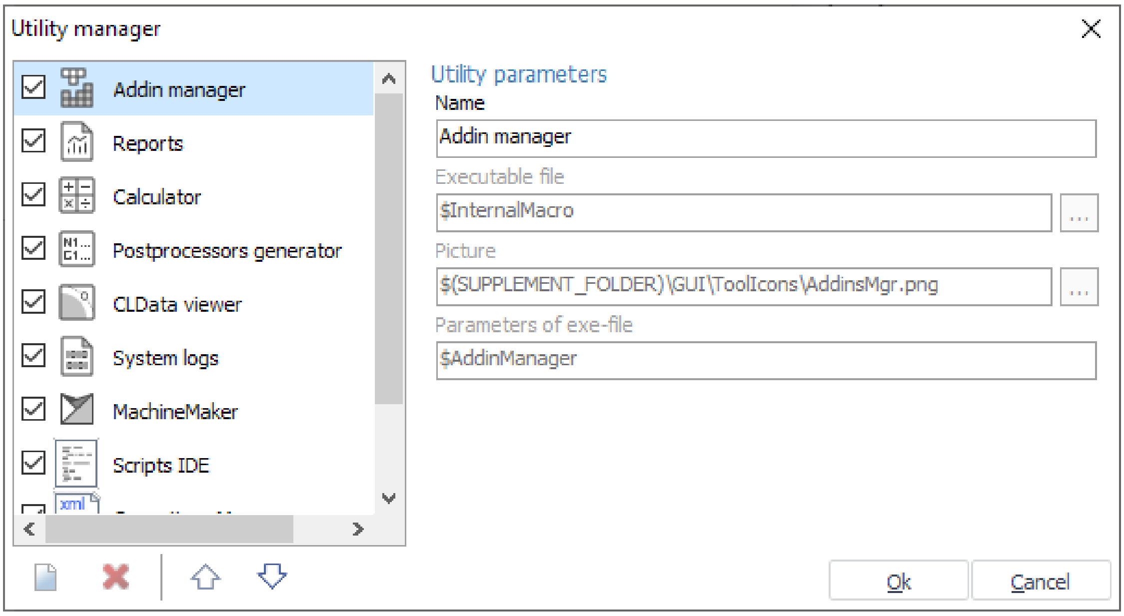
Button	Description
	Create script operation - shows the dialog to create a scriptable operation.

See also:

System's main window

3.1.10.1 Configure utilities window

Utility manager window allows you to customize **Utilities** drop-down menu of the main toolbar. You can drag or hide standard utilities if you do not use them (AddInManager, Reports maker, SprutIDE, Calculator, Postprocessors generator etc.) and add external utilities instead that you want.



You can add as utility

- any external executable file (*.exe, *.bat);
- Sprut-script file (*.spr, *.s);
- windows *.dll file which realizes simple plug-in of SprutCAM X interface which is below. The GUID of plug-in must be defined in Parameters of exe-file edit box.

```
IST_CAMPluginsEnumerator = interface(IUnknown)
  ['{719AC6C2-F83C-4C93-9E50-6AAFADBD8873}']
  function MoveNext: WordBool; safecall;
  function GetCurrent: TGUID; safecall;
end;

IST_CAMPlugin = interface(IUnknown)
```

```
  ['{80BBC39B-1E2B-4D46-B87E-3A3658A77063}']
  function Get_PluginID: TGUID; safecall;
  function Get_PluginCaption: WideString; safecall;
  function Get_PluginDescription: WideString; safecall;
  property PluginID: TGUID read Get_PluginID;
  property PluginCaption: WideString read Get_PluginCaption;
  property PluginDescription: WideString read Get_PluginDescription;
 end;

 IST_UtilitiesButtonCAMPlugin = interface(IST_CAMPlugin)
  ['{4B74BB21-9F48-4D62-9870-0A831C8AD2DA}']
  procedure OnButtonClick(const SenderApplication: IUnknown); safecall;
 end;
```

The plug-in dll should export two main finctions:

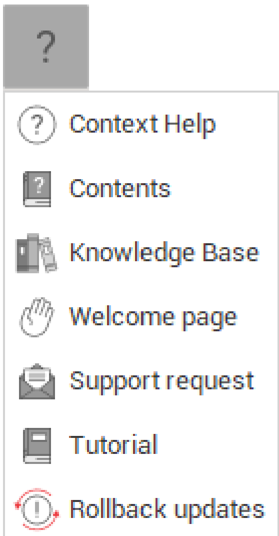
```
  function GetPluginsEnumeratorOfType(PluginInterfaceID: TGUID): IST_CAMPluginsEnumerator;
safecall;
  function CreateInstanceOfPlugin(PluginID: TGUID): IST_CAMPlugin; safecall;
```

See also:

System's main window

3.1.11 Help button of main panel

Help menu consist of actions that allow to get answers to emerging questions.



Button	Description
Context Help	Shows SprutCAM X help.
Contents	Shows contents of the help.
Knowledge Base	Knowledge Base
Welcome page	Opens a page with the actual information from SprutCAM X web site.
Support request	Prepare a message to the technical support service of SprutCAM Tech Ltd., e-mail: support@sprutcam.com
Tutorial	Runs SprutCAM X tutorial

See also:

System's main window

3.1.12 Process indicator

The process indicator runs when the system performs lengthy operations such as the geometrical model import, toolpath calculation, machining simulation etc. Left clicking on the indicator cancels these operations only after being reconfirmed.

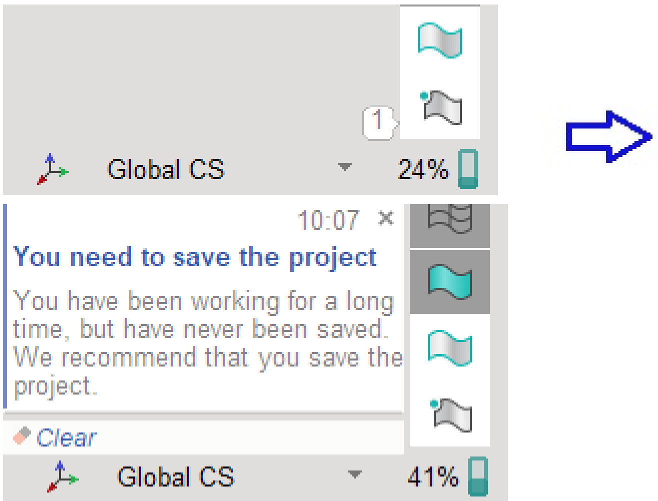
The process indicator is on the main window bottom.

See also:

System's main window

3.1.13 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, which briefly describes the essence of what happened. The icon in the corner also shows the total number of such notifications. By clicking on it, you can open a panel with a complete list of notifications.



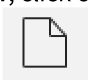
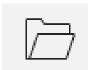
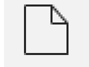
3.1.14 Multiproject interface

3.1.14.1 **About the functionality:**

Starting from version 17, **multiproject** interface support has been introduced, allowing you to combine multiple windows as "tabs" for simultaneous work on multiple projects. You can utilize the **multiproject** interface features to attach, detach open projects, and create new projects in separate instances. However, you can still work with **only one window**, without using the multi-project interface feature.

Options for opening an additional window:

Opening an additional window from the current system window:

1. With an open system window, click on the **New Project in a New Tab** button . Hover your cursor over the **New Project**  and select **New Project in a New Tab** from the dropdown list. A new window will appear in the project list.
2. With an open system window, click on the **New Project in a New Tab** button. Hover your cursor over the **Open Project**  and select **Open Project in a New Tab** from the dropdown list.
   Additionally, you can open a project by hovering your cursor over a specific project in the **Open Project**  dropdown list and selecting **Open Project in a New Tab** from the second dropdown list.
3. With the system window open, click on the **New Project with machine setup in new tab** button. Hover your cursor over the **New Project**  and select **New Project with machine setup in new tab**  from the dropdown list. A new window will appear in the

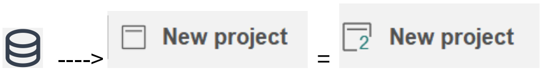
project list.

The overall workflow will look as follows:

Opening an additional window through external systems:

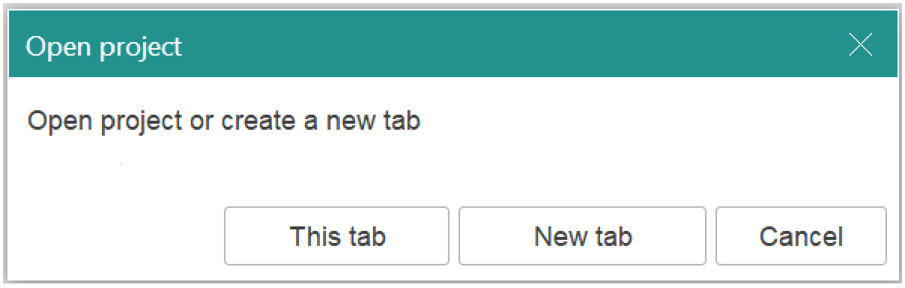
1. With the system window open, click on the **Export from External CAD System** option (in the specific CAD system window).
2. With the system window open, click on the **Export from Machine Maker** option (in the specific Machine Maker window).
3. With the system window open, click on the **Export from Project Library** option.
4. With the system window open, launch a project by double-clicking on the project icon or by dragging and dropping the project using the **drag and drop** function.

The overall workflow will look as follows:



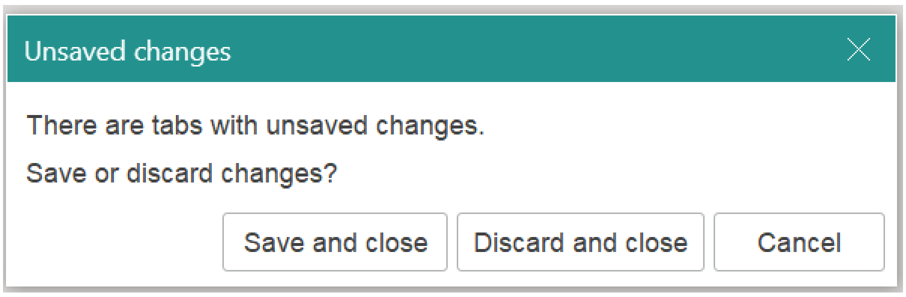
When opening an additional window through external systems, it's important to keep the following in mind:

1. If your window is **empty** (doesn't contain any operations, parts, etc.), it is considered empty, and when importing external files, they will be loaded into this window.
2. If changes have been made in the current window (and it's not empty), the system will provide you with the option to choose into which window the import should be performed.



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

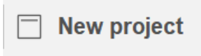
When **exiting** the system while working in multiple windows with unsaved changes, the system will suggest saving them. It will initially prompt to save the project from the current tab, and this will continue until all projects in all tabs are closed or saved.



3.1.14.2 Project navigation:

Working with the project list:

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

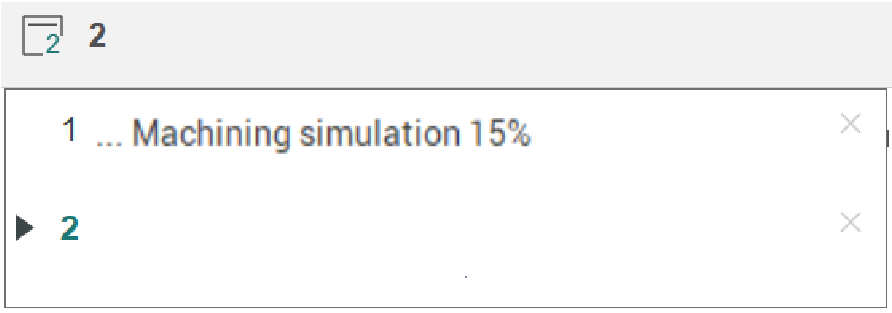


The icon represents the number of projects combined in the current system window.

A circular animated icon appears when a new tab is created and added to existing ones.

If there are more than one tabs present, hovering the mouse cursor over the project name panel for a while will trigger a popup list of tabs. This list allows switching between tabs or closing them.

If a calculation process is running in a tab, a brief description is provided (calculating trajectory, simulation, etc.).

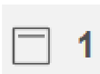


Working with project windows:

To detach a tab, hover the mouse cursor over the **multiproject**  , hold down the left mouse button, and drag it. You can also detach a tab from the list by holding the SHIFT key and dragging the tab.

When you release the button, the tab will detach. If the tab is released over another window of the system, it will attach to that window. Otherwise, it will become a separate window.

The overall workflow will look like this:

 **+ Dragging a tab =** 

and 

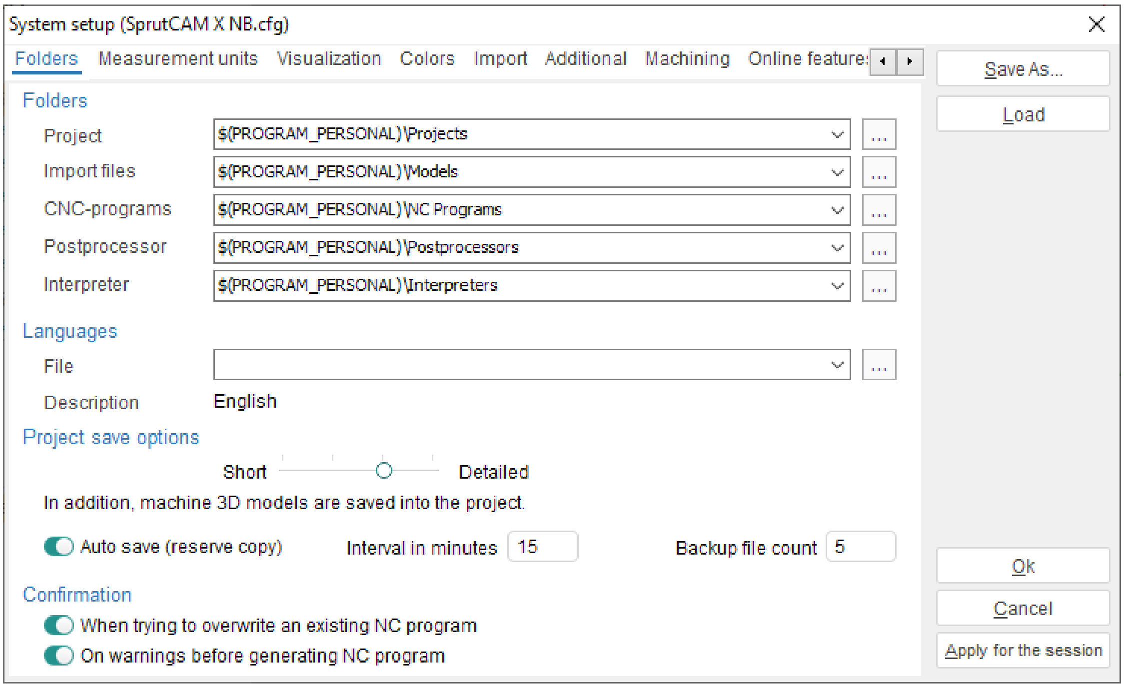
+ Dragging tab 2 onto tab 1 = 

The user can also copy operations between different windows using the keyboard shortcuts 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, and in such a case, the system will generate an error message.

3.2 System settings window

The system settings window can be opened by pressing the  button on the main panel.



All default parameters are saved in the *.cfg file. The user-defined parameters can be either saved or loaded by using the buttons on the right hand side of the window. User can create so many configuration files as he needs.

Buttons of the <System setup> window

- The <Save As...> button quits the window and saves the settings into the user-defined configuration file. The saved file becomes as current and the settings are applied for the current session.
- The <Load> button loads settings from the defined configuration file and make ones current. The file will be used for the next launching as default.
- The <OK> button quits this window with saving of settings into current configuration file and applies the settings for the current work session. The saved settings will be applied for next sessions too.
- The <Cancel> button quits this window and discards all the changes made.
- The <Apply for the session> button quits this window and applies the new parameters for the current work sessions only.

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

The window contains the following tabs:

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

See also:

<Folders> tab

<Measurement units> tab

<Visualization> tab

<Colors> tab

<Import> tab

<Additional> tab

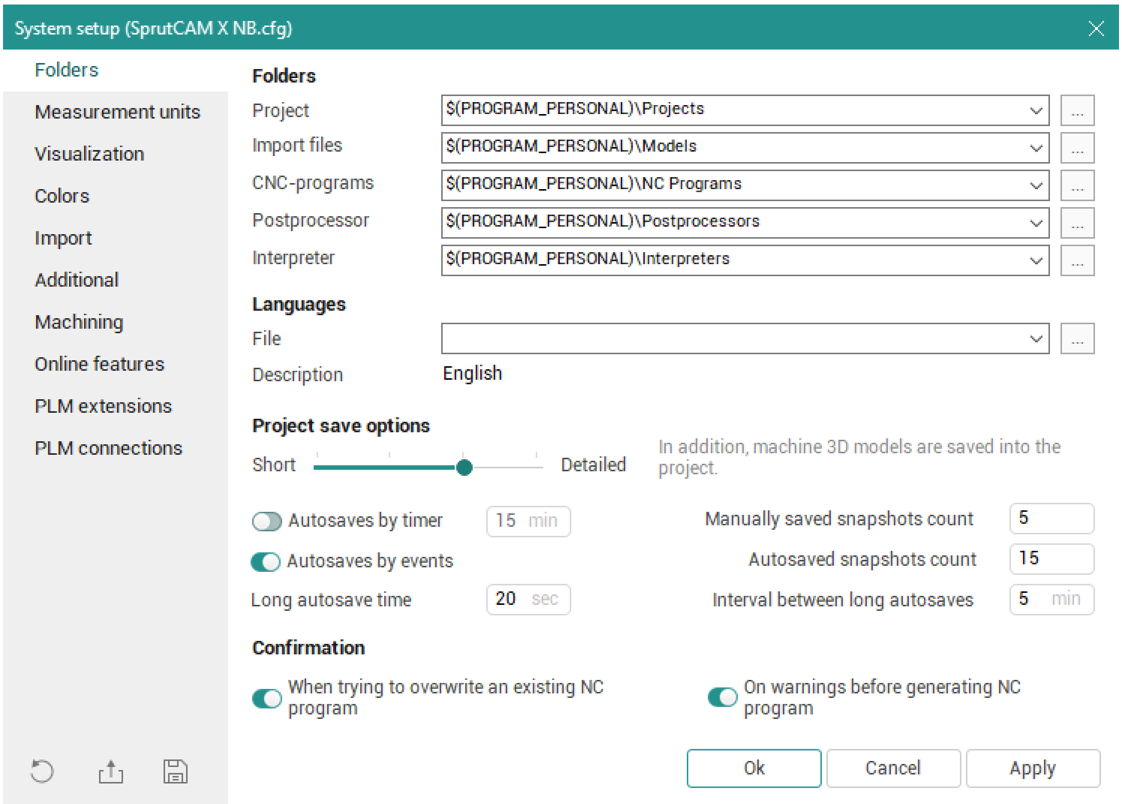
<Machining> tab

<Online features> tab

<PLM extensions> tab

<PLM connections> tab

3.2.1 <Folders> tab



In this window the user can set default paths for the SprutCAM X files.

- <Project> is the default folder for loading and saving projects.
- <Import files> is the default folder for loading geometrical models (IGES, DXF etc.) files.
- <CNC-programs> is the default folder for saving NC-programs generated by the postprocessor.
- <Postprocessor> is the default folder for loading postprocessors files.
- <Interpreter> is the default folder for loading interpreters files.

The paths can be entered manually as well as by using the path selection dialogue, which is accessed by using the ... button.

There are four pre-defined variables, which can be used for defining the corresponding directories (folders):

- <$(SPRUTDIR)> – the directory from where SprutCAM X was launched;
- <$(PRJDIR)> – the directory defined in the <Project> field;
- <$(PROGRAM_PERSONAL)> – the directory that created when installing program in the user's personal documents folder <My documents>.
- <$(PROGRAM_COMMON_DOCUMENTS)> – the directory that created when installing program in the public documents folder on the local computer <Public documents>.

When defining the real names of the directories used during the running of SprutCAM X, the defined variables will be substituted by the appropriate full path used at system start-up or the user defined (edited) settings.

System languages can be change on the <Languages> panel. The language change will be applied on current session.

Changing of the <Project save options> is to manage size of the project files. The project file size will be bigger if more detailed information is saved. Depending on the level of detail of the data stored in the project, there are the following methods, arranged in order from the minimum necessary information stored in the file to the most complete:

1. Source data only (geometrical model, coordinate systems, operations and their parameters) is saved in the project.
2. All data described in the previous paragraph, as well as tool path of operations, are saved.
3. All data described in the previous paragraph, as well as 3D models of the machine schema files and tools, are saved.
4. All data described in the previous paragraph, as well as intermediate workpiece states, are saved.

Autosaves by timer, Autosaves by events, **Manually saved snapshots count, Autosaved snapshots count, Long autosave time**, **Interval between long autosaves** - a group of options that allows you to configure the procedure for creating backup versions of projects. See Autosaves and project snapshots article for more details.

It is possible to switch off a confirmation message during an NC file generation into the existing file on the <Confirmation> panel.

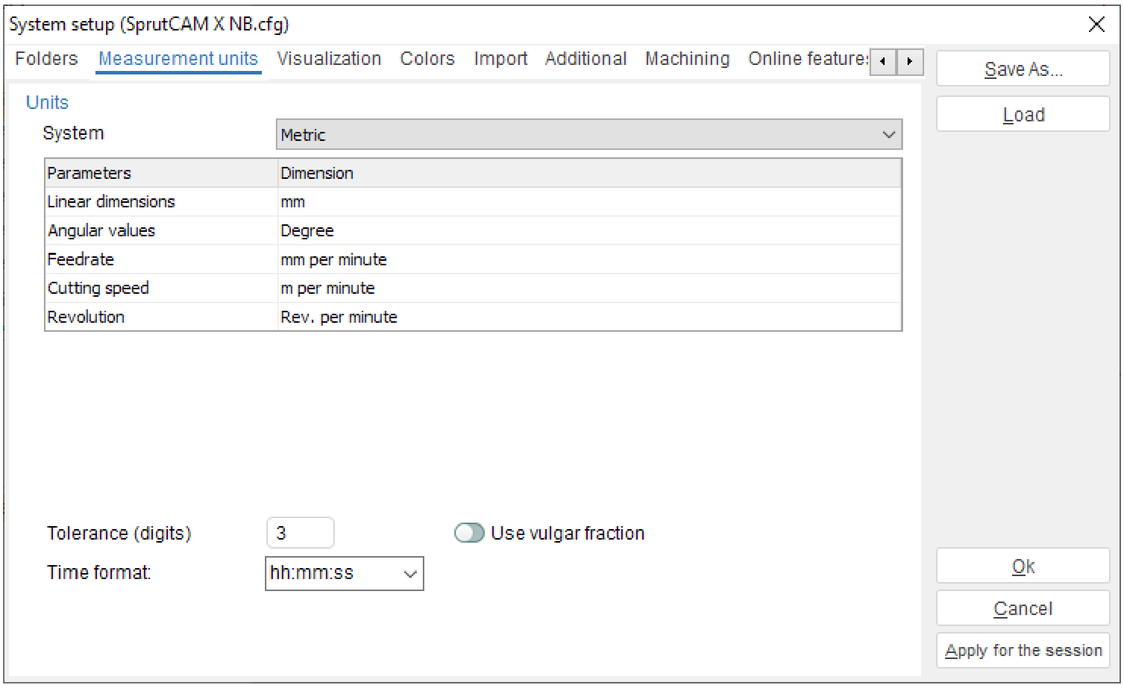
See also:

System settings window

Project recovering

Project snapshots manager

3.2.2 <Measurement units> tab



Allow define measurement units for the system.

Measurements are based on the units used in the imported model. Output data (NC) is created using the same units. Consequently, in order to obtain an NC program for a CNC milling unit in millimeters (inches), all measurements of a model must be in millimeters (inches).

Angular measurements are given in degrees with decimals.

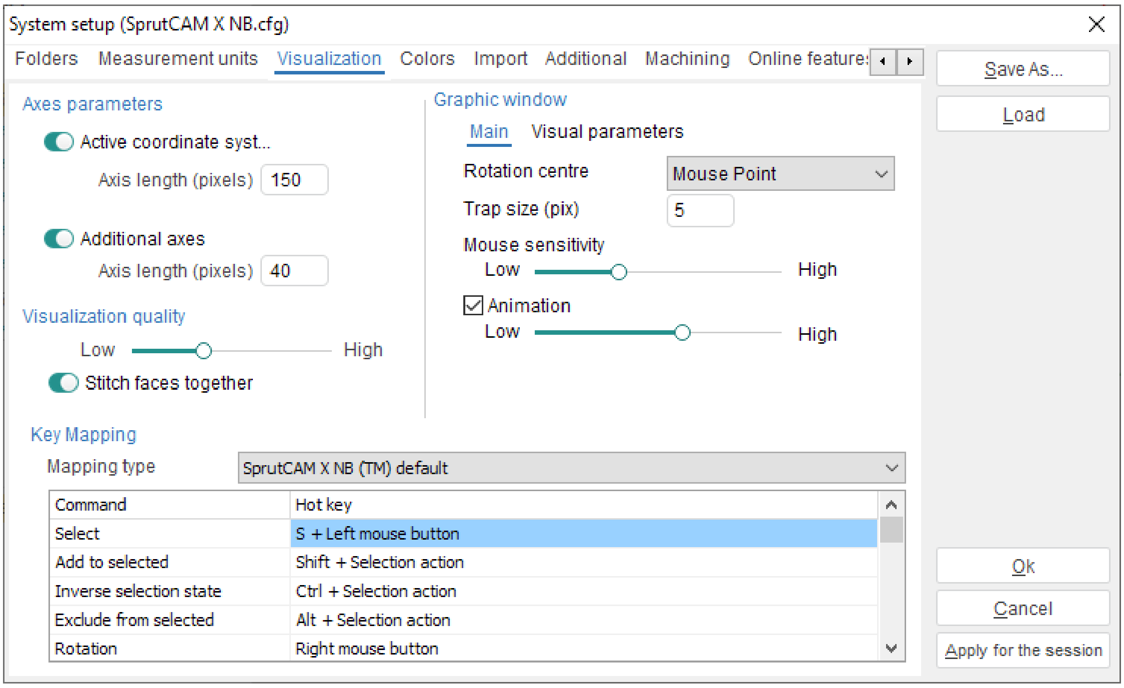
The <Default parameters> panel defines a file with description of initial parameters for new operations. The <Edit> button opens a window for the file editing.

- The <Tolerance (digits)> value defines digits count after decimal separator for values which are outputted into the NC-program. It is recommended to set the value to equal or greater for one than the maximal NC-machine tolerance (or digits count after decimal separator for the NC-program).
- The <Time format> field defines a format for the time output (for the machining time for example).
- The <Use vulgar fraction> – using in system vulgar fraction.

See also:

System settings window

3.2.3 <Visualization> tab



There are control means of graphical window settings.

Visibility of coordinate systems and their axes length can be changed on the <Axes parameters> panel. The <Active coordinate system> is drawn on the real space but <Addition axes> of the coordinate system are drawn on left button side of the graphical window.

On the <Visualization quality> panel it is possible to assign the default visual quality of objects. Moving the slider in the bottom part of the window will alter the visual accuracy. The higher visualization quality requires more computer resources. The accuracy value will be applied as default for all newly imported objects.

Note: *It is not recommend to setting high accuracy for lower specification computers due to possible negative effects on speed.*

The <Use simple OpenGL objects only> option is switched off as default. The option musts be switched on for rare cases only when surfaces can not be drawn right because of the display adapter or its driver does not support full commands list of OpenGL.

The <**Stitch faces together**> option allows, when triangulating surfaces for visualization, to take into account the stitching, so that there are no gaps between the surfaces.

Following parameters can be defined on the <Graphic window> panel:

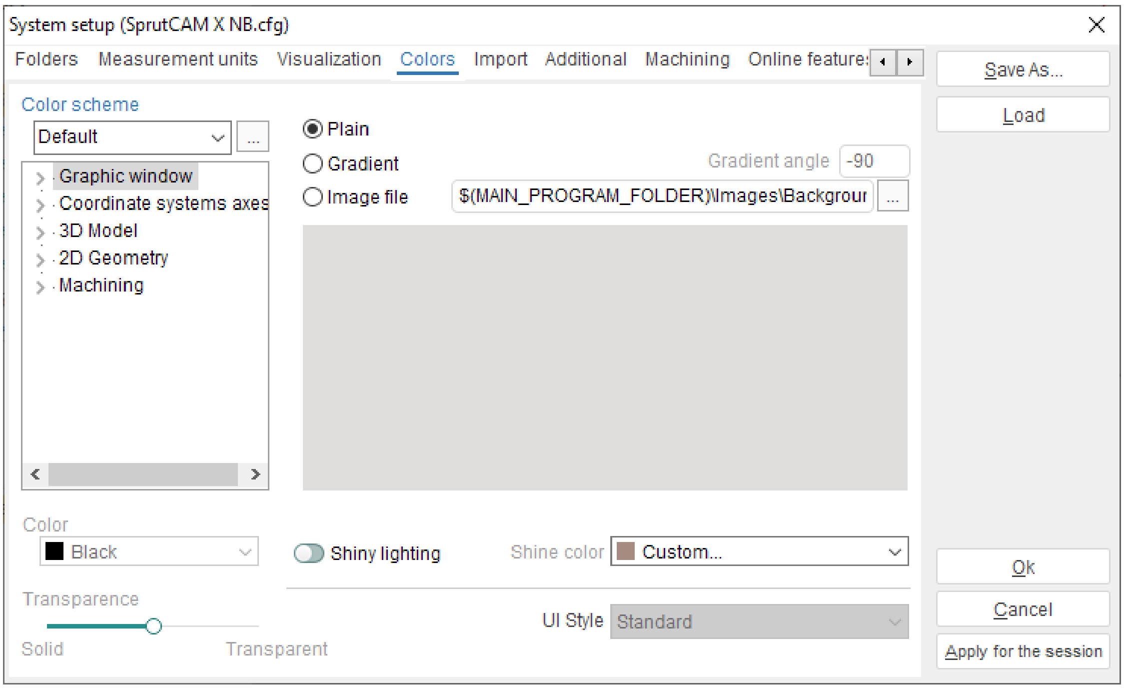
- <**Main**>
  - <Rotation centre> is the point of rotation centre of the graphical window.
  - <Trap size (pix)> is the trap size in pixels for the objects selection and highlighting.
  - <Mouse sensitivity> defines the rotation and scaling speed by mouse.
  - <Animation> assigns count of intermediate pictures when view parameters of the graphical window are instantly changed.
  - <Perspective> defines the objects distortion extent by perspective during visualization.
- <**Visual parameters**>
  - <**Rendering mode**> has three options:
    - <**Hardware Advanced**> - shader rendering
    - <**Hardware Standard**> - hardware accelerated classic rendering with OpenGL without shaders
    - <**Software (very slow)**> - software rendering with OpenGL without using a video card
  - <**Ambient occlusion**> - objects shading

<Key Mapping> area defines the hot keys for the visualization control. SprutCAM X has four different key mapping schemes. It is created for the convenience of users which works with other systems. On panel it is possible to change yey mapping scheme only. Hot key is read only.

See also:

System settings window

3.2.4 <Colors> tab



In the < Color scheme > field it is possible to select one of the established color schemes. By pressing the ... button it is possible to return values of colors to the schemes, installed by default, or, if the color scheme < Another > is selected to load in this scheme of a value of one of other color schemes.

Under the color scheme, as tree-like structure, values of color of a separate element are shown. These elements are broken on groups: < Graphic window >, < Coordinate systems axes >, <**3D** Model >, < 2D Geometry >, < Machining >. Having opened the necessary group, it is possible to edit values of color of any element of group. Color of a flowing element is assigned in a field < Color > by a choice of the necessary value from the falling out list. In a field < Mode > is established a condition of a featuring of an element: with a shade or wire. Migration of a slider < Transparence > it is possible to install a transparency of a flowing element.

There is can be selected background of graphic window:

- < Plain > – permanent background. Colour is assigned in < Graphic window > –> < First background color >.
- < Gradient > – gradient drawing of a graphic window. Colors of a drawing are assigned in group < Graphic window >: < First background color > and < Second background color >. In a field < Gradient angle > it is possible to set a gradient angle. The angle is set concerning a vertical axis.
- < Image file > – using graphic image as wallpaper. Available formats is < BMP > and < JPEG >.

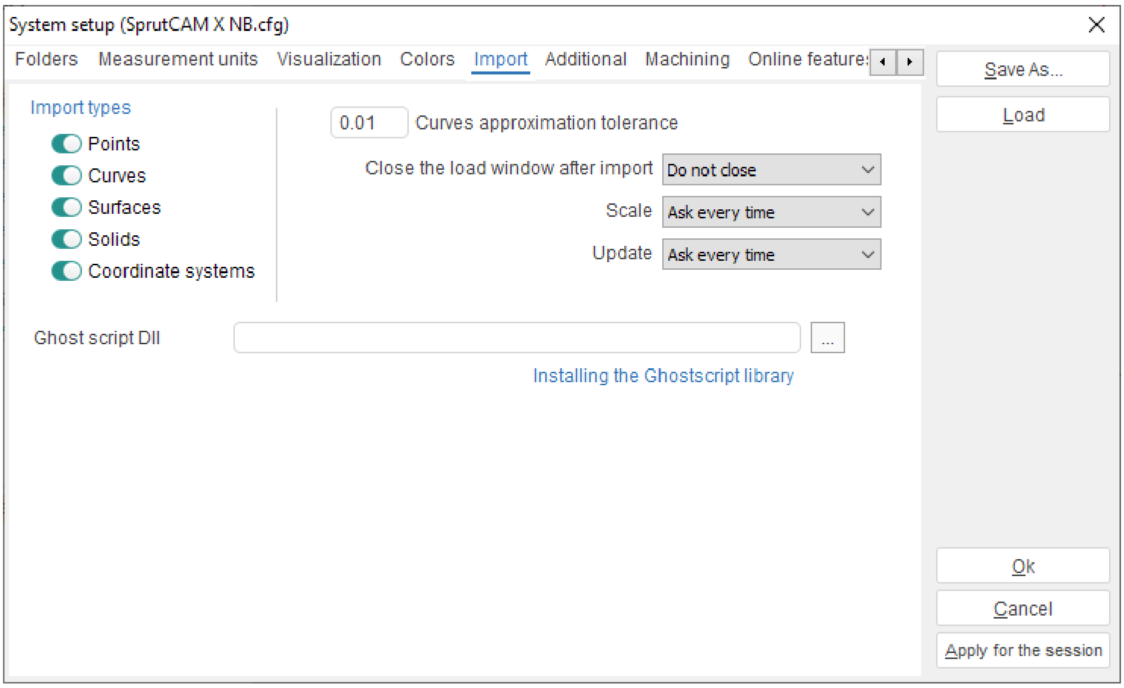
<**Shiny l** ighting > is enable additional light source.

See also:

System settings window

3.2.5 <Import> tab

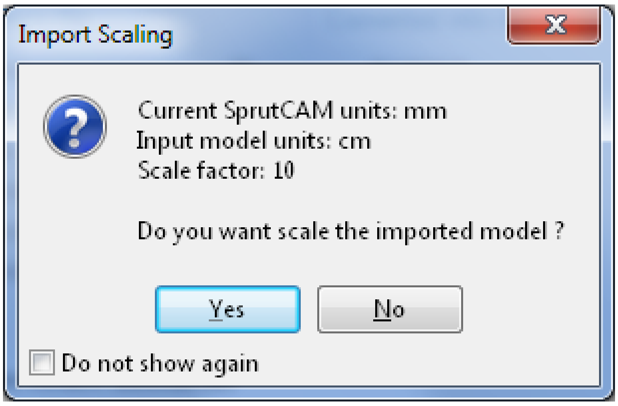
Here the general import parameters can be set.



In the <Import types> objects group, the user can define the types of geometrical objects, which can be imported from the geometrical data exchange (CAD) files. If the box opposite a specific type isn't selected, then the corresponding object will be ignored during the import process.

If during import a curve needs to be transformed, then approximation will be performed using the value specified in the <Curves approximation tolerance> **window.**

The <**Scale**> option is used to control the appearance of "Import scaling" window:



The <**Update**> option is used to control the appearance of "Model files update manager" window.

<**Ghost script dll**> is used for the PostScript files import.

See also:

System settings window

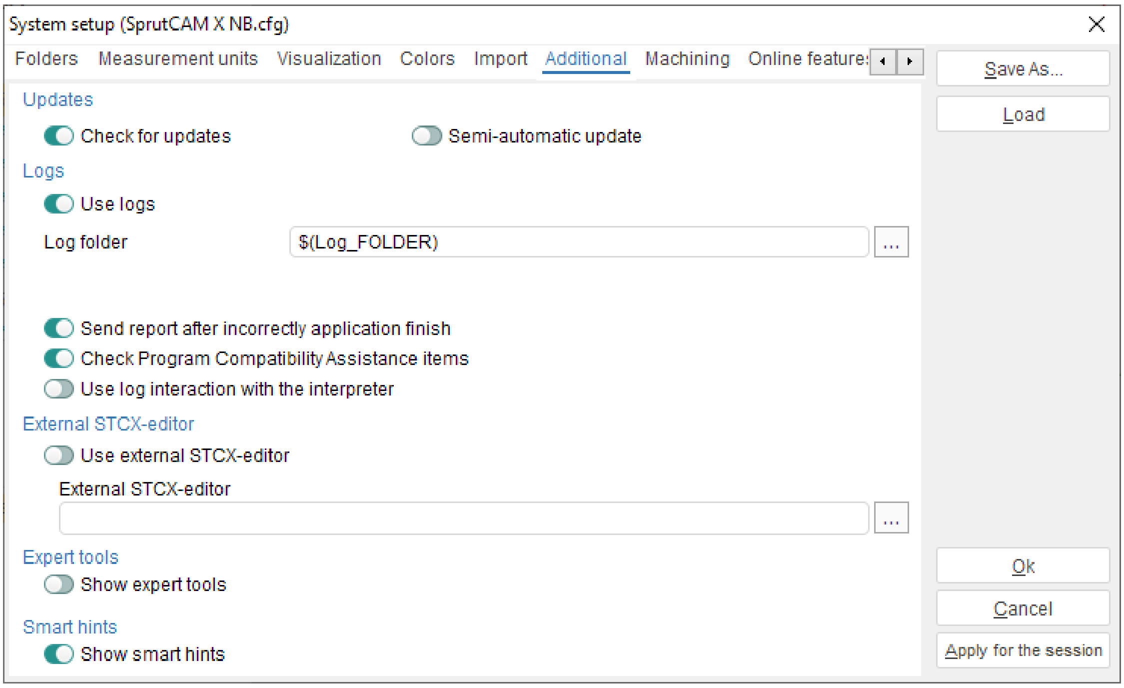
Importing objects from PostScript files

Geometrical model structure updating

3.2.6 <Additional> tab

Use < Additional > page to setup SprutCAM X events logging and SprutCAM X updating settings.

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.



To enable updates checking switch on < **Check for updates** > option.

Switch on < **Semi-automatic update** > to start downloading updates right after click the "Updates found" notification.

To enable logging check the < Use logs > option.

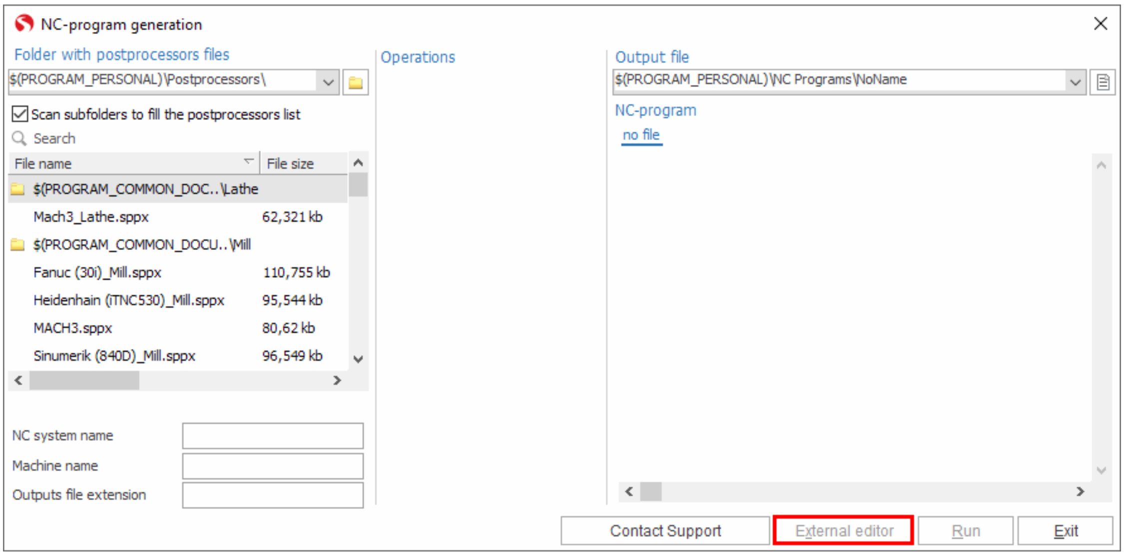
Enter the folder path in which you want SprutCAM X to save the logs into the < Log folder > field.

If you want SprutCAM X to send error reports automatically check the < Send report after incorrectly application finish > option.

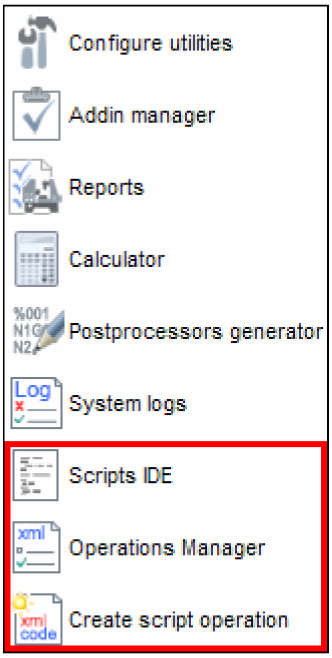
The <**Check Program Compatibility Assistance items**> option is used to exclude the SprutCAM X from Program Compatibility Assistant.

To log the interaction of SprutCAM X with the interpreter in G-code based simulation mode and G-code based milling operation use the option <**Use log interaction with the interpreter**>.

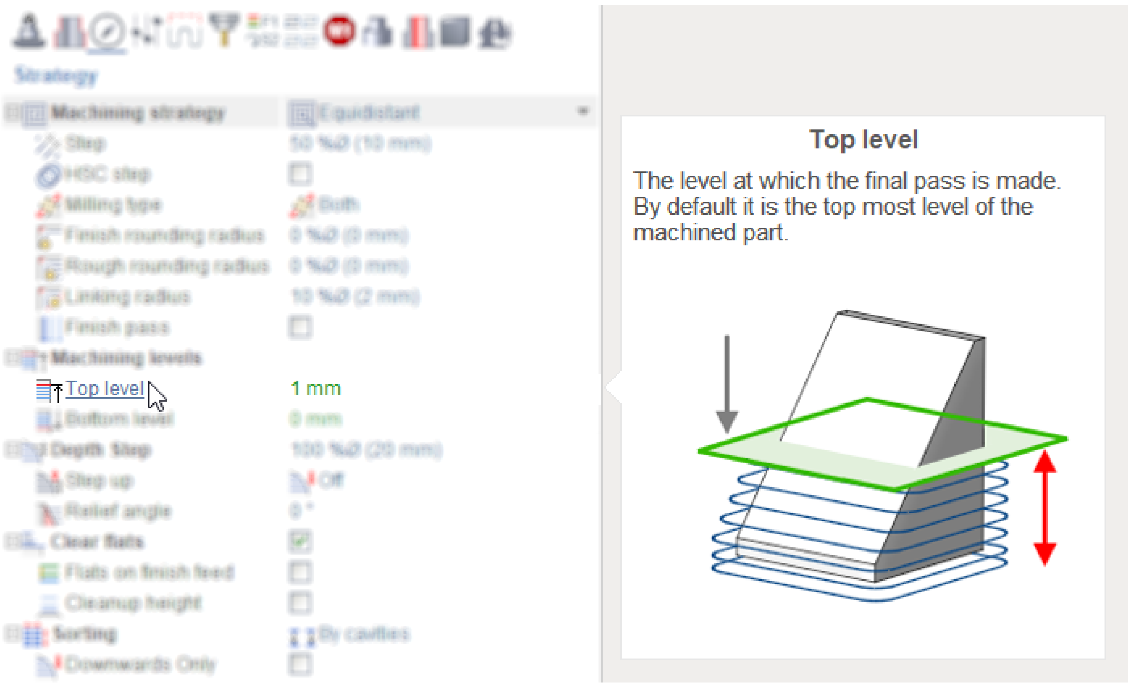
The <**External STCX-editor**> panel is affects "NC-program generation" window and allows you to select an external editor to edit the NC-program.



The <**Show expert tools**> option is used to make some tools visible, that will be useful to expert users.



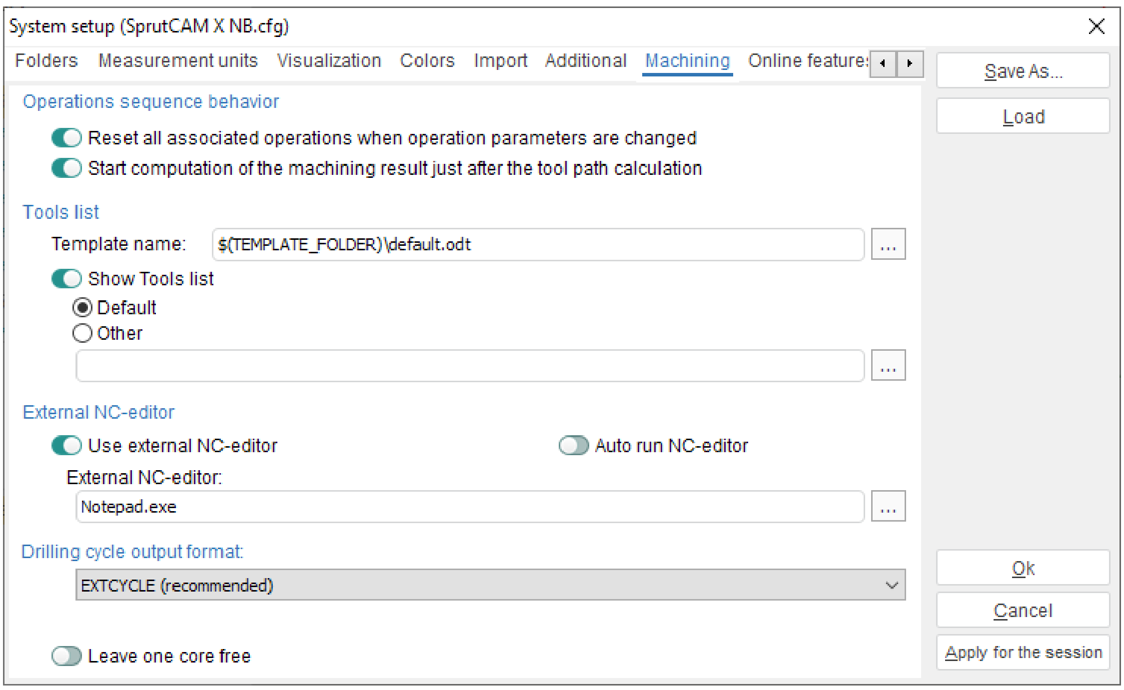
The <**Show smart hints**> option enables/disables the visibility of smart hints.



See also:

System settings window

3.2.7 <Machining> tab



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

Tools list can be created in HTML format.

When creating a tools list the system uses templates. Several templates are included into the installation set, from which it is possible to choose the required one. One can also create new templates, knowing HTML language is required. For more information contact the support desk.

When created in HTML document can be saved immediately, or viewed and corrected if necessary. To save a tools list without viewing, deselect <Show tools list>.

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

With the help of the <**Operations sequence behavior**> panel you can control how the operations will behave when changing the status to calculated / reset:

- The <**Reset all associated operations when operation parameters are changed**> option allows you to automatically reset the tool path of the operations when changing the input data that comes from previous operations of the technological process.
- The <**Start computation of the machining result just after the tool path calculation**> option allows you to perform a simulation in the background immediately after the calculation of the tool path.

<**External NC-editor panel**> is used to specify external application for viewing NC program. The system recognizes certain command line parameters in the command line:

- <$NCOUTPUTNAME$> – the name of the NC-program file that was created by the postprocessor,
- <$CD.<ItemName>$> – keywords of this kind will be replaced by appropriate item value of the custom data.

If the <**Autorun NC-editor**> checkbox is checked the specified NC-editor application would be automatically launched directly after the postprocessor generates the NC-program.

For compatibility with older versions of postprocessors the system provides the ability to change the output format of the drilling cycle (when not expanded toolpath output method is used for hole machining operation). Parameter with the same name can have the following values.

- <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 processing routine is not implemented.

The <**Lock Simulation Kernel**> option allows to simulate without removing the material. The tool moves along the path, but no material is removed. The option can be used on slow computers.

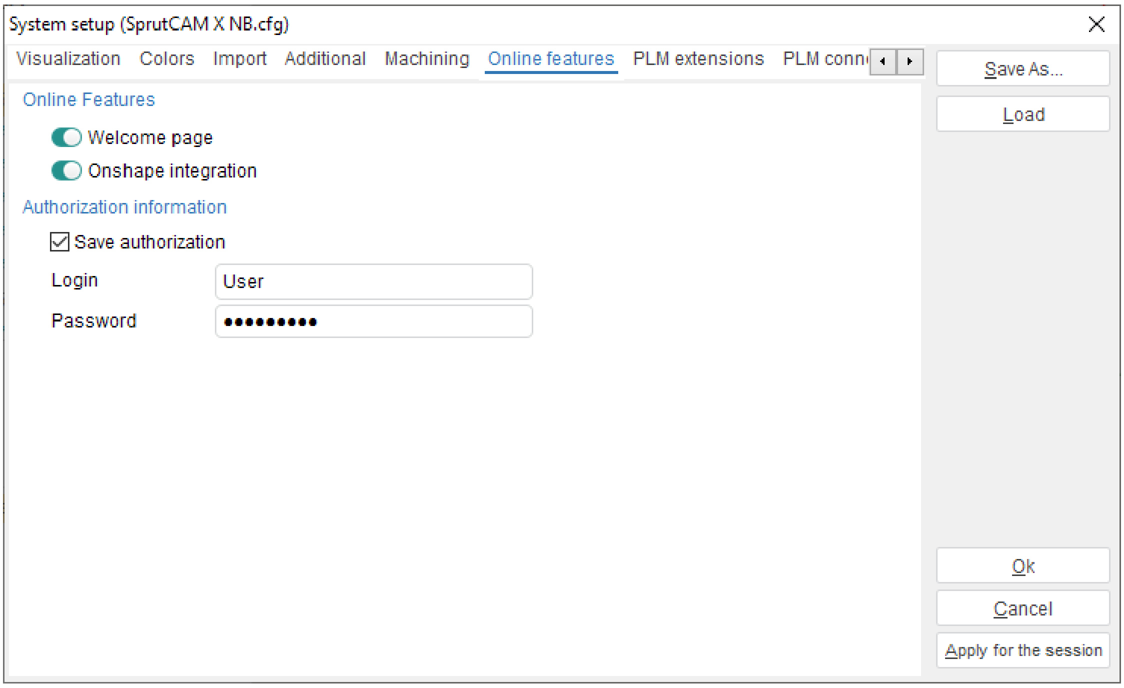
The <**Leave one core free**> option allows to leave one core free in multi-core computing (restart required).

See also:

System settings window

3.2.8 <Online features> tab

This tab is used to manage online features.



This <**Welcome page**> option sets the visibility of the welcome page at system startup.

The <**Onshape Integration**> option sets the visibility of the "Import from Onshape" menu item.

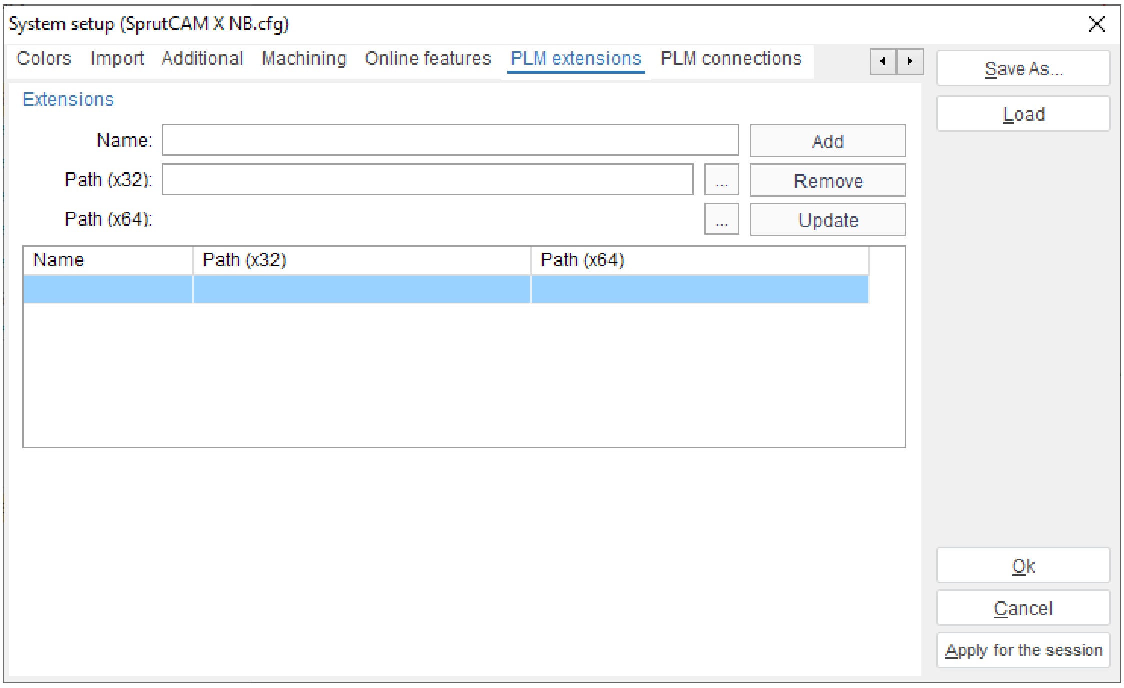
The <**Authorization information**> panel is required to save authorization information.

See also:

System settings window

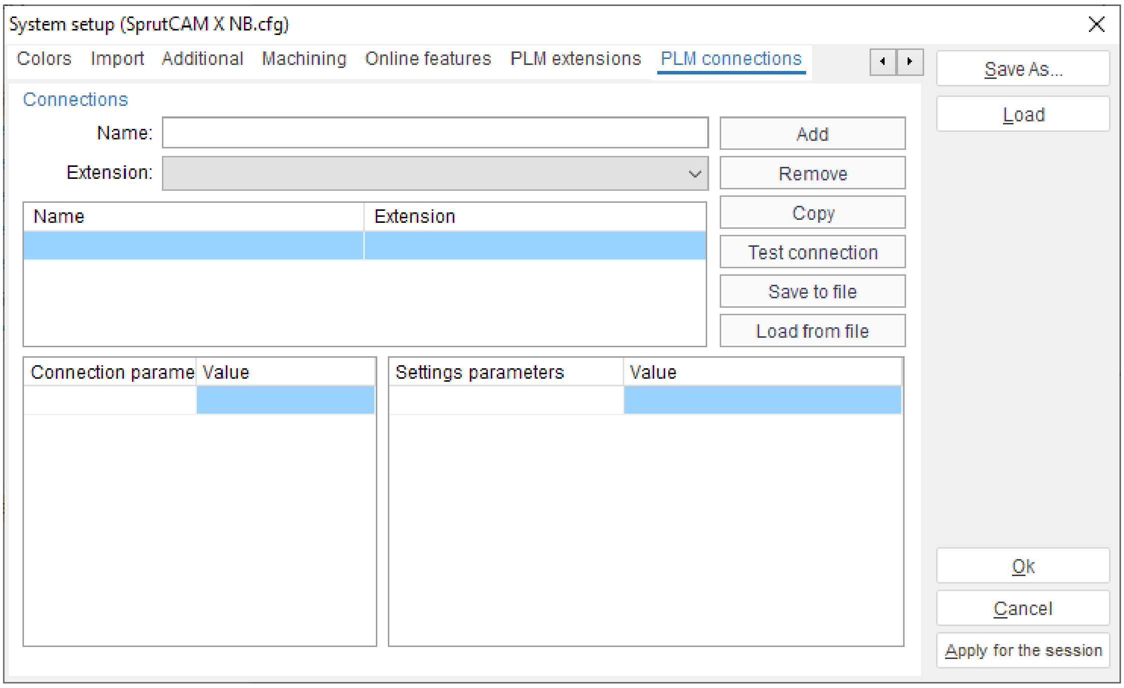
3.2.9 <PLM extensions> tab

This tab is used to configure PLM extensions for the Teamcenter PLM system. Features of configuration are described on the PLM extension setup. The Teamcenter PLM system integration module description is located at this Teamcenter PLM Integration Module.



3.2.10 <PLM connections> tab

This tab is used to configure PLM connections for the Teamcenter PLM system. Features of configuration are described on the Настройка PLM-соединения. The Teamcenter PLM system integration module description is located at this Teamcenter PLM Integration Module.



3.3 Exchange files

No content in this page. See child topics

3.3.1 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.

3.3.2 Importable files

In SprutCAM X 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 SprutCAM X with different CAD-systems. Add-ins permit generally SprutCAM X 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 SprutCAM X using any of the supported formats. SprutCAM X 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 SprutCAM X 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.

3.3.3 DXF export

The geometrical objects of SprutCAM X 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 SprutCAM X. Contours of 2D geometry are saved in the XY-plane irrespective of the real orientation of the local coordinate systems.

3.3.4 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 to legacy SprutCAM version 8.1 have 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 SprutCAM X 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.

3.3.5 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 SprutCAM X is installed, to the CNC unit can be performed by any available method.

3.3.6 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 G-code based simulation mode is enabled and in G-code based operations.

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

See also:

Creating your own interpreter

3.3.7 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.

3.3.8 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.

3.3.9 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.

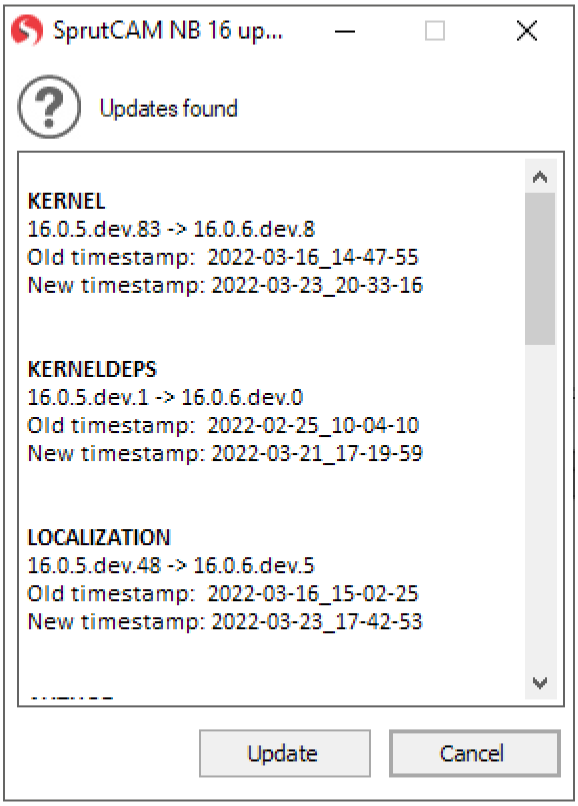
To control the machine setup files, use the "Save as a machine setup", "Import from the machine setup file" and "New project from the machine setup file" on the main panel.

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

3.4 Updating

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

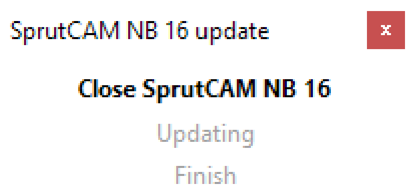
When updating systems find and successfully download modules, an updating prompt will be shown:



Press the <Update> button to run the updating process.

Updating process

The SprutCAM X updating process includes three steps:



- The <Close SprutCAM X> — system waits while the user saves all projects and closes SprutCAM X manually. SprutCAM X 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. SprutCAM X can ask an administrative account to complete this step.
- The <Finish> — system deletes all temporary data, used for update.

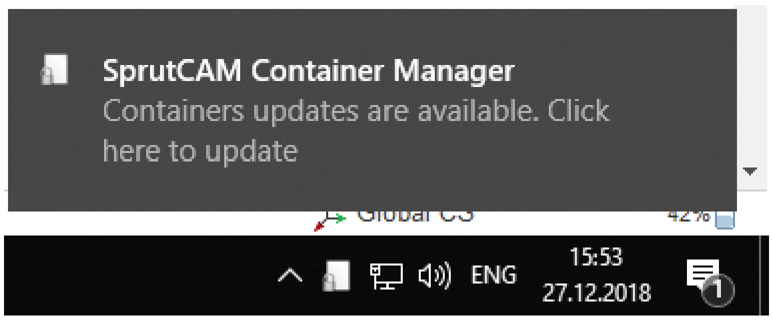
After these steps, the updating window will be closed and SprutCAM X will be started automatically.

3.5 Container manager

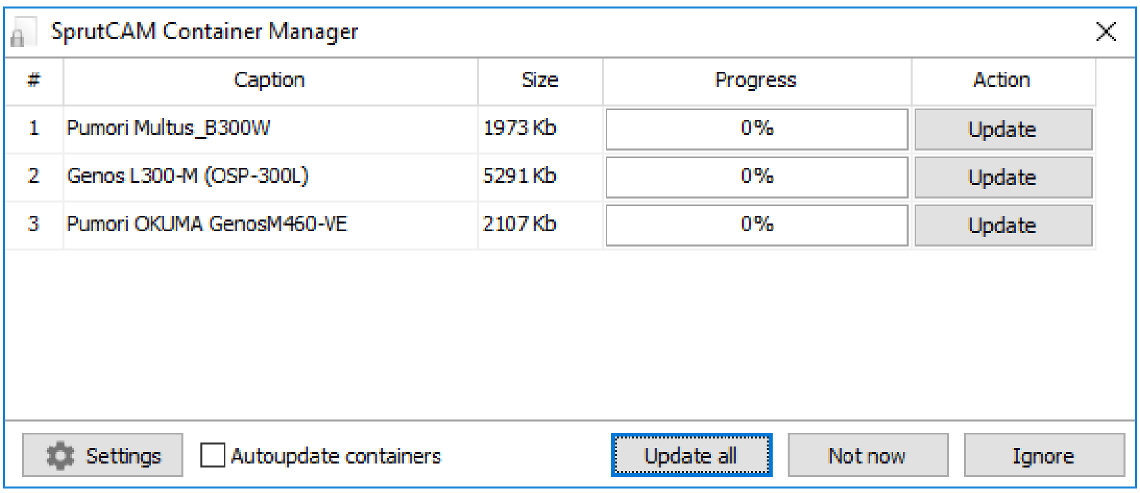
A container manager has been added to SprutCAM X. It automatically downloads and updates container files (.stfc), assuming that licenses for the respective containers are received.

Checking of the uninstalled containers and updates for container files occurs each time SprutCAM X is launched and while container license installation through license manager.

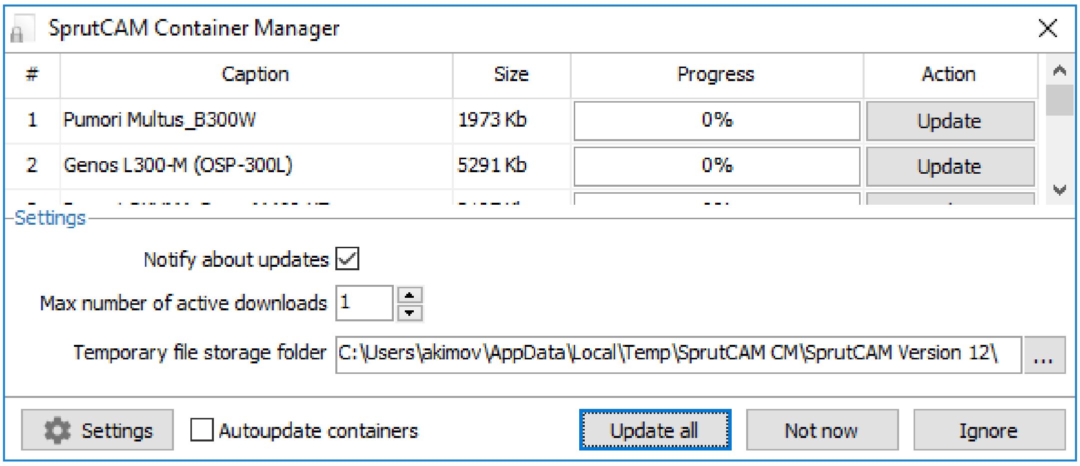
If there are not installed containers or updates, you will be offered to download them in a pop-up message:



When you click on a message or an icon in the tray, window with a list of containers that can be downloaded/updated appears:

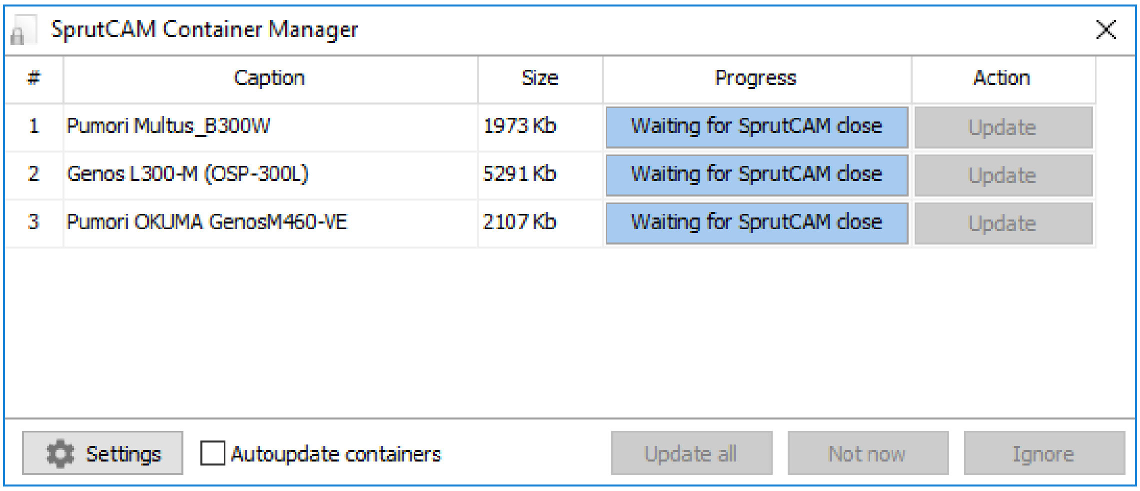


- The <**Update**> button starts the process of downloading and updating the corresponding container.
- The <**Update all**> button starts the process downloading and updating of all containers in the list.
- The <**Not now**> button cancels the containers downloading/updating and minimizes the container manager to tray. The system prompts you to update next time.
- The <**Ignore**> button cancels the containers downloading/updating and minimizes the container manager to tray. The system will no longer show a message about the availability of updates until the option <**Notify about updates**> is enabled on the container manager settings panel.
- The <**Autoupdate containers**> flag sets the mode of automatic containers downloading/ updating without notifying the user.
- The <**Settings**> button displays/hides the container manager settings panel:



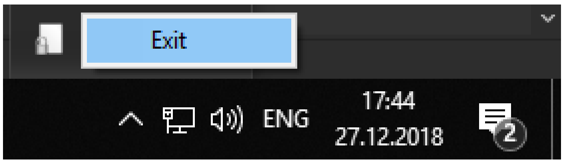
- The <**Notify about updates**> checkbox enables/disables user notifications about container updates. When you click on the <**Ignore**> button, the flag is removed.
- Field <**Max number of active downloads**> sets the number of simultaneous container downloads. If you have a slow Internet connection, 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 starts, the container files will be saved to the temporary directory specified in the settings panel. After the download process is complete, the container manager will wait for SprutCAM X to close in order to move the downloaded files to the SprutCAM X container storage directory (by default: C:\ProgramData\SprutCAM Tech\SprutCAM X\Version 16\Containers\).



At the end of the work, the SprutCAM X container manager will close automatically.

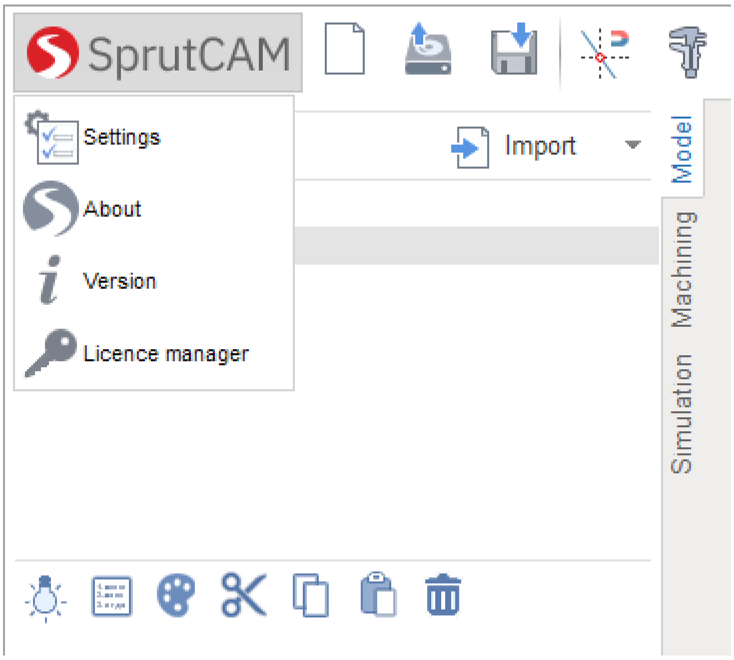
To force closing of the container manager, right-click on the tray program icon and select <Close> in the context menu:



In the case of the forced closure of the application, all active downloads will be terminated, previously downloaded containers will not be moved to the directory for storing SprutCAM X containers. The next time you start, you will need to re-download the containers, whose update was interrupted.

3.6 Licence manager

The license manager contains functions for working with licenses for SprutCAM X and licenses for its modules.



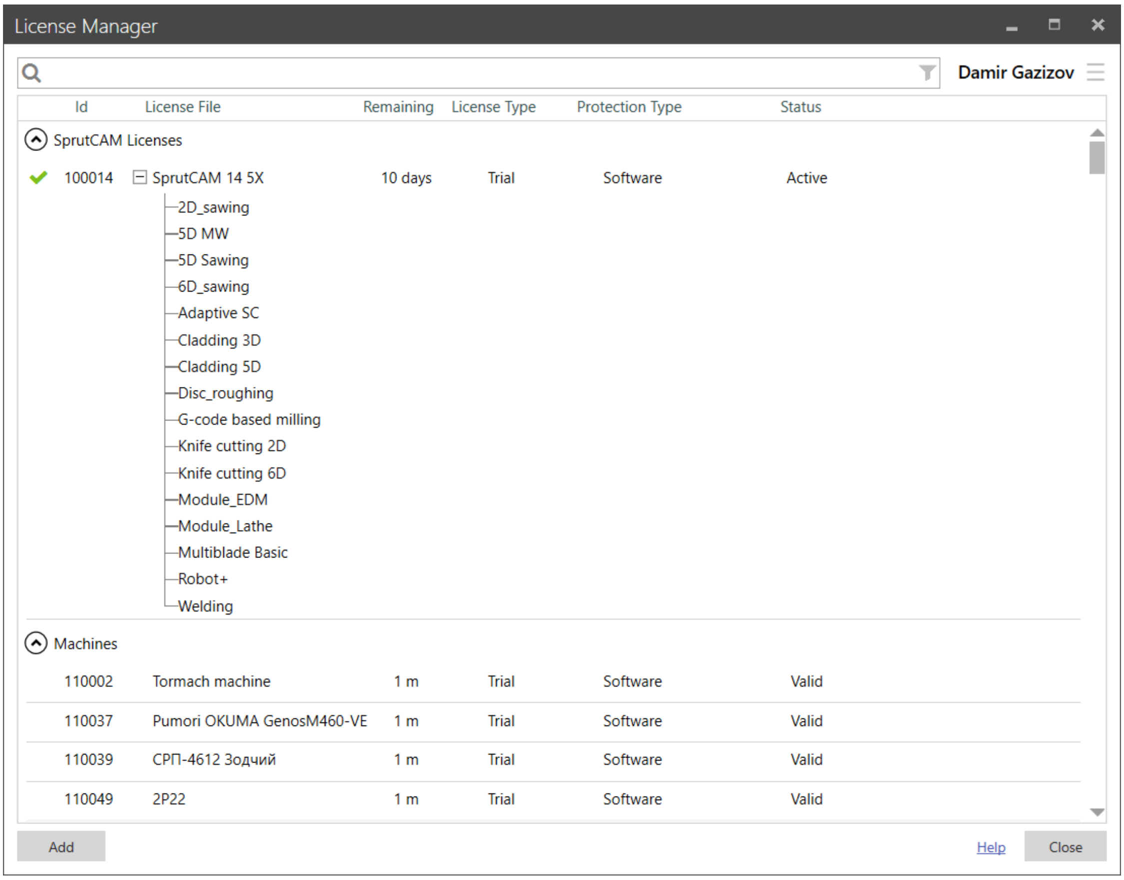
To start the manager, use the <**License Manage**r> item in the drop-down list of the main menu.

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

Each SprutCAM X 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 SprutCAM X dealer.

The main manager window is presented below:



The upper part of the window is occupied by the list of SprutCAM X 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) SprutCAM X licenses.

2) Postprocessors.

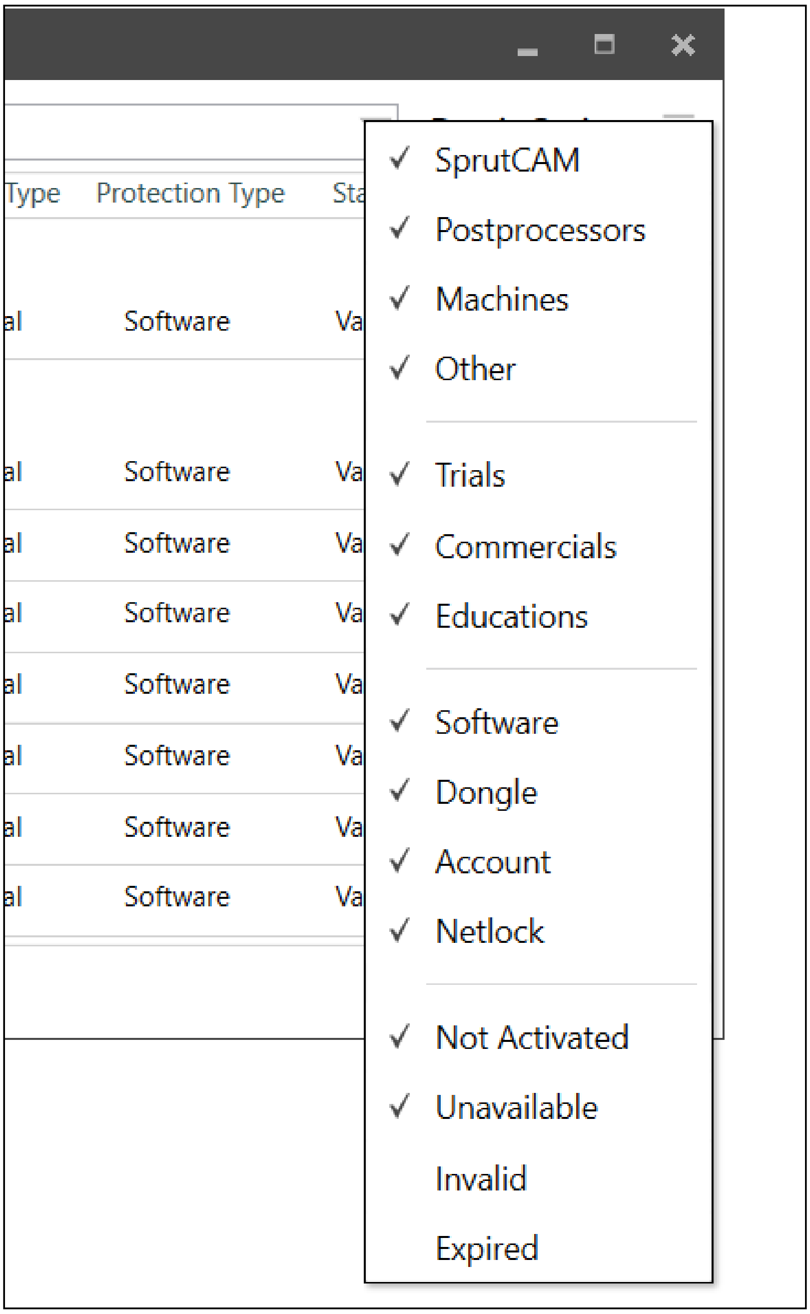
3) Machines

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)



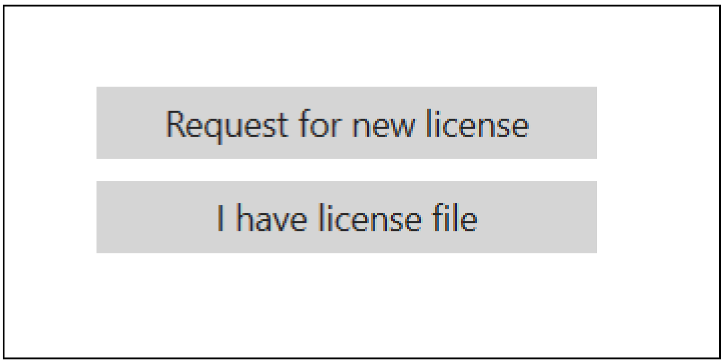
When you hover over the line with the license, the area with 3 buttons is shown on the right:

<**Activate**> - Performs license activation.

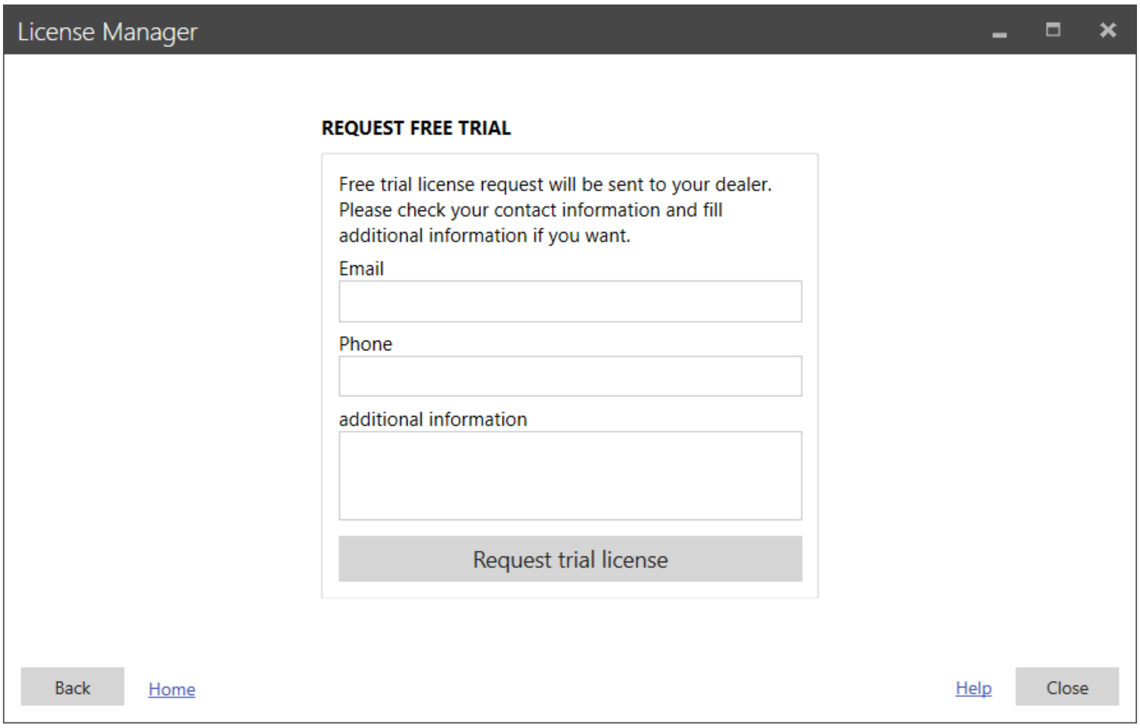
<**Refresh**> - Updates information on the current license.

<**Deactivate**> - Deactivate the current license.

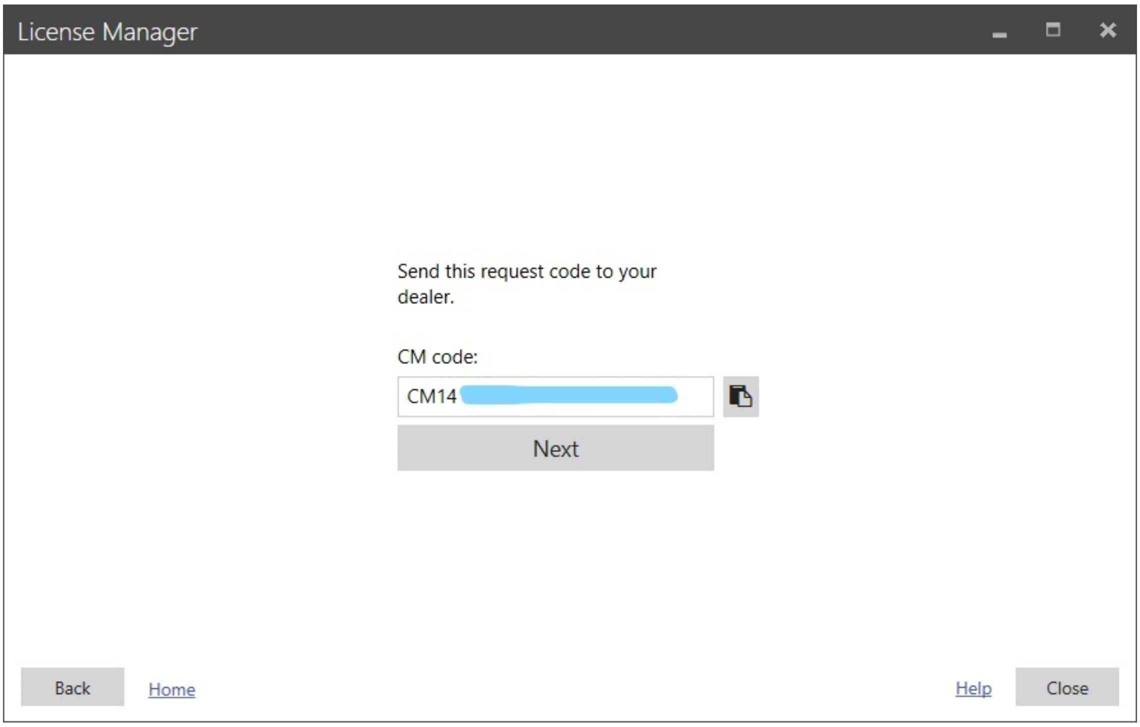
At the bottom of the window on the left is a button <**Add**>, by clicking opens this page:



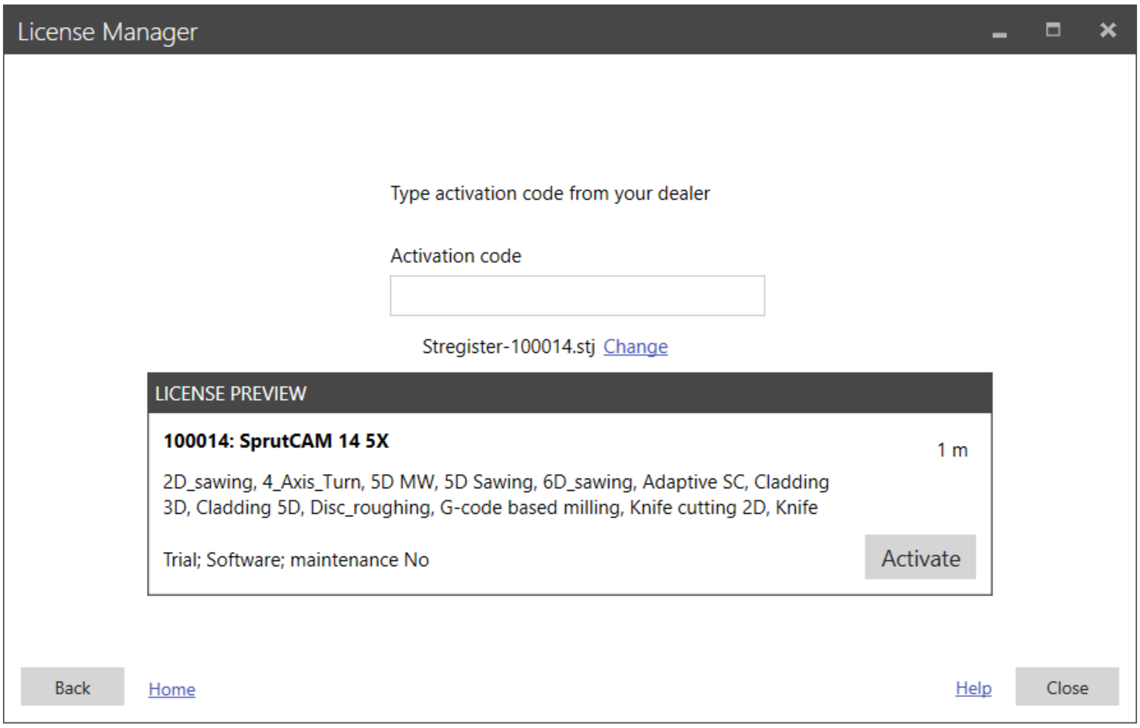
<**Request for new license**> - Opens a license request window



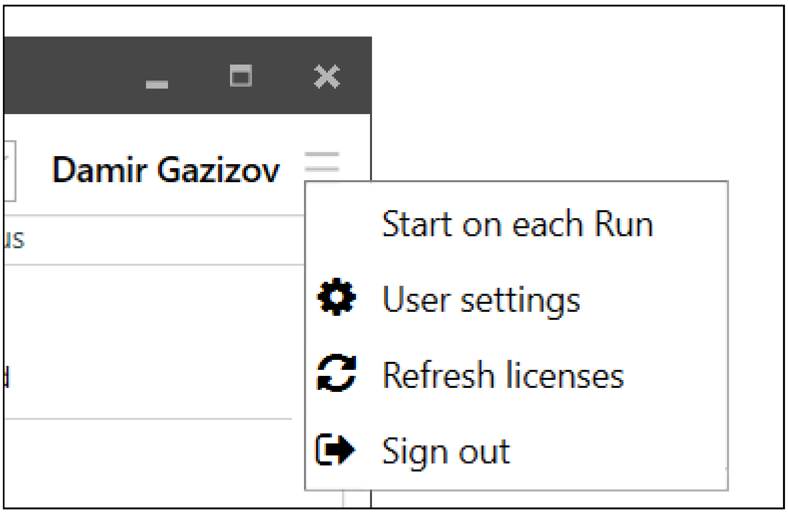
If you do not have an Internet connection, when you click the <Request a new license> button, a window opens with the ability to receive a CM code to activate an offline license:



<**I have license file**> - Open the license file selection dialog. This menu item is useful when installing SprutCAM X on a clean computer using an online installer.



In the upper right part of the window there is a drop-down menu with additional functions:



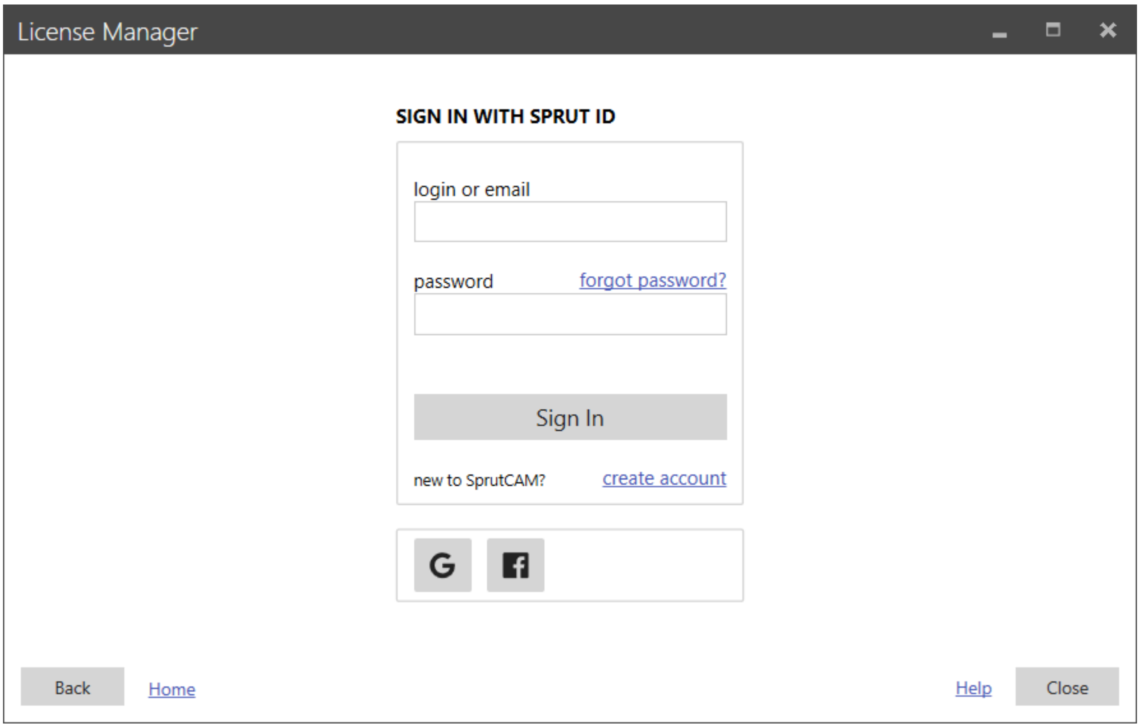
<**Start on each Run**> - Turns on/off the start of the license manager every time SprutCAM X starts.

<**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

Below is the login page:



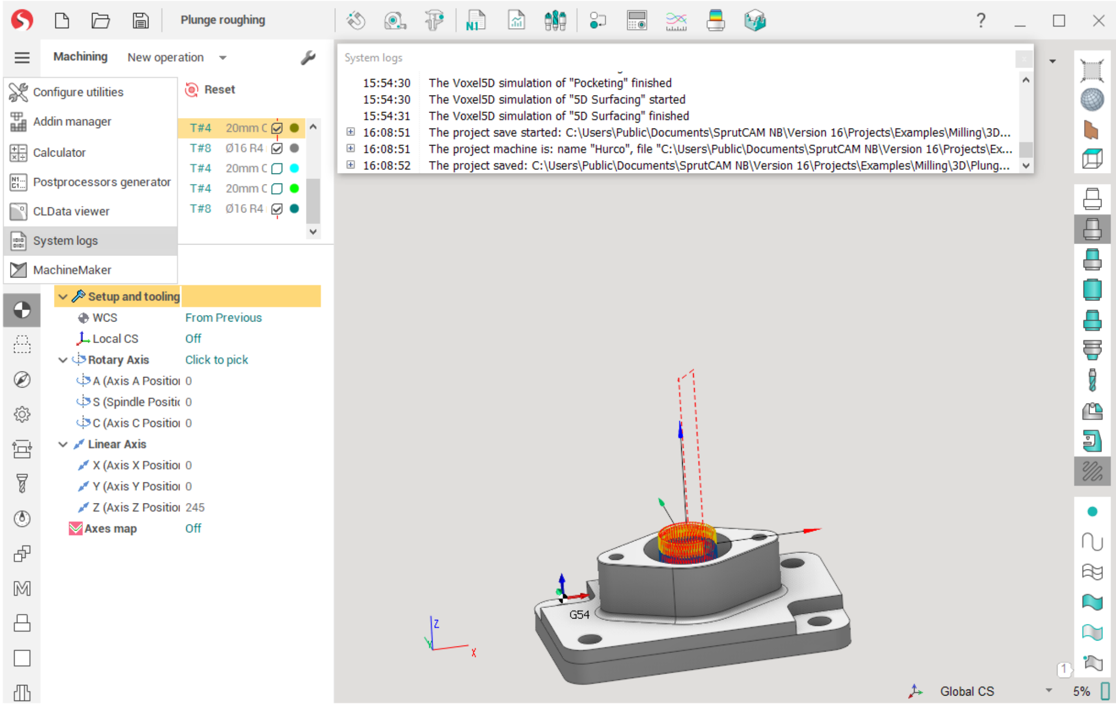
The page allows you to: enter your personal account, restore a forgotten password, register a new user, as well as the possibility of authorization through social networks.

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

3.7 System logs

There is a possibility to view a list of the main events that occurred in the application, starting from the moment of launch.

The events are displayed in a separate window, which can be called up from the Utilities/System logs menu.



Each event contains the time of occurrence and a description. If the description does not fit on one line, then at the beginning of the row the "Expand" button is displayed. Depending on the type of event, it can be drawn in different colors - red for errors, yellow for warnings and white for normal behavior.

This window can be useful, for example, in case of problems. Here you can track the details and try to understand the possible reasons.

4 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;

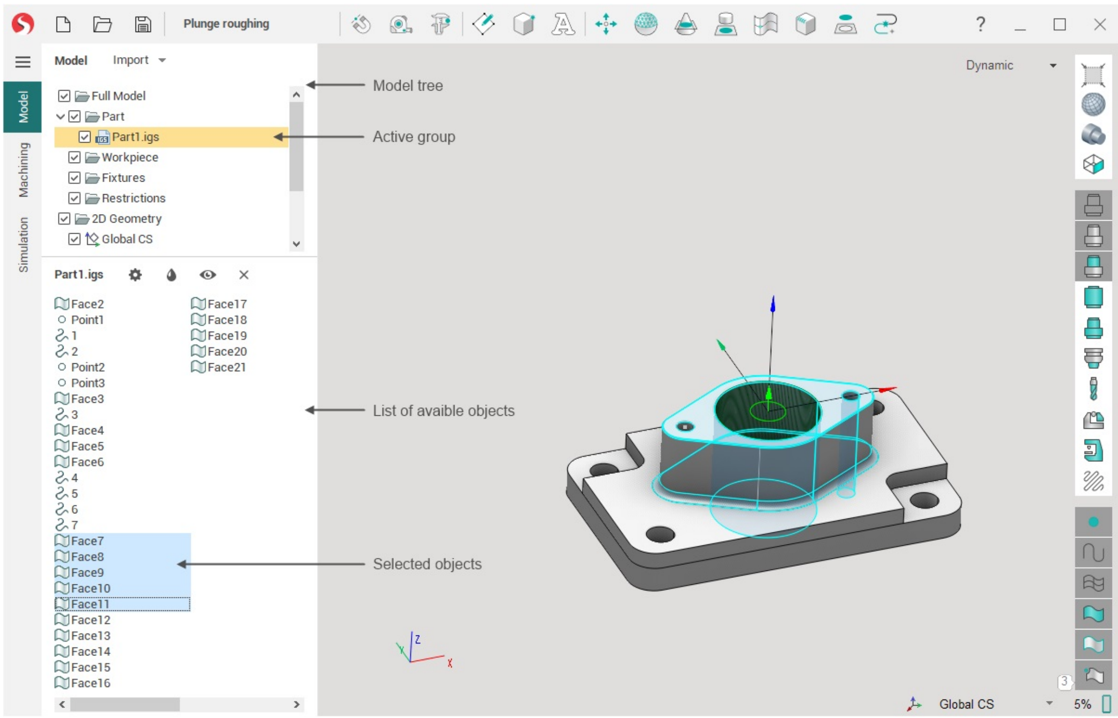
Access to elements of the model is performed both from the model tree window and interactively on the screen. Different functions can be launched via the pop up menu in the graphical window and the model tree window. Buttons for frequently used functions are put on the toolbar of the <Model> tab.

See also:

Geometrical model structure

Geometrical objects import

4.1 Geometrical model structure



A geometrical model in SprutCAM X 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.

See also:

Geometrical model preparation

Faces, meshes, curves and points

Groups (folders)

Active folder

Geometrical model structure window

Object selection

Intellectual object selection

Geometrical model structure editing

4.1.1 Geometrical objects types

No content in this page. See child topics

4.1.1.1 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.

See also:

Geometrical model structure

4.1.1.2 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.

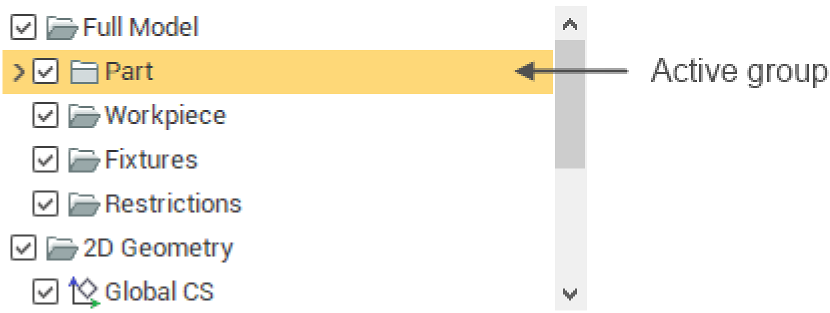
Note: When importing, groups can be automatically placed in special folders, if you add one of the following keys to the name:

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

See also:

Geometrical model structure

4.1.1.3 Active folder



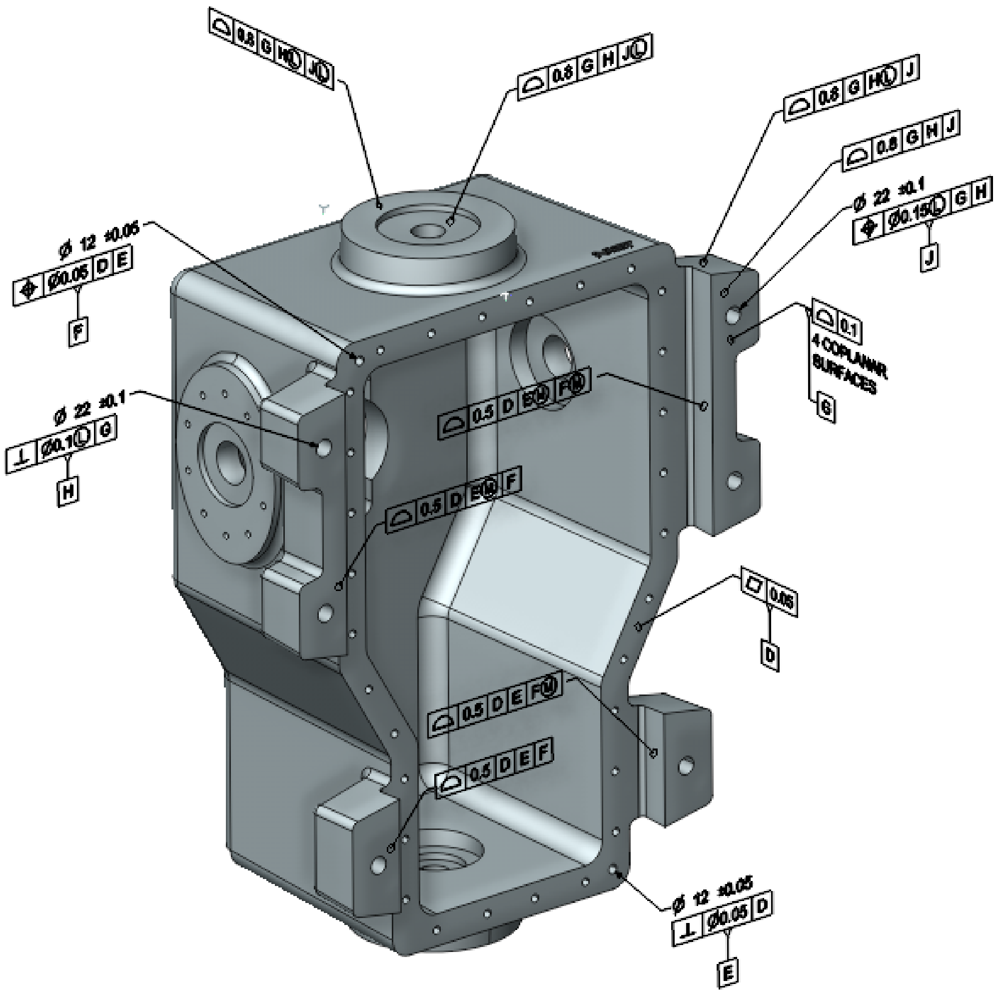
Only one group can be active at one time. Work (import, detailing, transformations and etc.) is possible only with the objects located in the currently active group. The group, which is active is treated as a single object, and is considered indivisible. In order to work separately with elements located 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.

See also:

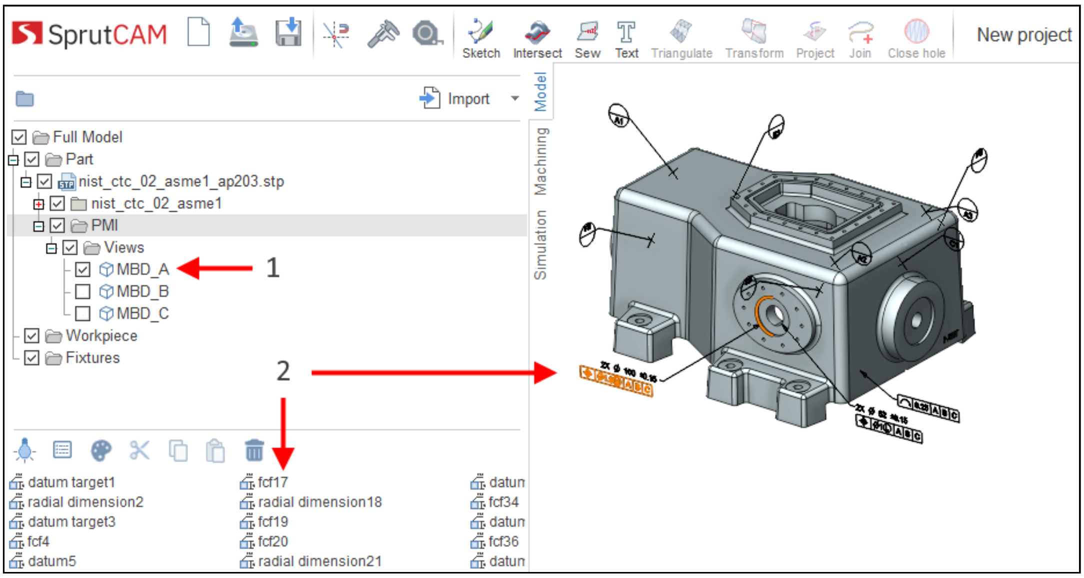
Geometrical model structure

4.1.1.4 PMI



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

PMI 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**

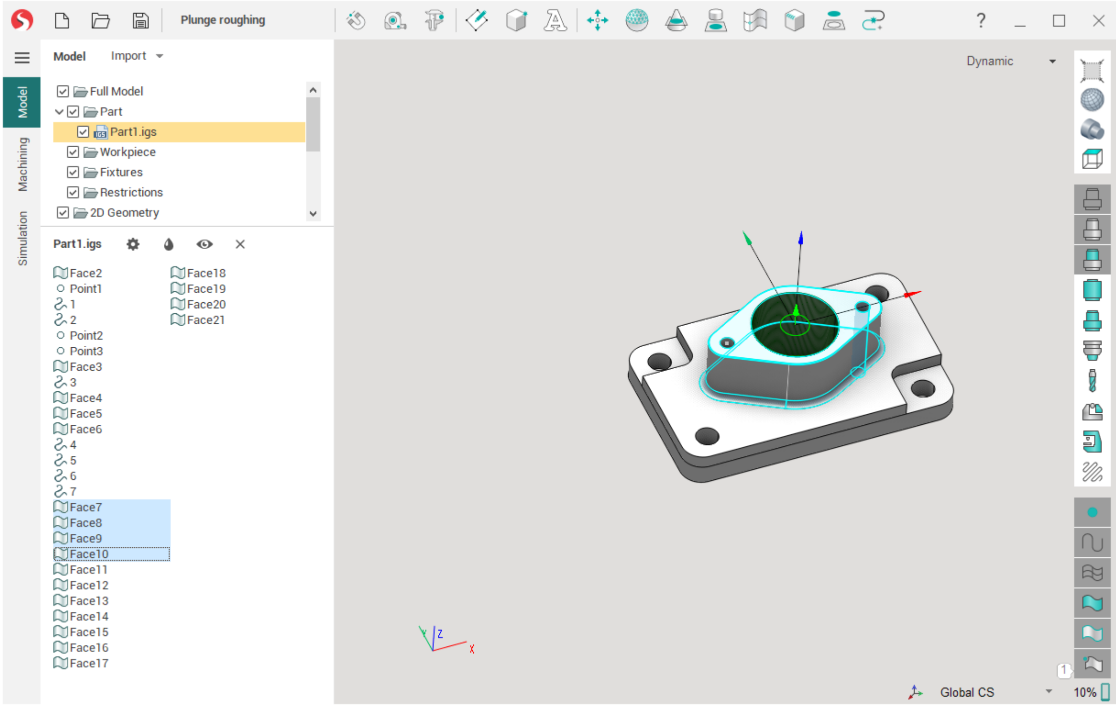
Some **PMI nodes** have connections to 3D model objects.. When you select a **PMI node**, they are also highlighted.

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)

4.1.2 Geometrical model structure window

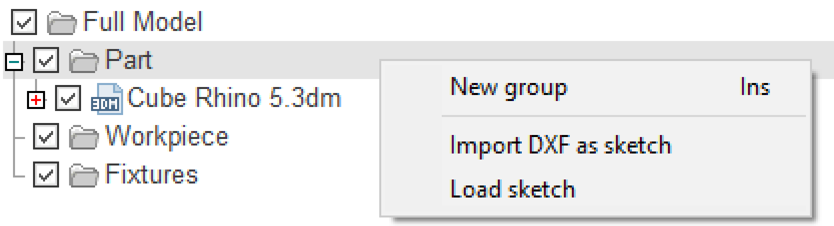
The model structure window consists of three panels: the <tools panel>, the <model tree> and a <list of available objects>.



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.

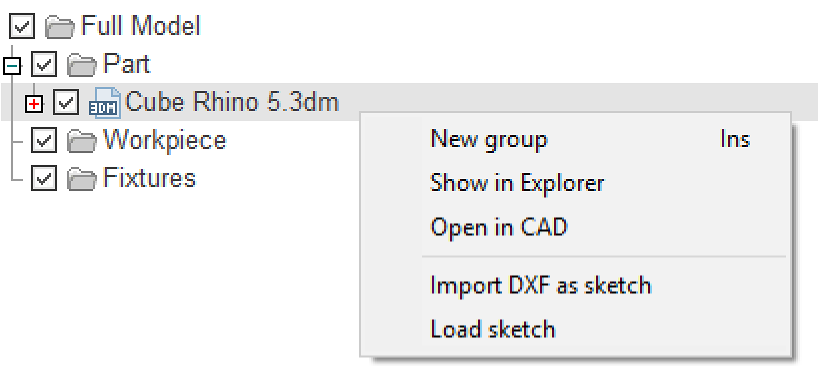
Clicking the right mouse button on a normal group in <**model tree**> brings up the following pop-up window:



Commands:

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

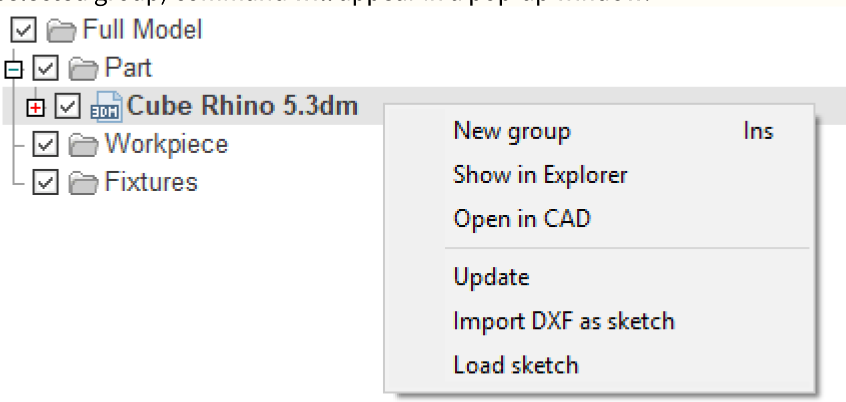
Clicking the right mouse button on the node of the imported model in the <**model tree**> causes the following pop-up window:



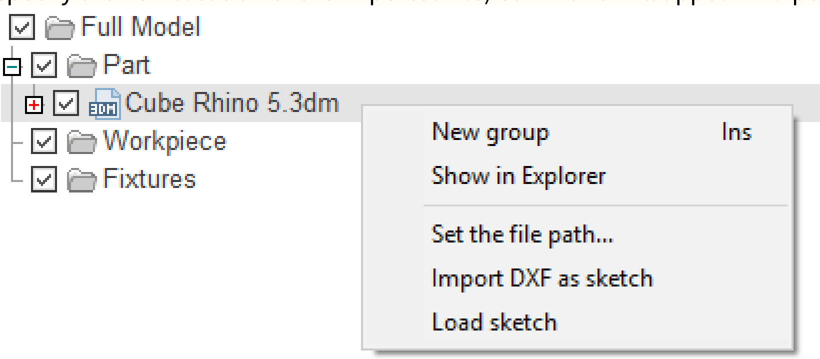
Commands:

- **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.

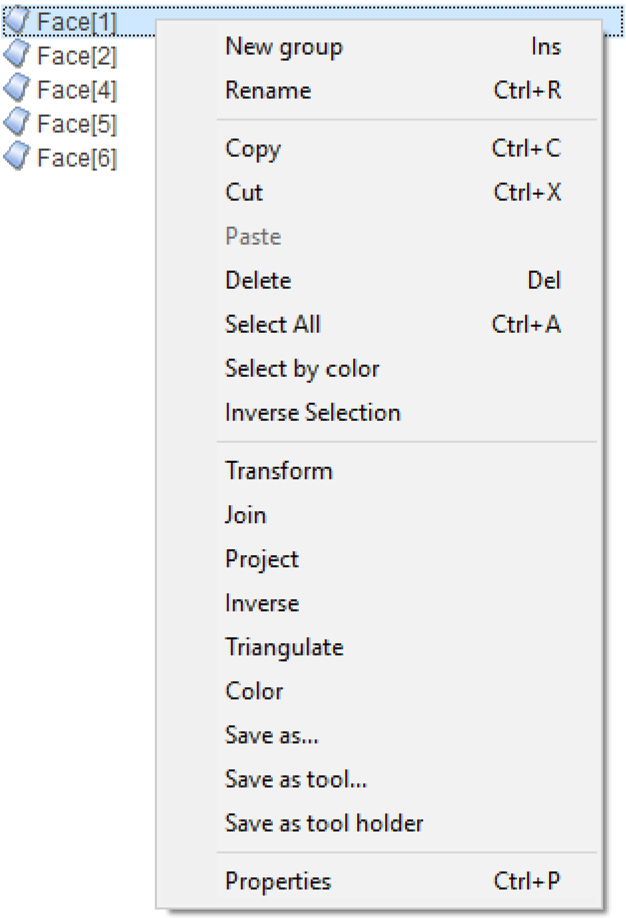
If the imported file has been changed, the name of the group will be allocated "**bold**", the toolbar button appears "Model files update manager" , and the Update (update only selected group) command will appear in a pop-up window:



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:

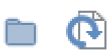


Commands:

- **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.

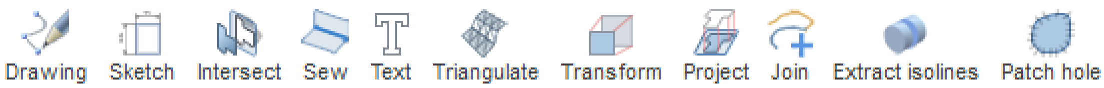
In the <list of available objects>, all groups and geometrical objects, which are a part of the active group, are displayed. That is, the objects which are currently available for selection and modification. Single left mouse clicking on any of the listed objects, selects that object.

On the <tool panel> you will find the following buttons:



- – New folder creation. The folder will be created inside the active one.
- – call the Model files update manager (visible only if changes in the imported model files are detected).

- – Set selected surfaces visible/invisible.
- – Viewing and editing properties of selected objects.
- – Setting color of selected objects.
- – Works with the exchange buffer (cut, copy, paste). The cut/copy function works with the highlighted objects. Insertion from the exchange buffer (paste) is performed into the active group.
- – Deletes selected objects.



- – Starts the SprutCAD drawing.
- – Getting curves as section of 3D model by plane.
- – Sew faces.
- – Creating texts.
- – Triangulating surfaces.
- – Spatial transformations of selected objects.
- – Outer borders projection on plane.
- – Joining curves.
- – Triangulation of selected curves or patching holes in meshes.

See also:

Geometrical model structure

4.1.3 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.

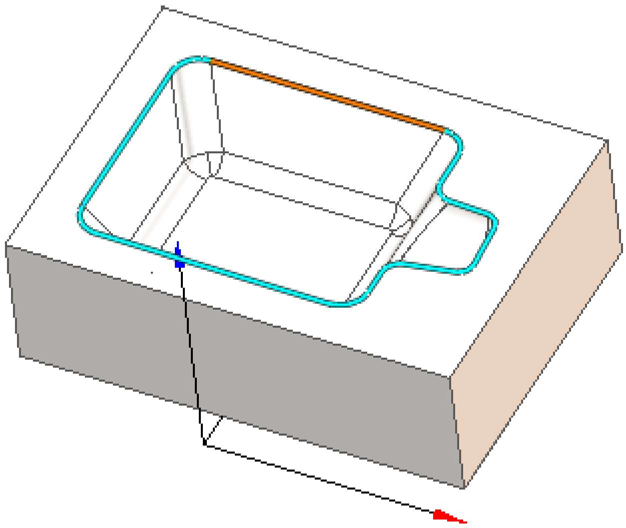
See also:

Geometrical model structure

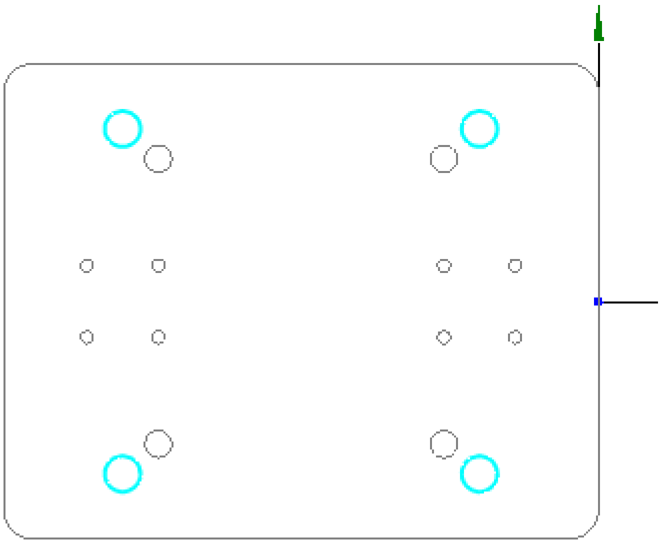
4.1.4 Intellectual object selection

In SprutCAM X you can quickly select design features and their patterns by simple double click on a 3d model entity.

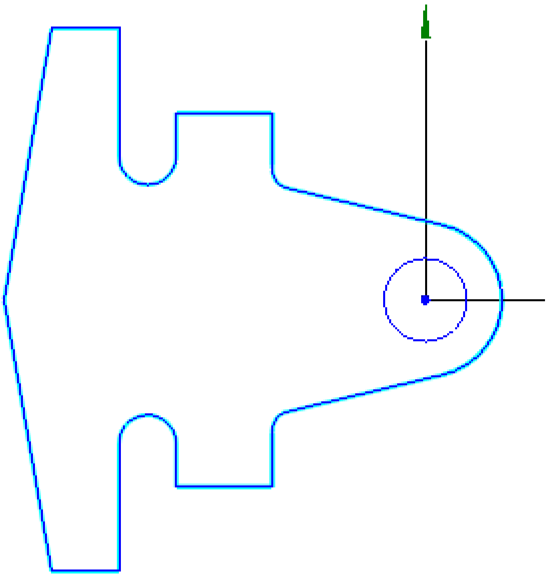
- Double click on a 3d edge automatically selects all tangent edges.



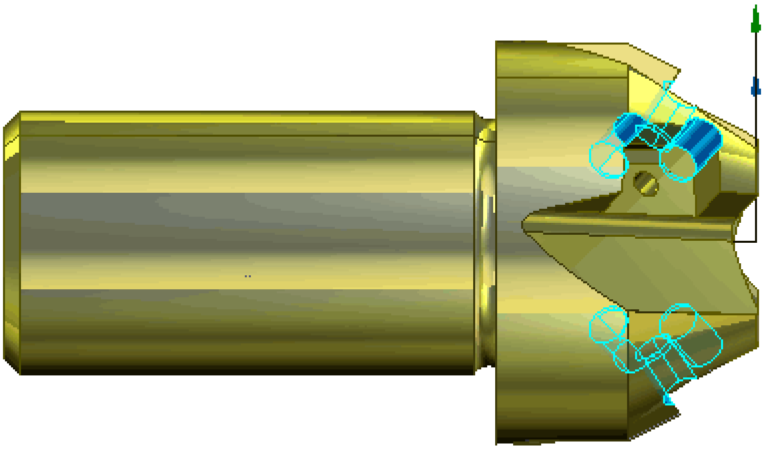
- Double click on a circle selects all the circles with the same diameter.



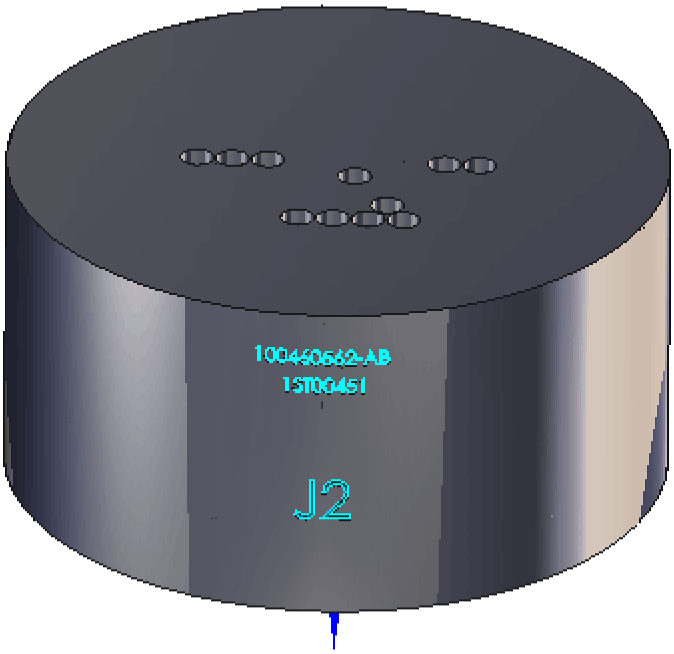
- Double click on a segment of a contour automatically selects all segments of the contour.



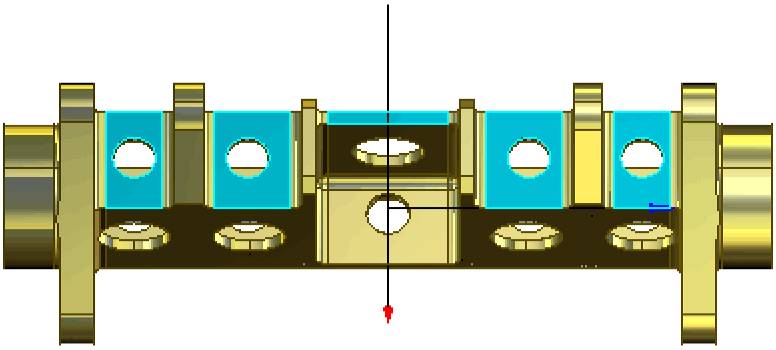
- Double click on an internal cylindrical face or hole automatically selects all the inner cylinders with the same diameter.



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



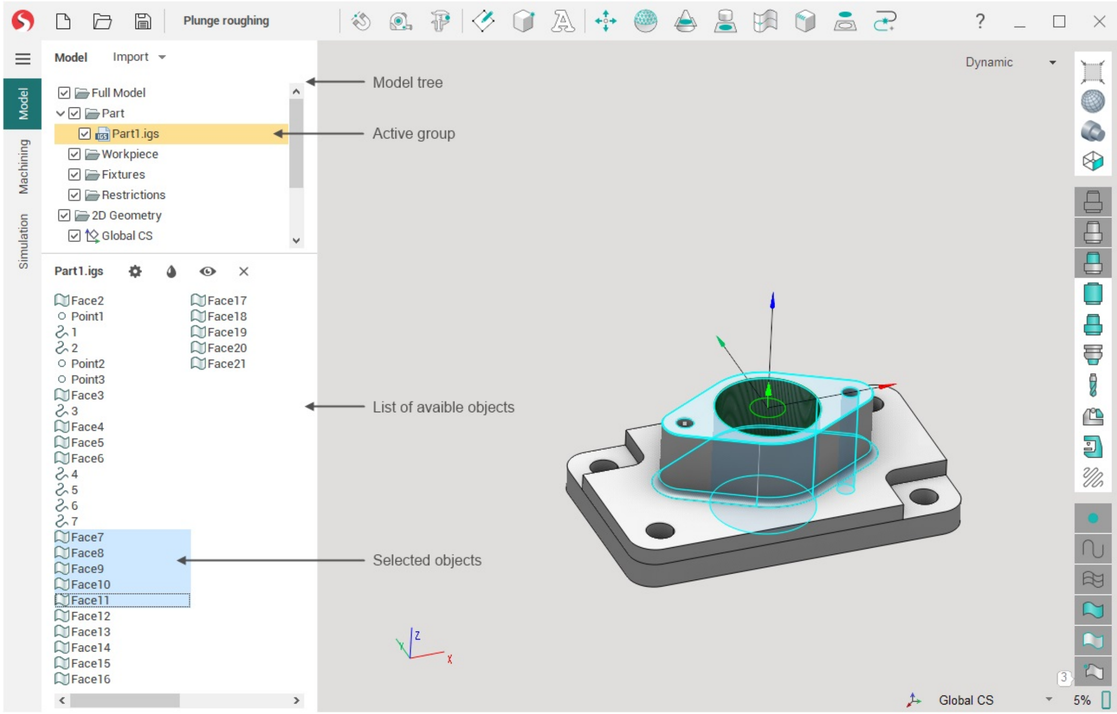
- Double click on a planar face automatically selects all coplanar faces.



See also:

Geometrical model structure

4.1.5 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.

See also:

Geometrical model structure

Creating a new group

Working with the exchange buffer

4.1.5.1 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.

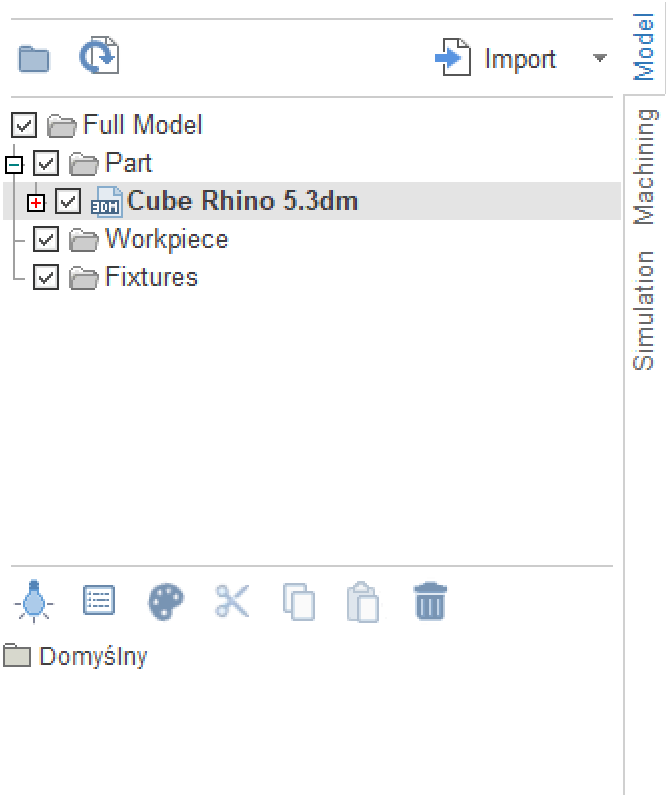
See also:

Geometrical model structure window

4.1.5.2 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 SprutCAM X 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>** .



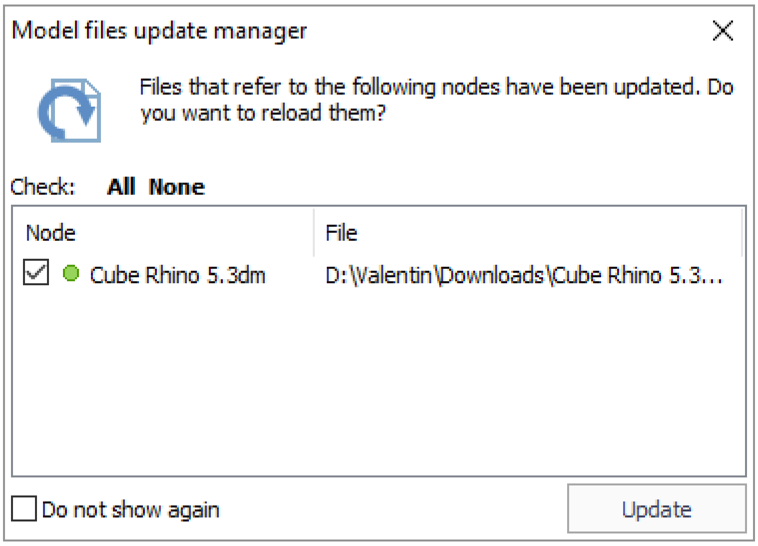
There are two options for the update:

- from SprutCAM X:
- via Model files update manager;
- via context menu **<Update>** .
- from CAD system in witch installed a SprutCAM X addin .

Updating from SprutCAM X

Model files update manager

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

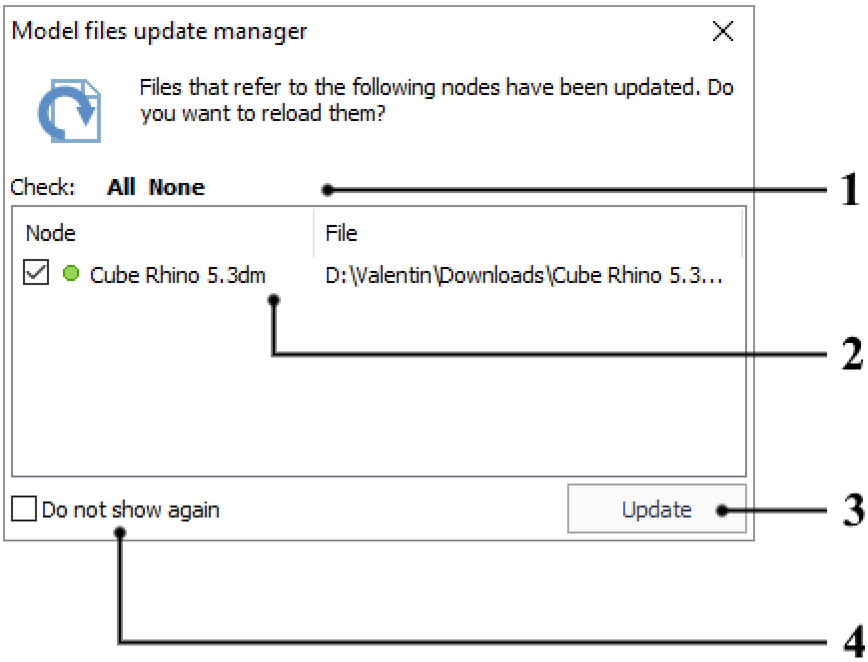


> If you just close the **Model files update manager** window, the next time it appears only when a new change is detected.

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

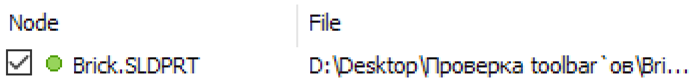


1. **<All>** button select all files, **<None>** button deselect all marks.

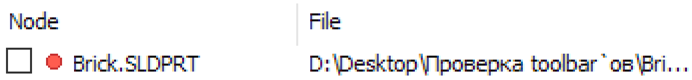
2. There is a list of all the imported files that were changed in this section. Set checkbox beside the file indicates that it is marked and after the pressing the **<Update>** button it will be updated.

> 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)

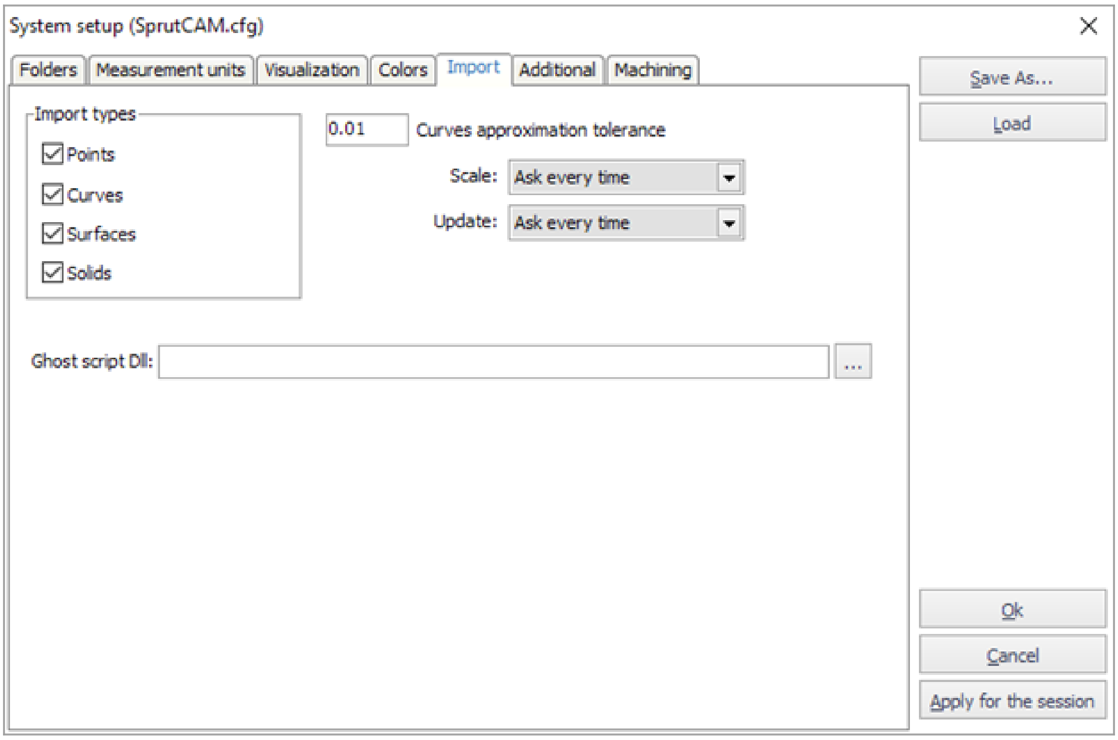


> Double click on the line will open a folder with the file.

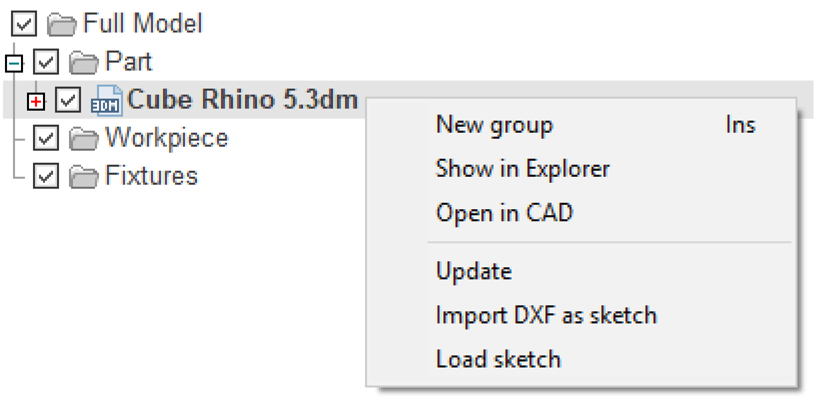
3. **<Update> / <Close>** button works as follows:

- 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>**).

4. If you check **<Do not show again>** - **Model file update manager** window will not be shown automatically. It's possible to change this setting by using the system menu (**Utilities / System setup / Import / Update**):



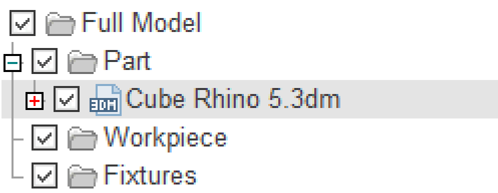
Context menu "Update"



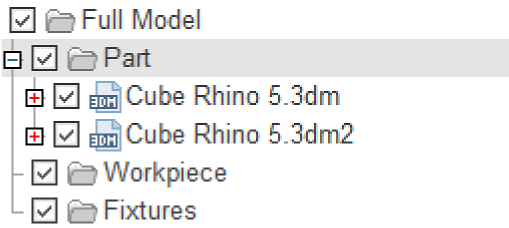
Updating from CAD system

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

- If you want to update, imported earlier model, this group need to make active or go to SprutCAM X and update from it;



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



See also:

Geometrical model structure window

4.1.5.3 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].

See also:

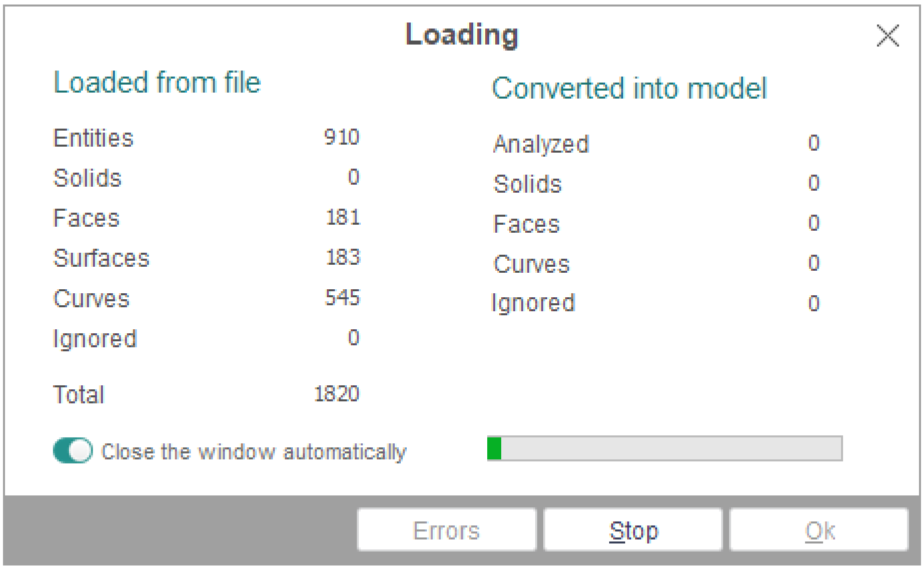
Geometrical model structure window

4.2 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 SprutCAM X 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.*

See also:

Geometrical model preparation

Importing objects from IGES files

Importing objects from DXF files

Importing objects from PostScript files

Importing objects from STL files

Importing objects from VRML files

Importing objects from 3dm files (Rhinoceros™)

Importing objects from Parasolid™ files

Importing objects form SolidWorks™ files

Importing objects form SolidEdge™ files

Addins for SprutCAM X

4.2.1 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 SprutCAM.

See also:

Geometrical objects import

Requirements for IGES file

Types of importable objects

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

4.2.1.1 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.

See also:

Importing objects from IGES files

4.2.1.2 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 SprutCAM X are identical to the colors used in the modeling program.

See also:

Importing objects from IGES files

4.2.1.3 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).

See also:

Importing objects from IGES files

4.2.2 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*.

See also:

Geometrical objects import

4.2.3 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).

See also:

Geometrical objects import

4.2.4 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: *SprutCAM X 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.

See also:

Installing the GhostScript Library

Geometrical objects import

4.2.4.1 Installing the GhostScript Library

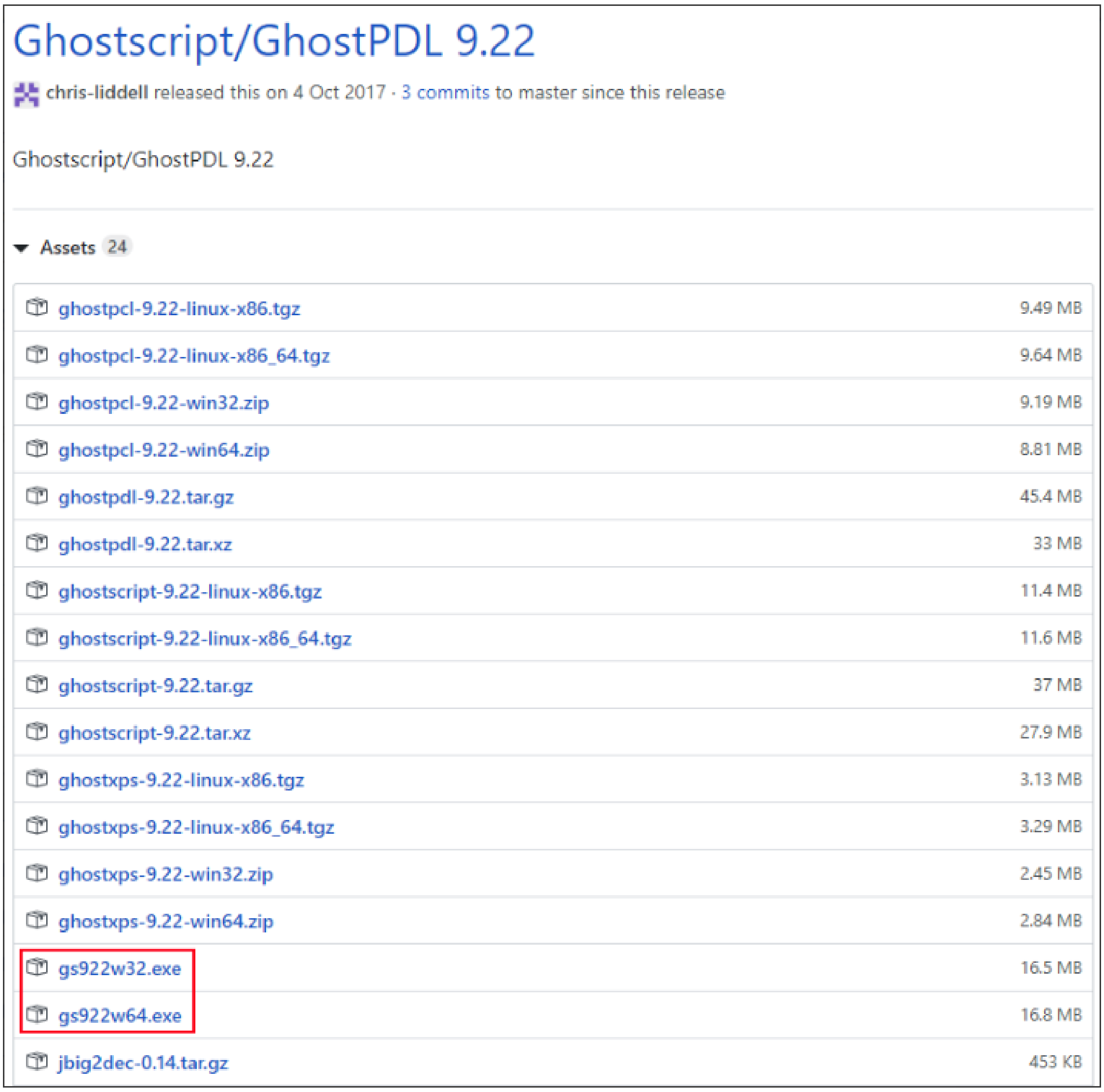
Ghostscript software is required to import PostScript files. This library was developed by Artifex Software, Inc. and is not included in the SprutCAM X distribution due to licensing restrictions. But you can download and install it for personal use.

SprutCAM X supports the Ghostscript library of version not older than 9.22.

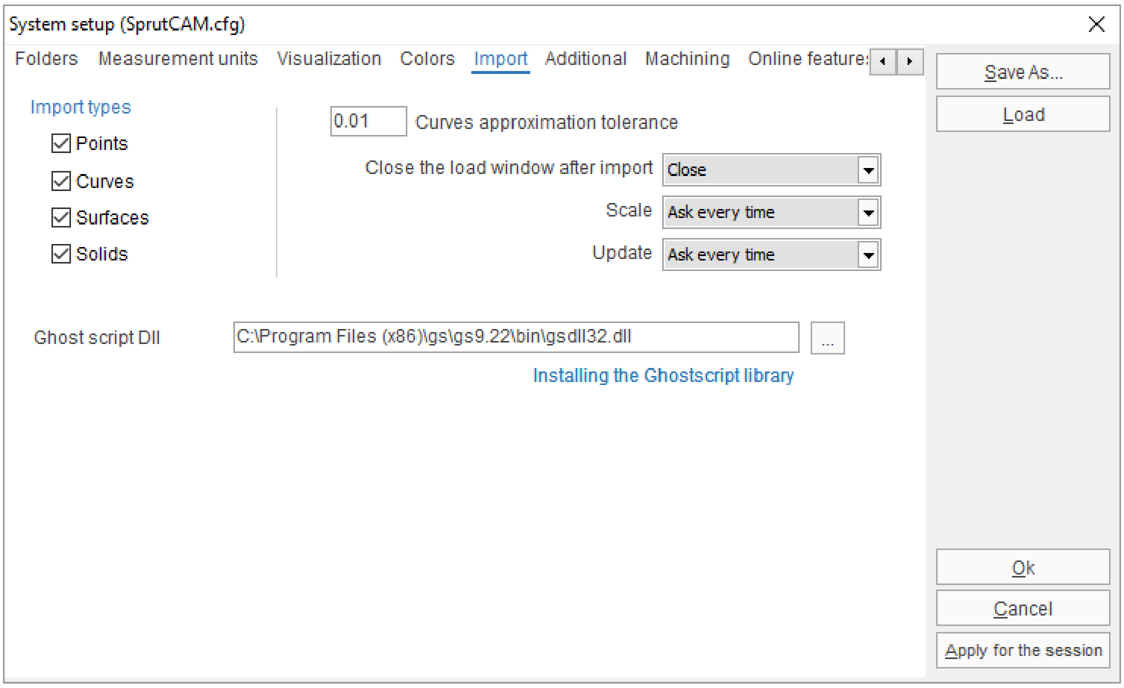
Download Ghostscript 9.22

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**":



See also:

Geometrical objects import

4.2.5 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>).*

See also:

Geometrical objects import

4.2.6 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.

See also:

Geometrical objects import

4.2.7 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.

See also:

Geometrical objects import

4.2.8 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.

See also:

Geometrical objects import

4.2.9 Importing objects from 3dm files (Rhinoceros™)

SprutCAM X performs direct reading of project files from the Rhinoceros CAD system versions 1.0 - 5.0 (*.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 SprutCAM addin "Rhinoceros Toolbar & import addin".*

See also:

Geometrical objects import

Rhinoceros toolbar & import addin

4.2.10 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.

SprutCAM X supports the **Parasolid™** data transmission format up to 30.2 versions.

See also:

Geometrical objects import

4.2.11 Importing objects from SolidWorks™ files

SprutCAM X
 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.

Note: *At the moment, imports files* ***without configuration*** *impossible.*

See also:

Importing objects from Parasolid™ files

Geometrical objects import

4.2.12 Importing objects from SolidEdge™ files

SprutCAM X

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.

See also:

Importing objects from Parasolid™ files

Geometrical objects import

4.2.13 Importing objects from SGM files (SPRUT)

3D models in from the SPRUT CAD system (*.sgm) is fully supported by SprutCAM X, without any limitations.

See also:

Geometrical objects import

4.2.14 Importing objects with SprutCAM X's addins

Addin is component or small program, which may be attach to SprutCAM X.

They provide the ability to import project files CAD.

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

See also:

Geometrical objects import

SprutCAM X Addins

Addin's list

4.2.15 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

	Format	Description	Number of points in a line
3	tffEulerXYZ	Coordinates of the point and Euler angles Rx Ry Rz with rotations around the movable axes	6
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 around the movable axes	6
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
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

	Format	Description	Number of points in a line
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 rotations around the movable axes	6
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
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

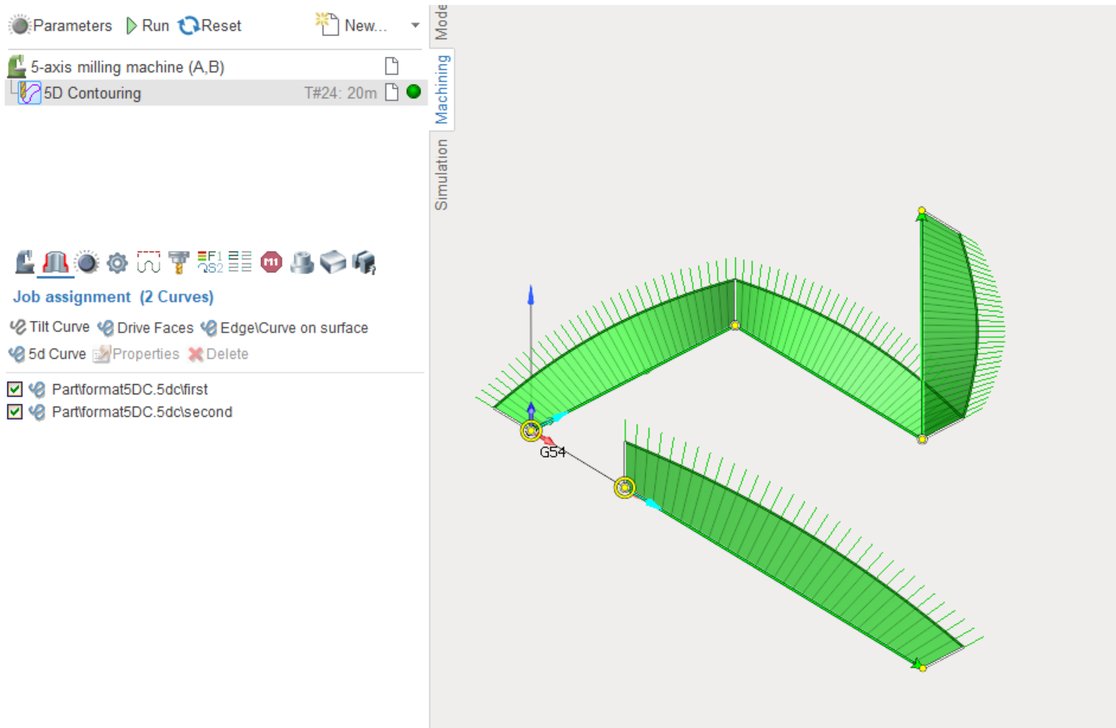
Example of a file that does not contain settings line:

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 

in 5D Contouring operation is shown on the following image :

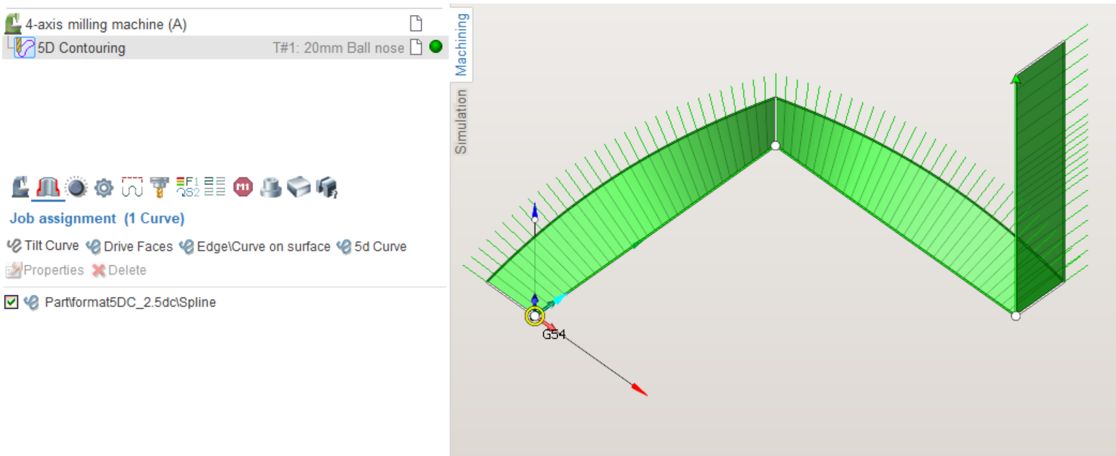


Example file, with settings line:

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:



4.2.16 SprutCAM X Addins

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 SprutCAM X
activation toolbar or the menu.

See also:

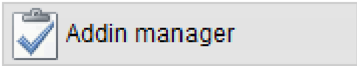
SprutCAM X Addin's Manager

Addin's list

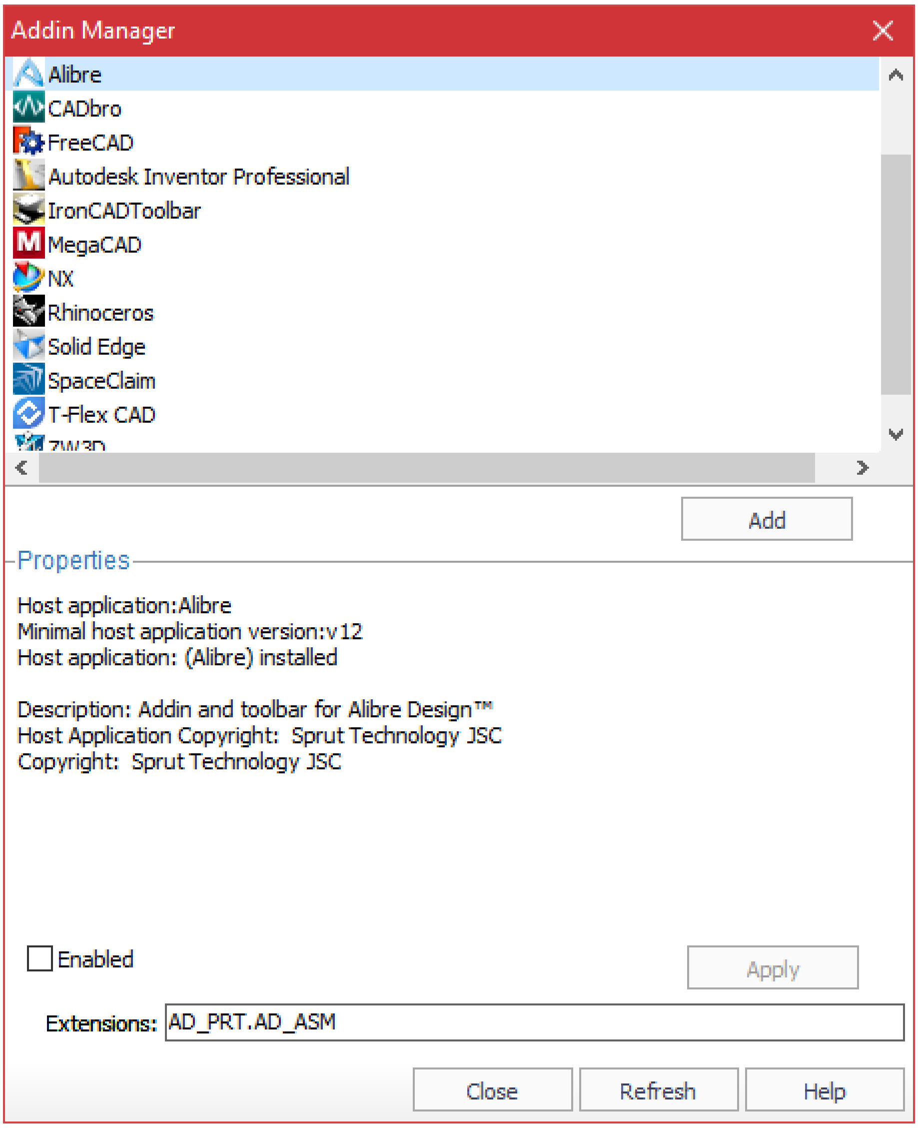
4.2.16.1 SprutCAM X Addin Manager

SprutCAM X® Addin Manager is a program designed to control SprutCAM X® addin's and external CAD translators.

You can run Addin's Manager from SprutCAM X- use menu item on main panel.



SprutCAM X Addin Manager window:



Addin's

Work sequence to install addin:

1. Select necessary addin from the top list.
2. Press <**Install**> button.
3. Edit parameters (custom).
4. Press <**Apply**> button to save changes (custom).
5. Press <**Close**> button (custom).

Work sequence to uninstall addin:

1. Select necessary addin from the top list.

2. Press <**Uninstall**> button.
3. 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 SprutCAM® 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.

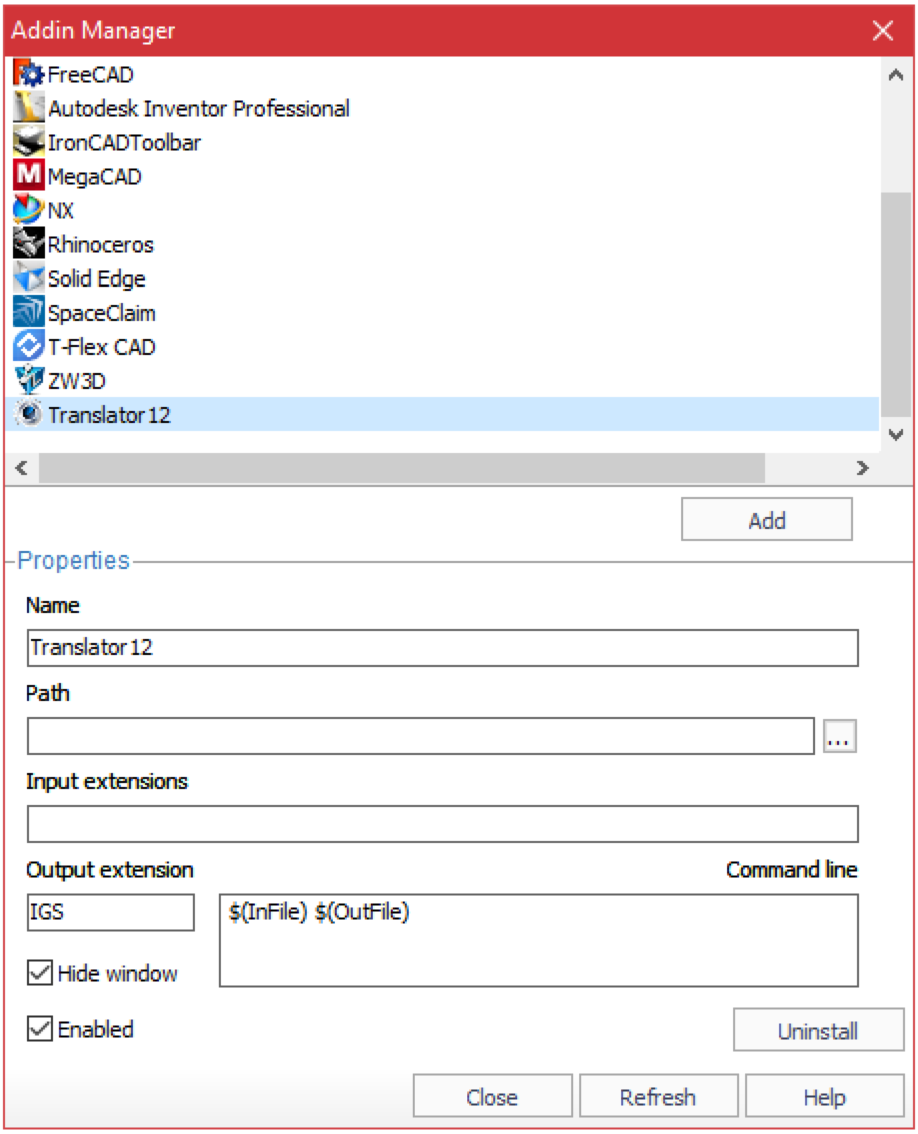
> In some cases there is a need to reload application (addin host) when addin properties are changed. For example after installation of the 'SprutCAM® toolbar for SOLIDWORKS® ' addin SOLIDWORKS® should be reloaded.
>
> All addin's must be in the 'Addins' folder (the folder for SprutCAM X® Addin Manager) and/or its subfolders. SprutCAM X® Addin Manager scans those folders to look for addin's and lists its. That is enough to place any addin (from SprutCAM Tech Ltd. or from any other source) to those folders and it will be accessible for SprutCAM X® Addin Manager (press <**Refresh**> button if SprutCAM X® 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 SprutCAM X 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 SprutCAM X.

So, to add an external CAD translator into SprutCAM X you should

1. Open the Addin Manager
2. Click on the <**Add**> button.
3. 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 SprutCAM X 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)

See also:

SprutCAM X Addins

4.2.16.2 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;

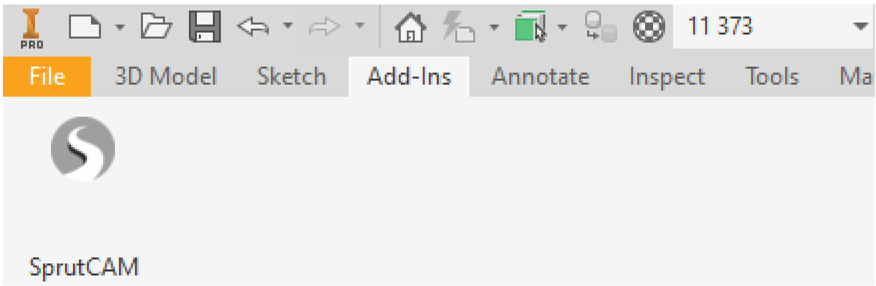
- KOMPAS™ toolbar & import addin ;
- NX toolbar & import addin;
- Rhinoceros™ toolbar & import addin ;
- SolidCAM™ toolbar & import addin ;
- SolidEdge™ toolbar & import addin ;
- SOLIDWORKS™ toolbar & import addin ;
- SpaceClaim™ toolbar & import addin ;
- T-Flex™ toolbar & import addin ;
- ZW3D™ toolbar & import addin ;
- Onshape™ connector plugin .

See also:

SprutCAM X Addins

Autodesk Inventor™ toolbar & import addin

The toolbar allows you to export geometric data from **Autodesk Inventor Professional™** to **SprutCAM X™**.



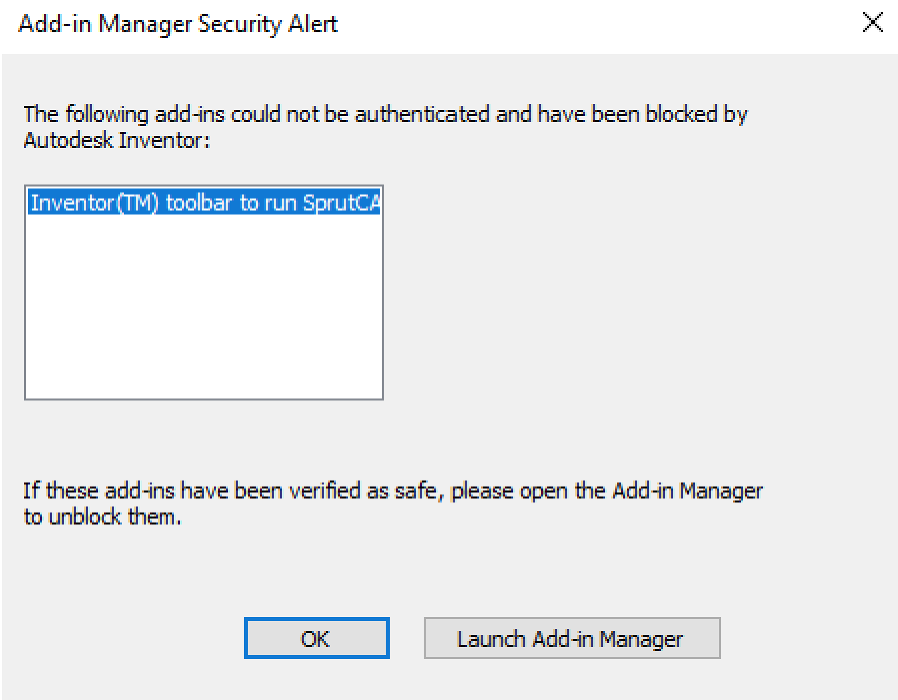
The addin allows you to import **Autodesk Inventor Professional™** project files.

Supported file extensions: IAM, IDW, IPT, IPN, IDE, PRT, ASM, SAT, STE, STEP, DWG, DXF, IGES, IGS.

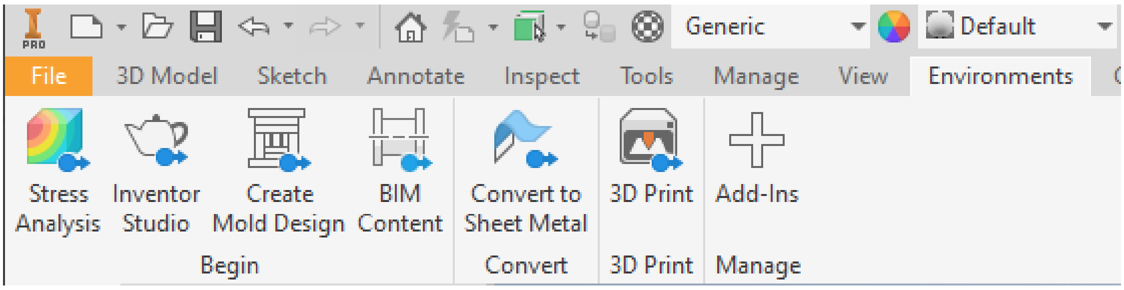
> The required application (**Autodesk Inventor Professional™**) must be installed on your computer for this option to work.

Autodesk Inventor Professional™2020 may have problems installing the toolbar:

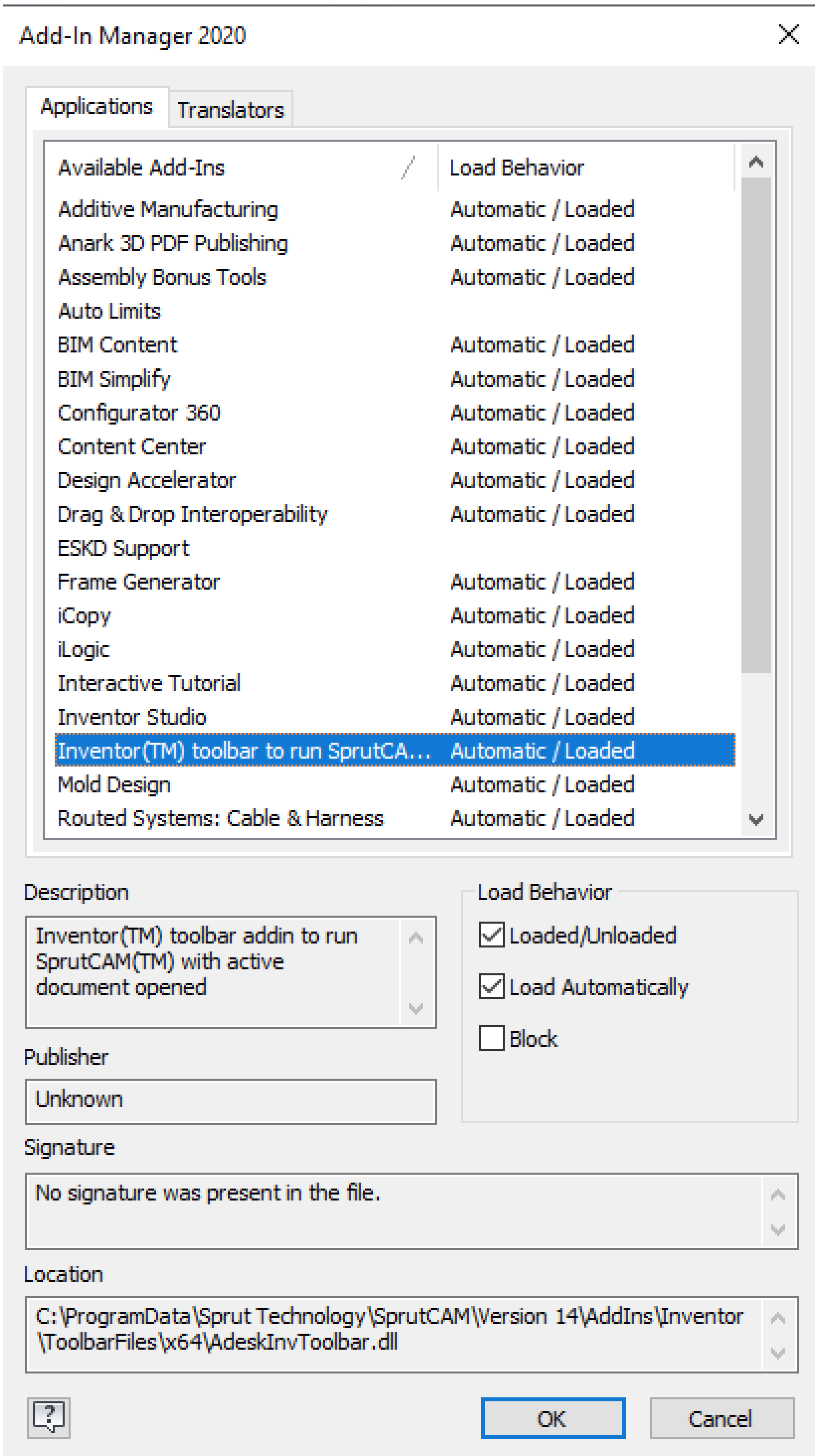
If at startup the following window appears:



you must manually enable the toolbar in **Autodesk Inventor Professional™2020**. 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 **SprutCAM X™ toolbar** and set the parameters as shown in the following screenshot:



Then the **SprutCAM X™ toolbar** should appear on the ribbon panel.

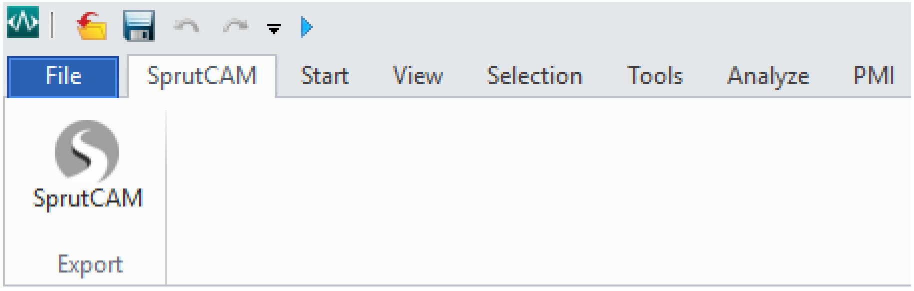
See also:

Addin's list

CADbro™ toolbar

CADbro

The toolbar allows you to export geometric data from **CADBro™** to **SprutCAM X™**.



⚠ The required application (**CADbro** ™) must be installed on your computer for the correct work of this option.

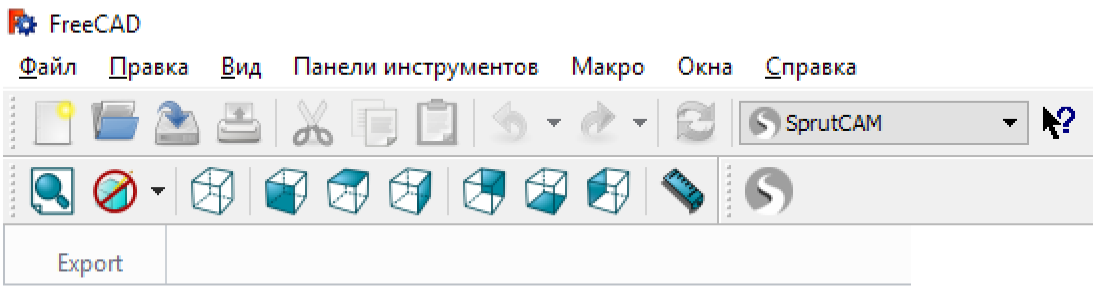
See also:

Addin's list

FreeCAD™ toolbar & import addin

FreeCAD

The toolbar allows you to export geometric data from **FreeCAD™** to **SprutCAM X™**.



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.

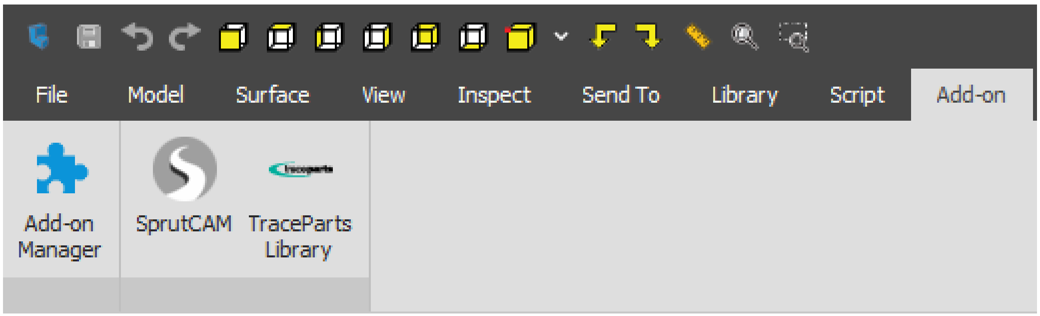
⚠ The required application (FreeCAD™) must be installed on your computer for this option to work.

See also:

Addin's list

Alibre Design™ toolbar & import addin

The toolbar allows you to export geometric data from **Alibre Design™** to **SprutCAM X™**.



The addin allows you to import **Alibre Design™** project files.

Supported file extensions: AD_PRT, AD_ASM, AD_SMP.

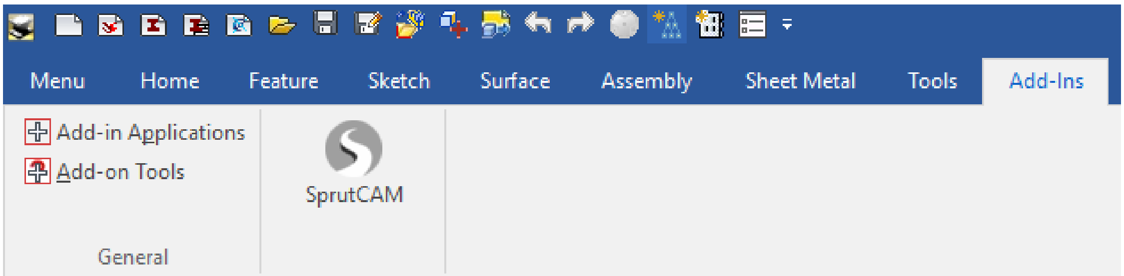
> ⚠ The required application (**Alibre Design™**) must be installed on your computer for this option to work.

See also:

Addin's list

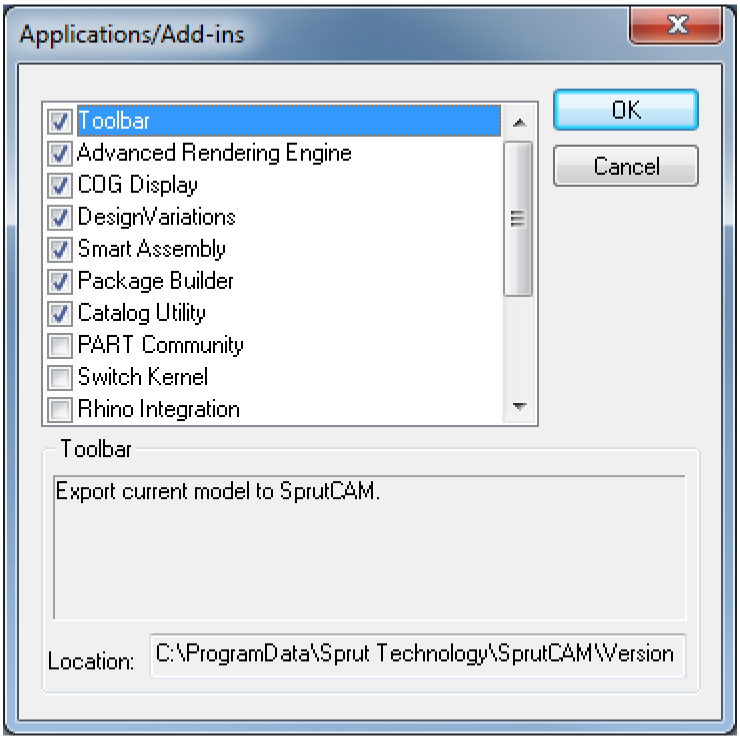
IronCAD™ toolbar

The toolbar allows you to export geometric data from **IronCAD™** to **SprutCAM X™**.



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.

See also:

Addin's list

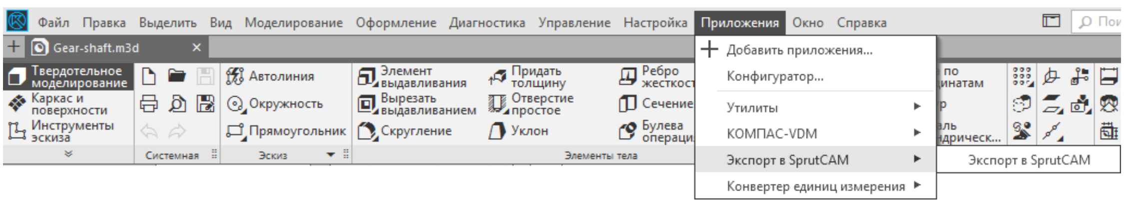
KOMPAS™ toolbar & import addin

Kompas(TM)x32
Kompas(TM)x64

The toolbar allows you to export geometric data from **Kompas™**, **Kompas LT™** or **Kompas Home™** to **SprutCAM X™**.

There are two versions for target CAD system: x32 and x64.

> ⚠ In versions of **Kompas V18+** does not work in a **demo-mode**.



The addin allows you to import project files **Kompas™**, **Kompas LT™** or **Kompas Home™**.

Supported file extensions: A3D, M3D, CDW, FRW.

> ⚠ The required application (**Kompas™**, **Kompas LT™** or **Kompas Home™**) must be installed on your computer for the correct work of this option.

Associativity

If operations that modify the whole body (such as moving, scaling, etc.) are used while model rebuilding in **Kompas™**, the associativity wouldn't work. These operations lead to a full reset of unique indexes, and they need to be re-assigned.

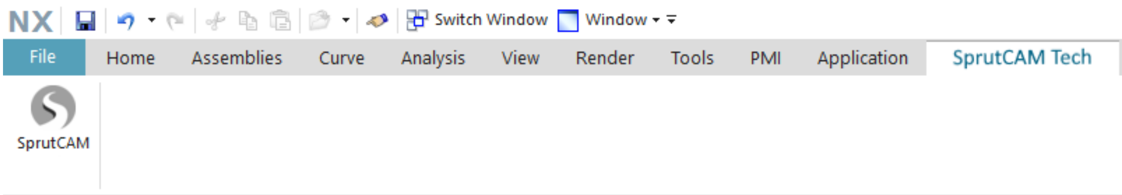
See also:

Addin's list

NX™ toolbar & import addin

The toolbar allows you to export geometric data from **NX™** to **SprutCAM X™**.

> ⓘ **NX™** version 8.5 and 11 are supported.



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.

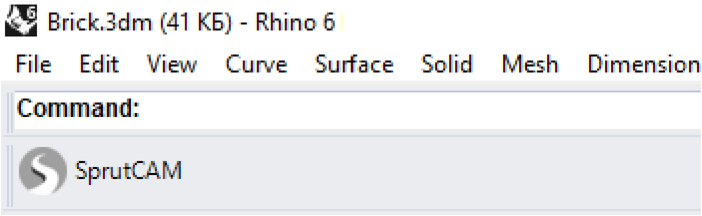
See also:

Addin's list

Rhinoceros toolbar & import addin

The toolbar allows you to export geometric data from **Rhinoceros™** to **SprutCAM X™**.

Rhinoceros™ version 3 - 7 are supported.
Rhinoceros™ version 3 must be SR3c (build 21-apr-2004) or later.



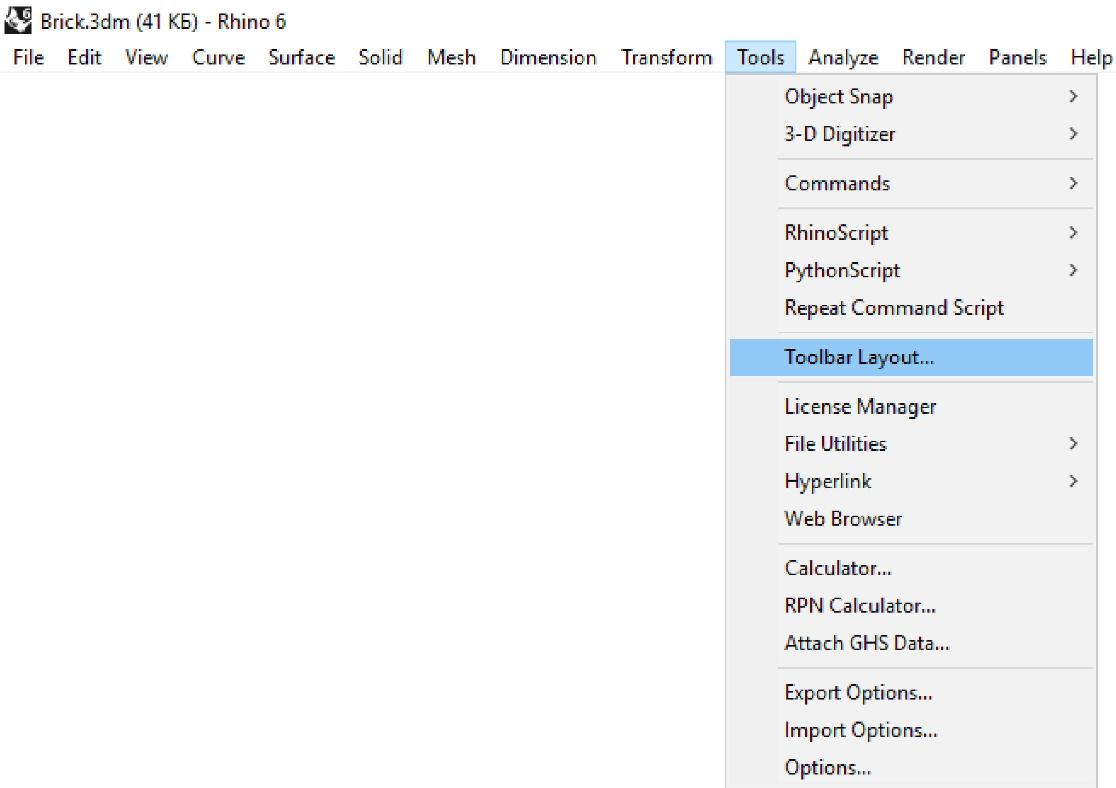
The addin allows you to import **Rhinoceros™** project files.

Supported file extensions: 3DM, RWS, 3DS, STP, STEP, RAW, WRL, VRML, AI, EPS, LWO, SPL, VDA, DWG, DXF, DGN, SLDPRT, SLDASM.

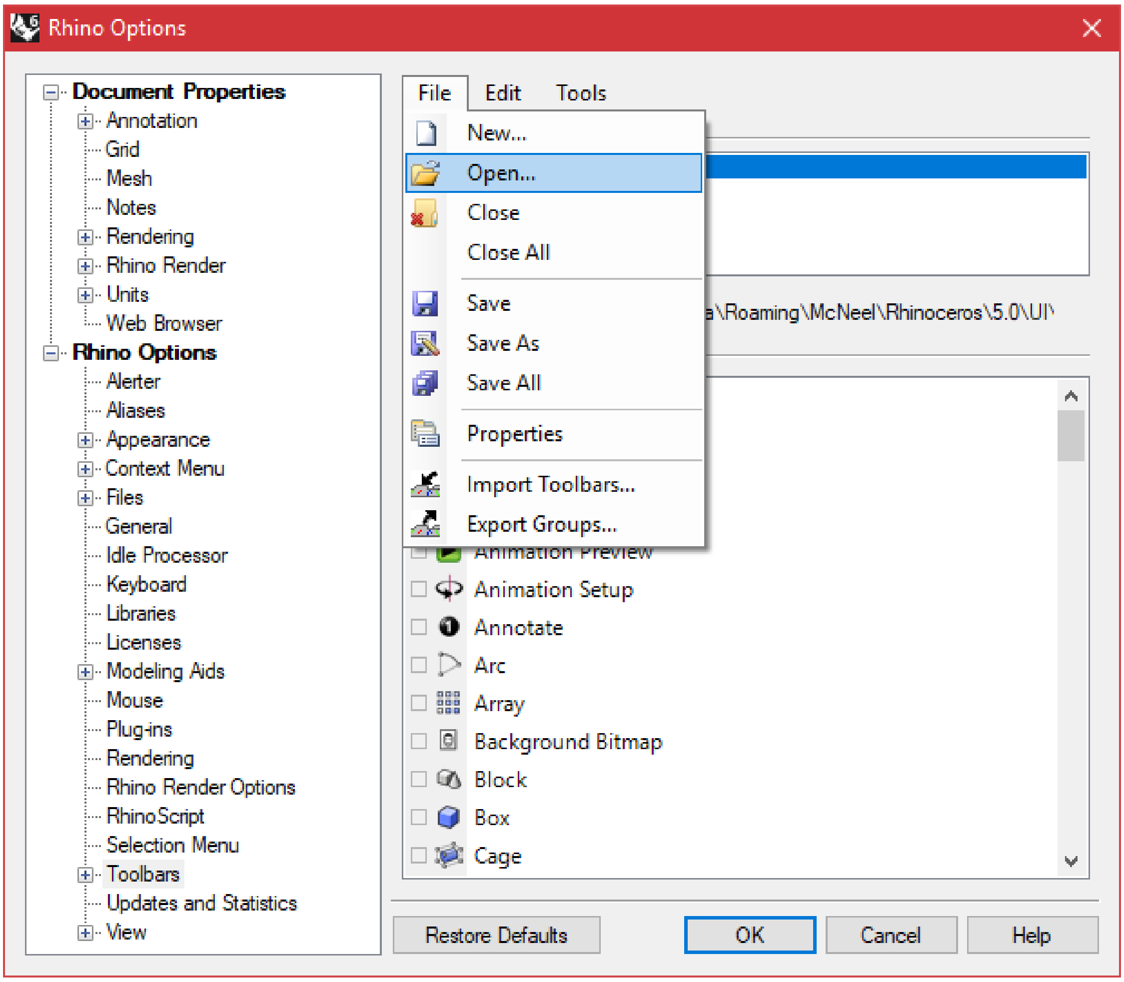
The required application (**Rhinoceros™**) must be installed on your computer for this option to work.
The **Rhinoceros™** application must be running and closed at least once in order for the data to be recorded in the registry. Otherwise, the toolbar will not be able to install.

Manual way to install the toolbar

Go into into **Tools** → **Toolbar Layout...**:



Click **File** → **Open...:**



Then specify the path to the **ToolbarForRhino5.ru** toolbar file. For example, in version 16 it will be here:

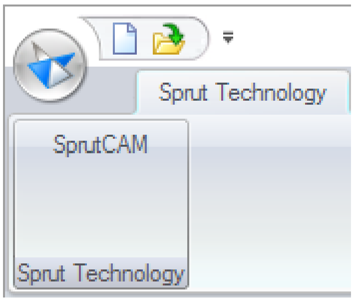
C:\ProgramData\SprutCAM Tech\SprutCAM X\Version 16\AddIns\Rhino\ToolbarForRhino5.rui

See also:

Addin's list

SolidEdge™ toolbar & import addin

The toolbar allows you to export geometric data from **SolidEdge™** to **SprutCAM X™**.



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.

See also:

Addin's list

SOLIDWORKS™ toolbar & import addin

The toolbar allows you to export geometric data from **SOLIDWORKS™** to **SprutCAM X™**.

The addin allows you to import project files **SOLIDWORKS™**.

Supported file extensions: SLDASM, ASM, SLDPRT, PRT, SLDDRW, DRW, X_B, X_T, STP, STEP.

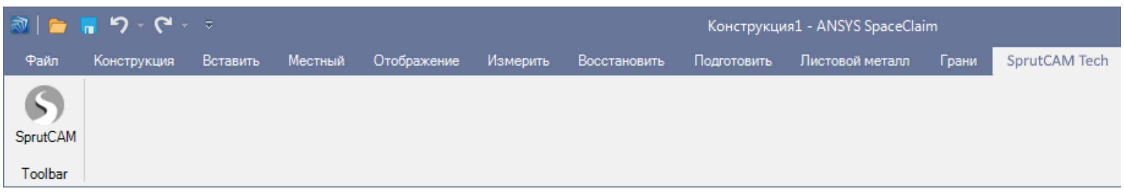
> ⚠ *The required application (SOLIDWORKS™) must* be installed on your computer for the correct work of this option.

See also:

Addin's list

SpaceClaim™ toolbar & import addin

The toolbar allows you to export geometric data from **SpaceClaim™** to **SprutCAM X™**.



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.

Associativity

Default exchange between **SpaceClaim™** and **SprutCAM X™** is performed by Parasolid files (associativity is maintained). But this module is not included in the standard package of components for **SpaceClaim™**, therefore the possibility of manual shifting for exchange file was realized. In this case, the associativity would be lost, but export function stays available.

Expansion change instructions:

1. Go to the folder where the addin files (depends on the version, the path can vary, e.g.: "C:\ProgramData\SprutCAM Tech\SprutCAM X\Version 16\Addins\SpaceClaim");
2. Copy SpaceClaimTranslator.xml file and rename the copy as SpaceClaimTranslator_UserConfig.xml;
3. Move this file to the directory above (for example: "C:\ProgramData\SprutCAM Tech\SprutCAM X\Version 16\Addins\SpaceClaimTranslator_UserConfig.xml");
4. Open SpaceClaimTranslator_UserConfig.xml file and change the extension at line <OutputExtension>:

a. X_T - for Parasolid (only it maintains associativity);
b. STP - for STEP;
c. IGS - for IGES.

- Save changes and Reinstall the toolbar in Addin Manager.

> ⓘ Deleting or renaming of SpaceClaimTranslator_UserConfig.xml file would lead to the default settings.

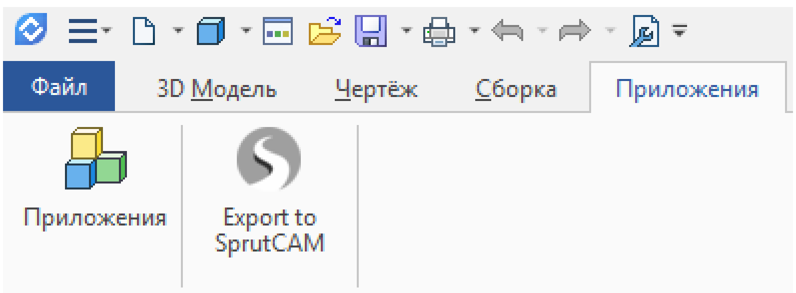
See also:

Addin's list

T-Flex™ toolbar & import addin

T-Flex CAD

The toolbar allows you to export geometric data from **T-Flex™** to **SprutCAM X™**.



The addin allows you to import **T-Flex™** project files.

Supported file extensions: GRB.

> ⚠ The required application (T-Flex™) must be installed on your computer for this option to work.

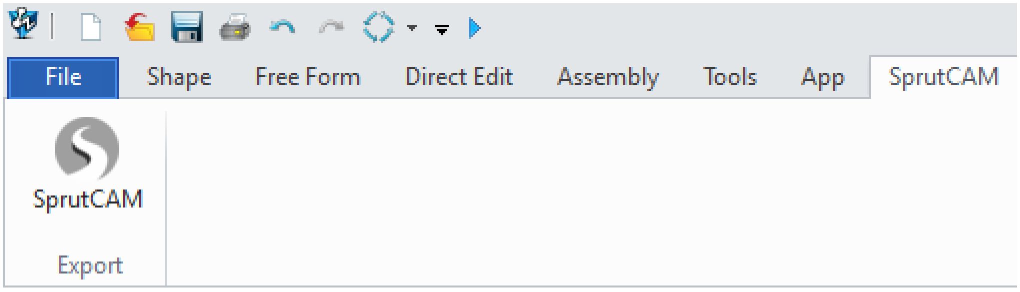
See also:

Addin's list

ZW3D™ toolbar & import addin

ZW3D

The toolbar allows you to export geometric data from **ZW3D™** to **SprutCAM X™**.



The addin allows you to import **ZW3D™** project files.

Supported file extensions: Z3.

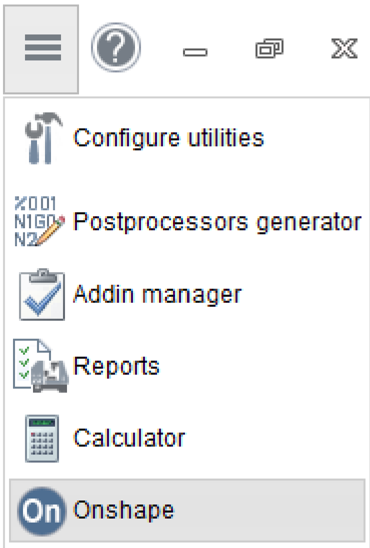
> ⚠ *The required application (ZW3D™) must* be installed on your computer for this option to work.

See also:

Addin's list

Onshape™ connector

Onshape™ is a new generation of full-cloud CAD designed specifically for modern agile design teams.



Onshape connector plugin can be started by clicking **<Onshape>** icon on the toolbar. After sign-in into Onshape account, plugin will show the list of available models. Selecting model from this list will start import process. If imported model is changed by Onshape, connector plugin will prompt to reimport model. Onshape connector plugin provides model associativity.

Onshape connector plugin can be disabled at **<**Configure utilities window**>.**

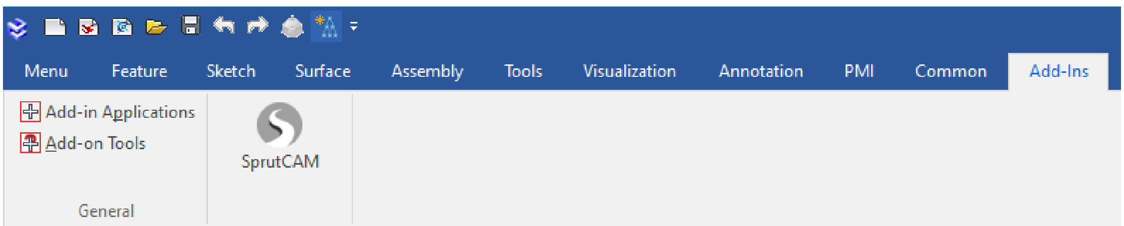
> ⓘ See the Onshape connector plugin workflow on YouTube.

See also:

Addin's list

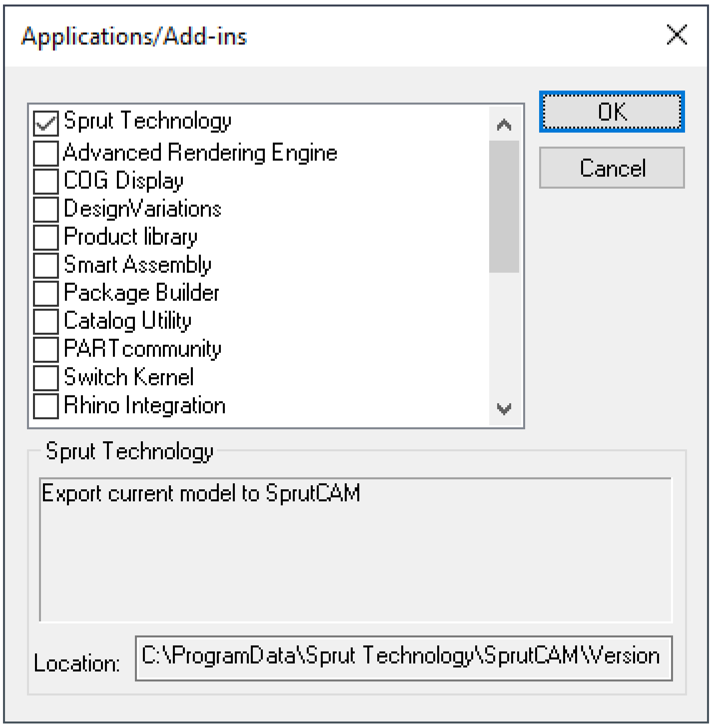
CAXA 3D™ toolbar

The toolbar allows you to export geometric data from **CAXA 3D™** to **SprutCAM X™.**



Once installed, the toolbar may not appear in **CAXA 3D™**. Then you need to press

in the dialog box to activate it:



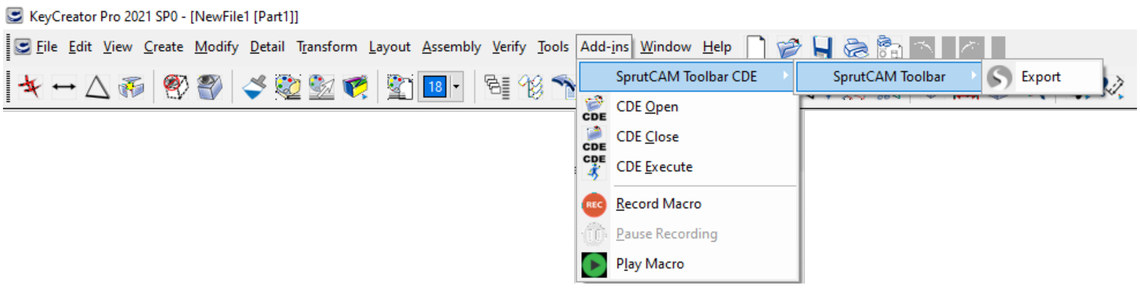
Supported file extensions: ICS, IC3D, ICSW, EXB.
The required application (**CAXA 3D™**) must be installed on your computer for this option to work.

See also:

Addin's list

KeyCreator™ toolbar

The toolbar allows you to export geometric data from **KeyCreator™** to **SprutCAM X™**.

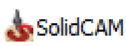


Supported file extensions: CKD.
The required application (**KeyCreator™**) must be installed on your computer for this option to work.

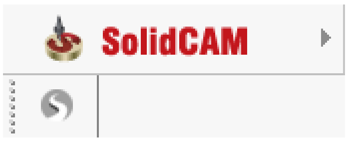
See also:

Addin's list

SolidCAM™ toolbar & import addin



The toolbar allows you to export geometric data from SolidCAM™ to SprutCAM X™.



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.

See also:

Addin's list

4.2.16.3 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
KOMPAS™	If operations that modify the whole body (such as moving, scaling, etc.) are used while model rebuilding in Kompas™, the associativity wouldn't work. These operations lead to a full reset of unique indexes, and they need to be re-assigned.
NX™	✓
Onshape™	✓
SOLIDWORKS™	✓
SpaceClaim™	✓
T-Flex™	✓

See also:

SprutCAM X Addins

4.3 Editing geometrical model

No content in this page. See child topics

4.3.1 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>.

See also:

Geometrical model preparation

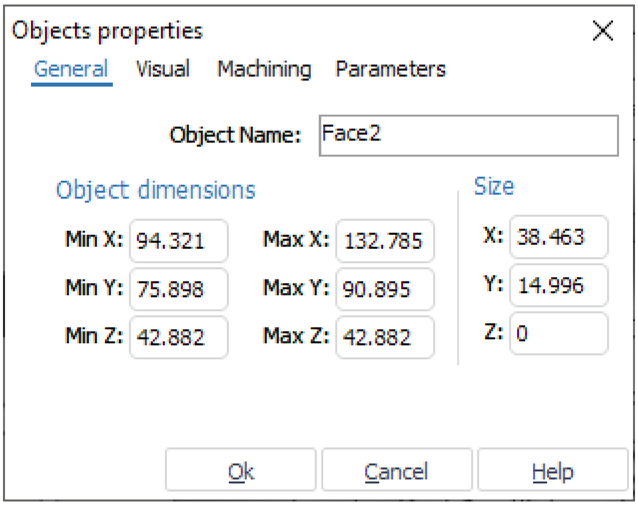
<General> tab

<Visual> tab

<Machining> tab

<Parameters> tab

4.3.1.1 <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.

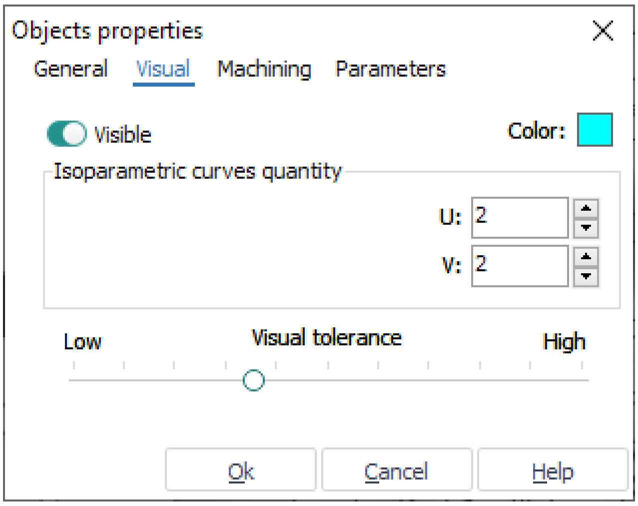
<Object Name> – name of the selected object. If several names are selected, then the field will be empty.

<Object dimensions> – overall dimensions of the selected object.

See also:

Geometrical objects properties

4.3.1.2 <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.

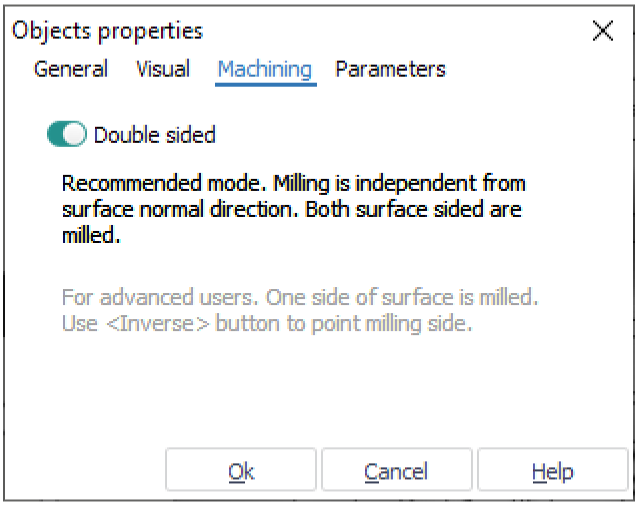
Note: *It is not recommended to set high visual tolerance on slower computers. Computer performance may be affected.*

The original visual tolerance, when loading the model, is defined in the system settings window.

See also:

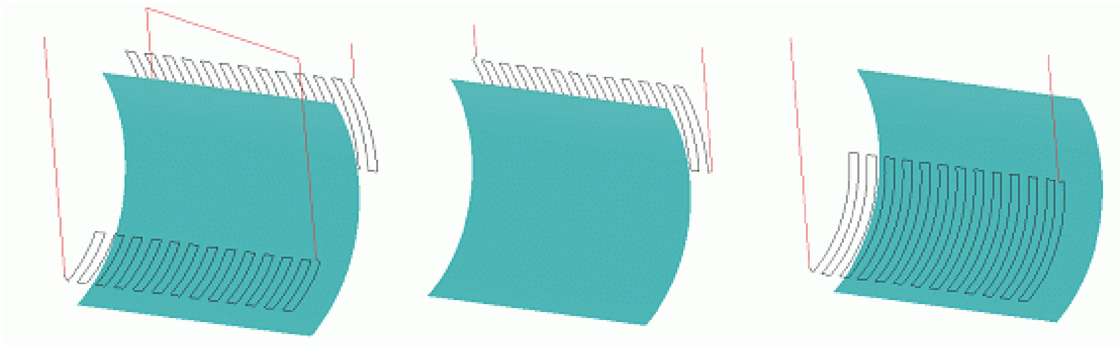
Geometrical objects properties

4.3.1.3 <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.



Double-sided (top part of the face does not permit to mill all the bottom part and the face does not have any gouge)

Single-sided (The bottom part of the face is not milled and it may be gouged)

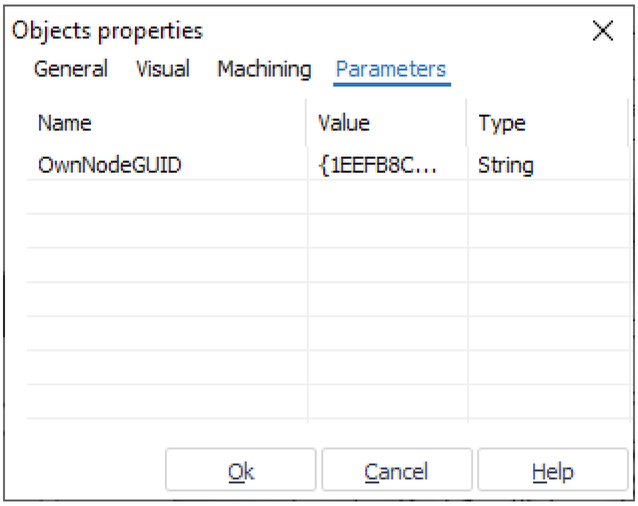
Inversed single-sided (The top part of the face is not milled and it is gouged)

Attention *: Incorrect direction of the normal vectors for non "double-sided" surfaces may cause faulty results during execution of machining operations.*

See also:

Geometrical objects properties

4.3.1.4 <Parameters> tab



On the <Parameters> tab other additional parameters are displayed. The parameters are imported or defined during designing.

See also:

Geometrical objects properties

4.3.2 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.

See also:

Geometrical model preparation

4.3.3 Delete

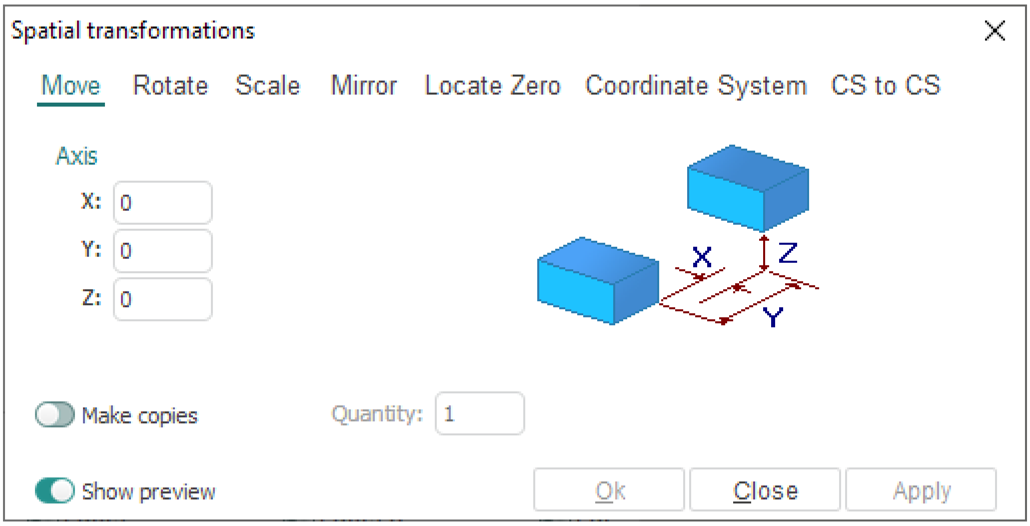
The  button deletes the selected objects. If the objects to be deleted are used in a machining operation, then confirmation will be requested. The function available on [Del] key too.

See also:

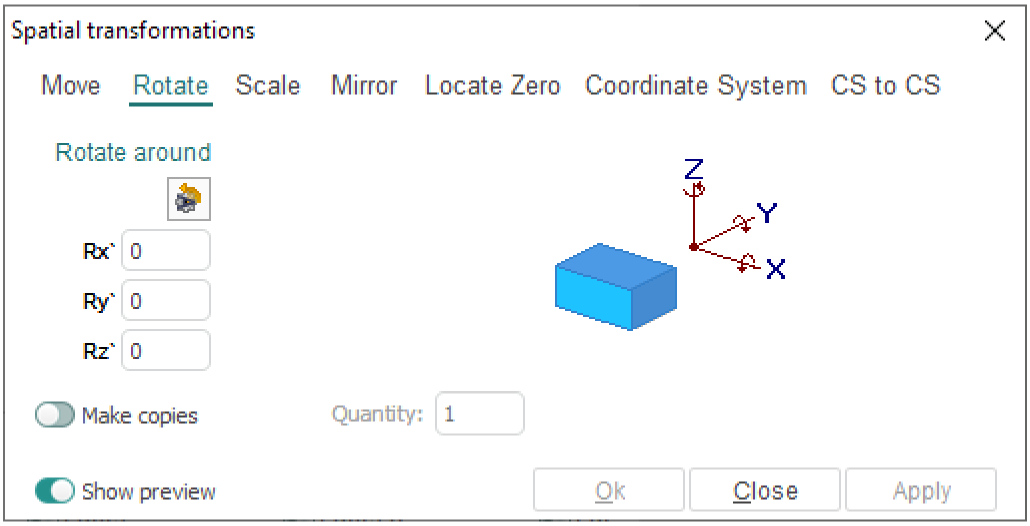
4.3.4 Spatial transformations

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.

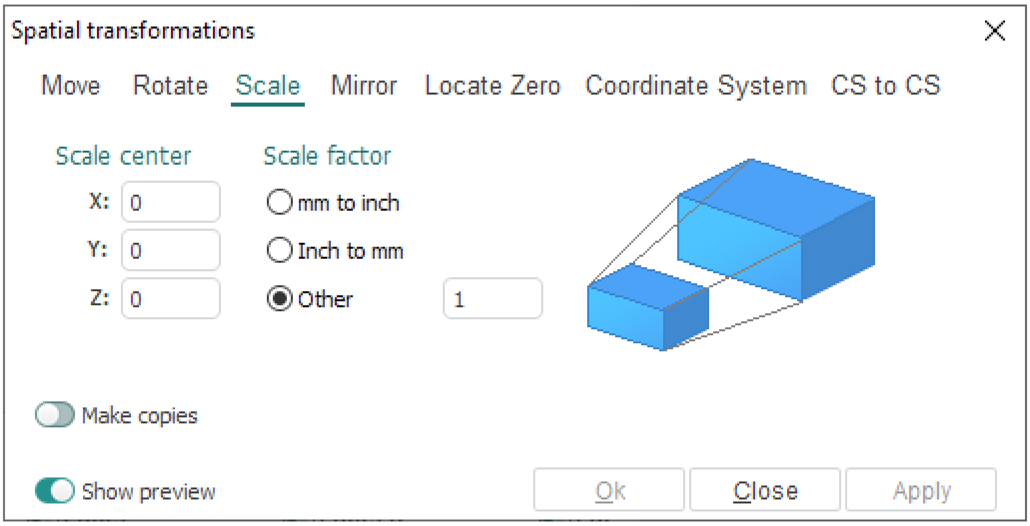
- On the <Move> page, the user can define the parallel transition of an object. In fields <X>, <Y>, <Z> setting shift values by axes. If there is no checkmark in the field <Make copies>, then the selected object will be transferred by the defined distance along each of respective axes. If the checkmark 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.



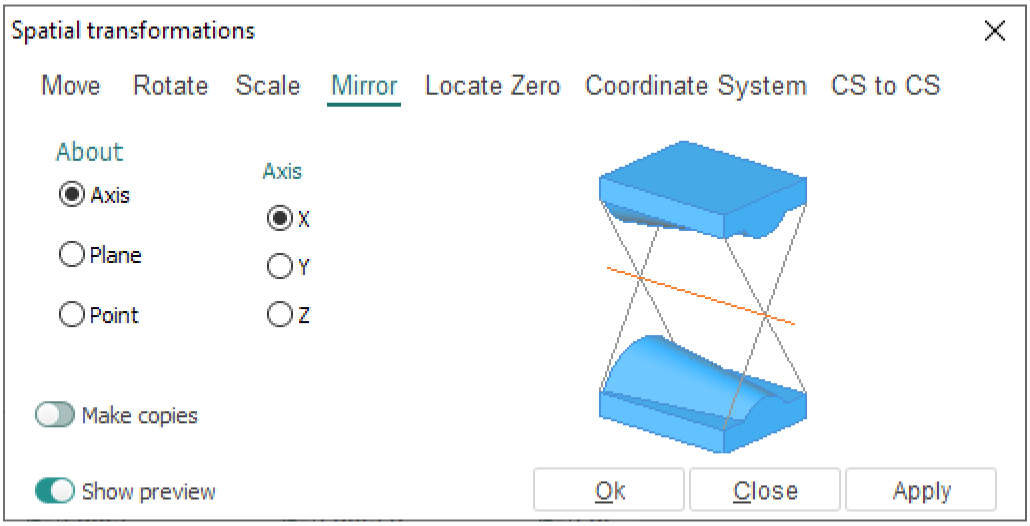
- On the <Rotate> page the user can rotate selected objects round the selected axis to the defined angle. The angle is assigned in degrees. Working with copies is incremental, that is, every subsequent copy is obtained by rotation of the previous one around the defined axis to the defined angle.



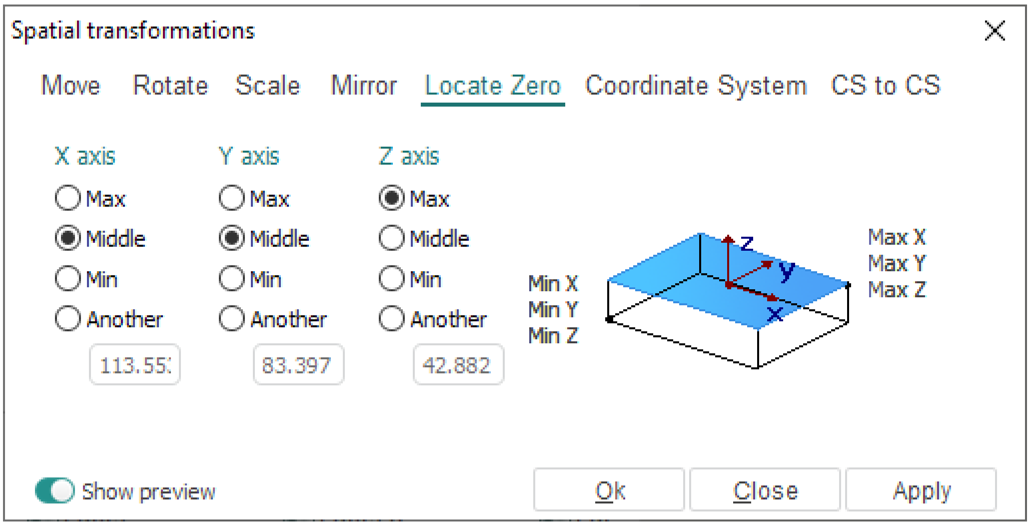
- On the <Scale> page the user can enlarge or decrease selected objects. In the field <Scale center>, the coordinates of the center point of scaling are defined. <Scale factor> can be one of two default: <mm to inch> or <inch to mm>, or arbitrary value: <Other>. When assigning a coefficient of scaling of more than one, then objects will be enlarged. If a coefficient of scaling is less than one, then objects decrease in size accordingly.



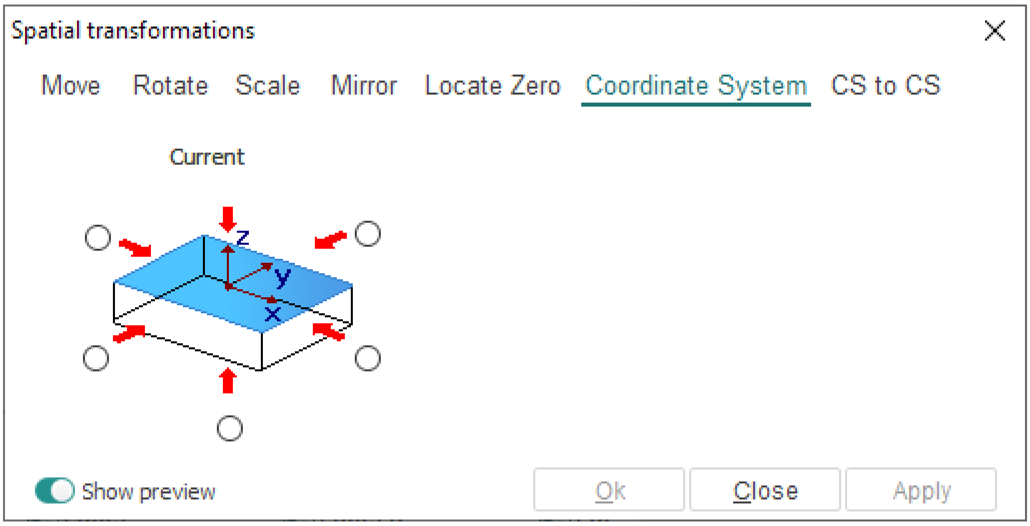
- On the <Mirror> page the user can obtain an object symmetrical to the selected one relative to an axis, plane or point.



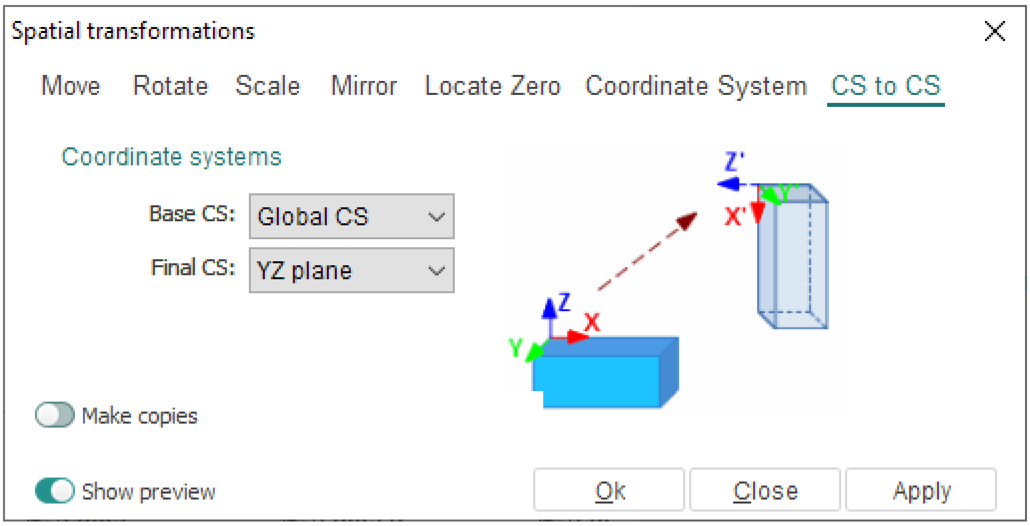
- On the <Locate Zero> page the user can perform a parallel transition of an object according to its spatial dimensions. The selected geometrical object will be shifted for such values along axes to have the transformed object relatively the current coordinate system according to the defined parameters.



- 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.

See also:

Geometrical model preparation

4.3.5 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.

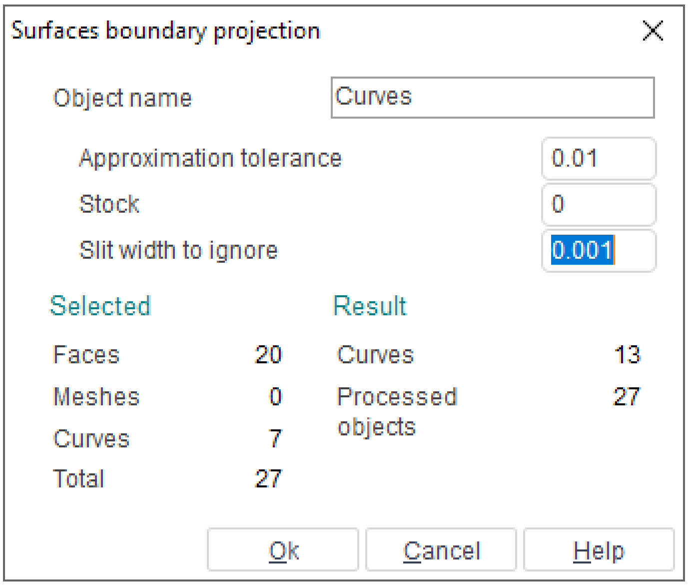
Attention: *Incorrect direction of the normal vectors for non "double-sided" surfaces may cause faulty results during execution of machining operations.*

See also:

Geometrical model preparation

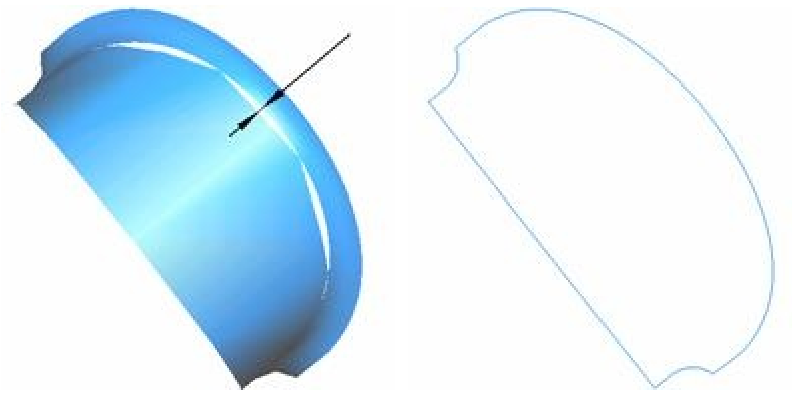
4.3.6 Outer borders projection

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.

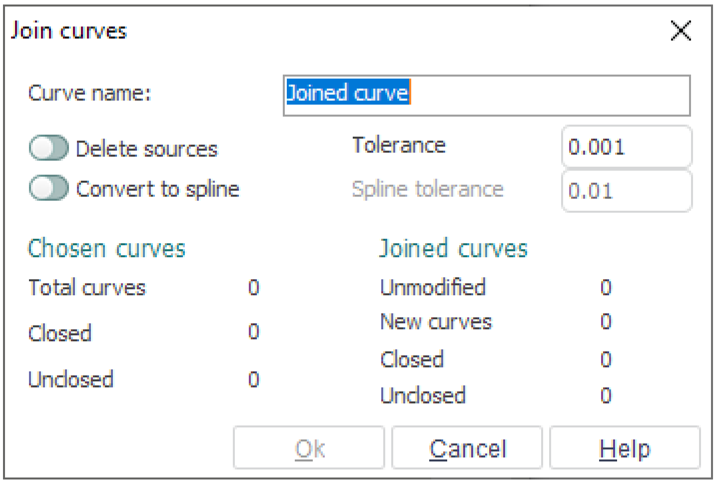
If the results of the defined parameters is correct, then the window should be closed using the <Ok> button. The boundary projection of the selected objects will be put into the active group. To cancel the projection function, press the <Cancel>button.

See also:

Geometrical model preparation

4.3.7 Curves joining

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.

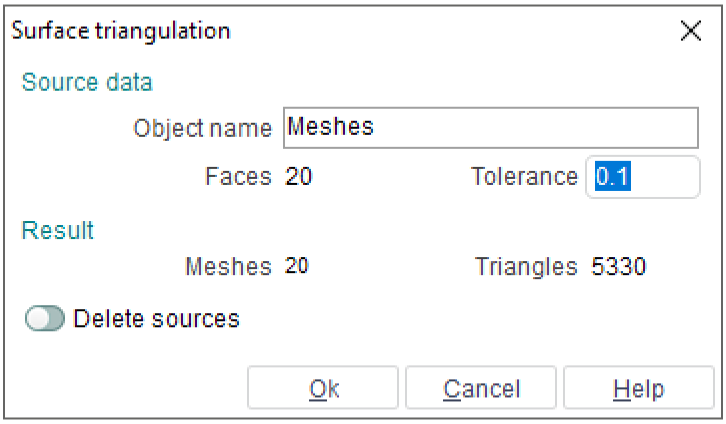
See also:

Geometrical model preparation

4.3.8 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.

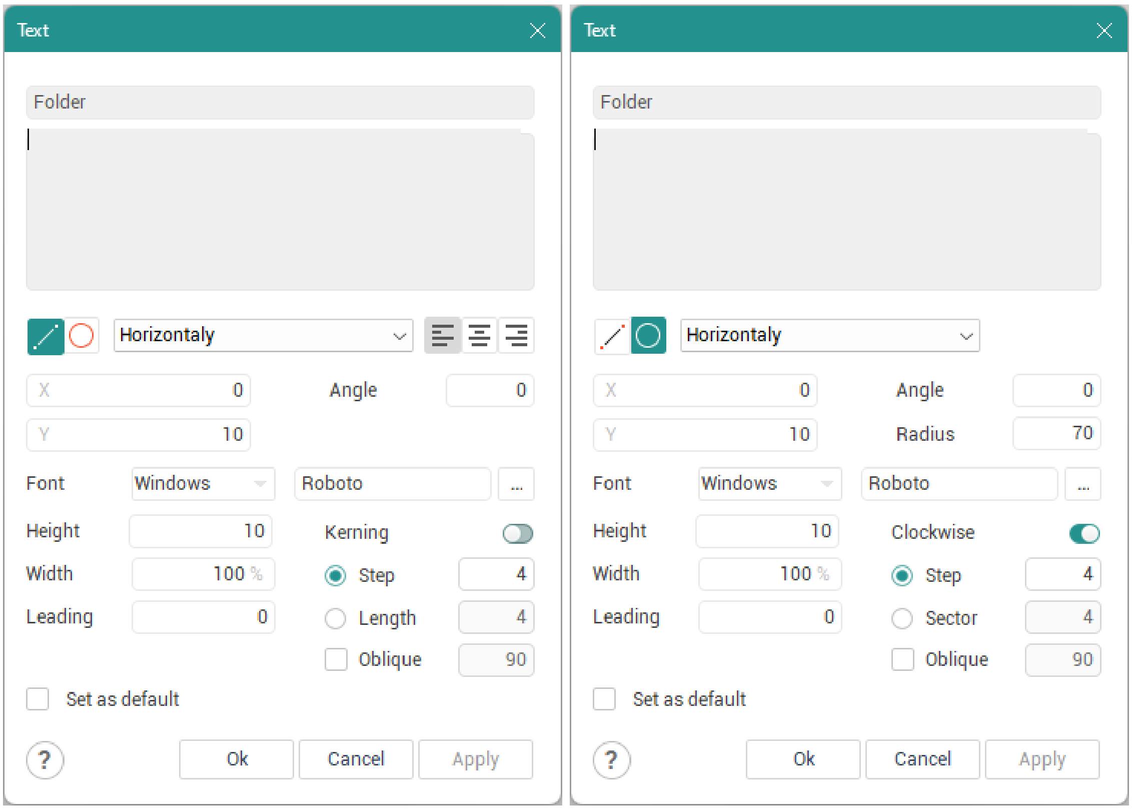
To cancel performing triangulation, press the <Cancel> button.

See also:

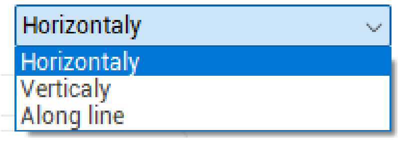
Geometrical model preparation

4.3.9 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.

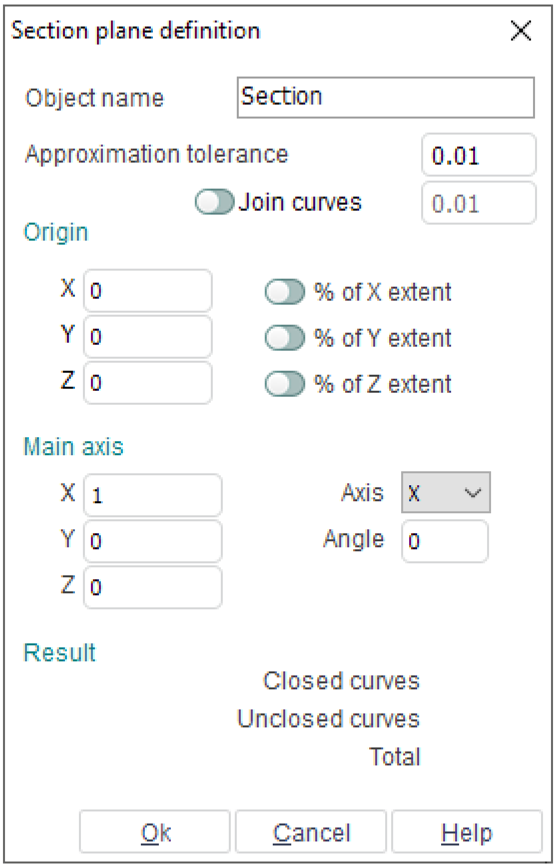
See also:

Geometrical model preparation

4.3.10 Creating sections

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.

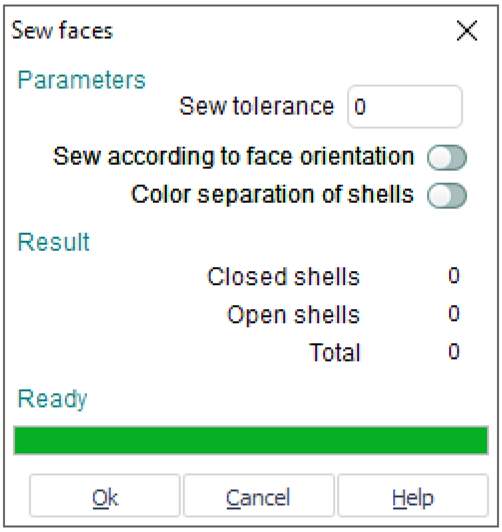
The section parameters can be edited. It is necessary to select the section and choose the <Modify Plane> item in the popup menu of the <Model> mode.

See also:

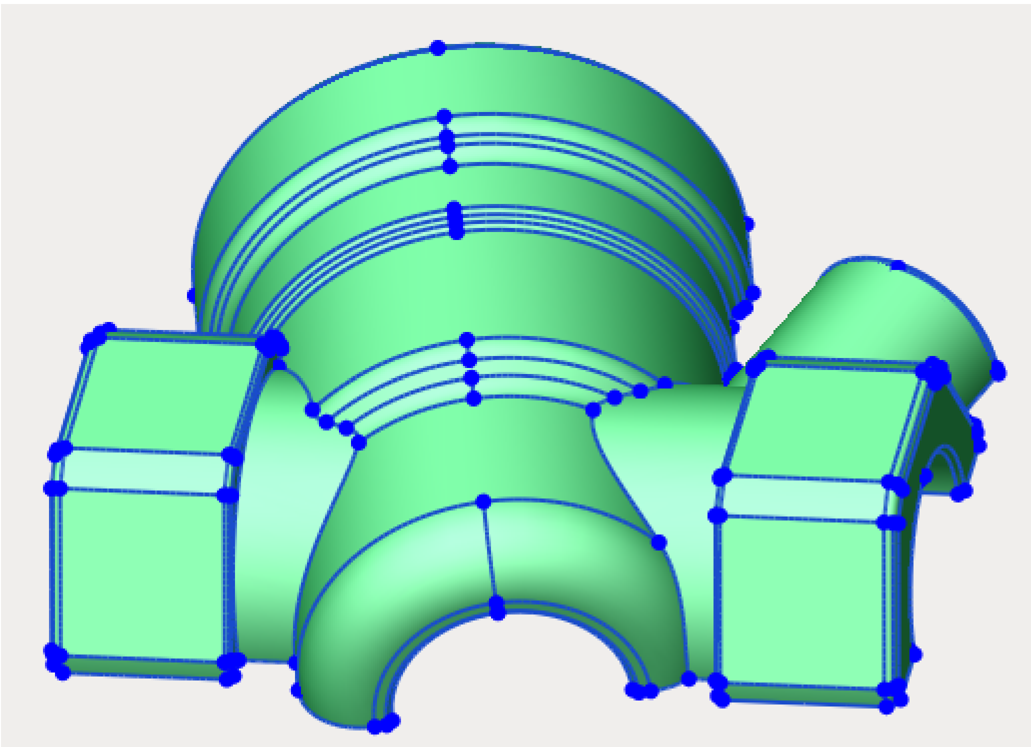
Geometrical model preparation

4.3.11 Sewing 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.

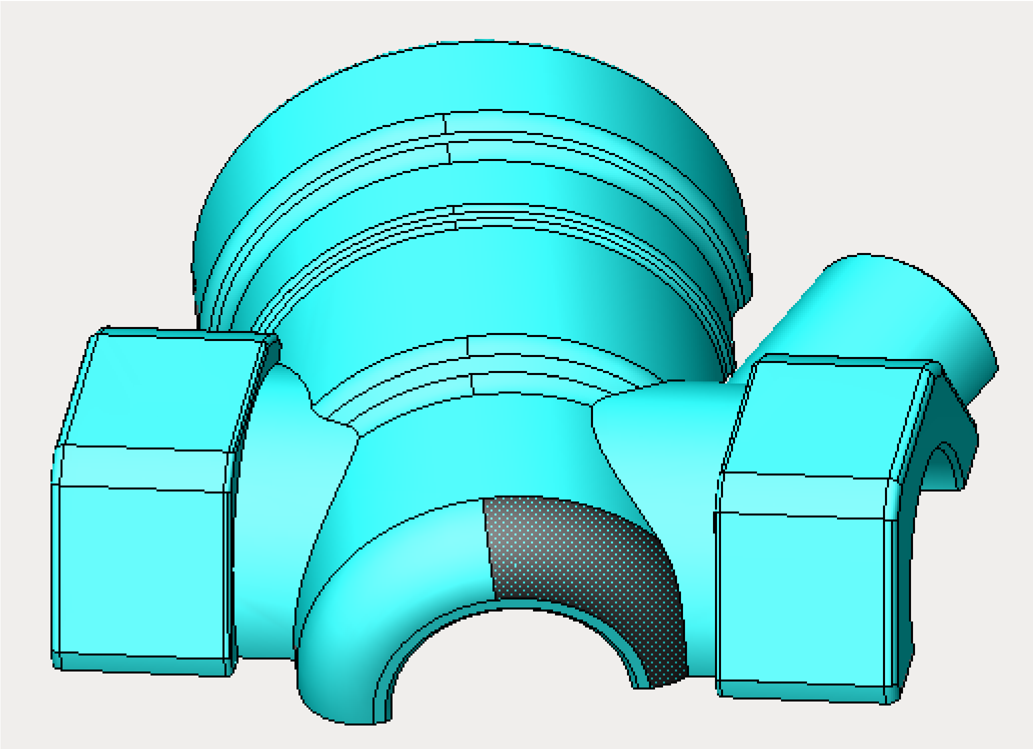


Parametric objects has some peculiar properties. They keeps permanent connection with source object. When this faces is modified, that edges and vertices will be reconstructed too. In consequence there is some limitations on actions with them. Its cannot be copied, deleted, transformed, etc. Therefore 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.

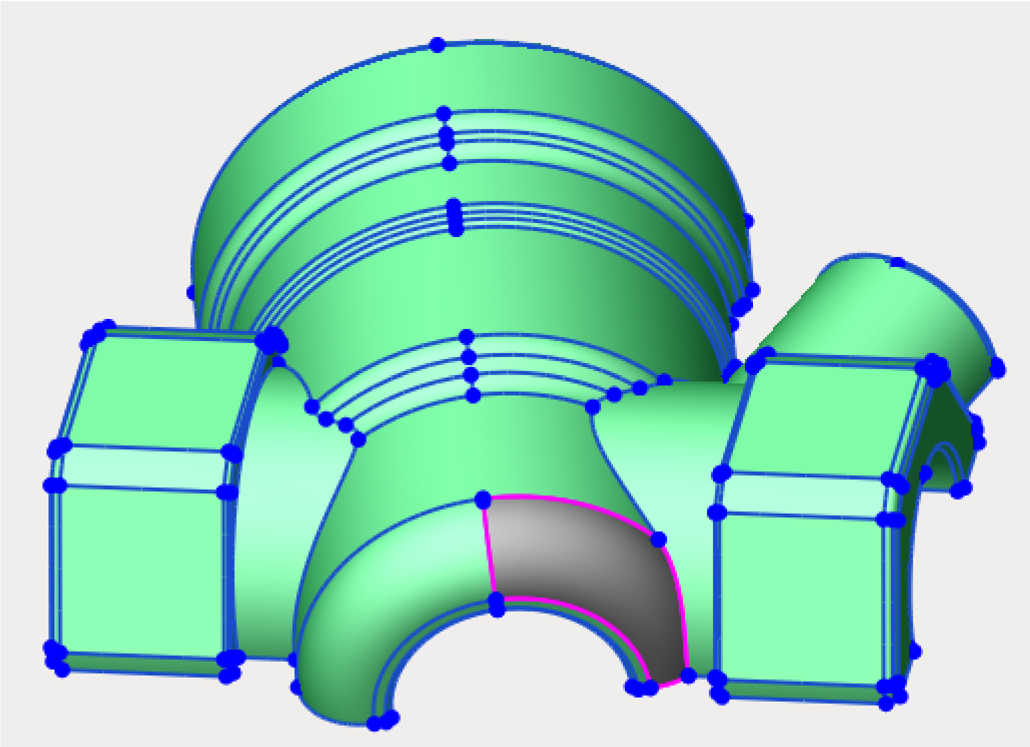
In <Sew tolerance> field assigned maximal value of deviation between edges and vertices of nearest faces. When tolerance is 0 this value taken automatically with account to overall dimensions of sewing objects.

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:

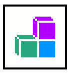


To solve these problems, the following tools were added (appear when selecting a face):

 - 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.

See also:

Geometrical model preparation

4.3.12 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 SprutCAM. Its used to save 3D models of single units of machines kinematic schemes. Extension of the file is *.osd.

See also:

Geometrical model preparation

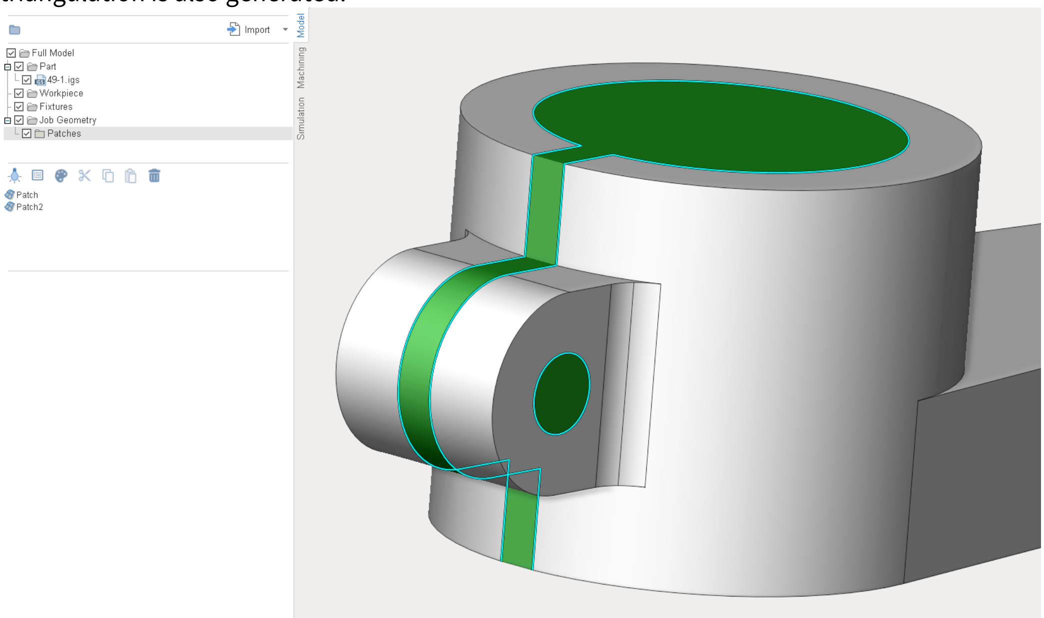
4.3.13 Patching holes

The patching holes function allows you to generate triangulation of the selected edges or meshes.

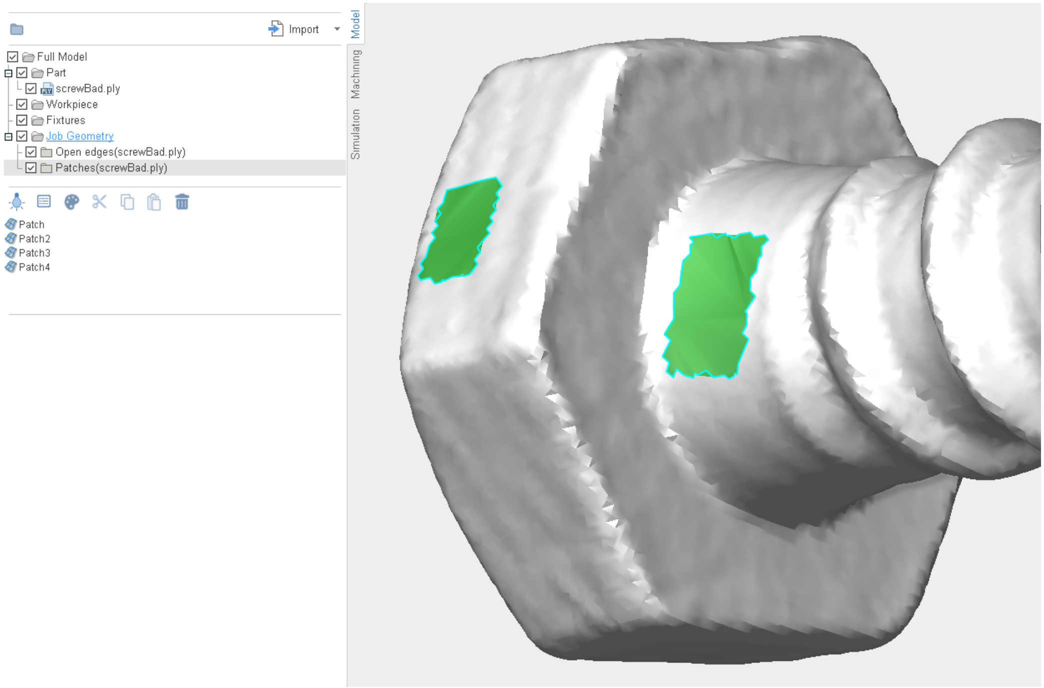
This function is called by pressing this button -  , on the “Model” page. By default, the button is not active. To activate it, you need to select edges or meshes for which you want to patch the holes in the graphics window or in the model tree.

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.



4.3.14 Extract isolines

The **Extract isolines** tool helping quickly extract isoline curves from face. 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 face for which you want to extract isolines in the graphics window or in the model tree. After activate it you can see the contours on the selected face. If you want to change the direction of the contour or make a spline, simply left-click on the selected face to open the edit button panel.

The panel has two buttons:

-  to form a spline from the isoline.
-  to change a isoline direction.

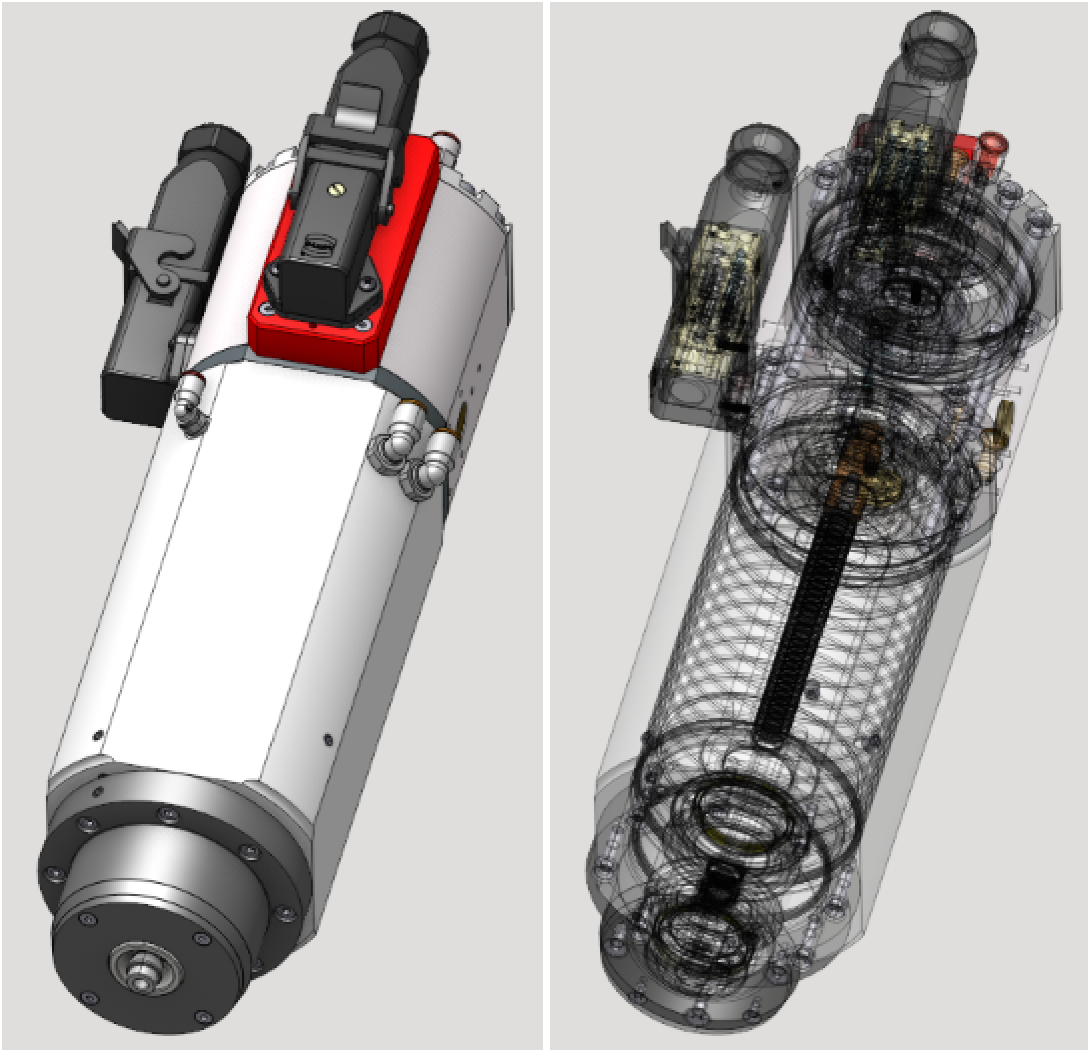
See also:

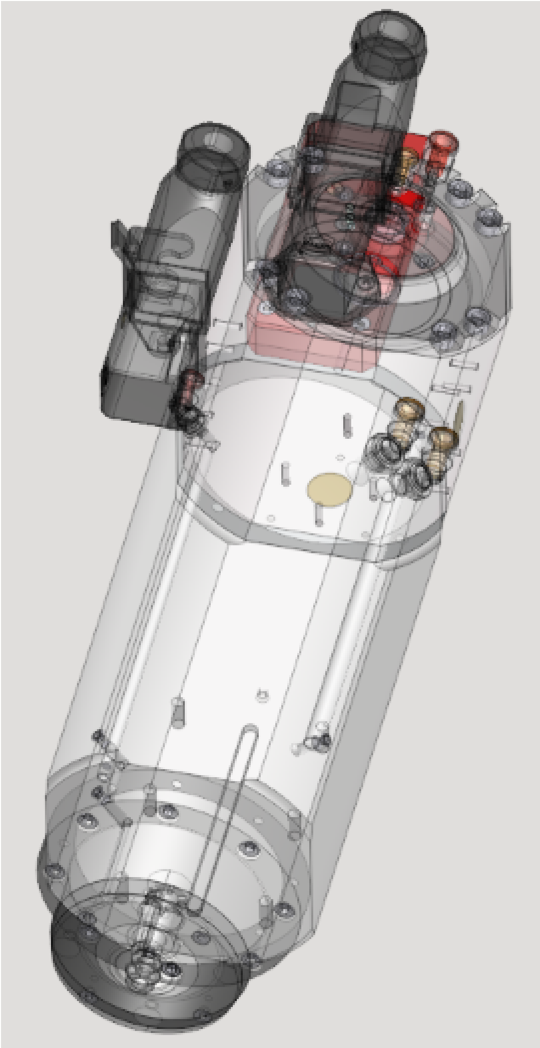
Geometrical model preparation

4.3.15 Simplifying geometrical model

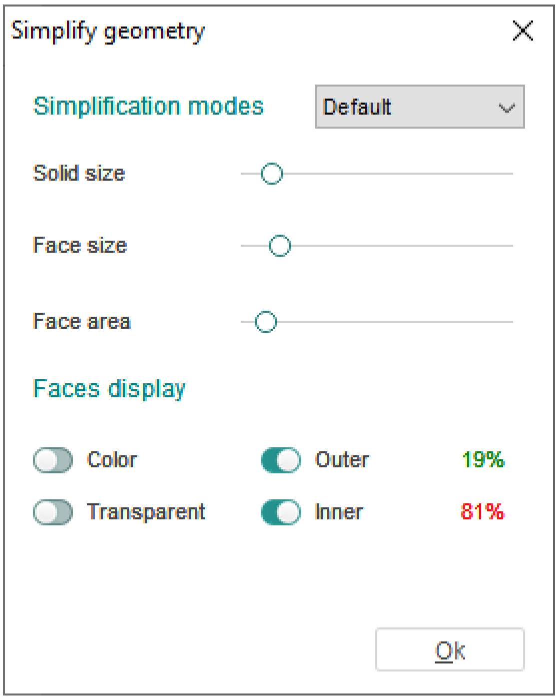
The <**Simplifying geometrical model**> function can be called from the context menu of the graphic window.

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.

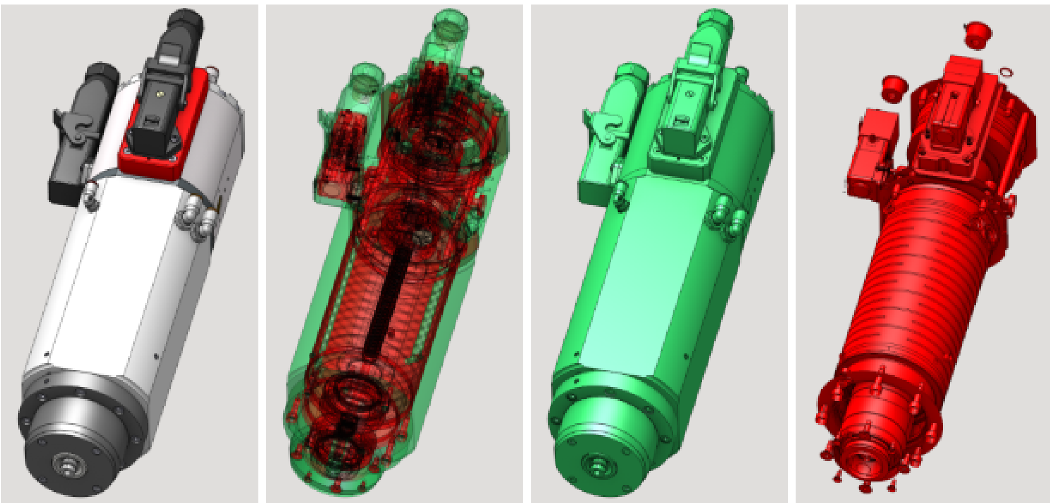
The <**Simplification modes**> block sets the parameter sets.

<**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.

4.3.16 Working with splines

4.3.16.1 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

4.3.16.2 Delete points

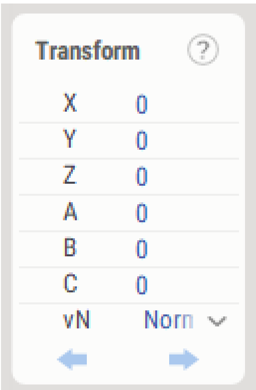
Right click on a point or press the <DELETE> key on keyboard to delete all the selected poitns

4.3.16.3 Undo and Redo

Press <Ctrl+Z> to undo the last action, press <Shift+Ctrl+Z> to redo the last action.

4.3.16.4 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.

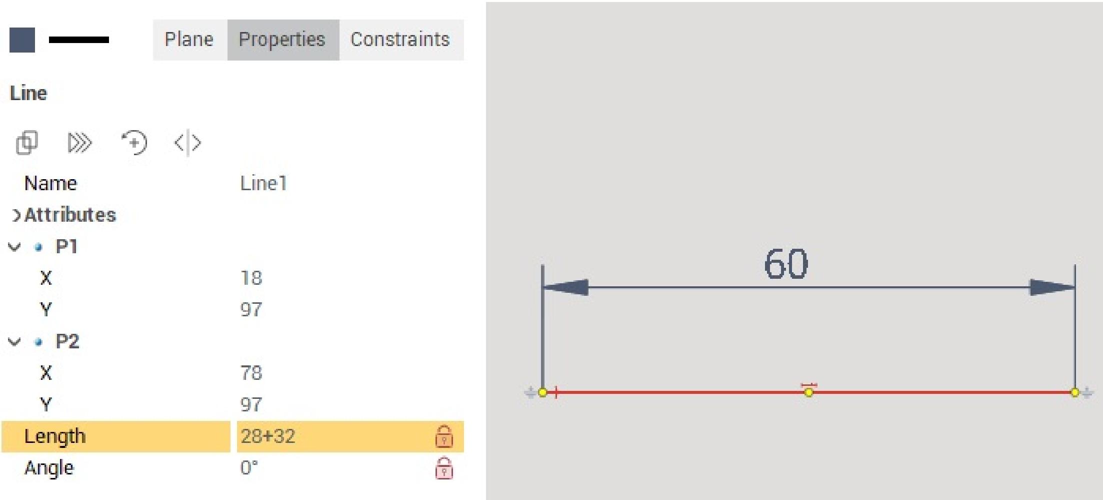


4.4 2D Geometry sketcher

4.4.1 Building and editing elements

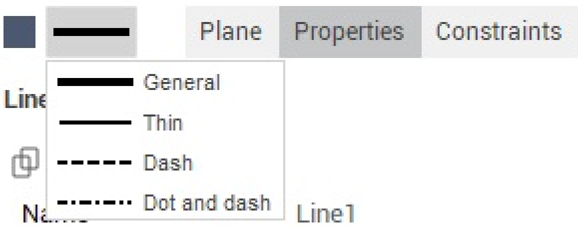
Almost all elements are built according to a typical scenario: the first nod point of the object is indicated, then the next ones and subsequent ones. When the last point is built, the construction of the object ends and the construction of the next object of the same type begins. In order to stay in 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.

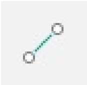


Also, the line type can be changed with a keyboard shortcuts **[Alt+1]**, **[Alt+2]**, **[Alt+3]**, **[Alt+4]**.

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.

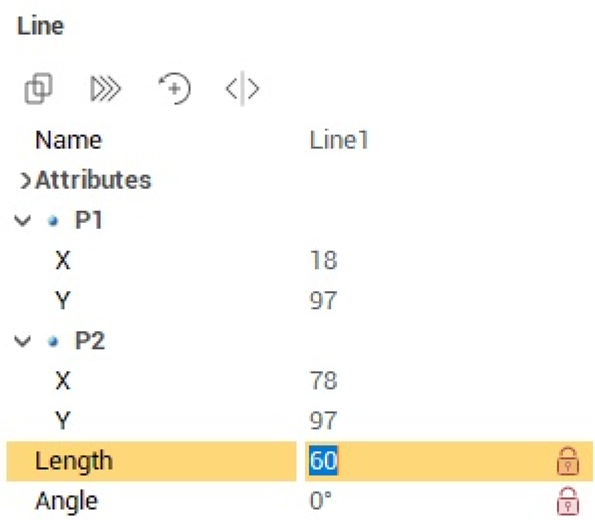
Exceptions are such elements as: chamfers, fillets, dimensions and hatching, which will be indicated in the description of these elements.

4.4.2 Line

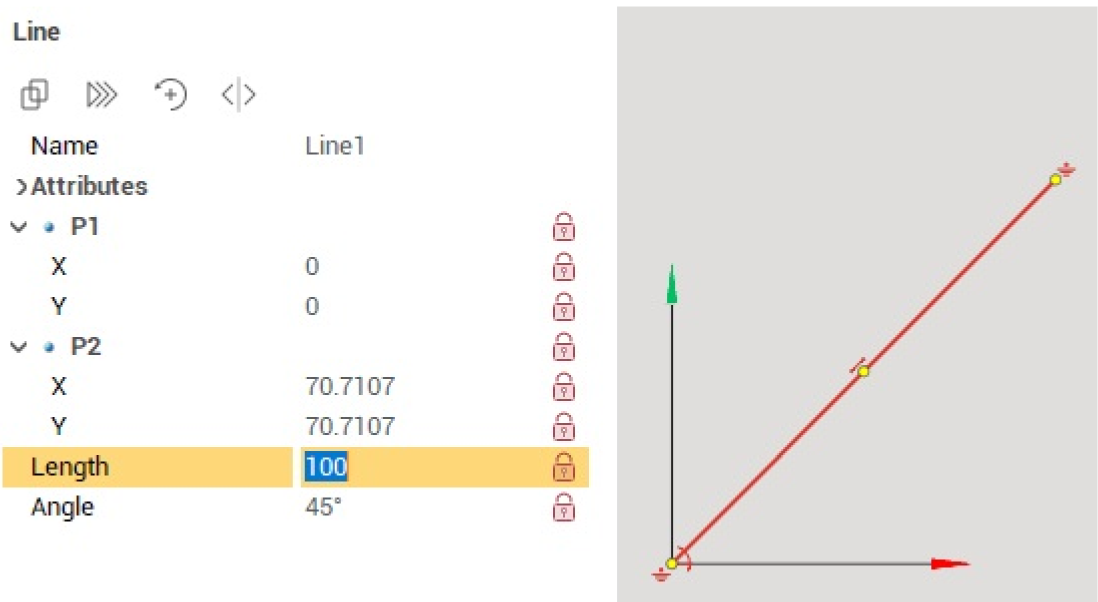


<Line> is built on two points and has a point lying on the line and divides it in half. The line is constructed by specifying the coordinates of the end points of the line in the **<Property inspector>**. Also, using the mouse, you can move the line beyond its midpoint without changing its length and 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:

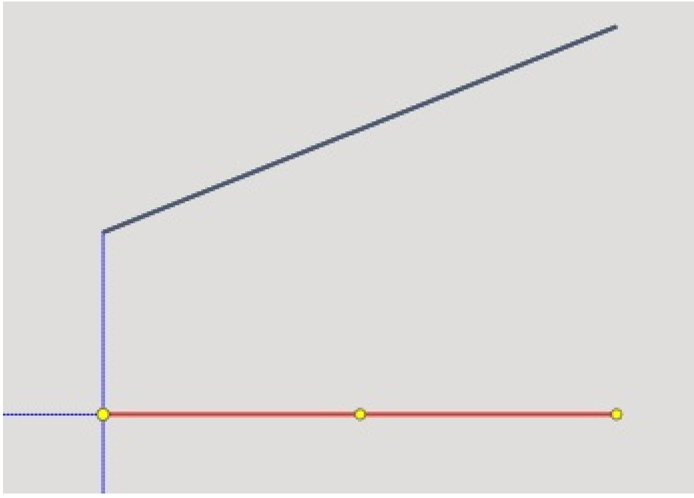


When editing a line, all these parameters can be fixed by clicking the “lock” symbol in the inspector, or the appropriate icon on the line itself.

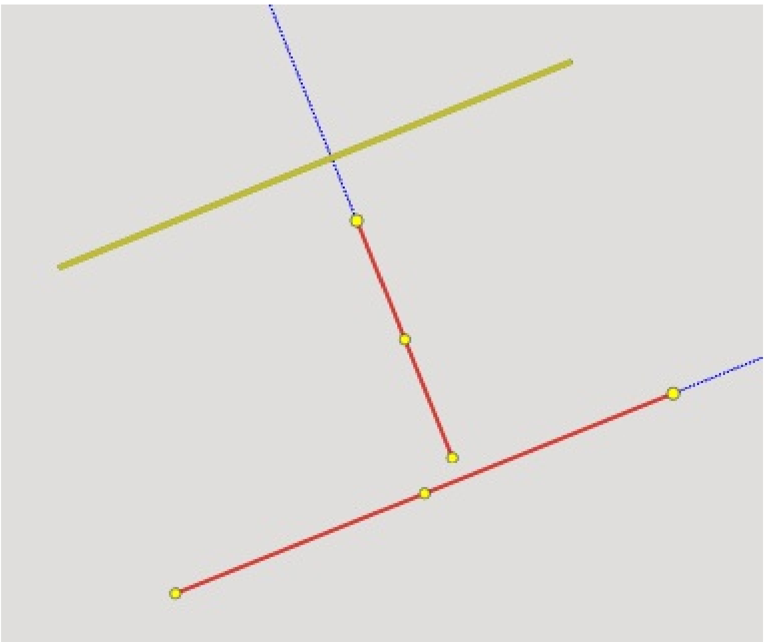


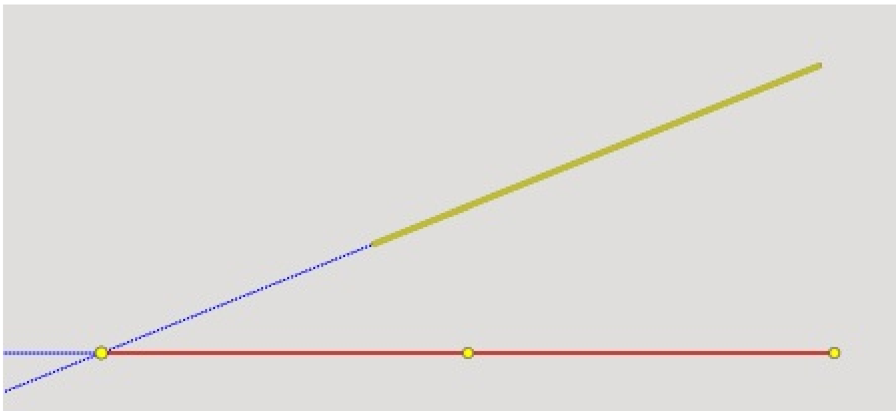
When building and editing a line, it is possible to use the help of guide lines. By default, guides are offered at 0°, 45°, 90° and the angle at which the segment was previously created in edit mode.

In addition, you can create guides by hovering over a point on a previously created object:

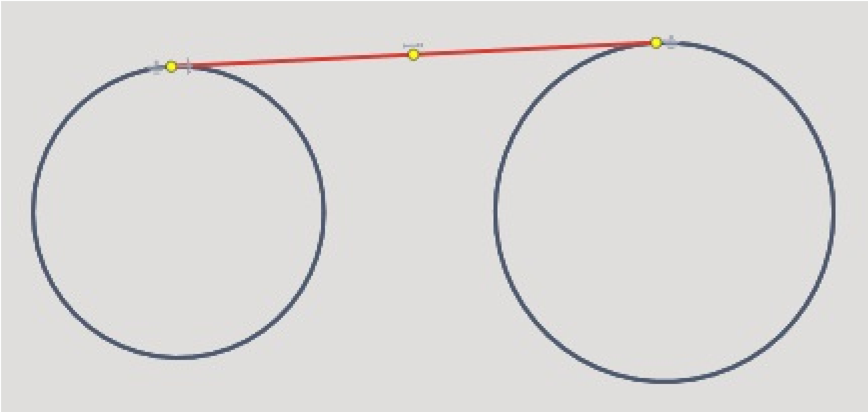


If you press the **[Ctrl]** key while drawing a line and hold the cursor over a previously created line, then perpendicular and parallel guides to this line, as well as a line that is a continuation, will be additionally proposed.





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.

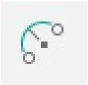


4.4.3 Arc

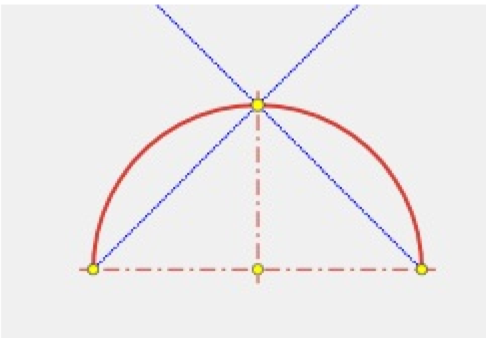
There are 3 construction options available for an arc:

- Two points and radius
- Center, starting point and angle
- By three tangents

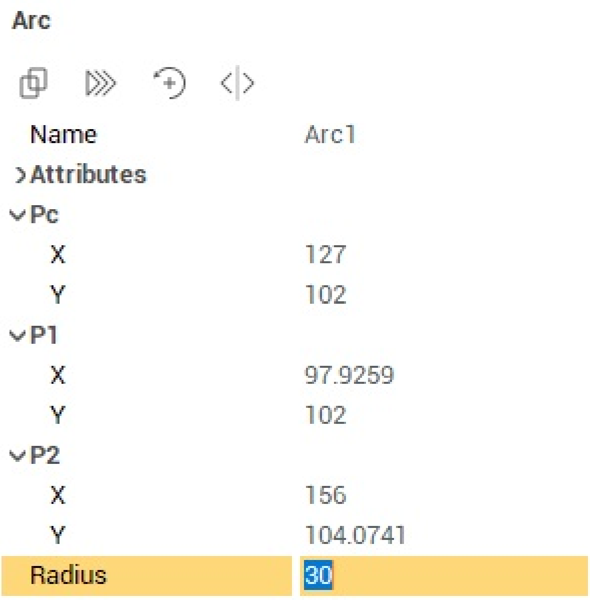
4.4.3.1 Two points and radius



To construct an arc, you must sequentially specify the first, second, and midpoint of the arc.



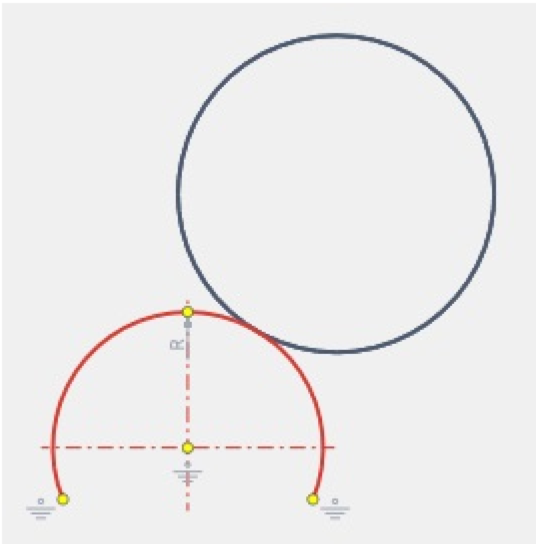
Arc parameters in the **<Property inspector>** window:



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.

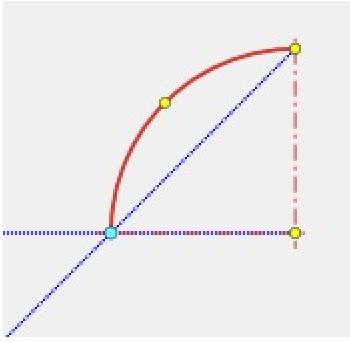


4.4.3.2 Center, starting point and angle

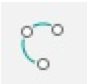
To construct an arc, you must consistently specify the center, first and second points of the arc.

Arc parameters and their fixation are completely similar to the first construction method.

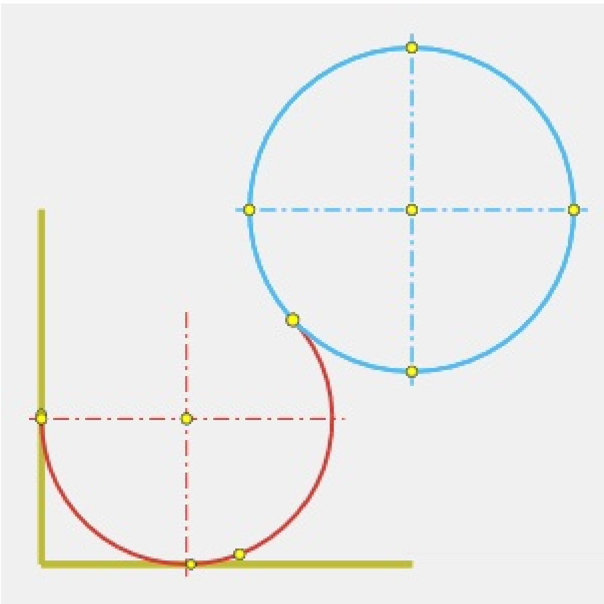
Guides are offered from the center (after specifying it), and then additionally from the first point of the arc.



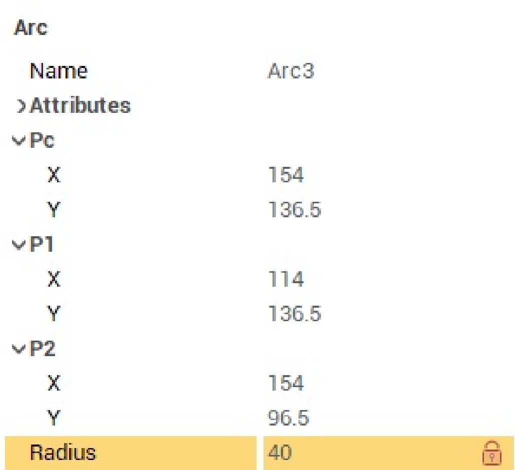
4.4.3.3 By three tangents

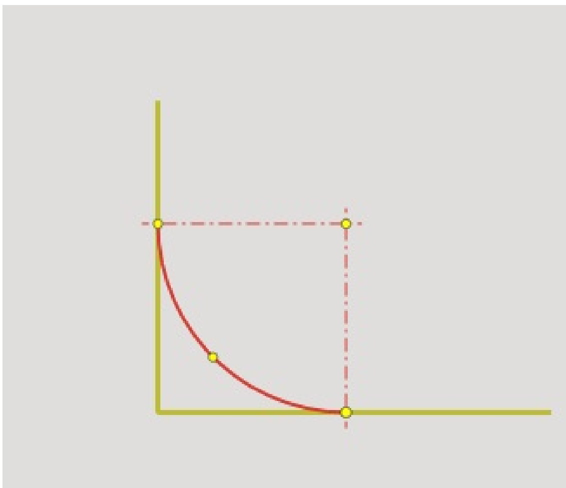


To construct an arc, you must sequentially specify three objects to which the arc should be tangent, with the **[Ctrl]** key pressed



Or specify two objects and enter the radius of the arc:

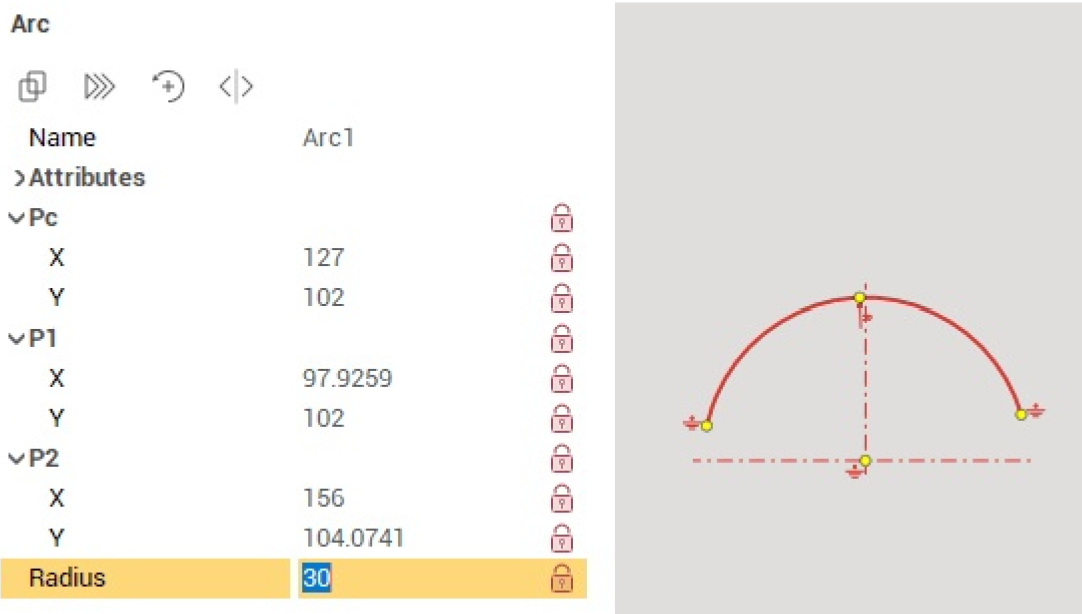




4.4.3.4 Edit arc

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.

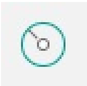
In addition, you can create guides by holding the cursor over a point on a previously created object.

4.4.4 Circle

There are 2 construction options available for a circle:

- Build Circle with Center and radius. Edit circle
- Circle by three tangents

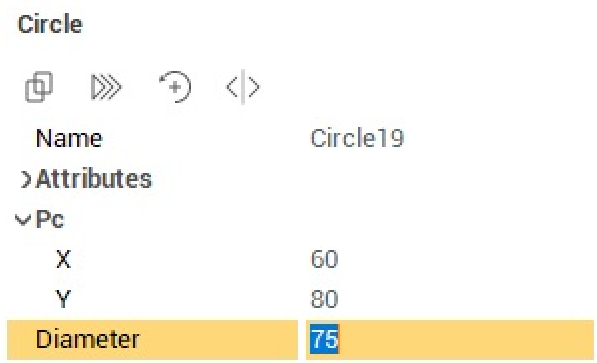
4.4.4.1 Build Circle with Center and radius. Edit circle



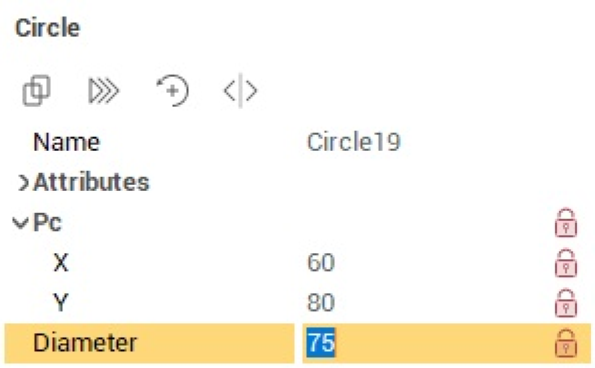
To build a circle, you need to specify the center point with the cursor, or enter the coordinates in the inspector. Specify the radius, in the same way, interactively, or by entering the diameter value in the inspector. You can also press **[R]** to switch to radius input mode or **[D]** to switch back to diameter input mode.

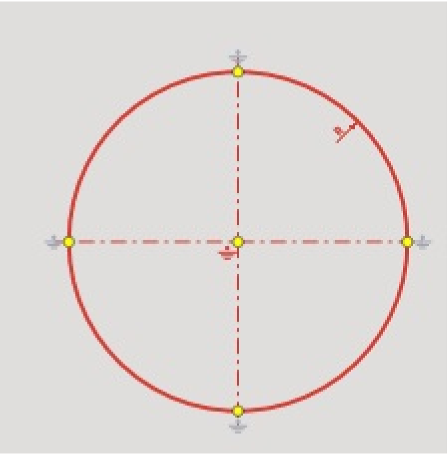
When editing a circle you can modify it directly in graphical window by moving circle points or change circle properties in inspector panel. Axis lines for the circle can be added by selecting **Line** and double-clicking on the circle.

Circle parameters in **<Property inspector>**:

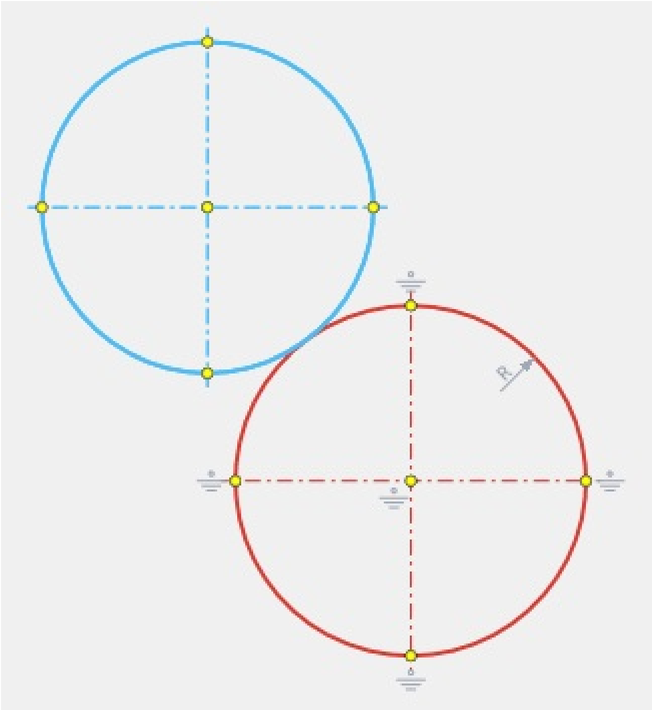


Circle parameter values can be fixed by clicking "lock" button in the editor field or by clicking constraint icon in the graphic window.





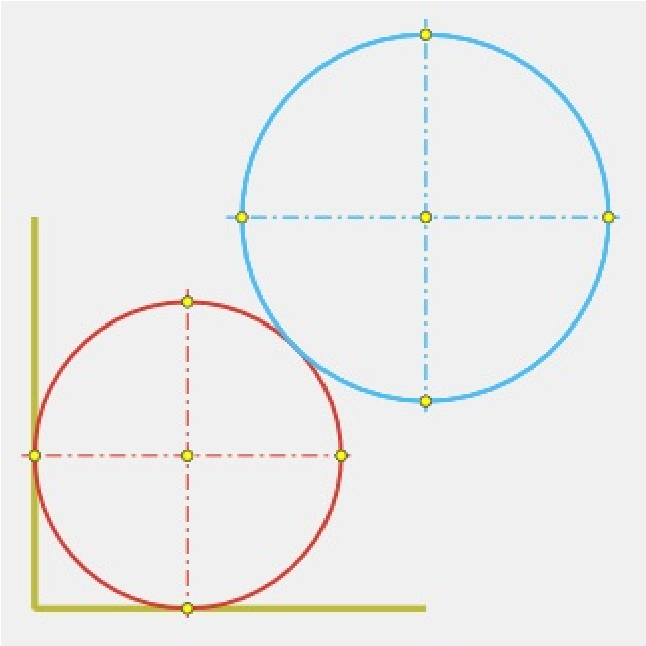
If it is necessary to construct a circle with a given center and a tangent to another object, then press the **[Ctrl]** key when specifying this object.



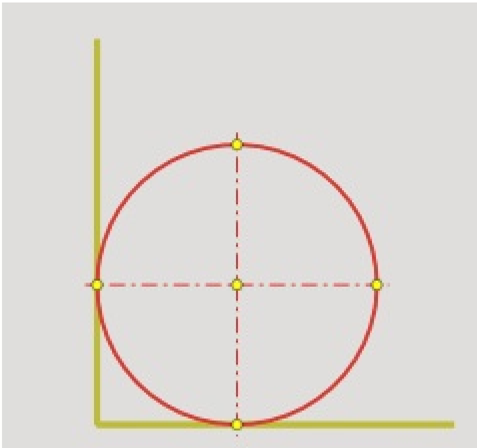
When defining or changing a circle, you can use the help of guide lines. By default, guides are offered at angles of 0°, 45°, 90° from the edited point.

4.4.4.2 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:

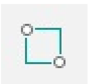


4.4.5 Rectangle

There are 3 construction options available for a rectangle:

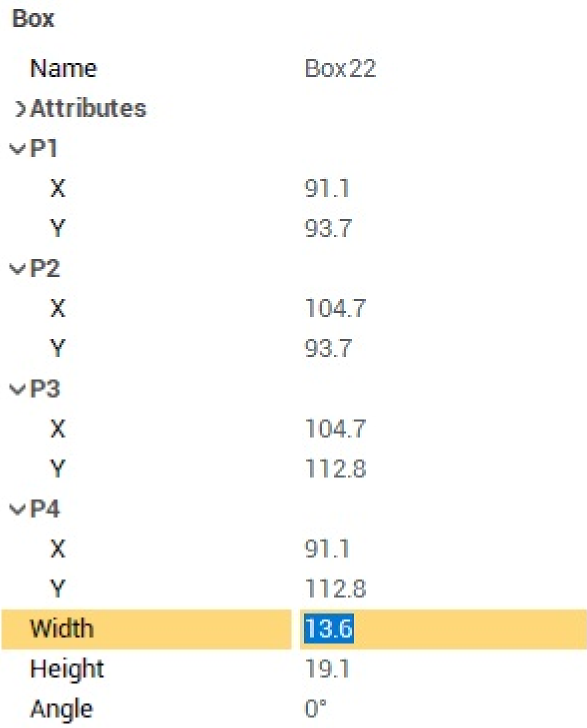
- By two points
- By side and height
- By center and point

4.4.5.1 By two points



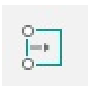
The rectangle is built by specifying two diagonal points interactively, or by specifying the coordinates of the first point and the width/height of the rectangle in the inspector. You can rotate a rectangle only by changing the angle of rotation in the inspector.

Rectangle options in the **<Property inspector>** window:

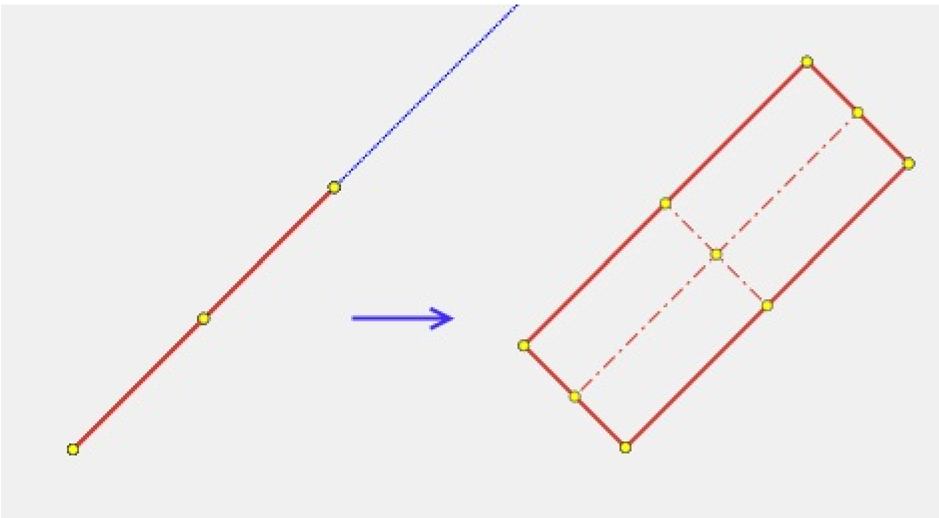


When defining a rectangle, you can use the help of guide lines. By default, guides are offered from the point being edited and the opposite point at angles of 0°, 45°, 90° and along a straight line passing through these points.

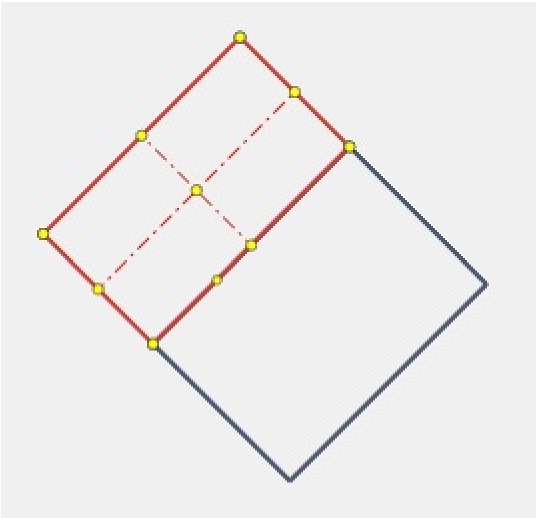
4.4.5.2 By side and height



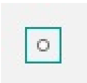
First, one side of the rectangle is specified, and then it is stretched in height. This allows you to build rectangles at the angle you need.



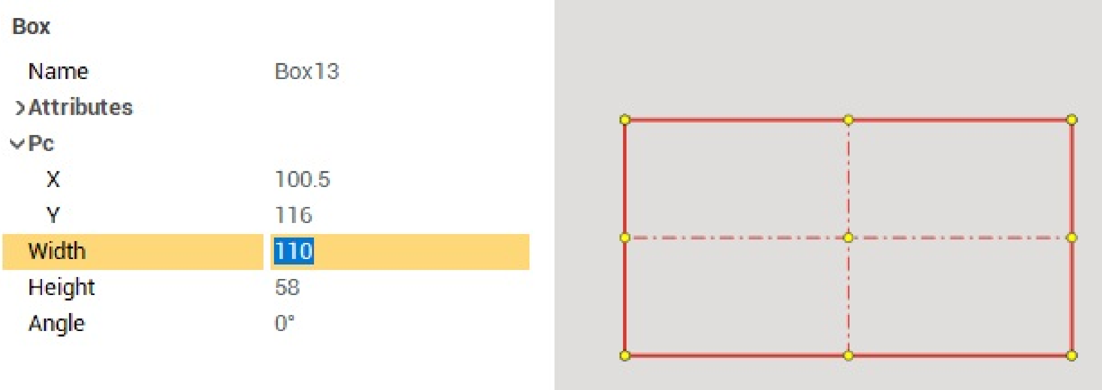
You can also specify an existing line and the side will be copied from this line.



4.4.5.3 By center and point



In this method, the center of the rectangle is specified first, and then one of the points of the rectangle.

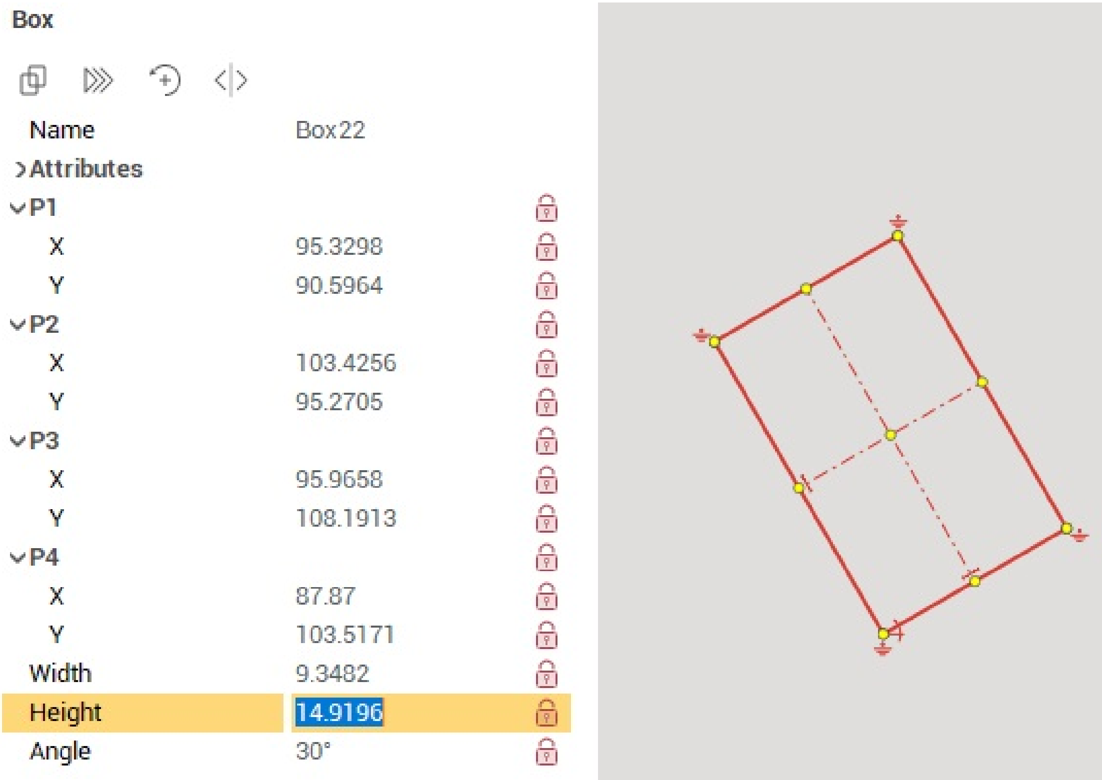


The inspector displays this center, as well as the width, height, and rotation of the rectangle.

4.4.5.4 Edit rectangle

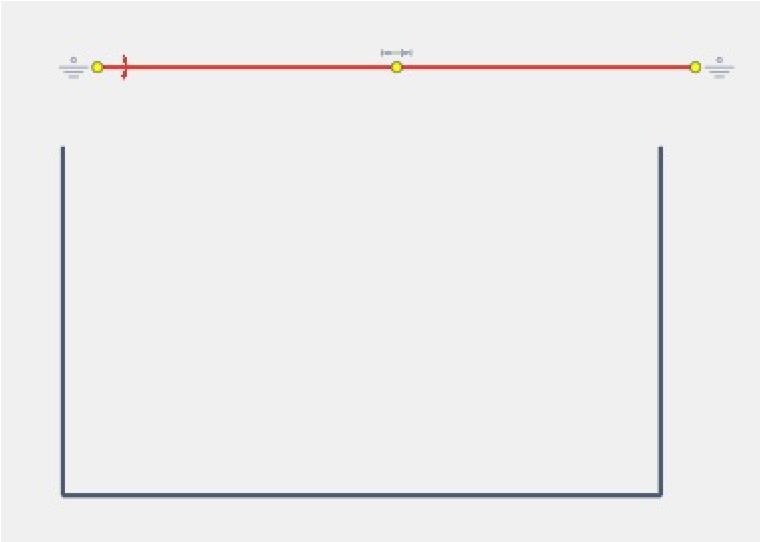
Selected rectangle object can be modified in graphical window or you can change it's parameters in the inspector.

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.

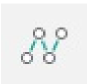


4.4.6 Contour

A contour is a set of lines and/or arcs connected by extreme points. There are two options for constructing a contour:

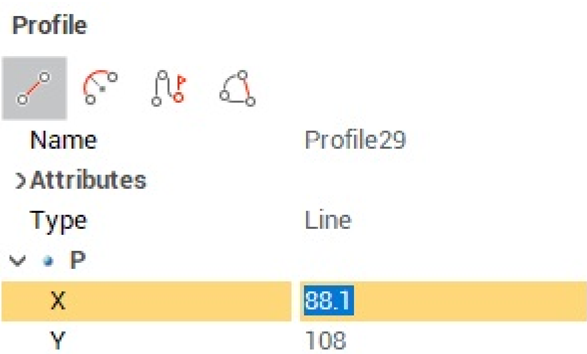
- Build by element
- By zone or border

4.4.6.1 Build by elements

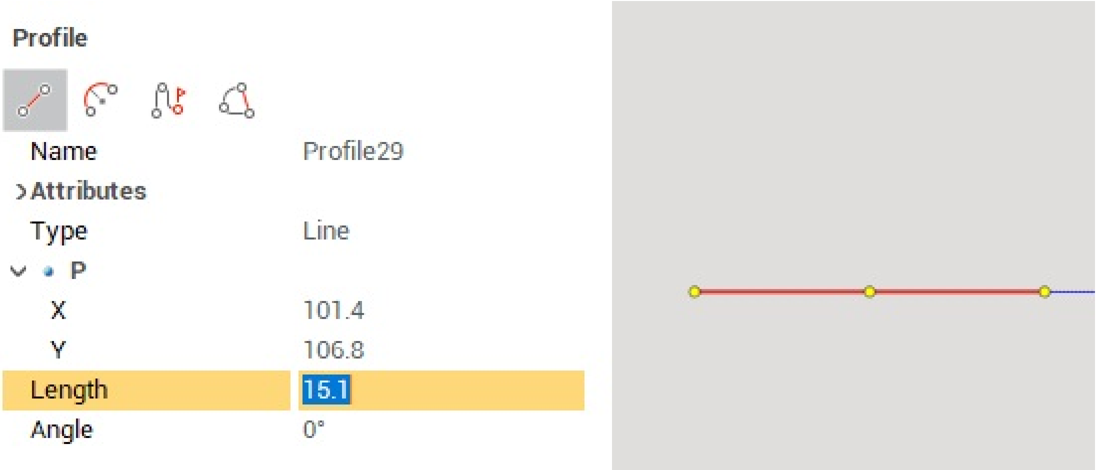


The building is carried out by successive construction of lines/arcs by specifying points interactively or through the inspector.

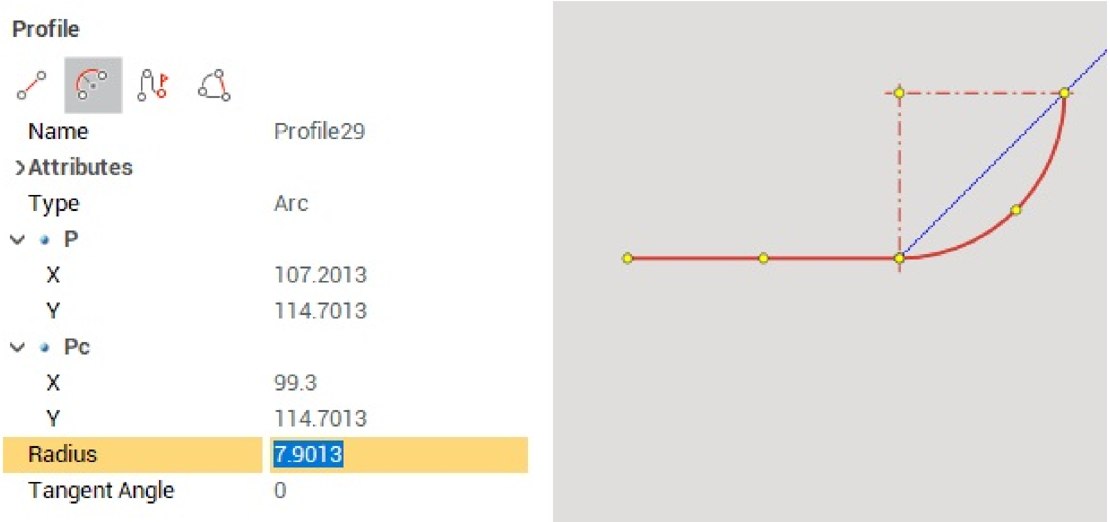
At the first step, you need to specify the starting point of the contour, or enter its coordinates in the inspector.



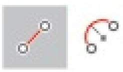
Next, you also need to specify the next point, or enter the length and angle of the line.



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:



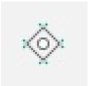
Or by pressing **[Ctrl+Space]**.

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.

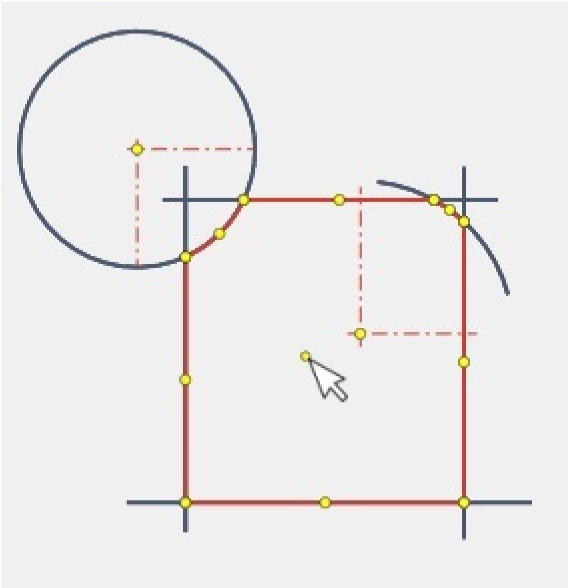
You can also end the construction by double-clicking the mouse button when building the last element

4.4.6.2 By zone or border

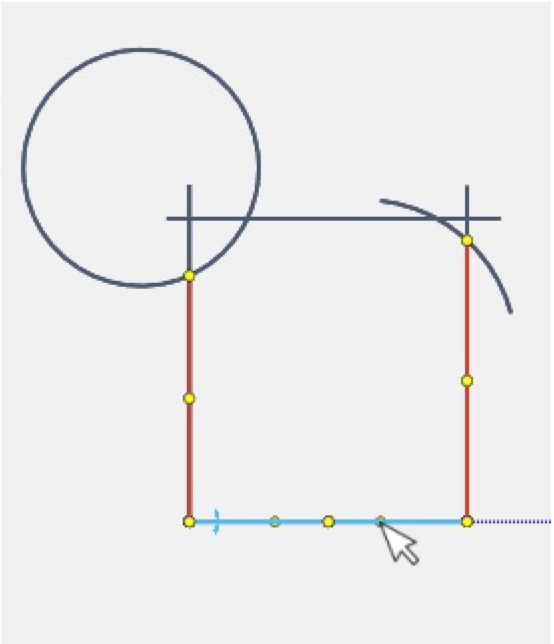


In this method, the contour is built on the already existing elements.

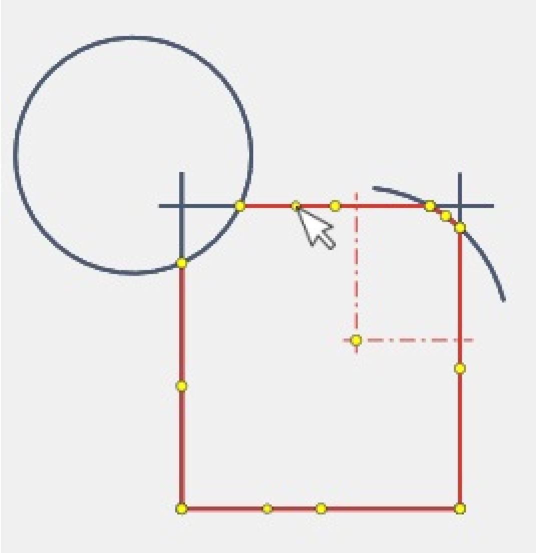
When specifying a point inside a group of elements, a cont will be created. The components of this contour are located in this zone.



When selecting the contour boundary, its elements will be created automatically along the chain to the nearest intersection with another object.



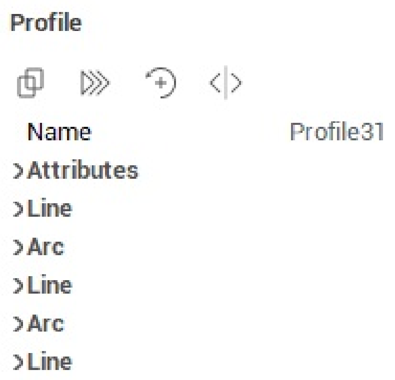
To continue the contour, select its next element. If the specified element is not connected to the previously built one, the system will try to complete it along the chain.

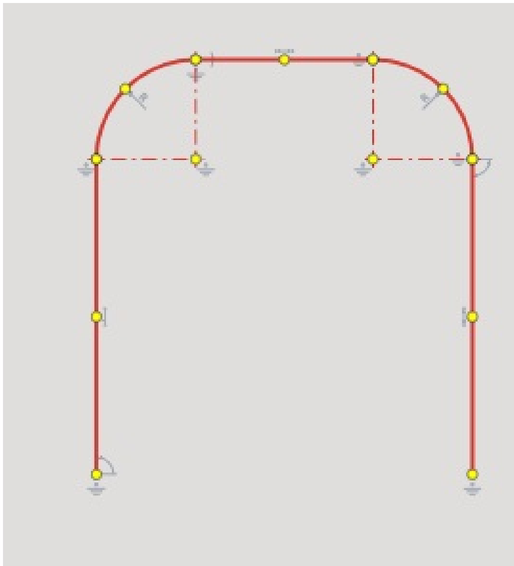


4.4.6.3 Edit contour

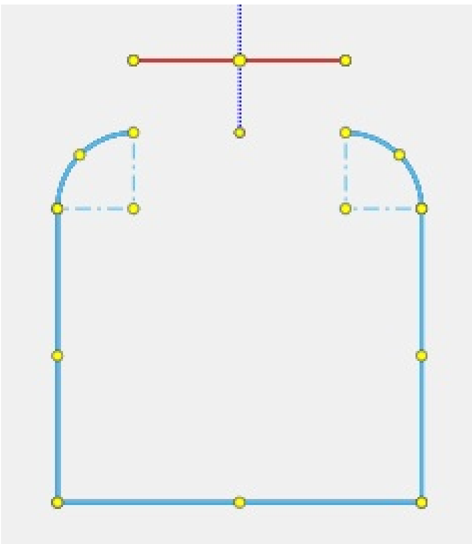
Selected contour can be modified by changing it's points in graphic window. You can change contour elements properties in the inspector.

In edit mode, the contour will be presented as a set of lines and arcs.



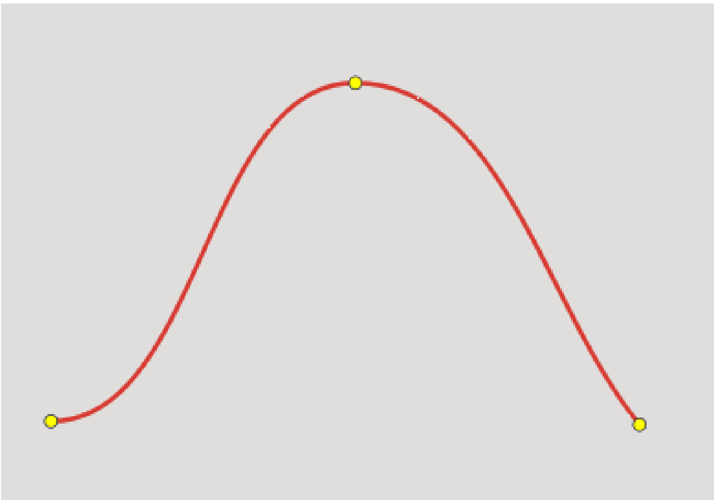


If you select the side of the contour with pressed **<Ctrl>,** only the highlighted contour element will be selected. If you separate an element from a contour the contour will be split into elements.



4.4.6.4 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.

Spline

Name	Spline4
> Attributes	
Spline precision	0.1
Firts point tangential	90°
Last point tangential	45°
Number of points	3
⌄ Spline points	
⌄ Point	
X	467
Y	278
⌄ Point	
X	523
Y	351
⌄ Point	
X	621.5
Y	320.5

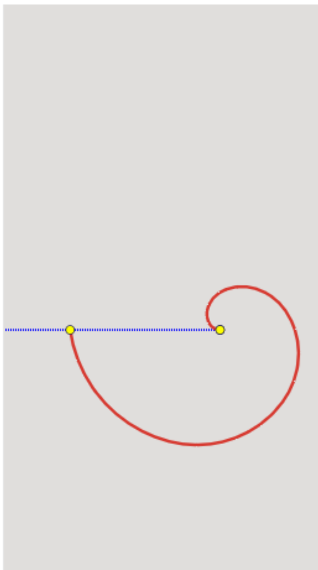
Click or double-click at last point to finish spline.

4.4.6.5 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.

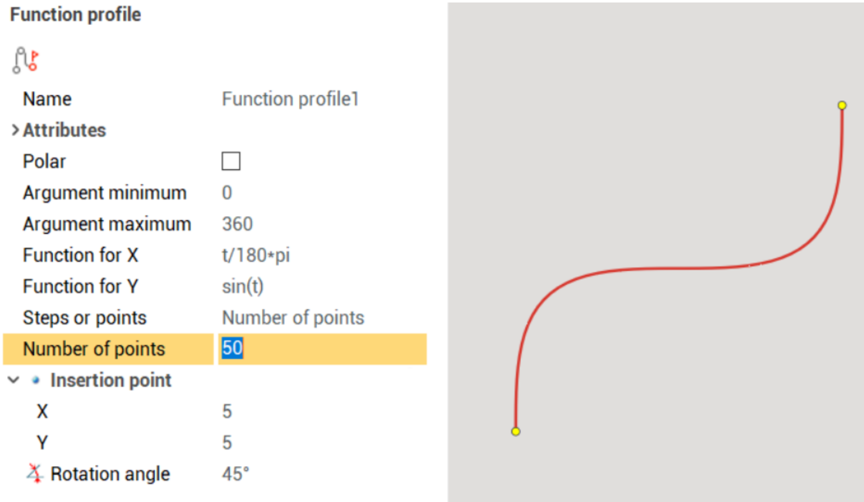
Archimedean spiral

Name	Archimedean spiral1
> Attributes	
Polar	☐
Counter clockwise	☑
Deviation	0.1
Number of loops	1
⌄ Center point	
X	211
Y	90
⌄ Starting point	
X	157
Y	90
⌄ Finishing point	
X	211
Y	90



4.4.6.6 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:

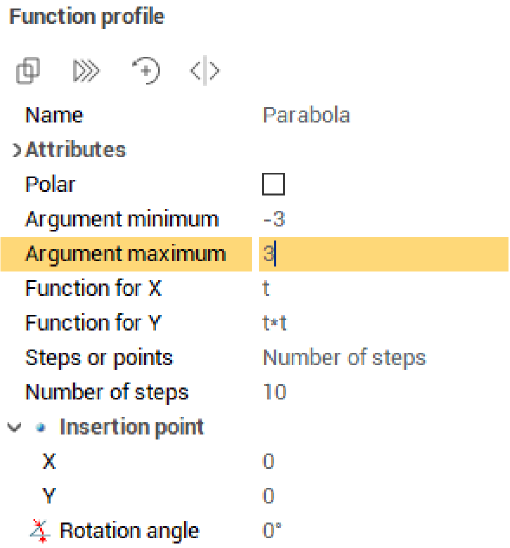
- sin(t) - sine of **t**
- cos(t) - cosine of **t**
- tan(t) - tangent of **t**
- cotan(t) - cotangent of **t**
- arctan(t) - arc-tangent of **t**

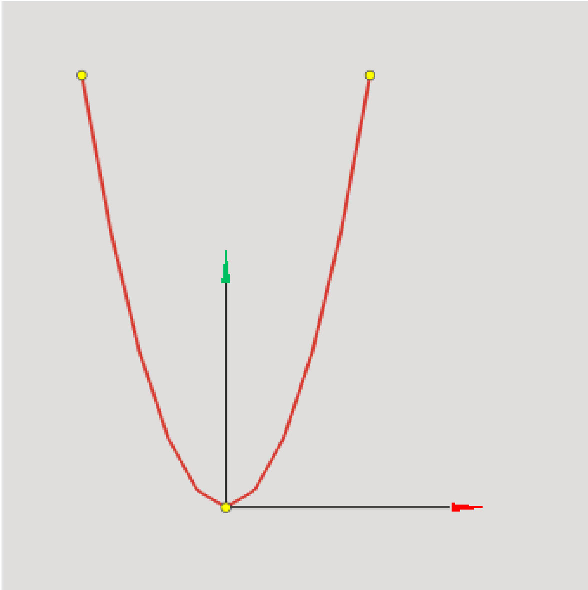
For trigonometrical functions the argument is in radians. **Pi** is the **π** number.

- exp(t) - **e** raised to the power of **t**
- ln(t) - natural logarithm of **t**
- sqrt - square root of **t**

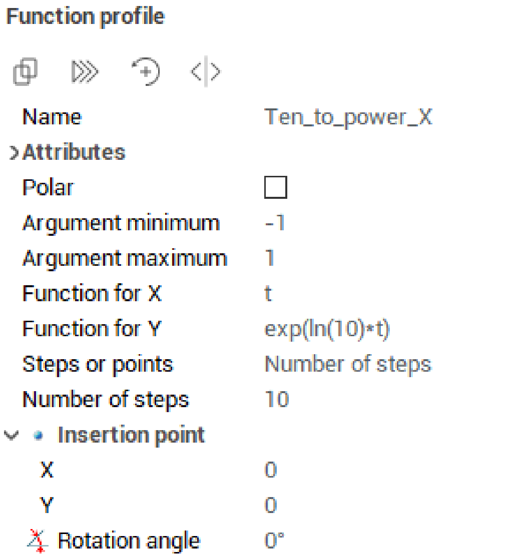
Examples of function profiles

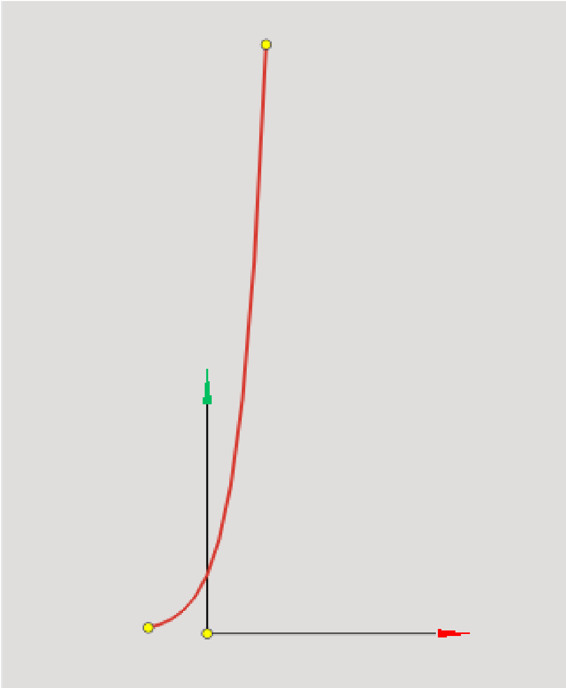
Quadratic function (parabola): $Y = X^2$





Exponential: $Y = 10^X$

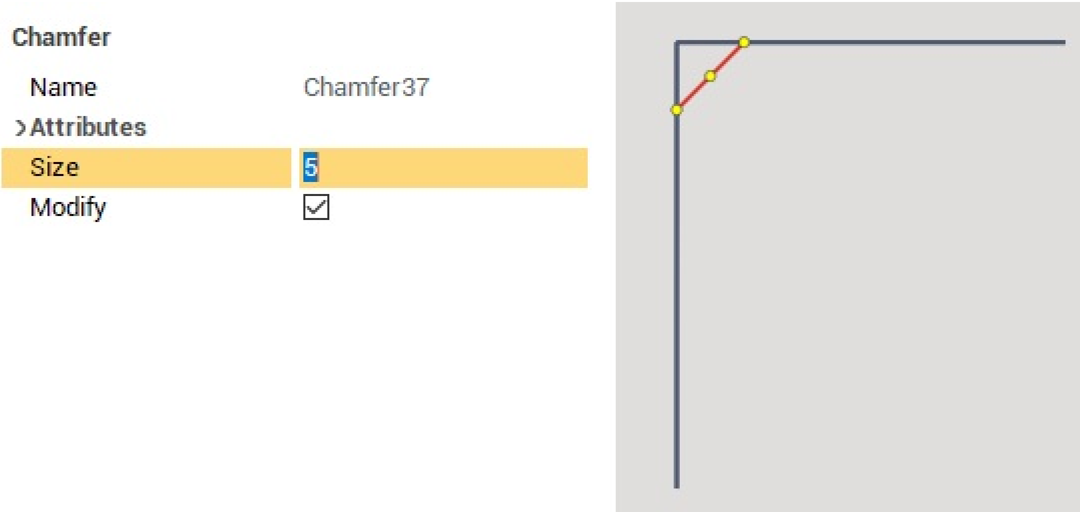




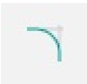
4.4.7 Chamfer and Rounding

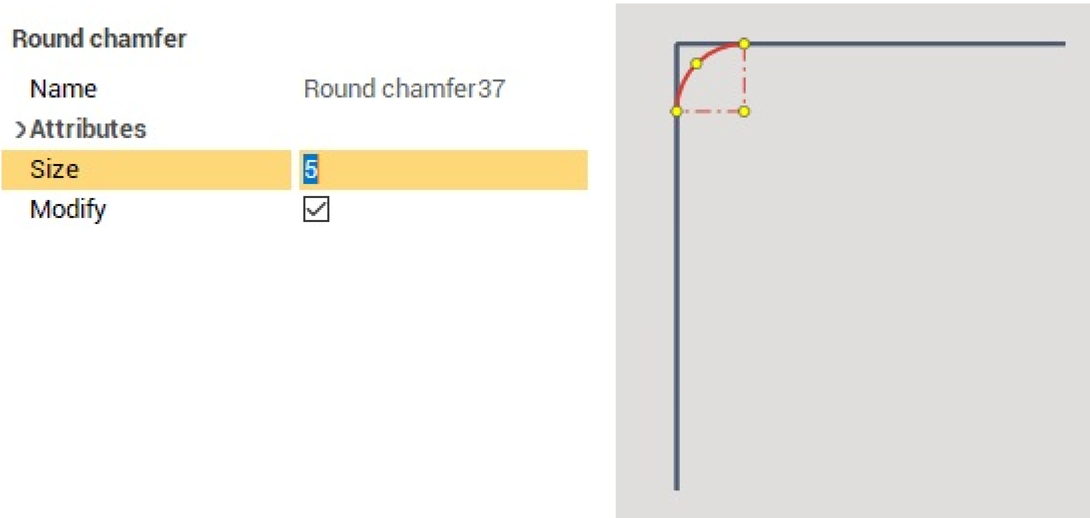
These two objects are built automatically. You just need to move the cursor to the intersection of two objects, enter the size value and click the mouse button to complete the construction. If for some reason the system cannot build the required element, then the elements must be sequentially selected.

Chamfer

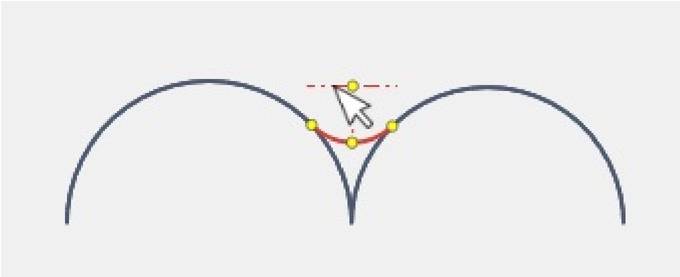


Rounding

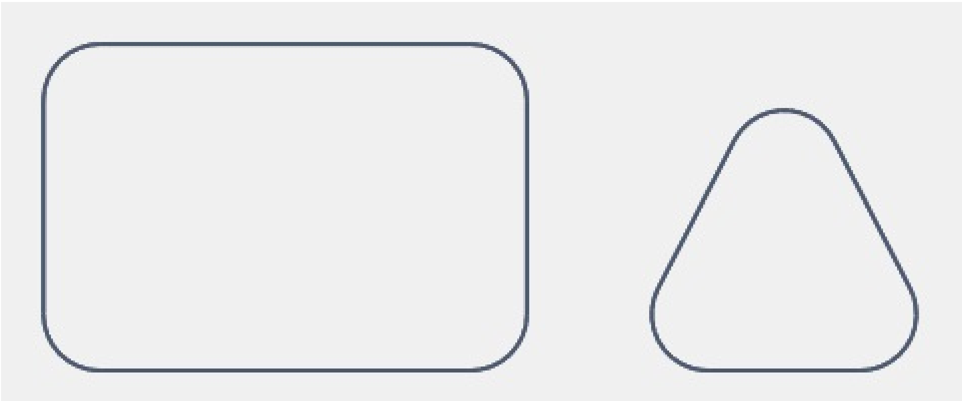




It is also possible to build chamfers and roundings between arcs.



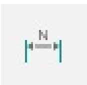
If you double-click on a rectangle or contour, then all corners of such element will be rounded.

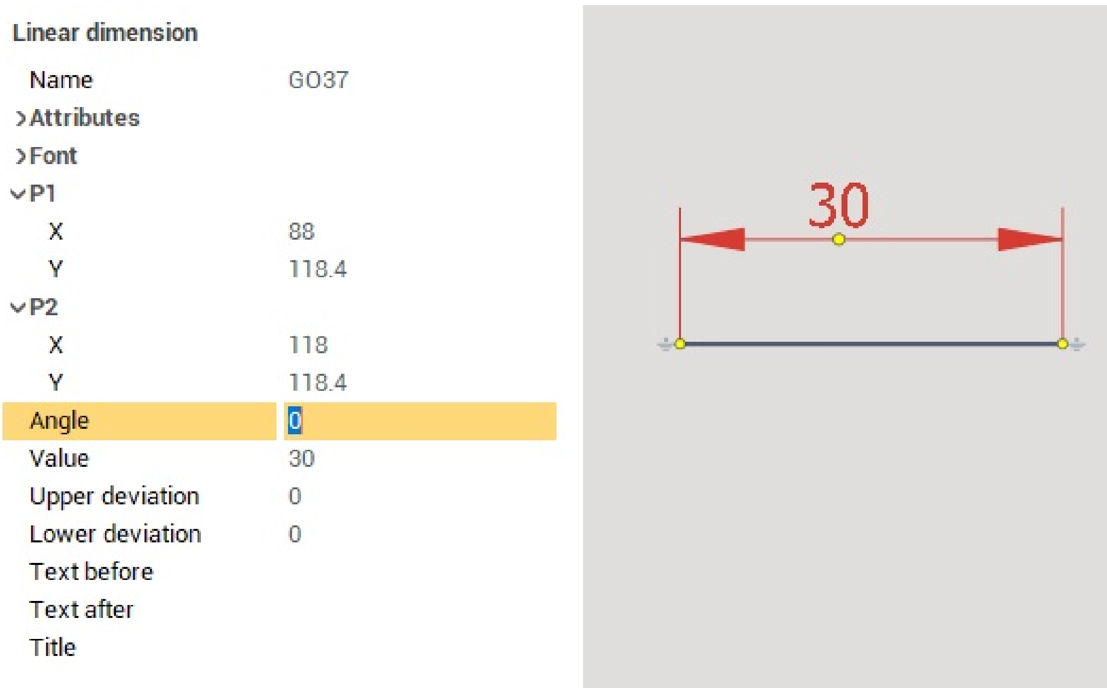


4.4.8 Dimension

There are three types of dimensions for dimensioning drawings: linear, radial and angular.

4.4.8.1 Linear dimension

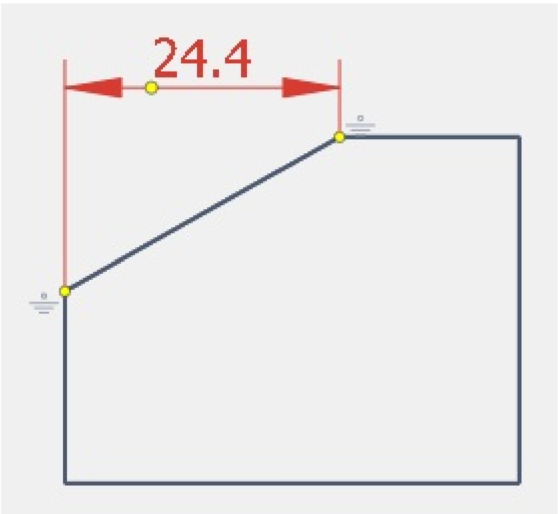




When constructing a linear dimension, you must specify the start and end points, the size of the leader, and the offset of the beginning of dimension text relative to the first extension line. The dimension value is calculated automatically. The inspector sets the slope of the dimension line relative to the x-axis, the values of the upper and lower deviations, the text before and after the dimension value, and the text font.

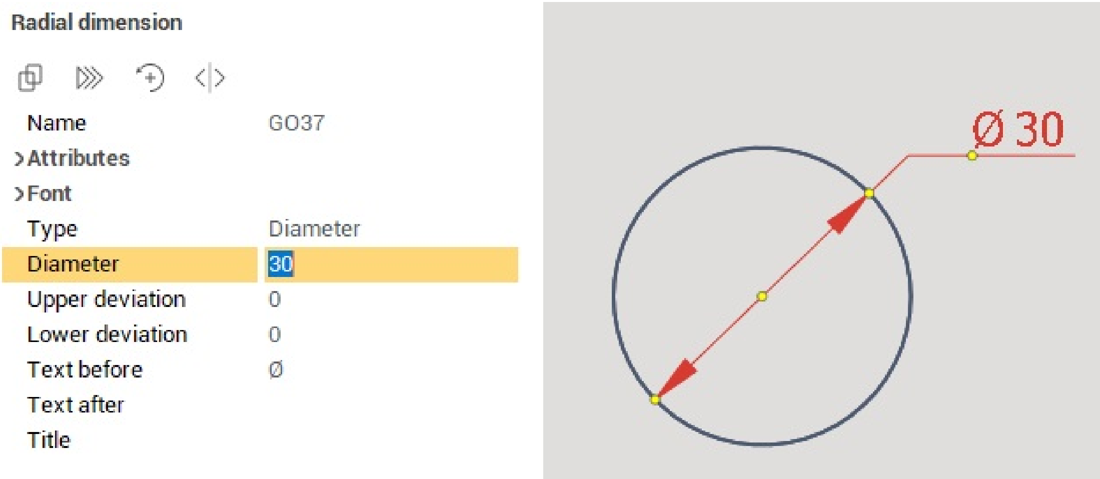
It is also possible to create a linear dimension by specifying an existing line. In this case, you do not need to specify the start and end points, and the system will have a link between the dimension and the line. Subsequent editing of a line or dimension will change both objects.

If the dimension line is not horizontal or vertical and needs to be normalized, press **[Shift]** while constructing the dimension.

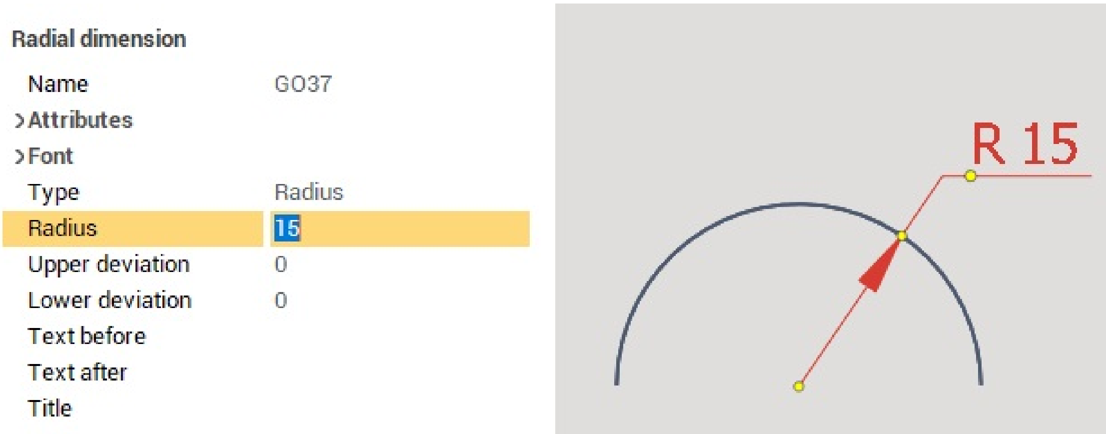


4.4.8.2 Radial dimension

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.



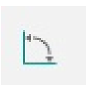
When dimensioning a circle, by default, a dimension of the **<Diameter>** type will be built, as in the picture above. When dimensioning an arc, a dimension of the **<Radius>** type will be built by default:



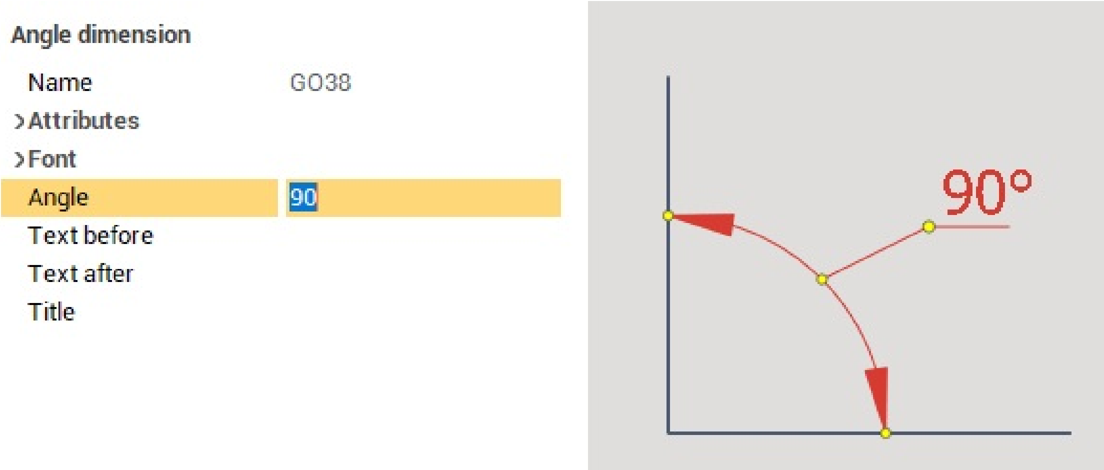
You can switch the type in the field of the same name at any time.

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.

4.4.8.3 Angular dimension

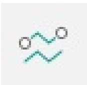


To build an angular dimension, you must sequentially specify two lines and the start point of the text. The dimension value will be automatically determined. The inspector sets the text before and after the size value and the font of the text.

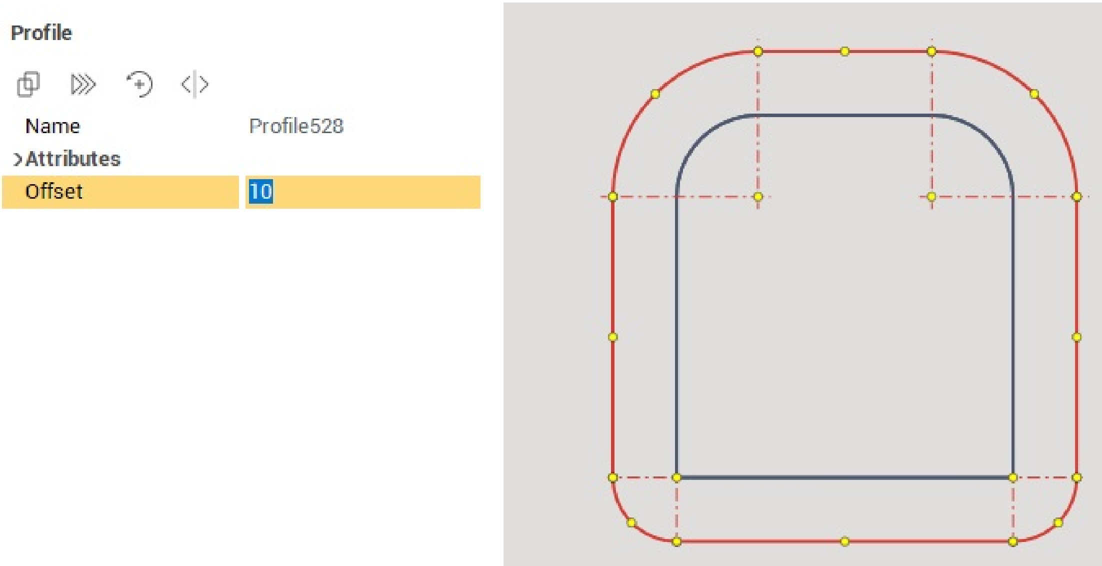


As a result of construction, the system will have a link between the dimension and the lines. Subsequent editing of a line or dimension will change both objects.

4.4.9 Offset

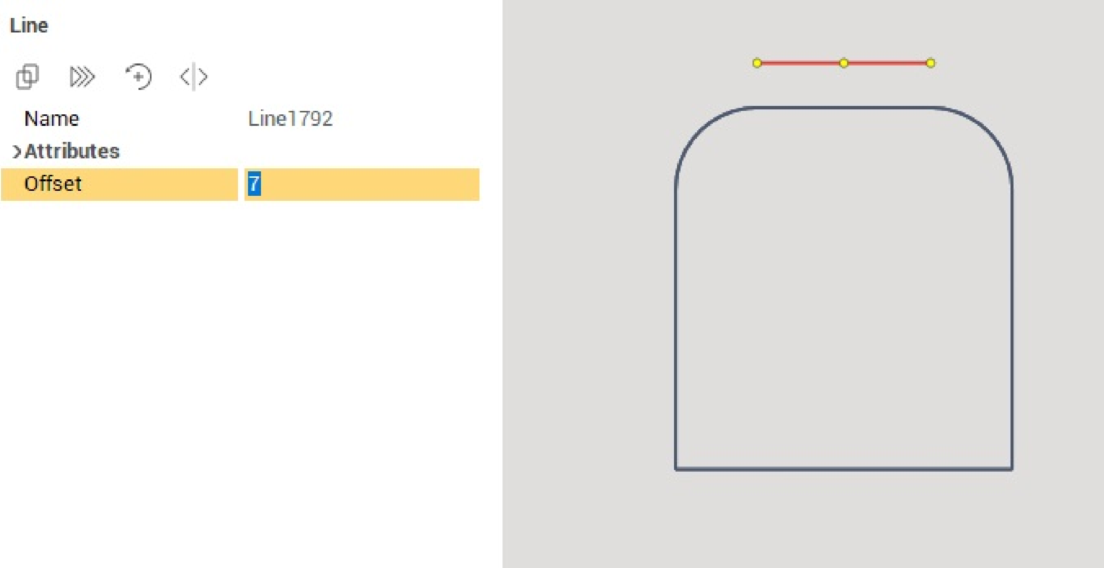


This function creates an offset object to the previously created one.

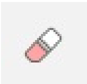


Offset can be built both interactively and by entering an offset value from the original object. The new object will be the same type as the original.

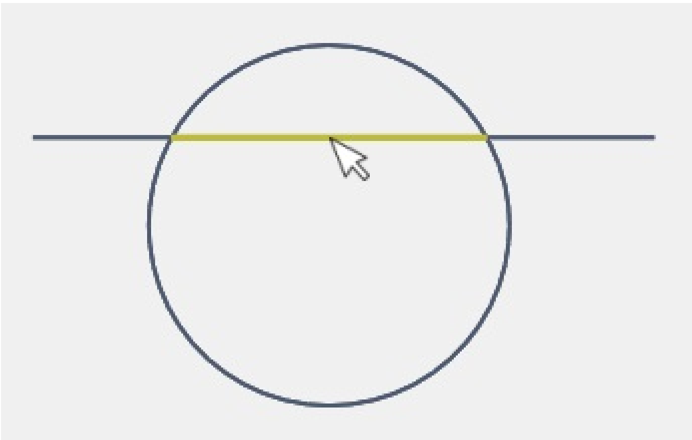
If **<Ctrl>** was pressed when selecting a compound object (rectangle, contour), then only the specified side (line or arc) will be selected to build the offset, and not the entire object.



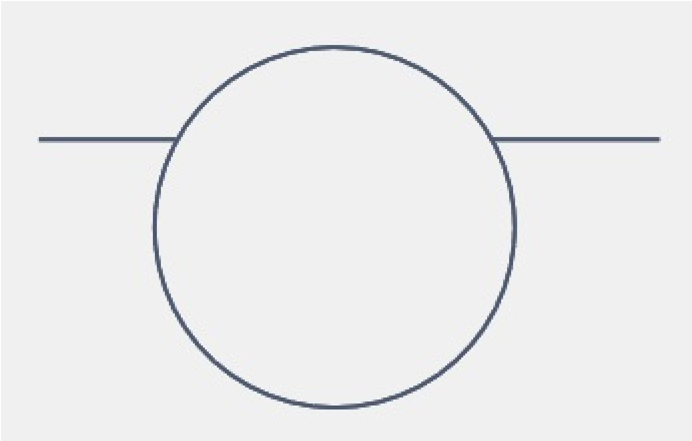
4.4.10 Trim function



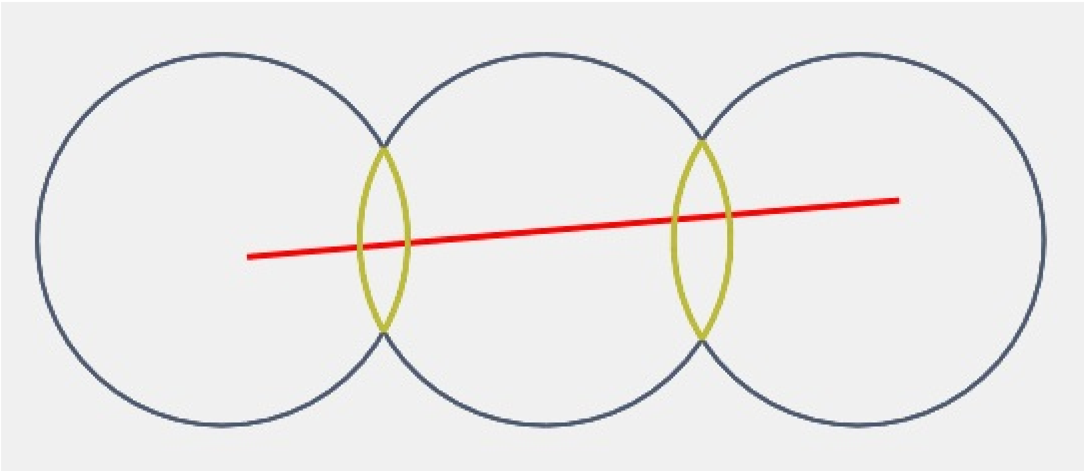
The function finds the intersection points of the specified element with others and highlights the area that will be deleted after confirmation by clicking.



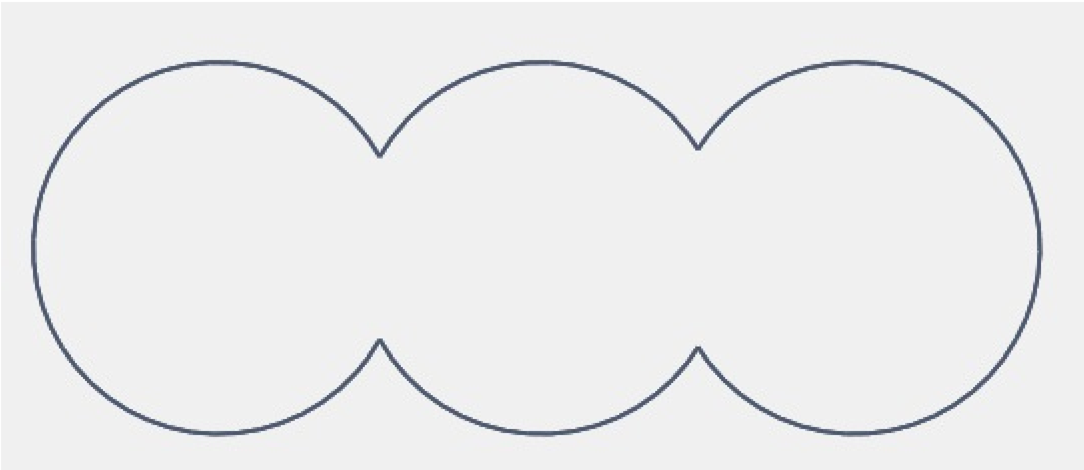
After confirmation:



You can also cut fragments of several elements at once by pressing the mouse button and dragging.



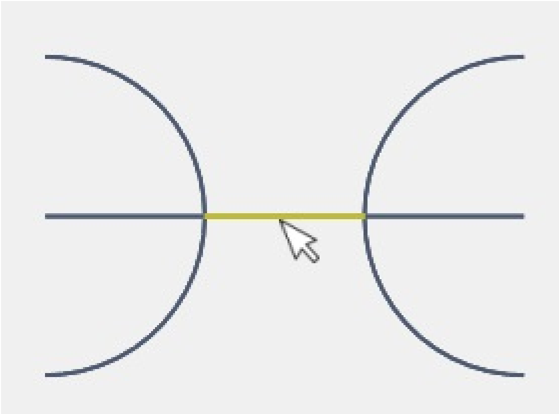
All intersections of objects with this imaginary line will be removed after releasing the mouse button:



4.4.11 Split object

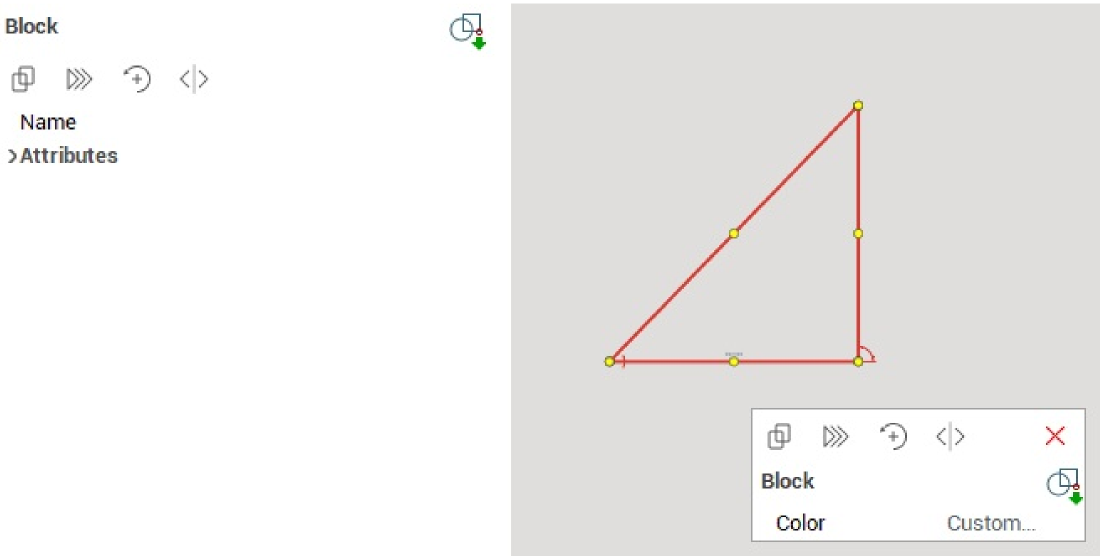
The function converts the specified compound element (rectangle, contour, named block) into its parts. If a block is pointed, it will be split into the elements from which it was built.

If you select an elementary object (line, arc, circle), this element will be split into several at the intersection points with other elements.



4.4.12 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.

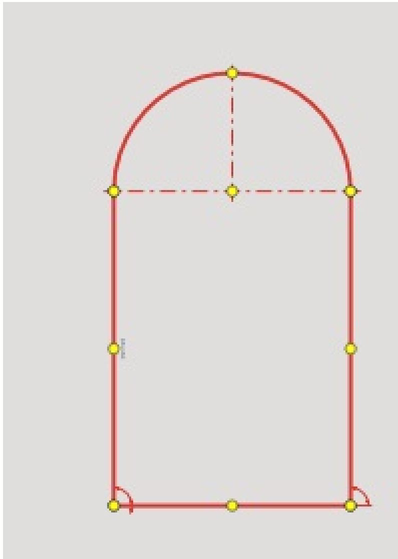


4.4.12.1 Copy

A copy of the block or single object is created in the same place as the original.

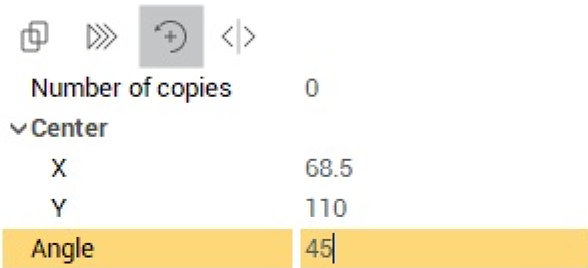
4.4.12.2 Shift

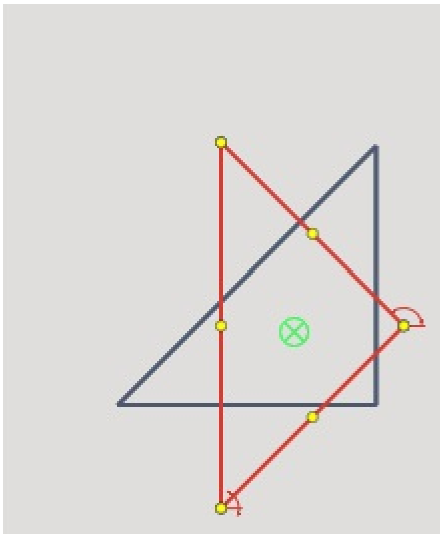
Shift a group of objects by a specified distance. In addition, you can specify the number of copies and each next copy will be shifted by the specified value from the previous one.



4.4.12.3 Rotate

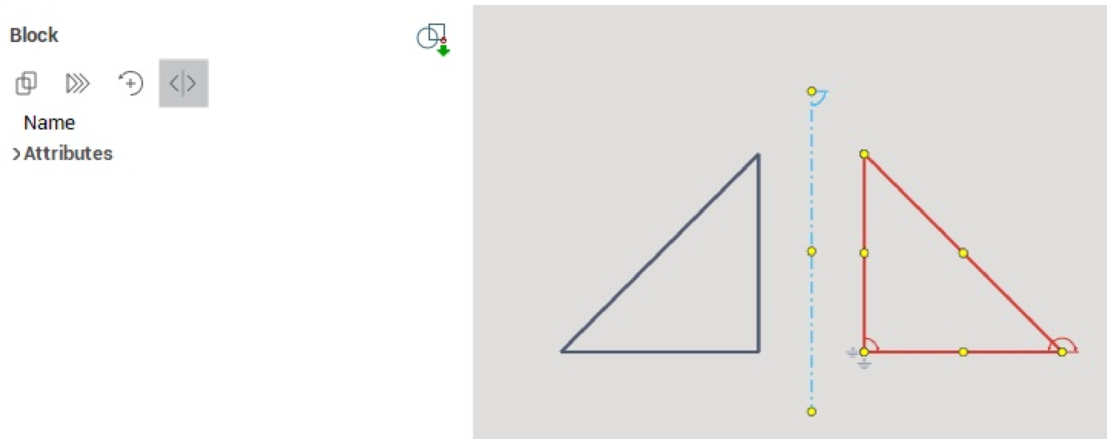
The object is rotated around the specified point by the specified angle. Rotate can be done interactively or by specifying the coordinates and angle in the inspector. In addition, you can specify the number of copies, and each next copy will be rotated by the specified value from the previous one.





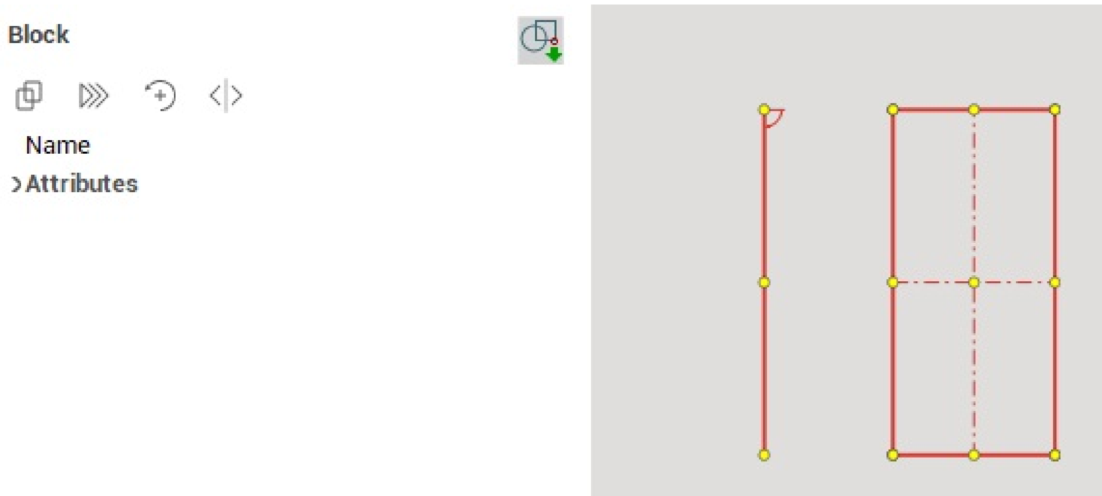
4.4.12.4 Mirror

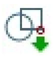
For mirror , you must specify the axis relative to which the mirror will be performed.

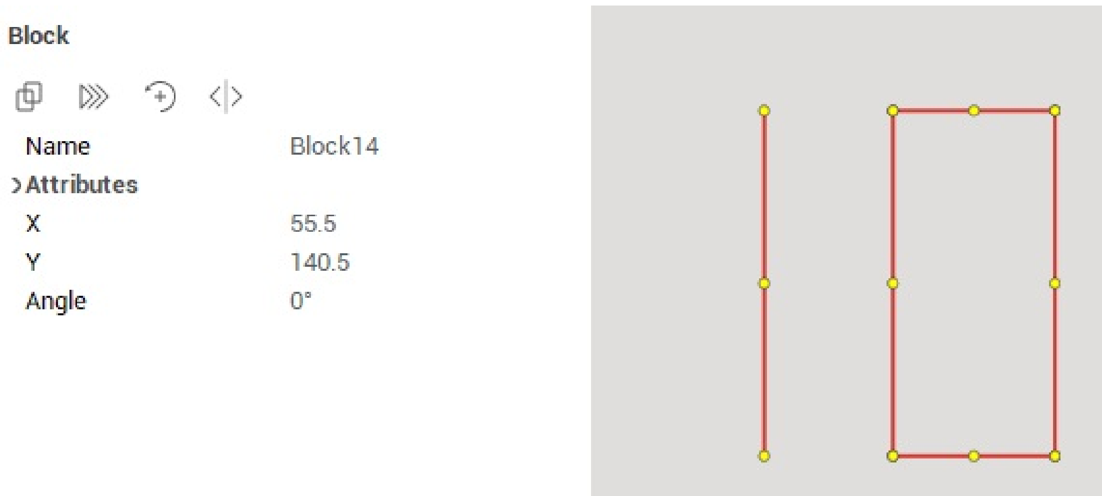


4.4.13 Named block

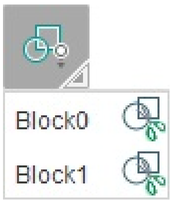
To create a named block, you need to select the necessary elements in the block. To do this, by pressing the mouse button and holding it down, select the elements with a rectangular frame, or sequentially select the elements with the **[Shift]** key pressed. The exclusion of elements from the block is also performed by specifying these elements with the **[Shift]** key pressed.



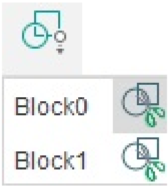
Click the  button in inspector. If necessary, you can enter its name or set the rotation angle of the block.



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.

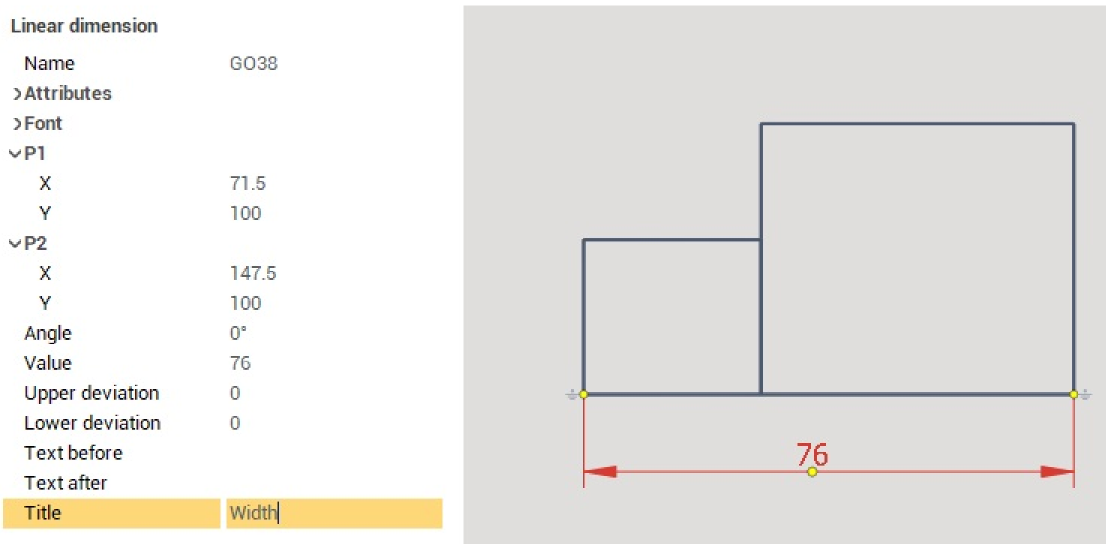


Subsequently, you can edit the original block by clicking the "scissors" in the menu. All copies of the edited block will also change.

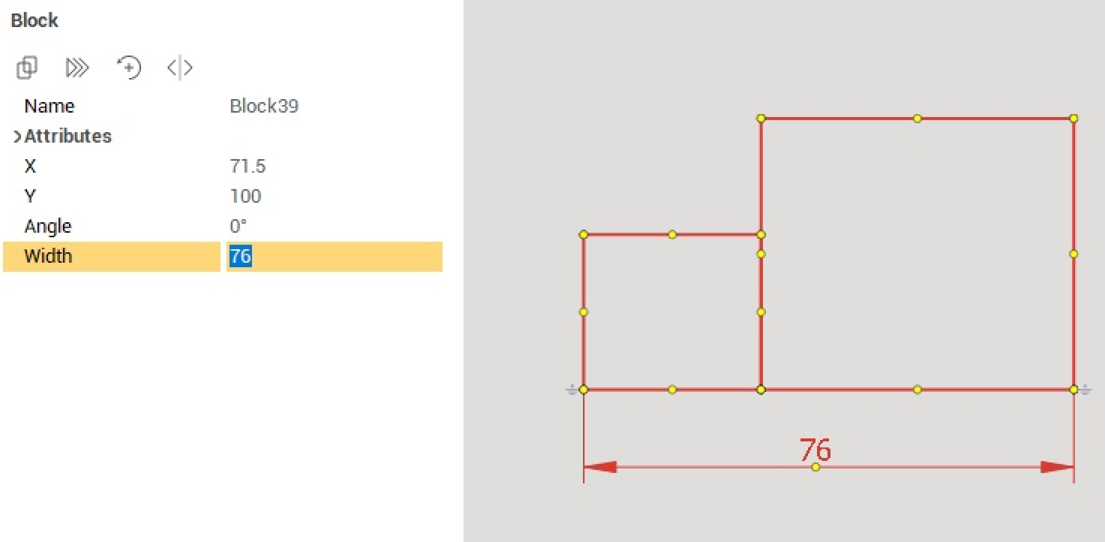


4.4.13.1 Block parameterization

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.



4.4.14 Constraints

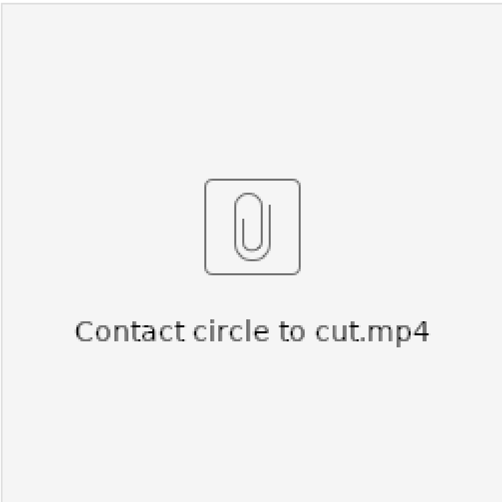
Constrains are created during the build process and are removed during editing when the link conditions are lost. For example, the line has ceased to be parallel or the touch has disappeared.

In the process of editing a block, the constrains are preserved and the objects change in accordance with them. That is, tangency, parallelism of lines, etc. is saved.

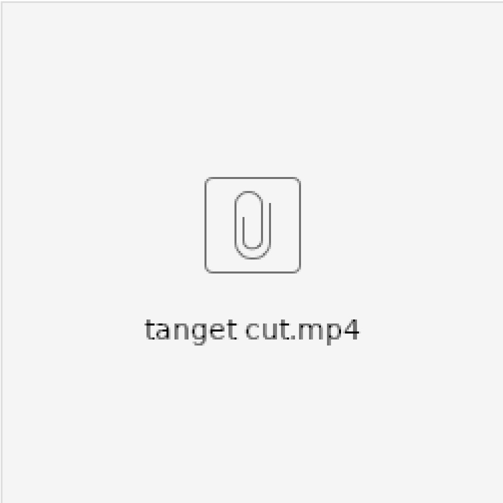
Constrain types:

- Tangency

  During construction, while pressing **[Ctrl]**, select the object to which this constrain 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

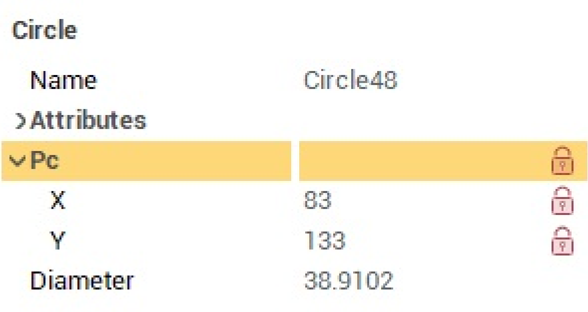
If during the construction process with pressed **[Ctrl]** move the cursor to the line, then auxiliary lines for construction will appear and a link will be set. You can also set the link in the inspector on the **<Links>** tab, and if the lines did not meet this criterion, they will be adjusted.

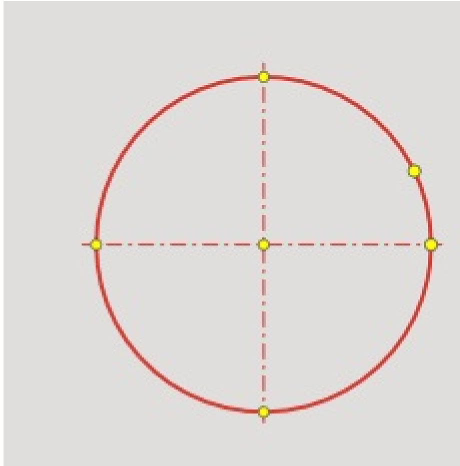
- Alignment

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.

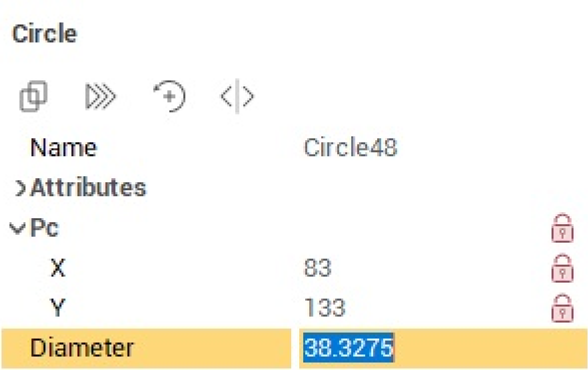
- Fixing

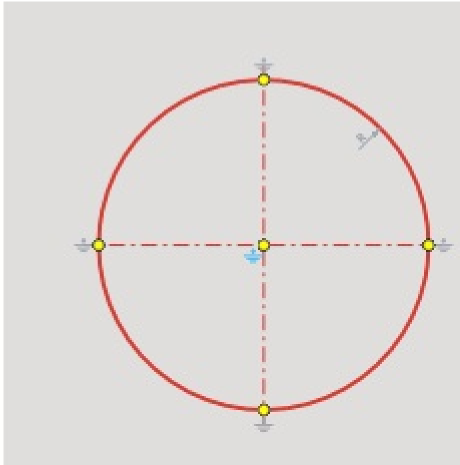
You can fix points by X, Y coordinates, separately or together. You can fix the length, radius, corners, height and width of the rectangle. You can fix it with the buttons in the inspector:





Or by anchors in the drawing:

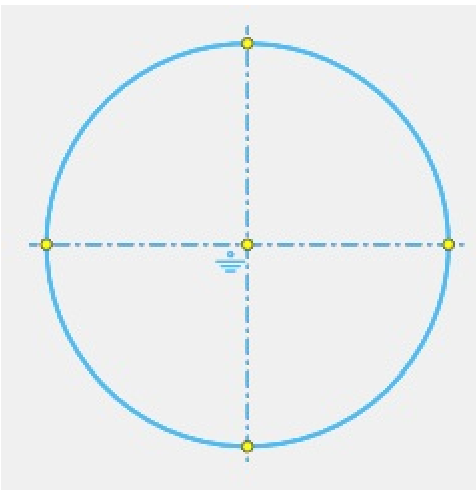




For dimensions, you can fix the value **<Value>, <Radius>, <Angle>** in the inspector. This affects how elements behave when edited in block mode - this size does not change.

- 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

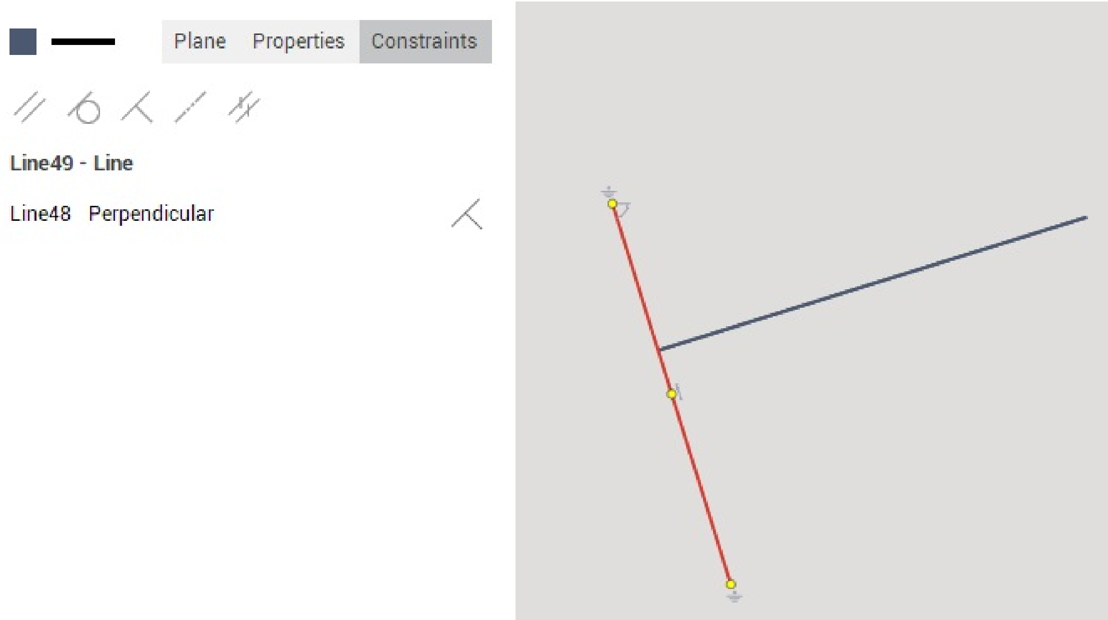
Such a connection is created during construction, when the points of the objects coincide. You can also set such a link in the inspector on the **<Links>** tab.

- Object-forming links

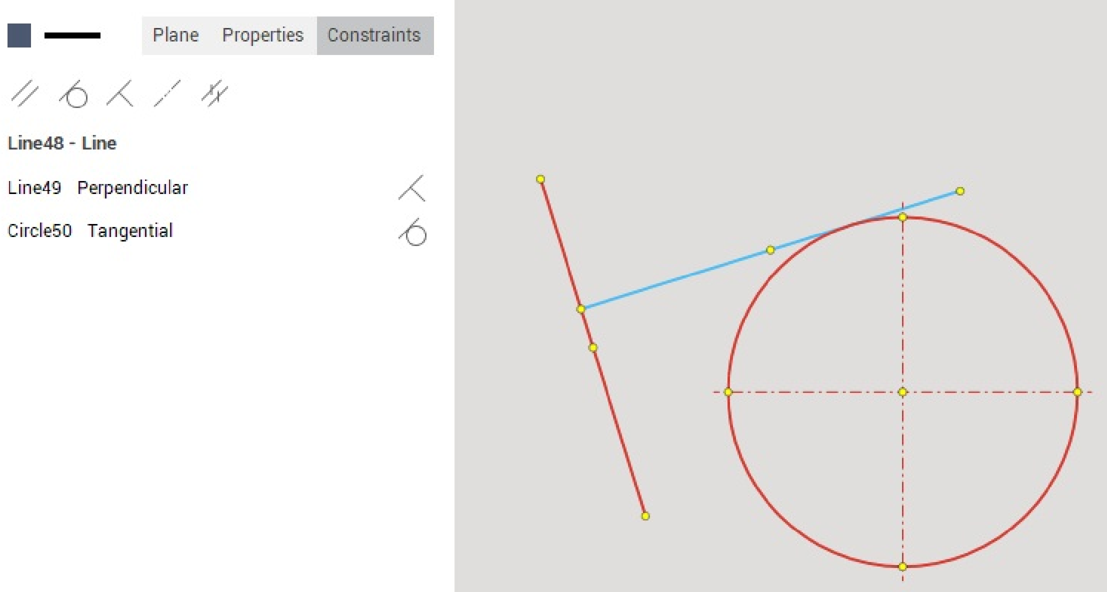
They are used to connect created complex objects, such as a contour, a radial dimension, an angular dimension, a rectangle, a fillet, a chamfer, a script. The link persists as long as the original object exists.

4.4.14.1 Editing links in a block

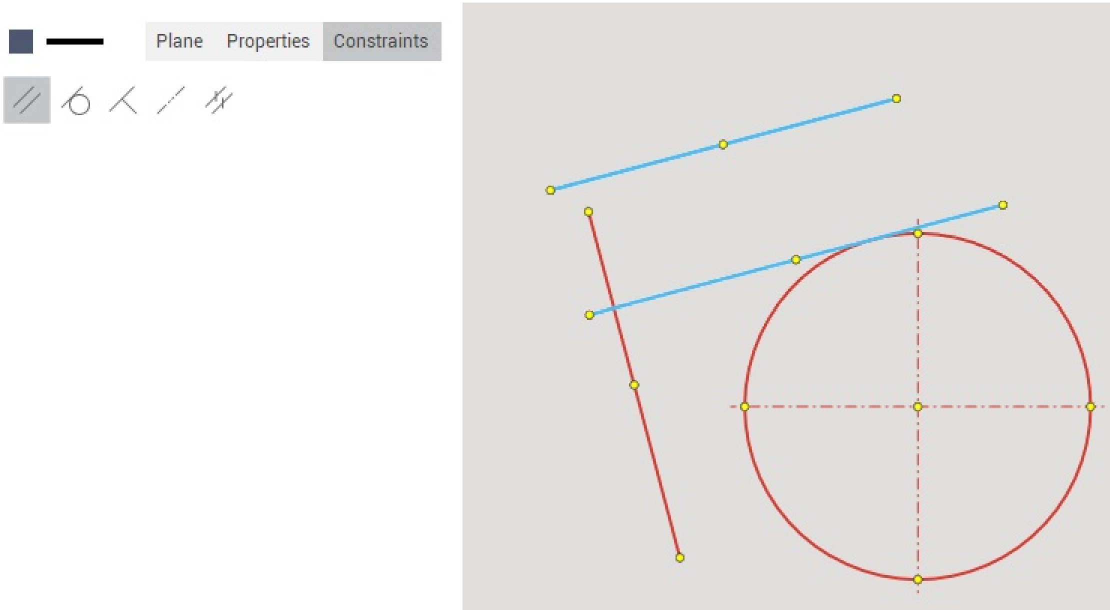
In the inspector, in the **<Constraints>** tab, a list of the active element's links is shown.



In block mode, you can disable / enable the restriction by clicking on the link image.



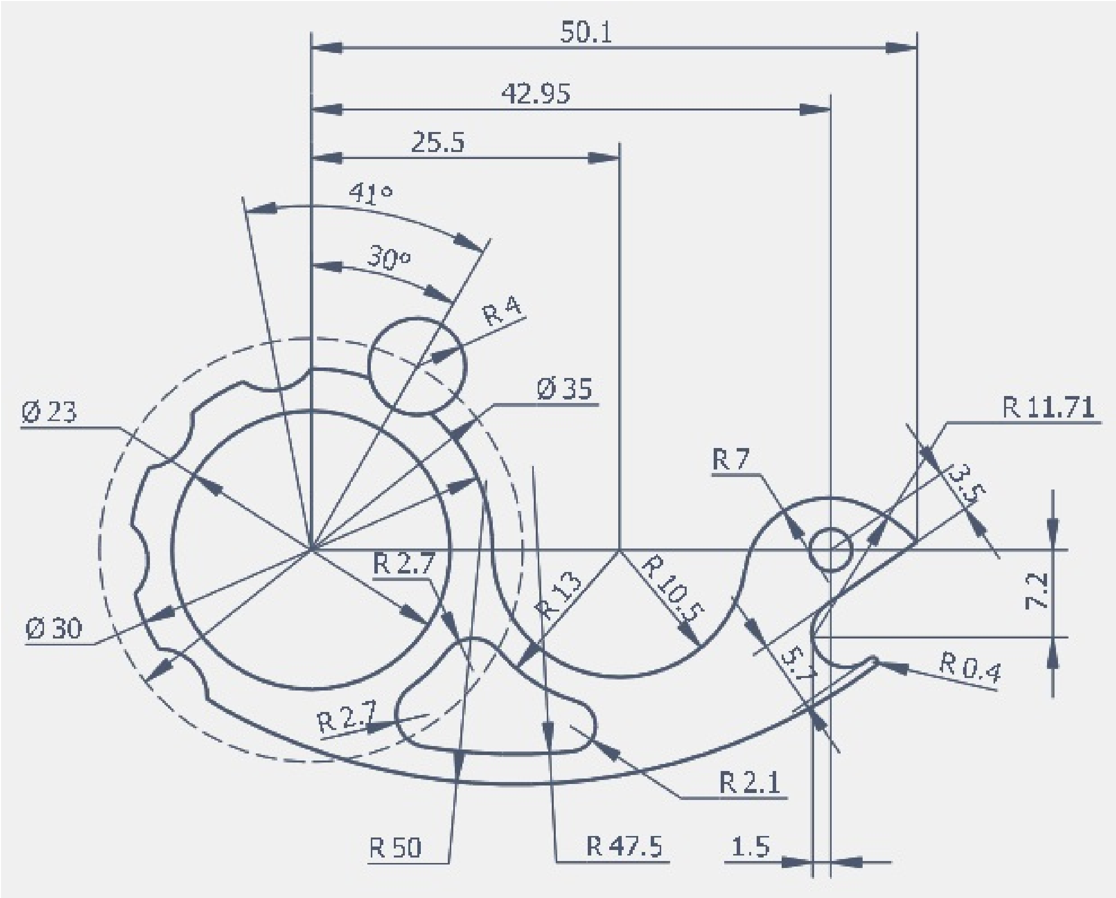
To create links **<Parallelism>, <Tangency>, <Perpendicularity>, <Coaxiality>, <Equality>** in block mode, press the corresponding button and sequentially select a pair of objects.

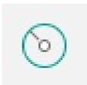


Releasing the button will return the link enable/disable mode. After exiting the block mode, disconnected links are permanently disabled.

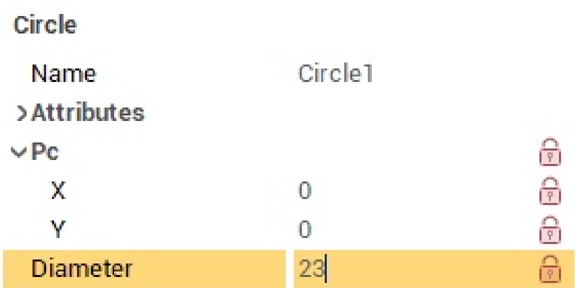
4.4.15 Drawing example

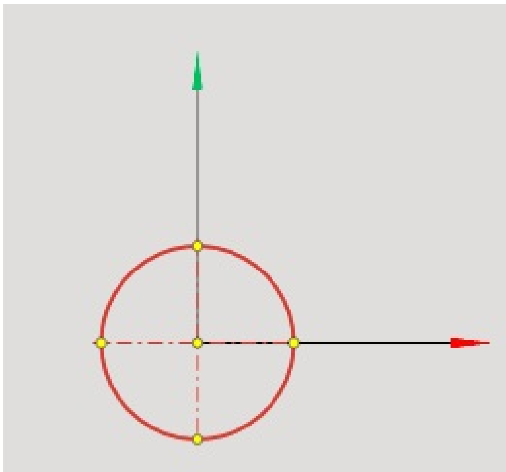
Let's look at the construction using this drawing as an example:



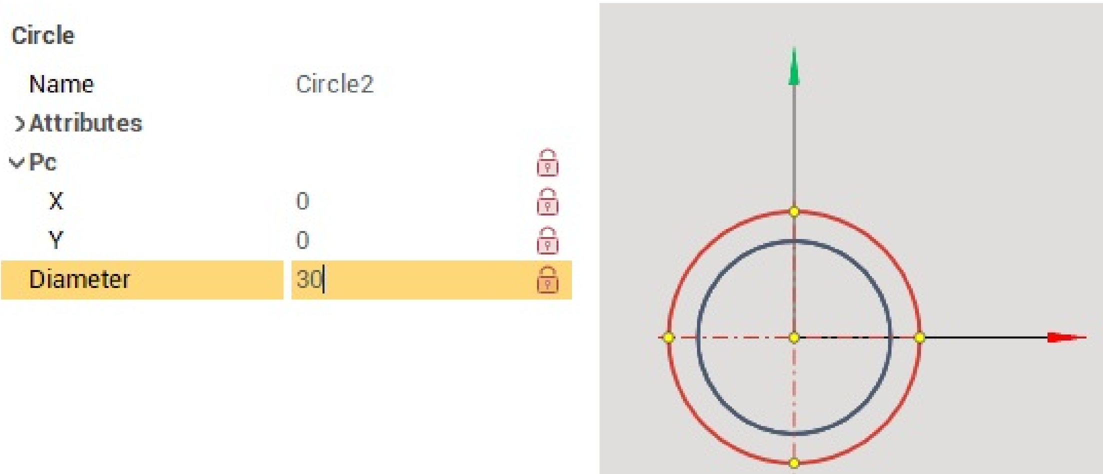
Press the button for drawing a circle 

In the inspector, enter coordinates 0, 0 and radius 23.

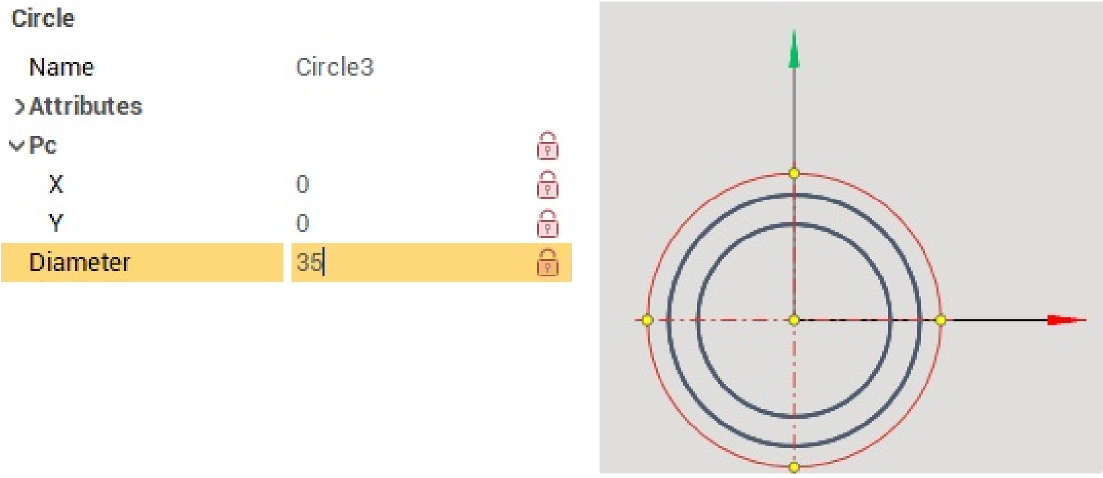


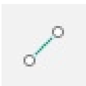


In the same way, we build the following circle:

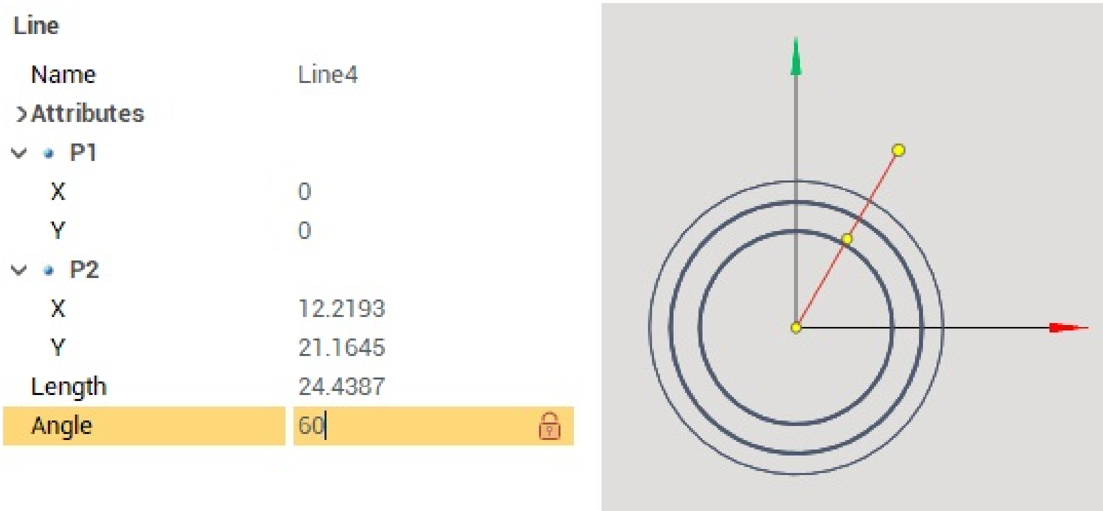


And the third, after pressing **[Alt + 2]** to switch the line type:



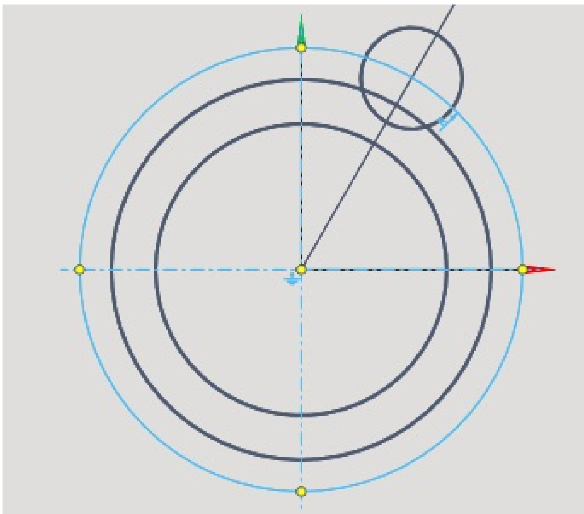
Next, switch to the line construction mode. 

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°:

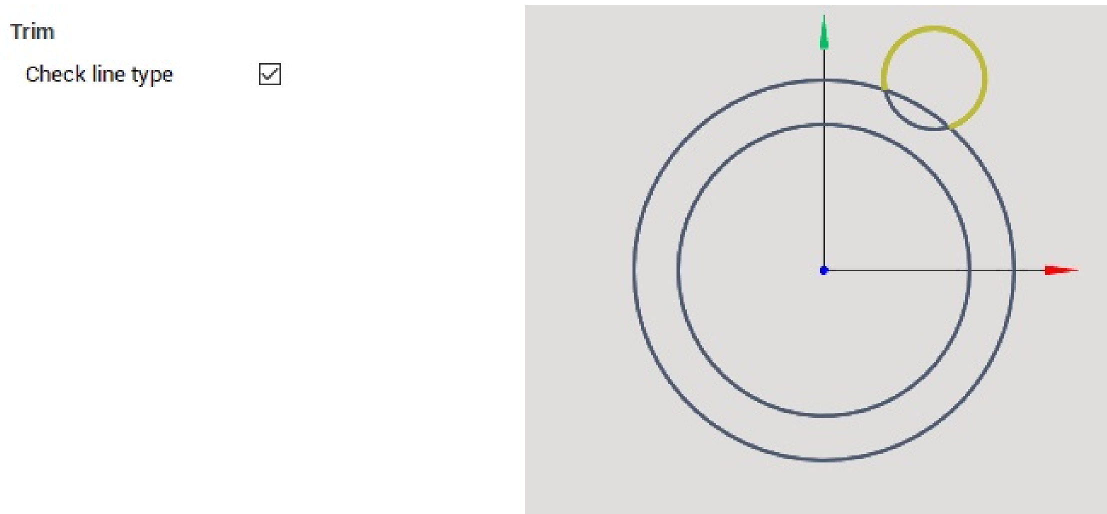


Switch back to the circle construction mode, press **[Alt+1]** to return to the main line construction mode, specify the intersection point of the segment and the third circle as the center, press the **[R]** key to switch to the radius input mode and enter 4:

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:

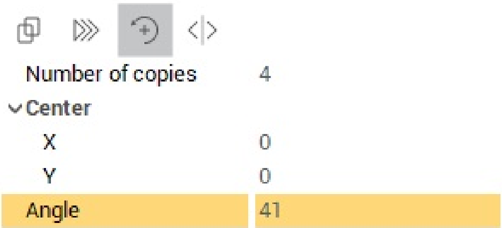


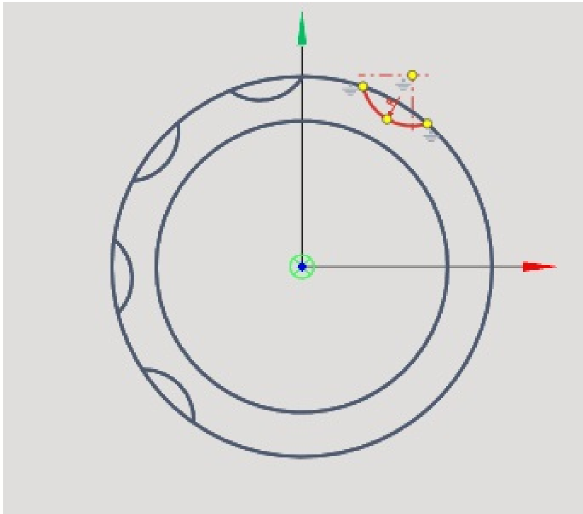
Switching to trimming mode  and delete the unnecessary part of the circle:



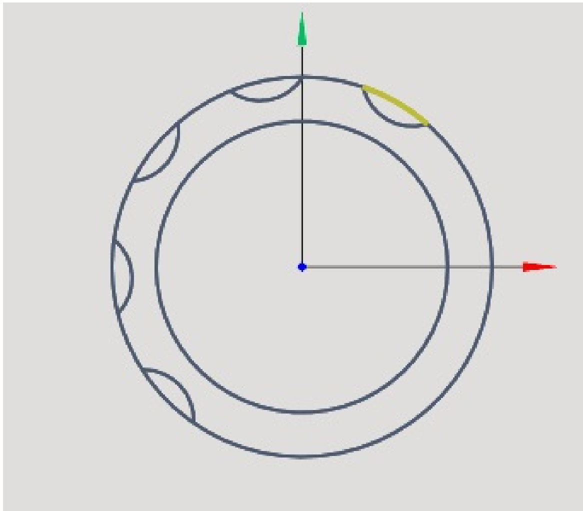
Select the resulting arc, switch to the object rotation mode 

Move the rotation center to the center of the circles, enter 4 in the **[Number of copies]** field, and 41° in the **[Angle]** field:





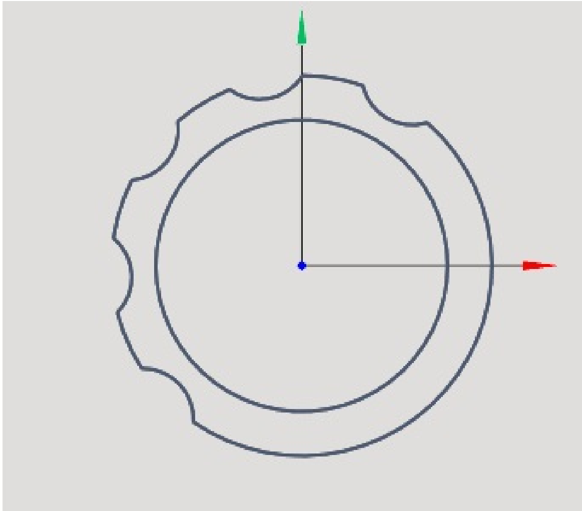
Next, turn on the trim function  and sequentially remove unnecessary sections:



As a result, we get this figure:

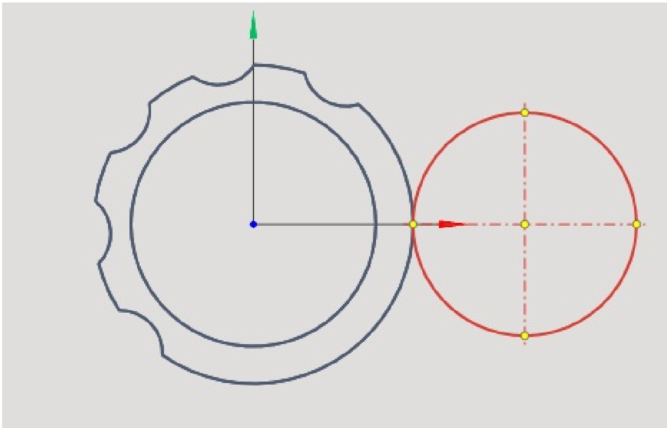
Trim

Check line type ☑



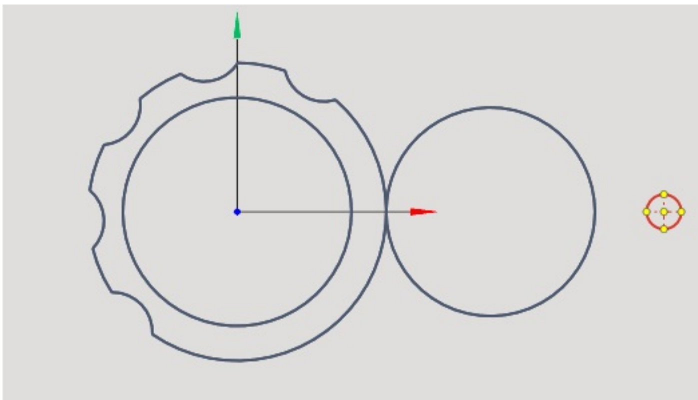
Turn on the construction of the circle again and enter the following values:

Circle	
Name	Circle50
›Attributes	
⌄Pc	
X	25.5
Y	0
Radius	10.5

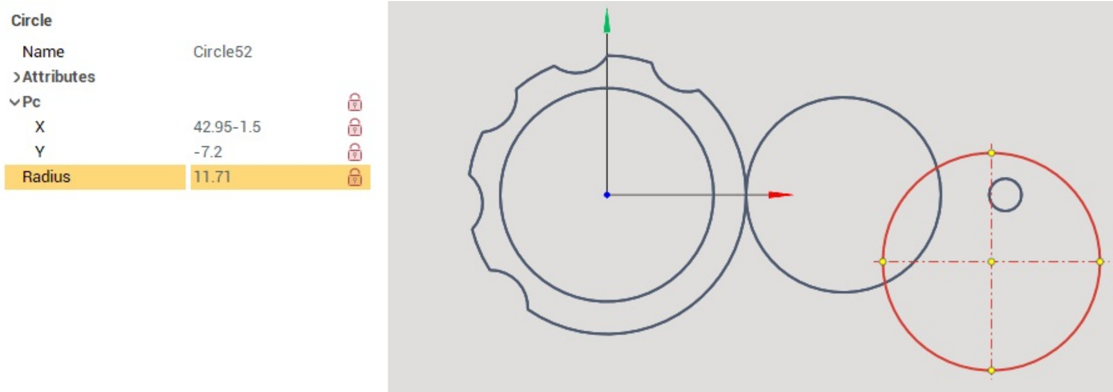


And the following circle:

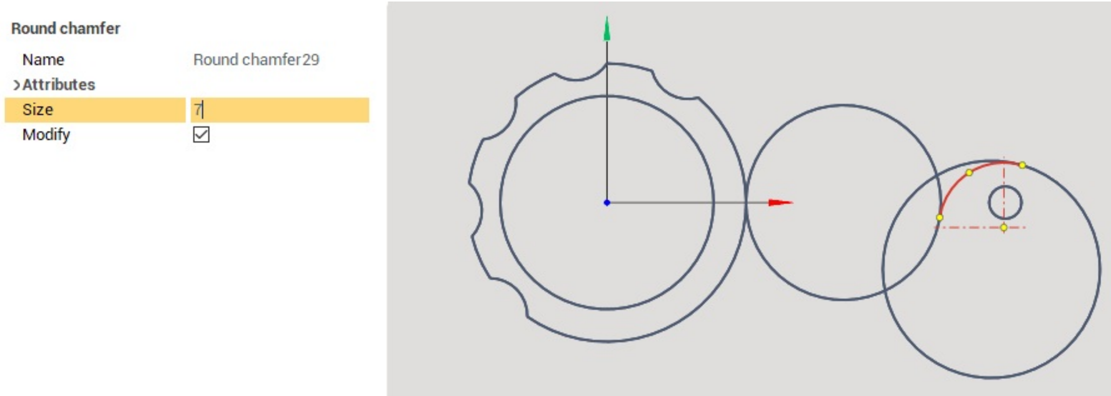
Circle	
Name	Circle51
›Attributes	
⌄Pc	
X	42.95
Y	0
Diameter	3.5



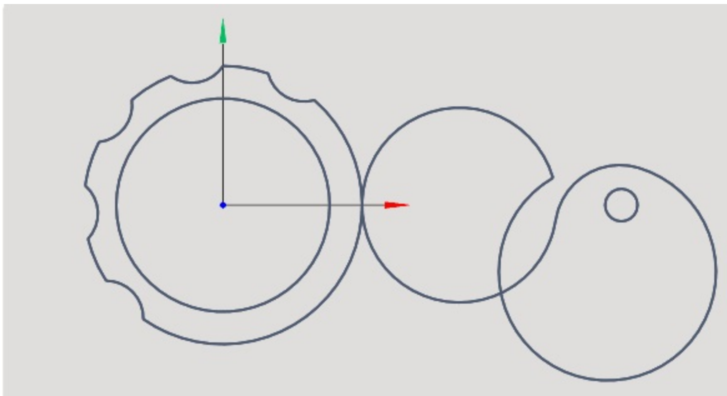
The following circle is the base of an arc with a radius of 11.71:



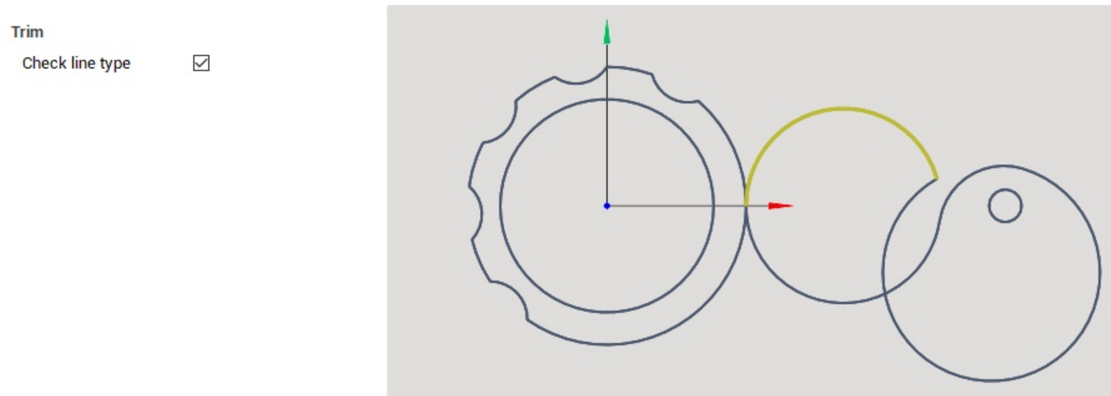
Turn on the construction of the rounding  and build an arc with a radius of 7:



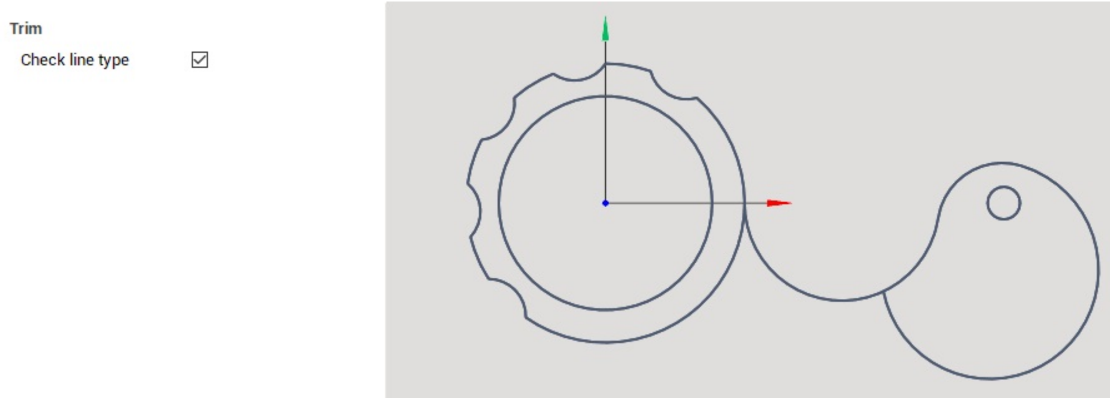
We got this:



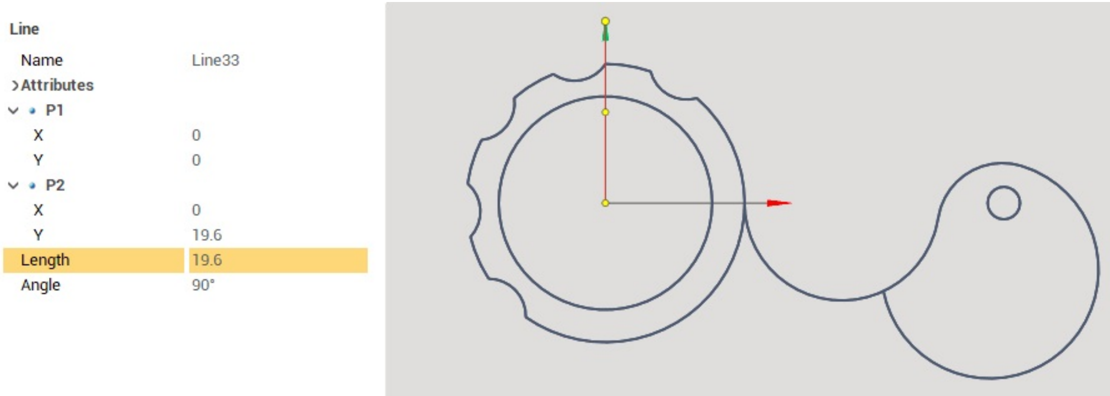
Turn on trim mode and remove two extra fragments:



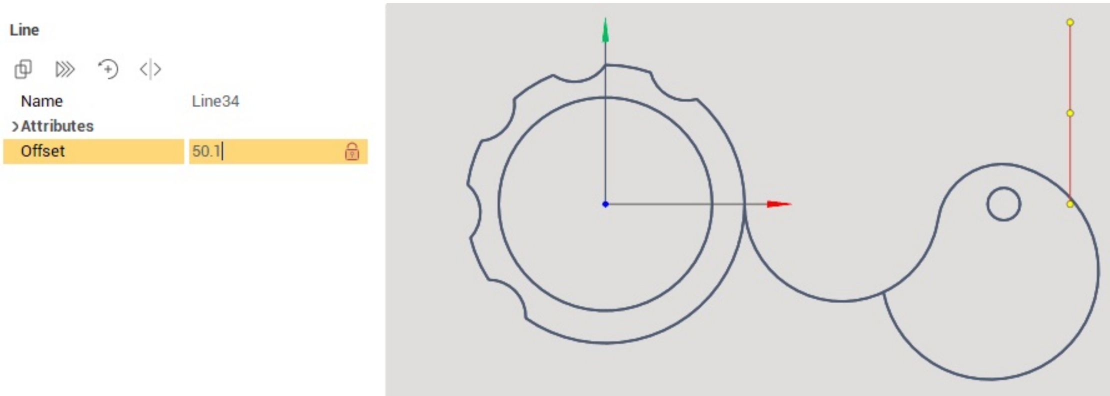
The result will be the following:



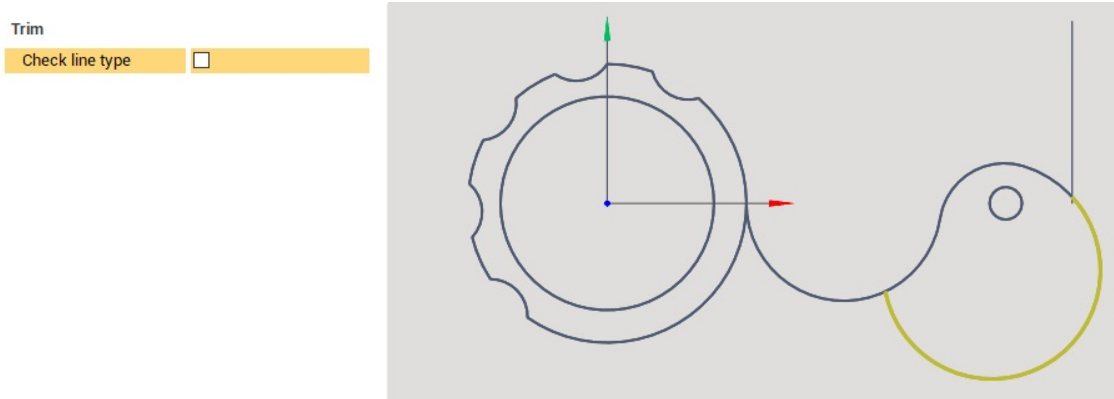
Build a vertical line:



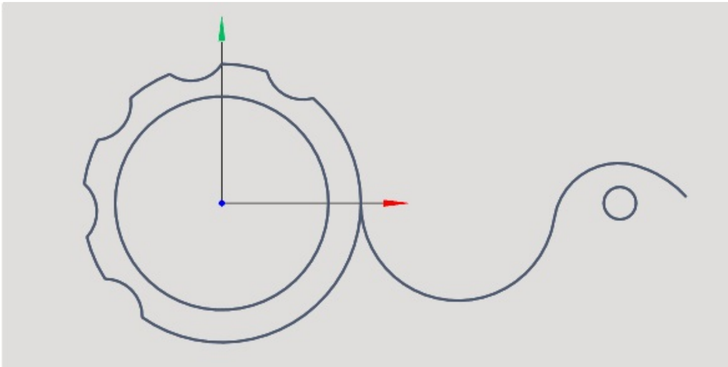
Turn on the construction of equidistant  and build a line parallel to the previous one at a distance of 50.1:



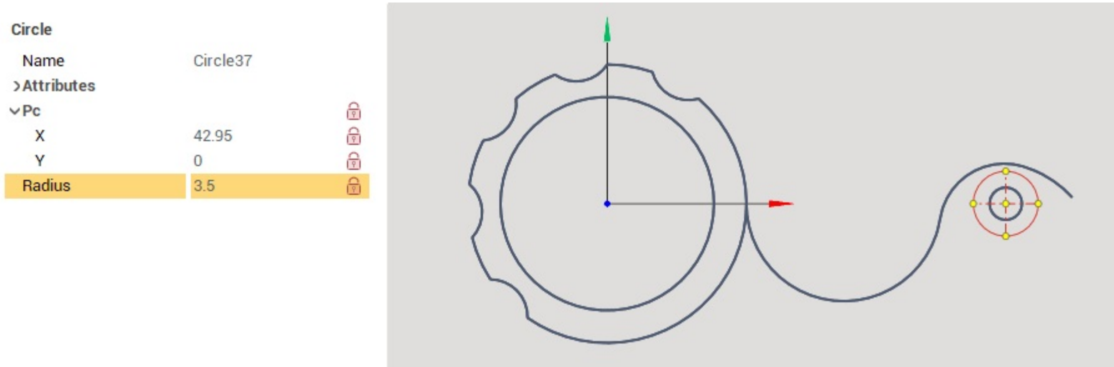
Turn on clipping and remove the excess part of the arc, having previously disabled the **<Check line type>** field in the inspector:



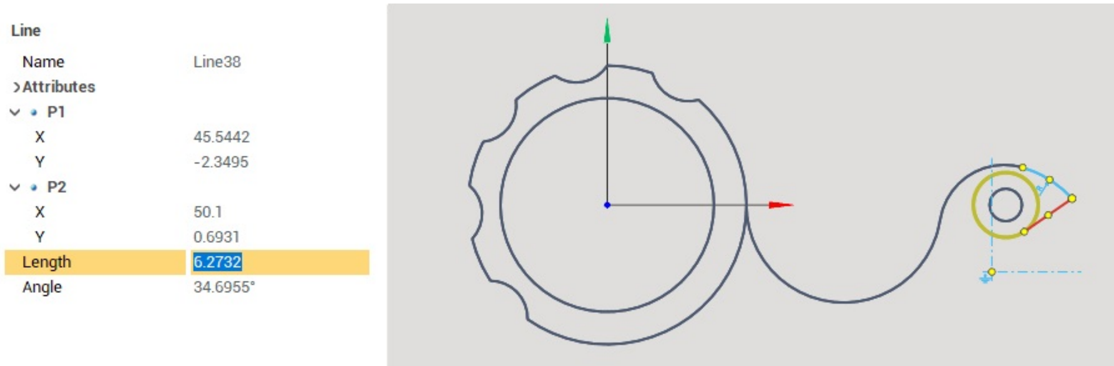
After that, we delete both time lines and we have the following result:



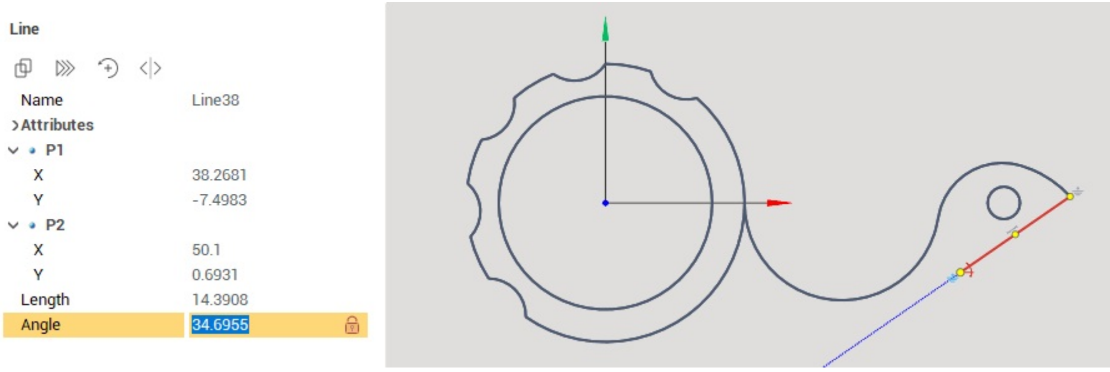
Now we build an auxiliary circle with a center at the point 42.95, 0 and a radius of 3.5:



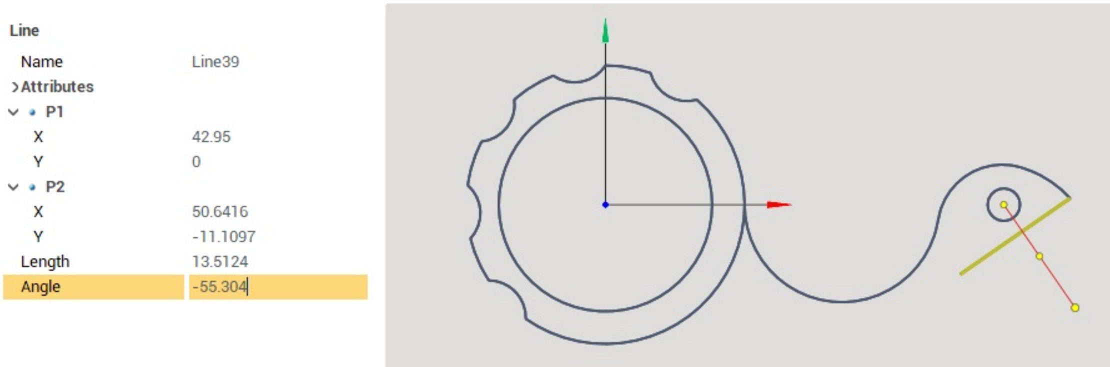
And then a line tangent to this circle, pressing **[Ctrl]** before selecting and connecting to the extreme arc:



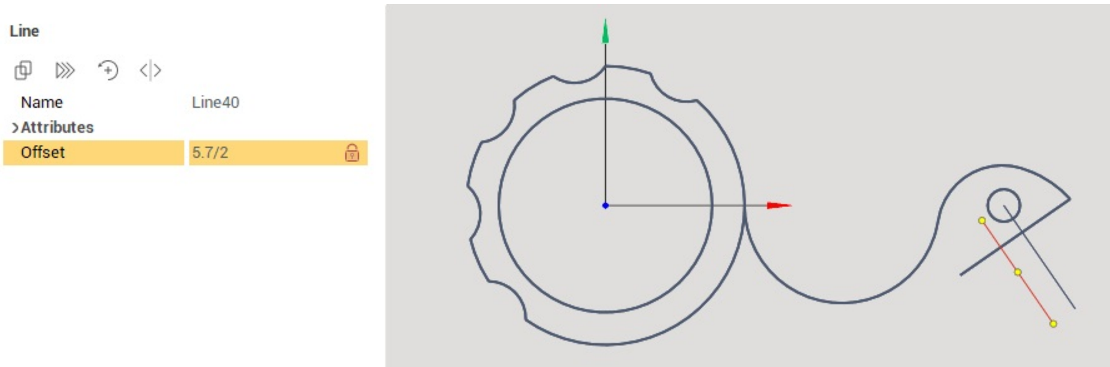
Delete the auxiliary circle, select the resulting line and fix the angle and lengthen it:



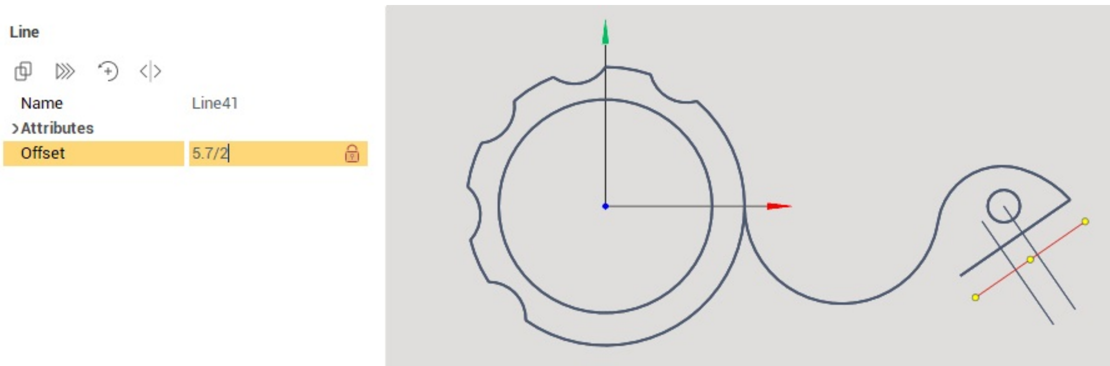
Next, we build another auxiliary line with a center from a small circle and perpendicular to the resulting line, for which we hover over the line with **[Ctrl]** pressed:



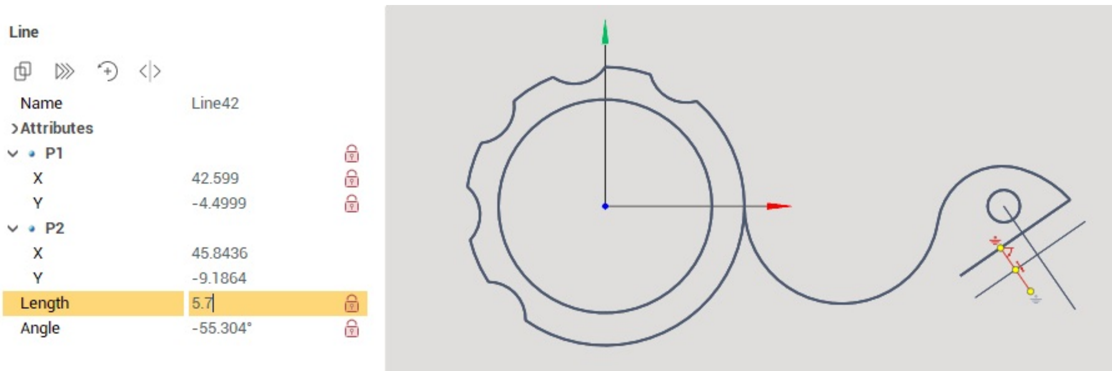
Turn on the construction of equidistant and build a line at a distance of 2.85 from it:



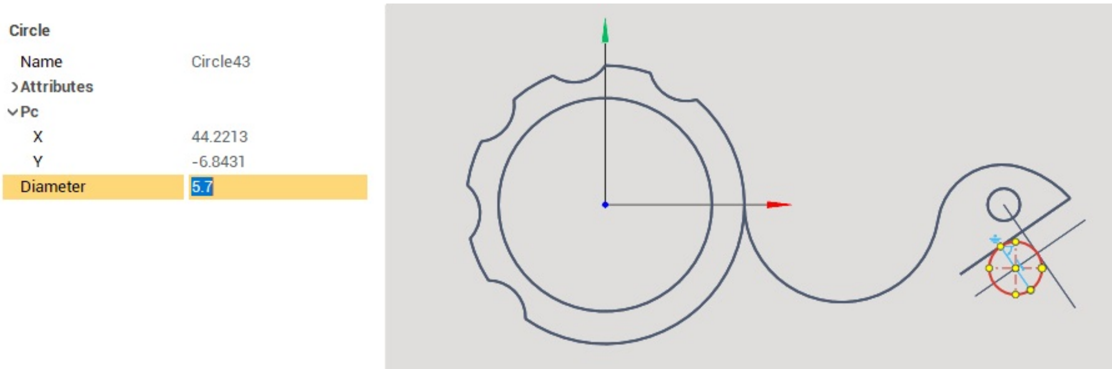
And another one at the same distance:



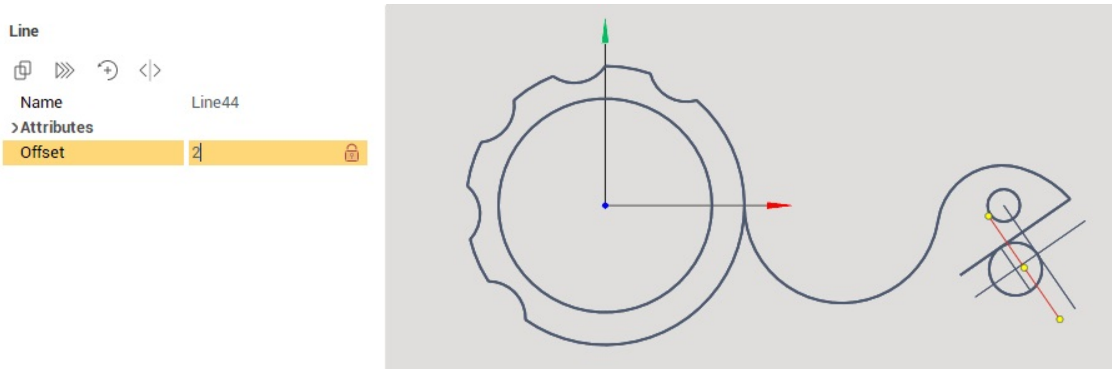
We correct the previous line, after fixing the angle in such a way that it remains on its line and has a length of 5.7:



We build a circle according to the received element:

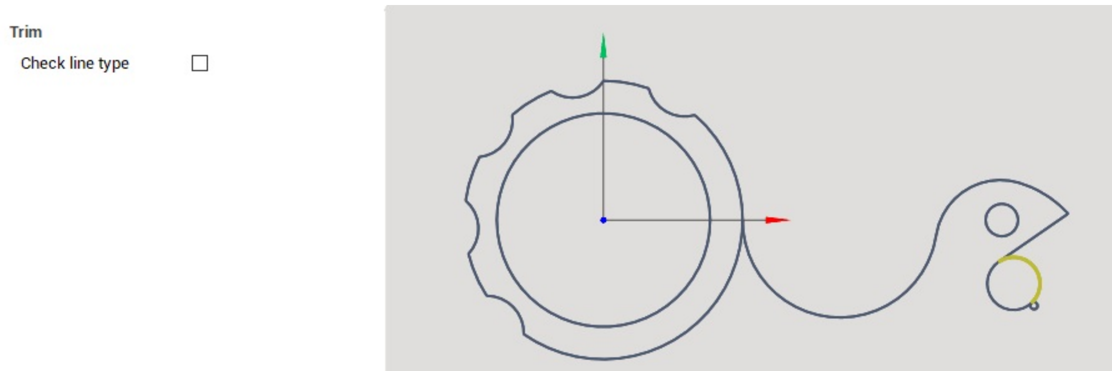


We construct another equidistant line:

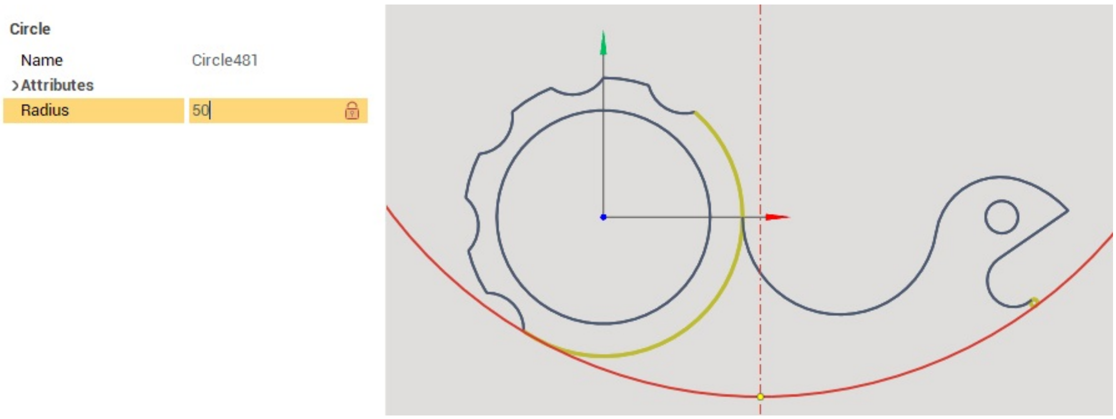


And a circle tangent to a circle and a line with a radius of 0.4:

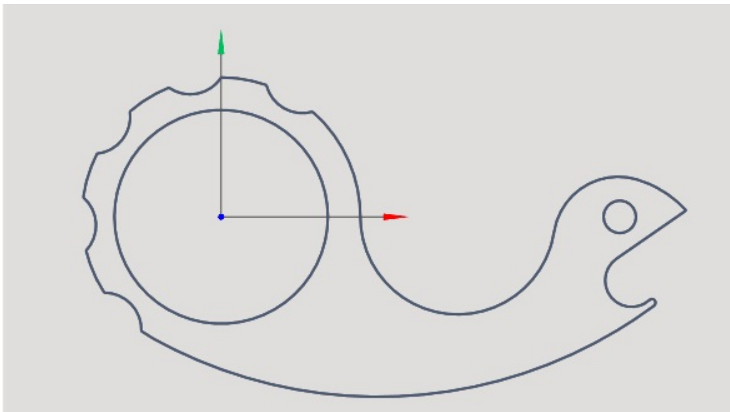
Unnecessary more auxiliary elements are deleted, the excess is cut off:



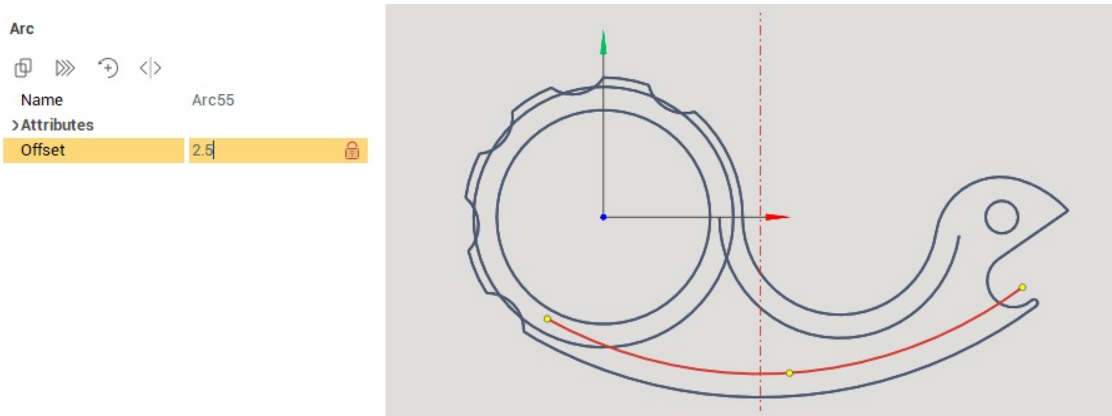
We build another circle, tangent to the previously constructed ones, with a radius of 50:



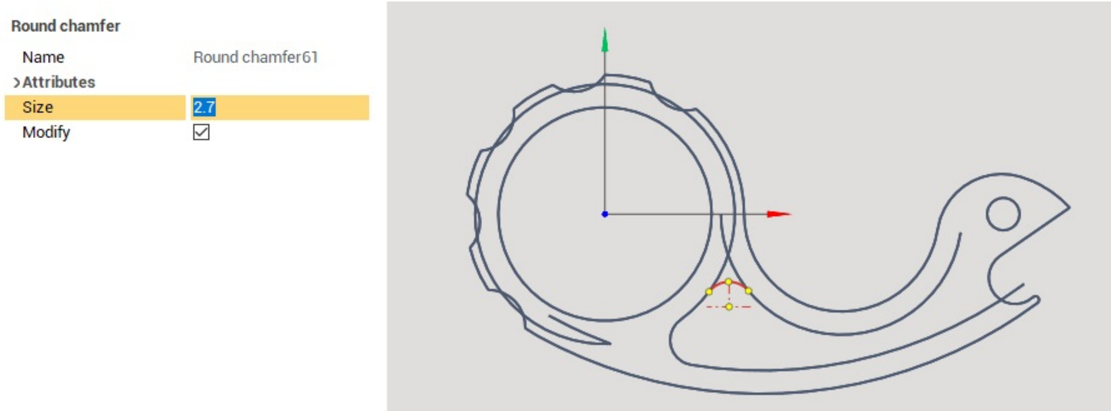
We cut off the extra sections of the circles and get the required figure:



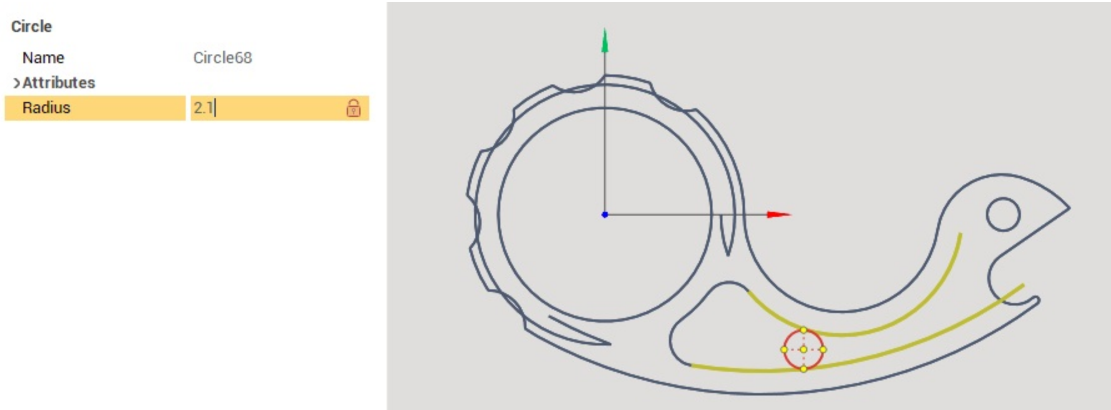
To build the inner figure, we build three equidistant arcs with a distance of 2.5 from the original ones:



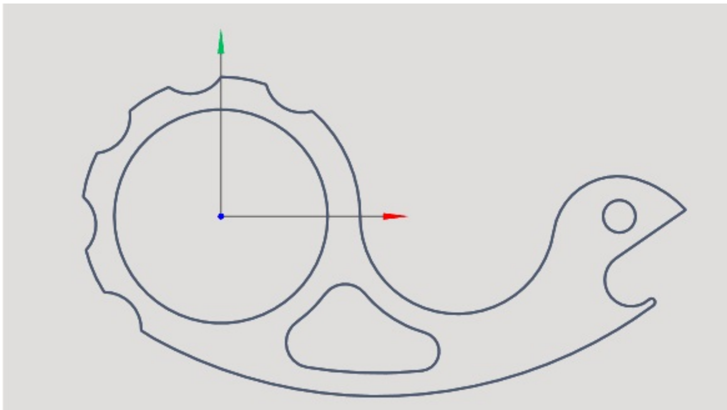
Turn on the construction of the rounding  and specify two elements with a size of 2.7:



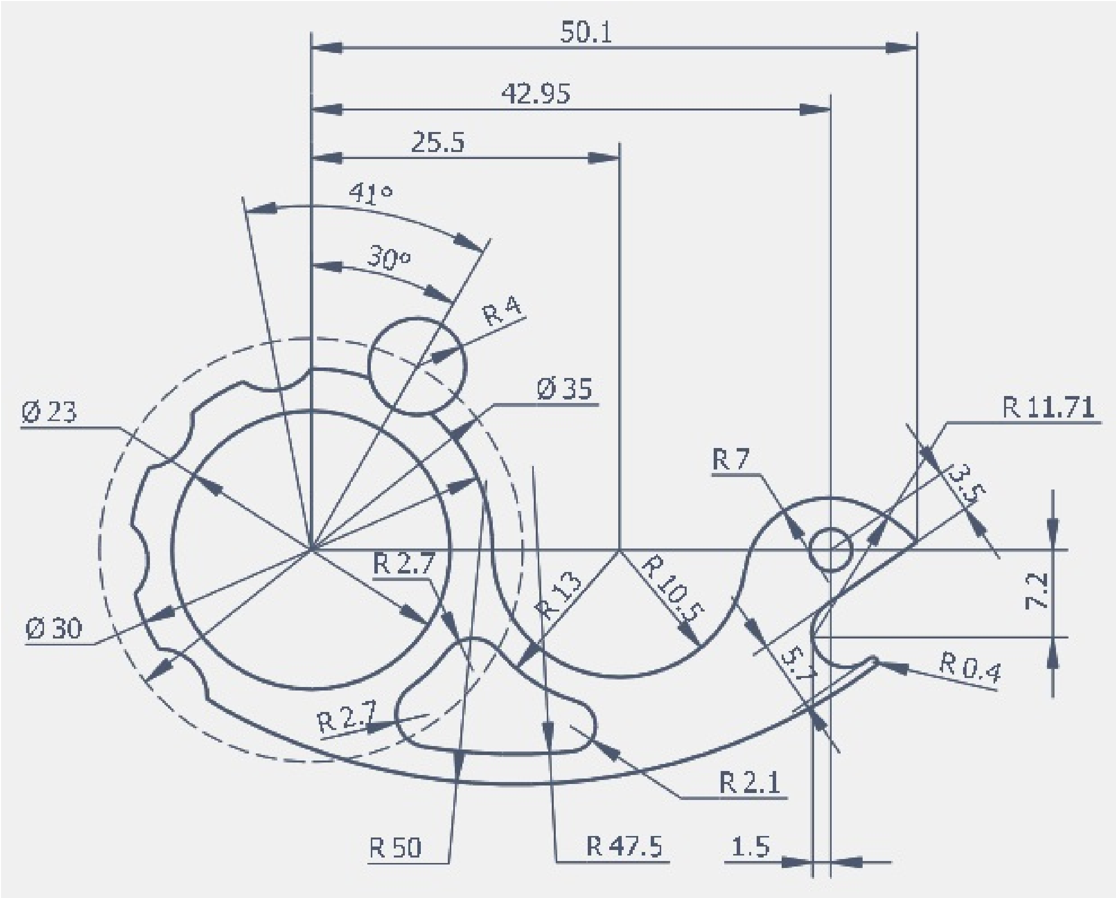
We construct the last element as a circle tangent to two arcs and with a radius of 2.1:



Delete trim elements and cut off unnecessary parts of the arcs, this completes the construction:

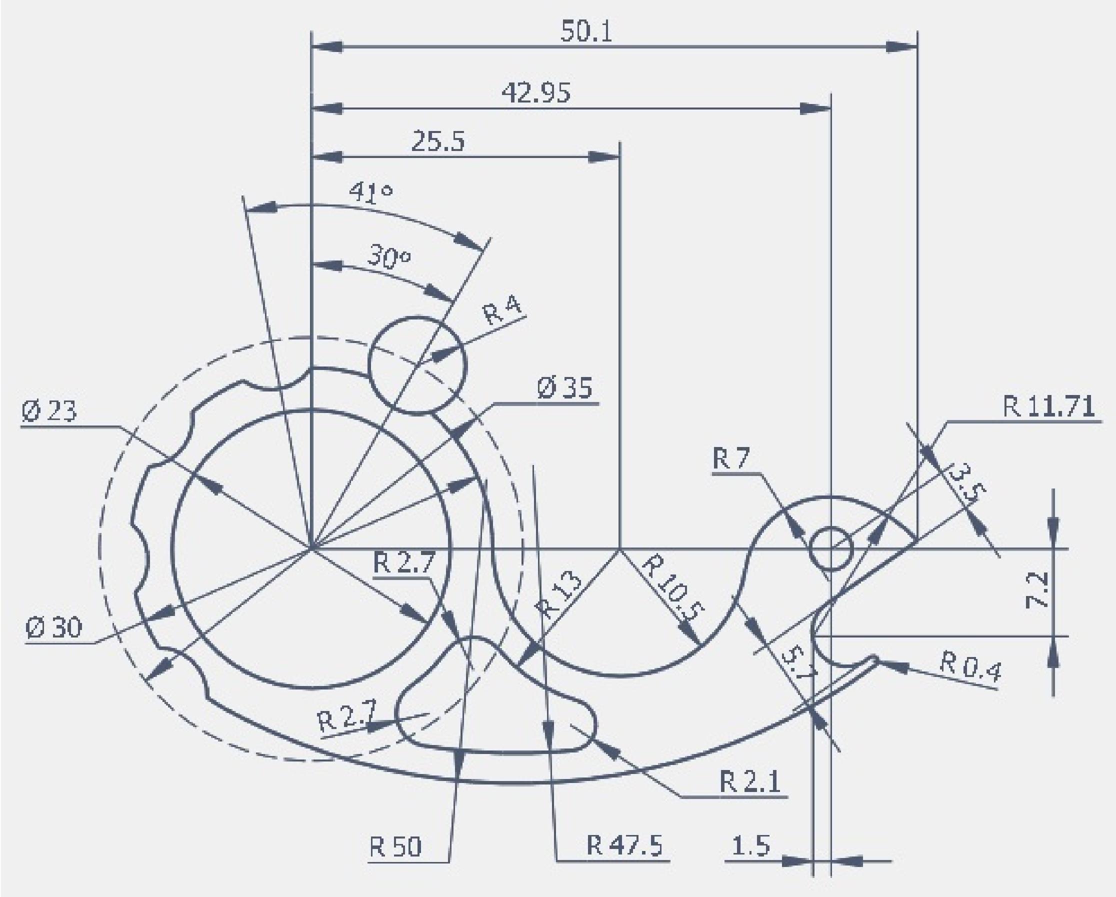


And our drawing after dimensioning:

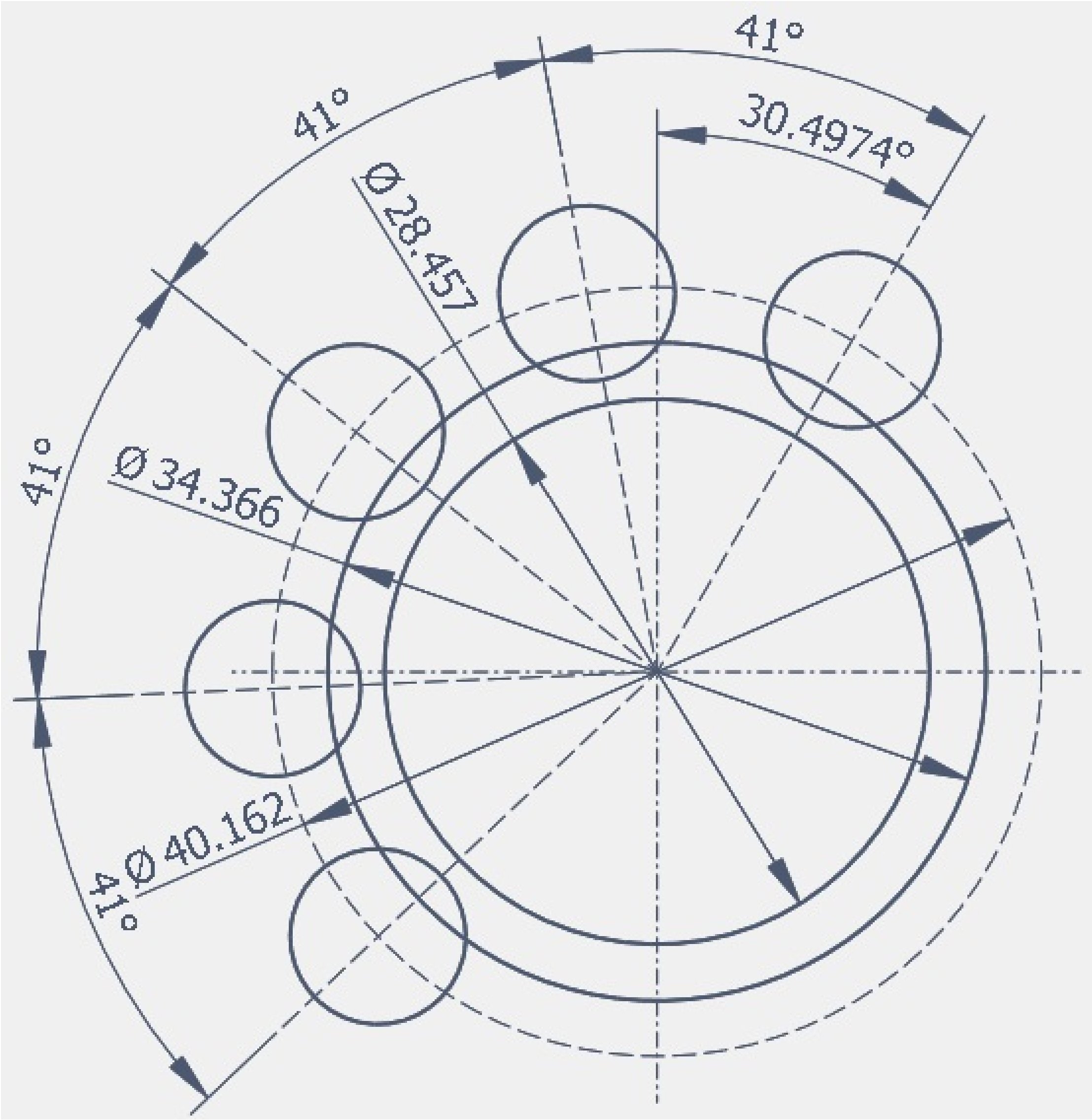


4.4.16 Example of building a drawing using constrains

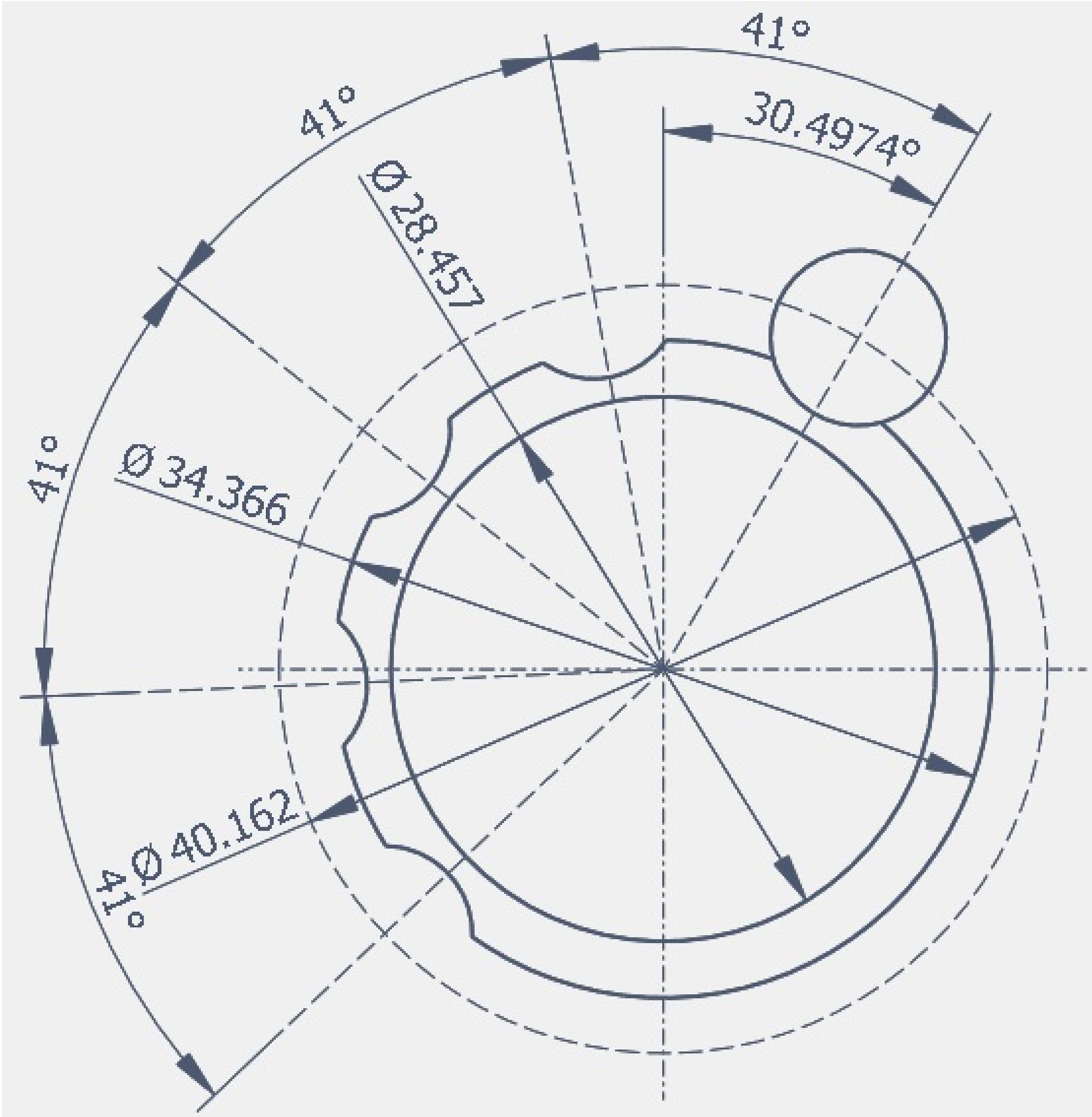
As example, let's take the same drawing as previously:



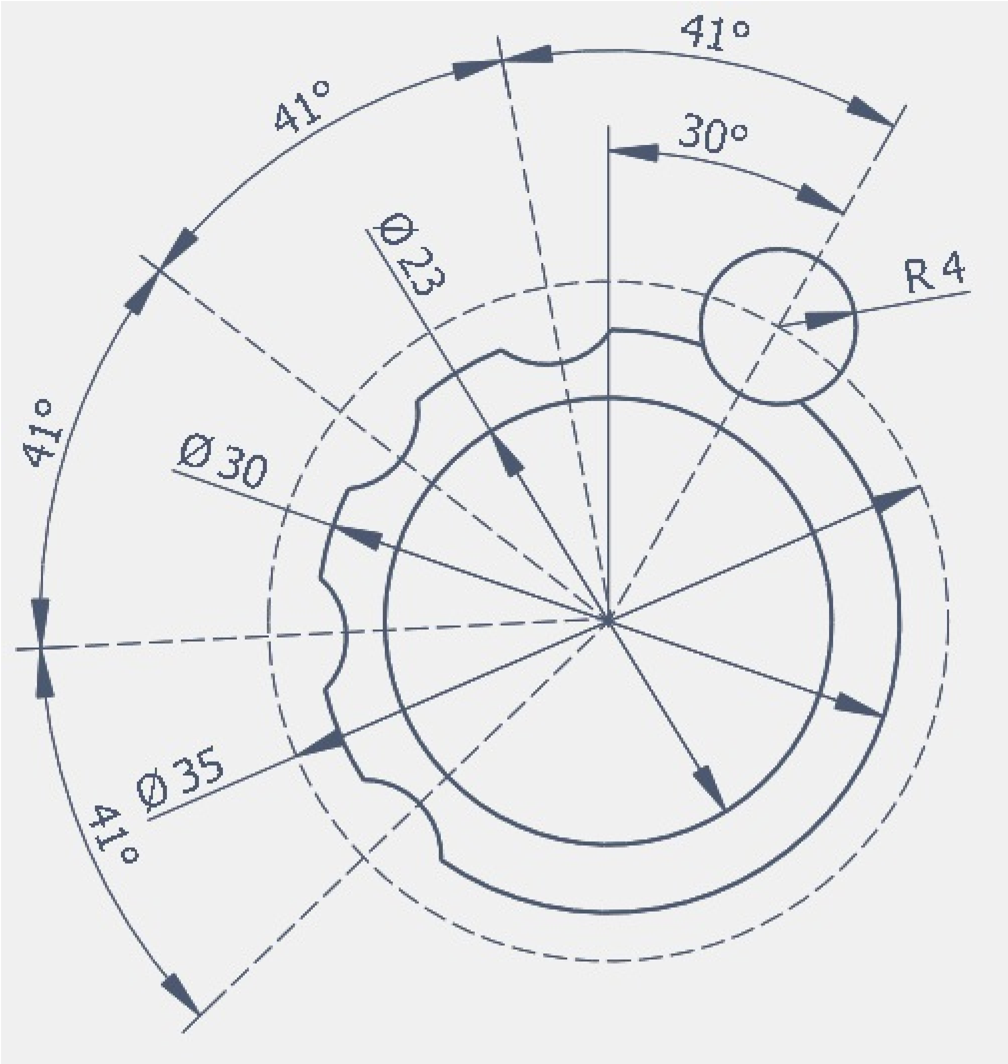
To begin with, from the point 0,0 we build three circles with an arbitrary radius, but more or less respecting the scale. From the center we draw a line at an arbitrary angle for the circle R4. We do not comply with the dimensions - then we will bring them to those indicated on the drawing. We build 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:



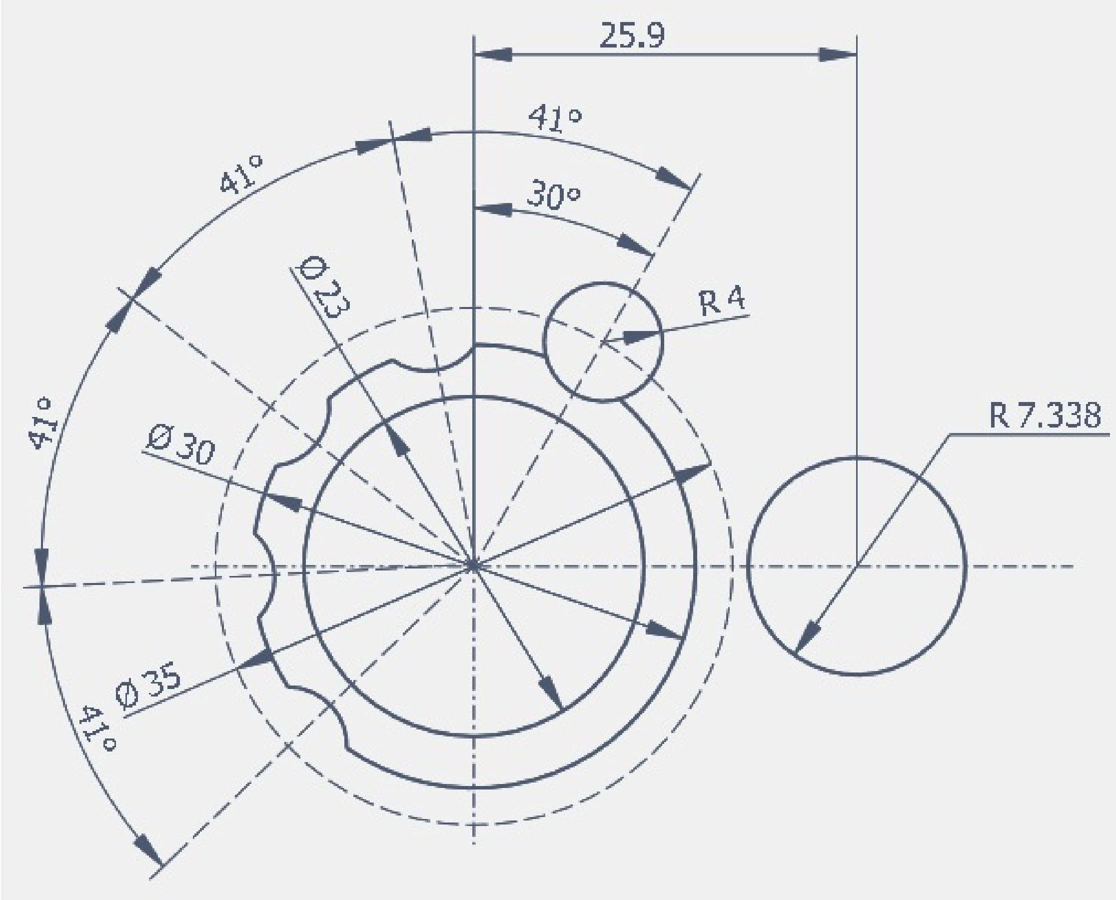
Trimming off the excess:



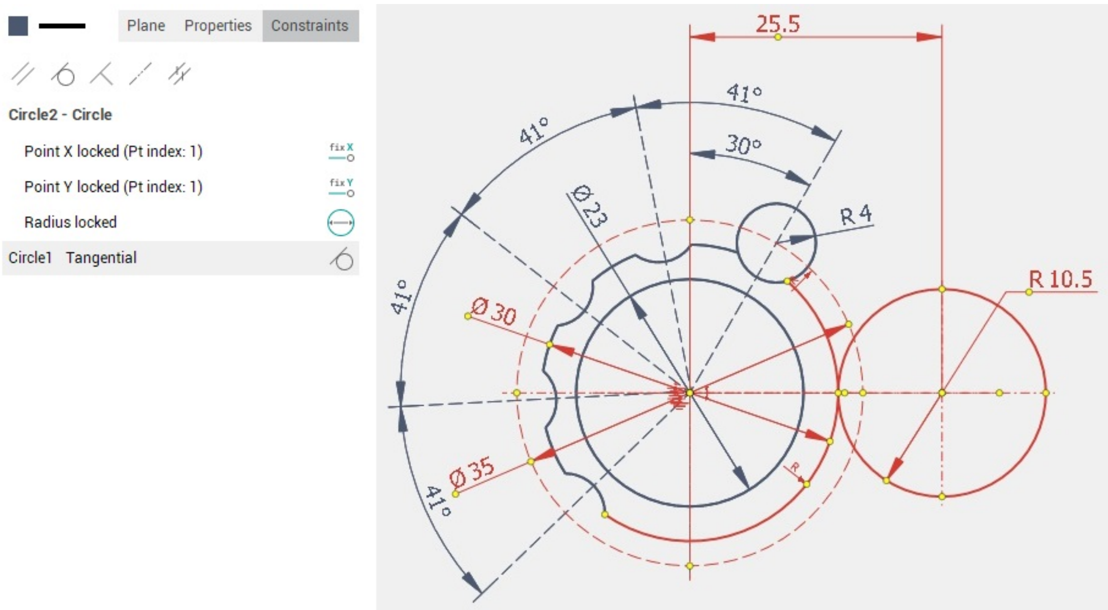
Now select the entire drawing with the key combination **[Ctrl-A]** and change the radius of the circle to 4, then change the angle to 30. After that, fix the angles of the segments and angular dimensions of 41°, after which we again select all the elements in the block and set the values of the diameters according to the drawing:



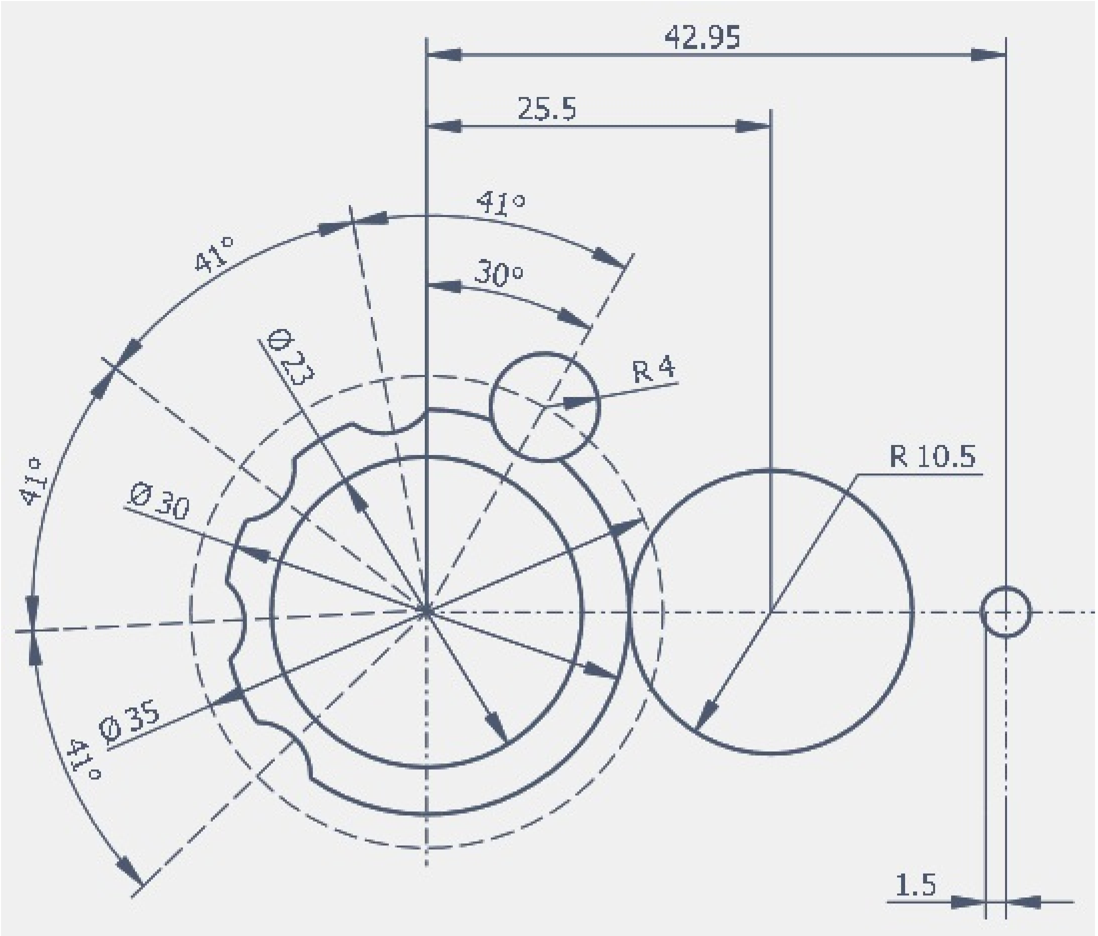
We fix the radii of the previously constructed circles and the resulting arcs. Next, we extend the circle axis to the right with a margin for further constructions. We build a circle with a center on the resulting line with a margin, then we cut off the excess and measure it:



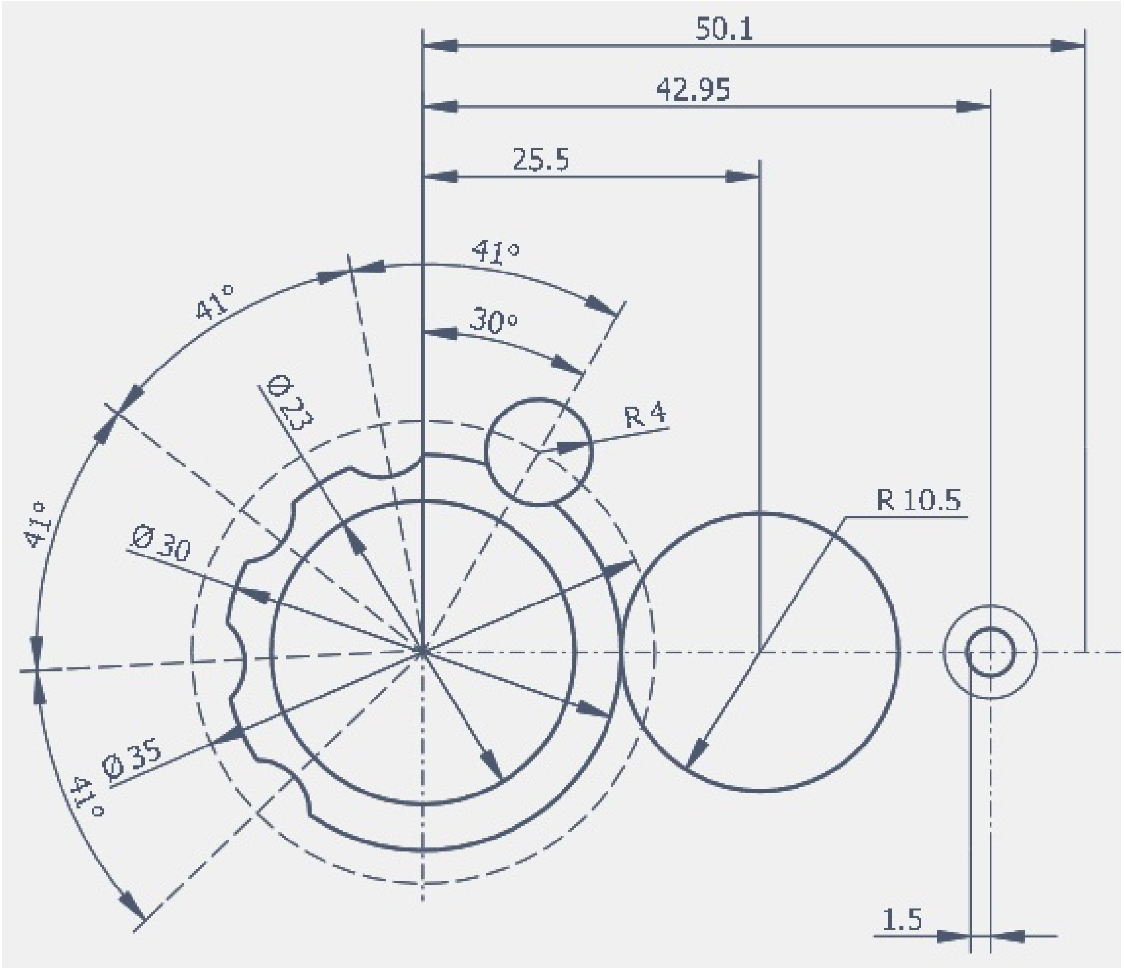
We fix the center of the arc Ø30, select a group of elements into a block, activate a new circle in the block. Next, in the inspector, in the **<Constraints>** tab, add the constraint **<Tangential to the circle>** and specify the part of the circle Ø30, set the radius according to the drawing and fix it:



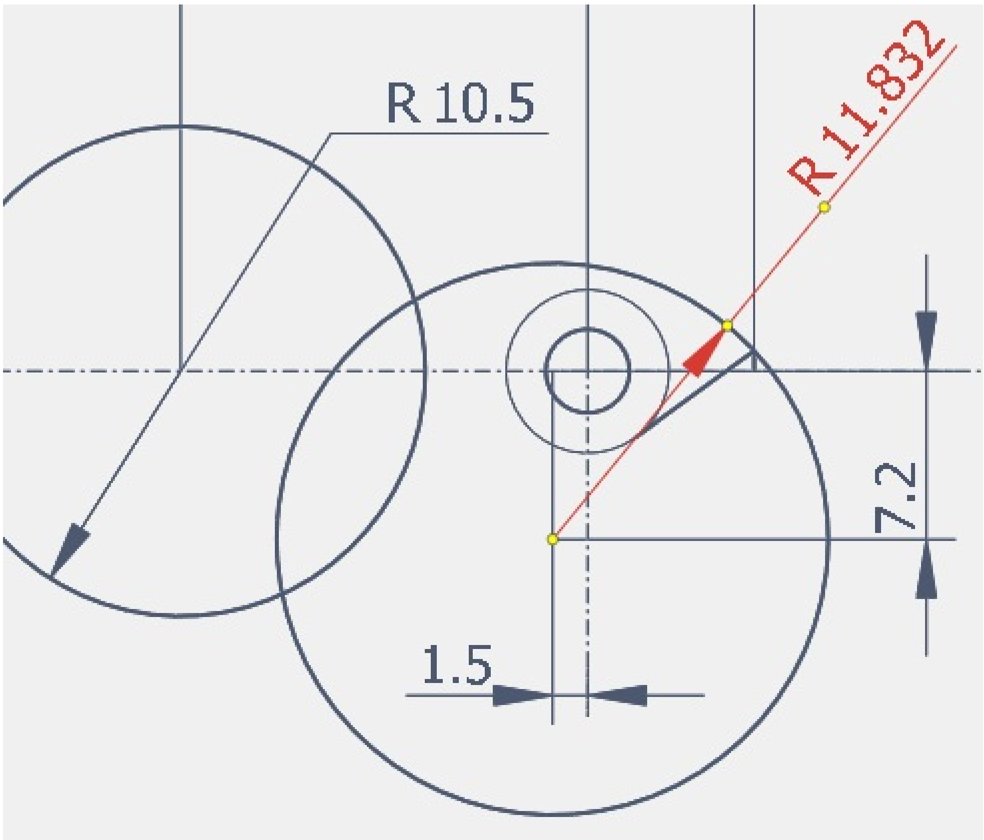
At the resulting circle, you can fix its center. Next, on the axis we build a small circle with a diameter of 3.5 and build a linear dimension from the zero point to its center. In the block, we expose according to the drawing and fix the center. We build a vertical axis for a circle of 3.5 and equidistant 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:



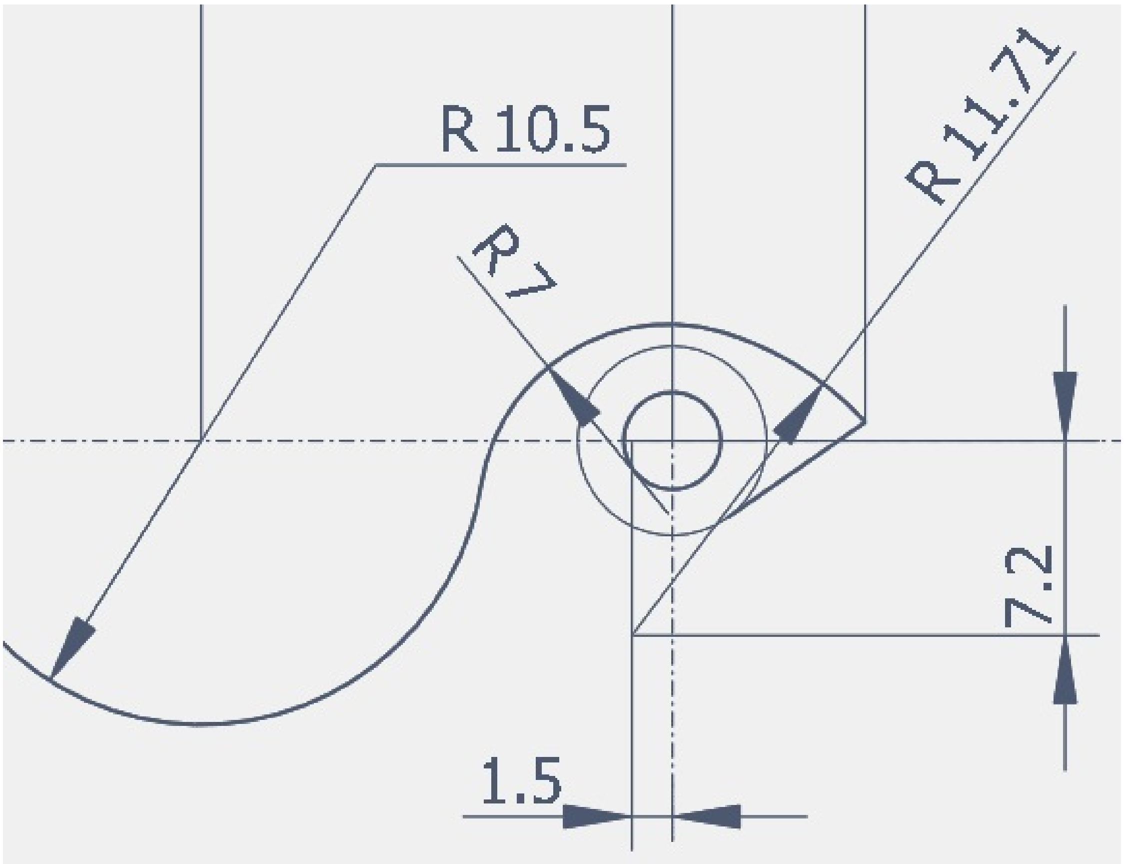
Now we build an auxiliary line for the size 50.1 from the zero point, adjust the size and build an auxiliary circle with a radius of 3.5:



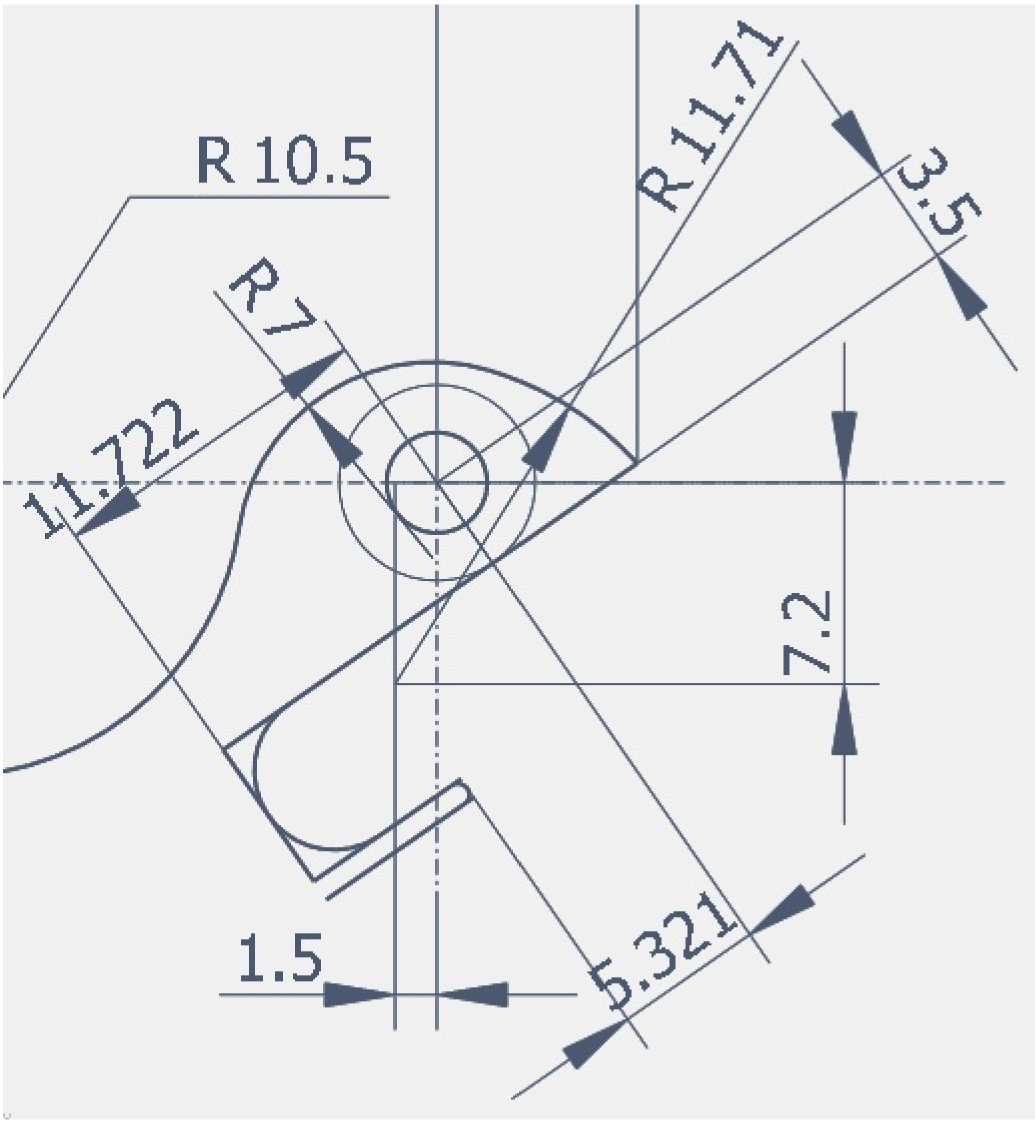
We build a vertical size and set it to 7.2 in the block, after fixing the top point. We build a line tangent to the auxiliary circle with a radius of 3.5, which would intersect the auxiliary line just above the axis. We cut off the auxiliary line and build a circle with the center at the resulting point and passing through the intersection point of the tangent and the vertical at 65.4. We measure the resulting circle:



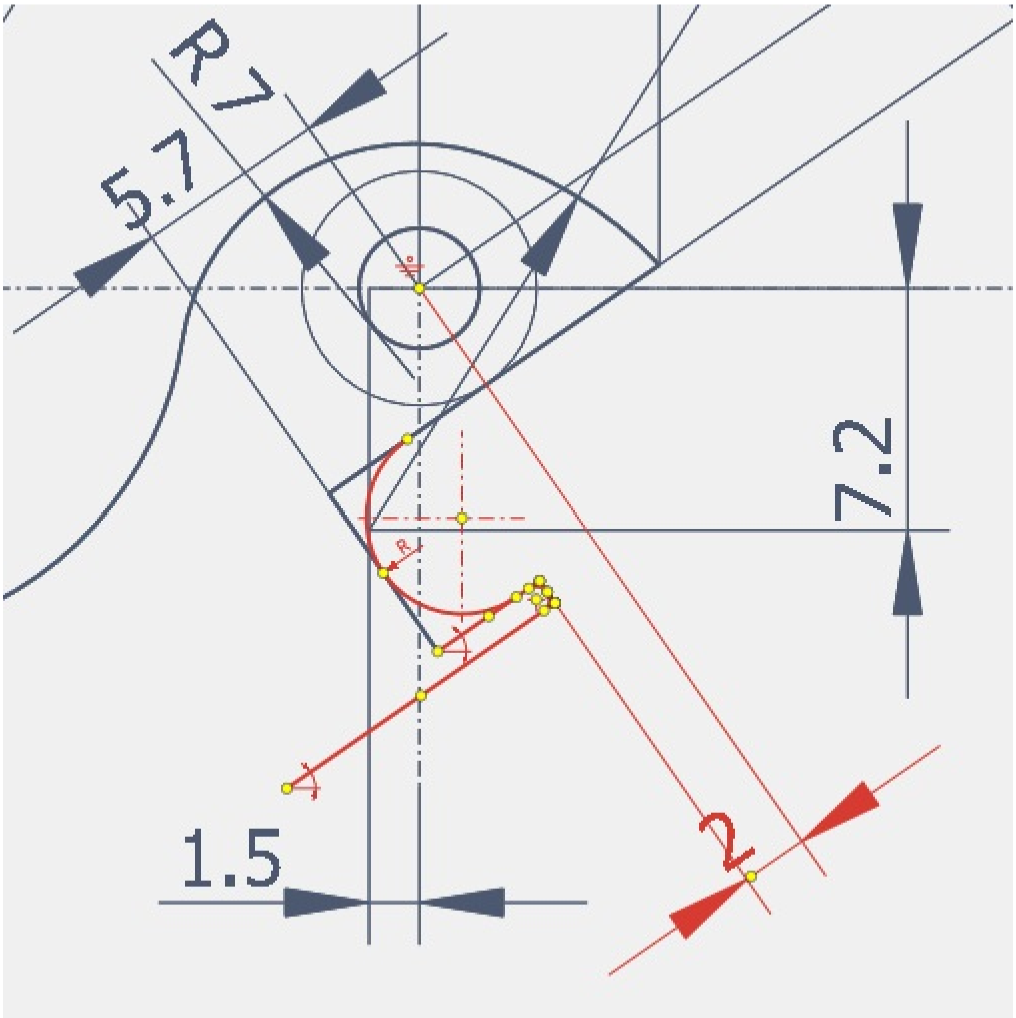
In the block, set the radius to 11.71, then build an arc with a radius of 7 tangent to the circles R10.5 and R11.71, cut off the excess.



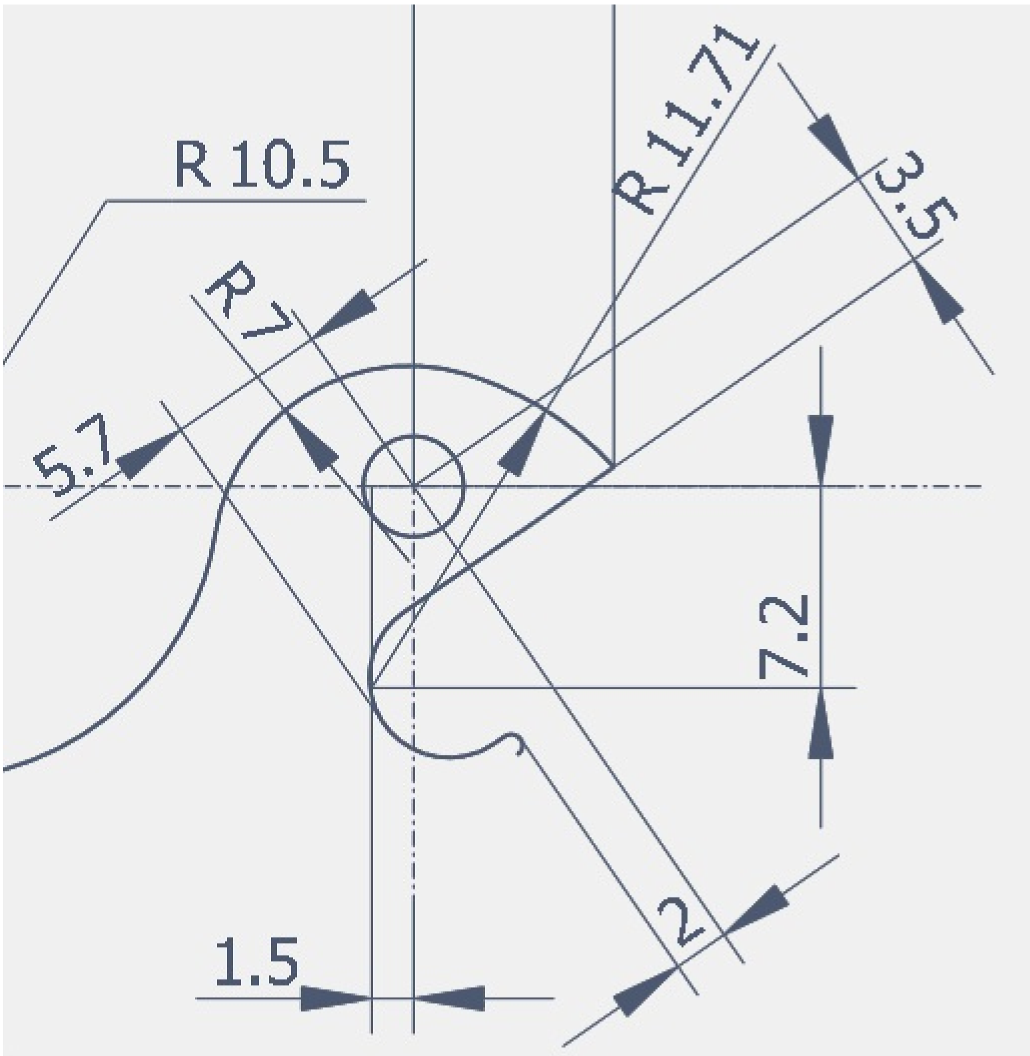
To build the arc R5.7 and R0.4, we extend the inclined line and build an equidistant line to it at a distance of 5.7, and from it another one at a distance of 0.8, connect the first two with a line and enter an arc between them. We build a time line passing through the center of a small circle and parallel 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.



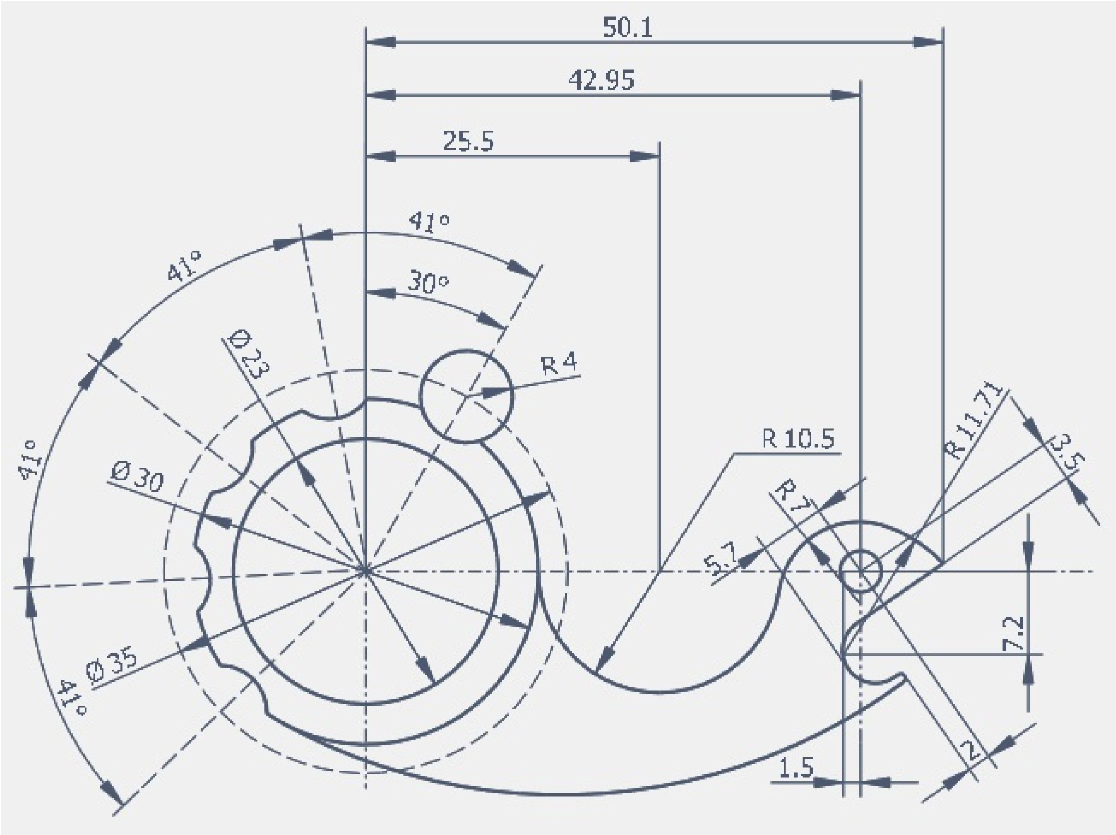
We fix the angles of the lines, and the points of the dimensions lying in the center of the circle and in the block are changed to the values from the drawing.



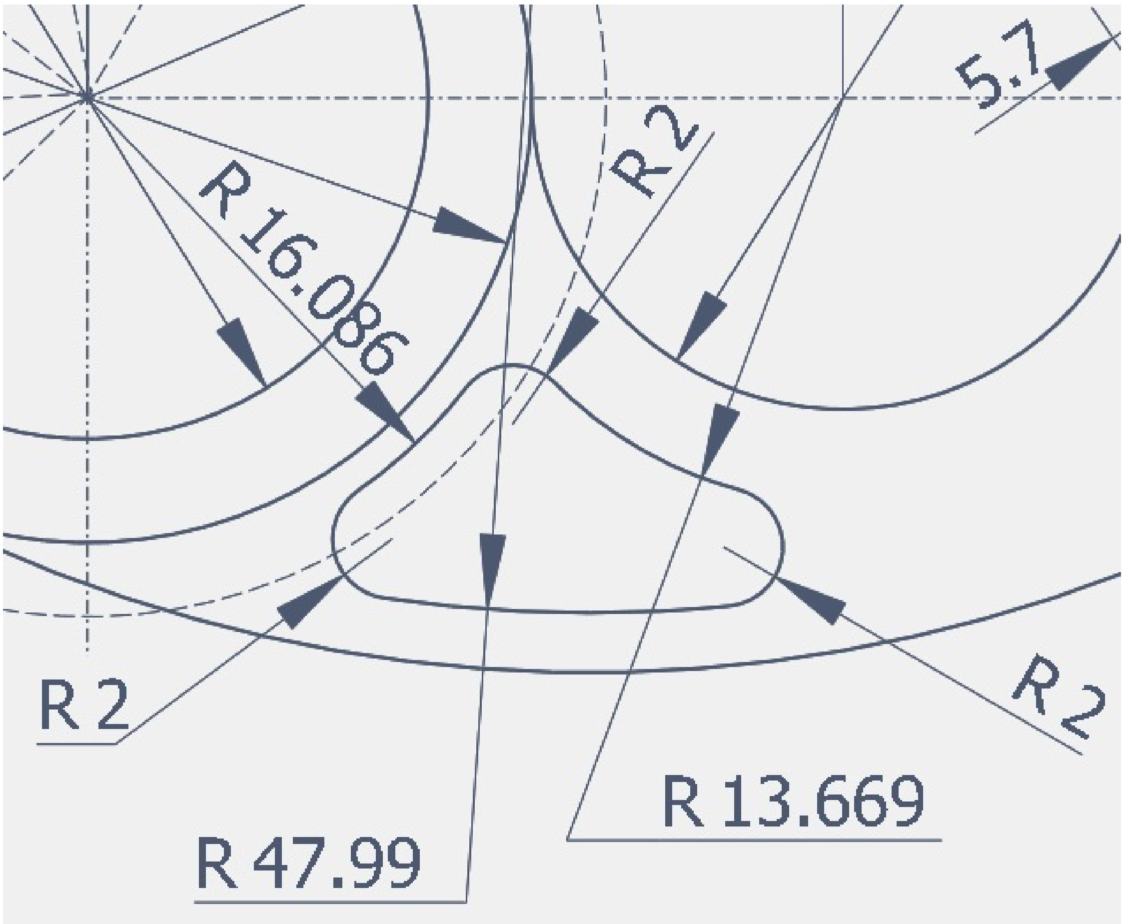
We cut off the excess or change the line type.



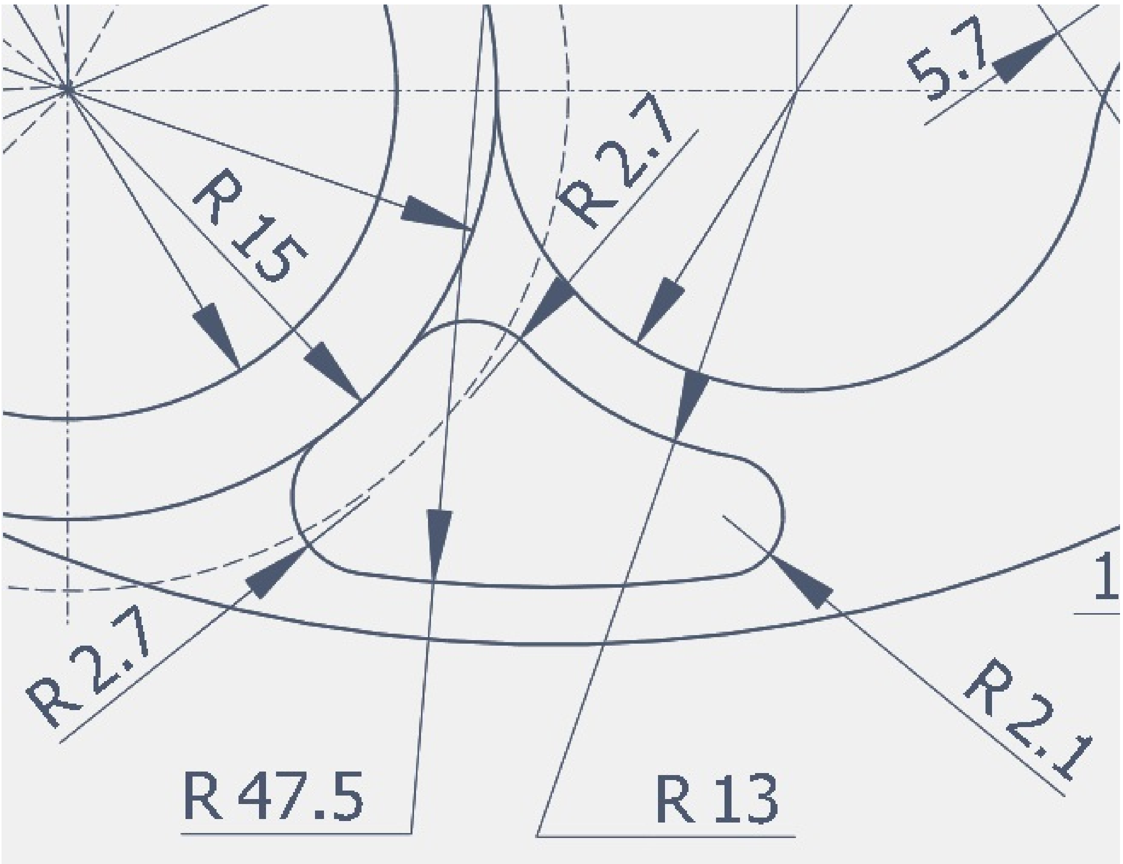
Conjugate arc R0.4 and left circle with R30 arc R50. We cut off the excess from the arc R0.4



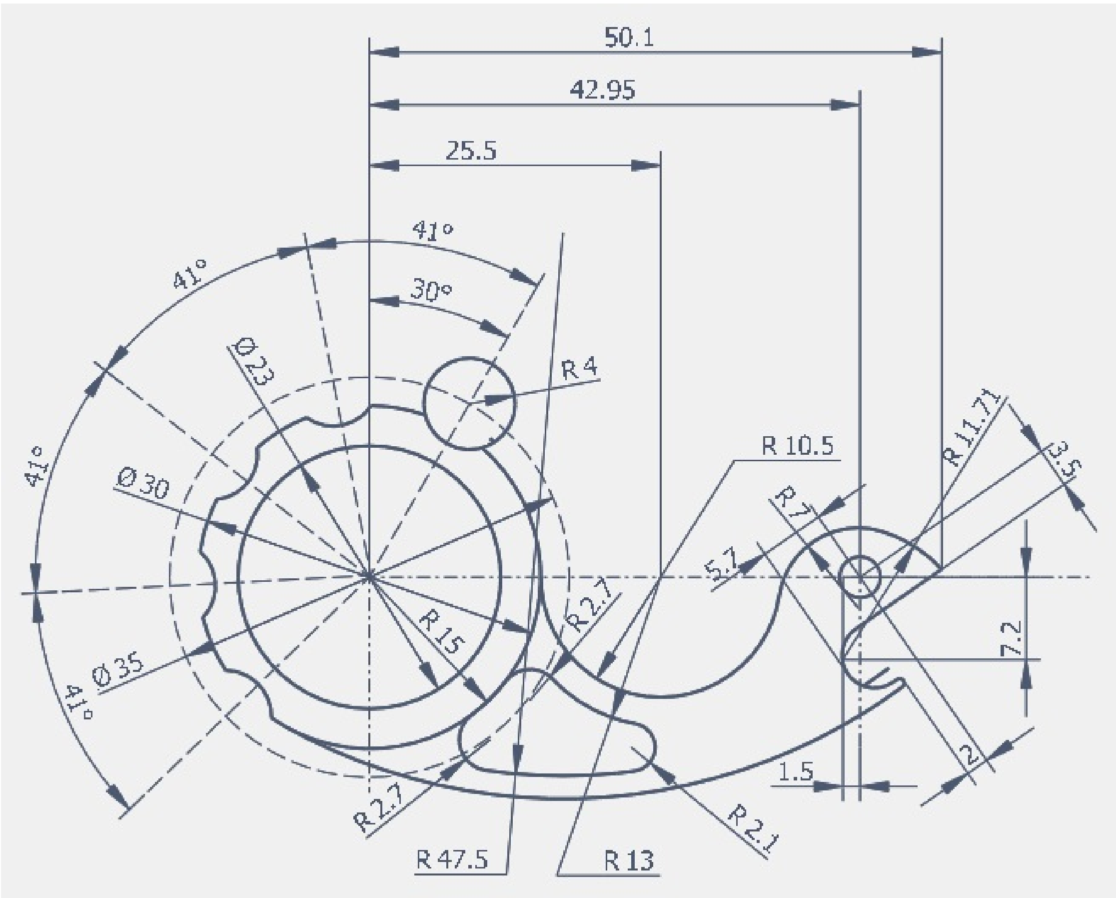
Now we build 3 arcs parallel to R50, R10.5 and R30, fix the centers and match them with arcs, set the dimensions and cut off the excess.



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:

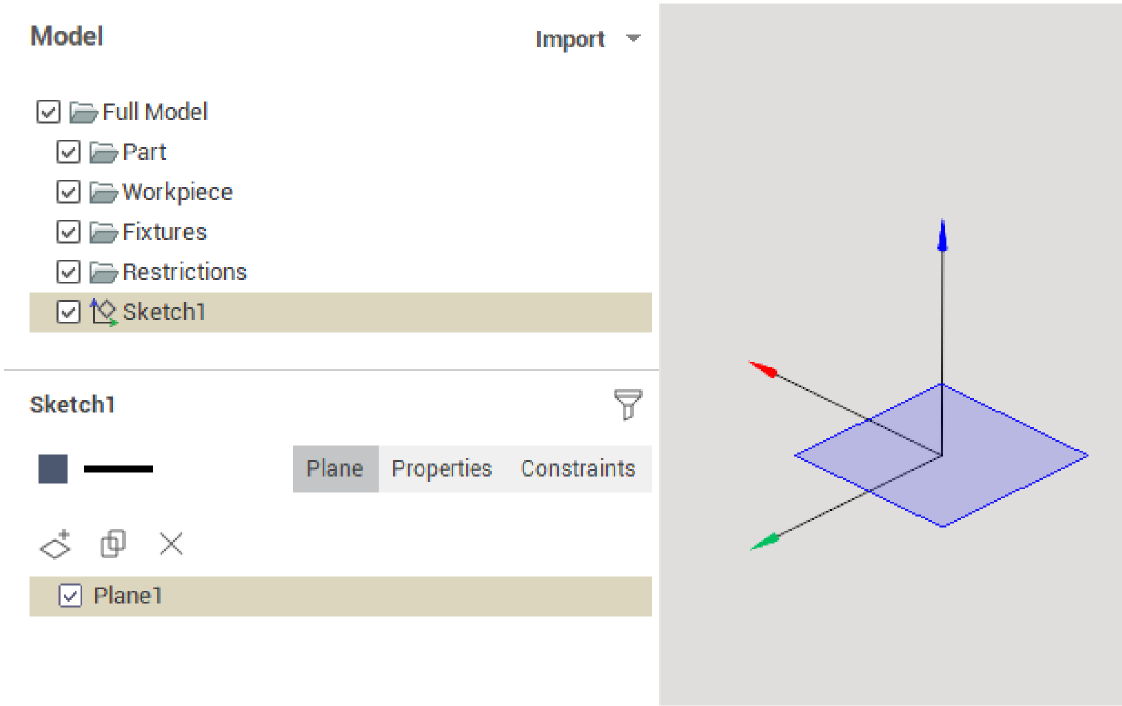


Thus, we got the required drawing.



4.4.17 Planes

Planes are controlled on the corresponding tab:

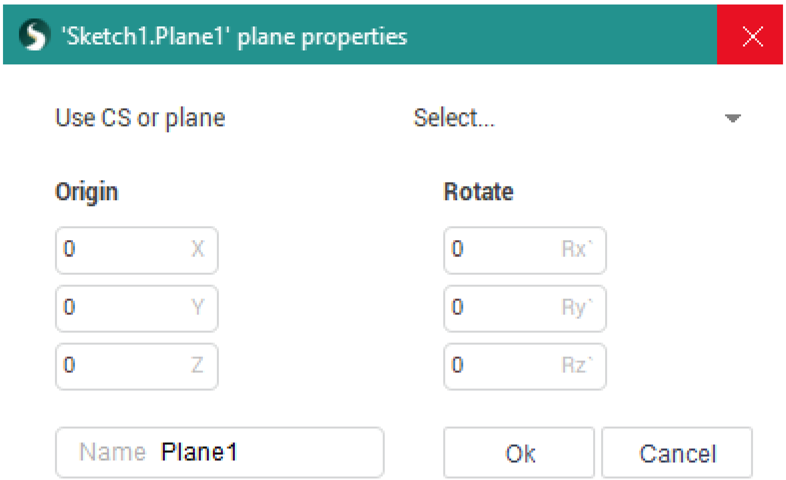


Here:

-  - 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.

5 Creating machining technology

No content in this page. See child topics

5.1 Common principles of technology creation

The process of **creating a machining technology** in SprutCAM X in general consists of the following consecutive steps:

1. Selecting or creating the **machine**. If needed, the machine schema in SprutCAM X format is created before that.
    - During the opening, SprutCAM X 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). See this tutorial for an example of the machine selection.
    - 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. SprutCAM X supports Multi parts projects, 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.

5.1.1 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.

See the robot XML description tutorial for an example of creating a machine schema for further use in SprutCAM X (not only robots can be defined in such a way).

5.1.2 Machining sequence

The **operation** is the basic unit of the technological process in SprutCAM X 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

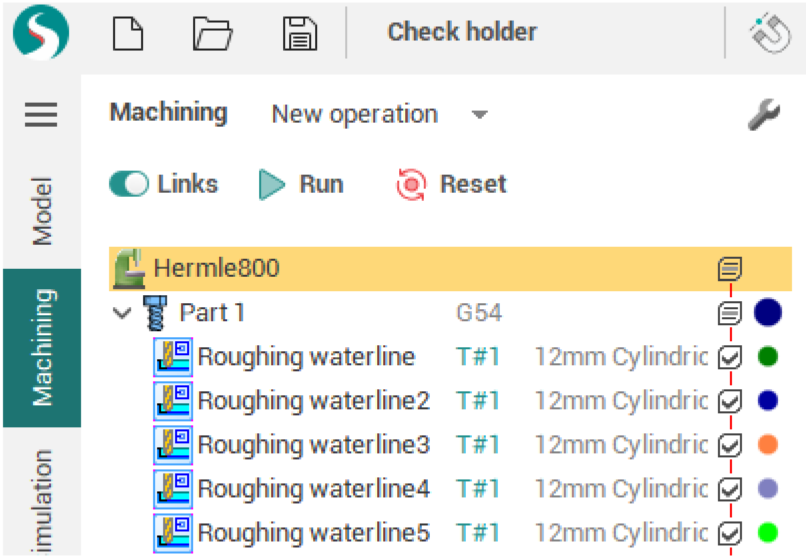
See the Defining machining sequence article for more info about the operations' common user interface.

5.1.3 Operations tree

The machining sequence in SprutCAM X 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.



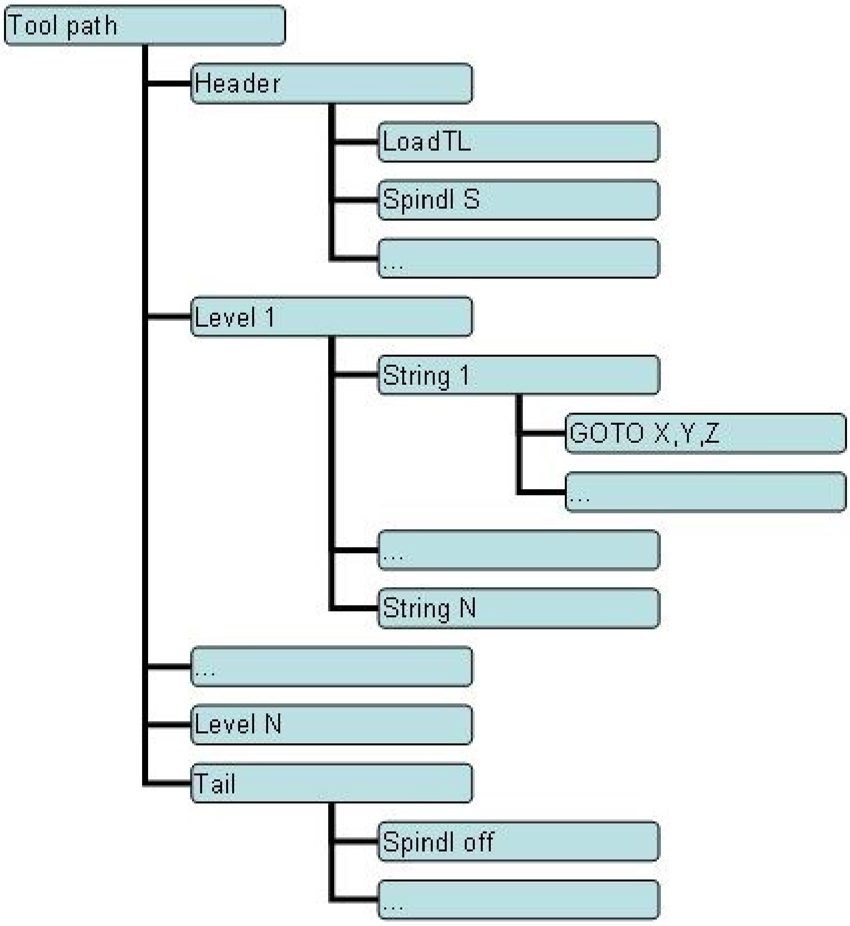
Operations tree example.

5.1.4 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 <Plane roughing operation> is shown below:

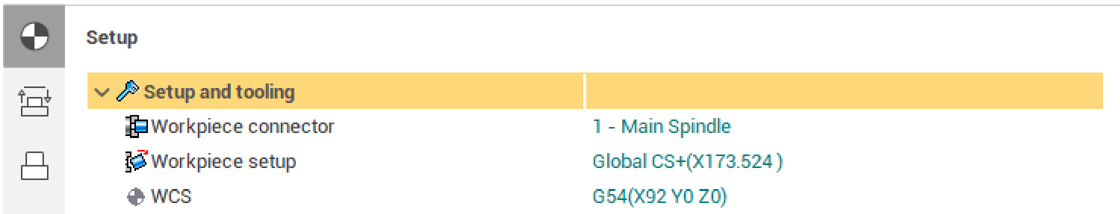


5.1.5 Setup stages

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.

See also:

Selection of a machine and its parameters definition

Multi parts projects

Defining machining sequence

Creating new operation

Executing operation

Generating CNC code

Machining report

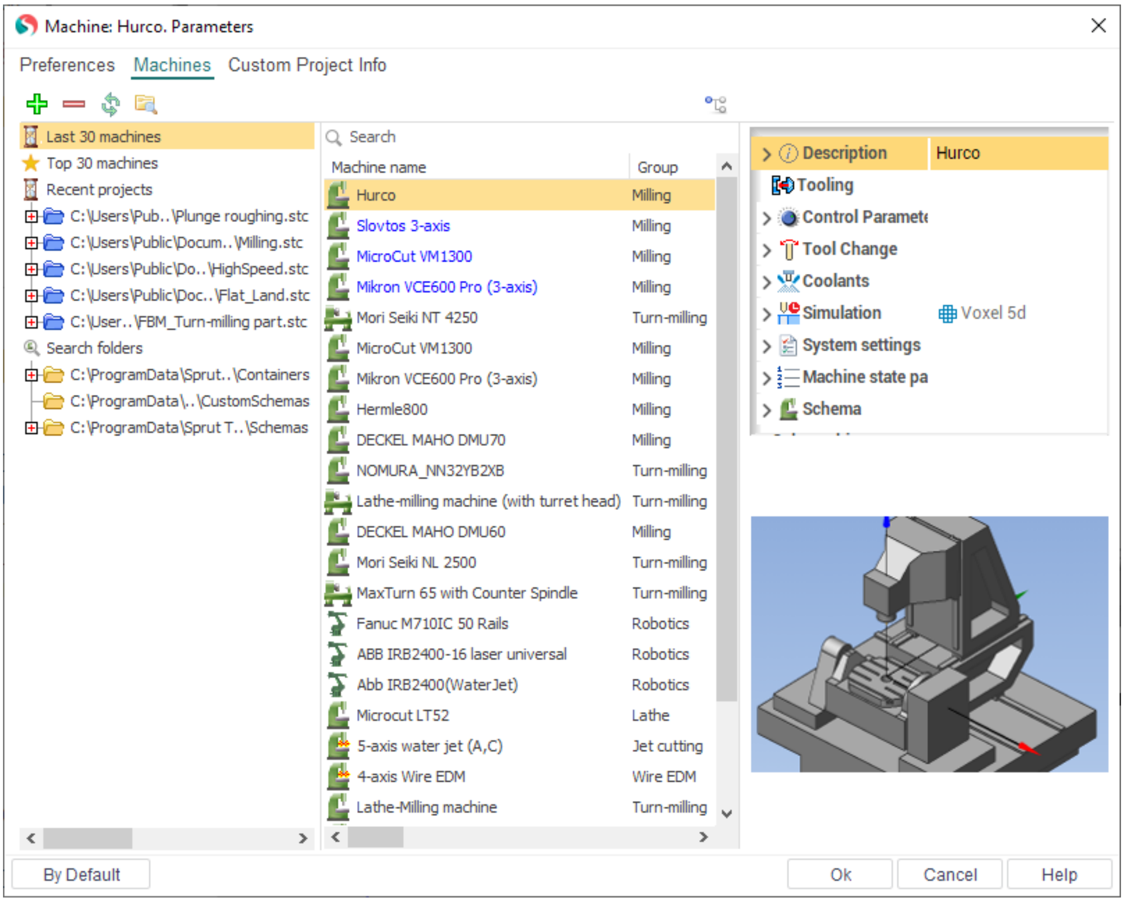
Operations setup

List of types of machining operations

5.1.6 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 G-code based simulation.

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.

See also:

Common principles of technology creation

5.1.7 Defining machining sequence

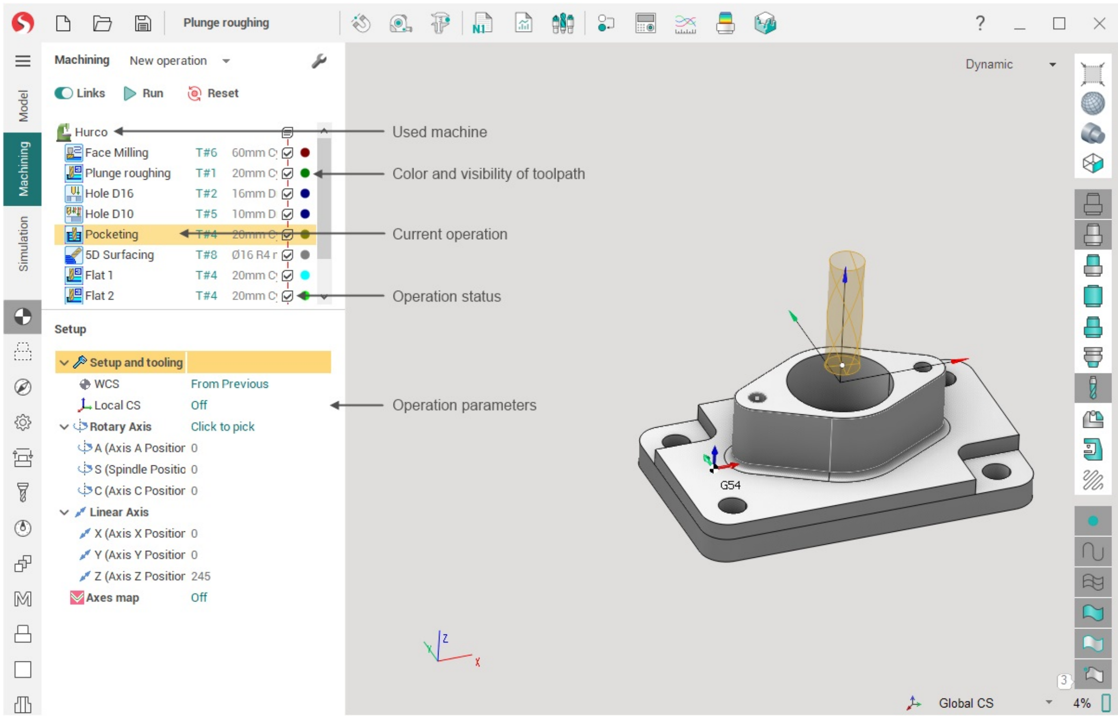
Before defining a machining sequence one should specify the produced part, the stock and the fixtures for the sequence. That is, one should define what is produced at first (the part), than what from it is produced (the workpiece) and only than how it is to be produced (the machining sequence).

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.

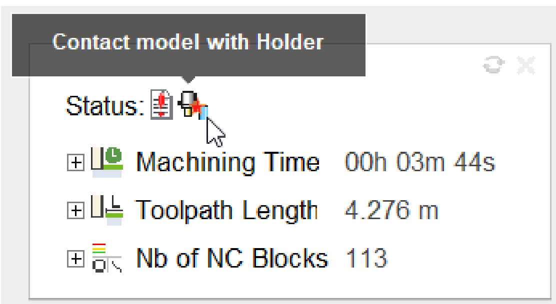
Current operation can be deleted, renamed, copied or cut to the clipboard by the standard keys or from the pop-up menu:

- [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.

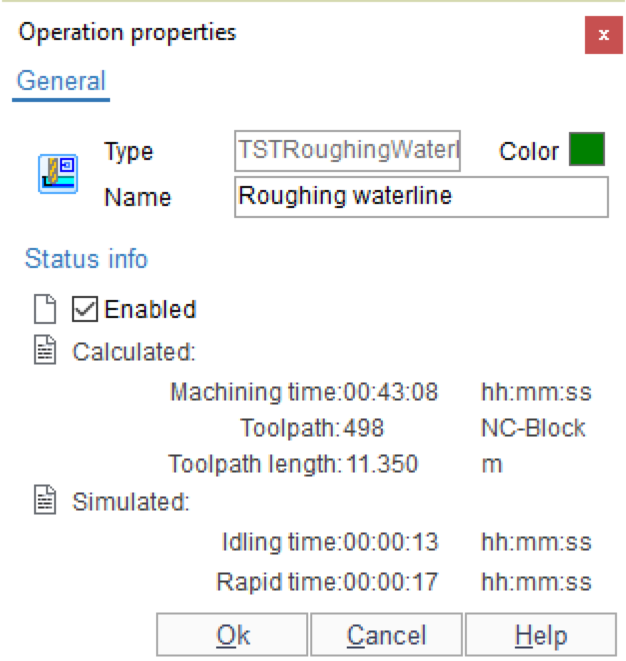
Mouse can change the structure of the job tree. To do it press and hold the left mouse button on the required operation and move it to the required place.

There is the icon near the every operation. This icon shows the operation status:

- – Operation is disabled. This operation is not calculated, is not output to the NC program and is not considered the rest re-machining operations are calculated. To switch on/off the operation one can either double click on its caption or select the "enable" command from the operation shortcut menu. The operation group is disabled if all operation inside are disabled;
- – Operation is not calculated (has no the toolpath);
- – Operation is calculated (has the toolpath), but not simulated yet. The operation group is calculated if all operations inside are calculated.
- – Operation is calculated and partially simulated, no simulation errors detected. Partially means that simulation has been performed without machine collision control, but gouge detection, tool holder collision and machine axes limits control performed successfully.
- – Operation is calculated and fully simulated without errors. A green mark means the simulation was performed with collision control of the machine and with all other checks. The operation group is simulated if all operations inside are simulated;
- – The errors were found while the simulations. The operations group is marked by error even, if one operation inside is marked by error. Click to the icon to show an operation status panel with detailed information about errors found.



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.

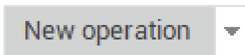


The window shows the operation type and the icon corresponding to the type, operation name, tool path color. If the operation is calculated then the number of the commands in tool path and the machining time is shown. If the operation is simulated then idle, work, rapid and auxiliary time is shown. If the errors were detected, while the simulation then the information about error is shown.

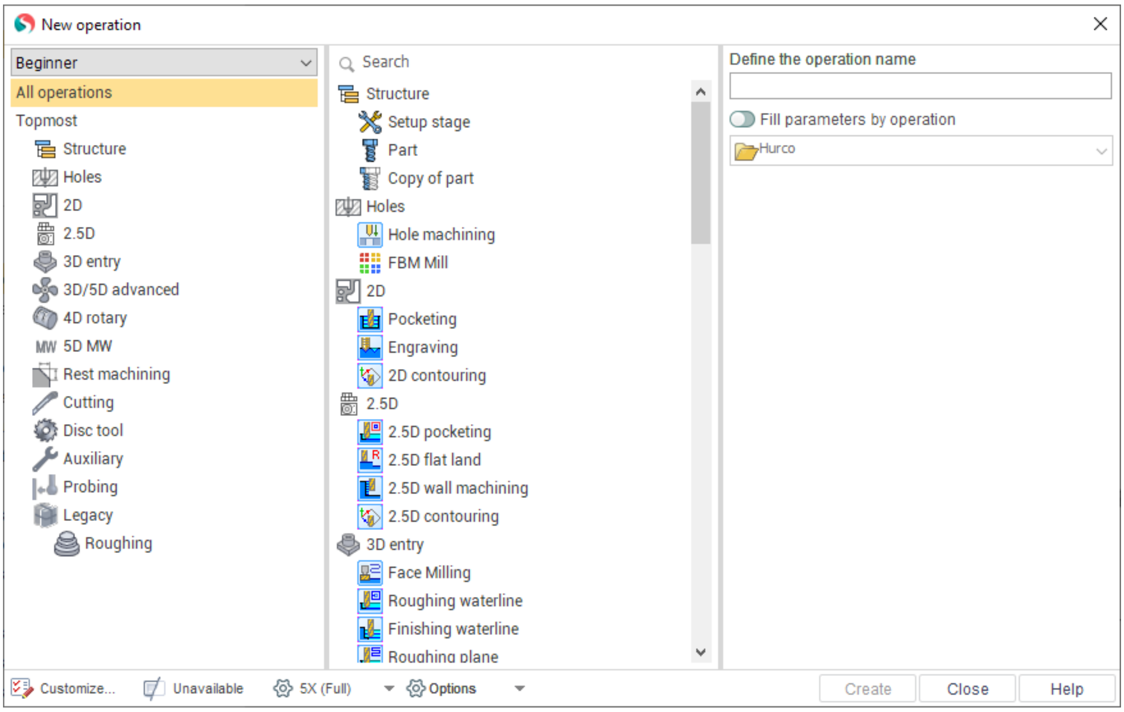
See also:

Common principles of technology creation

5.1.8 Creating new operation

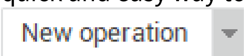
To open the window for creation a new machining operation just press the 

button.

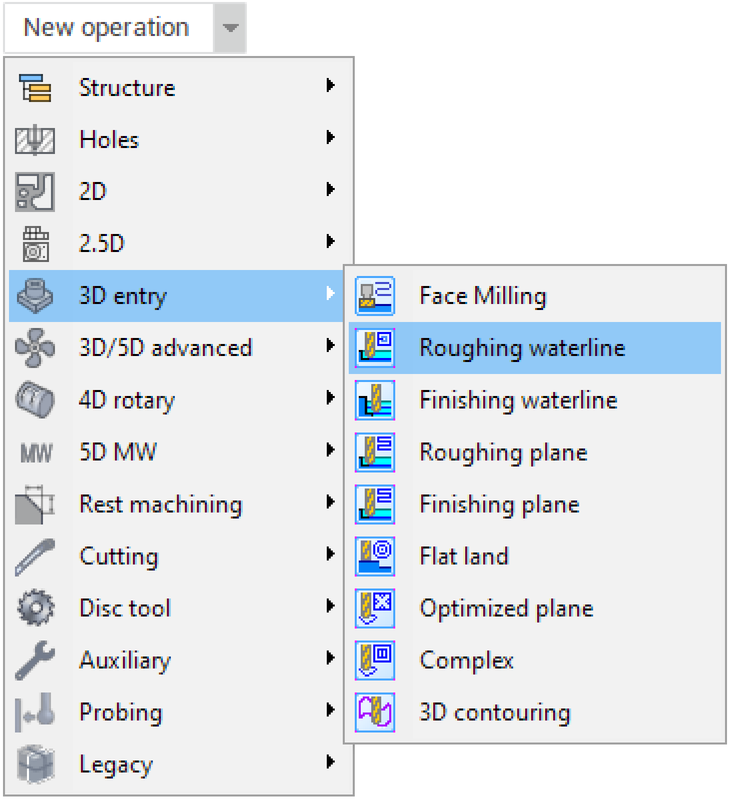


The window is divided into three parts. Left one is the list of operation groups. Here you can select one of the groups, and in the middle part of the window would be displayed a list of operations that belong to selected group. You can also select "All operations". Then all the available operations will be shown in the list of operations. For a quick search of desired operation you can also use the search field, that is located above the list of operations. When you enter characters in the list of operations remain visible only the ones that include the specified sequence of characters. 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.

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.

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 

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.

In the right pane of the new operation window displays information about the selected operation. Here you can specify the name of the newly created operation ("Define the operation name" field), and copy settings from another operation. It is enough to enable <Fill parameters by operation> check-box and select desired operation from the combo box. At the bottom of the information panel, there is a picture that explains the meaning of the operation. You can start or stop playback of video clip by clicking on the picture.

Depending on the presence or absence of the licence options, and depending on the machine, some operations may not be visible in the list. In order to become visible you can click the "Unavailable" button. In this case they will be gray in the list. When you select such operation, the reasons for which this operation is not available will appear in the Information panel.

By default, operations can be divided into groups such as "Roughing", "Finishing", "Rest machining", "Lathe", "5X", etc.

"Roughing" operations provide a selection of the entire workpiece material, which lies outside the machined model and outside prohibited zones. Generally roughing operations are used in the initial selecting of material in that cases where the shape and dimensions of the workpiece are significantly different from the shape and dimensions of the workpiece.

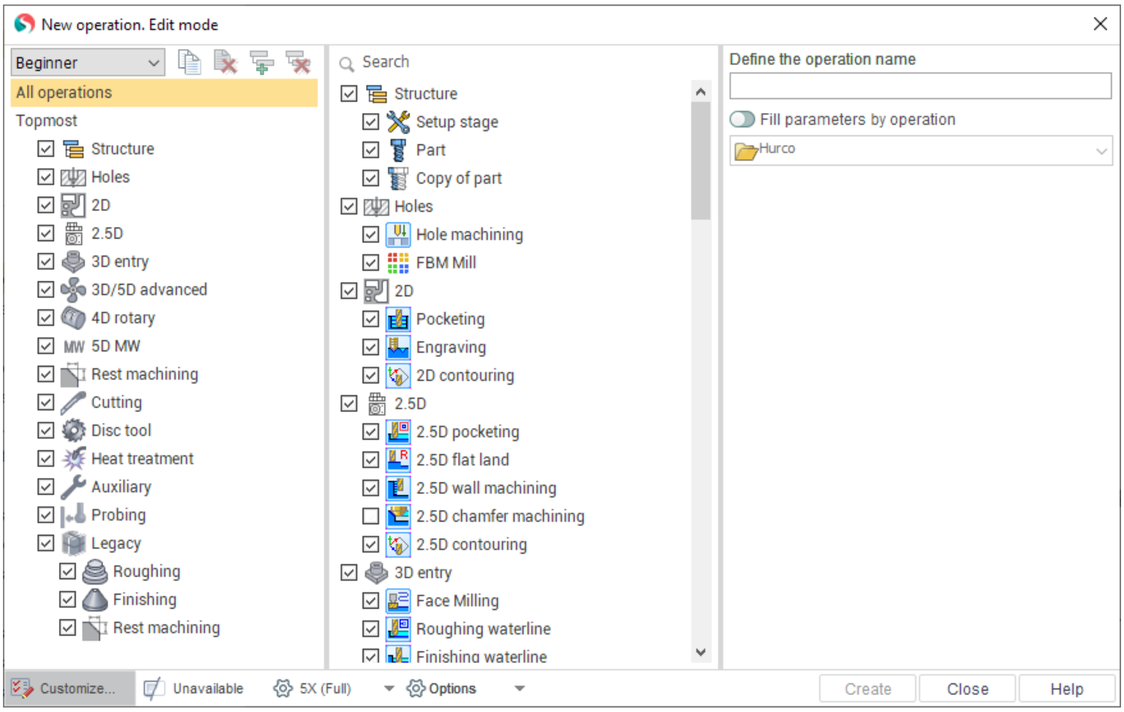
"Finishing" operations perform machining only of the final surfaces, without workpiece material removal. The finishing operations are normally used for the final clearance of a models surface(s) after previous machining (e.g. roughing). They can also be used if the workpiece and final model do not differ too much or if used workpiece is made of soft material.

"Rest machining" operations allow to perform machining only on those areas where material was left after previous machining operations. The rest machining strategies are identical to other strategies, only different values are set. The roughing remachining operations perform removal of the entire volume of residual material, and the finishing ones, machine the actual surface of the model only in un-machined areas. The rest milling operations allows to optimize machining for complex details.

They are designed to be used for roughing or finish rest milling using a tool of a different shape or smaller diameter than the tool used in previous operations.

Other groups are created. They are based on the same machining type of operations included into group or depending on other common parameters such as developer or licence.

However, the breakdown of operations on a groups is a rather nominal. You can always redistribute operations between groups, create new groups, hide unnecessary operations and groups. To do this, enter the edit mode by pressing the "Customize..." button.



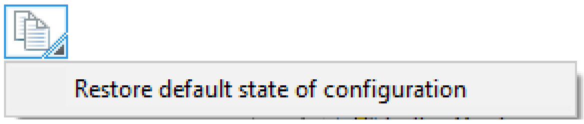
The window will change, there will be new buttons and check-boxes next to the names of operations and the groups. Also it will be available to drag and drop operations between groups. The check-boxes determine the visibility of operations and groups. To create a new or delete an existing group you have to press the appropriate button in the upper part of the window. Then you can click on the group name and change it to any other. You can also drag the group to a different location or even inside another group.

Creation of a new operation with your own settings is also possible. To do this, create an instance of operations, based on which you want to create a new operation, close this window, change the settings of the operation as you need and then select in the pop-up menu of the Technology page "Save as user operation ...". This will bring up a dialog box where you can specify the file name, in which the operation will be written, and press Ok. You can then re-open the window for creating operations and move added operation to the desired group.

The visibility of operations and a breakdown of them into groups in conjunction with the user operations allows fine-tune the system for your own purposes. Moreover, you can create several layouts for different purposes. Here they are called configurations of operations. For example, working with one machine requires one set of operations, the second machine - the other. The current configuration is selected in the drop-down list at the top left corner of the window. In the edit mode, you can create multiple configurations. For this purpose, next to the list, there are add and remove buttons.

 removes current configuration. Last configuration in the list can not be removed.

 adds the new configuration, all properties are copied from the current configuration.



Drop-down menu on the button to create a configuration allows you to restore all of the groups and the visibility of operations in its original form as laid down in the system default.

See also:

Common principles of technology creation

5.1.9 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 

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.

See also:

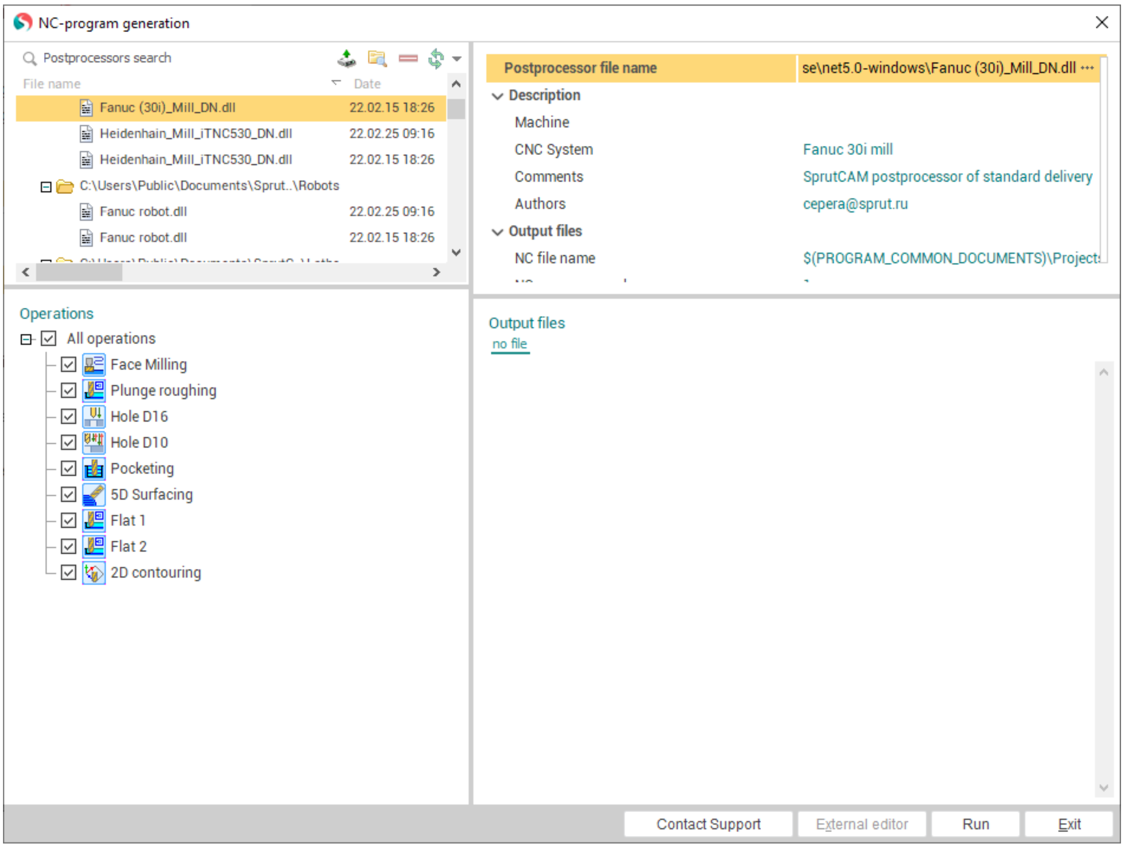
Common principles of technology creation

5.1.10 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

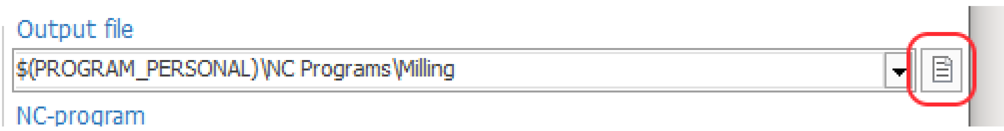
*.sppx extension. The output of CNC code is performed on a standard text file, so it can be easily transmitted from the computer, where the distribution package is installed to the CNC control using any standard methods of copying files and folders.

To generate the CNC code, press the  button. The NC-program generation panel will appear.

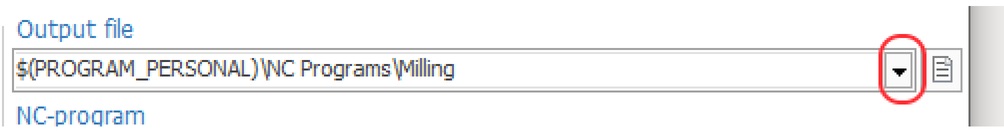


Here you can select the required postprocessor and run it to get an NC program. If the list doesn't contain the postprocessor you want, try to select another < Folder > with postprocessor files using the appropriate controls at the top of the panel. General information about the selected postprocessor 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 select file button to open dialog.

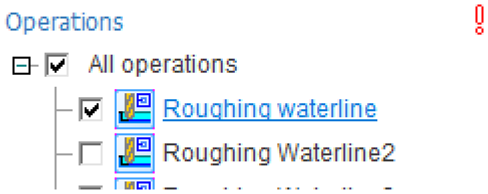


Use drop down button to see list of previous NC program files, or select output NC-program to the same folder as project or default folder. Default folder is set in System settings window.

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.

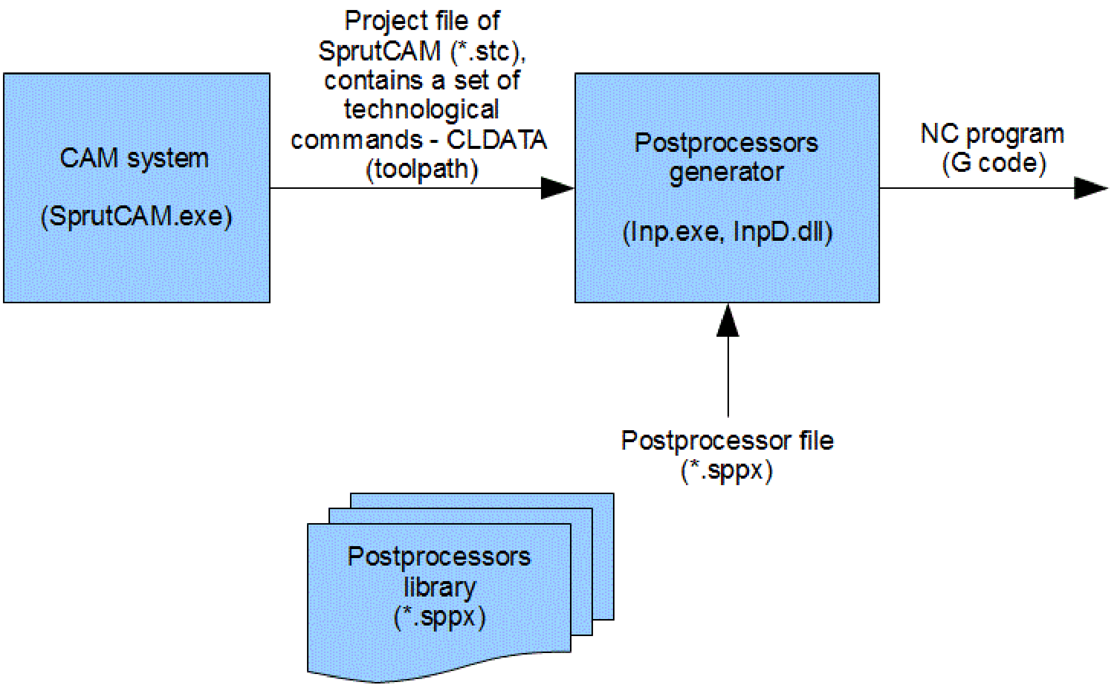
Please note, that the postprocessor only creates NC code for all calculated machining operations that are 'enabled' (, or status icon in the machining process list window) and also have **<Generate NC code>** property set at the time the postprocessor is run (the operations tree below postprocessor list).

> **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 SprutCAM X
. 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).

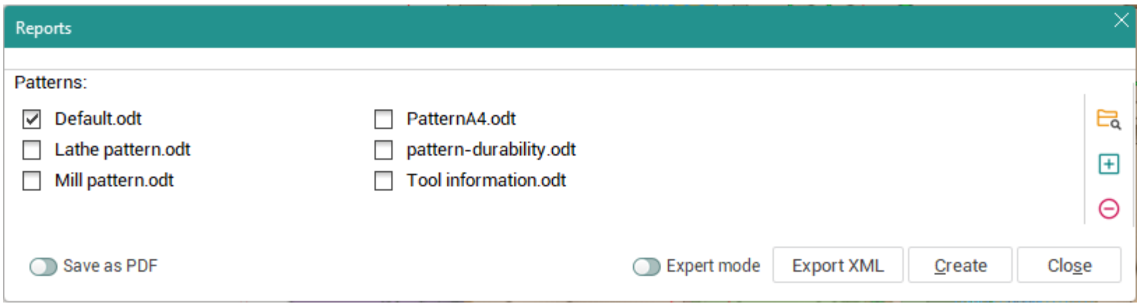


See also:

5.1.11 Machining report

SprutCAM X 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.

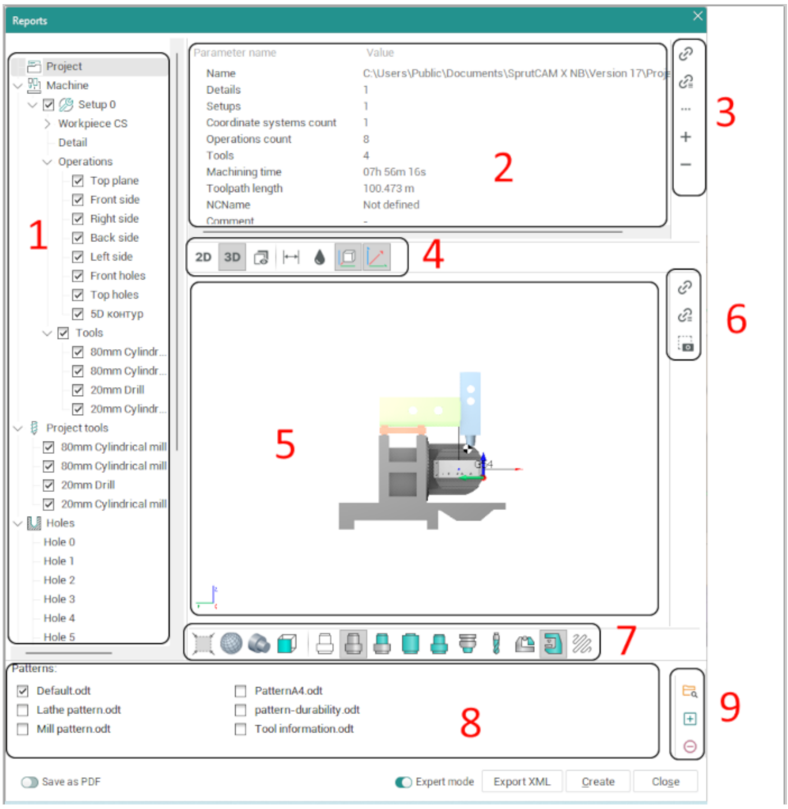


As default the biggest part or the machining report window is occupied by list of patterns from the default folder. It is possible to select several patterns and generate reports by the one click.

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.

SprutCAM X 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;

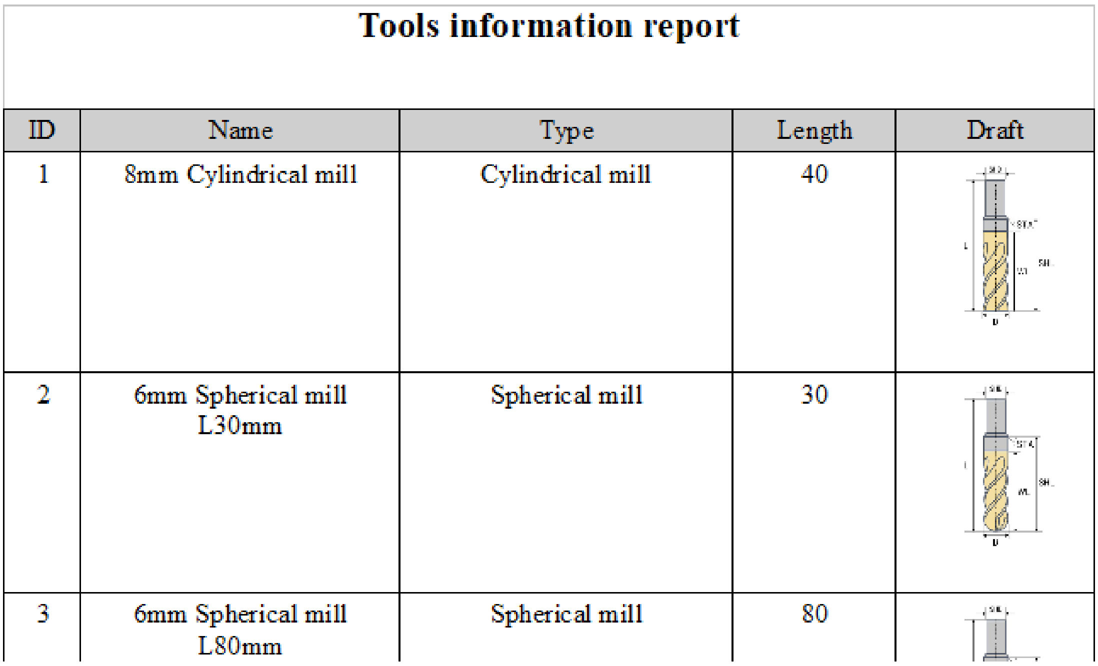
8 - Pattern list;

9 - The control buttons for add/remove reports and changing the default pattern folder.

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	

5.1.11.1 Machining report creation/modifing.

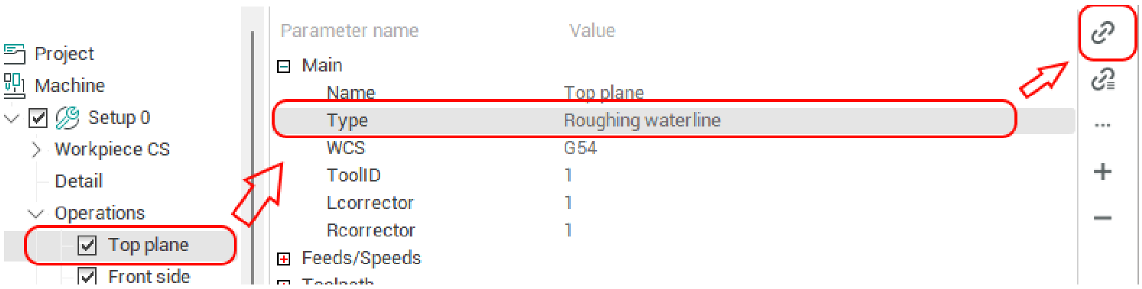
SprutCAM X report system generate reports based on templates in the Opendocument format (.odt).

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.

To add arbitrary variables to the report that you want to display, but which are absent in SprutCAM, you can use the following 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.

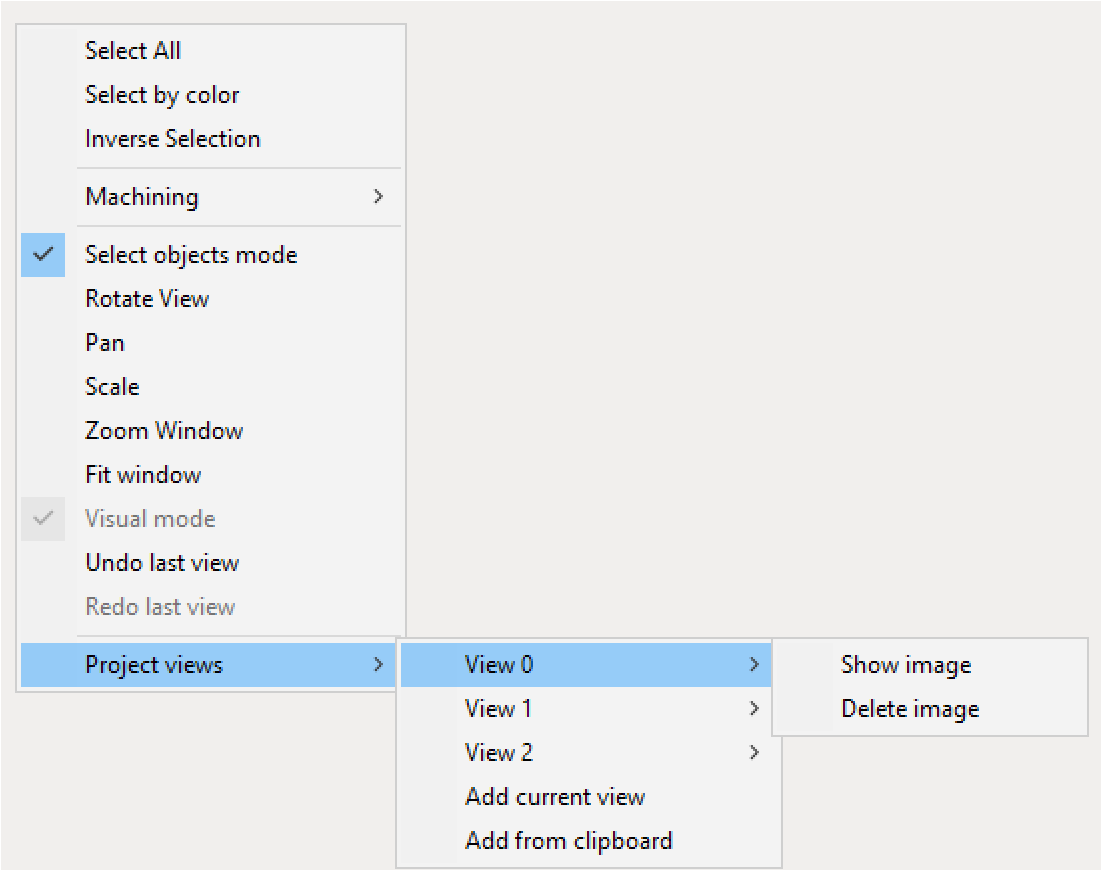
To view images added to the project, call the context menu in the SprutCAM X graphic window and select the **Project Views** option.

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 is recommended that you familiarize yourself with the supplied examples from the **SprutCAM X** distribution and exam lessions about Machining reports creation.

See also:

Common principles of technology creation

5.1.12 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 SprutCAM X 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.

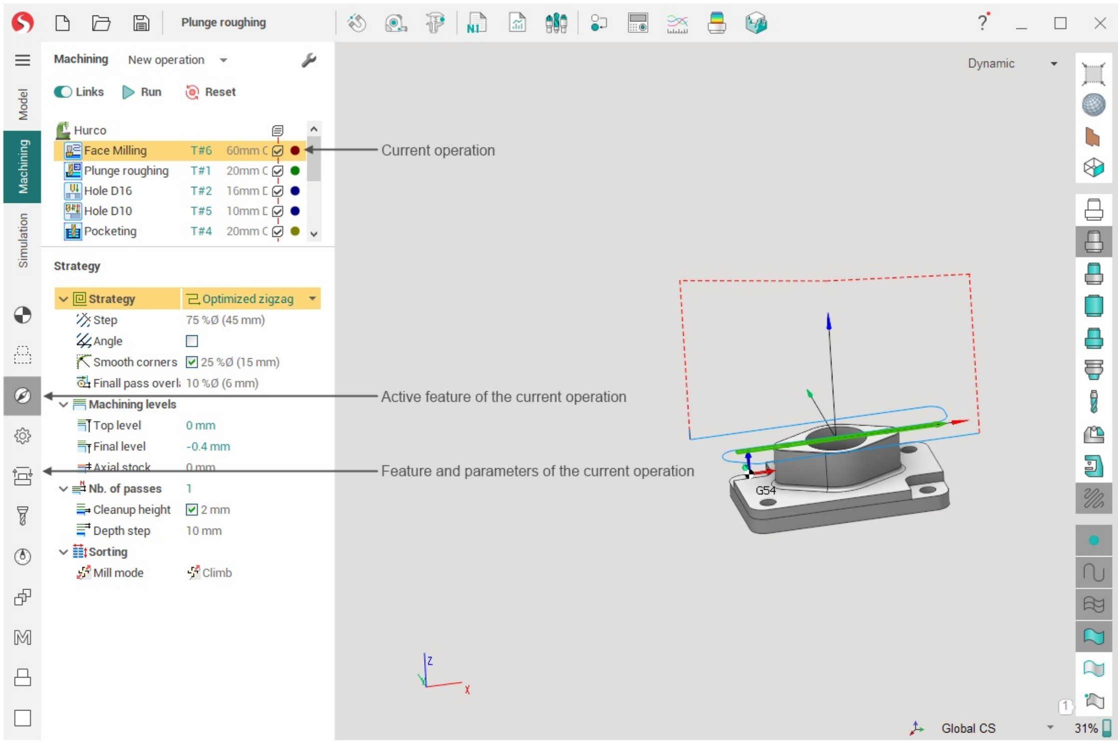
The <Import operation> command adds to the machining sequence one or more operations either from the SprutCAM operation file (*.sto) or from the any given SprutCAM X project file (*.stcp).

See also:

Common principles of technology creation

5.1.13 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.



See also:

Common principles of technology creation

Geometrical parameters of an operation

Defining part, workpiece and fixtures

Positioning of part at machine

Tool selection window

Tool change position

Operation local coordinate system

5 axes positioning

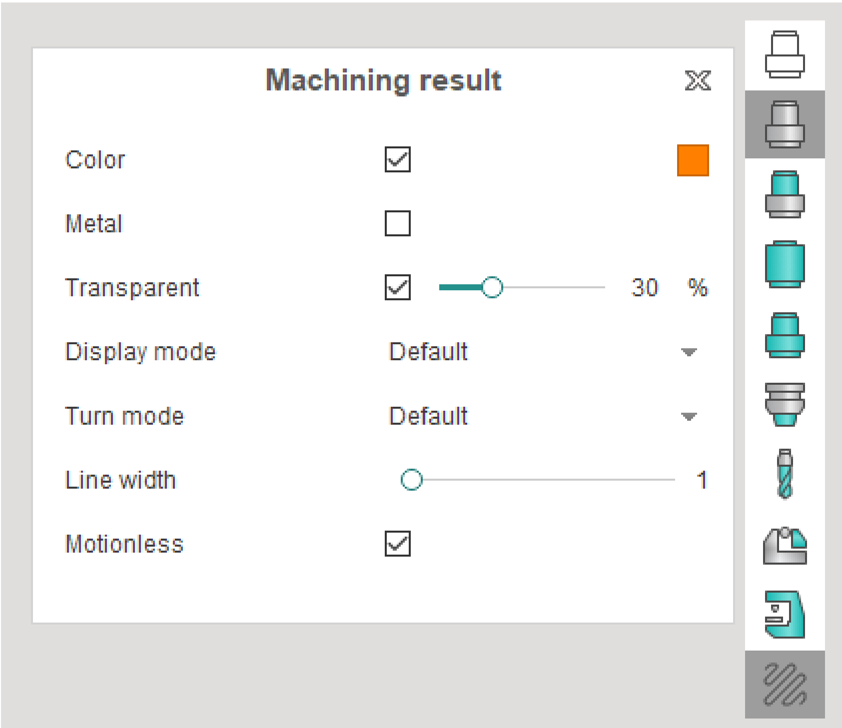
Approach and return rules

User operations

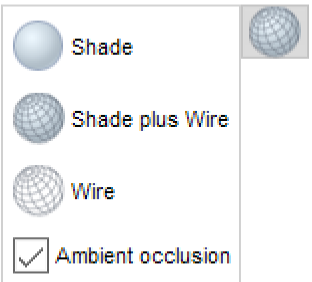
5.1.13.1 Geometrical parameters of an operation

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.



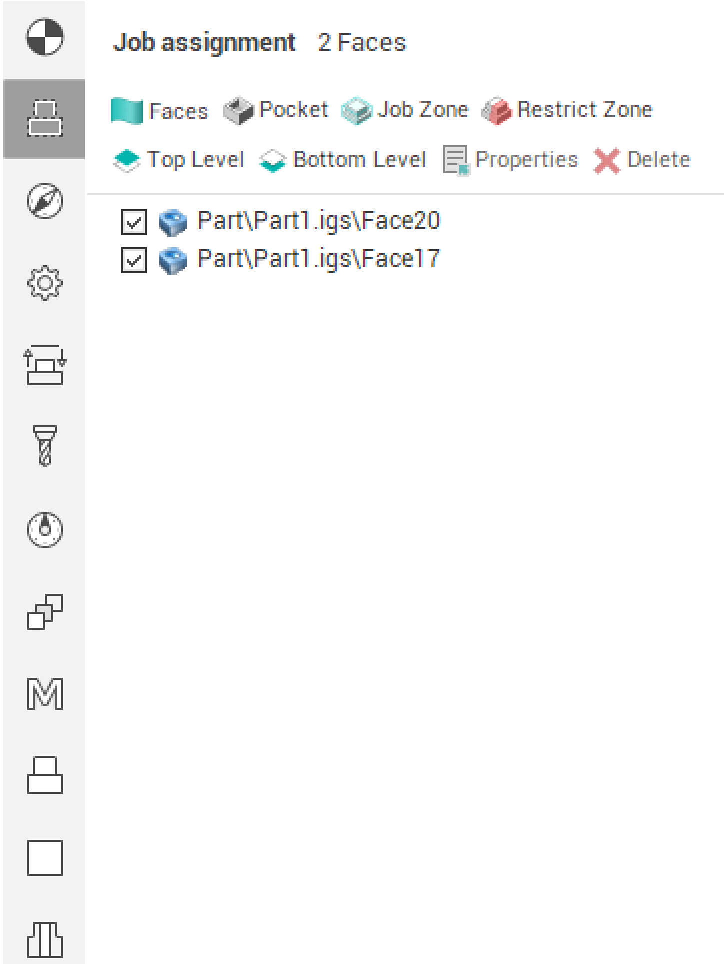
The rest of the modes force an object to be displayed irrespective to the global settings.

See also:

Operations setup

5.1.13.2 Defining part, workpiece and fixtures

Selecting a part, a workpiece or fixtures in the machining tree forces the bottom part of the window to take the following look.

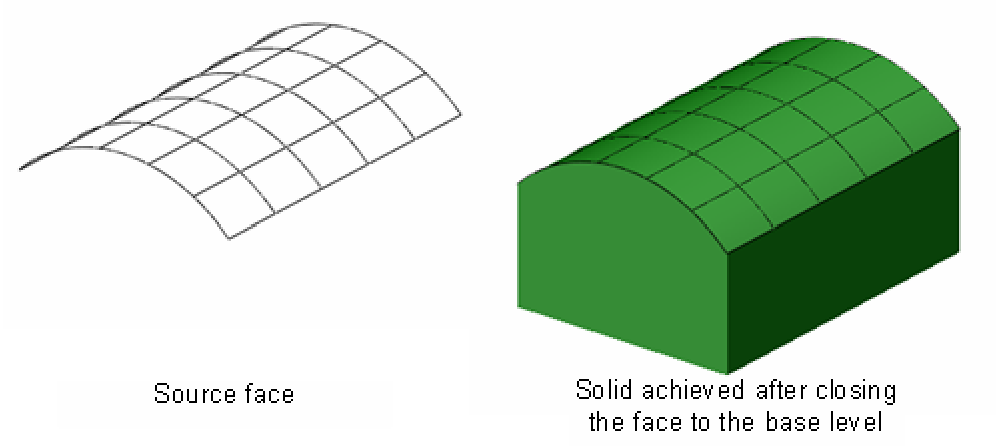


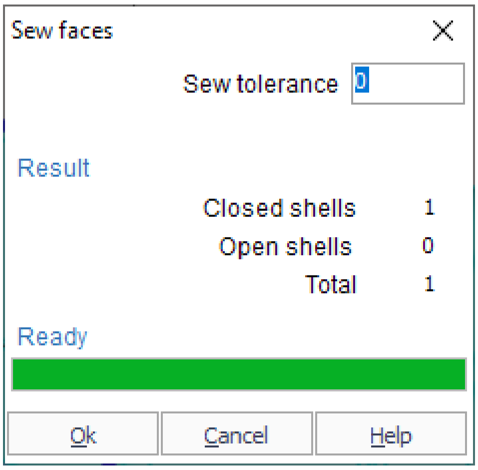
Each item of the list defines a surface, a solid etc to take into account while calculating toolpath. The item caption displays the full path to the item in the geometry model tree. If the source object of an item is removed or renamed the item is marked with the question sign. The icon left to the item 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.

The Faces 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.

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:

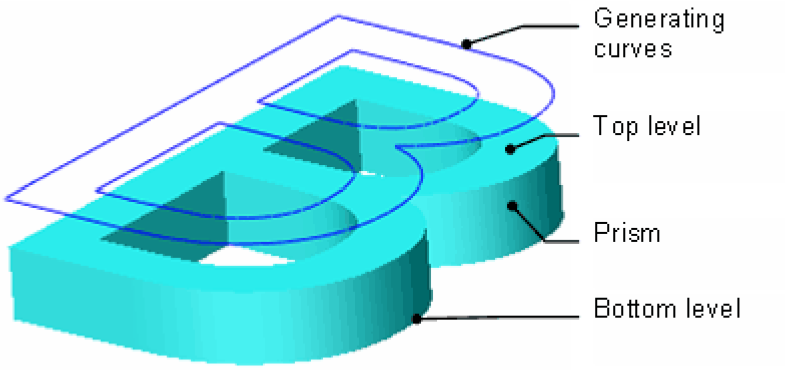
- By sewing faces. The added faces need to form one or more solids. The tolerance that the faces are connected with to each other is specified in the dialog that appears.
- By closing faces to the given Z level.

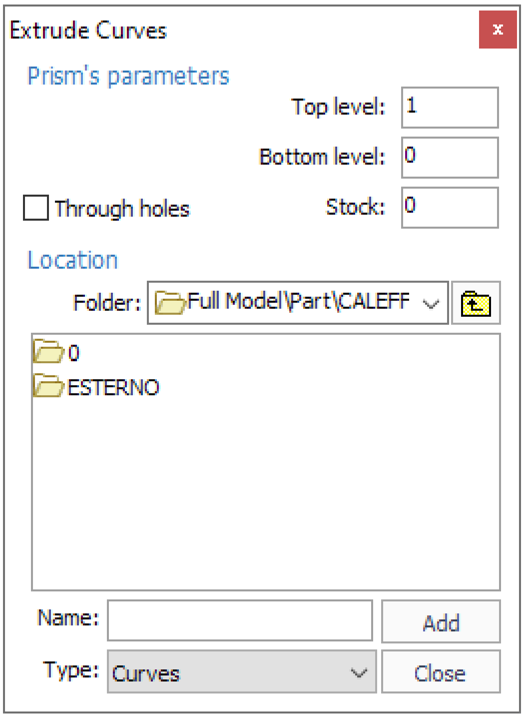




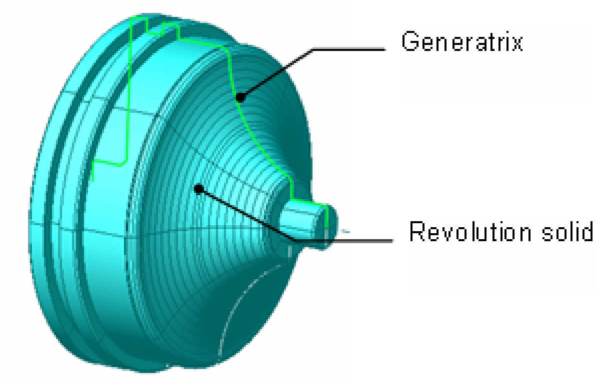
The Extrude button is used to make a prism. Closed curves as well as group of curves forming closed profiles can be used as the source objects. The specified curves are used as the base of the prism. In addition, the top and the bottom levels of the prism should be specified.

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.





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.



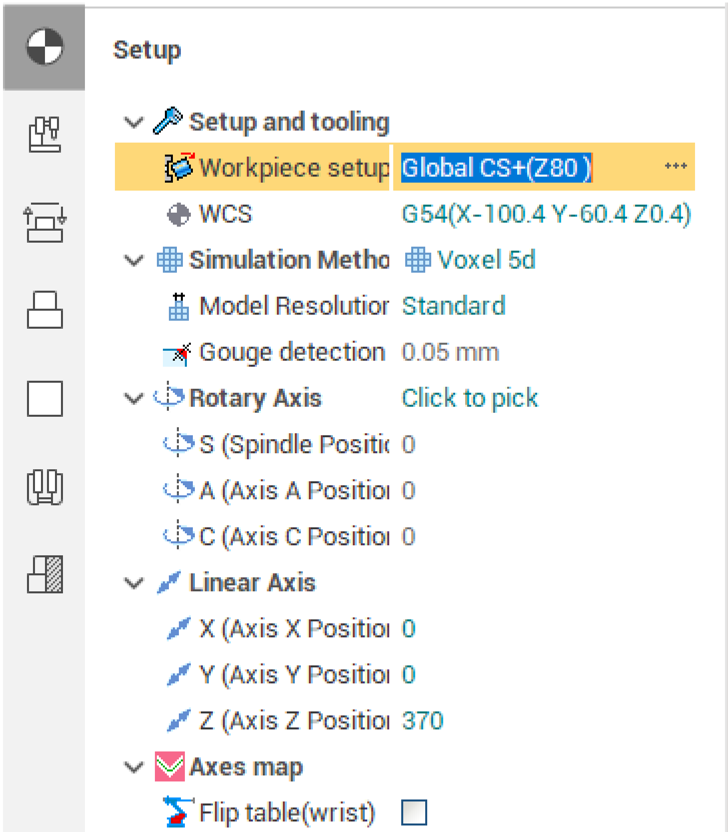
The special link items are introduced. Those are used to provide relations between operations as well as to define simple parametric features. At that, a link item is the reference to the source object, that in turn can be a reference to another object. Changing a source object forces all link items to be reset and/or recalculated. To add a link item to the list one should press the Reference button. The result depends on the parameter type being edited. If the part is defined then the reference to the part of the previous operation will be added. If the fixtures are edited then the reference to the fixtures of the previous operation will be added. If the workpiece is modified then the dialog will appear that offers the following items to be added.

-  – the workpiece of the previous operation.
-  – the machining result of the previous operation.
-  – the box around part.
-  – the cylinder around part.

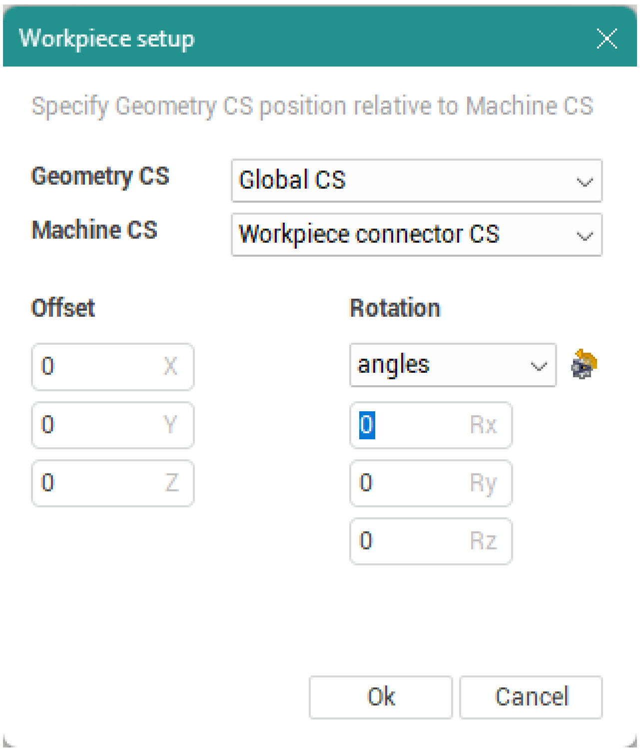
See also:

5.1.13.3 Positioning of part at machine

The position of the part at the machine is determined by the Workpiece Setup parameter in the Setup panel.



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).

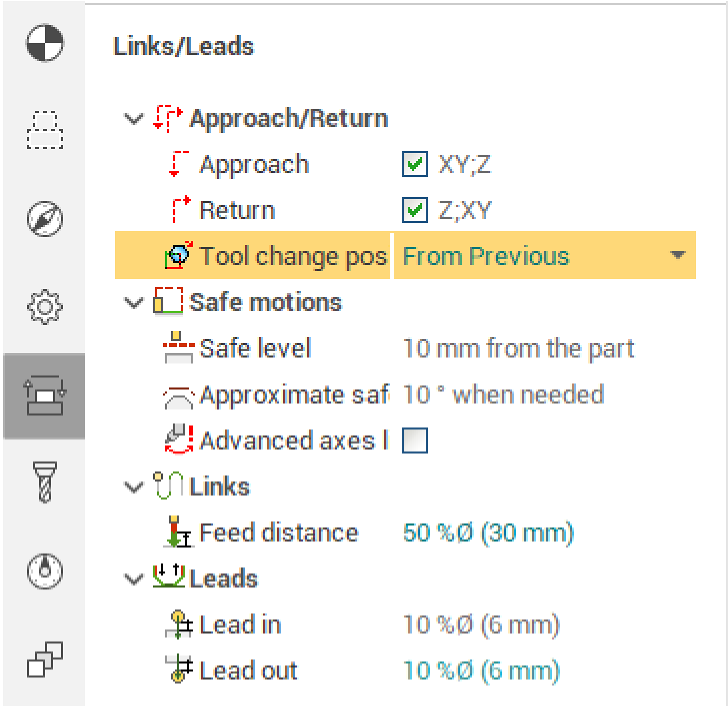
See also:

Operations setup
Positioning of part

5.1.13.4 Tool change position

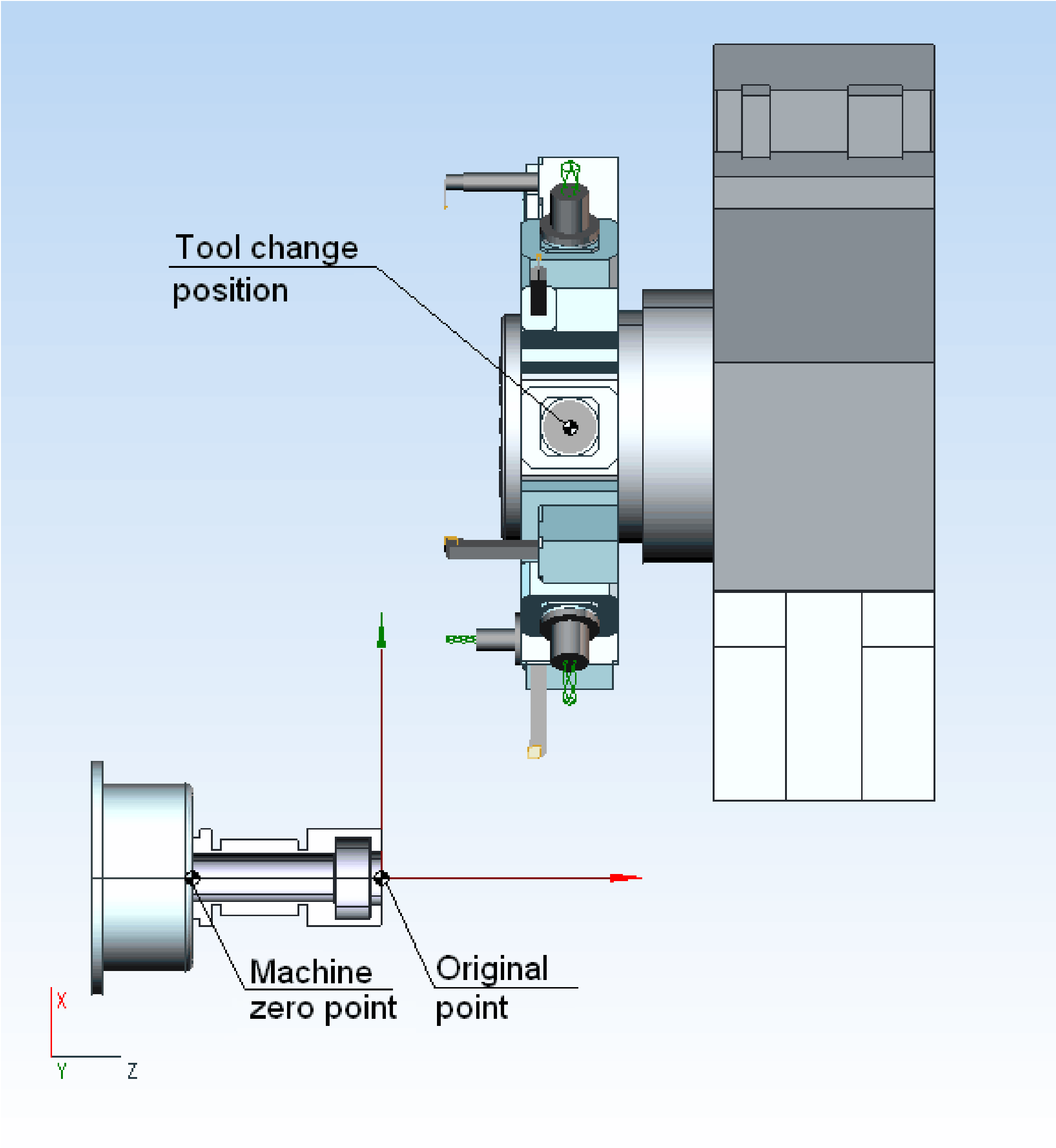
Tool change position is the machine position in which tool changing is performed. In SprutCAM X you can specify a default tool change position for a whole job list and override this setting in individual operations as you need.

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.

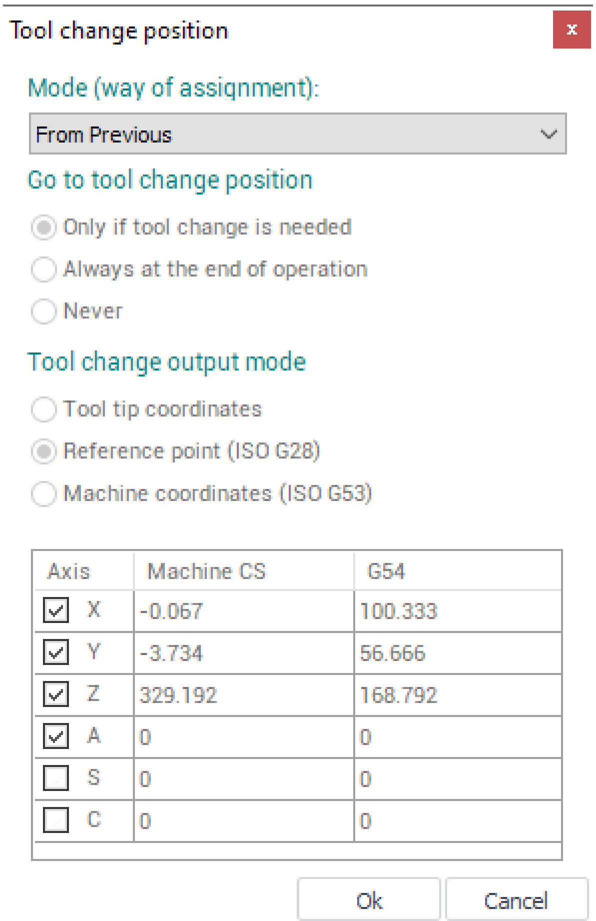


For an actual operation you can select a tool change position from the list of the following options

1. From the previous operation - the tool change position is inherited from the previous operation.

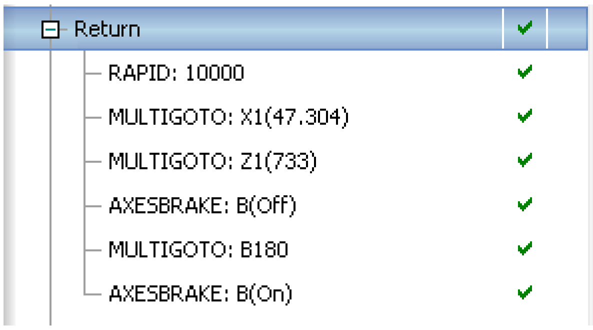
2. From machine - the tool change position is inherited from the position specified under the machine node.

3. Custom. If you select this item, a tool change position dialog appears.

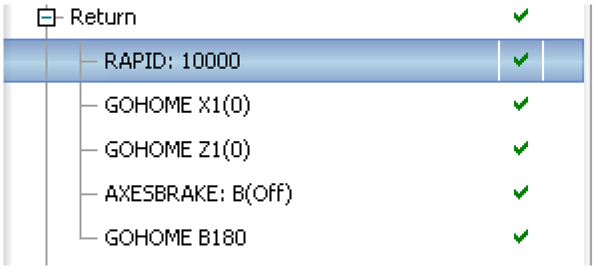


The tool change position dialog is used to define a tool change position as well as the tool change behavior for an actual operation in an interactive mode. Here you can set

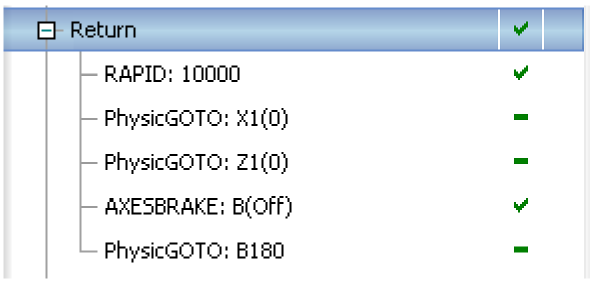
1. Mode (way of assignment) (From machine, From the previous operation and Custom).
2. Using of tool change position.
    - Auto (go to position for the tool change). If this option is selected the tool change position is used only when two neighboring operations use different tools with different tool 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. SprutCAM X 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 SprutCAM X 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).



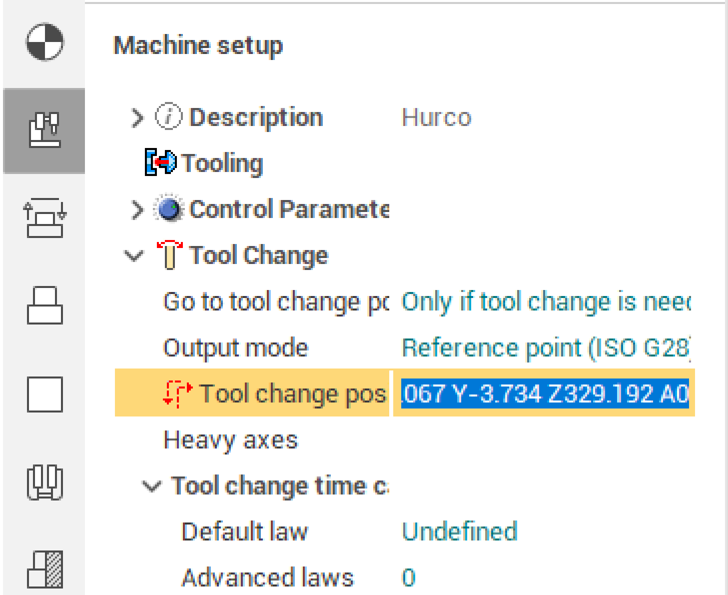
- Machine coordinates (ISO G53). When selected the option forces SprutCAM X to generate the PhysicGOTO commands for the tool change positioning. The PhysicGOTO commands represent coordinates in the machine coordinate system and can be used in the postprocessor to generate "Go to machine coordinate" commands (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>
```

See also:

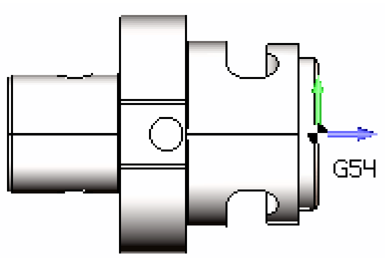
Operations setup

The point of tool interchange

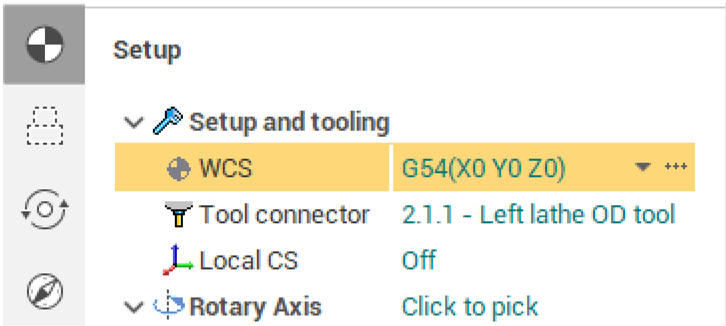
Mill-turn machining

5.1.13.5 Workpiece coordinate system (G54 - G59)

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. It looks like at the following picture:



The workpiece coordinate system is specified under the WCS parameter in the Setup panel.



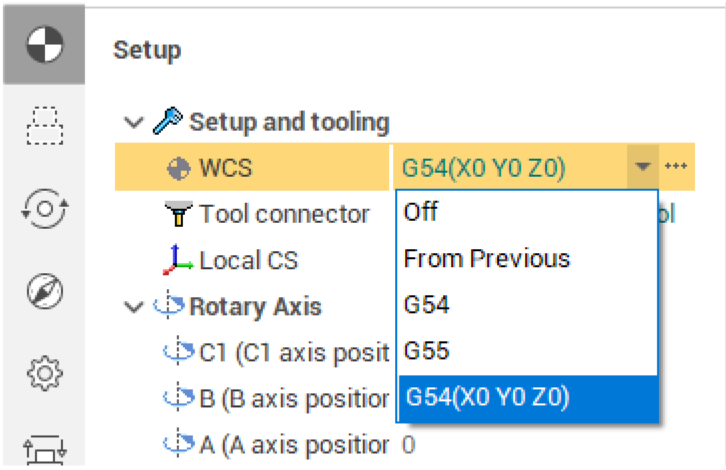
There are two ways you can define the Workpiece CS. You can select the Workpiece CS from the drop down list and you can set the Workpiece CS in the interactive mode by clicking on the ellipses button.

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.

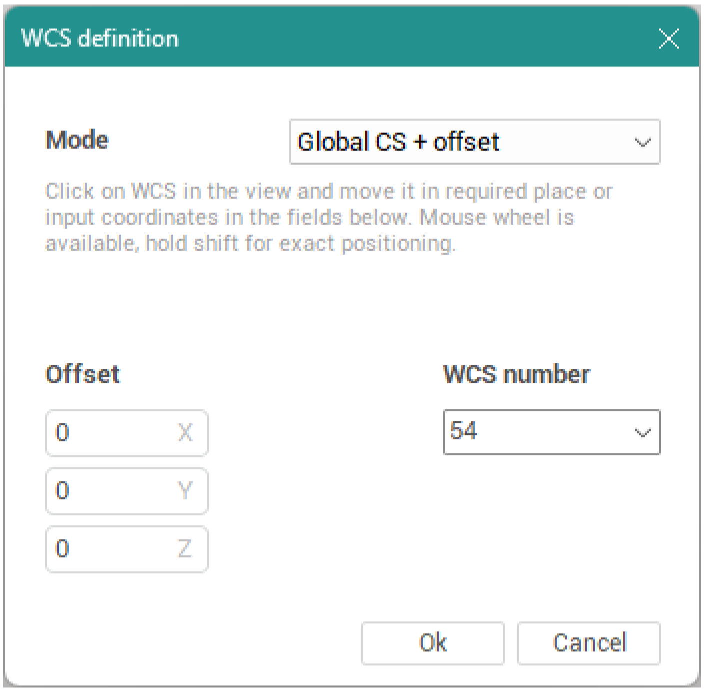
2. From previous - the workpiece coordinate system is taken from the previous operation.

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.



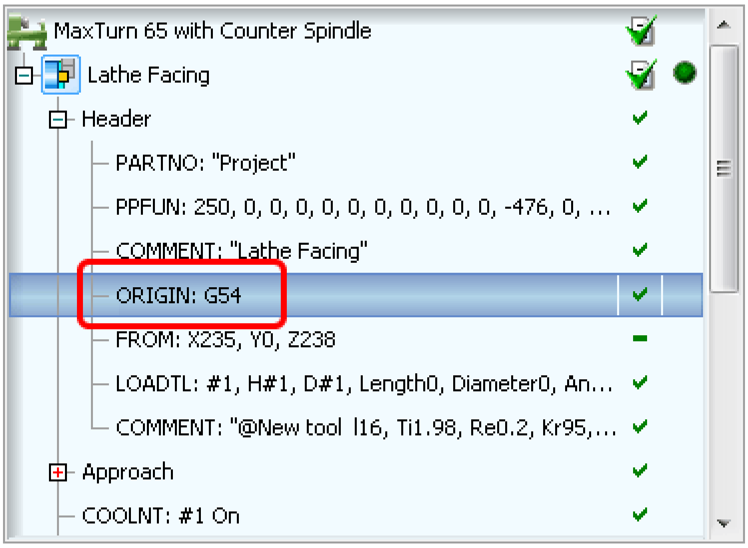
You can specify the WCS in the interactive mode by clicking on the ellipses button next to the WCS parameter.

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.).

Note that CAM modifies operation header CLData according to your changes. The <ORIGIN G54> command is added to the header in the example:



See also:

Operations setup

Geometrical coordinate systems

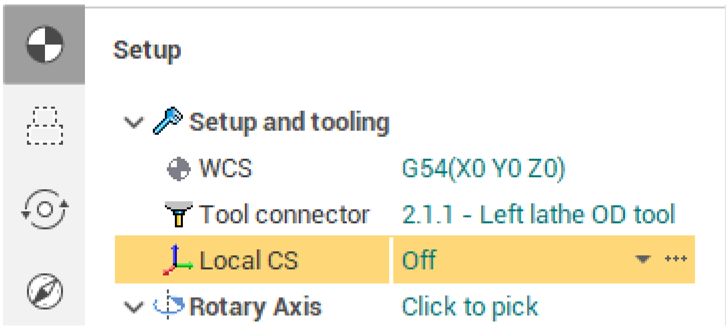
5 axis machining

Operation local coordinate system

5.1.13.6 Operation local coordinate system

An operation generates the toolpath in a local coordinate system. You can see the local coordinate system of an operation in the graphic view while working in the Technology and the Simulation modes. It looks like at the picture below.

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.

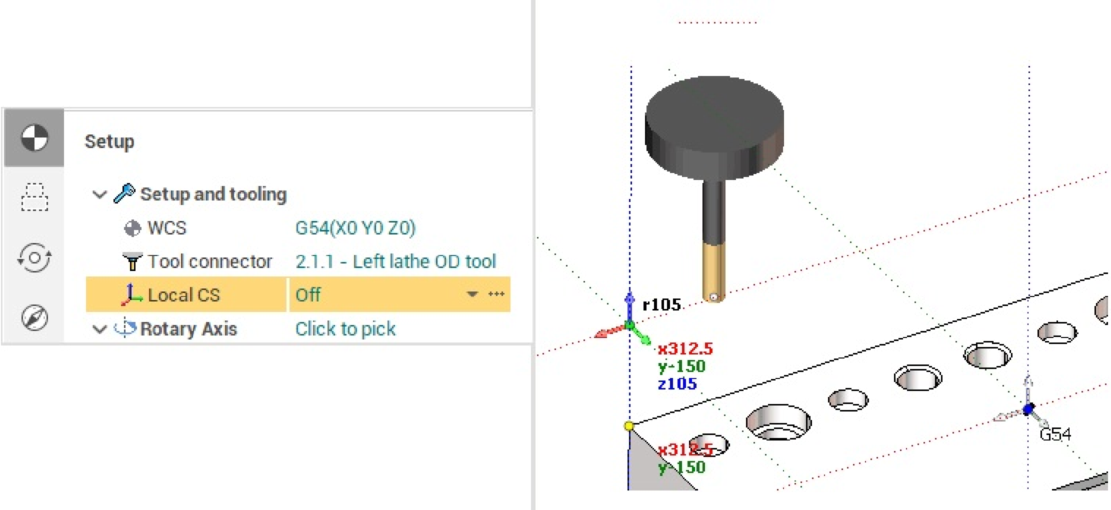
The drop down list contains the following options:

1. <Off> - the local coordinate system is disabled - the toolpath is generated in the workpiece coordinate system.

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).

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.

By clicking on the ellipses button next to the Local CS parameter you enter the interactive mode of Local CS definition. In this mode you can change the position of the Local CS origin in the graphic view using the standard drag&drop technique. Just hover the mouse pointer over the local CS 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.



After positioning the local coordinate system just click OK to apply the changes.

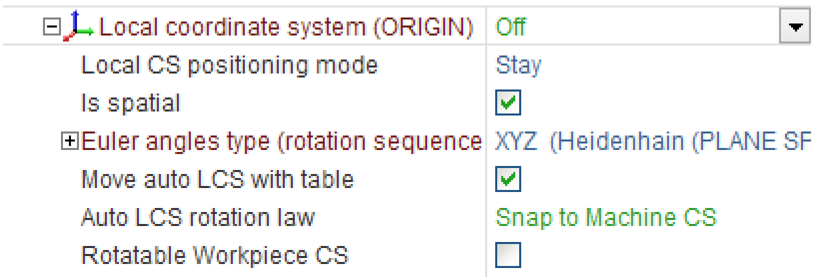
When you enable a local coordinate system SprutCAM X generates the Origin command in the header section of the CLData. The origin command contains all the data required for postprocessing toolpath. These are:

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.

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. SprutCAM X 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. SprutCAM X supports the following modes of the Local CS positioning.

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

See also:

Operations setup

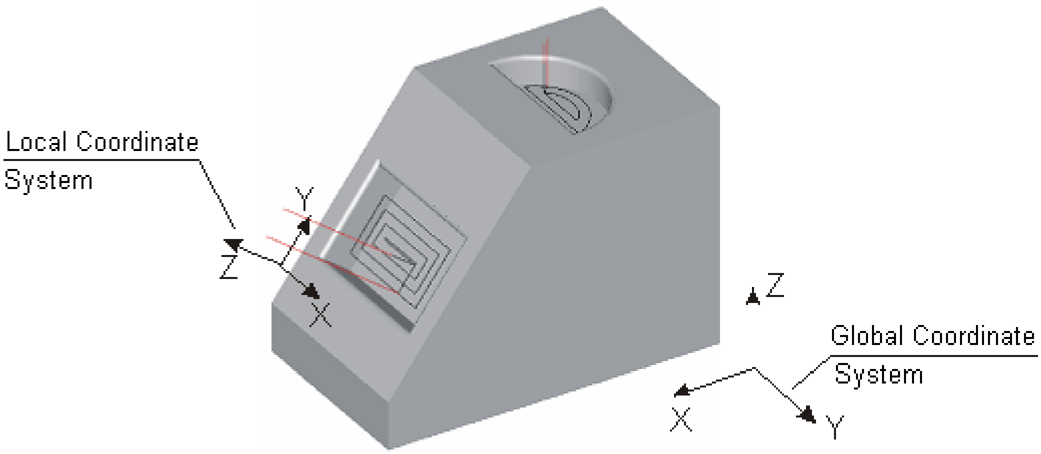
5 axis machining

Geometrical coordinate systems

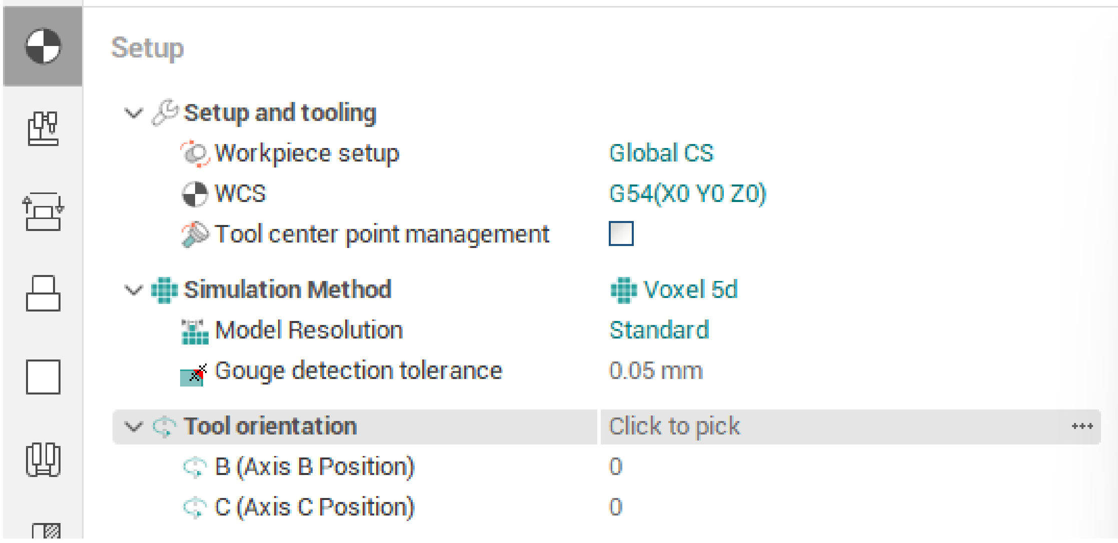
Machine coordinate system G54 - G59

5.1.13.7 5 axes positioning

5-axis positioning provides a convenient method of manufacturing parts that require milling on multiple faces by minimizing setups. The figure below shows an example of a part that requires milling from 2 different orientations. With 5-axis positioning, this entire part can be milled with a single program.



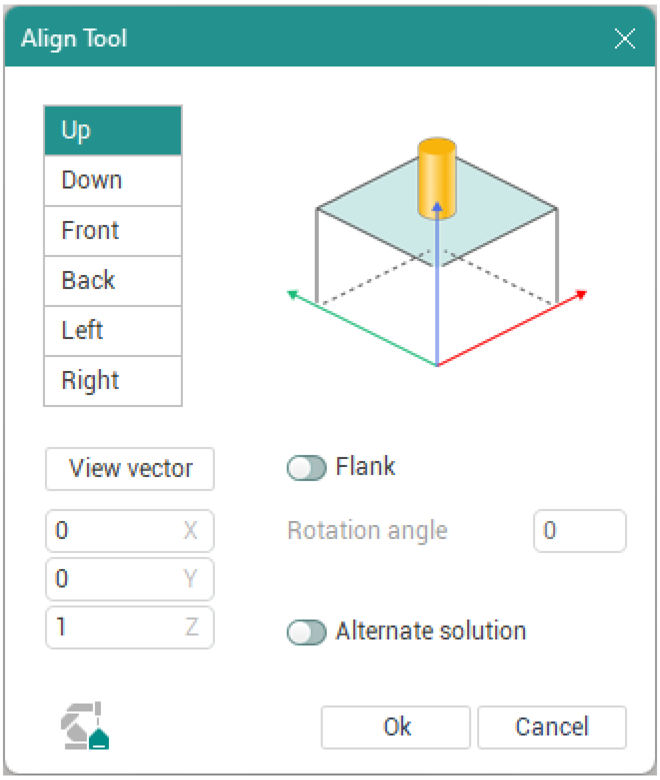
5-axis positioning is performed by setting the appropriate values to the **Tool orientation** parameter section in the operation Setup panel. All the milling operations may be used for part's processing from different sides. In the case the equipment doesn't allow to rotate the part or the axis of the tool, it is necessary to change the scheme of part's workholding.



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.

In SprutCAM X you can easily set the rotary axes position by clicking on the part face which the tool axis has to be aligned to. This method is performed with the following steps:

1. In the operation Setup panel select the Tool orientation parameter and press the ellipses button at the right of the "**Click to pick**" caption. The Align Tool dialog will appear:



In the dialog you can see and edit the orientation of the current tool axis in the Global coordinate system.

1. To align the tool with regard to a flat or a cylindrical surface just click on that surface in the graphic view. The 3d Model visibility button should be turned on.
2. Press Ok to apply the changes or press Cancel to discard the changes and close the window.

If you want to specify one of the standard orthogonal orientations, you can use the buttons with names corresponding to the direction you need: **Up**, **Down**, **Front**, **Back**, **Left**, **Right**.

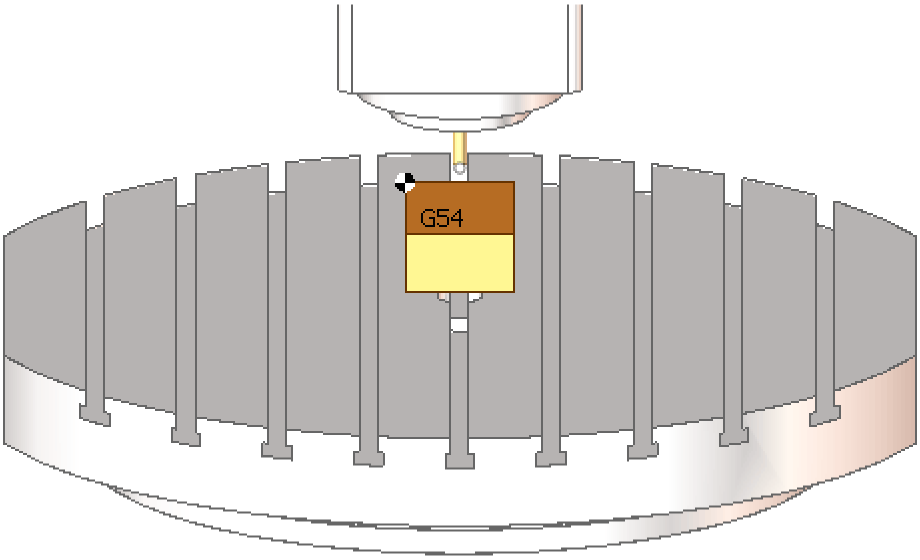
You can also orient the machine axis along the current view vector used in the graphics window by the **View vector** button.

The **Flank** option will help you orient the tool sideways in relation to the selected direction instead of along it.

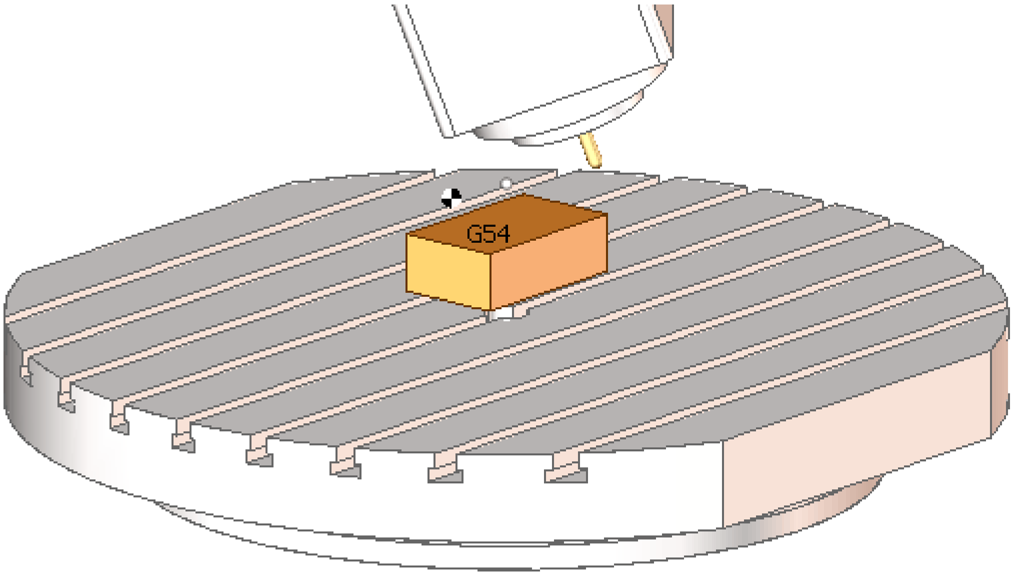
Use the **Alternate solution** option if the machine allows you to provide the same relative position of the tool relative to the workpiece in several ways.

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.

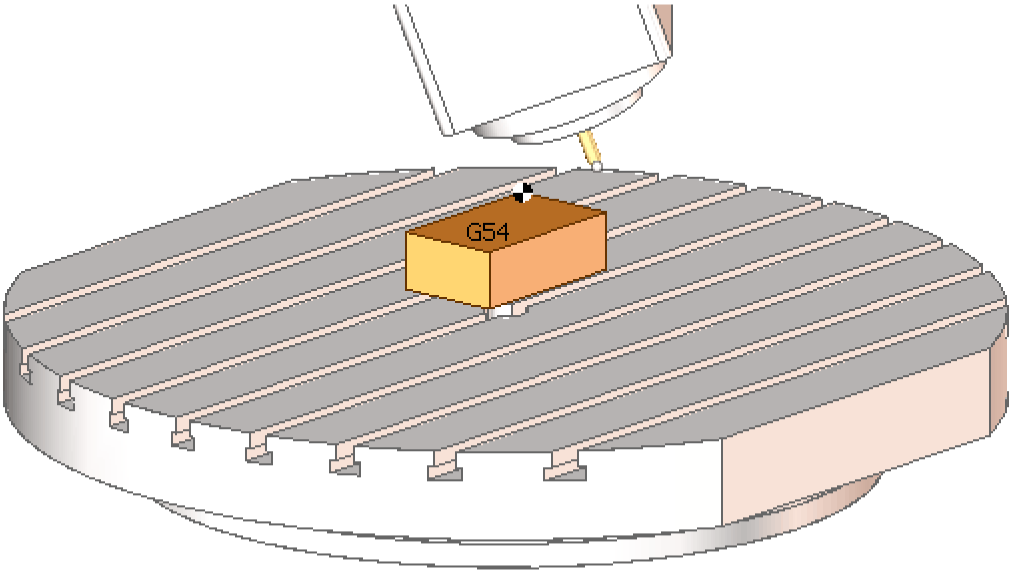
Many today controls require definition of the local coordinate system for 5 axis positioning. Without this definition the position of the workpiece zero point (G54) and the position of the tooling point is not updated after changing the position of rotary axes - the control behaves as it knows nothing 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.



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



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.

See also:

Operations setup

5 axis machining

Workpiece coordinate system (G54 - G59)

Operation local coordinate system Mill-turn Machining

5.1.13.8 User operations

Every technological operation of SprutCAM X 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:

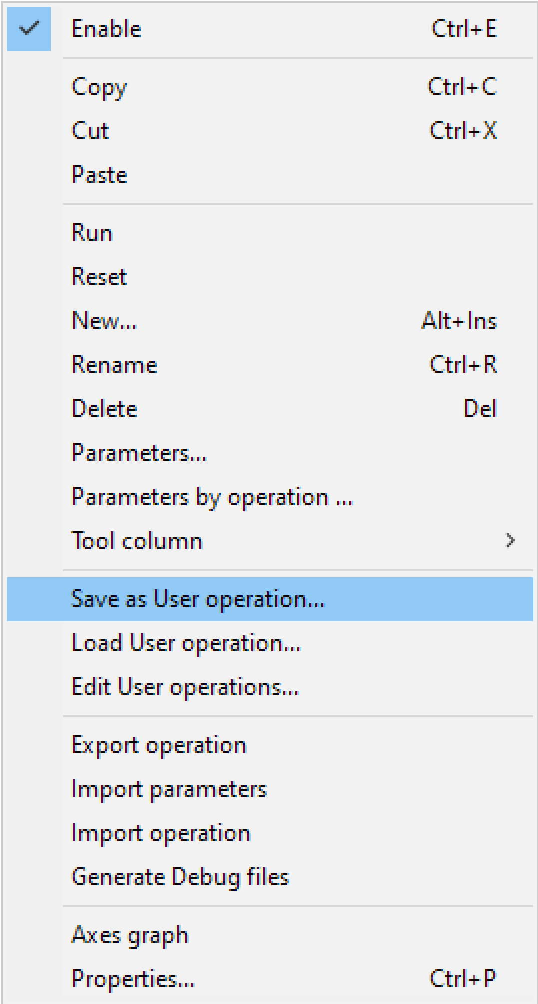
- Determining machining strategy for a specific element of a part.
- Determining operation which is able to implement the proper strategy.
- Creating operation and specifying the machined part's element as it's job assignment.
- Calculating toolpath.
- Analyzing toolpath.
- Modifying technological parameters 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 SprutCAM X operations have a lot of parameters. Therefore the process of the operation customizing may take a long time and significant effort. SprutCAM X provide the <User operations> feature to reduce the required time to design operations. It allows to specialize the universal SprutCAM X's operations

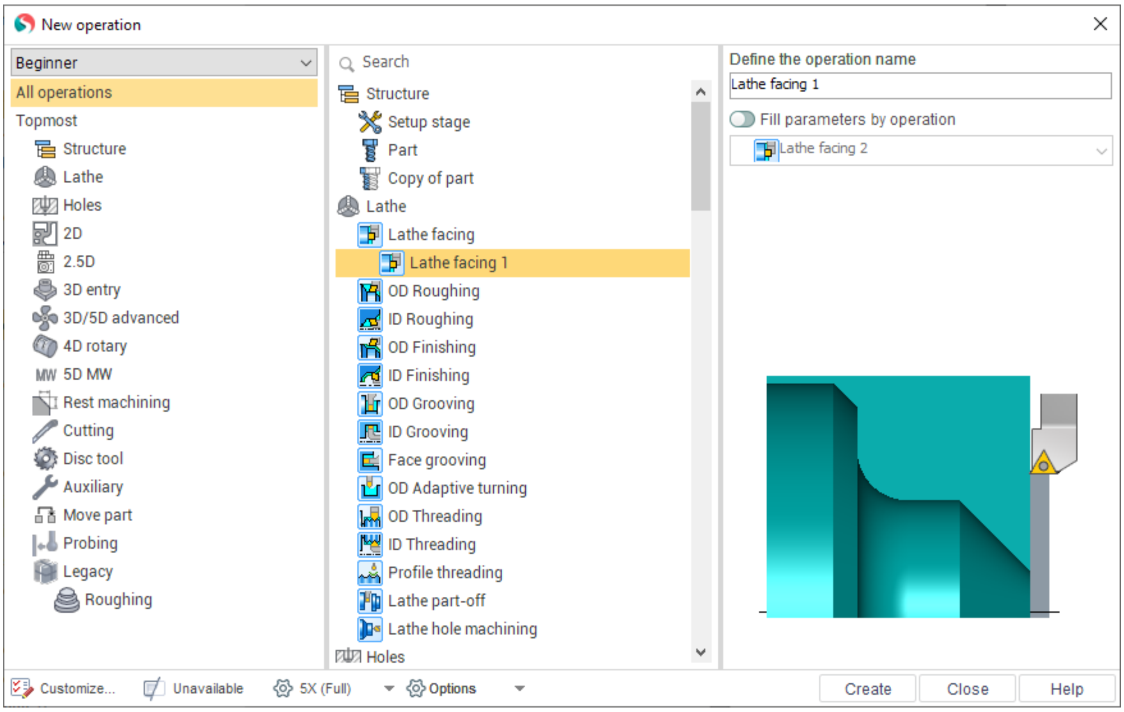
After setting up the operation for specific purposes it is possible to save it's parameters set under the certain name to the external *.stox file. When processing the next similar element possible to download this set of parameters and, thereby, greatly reduce the time for setting up the operation. 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.

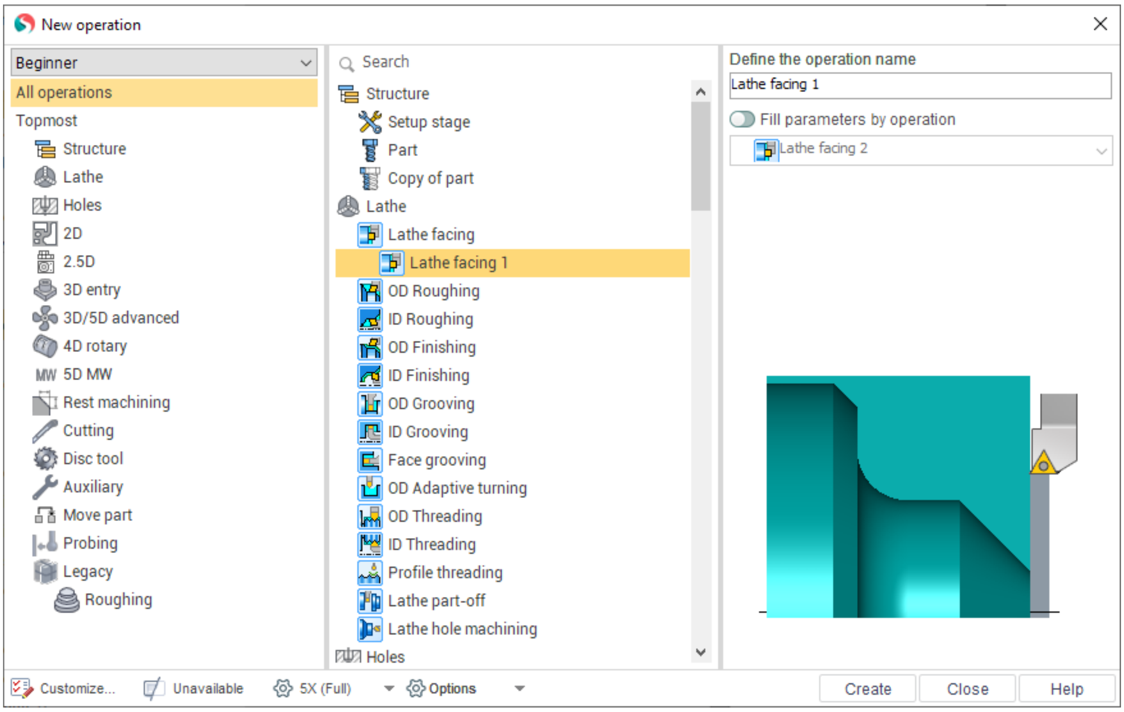
The parameters are located on the other tabs may vary and depend on the particular type of operation. All of them are also available to allow you to easily edit values without changing the parameters of the source SprutCAM X's operation from which this user operation was created.

Above the list of parameters are located a few buttons to control the display of these parameters.

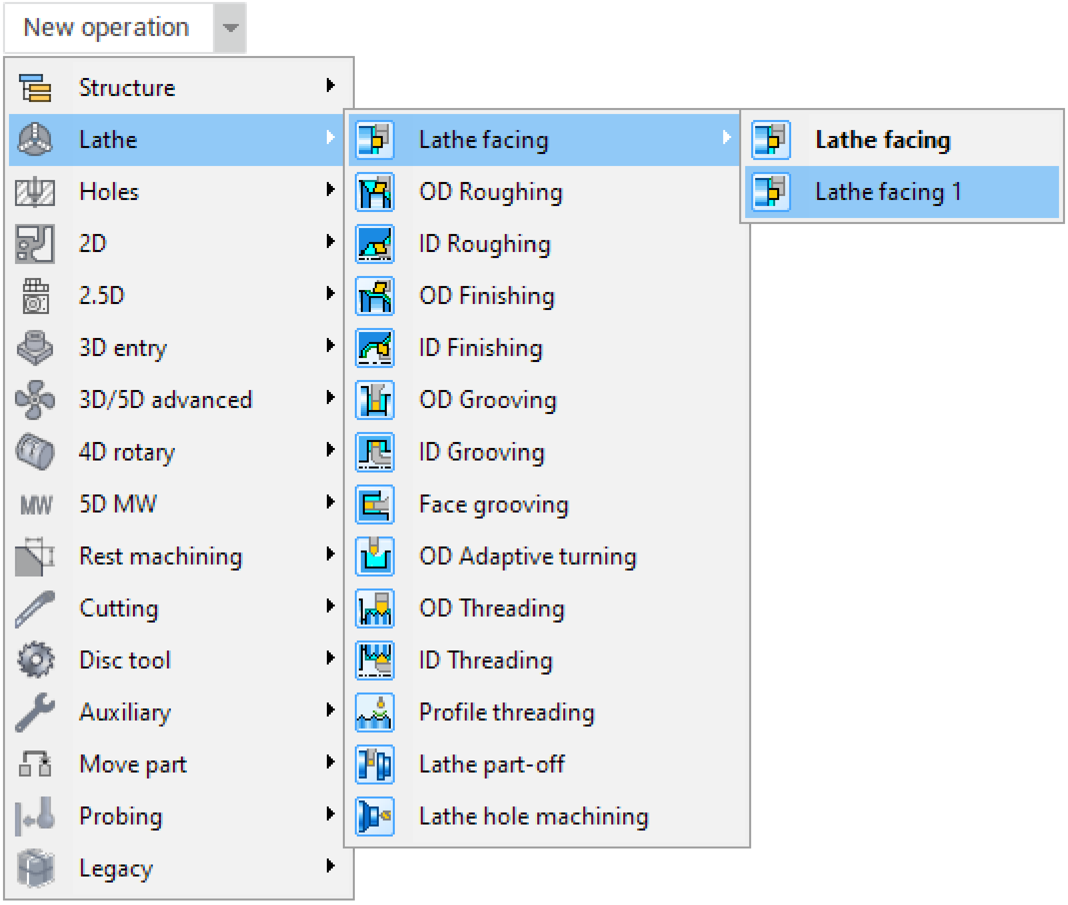
- allows you to hide / display the properties whose values coincide with the system default values for this type of operation. This makes it easier to distinguish between the important parameters that have been changed with respect to the source. In inspector values coinciding with the default values 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.
- f(x) 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.

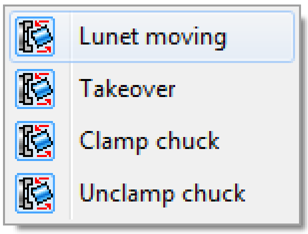
In the New operation window, select a previously created user operation.



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 SprutCAM X.

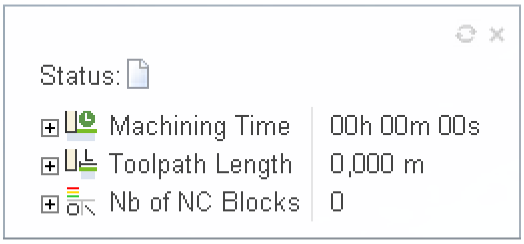


See also:

Operations setup

5.1.13.9 Operation status panel

When clicking on the operation status icon inside technology tree next to the operation name then the status panel appears.



Panel operation state contains three tabs and the status bar.

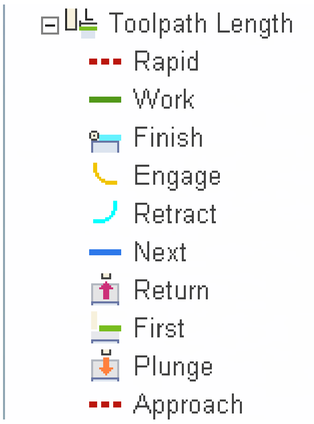
1. Status bar.

Status bar displays operation information and simulation errors in the icons.

1. Tabs contain information about toolpath processing time, the path type and it's length.

   - The toolpath processing time for all simulation steps.

   - The length of each path type.



   - Number of NC blocks for each node type.

5.1.13.10 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

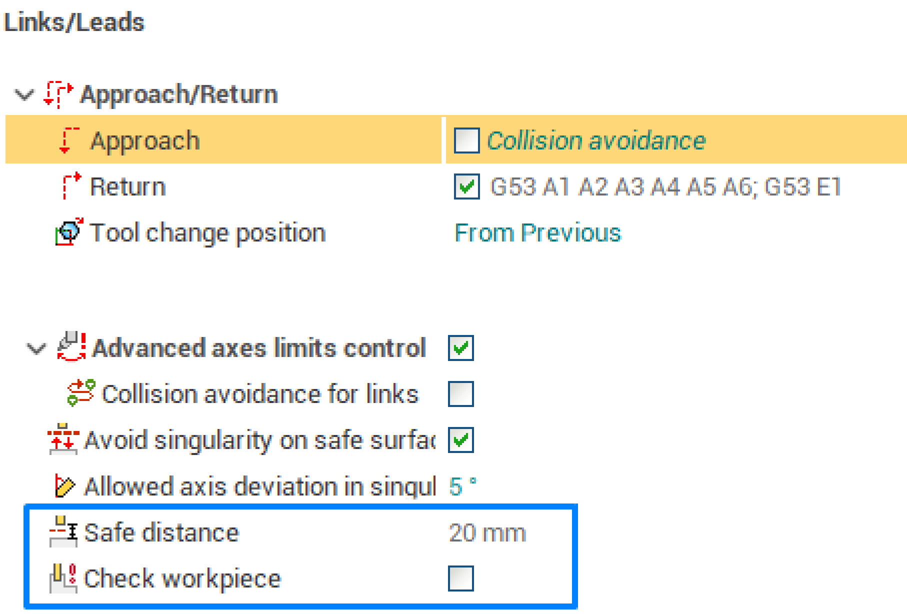
The approach/return rule is specified under the Approach/Return section in the operation **Links/ Leads** inspector panel. The check box indicates whether or not the approach/return is automatically generated to **avoid collisions**.

- ***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.



1. ***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 SprutCAM).

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.

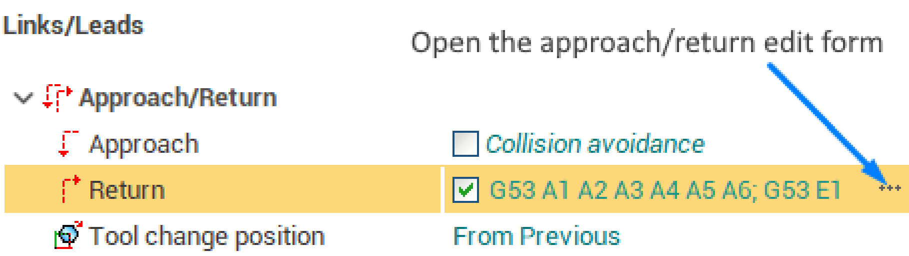
The LCS used by this command is not defined by the operations' parameters, instead, it corresponds to the **first point** of the toolpath, if the <**SLCS**> command is inside the **approach section** and to the **last point** of the toolpath, if the command is inside the **return section**. The **rotary 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.



See also:

Machine's approach/return list

Approach/return rules edit window

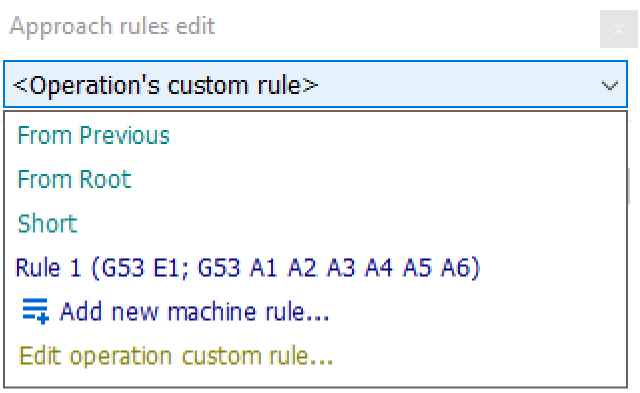
Submachine definition in the machine schemas

The list of the basic CL-data commands

Operations setup

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.

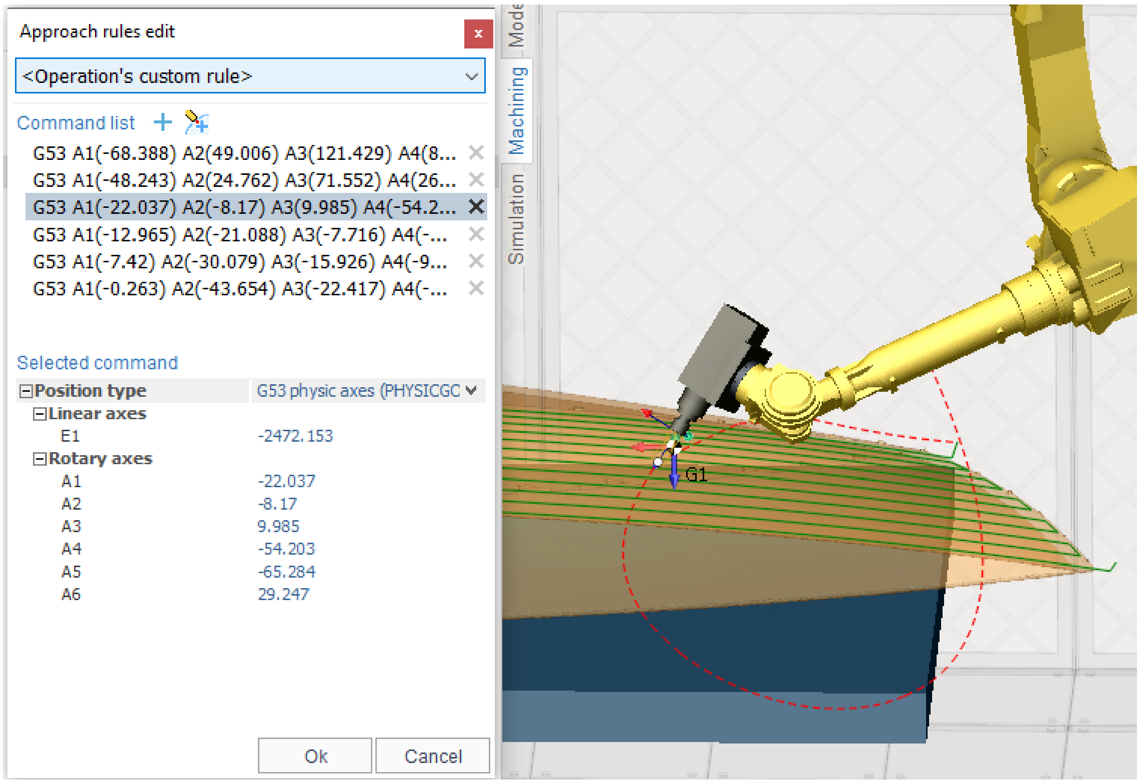
1. In the top most combobox you can select to edit operation's own approach/return or one of the items in the rule list.

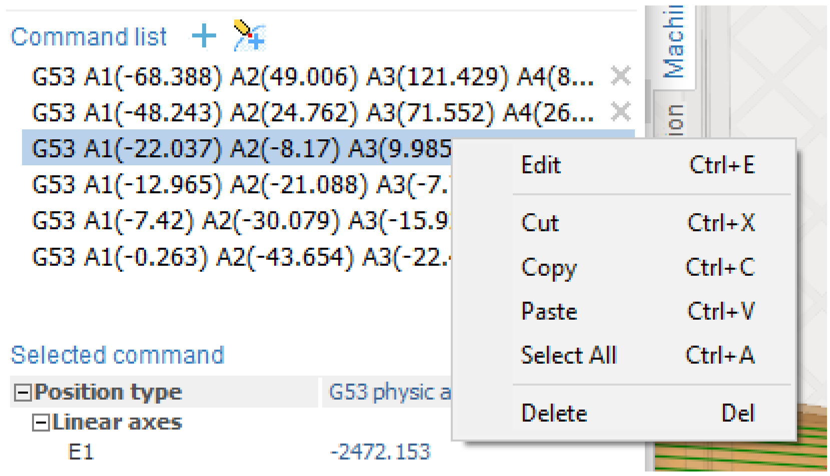
2. Below, in the center of the form, there is the list commands which define current selected rule. Each line (command) in that list represents a distinct point of the lead path. A point has the following format.

- 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 SprutCAM X 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).





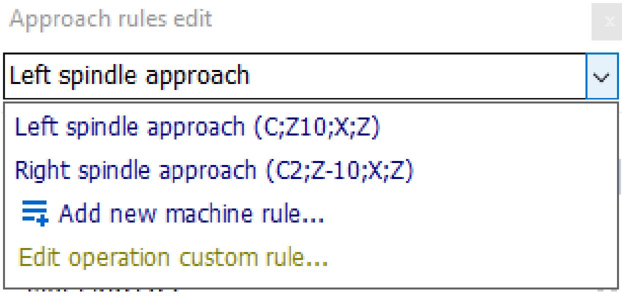
3. In the bottom area of the form there is a list of machine axes. To add a new reference point to the path you should either fill the coordinates of the appropriate axes with desired values, and/or set some of them to AUTO by clicking on the ellipses button next to an axis. To edit a reference point 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..**'



See also:

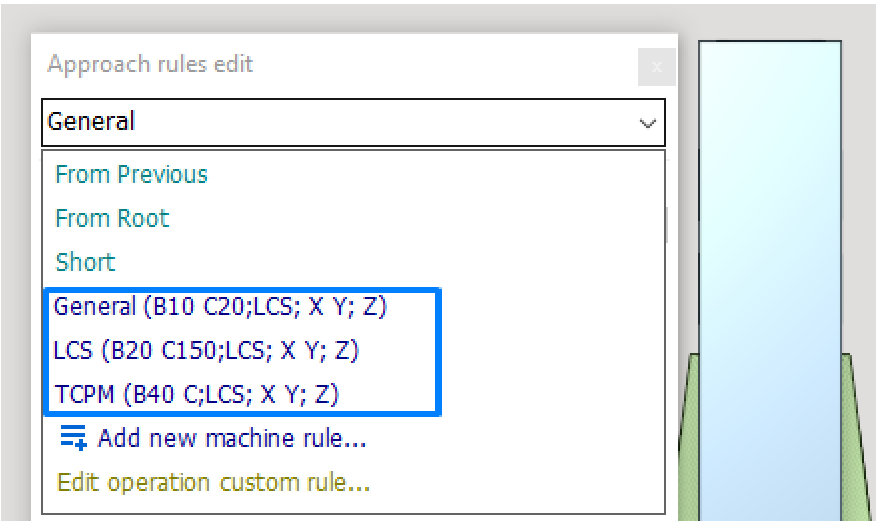
Machine's approach/return list

Approach and return rules overview

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 SprutCAM X.

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.

See also:

Approach and return rules overview

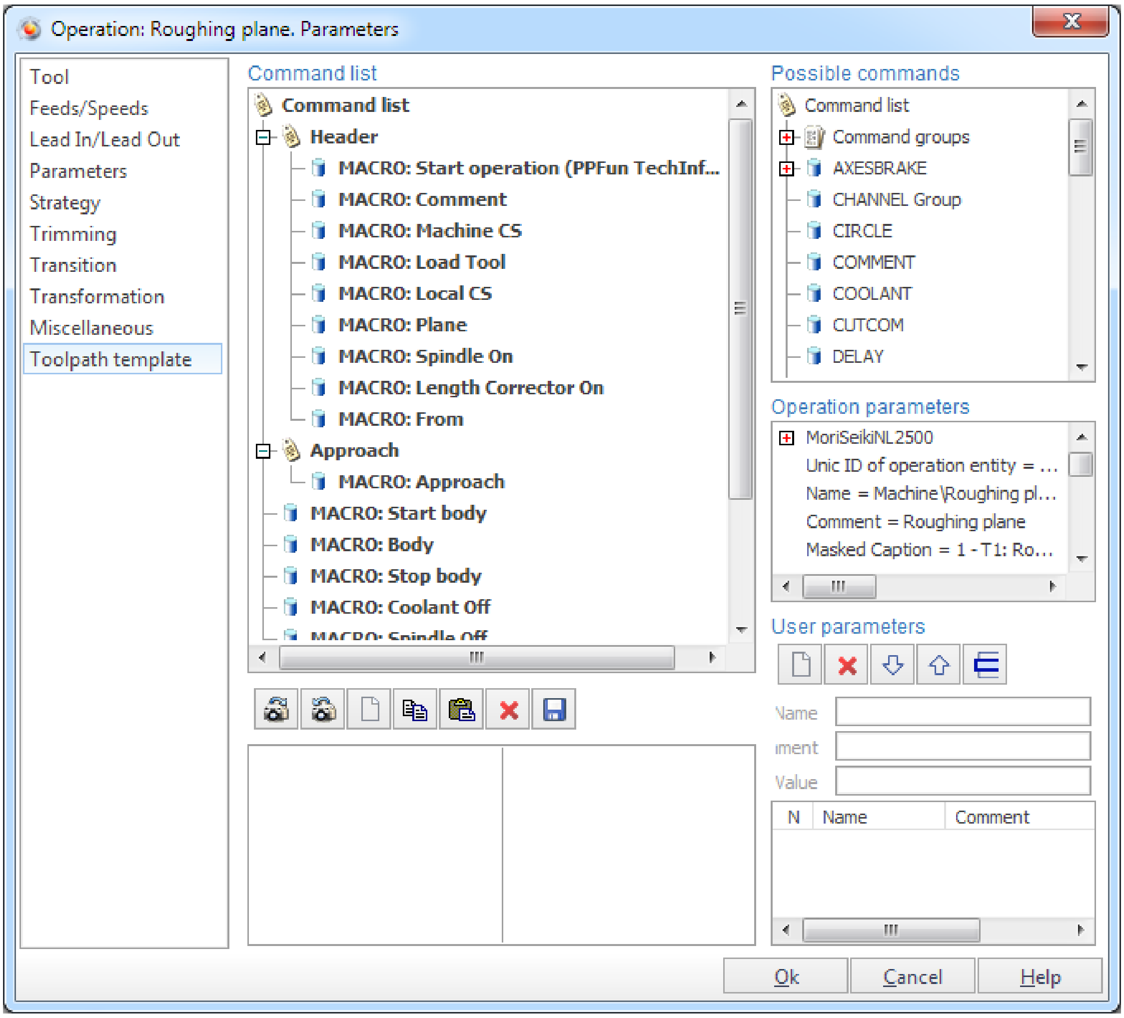
Approach/return rules editing

Submachine definition in the machine schemas

5.1.14 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 SprutCAM X 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.

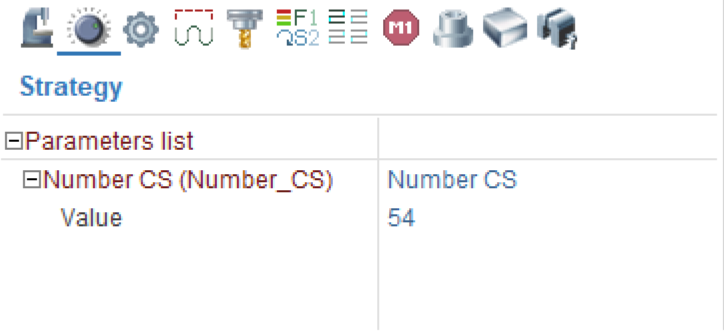
For adding one of commands needs to select a row with a corresponding name and drag it into wishful place in command list. The root element of the list of available commands MCDTree also may be inserted into command list, and inside of MCDTree unit other CLData command may be placed. Thus, commands in the list may be grouped by any other concept. For editing of the list the tools panel may be used.

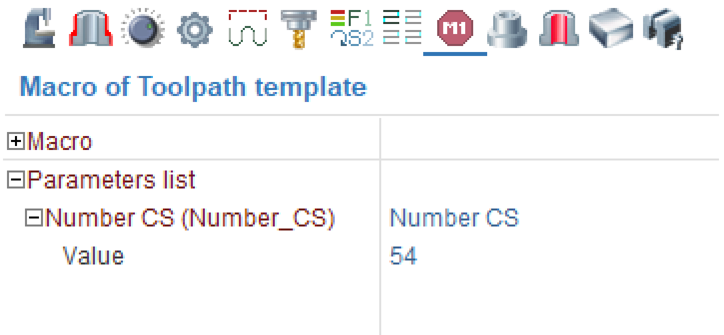
Any CLData command has special set of parameters (feed magnitude, coordinate position, comment string and so on). Destination of property inspector is on representation and editing pointed parameter, it display choosing command in list. To properties of technological operation may be assigned either 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:

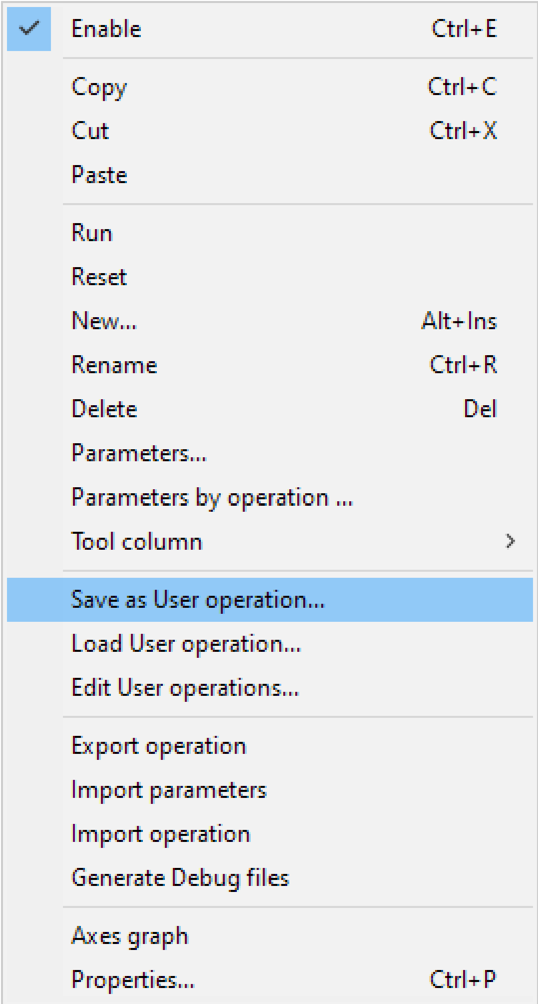
-  <Expand all>. It expands all the blocks in <Command list>.
-  <Collapse all>. It collapses all the blocks in <Command's list>.
-  <Add new command>. It adds the command, indicated from the list of <Possible commands> to the place which is indicated in the <Command list>.
-  <Copy command to clipboard>. 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.

The button of the right board:

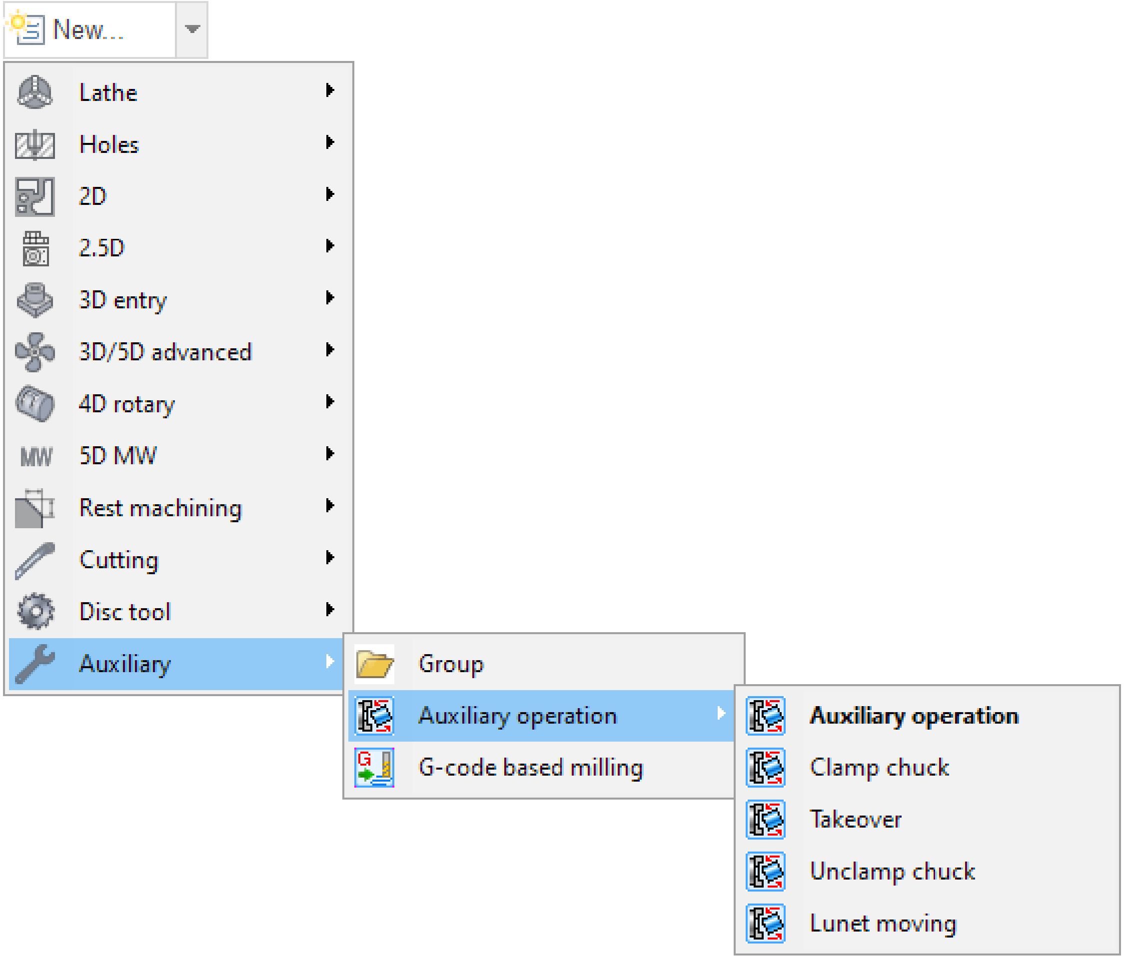
-  <Add new parameter>. It adds new parameter to the detailed place in the <Parameter list>.
-  <Delete parameter>. It delete parameter from the <Parameter list>.
-  <Move parameter down>. It lows down the parameter in the <Parameter list>.
-  <Move parameter up>. It raises up the parameter in <Parameter 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 SprutCAM X 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>.

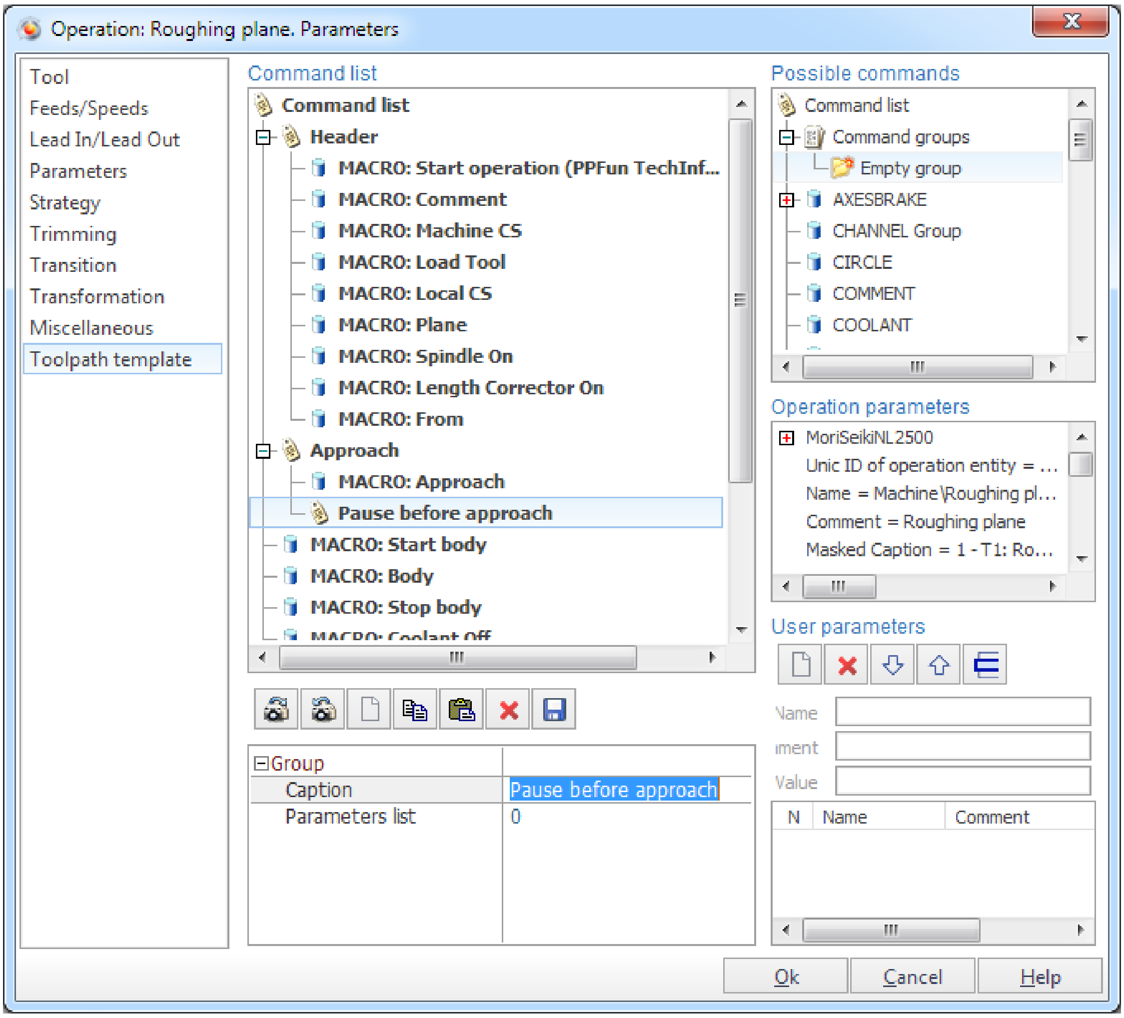


On choosing the <Save as User operation...> the current operation parameter's kit organize into the separate object which is kept safe in a special file independently from the current project and, so, may be used in other projects. After saving the default parameters become available during the creation of new operations (in the window of creation the new operation and drop down menu of new operation creation).

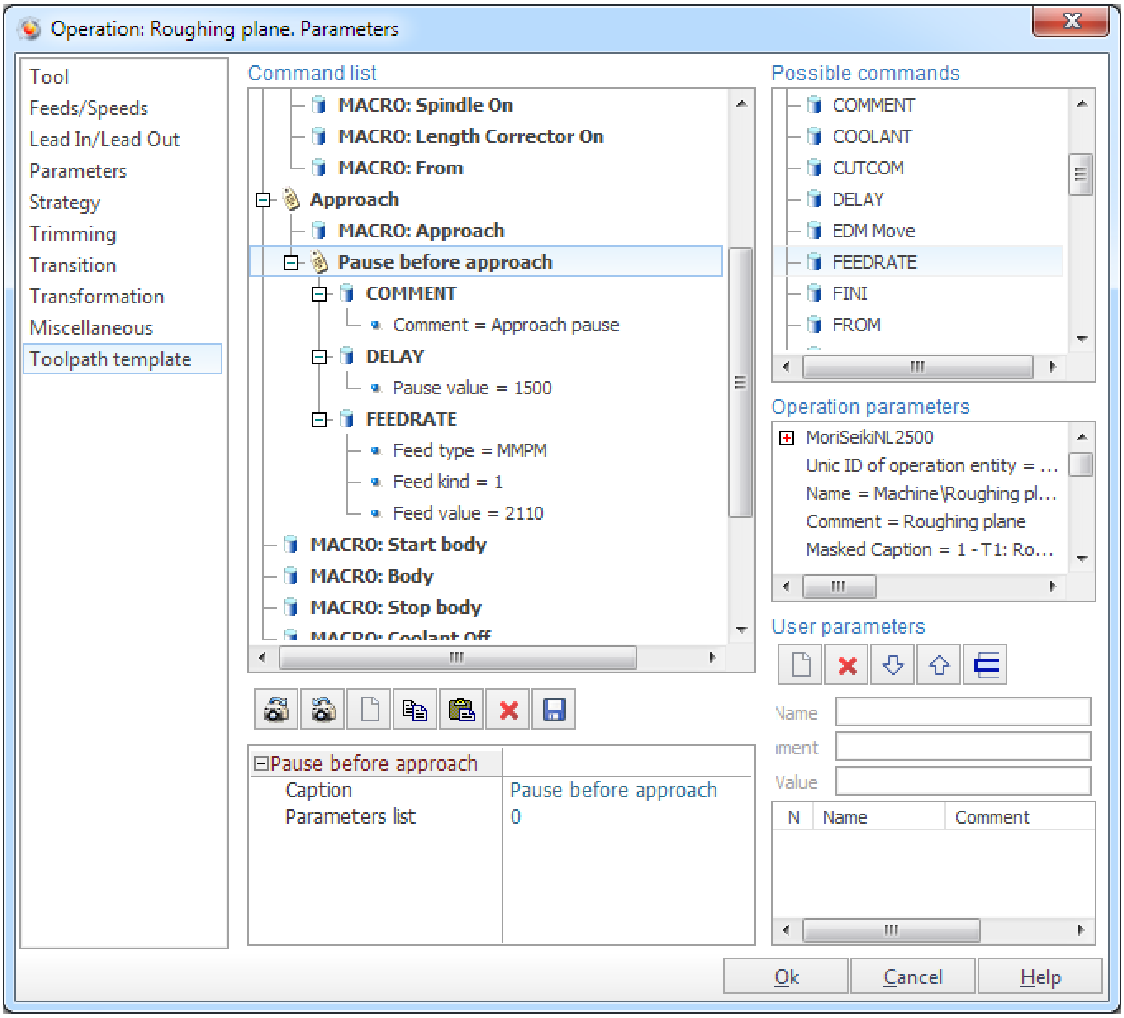


The default parameters may be applied to the operation not only during its creation but also to already existing operation. For that in the context menu of operation should be chosen <Load User operation...>. On the screen it will appear available for this type of operation list of defaults.

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.

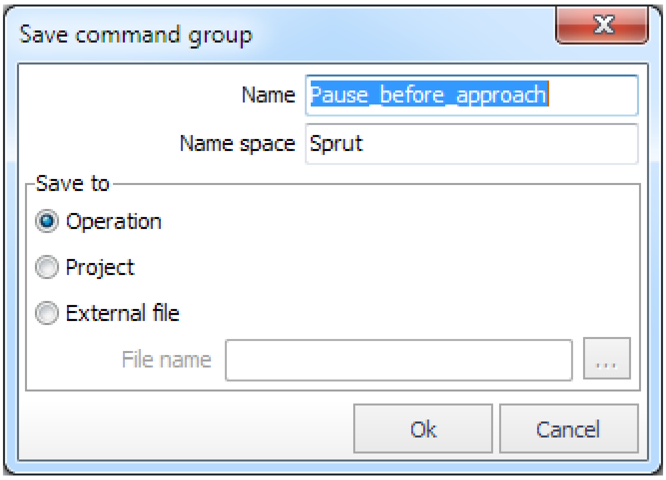


It is possible to add commands from <**Possible commands**> list to the group. So, you can form the set of commands for a specific task.



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.

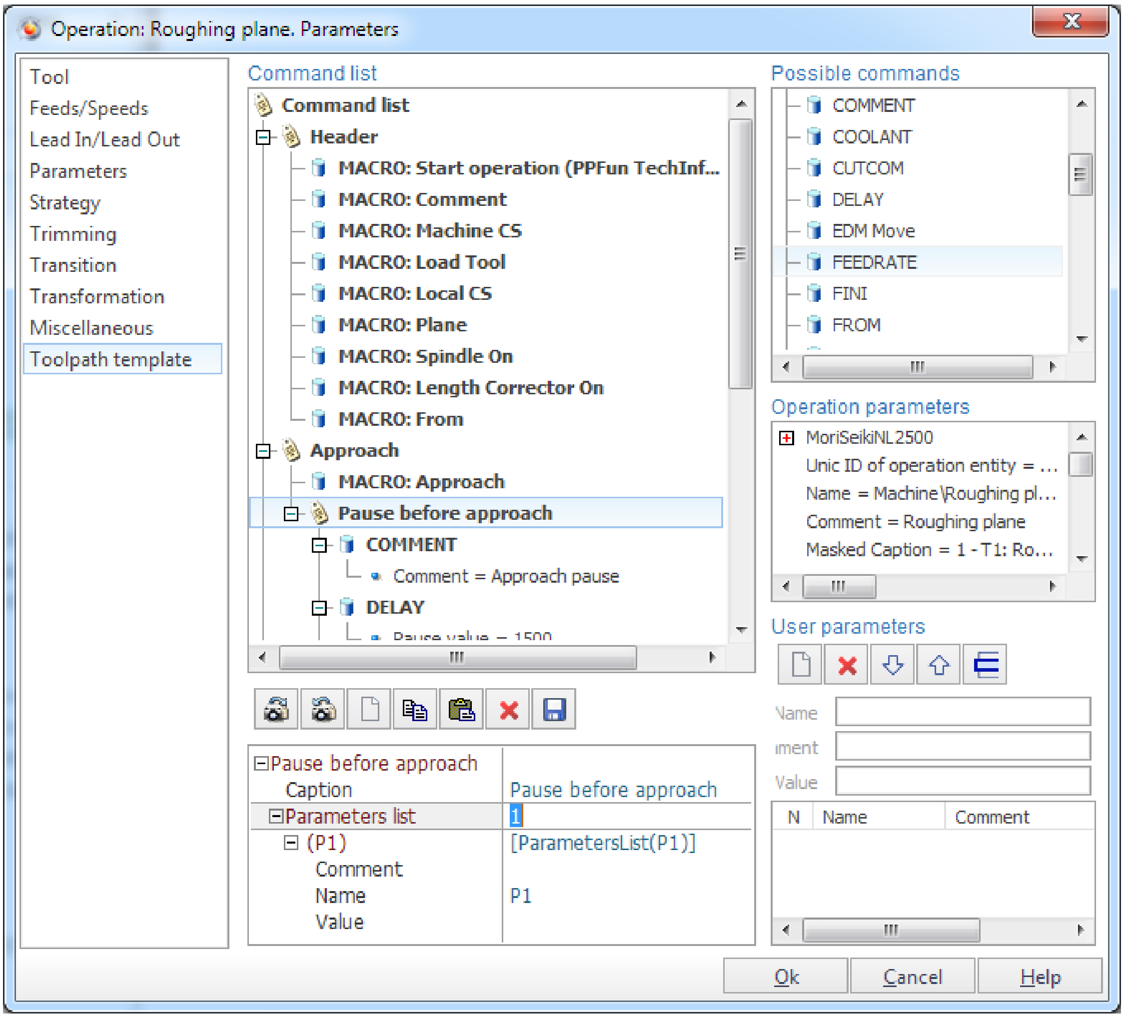
There are 3 group saving modes:

<**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.

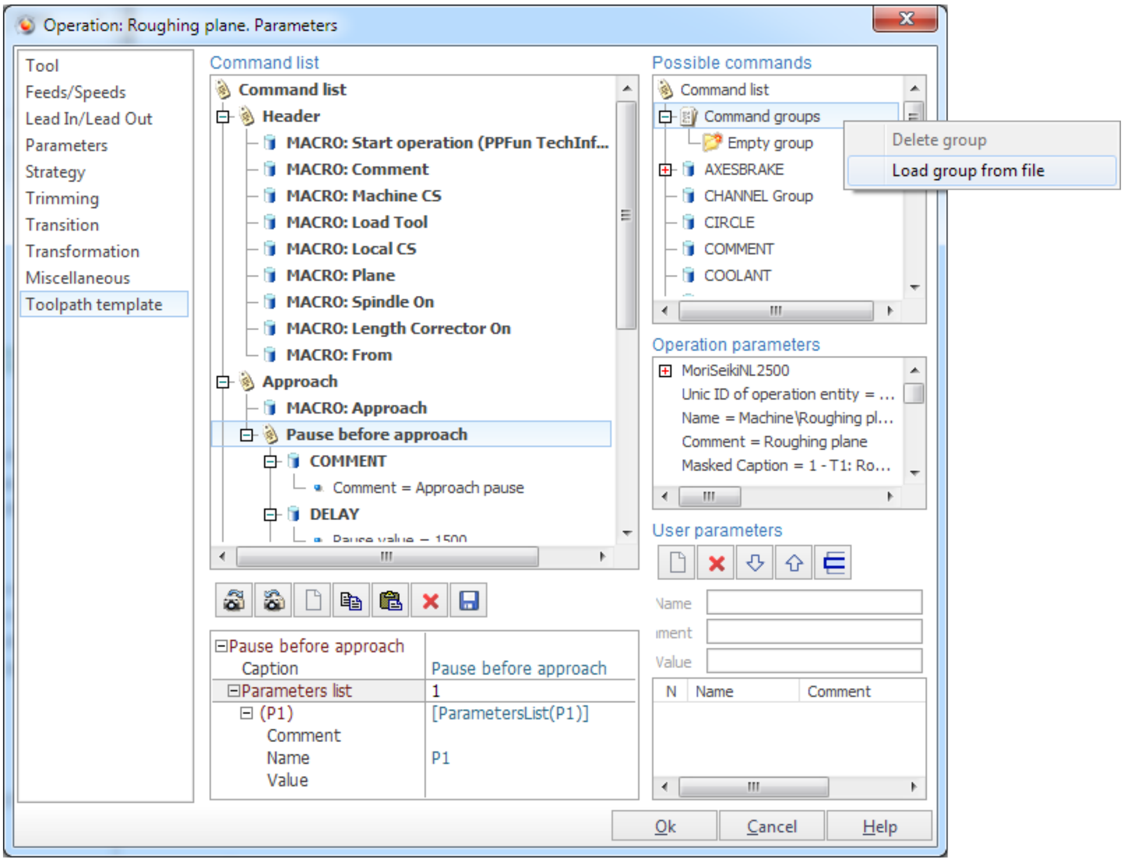
<**External file**>. Saves group in the external file. This allows you to use the group in all operations and projects. But it is necessary to remember that parameters specified on the <ParametersList> are unavailable in the other operations. So, it is impossible to use that parameters.

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.

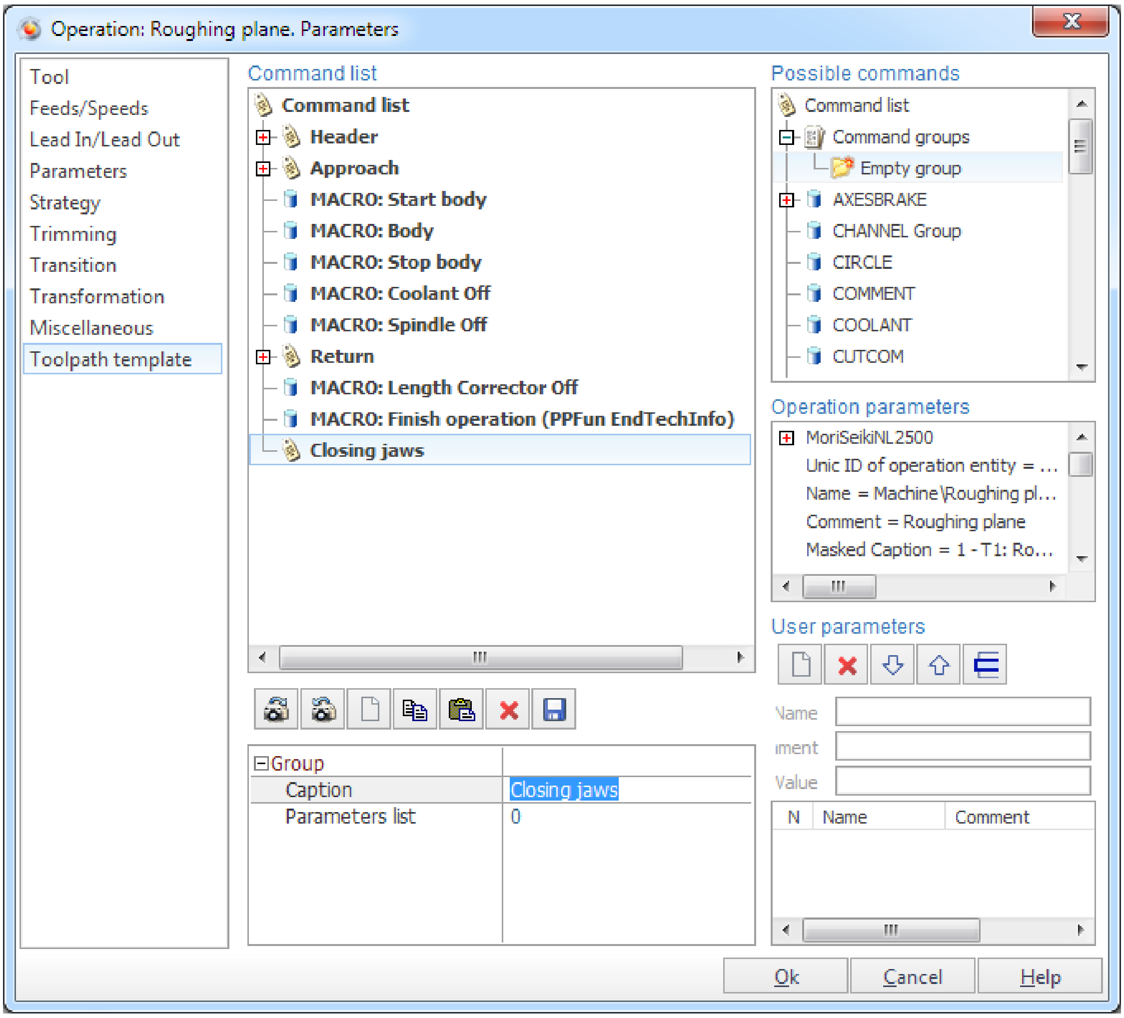
So it is possible to save all typical groups of commands to external files and even use them on the another computer. To load the group of commands from an external file, open the context menu by right-clicking on the list of possible commands and choose <**Load group from file**> item.



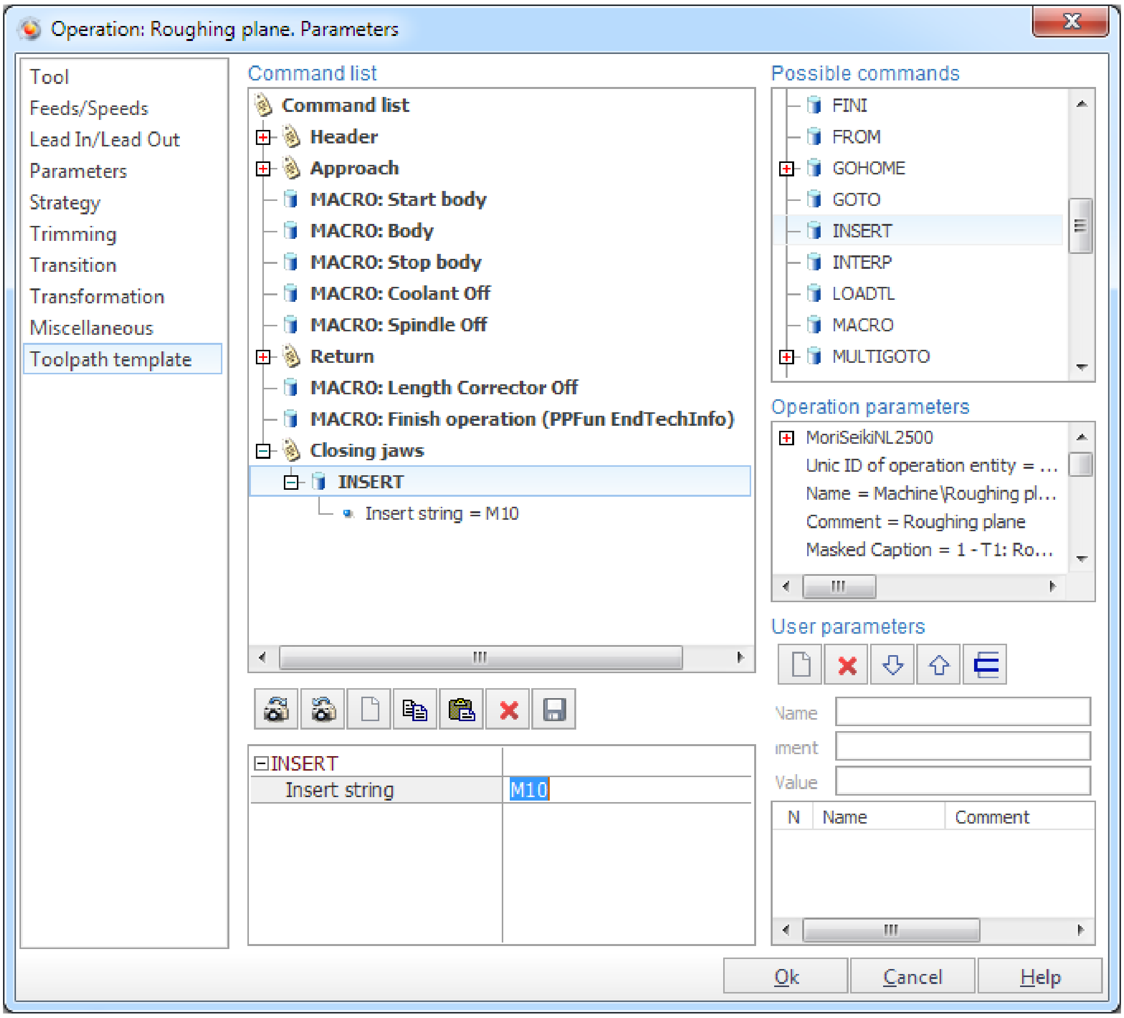
There is the example of using groups.

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.

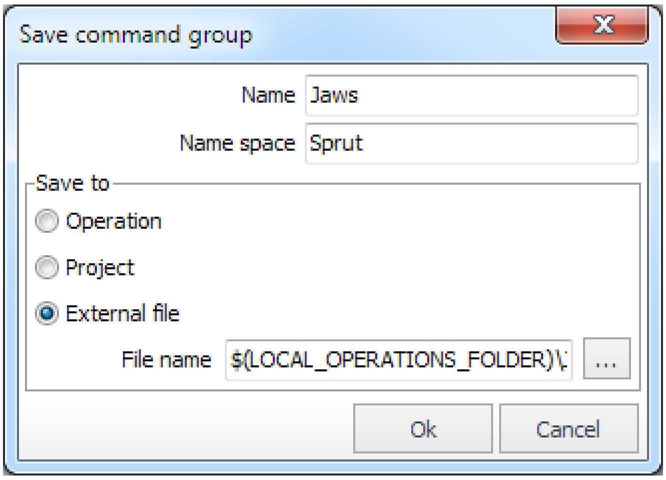
Step 1. Create the empty group in the command list and set the name <**Closing jaws**> for it.



Step 2. Provide the sending M10 command to the NC program. Add INSERT command to the group and set value = "M10".

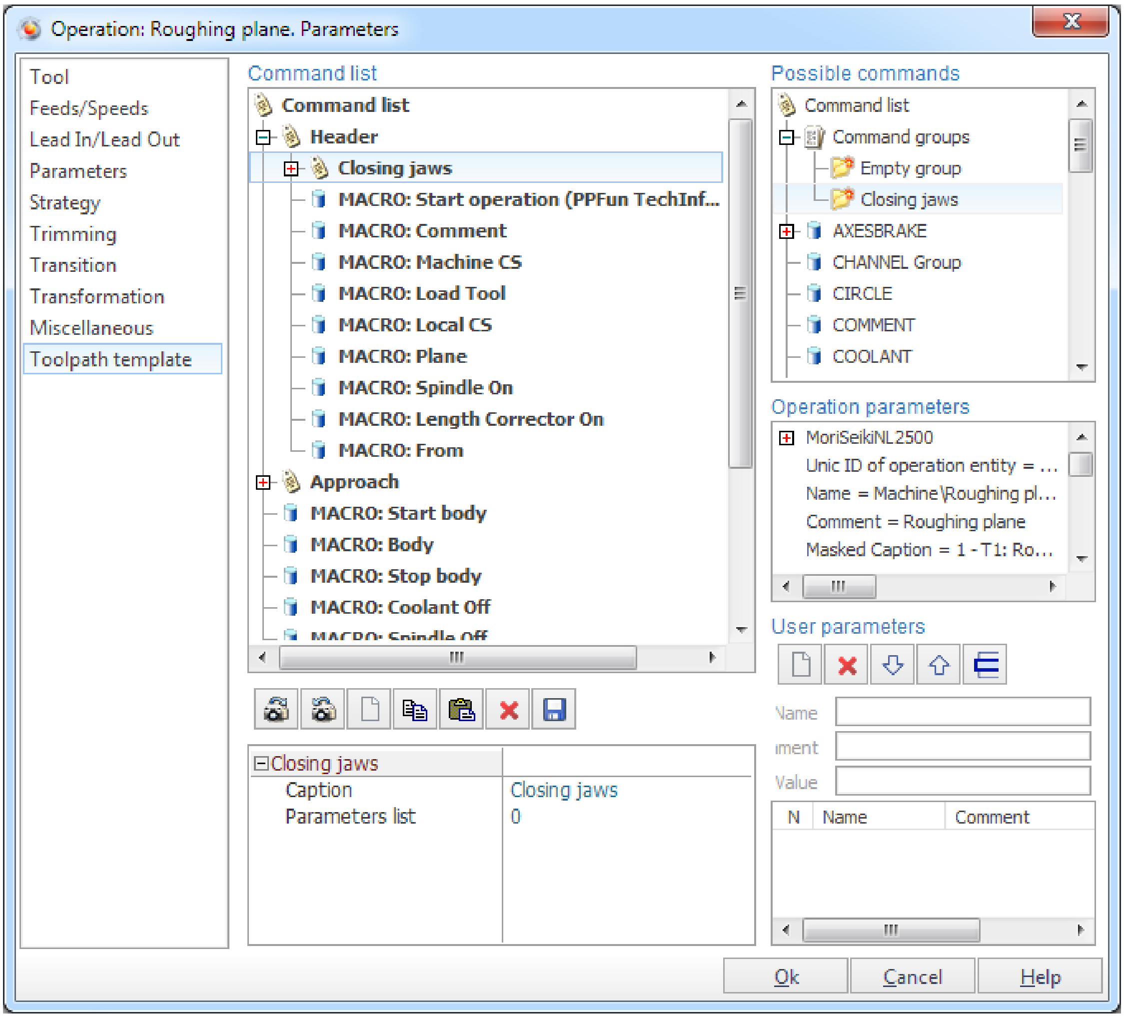


Step 3. Save formed group to the external file.

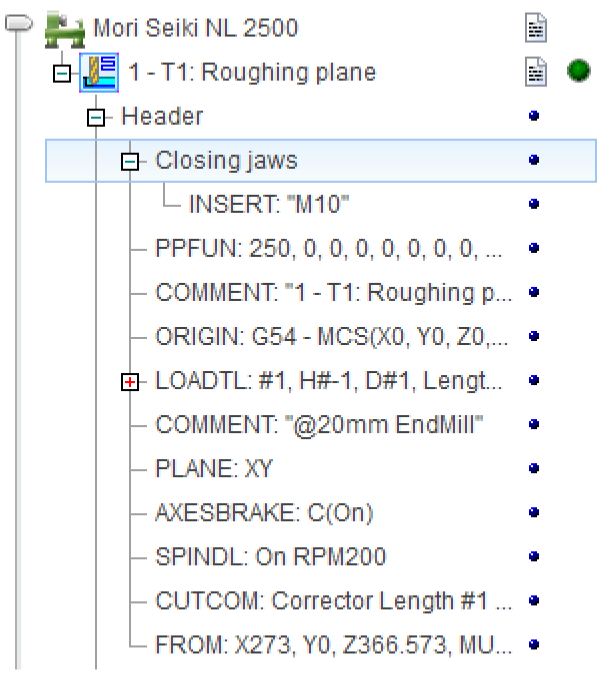


Formed group is available in the <**Possible commands**> list.

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>.

See also:

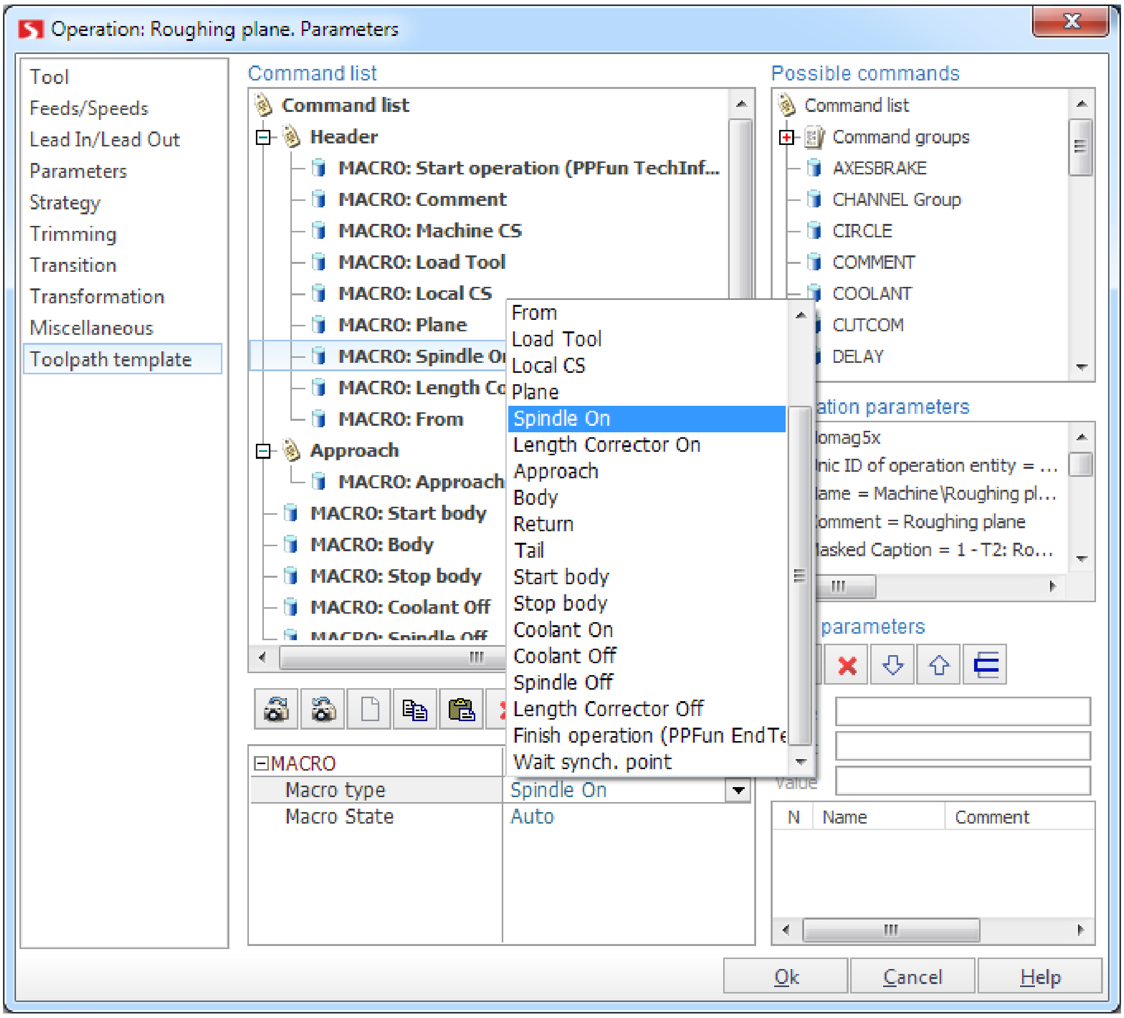
Common principles of technology creation

Operation default parameters

Macro commands in the tool path template

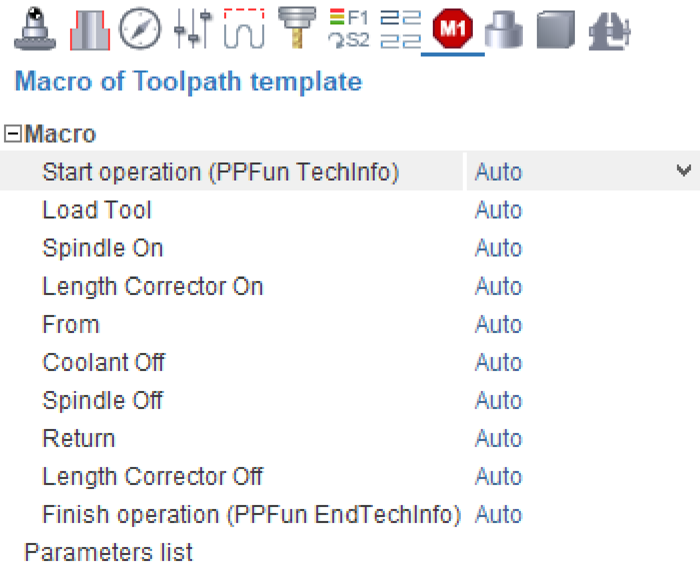
5.1.14.1 Macro commands in the tool path template

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 SprutCAM. 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.

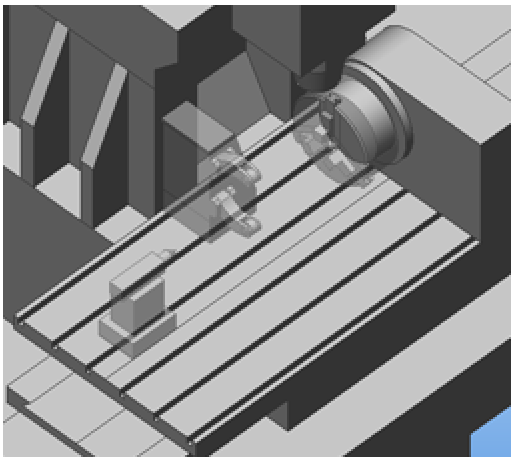
If the state is **disabled** then the CLData commands is not output. It allows to exclude the unnecessary commands without the appreciably template editing. For example, it is possible to output coolant switch off command even if it was not disabled in the previous operation. For the comfortable editing of the macro commands states it is located on the additional page of the operations properties inspector.



See also:

Tool path template

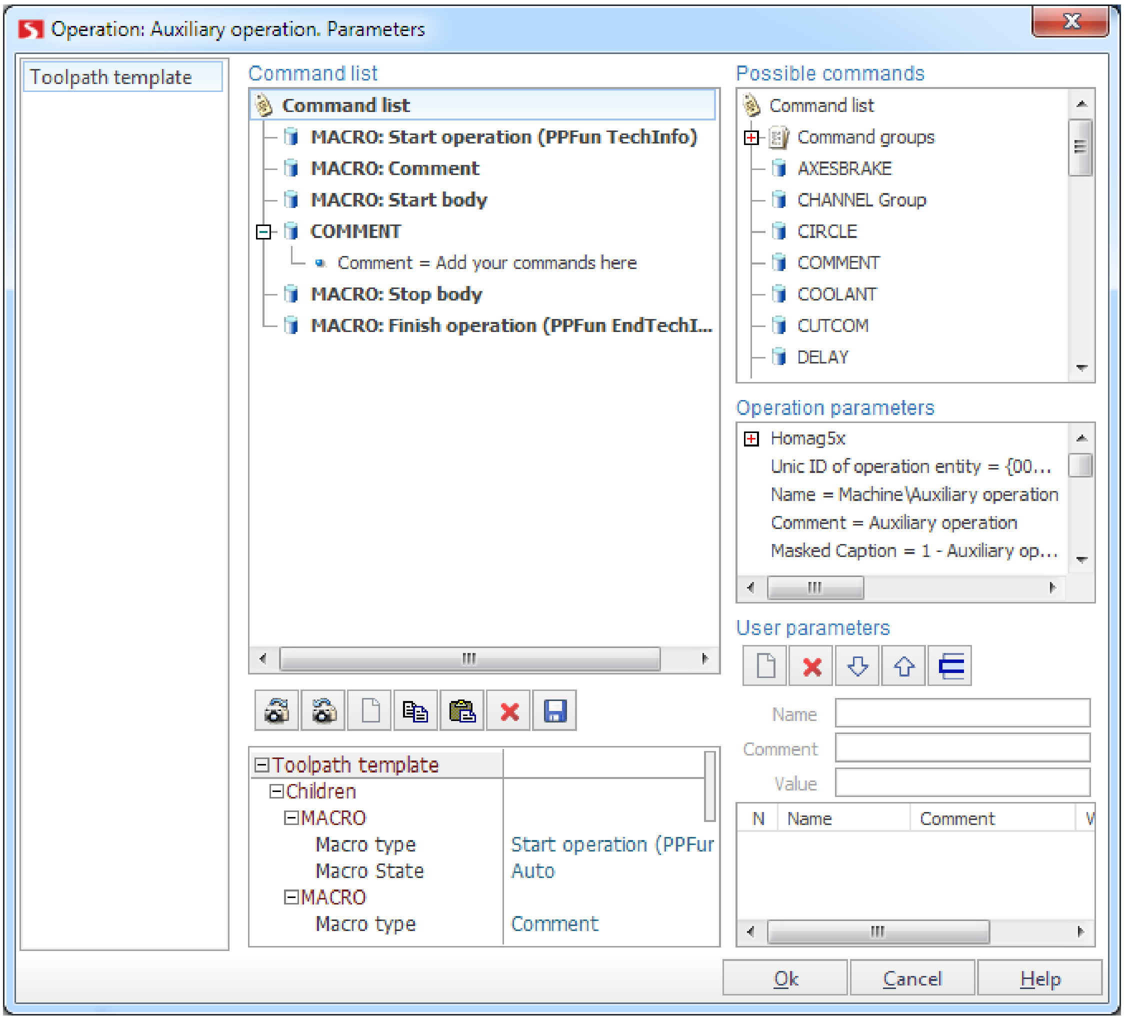
5.1.15 Creating of auxiliary technological operation



<Auxiliary operations> of SprutCAM X 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 SprutCAM X 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.



See also:

Common principles of technology creation

Toolpath template

5.1.16 Toolpath interpolation

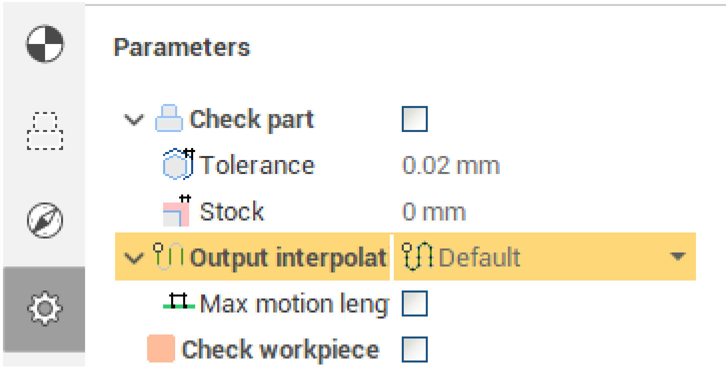
Using interpolation is one of the main ways to modify the path, allowing you to adjust smoothness and the concentration of points on trajectory.

Interpolation is available in operations:

- Disc cutting 6D
- Disc roughing
- Rotary roughing
- Sawing
- Morph
- 5D Surfacing

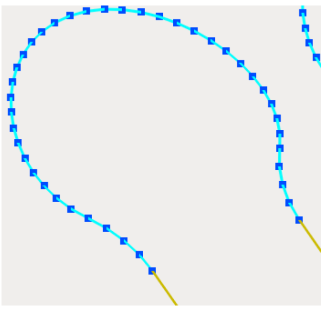
- Scallop
- 3D Helical
- Rotary machining
- 5D Contouring
- 6D Knife cutting
- Corners cleanup
- Pencil
- Cladding 5D
- 3D contouring
- Welding 5D
- Roughing drive
- Drive
- Plane
- Roughing plane
- Optimized plane
- Complex
- Combine
- Chamfering
- 5D by meshes

You can enable interpolation on the "Parameters" tab in the inspector window:

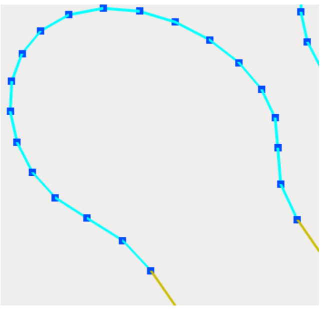


Interpolation options:

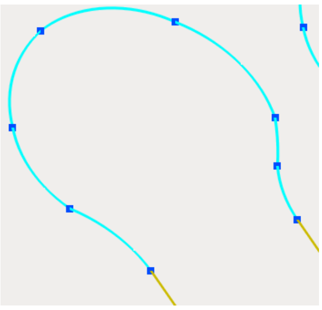
- **Default** - the system selects the type of interpolation depending on the parameters of the machine and the trajectory type. If the machine supports spatial arcs and the path of operation is 5 or 6 axial, then the "**Spatial arcs**" mode will be used. If the machine does not support spatial arcs, but supports arcs in the XY / YZ / ZX planes, and the path is 3 coordinate, then the "**Planar arcs / Helics**" mode will be used. If the machine does not support arcs, the "**Cuts**" mode will be used.
- **Off** - output only by cuts without changes.



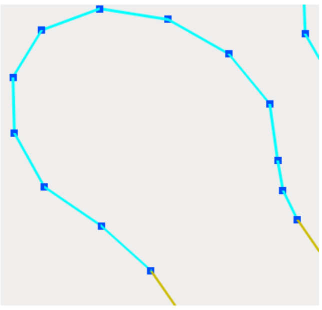
- **Cuts** - Cuts interpolation with a selected tolerance.



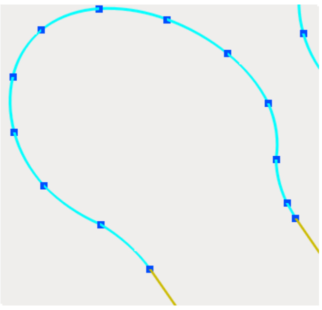
- **Planar arcs/helics** - arcs interpolation in the XY / YZ / ZX planes with the Circle command (depending on the operation).
- **Spatial arcs** - arcs interpolation using the "Multiarc" command, if supported by the machine.



- **Fixed length cuts** - output with cuts of a selected length. Valid only for a smooth path. Sharp corners can be cuts.



- **Fixed length spatial arcs** - output by "Multiarc" arcs of a selected length. Only permissible for a smooth trajectory.



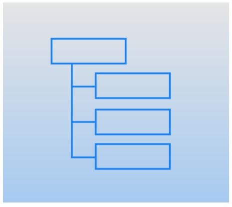
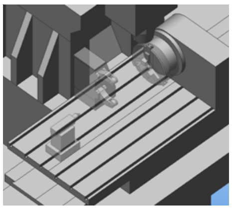
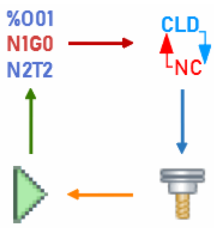
- **Spatial arcs(old method)** - output arcs with the "Multiarc" command (old method) in those operations where it was previously.

5.2 List of types of machining operations

Machining operations can be divided into two groups: roughing and finishing. The main difference between them is that the roughing operations perform clearance of thestock

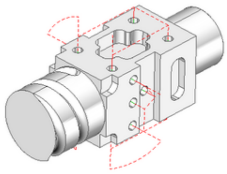
material and the finishing ones only perform surface machining. Rest milling operations differ from the others only by the default parameter values that are set during their creation.

The list of all machining operations with a short description is shown below. The types of machining operations listed here are divided into the same groups that they appear, in the new operation creation window.

Operations 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.
Auxiliary operation	
	<Auxiliary operations> of SprutCAM X 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 operation, G-code based lathe operation	
	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.

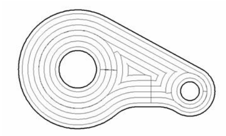
Roughing mill operations

Hole machining



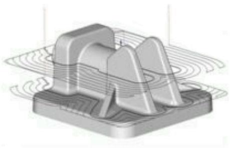
Creates a set of machining commands for holes. These include drilling, boring, centering, tapping or thread milling. The operation can be used both for hole machining and for preliminary drilling in tool plunge points in pocketing and waterline roughing operations. Hole machining operation can be used 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.

Pocketing



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.

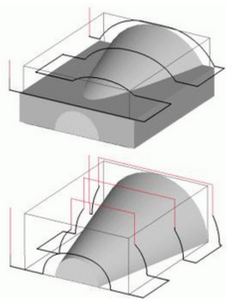
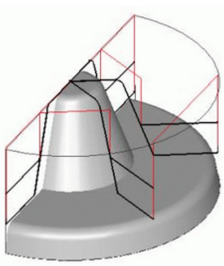
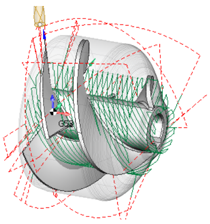
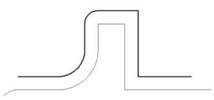
Waterline roughing operationWaterline roughing operation



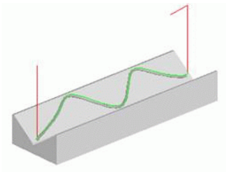
Waterline removal of stock material of a workpiece, which lies outside the 3D model. As in pocketing, the main part of the material is removed by the horizontal (XY) movements of the tool. The operation is often used for primary rough machining of complex models, which have considerable geometrical difference to the workpiece.

Plane roughing operation

Plane roughing operation

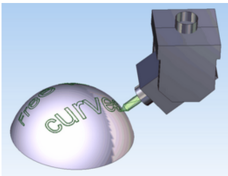
	Plane removal of stock material of a workpiece, which lies outside the 3D model. The sections lie in vertical parallel planes. To limit the pressure on the tool, machining can be performed with small preset Z depths. The finished operation is usually closer to the finished model than using the waterline 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.
Drive roughing operationDrive roughing operation	
	As in the plane operation, removal of the stock workpiece material that lies outside the volumetric model is performed by separate cuts. Depending on the operations parameters, cuts lie either in the vertical plane or in vertical mathematical cylinders, the shape and location of which are defined by the drive curves. To limit the pressure on the tool, machining can be performed preset smaller into Z depths. In some cases, the model after rough machining is very close to the finished model, but because of the uneven nature of the material being removed it is not always possible to reach an optimal machining time. The operation is best used only with certain workpiece and machined model shapes.
Roughing rotary machining	
	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.
Finishing mill operations	
2D contouring	
	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.

3D curve milling3D curve milling



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.

5D contour and 6D contour operations5D contour operation



5D contour operation is designed to generate the continues 5-axis tool path. Where are the three way to generate the work path depends on the way how the job assignment is set

1. The passes along the curves that is lie on the part surface.

2. The passes along the isoparametric curves of the defined surfaces..

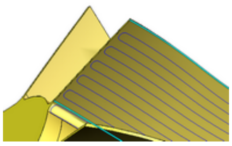
3. The passes along the edges of the part.

1. The passes along the curves that is lie on the part surface.

2. The passes along the isoparametric curves of the defined surfaces..

3. The passes along the edges of the part.

Morph operationMorph operation

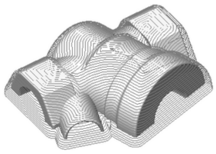


Morph operation generates a toolpath that smoothly morphs between two specified curves with high speed links. Available strategies include: Across, Along, Spiral.

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.

Scallop operation

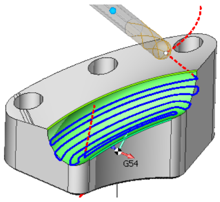


Scallop (or 3d constant step-over) tool path starts with curves lying on the part surfaces and repeatedly offsets them inwards until the curves collapse. consistent step-over across the part surfaces is guaranteed The tool path is well suited for high speed machining of complex molds and sculptured models.

Features:

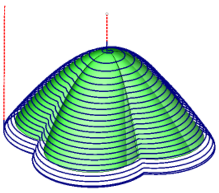
- Lightning fast toolpath calculation.
- One entry, one exit. It is possible to machine a whole part with a single continuous spiral-like toolpath with only one entry and one exit points.
- High speed machining. It is possible to generate a toolpath with rounded corners.

5d surfacing operation



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).

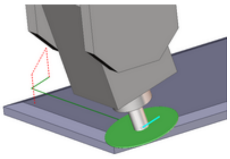
Helical operation



Helical machining operation are useful for machining of cylindrical parts without an undercuts. The entire model is selected as the job assignment.

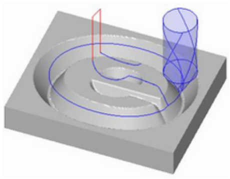
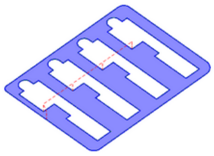
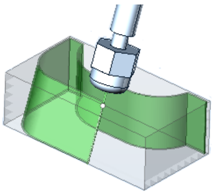
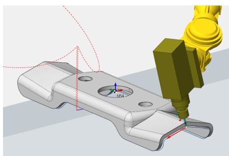
The operation can generate a single-pass spiral like path for the entire model. If there are model areas that can not be processed without a transition, it will be processed after the processing of the current pass. The operation does not control the height of the scallop and does not ensure a uniform height change.

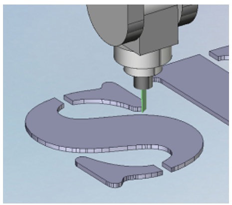
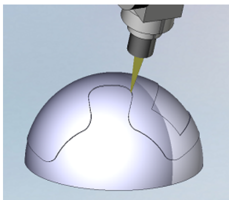
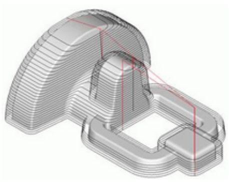
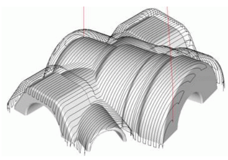
Sawing operationSawing operation

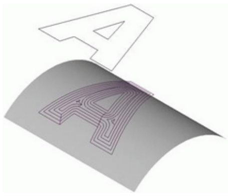


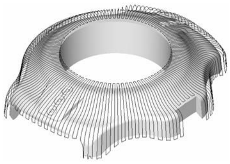
The Sawing operation is specifically designed for the fast programming of a saw blade for up to 5 axes milling of wood, marble, granite, stone and similar materials.

Benefits: Automatically calculates the correct saw inclination, approaches and sawing motion.

Engraving operation	
	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.
Jet cutting operation	
	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.
Jet cutting 4D operation	
	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.
Knife cutting 2D	

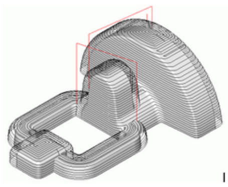
	<Knife cutting 2D> operation designed to programming cutting of sheet material with the tool like knife (**it can be knife, band-saw, disk-saw etc.**). A special transition formed in the sharp corners of toolpath that avoids the bendings of the material because of the sharp turn of the knife. The operation 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.
Knife cutting 6D	
	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.
Waterline finishing operationWaterline finishing operation	
	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.
Plane finishing operation Plane finishing operation	
	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.
Drive finishing operationDrive finishing operation	





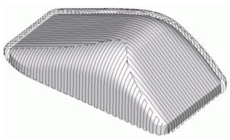
As in the plane operation, surface machining of a volume model is performed by separate strokes. Depending on operation parameters, the strokes lie either in vertical planes or vertical mathematical cylinders, the shape and location of which are defined by drive curves. The operation gives best results when machining separate areas of a detail with complex rounded wavy surfaces. It is best used for rest milling of surface areas of specific shapes, for machining of some models with smooth changing of surface geometry. And also, for milling inscriptions and drawings on a freeform model surface.

Combined operationCombined operation



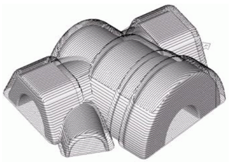
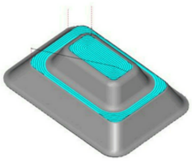
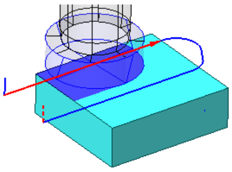
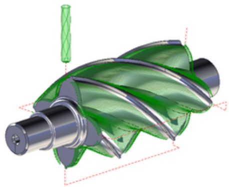
The toolpath for the surface machining of a volume model are formed in two stages. Firstly, the horizontal toolpaths (waterline), and then, for the remaining areas the toolpaths are created by using the rules for the drive operation. Because of this, both flat and steep areas are machined equally well. An even scallop height can be obtained when using a fixed step-over. Combined machining provides easier conditions for the tool, this allows using longer tools with a small diameter. The operation performs quality finish machining regardless of the model surface complexity, and also minimizes the machining time.

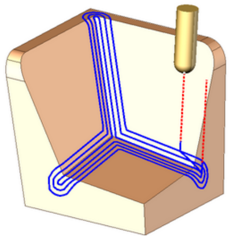
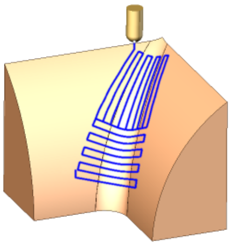
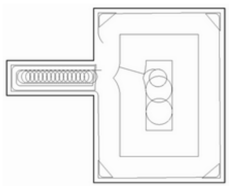
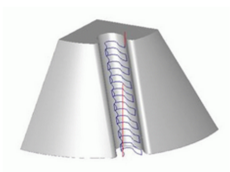
Optimized plane operationOptimized plane operation

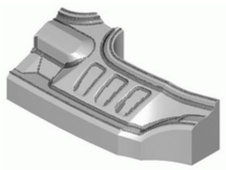
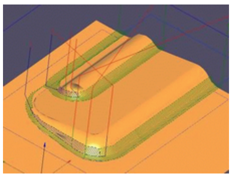
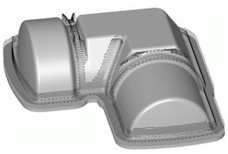


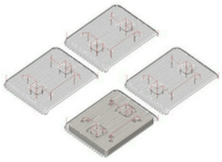
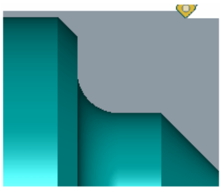
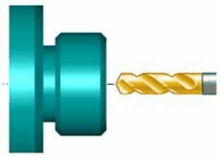
Two plane operations with mutually perpendicular toolpaths are created at a time for surface machining of a 3D detail. The default parameters of this operation are set so that every operation would machine only those surface areas of the model, where it can achieves an optimal result. This means that 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.

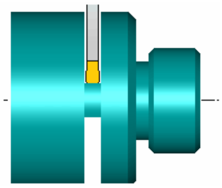
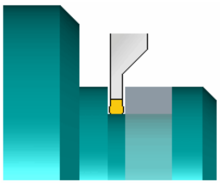
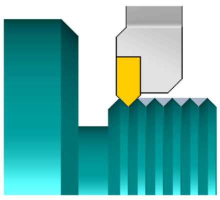
Complex operationComplex operation

	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.
Flat land machining operationFlat land machining operation	
	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.
Face milling operationFace milling operation	
	Face milling operation removes stock on a given horizontal plane with one of the following strategies: • One way, • Zigzag, • Optimized zigzag, • Spiral.
Rotary machining operationRotary machining operation	
	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.
Rest milling operations	

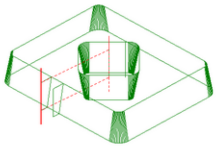
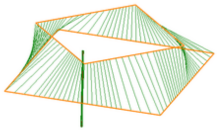
Pencil	
	The rest machining operation generates passes along inner corners of the part.
Corners cleanup operation	
	The rest machining operation takes the diameter of the previous tool as a parameter and generates passes where the previous tool would leave unmachined material.
Area rest millingArea rest milling	
	The area rest milling operation performs rest milling of remaining material. That is, it performs a waterline rest milling inside the defined area or pocket. The shape of the area is formed from curve projections on the horizontal (XY) plane. The operation is used for the remachining of residual material using a tool of smaller diameter than the previous tool for 2 & 2.5D machining of pockets and isolated areas.
Waterline rest millingWaterline rest milling	
	Rest milling of a models surface using a waterline finishing operation. Model surface areas, insufficiently machined by the previous operations are milled using horizontal passes of the tool. A good result can be achieved when machining almost vertical areas.
Plane rest millingPlane rest milling	

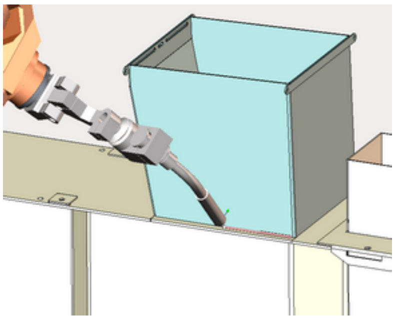
	Rest milling of model surface using plane finishing operations. Model surface areas, insufficiently machined by the previous operations are milled using passes lying in vertical parallel planes. The operation is intended for use when remachining slightly sloping areas and areas close to the vertical that are perpendicular (or close to) the toolpath.
Drive rest millingDrive rest milling	
	Rest milling of a surface model by drive finishing operations. A model's surface areas, insufficiently machined by the previous operations are milled using passes lying either in vertical planes or mathematical cylinders. By default, the drive curve should be formed along the unfinished areas, which allow rest milling to be performed with minimum number of passes. The operation gives the best results when machining non-vertical areas.
Optimized plane rest millingOptimized plane rest milling	
	Rest milling of a surface model by optimized plane-finishing operation. Models surface areas, insufficiently machined by a previous operation are milled by using 2 plane operations with mutually perpendicular toolpaths. Because each operation only machines its optimal area, good results are achieved. Rest milling using plane-optimized operation is recommended for use where there are relatively large unfinished areas.
Complex rest millingComplex rest milling	
	Rest milling of a surface model using the complex finishing operation. A model's surface areas, insufficiently machined by previous operations are milled by two operations: plane and waterline. Flat areas are machined by the plane operation and areas close to vertical by the waterline. The operation allows quality rest milling of free-form areas at any angle of the model surface.
Multiply group	

	The operation makes spatial transformations of a toolpath of any operations with copying or multiplying by a scheme. It is expedient to apply to machine models with repeating fragments. The operation allows to reduce time of calculation and debugging of an NC-program.
Turning operations	
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.
Lathe Facing	
	The operation is designed to machine the uneven ends. It is used to prepare the base surface before the drilling or before another turning cycle. The operation can be uses either for finishing or the roughing machining.
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.
Lathe part-off	

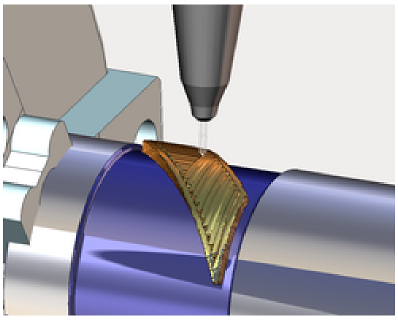
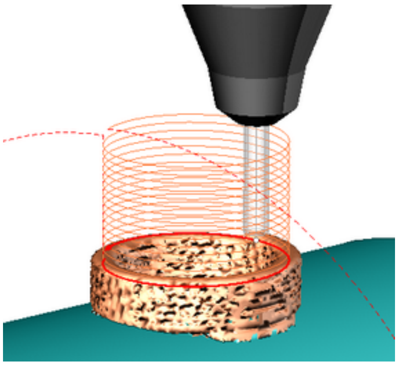
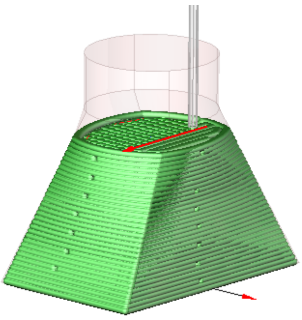
	The operation cut out the part with the additional chamfer or rounding machining. The size of a groove, the chip breaking parameters and the delay values can be set.
OD finishing, ID finishing	
	The operation is designed for the final finishing machining. The machining is performed by the offset motions 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 and Face grooving	
	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 threading, ID threading, Profile 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

Wire EDM
Wire EDM 2D Contouring

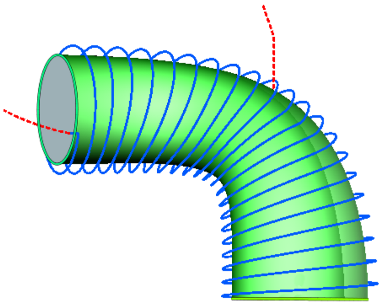
	The <Wire EDM 2d Contouring> operation is designed for wire path generation along flat contour on 2d contouring as well as along flat contour with wire slope angle on taper or 3d contouring. Resulting wire path is based on contours, which lays in one plane.
Wire EDM 4D Contouring	
	The <Wire EDM 4d Contouring> 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.

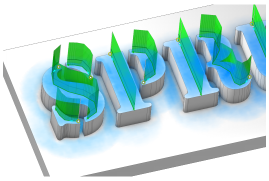
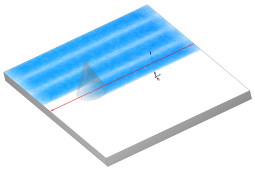
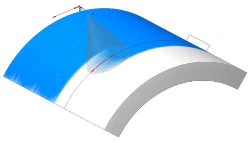
Welding operations	
Welding 5D	
	It implements the functional of automatic weld seam geometry calculation without reference to a particular type of welding equipment (i.e., does not generate the specific commands to the laser, electric arc, gas burners, ultrasonic device, etc.). It is enough to add the 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.

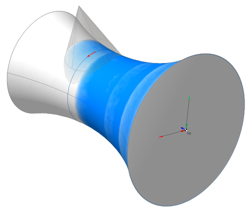
Additive operations
Area cladding

	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 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

	"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.

Spraying operations	
Contour spraying	
	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. 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	

	Rotary spraying operation based on Rotary finishing operation

See also:

Common principles of technology creation

5.3 Basic technology terms

No content in this page. See child topics

5.3.1 Operations 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).

See also:

List of types of machining operations

5.3.2 Part

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.

See also:

Job Assignment

Technological operation list

5.3.3 Job assignment

The job assignment defines the surfaces and the areas of surfaces to be machined by an actual operation. Depending on the operation type, the job can be defined by different methods.

For the curve machining operations (such as 2D contouring, 3D curve milling), a job assignment can be defined by a set of curves, or groups that contain the curves, and also by points, that define circle centers. All geometrical objects of other types will be ignored. The tool toolpath will be created with regard to the user-defined options, e.g., tool offset left/right/center from the curve. An additional stock can be added for every curve, as well as the operations stock. The stock amount will only be applied if the tool edge is touching the contour (left or right). When the tool travels along the center of the curve then any stock values are ignored, and no offset toolpath is calculated.

In the engraving and pocketing operations, the job assignment is formed as an area at the top machining level. The area at the top machining level can be created from projections of curves onto the horizontal plane, any additional stock is also considered. Every curve can define either a ridge, ditch or an inversion curve, the thickness is defined by the additional stock value. Closed curves can also define a ledge, cavity or an inversion area, also considering the additional stock. Thus, the order that the objects appear in the list has considerably effect on the appearance of the resulting area. The toolpath is calculated so as to remove the workpiece material from outside of the model, taking into consideration any lateral angle applied for the operation.

For most of volume machining operations, the job assignment is defined by a set of solids, surfaces and mesh. The objects can be present in any combination and be grouped in an arbitrary way. All curves that are in the machining model will be ignored. When defining the job assignment, every geometrical object or its group can have an additional stock amount defined, which will be added to the operation stock. The tool toolpath is calculated to remove the workpiece material, which lies outside the job assignment, considering any stock amounts. This means that the tool will never cut the solids, surfaces or meshes that make up the job assignment.

The job assignment for the drive operations define a shape of the toolpath in the XY plane while the Z coordinate of the toolpath is calculated to guarantee the used tool touches the part.

The job assignment is formed from the curves projections onto the horizontal plane taking into account the prescribed stocks as well as the ways the actual curves (or curve groups) have to be included into the resulting area.

Each curve projection can form either a ridge or a ditch or an inversion curve whose width is determined by the specified stock. Furthermore, each curve projection can represent also a body, a hole and an inversion area. In those cases, the specified stock is also taking into account. The objects are included into the resulting area by the order they are listed using Boolean operations. Thus, the order the objects are listed has a great influence on the result. The way the tool will clear the resulting area itself is determined by the operation strategy.

If the job assignment for a drive operation is not specified then the system uses the part envelope as the job area.

The job assignment for turn operations introduces several ways to specify one or more fragments of the turn generatrix of the part to be machined.

See also:

Defining part, workpiece and fixtures

Part

5.3.4 Workpiece

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.

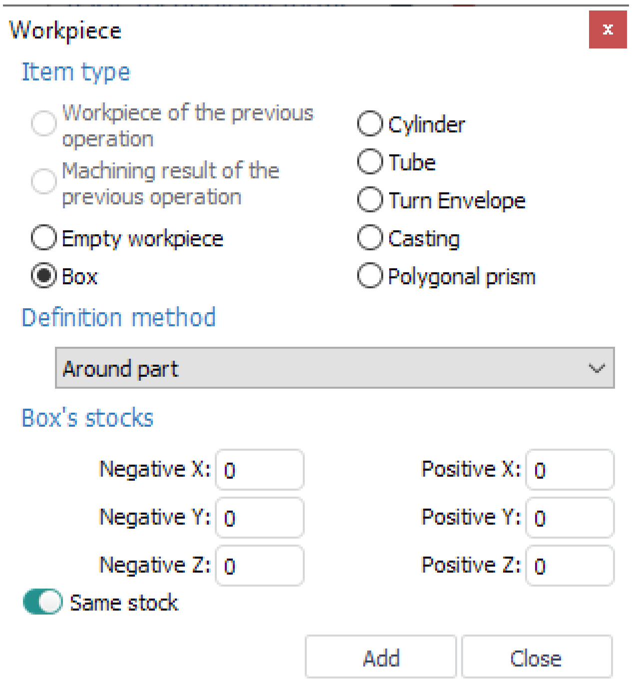
For elements < A cylinder swept around the part > radius of cylinder is calculated by part faces not a part box.

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 >.

Workpiece definition window if shown below.



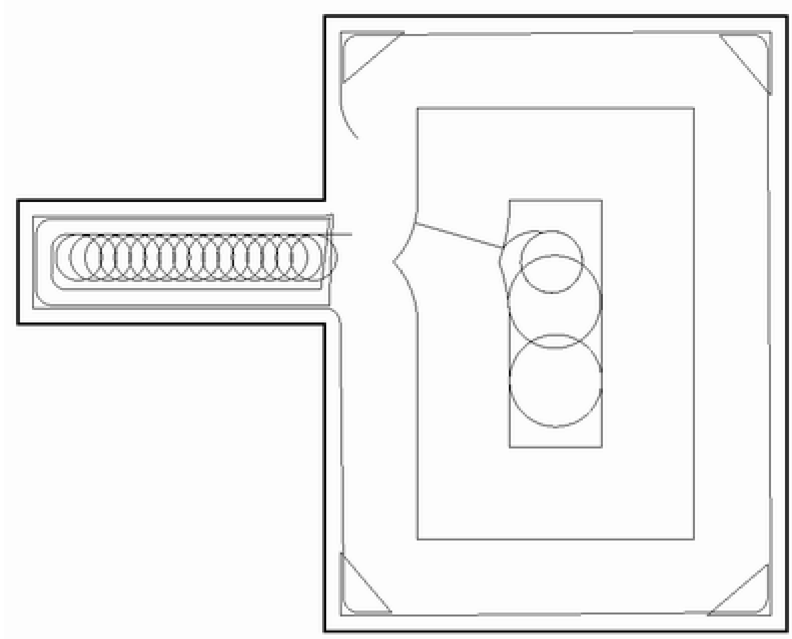
See also:

Defining part, workpiece and fixtures

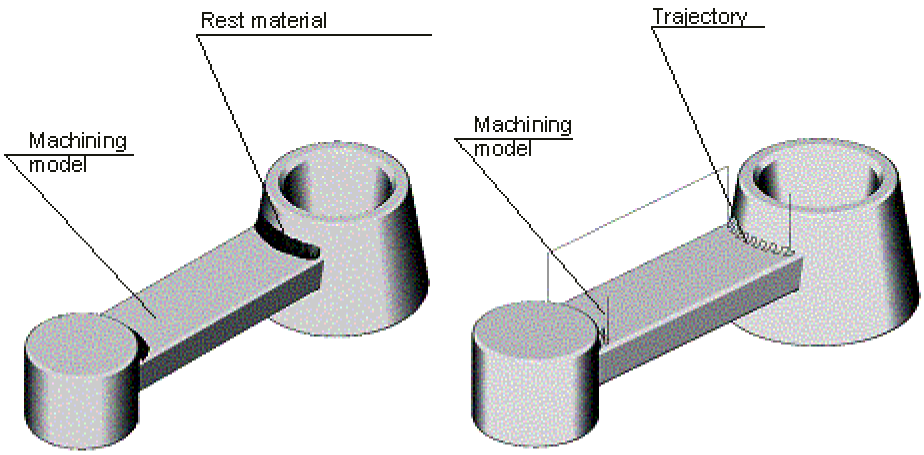
Part

5.3.5 Rest machining of remaining material

To machine only the material leaved by preceding operations one should define the operation workpiece as the machining result of the previous operation. The roughing operations use that workpiece to generate the toolpath removing only the excessive material. The finishing operations by default ignore the workpiece and machine all the given surfaces irrespective of the rest stock. To generate a finishing toolpath that eliminates cuttings of the already machined surfaces one should check the <Check workpiece> box and set the value to the <Stock to ignore> in the operation strategy panel.



It is recommended to set the value of the stock to be not less than the height of the obtained scallop for previous operations. One should also note that, if running rest milling with zero stock, and the value of the ignored layer of the remaining material is set to less than the stock for previous operations, then the whole model will be rest milled. This is because the stock material of the previous operations exceeds the size of the ignored unmachined areas.

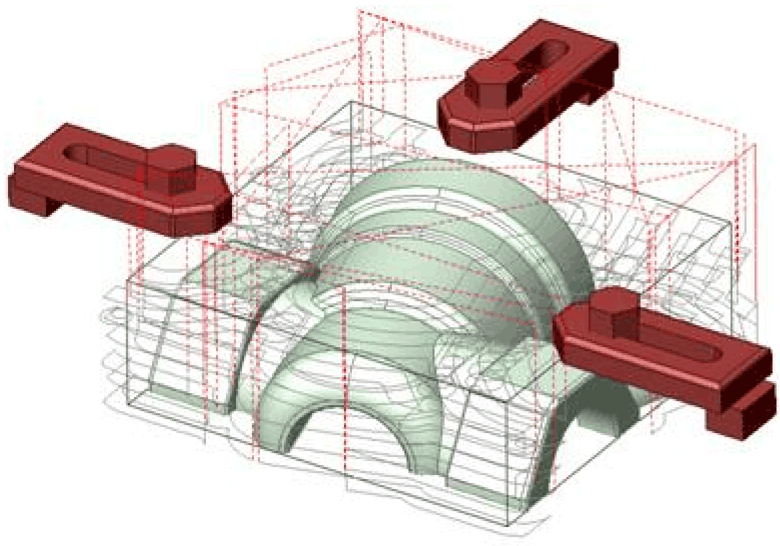


See also:

Operation of the re-machining of the rest material

5.3.6 Fixtures

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 fixtures of an operation are set to the <Fixtures of the previous operation> by default. If no <Fixtures> are defined then check for collisions is not performed.



All tool movements, irrespective of the operation type, can only be performed outside the borders of the fixtures. The finishing operations can machine only those areas of the model being machined that are located outside the restricted model. The roughing operations can remove material only from outside of the restricting model.

Note: Although the restricting model is not machined, there can be moves between work passes performed along it, and the rough operations perform material removal located outside the restricted model. Therefore, a guaranteed gap between the tool and the real restricting objects should be considered either directly in the geometrical model of the restricting objects, or by the additional stock value. If a restricting model is not defined, then it will be considered that machining can be performed in all areas.

See also:

Defining part, workpiece and fixtures

5.3.7 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.

See also:

Defining part, workpiece and fixtures

5.3.8 Drill points

In the waterline roughing and pocketing 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.

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.

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.

See also:

Hole machining

Job assignment for hole machining operation

5.3.9 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.

See also:

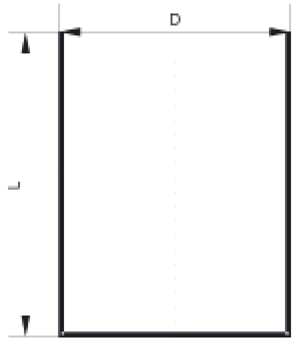
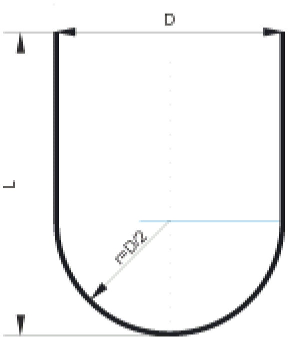
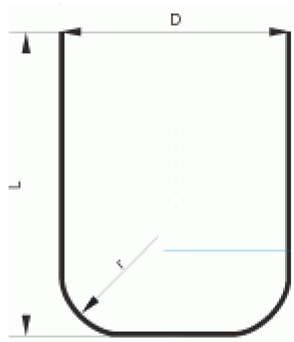
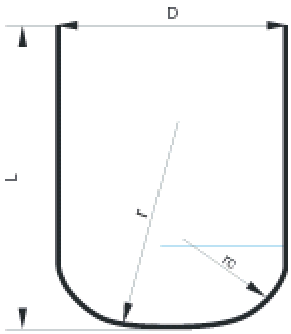
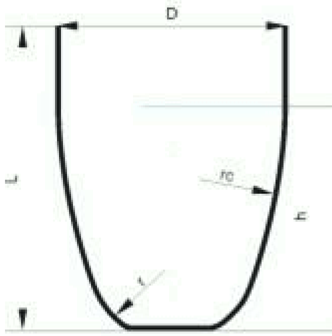
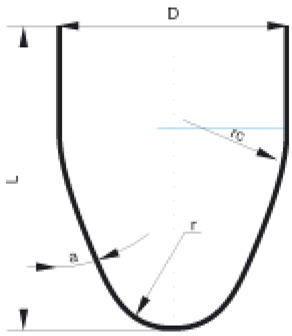
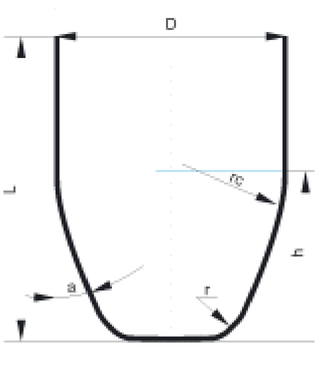
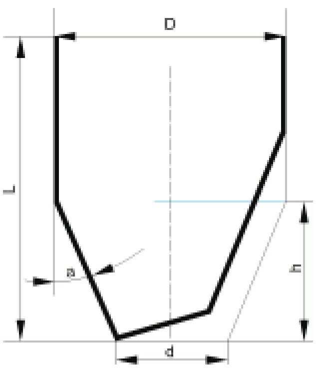
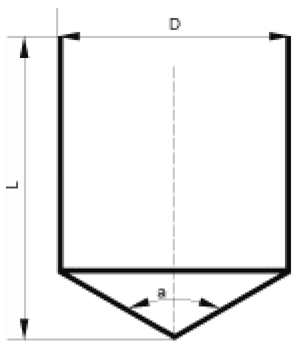
Mill tool
Lathe tool

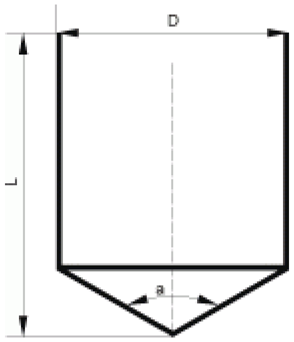
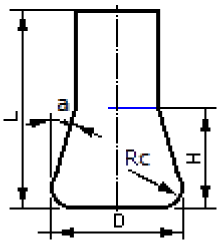
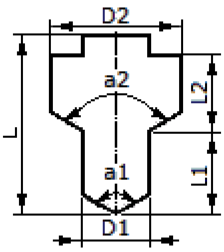
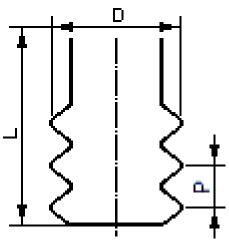
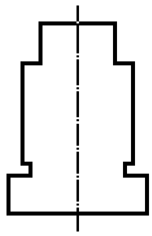
Tool overhang

5.3.9.1 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:	**Spherical mill:**	**Torus mill:**
length, diameter;	length, diameter;	length, diameter, rounding r

		
Double radial mill: length, diameter, rounding radius at the cylindrical part, rounding radius of the peak;	**Limited double radial mill:** length, diameter, rounding radius at the cylindrical part, rounding radius at the peak, height;	**Conical mill:** length, diameter, rounding radius at the cylindrical part, rounding radius of the peak, angle;
		
Limited conical mill: length, diameter, rounding radius at the cylindrical part, rounding radius at the peak, angle, height;	**Engraver:** length, diameter, angle, height, peak diameter;	**Drill:** length, diameter, grinding angle;
		
Center drill:	**Undercut mill:**	**Two stage drill:**

		
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>.

See also:

Definition operation parameters

5.3.9.2 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).

See also:

Technological operation list

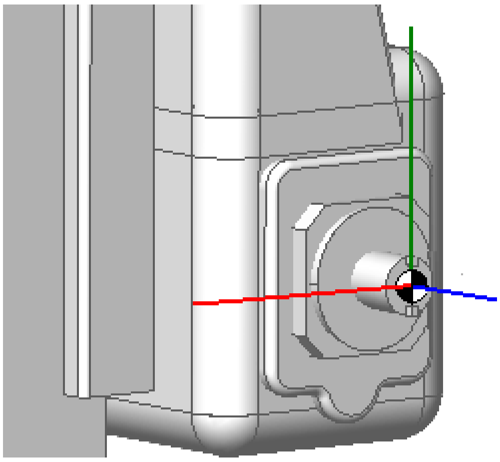
5.3.9.3 Tool overhang

Tool overhang is distance from tool base point to tooling point.

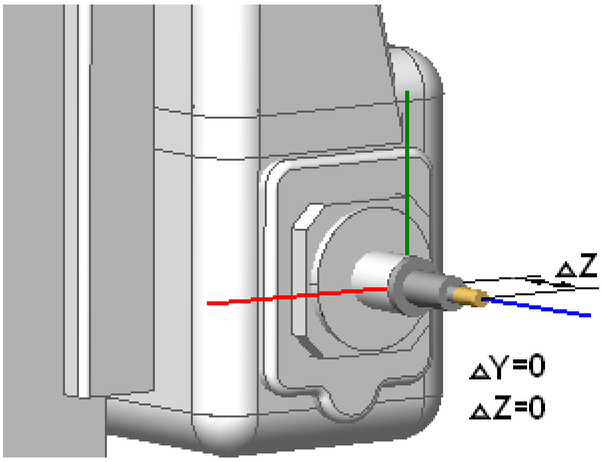
Tool base point is base point of instrument block (tool and tool holder) at machine coordinate system.

For mill machine tooling point lie along tool axis.

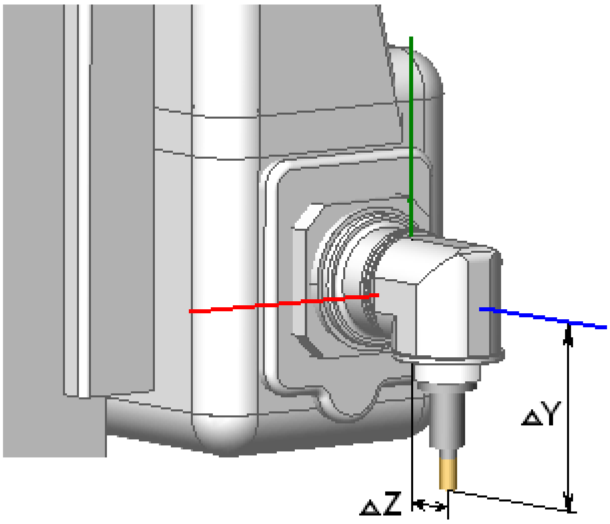
Some overhang definitions for mill machines are shows below.



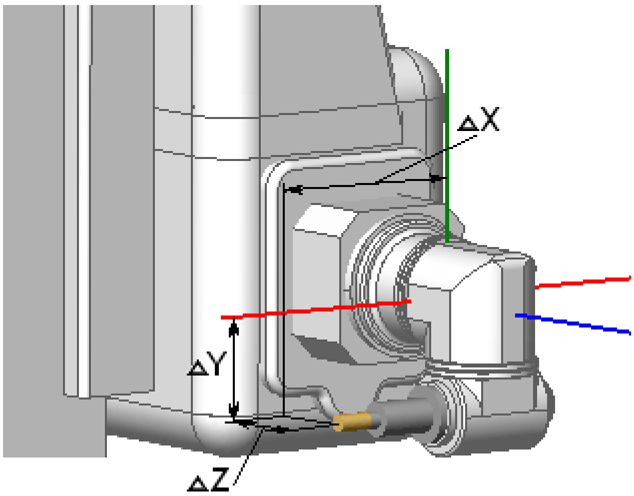
Tool base point



Axial tool overhang



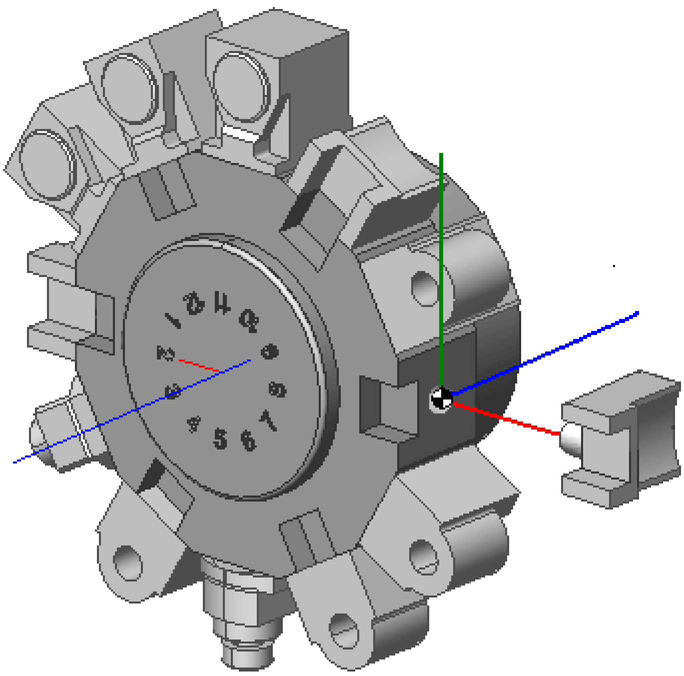
Overhang for rotary head



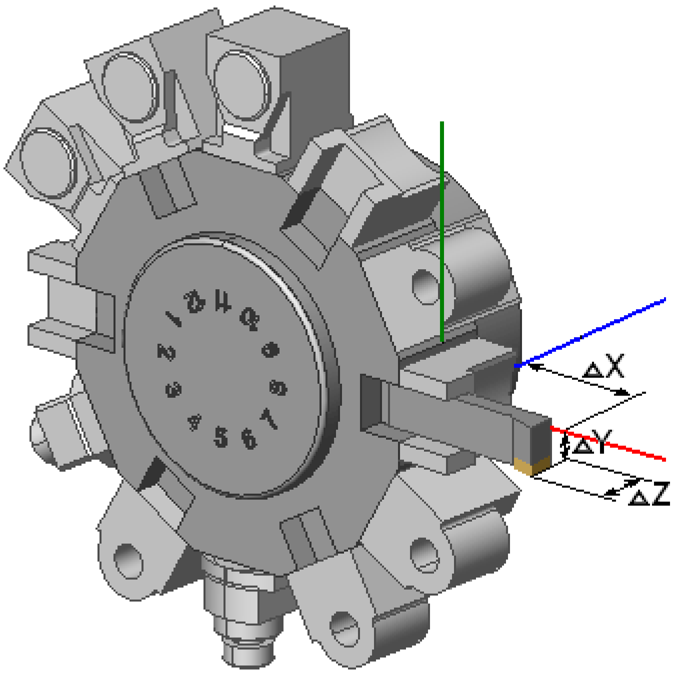
Overhang for 2 rotary heads

For lathe machines base point lie on turret base plane along tool fix system axis.

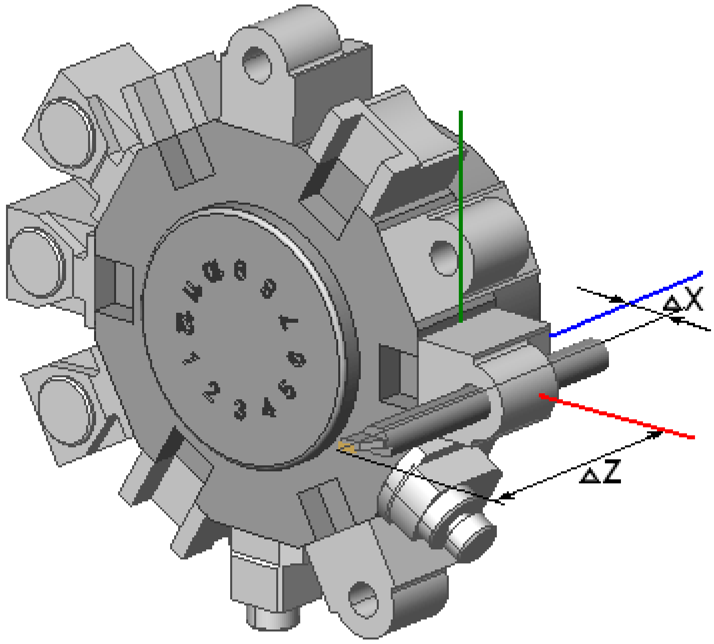
Some overhang definitions for mill-lathe machines are shows below.



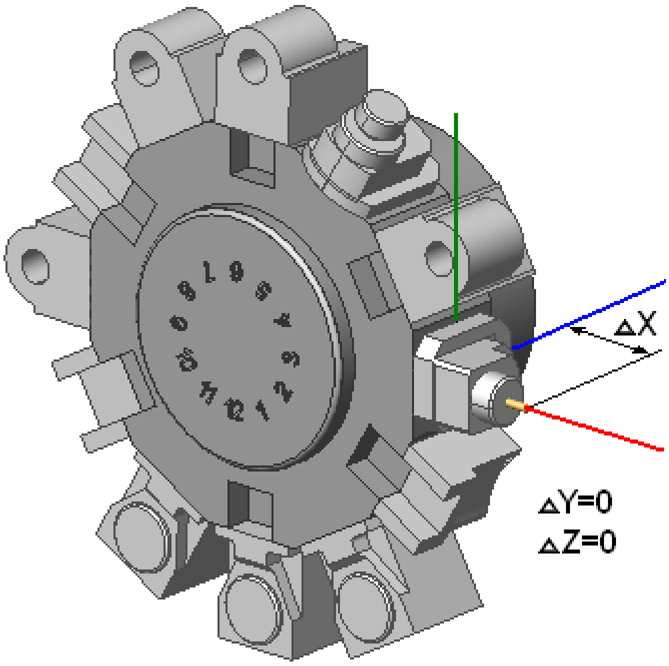
Base tool point on turret



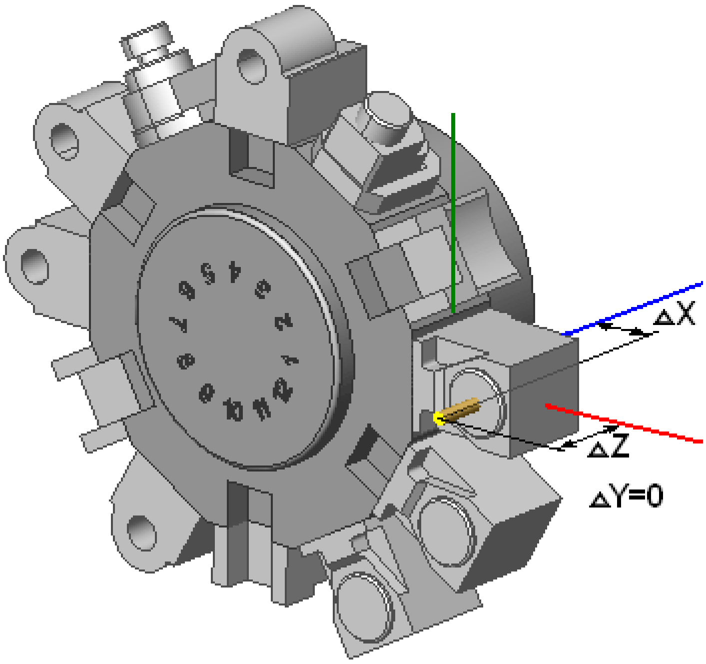
Straight turning tool overhang



Boring bit overhang

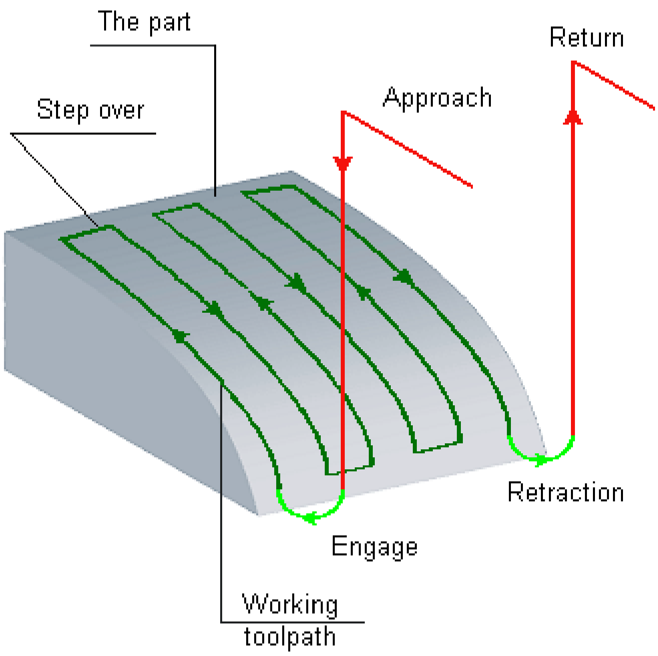


Base point for radial milling head



Base point for axial mill head

5.3.10 Tool movement trajectory areas



- Approach
- Engage
- Working toolpath
- Step-over
- Retraction
- Return

Some operations also include a tool plungeplunge move.

5.3.10.1 Approach

Approach means the path the tool takes to arrive at the first point of an engage or plungeplunge move either from a tool change position or from the last position of the previous operation.

Usually, an approach move consists of two steps:

1. Tool movement with rapid feedrate at either the <Safe plane> (for 3D operations) or, at the <Safe coordinate> (for 5D operations).
2. Tool movement with rapid feedrate to the first position of either the engage or the plungeplunge move of the first working toolpath, this is either along the Z axis (for 3D operations) or along the tool axis (for 5D operations).

See also:

Tool trajectory parts

Approach and return rules

5.3.10.2 Engage

Engage defines the path the tool takes to arrive at the first point of the working toolpath. The engage movement always lies on an imaginary plane that contains the work pass. This means that for the waterline operation's moves, they are performed in the horizontal plane and for the plane operations in the vertical plane and for the drive operations on vertical mathematical cylinders defined by the appropriate curves etc.

The available engage methods depends on the selected operation typeoperation type. The engage method can be defined in theapproach and retraction modes windowapproach and retraction modes window.

See also:

Tool trajectory parts

5.3.10.3 Working toolpass

A working toolpath describes the tool movement when machining a model feature (surface or curve). For the plane operations, the working toolpath is a single X/Y pass, for the waterline operations – a single horizontal pass etc.

See also:

Tool trajectory parts

5.3.10.4 Stepover

Step-over describes the toolpath section between work passes. For the plane operations it will be an X/Y axis step-over, and for the waterline – Z-axis. A step-over depending on the selected machining typemachining type can be performed either in contact with the model, or via the safe plane, or by the distance needed for the approach and / or retraction moves. Step-over cannot contain more than one approach and retraction element.

See also:

Tool trajectory parts

5.3.10.5 Retraction

Retraction describes a sequence of tool transitions, which allows its smooth departure from the last point of a working toolpath. As with an approach move, a retraction always lies on an imaginary plane that contains the work pass. This means that for the waterline operation's moves, they are performed in the horizontal plane and for the plane operations in the vertical plane and for the drive operations on vertical mathematical cylinders defined by the appropriate curves etc.

The available retraction methods depend on the selected operation type . The retraction method can be defined in the approach and retraction modes windowapproach and retraction modes window.

See also:

Tool trajectory parts

5.3.10.6 Return

Return describes the rapid feed path the tool takes away from the last point of a retraction or a working toolpath. The retraction, just as with an approach, is usually performed in two steps:

1. Rapid tool movement from the last point of retraction along the Z axis up to the 'Safe plane' (for 3D operations) or along the tool axis up to the safe coordinate (for 5D operations).
2. Rapid tool movement to the tool change point.

See also:

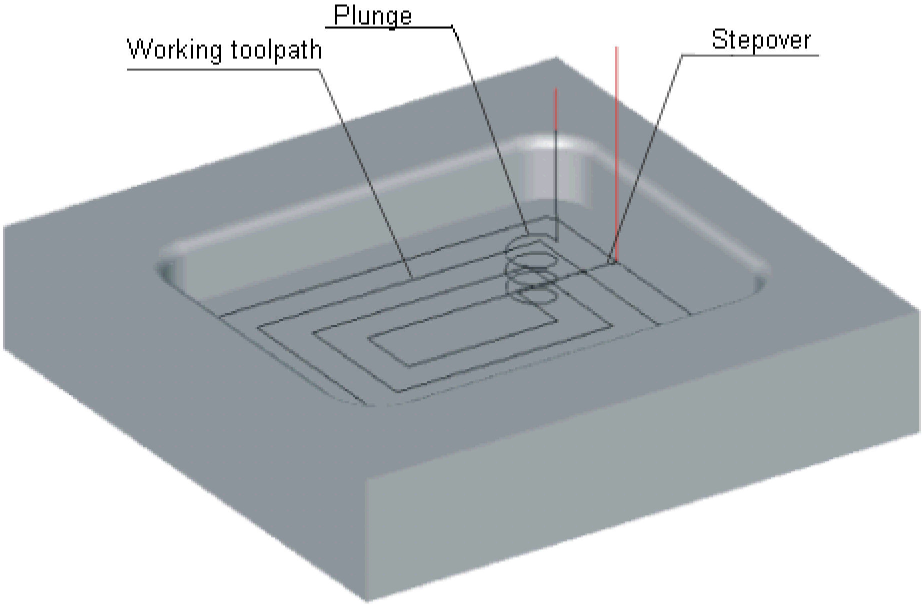
Tool trajectory parts

Approach and return rules

5.3.10.7 Plunge

A tool plunge normally describes a sequence of tool moves within the workpiece body, that allows lowering the tool from one machining level to another. Plunge moves are used in the waterline roughing and pocketing operations.

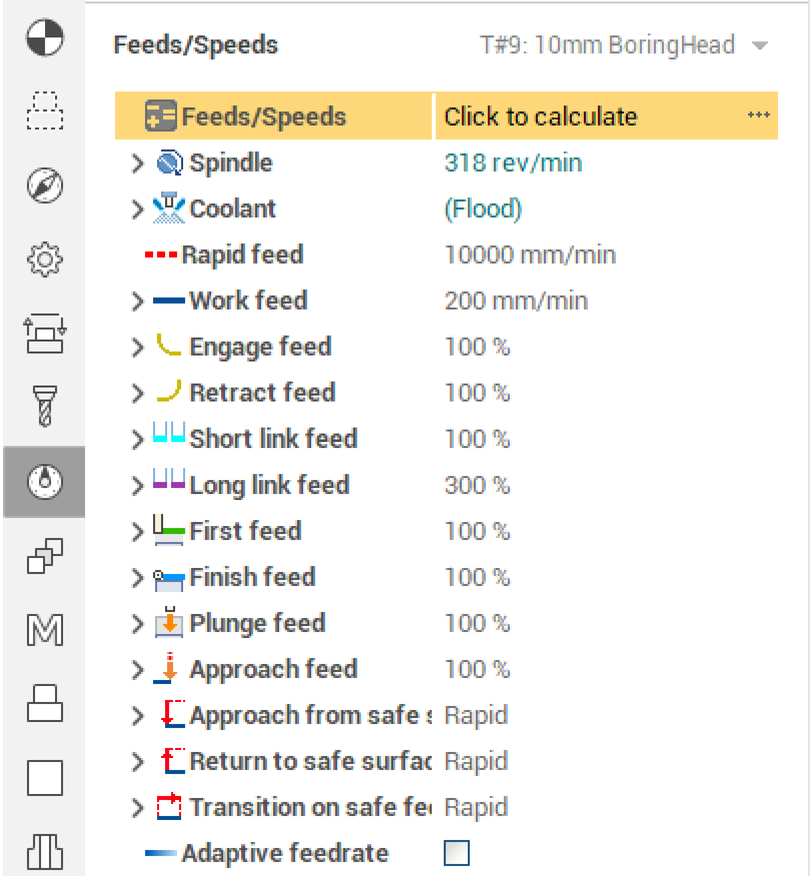
They are applied when a tool approach from outside the periphery of the workpiece is not possible. A plunge move can only be defined for operations that perform waterline stock removal (waterline roughing and pocketing operations), for all other operations there are no plunge methods available. The plunge method is defined on the Lead in / Lead out page.



See also:

Tool trajectory parts

5.3.11 Feed types



In machining operations the user can set feed values for every type of tool movement. The number and the set of definable feeds depend on the operation type. Feeds can only be defined for those types of moves that are available for the current operation. It is possible to define the following feed types:

- <Rapid feed> – feed at which all rapid tool transitions (positioning) are performed. The value is used for the calculation of the machining time and also during creation of NC code for controls that it is necessary to define the rapid feed value. When creating an NC program for machines that positioning speed depends on the drive speed, the rapid feed value is ignored;
- <Work feed> – feed at which the main work passes of an operation are performed. In roughing operations material removal will be performed, in finish – detail surface machining;
- <Approach feed> – feed, on which approach to the work pass beginning is performed;
- <Retract feed> – feed, on which retraction after completion of a work pass is performed;
- <Plunge feed> – on this feed, plunging to a lower machining layer in waterline roughing and pocketing operations is performed;
- <Feed to next> – feed, on which step-over along the being machined surface to the next work pass is performed;
- <Retrace feed> – feed, on which return to the previous work pass, along the toolpath of the earlier completed step-over is performed;
- <Finish pass feed> – feed, on which work passes along the detail surface in rough operations are performed. It is advised to use when need to obtain a surface of high quality after a roughing operation;
- <First pass feed> – feed of the first from the workpiece surface machining stroke in the rough operations. It is recommended to assign, for example, with different machinability of surface and workpiece core;
- <**Transition on safe surface**> – feed for a toolpath that transitions on a safe surface;

- <**Return to safe surface**> – feed for a toolpath that return to safe surface;
- <**Approach from safe surface**> – feed for a toolpath that approach from safe surface;
- <**Long transition**> – feed for long transitions between passes;

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.

If in the used SprutCAM X configuration there is the cutting modes calculation module, then feed value can be calculated automatically, regarding the workpiece material, tool and operation parameters. When using the calculated feed, the real feed values when moving up, horizontally and down will be calculated by multiplying the obtained feed value onto the corresponding coefficients.

Rapid feed can be assigned by the permanent value only. The work feed can be either permanent or variable; its real value can be assigned manually or calculated automatically by the cutting modes calculation module. All other feeds are assigned either analogously, or in percents from the work feed. When assigning in percents from the work feed, the feed type will be set just as for the work feed, and numeric values will compound the defined percent from the corresponding values of the work feed. For example, upon setting the approach feed equal to 50% from the work feed, the approach will be performed at half the feed speed of the main work passes.

See also:

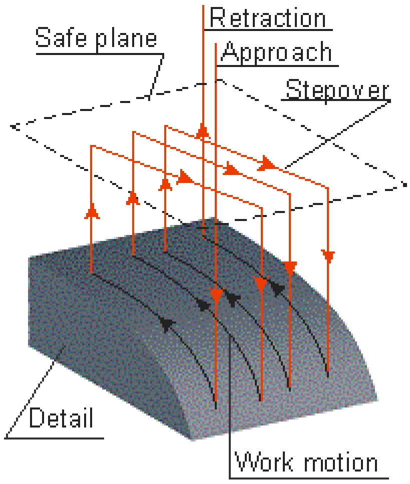
Approach type

5.3.12 Safe plane

Safe plane is a horizontal plane, located on such level, so that any tool transitions above that plane would not lead to tool collision with the detail being machined or any machining equipment. All horizontal transitions on the rapid feed are performed in safe plane.

The safe plane level should be set to be higher than the top machining level and than the workpiece or machining equipment.

The safe plane level can be assigned in the <Parameters> page.



See also:

Upper and Lower machining levels

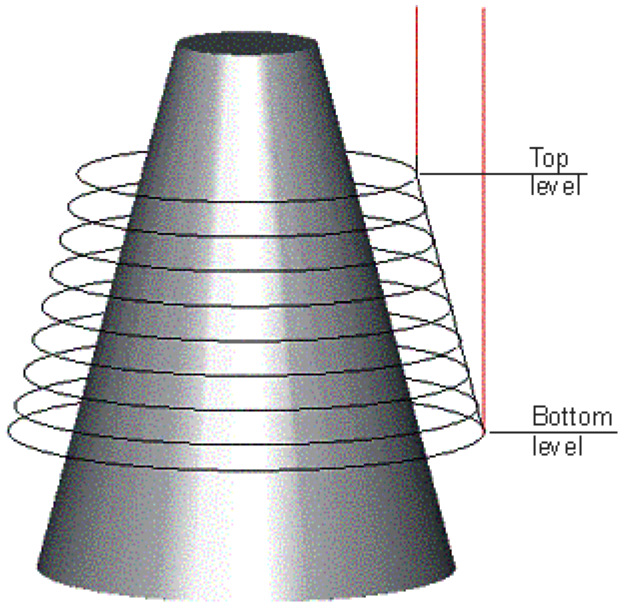
5.3.13 Top and bottom machining levels

The top and bottom machining levels define the machining range along the Z axis. Only those areas of detail surface will be machined, which lie between the top and bottom levels.

If the workpiece or the restricting model are assigned by areas, which lie in the basis of a prism, then it is considered that this rule extend to all machining levels, i.e. between the bottom and top machining levels.

Off course, the top machining level cannot be lower than the bottom one.

The top and bottom machining levels can be assigned in the <Parameters> page.



See also:

Safe plane

Definition machining strategy

5.3.14 Tolerance

Machining tolerance is assigned in the system by the maximum deviations of the approximated tool movement toolpath from the ideal one.

< Outer deviation > defines the maximum allowed tool deviation away from the surface of the detail being machined (inwards to the tool).

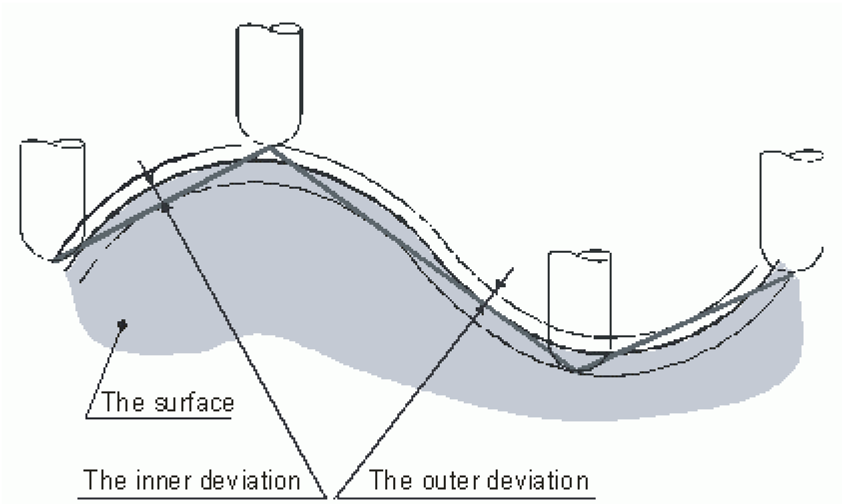
< Inner deviation > defines the maximum allowed cutting of the tool into the detail being machined (outwards from the tool).

For deviation from the detail, the positive direction is outwards from the detail surface (inwards to the tool), and negative for deviation into the detail (outwards from the tool). Thus, the machining tolerance is equal to the sum of deviations outside and inside the detail. One should note that increasing the tolerance (decreasing the sum of deviations) would increase the calculation time and the size of the NC program. And vice versa, the higher the sum of the deviations inside and outside the detail, the more rough will be the toolpath. The sum of the deviations must be more than zero, otherwise it will be impossible to construct an approximated toolpath.

In most cases it is more convenient to assign deviations inside the detail equal to zero, and outside the detail equal to the desired machining tolerance. With these parameters, the minimal thickness of the remaining material layer will be equal to the defined stockstock. If deviation inside the detail is not zero, then the thickness of the remaining material layer will be less than the defined stock equal to the deviation inside the detail value.

Note: It is recommended to enter positive deviation and stock values.

Maximum deviations can be defined in the < Parameters > < Parameters > page.



See also:

Stock

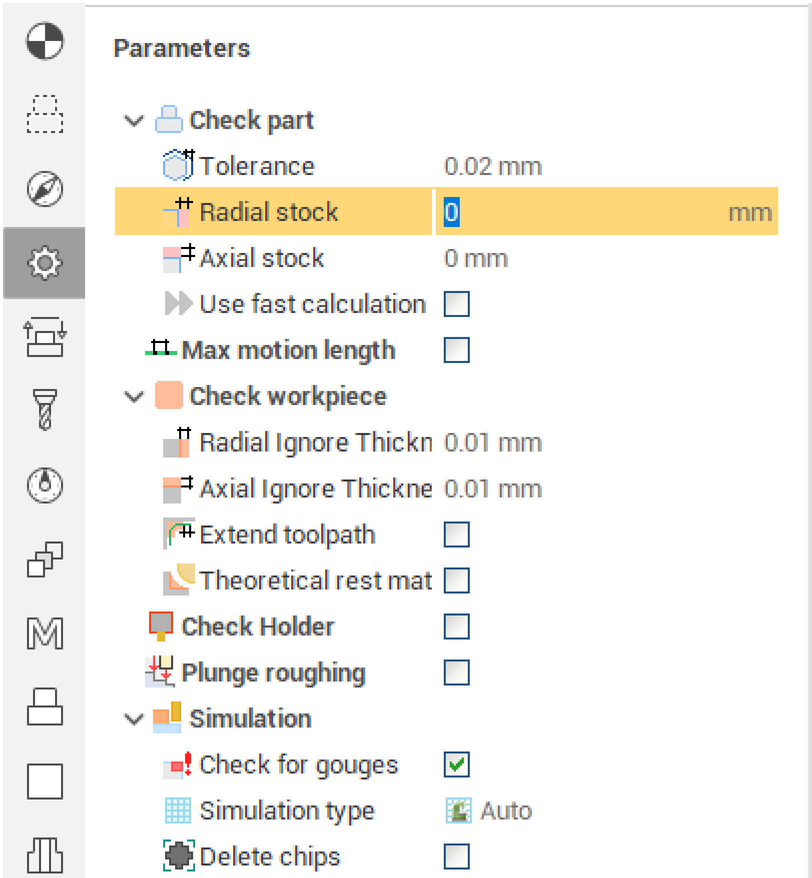
5.3.15 Stock

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.



See also:

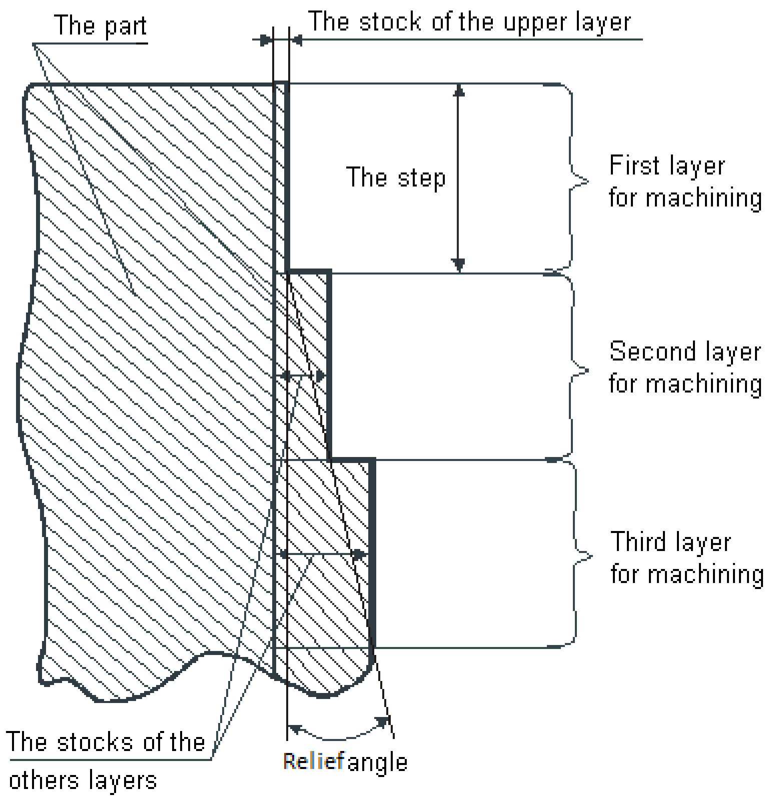
Definition operation parameters

5.3.16 Relief angle

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.

The relief angle can be assigned on the <Parameters> page.



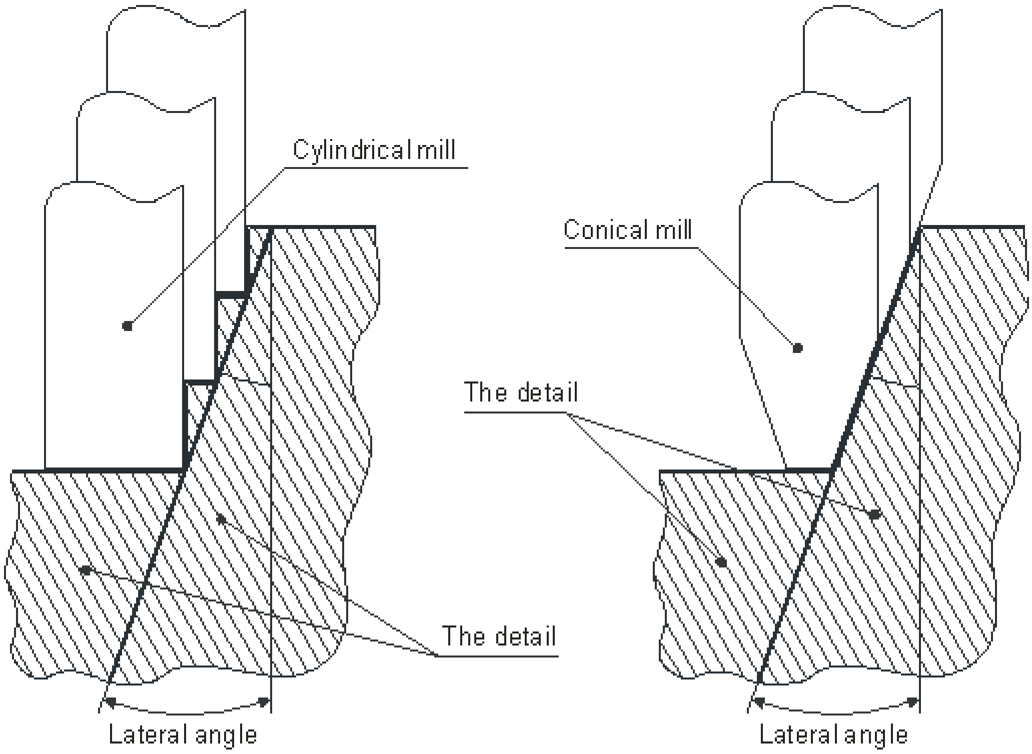
See also:

Lateral angle

Definition machining strategy

5.3.17 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.

See also:

Relief angle

5.3.18 Machining step

The machining step defines the distance between two neighboring work passes of the tool. Depending on the operation type the step can be assigned in vertical and/or in horizontal direction. In the roughing operations, normally, the user can assign both the vertical and the horizontal steps; in finish operations – the step for the vertical direction, in plane and drive – in the horizontal. The step value defines the height of the remaining material scallop between two neighboring passes.

For more convenience, the step value can be assigned by several methods.

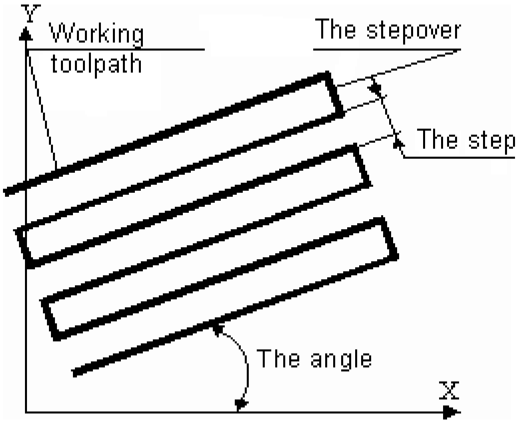
- **By the real step value.** The value is assigned by the absolute value and does not change upon correction of other parameters;
- **In percents from the mill diameter.** The real step value makes the defined percent from the tool diameter and accordingly alters upon changing the tool;
- **By the number of passes.** The whole machining range is divided into the defined number of equal parts. The real step value will change upon alteration of the machining range. This means that the step value in the vertical direction will alter when the top and/or the bottom machining levels are changed;
- By scallop**.** The step between neighboring tool passes is not permanent, it depends on the geometrical parameters of the tool and the shape of the model surface of the being machined.

The step value is selected so that the height of the scallop of the remaining material between neighboring work passes does not exceed that defined. When choosing the step by the height of the scallop in the vertical direction it is also necessary to additionally assign the maximum step, and in the horizontal – the minimal. In case if the calculated step value exceeds the defined limits, then its maximum value will be used;

All the above methods arrive at the same results. Upon changing the value for calculation of the real step, the system will automatically recalculate all values for definition of the same step by other methods. It allows, for example, estimating the real step value when assigning it as a percentage of the tool diameter.

Irrespective of the assignment method, the real step value must be more than zero.

The step in the vertical direction can be assigned in the <Parameters> page, and in the horizontal – on the <Strategy> page.

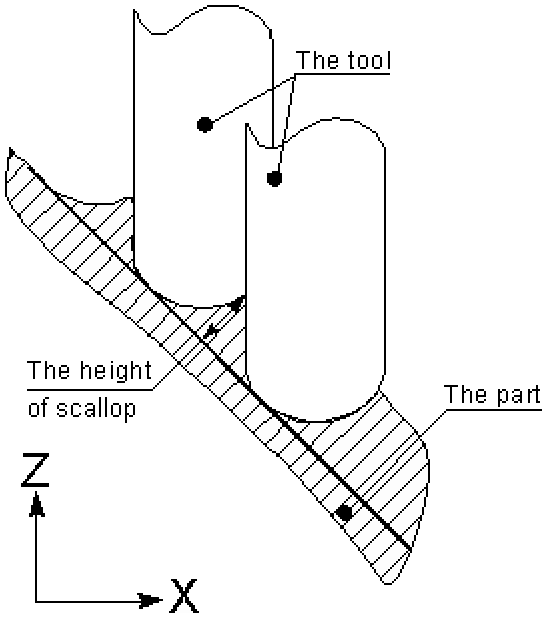


See also:

Definition operation parameters

5.3.19 Selection step by scallop height

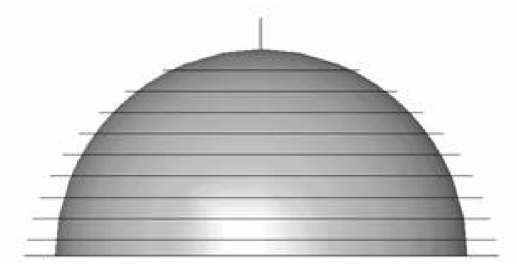
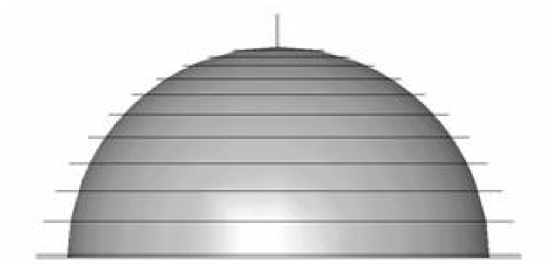
When machining surfaces, whose radius of curvature does not coincide with the profile radius of the tool, then there appears a scallop of residual material between the neighboring passes. Its height depends on the surface shape of the model being machined, the type and size of the tool and the distance between the neighboring passes of the mill.

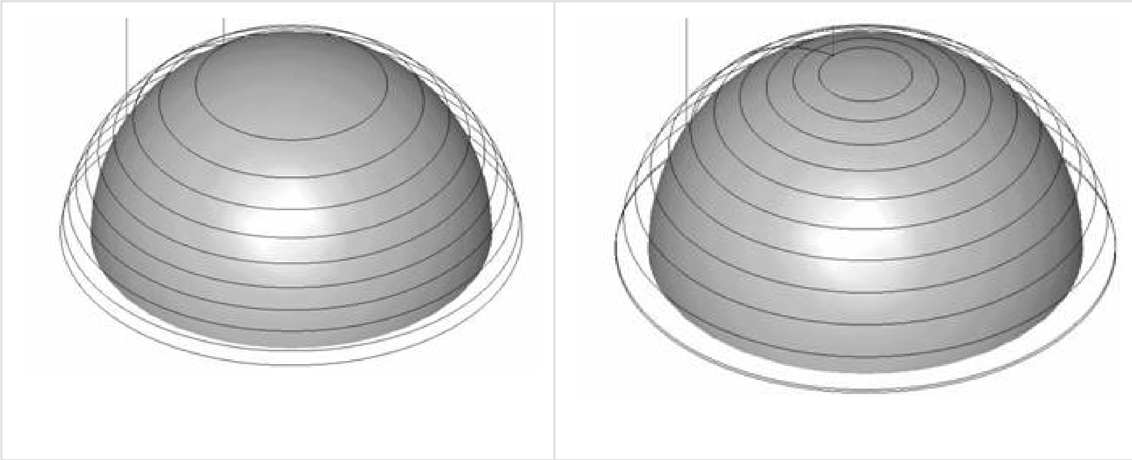


In operations it is possible to assign the step by the maximum height of the scallop. With this, the distance between the neighboring tool passes will be automatically selected so that the height of the scallop does not exceed the defined.

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.

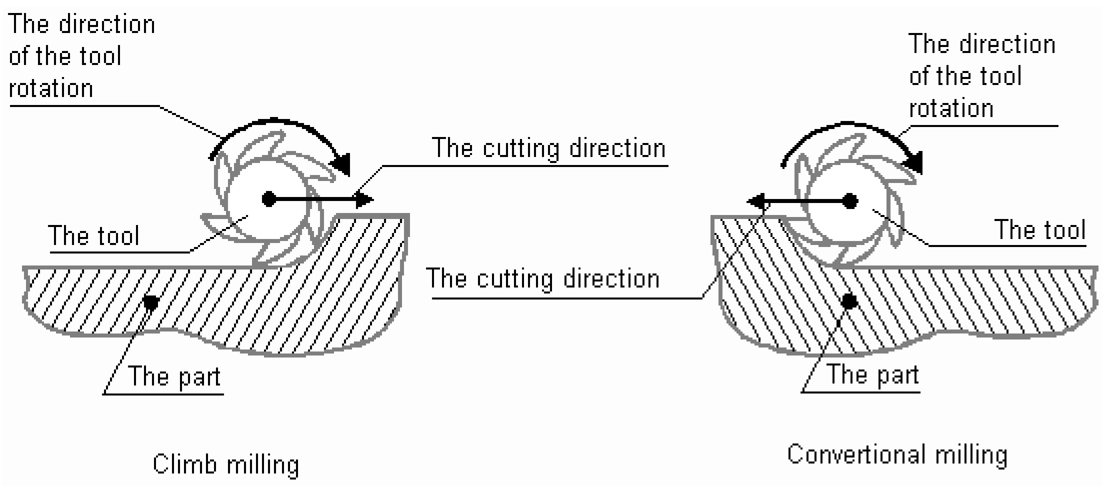
See also:

Definition machining strategy

5.3.20 Milling types

In the system it is possible to generate tool movement trajectories of climb or conventional milling.

If the milling type is not important, then it is advised to set the mode without considering the milling type. This mode allows considerable reduction of non-cut passes and consequently decreases the machining time.



Milling type can be defined in the <Strategy> page.

See also:

Definition machining strategy

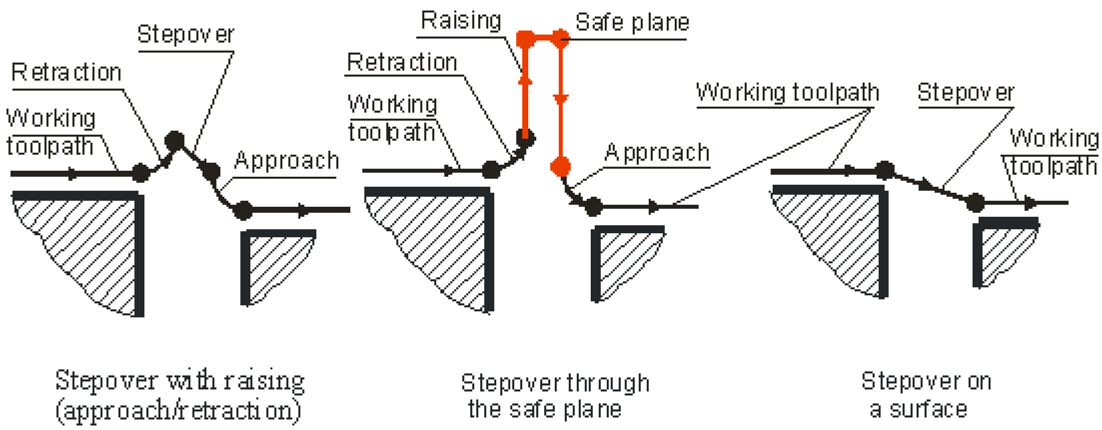
5.3.21 Stepover method

<Tool stepover> is a tool movement between the contours being machined. One should not mistake it with the machining term <Stepover>.

If the model is machined using one tool, and the machining toolpath represents by itself a tool transition along several contours, then it is necessary to create machining conditions for the transition from one contour to another.

In volume machining operations the step-over from one work pass to another are performed by the following methods:

1. <On surface>. Tool stepover is performed without retracting from the model being machined. With the small distance between the end points and beginning of the neighboring work passes, shorter machining times are achieved;
2. <Retract-approach>. At the end point of the work pass there will be a retraction using the defined method, then a stepover at the work feed to the first point of the next approach, then the approach according to the defined method to the first point of the next work pass. Such a stepover takes more time, but the stepover is performed without touching the machined surface;
3. <Via safe plane>. In the end point of a work pass there will be a retraction using the defined method. Then the tool will rapid to the safe plane. Stepover at rapid feed at the safe plane. Then, tool lowering and approach according to the defined approach method;



Because the step-over methods <On surface> and with <Retract-approach> are the optimal only with the short length stepover, if it is necessary to perform a stepover of a longer length, then regardless of the defined stepover type, the system will automatically generate a stepover via the safe plane.

In the curve machining operations, SprutCAM X gives the user the following step-over methods:

1. <Via safe plane>
2. <**Tool step-over around the workpiece and at the defined Z height**>
3. <By Z>
4. <Around workpiece>

Via safe plane

This method is the most frequently used one; but it is not the optimum relative to the machining complexity. The safe level can be assigned in the <Operation parameters> window on the <Parameters> page. Stepover is normally performed at the rapid feed (G00).

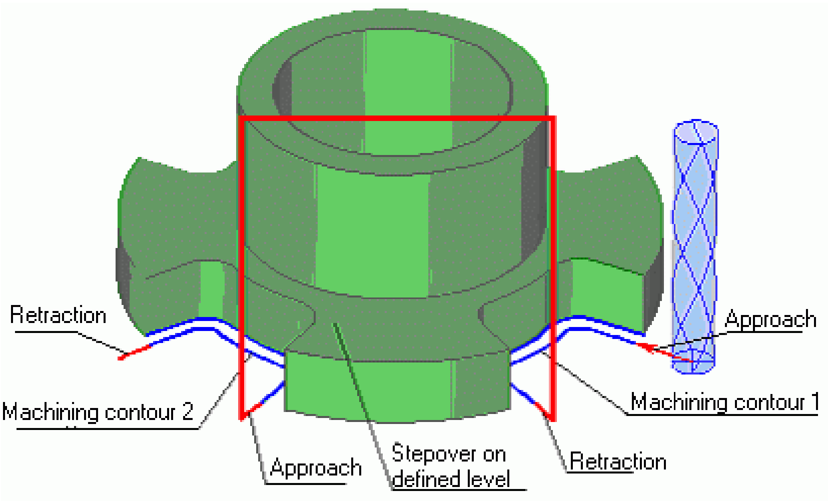
Tool step-over around the workpiece and at the defined Z height

If on the machining <Strategy> page, the user does not activate the Idling minimization mode, then the contour machining order will be defined by the order in which the contours are located in the list of the <Model> page. To define an optimized sequence of contour machining using the minimum length of idle moves, the user should select Idling minimization mode on the machining <Strategy> page.

SprutCAM X in the 2D machining mode allows the user to work either using or not using a workpiece. If a workpiece is not used, then the stepovers between work contours will be performed either by the safe plane, or at the defined Z level.

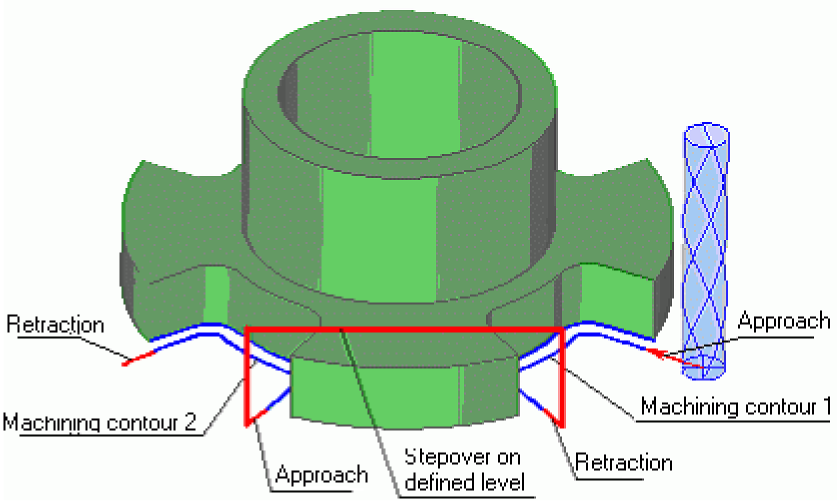
Using a workpiece will provide a safer work mode because the system will automatically create a step-over toolpath with control of tool collision with the workpiece, also in this case, SprutCAM X will give the user more possibilities to optimize the machining toolpath.

The workpiece can be defined on the <Model> page. For a complete description on how to create a workpiece, refer Workpiece.



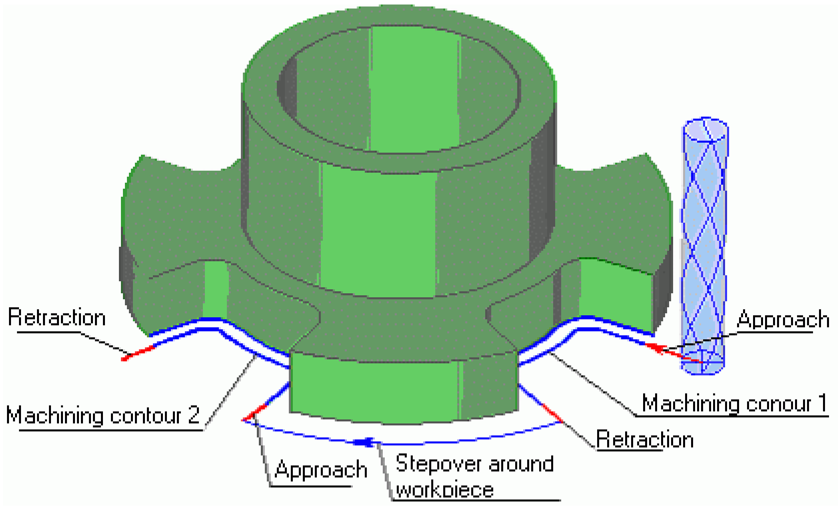
By Z

The plane level for transitions can be assigned on the machining <Strategy> page relative to the zero of the current coordinate system.



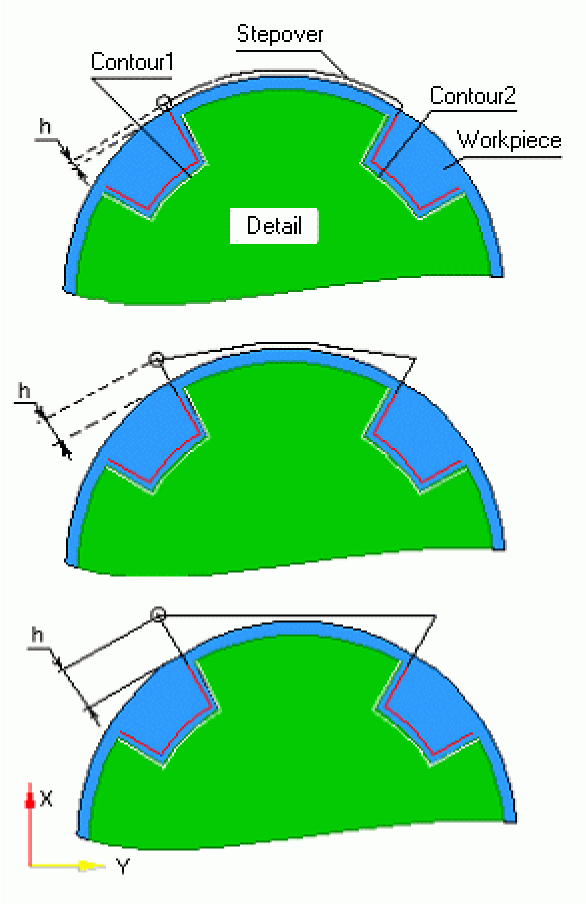
Around workpiece

The toolpath will be created using the shortest route around the workpiece profile on the plane of the work contours location. To assign the feed, in the <Feedrate> page the user should choose the Feed to next mode and in the feed value assignment window define the required value. The workpiece must be defined.



The step-over toolpath depends on the workpiece shape and will be formed as a curve around the workpiece area that is found. The shape of the step-over curve is also affected by the distance from the workpiece to retraction and approach points to contours.

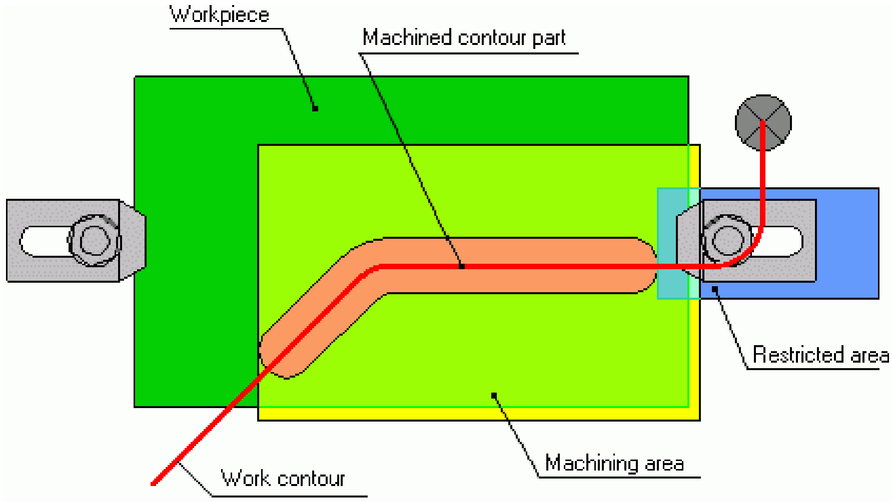
An example of toolpath alteration, depending on the value of that figure (h) is shown in these pictures. The toolpath can change its look from a straight line to the curve that repeats the shape of the workpiece as shown on the picture. Concave areas of the workpiece pass along the shortest curve, convex – along the rounding one.



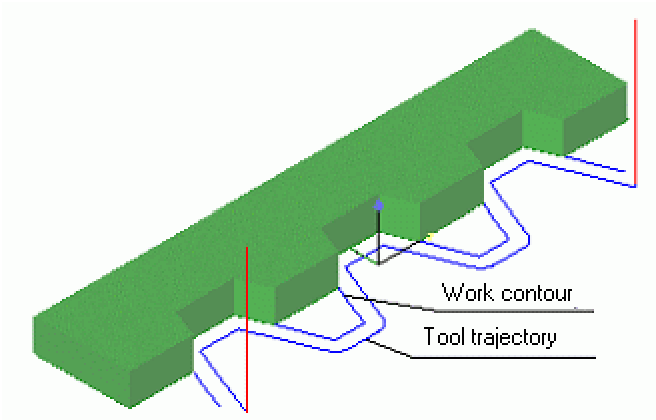
The step-over curve is constructed touching the edge of the tool against the workpiece, therefore in practice, when assigning a workpiece; the user should add an additional stock to the workpiece on the <Model> page.

In order to obtain the desired step-over curve the user should use different values of h for the approach and retraction points, and also alter the workpiece profile.

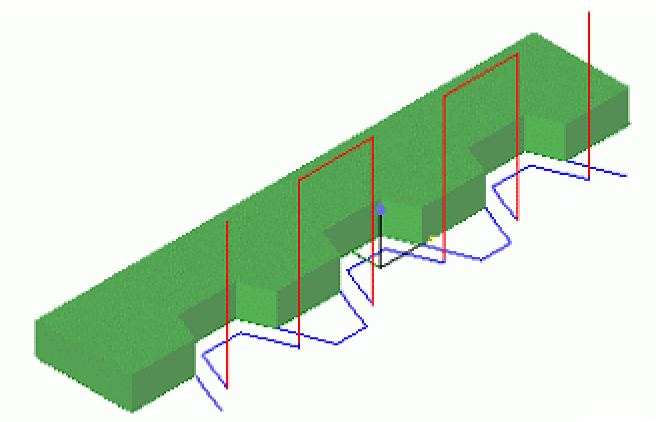
One should pay special attention to the case where the contour has areas that go beyond the workpiece area. In SprutCAM X there is a rule – if a workpiece has been defined, the machining toolpath will be formed within the workpiece area only. If machining and restricted areas have also been defined, then the toolpath will be formed within the workpiece area, inside the available machining areas and outside the restricted areas (see pic.).



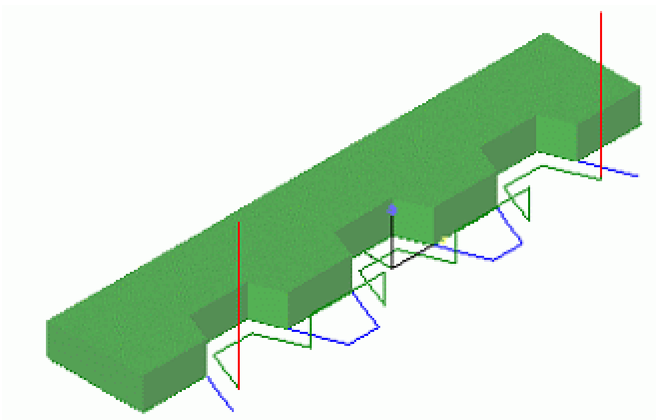
In the picture the toolpath is formed regardless of a workpiece, using the activation and deactivation area of compensation by tangent. As can be seen in the picture, the toolpath is formed equidistant to the work contour:



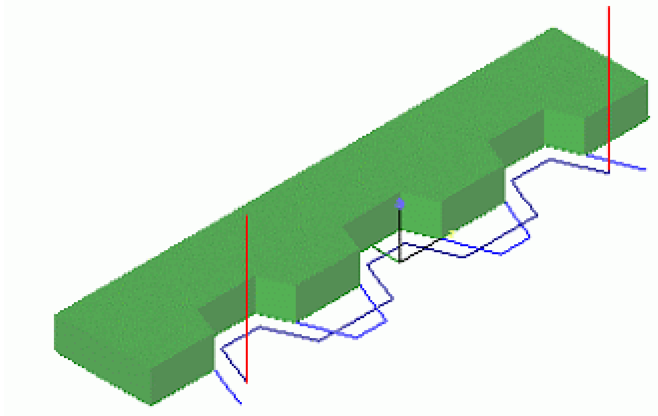
With a workpiece (bar) defined. The toolpath has been formed regarding the workpiece, i.e. within the workpiece area. Outside of the workpiece, machining will not be performed. To every area the system has automatically added the compensation activation and deactivation areas. Tool transition from area to area is performed at the safe plane:



This operation has the same parameters as the previous one; the difference is in step-overs. This operation uses step-over at the defined level:



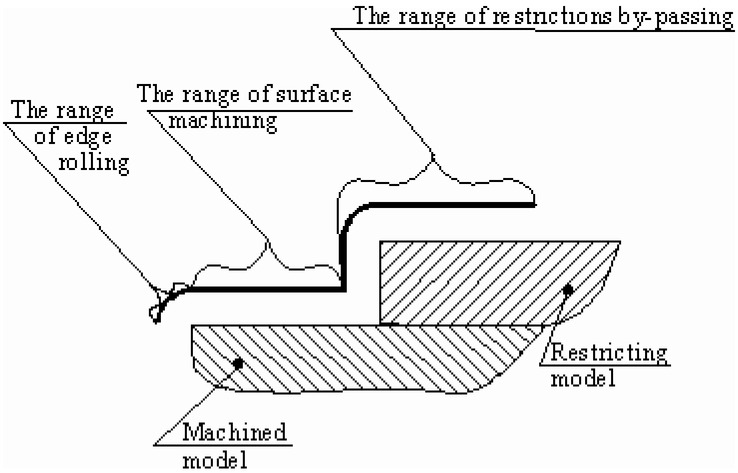
In this operation, the system uses step-over round the workpiece. One should note that the compensation activation and deactivation blocks will automatically be added to the corresponding areas of the work contour:



5.3.22 Roll type

In the finishing operations, when machining a detail surface it is possible to mark out the following toolpath sections:

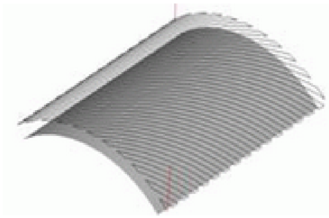
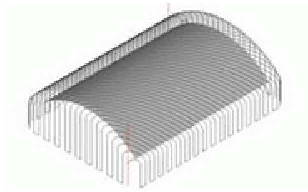
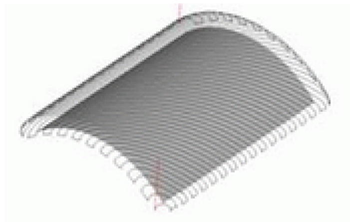
- machining area of form-creating surfaces;
- edges rounding area between these surfaces;
- bypass area of restricting model.



Quite often (in machining with zero stock for instance) the edges between neighboring surfaces are formed when machining the surfaces themselves, and do not require additional machining. In this case, it is enough to include only the machining areas of the form-creating surfaces. Due to exclusion of unnecessary areas, the length of the resulting toolpath decreases and consequently, decreases the machining time on a milling unit. Defining the roll type as surfaces only activates this tool movement toolpath creation method.

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.

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. Besides this, in the plane and drive operations all the vertical toolpath areas will be added.

		
Surfaces only	**With edges**	**With restricting model**

The roll type can be defined in the <Strategy> page.

See also:

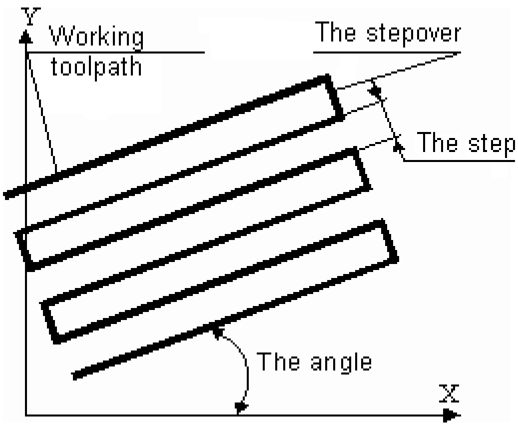
Definition machining strategy

5.3.23 Work pass angle in plane operations

In the plane operations, tool work passes lie in the parallel vertical planes. The orientation of these planes in space is defined by the angle of work passes. The angle is defined in degrees and counted along the X axis in the horizontal plane counter-clockwise.

The angle value also affects the order, in which work passes will be joined. For example, with the work passes angle equal to 90 degrees during joining, the number one priority will be the queue of passes towards increment of the X axis, and with 270 degrees – its decrease.

The angle of work passes for plane operations can be assigned in the <Strategy> page.



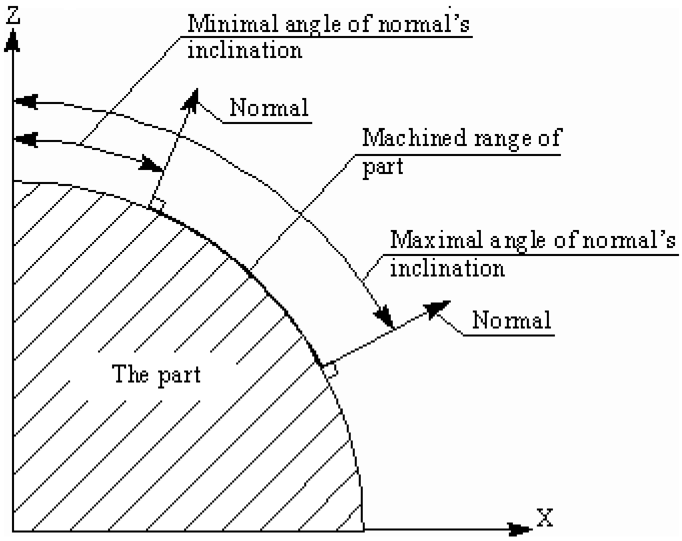
See also:

Definition machining strategy

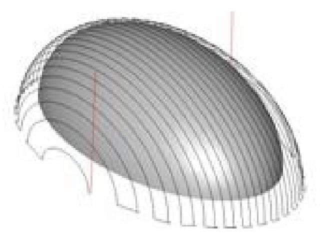
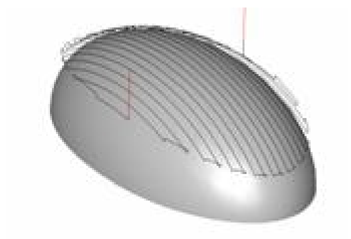
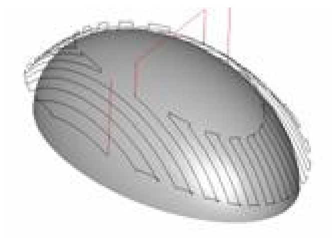
5.3.24 Maximum slope angle of normal

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, as is shown in the picture (below).

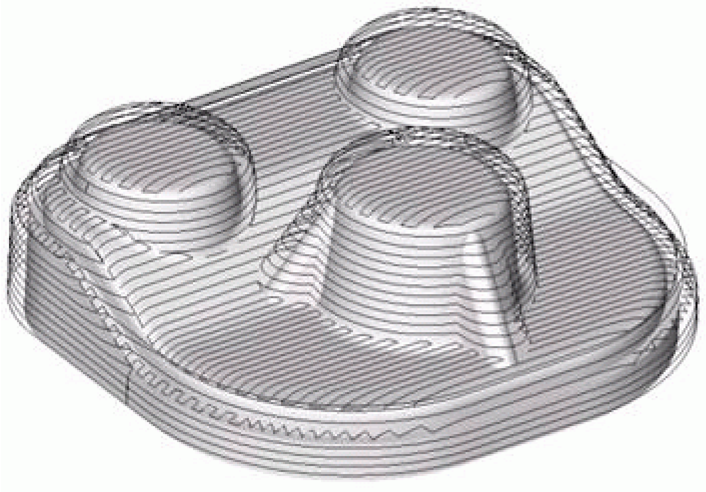
The maximum values of the normal slope angle can be assigned from 0 degrees (horizontal area, normal is vertical) to 90 degrees (vertical area, normal is horizontal).



It is understood that the plane machining method is optimal when milling surfaces that are closer to the horizontal, and the waterline machining gives best results when machining surfaces that are closer to the vertical. Use of the maximum slope angle of the normal allows the user to machine the horizontal surface areas by plane method and the vertical – by waterline.

		
Without limitations	**Limitation from 0 to 45**	**Limitation from 30 to 60**

An example of toolpath machining, obtained by joint use of the plane and waterline-finishing operations is shown on the picture. The plane operation machines surface areas that have a slope within the range of 0 to 45 degrees, and the waterline with slope within the range of 45 to 90 degrees. These parameters are set as default in the complex finishing operation.



The maximum slope angle of the normal can also be used for milling horizontal areas using a cylindrical tool or for machining surface areas with a slope angle equal to the side angle of the tool e.g. conical mill.

The maximum angle of the normal for the finishing operations can be assigned on the <Strategy> page.

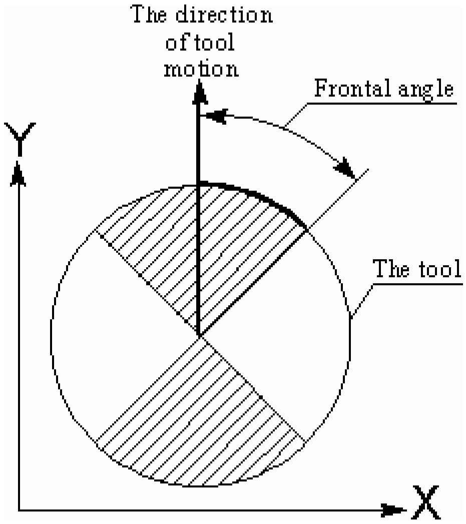
See also:

Definition machining strategy

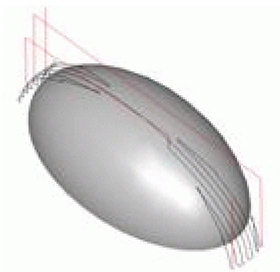
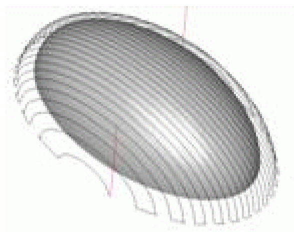
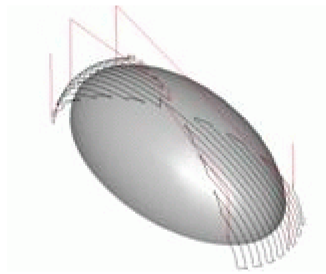
5.3.25 Frontal angle

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.

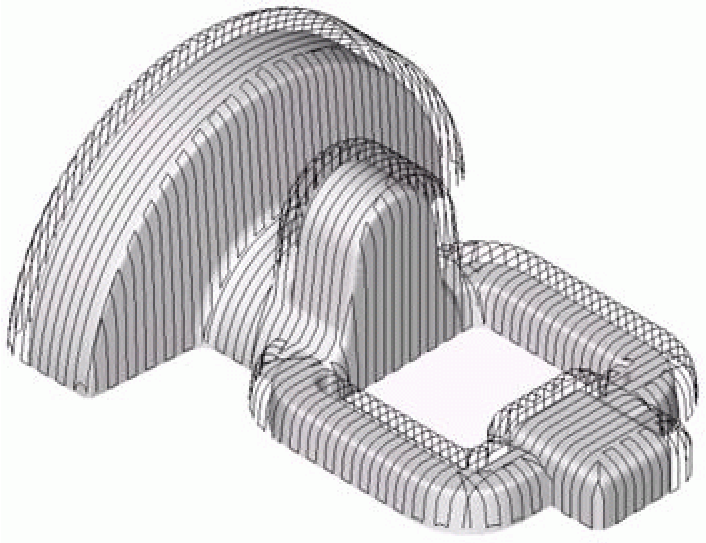
In order to obtain the optimal toolpath in plane finishing and drive operations, it is possible to define a limitation of the frontal angle. The frontal angle is the angle between projections onto the horizontal plane of the tool movement direction and the normal to the surface at the cutting point.



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.

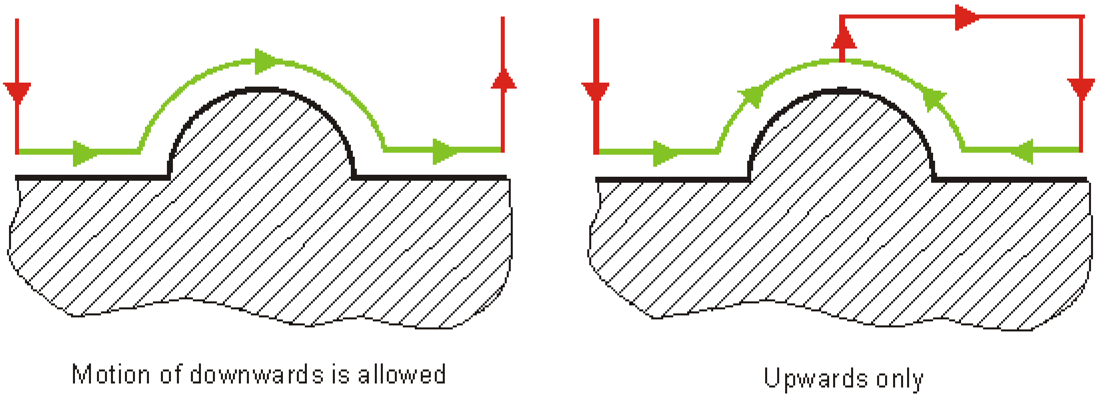
See also:

Slope angle of a surface

List of types of machining operations

5.3.26 Machining upwards only

When using plane finishing or drive operations and downward movement of the tool is not desired, it is recommended to use the machining upwards only mode.



If this mode is active, then upon reaching the top point of the work pass, the tool moves to the beginning of the next pass via the safe plane, from where again it moves upwards only.

By default the mode is off.

The machining upwards only mode for the plane finishing and drive operations can be defined in the <Strategy> page.

See also:

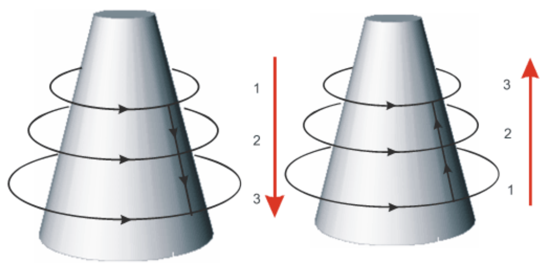
Definition machining strategy

5.3.27 Machining direction

In the waterline finishing and combined operations it is possible to define the priority direction of work passes joining in the resulting toolpath. The direction of machining either can be downwards or upwards.

For models, which have surface areas close to the vertical, machining downwards is recommended. Machining upwards is advised for use on models with form-creating surfaces which are closer to horizontal.

Machining direction can be assigned in the <Strategy> page.



See also:
Machining upwards only

Defining the machining strategy of mill operations

5.3.28 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:

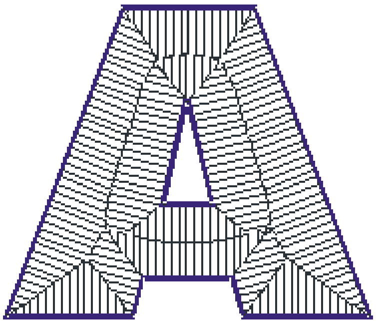
- <On curve> – performs only one pass along the area borders. That is, performs machining of the detail so that the tool axis is always on one of the curves, which limit the drive area.
- <Along curve> – the first pass is performed along the area borders, and all following passes – equidistant in the 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.

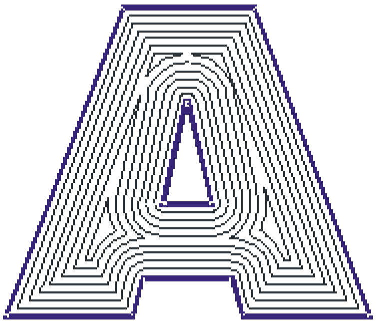
The number equidistant passes when machining along curve, and the length pf passes when machining across curves, in general is limited by the form and the dimensions of the drive area. Work passes are constructed until the entire model, which is inside the drive area, has been machined.

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.

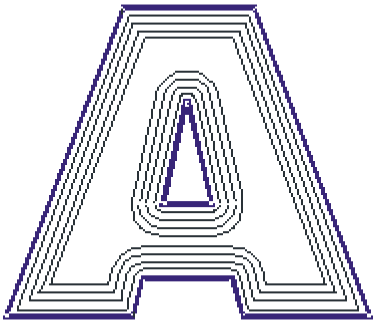


Across Curve

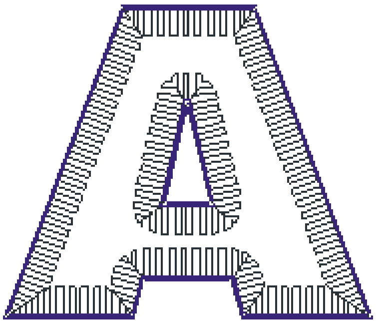


Along Curve

On Curve



Along Curve with using width



Across Curve with using width

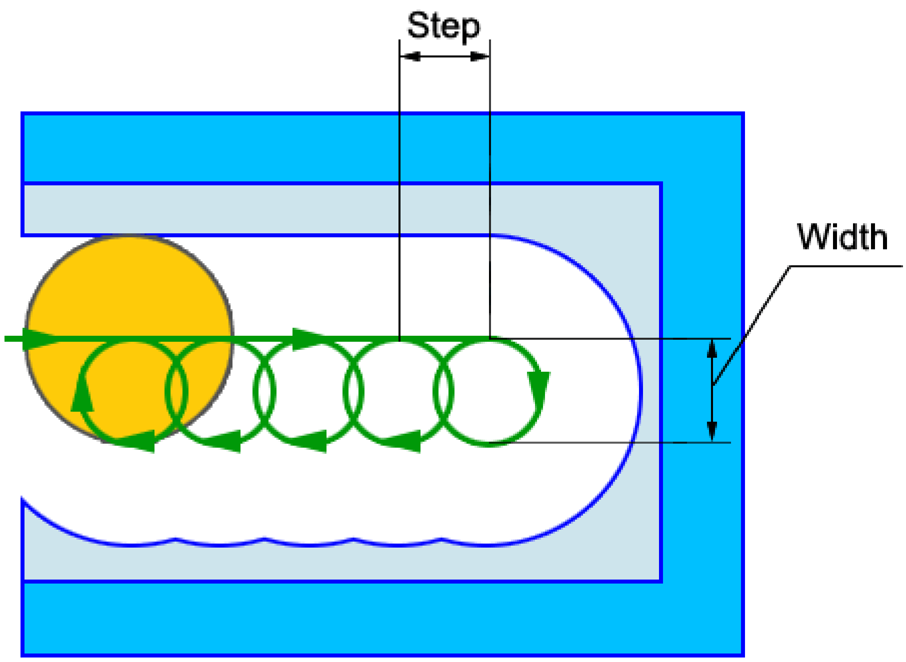
The machining method for the drive operations can be assigned in the <Strategy> page.

See also:

Definition machining strategy

5.3.29 Trohoidal machining

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. SprutCAM X 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;

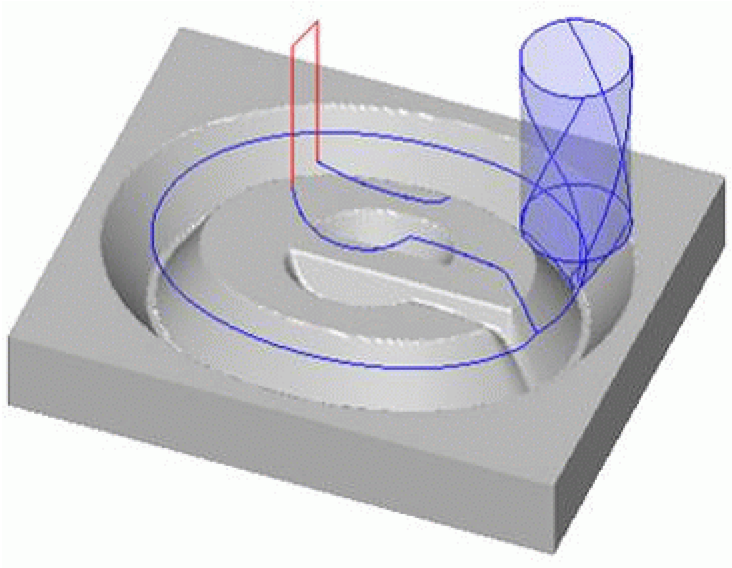
Attention: If machining step great then half of tool diameter or tool have corner radius, then can be rest of material islands. You must visually check this machining at <**Simulation**> mode.

See also:

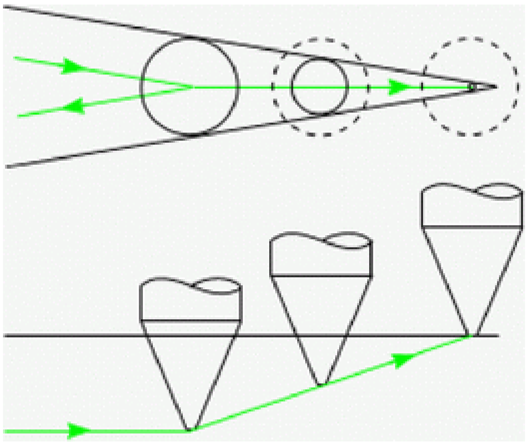
Definition machining strategy

5.3.30 Three-dimensional toolpath

3D machining of corners is designed for removal of residual material in corners, and also in other areas, where it is impossible to perform by using the tool with the defined geometrical parameters on the current machining level. If the tool has a profiled part, with a diameter gradually decreasing to the end point, then upon increasing the Z coordinate of the tool, machining of more "narrow" areas will be possible.



The tool toolpath during 3D corner machining represents by itself a curve, along which the tool touches the defining contour of the model at several points simultaneously. The Z coordinate of the tool is defined regarding the tool geometry and the distance to the model contour at every point.



Using the 3D corner machining, the user can create sharp inner corners on the model being machined and machine smaller width areas by a single three-dimensional pass of the tool.

The 3D corner machining function for the engraving and pocketing operations can be activated on the <Strategy> page.

See also:

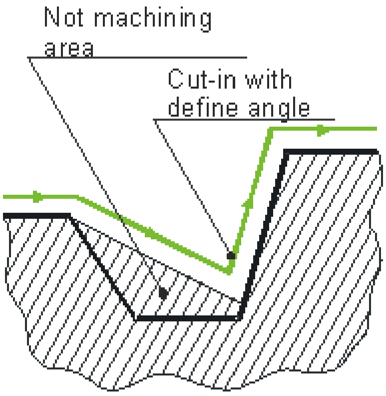
List of types of machining operations

5.3.31 Descent types in plane roughing operations

In the plane roughing and drive operations, it is possible to limit a tools downward movement. The limitations may be based on the peculiarities of the cutting tool or due to hard machining of the workpiece that can only cut into material at a limited angle.

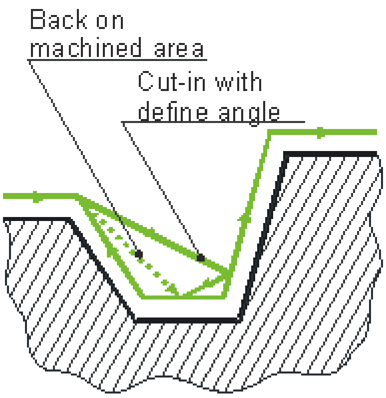
One of the following descent types can be used:

- <Machining strictly upwards>. Tool movement down inside the material being machined is absolutely restricted. The material, left in hollows will not be machined. Such a method is advised for use on convex models. A model that has hollows or pockets will require subsequent rest milling.



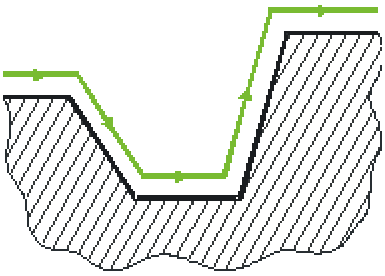
Control cut-in angle without re-machining shadow areas

- <With the defined maximum cutting angle> – without machining of shadowed areas. If necessary, the tool may move down, but within the defined angle. The material, which is left in hollows, will not be machined. Consequently, details with abrupt walls in hollows will require rest milling. The maximum cutting angle must be set within 0 (similar to machining strictly downwards) and 90 degrees (like without control).



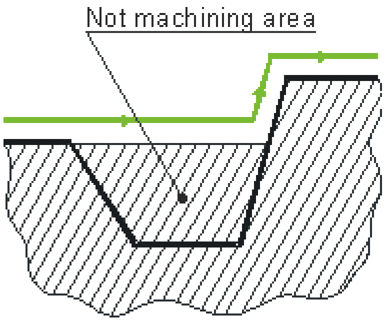
Control cut-in angle with re-machining shadow areas

- <With the "defined maximum cutting angle with machining of shadow areas"> the tool, as in the previous case, can cut into the workpiece material within the defined angle, but the material remaining in hollows will subsequently be removed. Machining is performed using reciprocal moves with simultaneous plunging at the defined angle. The maximum descent angle must be more than zero.



Allow down motion

- <Without descent control>. The tool movement direction is not controlled. The machined detail may contain unmachined areas because of the tool geometry.



Machining upwards only

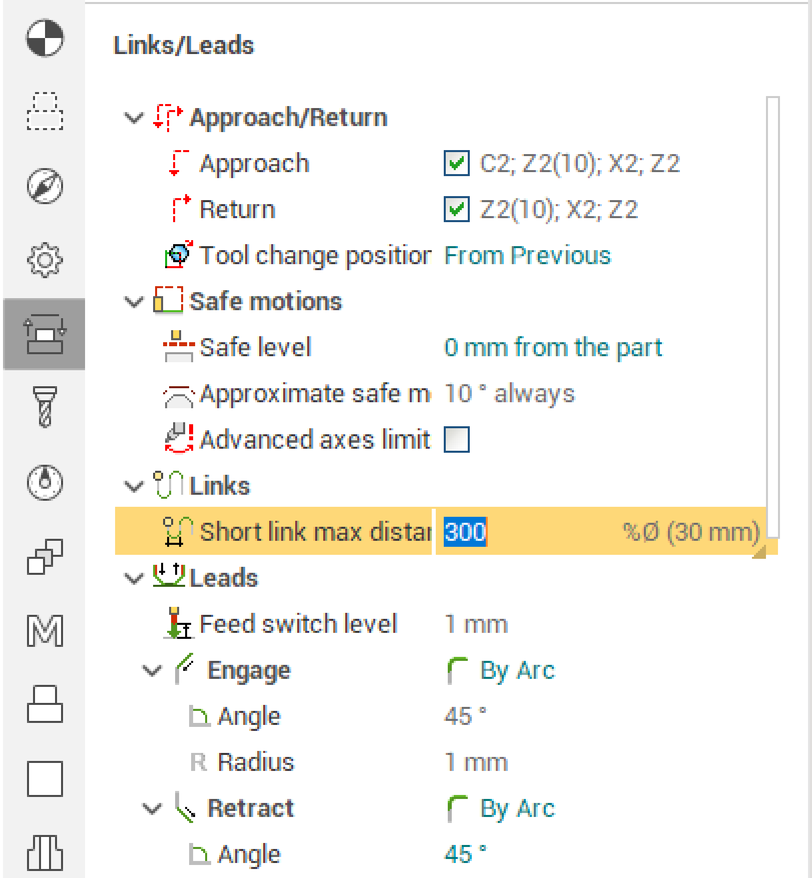
The descent type for the plane roughing and drive operations can be assigned in the <Strategy> page.

See also:

Definition machining strategy

Relief angle

5.3.32 Short link



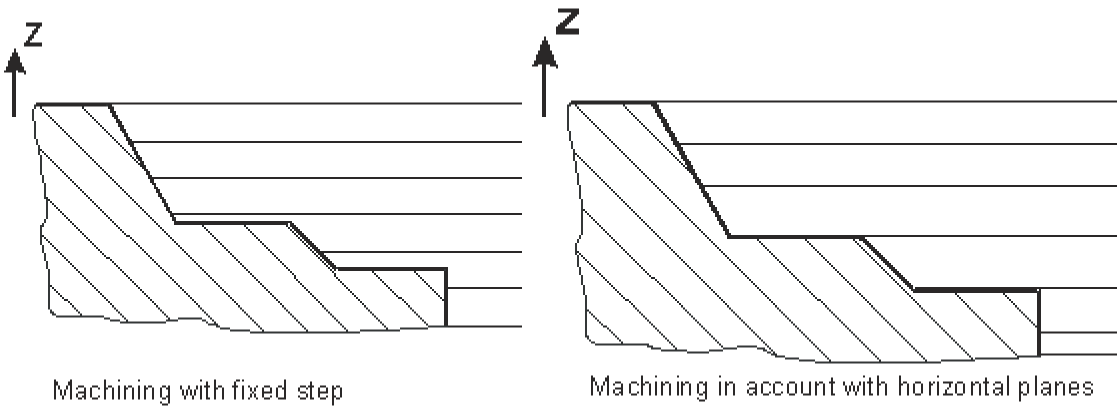
Short link parameter defines the work path link method. If the link from one work path to the next work path is more than this value then the link is performed via the safe plane. Else the link is performed on the surface.

See also:

Definition machining strategy

5.3.33 Machining horizontal planes (Clear flats)

Upon activation of this function, there will be additional passes of the tool on those areas, where there are horizontal planes. This allows the user to exclude additional machining of these surfaces. The function is available in the waterline roughing, waterline finishing and combined operations.

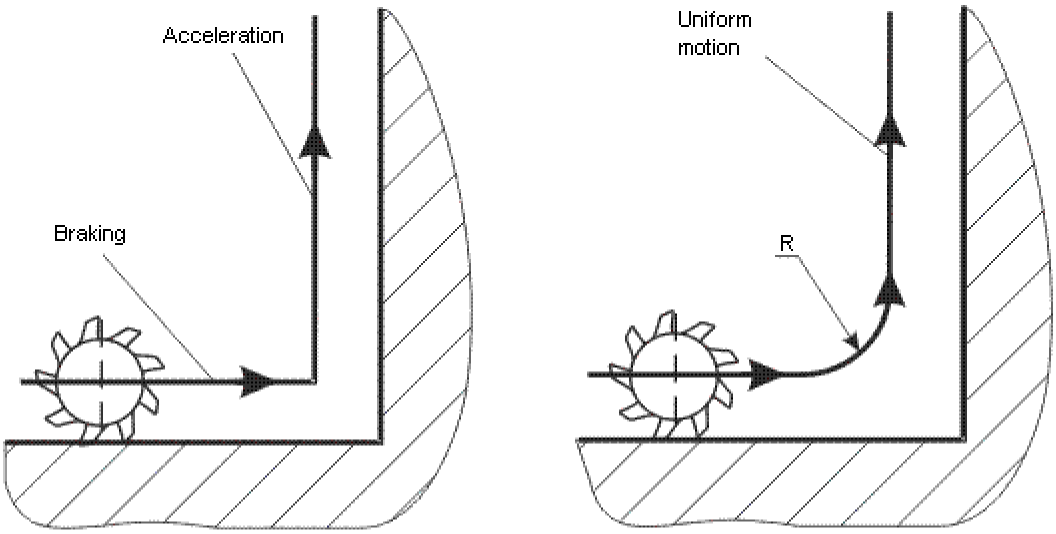


In the waterline machining, the tool passage levels are defined by the step value, which assigns the thickness of the material being removed. Upon activating the clear flats function, the step becomes variable. If on the layer being machined there are horizontal planes, then the value of the step being executed will be corrected so that the tool will pass along the available planes. The level of the next pass will be calculated from the last level according to the defined step.

See also:

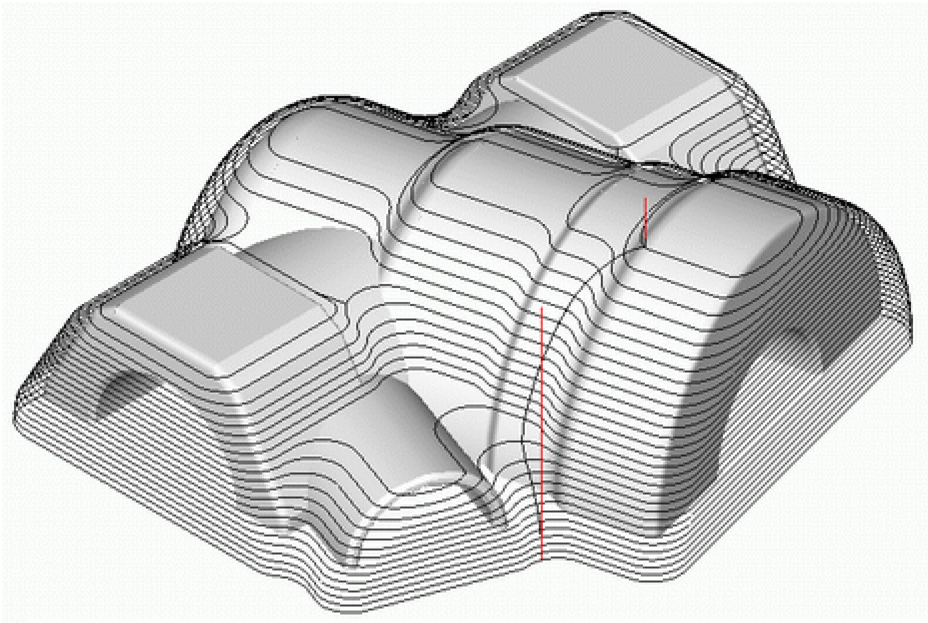
Definition operation parameters

5.3.34 Corners smoothing



When there is a sudden change of the tool movement direction, the milling unit 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 at the defined radius for machining inner corners of the model.

An example of machining with corner smoothing:

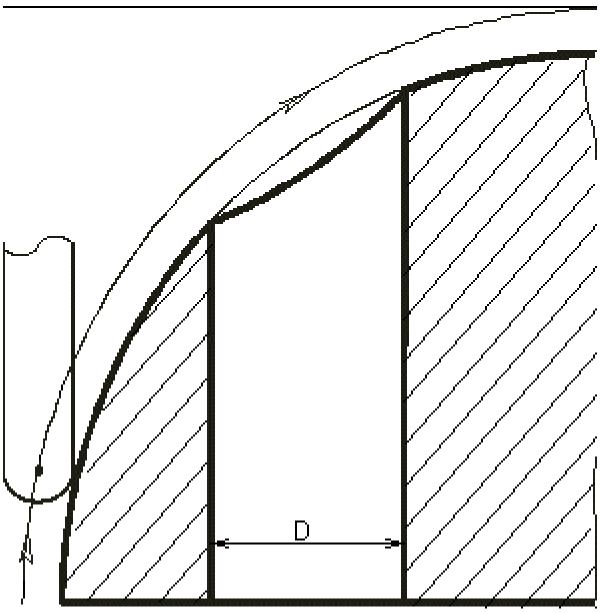


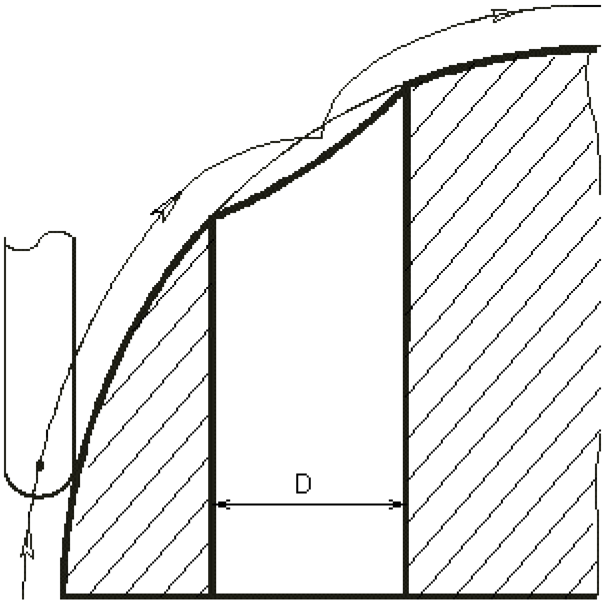
See also:

Definition machining strategy

5.3.35 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.





See also:

Definition machining strategy

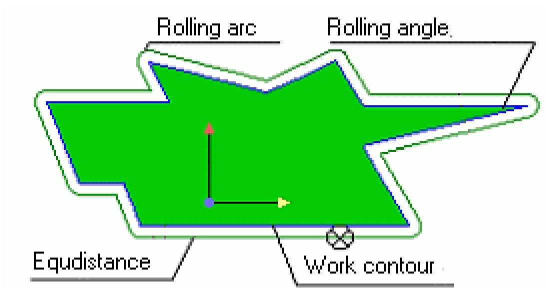
Job assignment for hole machining operation

Defining holes by coordinates

Defining holes by using a geometrical "point" object

Automatic hole recognition

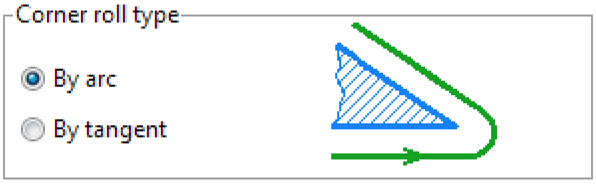
5.3.36 External corner roll types



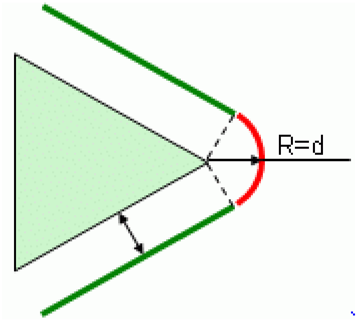
There are two types of external contour corner rolling in SprutCAM X:

- <By arc>
- <By tangent>

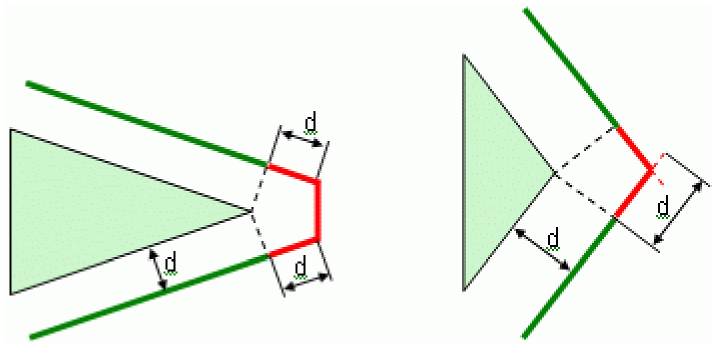
Corner rolling is possible only when SprutCAM X creates an offset toolpath. Selection of the roll type can be made on the <Strategy> page. Select the Corner roll type area by choosing either the By arc or By tangent mode.



When rolling by arc, an arc whose radius is equal to the offset value will be inserted into the toolpath corner.



In <By tangent> mode the lines are used to roll the corner. The tool path before and after the corner is extended on the offset length. If the extended path is intersected then it is truncated in the intersection point. Else the connection line is added.



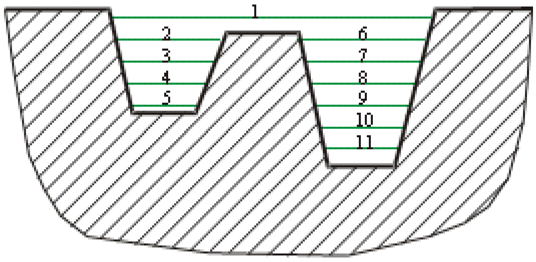
See also:

Definition machining strategy

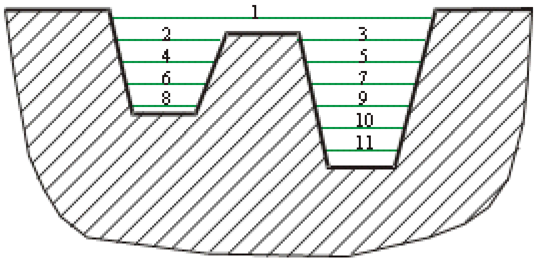
5.3.37 Machining order (by depth or by contours)

When machining multiple contours in several passes, two options exist for controlling the order of depth machining. When machining by depth, after completion of the first pass along the first contour the system then performs execution of the first pass along the next contour. The execution of the second pass is performed after all contours have been machined. When machining by contours, the system first performs all passes along the first contour, then all passes along the second contour etc.

- <By cavities>



- <By layers>

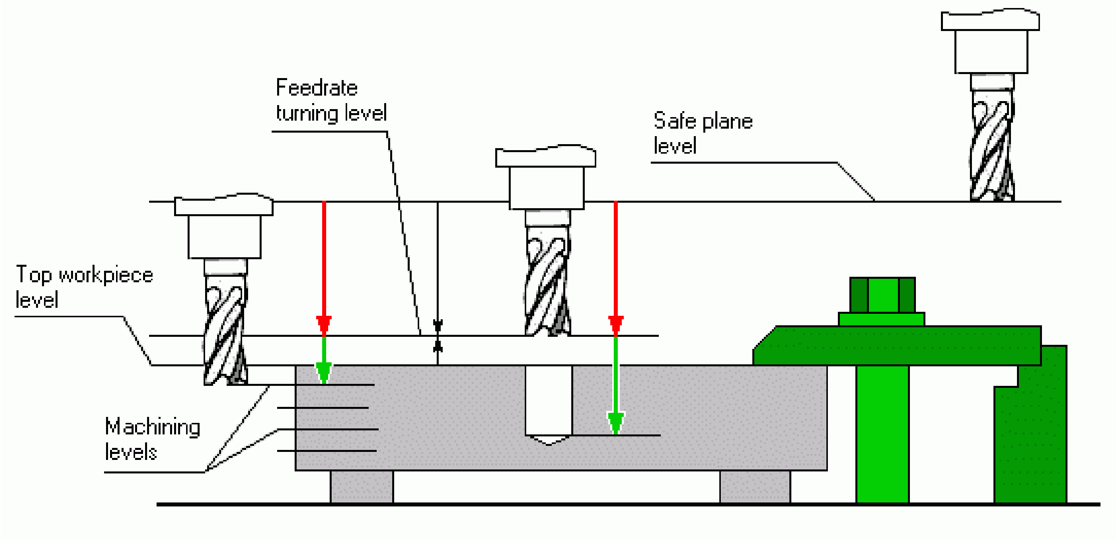


See also:

Definition machining strategy

5.3.38 Tool plunge

Tool plunge is a movement of a tool from the plane level (depth) of tool step-over along the Z axis from the top machining level, which can be assigned in the parameters assignment window. If the step-over plane level coincides with the top machining level, for example, during step-over around the workpiece, a plunge move will not be applied. A plunge consists of two parts: a transition at rapid feed and a transition at work feed. If required, the user may use only one mode – e.g. only the rapid move, or, only the work feed move. Tool plunge must be performed either outside of the workpiece area or into a previously drilled hole. The plunge parameters can be assigned on the Toolpath page. The assigned parameters will apply for the entire operation irrespective of how many contours are being machined in the operation.



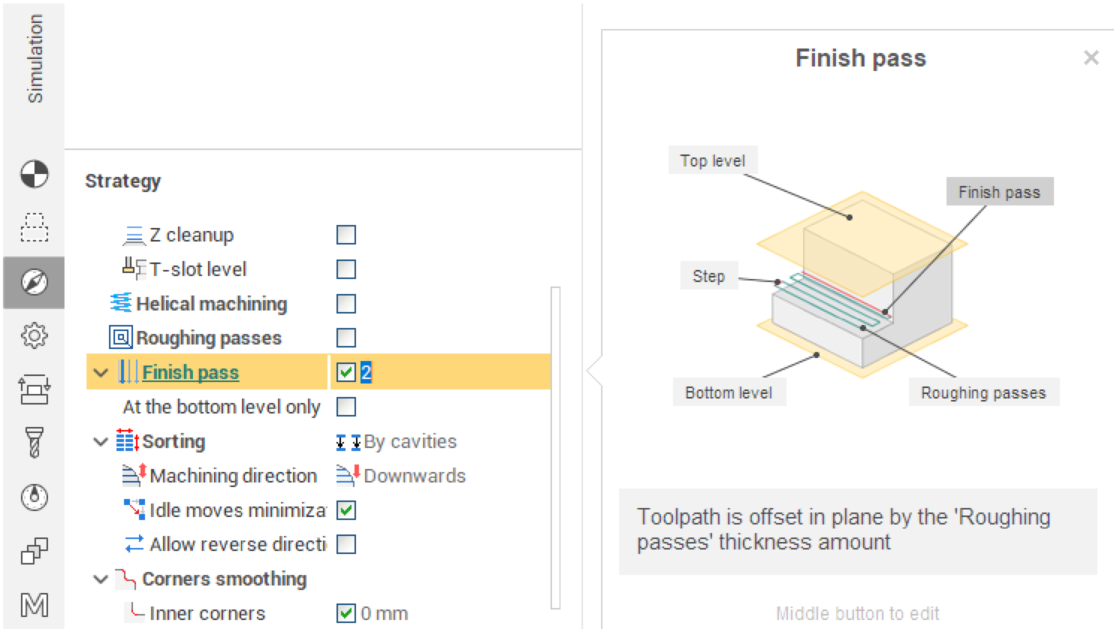
The feed switchover level can be assigned by two methods: as an absolute value in the current coordinate system (just like assigning the safe plane, and as the top and the bottom workpiece levels), or as the distance from the top workpiece level. The assignment can be performed in the < Lead in / Lead out> window.

See also:

Plunge

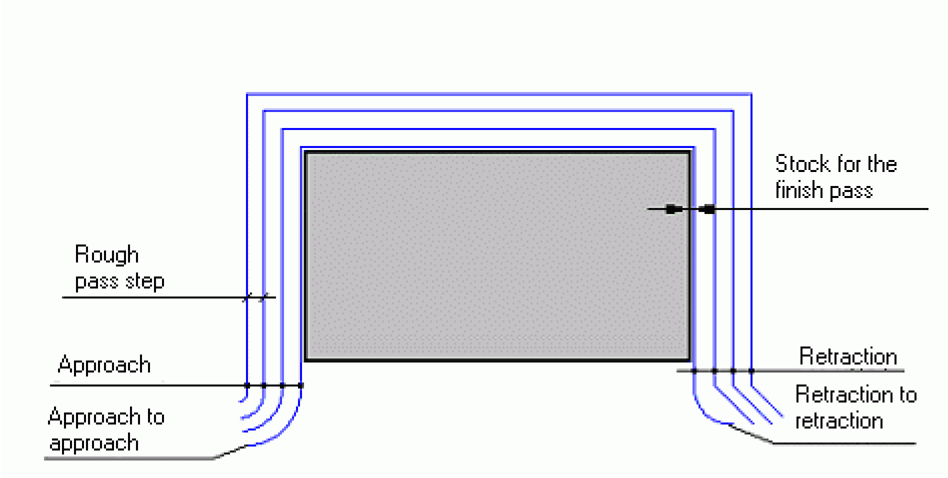
5.3.39 Assigning finish pass in the XY plane

A finish pass for closed and open curves in 2D machining operations can be set up in the <Strategy> window.



When activating this option, the additional stock will be left on the model, which will be removed by the last finishing pass. It will allow the user to obtain higher quality surface finish.

The value of the stock can be defined as the absolute value or as a percentage of the tool diameter.



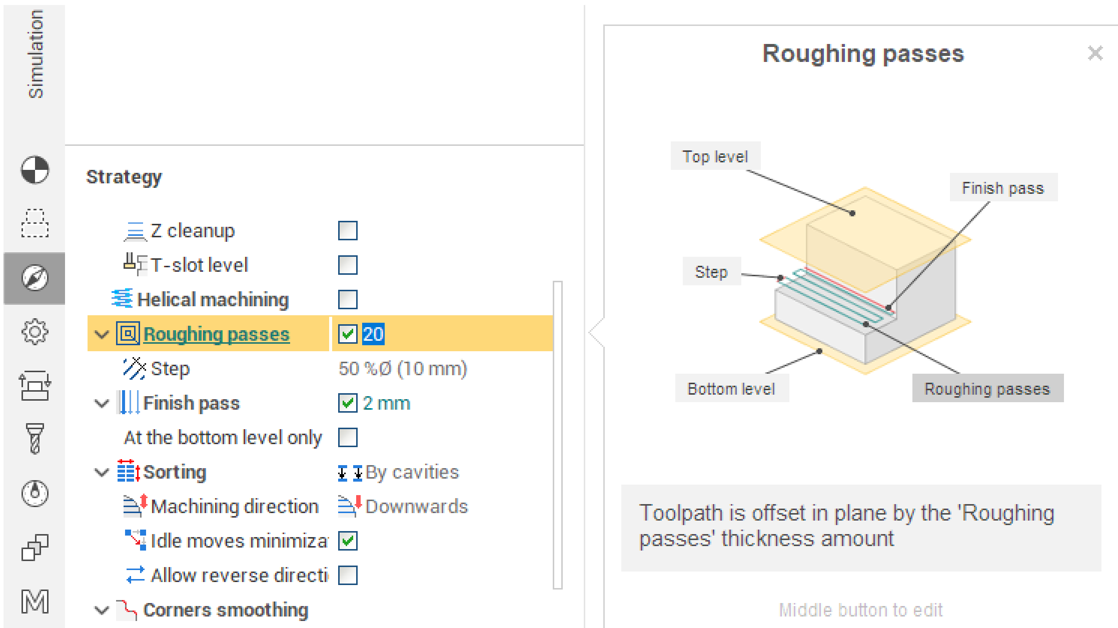
See also:

Definition machining strategy

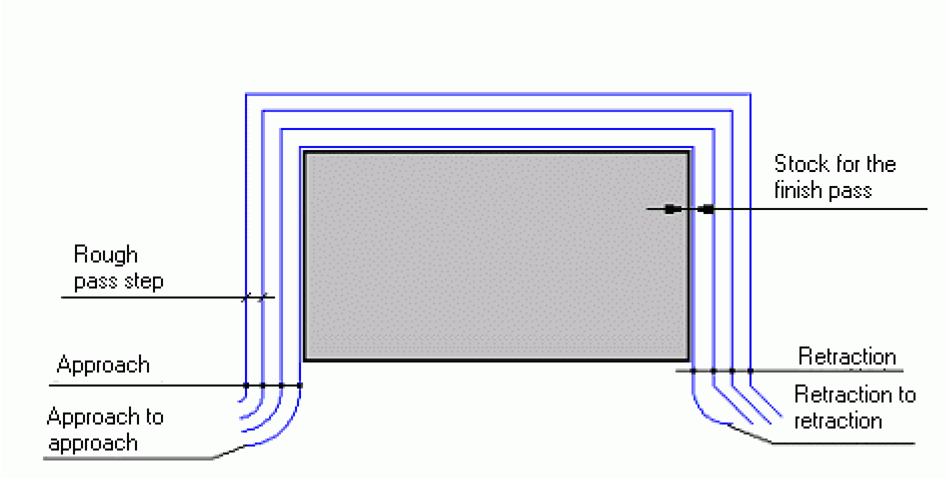
List of types of machining operations

5.3.40 Assigning rough pass in the XY plane

Roughing passes for closed and open curves in 2D machining operations can be set up in the <Strategy> window.



When activating this option, the stock from the model will not be removed in one go; instead, it will be done according to the assigned parameters. The value of the stock can be defined as the absolute value or as a percentage of the tool diameter.

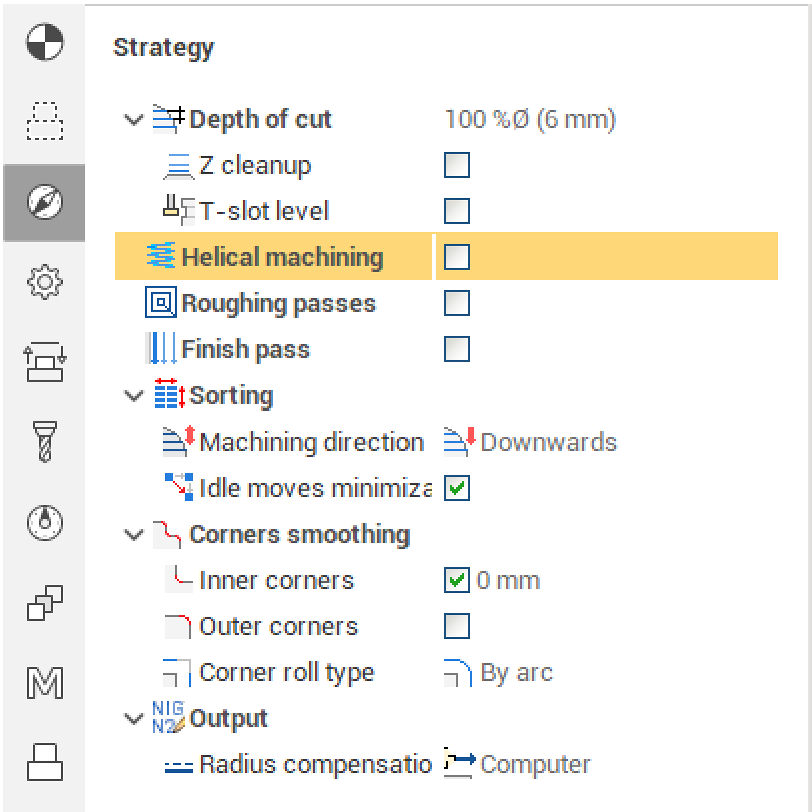


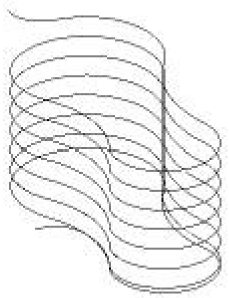
See also:

List of types of machining operations

5.3.41 Helical machining

When machining 2D curves it is possible to obtain a spiral-like toolpath. Thus, in the resulting NC program, a block G2/G3X:Y:Z:R: type can be inserted (depending on the postprocessor used). At the bottom machining level, the user can assign a finish pass along the entire curve. The user can also assign a smoothing value for linking the spiral and the finish pass at the bottom level. Smoothing is defined as a radius value.





The number of spiral loops depends on the Z step assigned in the <Parameters> page. If a tick is placed in the <Clean pass> box, the system will generate a horizontal toolpath area for bottom machining.

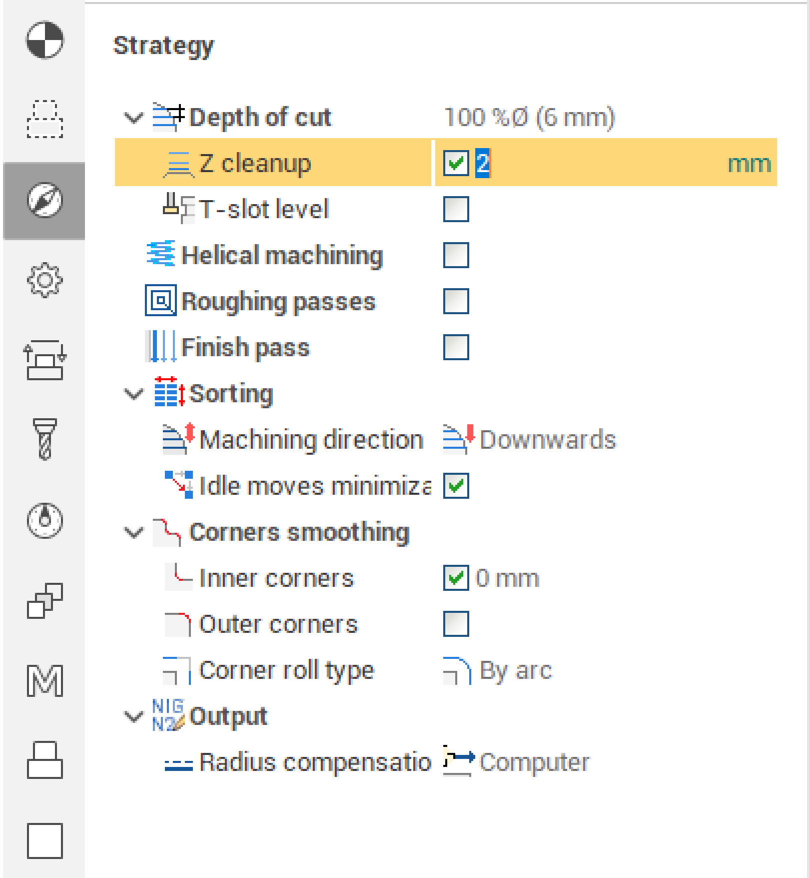
With <Smoothing> mode active, the tool transition onto the horizontal plane (bottom level) will be performed without a break along the curve using the defined radius. This function is especially useful for high speed machining.

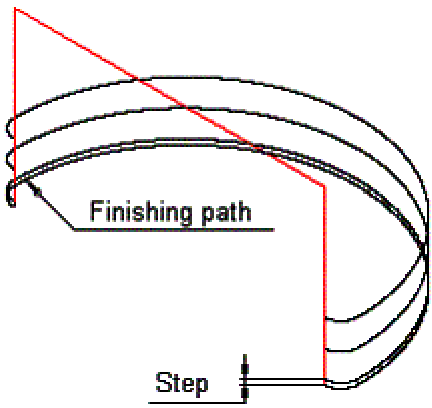
See also:

Definition machining strategy

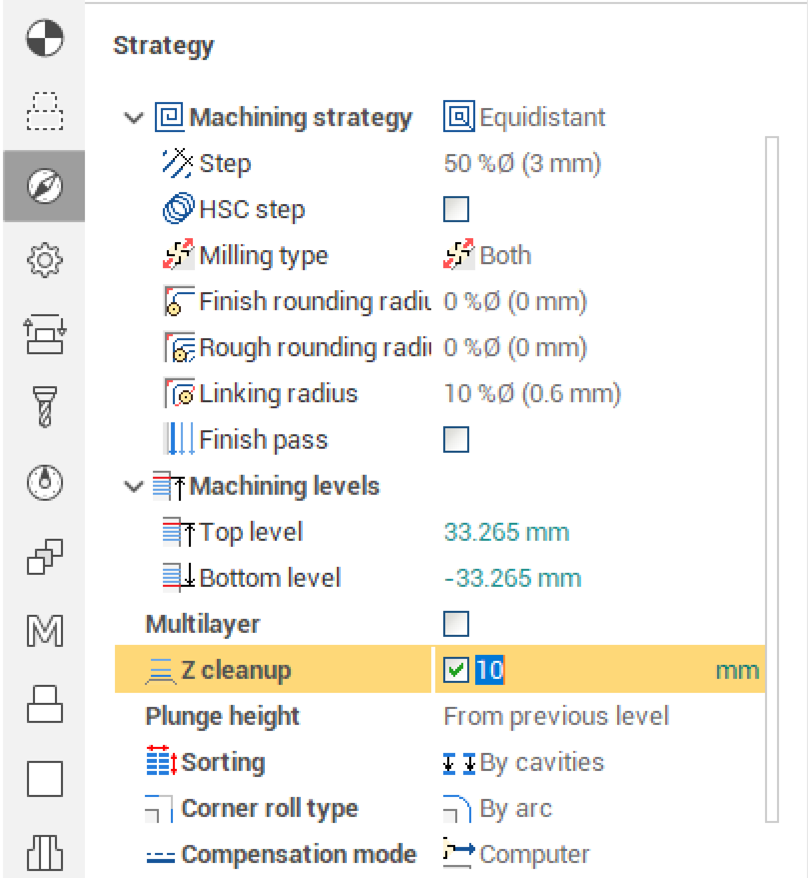
5.3.42 Z cleanup

In 2D and 2.5D machining operations, it is possible to activate the clean pass mode in depth, e.g. at the bottom level. This is assigned on the < Parameters > page in the < Operation parameters > window. All previous passes will be automatically allocated equally along the Z axis.





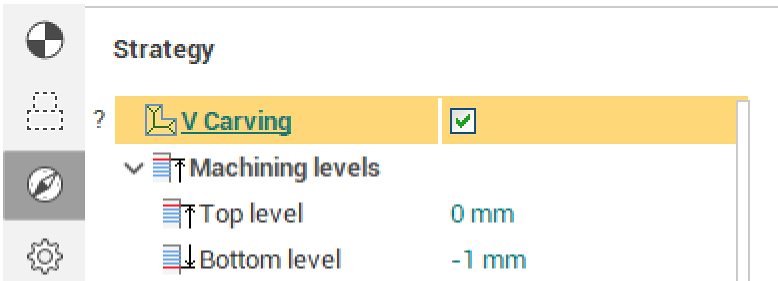
In Waterline roughing and Flat land machining machining operations, for the definition of width machined material layer on finish pass, it is necessary to switch on an <Z cleanup parameters> and to enter a stock value on finish pass.



See also:

Definition machining strategy

5.3.43 V Carving

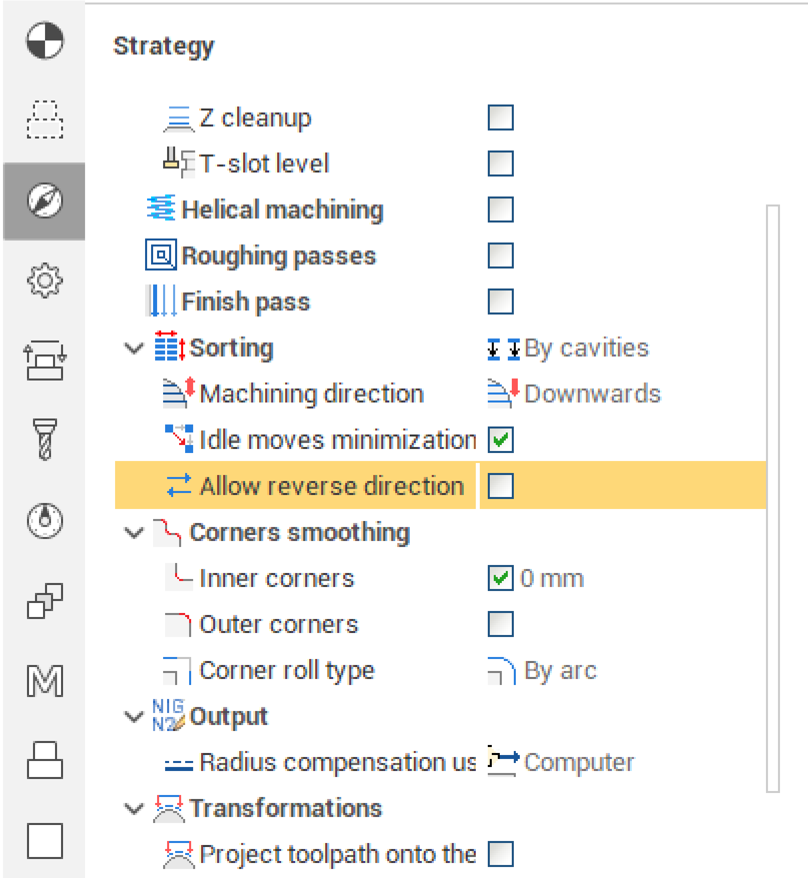


The <**V Carving**> mode allows the system to create a 3D toolpath for material removal in all areas that are not accessible for machining on the current level (e.g. inner corners).

See also:

Defining the machining strategy

5.3.44 Allow reverse direction

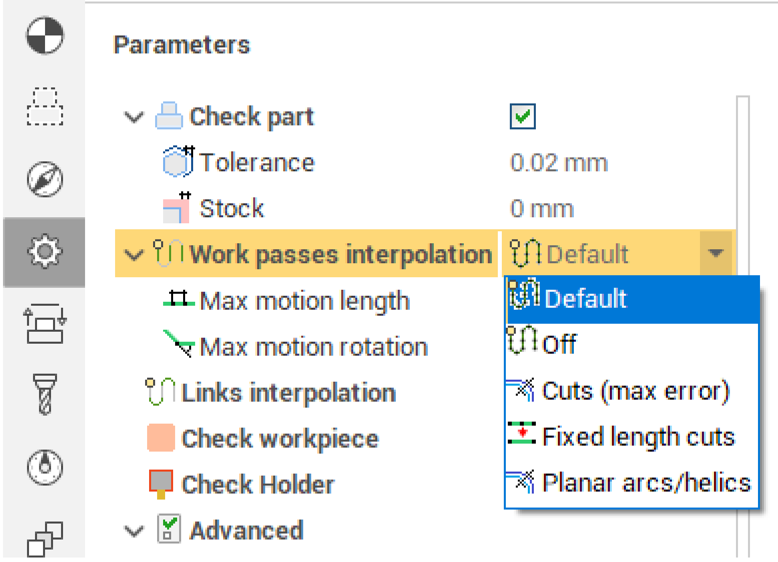


<Allow reverse direction> in the curve machining operations allows the tool to reverse its cut direction along a curve if it will decrease the overall amount of tool movements.

See also:

Defining the machining strategy

5.3.45 Work passes interpolation

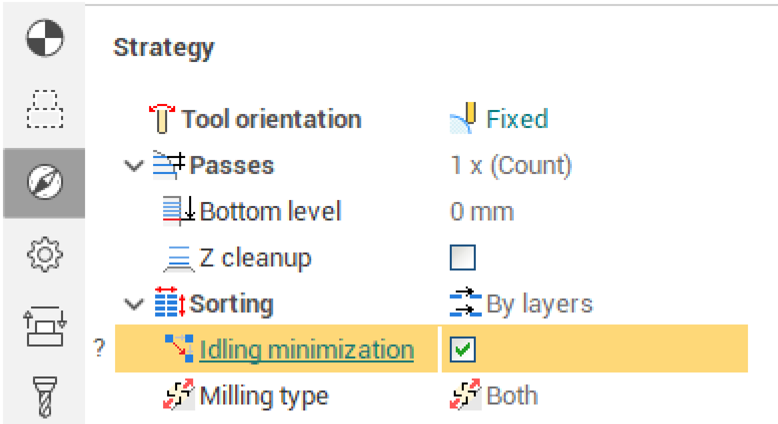


<Work passes interpolation> allow approximate toolpath by arcs.

See also:

Defining the machining strategy

5.3.46 Idling minimization

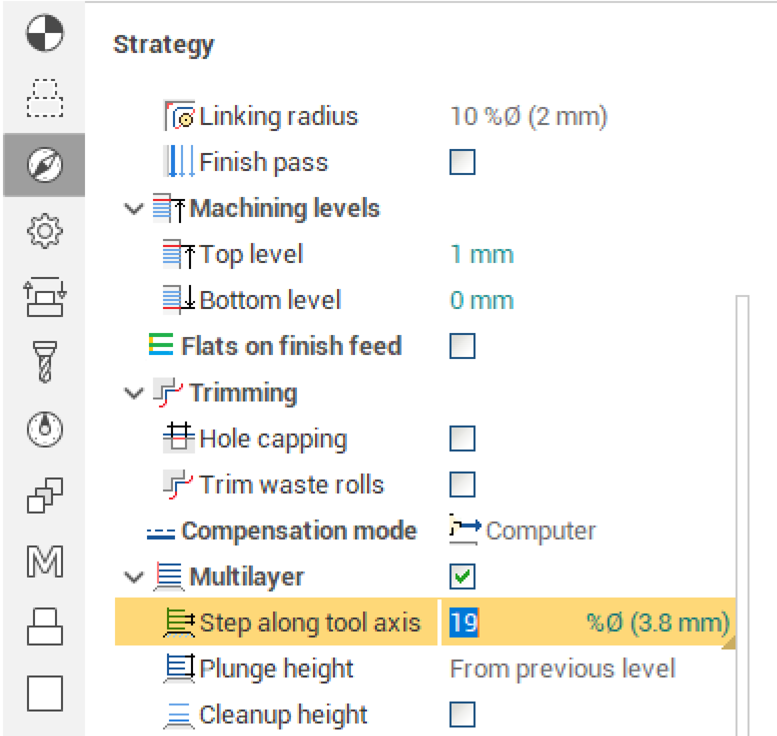


With <Idling minimization> active, the total distance that the tool moves for machining all selected curves, is kept to the minimum. Otherwise, the machining will be performed according to the order defined on the <Model> page.

See also:

Defining the machining strategy

5.3.47 Machine by layer

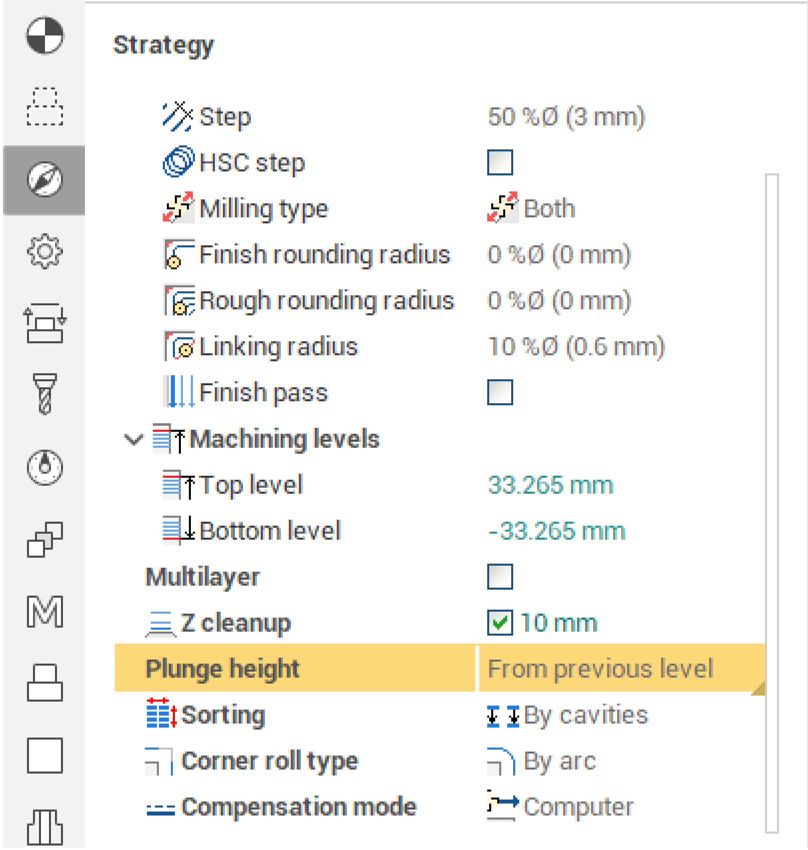


The material may be deleted for some passes. For this purpose, it is necessary to switch on <**Machine by layers**>. The amount of passes defines by the layer height and step on axis Z, given in the panel. The step on axis Z may be given by an absolute value or in percentage of diameter of the tool.

See also:

Defining the machining strategy

5.3.48 Plunge height



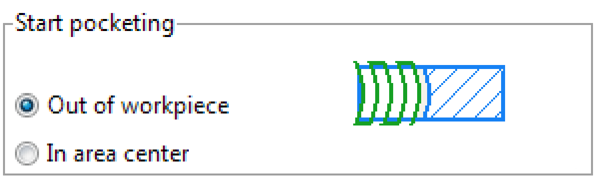
The height of the layer is set in the <Plunge height> panel; in particular, it defines a value of level Z on which the given type of plunge will join.

- <From previous level> – Plunge will join at once at driving downwards to the following level
- <From level (Z)> – the scheme of plunge will be powered up always at driving downwards from the given level.
- <Height (H)> – the scheme of plunge will join on distance H from a treated plane.

See also:

Defining the machining strategy

5.3.49 Start pocketing

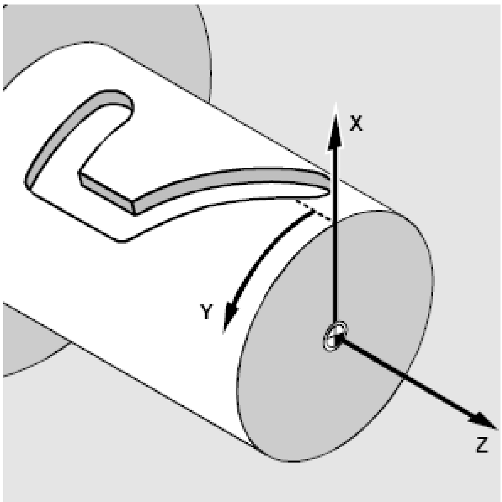


The strategy of the tool motion within the limits of one layer is defined in the <Start pocketing> panel. If to start machining <In area centre> the path length in a plane will be less, rather than if to start machining <Out of workpiece>. In this case the tool will plunge in a material even in the event that it is possible to lift down behind. As investigation, a machining time considerably may increase at the expense of application of the plunge scheme.

See also:

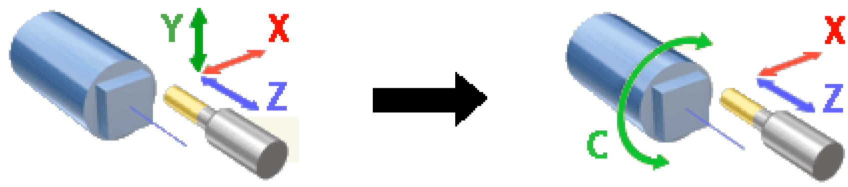
Defining the machining strategy

5.3.50 Cylindrical interpolation



The cylindrical interpolation is available in operations: 2D contouring, pocketing, 2.5D pocketing, 2.5D wall machining.

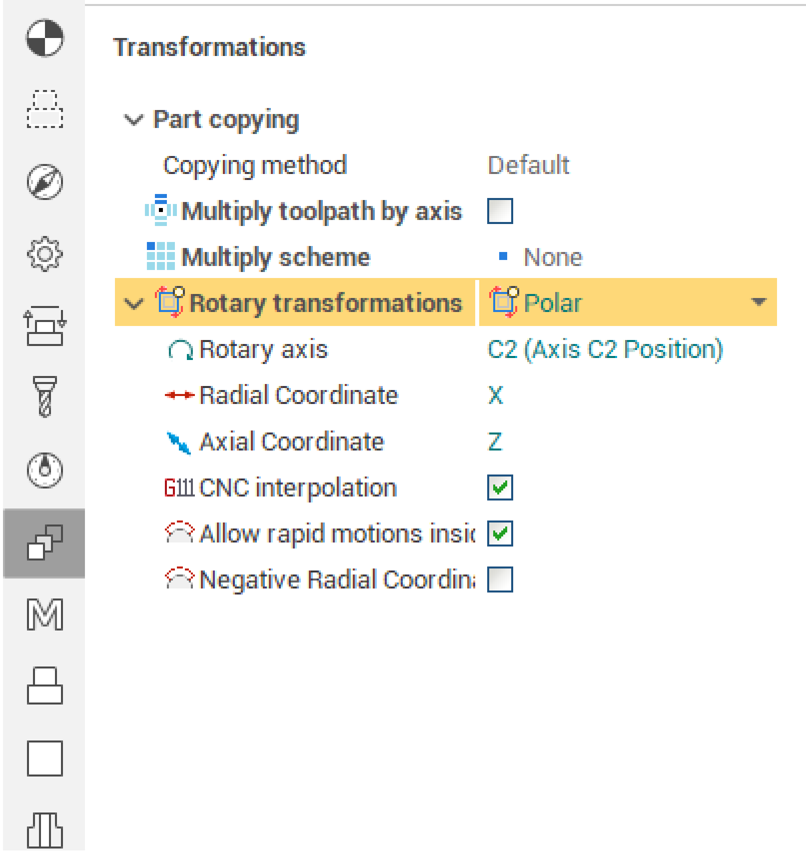
The cylindrical interpolation gives the possibility to mill the side surface of cylinder by programming the unrolled curves. The unrolled curves are programmed in the [X,Y,Z] coordinates, but the cylinder milling is performed in [X,C,Z] coordinates. So the cylindrical interpolation makes the transformation [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

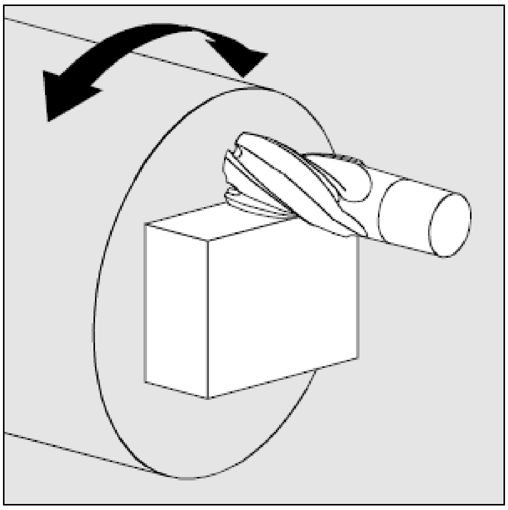
The corresponding commands for the well known controls are shown in the table below.

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.

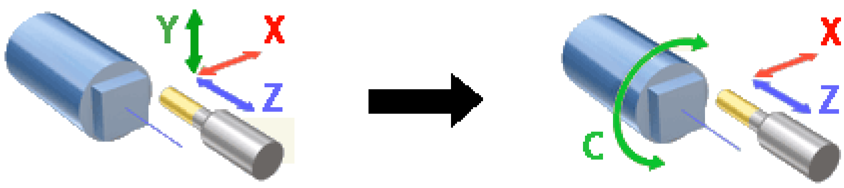
If <Allow rapid motions inside interpolation block> is checked then the interpolation switch on in the beginning of the tool path and switch off in the end of tool path. Else CNC interpolation is started before the work feed and closed before the rapid feed motion.

5.3.51 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.

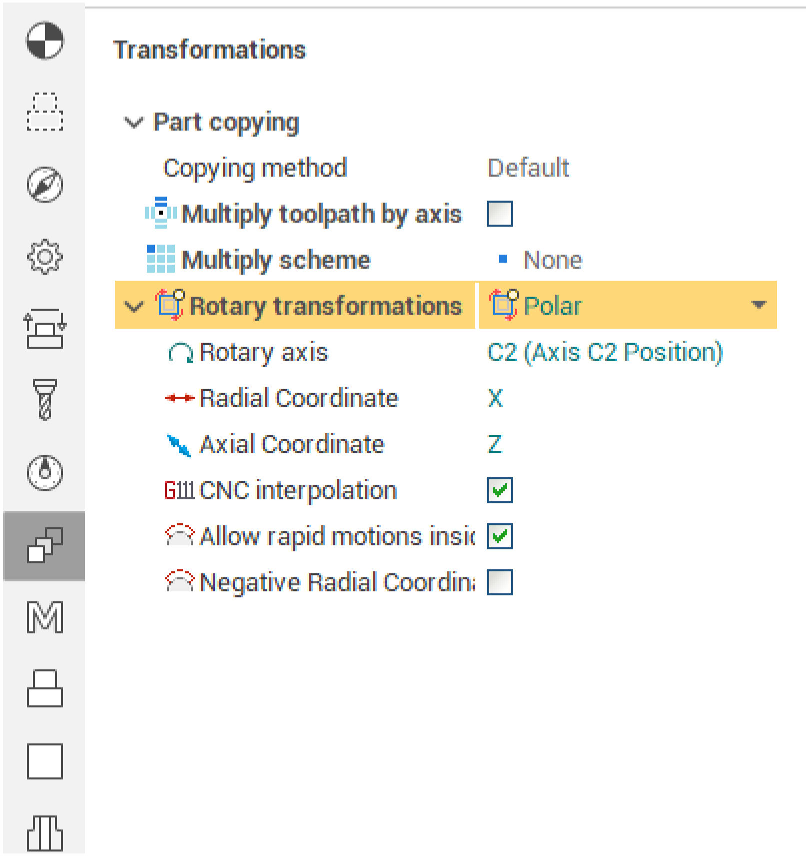


In simple mode SprutCAM X generates the G-code in [X,Y,Z] coordinates. If polar transformation is active then the same g-code is generated in the [X,C,Z] coordinates. So polar interpolation transforms [X,Y,Z] => [X,C,Z]

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	-	-

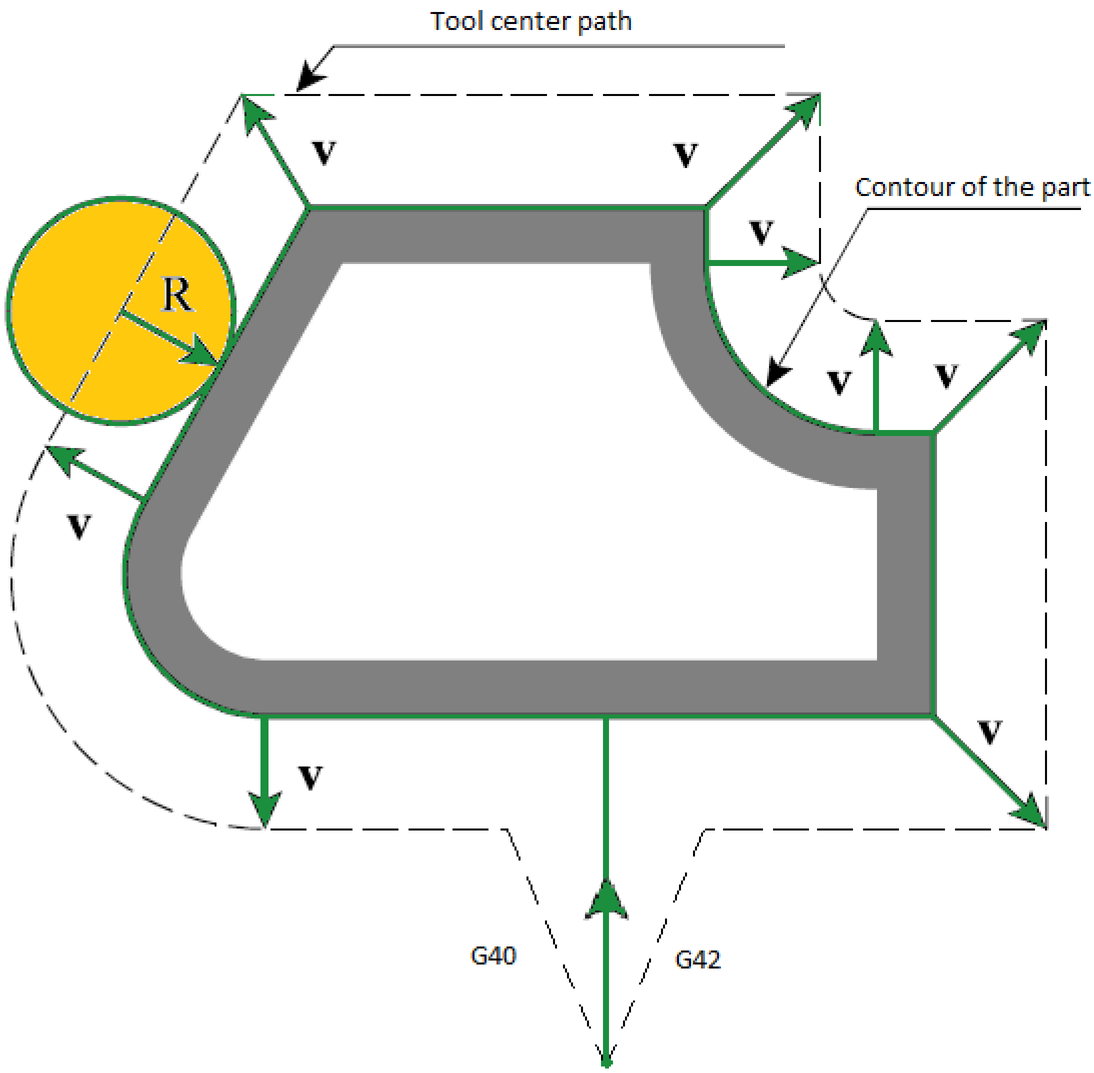
If the machine variable Machine –> Control parameters –> Rotary transformations –> CNC support polar interpolation is set then CNC interpolation tick is available. If this parameter is on then the G-code generated with the commands to switch on/off the polar interpolation. Else the G-code is generated in the [X,C,Z] coordinates.

5.3.52 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).

5.3.53 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.



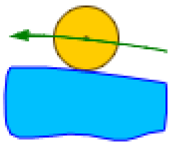
In order to correctly calculate the trajectory of the tool center CNC needs to know the direction in which to move the tool for each frame of the trajectory. There are special commands that set the direction of correction in the NC-program. Usually this command G41 - the correction to the left of the 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.

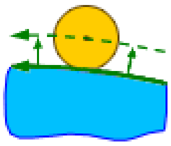
CAM-system can generate a trajectory with tool radius correction. On the "Lead In/Lead Out" page is a panel where you can enable the use of correction, as well as to select the types of compensation switch lines.

There are 5 different types of the correction:

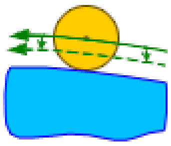
- <Computer>. In this case, the CAM-system itself calculates the trajectory with the size of the selected tool. In the program is generated a trajectory for the tool center. Correction turn on and of commands does not appear. This does not allow the operator to affect the correction at the machining 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.



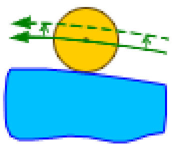
- <NC control>. In this case, the trajectory calculation is made without taking into account tool radius. In the NC-program displays a contour of the part, as well as the turn on and off compensation commands of the appropriate sign. This allows the operator to adjust the trajectory, taking into account 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).

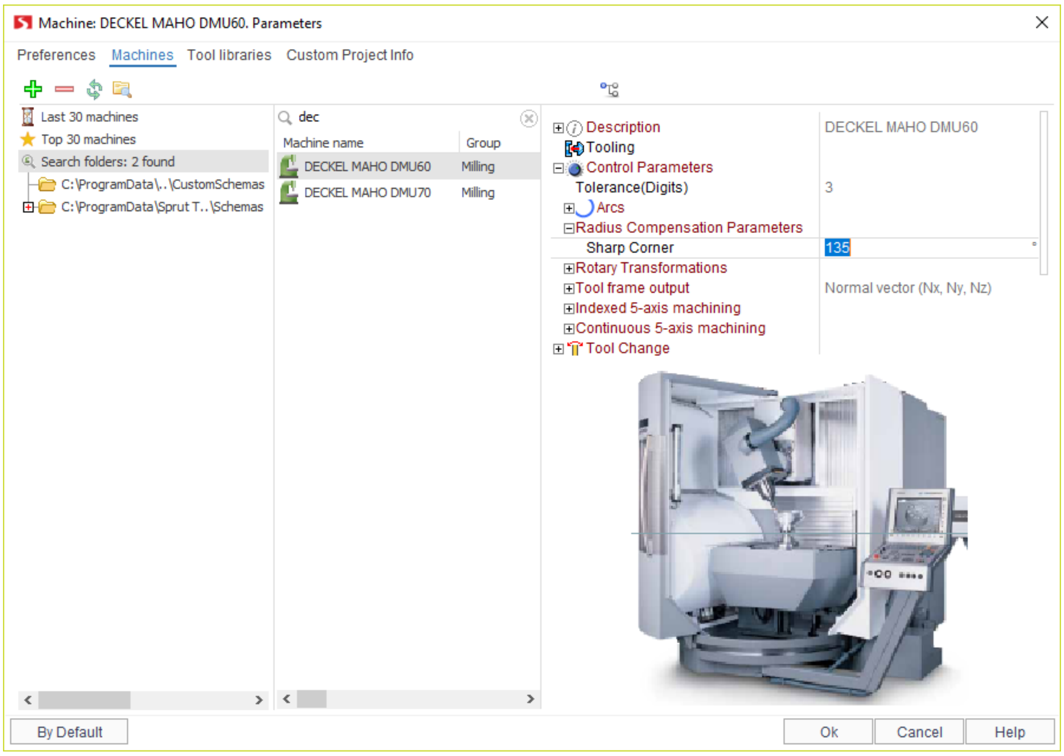


Radius compensation simulation options

SprutCAM X 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. SprutCAM X 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.

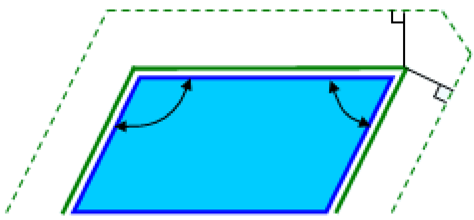
Different NC machines can use different methods for applying / canceling compensation. SprutCAM X have several options which can be used to 'tune' SprutCAM X's tool radius compensation so that it matches those used by the machine control. These options are available in the <Machine: ... Parameters> window on the <Machines> tab. There is a node called <Control parameters> –> <Radius

compensation> a property editor, the properties are used for tuning the SprutCAM X 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.

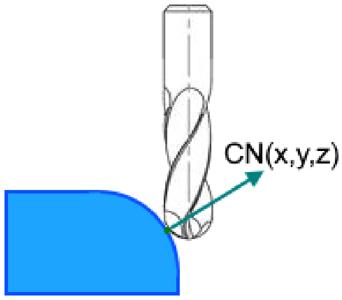


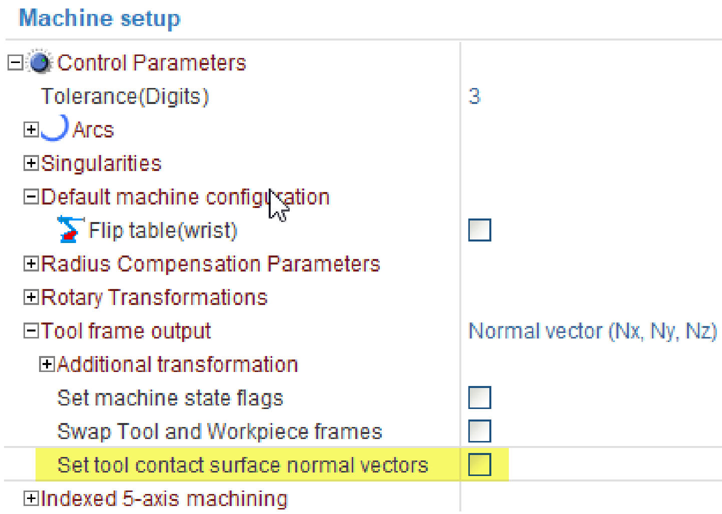
Related reference:

The motions of CRC switch on/off

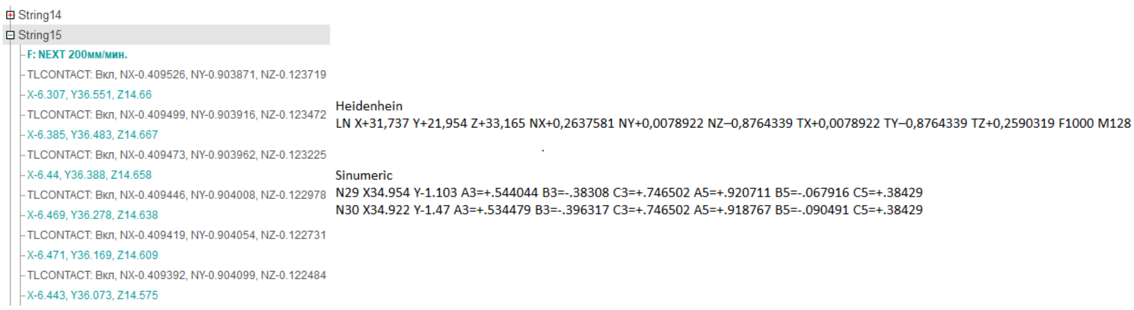
5.3.54 Tool 3D compensation

For the surface finishing operations it is possible to output to the G-code the normal vector of the surface at the point of contact with the tool. This allows to perform the so-called “3D compensation”, i.e. slight displacement of the tool at each point of the path away from the part.





You need to enable the "Set tool contact surface normal vectors" flag in the settings of the machine schema in order to see this information in a G-code. In this case, a new TLCONTACT command (NX, NY, NZ) will appear in CLData for each path point, which contains the necessary normal vector for subsequent path points. In the postprocessor, depending on how 3D correction is supported in a particular CNC, you need to either directly output the contact normal to the G-code frame or manually shift each point of the path in the direction of this vector by the correction amount.



5.3.55 Toolpath multiplying

The Multiply toolpath features of the **SprutCAM X** 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.

There are 2 different ways to perform it: multiply toolpath by axis and multiply scheme.

The <Multiply toolpath by axis> feature more suitable for lathe-mill machines. It can multiply toolpath only by one machine axis (frequently it is the rotary axis, for example C-axis of lathe milling machine, when coordinate system does not rotate with the workpiece together when changing axis value). This kind of transformation is possible if source set of toolpath commands may be converted correctly with only one axis value replacing. So you need to define in parameters the name of the axis to convert, the step to increment value of this axis and the count of repeats. Frequently source portion of NC code that used for transformation does not contain the value of axis you want to use for multiplying. For example when we have indexed rotary machining, C rotary axis presented only inside approach part of the toolpath to provide desired tool-workpiece orientation but the main part of machining toolpath is fixed in a space and contains X, Y, Z movements only. So this unchanged part of toolpath can be formalized as a subroutine inside NC-code and can be called so many times as you need with changing only C axis orientation before each call. To make such transformation it is enough to enable Formalize as subroutine parameter and input desired subroutine ID (name or number).

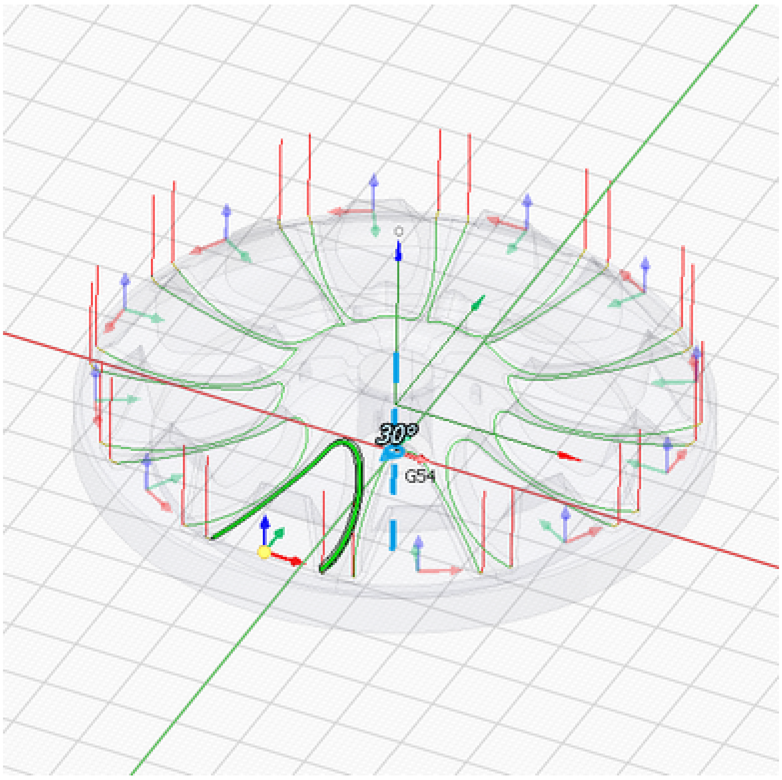
We can have more complicated case of transformation when many coordinates of source toolpath must be changed at the same time (for example X,Y and Z) almost inside each NC block to repeat machining at the neighbor place. For 5-axis machines with TCPM (when coordinate system rotates with the workpiece together) only this kind of transformation is possible. In this case you need to use Multiply scheme transformation instead. This transformations works another way. It converts not ready toolpath commands, but it transforms source geometrical curve of toolpath in space and only then converts it to the toolpath commands.

Multiply scheme provide different spatial strategies of transformation:

- Two dimensional array;
- Two dimensional array (manually);
- Round array;
- Round array (most distant);
- Round array (manually);
- Level array;
- Axis symmetry

Instance count, steps, center point and sometimes angle must be determined depend on selected strategy.

When selecting Multiply scheme parameter of the operation inside inspector then its graphical visualization appears on the screen. You can directly see an amount of transformed items and their orientation. You also can drag some hotspots (for example center point) with mouse button and input multiplying step values.



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.

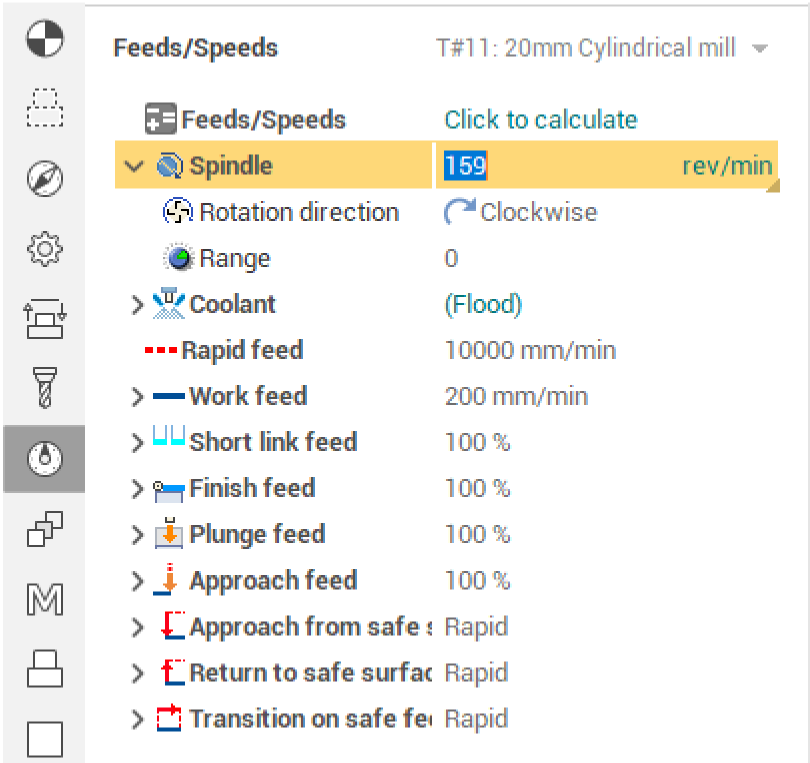
Use feature CS parameter allows to perform machining of each multiplied item with their own local CS. Before calling the each next item's machining the ORIGIN command will be added that will be converted by the postprocessor to PLANE SPATIAL, CYCLE800, G68.2 etc. transformation command depending on used type of controller.

5.3.56 Speeds/Feeds calculation

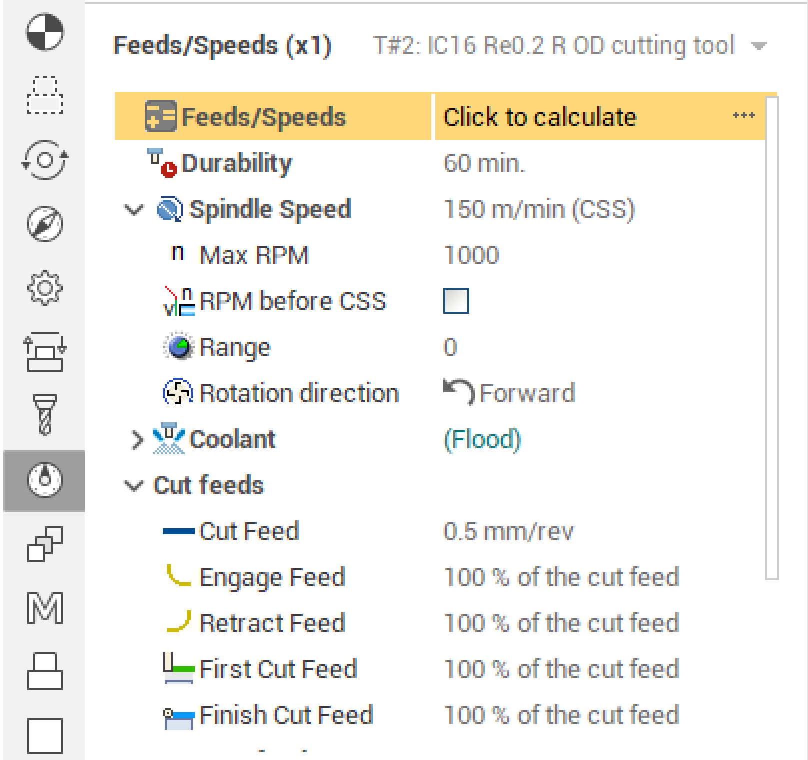
In <**Feeds/Speeds**> form you can call <**Feeds/Speeds calculation**> window for calculating feed and speed of cutting conditions of operations.

You can perform calculations for both milling and turning operations.

Milling operation:



Turning operation:



5.4 Feature based machining

Feature Based Machining

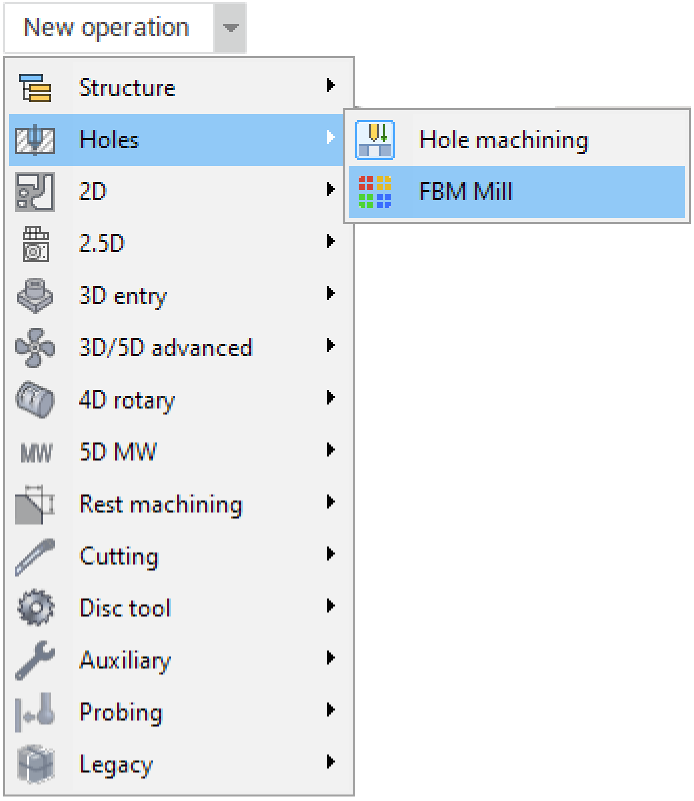
A Manufacturing Feature is a component of a part that can be manufactured with a standard machining process. Examples of features include holes, pockets, slots, grooves, chamfers.

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

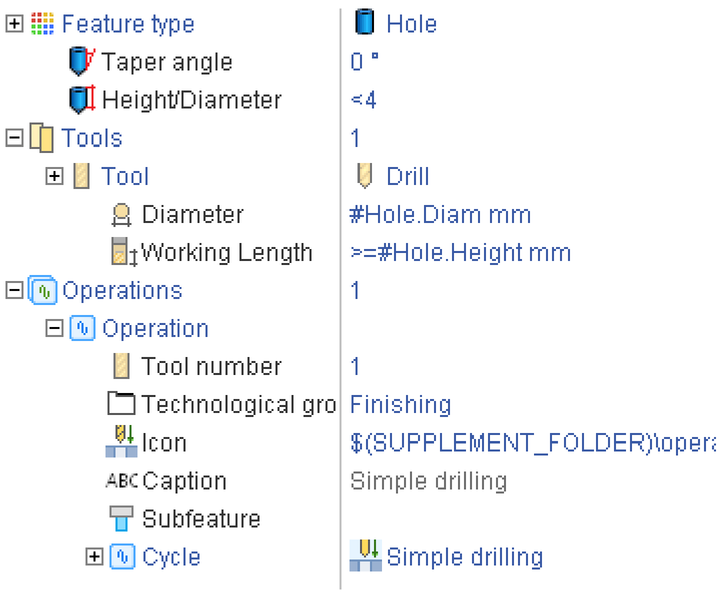
The FBM Mill Group is a group operation in SprutCAM X that allows you to do the Feature Based Machining.

When you create a new FBM Mill Group operation SprutCAM X 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 SprutCAM X 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.

5.4.1 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, SprutCAM X 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 SprutCAM X 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.

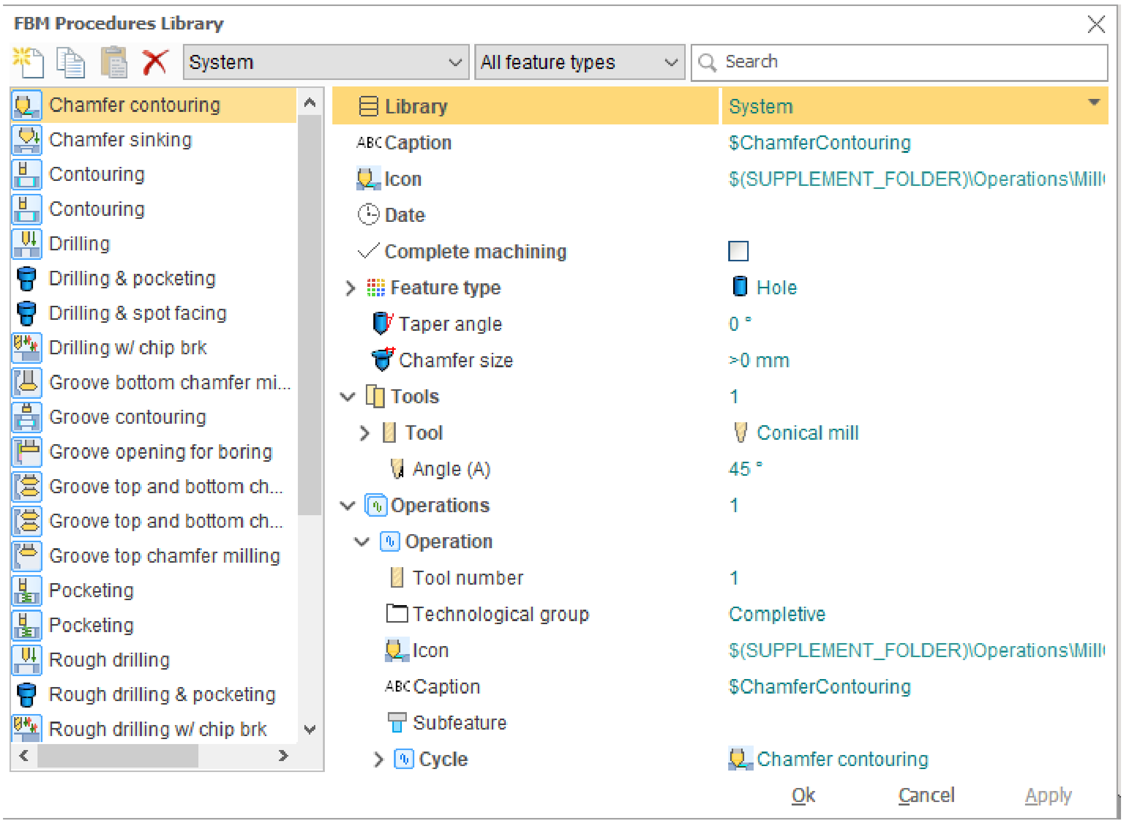
The Technological Group defines the class to which the operation belongs. There are the following technological groups:

- 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.

5.4.2 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 SprutCAM X installation to provide the most universally used complete and atomic procedures.

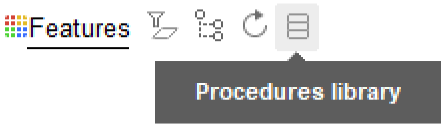
The All Users folder contains procedures accessible to all users of the PC.

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 SprutCAM X 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.

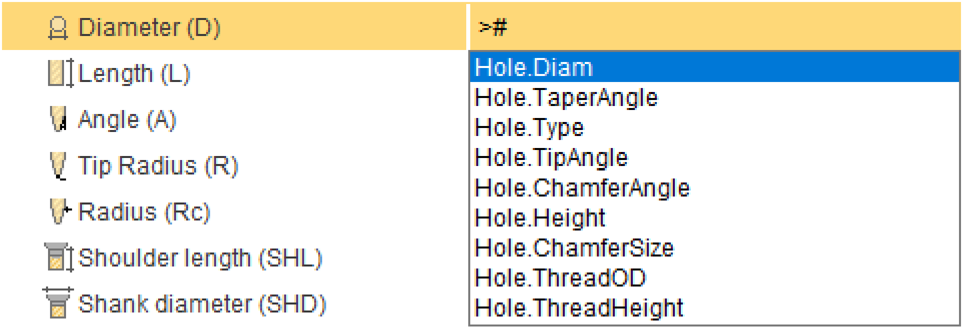
Editing the Feature Constraints

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

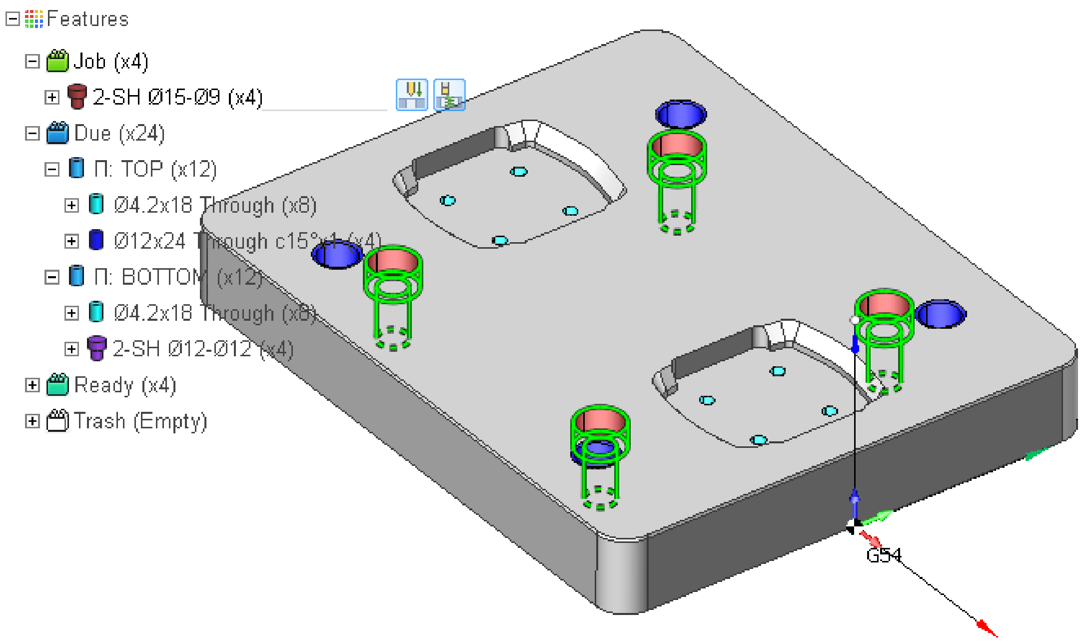
Edited procedures are marked with the bold font in the list.

- 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.

5.4.3 Feature Tree UI

After SprutCAM X 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 SprutCAM X 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, SprutCAM X marks the other one as Ready automatically.

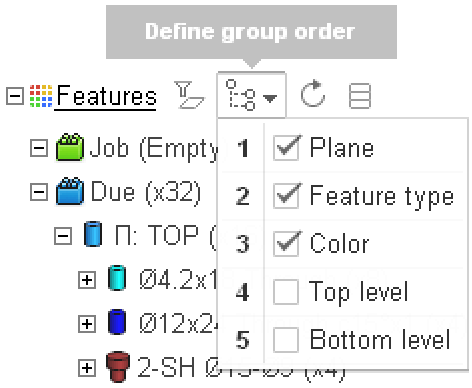
Changing the grouping scheme

SprutCAM X 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.



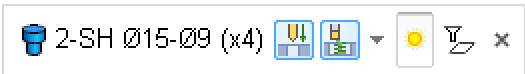
Selecting features

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

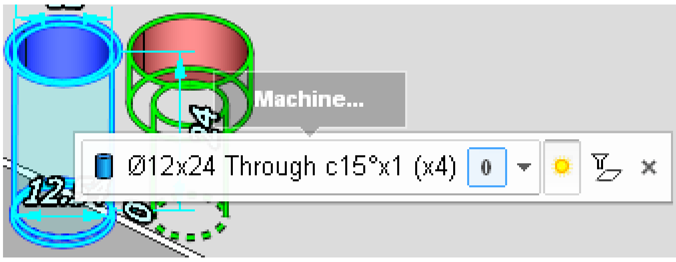
The Delete Features operation is context aware.

- 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.

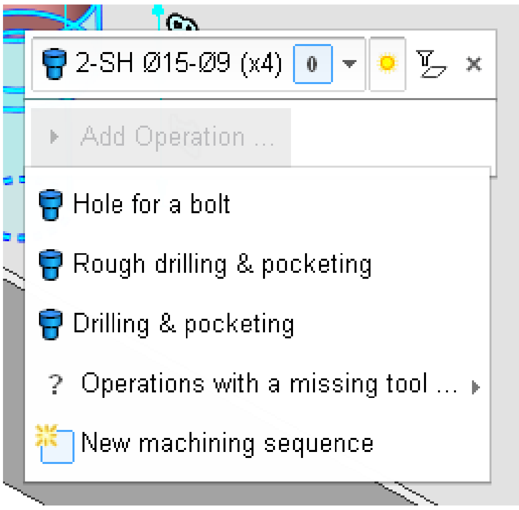
5.4.4 Assigning procedures to features UI

To select a machining procedure for a feature,

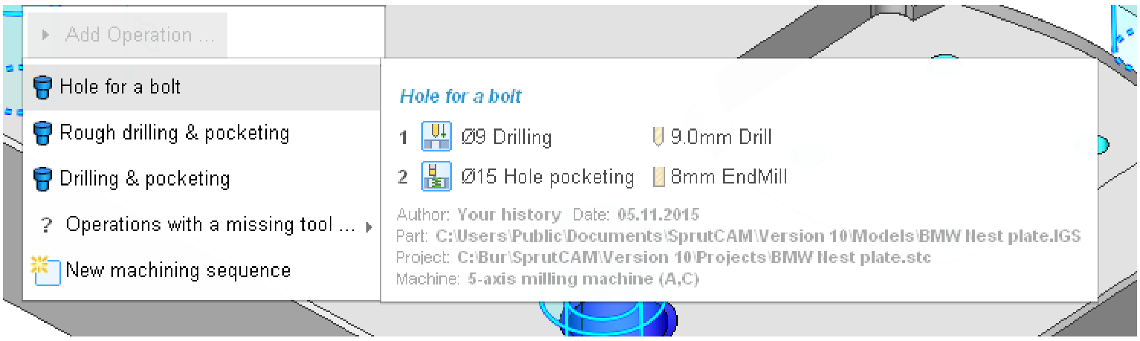
- Either click on the  icon in the tree. This icon appears when you highlight a node in the tree;
- Or click on the Machining procedure combo in the action bar.



- 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.



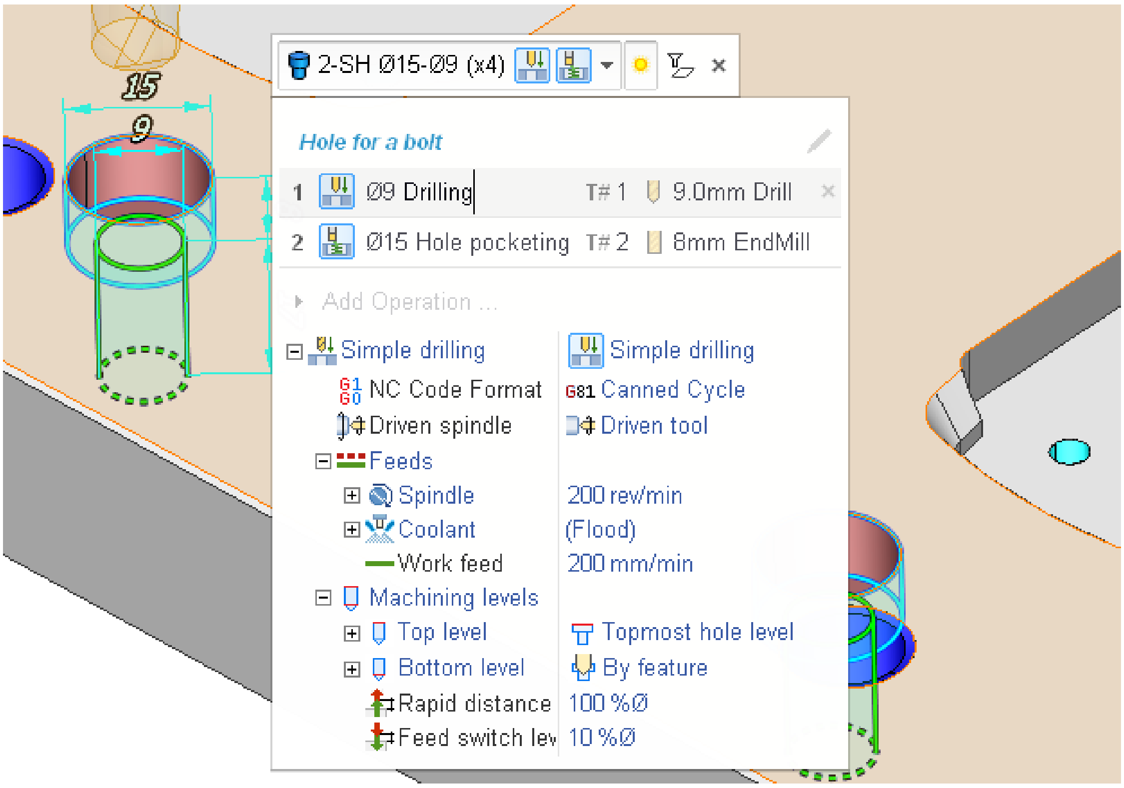
Along with that you will see some additional information about the origins of the suggested procedure: the author, the date of creation, the part it was created for, the project it was used in, the machine which was used in that project.

The operations for which SprutCAM X 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 SprutCAM X updates the resulting operation list, while the selected procedure becomes available for editing.

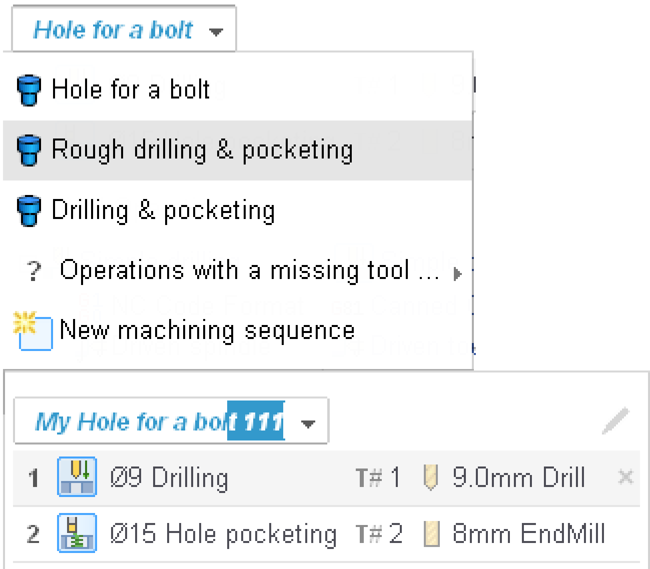
5.4.5 Procedure Editing UI



The editing of machining procedures is done in this little panel you can see above.

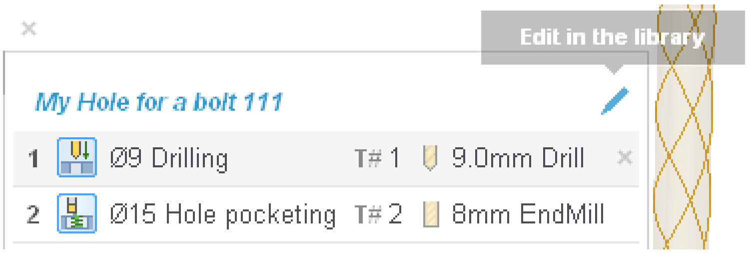
The Machining procedure selector

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.



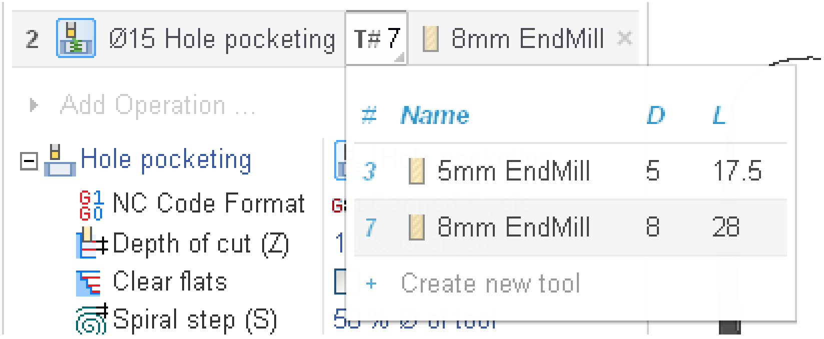
The operation list

Under the header there is the list of the operations constituting the machining sequence.

Each entry in the list consists of the following fields.

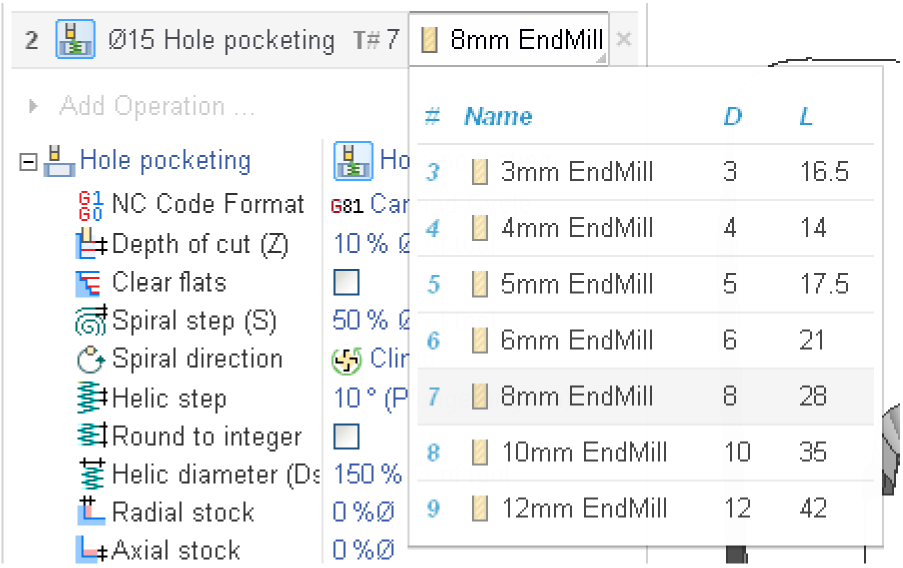
1. The sequence number. You can drag and drop operations by this number.
2. The Icon.
3. The subfeature prefix (present only if an operation machines a subfeature inside a composite feature).
4. The Operation name, It’s an edit-box, you can change the operation name any time you want.
5. The tool number 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.

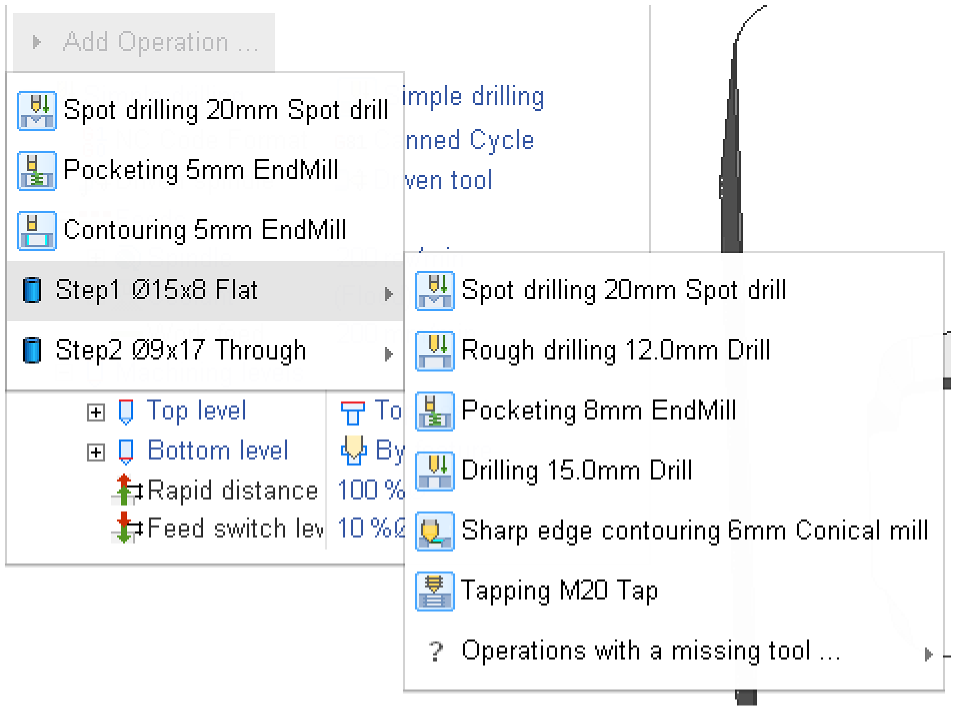
6.The tool selector. In this combobox you change the tool selected in the previous field. If the tool is used in several operations, all those operations are affected.



7.The Delete button.

The Add Operation… menu

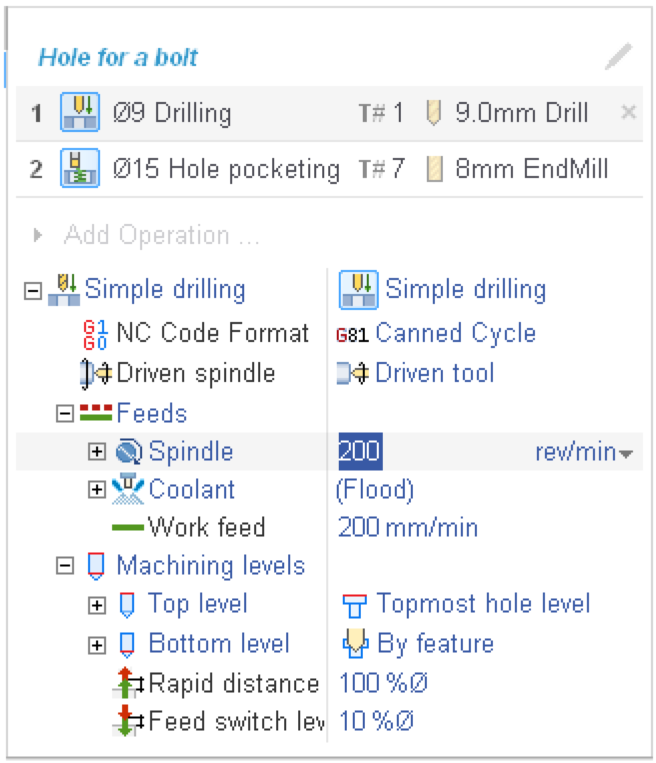
Under the operation list there is the Add Operation… menu.



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.

5.4.6 Operations generation parameters

SprutCAM X 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 SprutCAM X 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.

5.4.7 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.

5.5 Mill machining

Milling machines are used to mill flat and form surfaces, revolution solids, gears and the like workpieces of metal and other materials by the milling cutter. The machining is performed by cutter rotation in the machine spindle and the workpiece fastened on the table performs the feed motion either arcs or cuts.

The number of coordinates that are handled by the NC defines milling machining kind

There are some kinds of mill machining by number of continuous axes along that can be simultaneously moved tool:

- Operations for 2/2.5-axes milling .
- Operations for the 3-axes milling .
- Operations for 4-axes and 5-axes milling

You can create all of this kind of operations at SprutCAM.

See also:

Types of machining operations

Operations for the 2/2.5-axes milling

Operations for the 3-axes milling

Operations for the 4-axes and 5-axes milling

Multiply group

5.5.1 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 SprutCAM 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 curve millingoperation 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.

See also:

Mill machining

List of types of machining operationsMultiply group2D contouring

3D curve milling

Engraving operation

Pocketing

5.5.2 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.

See also:

Mill machining

Hole machining

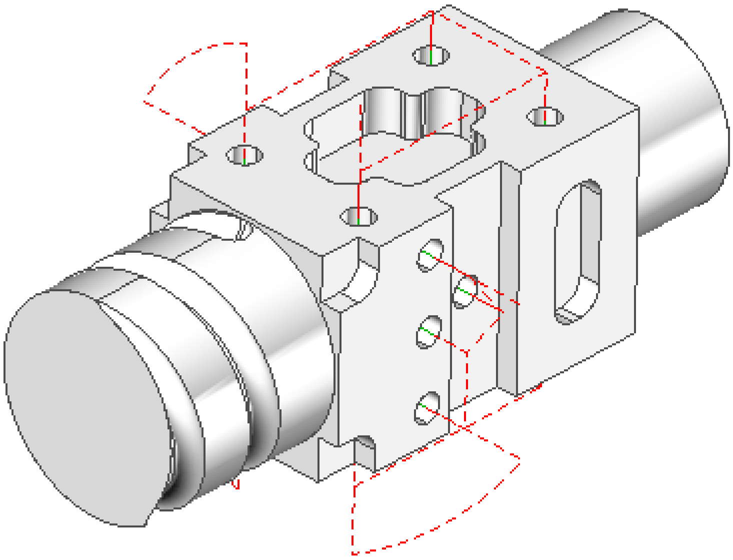
2D contouring

Engraving operation

Pocketing

2.5D machining operations

5.5.2.1 Hole machining operation



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 waterline roughingwaterline roughing operations. For this, the system will use either user defined drill points, or points generated automatically by the waterline operations. The list of holes can also be created automatically from a geometrical model using the holes recognition function.

For drilling the tool plunge points during pocketing and waterline-roughing operations it is necessary that when creating a hole machining operation the user define the operation prototype for which to perform the pre-drilling. The prototype operation will contain the list of drilling points and their depth for the hole machining operation.

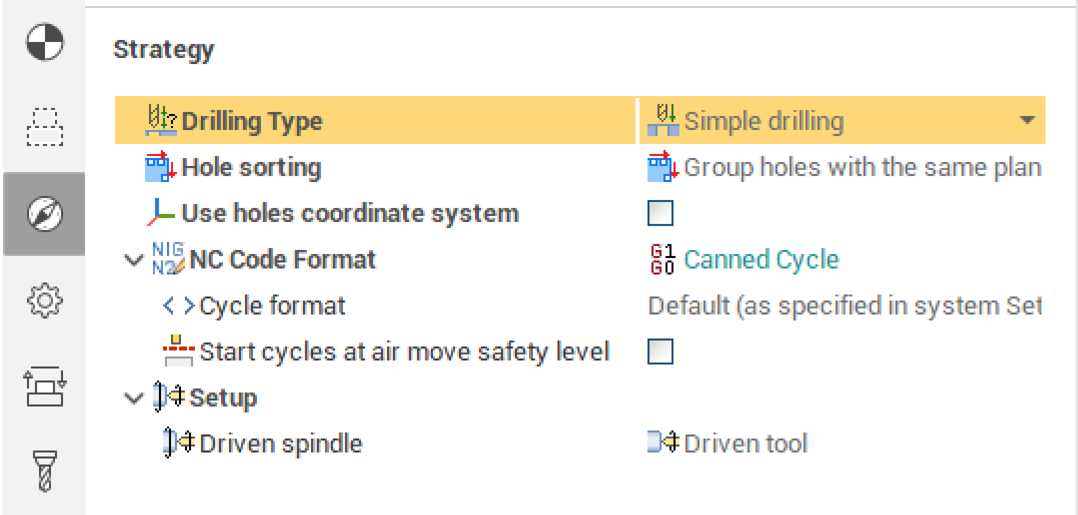
Holes can be automatically recognized, selected and added through graphic interface, specified by center points or by manual coordinate input. Coordinates for holes centers are assigned by points, which can be imported from files or defined in the 2D geometry mode. The list of points with their parameters (Z rapid height and drilling depth) is formed in the Job assignment window. In the same window the user can access the function of automatic recognition of holes in the model.

The hole top level and bottom level for every point can be assigned by the user or calculated automatically. When the top level is calculated automatically, it will be determined according to the workpiece model, and the bottom level definition based on the model being machined.

In most cases the diameter of the tool should be defined equal to the diameter of holes. And when machining holes by spiral and circular strategies, the diameter of the tool should be smaller than the hole diameter. All holes of an operation are machined using one tool and one cycle. To machine holes of different diameter or different types of cycles one should create several operations. Excluded from this are the spiral and circular machining options.

You can use the **operation properties inspector** to setup general operation parameters.

Hole machining mode and other additional parameters are set on the Strategy tab.



The dialog interface and parameter list change according to selected <Drilling type> option.

The order of the holes machining depends on the order in the job assignment list if the Holes sorting is set to By list. if **Optimal** is selected then the system minimizes the length of the idle motions. If Group holes with the same plane item selected then system will optimize machining order inside groups of holes that lie at the same planes.

If Use holes coordinate system is off then alll holes machined in the operation's Workpiece coordinate system (G54 – G59) or Local coordinate system. If Use holes coordinate system is on then the ORIGIN command is output before the each hole machining (or before each hole with changed plane). 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.

NC code format defines the way of the g-code output:

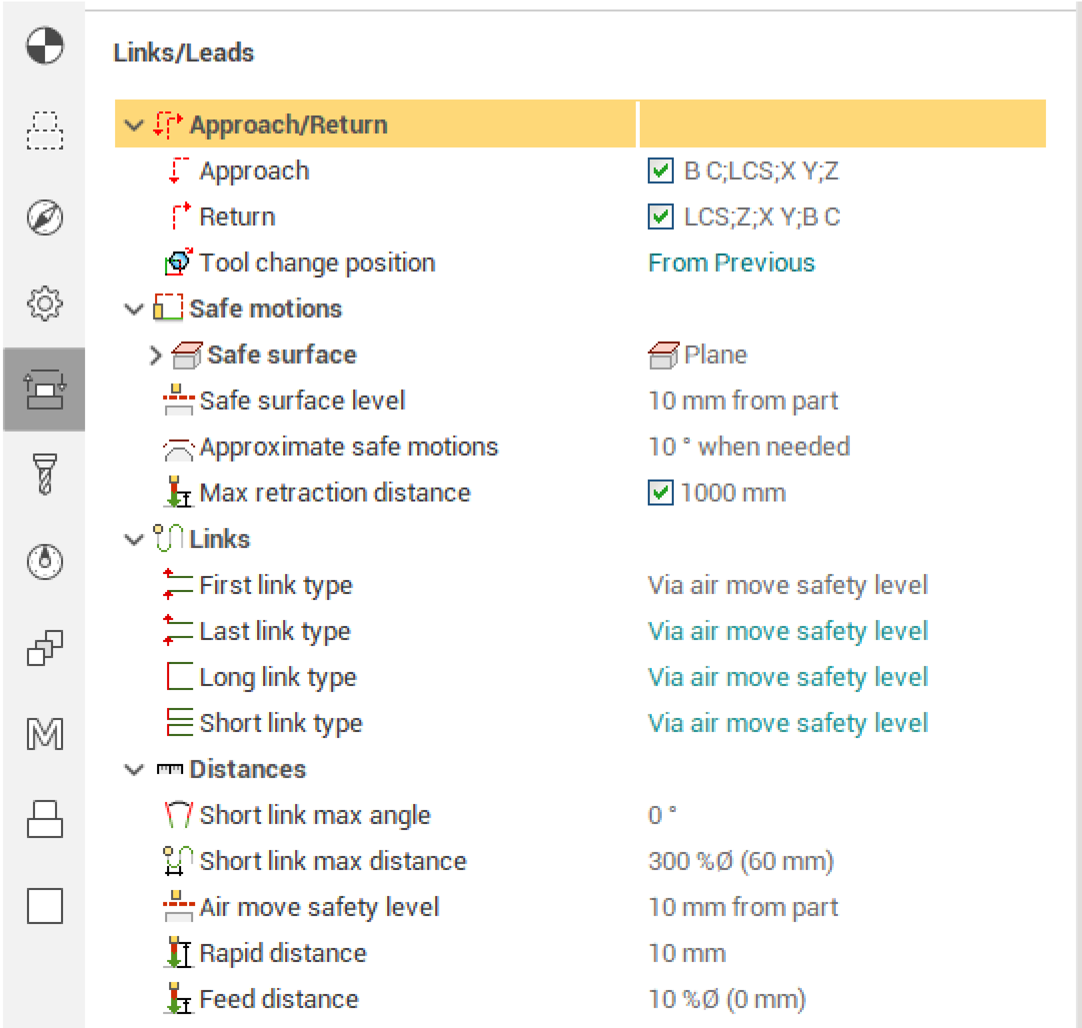
- Long Hang. All motions are output as the elementary movement commands (lines and circle arcs). Use this option for special cases then machine's CNC-system can't form canned cycle movements (for example, some CNC-systems do not support canned cycles at non-orthogonal planes).
- Canned cycle. The cycles is output. Every cycle contains a full set of motions to machine the hole within itself. The way of machining depends on the used CNC. See your CNC manual for the detailed information.

For compatibility with older versions of postprocessors the **Cycle format** property provides the ability to change the output format of the drilling cycle (when **Canned cycle** output method is used). It can have the following values.

- Default (as specified in system Setup). The cycle format will be used, which is specified in the system settings. 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.

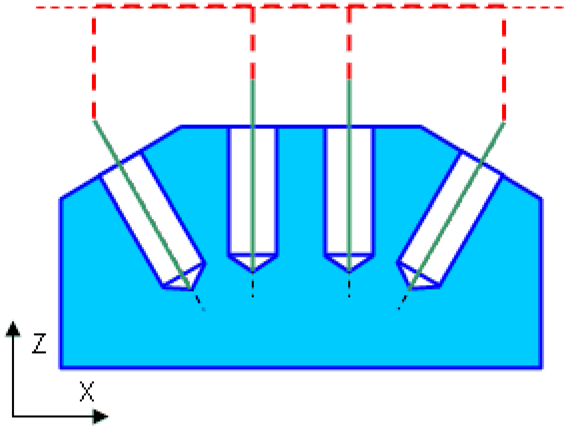
The **Start cycle at air move safety level** option defines the position at which the each cycle toolpath will start. If it is disabled the the first cycle point located at the **Rapid distance** from the top level of hole. If it is enabled then the first cycle point is at the **air move safety level** 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).

At the **Links/Leads** page you can flexibly adjust transitions between holes.

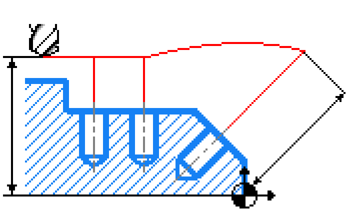
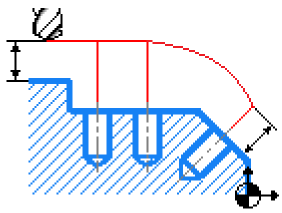


Using the **First link type**, **Last link type**, **Long link type** and **Short link type** parameters you can choose an appropriate way of transition separately for each case. There are the following values.

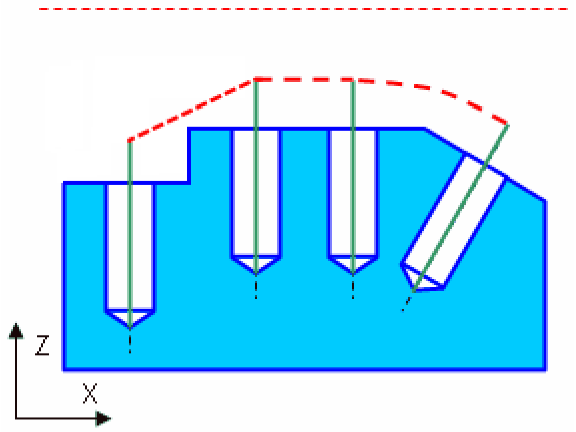
- **Via safe surface**. The toolpath continues until the hole axis intersects with the specified safe surface (plane, cylinder or sphere). If the intersection point goes far beyond the dimensions of the part, then outside the gabarites, the point is projected onto the safe surface at the nearest distance. You can explicitly set the **Max retraction distance** from the first cycle point.



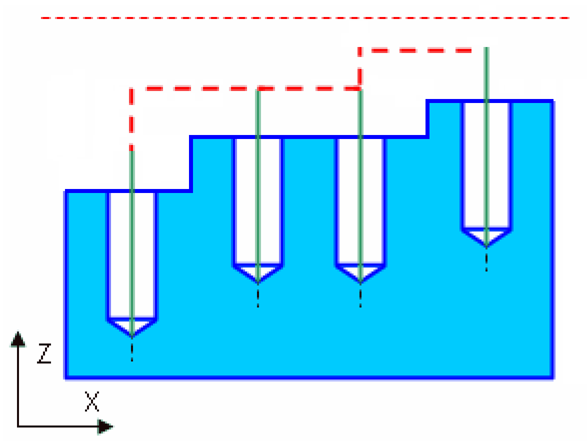
- **Via air move safety level**. The toolpath continues along the hole axis up to the **air move safety level**. The air move safety level can be definied as **absolute** distance from the origin of CS or **incrementally from the part** (plus workpiece and fixtures) top level.

Absolute safe level	Incremental safe level
	

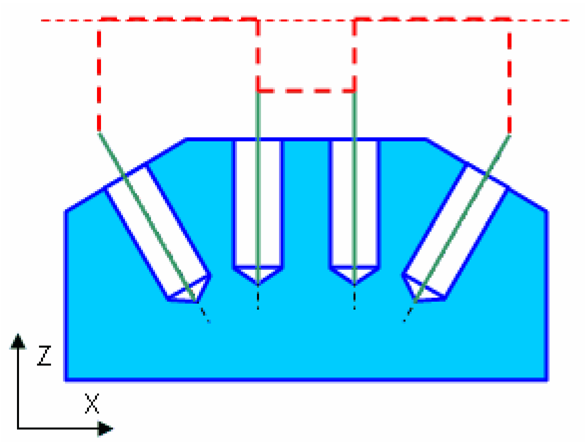
- **Straight**. Straight transition between holes' first cycle points (which are either at rapid distance a or at air move safety level depend on **Start cycle at air move safety level** option).



- **Orthogonal**. The same as the **Straight** transition but one intermediate orthogonal point appears if the holes are on different height.



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.



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.

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.

For the spiral (thread milling) and hole pocketing cycles it is possible to generate the g-code with the radius compensation. The radius compensation works like in other operations.

Use Feeds/Speeds tab to setup cutting conditions: spindle revolution rate, cooling, feed rates for different motion modes(approach, retract, work feed and the like). Auxiliary transitions (non-cut transitions) are performed either with rapid feed rate or with work feed rate, this option is controlled by the All non-cut feeds as rapid check-box. Work feed rate motion for non-cut transitions is useful then machining non-orthogonal plane holes as some NC-systems control only the start and end positions of the tool when performing rapid motions.

Multiply tool path by axis simplifies the machining of the repeated part holes. It works like in other operation.

See also:

Types of machining operations

Operation for 2/2.5-axes milling

The ways of the holes machining

The ways of the holes machining

Hole machining operations realizes the wide range of the holes machining cycles. The cycle selection is performed on the <Strategy> page of the parameters dialog.

The next cycles are supported:

- Drilling cycle (G81, W5DDrill(481))
- Drilling cycle (G82, W5DFace(482))
- Drilling with chip removing cycle (G83, W5DChipRemoving(483))
- Drilling with chip breaking cycle (G73, W5DChipBreaking(473))
- Tapping cycle, tapping with chip breaking and removing (G84, W5DTap(484))
- Drilling cycle (G85, W5DBore5(485))
- Drilling cycle (G86, W5DBore6(486))
- Drilling cycle (G87, W5DBore7(487))
- Drilling cycle (G88, W5DBore8(488))
- Drilling cycle (G89, W5DBore9(489))
- Thread milling cycle (W5DThreadMill(490))
- Hole pocketing cycle (W5DHolePocketing(491))

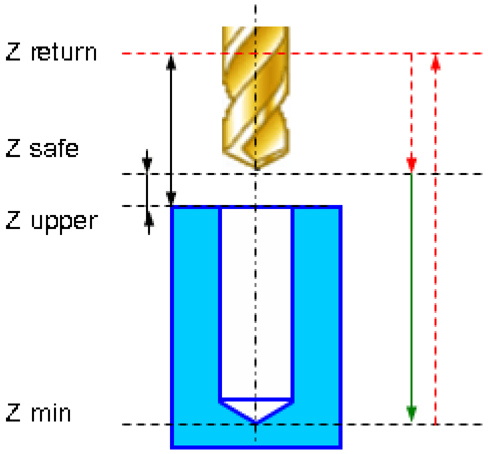
See also:

Types of machining operations

Hole machining operations

Drilling cycle (G81, W5DDrill(481))

Drilling cycle drills holes with rapid approach to the safe level and rapid retract the safe plane level.



Drilling cycle <G81> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- Rapid tool return to the <Z return> level.

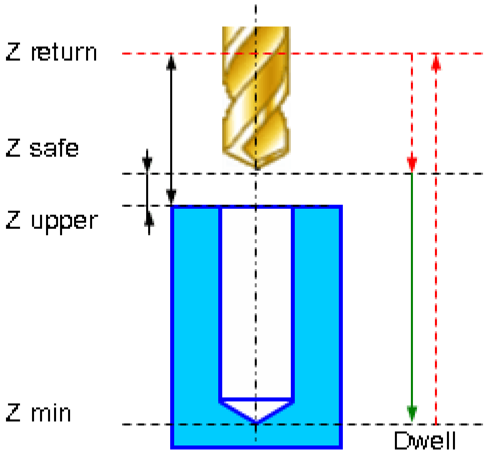
See also:

[Hole machining operation](#)

[The ways of the holes machining](#)

Drilling cycle (G82, W5DFace(482))

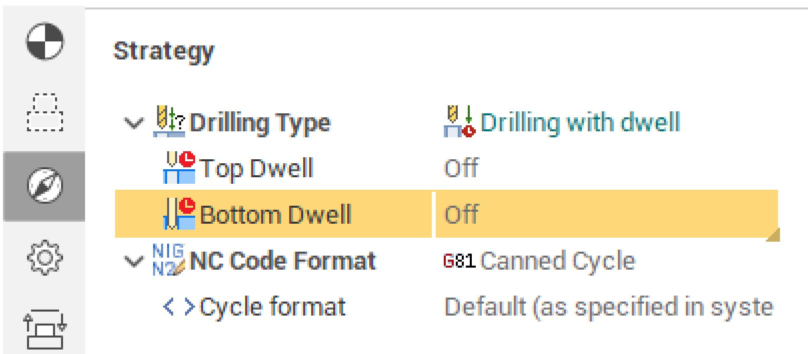
Drilling cycle drills holes with rapid approach to the safe level, dwell at hole bottom level and rapid retract the safe plane level.



Drilling cycle <G82> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- <Dwell> at the <Z min> level.
- Rapid tool return to the <Z return> level.

The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



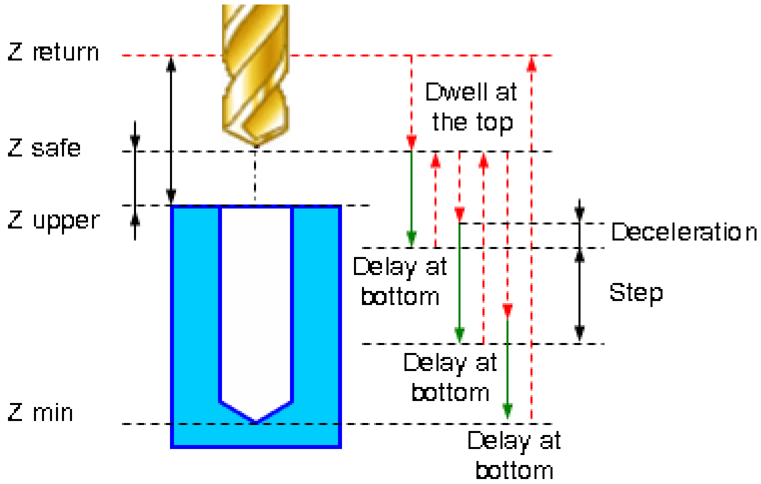
See also:

[Hole machining operation](#)

[The ways of the holes machining](#)

Drilling with chip removing cycle (G83, W5DChipRemoving(483))

Drilling with chip removing cycle performs tool motion to the hole center at the <Z return> level and consequent cyclic drill with tool retraction to the <Z safe> level.



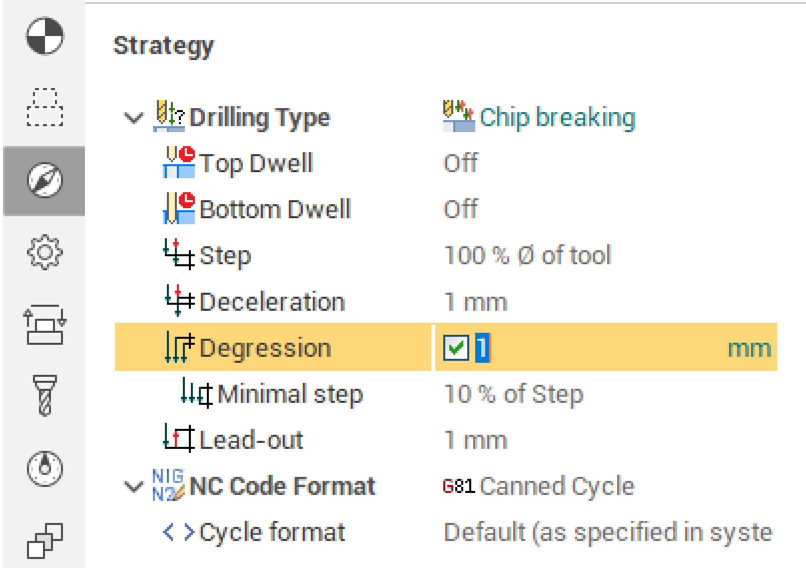
Cycle consists of:

- Rapid approach to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate motion to the <Step depth S>.
- Dwell for the <Delay at the bottom> time.
- Rapid return to <Z safe> level.
- Dwell for the <Dwell at the top> time.
- Rapid motion to the previous depth level, with a <Deceleration (Dcl)>.
- Work feedrate to the <Deceleration (Dcl)> with <Step (S)>.
- Dwell at for the <Dwell at the bottom> time.
- Repeat previous five iterations until the full hole depth is reached.
- Rapid tool return to the <Z retract> level.

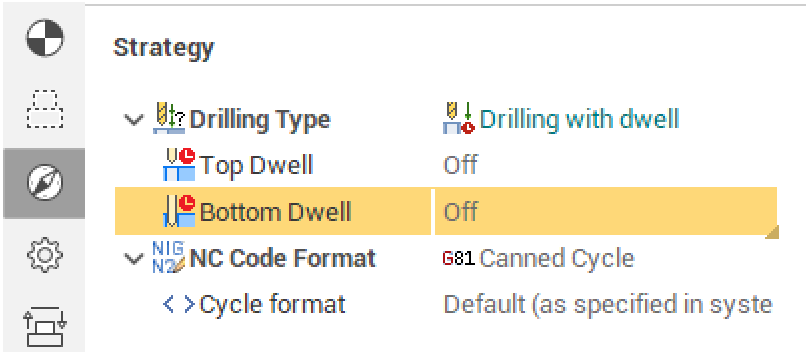
Chip breaking parameters panel defines the step and deceleration. The <Step> can be specified by different ways:

- <Distance>. The step is equal to the input value.
- <Count>. The value defines the quantity of the tool pecks. The step is calculated as the hole depth divided into the peck count.
- <Percent>. The step is specified in the percent of the tool diameter.

If the <Degression> is checked then the depth of every following peck is reduced on the defined value, else the step is constant. The step reduction occurs until its value is not less than <Minimal step>. Minimal step is a percentage of the first step value.



The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



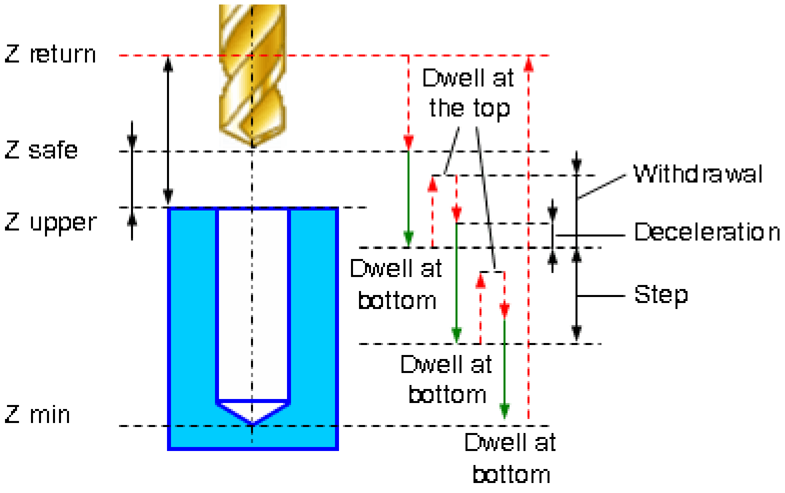
See also:

[Hole machining operation]

[The ways of the holes machining]

Drilling with chip breaking cycle (G73, W5DChipBreaking(473))

Drilling with chip breaking cycle performs tool approach to the hole center at the <Z return level>. When cyclic drilling is performed with tool retraction for chip breaking.



The cycle consists of:

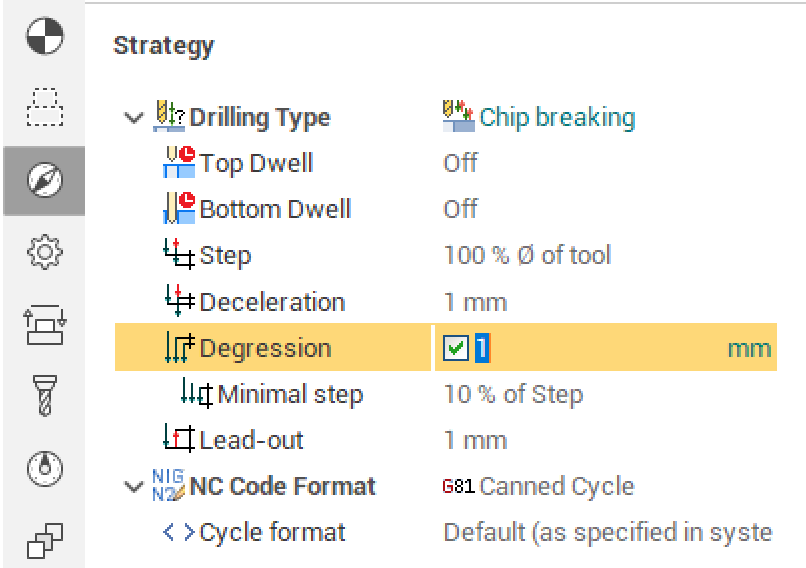
- Rapid approach to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate motion to the <Step depth S>.
- Dwell for the <Delay> at the bottom time.
- Rapid tool retraction for the <Withdrawal distance (Ld)>.
- Dwell for the <Dwell at the top> time.

- Rapid motion to the previous depth level, with a <Deceleration (Dcl)>.
- Work feedrate to the <Deceleration (Dcl)> with <Step (S)>.
- Dwell at for the <Dwell at the bottom> time.
- Repeat previous five iterations until the full hole depth is reached.
- Rapid tool return to the <Z retract> level.

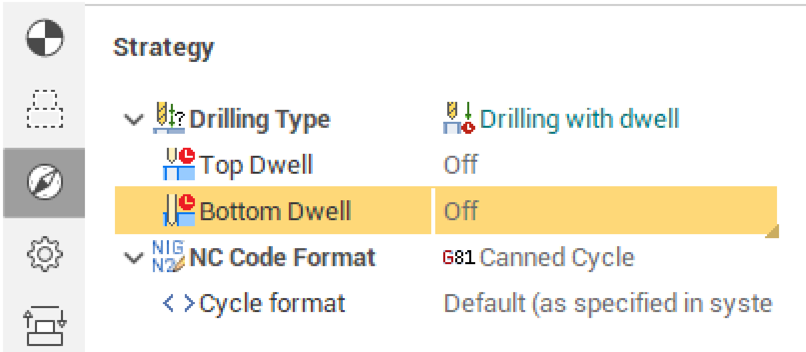
<Chip breaking parameters> panel defines the <Step>, <Deceleration> and <LeadOut>. The step can be specified by different ways:

- <Distance>. The step is equal to the input value.
- <Count>. The value defines the quantity of the tool pecks. The step is calculated as the hole depth divided into the peck count.
- <Percent>. The step is specified in the percent of the tool diameter.

If the <Depth degression> is checked then the depth of every following peck is reduced on the defined value, else the step is constant. The step reduction occurs until its value is not less than <Minimal step>. Minimal step is a percentage of the first step value.



The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



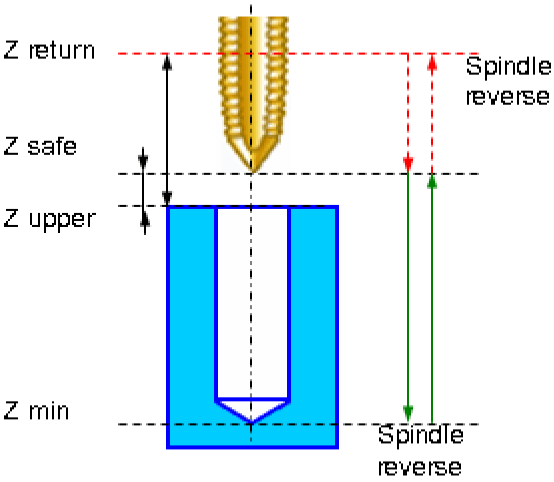
See also:

Hole machining operation

The ways of the holes machining

Tapping cycle, tapping with chip breaking and removing (G84, W5DTap(484))

Tapping cycle performs rapid approach to the <Z return> level, thread tapping with subsequent retraction at work feedrate with reverse spindle rotation.



<G84> tapping cycle includes:

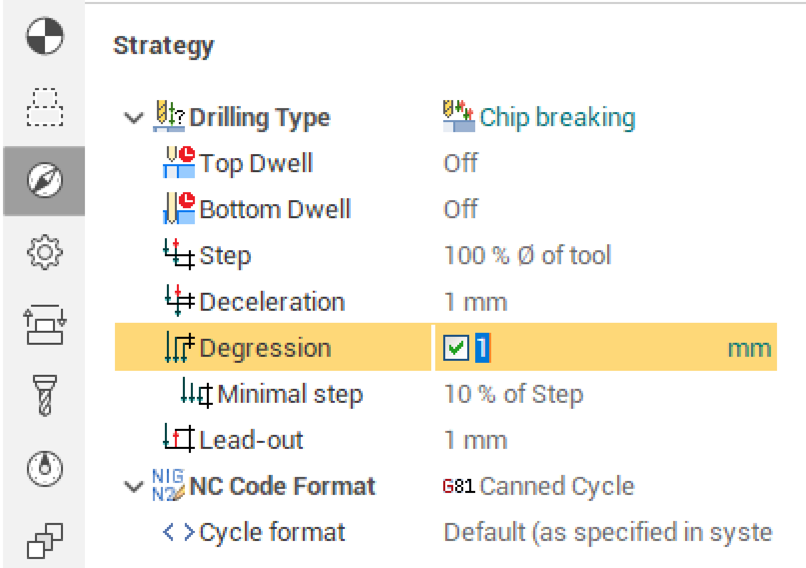
- Rapid approach to the hole center at the <Z return> level.
- Rapid travel to the <Z safe level>.
- Work feedrate motion to the <Z min> level and then <Spindle reverse>. If you select the tapping with chip removing or breaking strategy, the finish depth of the hole will be reached in several iterations.
- Work feed travel to the <Z safe> level.
- Rapid retract to the <Z return> level.
- Restore spindle rotation direction and speed.

The cycle parameters are defined in the <Thread parameters> panel. The <Thread pitch> defines the pitch in millimeters or inches. It depends on the current measurement units. The <Spindle position> is used for the multistart threads and defines the start <Spindle position> in degrees.

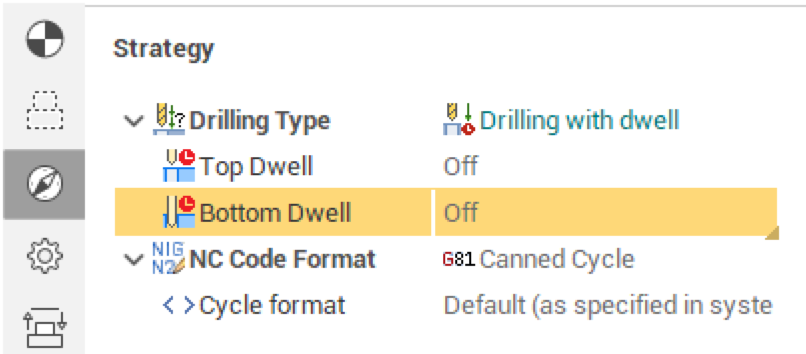
<Chip breaking parameters> panel defines the <Step>, <Deceleration> and <LeadOut>. The step can be specified by different ways:

- <Distance>. The step is equal to the input value.
- <Count>. The value defines the quantity of the tool pecks. The step is calculated as the hole depth divided into the peck count.
- <Percent>. The step is specified in the percent of the tool diameter.

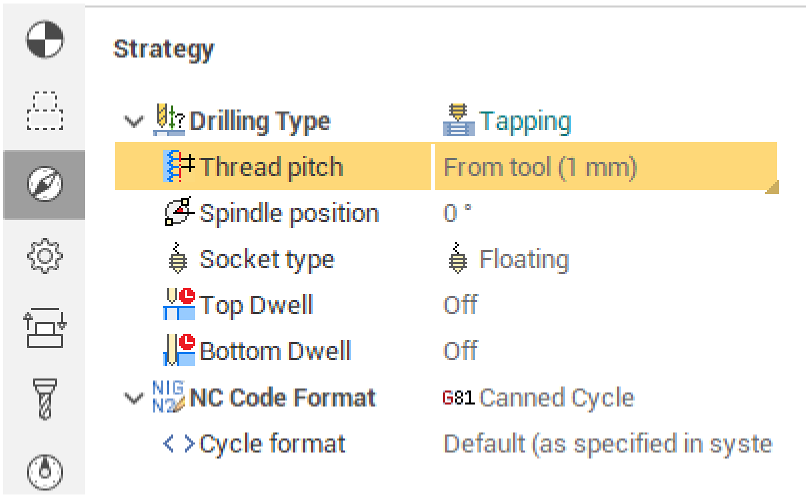
If the <Depth degression> is checked then the depth of every following peck is reduced on the defined value, else the step is constant. The step reduction occurs until its value is not less than <Minimal step>. Minimal step is a percentage of the first step value.



The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



Some numerical controls has different cycles for the different socket type. So the socket type can be defined as floating or fixed.



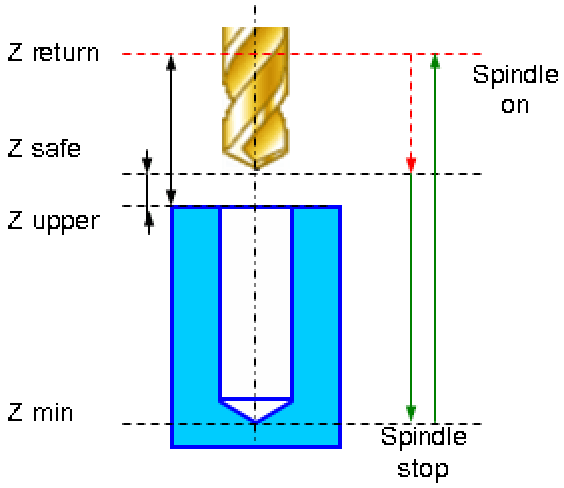
See also:

[Hole machining operation]

[The ways of the holes machining]

Drilling cycle (G85, W5DBore5(485))

Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and work feedrate retract to the <Return> level.



Boring canned cycle <G85> consists of:

- Rapid travel to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the <Z min> level.
- <Spindle stop>.
- Work feedrate return to the <Z return> level.
- Restore spindle rotation direction and speed.

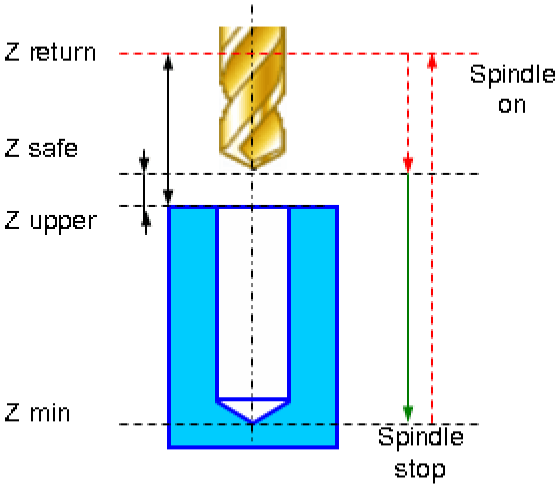
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G86, W5DBore6(486))

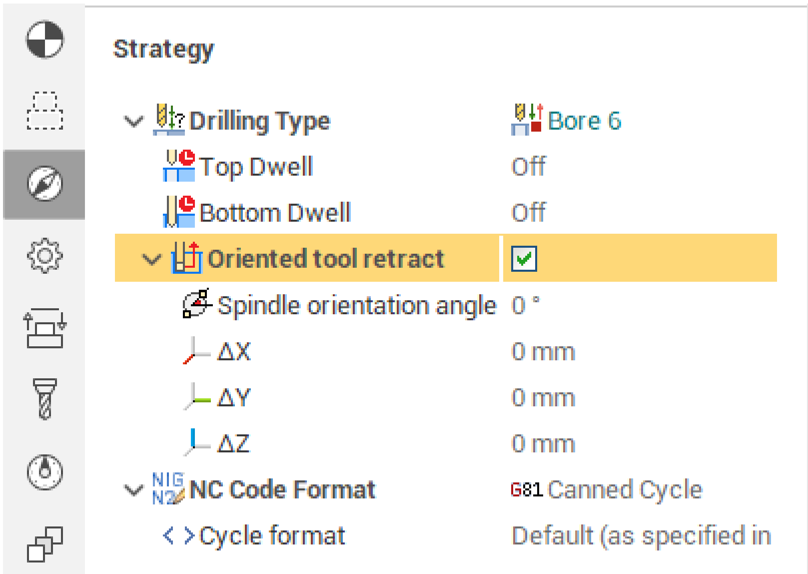
Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and rapid retract to the <Return> level.



Boring canned cycle <G86> consists of:

- Rapid travel to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the <Z min> level.
- <Spindle stop>. If <Oriented 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.
- Rapid return to the <Z return> level.
- Restore spindle rotation direction and speed.

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.



To edit the following parameters are available: the angle of the oriented spindle stop in degrees and the coordinates of tool offset after a stop.

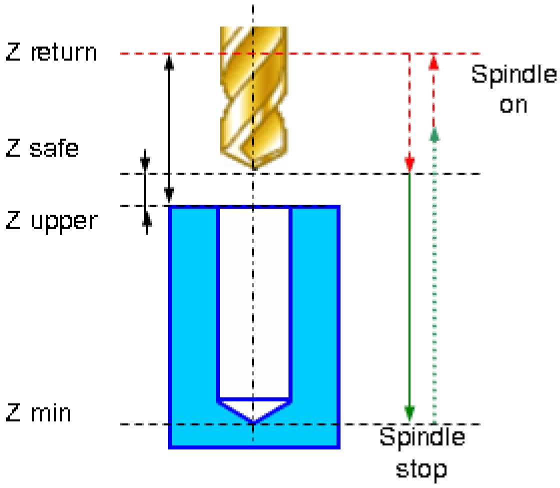
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G87, W5DBore7(487))

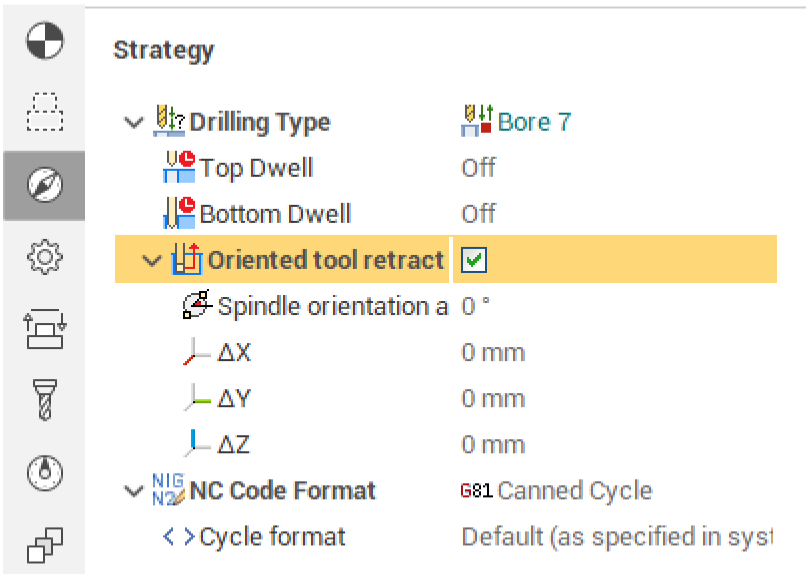
Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and manual retract to the <Return> level.



Boring canned cycle <G87> consists of:

- Rapid travel to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the <Z min> level.
- <Spindle stop>. If <Oriented 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.
- Manual retract to the <Z return> level.
- Restore spindle rotation direction and speed.

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.



To edit the following parameters are available: the angle of the oriented spindle stop in degrees and the coordinates of tool offset after a stop.

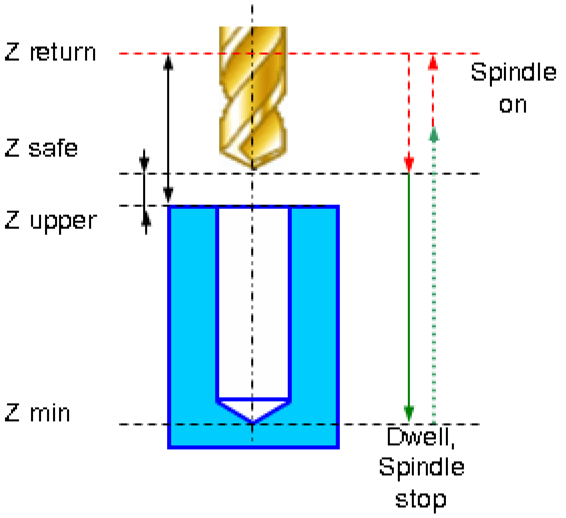
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G88, W5DBore8(488))

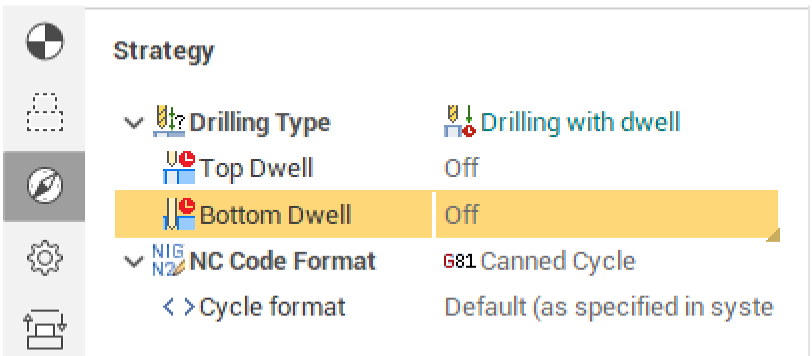
Drilling cycle type drills holes with rapid approach to the safe level, dwell at hole bottom level, spindle stop and manual retract to the safe plane level.



Drilling cycle <G88> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- <Dwell> at the <Z min> level.
- <Spindle stop>.
- Manual tool retract to the <Z return> level.
- Restore the spindle rotation direction and speed.

The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



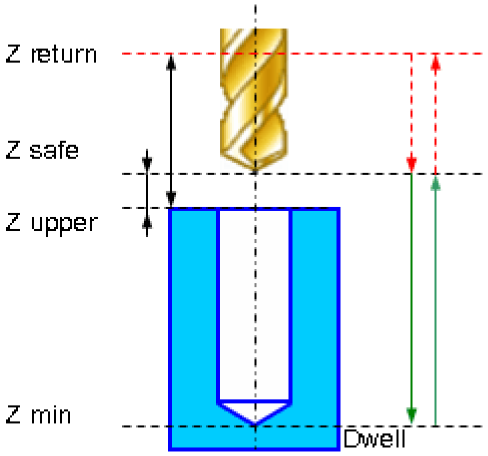
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G89, W5DBore9(489))

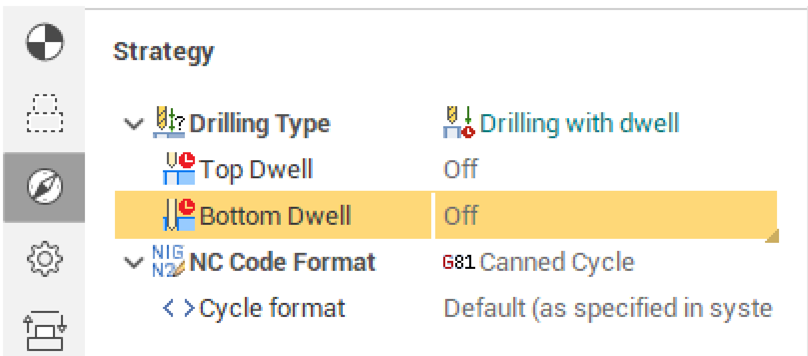
Boring cycle type bores holes with rapid approach to the safe level, dwell at hole bottom level, spindle stop and manual retract to the safe plane level.



Boring cycle <G89> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- <Dwell> at the <Z min> level.
- Work feedrate return to the <Z safe> level.
- Restore the spindle rotation direction and speed.

The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.

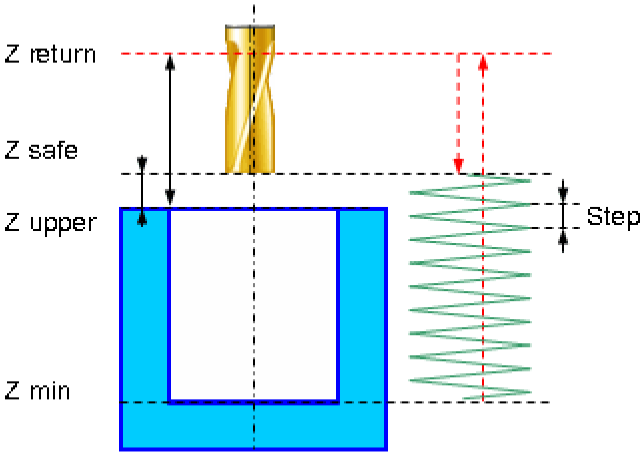


See also:

Hole machining operation

Thread milling cycle (W5DThreadMill(490))

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.

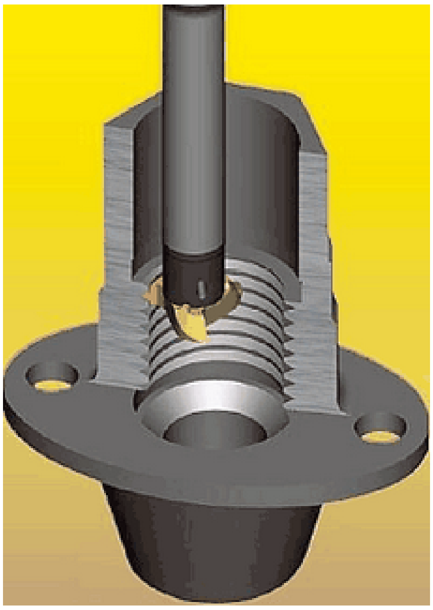


Spiral machining includes the following steps:

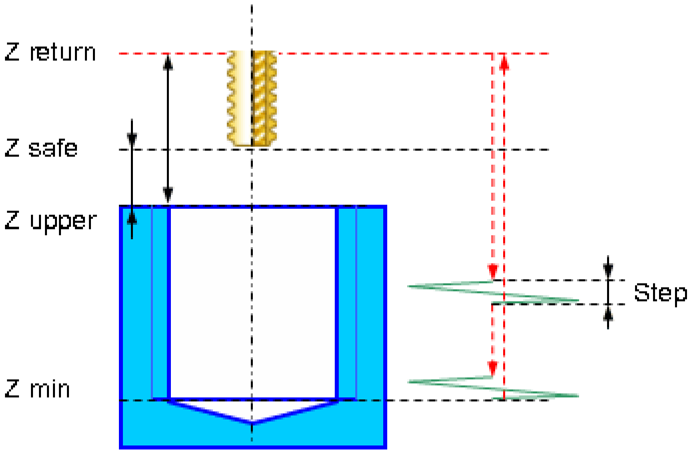
- Rapid approach to the hole center at the <Z retract> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the spiral start.
- Work feedrate spiral motion to the <Z min level>.
- Optional circular pass on the bottom level. Circle diameter equals spiral diameter.
- Return to the hole center.
- Rapid travel to the <Z safe> level.
- If additional roughing and finishing passes are applied previous five steps are repeated until desired hole diameter is reached.
- Rapid travel to the <Z retract> level.

<Threadmilling> provides the following advantages over traditional tapping:

- blind и through, left and right threads are machined by the same tool;
- different threads with the same pitch are machined by the same tool;
- all precision parameters are secured by the same tool;
- accurate threading is machined to the full depth of the blind hole as the mill has no chamfer;
- different materials are machined by the same tool;
- high reliability of the machining because of good chip handling;
- high efficiency of threadmilling due to higher cutting speed and feedrate;
- low spindle torque even for coarse thread machining.




For threadmilling both single-cutter tools and multi-cutter ones allowing to machine several thread turns in one pass. Multi-cutter tool machining is much similar to spiral machining.

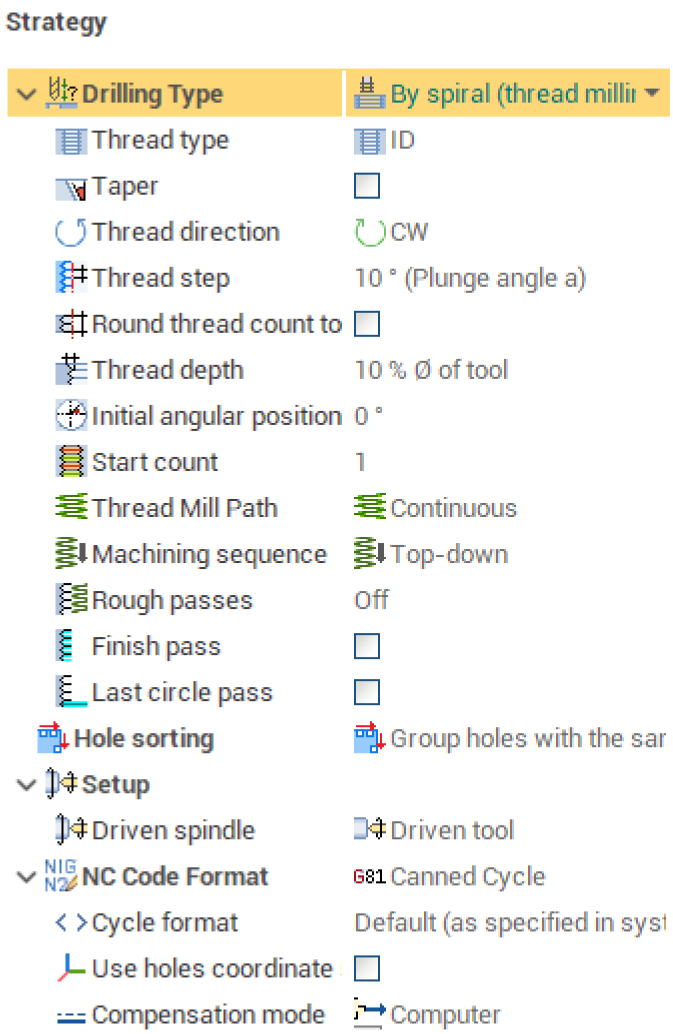


When using the multi-cutter tool threadmilling machining includes the following steps:

- Rapid approach to the hole center at the <Z retract> level.
- Rapid travel to the <Z safe> level.
- Rapid travel to the tool cutting edge length distance which is determined by the number and size of the mill tooth size (thread pitch).
- Work feedrate travel to the start of the spiral.
- Machining along one spiral turn with step equaling thread pitch.
- Retract to the hole center.
- If one spiral turn is not enough to machine the threading to the full hole depth descend to the cutting-edge length and spiral motion are repeated until desired threading depth is reached.
- Rapid return to the <Z retract> level.

If additional roughing and finishing passes are applied then the above steps are repeated until specified thread depth is achieved.

The spiral parameters panel defines the parameters for the spiral hole machining and thread milling.

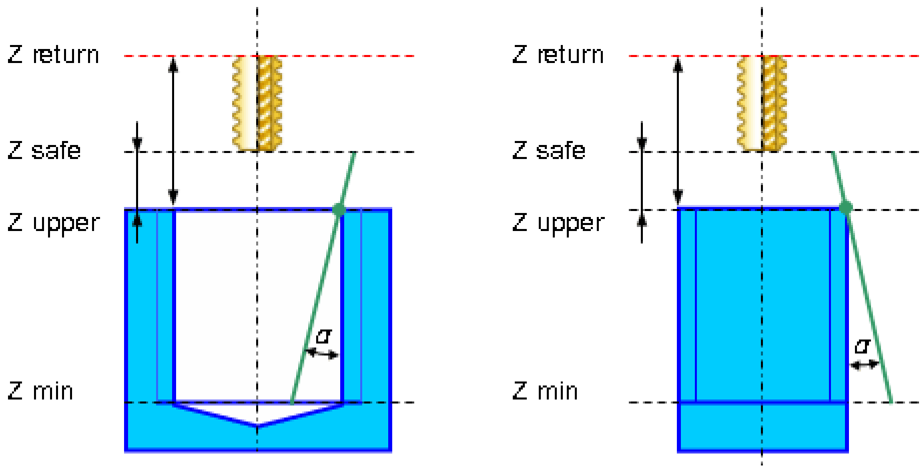


<Thread type> parameter specifies whether the threading is <External> or <Internal>.

There are cases when technology requires that threading is machined upside down and there are cases when threading is done from the bottom to the top. it defines in the <Machining sequence> field.

Thread kind, right or left, is determined by the <Thread spiral direction> parameter. For spiral machining it is convenient to define the spiral direction according to the spindle rotation direction. When <Follow> direction is specified the tool rotation and the spiral directions coincide, for the <Counter> direction they are opposite. The tool rotation is defined on the <Tool> page of the operation parameters dialog.

To machine the conical threads it is need to set <Taper angle> tick and specify the conic angle in degrees. The taper angle is measured from the top level of the hole (lug). Positive angle direction for tapered thread machining of the hole is the direction to the center of the hole. Positive angle direction for tapered thread machining of the lug is the direction from the center of the lug.



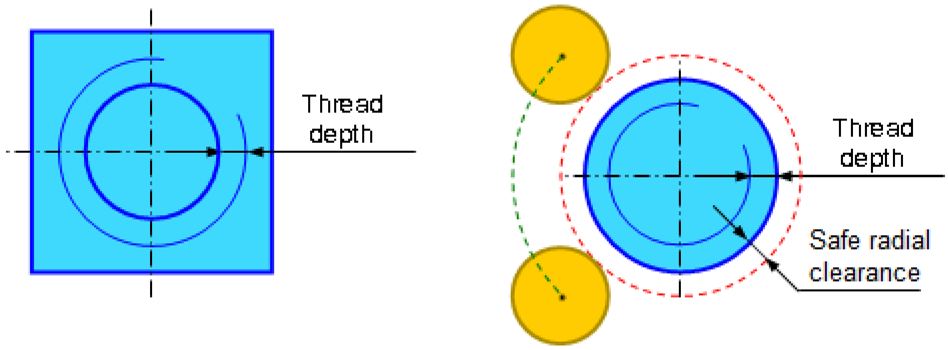
The <Spiral step> defines the spiral step for the spiral hole machining or the thread pitch in the case of the thread milling. If the <Plunge angle> mode is selected then the step is calculated with the using of the spiral angle, tool and hole diameters.

<Round turns count to integer> can be very useful for the spiral hole machining. If it is set then the step is recalculated to generate the integer coils number. The coil numbers is rounded to the nearest value to provide the required step. This option can not be used for the thread milling because it approximates the step.

<Last circle pass> specifies whether the circle motion is performed when the bottom of the hole is reached. if it is set then additional pass along the circle is performed on the bottom of the hole. The circle radius is equal to the spiral radius. This option must be disabled for the thread milling.

Multi-start thread is machined if <Thread start count> parameter is greater than 1. If start count is 1 single-start thread is machined.

The <Thread depth> defines the distance between the inner and outer diameters of the thread. It works differently for the inner and outer threads. If hole is machined (inner thread), then the diameter that is defined in the job assignment defines the inner diameter of the thread. The outer diameter is calculated as the sum of the inner diameter and the thread depth. If the boss is machined (outer thread) then the diameter that is defined in the job assignment defines the outer diameter of the thread. The inner diameter of the thread is calculated as the difference between the outer diameter and the thread depth.

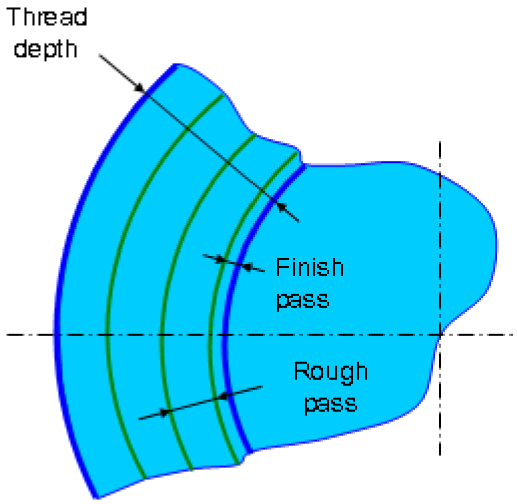


The <Thread depth> can be specified in current measurement units (mm or inch) on in the percents of the tool diameter.

In addition, for the correct machining of the external (OD) thread, it is need to specify the safe radial clearance value. It defines the radius at which the tool could safely bypass the boss if necessary.

The thread milling can be performed in a few passes. Switch on the rough or/and finish passes to do it. If the <Finish pass> is checked the field near defines the stock for the finish pass. in this case the additional pass is generated before the last pass. The thread depth remained after the subtraction of the finish pass can be removed by rough passes. The step for the rough passes can be defined by a few ways:

- <Off.>. the rough passes are not performed. So the all stack is removed in one pass.
- <Distance>. The step is defined by the absolute value (mm or inches).
- <% D of tool>. The step is defined in the percents of the tool diameter.
- <Count>. The step is specified by the pass count. In this case the step is equal the thread depth divided into the count.

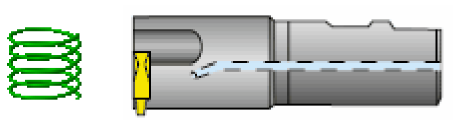


The <Lead in / Lead out> field allow to select the way of approach to the starting point or retract from the end point. The option has following items:

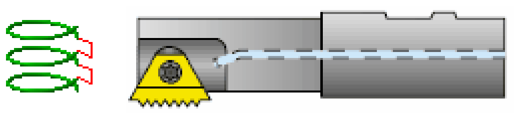
- <Direct>. For the case of internal machining the approach is direct from the center of the hole to the beginning of the working pass. In the case of external machining approach implemented by a straight line, which starting point is at the safe radial clearance distance from the outer diameter of the thread.
- <By arc>. Approach by arc allows to get a smoother start and end of thread. Arc radius and angle can be specified in the appropriate fields at the same panel. Note that the radius can be negative. It may be necessary for smooth plunging in the case of external machining, if you want to get the curvature direction of the arc coincides with the curvature direction of toolpath.

<Path type> parameter is used defines the toolpath type according to the used tool type. It can be one of the following:

- <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.

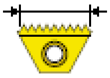


- <Transition along the axis>. This type is used for multi-cutter tool which forms multiple thread turns for one turn of the trajectory spiral. The trajectory consists of subsequent spiral turns connected with rapid cutting-edge long transitions along the spiral axis.

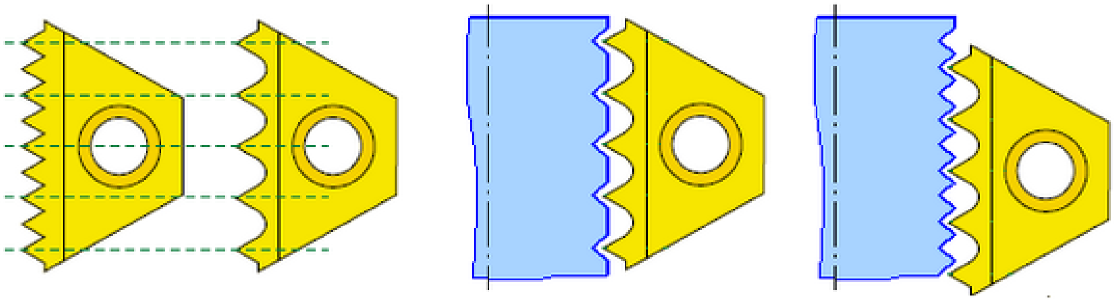


It is need to use the continues path type for the spiral hole machining cycle

<Cutting edge length> is used only for <Transition along the axis> toopath type. 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.



Fine pitch threads are threads with small pitches. It is difficult to produce multitooth inserts for small pitches because of the small radius between the teeth. It is developed inserts where every second tooth was dropped to enlarge the radius between the teeth. In this cases the tool needs to make a few coils. For example if the insert pitch is greater in two times than the thread pitch then two coils must be performed. The turn count defines it. In the most cases it's one, that means the insert pitch is equal to the thread pitch.



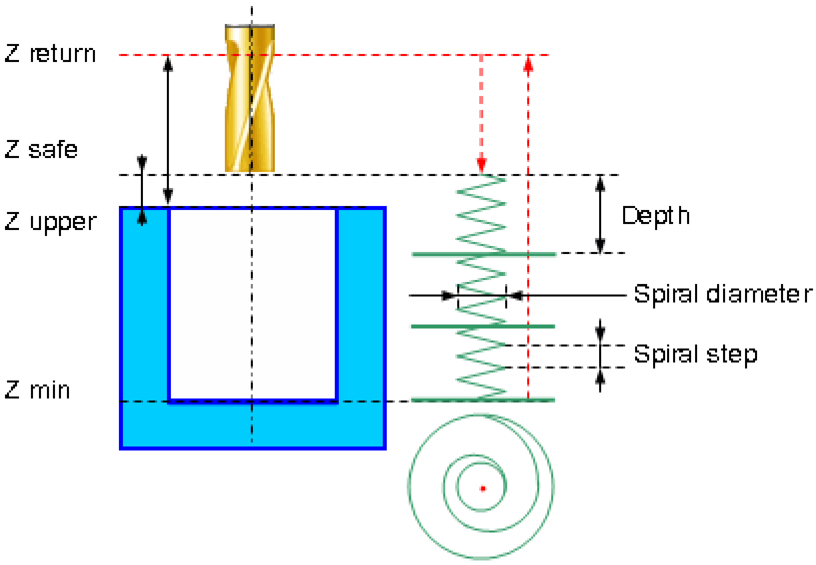
See also:

Hole machining operation

The ways of the holes machining

Hole pocketing cycle (W5DHolePocketing(491))

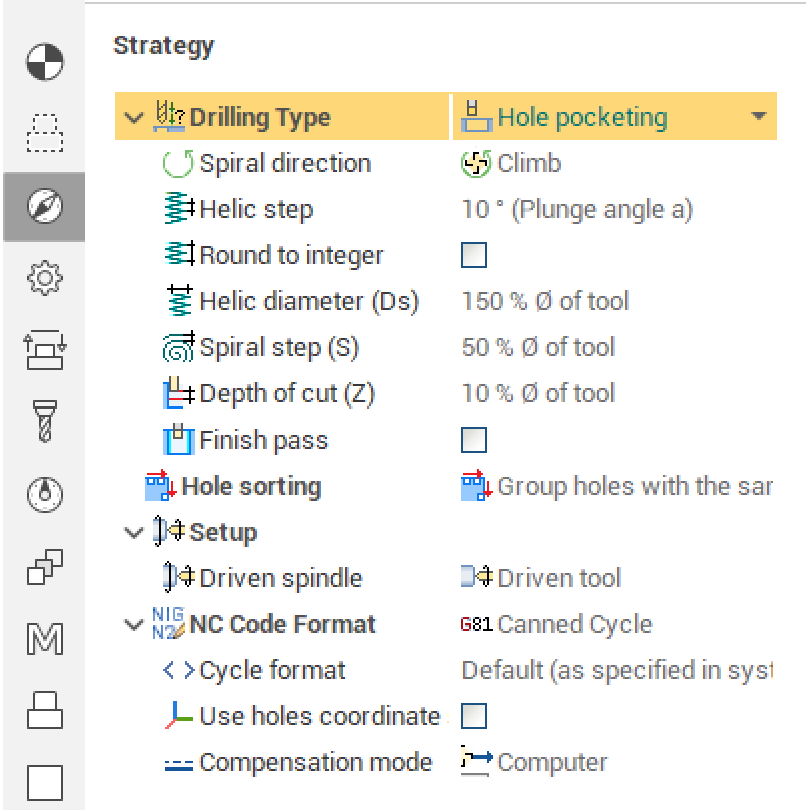
The cycle is used to machine holes which diameter is greater than the tool diameter. The pocketing is performed by layers. The tool cuts in along a spiral to each layer and then expands the hole to the desired diameter by moving along Archimedes spiral with finishing pass along the circle. The Archimedean spiral is approximated by the circle arcs.



Hole pocketing cycle includes the following:

- Rapid approach to the center of the hole at the <Z retract> level.
- Rapid motion to the <Z safe> level.
- Work feedrate spiral cut-in to the <Z machining depth>. Spiral diameter <Ds> is specified in the percents of the tool diameter. The plunge is defined by the Angle <a> or Step <Hi>.
- Archimedes spiral with <Step S> 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.
- Rapid travel to the <Z retract> level.

Pocketing parameters panel defines the parameters of the hole pocketing cycle.



Spiral direction defines the torsion direction. It can have the next values:

- <Right>. The Spiral is twisted right. The tool is rotating clockwise if watched from above.
- <Left>. The spiral is twisted left. The tool is rotating counter clockwise if watched from above..
- <Counter>. Spiral twist direction is determined by the spindle rotation direction and corresponds to the up cutting milling. When counter pocketing milling tool rotation direction and spiral direction are opposite to each other.
- <Follow>. Spiral twist direction is determined by the spindle rotation direction and corresponds to the down cutting milling. When counter pocketing milling tool rotation direction and spiral direction are coincident.

Note: The direction of the tool rotation is defined on the **Tool** page of the operation parameters dialog.

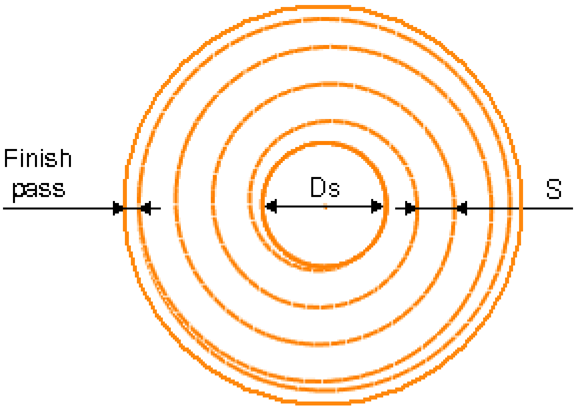
The spiral step is defined in the field with the same name in the Pocketing parameters panel. If the Plunge angle (a) is selected in the box the step value depends on spiral radius and specified as the plunge angle in degrees. If the distance is selected then the value is a step in the current measurement units (mm or inches).

The round turn count to integer recalculates the step value to get the integer value of the coils.

The depth of cut (Z) defines the distance between the horizontal layers. It can be specified by different ways:

- <Distance>. The depth is defined as absolute value in the current measurement units (mm or inch).
- <% D of tool>. The depth is defined as the percents of the tool diameter.
- <Count> The value defines the layers quantity. So the step is calculated as the hole depth divided into the layers count.

The plunge spiral diameter (Ds % of tool) is specified in the percents of the tool diameter. The Archimedean spiral step (Ds % of tool) is specified in the percents of the tool diameter also.



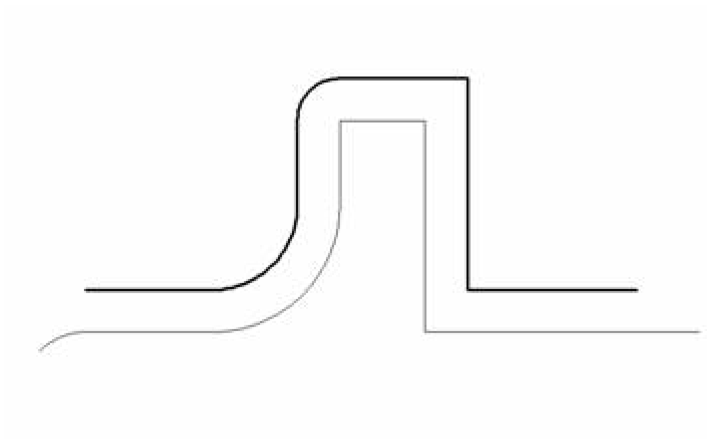
If the finish pass is enabled then the additional circle pass is generated before the final pass on every layer. The value near the Finish pass defines the stock for the final pass in current units (mm or inch). This feature allow to remove the equal stock of final pass.

See also:

Hole machining operation

The ways of the holes machining

5.5.2.2 2D contouring



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.

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.

Machining is performed in a series of horizontal passes of the tool. The passes differ from each other in the Z depth they are located at. The number of passes and their depths by Z depend on machining levels and the step defined on the parameters page. It is also possible to define a different Z depth for the last pass.

In the same window the user may define the machining tolerance and the stock. For contours, which are machined from left or right, the stock is laid off towards the tool, and when machining using the tool center it is ignored.

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 pass es 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>, <with return>.

When setting the machining order <By Contours> each contour will be machined to full depth before moving to another contour. When setting <By Depth>, each contour will be machined at the current cutting depth, the tool will move to the next depth only after all contours are machined.

Selecting <Idling Minimization> optimizes the order that contours are machined in. When deselected the contours are machined in the order that they appear on the model page.

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.

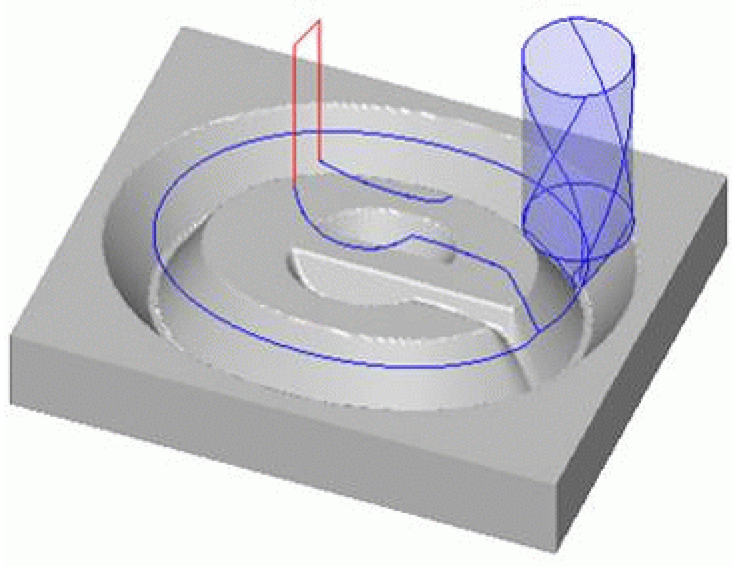
Note: *If you need to dictate the order of contour machining and the direction of their machining, then it is recommended to turn off the <**Idling Minimization**> mode and restrict the use of <**Reverse direction**> this will ensure that the order and the direction of machining will correspond to the order defined on the <Model> window.*

See also:

Types of machining operations

Operation for 2/2.5-axes milling

5.5.2.3 Engraving operation

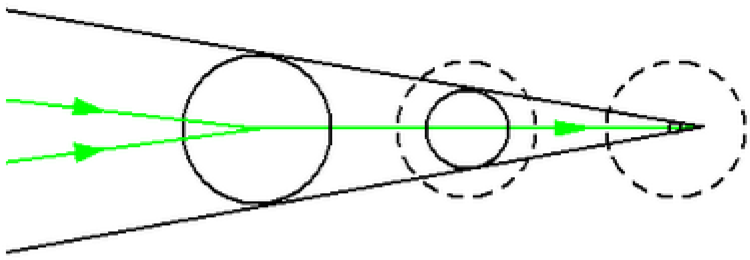


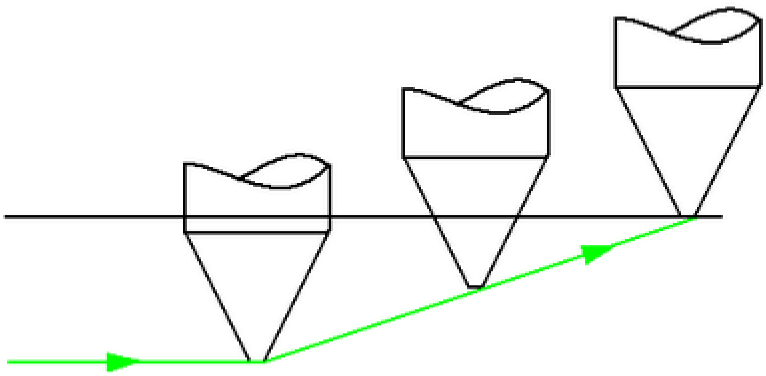
This operation is designed for the engraving of drawings and inscriptions on flat areas, and also for performing a finish pass along the side walls of pockets and contours using 2 / 2.5D machining.

The model being machined is formed from 2D curves in the horizontal (XY) plane. The model is created by the addition of curves or groups of curves into the resulting model. Any curve can define a scallop, groove or inverse (reversed) curve of a defined thickness, besides this, closed curves can be added as ledges, hollows or inverse areas. Additional stock, which will be added to the operation stock, can be assigned separately for every curve or a group of curves. Machining is performed along the outer contour of the created model with regard to the defined side angle (i.e. the edge of the model is not always vertical).

Only those areas, which lie within the workpiece or outside of restricted areas, will be added to the resulting toolpath. The workpiece, machining areas and restricted areas are defined by projections of closed curves. If the workpiece or restricted areas are not defined, then machining of the entire model will be performed.

Horizontal passes of the tool are used to form the main edge of the model using the defined step between passes. In order to form sharp inner corners and for machining of smaller width areas it is advised to use the 3D clearance option. This means that when working with a profiling tool, the diameter of which decreases towards the end point, machining of more "narrow" areas with simultaneous increasing Z value is possible.





If the operation is performed in the local coordinate system or if using a swivel head, then the model being machined is formed from curve projections onto the horizontal plane of the local coordinate system, the main work passes are parallel to the same plane, and during 3D area clearance the tool will be raised to the appropriate value along the Z axis of the local coordinate system of the operation.

The joining order of separate passes depends on the defined machining direction (upwards or downwards). Step-over between passes can be performed along a contour, with generation of intermediate approaches/retractions or via the safe plane.

See also:

Types of machining operations

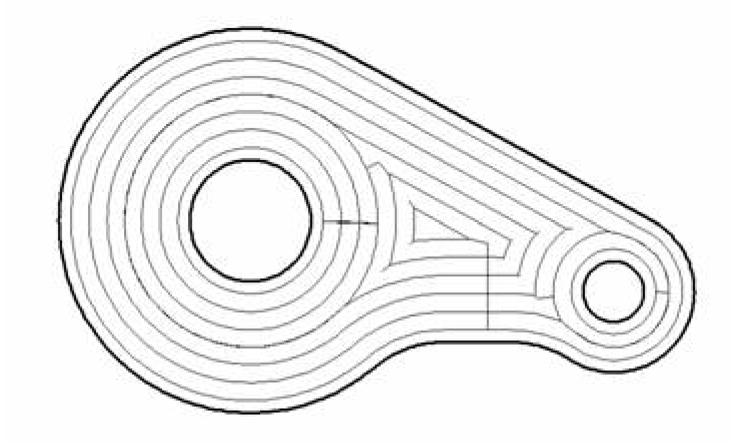
Operation for 2/2.5-axes milling

4-axis milling with using of the engraving and pocketing operations

Using design features in an Engraving/Pocketing operation

Job assignment for engraving and pocketing operations

5.5.2.4 Pocketing



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. Any curve can define the scallop, groove or inverse curve of a defined thickness; closed curves can also be added as ledges, hollows or inverse areas. Additional stock, which will be added to the operation stock , can be assigned separately for every curve or a group of curves. Side edges of the model are not always vertical; the angle of its slope is defined by the value of the side angle. This allows using the pocketing operation for rough material removal before the engraving operation.

The operation performs removal of the entire material, which lies within the model being machined and outside any restricted areas or a workpiece. The workpiece, machining areas and restricted areas are defined by a projection of curves.

The material is removed layer by layer, with assigned step between the layers . Depending on the defined strategy , the material of every layer can be removed by spiral paths, starting from the center working out or from the outside working in. Area clearance using parallel moves can also be used. Transition to the next machining layer can be performed using any of the plunge methods (axial, by spiral, zigzag), or via drill points. A search for an appropriate diameter and depth hole will first be made in the list of operation holes, and then in the open list of holes for the machining process. If no appropriate hole is found, then the system will select coordinates for the hole automatically, using optimum settings. Where possible the coordinates for the center of the new hole, are rounded. If when the operation is created, the hole machining operation was chosen as its prototype, then the list of holes will be copied into the operation and used when searching for appropriate hole for tool plunging.

When using a local coordinate system or a rotary axis , the model being machined is formed from curves projected onto the XY plane of the local coordinate system , machining layers are parallel to the same plane.

If using a profiling tool, the diameter of which gradually decreases towards the end point of the tool (e.g. engraving), then it is possible to use the 3D clearance option for more accurate creation of the side surface of the model simultaneously with material removal.

See also:

Types of machining operations

Operation for 2/2.5-axes milling

5.5.2.5 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.

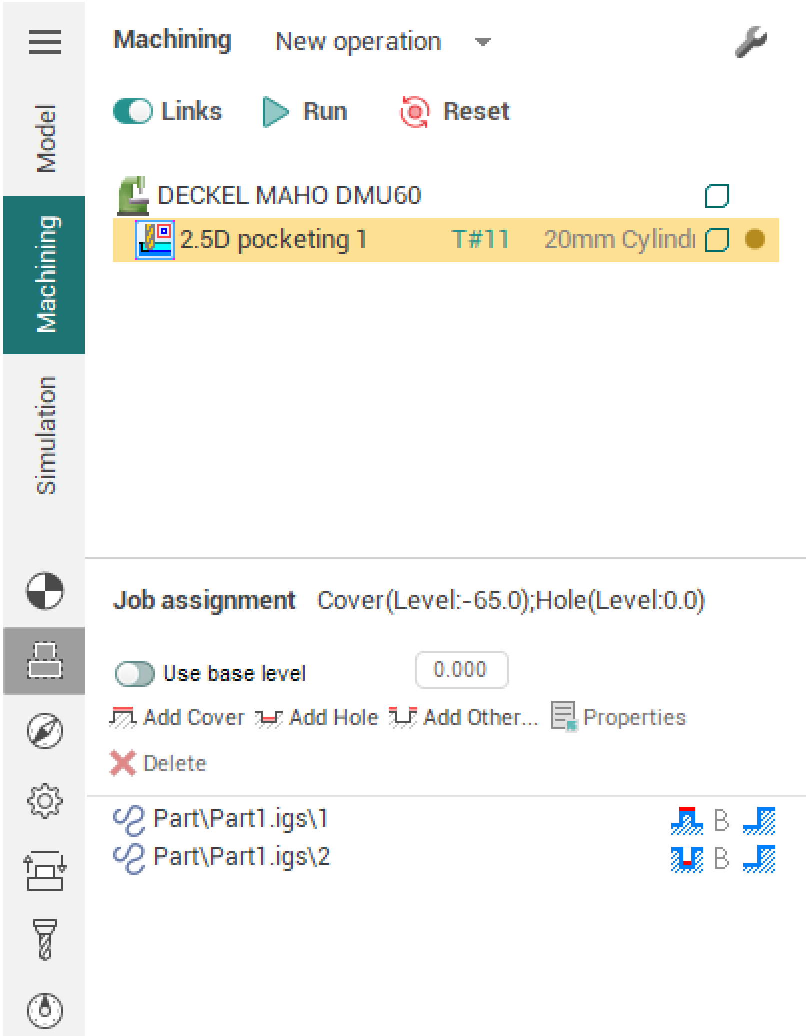
In the 2.5D operations, the system allows the user to visually create the geometry of a model by using flat profiles. If the parameters of a profile are changed, the displayed model is updated automatically. The operation processes the list, which consists of an arbitrary number of profiles and curve 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.

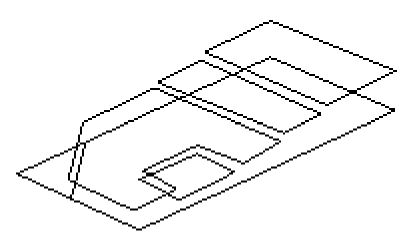
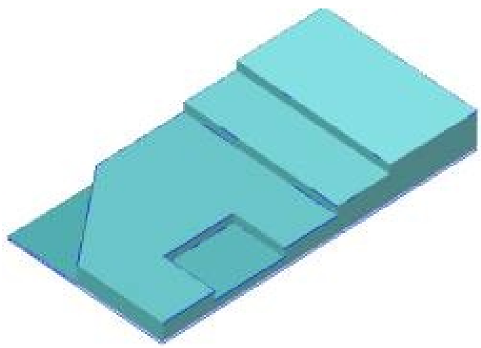
Machining is performed by a series of horizontal passes of the tool. The passes differ from each other only by the depth at which they are located. The number of such passes and their Z depth depend on the machining levels and the step defined on the parameters page in the <Operation parameters> window.

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.



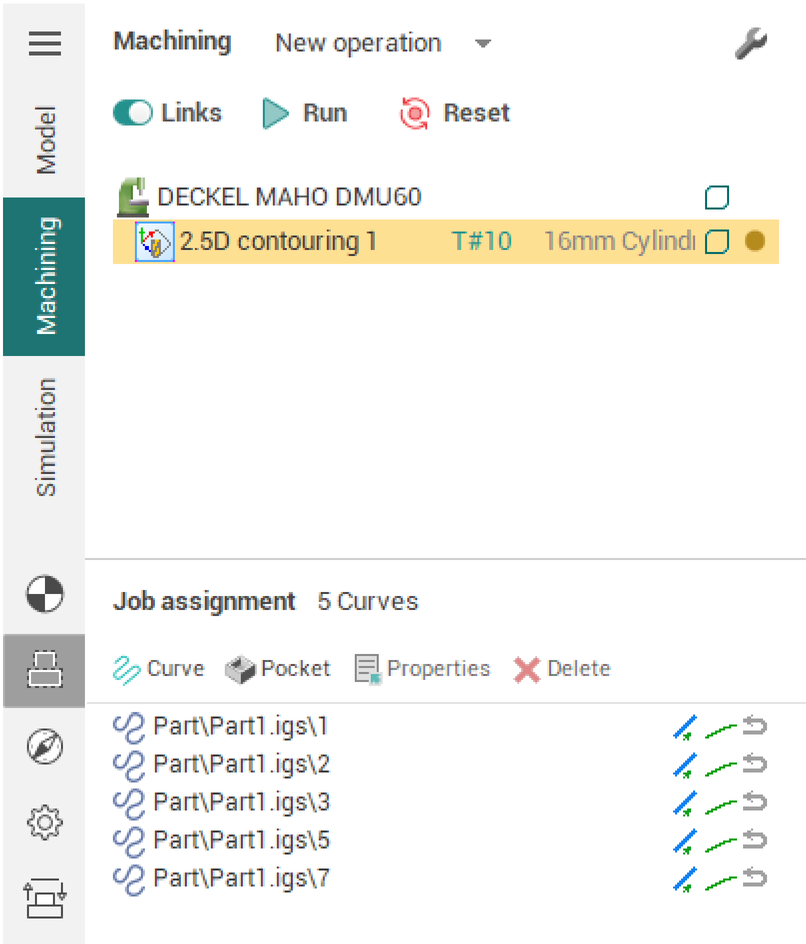
The system dynamically displays the 3D model in the graphic window, updating as any alterations to the parameters for elements are made in the Model window.

	
Curves that assign the model	**Created model**

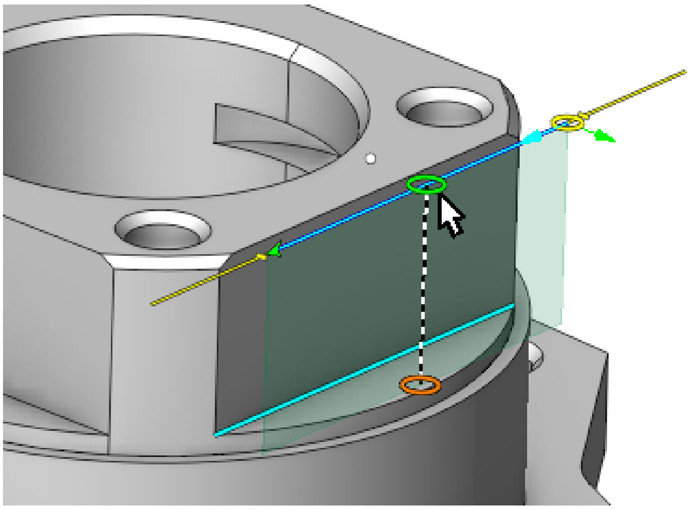
See also:

2.5D contouring

This operation extends the capabilities of the 2D contouring operation in terms of machining multi-level contours of parts. All the functionality of the 2D contouring operation was preserved.



For each contour specified in the job assignment of the operation, the top and bottom machining levels can be set, which affect only this contour. Setting machining levels is done interactively with the mouse cursor. To set them, click in the graphics window on the contour that is selected in the list of job assignments. On the contour line, next to the cursor, a graphic mark appears in the form of a small circle. Moving the cursor with this cursor in the direction perpendicular to the plane of the curve, we can set the top and bottom level for the contour. During that process the cursor position is snapped to the boundaries of the geometric object to which the contour belongs. The exact level value is entered in the input field, which is fixed by pressing the cursor again on the graphic label for setting the machining level.

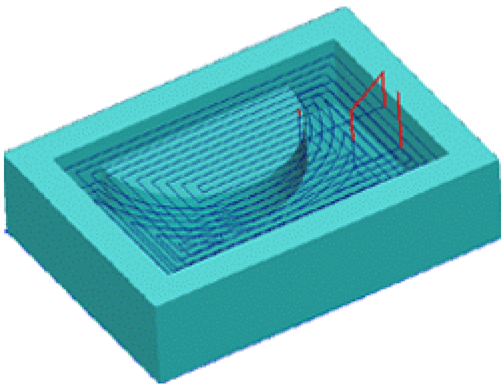


See also:

2D contouring

2.5D machining operations

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.

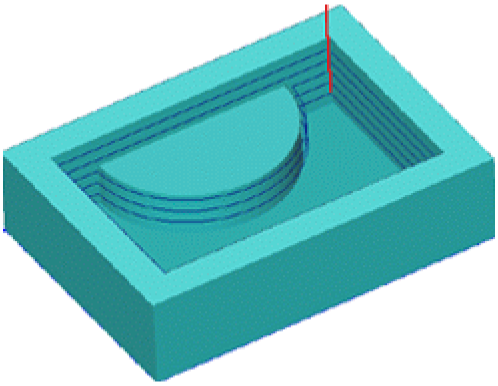
Material is removed layer by layer, using the defined step between layers. Depending on the defined strategy, the material of a layer can be removed using spiral strokes, starting at the center moving outwards, from the outside inwards, or by parallel passes. Plunge to the next machining depth can be performed either by one of plunge methods (axial, spiral, zigzag), or through drill pointsdrill points. 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).

See also:

Types of machining operations

2.5D machining operations

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.

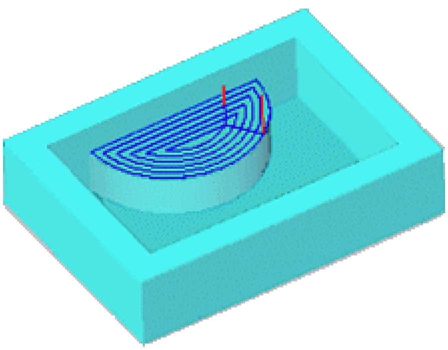
The material is removed in layers, using the defined step between layers. In the operation strategy the user can define the milling type, the corner roll type and corner smoothing if required.

See also:

Types of machining operations

2.5D machining operations

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.

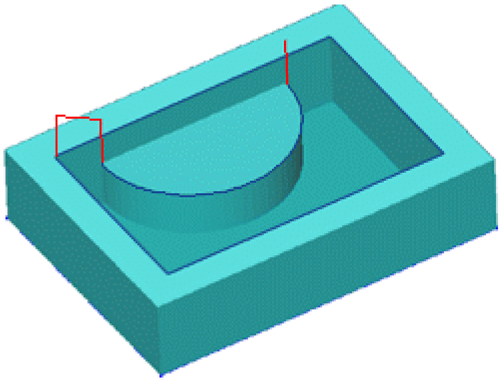
Depending on the defined strategy, the material of a layer can be removed using spiral strokes, starting at the center moving outwards, from the outside inwards, or by parallel passes. In the operation the user can also define milling type, corner rolling and corner smoothing.

See also:

Types of machining operations

2.5D machining operations

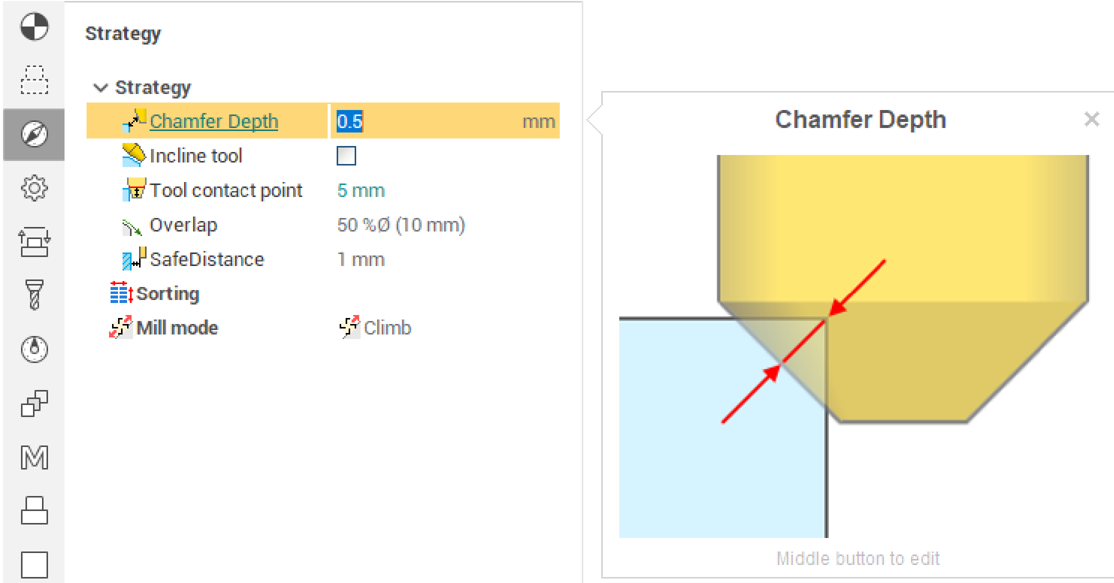
2.5D chamfer machining operation



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.

See also:

Types of machining operations

2.5D machining operations

5.5.3 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 SprutCAM X it is required the presence of three-dimensional models of processing parts.

See also:

Mill machining

3D curve milling

Flat land machining operation

Waterline roughing operation

Plane roughing operation

Drive roughing operation

Waterline finishing operation

Plane finishing operation

Optimized plane operation (plane-plane)

Drive finishing operation

Combined operation (waterline-drive)

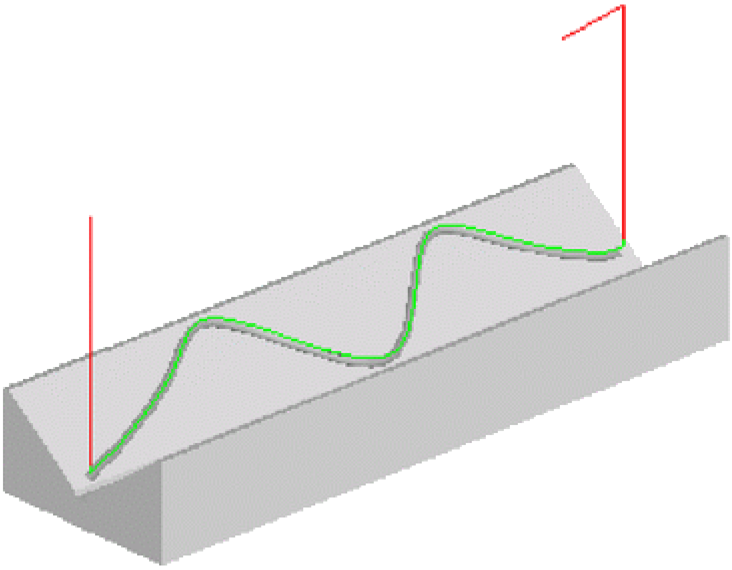
Complex operation (waterline-plane)

Rest milling operations

Plunge roughing

Waterline undercut operation

5.5.3.1 3D curve milling



This operation is designed for performing machining along any spatial curves.

The model being machined is either an imported model or created using 2D geometry. Every profile in the list can have its own machining method: either the tool center passes along the contour or, the left or right edge touches the contour. If the tool is offset (left or right) then it is possible to define an additional stock, a positive stock value is added to the machining side. If the tool follows the center of the contour then the stock value will be ignored. The Z coordinate for every point of the toolpath will be calculated according to the Z coordinate value of the corresponding curve and the defined offset value.

When machining a curve from left or right and complying with the condition that the tool contour touches one section, the mill contour theoretically may cross over the curve to another section. These areas correspond to equidistant loops of a horizontal projection of the curve. That means that, when machining such areas one may get a gouge in the model. In order to avoid this, the described faulty toolpath sections are automatically detected and deleted.

If in the operation there is a defined workpiece and/or restricted areas, then only those curves that lie within the workpiece and outside the restricted areas will be machined. If no workpiece or restrictions are defined then all selected curves will be machined.

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.

In the same window the user may define the machining tolerance and the stock. For curves that are machined from the left or right, any stock amount is added in the offset direction of the tool, and when machining along the tool center – it is ignored.

If the operation is performed using a local coordinate system or if using a rotary head then the toolpath will be created according to how the milling cutter touches the curve (left/right/center). The tool will be parallel to the Z-axis of the local coordinate system of the operation. This corresponds to the construction of equidistant curves in the XY plane of the local coordinate system that are equal to the tool radius plus stock, and the Z value is equal to the Z coordinate of the corresponding point on the source curve in the local coordinate system.

The initial machining point for an open curve corresponds to its first or last point (depending on the selections on the Model page for side of machining and <Inverse> and also the status of <Allow reverse direction> option). For closed curves, if an initial point has not been defined on the Model page, the first machining point will be selected automatically, for the minimization of tool movements.

When joining passes into the resulting toolpath, the defined approach method will be added to the beginning, and the retract method at the end. The joining order depends on the combination of the settings: <curve/offset>, <compensation>, <with return>.

When setting the machining order <By Contours>, each contour will be machined to full depth before moving to another contour. When setting <By Depth> each contour will be machined at the current cutting depth, the tool will move to the next depth only after all contours are machined.

Selecting <Idling Minimization> optimizes the order that contours are machined in. When deselected the contours are machined in the order that they appear on the model page.

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.

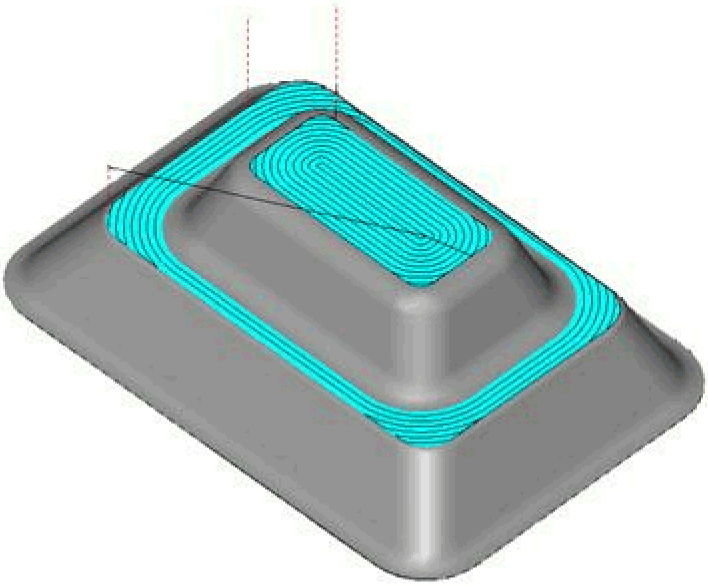
Note: *If you need to dictate the order of contour machining and the direction of their machining, then it is recommended to turn off the Idling Minimization mode and restrict the use of reverse direction. This will ensure that the order and the direction of machining will correspond to the order defined on the Model page.*

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.2 Flat land machining operation



The operation is expedient to machine a model with horizontal flats. Machining consists of series of horizontal tool passes on miscellaneous levels.

Surfaces and meshes defines the machining model. An additional stock value may be set for each geometrical object or objects group. The value will be added to the main stock of operation for machining.

All horizontal segments will be recognized automatically during elements adding in the model for machining. At model preview, these segments are drawn by other color for clearness. All other surfaces of a machining model is inspected, as well as restricted model. The rule allows to avoid part gouges.

The milling type (climb or conventional) is available during a toolpath calculation. It is possible to skip holes in a machining model the size less indicated, to keep them for further machining (holes capping).

Using of finishing pass (by vertical and horizontal) allows receiving more excellence quality of a part surface because of a small previously left finish stock.

Material removing may be realized with using of high speed cuts.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.3 Face milling

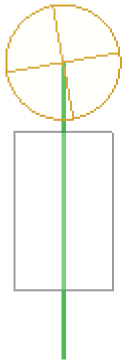
The face milling operation removes stock on a given horizontal plane with one of the following strategies:

- One pass,
- One way,
- Zigzag,
- Optimized zigzag,
- Spiral.

The part geometry is ignored, only the workpiece geometry is taken into account. There is no gouge checking.

The stock can be removed in multiple Z-passes. The Final level of machining by default is the top most level of the part while the top level of machining is the top most level of the workpiece, but it is possible to override these values. The number of passes is determined automatically based on the given cleanup height and the depth step. However you can specify the number of Z-passes explicitly. In this case the depth step will be calculated based on the number of passes and the difference between the top and the final level of machining.

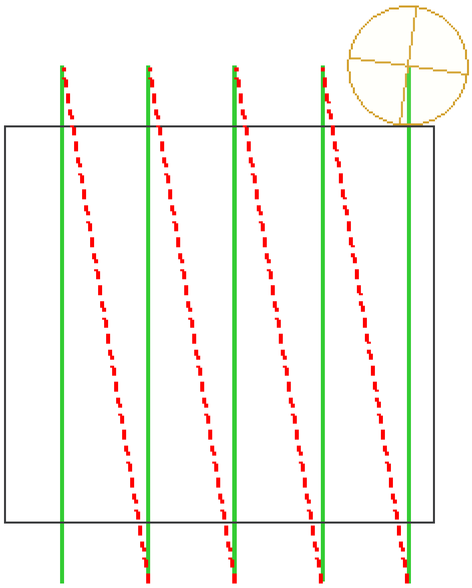
One pass strategy



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.

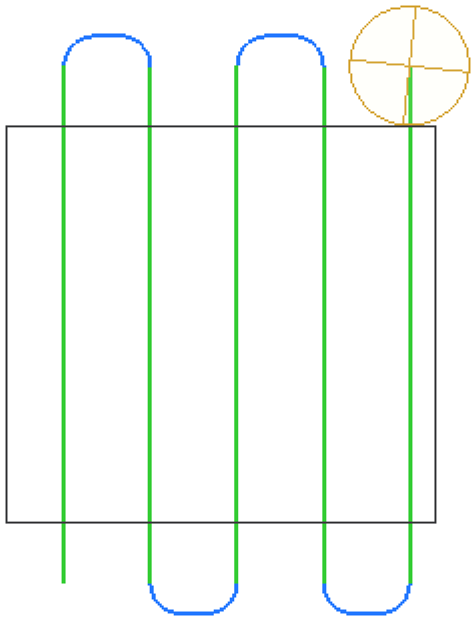
One way strategy



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.

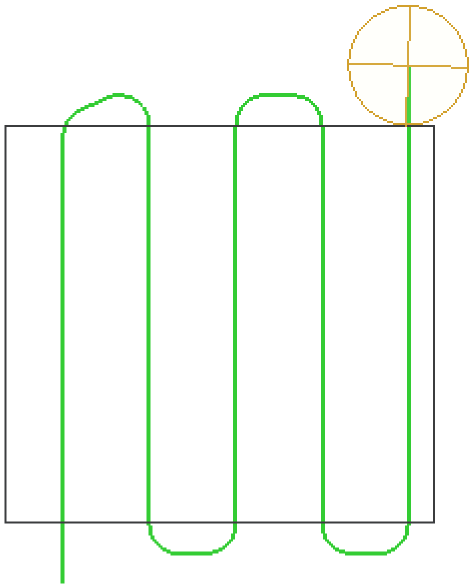
Zigzag strategy



The tool moves along zigzag trajectory. The work passes start and end outside the workpiece material. The link moves are made outside the material as well. A different feed rate may be used for the link moves.

The strategy is optimal for finishing face milling.

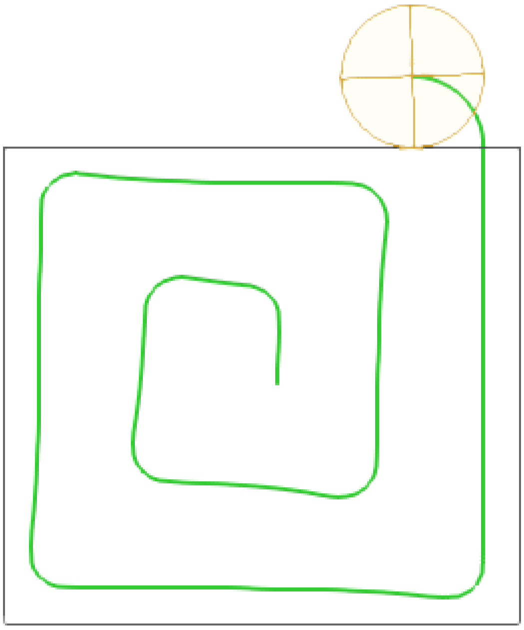
Optimized zigzag strategy



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.

Spiral strategy



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.

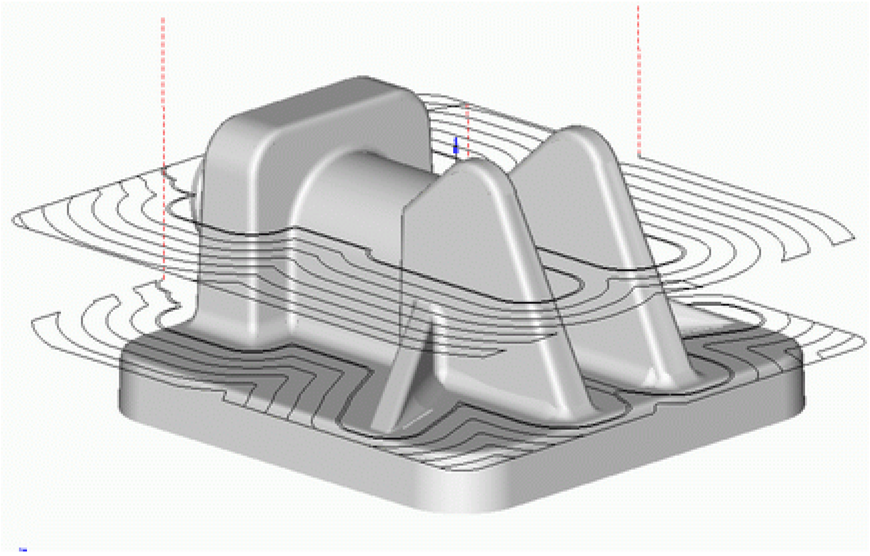
The maximum allowed width of cut and the tool engagement angle are never exceeded.

The strategy is very good for removing large volumes of material in minimal time.

Operation parameters

For more information on the operation parameters refer to the online help integrated into the operation parameters inspector.

5.5.3.4 Waterline roughing operation



The waterline roughing operation is used for preliminary rough machining of models of a complex shape, which have significant differences to the 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.

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. The material is removed using horizontal passes of the tool layer by layer. The step (or depth of the layer being removed) can be fixed or calculated according to the defined height of the scallop. Either depending on the selected strategy, the material for every layer can be removed using spiral passes, directed towards or out from the center, and by using parallel passes.

Transition to the next machining depth can be achieved either by using one of the plunge methods (axial, by spiral, zigzag), or through drill points. If the latter method is used, a search is made for a hole of an appropriate depth/diameter. The search will first be made in the list of holes for the operation, and then in the open list of holes for the machining process. If no appropriate hole can been found, then the system will select appropriate coordinates for it automatically, at an optimal position. The coordinates for the center of the new hole, if possible, are rounded. If when the operation was created, the hole machining operation was chosen as its prototype, then the list of holes will be copied into the operation and used when searching for appropriate hole for tool plunging.

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 that are perpendicular to the horizontal plane of the local coordinate system.

There is can to be used quick calculation method also.

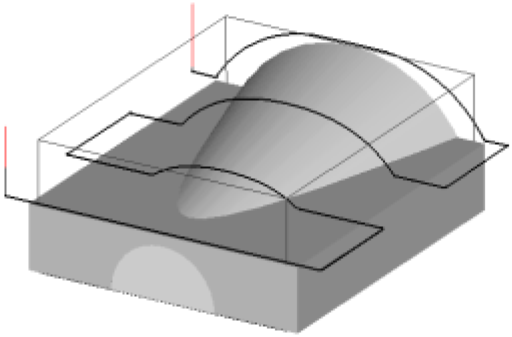
For roughing waterline operation Adaptive feedrate feature available.

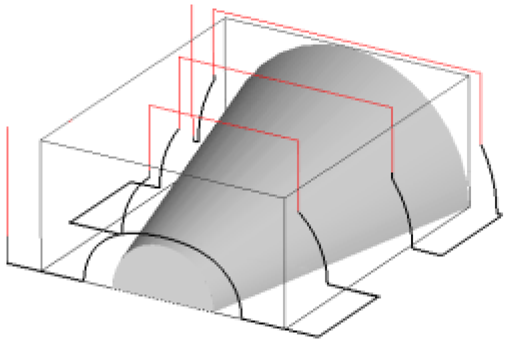
See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.5 Plane roughing operation





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.

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.

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. The work passes of the operation lie in parallel vertical planes. The positions of the planes are defined by the angle between these planes and the Z-axis. The step between the planes of neighboring work passes can be either fixed or calculated according to the defined height of the scallop.

To limit pressure on the tool, the depth of material removed can be defined. If the depth of the material being removed from the workpiece exceeds the 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 that are perpendicular to the horizontal plane of the local coordinate 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, and then the system first removes the entire material at the first depth before starting on the next one.

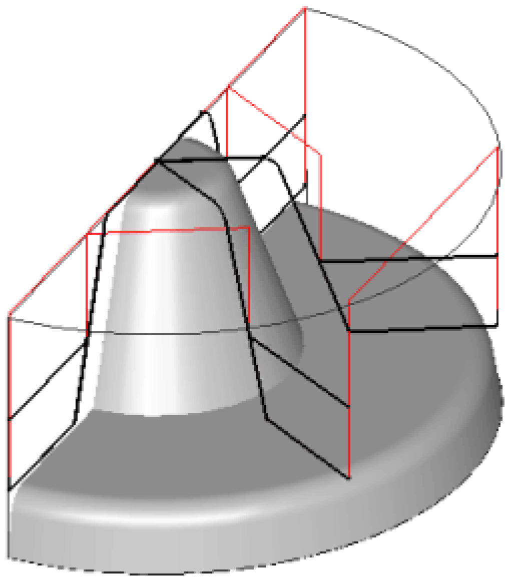
There is can to be used quick calculation method also.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.6 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.

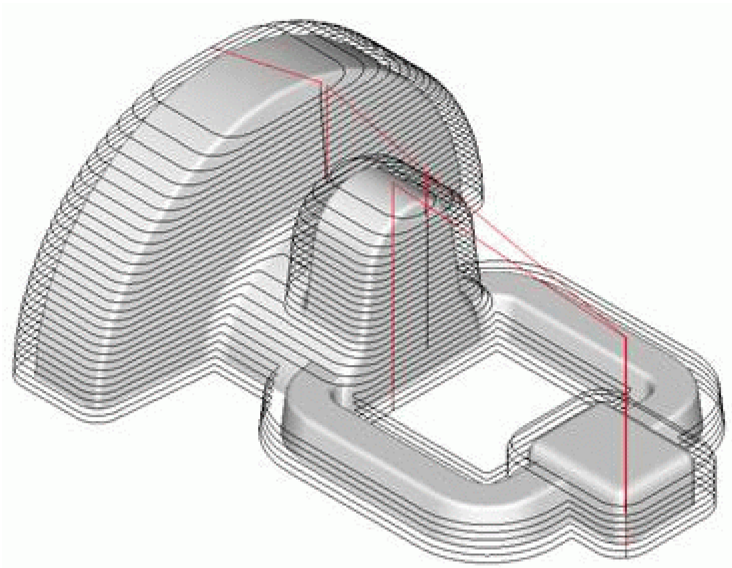
There is can to be used quick calculation method also.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.7 Waterline finishing operation



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 or drive finishing operations.

The model for the waterline finishing operation is defined by a set of solid bodies, surfaces and meshed objects. 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.

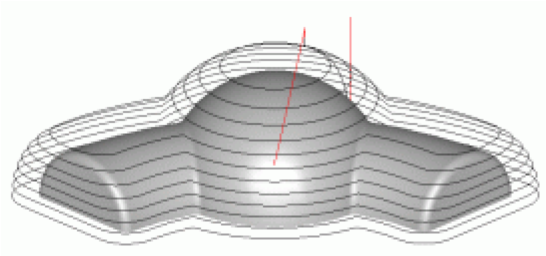
Machining of the model surface is performed using horizontal passes. The step between the passes of neighboring toolpaths can be either fixed or calculated according to the defined height of the scallop.

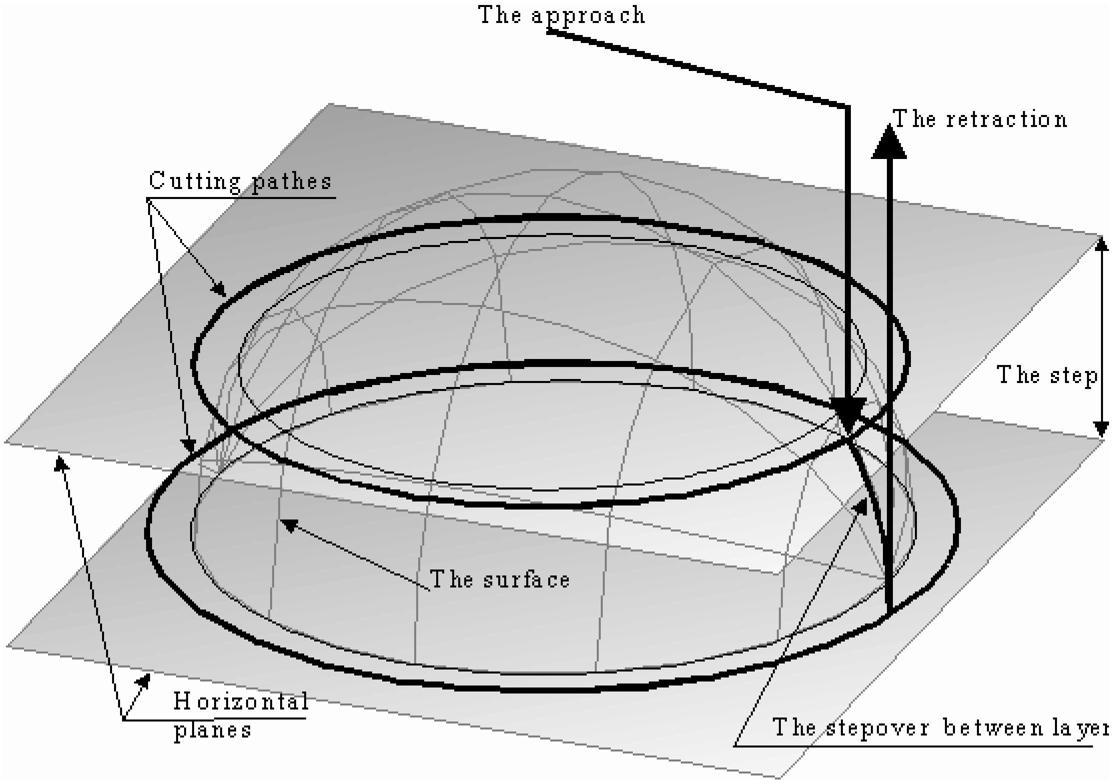
When using a local coordinate system or a rotary axis, the position of the model being machined will not change, the tool rotation axis is parallel to the Z axis of the local coordinate system, and all work passes are located parallel to the horizontal plane of the same system.

The areas of the model surface 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 steep areas with a slope angle of the normal to the Z-axis more than 45 degrees, then it is advised to set the values for the minimum and maximum slope angles to 45 and 90 degrees respectively.

It is also possible to restrict machining of the areas of the restricting model and areas of edges rounding from 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.



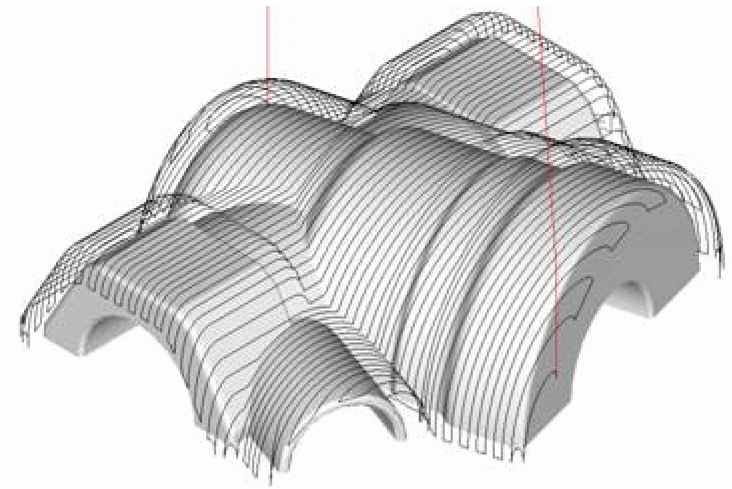


See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.8 Plane finishing operation



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 or another plane operation with a toolpath, which is perpendicular to that of the first operation.

The model to be machined by the operation is assigned as a set of solid bodies, surfaces and meshed objects. 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 toolpaths for the operation lie in parallel vertical planes. The positions of the planes are defined by the angle between these planes and the Z-axis. The step between the planes of neighboring work passes can be either fixed 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, perpendicular to the horizontal plane of the local coordinate system and parallel with the X axis of the same system, at the defined angle.

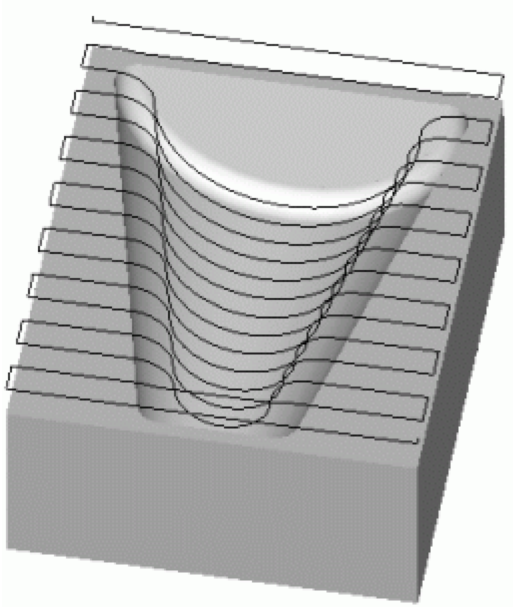
The areas of the model's surface being machined can be limited depending on the slope angle of the normal to the Z-axis. If for example, the user needs to machine flat areas with the slope angle of the normal to the Z-axis less than 45 degrees, then it is advised to set values of the minimum and maximum slope angles to 0 and 45 degrees accordingly.

In order to only machine areas with a small deviation from the surface normal to the plane of the work pass, it is advised to limit the frontal angle. For example, if it is needed to perform machining using two perpendicular plane operations, then it is advised to set the value of the frontal angle equal to 45 degrees. If machining is performed using a series of three plane operations, then set it to 30 degrees.

Note: *In order to avoid repeated machining of horizontal areas, the user should set the minimum value of the slope angle of the normal equal to 0 only for just one operation, for others – set it higher (e.g. 1 or 2).*

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.



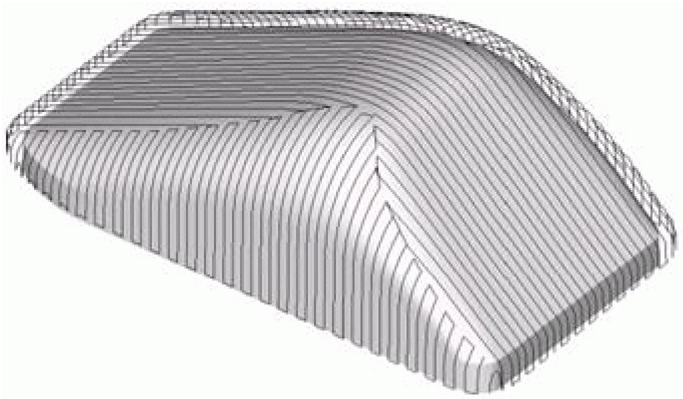
There is can to be used quick calculation method also.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.9 Optimized plane operation (plane-plane)



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.

The default set of parameters for the optimized plane operation is identical to the plane finishing operation.

The model being machined by the optimized plane operation is defined by a set of solid bodies, surfaces and meshed objects. 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 work passes of the operation lie in two parallel vertical planes. The planes of different operations are perpendicular to each another. The positions of the planes are defined by the angle between these planes and the Z-axis. The step between the planes of neighboring work passes can be either fixed or calculated according to the defined height of the scallop.

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 that are perpendicular to the horizontal plane of the local coordinate system.

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.

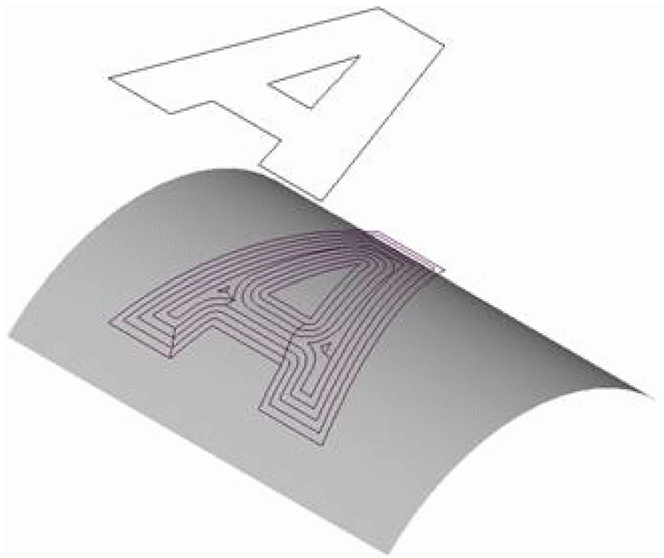
Note: *In order to provide a good finish at the border area(s), it is recommended to "overlap" the toolpaths for the operations. For example, set the value for the* frontal angle *in the first and second operations to 46 degrees.*

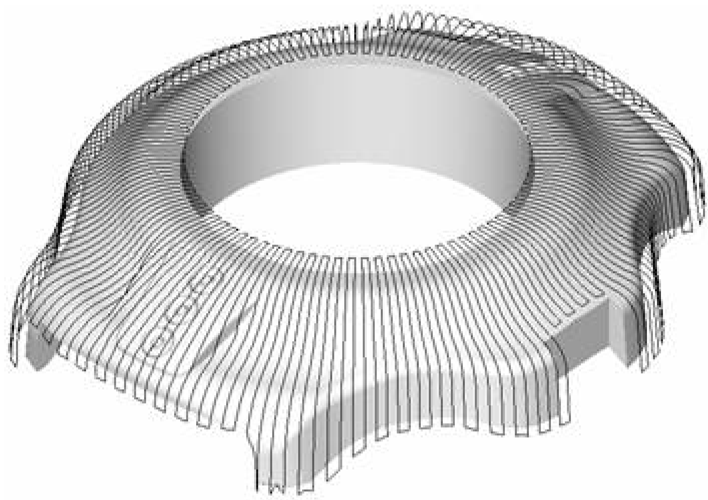
See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.10 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.

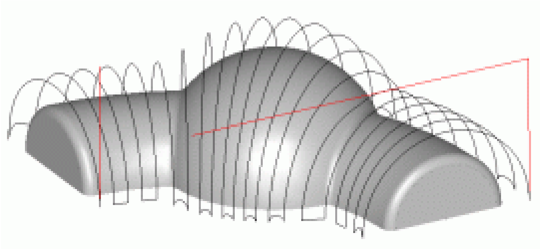
When using a local coordinate system or a rotary axis, the position of the model being machined will not change, the tool rotation axis is parallel to the Z axis of the local coordinate system, and all work passes are located in planes or mathematical cylinders, perpendicular to the horizontal plane 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.

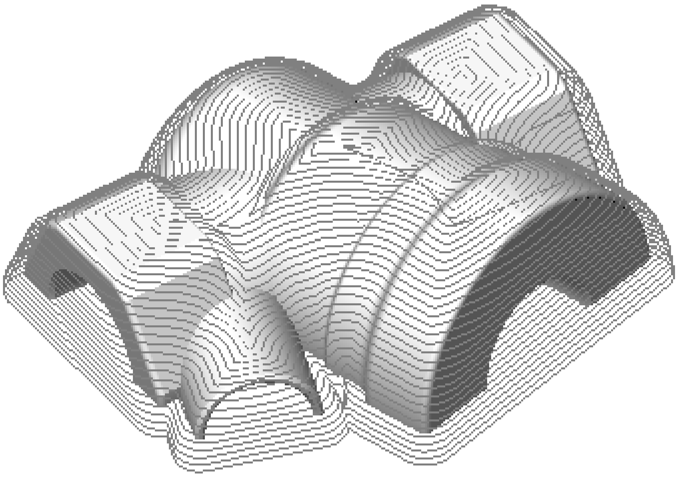


See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.11 Scallop finishing operation



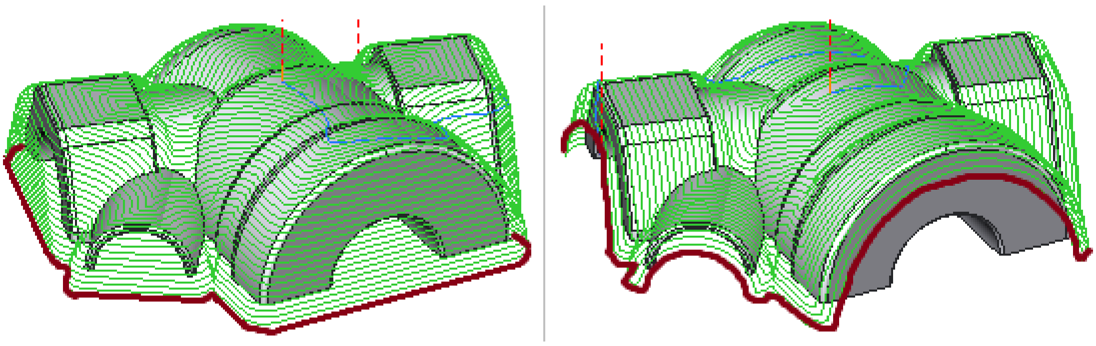
The Scallop toolpath starts from the curves lying on the part surfaces and is generated by repeatedly offsetting those curves inwards until the curves collapse. So basically it is an equidistant toolpath except that the offset is made in 3d space on the machining surfaces. The toolpath achieves a consistent 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.

Starting curves

There are two options currently available:

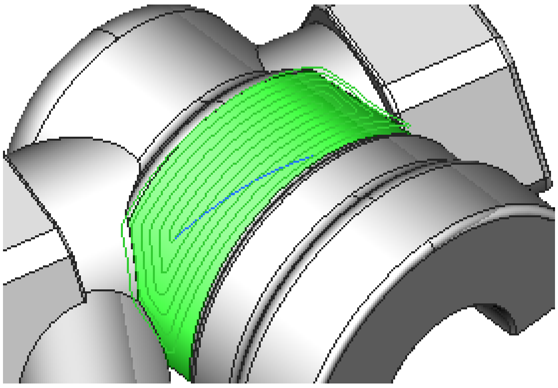
- start from the bottom, and
- start from the top of vertical walls.

For the first option the toolpath starts at the bottom level of machining, for the second option the toolpath starts from the silhouette curves.

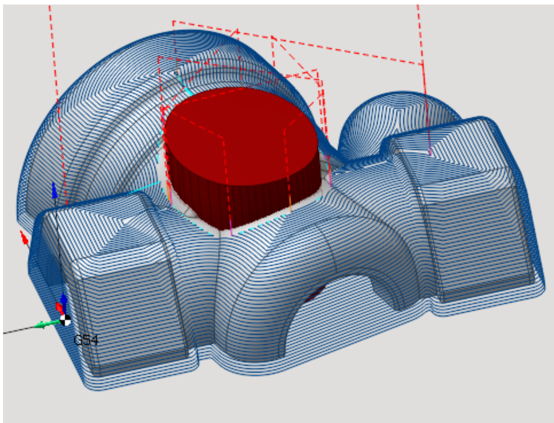


Job assignment

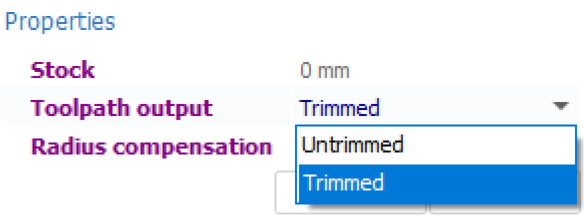
You can specify machining surfaces in the job assignment. The starting curves will be detected as the curves of contact of the cutter with those surfaces in this case.



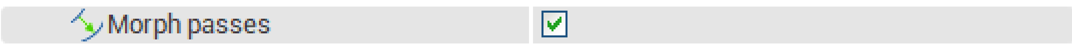
You can also use the Job Zone to define the starting curves.

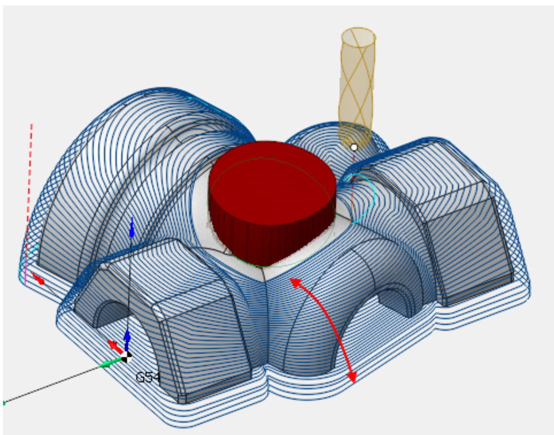


You also can add **Restrict Zones** with determine restricted or cutted toolpath depending on the option you choose.



Morph Passes option calculates toolpath between open areas.





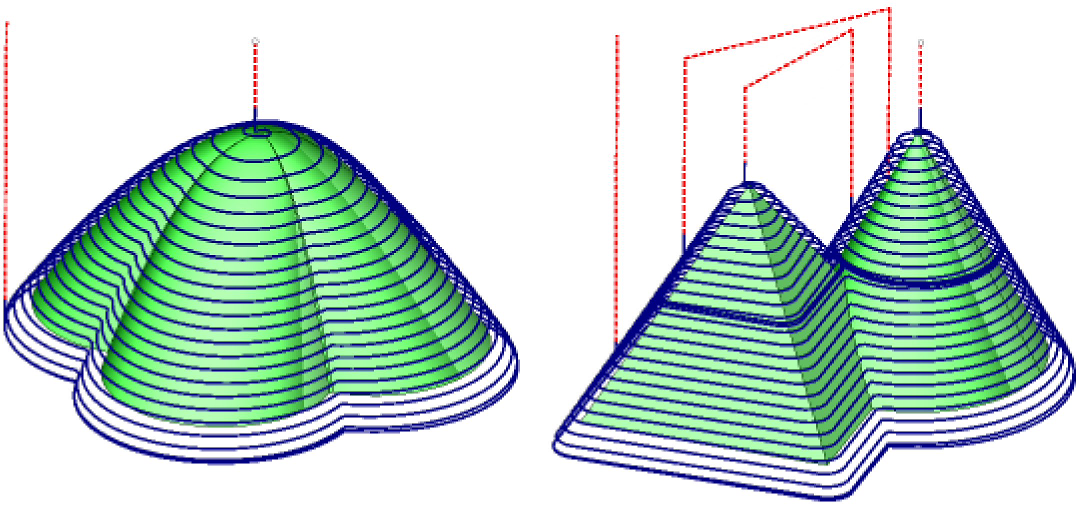
Strategies

The toolpath can be generated both from the inside out and vice-versa, from the outside in.

It is possible to generate a spiral toolpath instead of parallel passes to minimize linking.

It is possible to smooth sharp corners in the toolpath.

5.5.3.12 Helical operation



The operation generates continuous helical passes with the given vertical stepover between the top and the bottom level.

Parameters

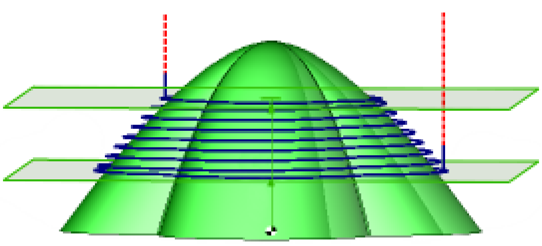
Step

The vertical distance between any two adjacent complete helix turns.

Strategy

Use the Start From parameter to choose between the top-down and bottom-up machining. Use the milling type parameter to choose between climb and conventional milling.

Job zone



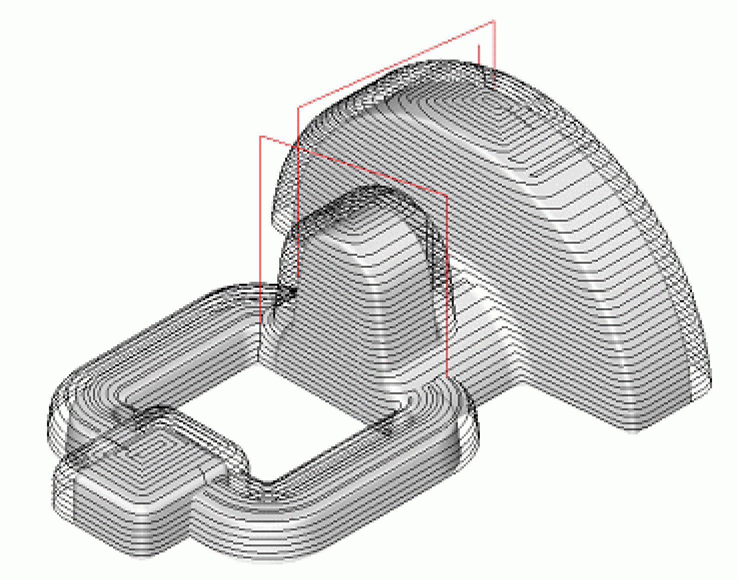
The whole part is being machined. The Top and the Bottom levels of machining can be specified.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.13 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.

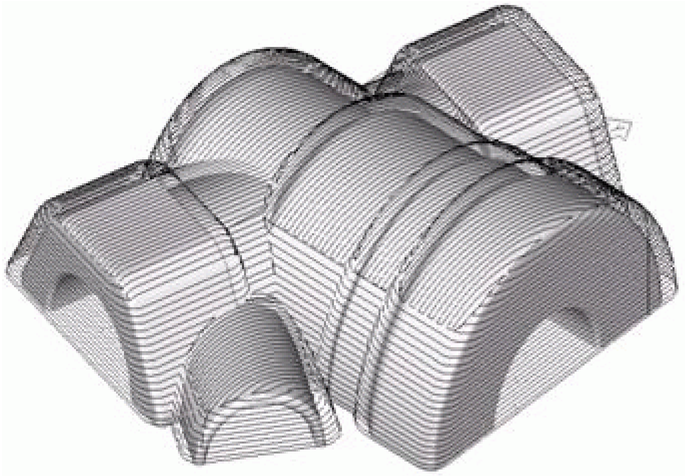
Joining of the work passes into a single toolpath can be performed going downwards or upwards. Transition between neighboring work passes can be performed on the surface, using retract and approach moves or via the safe plane.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.14 Complex operation (waterline-plane)



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.

A model being machined for the complex operation is assigned by a set of solid bodies, surfaces and meshed 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.

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 work passes of the operation lie in two parallel vertical planes. The planes of different operations are perpendicular to each another. The positions of the planes are defined by the angle between these planes and the Z-axis. The step between the planes of neighboring work passes can be either fixed or calculated according to the defined height of the scallop.

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 that are perpendicular to the horizontal plane of the local coordinate system.

It is also possible to restrict machining from entering areas of the restricting model and areas of edge rounding in the resulting toolpath.

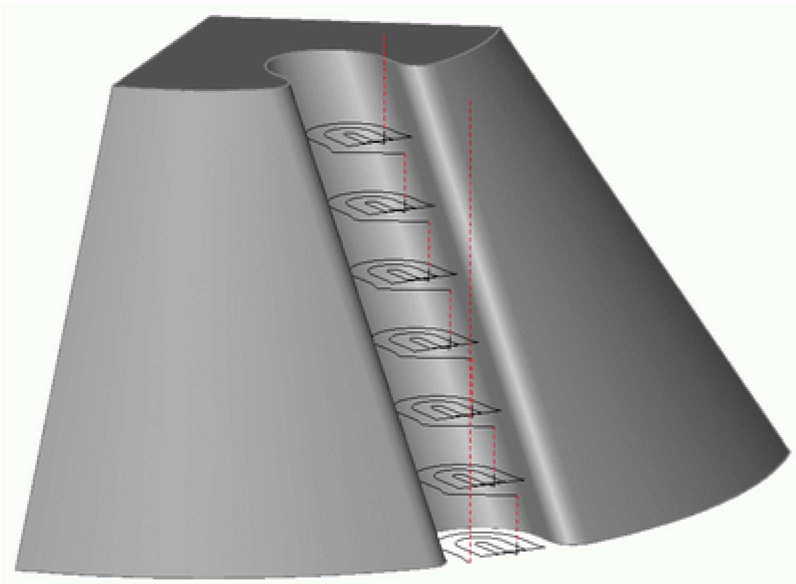
Joining of the work passes into a single toolpath can be performed going downwards or upwards. Transition between neighboring work passes can be performed on the surface, using retract and approach moves or via the safe plane.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.3.15 Rest milling operations



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.

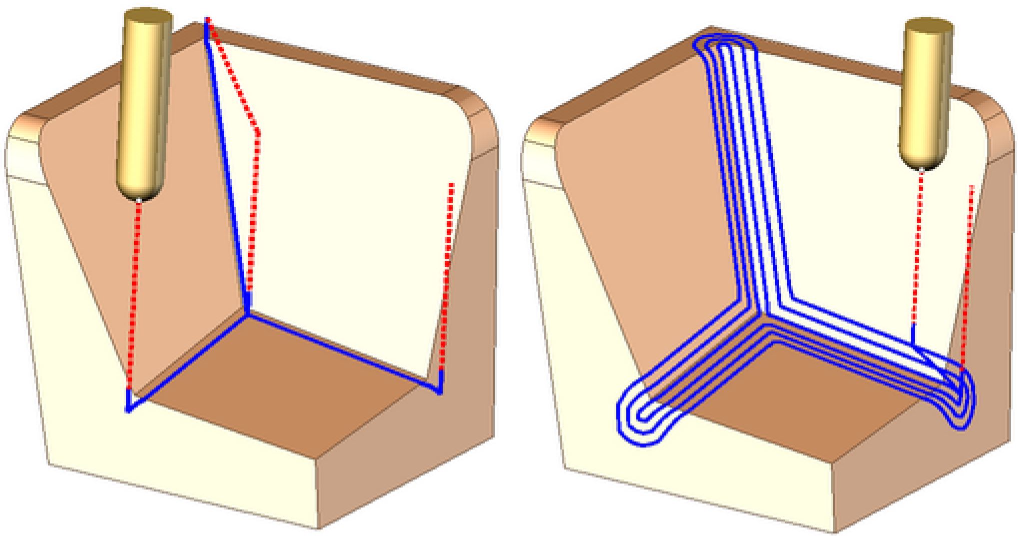
All machining characteristics for the rest milling operations are similar to the normal machining operations of the same type.

See also:

Types of machining operations

Operations for the 3-axes milling

Pencil operation



The rest machining operation generates passes along inner corners of the part.

Strategies

One pass

One pass generates a single pass along every inner corner.

Parallel passes

The Parallel passes strategy generate multiple passes along inner corners of the part. The passes represent offsets along the machining surfaces of the passes like the One pass strategy would generate. The number and the step-over between passes can be set.

The machine by strokes feature allows separation of the toolpath into regions so that smoothly connected regions are machined separately from one another. The connection angle defines the angle between passes that is considered to be a smooth connection.

Job zone

Machining surfaces

The passes are generated only in the places where the tool contacts with the machining surfaces. If the machining surfaces are missing, the toolpath is generated against the whole part.

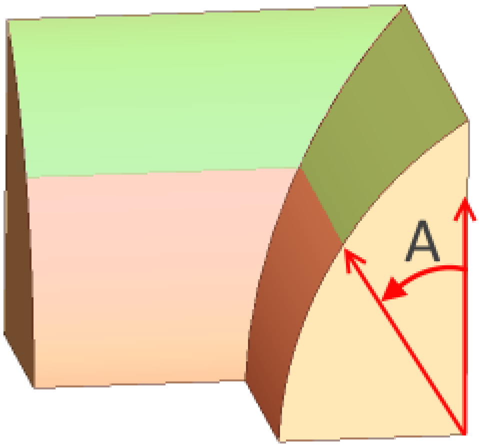
Job zone

Use the job zone to trim the passes outside the specified containment areas.

Restring zone

Use restrict zones to easily create restriction geometry from curves and edges.

Machining slope



Use the Machining slopes parameter to machine only steep or only shallow areas. Use the steep/ shallow split angle parameter to set the slope angle which separates the steep from the shallow.

Bottom level

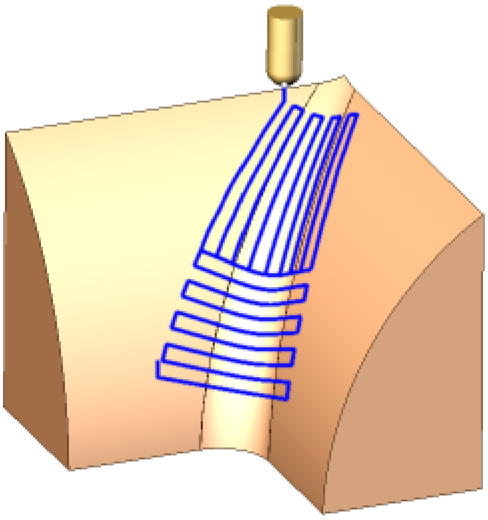
A bottom level can be set for the passes

See example of operation on YouTube:

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=ANKfyNN65uQ

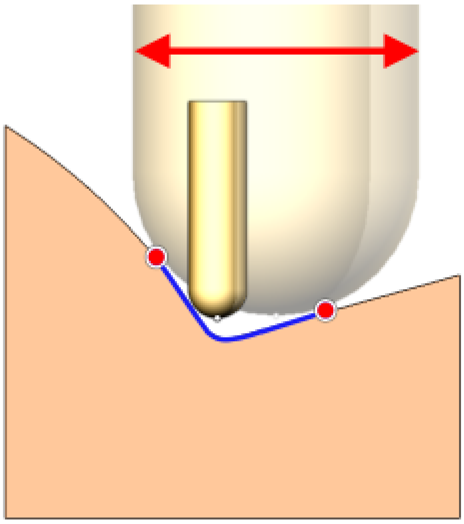
Corners cleanup



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.

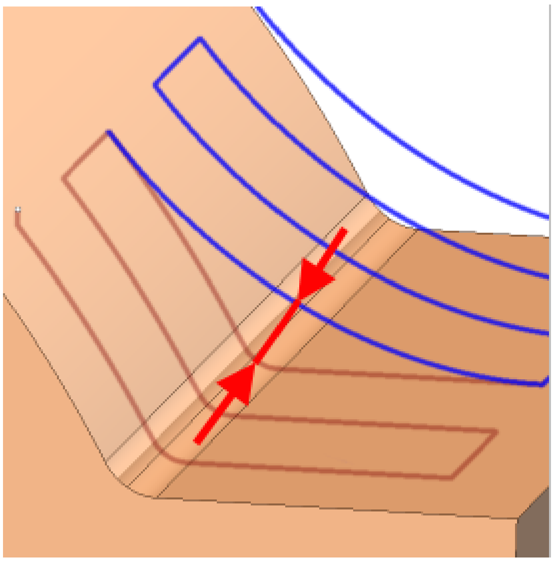
Parameters

Previous tool diameter



The diameter of a spherical mill which is used for the rest material calculation.

Cut depth



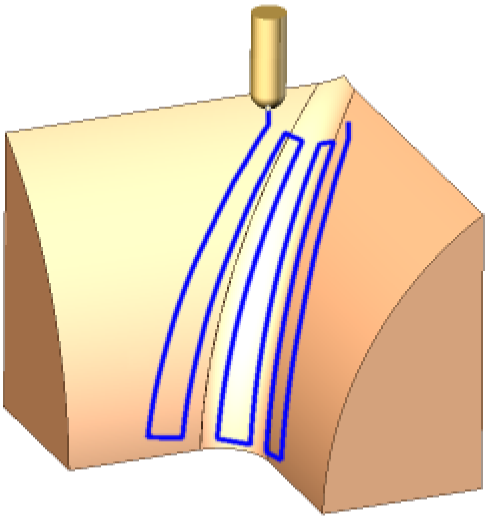
The maximum cut depth for a cut

Step

The maximum step-over between passes

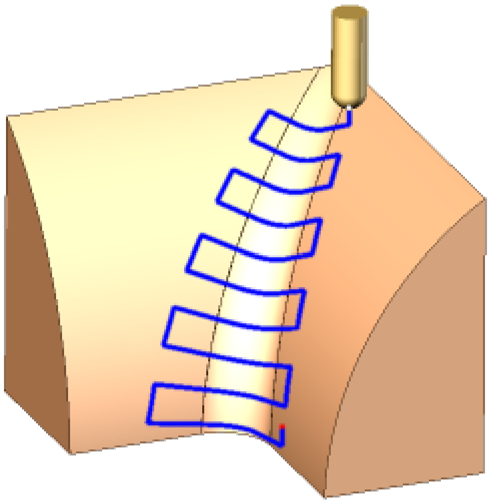
Strategies

Along



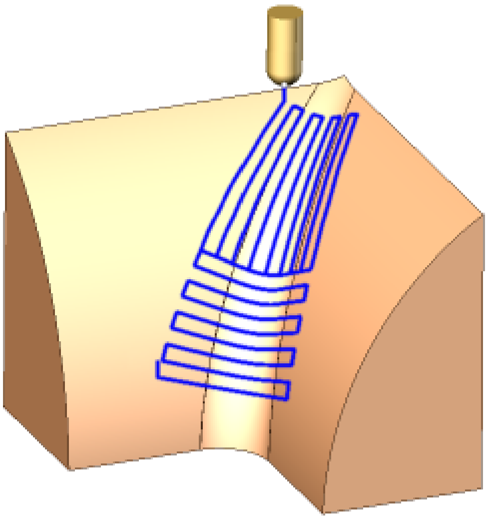
The passes are generated along the corners.

Across



The passes are generated across corners

Combined



For shallow areas - along passes are generated, for steep areas - across passes are generated.

Job zone

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 against the whole part.

Bottom level

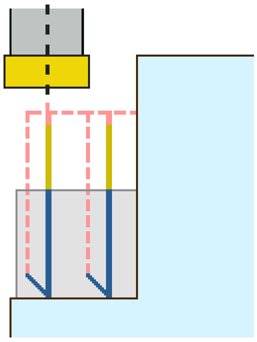
A bottom level can be set for the passes

See example of operation on YouTube:

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

http://youtube.com/watch?v=ANKfyNN65uQ

5.5.3.16 Plunge roughing



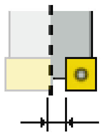
Plunge roughing option generates vertical passes from Roughing Waterline, Plane and Pocketing operations result.

This option requires a tool that is designed to handle axial cutting.

Parameters

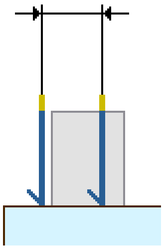
Core radius

Radius of the central non-cutting part of the tool.



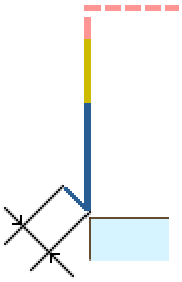
Step

Step between passes.



Pull back distance

Distance for which tool pulls back away from the model.

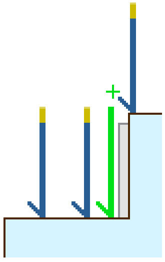


Additional pass

Creates an additional pass if can't process the workpiece on the current pass.

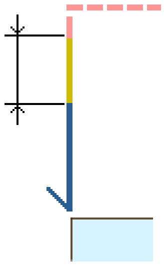
This function is useful when plunge option cannot process corner areas.

Step will be ignored for additional passes.

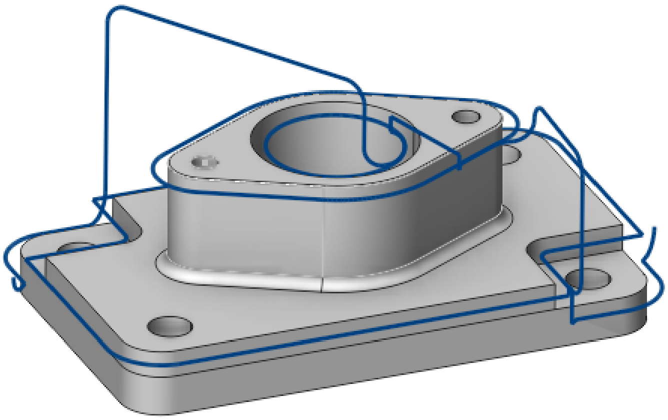


Feed distance

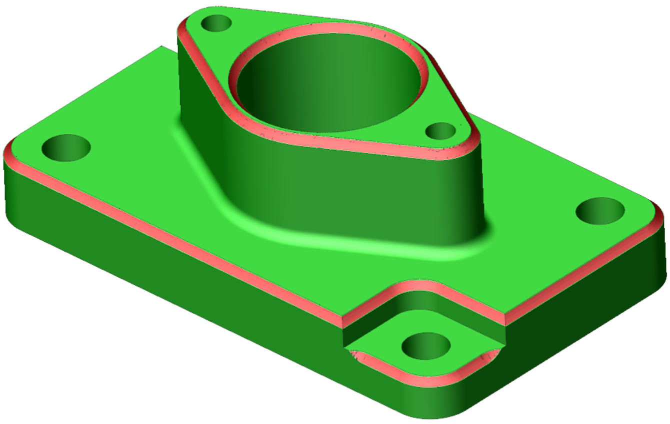
Distance (along the tool axis) where vertical motion switch to work feed.



5.5.3.17 Chamfering

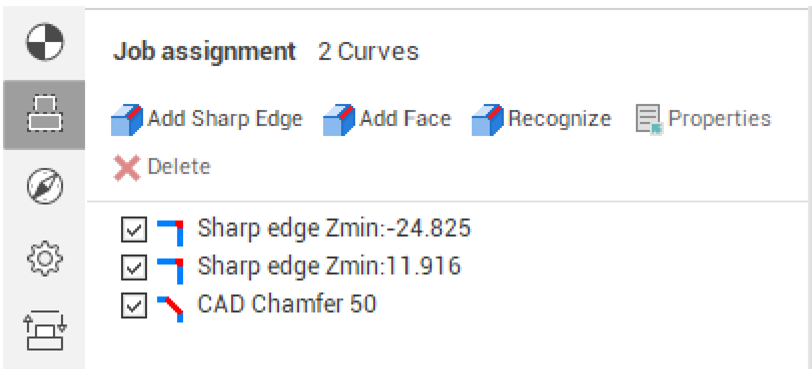


This operation generates a chamfering trajectory on specified edges, chamfers or fillets from the 3D model. It is most often used as the finishing for edge beveling or for deburring along the edges of the part that were formed after milling. The operation allows to machine chamfers and at the same time not to damage the part where there are no chamfers. Cylindrical end mills, conical mills, spherical mills are available for chamfering.



Job assignment

In this operation, it is possible to handle chamfers both present on the 3D model and defined using sharp edges.



Add Sharp Edge

To machine chamfers based on edges, you need to add them using the “Add Sharp Edge” button on the “Job assignment” tab. If you select a face instead, then all sharp edges of this face are automatically added.

Add Face

To machine chamfers which are present on a 3D model, you need to select faces marked as chamfers and add them using the “Add Face” button.

Recognize

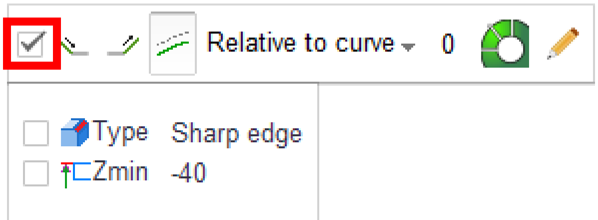
It is also possible to automatically recognize and add chamfers using the “Recognize” button.

Chamfer types

For convenient use chamfers can be one of the four types:

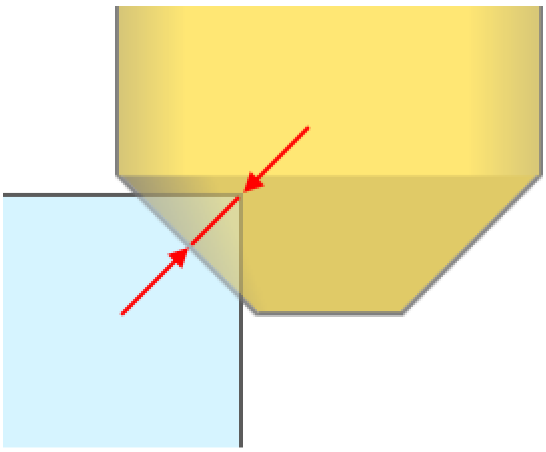
1. Sharp edges
2. CAD chamfers
3. Fillets
4. Hole chamfers

To highlight chamfers with similar parameters, you can use the auxiliary panel in the graphics window that appears when a job assignment curve is selected.



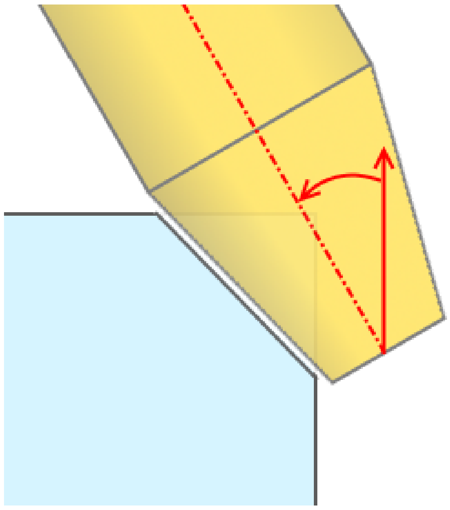
Main parameters of the operation

Chamfer depth



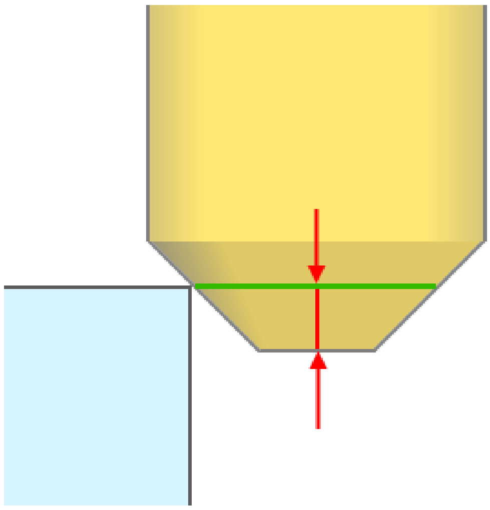
Specifies how far the tool will plunge into the part. If a “CAD Chamfer” is machined, then the tool passes over the surface of the chamfer.

Incline tool



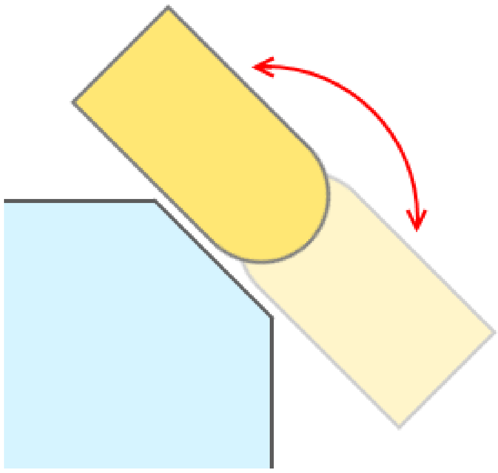
This parameter allows tilting the tool to chamfer with its side.

Tool contact point



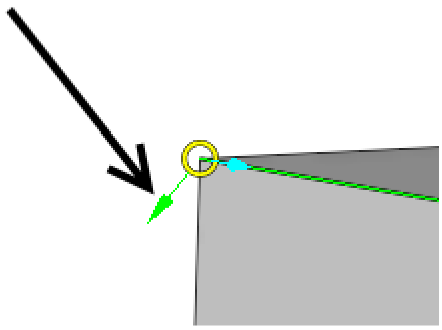
It is the point on the tool with which it should mainly touch the geometric objects specified in the job assignment of the operation. The contact point is determined by the distance from the tool tip. It can also be changed interactively in the graphics window.

Inverse tool axis direction

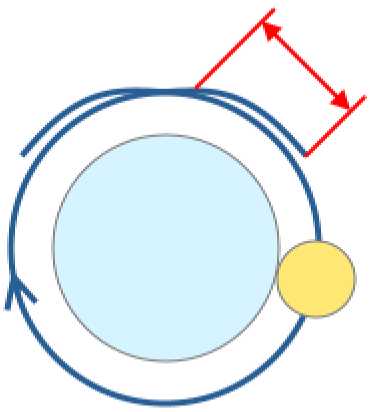


This parameter appears only when the "**Incline tool**" option is enabled. It allows you to machine the chamfer at a different angle to the axis of the tool.

The same effect can be achieved by changing the side if clicked on the direction arrow in the graphics window. In this case, the direction of the tool axis changes for each curve individually.

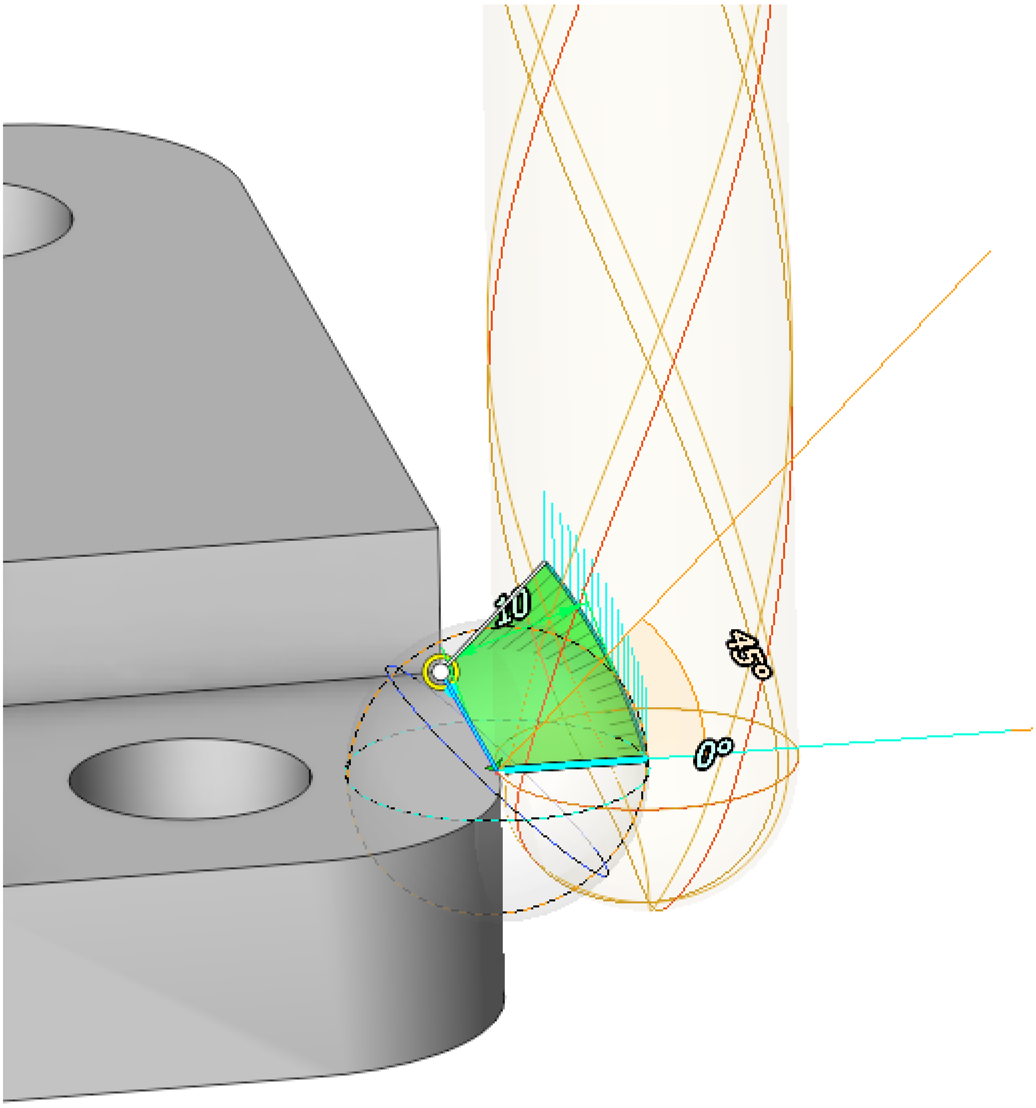


Overlap



Allows extending the trajectory in closed chamfers so that there are no burrs left at the junction.

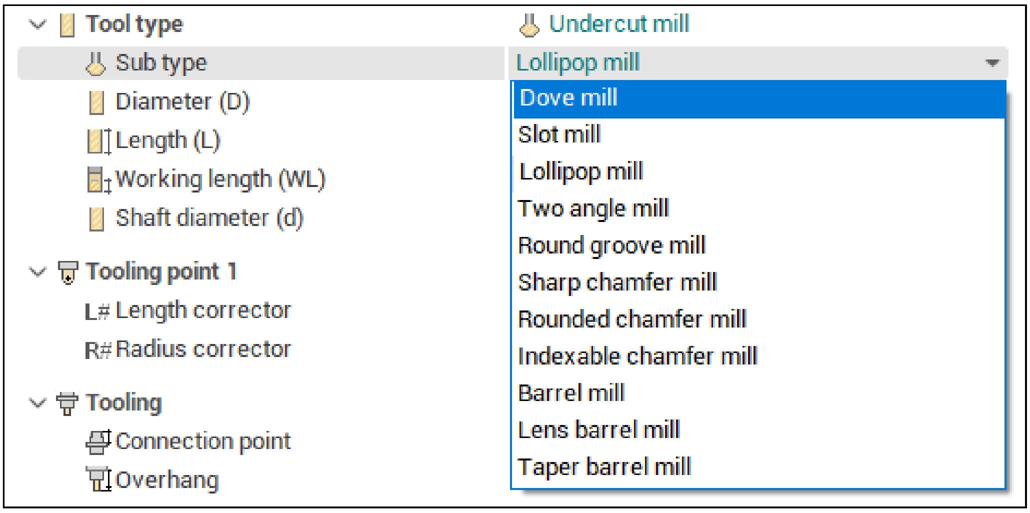
Interactive tuning



In some special cases, you may need to change the normal of the plunge into the part. To do so, you can use the interactive normals that appear when you select the job assignment curve in the graphics window.

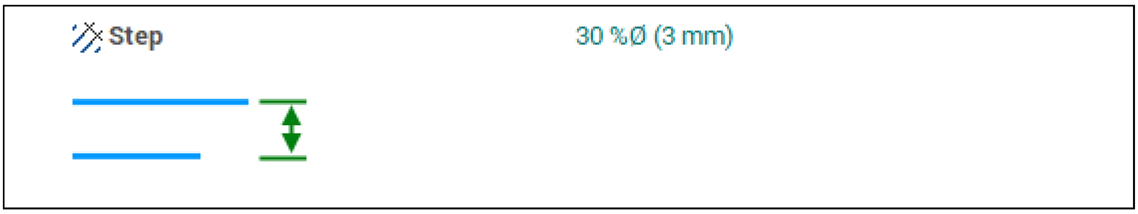
5.5.3.18 Waterline undercut operation

This operation generates a waterline trajectory for undercut tools.

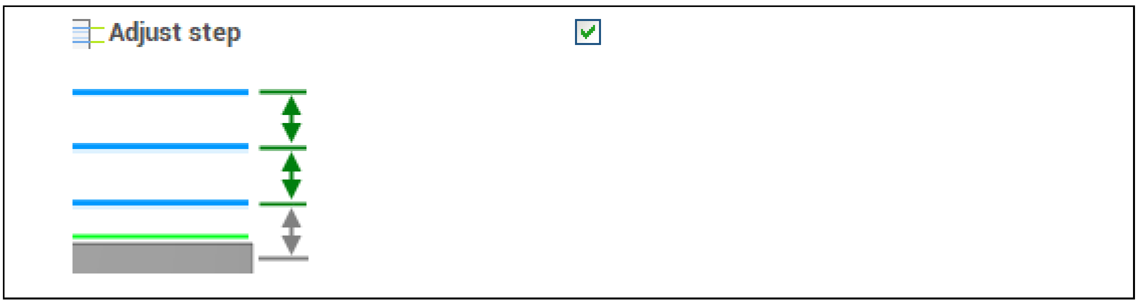


The toolpath can be defined with specific parameters such as:

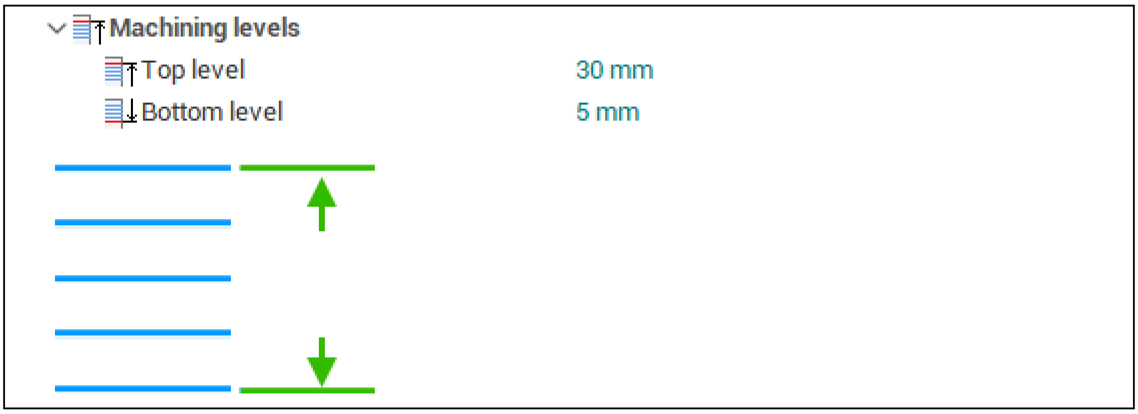
Step between passes.



Additional pass on the upper or lower level.

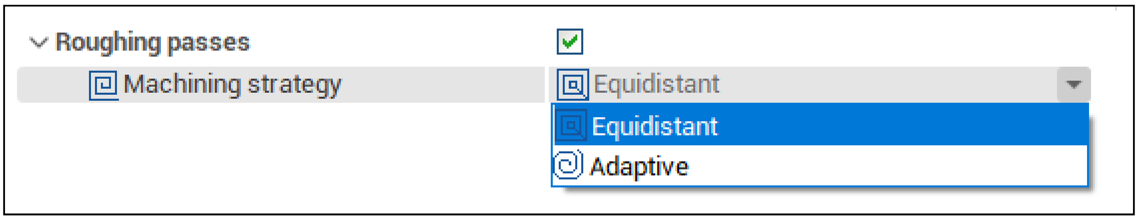


Top and bottom levels.



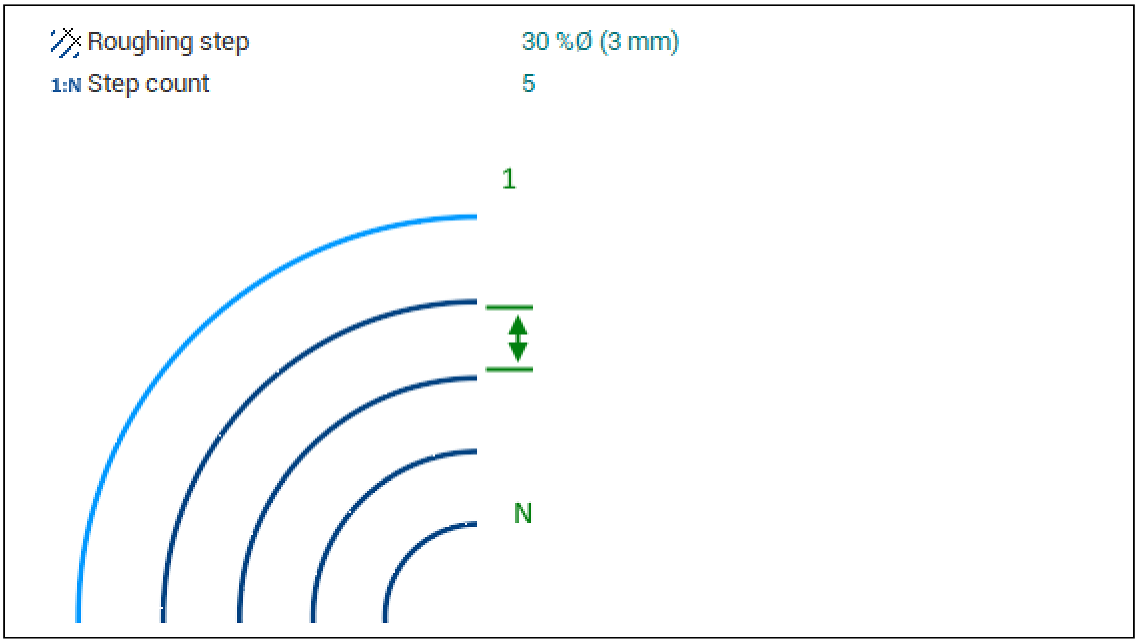
The operation can also generate rough passes and has two strategies for doing so.

Equdistant and Adaptive.

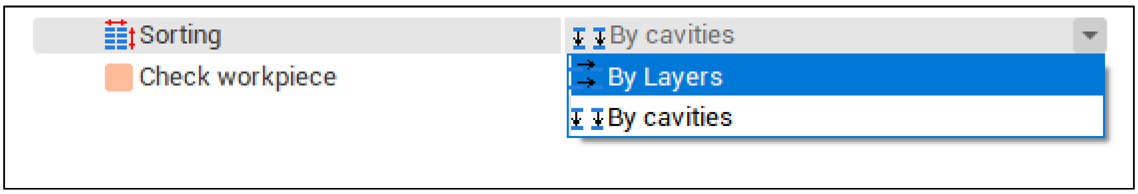


The roughing toolpath can be defined by specific parameters:

Step of roughing passes and number of steps for equidistant strategy.



And two sorting types by layers and by cavities.



The adaptive strategy has the same parameters as the Waterline roughing operation.

See also:

Types of machining operations

Operations for the 3-axes milling

5.5.4 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. SprutCAM X 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 machiningContinues 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 >< Rotary machining >
 operation.
6. <Holes machining> performs the machining of the holes with the free axes direction.

See also:

5 axis machining

Mill machining

Indexed 5-axes milling

Rotary machining with using of the cylindrical interpolation

4-axis milling with using of the engraving and pocketing operations

Hole machining operation

5D contour and 6D contour operations

Rotary marching

5.5.4.1 5 axis machining

SprutCAM X can generate toolpath for 5 axis milling machines based on the machine kinematics and the parameters of the used control. SprutCAM X 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.

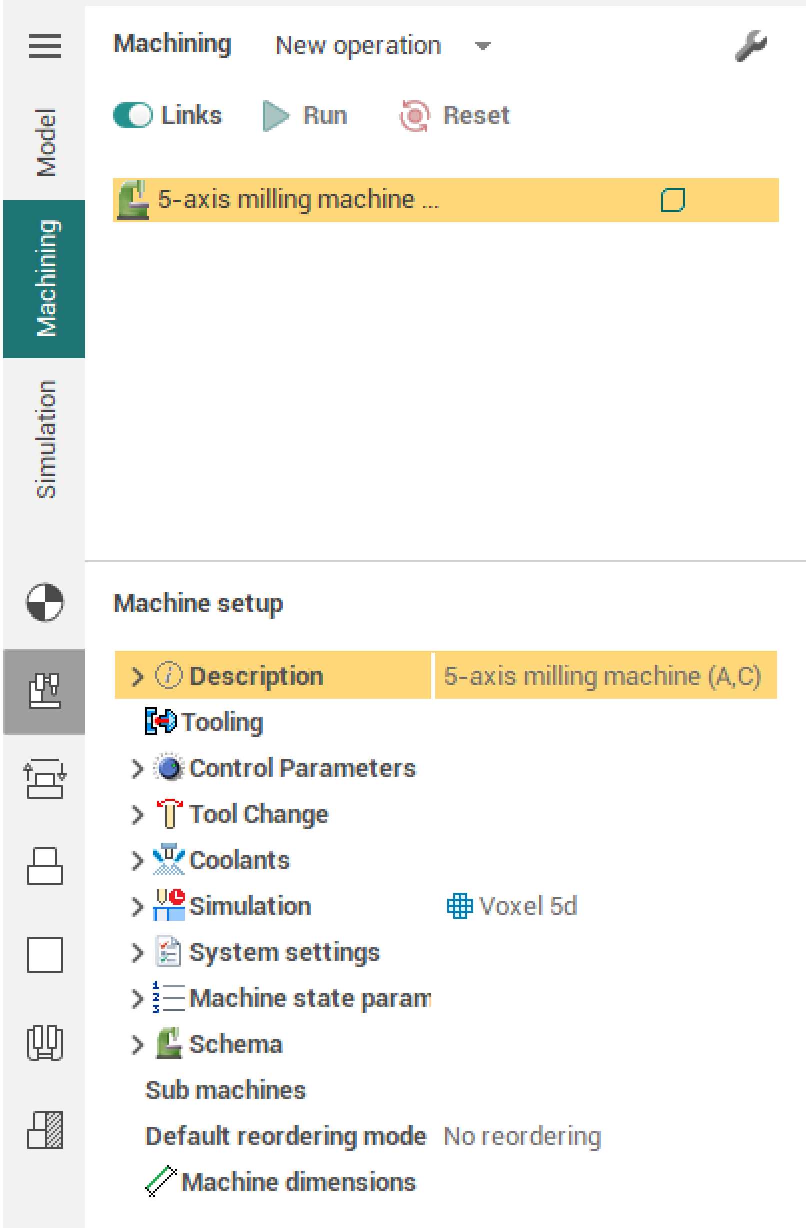
Most controls have two modes for the five axis machining. Those are the default "3 axis" mode and "Tool Center Point Management (TCPM)" mode. The controls behave differently in these modes. Anyway the behavior of the control is composed of the three options.

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.

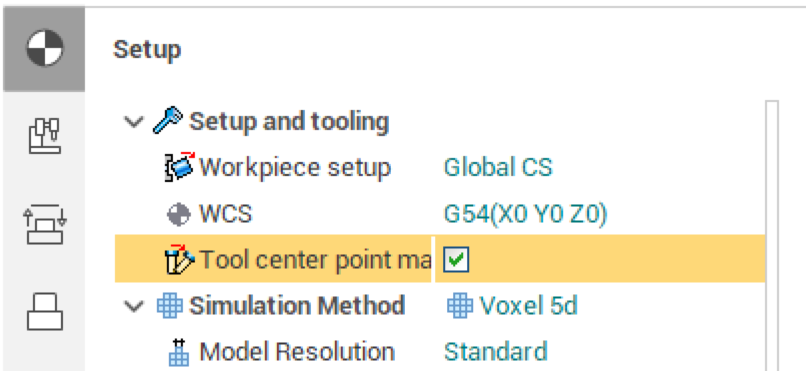
3. Does the axes of the workpiece coordinate system rotate with the rotary table or they stay fixed.

In SprutCAM X 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 SprutCAM X 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 5 axes positioning topic.

Advanced axes limits control feature available for multiaxis operations.

See also:

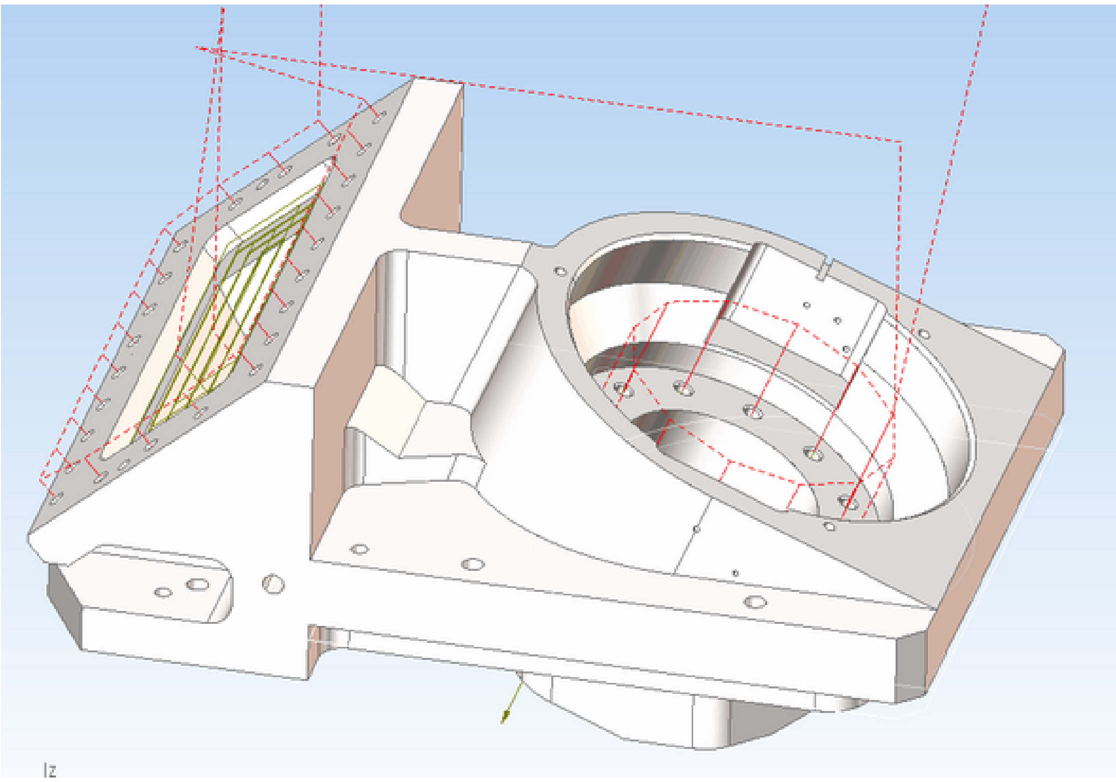
Operations setup

Workpiece coordinate system (G54 - G59)

Operation local coordinate system

5 axes positioning

5.5.4.2 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.

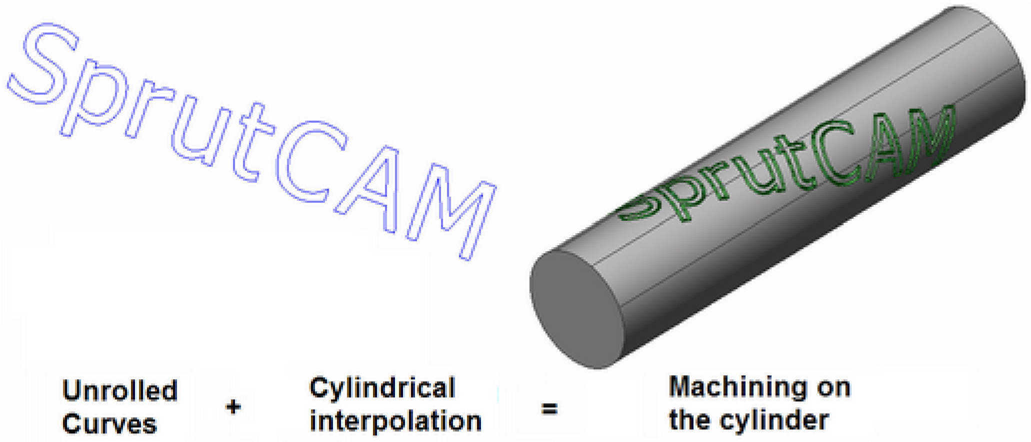
Advanced axes limits control feature available for multiaxis operations.

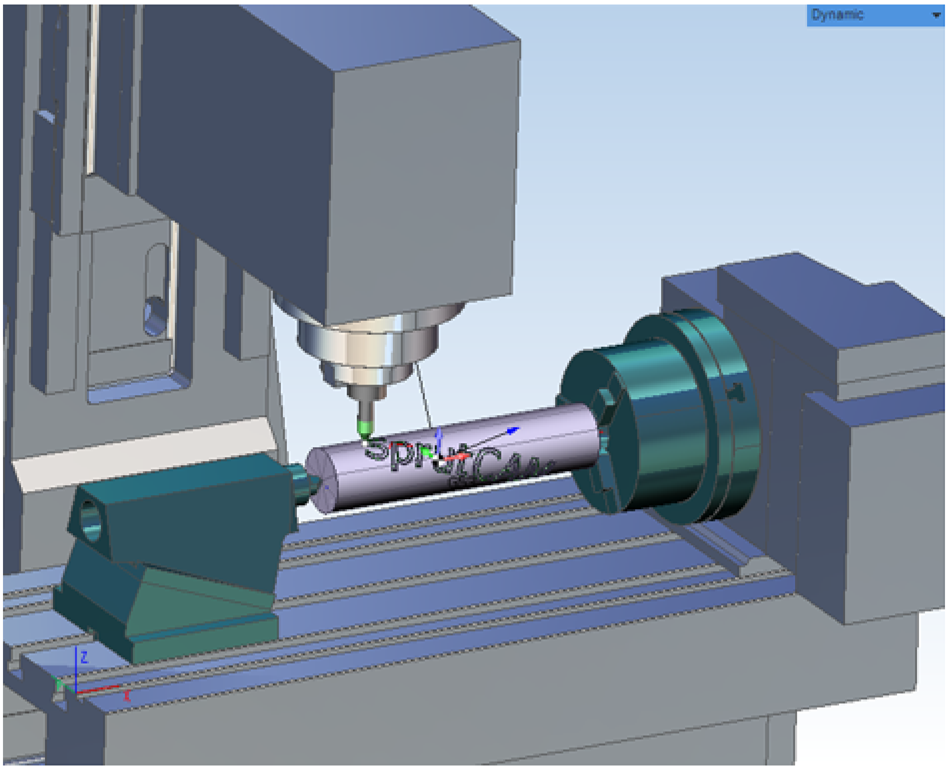
See also:

Operations for 4-axes and 5-axes milling

5.5.4.3 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.





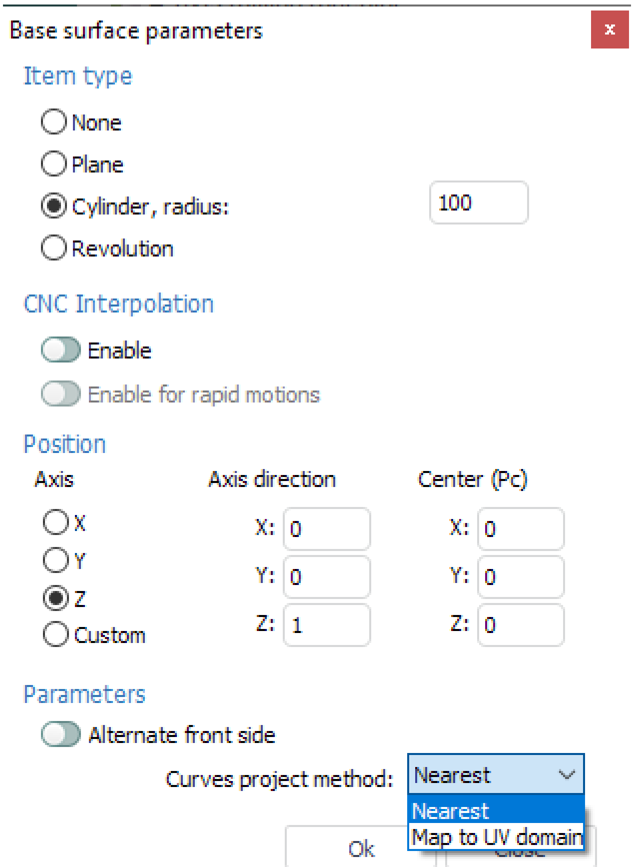
See also:

Operations for 4-axes and 5-axes milling

5.5.4.4 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.

See also:

5 axis machining

Operations for 4-axes and 5-axes milling

Engraving operation

Using design features in an Engraving/Pocketing operation

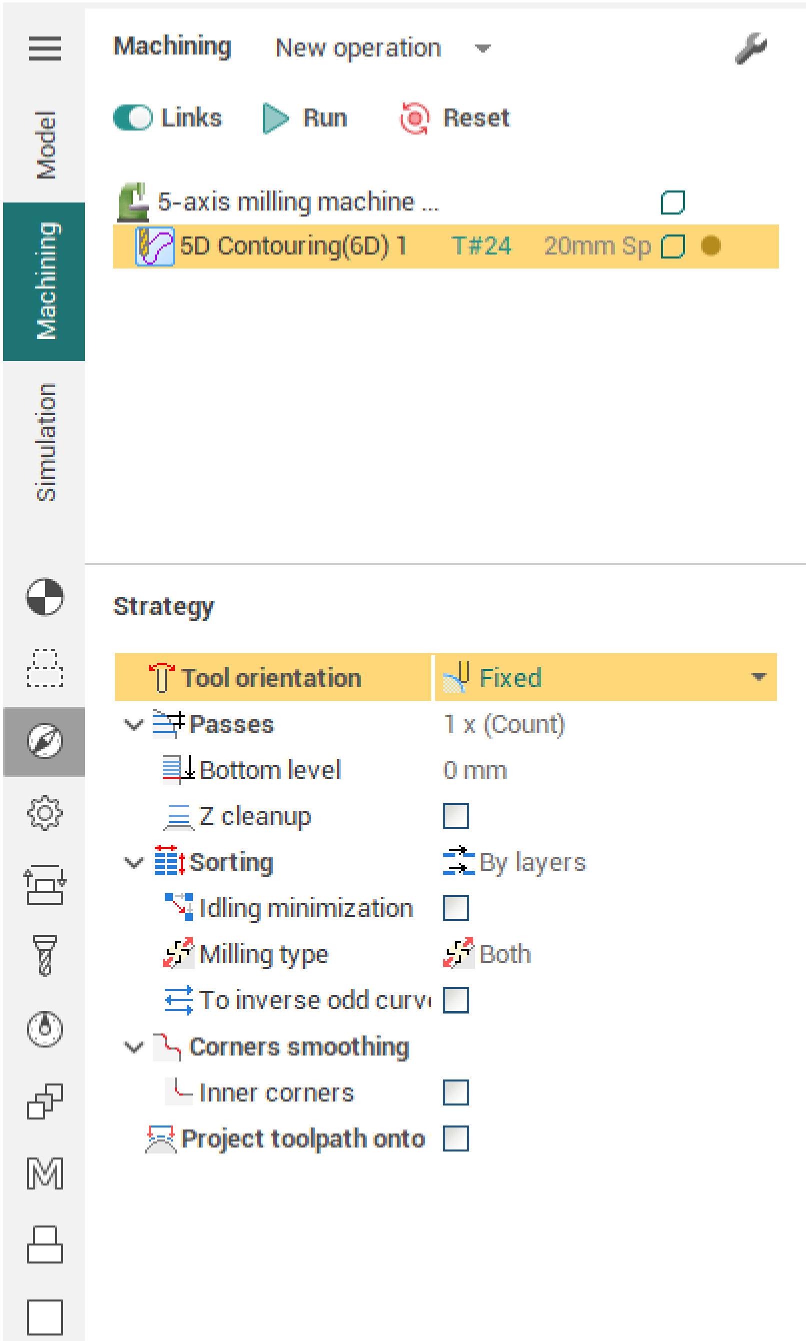
Job assignment for engraving and pocketing operations

5.5.4.5 5D contour and 6D contour operations

5D and 6D contour operations are designed to generate the continues 5-axis tool path. There are few ways to generate the toolpath depends on the way that job assignment is set:

1. The passes along the curves that is lie on the part surface.
2. The passes along the isoparametric curves of the defined surfaces..
3. The passes along the edges of the part.
4. Using custom vectors feature.

The tool, the feeds and speeds, the lead-in and lead-out are defined like in all other milling operations. The way of the rough passes generation and the way of its joining are defined on the strategy page of the parameters dialog.

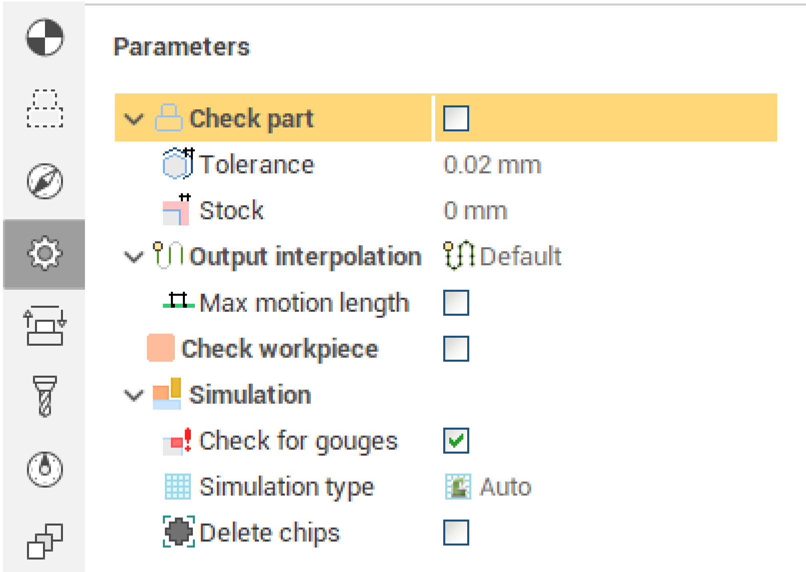


The <Levels> and <Z cleanup parameters> panels allows to generate the additional rough passes under every finishing pass. The <Bottom level> defines the distance from the machined surface. If the value is positive then the tool will undercut the surface. if the value is negative then the tool will over-cut the surface. The <Top level> defines the maximal stock that must be removed by the rough passes. The <Step> defines the layer thickness that is removed at one pass. The <Cleanup height> defines the stock that must be removed at the finishing pass. The <Stock> allows to move both rough and finishing passes from the machined surface. The positive value move the passes away from the surface. The negative value move the passes near to the surface.

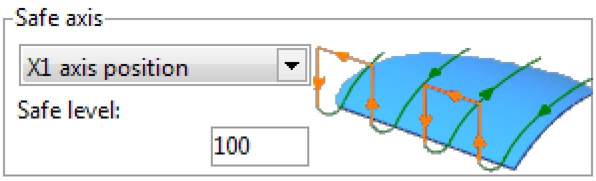
The deviation defines the <Tolerance> of the tool path approximation.

<Milling type> defines the passes joining strategy. In <Climb> mode the machining direction depends on the curves and surfaces. In the <Conventional> mode all passes are inverted. The <Both> mode makes the zigzag tool path.

Sometimes the loops are appear inside the pass. These collision is not checked by default. It's needed to set the <Check part> tick to control the loops. In this case every point of the tool path will be checked. If the path segment is over-cut the part then it will be excluded from the resulted tool path. The <Check workpiece> works like in other mill operations.



The way how to generate the transitions between the passes is defined on the transition folder.



The safe axis is one of the main parameters. It defines the axis name and position, where the rapid motions can be performed. For example if the Axis X is selected and the value equal 100 then it means that the safe plane can be defined as X=100. Parameters <Step-over type> and <Short link> work like in other mill operations.

See also:

5 axis machining

Operations for 4-axes and 5-axes milling

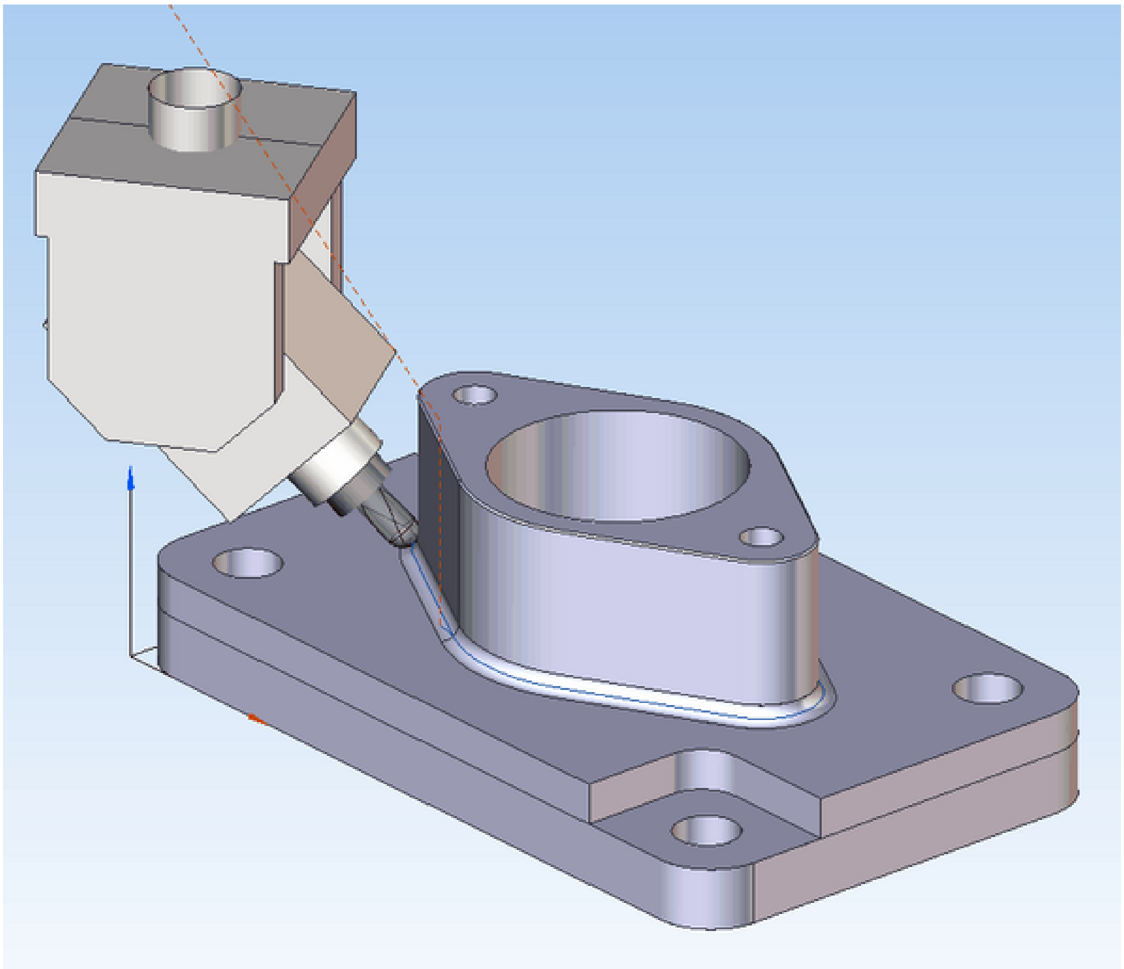
5-axis milling along the isoparametric curves

5-axis milling of the profile on the surface

5-axis milling of the ruled surfaces by the flank of the mill

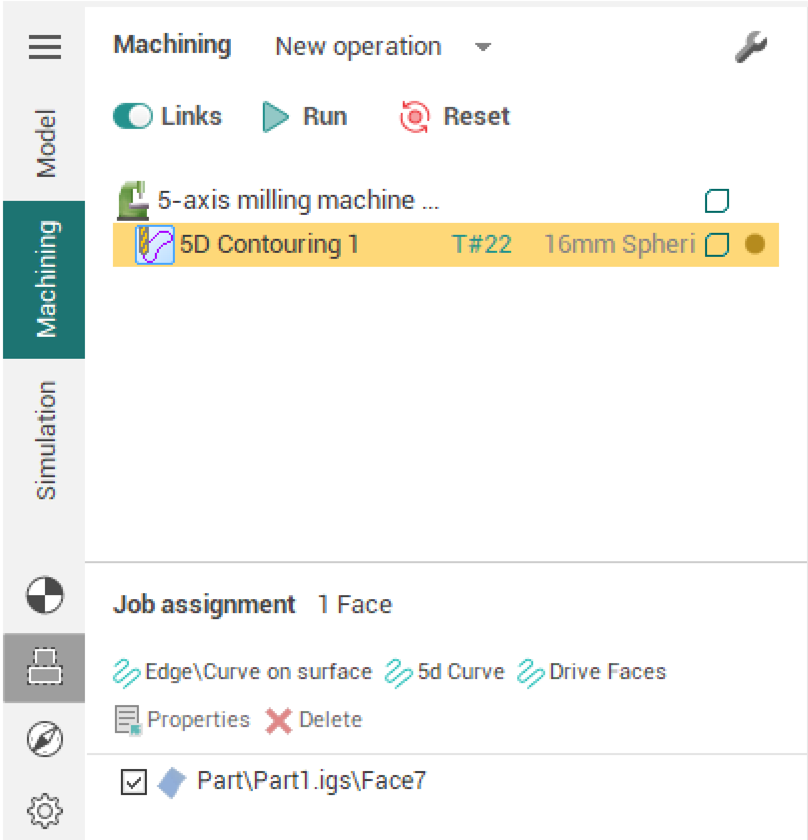
Using custom vectors feature

5-axis milling along the isoparametric curves

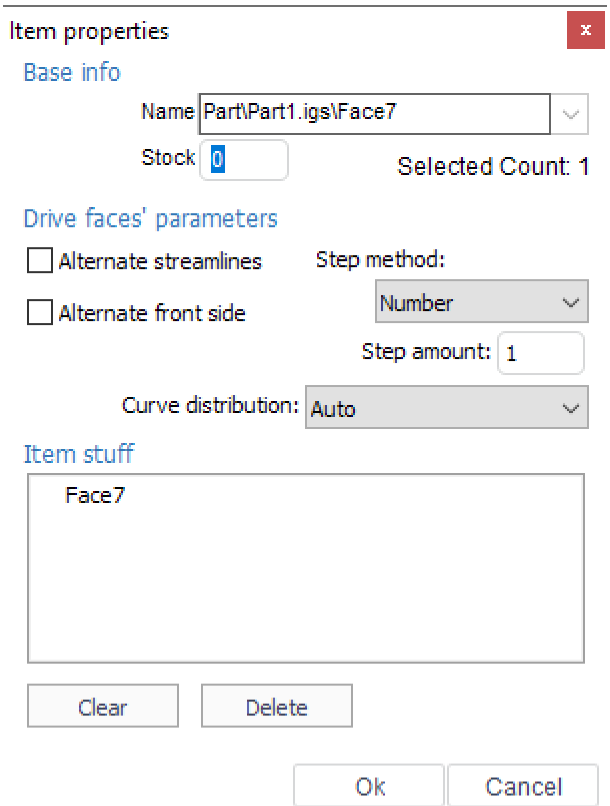


The isoparametric curves of the surfaces depends on its way of creation. Frequently its reflect the topology of the surface and so is very useful is machining as a work passes. Different fillets can be machined this way very well.

The surfaces, the isoparametric curves of that must be used, is added to the job assignment of the 5D contouring operation. 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 alone click on the <Drive faces> button then it will be machined together.

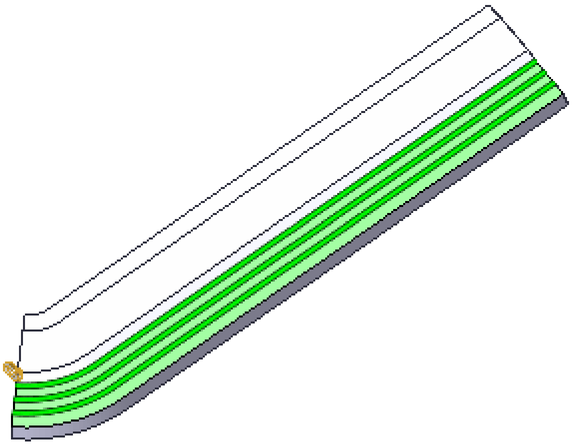


The quantity of the passes for the added surface is defined in the <Item properties> dialog. Select the item and click the <Properties> button to open it.

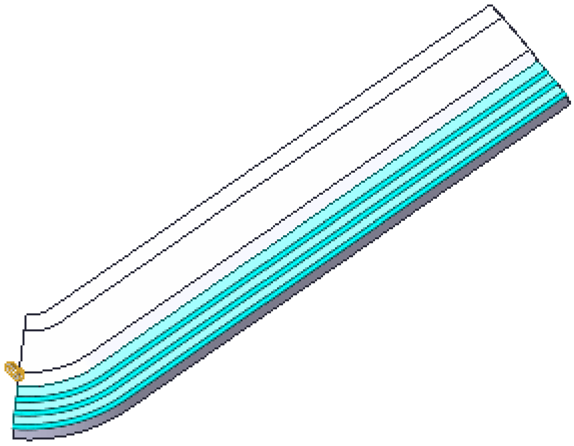


The pass count can be defined directly, or by the step value, or using the scallop value. The <Alternate streamlines> tick changes the passes direction to the orthogonal ones. The <Alternate front side> changes the machined side of surface to the back one.

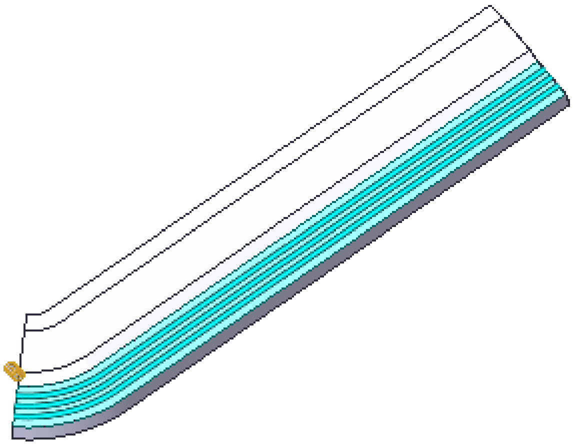
- <Curve distribution> – this option controls the way SprutCAM X produces isoparametric curves on the guide surface:
- <From Start> – the first curve is placed on the left edge of the surface and other curves are placed to the right of the first one.



- <From End> – the first curve is placed on the right edge of the surface and other curves are placed to the left of the first.



- <From Center> – the first curve is placed in the middle of the surface, other curves are placed to the left and to the right of it.

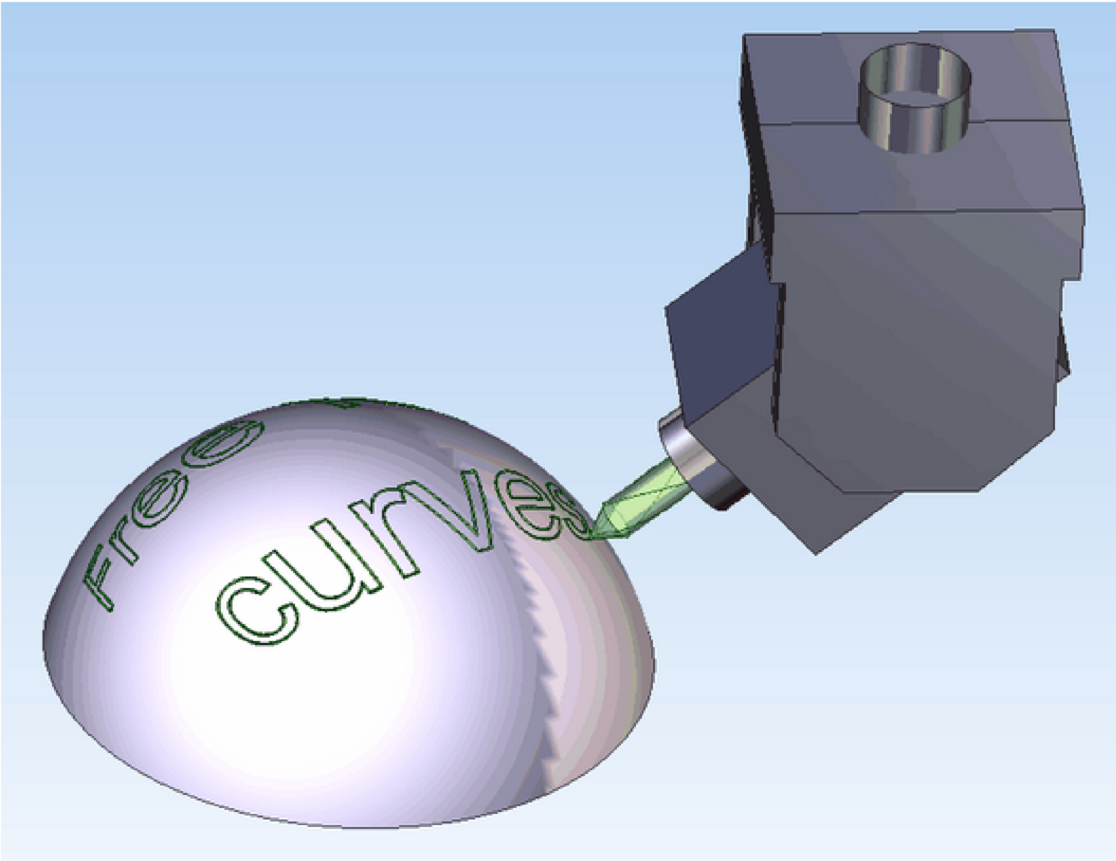


- <Auto> – curve placing method is defined by SprutCAM X based on the machining method (Face, Flank) and tool axis orientation.

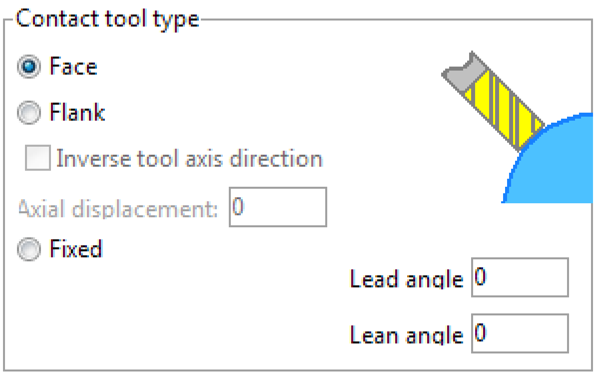
See also:

5D contour and 6D contour operations

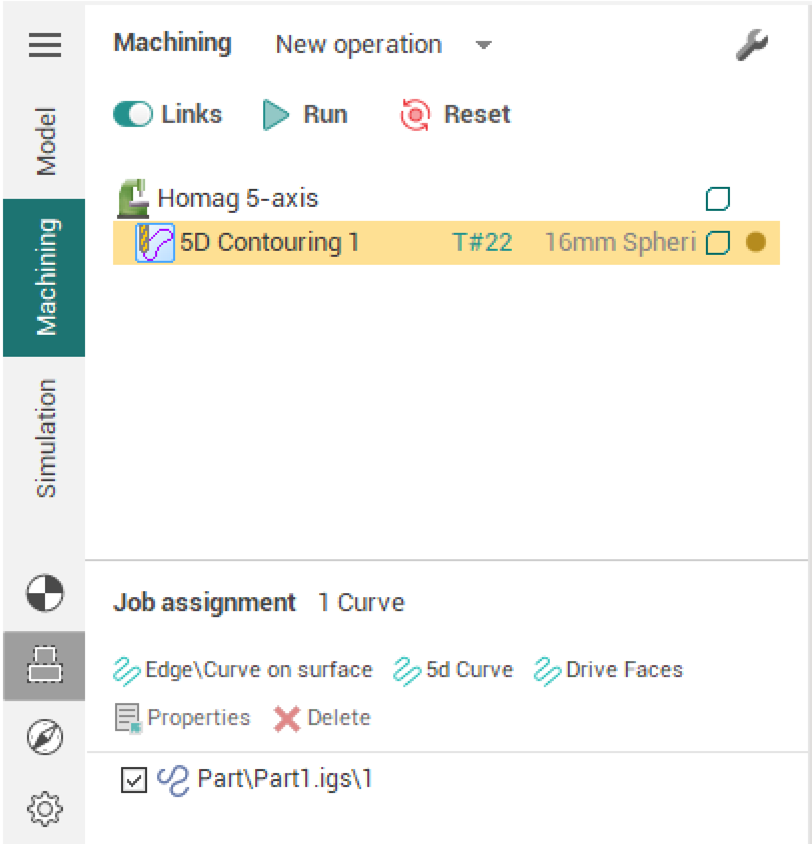
5-axis milling of the profile on the surface



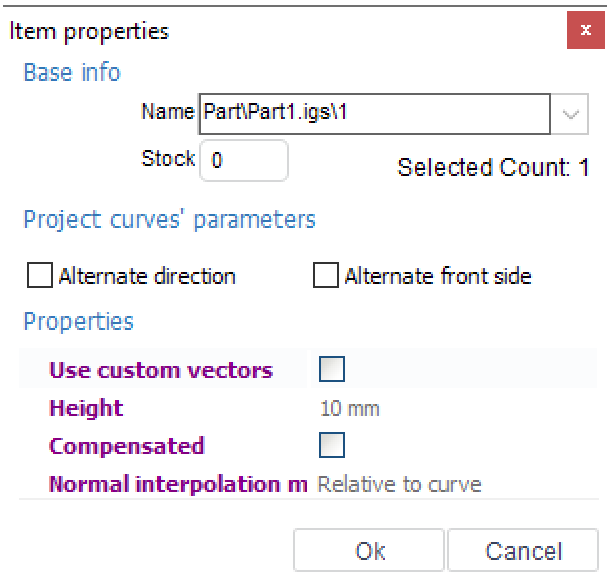
There is the possibility to generate the 5-axis tool path as the projection of the arbitrary curves on the part surface. In this case the tool tip will move in touch with the surface. The passes can me moved away or moved near by the stock or by the levels. The tool axis is located perpendicular to the surface. The tool axis can be inclined using the <Lead> and <Lean> angles. These parameters are defined in the parameters window on the <Strategy> page.



The surfaces or the solid, on that the curves are projected, must be defines as the part of the of the operation. The projected curves is added to the job assignment by the <Project Curves> button.



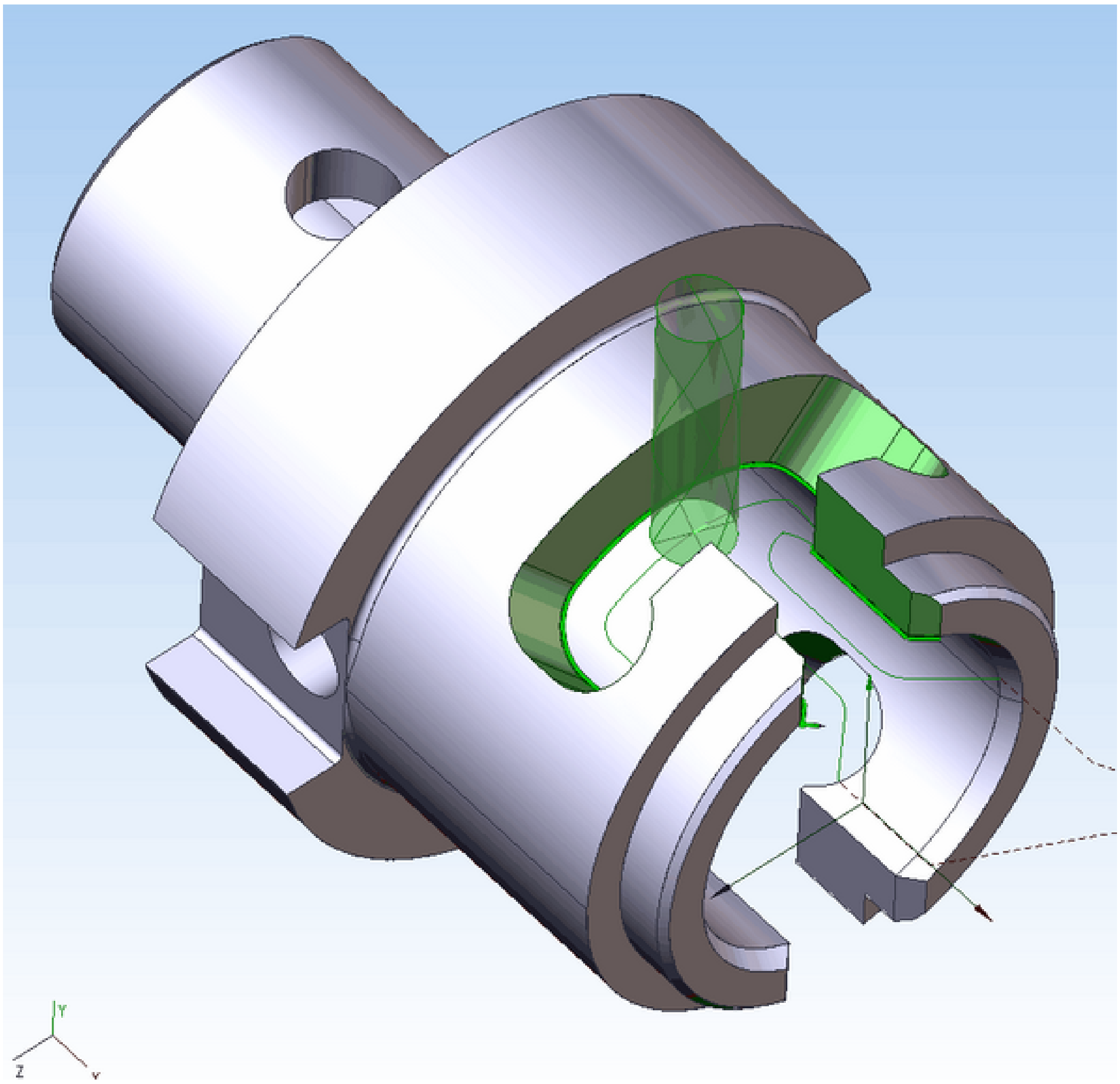
The item properties window allows to change the <Direction> and the <Side> of the machining. Select the items and click <Properties> button to open it.



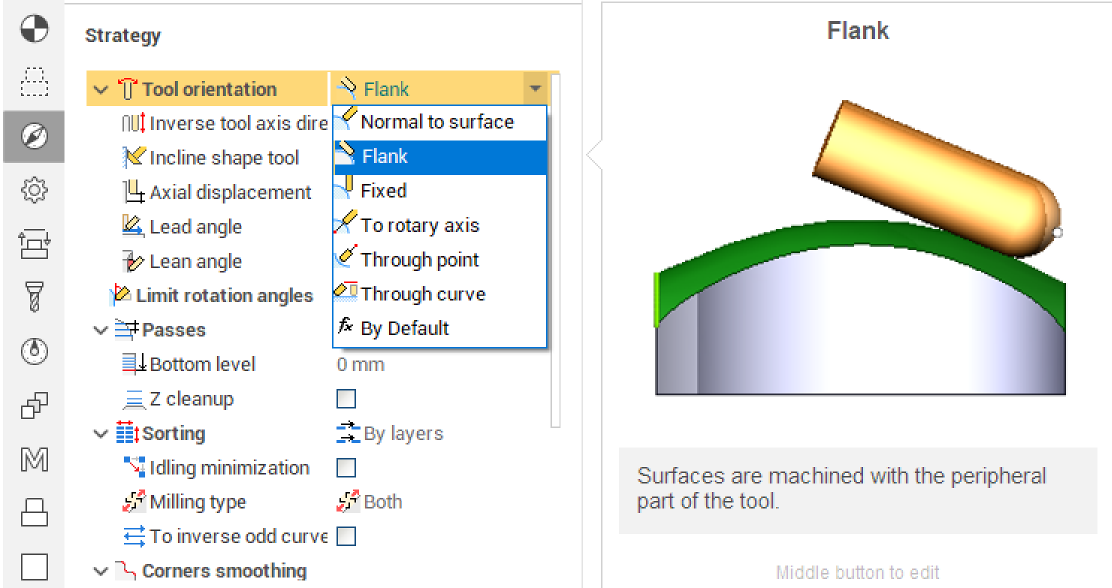
See also:

5D contour and 6D contour operations

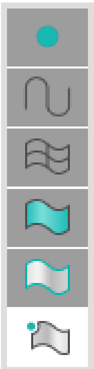
5-axis milling of the ruled surfaces by the flank of the mill



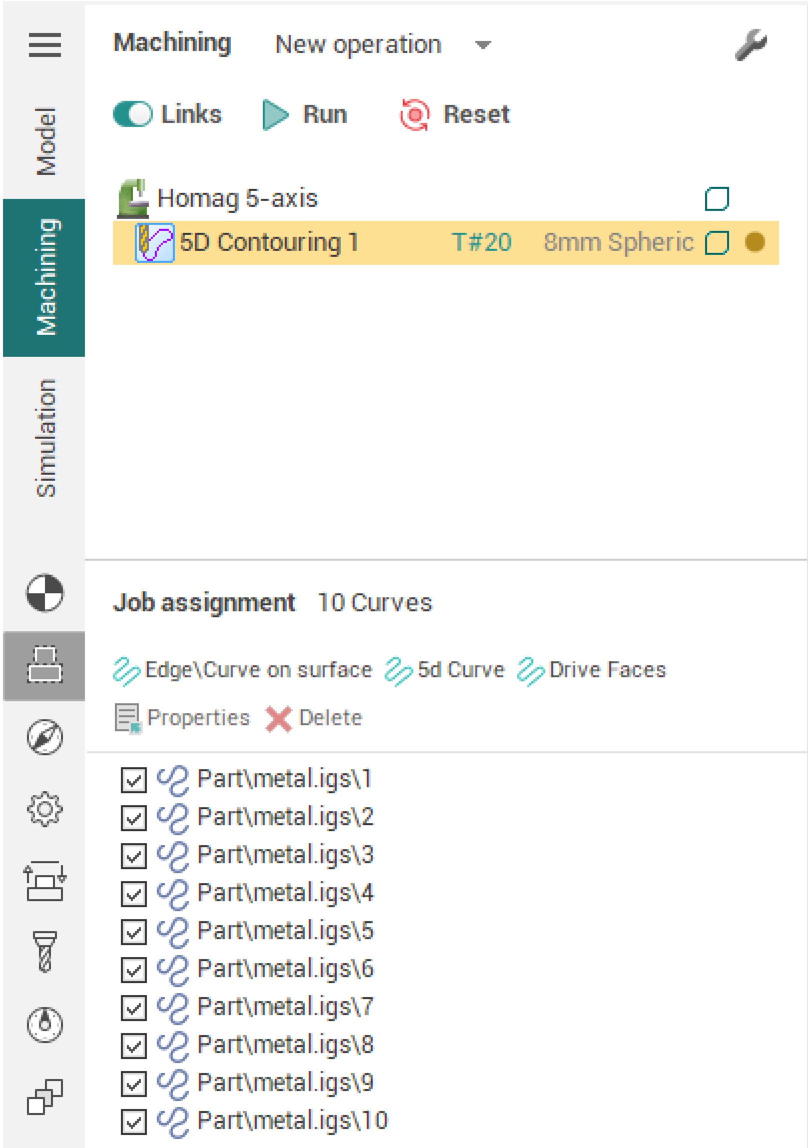
Set the <Contact tool type> parameter to the <Flank> position to machine the ruled surfaces by the flank of the mill. The <Inverse tool axis direction> tick changes the tool axis direction to the opposite one. <Axial displacement> defines the additional shift along the tool axis.



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.

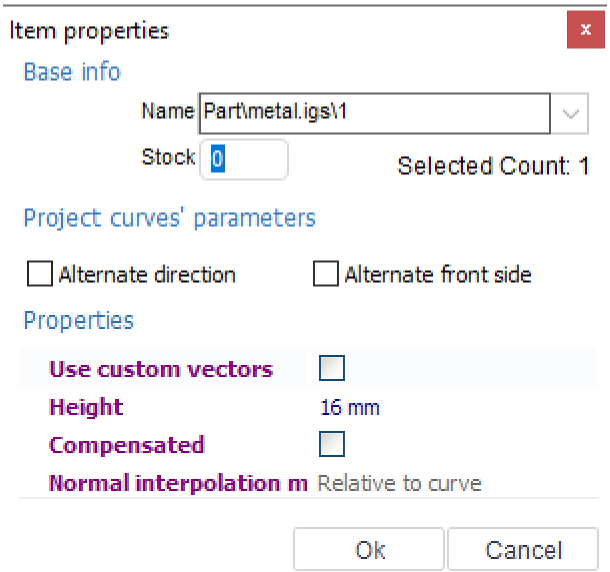


If the edge selection filter button is down but the edges is still not highlighted then sew the faces.



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.

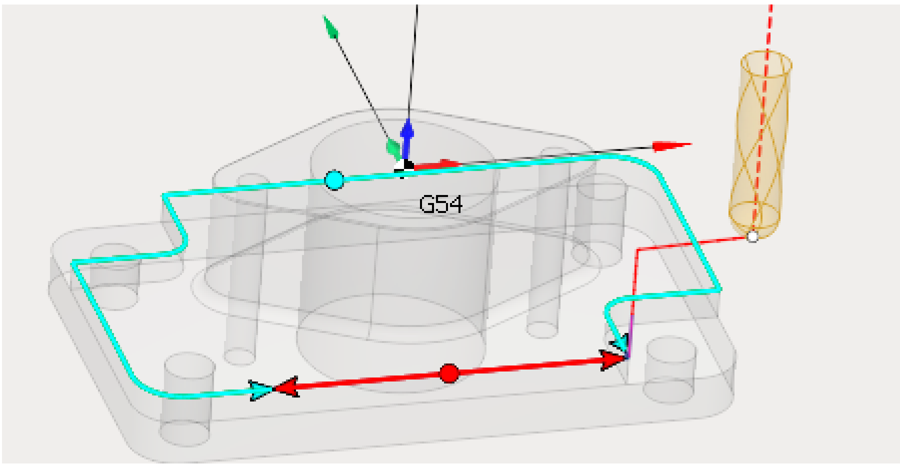


See also:

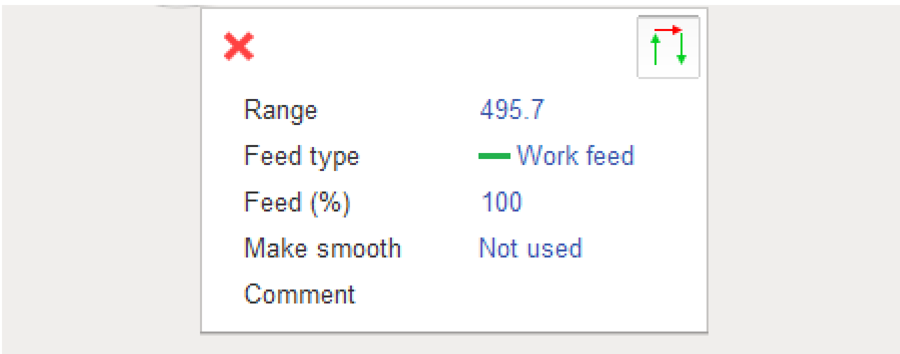
5D contour and 6D contour operations

Feeds control feature

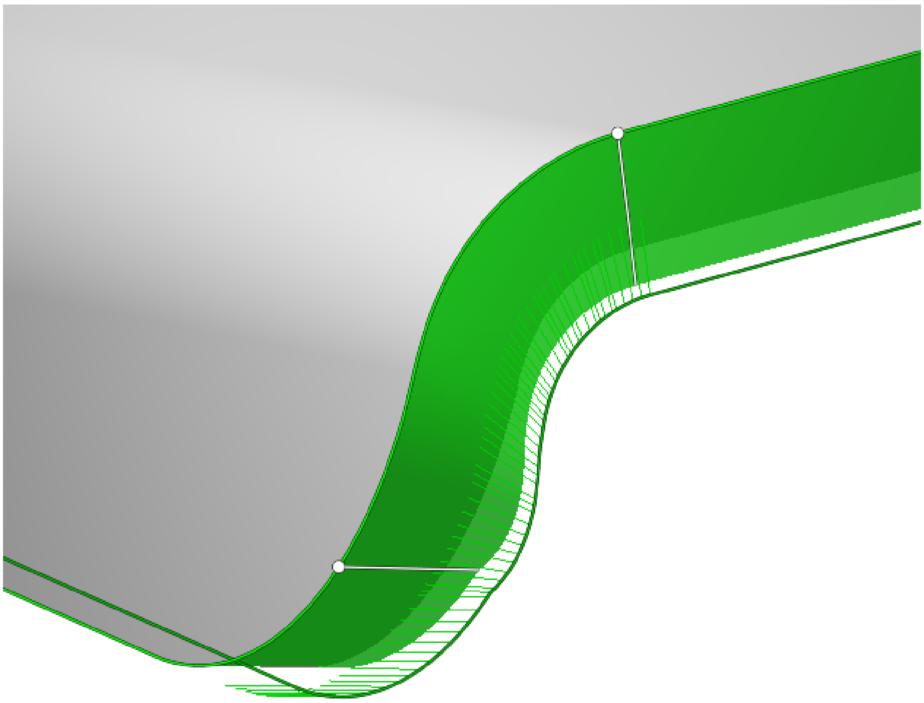
This function allows you to create feed value contour. In order to have the ability to edit feed value you need to click **Feed control mode**.



In the pop-up panel, you can control the range, feed type, speed and interpolation type.

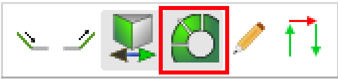


Using custom vectors feature



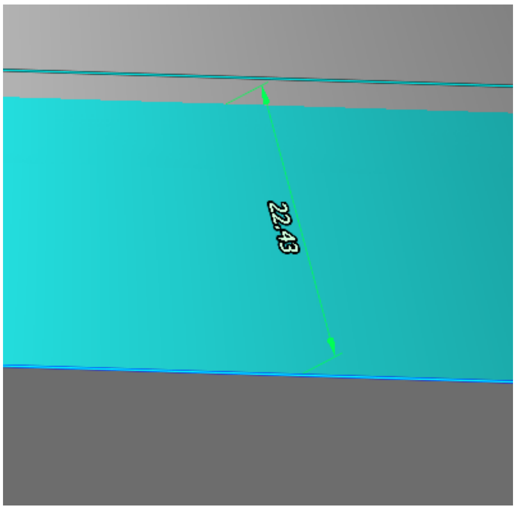
Vectors allow to set the orientation of the tool at any point of the contour. In order to have the ability to edit vectors you need to click **Use custom tool vectors**.

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, SprutCAM X will try to set the vectors automatically.



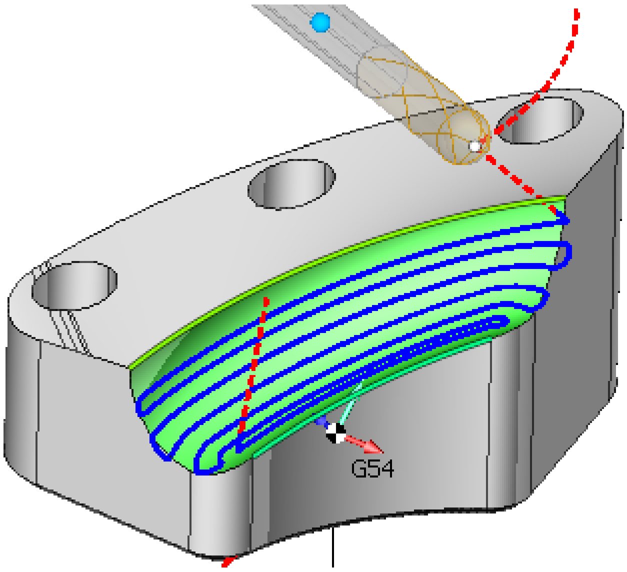
Using **Custom vectors** allows you to edit the original surface to be processed. Editing methods:

- 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.

5.5.4.6 5d surfacing operation



The finishing operation allows the surface to be machined with a 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).

Typical workflow

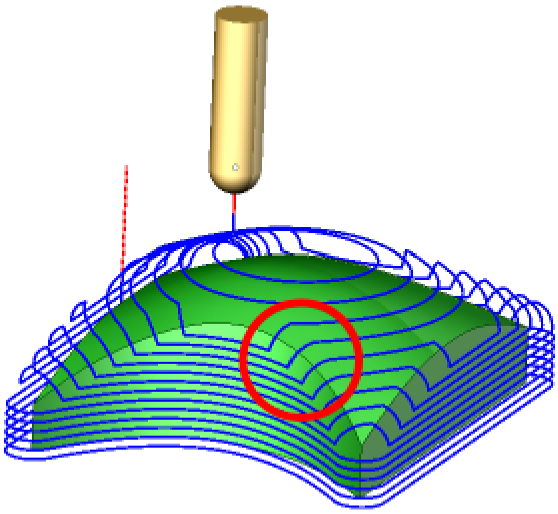
1. Create the operation
2. Add the Machining surfaces to the Job assignment
3. Select the toolpath strategy
4. Choose between the tool center and tool contact calculation modes.
5. If needed, change the step-over and/or the toolpath margins.
6. Select the tool axis orientation mode.

7. If required, turn on the roughing passes.

8. Calculate the toolpath.

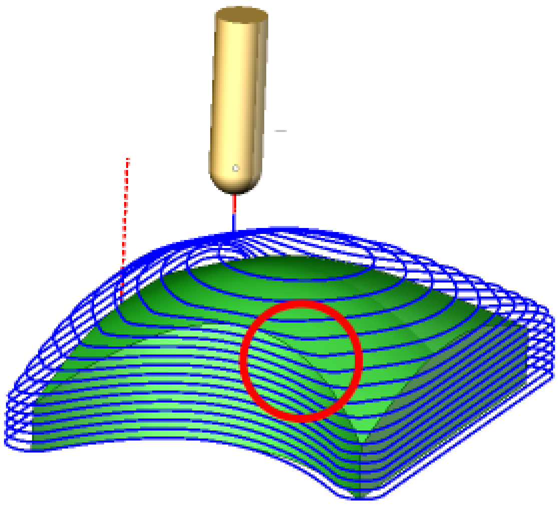
Comparison of the toolpath based on the center and the tool contact point.

1. 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.

2. Calculation based on tool center



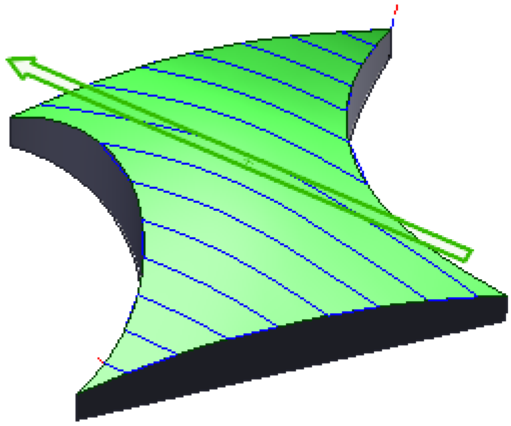
In this mode the work passes are generated in such a way that at first the machining surfaces are being offset either by the tool shape itself for 3 axis machining or by the tool radius along the surface normal for 5 axis machining and only then the sections of the offset surfaces are calculated. For 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.

Strategies

Parallel to plane

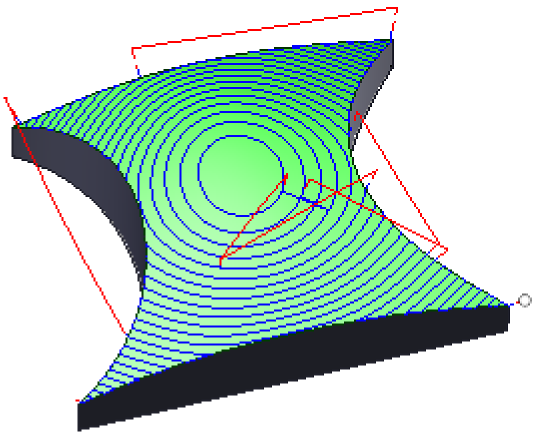
The passes are generated as a result of intersection of the machining surfaces and parallel planes surfaces. Three options are to choose from.

1. Parallel to vertical plane



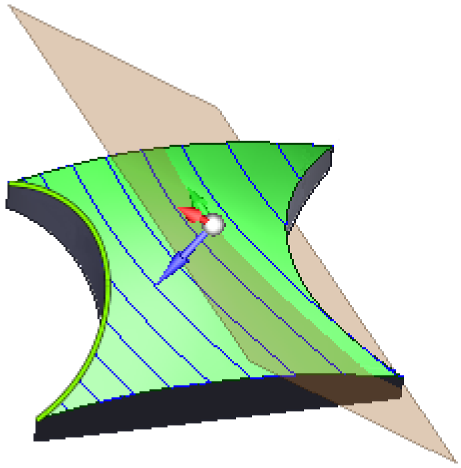
The planes are parallel to the tool axis, as in the Plane toolpath. Additionally the angle of rotation of the planes around the tool axis can be specified.

2. Parallel to horizontal plane



The planes are perpendicular to the tool axis, as in the Waterline toolpaths.

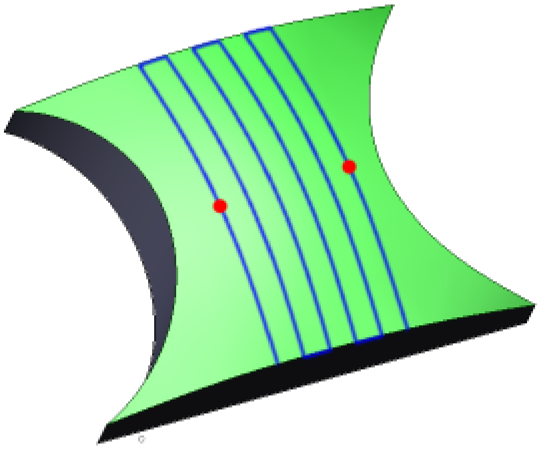
3. Parallel to 3d plane



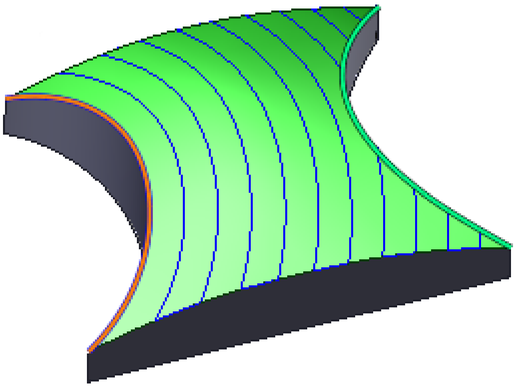
The planes can be freely oriented in space regardless and independently from the tool axis orientation.

Margins

It is possible to limit the generated passes by two points.



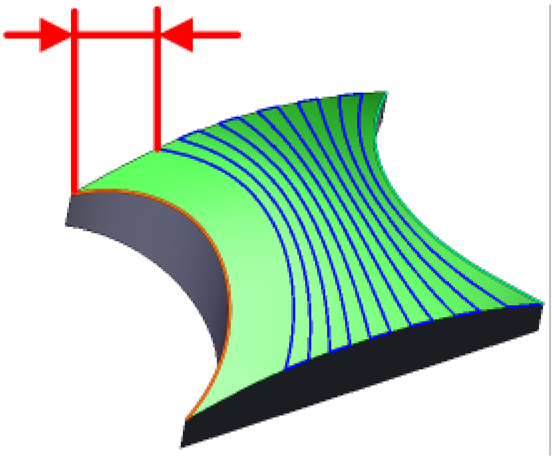
Parallel to curve



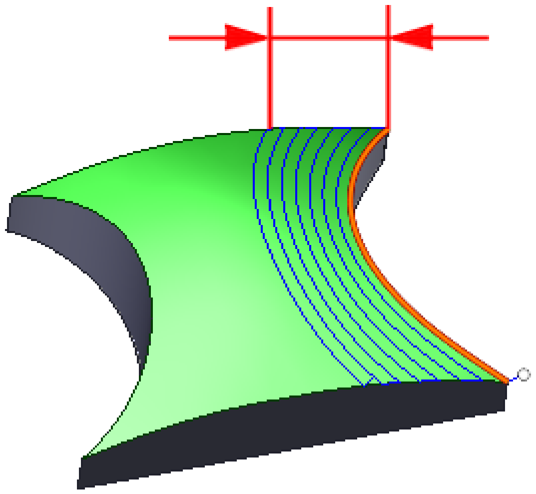
The passes are generated by finding points on the machining surfaces that lie on the same distance from the First Curve. Unlike with the Scallop toolpath the step-over between passes is not guaranteed to be constant.

Margins

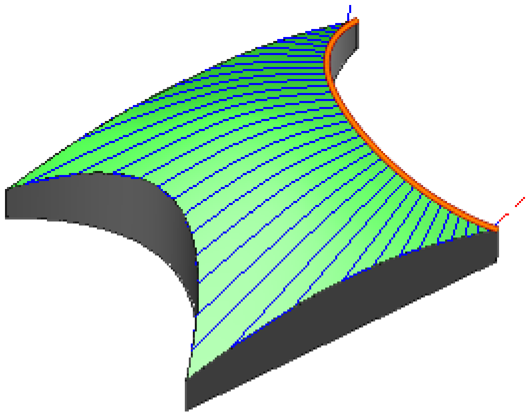
Use the Start margin to set the starting offset for the first generated pass.



Use the Zone Width to limit the number of generated passes. You can either specify an exact value for the zone width or define the job zone with a point.

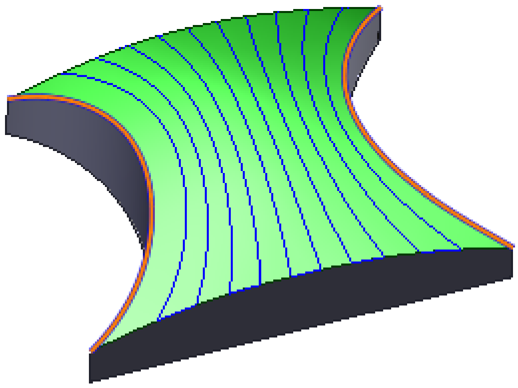


Across curve



The passes are generated by sectioning the machining surfaces with planes perpendicular to the First curve.

Morph between two curves

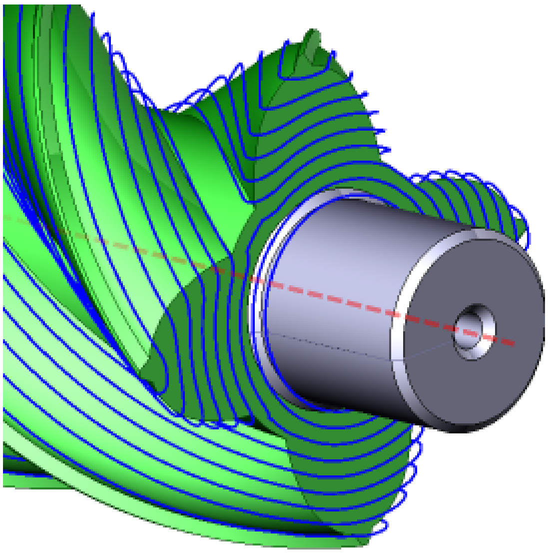


The passes are calculated by finding points on the machining surfaces which satisfy the criteria that the ratio of the distance from the given surface point to the First curve to the distance from the point to the second curve stays the same for a given pass.

Margins

Use the Start margin and the End margin to set the offsets from the generated passes to the First and the Second curves respectively.

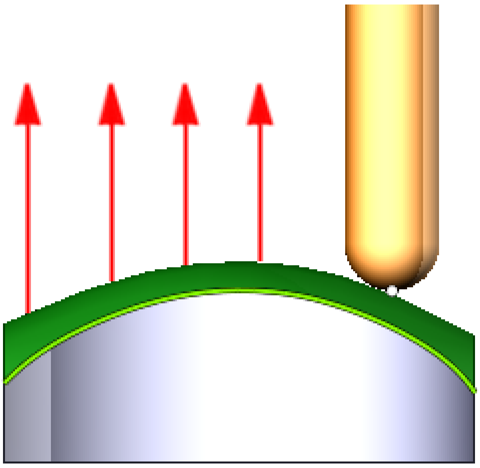
Around rotary axis



The passes are calculated as sections of the machining surfaces with the series of cylinders around the rotary axis.

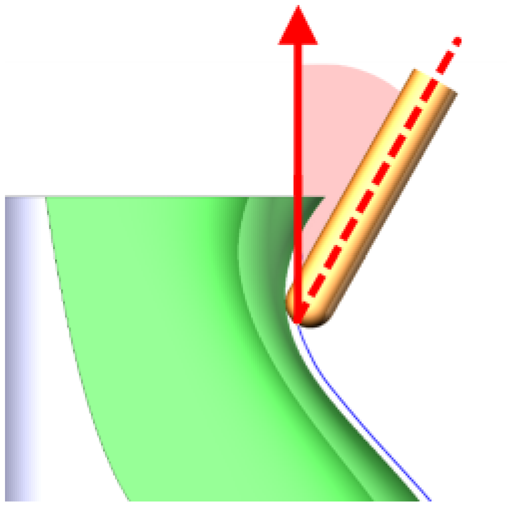
Tool axis orientation modes

Fixed

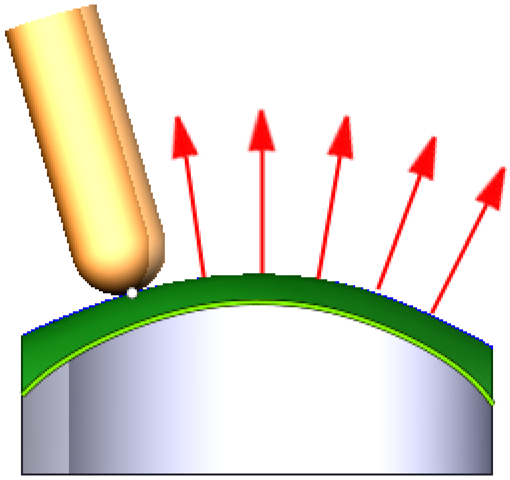


In this mode the tool axis orientation stays the same for the entire toolpath (unless the vertical clearance angle is specified). Basically what you get is a conventional 3 axis milling toolpath.

The Vertical clearance angle feature automatically tilts the tool away from the wall surfaces in places where the slope of a surface is steep or negative (a surface normal looks down the tool axis). An additional clearance angle can be used. It allows simple machining of undercut areas.

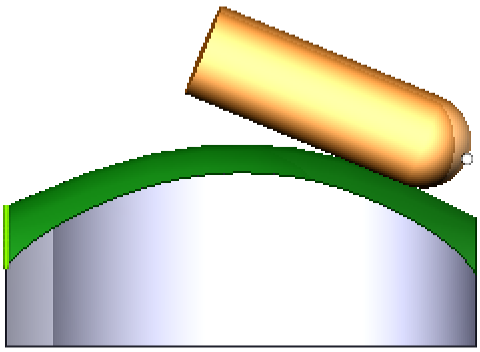


Normal to surface



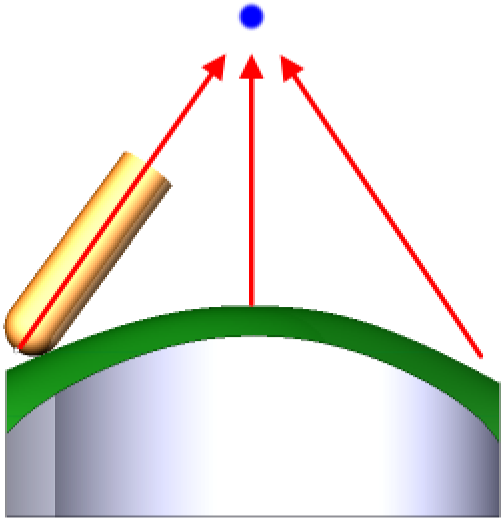
The tool is oriented by normal to machining surfaces. Additionally the lead and lean tool angles can be applied to further tilt the tool along or to the side from the cutting direction.

Flank



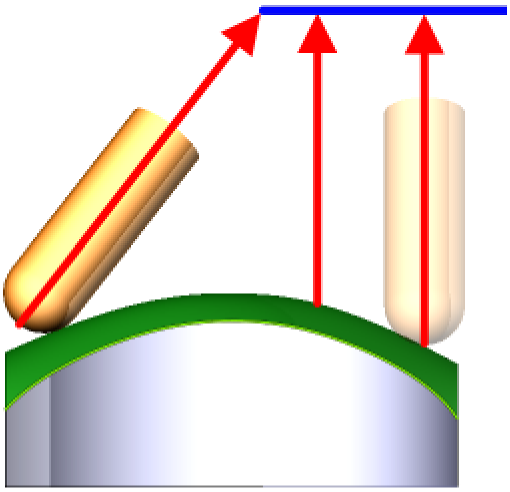
The tool contacts machining surfaces with the peripheral part (cylindrical part for the cylindrical mills). Additionally lead and lean angles can be applied. The strategy can be used for swarf milling.

Through point



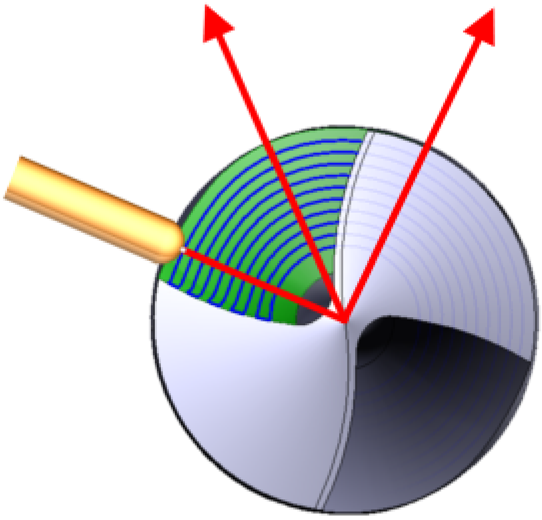
The tool axis is oriented to the specified point.

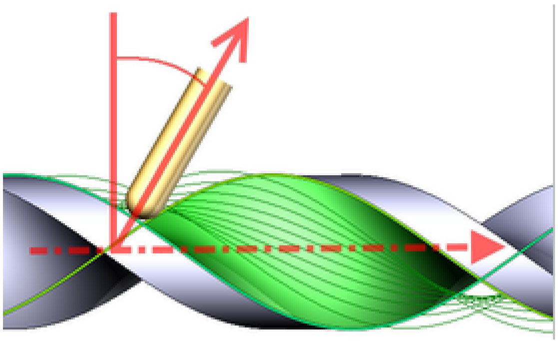
Through curve



The tool axis is oriented to the nearest point of the specified Tilt curve.

To rotary axis



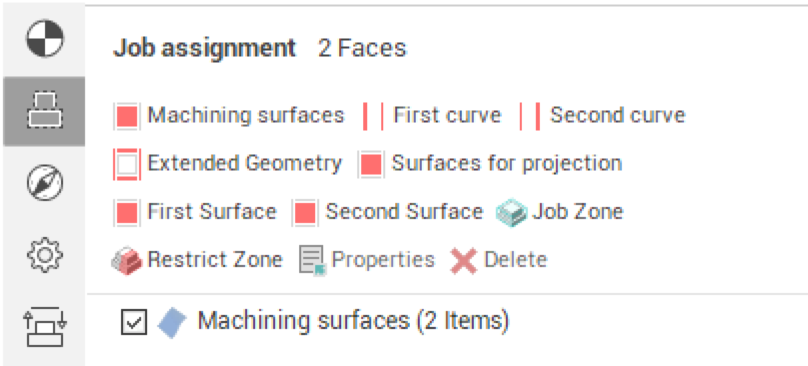


The tool axis is directed to the rotary axis, as in the rotary machining. Additionally the side angle to the rotary axis can be specified

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.

Job assignment



Machining surfaces

The machining surfaces define where the toolpath will be calculated

First curve

The First curve is used in the Parallel to curve strategy to define the curves parallel to which the passes are calculated and in the Morph between two curves strategy to define the first curve. You can select one or more not necessary connected curves or edges as the First curve.

Second curve

The Second curve is used in the Morph between two curves strategy. You can select one or more not necessary connected curves or edges as the second curve.

Tilt curve

The tilt curve is used for the Through curve tool axis orientation mode.

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.

Restring zone

Use restrict zones to easily create restriction geometry from curves and edges.

Gouge control

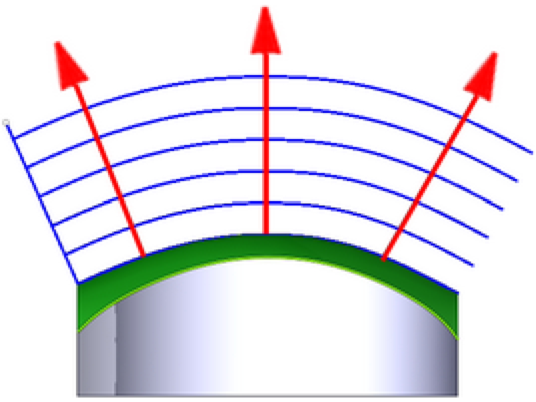
By default the Check part option is enabled. It means that the operation generates a gouge-free toolpath. Often when the geometry of the machining surfaces is simple and the minimal curvature of the machining surfaces is larger than the tool radius you can disable the Check part option to speed up the toolpath generation.

Roughing

Enabling the roughing option turns on the roughing passes. The number of layers and the step between layers can be set. The Check workpiece option can be used to eliminate air cutting.

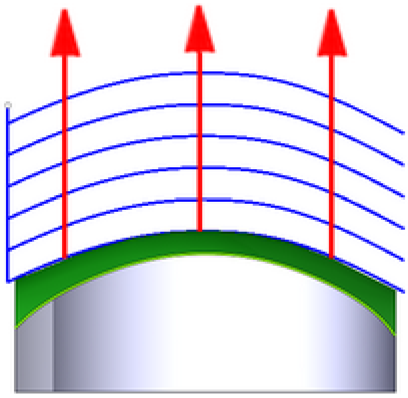
There are several different modes of calculation of the roughing passes.

1. By surface normal



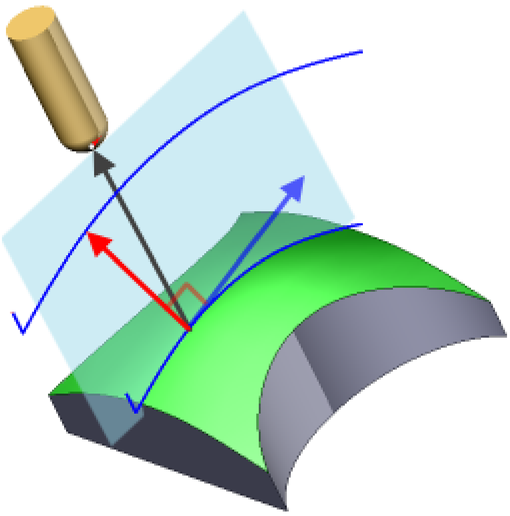
The roughing passes are calculated as offsets from the finishing passes along the surface normal.

2. Along tool axis



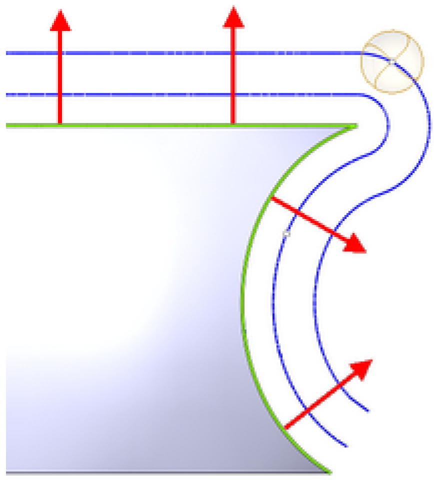
The roughing passes are calculated by simple shift of the finishing passes along the tool axis.

3. In tool plane



The roughing passes are calculated as offsets from the finishing passes in the frontal tool direction. It works best for the parallel to vertical plane strategy and when machining surfaces with the front of the tool.

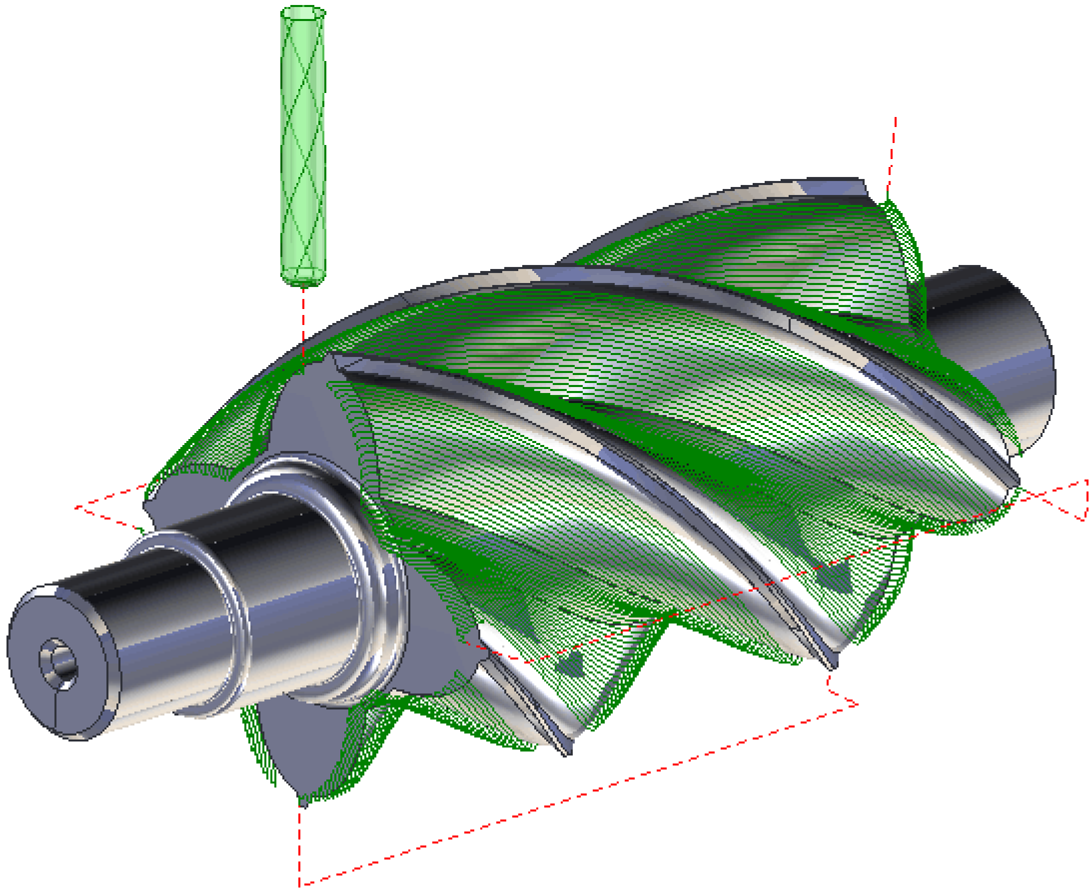
4. Perpendicular to tool axis



The roughing passes are calculated as offsets of the finishing passes in the plane perpendicular to the tool axis. The mode works better for the parallel to horizontal plane strategy, and when machining surfaces with the peripheral part of the tool.

5.5.4.7 Rotary operation

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.

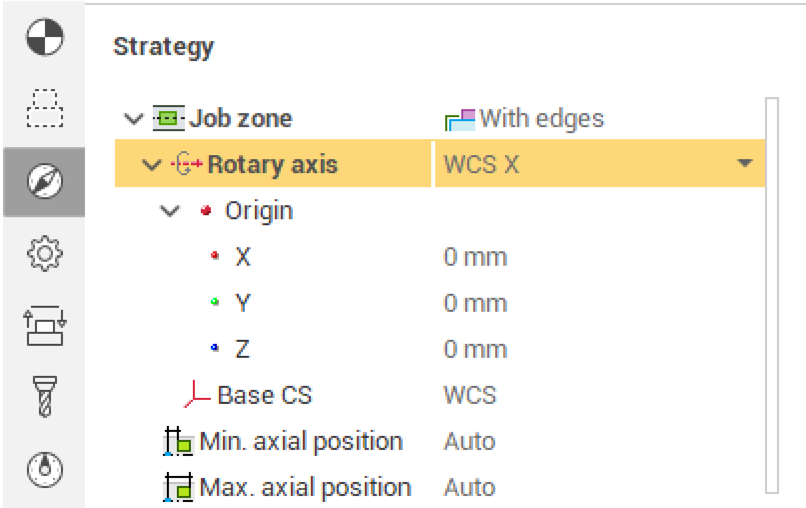


The main peculiarity of the operation is that it uses the 4th rotary axis together with the linear axes. The 5th axis (when it exists) is fixed. Sometimes the 5th axis can be used also.

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.

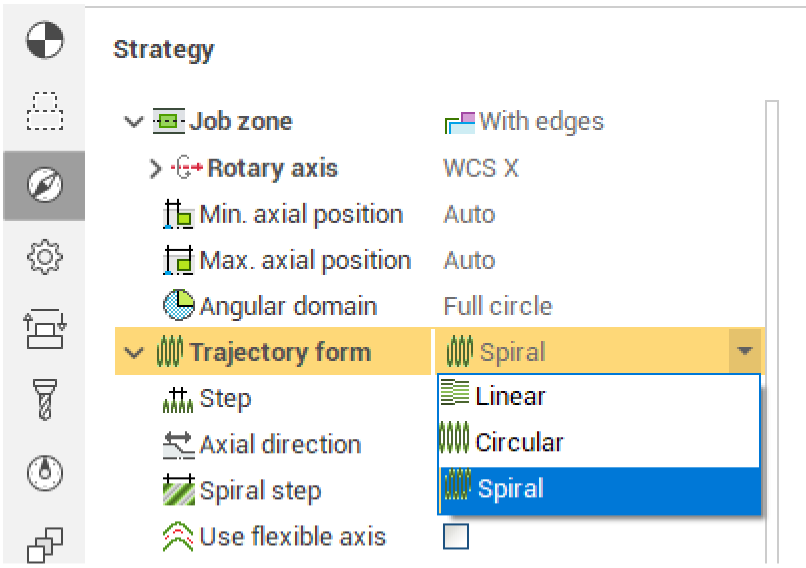
The workpiece can be defined as the box, cylinder, solid of revolution, rest material, or free-form solid that uses faces and meshes of the geometry model. The operation check on the objects that is specified in the fixtures folder.

The rotary axis position can be defined in the properties inspector that is located as shown in the next picture.

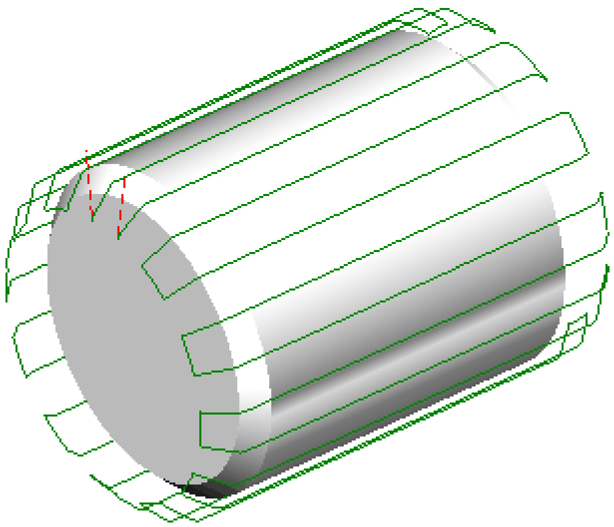


The <Origin> field defines the point that lies on the rotary axis. The orientation group defines the vector of the rotary axis. If the mode is set to the <Along X>, <Along Y> or <Along Z>, then the axis of rotation will be oriented along the corresponding axis of the coordinate system. If mode is <Custom> then axix vector must be defined manually in the fields X,Y,Z of custom direction group.

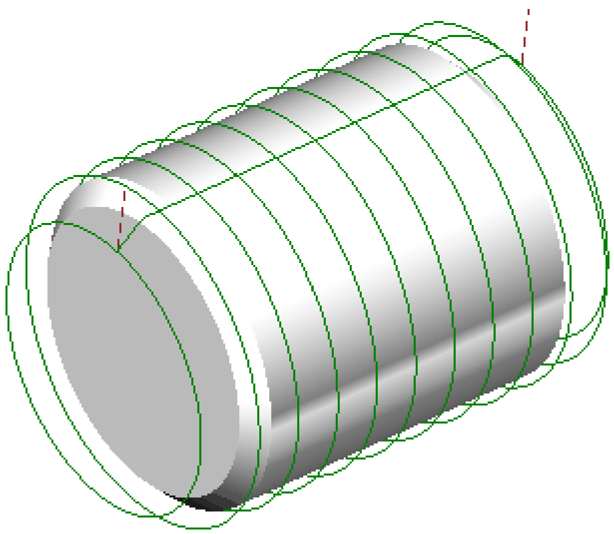
The machining is performed by the series of the passes. The passes can be different shape and located by different ways. The shape and the location of the pass is depended on the Trajectory form that is defined on the strategy folder in the parameters dialog.



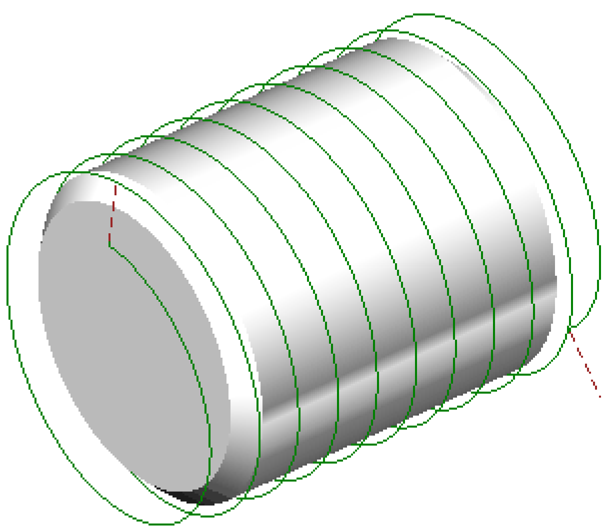
if <Linear> is selected then working tool path is parallel to the axis.



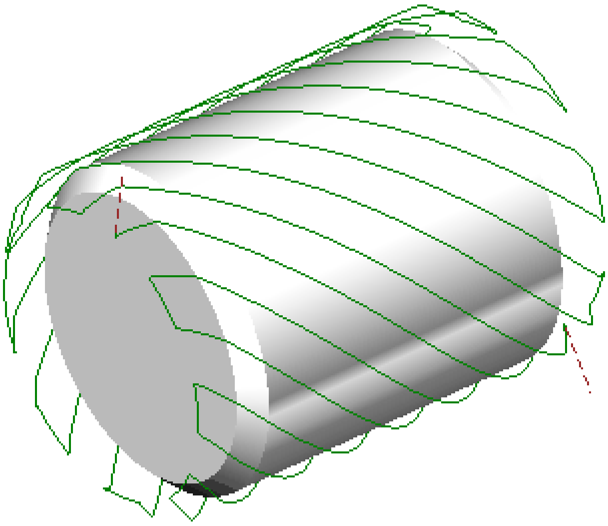
if <Circular> is selected then working tool path lies the plane that is perpendicular to the axis.



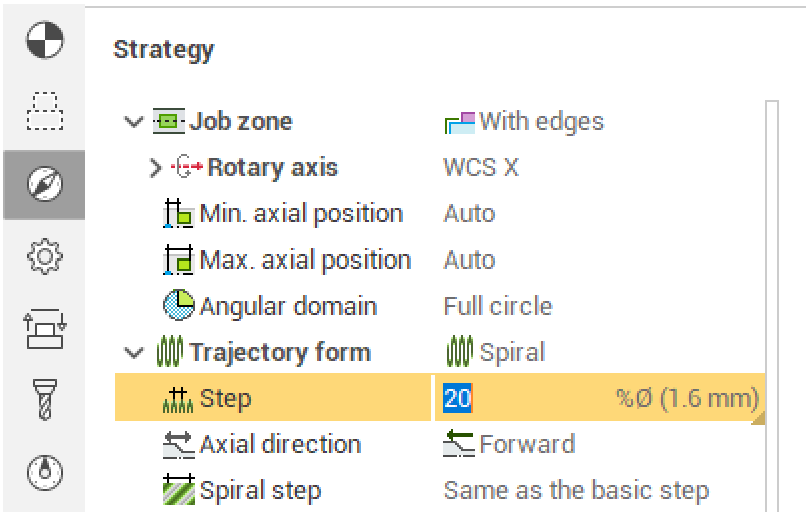
if <Spiral> is selected then working tool path has the helical shape. if the step definition mode is set to the <Same as the basic step>, then the tool path is a one continues helic. The spiral step can be also specified as an absolute value, percents of the tool diameter or the angle in degrees.



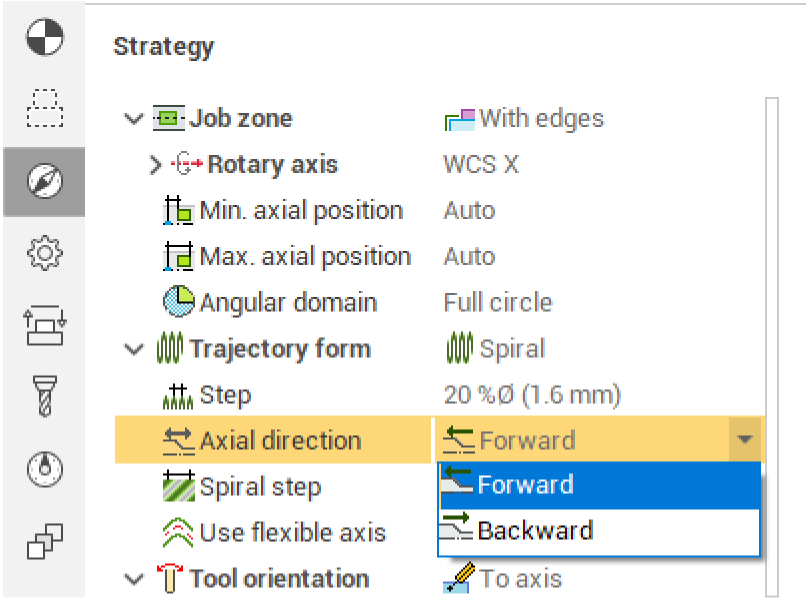
If the <Spiral step> is not equal to the step of machining then tool path is a series of the helical curves The distance between the curves in the series is equal to the machining step. The spiral step value can be positive or negative. The sign defines the torsion direction.



The step on machining is defines on the same page. It can be specified as absolute value or in percents on the tool diameter.

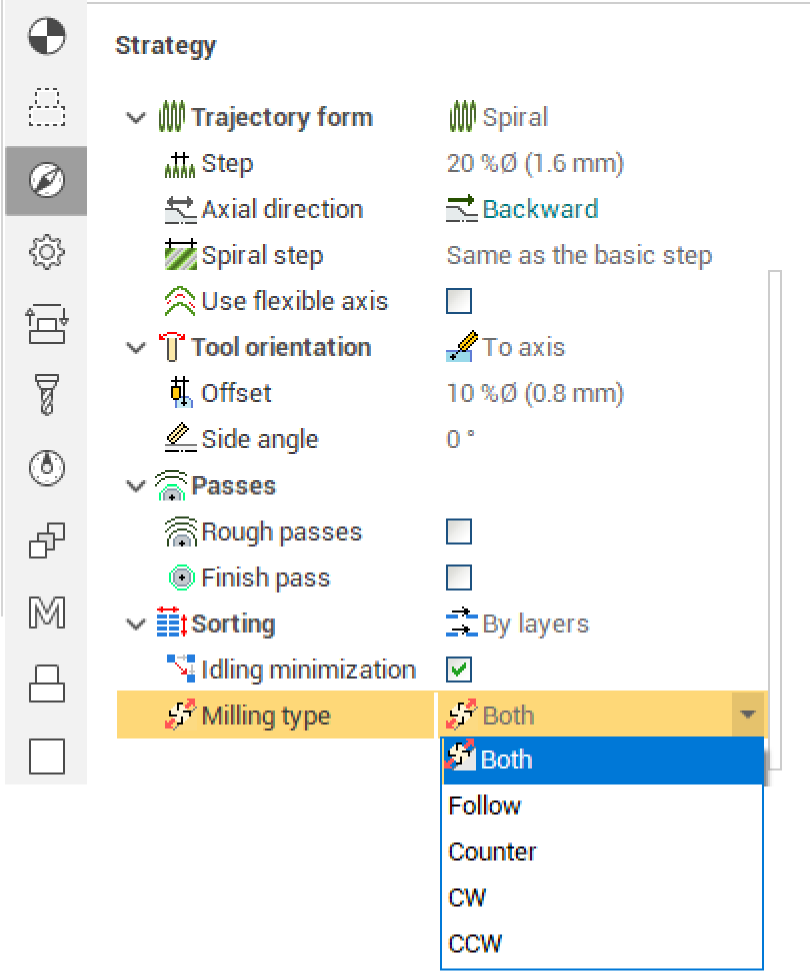


The <Axial direction> defines the order of machining. If the Axial direction is <Forward> then the tool passes is ordered in the rotary axis vector direction else the order is reversed.

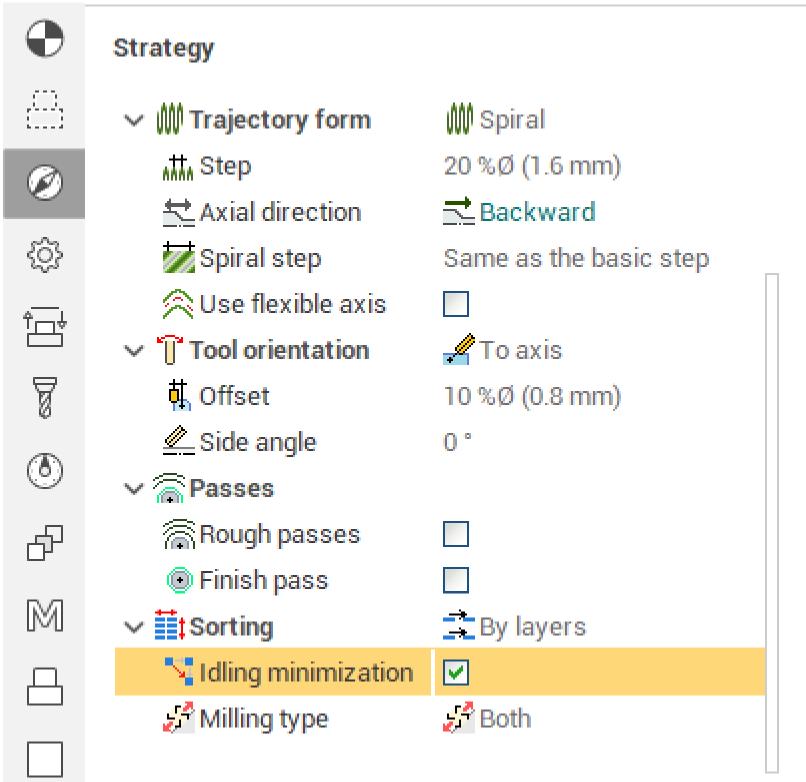


The milling type defines the direction of the part rotation around the rotary axis.

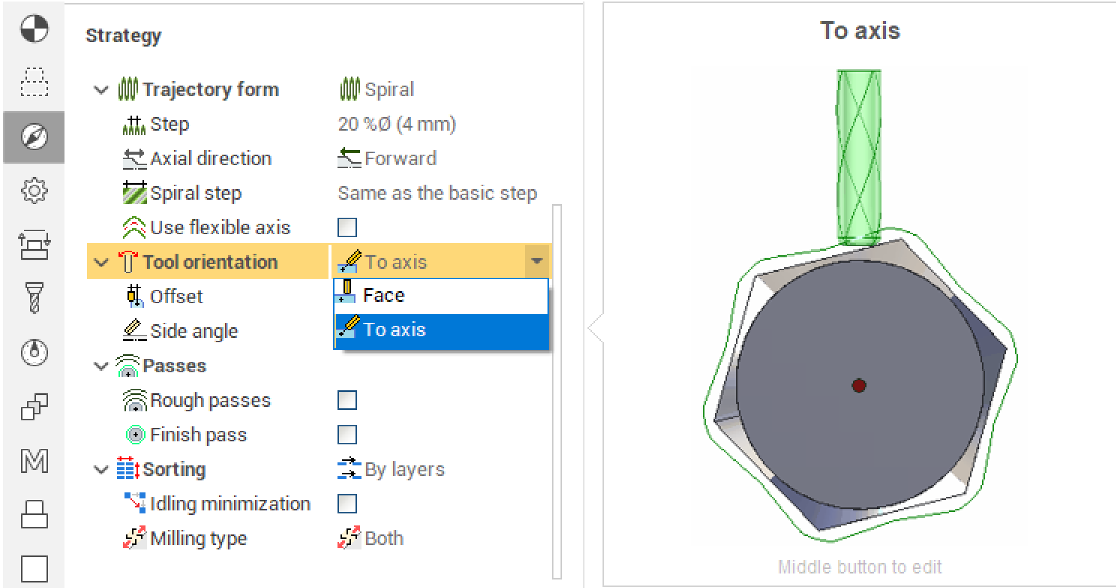
- <Both>. The rotation is alternated at every working tool path. It allows to reduce the idle passes and the machining time that is the save.
- <Climb>. The part rotation direction depends on the tool rotation. It gives the climb milling type.
- <Conventional>. The part rotation direction depends on the tool rotation. It gives the conventional milling type.
- <Clockwise>. The part rotation direction does not depends on the tool rotation.
- <Counterclockwise>. The part rotation direction does not depends on the tool rotation.



The length of the idle tool path cam be minimized if set the tick in the Idle parameters.



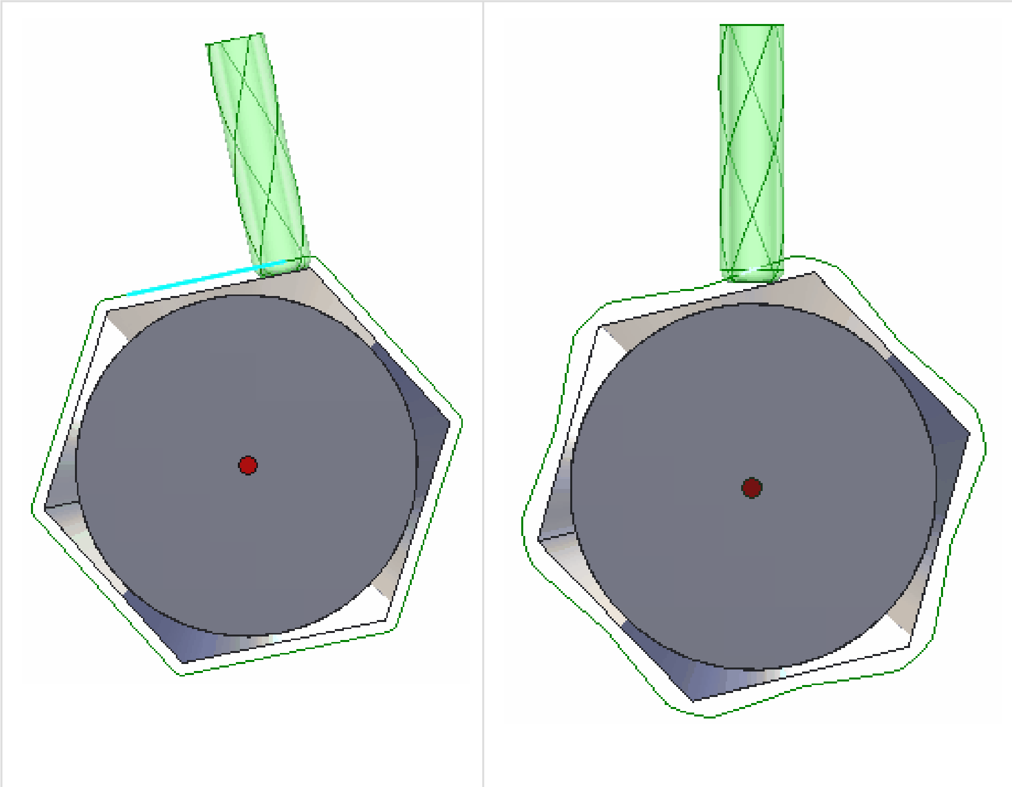
The <Contact tool type> defines the way how the tool axis is calculated.



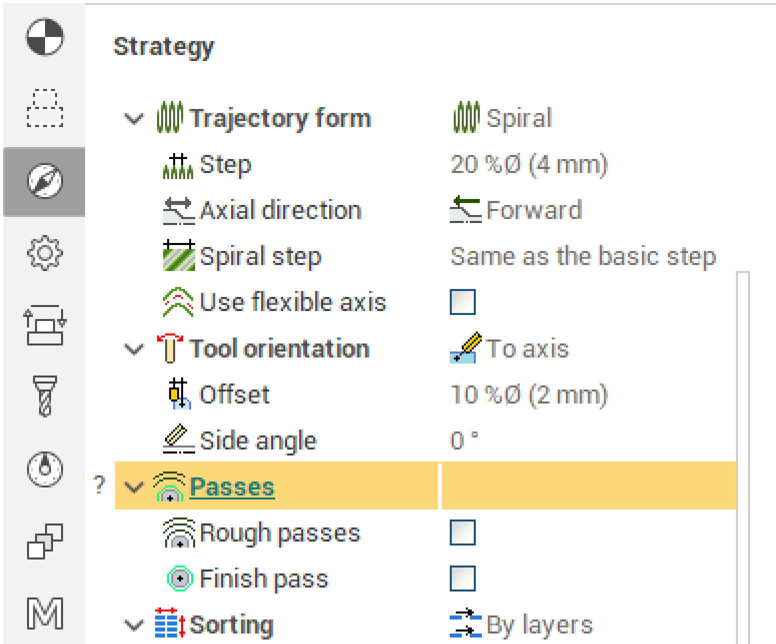
if <Face> is selected then the tool axis is parallel to the surface normal in every tool path point. The <Lead Angle> can be defined additionally. It allows to improve the cutting conditions in the contact point. It can be positive or negative.

if <To axis> is selected then 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 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.

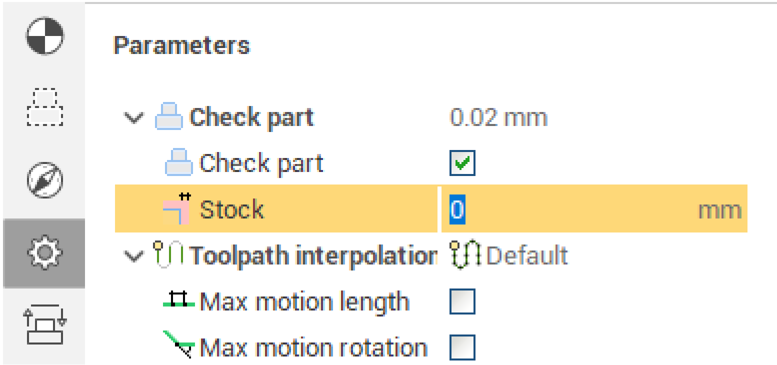
Tool orientation	
Face (surface normal)	To the axis



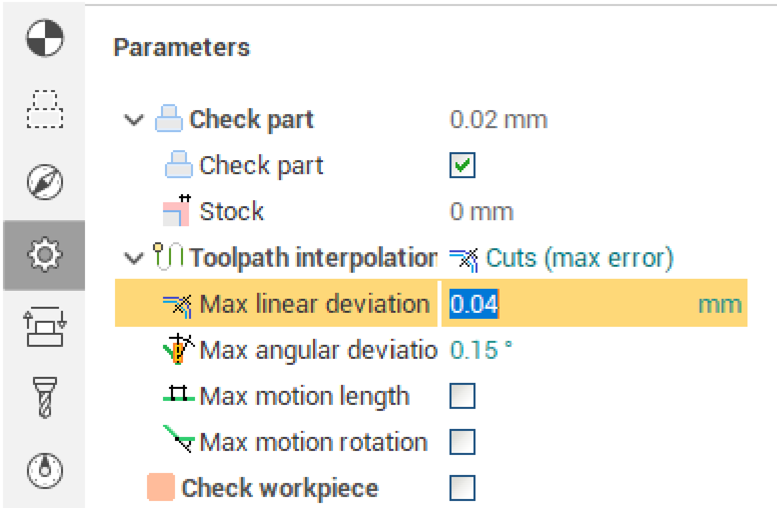
The additional rough passes can be defined on the <Radial layers> panel. The number of the passes is calculated using the <Stock> and the <Step>. Step can be defined as the absolute value, percents of the tool diameter or count. If count is selected then step equal to stock divided into count. <Finish pass> tick generates the additional cleanup pass. The stock that is near defines the stock for the last finish tool path.



The stock parameter shifts the rough and finish passes from the surface. It can be positive or negative.

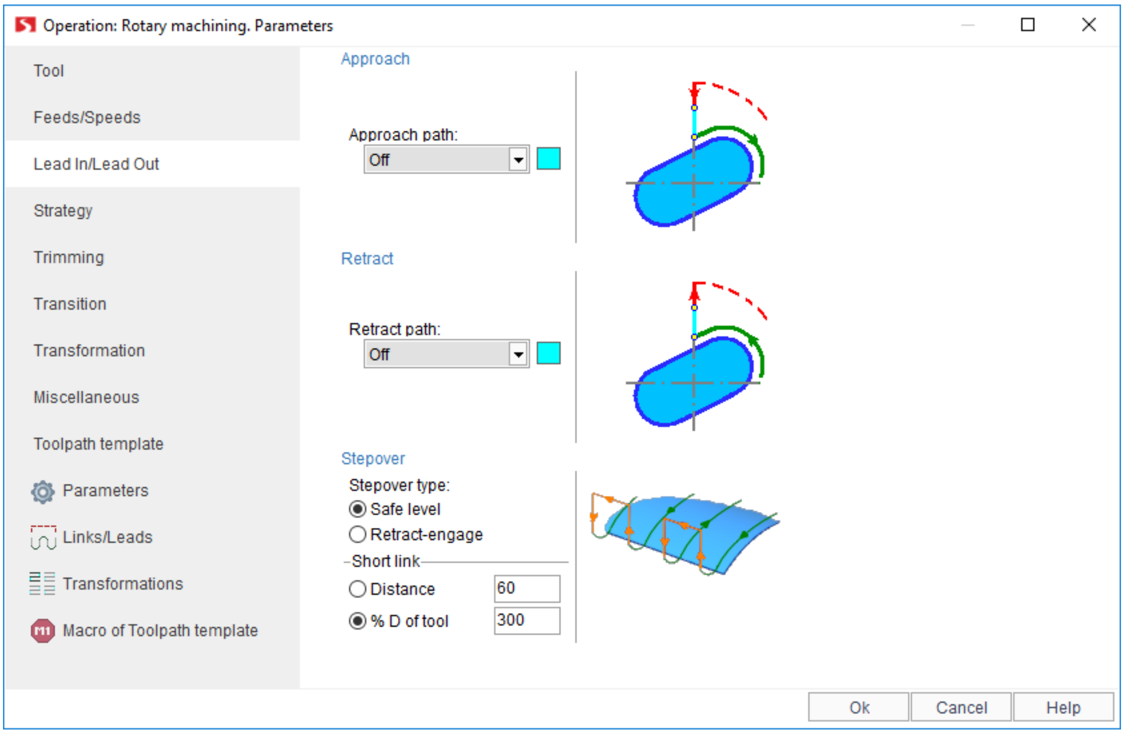


Deviation works like in all milling operations. The higher tolerance the much time need to calculate the tool path.



The roll type and the checked geometry works like in the other milling operations.

The way of the approach to the first point of the working tool path and retract from the last point of the working tool path are defined on the <Lead In/Lead Out> page of the parameters dialog. This page defines the way of the transition between the working passes.



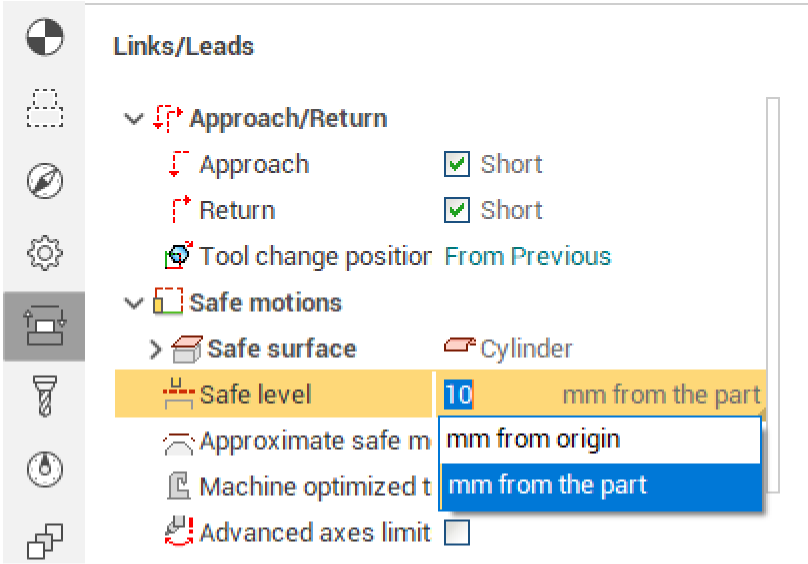
The next approaches and retractions are available:

- <None>. The tool goes down/up to the first point of the working tool path without additional engage or retraction.
- <Normal>. Additional cut is added to the first or last point of the working tool path. The cut is directed as the surface normal in the current point. The cut length is specified in the <Distance> field.
- <Tangent>. Additional cut is added to the first or last point of the working tool path. The cut is directed as tangent to the tool path in the first (last) point. The cut length is specified in the <Distance> field.
- <Angle to tangent>. The additional cut is directed under defined angle to the working tool path in first/last point. The cut length is specified in the <Distance> field. The <Angle> is specified in the degrees. It lies in the plane that is perpendicular to the part rotary axis.
- <Arc>. The additional arc is added to the first/last point of the working tool path. It lies in the plane that is perpendicular to the part rotary axis. The arc radius is specified in the <Distance> field. The arc sector is specified in the degrees in the <Angle> field. The radius and angle can be positive or negative.

There are three ways of the transition between the subsequent tool passes. The way is defined on the <Stepover> panel.

- <The shortest stepover>. This way is used automatically if the distance between the last point of the current pass and the first point of the next pass is less than the value defined in the <Short link> panel. This value can be specified as absolute distance or in the percents of the tool diameter. If the distance is more the than the short link size than the one of the next strategies is used:
- <Retract-Engage>. In this case the retraction is added to the end of the current pass and the engage is added to the start of the next pass. The link between the last point of the retraction and the first point of engage is the shortest.
- <Safe level>. In this case the link between the last point of the retraction and the first point of engage is performed with the go up to the safe level.

The value of the safe level is defined on the <Transitions> page of the parameters dialog.



The <Safe level> in the current operation defines the cylinder radius. The axis of the cylinder is equal to the rotary axis. It is assumed that any motions out of this cylinder would not make the collision with the part or fixtures. The safe level can be defined by absolute or increment value. In the first case it is equal to the cylinder radius. In the second case the value is added to the point that is the farthest from the rotary axis.

The tool goes down to the defined level on the rapid feed, after that the tool goes on the approach feed. The level where the feed is changed is defined on the <Feed switch level> panel. This level can be specified as absolute or incremental value. if value is absolute then it is a distance from the rotary axis. if value is incremental then it is a distance from the first point of engage.

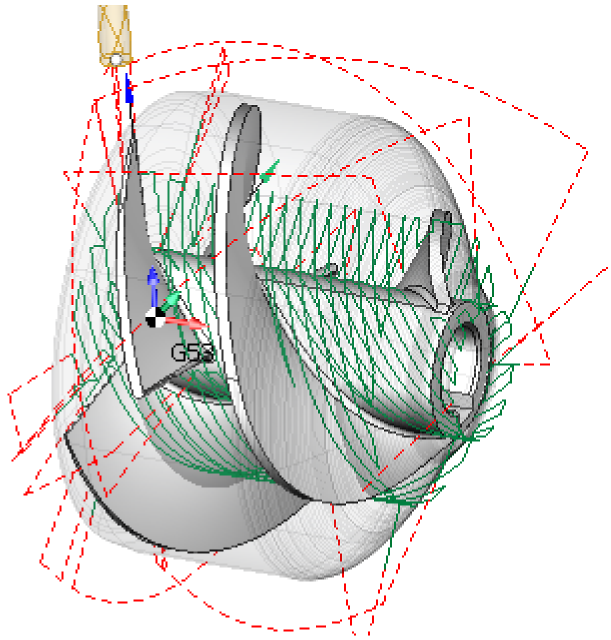
The possibility to multiply tool path by axis is available for the rotary machining operation also. It allows to reduce the calculation time if the part elements are repeated.

See also:

5 axis machining

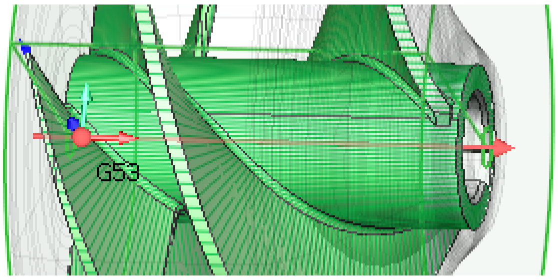
Operations for 4-axes and 5-axes milling

5.5.4.8 Roughing rotary operation



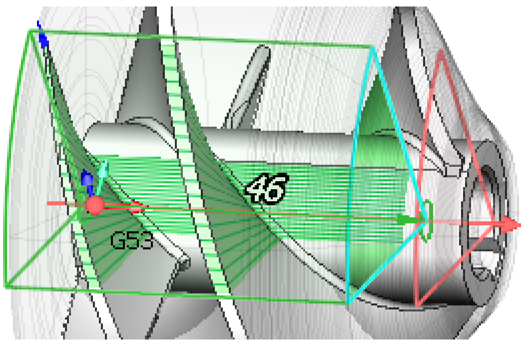
Roughing rotary is a 4 axis toolpath that removes the workpiece material layer by layer. It is similar to the Roughing Waterline except that the machining layers are not planes, but cylinders around the rotary axis.

Rotary axis



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.

Job zone



The job zone is defined by:

- the minimal and maximal axial positions,
- the angular sector,
- top and bottom levels of machining.

Machining parameters

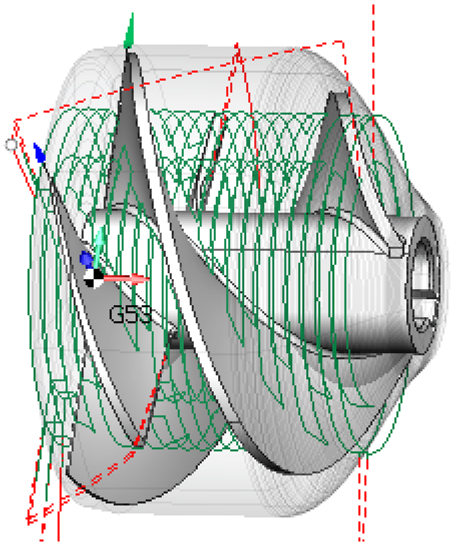
The depth step can be set as an exact value as well as the number of layers.

The machining step defines the maximal distance between the machining passes in a layer.

Machining strategies

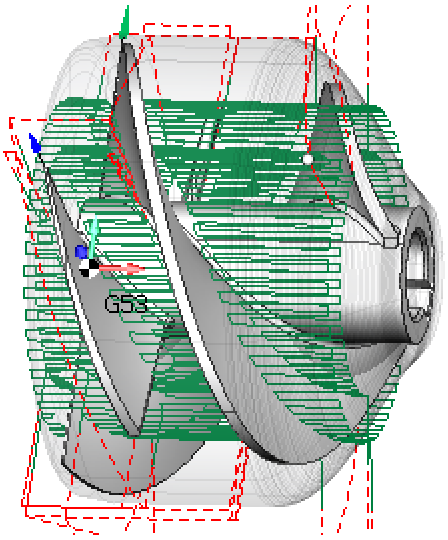
Currently three strategies are available.

1. Circular



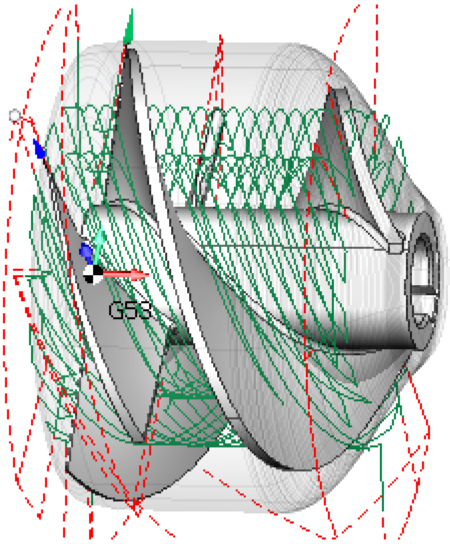
The circular strategy generates passes around the rotary axis that represent 4 axis arcs.

2. Linear



The linear strategy generates passes along the rotary axis that represent linear cuts. The angular step-over between passes on each layer is the same, what means that the real step-over gradually decreases when approaching the bottom layer of machining

3. Spiral

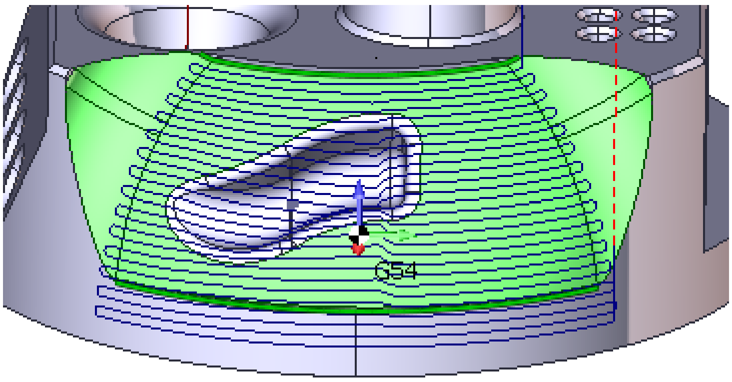


The Spiral strategy generates helical passes. The pattern is well suited for machining parts like screws and impellers.

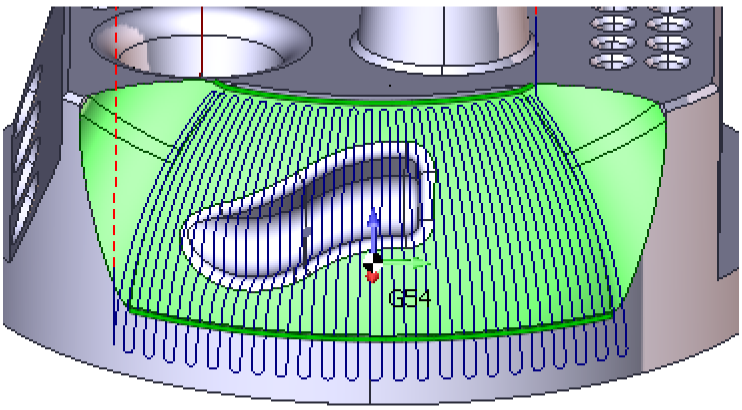
5.5.4.9 Morph operation

Morph generates passes between two curves. The possible strategies include:

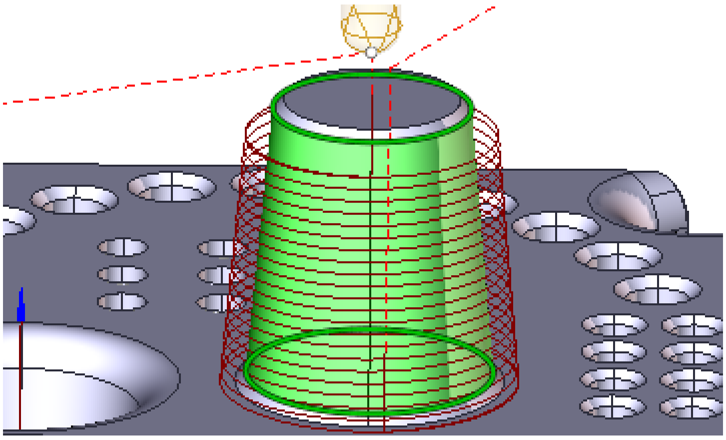
- Along curves;



- Across curves;

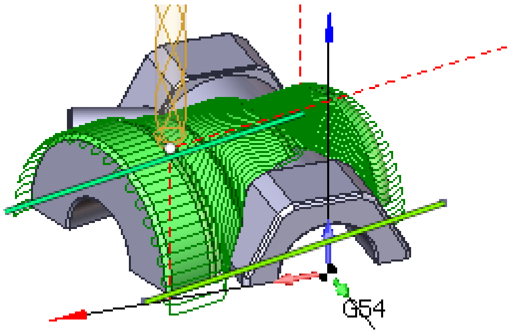


- Spiral.

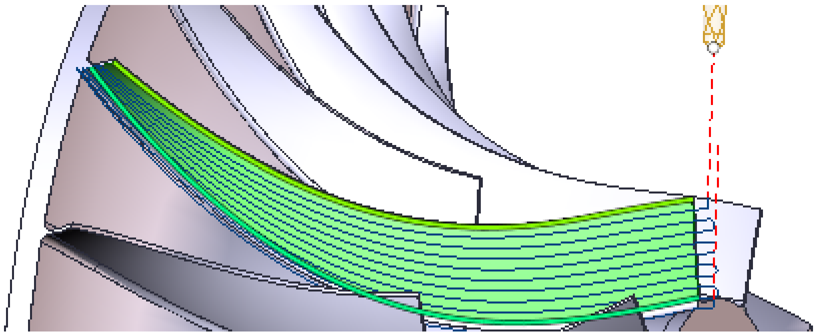


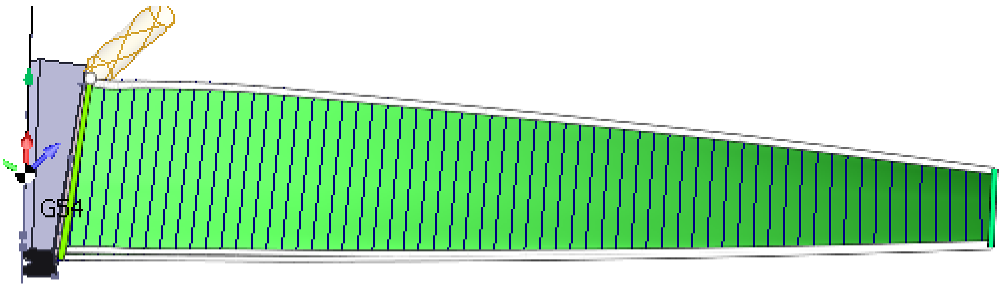
Extensive tool axis orientation modes and tool axis lean options of Morph make it a rather versatile toolpath. The tool axis orientation modes include:

- Fixed (3d);

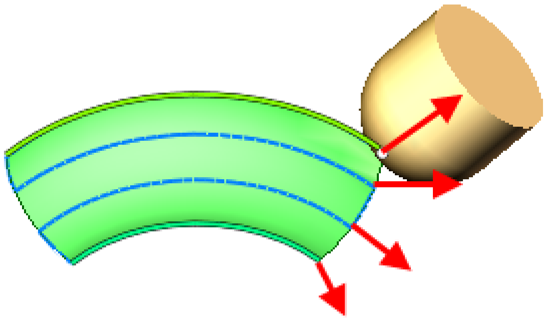


- Normal to curves;

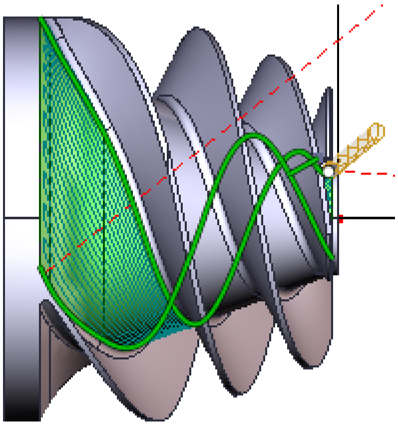




- Normal to surfaces;



- To rotary axis.



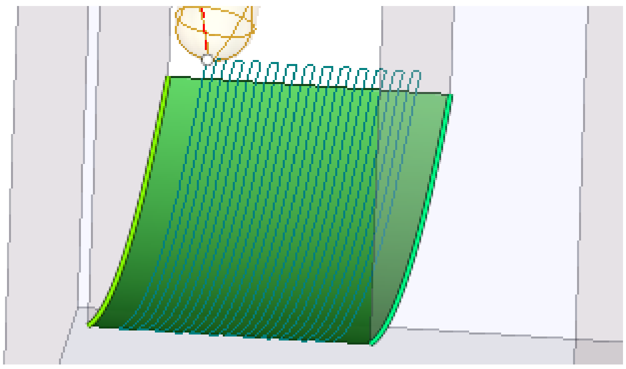
Job assignment

In Morph you have to specify:

- First curve and Second curve,
- Machining surfaces,
- and, optionally, sync lines between the first and the second curves.

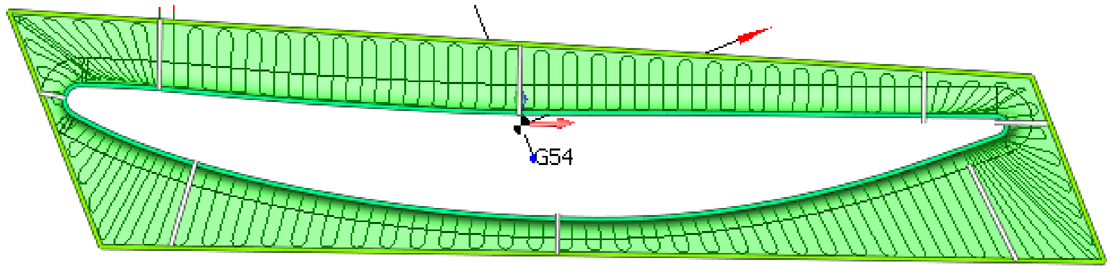
The machining surfaces are used to restrict the area of machining. Morph generates passes only where a tool has contact with those surfaces. Morph generates passes not between

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.



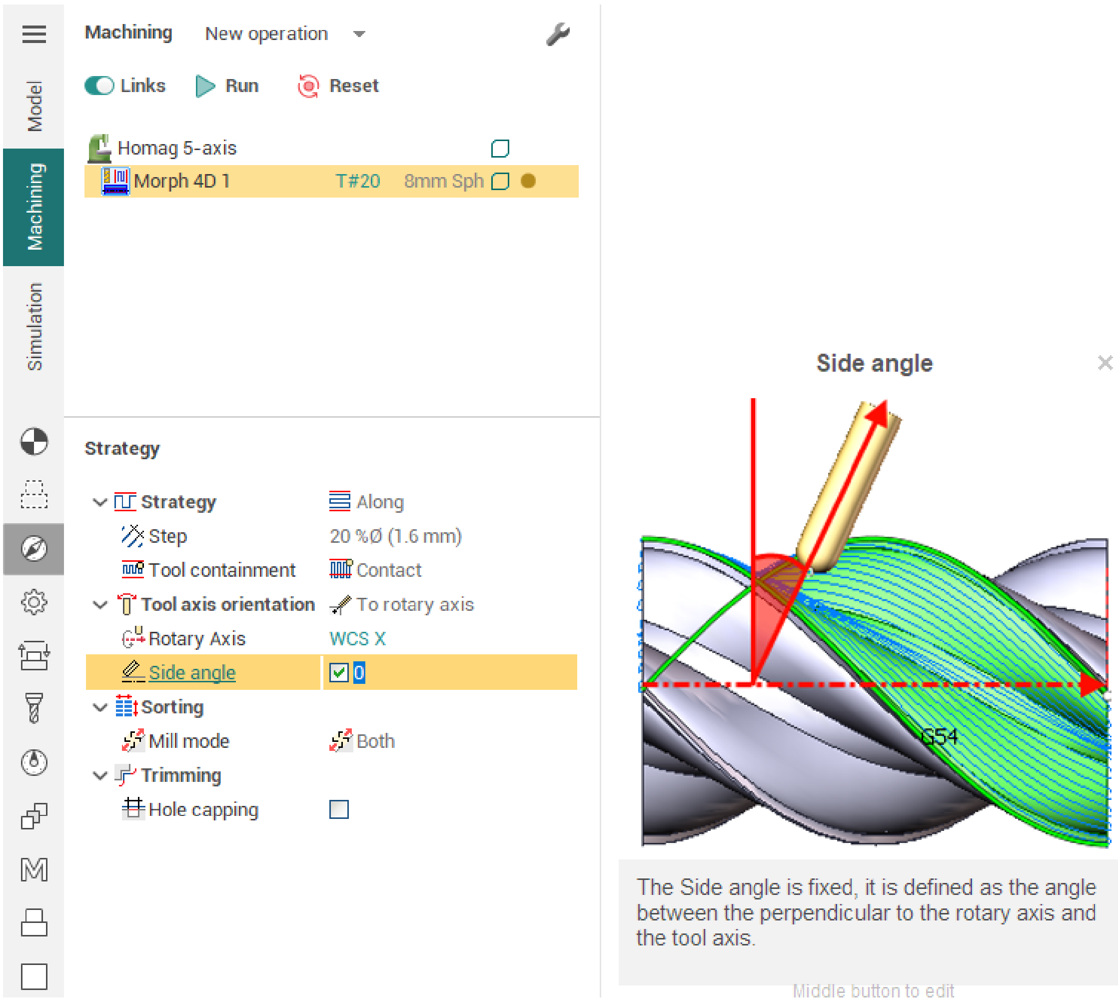
Sync lines are used to improve the quality of morphing in difficult cases, especially when machining closed contours. A sync line specify two corresponding points of the first and the second curve.

You can use any types of curves as sync lines: edges, 3d curves, 2d geometry curves. They do not have to connect the points on two curves precisely, so you can draw sync lines at the 2d geometry tab.



Parameters

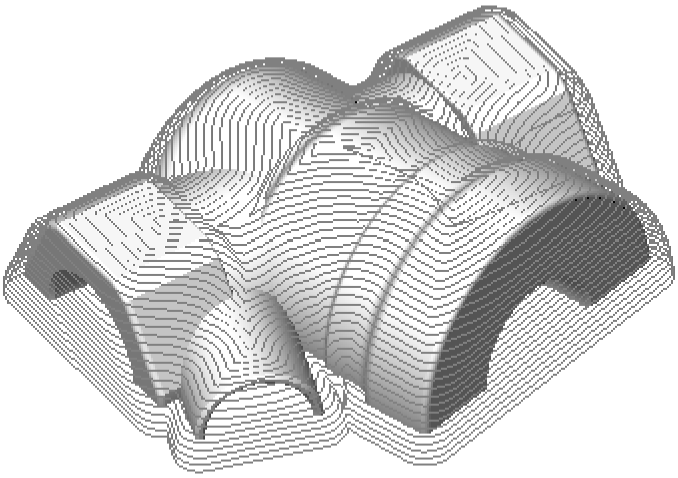
Parameters of morph are accessible in the parameters inspector. The documentation on parameters is available through the hint. Just click on the question mark next to a parameter and you will see the description of it.



This operation is available for SprutCAM's configurations:

- 3x Mill
- 5x Mill
- Robot
- Expert
- Master
- Pro

5.5.4.10 Scallop finishing operation



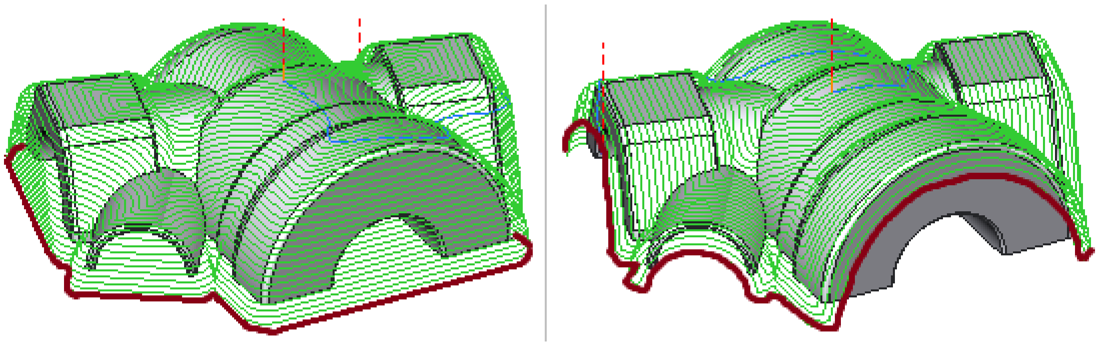
The Scallop toolpath starts from the curves lying on the part surfaces and is generated by repeatedly offsetting those curves inwards until the curves collapse. So basically it is an equidistant toolpath except that the offset is made in 3d space on the machining surfaces. The toolpath achieves a consistent 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.

Starting curves

There are two options currently available:

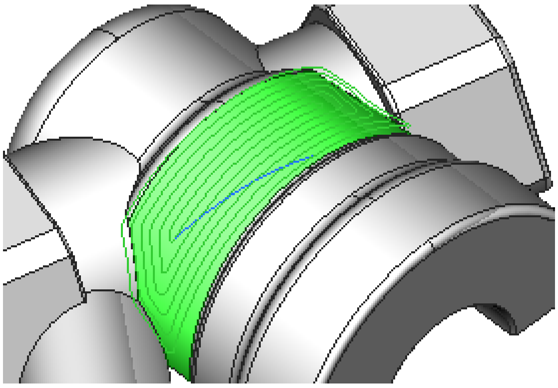
- start from the bottom, and
- start from the top of vertical walls.

For the first option the toolpath starts at the bottom level of machining, for the second option the toolpath starts from the silhouette curves.

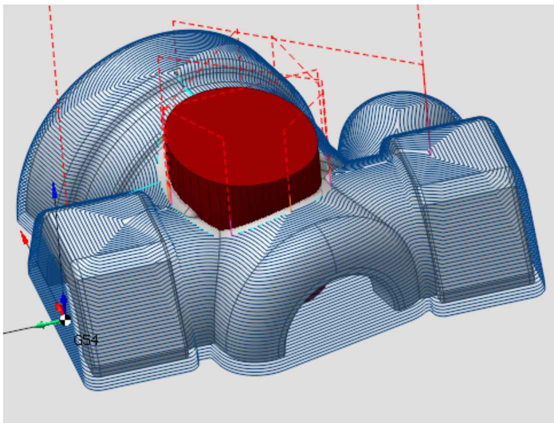


Job assignment

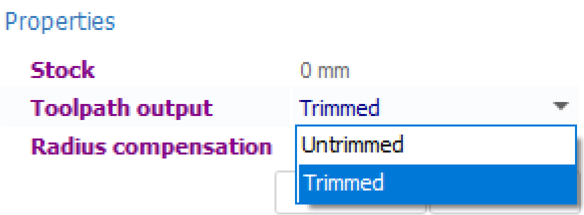
You can specify machining surfaces in the job assignment. The starting curves will be detected as the curves of contact of the cutter with those surfaces in this case.



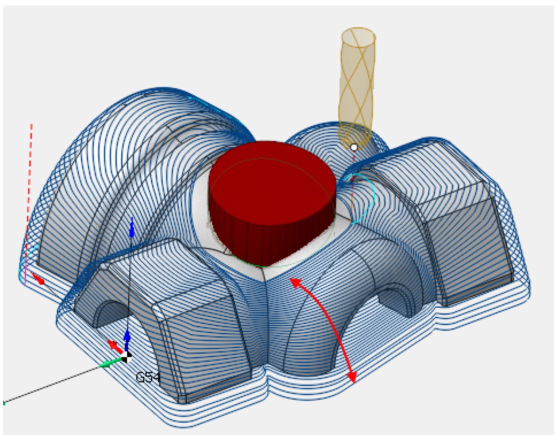
You can also use the Job Zone to define the starting curves.



You also can add **Restrict Zones** with determine restricted or cutted toolpath depending on the option you choose.



Morph Passes option calculates toolpath between open areas.



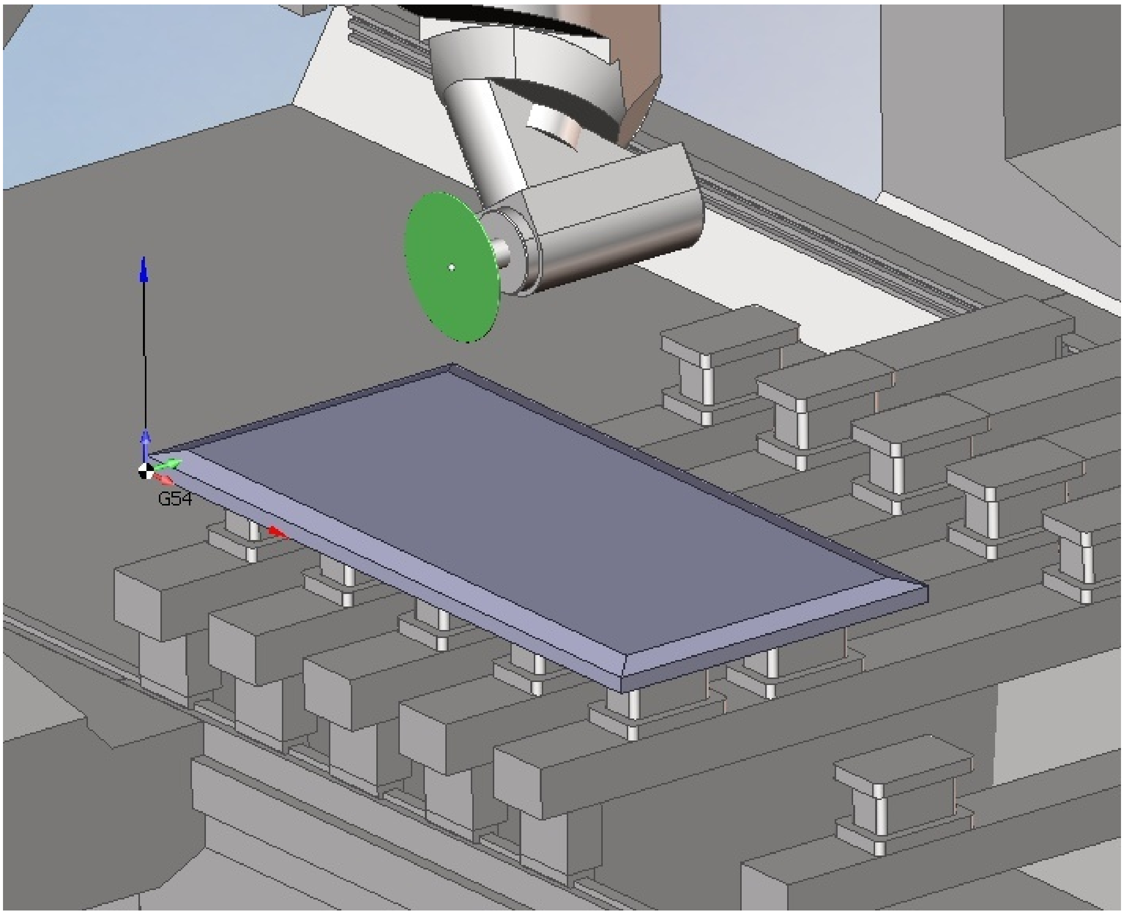
Strategies

The toolpath can be generated both from the inside out and vice-versa, from the outside in.

It is possible to generate a spiral toolpath instead of parallel passes to minimize linking.

It is possible to smooth sharp corners in the toolpath.

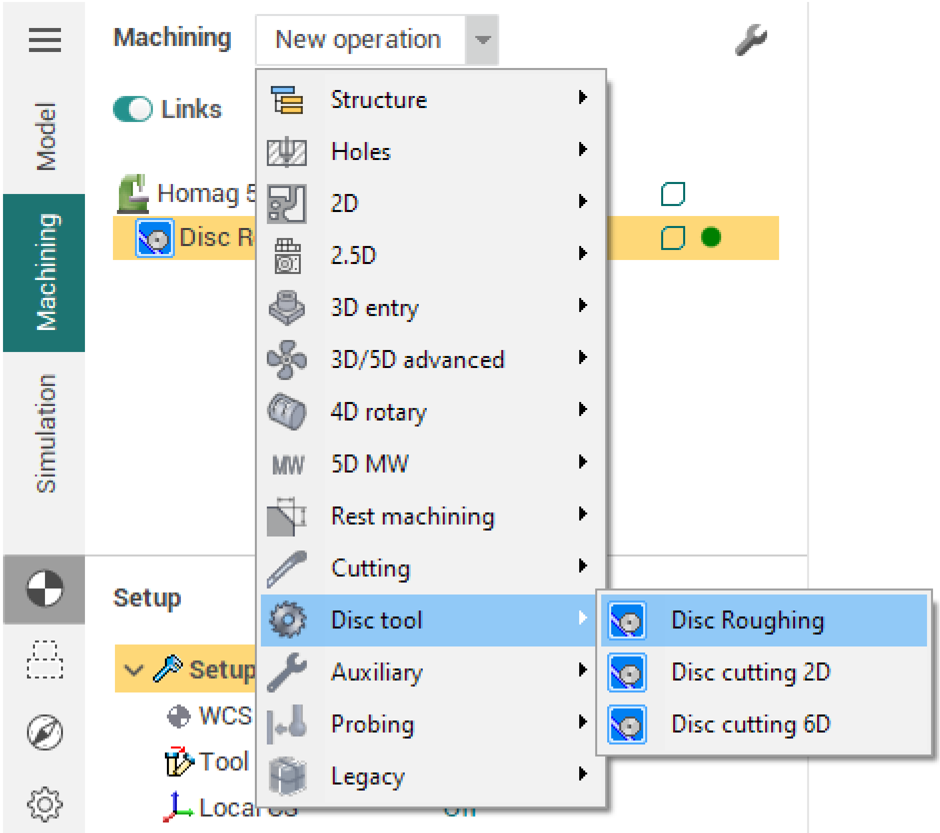
5.5.4.11 Saw operation



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.

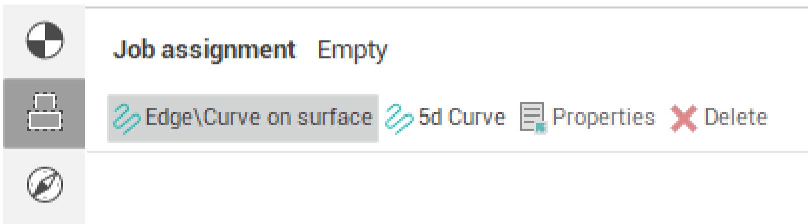
Way of use:

1. Create sawing operation.

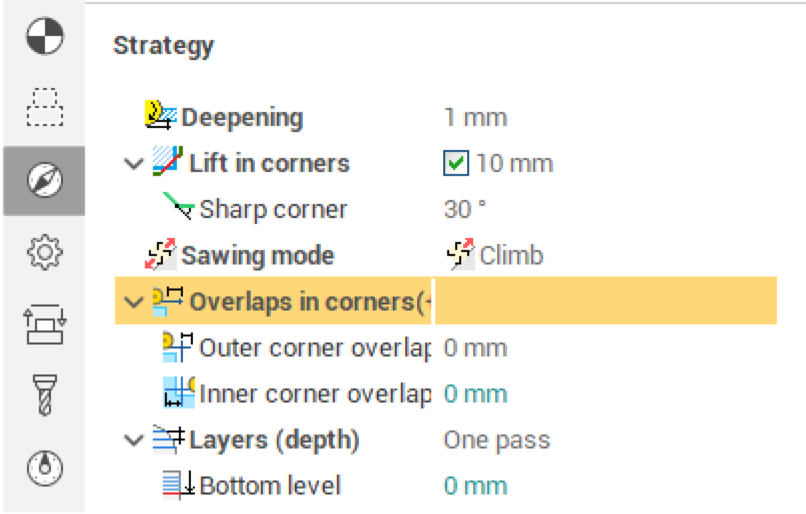


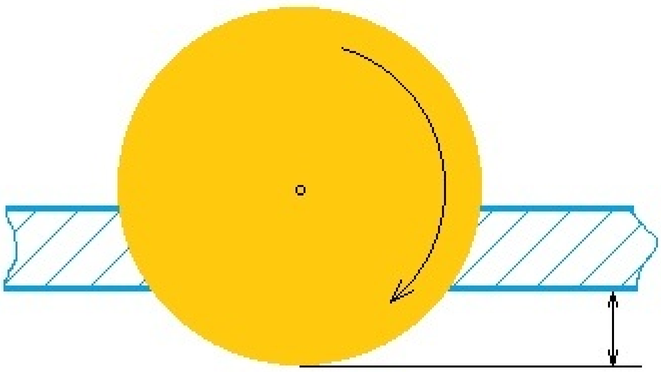
2. Define the required tool size and speeds, the same way as it can be done for all other operations.

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.

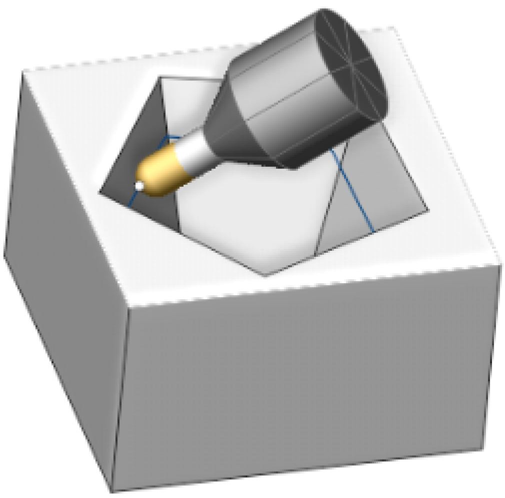




See example of operation on YouTube:

http://www.youtube.com/watch?v=op5vH1P-V3s

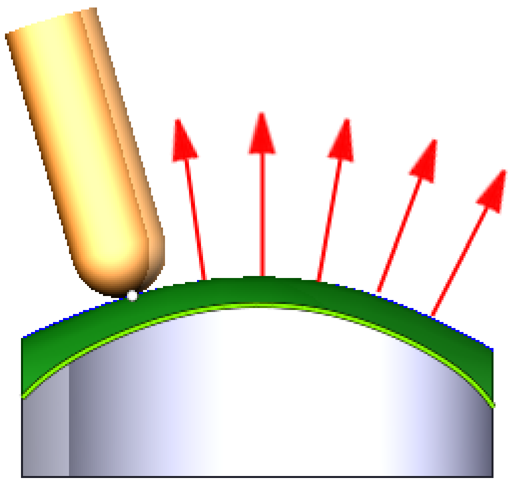
5.5.4.12 5 axis tool path conversion



This option converts 3 axis toolpath into 5 axis for such operations as: Scallop, Morph and Geodesic. This feature allows to avoid collisions and use minimal length tool.
5 axis tool path conversion can be used with spherical mill tool.

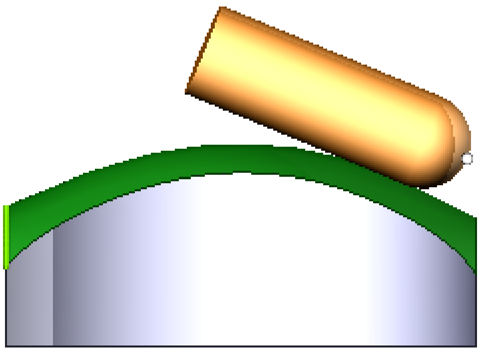
5 axis toolpath conversion options:

Normal to surface



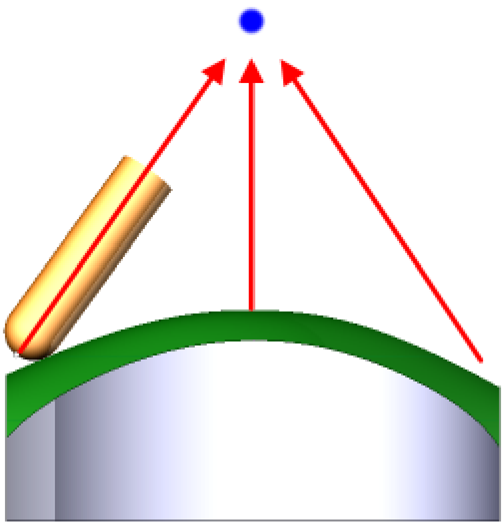
The tool is oriented by normal to machining surfaces. Additionally the lead and lean tool angles can be applied to further tilt the tool along or to the side from the cutting direction.

Flank



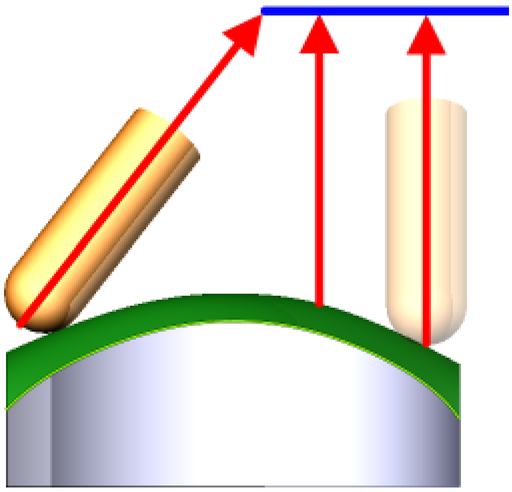
The tool contacts machining surfaces with the peripheral part (cylindrical part for the cylindrical mills). Additionally **lead** and **lean** angles can be applied. The strategy can be used for swarf milling.

Through point



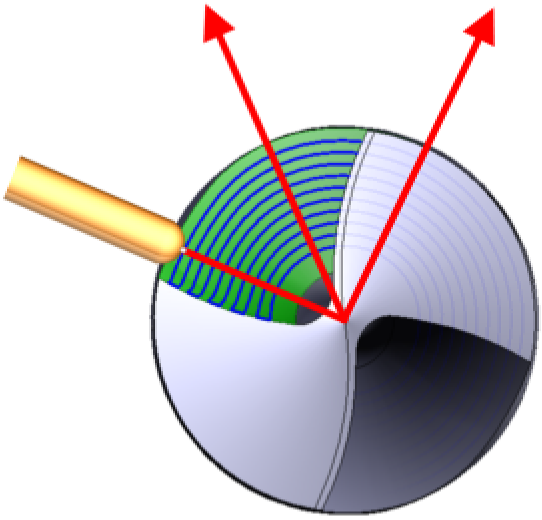
The tool axis is oriented to the specified point.

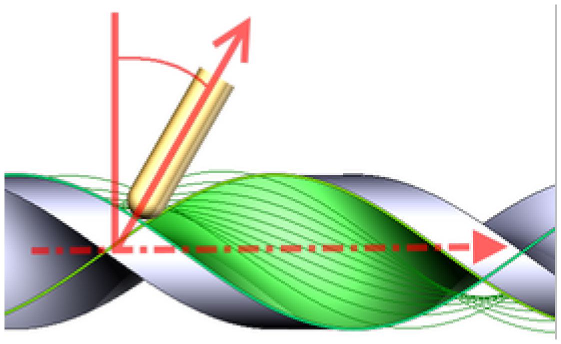
Through curve



The tool axis is oriented to the nearest point of the specified Tilt curve . Additionally **lead** angles can be applied.

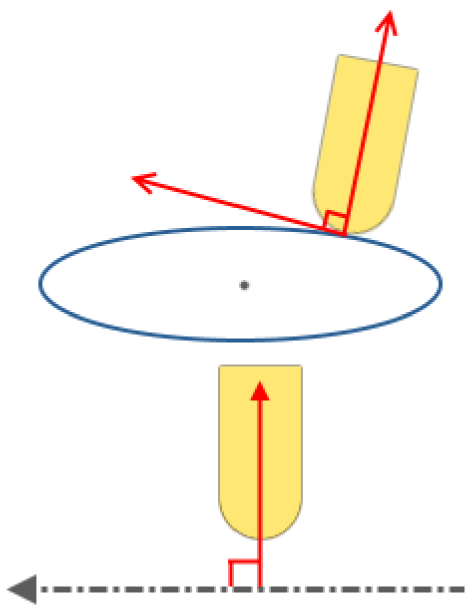
To rotary axis





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



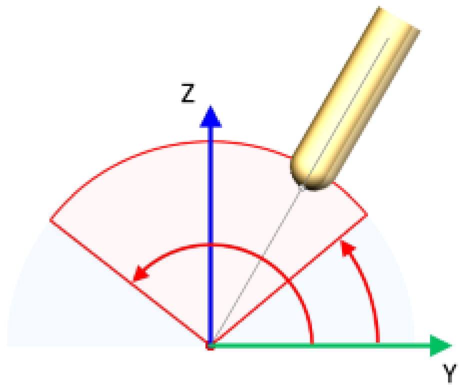
The tool axis is oriented to the perpendicular to toolpath and perpendicular to the rotary axis. Additionally the side angle to the rotary axis and **blend distance** can be specified.

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.

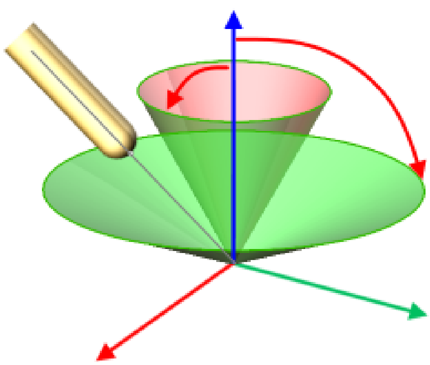
Limit Rotation angles

Limit around axis



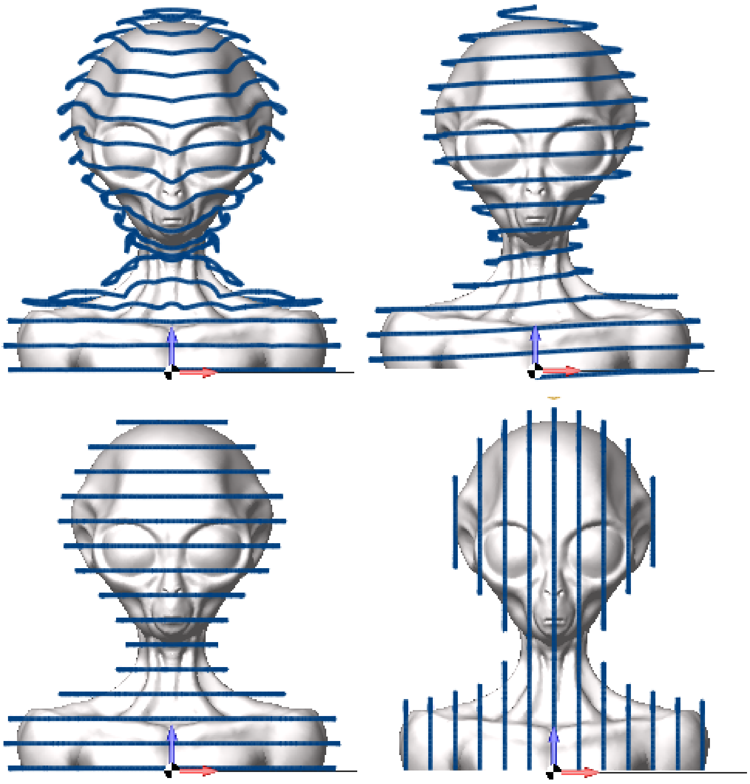
With this option you can limit the tool on the XY plane, XZ plane or YZ plane between two angles.

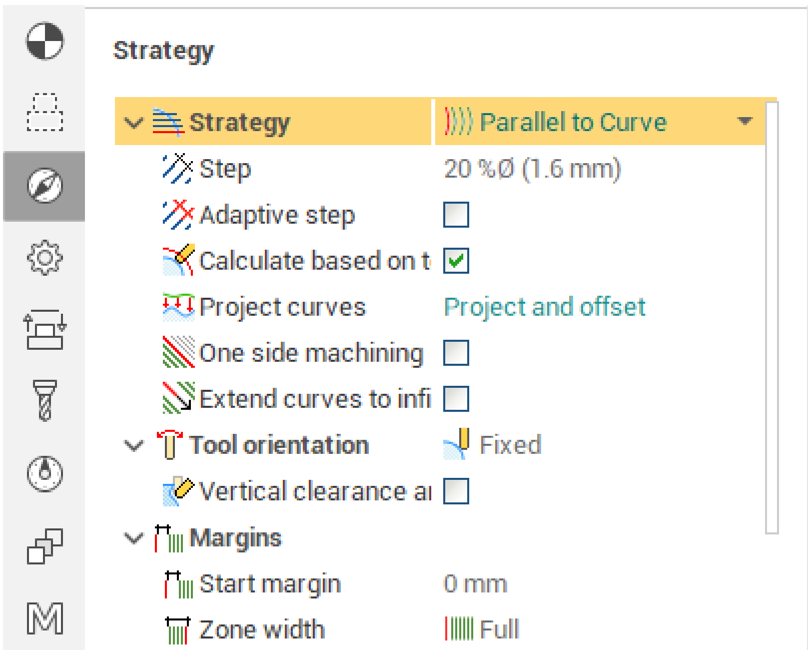
Conical limit



With this option, you can limit the tool at an angle along one axis between two cones.

5.5.4.13 5D Meshing operation





Overview

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 finishing operation, Helical operation and Waterline strategy from 5d surfacing operation. The operation allows generating machining toolpath on the 5 axis machines.

For setting spherical tool normals the 5 axis tool path conversion option is available.

Supported model types

- Surface models
- Triangular mesh models

Supported tool types

- Spherical mill
- Conical mill with a spherical end
- Lollipop mill

Strategies

- Scallop
- Helical
- Waterline
- Plane

Tool axis orientation modes

- Fixed
- Normal to Surface
- To Rotary Axis
- To/From Curve
- To/From Point
- Perpendicular to Toolpath

Collision avoidance

- Trim toolpath
- Side Tilting
- Frontal Tilting

Job assignment

- Faces of the part
- Job Zones
- Start Curves

Quick start

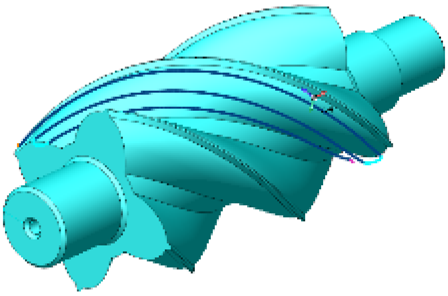
Just create the operation and press Run. The operation will generate a scallop pattern on the whole part starting from the bottom level. The tool will stay in the vertical orientation unless there is a collision with the part, otherwise it will be tilted to the side.

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

5.5.4.14 Rotary waterline operation



Operation based on the **5D surfacing** operation and preconfigurated for a rotary machining.

The strategies for 5th axis control is blocked.

Operation allow machining models like a screw and body rotation and available for **4x** and **5x** configrations.

Unlike **5D surfacing** operation for calculation with empty **Job assignment** to calculate will be used all model.

Contact tool type is predefined as "**To Rotary axis**".

Strategy set as "**Around rotaty axis**".

Margins define as minimal radius for start machining(**R min.**) and width or machining zone (**Zone width**).

Blocked incline angles, avoid singularity and axis deviation.

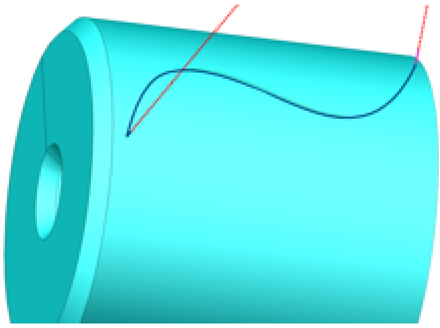
See also:

5D surfacing operation

5 axis machining

Operations for 4-axes and 5-axes milling

5.5.4.15 4D contouring operation



Operation based on the **5D contouring** operation and perconfigurated for rotary machining and available for **4x** and **5x** configurations.

Tool orientation set as default "**To Rotary axis**".

Tool orientation "**Through point**" and "**Through curve**", **Advanced axes limits control**,**Side angle**,**Lead** and **Lean angles** options are blocked.

Safe surface set as "**Cylinder**".

See also:

5D contour and 6D contour operations

5 axis machining

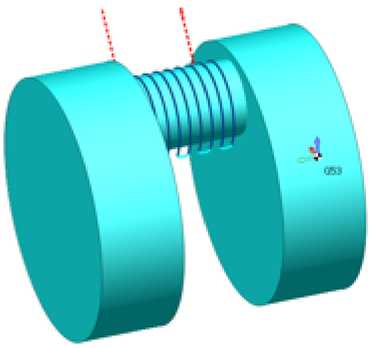
Operations for 4-axes and 5-axes milling

5-axis milling along the isoparametric curves

5-axis milling of the profile on the surface

5-axis milling of the ruled surfaces by the flank of the mill

5.5.4.16 4D surfacing operation



Operation is based on the **5D surfacing operation** and preconfigurated for rotary machining.

Contact tool type mode set as "**To rotary axis"**.

Blocked incline angles, avoid singularity and axis deviation.

Safe surface set as **Cylinder**.

For a holder collision is available "**Trim**" method only.

Operation available for **4x** and **5x** configurations.

Strategies for 5th axis control is blocked.

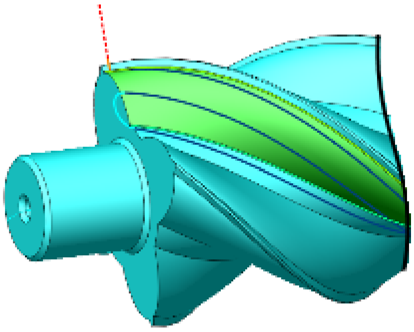
See also:

5D surfacing operation

5 axis machining

Operations for 4-axes and 5-axes milling

5.5.4.17 4D Morph operation



Operation based on the **Morph** operation and preconfigurated for rotary machining.

Operation is available for **4x** and **5x** configurations.

Tool orientation type is set as "**To Rotary Axis**".

For holder collision available "**Trim toolpath**" method only.

Safe surface set as "**Cylinder**" by default.

Blocked Lean angles.

Strategies for 5th axis control is blocked.

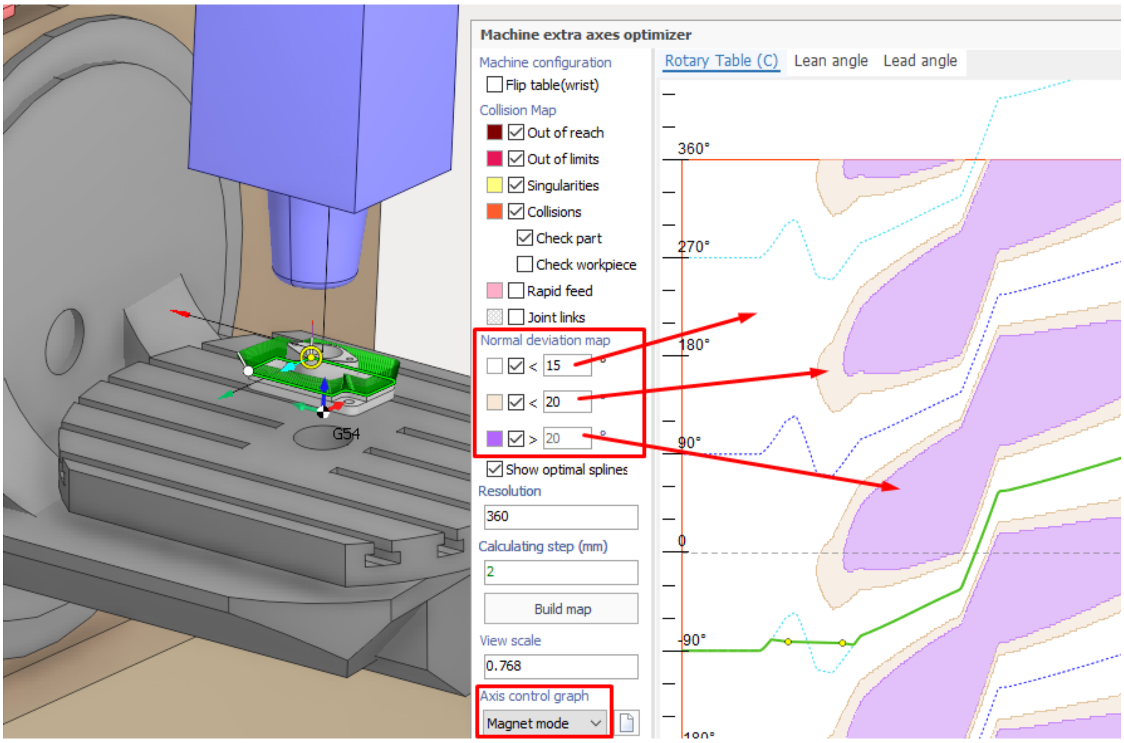
See also:

Morph operation

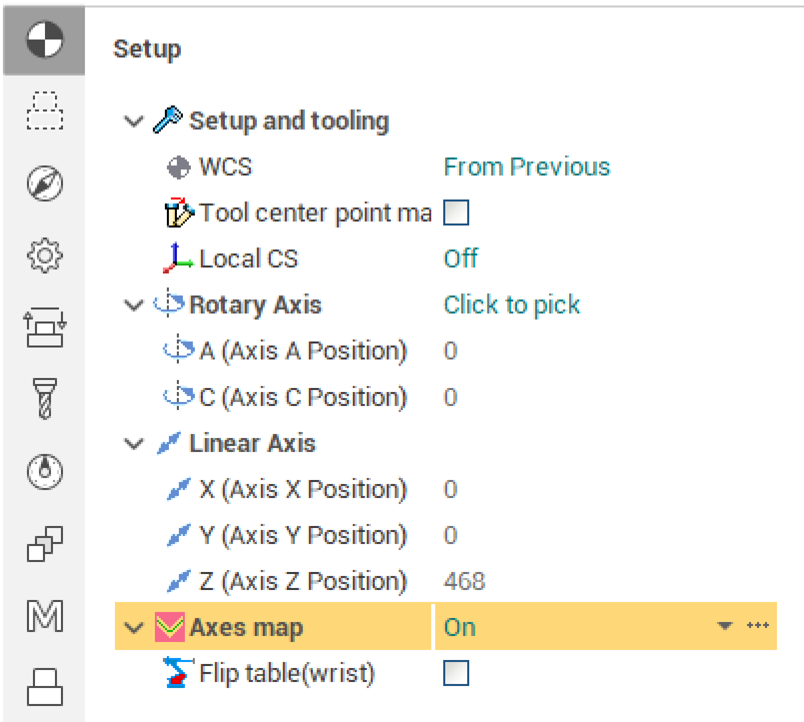
5 axis machining

Operations for 4-axes and 5-axes milling

5.5.4.18 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.

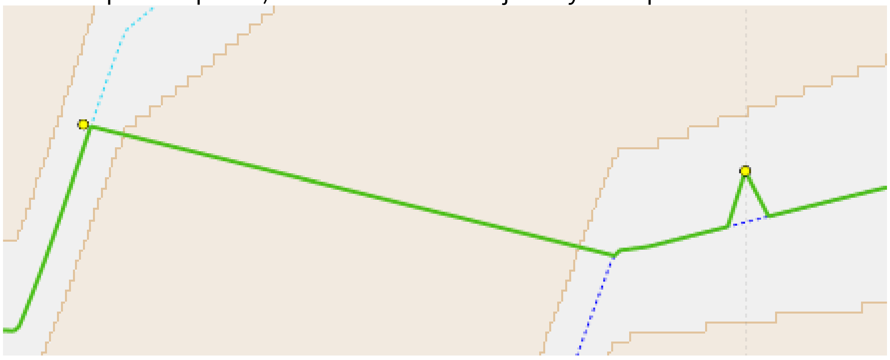
Spline editing modes

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

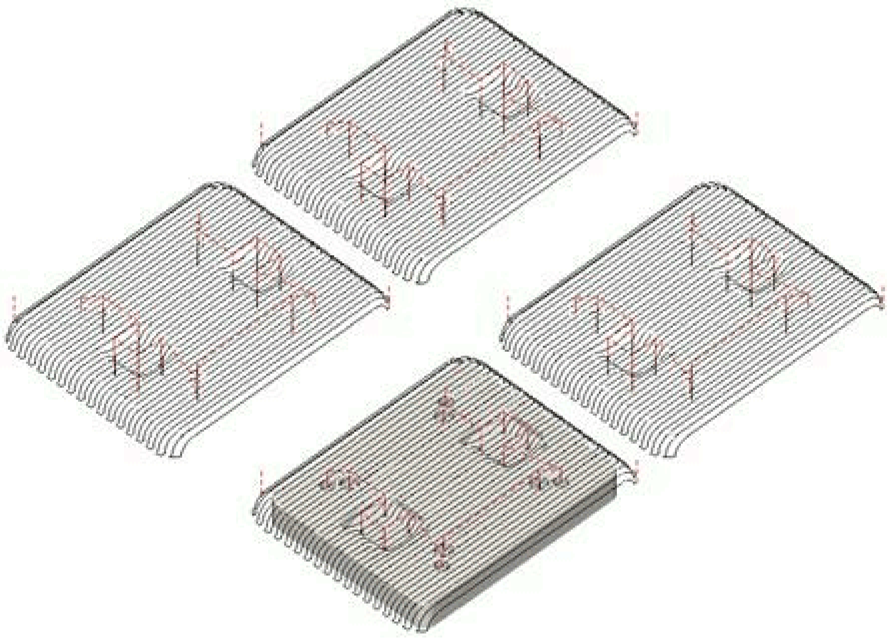
The Lead/Lean angle are the important parameters in 5-axis machining which define the inclination of the tool along the toolpath and to the side (in the perpendicular plane) of the toolpath, respectively. Switch to the Lead or Lean angle tab to control the values of these parameters along the operation toolpath. The user interface is the same as for the robot extra axes optimizer.

See also:

5 axis machining

Robot axes map

5.5.5 Multiply group



This method of copying is outdated, it's recommended to use Part copies.

The operation is intended for machining of sample pieces with repeating patterns. It allows to calculate a tool path for one pattern by any combination of operations and then to repeat this machining a necessary amount of times for other patterns.

Source operations must be added into the multiply group (by analogy with an operations group). A toolpath of the operations will be transformed or multiplied. The transformation mode and the copying scheme are arranged on the Strategy page of the operation parameters window.

The following spatial transformation types and copying schemes are available:

- <Two dimensional array>. Repeating patterns of a toolpath place as a rectangular grid, with distance between elements to the equal values given in fields X and Y. The Angle value sets an inclination angle of a grid relative to horizontal;
- <Two dimensional array (manually)>. Repeating patterns of a toolpath place manually. The count of elements and coordinates of each element are set by user;
- <Round array>. Repeating patterns of a toolpath place along a circle, with the centre in point X and Y, radius R and the angle pitch given in the appropriate field;
- <Round array (most distant)>. Repeating patterns of a toolpath place along a circle, with the centre in point X and Y, radius R and the angle pitch given in the appropriate field. Elements are sorted more distantly from each other;
- <Round array (manually)>. Repeating patterns of a toolpath place along a circle, with the centre in point X and Y, radius R. The count and the angle of each element are set by user;
- <Axis symmetry>. Repeating patterns of a toolpath symmetrical about the given axis. A point and an angle within a horizontal plane set the axis
- <Point symmetry>. Repeating patterns of a toolpath symmetrical about a centre point. Coordinates X and Y set the point of the centre of symmetry;

The amount of rows, lines and columns for all copying schemes (without manually setting elements) is set in the <Columns> and <Lines> fields.

It is possible to set the machining order <By blocks> or <By operations>.

If the machining order is by blocks then repeating patterns of a sample piece will be machined, sequentially one by one. That is, the first pattern will be machined by all indicated operations all over again, and then the same set of operations will be repeated for the subsequent patterns in the appropriate place etc. If the machining order is by operations then all patterns will be machined by the first operation all over again, next by the second operation etc. Machining by blocks is expedient for applying the order of machining, if machining is manufactured by one type of the tool or patterns are located on big distance one by other. An order by operations is optimum for machining by different tools.

Also, it is possible to set the milling type (climb or conventional).

Other way for machining multiply elements is use option multiply toolpath by axis for operation.

See also:

Mill machining

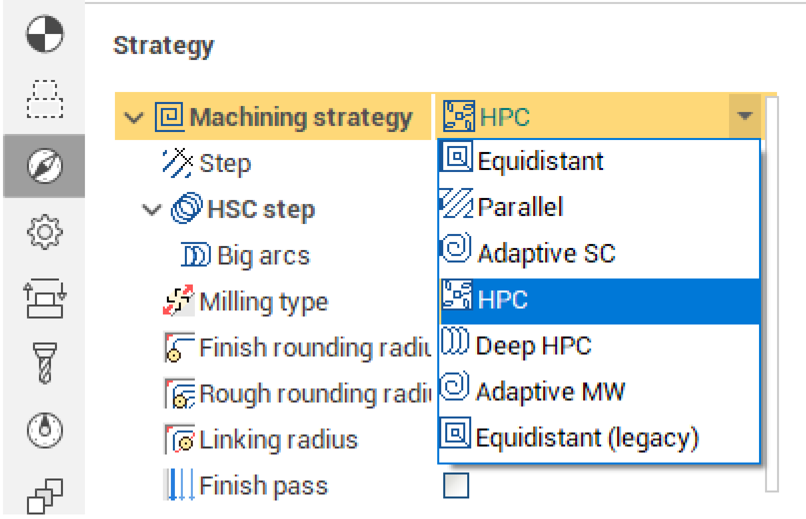
5.5.6 High performance cutting (Sprut HPC)

The high performance cutting (Sprut HPC) strategy is designed for the efficient removing of material in the open and closed pockets.

This strategy is available in the following operations:

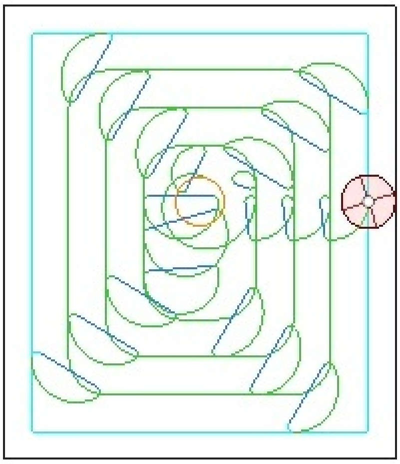
- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

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

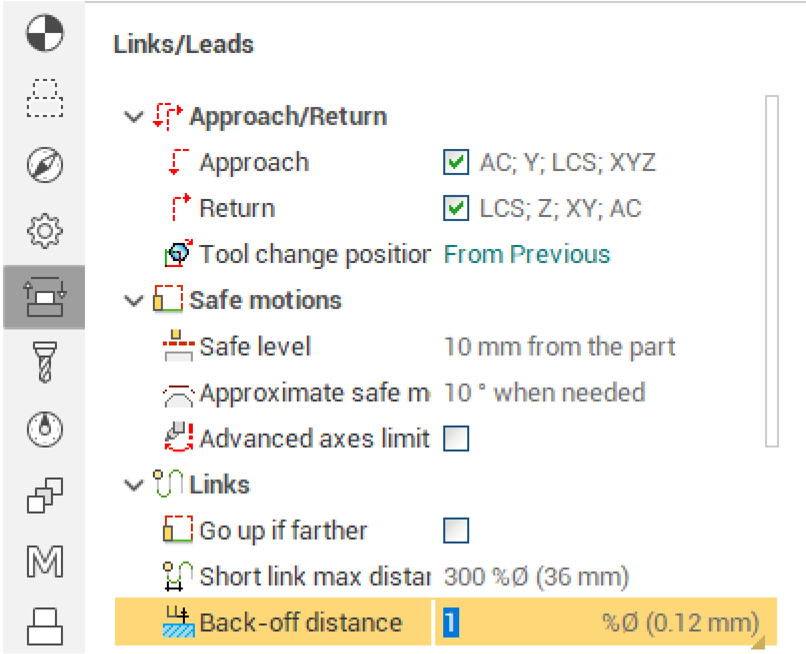


Tool path length has been reduced by up to 20% compared to the legacy version of the program.

New trochoidal arcs avoid tool overload without requiring feed rate reduction.

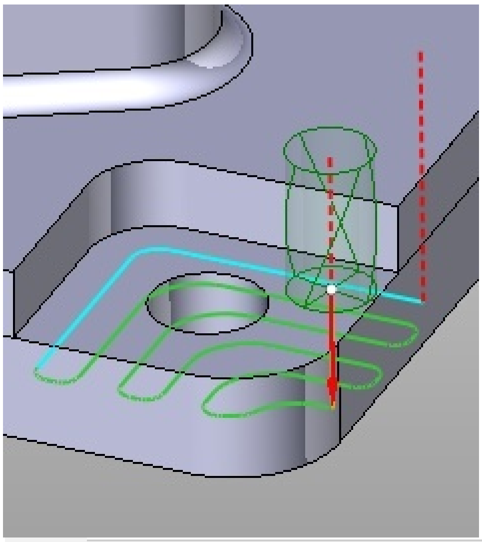
It makes for a much smoother machining process.

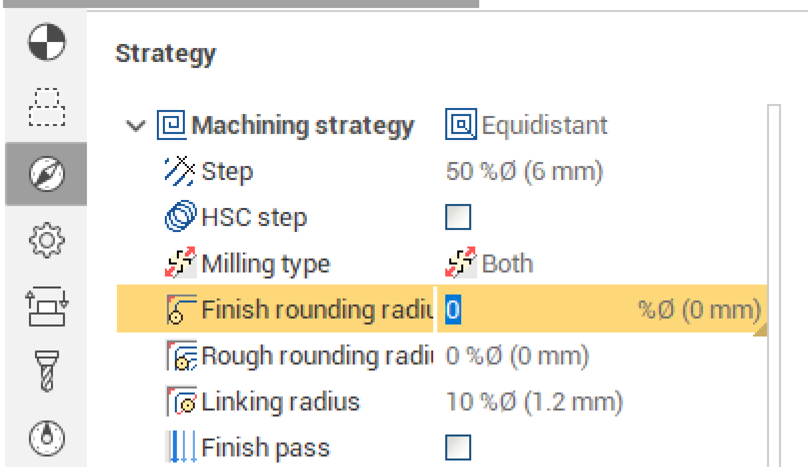
Back-off distance parameter



The tool can be lifted above the already machined surface when it moves to the next trochoidal arc start position.

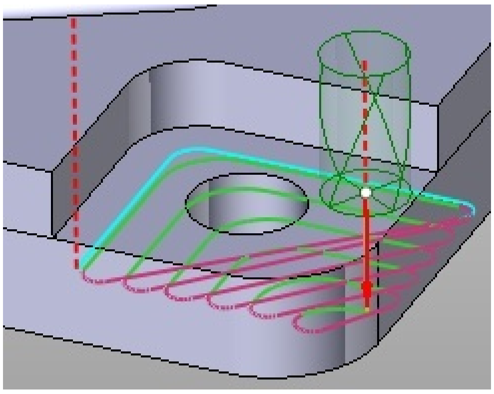
Rounded links in zigzag mode





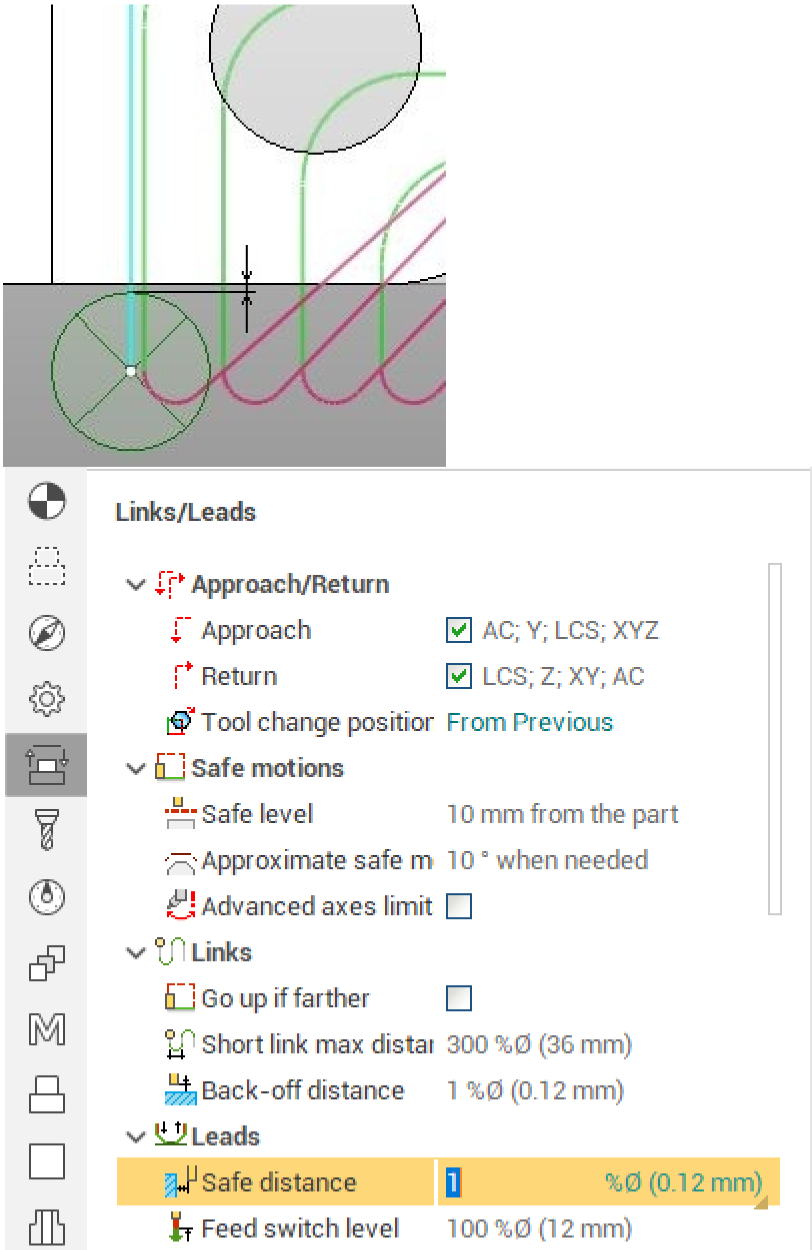
The ‘Finish rounding radius’, ‘Rough rounding radius’ and ‘Linking radius’ value is used for rounding of the links.

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

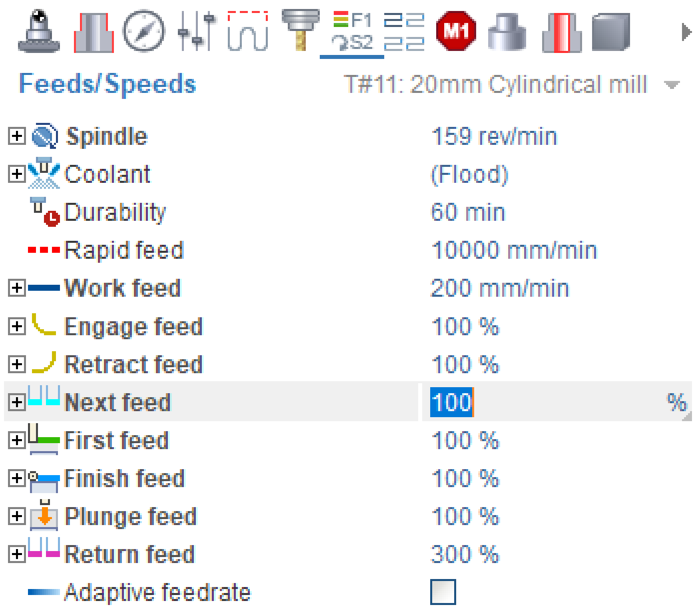


Safe distance is used to move the tool down/up from/to the safe surface.

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.

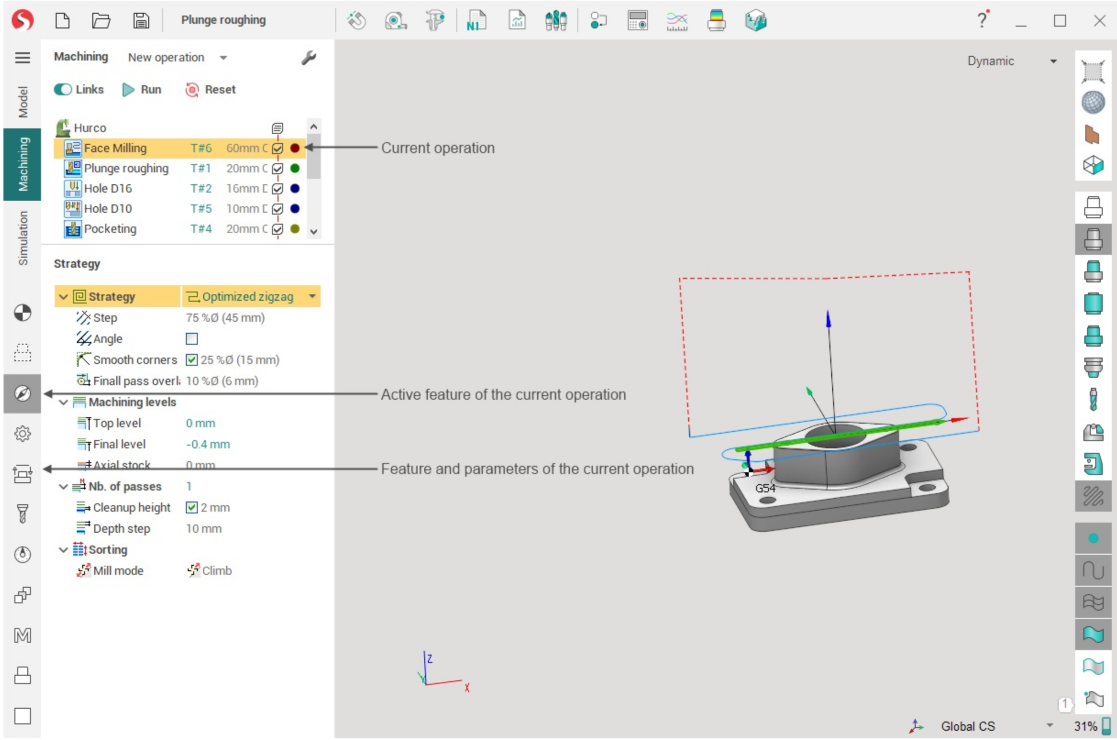
Rapid feed links



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.

5.5.7 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.



See also:

Common principles of technology creation

Geometrical parameters of an operation

Defining part, workpiece and fixtures

Positioning of part at machine

Tool selection window

Tool change position

Operation local coordinate system

5 axes positioning

Approach and return rules

User operations

5.5.7.1 Using design and machining features in Job Assignment

No content in this page. See child topics

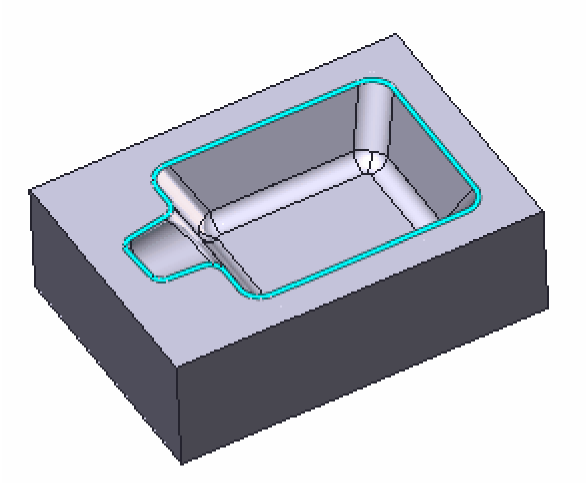
Job zones

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 SprutCAM offer an easy and intuitive way to define open machining areas using only source 3d model entities.

Using closed job zones

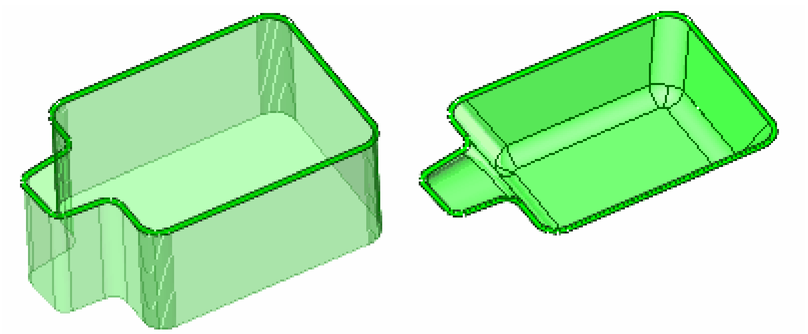
As a source geometry for closed job zones can be used closed chains of curves, edges and vertical walls. The best practice of using closed job zones includes the following steps.

1. 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.

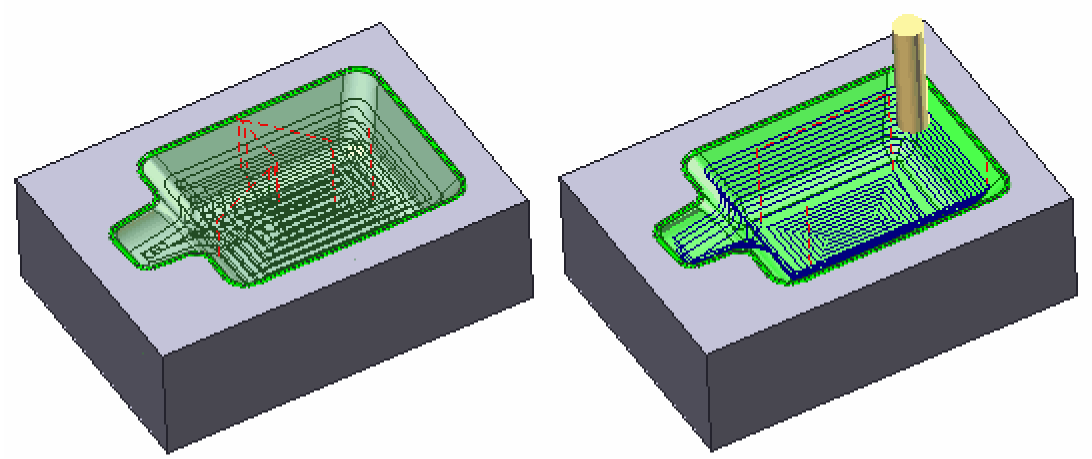


2. At the <Job Assignment> panel press the <Job Zone> button.

3. Doing so you will add a new <Job zone> into the <Job assignment> of an operation. According to the operation type a <Job Zone> may look different. There is the <Job Zone> for a <Roughing Waterline> operation at the left picture and the <Job Zone> for a <Finishing Drive> operation at the right picture following.



4. Generate toolpath. The trajectory should be contained inside the <Job Zone limits>.



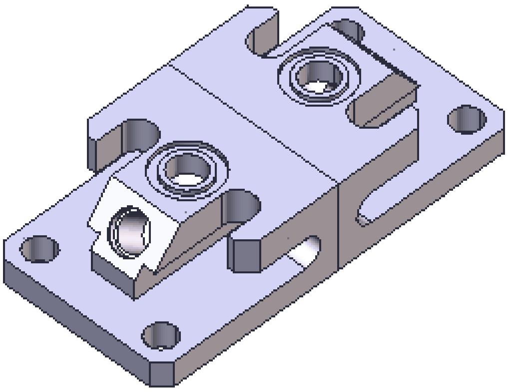
Using open job zones

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.

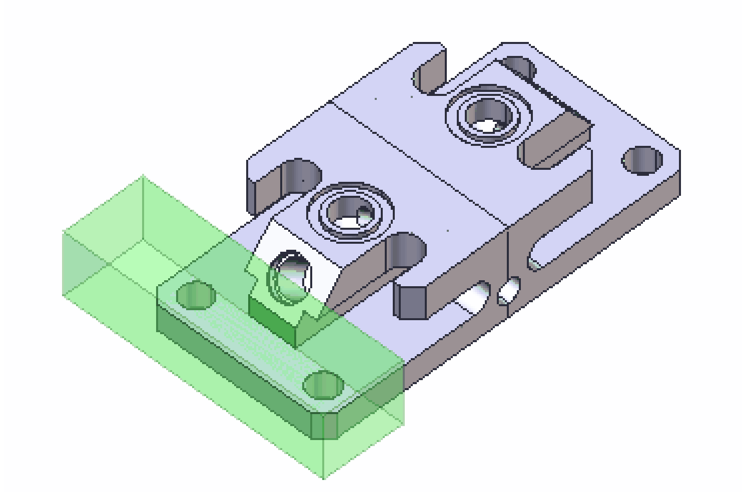
The best way to understand how and when to use open job zones is to look at practical examples.

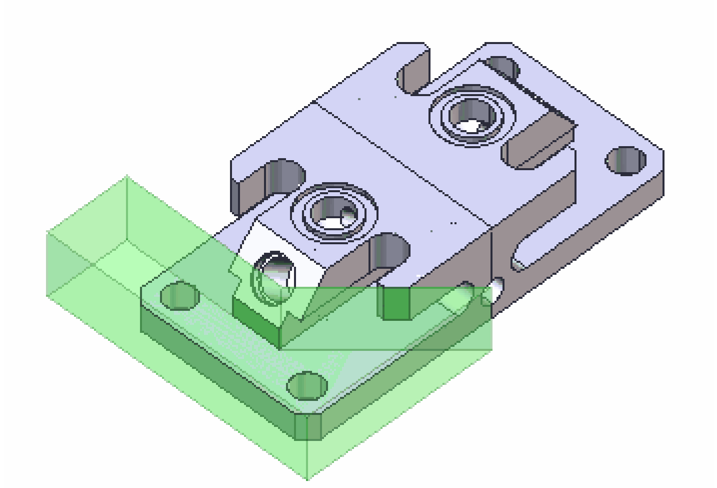
Using open job zones in a roughing operation

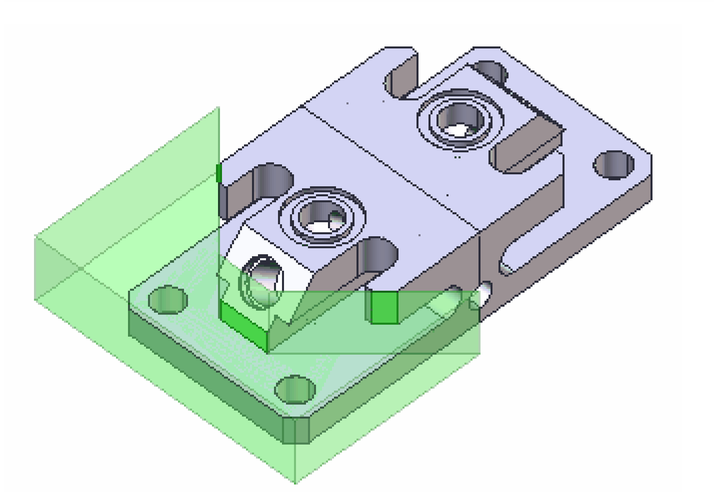
At the picture below you can see a complicated prismatic part consisting of many simple features. The workpiece for the part is a simple box. In the first stage we want to use a big tool to cut excessive material from the left side of the part using a roughing waterline operation.



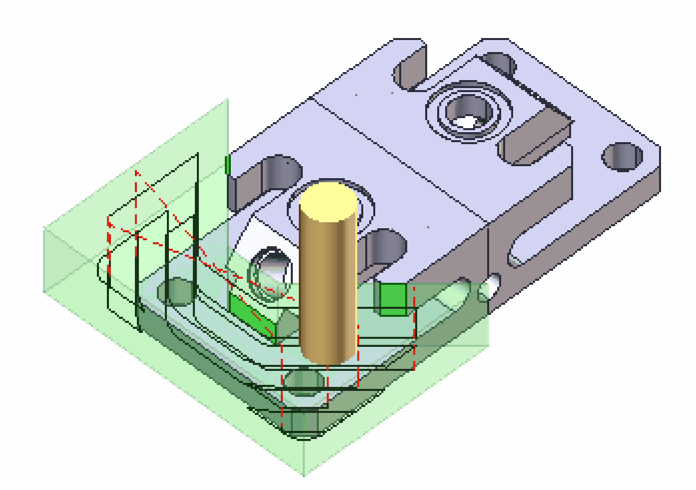
To accomplish this task and generate efficient toolpath without waste movements we have to define an open job zone, constructed from three vertical walls at the left side of the part. To do so we just consequently select the desired walls and add them into the Job assignment by pressing the <Job Zone> button at the <Job Assignment> panel.





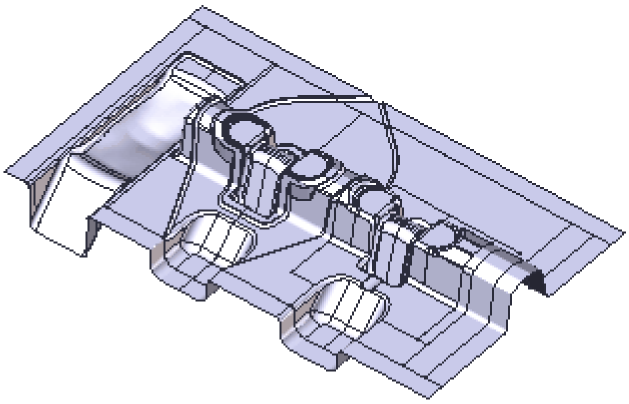


After that we generate toolpath. The result is depicted below:

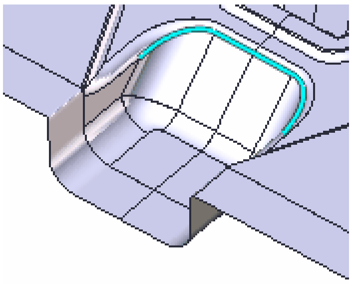


Using open job zones in a finishing operation

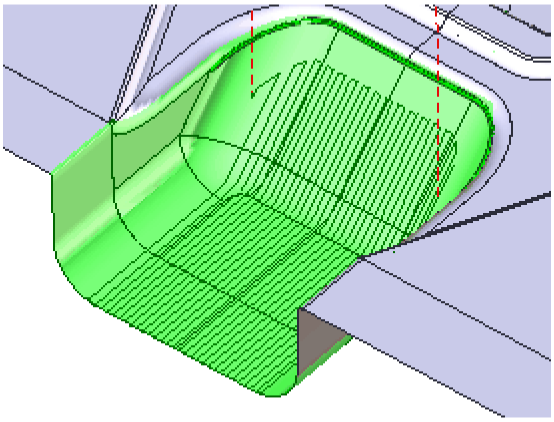
At the picture below you can see a die with three open freeform pockets. We will machine one small pocket using <Finishing plane> operation.



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.



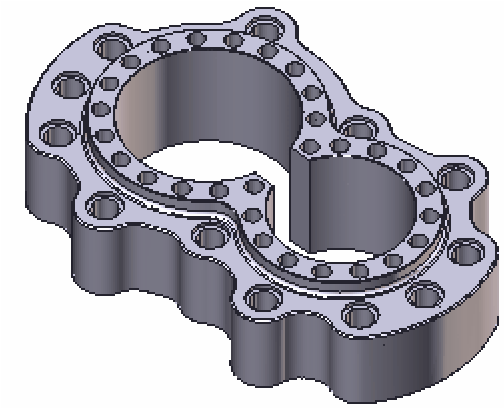
See also:

Mill machining

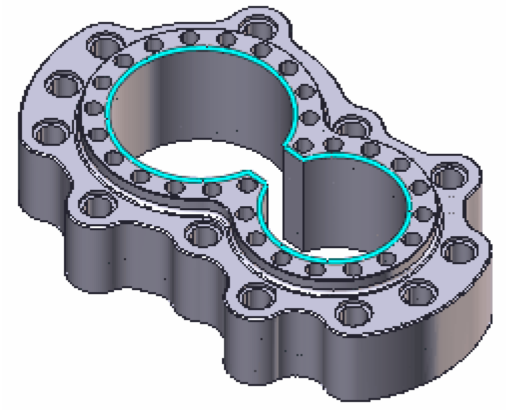
Restrict zones

In addition to Job Zones in SprutCAM X 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.

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.

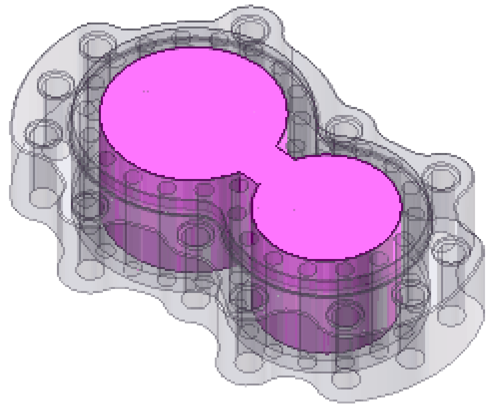


At first we select the edges chain around the window.

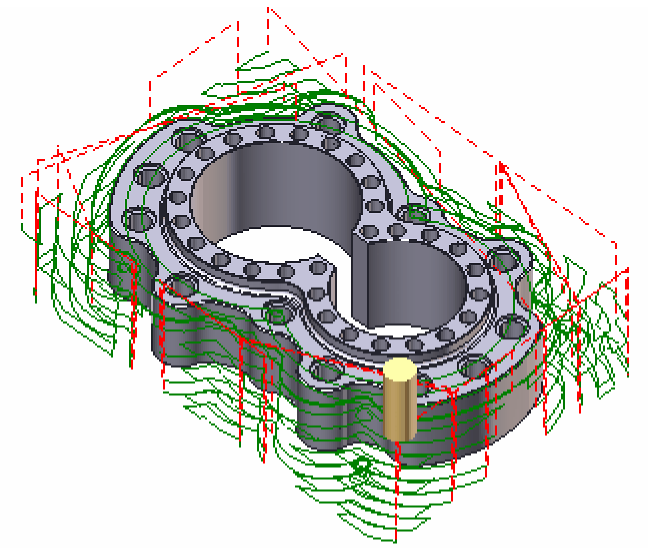


After that we add a new <Restrict Zone> into the <Job Assignment> of the <Roughing operation>.

Restrict zones are colored with the same color as the <Fixtures> of an operation.



After generating toolpath we will see the following picture. As you can see the inner window has been leaved uncut.



See also:

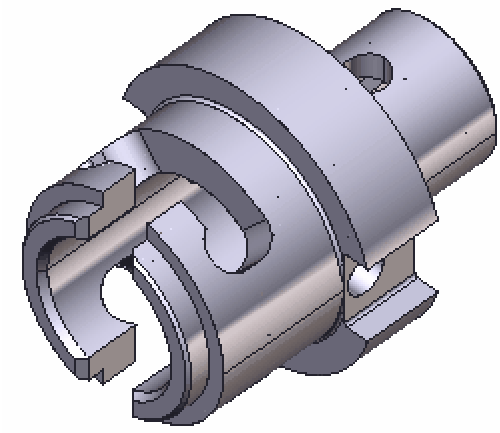
Mill machining

Top and bottom levels

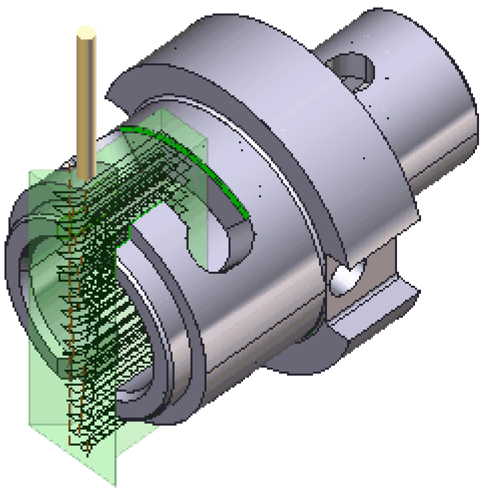
The common task while machining prismatic and turn-milling parts is to define the <Top> and <Bottom level> parameters of an operation based on the geometrical position of the machined feature. The most straightforward way to accomplish this task is to use the corresponding items of the <Job Assignment>.

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.

In the following picture you can see a turn-milling part. We will machine the front open slot.

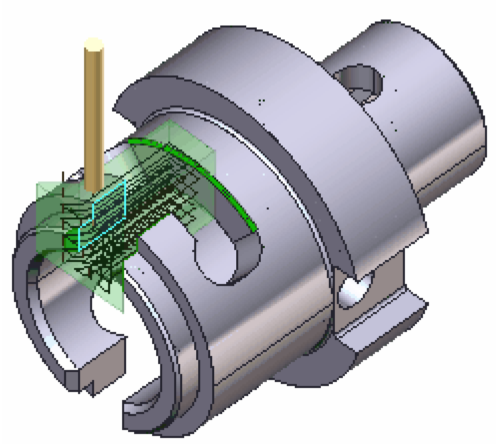


Using open job zones, we construct the machining zone in the XY plane. As you can see in the picture below, we have to limit the toolpath by Z. Both top and bottom levels have to be adjusted.



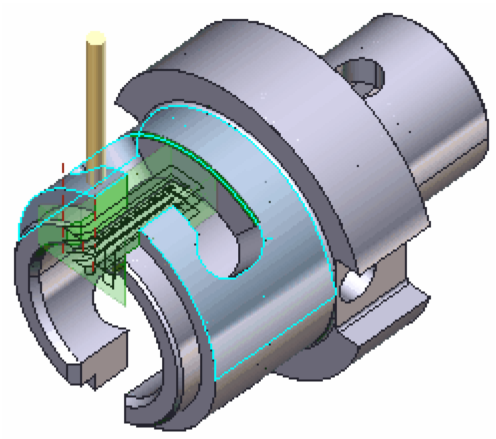
To accomplish this task we will add the side face of the slot as the <Bottom Level> item into the <Job Assignment>.

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>.

The resulting toolpath is pretty good:



See also:

Mill machining

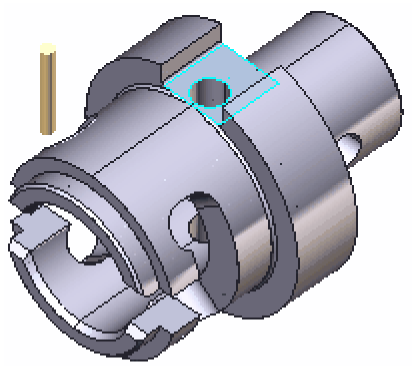
Pocket feature

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, SprutCAM X 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.

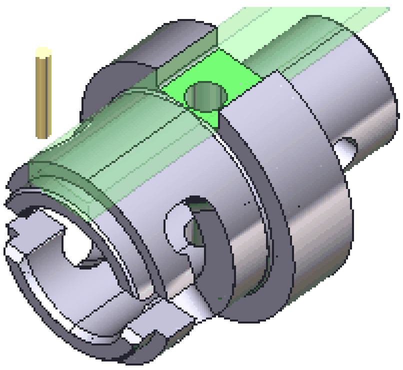
Using pocket feature

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.



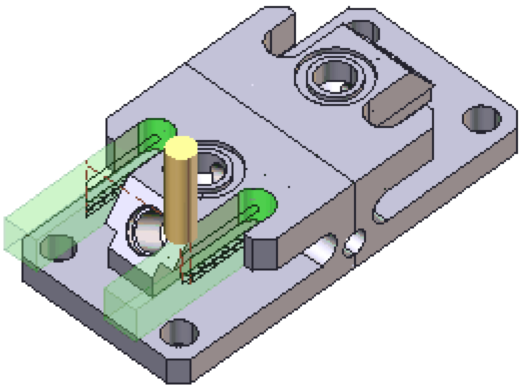
2. We select the bottom face of the slot and press the <Pocket> button at the <Job Assignment> panel.

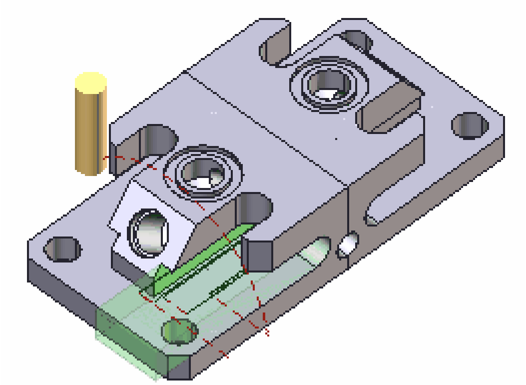
3. The pocket was converted to an open <Job zone> and <Top level> and <Bottom level>.

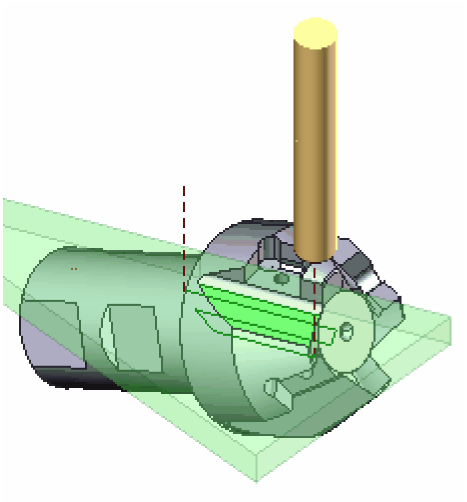


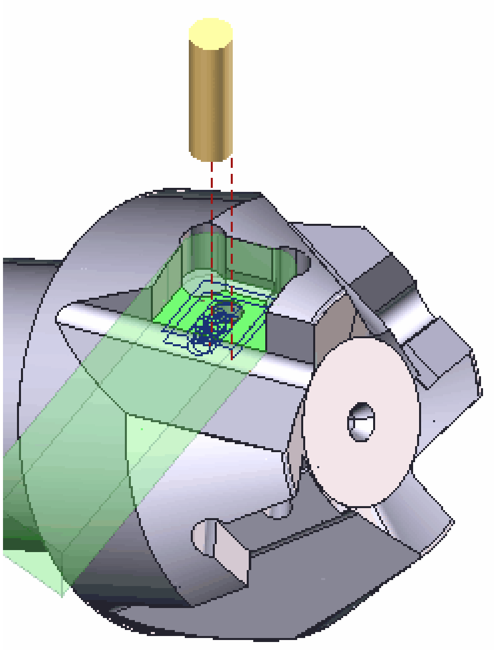
Typical uses of a pocket feature

Using a <Pocket> feature in a <Roughing operation>:

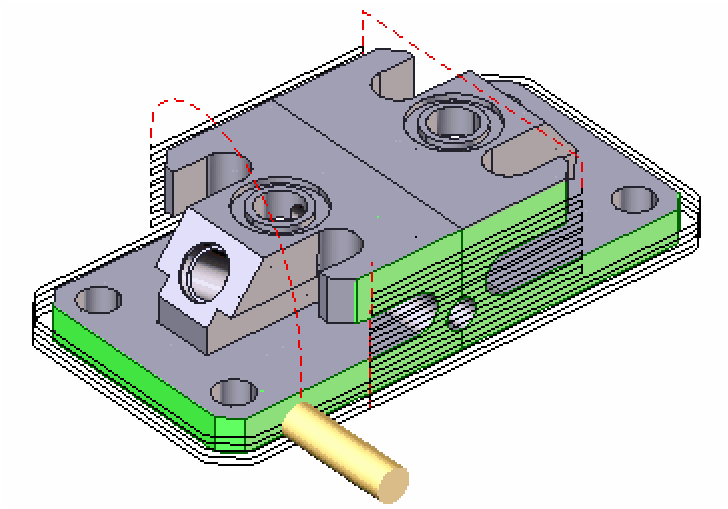




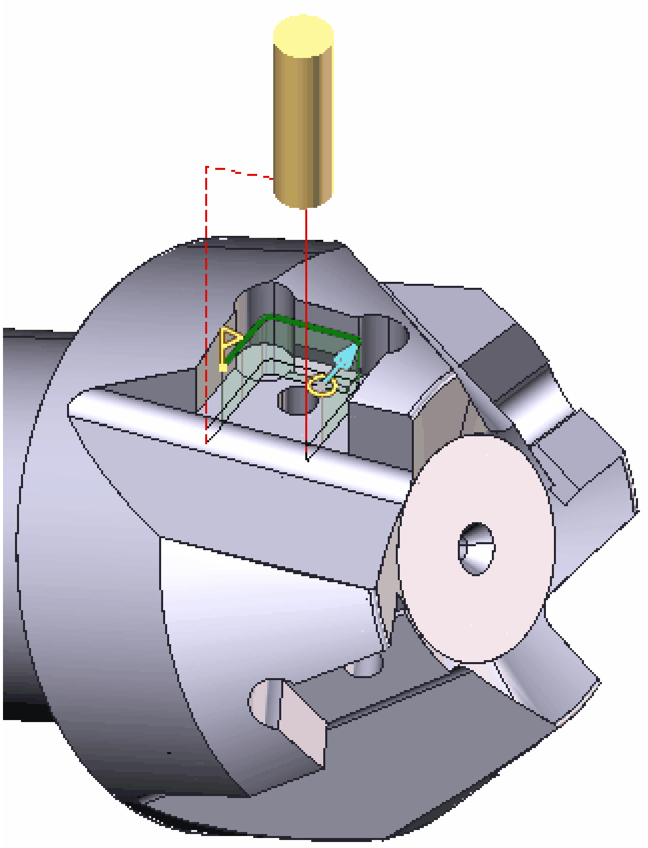


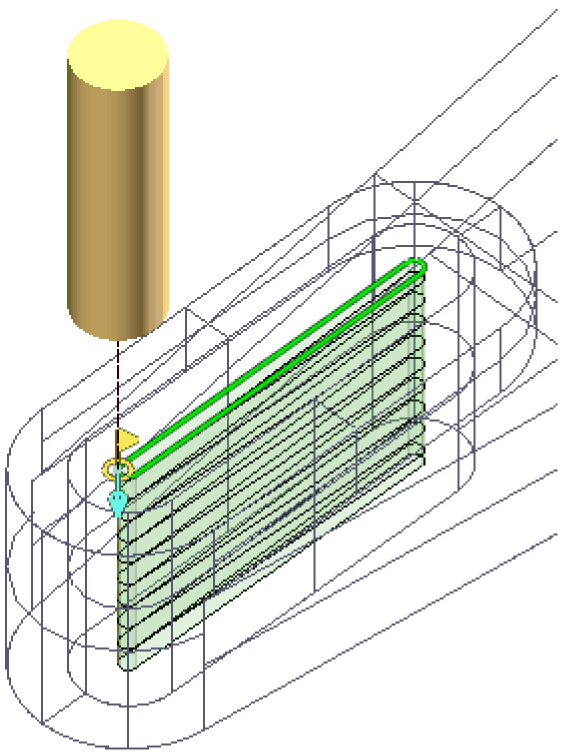


Using a <Pocket> feature in a <Finishing operation>:



Using a <Pocket> feature in a <2d contouring operation>:





See also:

Mill machining

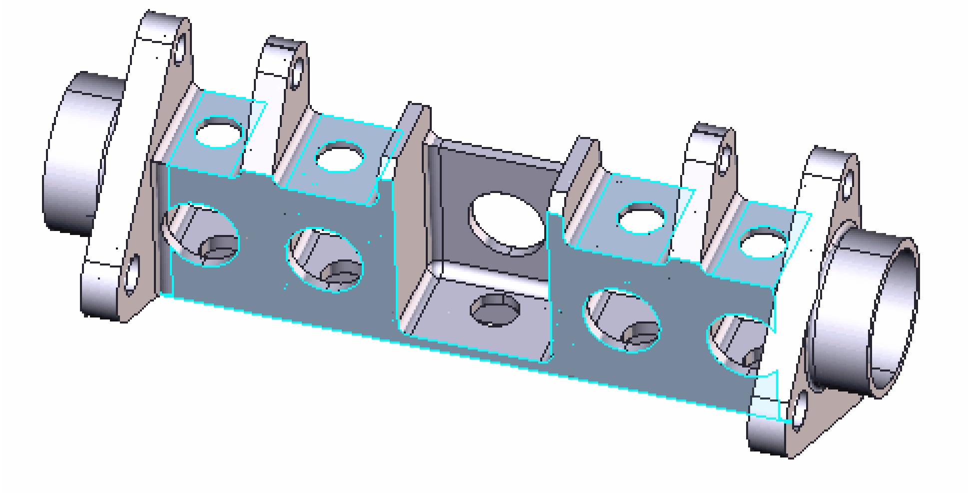
Creating operations for selected features

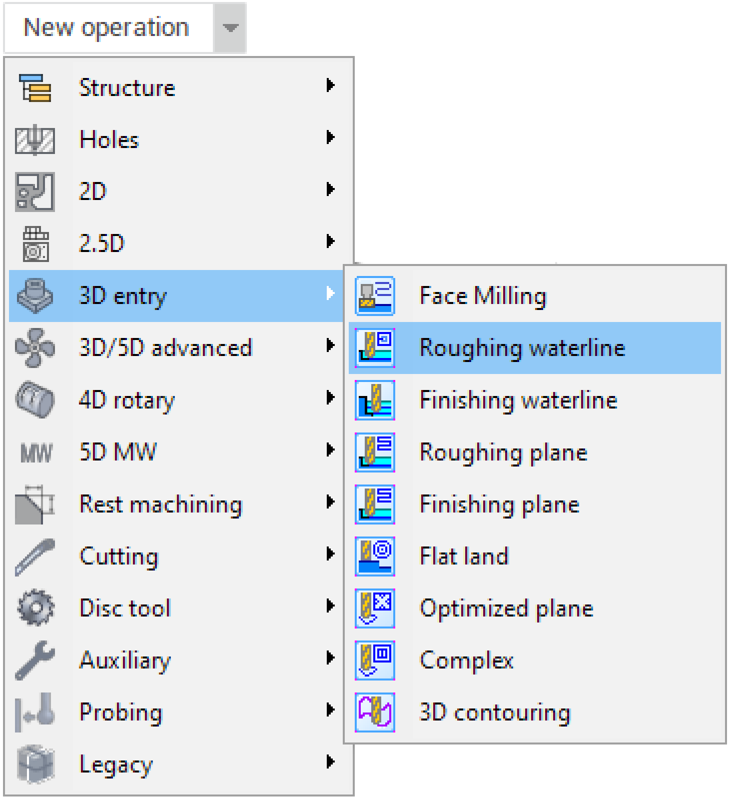
Creating technology for a prismatic part consisting of many simple shapes is a time consuming process. To simplify this task SprutCAM X offers the possibility to create operations for selected geometry. This is very straightforward. At first you select what you want to machine in the graphic view, and than you create the operation which will machine the selected geometry. Based on the operation type created SprutCAM X 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 SprutCAM X 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, SprutCAM X automatically measures it and highlights all the holes with the same diameter. Now if you create a hole machining operation SprutCAM X 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, SprutCAM X will create not one hole machining operation but several operations according to the number of planes in which holes are positioned.

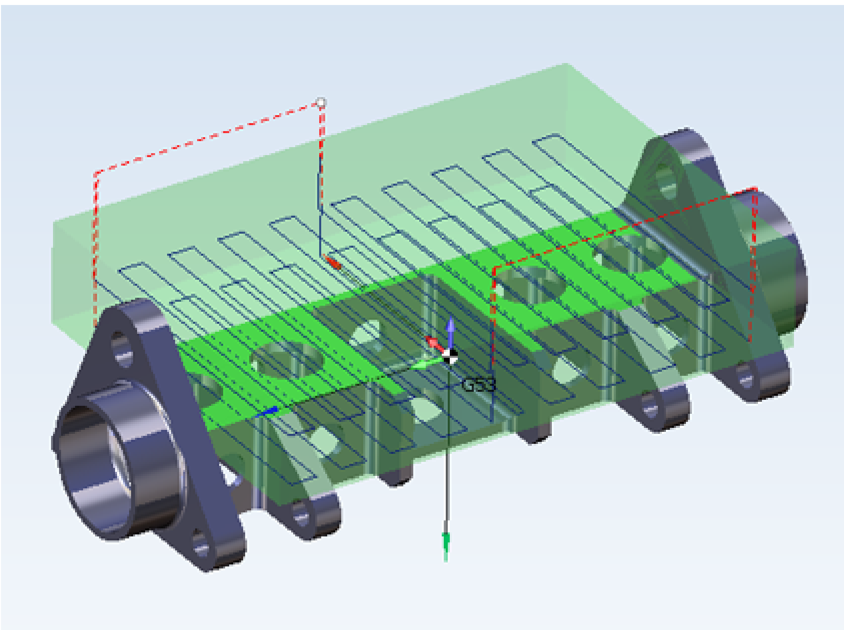
Example of fast creating pocketing operation:

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.





SprutCAM X automatically creates two roughing operations with pockets in the job assignment. The generated toolpath can be seen at the picture below:



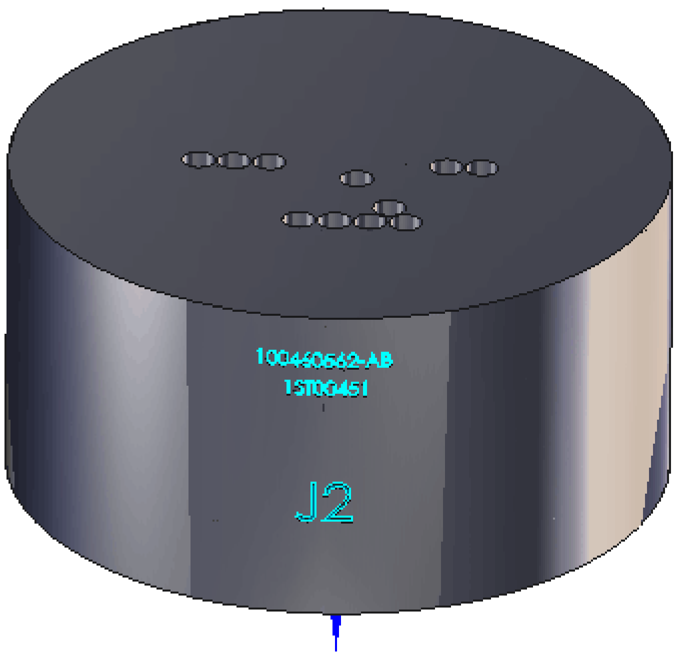
See also:

Mill machining

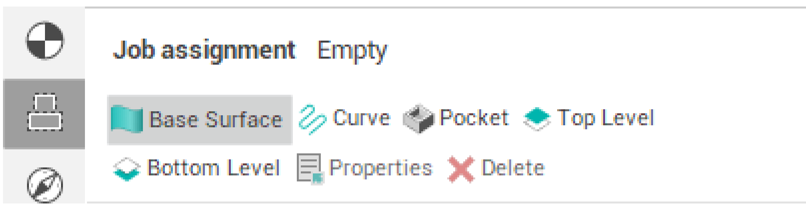
Using design features in Engraving/Pocketing operation

It is possible to create <2D Contouring>, <2D Pocketing> and <Engraving> 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.

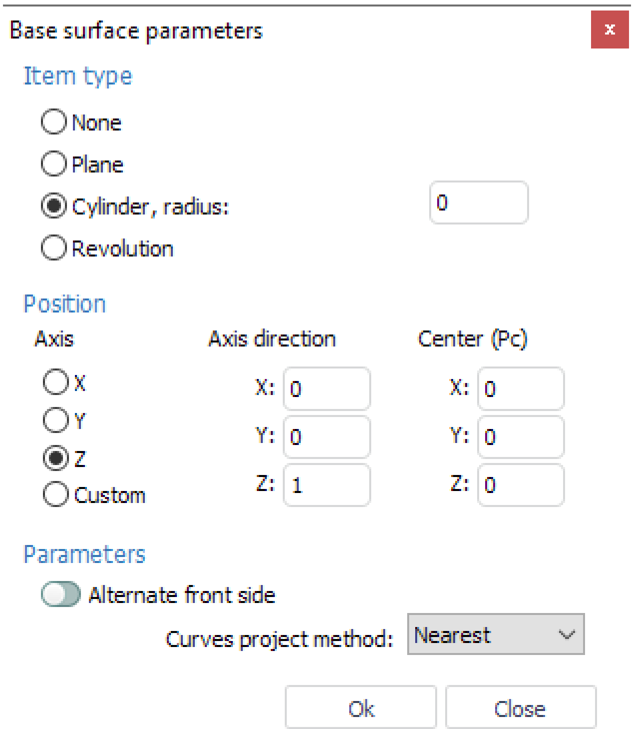
Let's look at the picture bellow. You can see a cylindrical part with a text on the outer diameter. We will engrave this text:



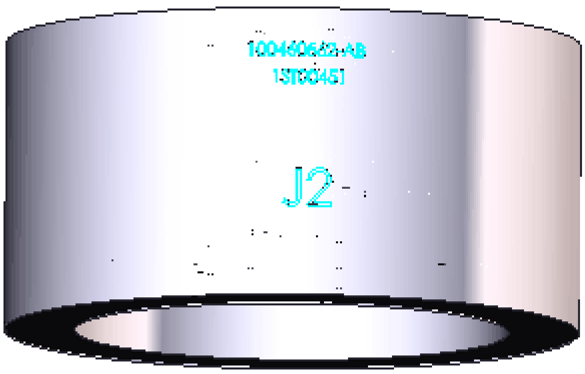
To accomplish this task we create a new engraving operation and press the <Base surface> button at the <Job assignment> panel of the operation:



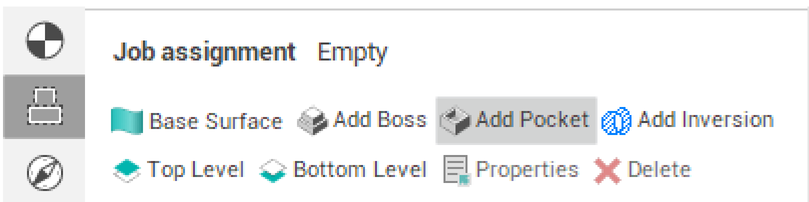
The base surface parameter dialog appears. We select the outer diameter as the base surface by simple click on it in the graphic view.



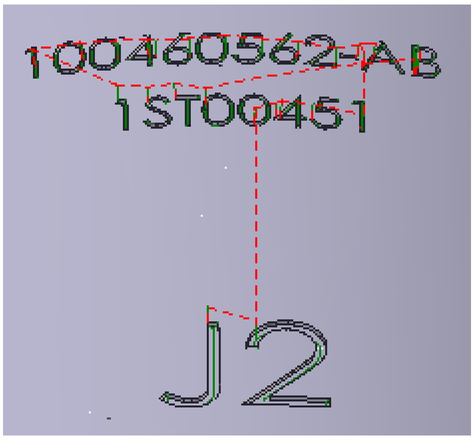
Than we close the dialog and select the letters we want to v-carve by double click on one of the letters in the graphic view.



After that we add selected surfaces into the <Job assignment> by pressing the <Add pocket> button:



After setting parameters of the operation and generating toolpath we will see the following picture:



See also:

Mill machining

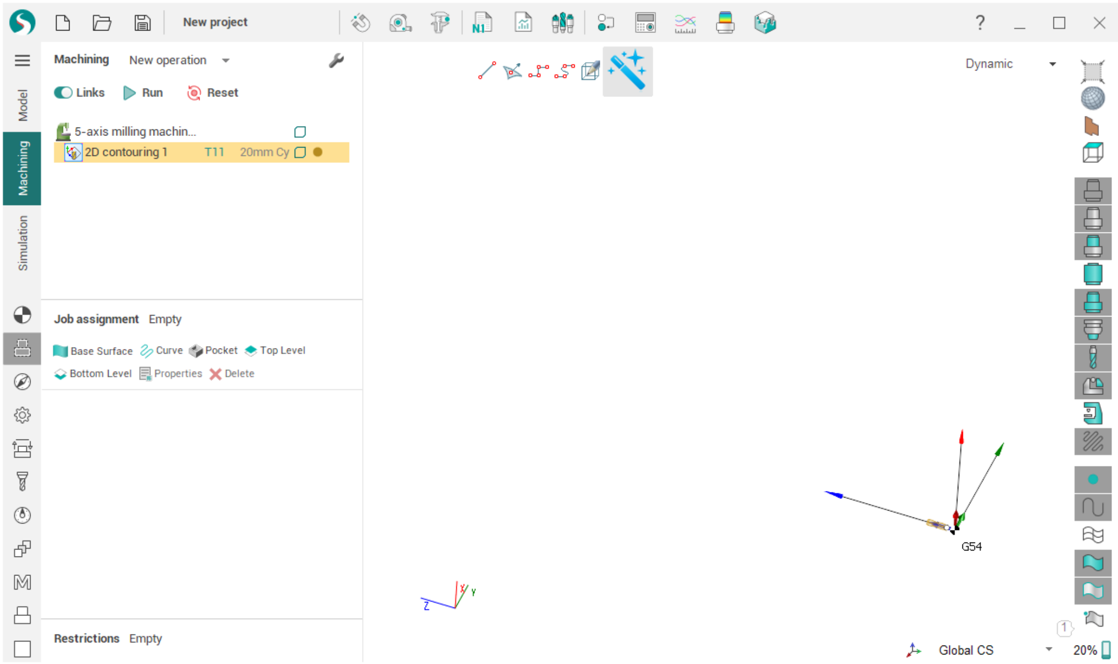
Engraving operation

4-axis milling with using of the engraving and pocketing operations

Job assignment for engraving and pocketing operations

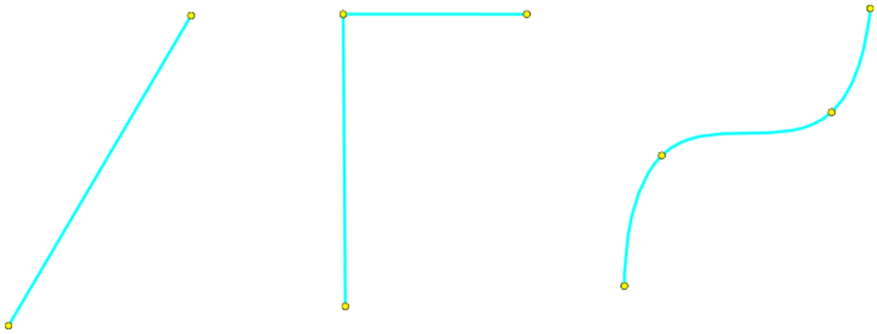
Spline curves drawing

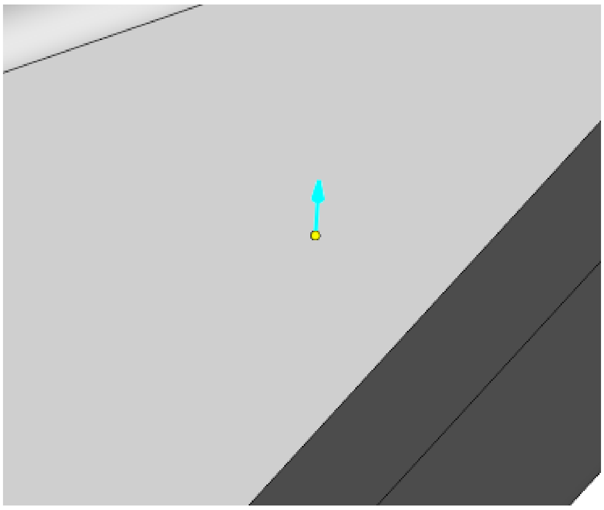
This feature is experimental, and still under the development. Live curve drawing toolbar added to the graphical window, which is visible while Technology mode is active.



It allows to draw auxiliary curves:

- straight lines
- polygons
- splines.



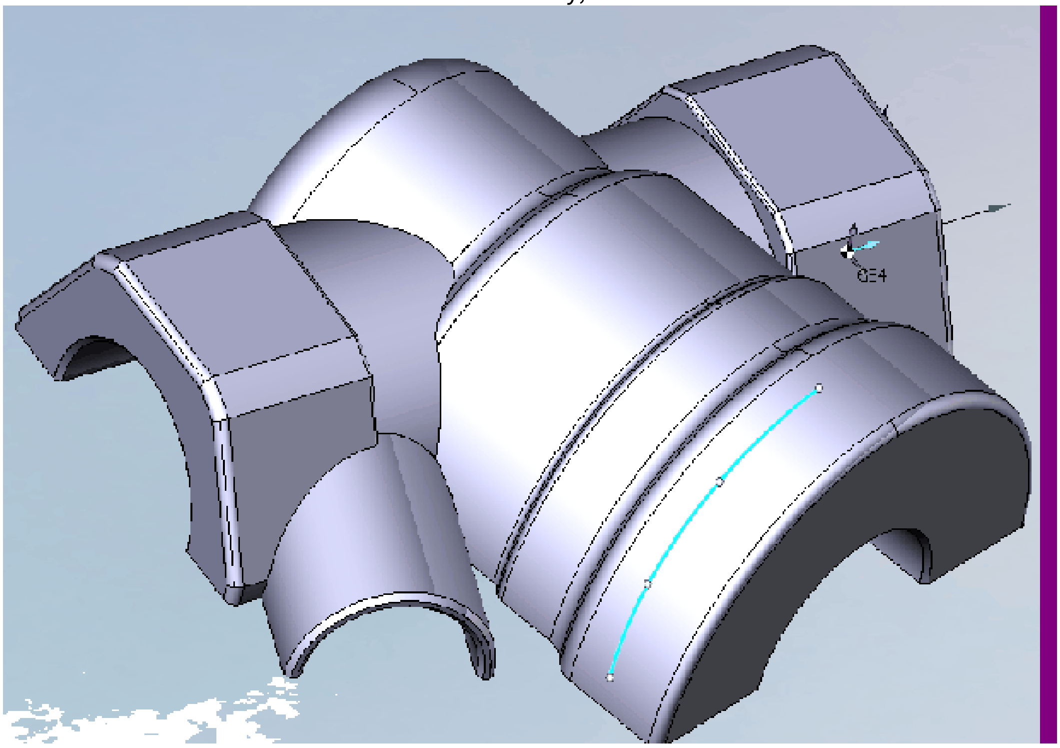


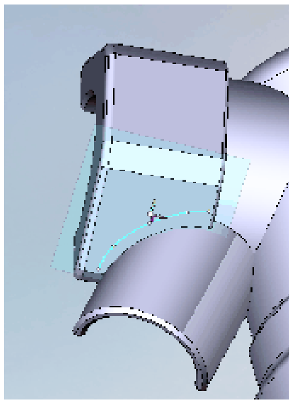
Directly on the surfaces of objects that are visible on the screen. Nodes of these curves can be dragged, deleted and added just by simple mouse click.

There are 3 drawing modes:

1. Free. Points are selected by the next rule
    a. In case, there is no snap object under the cursor, and it is the 1st point. The point will have undefined Z.
    b. 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);
    c. In case we snap surface under cursor surface's, the nearest point will be selected.

2. On surface. Points will be selected from surface only;



3. 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).

See also:

Mill machining

Engraving operation

4-axis milling with using of the engraving and pocketing operations

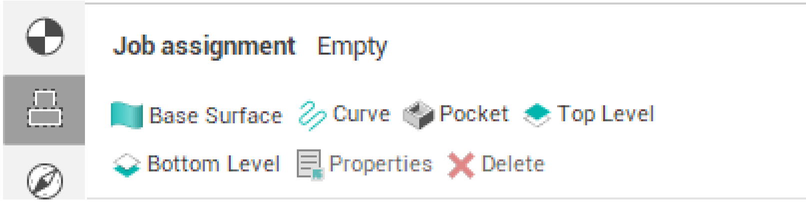
Job assignment for engraving and pocketing operations

5.5.7.2 Job assignment for 2D and 3D curve machining operations

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.

Adding new items

In the 2D contouring job assignment you can use the following items: **curve**, **pocket**, **top level**, **bottom level** and **base surface**.



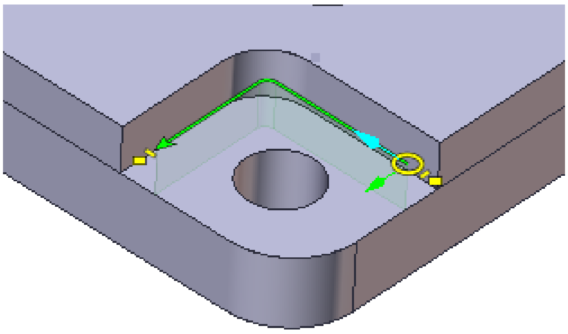
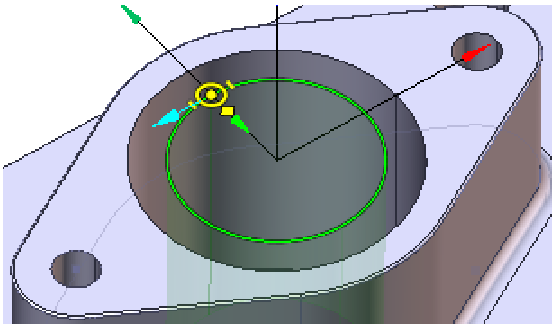
To add a new contour into the job assignment just select curves, edges or vertical faces you want to machine and press the <Curve> button. The connected entities will be joined into contours.

To add a pocket into the job assignment just select either a bottom or a side face of the pocket and press the <Pocket> button. A pocket defines not only the contour, but also top and bottom levels.

You can define top and bottom levels of machining in the Job assignment. Just select a geometry entity in the graphic view and press either the <Top Level> or the <Bottom Level> button.

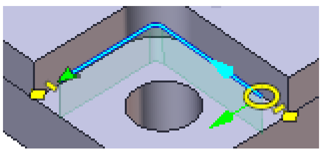
Contour features

A job assignment item looks like at the pictures below.

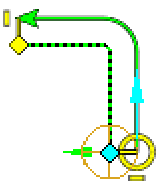



An item consists of the following features.

1. Contour itself. The bold green line represents the curve which will be output into the CLData.

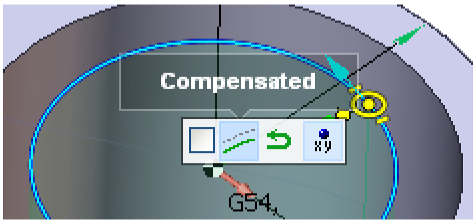


If a contour is compensated and the <**Radius compensation**> is set to <**Control**> than an additional dashed line is displayed, which represents the tool center curve (the curve along which the tool will be moved).



Accessing contour parameters

To access contour parameters just left click on a contour in the graphic view. You will see the action bar.



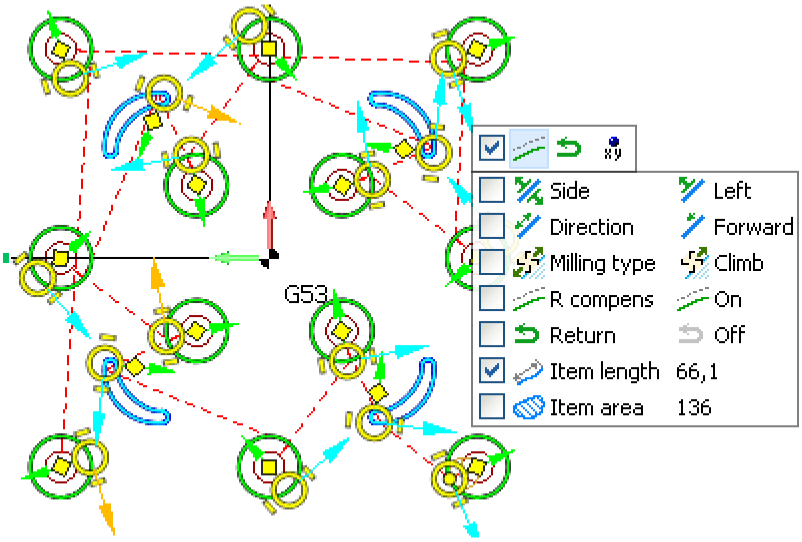
Here you can switch the <**Compensated**> flag of a contour and its <**Machining with return**> flag. For closed contours you can also switch the <**Fix start point**> flag.

Changing parameters of several contours together

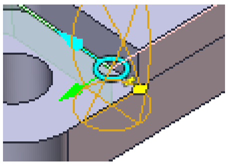
To do this you should select several contours and edit parameters of one of them. When you make a change to a parameter of one contour this change applies to all the selected contours. In such a way you can change the <**Compensated**> flag, the <**Machining with return flag**>, the <**Fix start point**> flag, the contour direction, the contour side.

Selecting multiple contours

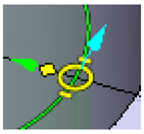
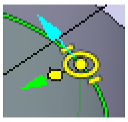
- To select several contours you can hold the [**Ctrl**] key on the keyboard and select them with the mouse.
- To select all contours in the job assignment just double click on any of them in the graphic view.
- To select contours by some criteria click on a contour, than on the check box in the action bar that appears and check the criteria in the properties tree that appears.



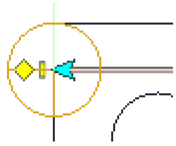
2. **Contour start point** (yellow circle). To change start point position just drag it with the mouse. You can also change start point position by X and Y dimensions. Just left click on the point and edit the dimensions that appear.



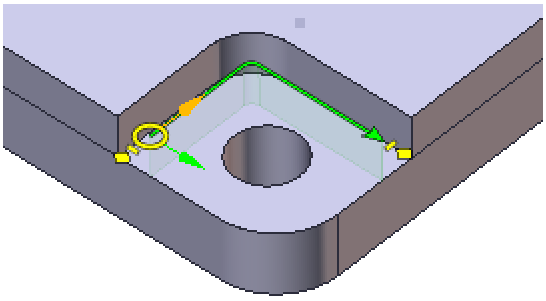
The default start point position for closed contours is <**Auto**>. It means the position of a point is determined by the operation during toolpath generation. You can distinguish auto start points from regular ones by their visual representation. Auto start points are depicted without yellow dot inside, regular ones points are displayed with a yellow dot inside.

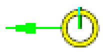
3. **Contour terminate point** (green arrow for an open contour and yellow dot in the middle of the yellow circle for a closed contour). You can drag the terminate point with the mouse, and you can change its position by X, Y dimensions.



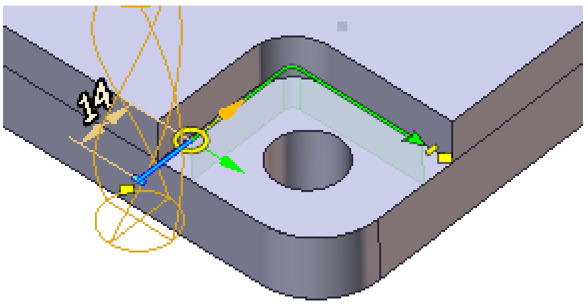
4. **Contour direction**. This is an aqua colored arrow attached to the start point at the above images. To change the contour direction just click on this arrow. The direction arrow can be of two colors: the aqua and the orange. The color of the direction arrow indicates milling mode of a contour, It's 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.



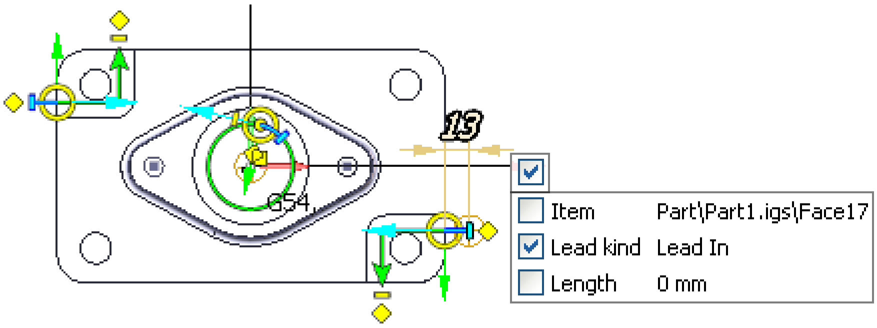
6. **Contour overlaps**. These are yellow dashes next to contour start and terminate points at the images above. An overlap is used to extend a toolpath beyond the contour itself. To change the overlap value you can either drag the yellow dash with the mouse or click on it and edit the dimension value which appears.



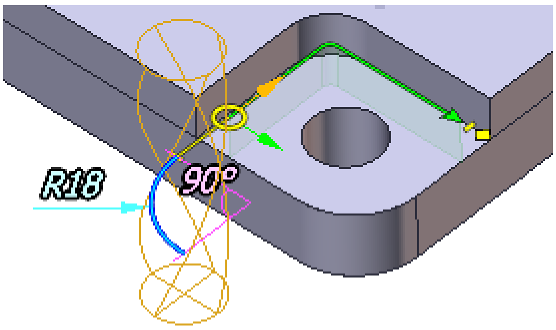
Editing multiple overlaps in one click

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 SprutCAM 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.

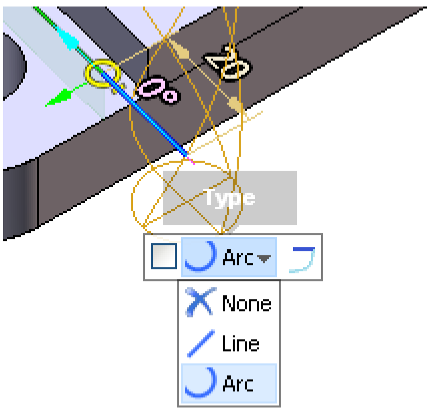
To select overlaps by a criteria you should click on one overlap, than on the check box in the action bar which appears and check the criteria on which you want select overlaps.



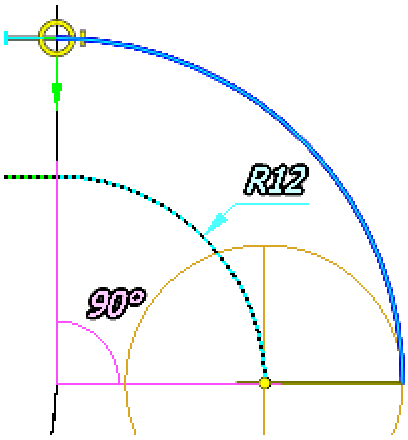
7. **Contour leads**. These are yellow diamonds at the above pictures. To change a lead just drag the corresponding diamond with the mouse. After that you can edit lead dimensions that appear.



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.

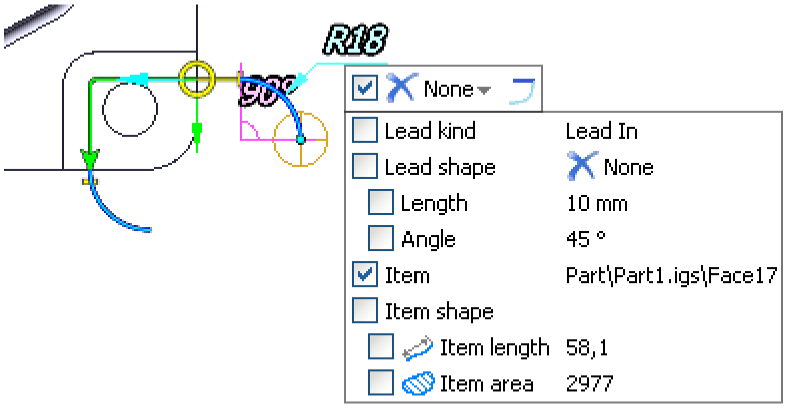


For **compensated** contours and <**Control**> radius compensation an additional dashed line is displayed. This line represents the tool center curve. Lead dimensions are specified for the tool center curve. This makes lead parameters independent of the current compensation type. If you want to change 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.

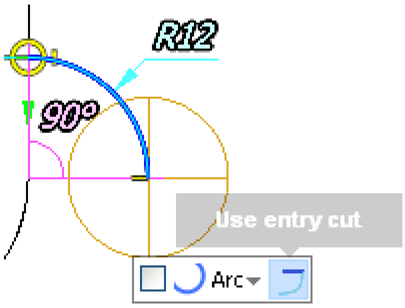


Editing multiple leads in one click

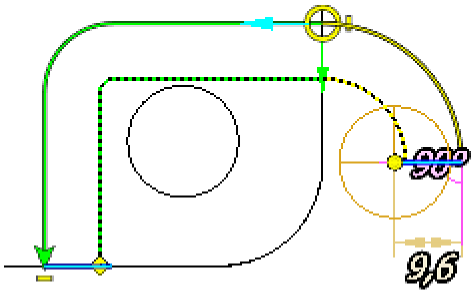
You should know that when you edit a lead parameter this change applies to all selected leads. To make the process of editing multiple leads quick and simple SprutCAM automatically highlights and selects all identical leads. You can disable this behavior by clicking on the magic wand button at the top 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.

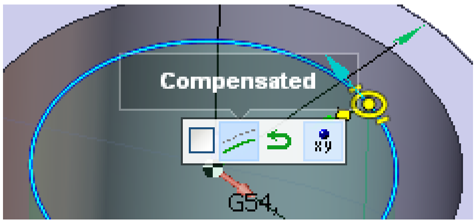


Here is an example of entry lines with <**Control**> radius compensation.

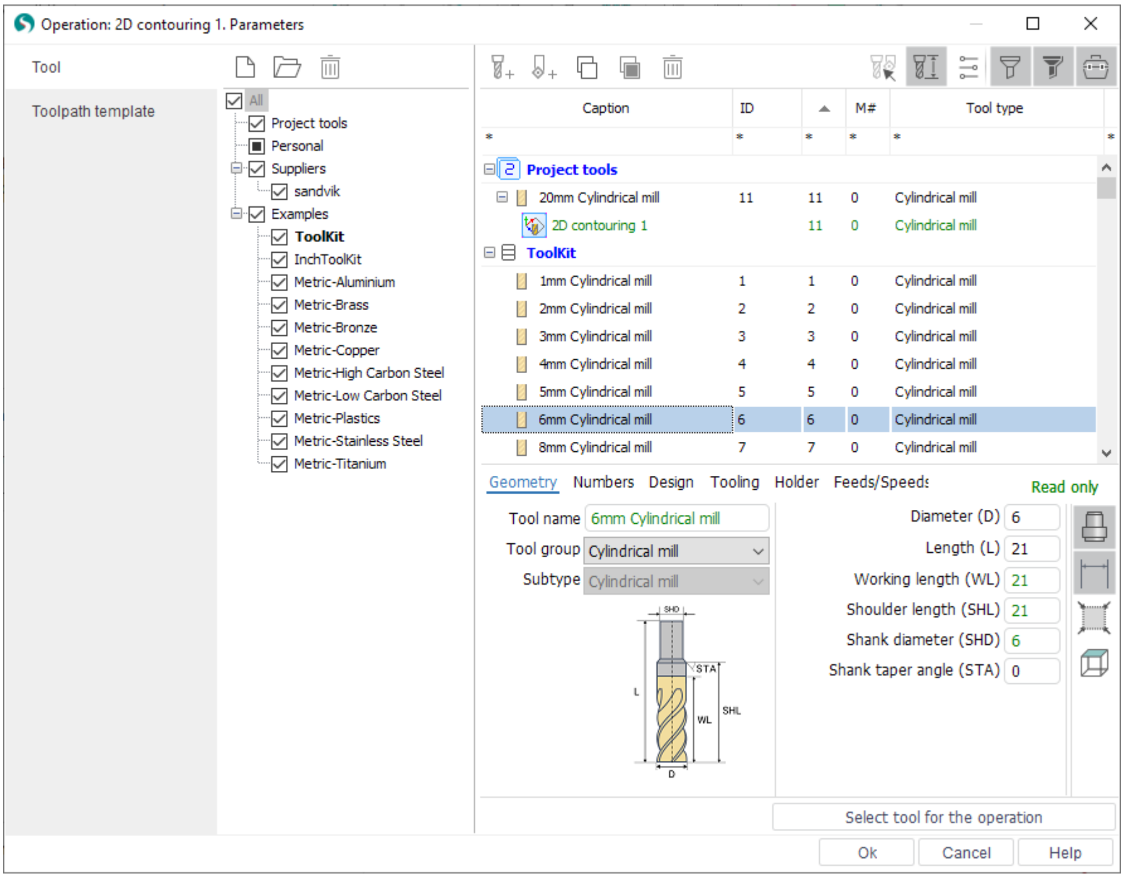


Contour offsetting/radius compensation

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.

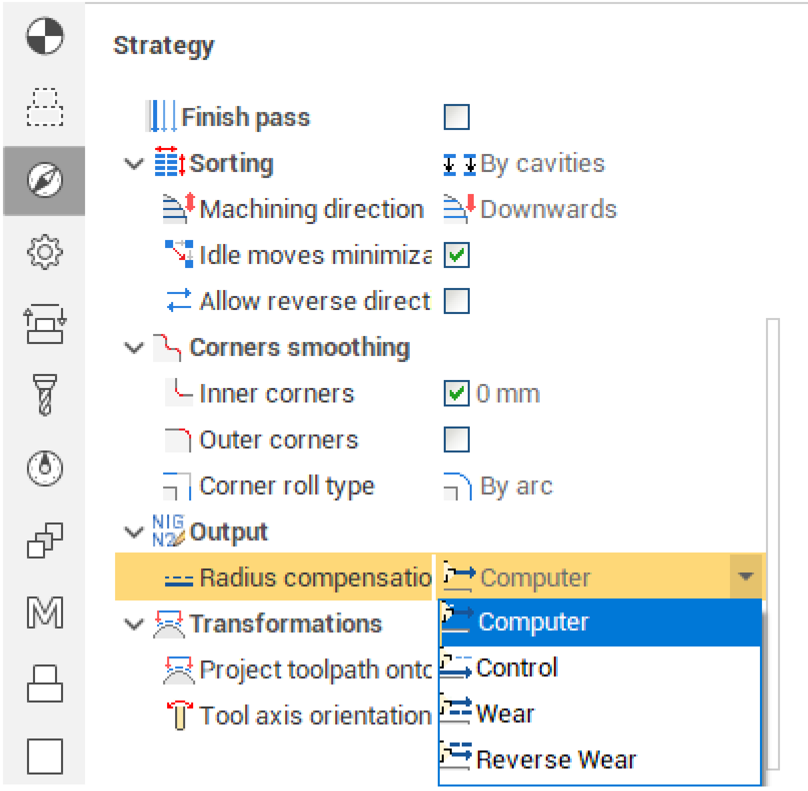


The tool maximal radius is used by default as the contour offset radius. But you also can specify the tool contact point explicitly in the Tool parameters.



In that case the contour offset distance will be taken from the tool contact point.

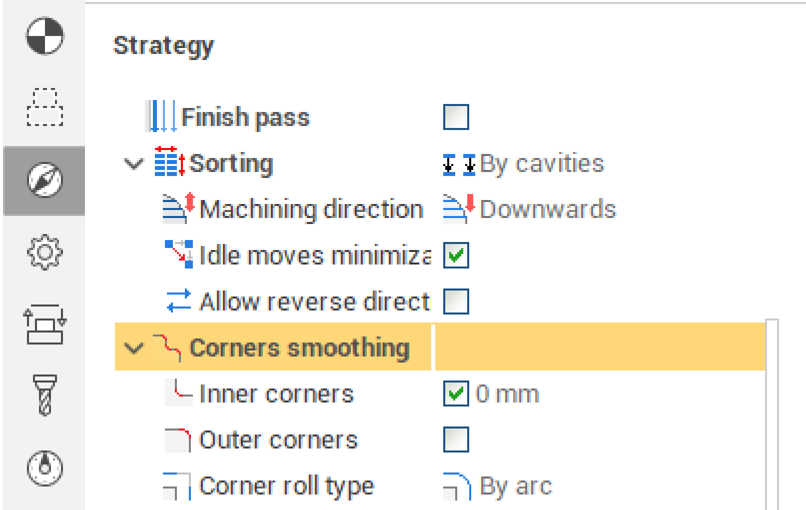
The method of radius compensation is determined by the operation parameter <**Radius compensation**>. You can select it either in the parameters inspector or in the operation parameters windows.



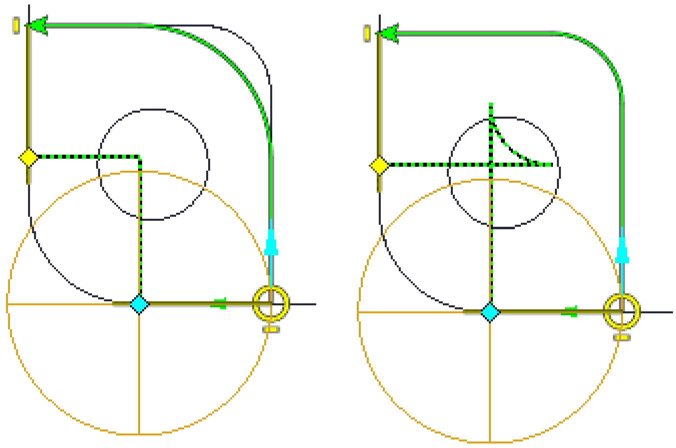
There are four types of radius compensation: Computer, Control, Wear, and Reverse Wear.

Computer radius compensation. When compensation type is computer SprutCAM offsets contours by the compensation radius and outputs them into the CLData. This method of radius compensation is most robust, as SprutCAM automatically removes all the loops and self intersections from the contour offset curves.

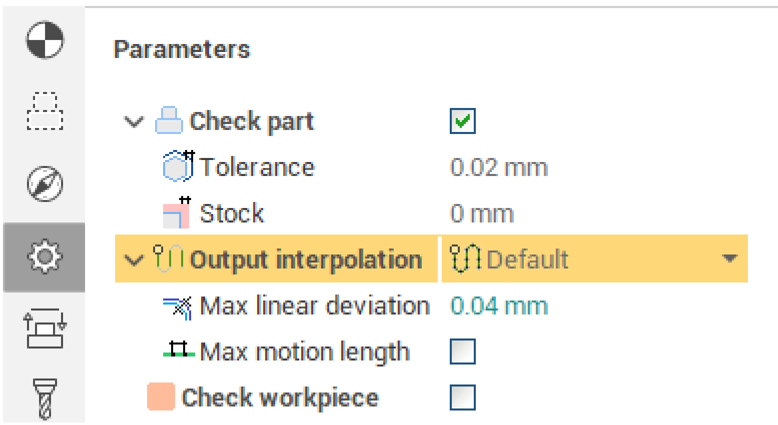
Control radius compensation means that the used CNC control is capable of doing radius compensation itself and SprutCAM should output into the CLData not the tool center curves but the part line curves. SprutCAM also generates compensation switch commands (G41, G42, G40 in terms of G-codes) in this mode. This method is generally used when you don't know what tool diameter will be used on the shop floor. But this method has also its shortcomings, as not all contours can be successfully compensated by a CNC control. To make this process more robust, SprutCAM automatically rounds all internal corners of compensated contours with the used compensation radius value. This behavior is controlled by the <**Inner corners smoothing**> parameter.



By default <Inner corners smoothing> is enabled and is equal to 0 mm. The corner smoothing values are specified for the tool center curves, so when the control radius compensation is used SprutCAM smooths contours with the radius you set in the box plus the compensation radius. To output unmodified contours into CLData just disable the Inner corners smoothing option.



To delete the noise from the output contours and improve the stability of control radius compensation you can also interpolate contours by enabling <Arc interpolation>.

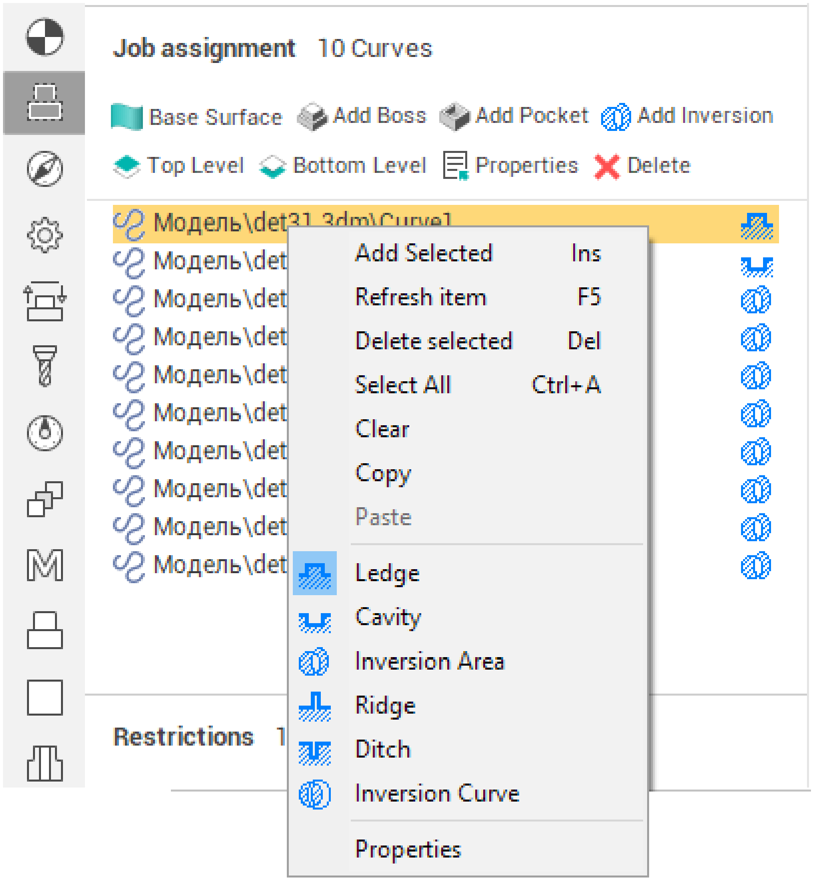


The <Wear> and <Reverse wear> radius compensation types are combinations of the <**Computer**> and <**Control**> radius compensation methods. SprutCAM outputs tool center curves into the CLData and generates compensation switch commands. These settings are generally used if you want to accommodate the tool wear on the shop floor. When using <**Wear**> radius compensation SprutCAM generates "inverted" compensation switch commands (G42 instead of G41 and vice verse). This allows you to use a smaller tool on the shop floor than you specify in SprutCAM (A mill becomes thinner and smaller due to wearing). The <Reverse wear> compensation method can be used as replacement of the <Control> radius compensation, but in that case on the shop floor you can use only tools which are bigger than the tool you used in SprutCAM.

See also:

Mill machining

5.5.7.3 Job assignment for engraving and pocketing operations

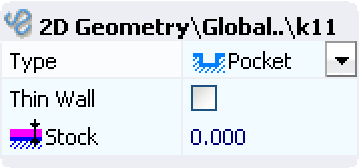


For the engraving and pocketing operations, the model is defined by curves as for the 2D and 3D curve machining operations. However, unlike the curve machining operations, the system forms a model for machining from the defined curves. A model for machining represents a flat area, which only exists where 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:

- <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.

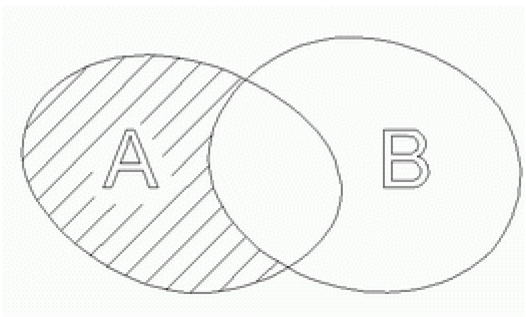
A <Ledge>, <Cavity> or <Inversion area> can only be defined by closed curves. When an open curve is selected, then these types will be unavailable. If a closed contour consists of several fragments, then it must be joined into a single curve (see Curve joining). A <Ridge>, <Ditch> or <Inversion curve> can be defined by any curve, either closed or not. To define their thickness, use additional stock.

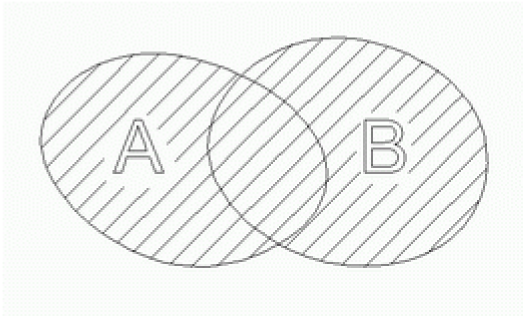
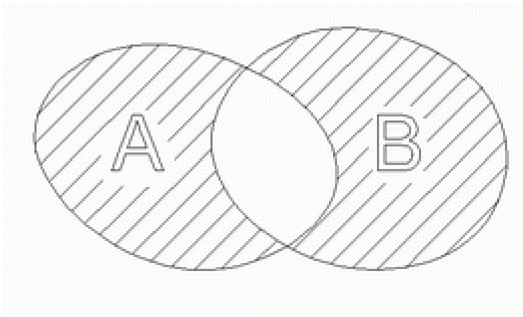
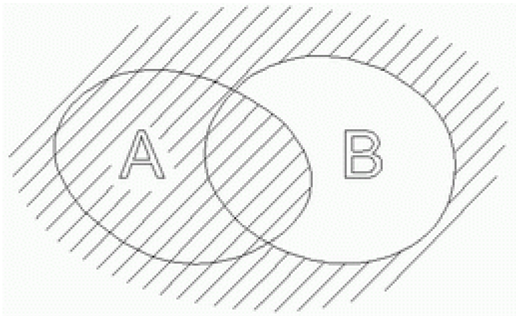
After the curve addition the window of curve parameters can be called by selecting the curve in graphic window and pressing right mouse button. The window is intended to view and edit the item parameters.



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.

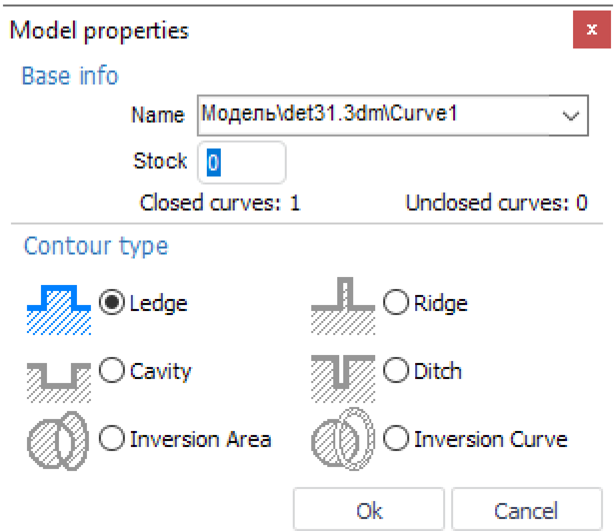
List contents	Resulting area
A – ledge B – cavity	

A – ledge B – ledge	
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.

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.*

See also:

Mill machining

Engraving operation

4-axis milling with using of the engraving and pocketing operations

Using design features in an Engraving/Pocketing operation

5.5.7.4 Job assignment for volume machining operations

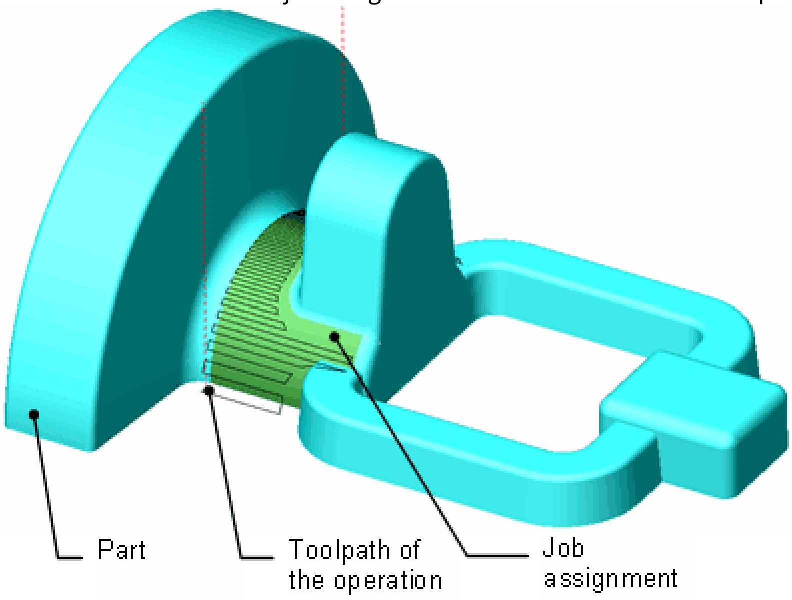
The job assignment of an operation defines the list of elements to be necessarily machined by the operation, while the part in turn is only checked to not produce tool travels involving part gouges; the part geometry itself is not used to generate toolpath. The volume machining operations use faces and meshes as job assignment. This is the list of the volume machining operations:

- hole machining operation;
- waterline roughing operation;
- plane roughing operation;
- drive roughing operation;
- waterline finishing operation;
- plane finishing operation;
- drive finishing operation;
- combined operation;
- plane optimized operation;
- complex operation;
- waterline rest milling with clearance;
- waterline rest milling;

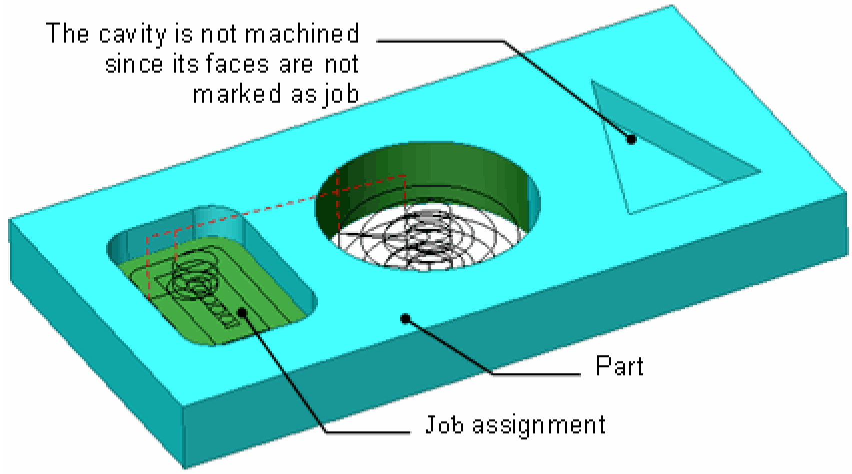
- plane rest milling;
- drive rest milling;
- plane optimized rest milling;
- complex rest milling.

- hole machining operation;
- waterline roughing operation;
- plane roughing operation;
- drive roughing operation;
- waterline finishing operation;
- plane finishing operation;
- drive finishing operation;
- combined operation;
- plane optimized operation;
- complex operation;
- waterline rest milling with clearance;
- waterline rest milling;
- plane rest milling;
- drive rest milling;
- plane optimized rest milling;
- complex rest milling.

The default job assignment for a volume machining operation is the <Current part>. The same item can be also added to the job assignment by pressing the <Reference> button. As one can see from the name of the item, all the faces of the part will be machined. If only several faces have to be machined those should be added to the job assignment explicitly by pressing the <Add Faces> button. The order the items are listed in the job assignment does not influence the toolpath will be generated.



The job assignment for roughing operations specifies the part fragments to be machined. At that the material that put obstacles to the tool will be also removed. This means that it is no need to specify all the faces of the cavity to be machined as job assignment. It is enough only to indicate the lowest face of the cavity



The job assignment also defines the automatic machining levels. For example, if a wall of a hole is specified in the job assignment then the lowest level to machine will be determined by the lowest point of that surface.

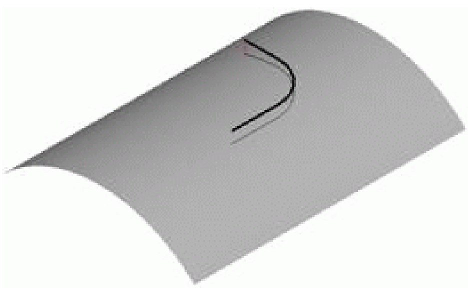
See also:

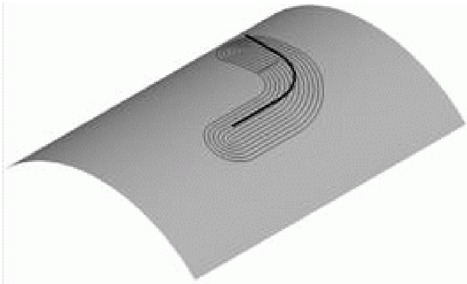
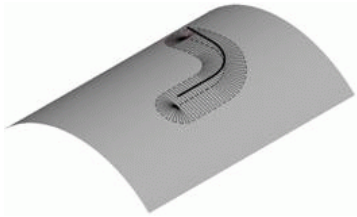
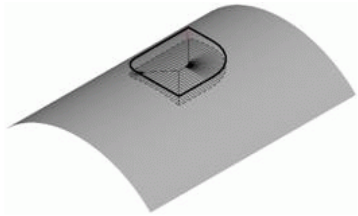
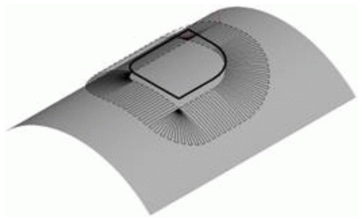
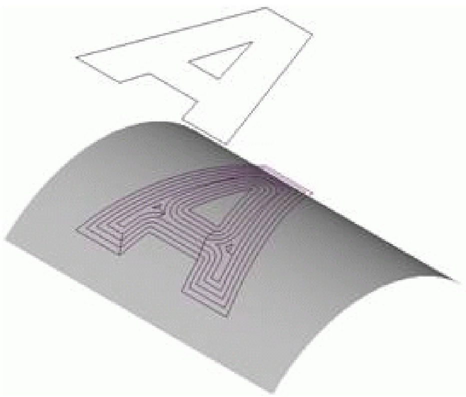
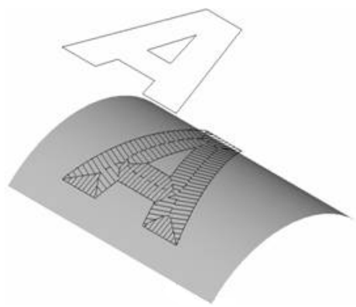
Mill machining

5.5.7.5 Job assignment for drive operations

In the finish and roughfinish and rough drive operations the tool transition rule in a plane is defined by job assignment. The task for the user is to create a drive area, in a similar way as creating an area for the engraving and pocketing operations. The drive area formed by the curves can be dynamically displayed in green in the graphic window.

There are two main types of tool movement in the drive operations: along or across the curves of the drive area. When machining along drive curves, the shape of the toolpath is created as an offset of the drive curve. When machining across the drive curve, the paths are created perpendicular to the drive curves. The side of the curve, on which the machining is performed is defined by the type of the drive curve. In the following chart, there are examples on how drive curves and their types affect the machining strategy.

Drive Curve type	Toolpath when machining along the drive curve	Toolpath when machining across the drive curve
Ditch		Not recommended

Ridge		
Cavity		
Ledge		
Outer curve – cavity, Inner curve – ledge		

If no curve is defined on the drive curves tab, then the drive area will be calculated by the system. The default method of area calculation depends on the type of drive operation.

In the default for the drive roughing operation, the drive curve will take the shape of the outer border of the workpiece. So, if for example the shape of the workpiece is a box, then the drive curve will be a rectangle, and machining will be performed inside the rectangle and either parallel or perpendicular to its sides.

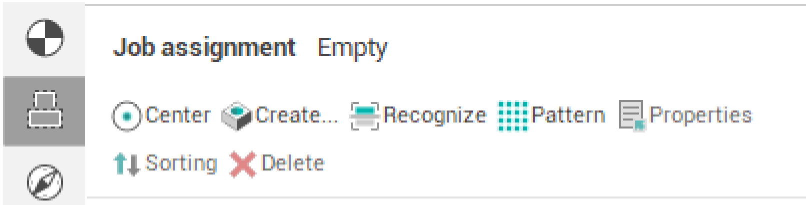
For the drive finishing operation, the default drive curve the system will use is the outer border of the model being machined, constructed using the method described in the Outer borders projection chapter.

For drive rest milling, by default the drive area is calculated so that machining is performed along the unfinished areas. For this purpose, first, the system detects the areas with residual material, and then the obtained area will be finished. Machining can be performed either along or across the unmachined areas.

See also:

Mill machining

5.5.7.6 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 Job Assignment dialog window's <Inverse> field 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>.

See also:

Mill machining

Defining holes by coordinates

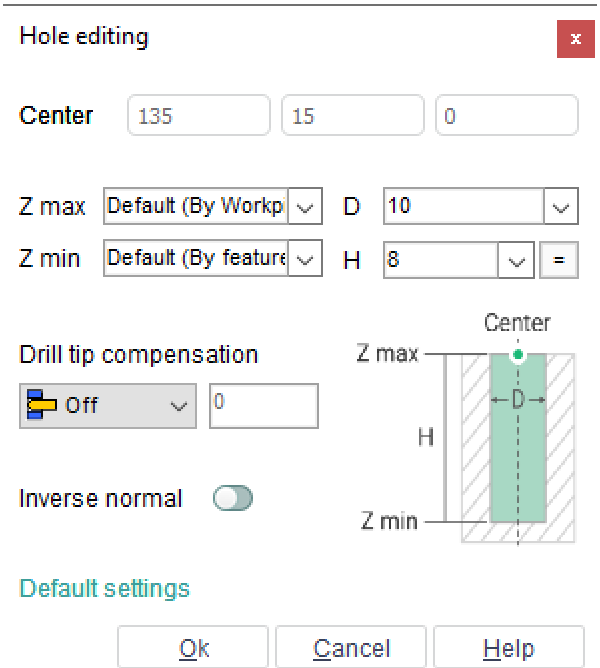
Defining holes by using a geometrical point object

Automatic hole recognition

Creating hole pattern

Defining holes by coordinates

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.



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.

<**Z max**> - top level mode:

- <By workpiece>;
- <By feature>;

- <Default> – parentheses will indicate the default value selected in the <**Default settings**> window;
- <Manual> – user set value manually.

<**Z min**> - bottom level mode:

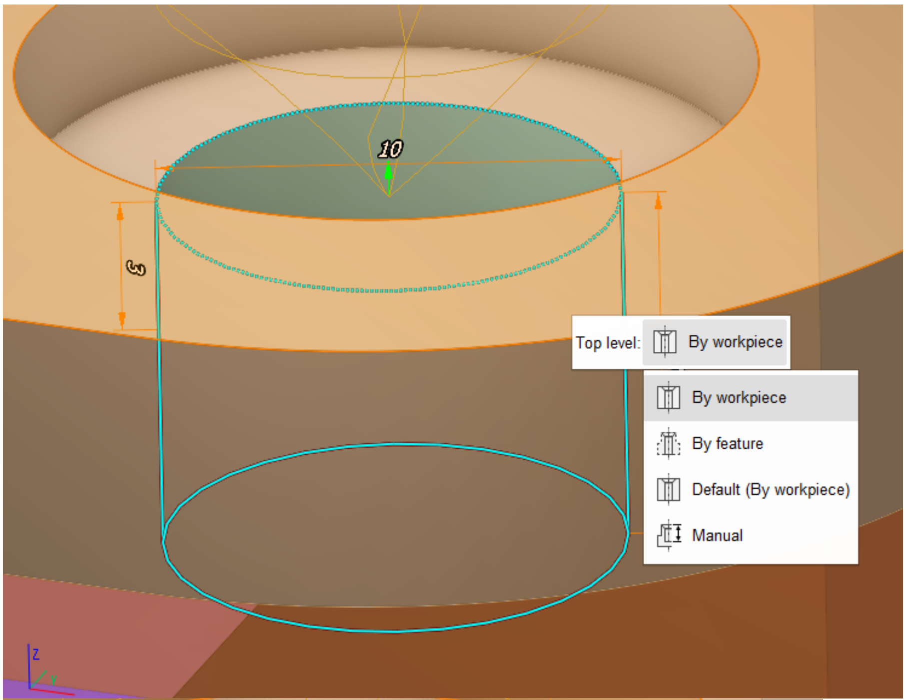
- <By feature>;
- <By model>;
- <Default> – parentheses will indicate the default value selected in the <**Default settings**> window;
- <Manual> – user set value manually.

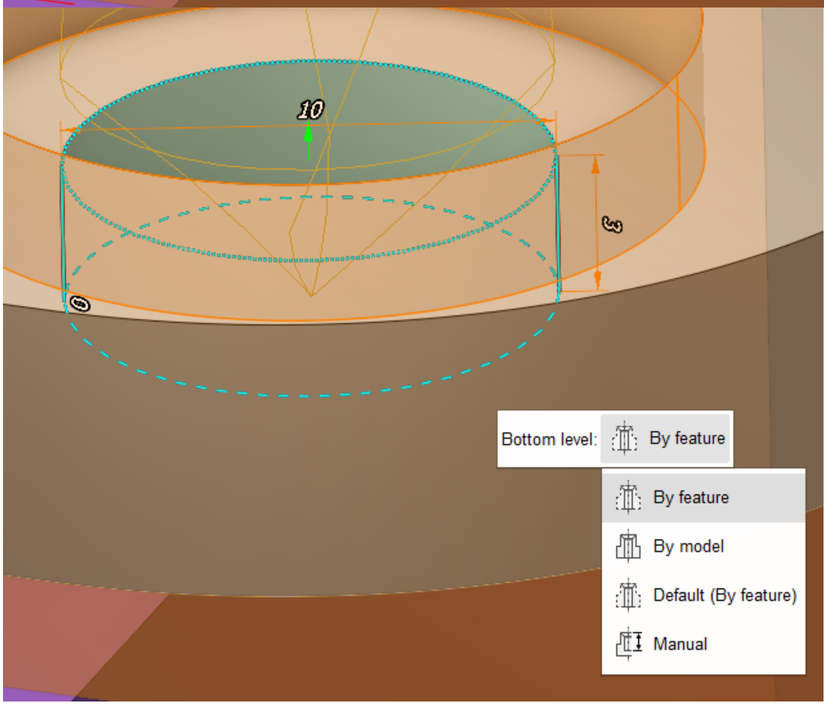
<Drill tip compensation> - choose the way the hole depth is specified:

- <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;
- <Auto> – hole depth is defined by the system based on whether the hole is blind or through.

<**Default settings**> button allows you to open window to set default values for <**Z max**> and <**Z min**>. 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:



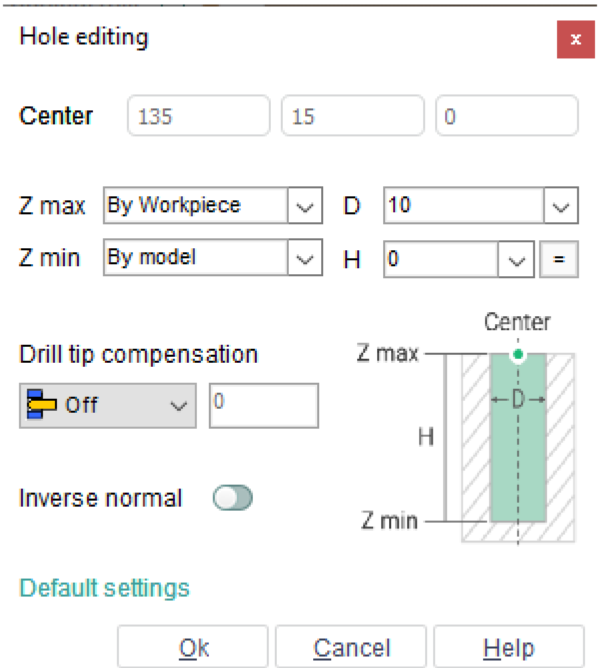


See also:

Job assignment for hole machining operation

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.



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.

See also:

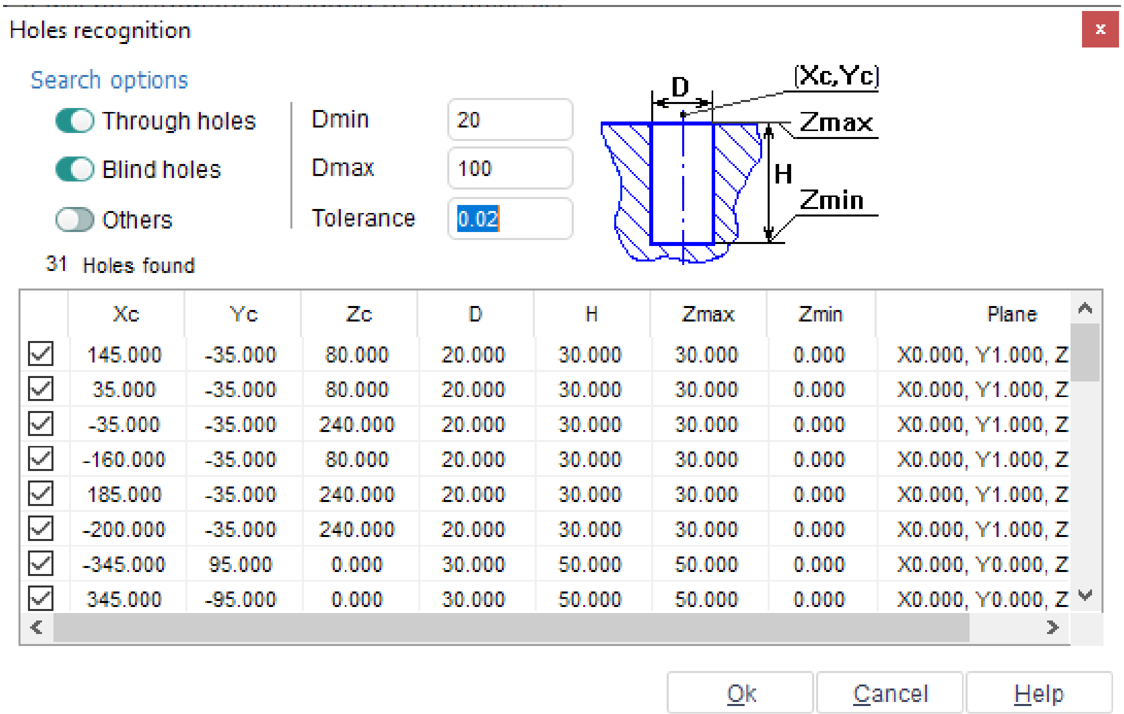
Job assignment for hole machining operation

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.



	Xc	Yc	Zc	D	H	Zmax	Zmin	Plane
☑	145.000	-35.000	80.000	20.000	30.000	30.000	0.000	X0.000, Y1.000, Z
☑	35.000	-35.000	80.000	20.000	30.000	30.000	0.000	X0.000, Y1.000, Z
☑	-35.000	-35.000	240.000	20.000	30.000	30.000	0.000	X0.000, Y1.000, Z
☑	-160.000	-35.000	80.000	20.000	30.000	30.000	0.000	X0.000, Y1.000, Z
☑	185.000	-35.000	240.000	20.000	30.000	30.000	0.000	X0.000, Y1.000, Z
☑	-200.000	-35.000	240.000	20.000	30.000	30.000	0.000	X0.000, Y1.000, Z
☑	-345.000	95.000	0.000	30.000	50.000	50.000	0.000	X0.000, Y0.000, Z
☑	345.000	-95.000	0.000	30.000	50.000	50.000	0.000	X0.000, Y0.000, Z

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.

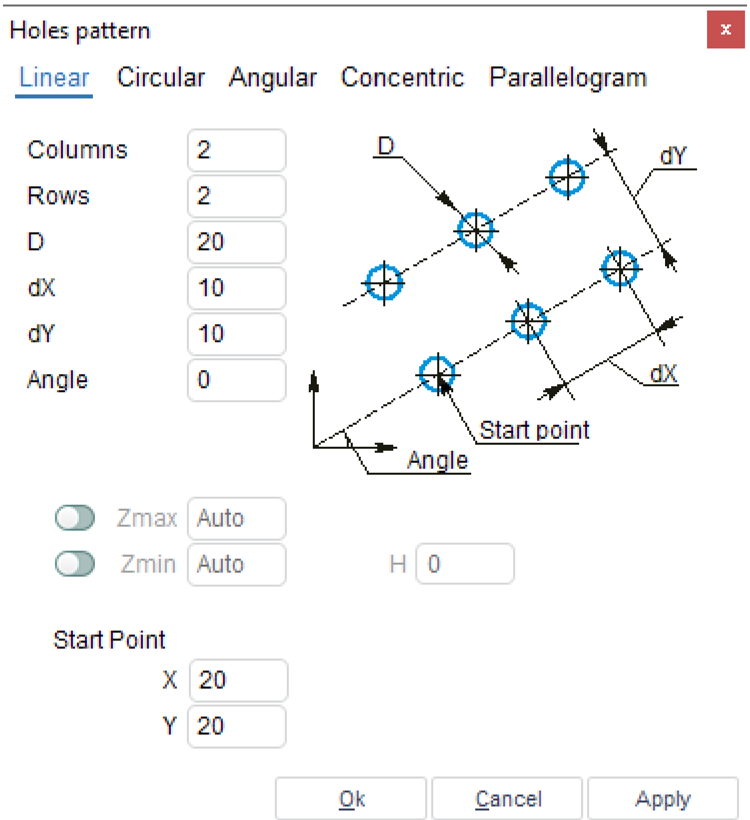
See also:

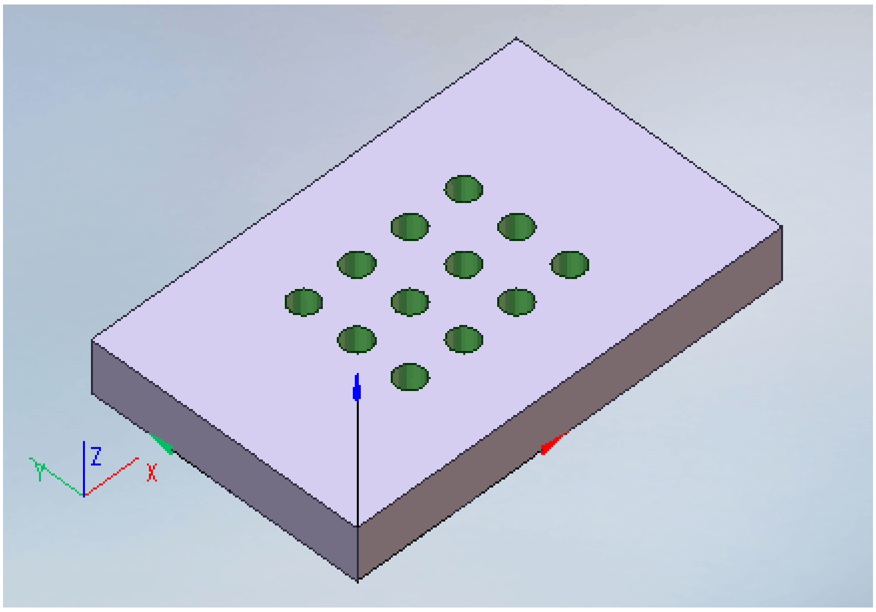
Job assignment for hole machining operation

Creating hole pattern

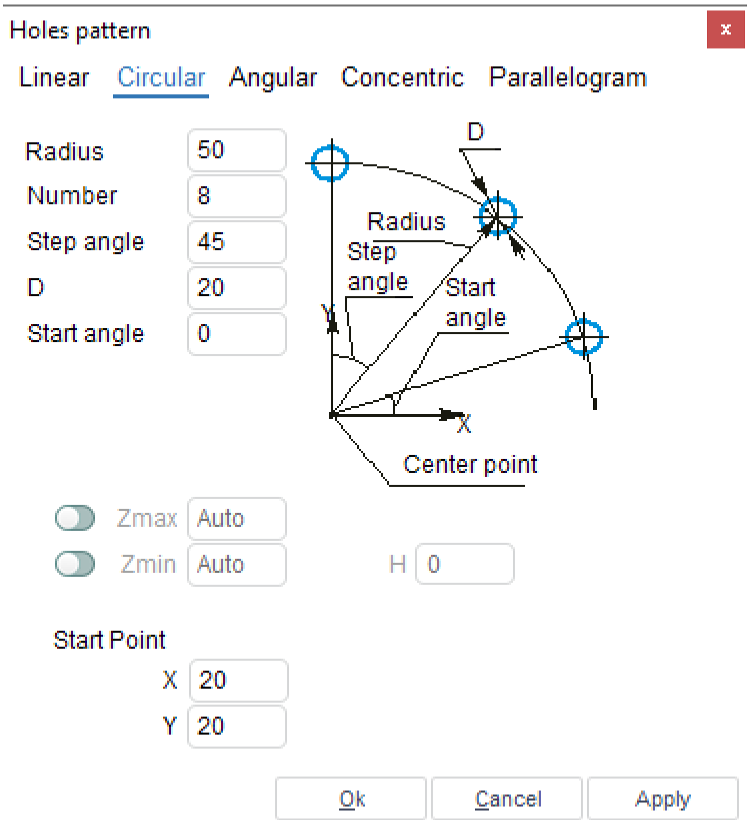
The system uses five types of pattern: <Linear>, <Circular>, <Angular>, <Concentric> and <Parallelogram>.

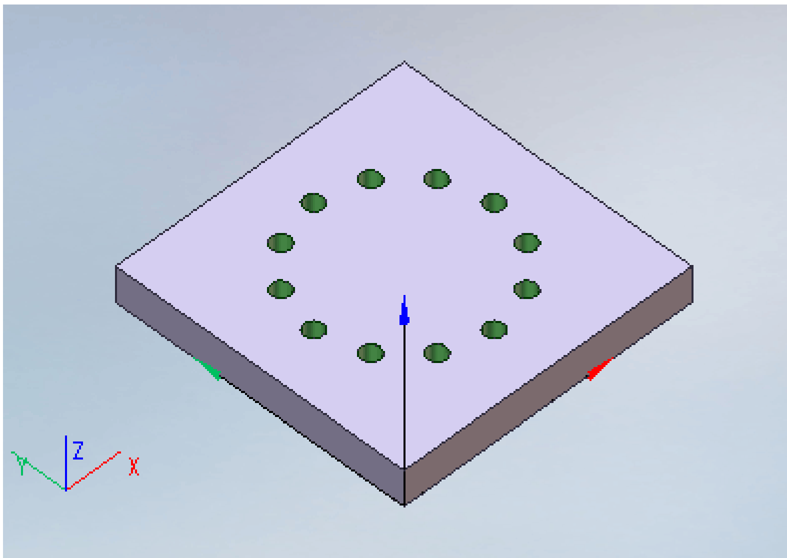
- On the <Linear> page user can create linear holes pattern:



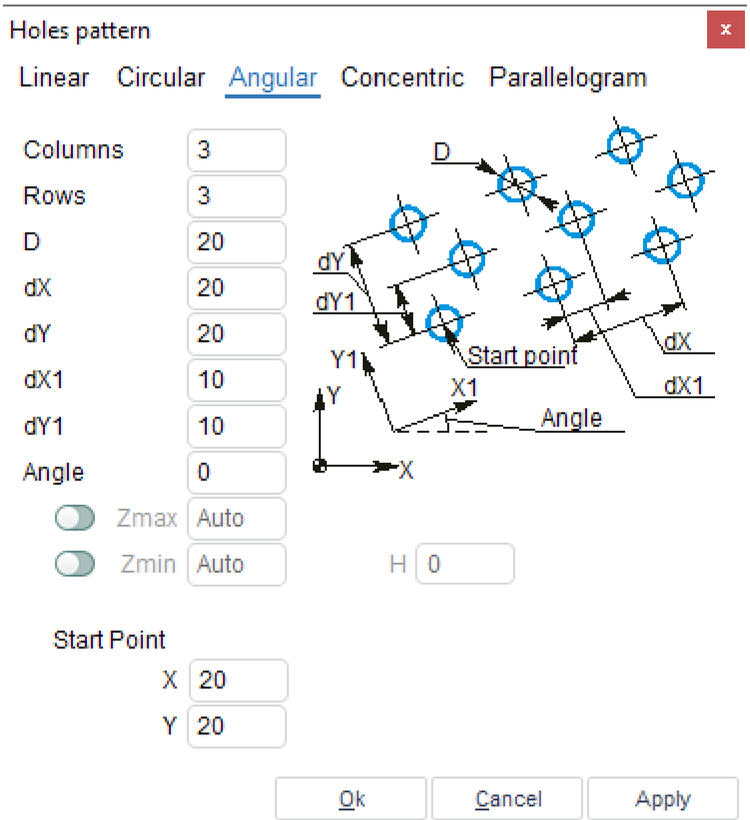


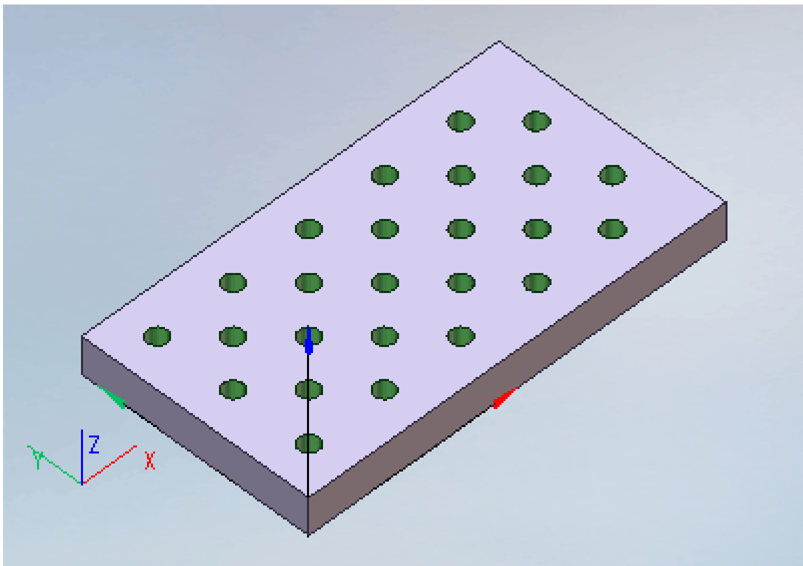
- On the <Circular> page user can create circular holes pattern:



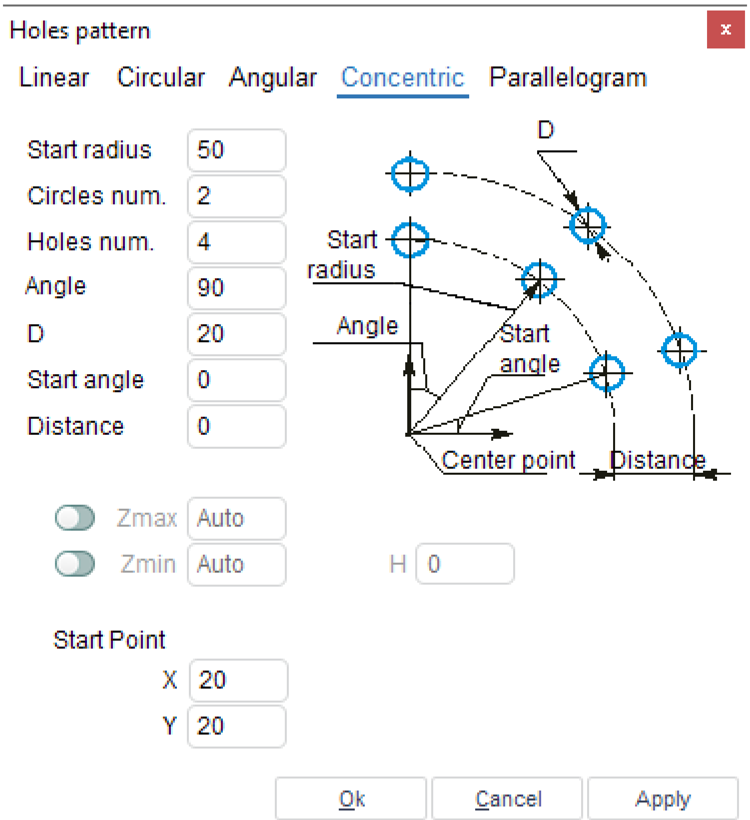


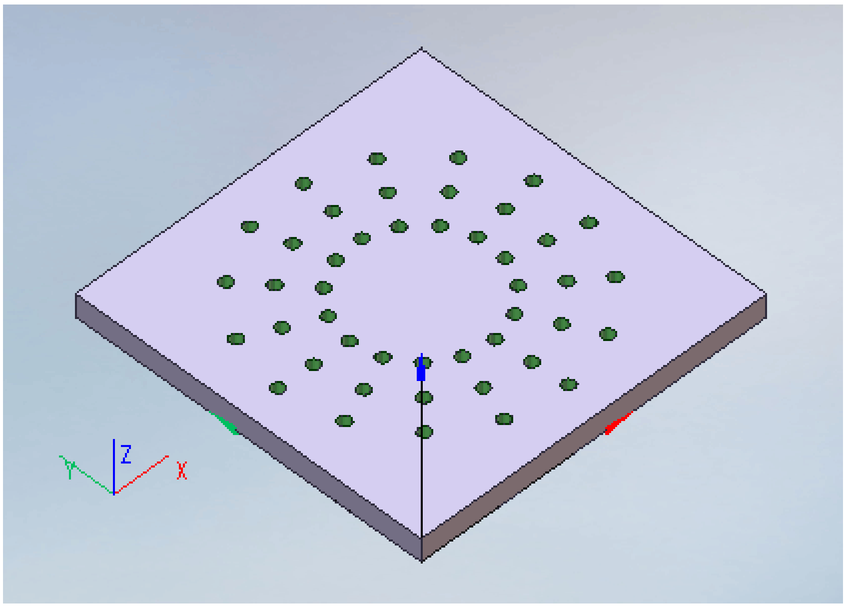
- On the <Angular> page user can create linear holes pattern:



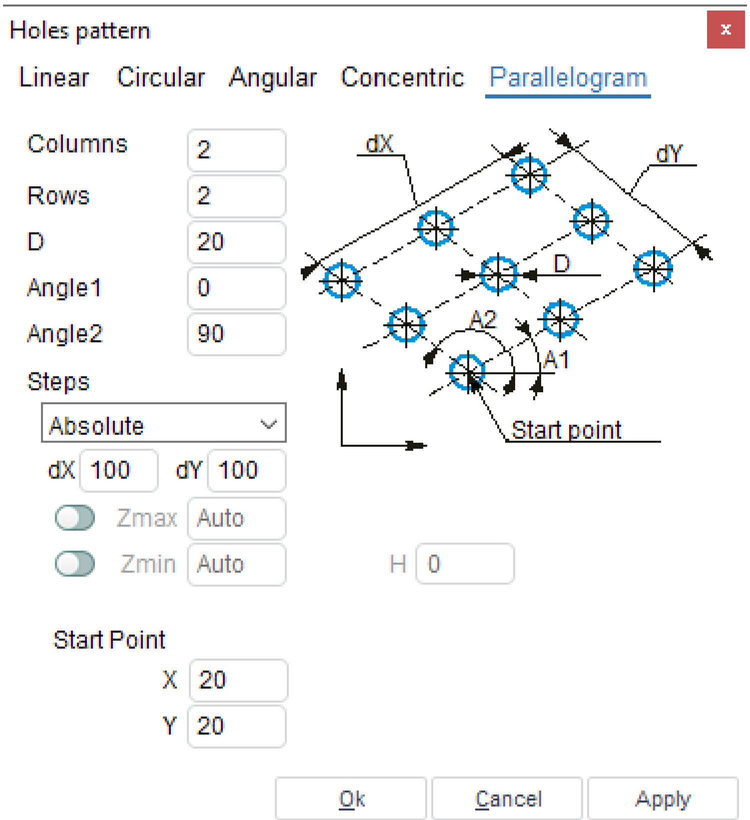


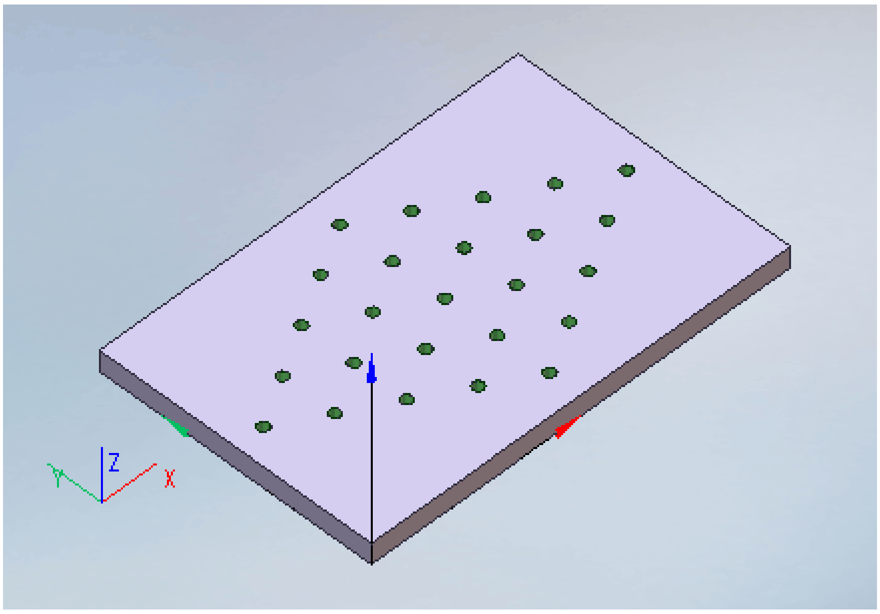
- On the <Concentric> page user can create concentric holes pattern:



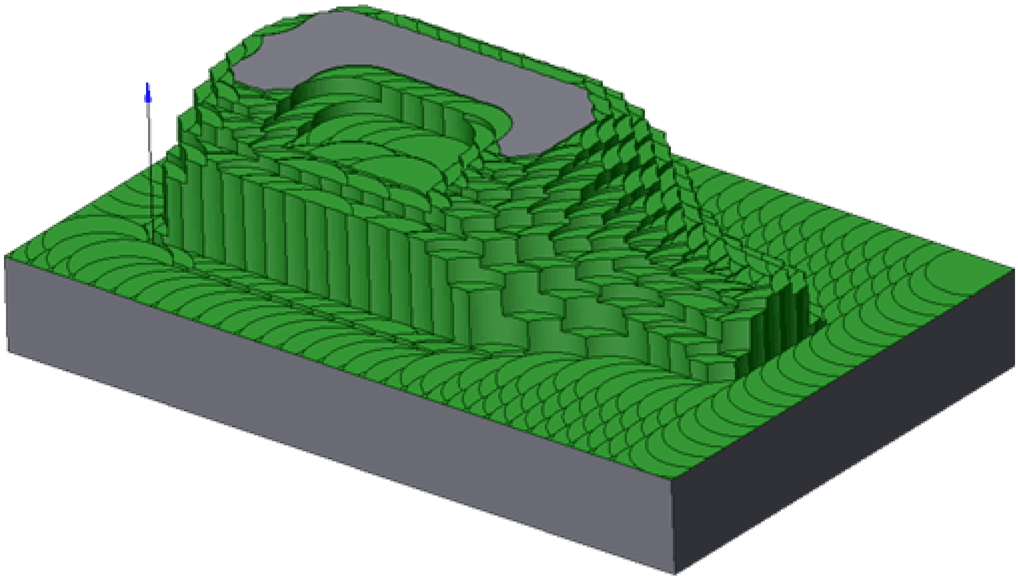


- On the <Parallelogram> page user can create parallelogram pattern:





Using the hole patterns together with automatic determination of hole levels allows someone perform roughing machining of the part by the axial plunging strategy.



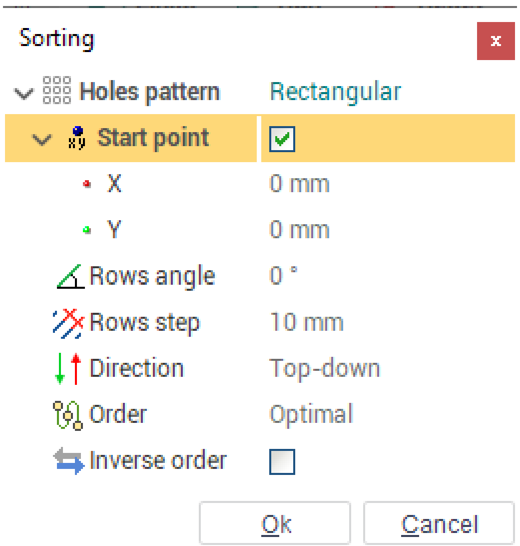
See also:

Job assignment for hole machining operation

Holes sorting

The holes sorting window allows you select one of the type of sorting using some parameters to sort holes list.

The system has four pattern of holes sorting: <**Rectangular**>, <**Circular (Rings)**>, <**Circular (Sectors)**>, <**Optimal**>.



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.

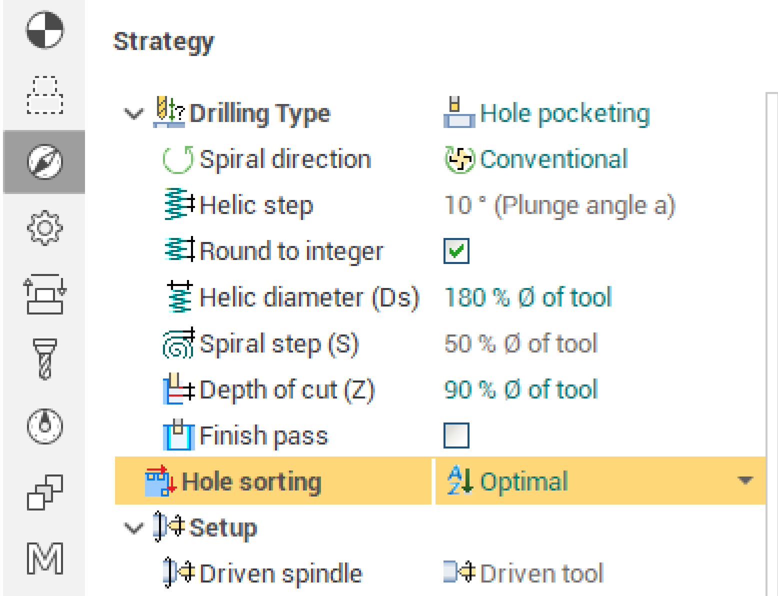
The pattern **<Circular (Rings)>** allows you to sort the holes along the rings located from specified center. This type of sorting has the following parameters:

- 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**>)

The pattern **<Circular (Sectors)>** allows you to sort the holes using sectors of a circle. This 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**>)

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.

5.5.7.7 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. SprutCAM X allows the construction of such models using 2D contours and automatically displays the volume model.

After the curve addition the window of curve parameters can be called by selecting the curve in graphic window and pressing right mouse button. The window is intended to view and edit the item parameters.

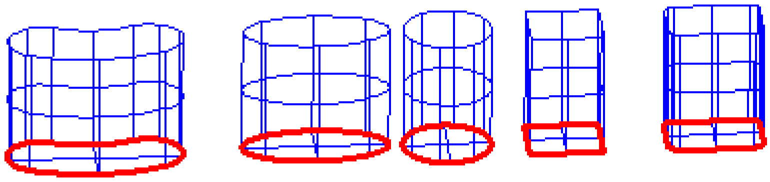
The window is closed automatically when the cursor leaves the icon.

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.

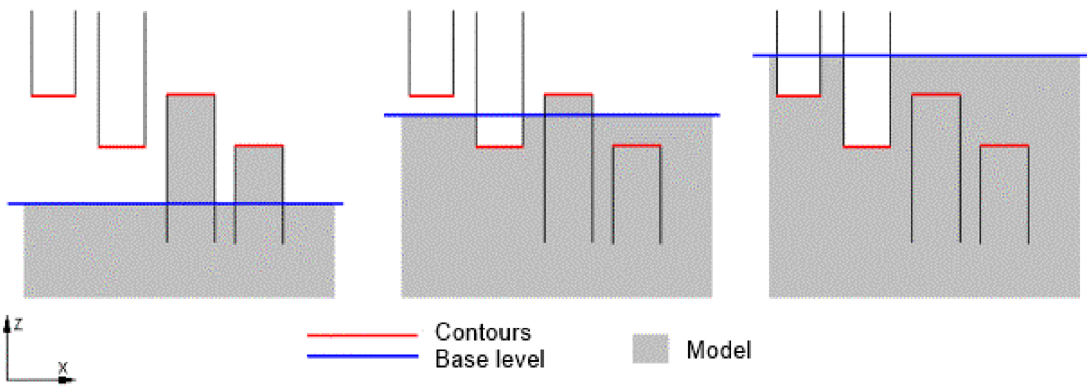
The '3D model' is constructed from 2D (flat) contours lying on different levels. There are two methods to add such areas: a 'cover' – "adds" material from the very bottom to the area level, and a 'hole' – removes the entire material from the top to the area level. This means that the 'side wall' for 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).

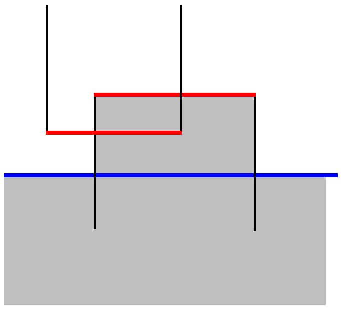


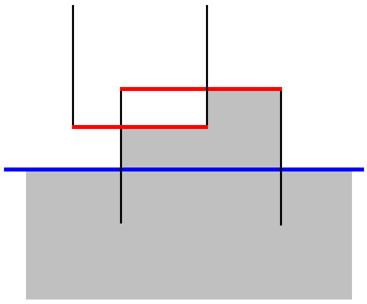
Of course, if hole-forms are located in empty space (or above the base level without other constructions), then they cannot press anything out, Likewise, cover-forms, which are located inside material (or below the base level without holes) cannot fill anything. Whereas, cover-forms located above the 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.

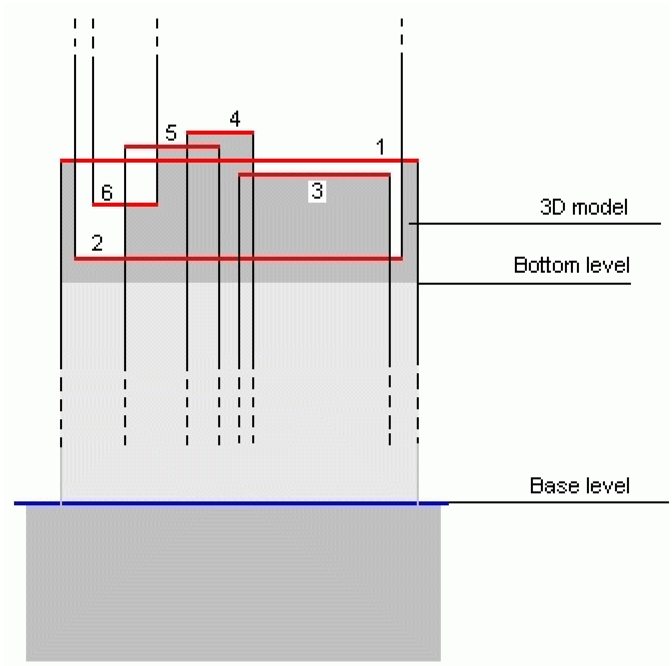




It is obvious that in the first case the hole did not reach the "sand" level, i.e. there was nothing to press out, and so the cover was untouched. In the second case, the cover was created, and then the hole pressed out a 'part' of the cover.

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.

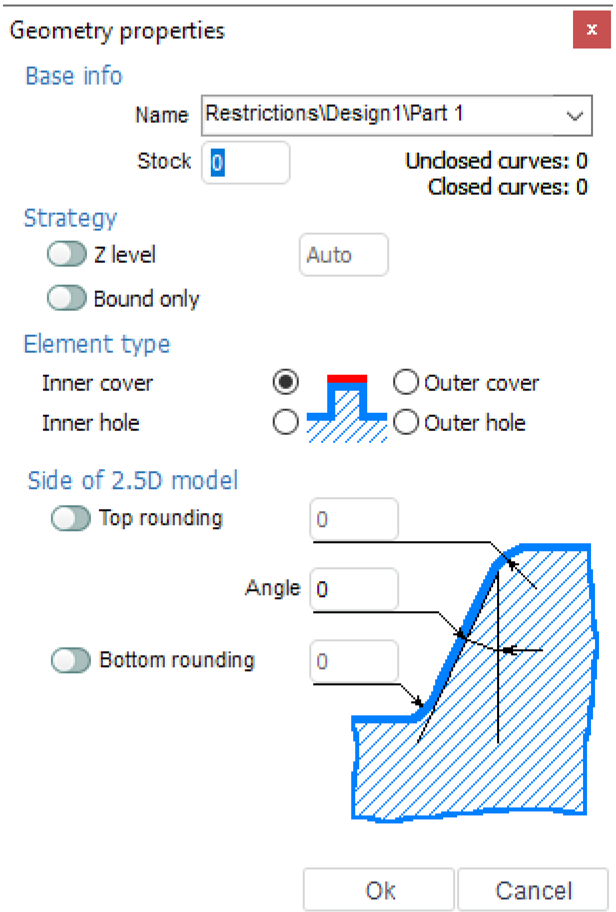
An example of such model is shown on the picture.



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.

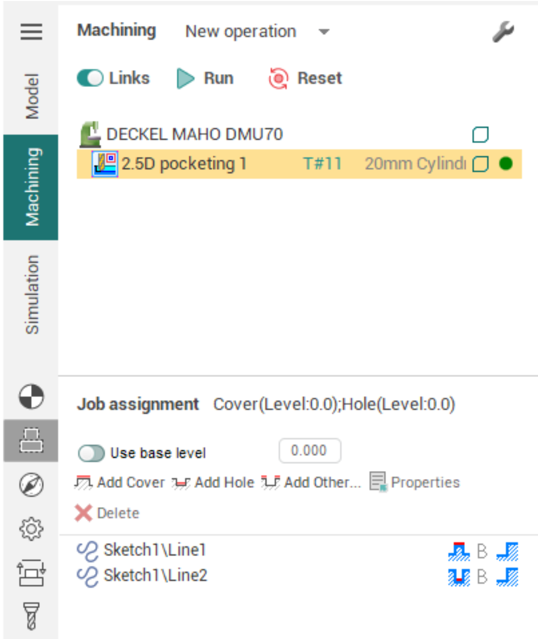
The properties window can be used to change the item parameters. The window can be opened by double click on the item or from the pop-up menu. The dialog is shown below:



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.

1. In the "2D Geometry" mode construct two rectangular intersecting contours;
2. Switch to the "Machining" tab and select the "2.5D area pocketing" operation;
3. 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;
4. 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;
5. 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;
6. Assign its value equal to -20.

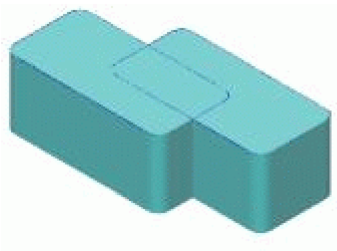


1. Put a tick in the Preview box. The button must be pressed. There should be a model similar to that shown in this picture:

We can represent parameters and their conditions in a chart:

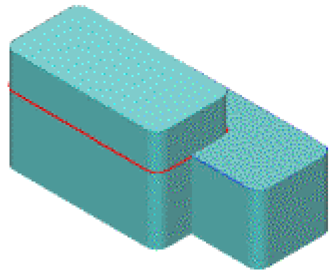
1. Contour 1	cover	level 0
2. Contour 2	cover	level 0

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;



1. 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.

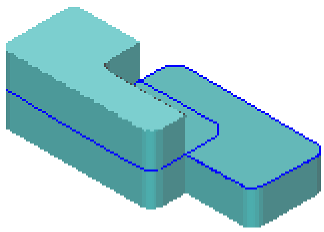
1. Contour 1	Cover	level 10
2. Contour 2	Cover	level 0



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

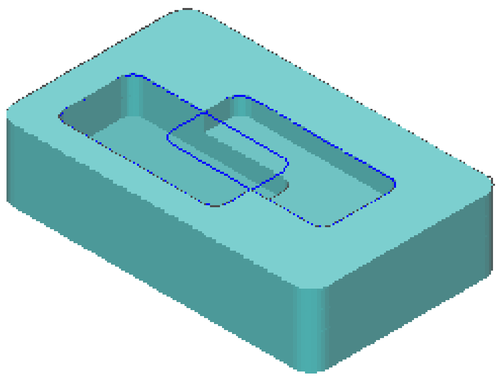
The result is shown on the picture



1. 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>.

1. Activate the Base level mode

1. Make the base level 0;
2. Create a list of contours with these parameters:

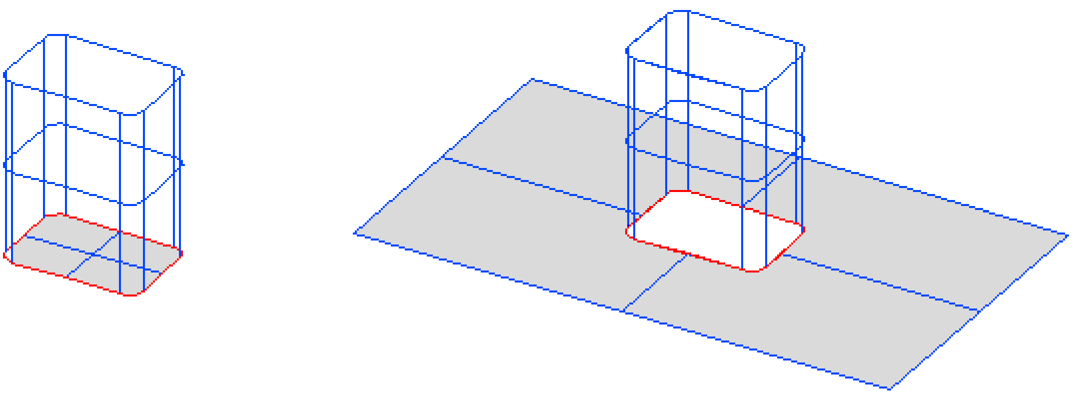
1. Contour 3	hole	level -10
2. Contour 1	cover	level 0
3. Contour 2	hole	level -5

The result should be similar to the one shown.

? Неизвестное вложение

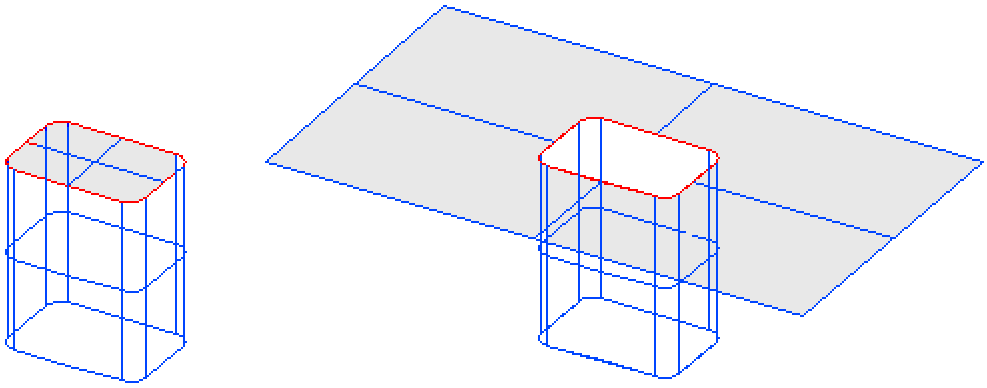
Model example with use <Outside> parameter.

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

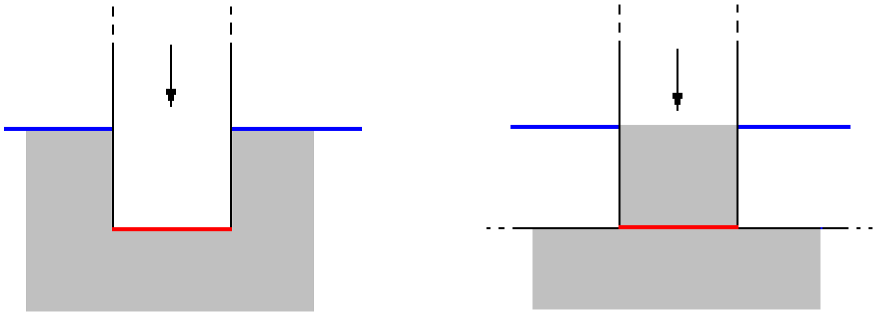
Forms for pressing out holes



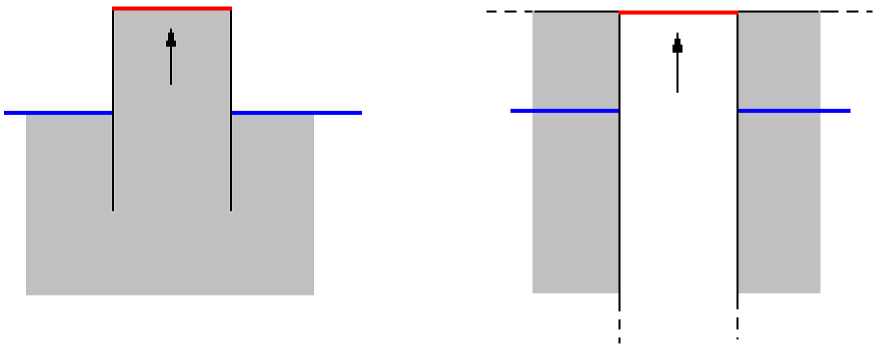
Inside Outside

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>.



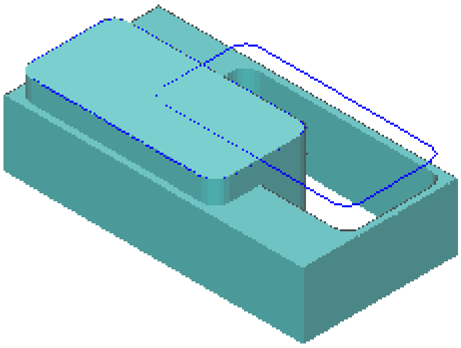
Cover pulling is shown in this picture. Left is <Inside> parameter, right is <Outside>.

Here is an example of using <Outside> parameter.

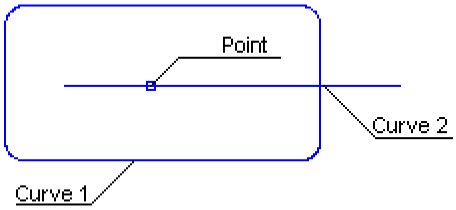
1. Create a list of contours with parameters defined as below;

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.



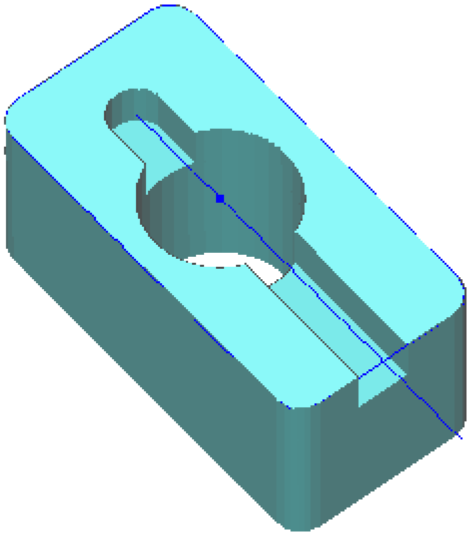
1. In 2D geometry create one closed contour, one open and one point, similar to the picture below:



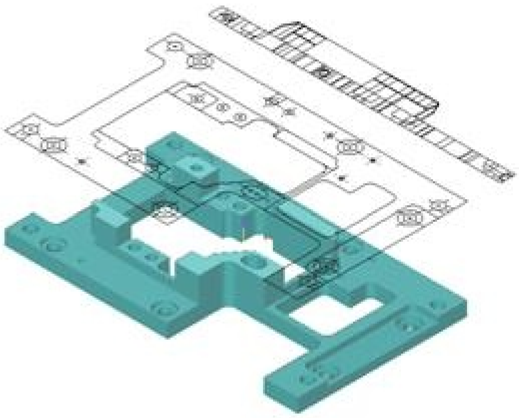
1. Create a list of elements with the parameters defined below:

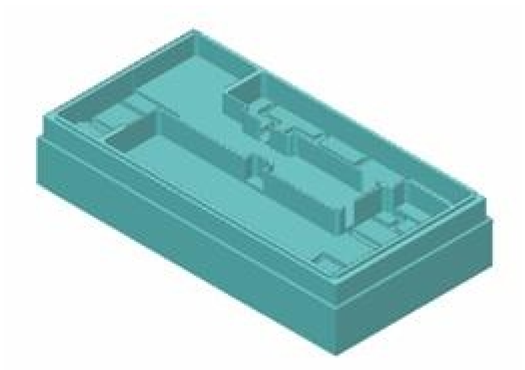
1. Contour 1	Cover	level 0	
2. Point 1	Hole	level -20	additional stock 8
3. Contour 2	Hole	level -5	additional stock 3

The result should be similar to the picture below:



Examples of visual model construction for 2.5D machining.





See also:

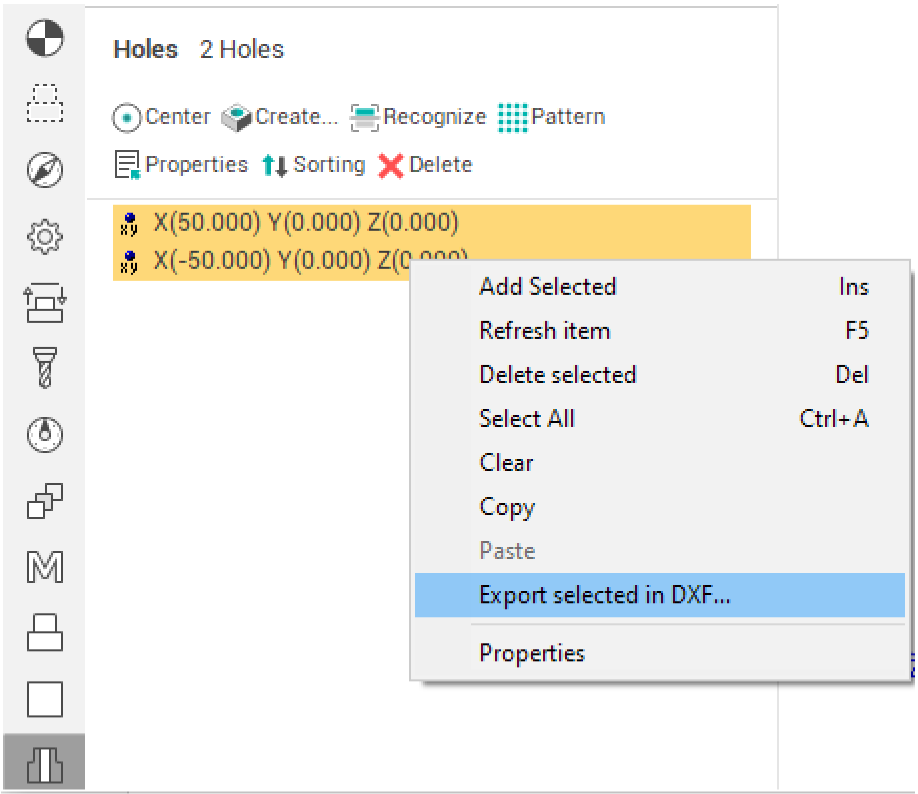
Mill machining

5.5.7.8 Tips & Tricks in the roughing waterline and pocketing operation

In the waterline roughingwaterline roughing and pocketing operation, the holes list defines the positions where vertical tool plunging is allowed because they are already drilled. The holes list is used for these operations by setting the plunge method to < Through drill point > in the toolpath window. If, when the toolpath is being calculated, the tool cannot approach an area from outside, then the system searches for an appropriate point in the holes list and if an appropriate hole is found, then it will be used for the vertical tool plunge. If a suitable hole isn't found, then one will be created automatically and added to the holes list.

For the fast creation of an operation that will provide preliminary drilling for tool plunging, it is necessary that when the hole machining operation is being created, the user select the pocketing or waterline roughing operation as the prototype. By doing this, all of the holes of the prototype operation will be copied to the newly created operation. And vice verse, to use the holes obtained for tool plunging, the operation can be defined as a prototype for the waterline roughing or pocketing operations.

Select holes in the holes list and use the context menu < Export selected in DXF > item to export the list into the DXF-file.



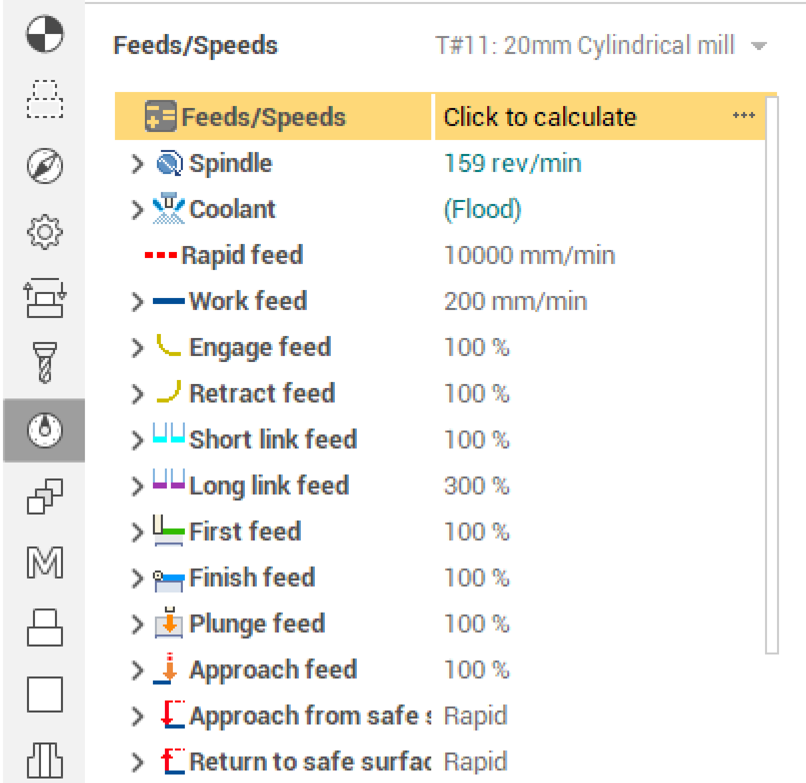
See also:

Mill machining

5.5.7.9 Cutting conditions of mill operations

The definition of the cutting modes for the current operation can be performed in the <Operation parameters> window on the <Feedrate> page. The window opens by pressing the <Parameters>

button. 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.

- <Rotations per minute>. Defines the spindle rotation speed in rotations per minute. The parameter will be recalculated when altering the cutting speed or the diameter of the tool;
- <Cutting speed>. Defines the spindle rotation speed in meters or feet per minute, depending on the selected measurement units. The parameter will be recalculated if the rotation speed or the diameter of the tool is altered.
- <**Spindle rotation range**>. Defines the range of the spindle rotation. Use this parameter only for the old machines, that have the spindle speed range switcher.

The user can also define the federates for various areas of the toolpath. The number of feed type options in the drop down menu will vary depending on the current operation type. Different operation types will have different options available.

- <Rapid feed> is mainly used for transitions at the safe plane. 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. For most CNC controls, the value of the rapid feed is not used in the NC program, but this value is always used by the system to calculate the machining time.
- <Work feed> defines the feed at which the work feed will be performed. This is the main feed value. All other feed can be defined as a percentage of this value.

- <Engage feed> defines the feed at which the approach to the machined object is made.
- <Retract feed> defines the feed at which the retraction from the object being machined.
- <**Short link feed>** defines the feed of link if its length is less than Short link max distance.
- <**Long link feed**> defines the feed of link if its length is more than Short link max distance.
- <First pass feed> defines the feed that is used for the initial cut of the tool in the workpiece.
- <Finish pass feed> for roughing operations this defines the feed used when cutting along a surface.
- <Plunge feed> defines the feed at which the system performs a vertical (Z) plunge to the next machining level.
- <Approach feed> defines the feed at which the approach move of a tool.
- <**Approach from safe surface feed**> defines the feed at which the tool moves from the safe plane to the feed level.
- <**Return to safe surface feed**> defines the feed at which the tool moves from the part to the safe plane.
- <**Transition on safe surface feed**> defines the feed at which the tool moves along a safe plane.
- <Measurement units>. A feed can be assigned either in millimeters per minute, in millimeters per revolution of the spindle or in millimeter per tooth. if mm/tooth is set then Feed is calculated with using of the current RPM and teeth number. Teeth number is set on the "toolHTML._352" page.

- <Coolant>. If coolant is enabled then the appropriate command will be added to the NC program to control the switching on/off of the coolant.

If the <Cut feed> option is selected, then the feed value can be defined in the appropriate field. Using this option means that the feed does not depend on the tool movement direction.

If the <Smart cut feed> option is selected, then the feed will be calculated for every move by taking into consideration the parameters entered by the user for the work feed direction (feed when moving vertically up, horizontally, vertically down) and the angle between the movement direction and the vertical.

It is possible to define federates as a percentage of the work feed by ticking in the <% of work feed> box. When selected, the feed type (<Cut feed> / <Smart cut feed>) and <Measurement units> will be set as they were for the work feed, and all other feed values will be calculated as a percentage of the work feed value.

It is also possible to call calculator for speeds and feeds.

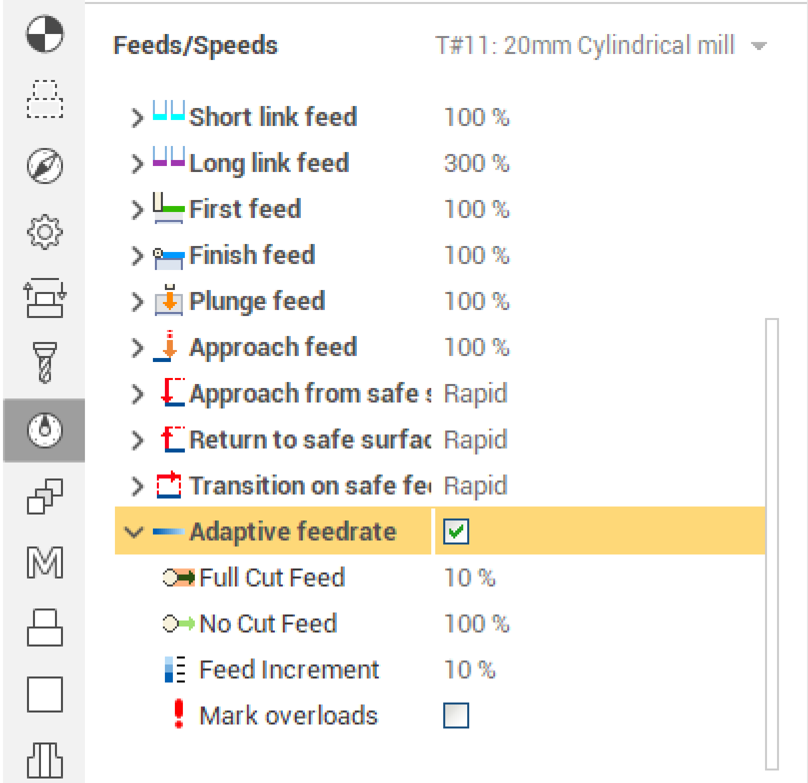
For roughing waterline operation Adaptive feedrate feature available.

See also:

Mill machining

Adaptive feedrate

Adaptive feedrate feature allows to change the feed of tool movements dynamically according to the real load on the tool (amount of the workpiece removing material).

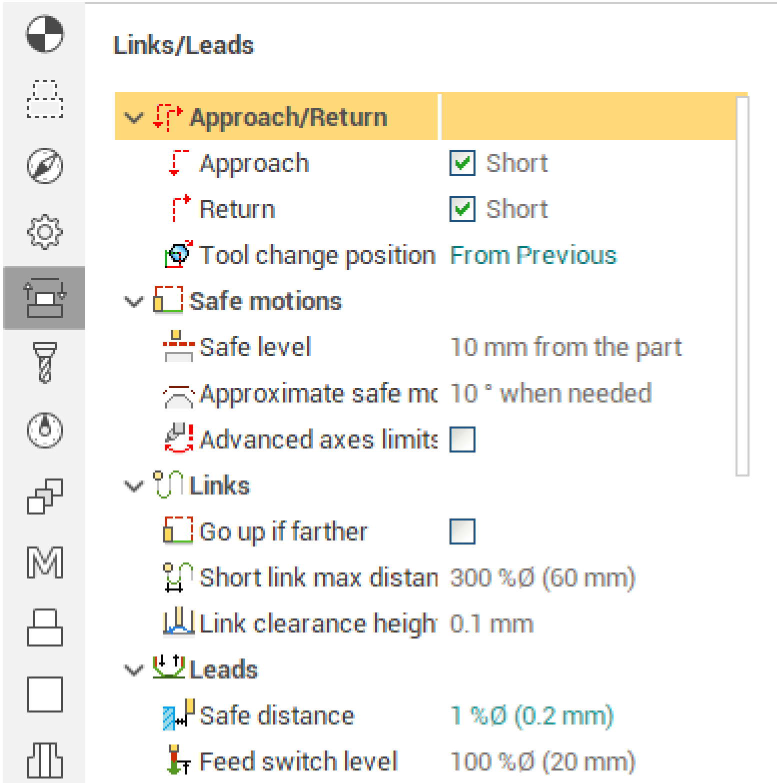


See also:

Mill machining

5.5.7.10 Approach, retraction, and plunge methods

Use the <**Links/Leads**> dialog to specify parameters that define tool approach to cutting passes and tool retraction from cutting end point. The actual dialog window is changed according to the current operation.



See also:

Mill machining

Approach and retraction of mill operations

Plunge methods of mill operations

Compound approaches and retractions of mill operations

Controlling tool's transition between operations

Approach and retraction of mill operations

To provide more flexibility and control at the start and end of a toolpath, there are several options for tool approach and retraction in the system. Approach moves are added at the beginning of every toolpath, retraction moves are added at the end. Feeds that differ from the work feed can be applied to these moves. If an operation uses cutter radius compensation, then it will be activated at the beginning of an approach move and canceled at the start of the retraction move.

The <Approach> of a tool is performed as follows:

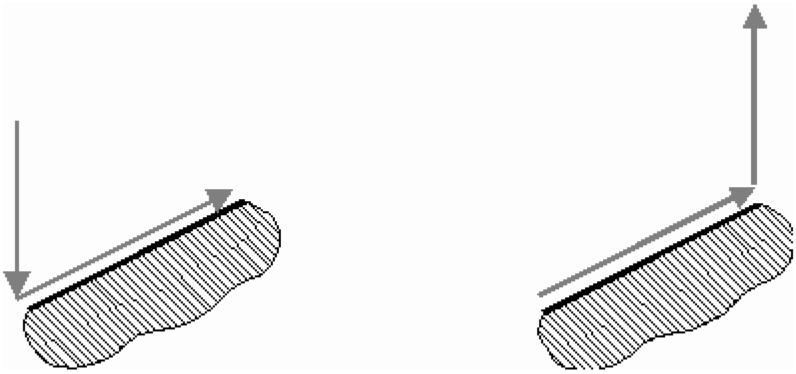
- Tool approach to the plunge point at the safe plane for the operation.
- The Z-axis rapids to the safe level or safe distance before beginning the approach move (depending on settings). The safe level is measured from the Z top level in the current coordinate system. The safe distance is the Z distance above the pass depth.
- The Z-axis feeds down to the beginning of the defined approach.
- The user-defined approach is applied to the machined model.
- Work pass starts.

<Retraction> of the tool is performed as follows:

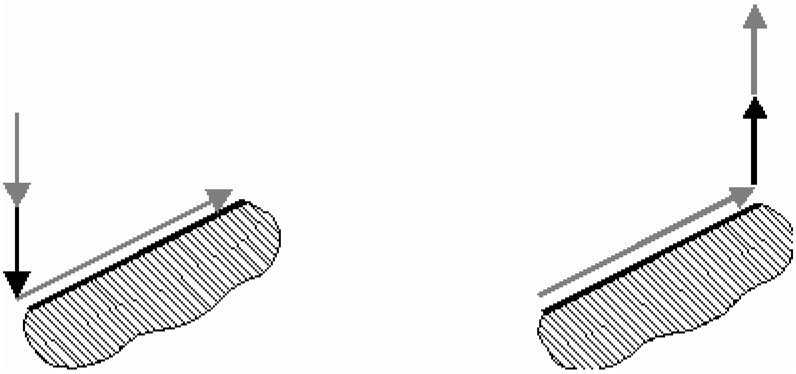
- End of the work pass.
- Retraction of the defined type at the work feed.
- Vertical rising of the tool to the safe plane at rapid feed.

In the system there are the following approach/retraction methods available:

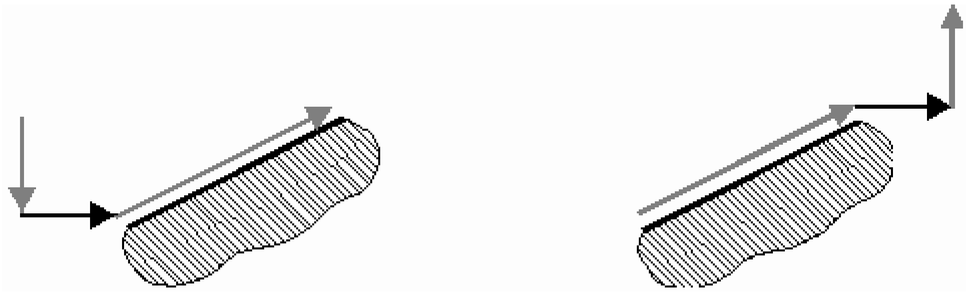
- <Without approach>. The approach and retraction toolpath parts is not generated.



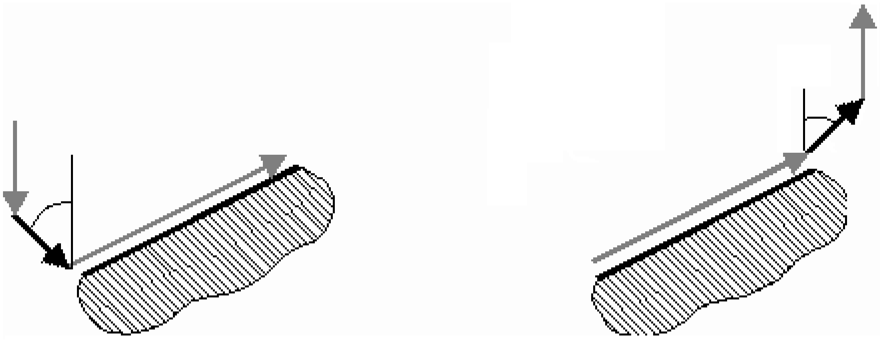
- <Vertical>. The approach is performed vertically to the first point of the work pass. The retraction – vertical from the last point of the work pass.



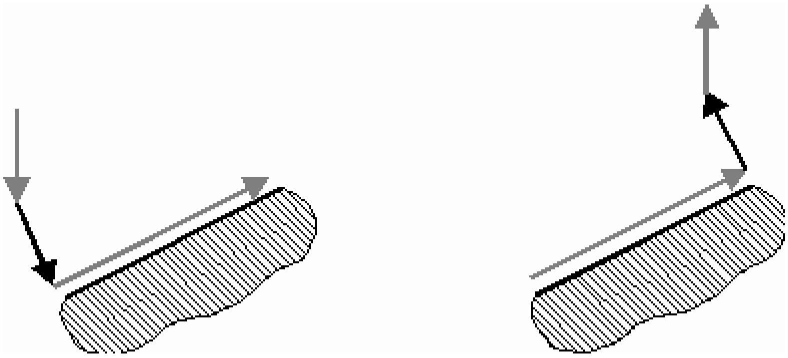
- <Horizontal>. The approach is performed horizontal to the first point of the work pass. The retraction – horizontal from the last point of the work pass.



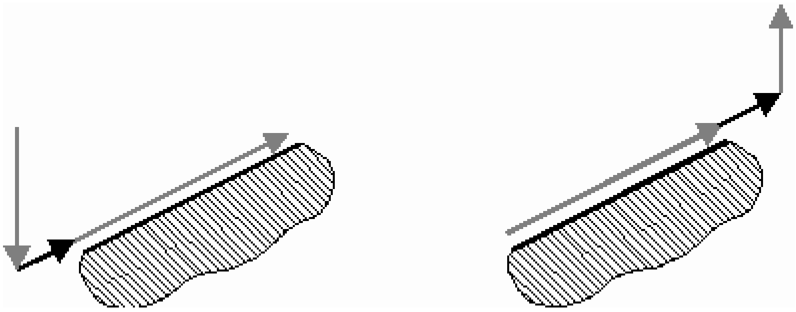
- <Angle to Z-axis>. The approach is performed by angle to Z-axis to the first point of the work pass. The retraction – by angle to Z-axis from the last point of the work pass.



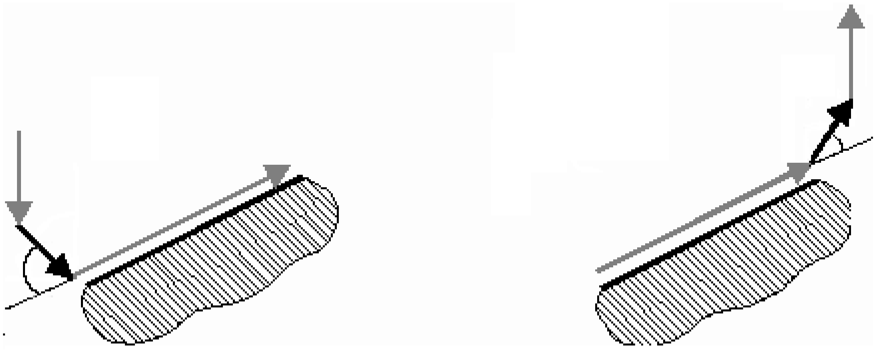
- <By normal>. The approach is performed along the normal to surface at the first point of the work pass, the retraction – from the last point



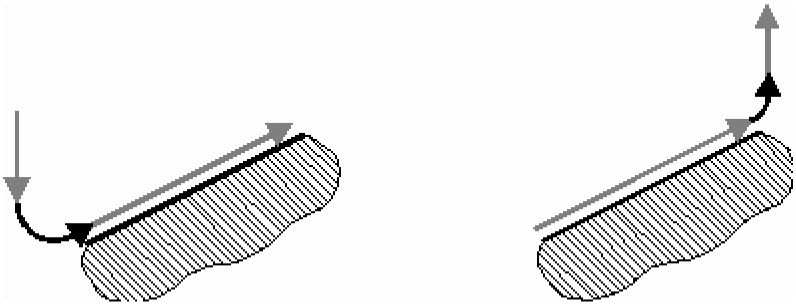
- <By tangent>. The approach is performed tangentially to the first machining point and the retraction tangentially from the last point



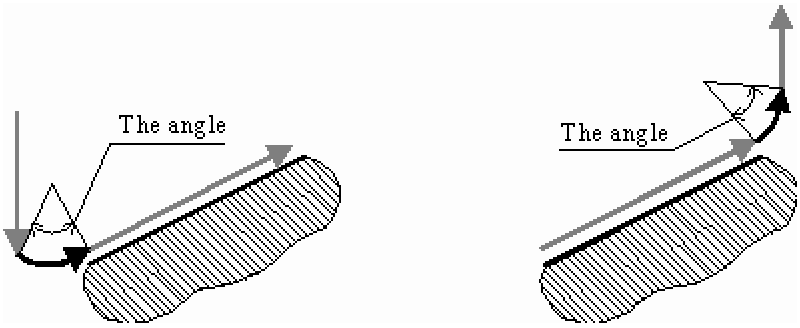
- <Angle to tangent>. The approach is performed by angle to tangent to the first machining point, and the retraction by angle to tangent from the last point



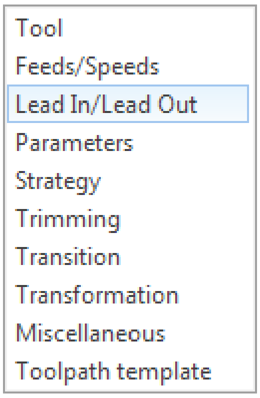
- <By arc>. The system adds an arc to the first point on the curve using the defined radius. The arc lies in the vertical plane and is tangent with the first toolpath applied to the contour. The angle of the curve is defined by user. The tool plunge is performed at the vertical end of the arc, and then moves along the arc and then onto the work pass. Retraction is performed in reverse.



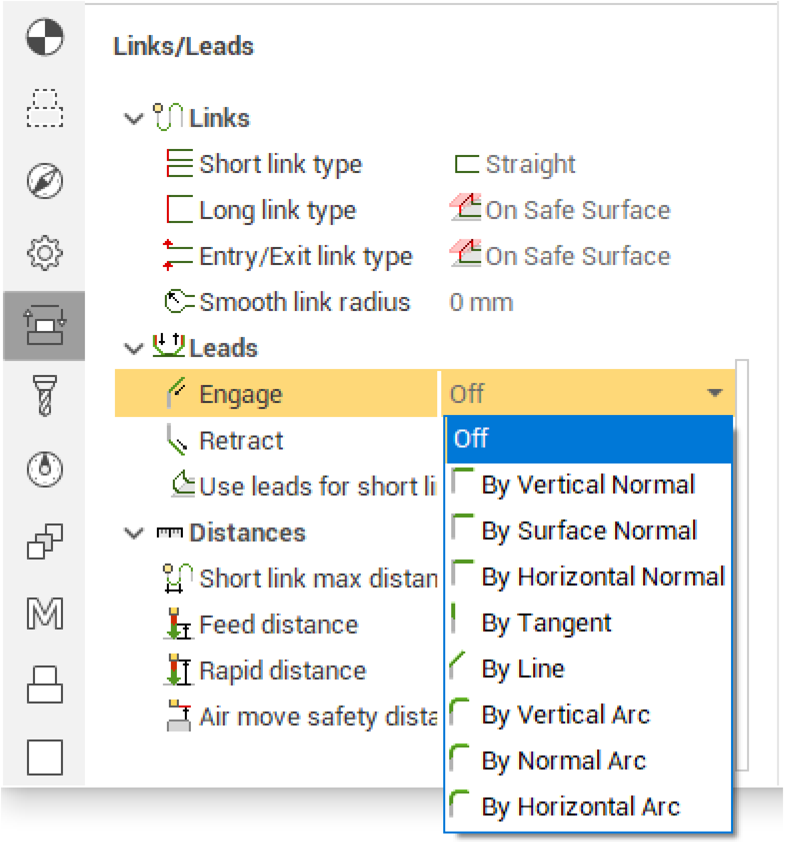
- <By arc (Angle)>. The system adds an arc to the first point on the curve using the defined radius. The arc lies in the vertical plane and is tangent with the first toolpath applied to the contour. The angle of the curve is calculated so that the tangent on the other side of the arc is vertical. The tool plunge is performed at the vertical end of the arc, and then moves along the arc and then onto the work pass. Retraction is performed in reverse.



The definitions of the geometrical parameters for approach moves are made in the <Lead In/Lead Out> page in the <Operation parameters> window.



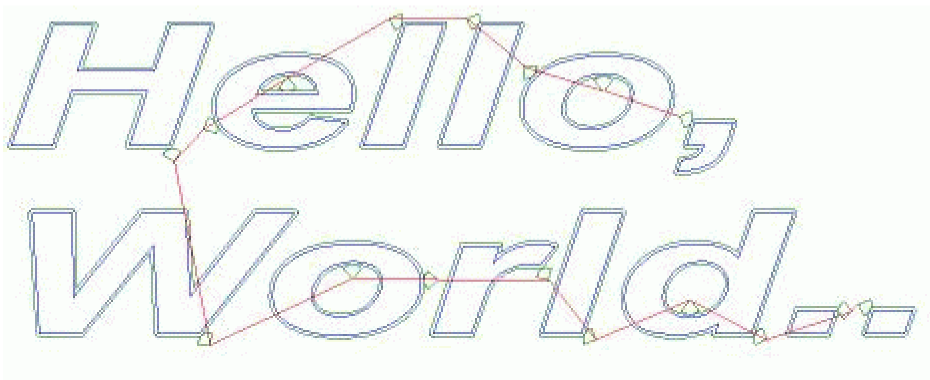
The page opens by pressing the <Parameters> button. In the schematic pictures the rapid toolpath is marked in red, work feeds in green. The required type of approach and retraction moves for an operation can be selected in this view. Depending on which type of approach/retraction is selected, the system updates the graphic and the fields for the parameters of the selected type. Dimensions of moves can be specified either in the current units of the system (mm or inches), or a percentage of the tool diameter. Way to specify the value switches buttons next to the input field.



Quick-clicking the left mouse button on the button switches the way to the next immediately. If when you press you hold the button release on the half-second, then a context menu appears, where you can select the desired method of setting value.

By using the <Safe level/Safe dist> options, the user can define the method for the tool to change from rapid to feed. The safe level is defined by the Z datum of the current coordinate system. The safe distance is defined relative to the approach of the defined type at the work pass height.

When machining 2D curves, the system performs automatic selection of the approach point. If the approach point is not defined, then approach will be performed at an outer corner or on the longest section as shown in the picture below.



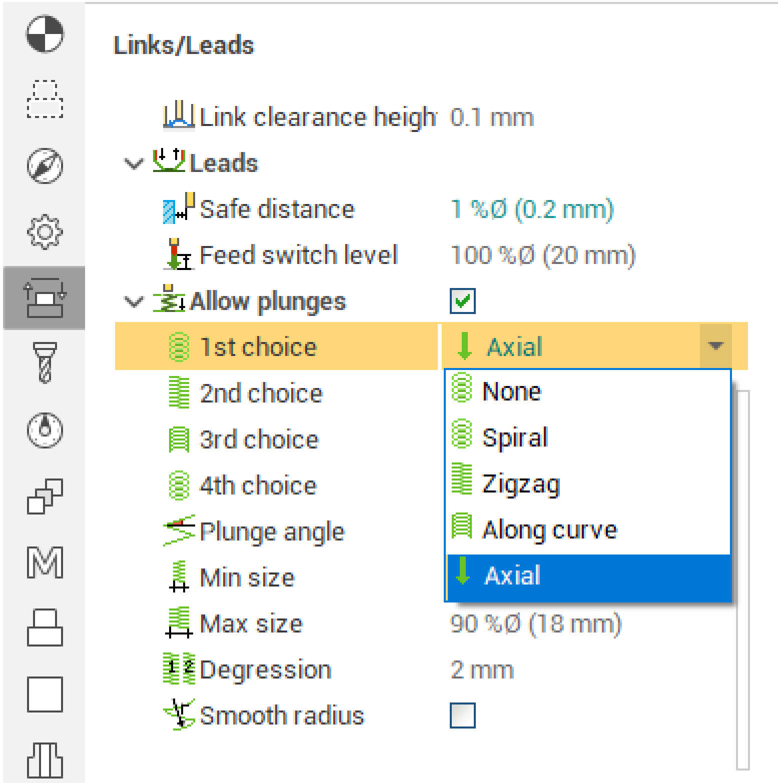
See also:

Approach, retraction, and plunge methods

Plunge methods of mill operations

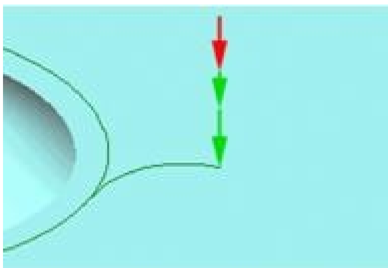
When it is impossible to approach the machining area from the outside, the system automatically generates a plunge movement to the first point of the work pass. A plunge is a toolpath section along the Z axis within the workpiece body. The <Plunge> is performed as follows:

- 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.

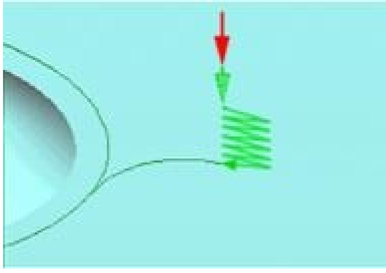


The following plunge methods are available in the system:

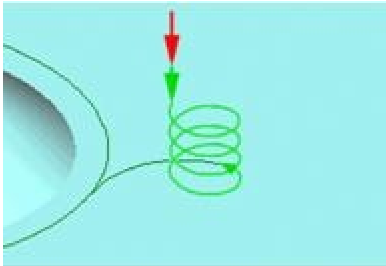
- <Axial>. Performed along the vertical straight to the first point of approach.



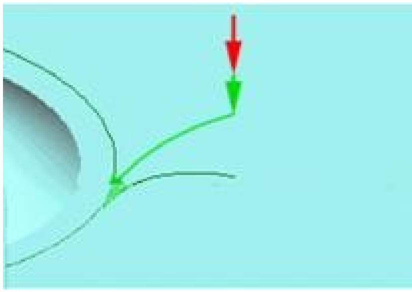
- <Zigzag>. The tool performs reciprocal movements along a straight section, connected to the first approach point. The length of the section is a user-defined option.



- <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.

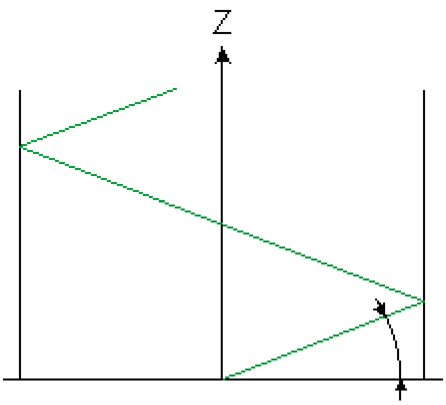


<Along approach curve>. The tool plunges along the approach curve. The plunge move is completed at the end of the approach move.

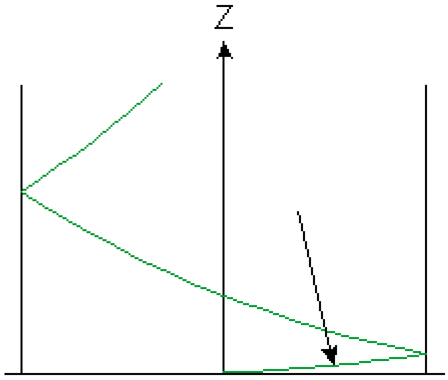


For the last three plunge types (<Zigzag>, <By spiral> and <Along approach curve>) the system applies the selected Z movement rule. Two motion types are available: <Angular> and <Radial>:

- <Angular>. The speed of the vertical movement of the tool is constant. The parameter is defined as the angle between the XY plane and the vertical tool (Z) plane.



- <Radial>. The vertical tool movement is performed according to the sinusoidal rule, where the depth variation speed at the last point is equal to zero. The parameter is defined by an arc radius, the center of which is located parallel to the Z-axis from the starting point of the toolpath.



You can define more than one type of plunge for one operation by set check field for this plunge type. At this case, you can arrange preferred types manually using button  . Upper plunge type at list is more preferred for system.

See also:

Approach, retraction, and plunge methods

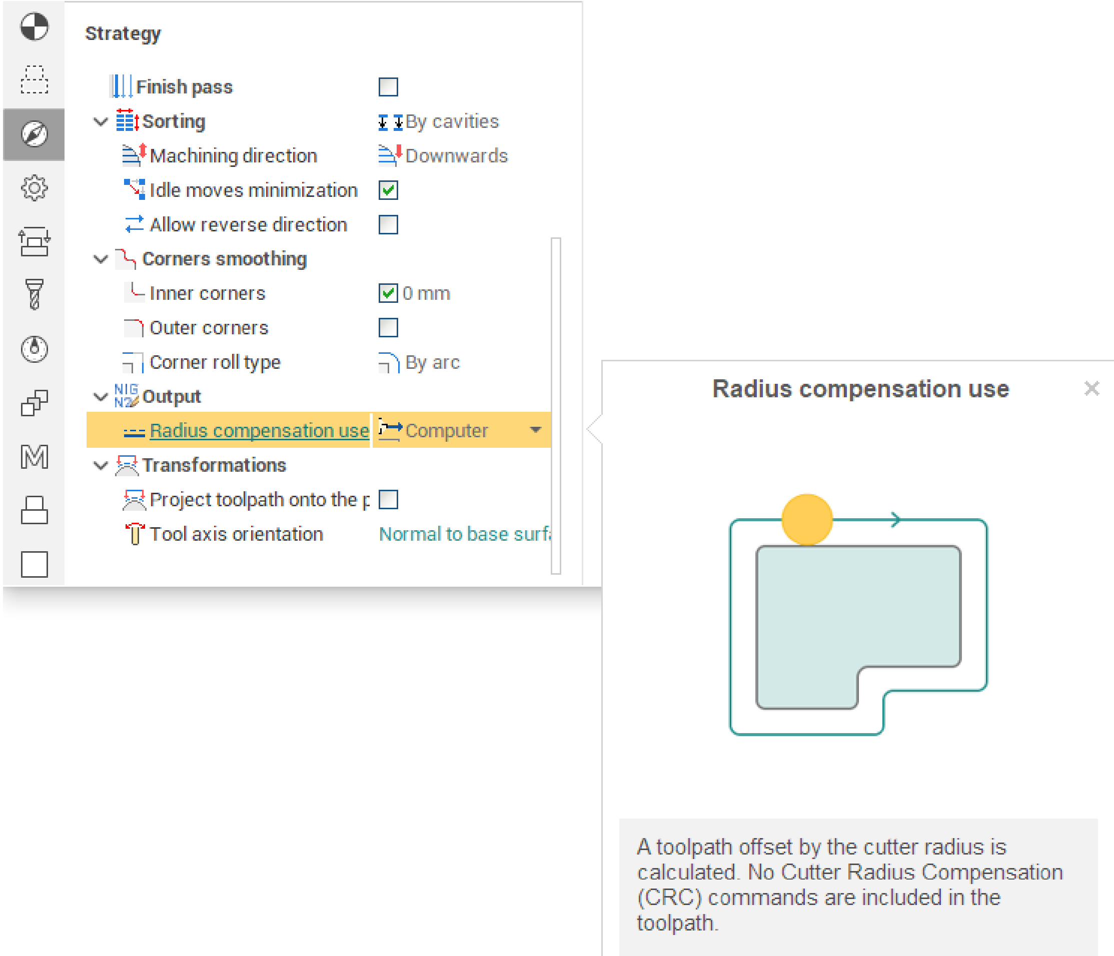
Compound approaches and retractions of mill operations

An approach move is performed in the same plane as the contour being machined. An approach move is a part of the machining toolpath, which is added to the starting point of the contour being machined. It can consist of three areas – toolpath inclusion area, the approach area itself and the area that represents an add-on to the contour being machined. Using these three methods, the user can obtain an optimal approach toolpath in every case.

Activation of tool radius compensation

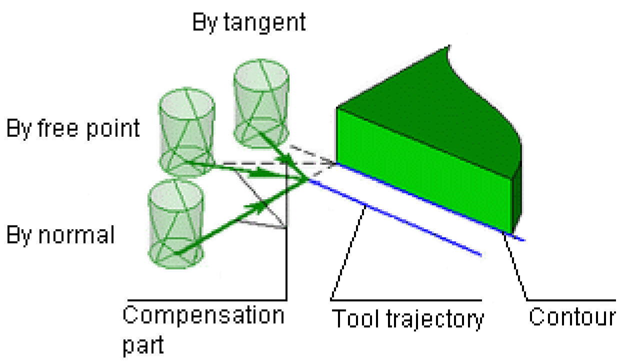
The activation and cancellation of cutter radius compensation can only be performed on linear tool movements, as either a tangent or normal (perpendicular) move to the next toolpath move, or from an arbitrary point. Immediately following the compensation activation area of the toolpath, there can be either an approach area, or a contour extension area, or the contour itself, depending on the conditions and options selected by the user. The compensation activation area is used to define the linear move in the NC code that is used to apply cutter radius compensation. It would also include the <G41> or <G42> commands as well as the offset numbers used (usually defined with a <D> or <H>). The tools used to define these parameters are found on the <Toolpath> page. Using the options in the <Compensation switch cut> area, the user can select the required method for applying the cutter compensation for the CNC control. The length of this move can be defined in the <Distance> field. In the "Use" combo box is selected in some cases it is need to add the compensation switch cut. There are several options.

- When compensation is on. Compensation switch cut is added to the toolpath only if you've enabled the tool radius correction. Correction can be enabled or disabled for the entire operation on the "Lead In/Lead Out" page with an appropriate switch. To enable the correction you should also set the Use compensation flag for the specific contour in the Job assignment window.
- Never. Compensation switch cut will not be added.
- Always. Compensation switch cut will always be added, even if the tool radius correction is turned off.



The compensation activation area is formed by the Milling unit's drive system; the older units form a simple linear transition, more modern CNCs can create a toolpath with control over tool contact with the workpiece.

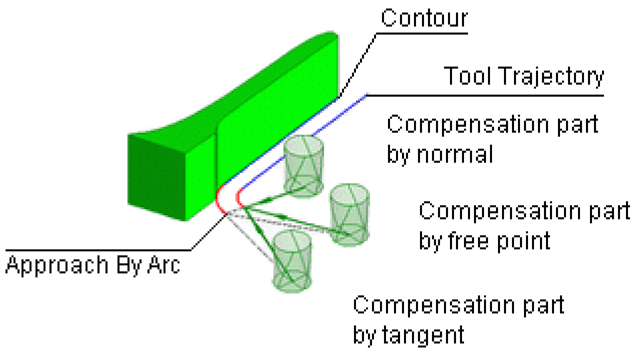
The picture below shows the available methods for the additional moves required for applying cutter radius compensation. These moves are joined to the start of the toolpath. In this picture the approach and additional approach moves are not used. The dashed lines show the tangent, arbitrary and normal (perpendicular) methods of compensation application moves. The lines with arrows are the toolpaths that will be produced at the CNC machine based on the radius value of the tool that is entered into the CNC control by the operator.



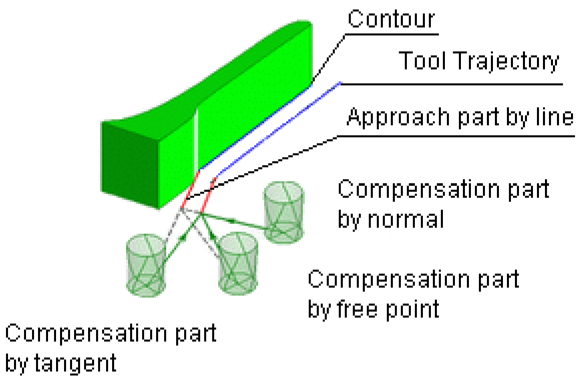
Approach

Rather than plunge the tool into the workpiece at the start point on a contour, it is possible to add an approach move. The choices are none, tangent, normal and from start point. This approach path would connect directly to the contour itself, or, if an <Additional approach> move is selected it will connect to this. If a <Compensation switch cut> is also selected, then this move will be added before the approach move, and the compensation command will be output in the NC code. Accordingly, if compensation activation (Model page) was not activated, then the tool will follow the center of the approach curve. The approach can be defined in the <Toolpath> window. In the <Approach path> field, the user can choose the approach type: <Without approach>, <Arc>, <Normal>, <Tangent>, <Angle to tangent>.

The picture below shows an arc approach move. The end of the arc touches the models contour. Actually, the approach area forms part of the machining contour. The question of whether to use an approach move or not has to be decided by the user. It depends on the specific conditions and requirements of the model being machined.



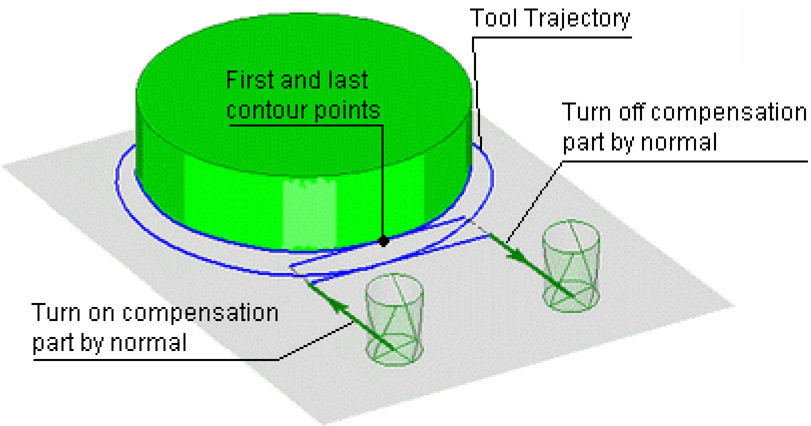
The next picture shows an approach using angle to tangent. This also uses a compensation switch cut as in the example above.



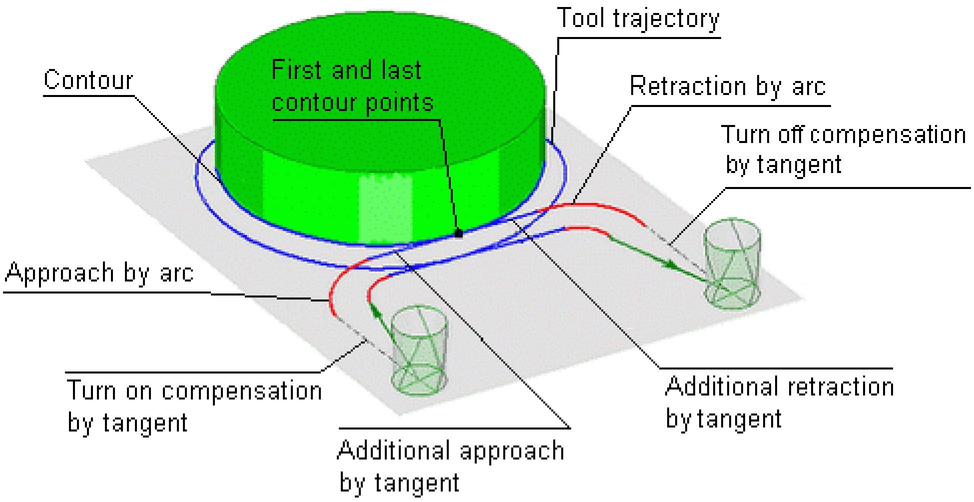
Contour extension area.

It might be required on occasions, to extend either the start of a contour, or the end, or both. To achieve this in SprutCAM X we use the <Additional approach> and/or <Additional retraction> options. An additional approach is added at the start of a contour, additional retraction is added to the end. In general, it is used when machining closed curves, when the start and end points of the contour are coincident. When machining starts and ends at the same point on a contour then a <Witness mark> can be left on the contour. This is due to the uneven loads on the tool due to stock removal.

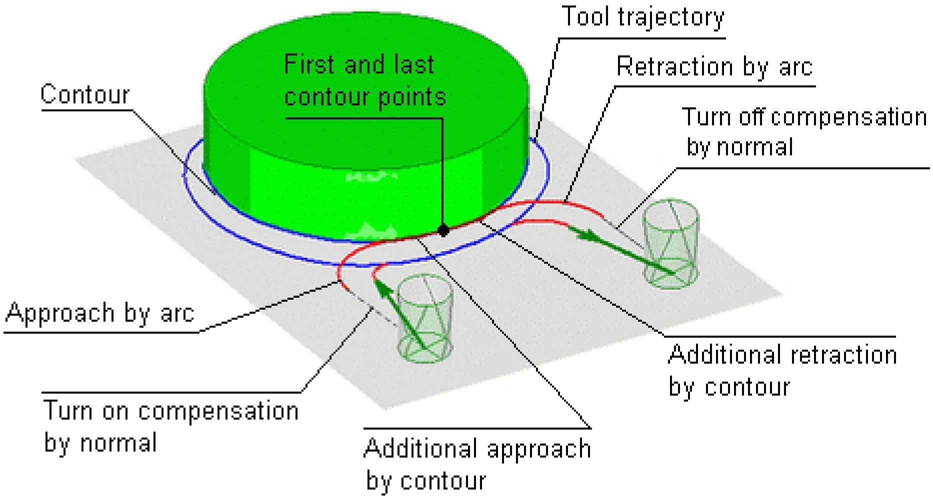
That extension area can be formed by two methods; either tangent to the starting and the end points or along the contour. When using the <Tangent> mode, one should note that in some cases the tangent can be pointing towards the model and thus take the necessary actions to prevent damage to the model. The additional approach and retraction moves are defined in the <Toolpath> window. In the <Approach type> area the user can choose the approach type: <None>, <Along curve> and <Tangent>.



The picture above shows the situation where the additional approach and retraction areas are defined using the <Tangent> method. The approach and retraction moves are not used and a <Normal> <Compensation switch cut> is being used. An additional tangent move at the end and a normal compensation switch is used to cancel cutter radius compensation.



The picture above shows a similar situation to the previous example, but with approach and retraction moves as well.

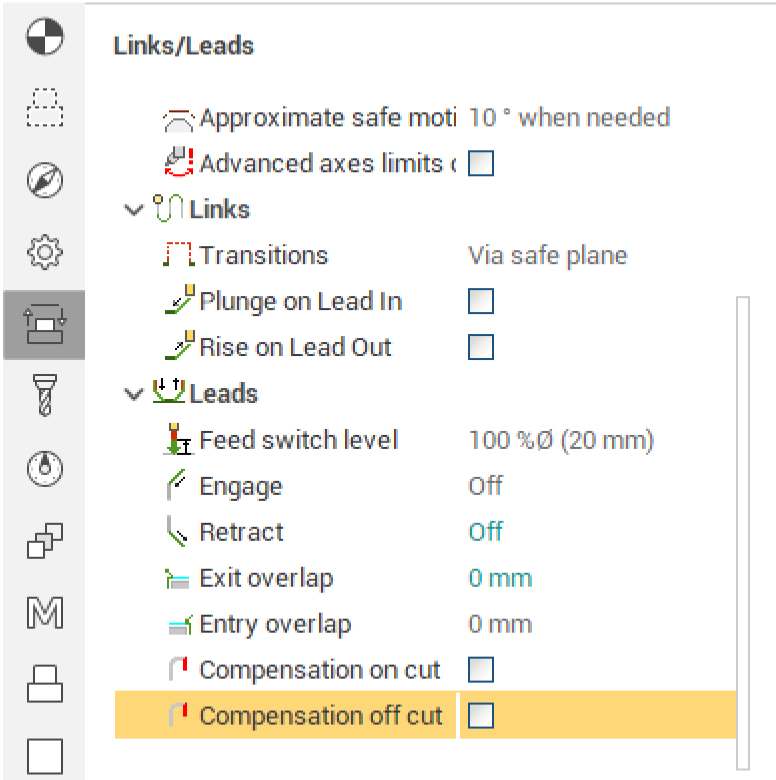


This example has the along contour option selected, the <Additional approach> and <Additional retraction> options are set to <Along curve>.

Tool radius compensation switching.

On Lead In/Lead Out page can enable or disable the use of correction on the tool radius. Depending on the type of operation the panel of correction can change his appearance.

The panel of correction in the contouring operations has a switch that allows you to specify only two options: "correction is on" and "correction is off". If the correction is on, it becomes available input field where you can set the value of the compensation radius. This value does not affect the toolpath calculation and the coordinates displayed in the NC-program. It only sets a value that is used in the simulation mode to simulate the behavior of CNC control. CNC, with the inclusion of correction, shifting tool from the programmed path by an amount specified in the corrector to the tool radius. Compensation radius can be specified either in the current units of the system (mm or inches), or a percentage of the tool diameter.



The correction panel of Waterline and some other operations, has a combo box that contains several possible modes of correction. Depending on the correction mode radius automatically takes either zero or a value equal to the tool radius.

See also:

Approach, retraction, and plunge methods

Tool radius compensation

Controlling tool's transition between operations

SprutCAM X generates auxiliary tool motions including transition from the tool change point, tool motions between operations and return to the tool change point using the operation start, operation end points, and the tool change point defined in the NC.

If additional control over the tool motion on these parts of the toolpath it is available to specify Approach and return rules for the operation.

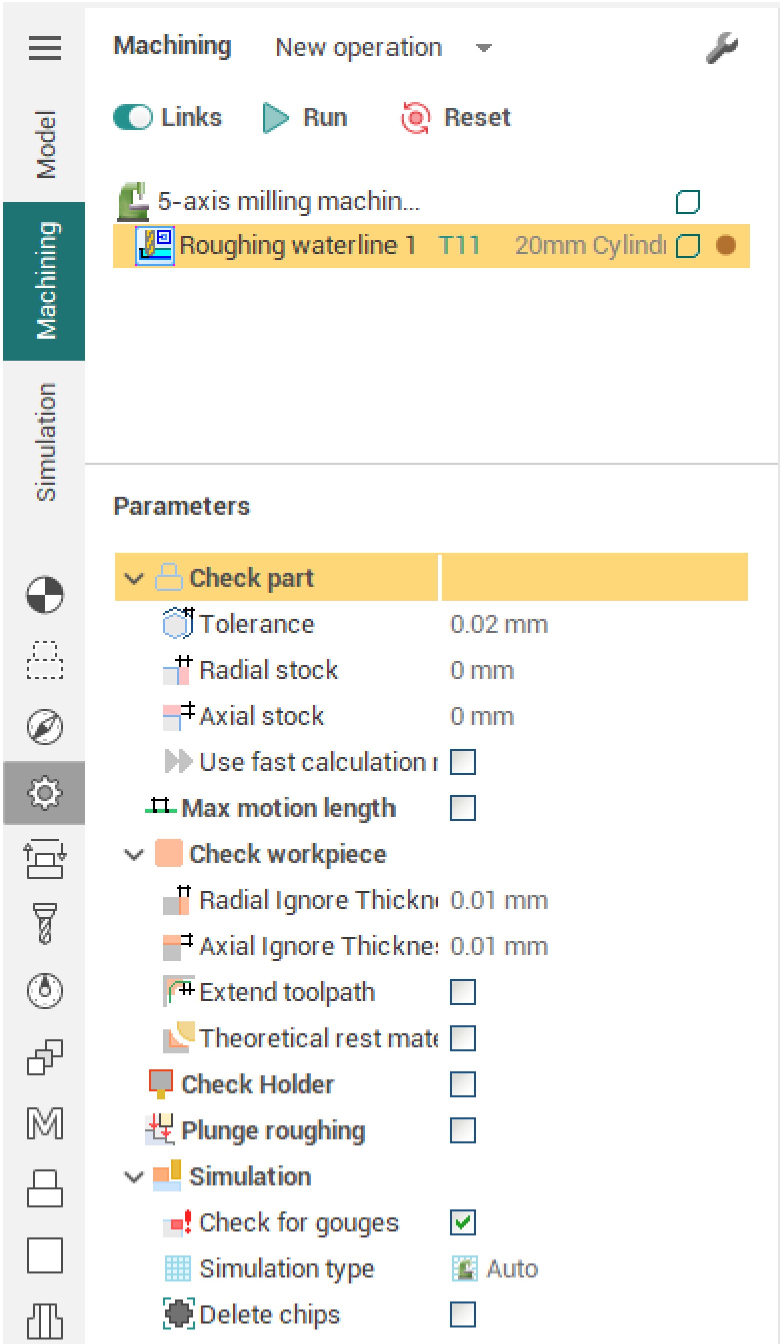
Specify tool change point for the operation if tool is changed in a point different from machine tool change point.

See also:

Approach and return rules

5.5.7.11 Assigning parameters of mill operations

The main parameters for the current operation can be defined in the < Operation parameters > window. It opens when the < Parameters > button is pressed.



This window have some sheets:

- < Tool > – for setup tool parameters.
- < Feeds > – for setup tool feeds.
- < Lead-In/Lead-out > – for setup approach, engage, return, retract and plunge.
- < Parameters > – for specific operation parameters setup.
- < Strategy > – for operation strategy setup.

Each operation type has its own help graphic and parameter list. Depending on the type of operation selected, the number of available parameters may vary. Any of the fields can be used for calculations; the user can enter any mathematical expression. To view the calculation result, it is necessary to point the mouse at the required field; the result will be shown in the tool tip text.

The following can be defined on the < Parameters > page in the < Operation parameters > window:

- <Geometrical coordinate systems> of an operation, defines the position of the workpiece and zero adjustment for the milling machine. All coordinates for the NC program will be calculated in the defined coordinate system. Any previously created coordinate system can be selected as the coordinate system for the current operation. By default, when a new operation is created, the currently active coordinate system is used.
- <Rotary axis position> can be defined if there is a rotary head on the milling unit, and its position is defined in the system settings. In this case, at the beginning of every operation, the < ROTABL > command for positioning of the rotary head will be inserted into the CLDATA program. When using the rotary head, its position must be synchronized with the coordinate system for the operation. When the window is closed this condition will be verified. If it is not synchronized, then SprutCAM X will attempt to select a coordinate system that will match the defined position of the rotary head. If a suitable coordinate system is not found, then the system will suggest creation of a new coordinate system, which will correspond to the defined position of the rotary head.
- <Machining levels> defines the range (depth) for machining along the Z-axis. If a tick is placed next to a field that defines the machining level, then the level displayed in the field will be used, otherwise, the dimensions of the model being machined will be used.
- <Safe plane> defines the level at which rapid movements of the tool can take place.
- <Deviation> defines the maximum deviation allowed for the approximated toolpath. The default machining tolerance for all operations is defined in the system settings window (Options –> Machining tab).
- <Stock> – the amount of material that is left after an operation, for further (finish) milling. By default, for finish operations the stock is set equal to 0, and for the rough is calculated by internal algorithms.
- <Z Stock on> – can be defined only for the engraving and pocketing operations.
- <Lateral angle> – available only in the engraving operations and defines the side surface of the model. Unlike the draft angle, this parameter is not considered when machining restricted areas.
- <Z step by Z> is available in all rough operations and in the waterline roughing operation, and conforms to the thickness of the material layer, removed for each pass. By default it is calculated by the system according to the tool parameters of the operation and the workpiece dimensions. The step can be assigned in millimeters(inches), as a percentage of the tool diameter or calculated with regard to the required number of passes. When defining the step by scallop, it will be calculated for every layer according to the amount of the required scallop height.
- < Clear flats >. With this function active, the system will perform additional passes at those levels, where there are horizontal areas.
- < Relief angle > is available only in the waterline roughing and the engraving operations. It defines the minimal horizontal offset between the layers being machined. It is used to prevent the side of the tool touching the side of a deep-machined area.
- < Draft angle > is available only in the waterline roughing and the engraving operations. It defines the minimal horizontal offset between the layers being machined. It is used to prevent the side of the tool touching the side of a deep-machined area.
- <Z cleanup> with increment on the Z-axis for 2D and 3D curve machining operations. It defines the value for additional stock for a finishing pass. This gives a better surface finish and reduces cutter push off.

When the < By default > button is pressed, the system will set all values to their default state. When the < Ok > button is pressed the alterations will be applied for the operation, otherwise the operation parameters will not be changed.

See also:

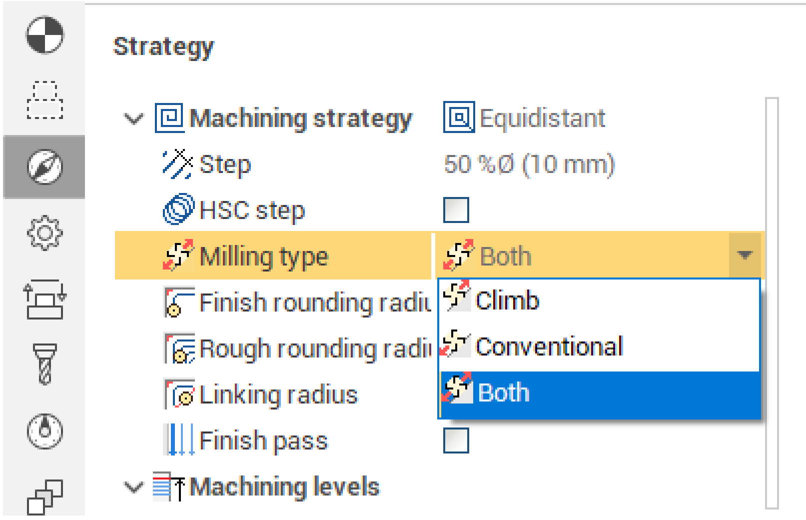
Mill machining

5.5.7.12 Defining the machining strategy of mill operations

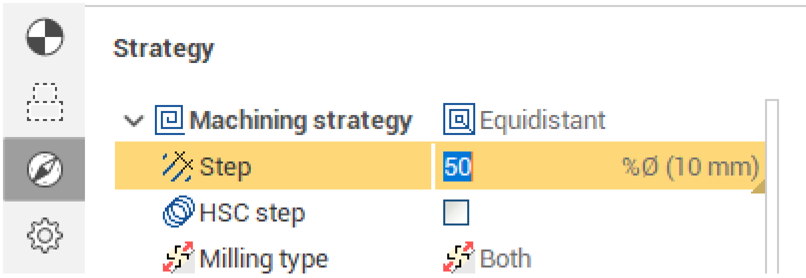
The definition of many parameters, which define the machining strategy for a model, can be performed in the < Operation parameters > window on the < Strategy > page. The window for altering these parameters is opened by pressing the < Parameters > button. This page gives access to a variety of fields and their explanatory graphics. Depending on the type of operation selected, the number of available parameters may vary. Any of the fields can be used for calculations; the user can enter any mathematical expression e.g. "10*sin(45)". To view the calculation result, it is necessary to point the mouse at the required field. The result will be shown in the tool tip text.

The following parameters can be defined on the page:

- < Type of milling > 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.

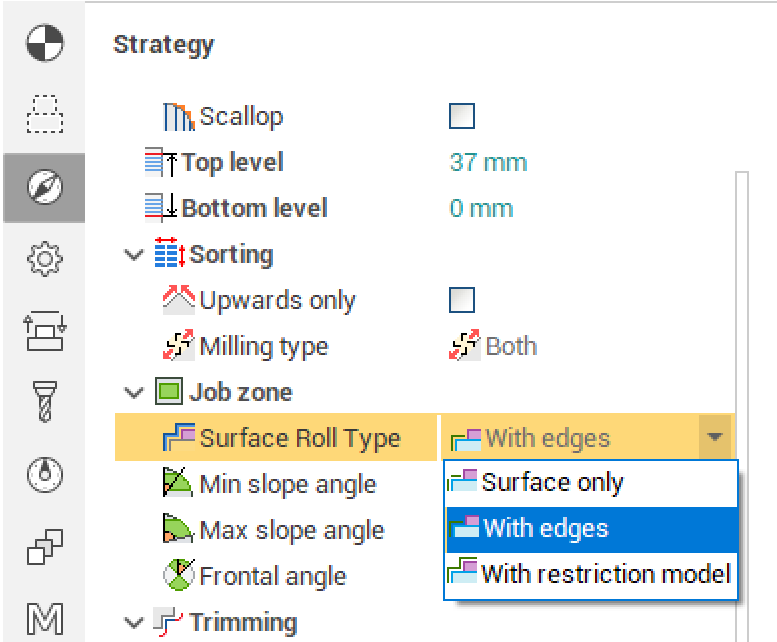


- < Step > of machining defines the distance between successive depths of the tool for the plane, drive and combined operations . The step can be either assigned by as an absolute value, as a percentage of the tool diameter or calculated at every step relative to the required scallop height .

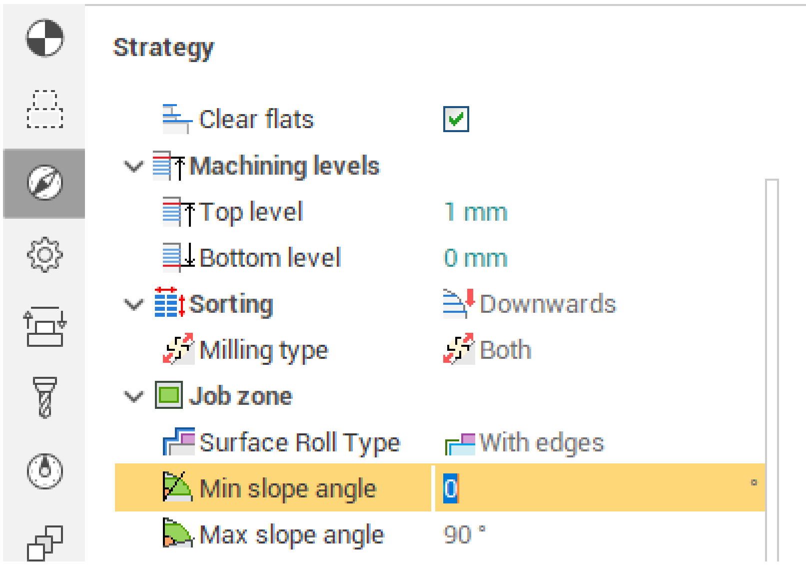


- < Stepover type > defines the toolpath when moving from one machining pass to the next.

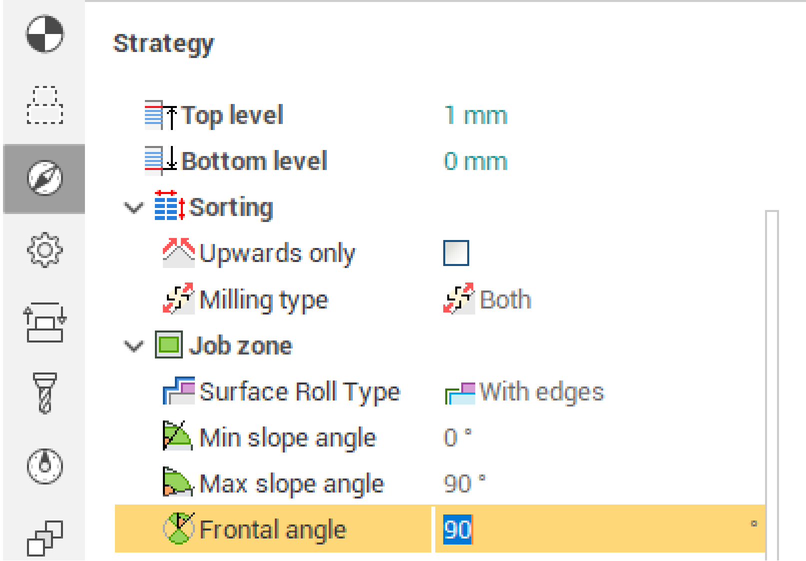
- < Roll type > defines the necessity of avoiding peaks and edges of the model when machining using the volume machining finishing operations .



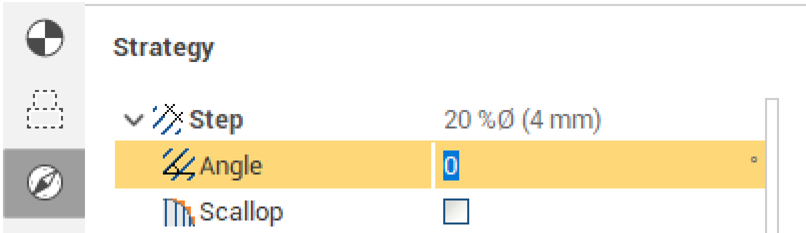
- < Surface slope > in the volume machining finishing operations limits the surfaces being machined depending on their slope angle.



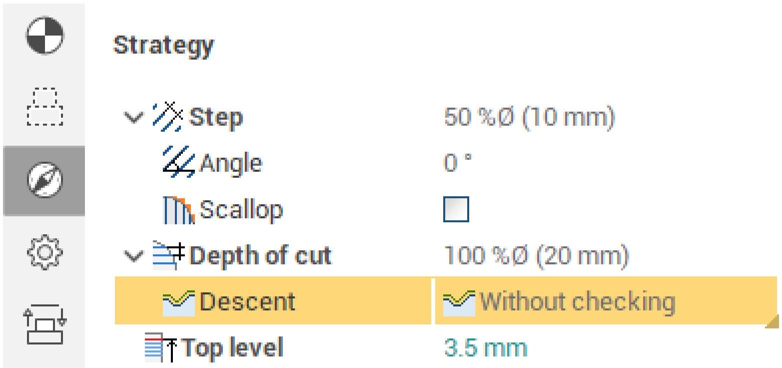
- < Frontal angle > in the plane and drive finishing operations limits the surfaces being machined depending on the angle between the cutting tools face in the cut direction, and the models surface(s).



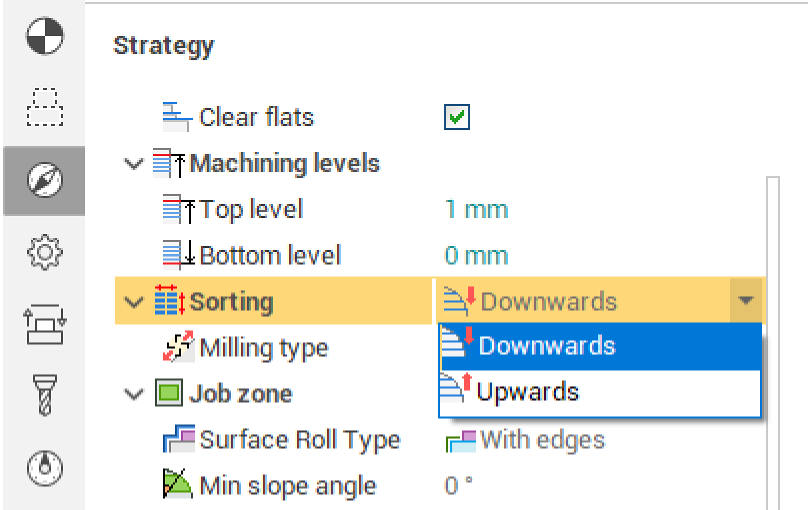
- < Angle > assigns the cut angle direction for the plane operations .



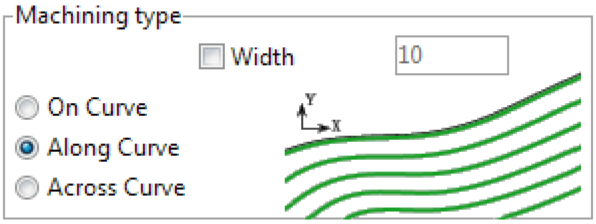
- < Descent type > defines the tool descent strategy for the plane and drive roughing operations.



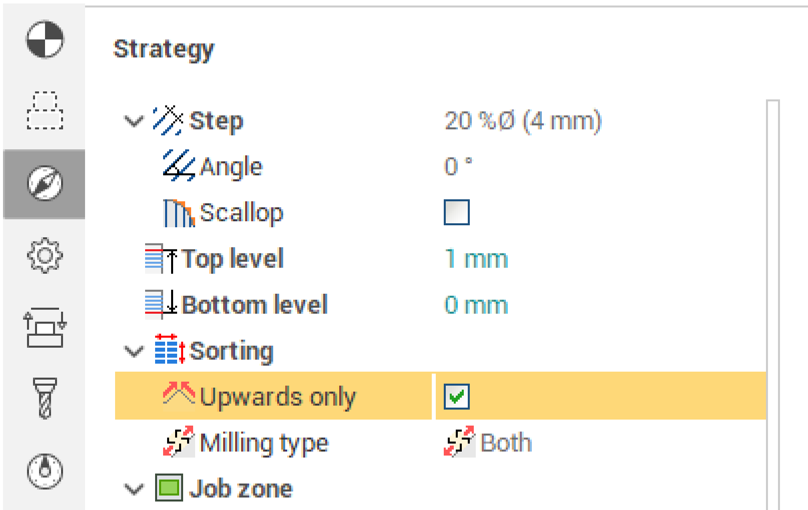
- < Machining direction > during the finish machining by depths defines the machining sequence, either upwards or downwards.



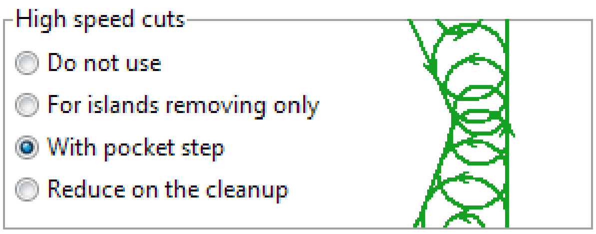
- < Machining type > in the drive operations defines the strategy for the work passes formed either along or across the drive curves.



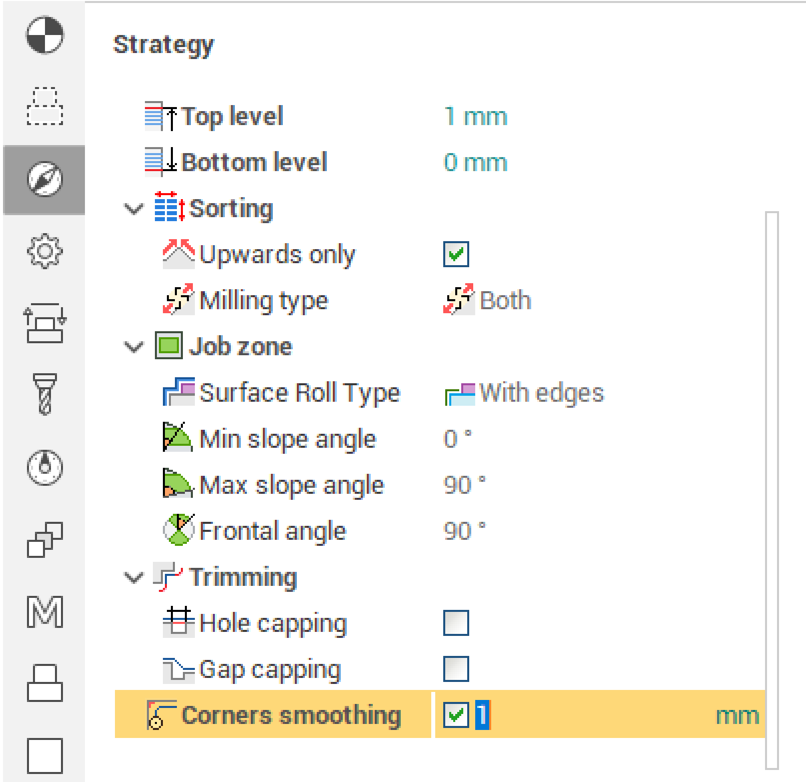
- < Upward only > in the plane and drive finishing operations restricts tool movements on the model to the upward (Z+) direction.



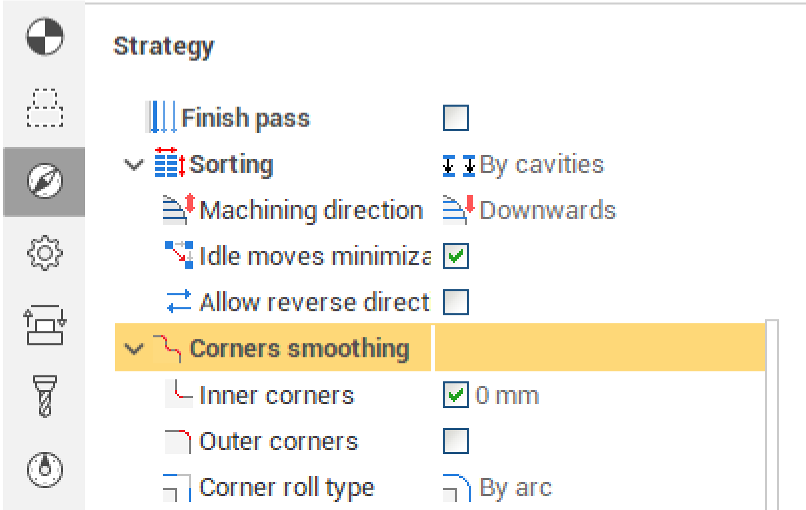
- < High speed cuts > allows to reduce the NC data much in comparison with the trochoid and at the same time secure the tool.



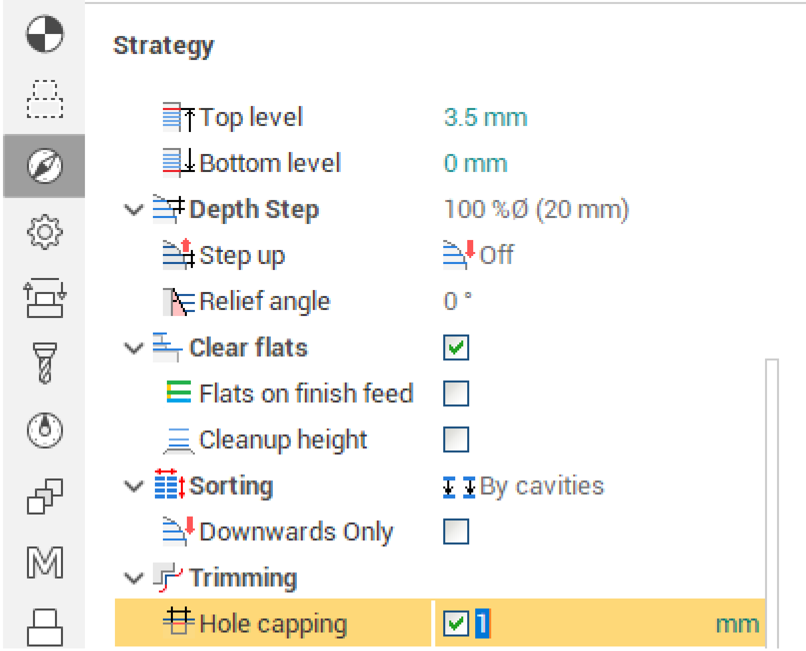
- < The corner-smoothing mode > virtually is in most operations, and provides toolpath 'rounding' when machining inner corners. This lessens vibrations and increases the machining speed.



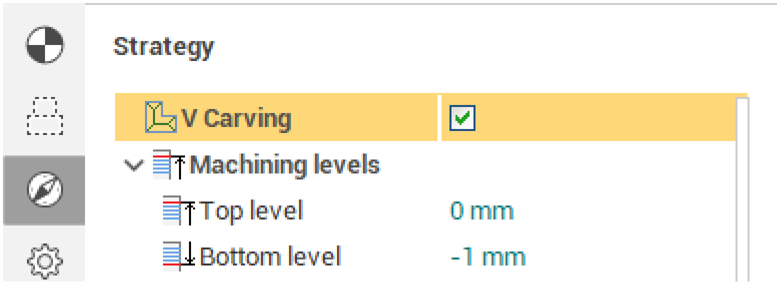
- < Corners smoothing > for 2D contouring provides toolpath 'rounding' when machining inner and outer corners.



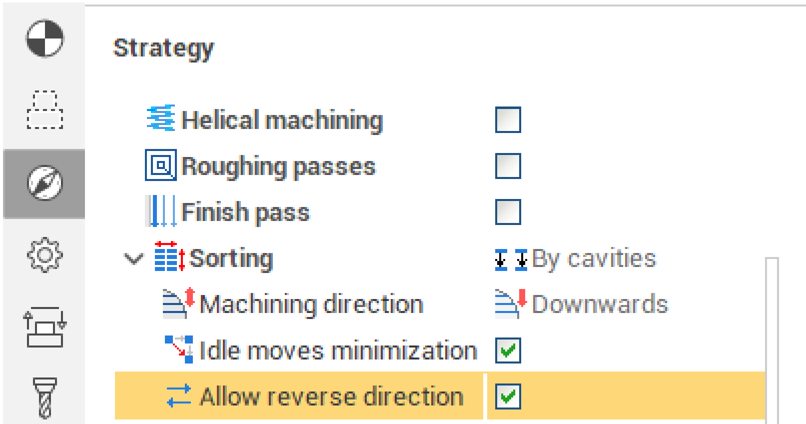
- < Hole capping > in the volume machining operations , this allows the system to ignore holes in the model that have a size that is less than the defined size, leaving them for further machining.



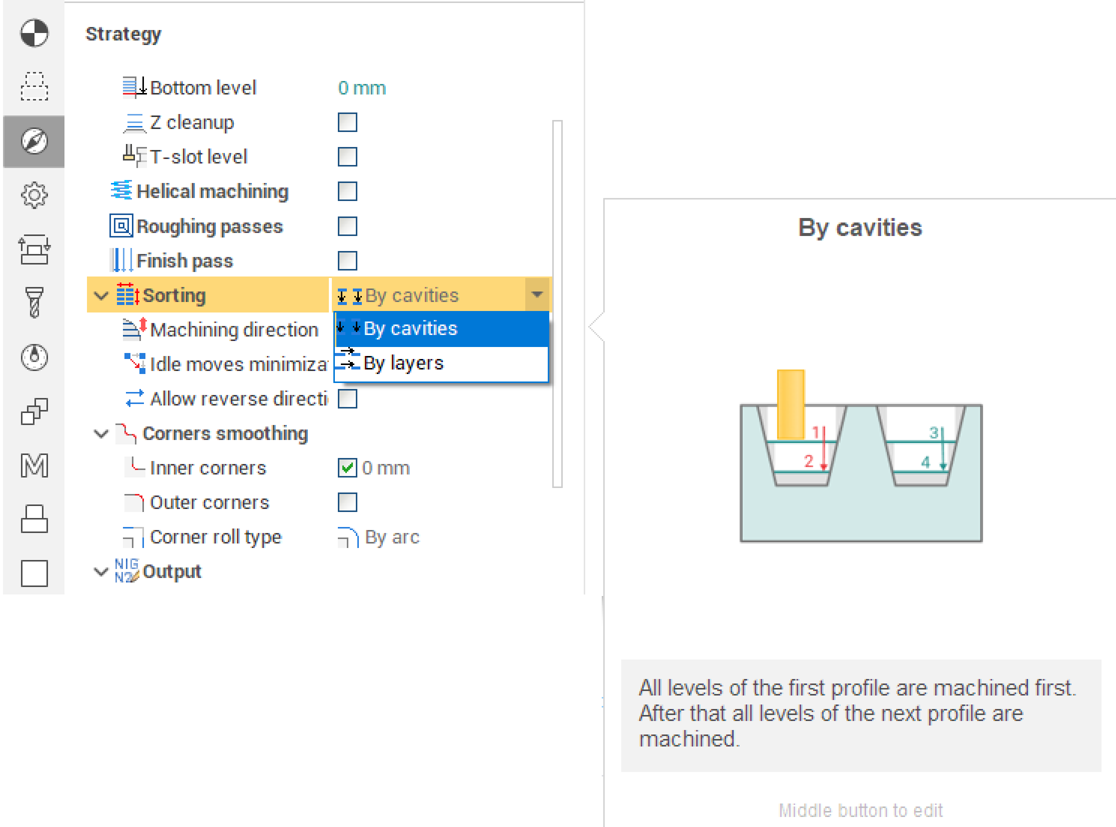
- < The allow 3D toolpath > mode allows the system to create a 3D toolpath for material removal in all areas that are not accessible for machining on the current level (e.g. inner corners).



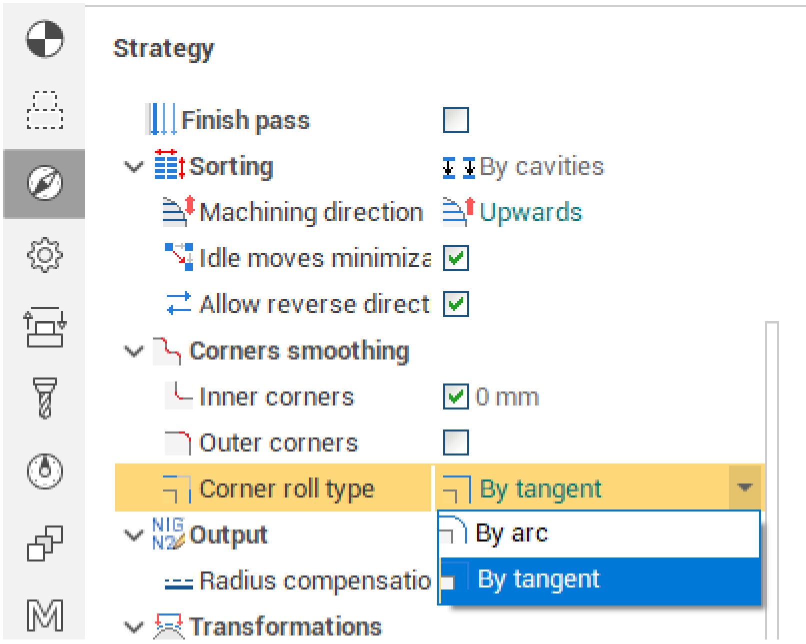
- < Allow reverse direction > in the curve machining operations allows the tool to reverse its cut direction along a curve if it will decrease the overall amount of tool movements.



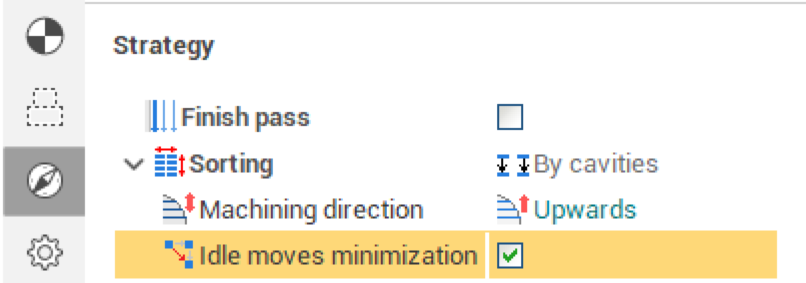
- < Machining order > defines the machining sequence in the curve machining operations (by contours, by depth).



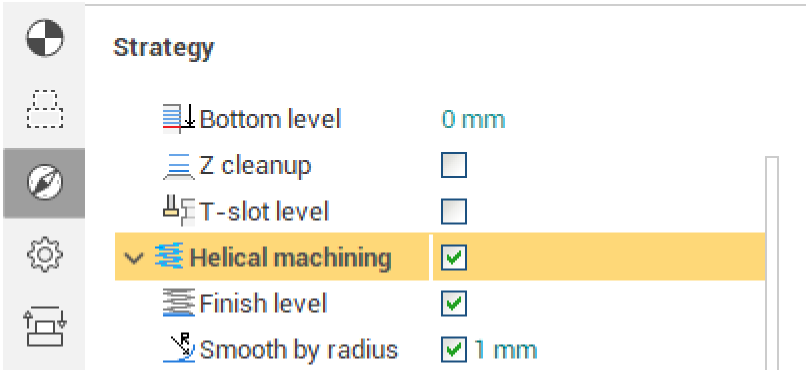
- < Corner roll type > defines the outer corner angle bypassing method in the curve machining operations .



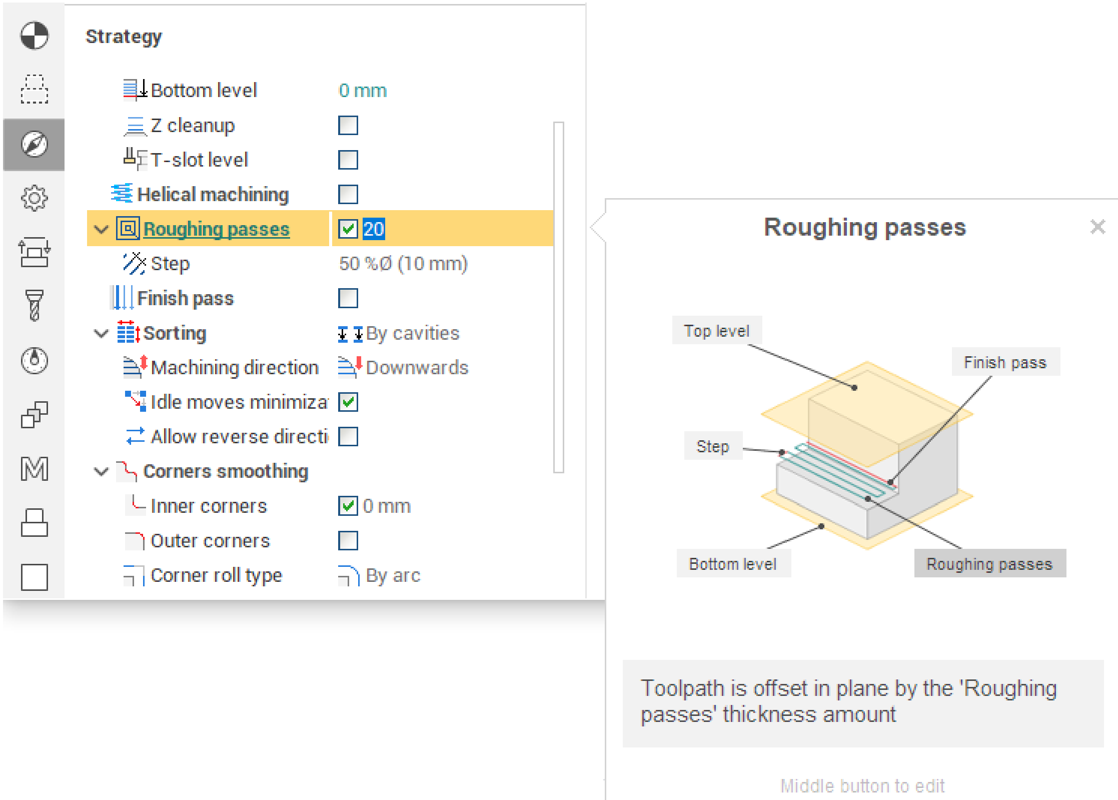
- With < Idling minimization > active, the total distance that the tool moves for machining all selected curves, is kept to the minimum. Otherwise, the machining will be performed according to the order defined on the Model page .



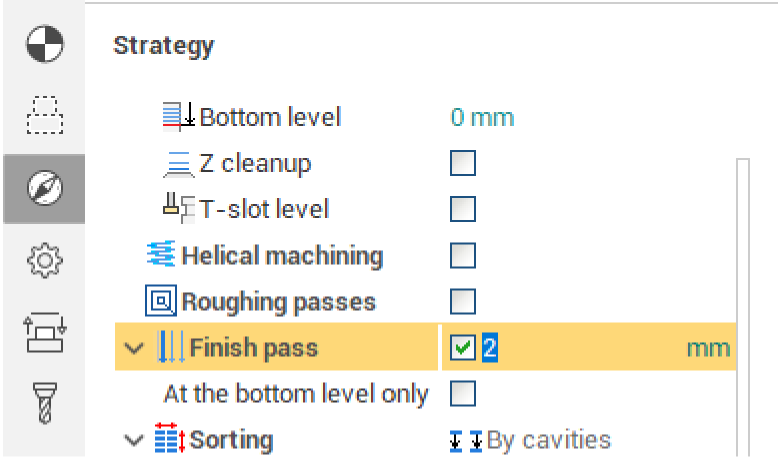
- < Helical machining > allows the user to obtain a spiral-like toolpath when machining a 2D curve .



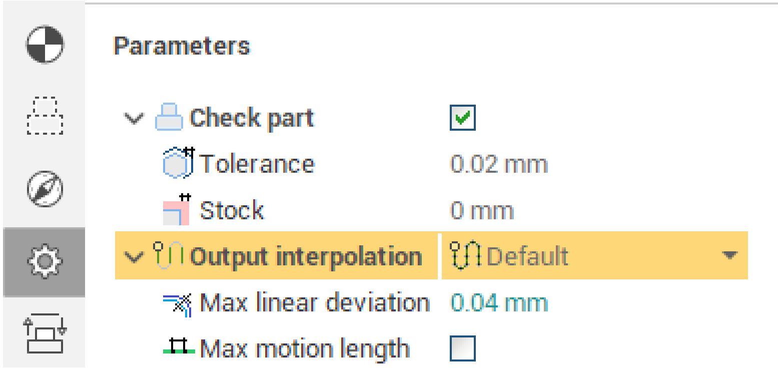
- < Roughing XY paths > allows the use of several XY passes to remove the stock material instead of a single pass. This can improve the surface finish by reducing the load on the tool.



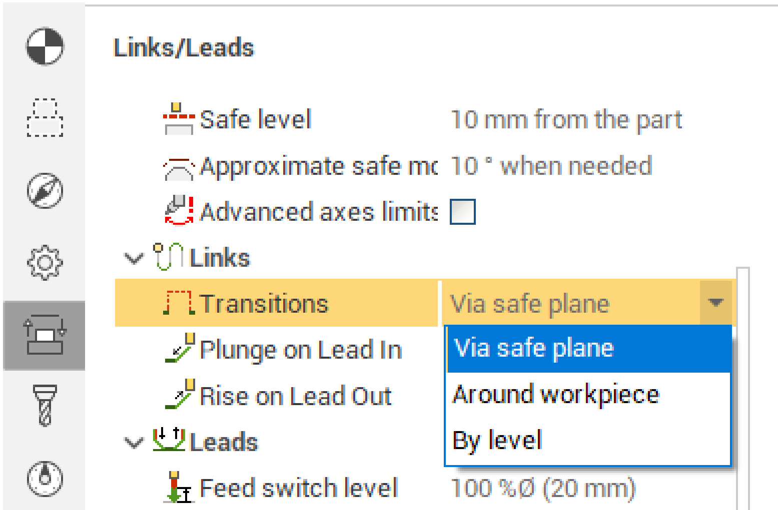
- < XY cleanup allows > the user to obtain a higher quality surface finish by leaving only a small user defined stock for an automatic finish cut.



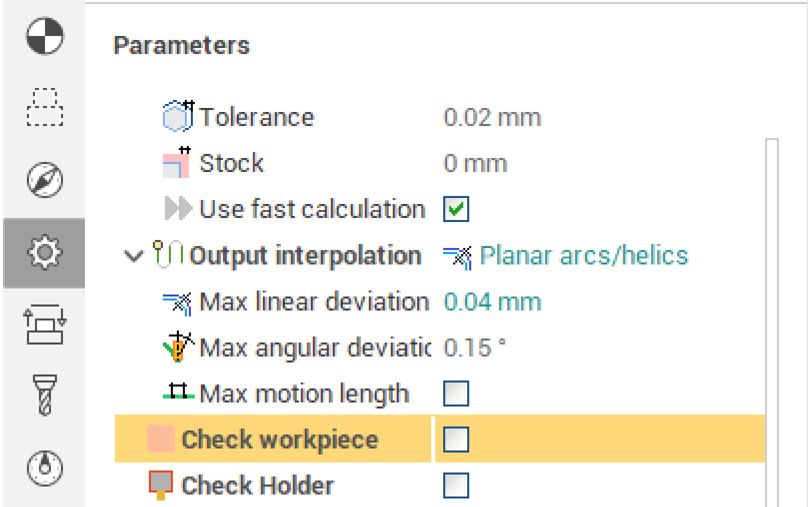
- < Work passes interpolation > approximate toolpath by arcs.



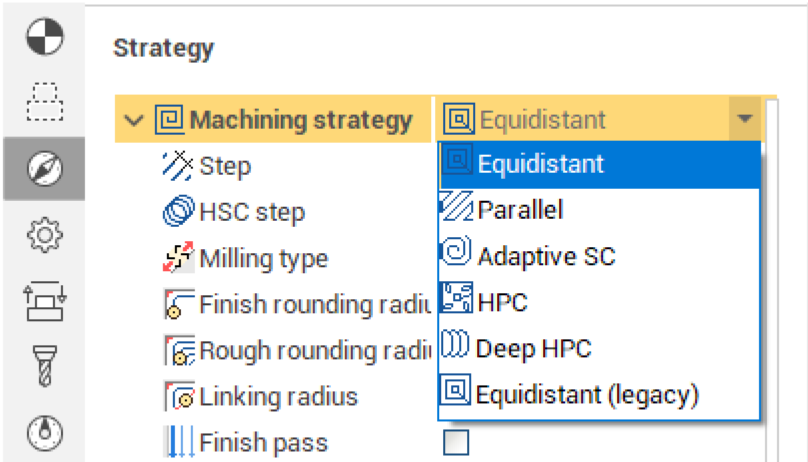
- < Transition > allow select stepover method for curve machining operation.



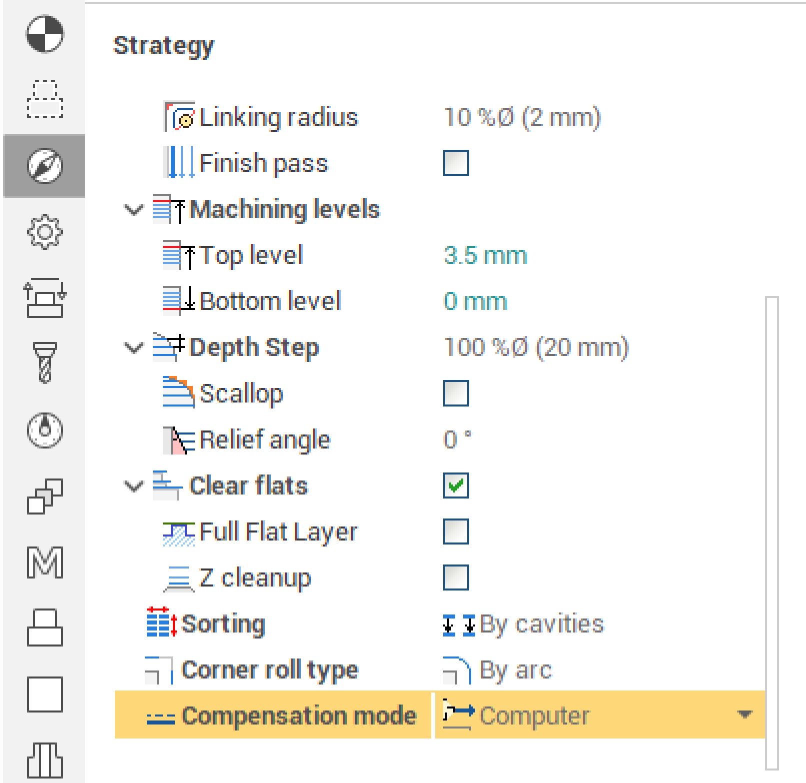
- < Check geometry > for finishing operations. This operations not check rest material by default and machining all surfaces from job assignment. For skip already machining patches you must set check < Check workpiece > or < Check part > and setup thickness value.



- < Machining strategy > allows the user to obtain a parallel-like or equidistant-like toolpath when machining for pocketing operations.



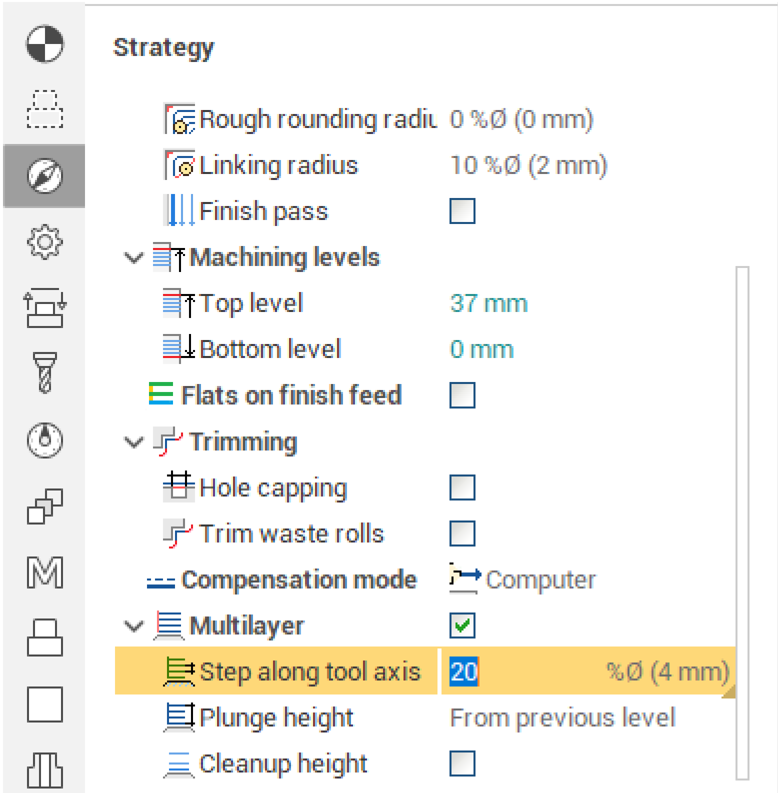
- Tool < compensation > is available for pocketing operations.



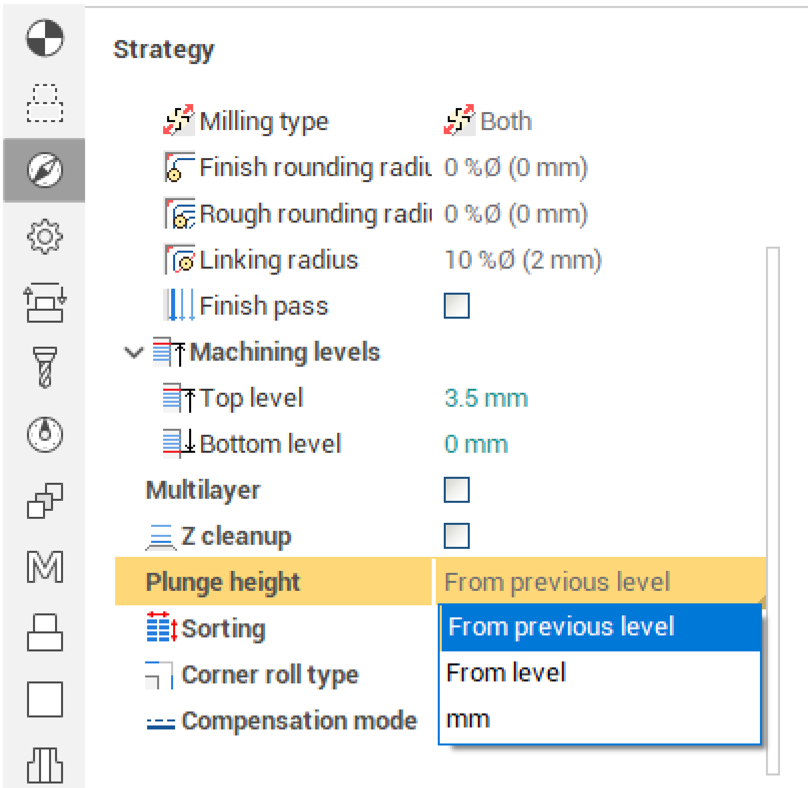
The strategy of flat land machining on axis Z.

In < Flat lands finishing > and < 2.5D flat land > operations the strategy of removal of a material layer above a horizontal plane is defined in panels: < Plunge Height >, < Machine by Layer >, < Z cleanup >.

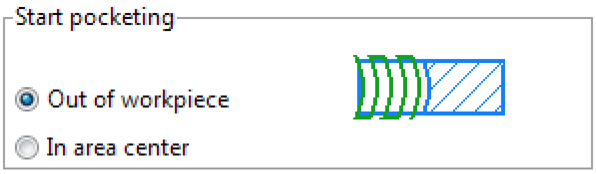
- < Machine By Layer > The material may be deleted for some passes. For this purpose, it is necessary to switch on < Machine by Layer >.



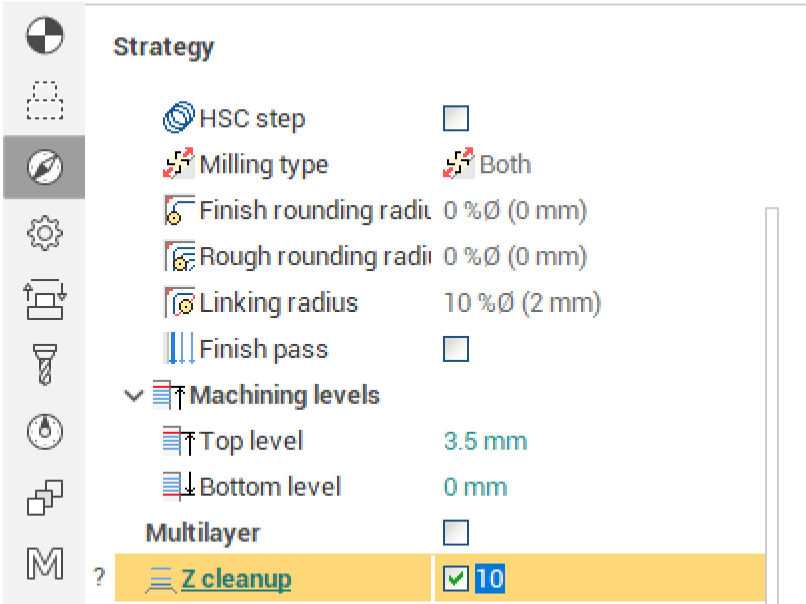
- < Plunge Height > The height of the layer is set in the < Plunge Height > panel; in particular, it defines a value of level Z on which the given type of plunge will join.



- < Start Pocketing > In the panel is defined the strategy of the tool motion within the limits of one layer.



- < Z cleanup > For the definition of width machined material layer on finish pass, it is necessary to switch on the < Z cleanup > option and to enter a stock value on finish pass.



On the < Strategy > page in the < Operation parameters > window for the hole machining operation the user can define the cycle type for hole machining (drilling, boring, tapping etc.) and its parameters. The set of parameters depends on the cycle type. Points for drilling can be defined in the < Model > window, or be automatically imported from a corresponding roughing operation. All holes for an operation are machined using one tool and a single cycle type. To machine holes using different cycles it is necessary to create a new operation for each cycle type. The order of hole machining can be defined either by the list, defined on the < Model > page, or be optimized so that total tool movements between holes is minimal.

See also:

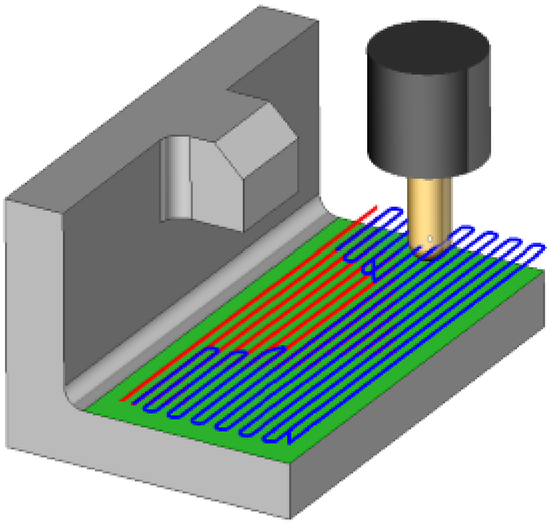
Mill machining

5.5.7.13 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.

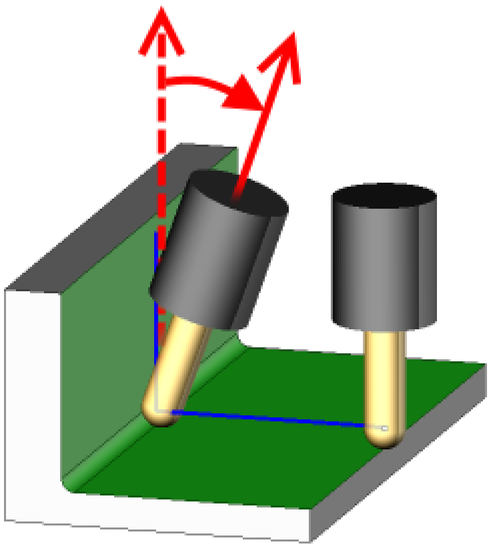
Strategies

Trim toolpath



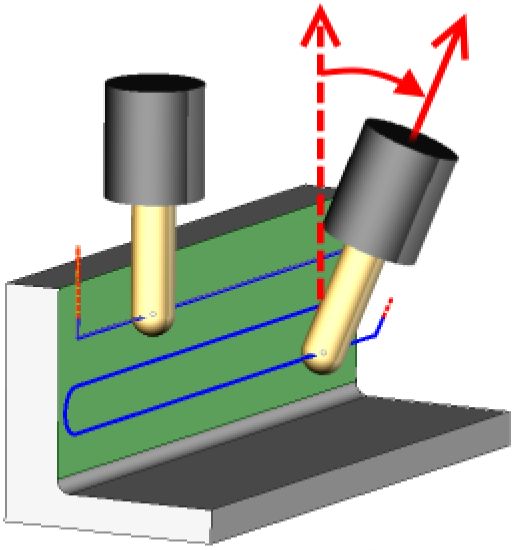
The colliding segments are excluded from the toolpath.

Frontal tilting



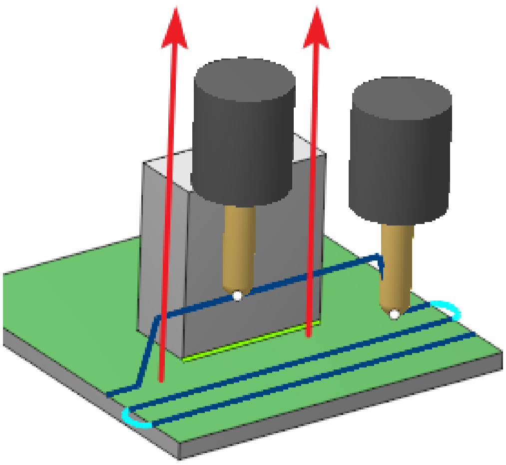
The tool tilts in the frontal direction to avoid collisions.

Side tilting



The tool tilts sideways to avoid collisions.

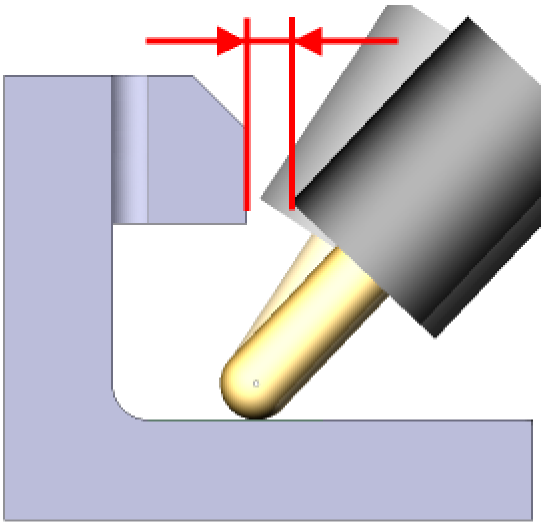
Offset along tool



The tool retracts along its axis to avoid collisions.

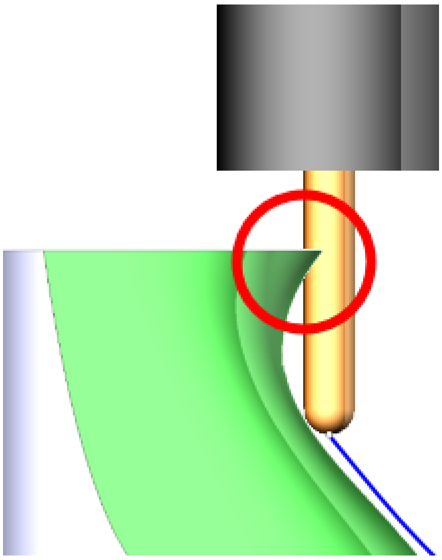
Parameters

Clearance



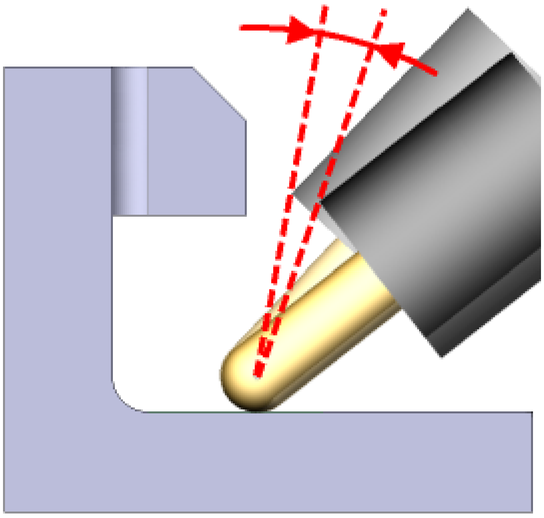
The minimum allowed distance between the holder and the part.

Check tool



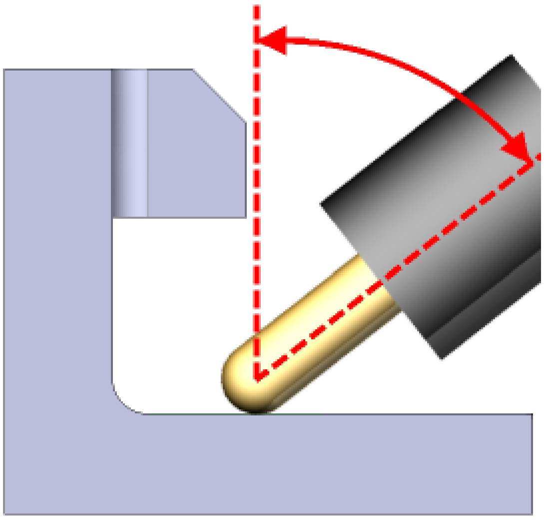
The option enables checking of the tool collisions alongside with the holder collisions.

Additional tilt angle



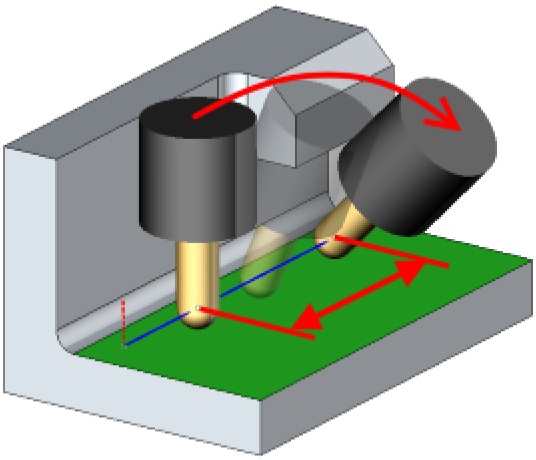
The additional clearance angle used for tilting to guarantee a safe collision-free toolpath. The greater the value is, the farther the tool tilts away from the collision.

Max. tilt angle



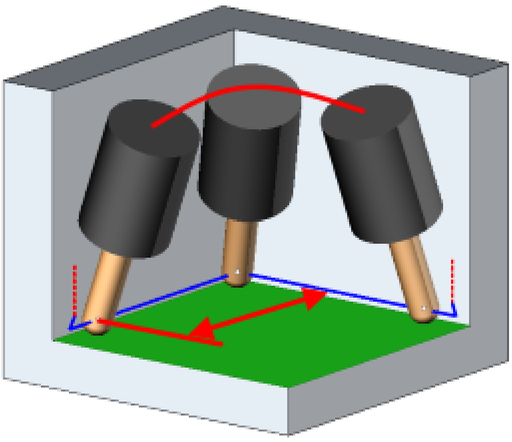
The maximum allowed tilting angle.

Smooth factor



The parameter controls smoothness of the tilting. The greater the value, the smoother the tilting is. 0 means no smoothing.

Blend distance

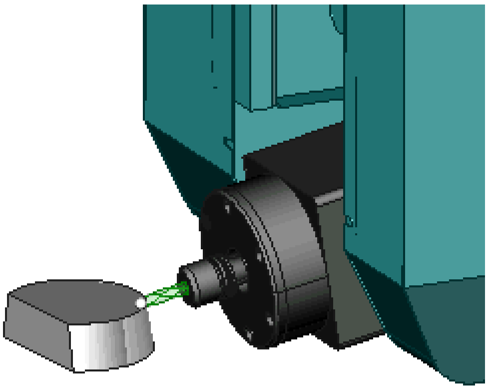


The blend distance is the distance between two points of the toolpath before and after the current tool position used for the calculation of the current tool movement direction which in turn is used to calculate the current tool tilting direction. Use greater values when machining irregular shapes with a lot of small bumps to eliminate unnecessary tool axis oscillations. Use values greater than the holder radius when machining inner corners.

See also:

Mill machining

5.5.7.14 Advanced axes limits control



Option allows to avoid 2 kinds of problem in the 5-axis toolpath.

1. When it's possible, it excludes the overturn in the middle of work pass. Overturn is performed on the rapid motions.

2. It generates the smooth path in the singular zones. Singular zone is a machine position where the one of the machine axis can not be calculated. For example, if A-axis is equal zero, then C-axis can be any.

Option available in most 5-axes operations.

See also:

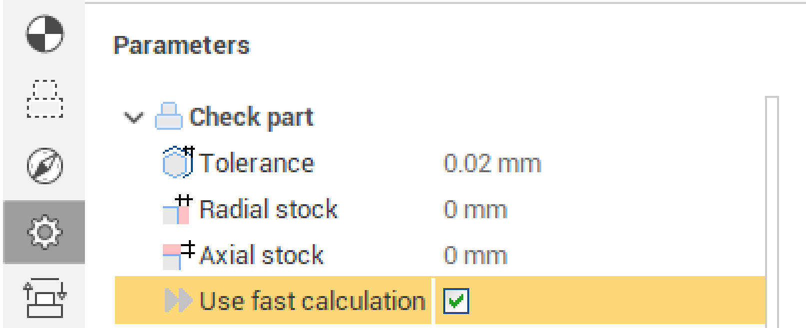
Mill machining

5.5.7.15 Miscellaneous parameters of mill operations

No content in this page. See child topics

Fast calculation method

The < Use fast calculation method > option is available for most 3d milling operations . The option is found in the Miscellaneous operation parameters tab.



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 < Deviation > 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 SprutCAM X gives you the alternative between speed of approximate calculation methods and the quality of exact calculation methods.

See also:

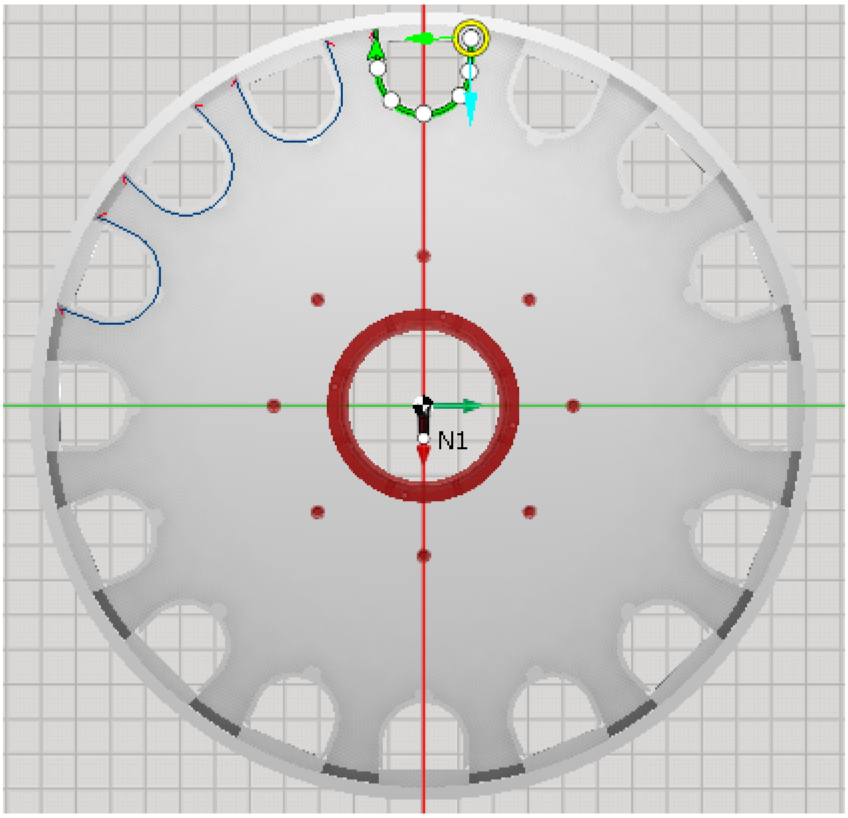
Mill machining

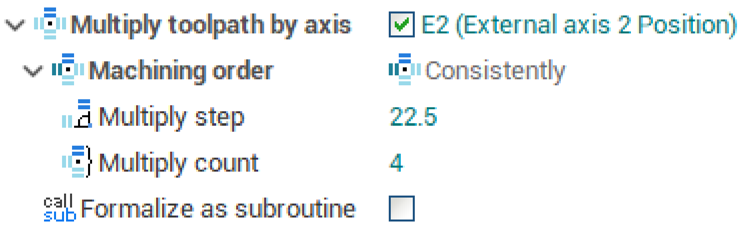
5.5.7.16 Transformation

<Transformation> – parameter's kit of operation, which allow to execute converting of coordinates for calculated within operation the trajectory of the tool.

On the page it is possible the setting of the following parameters:

- <Multiply toolpath by axis> allows to replicate the trajectory received on the computation of the operation in direction of one axle of the machine including rotary axle, if it presents in the machine.





The coordinate system axle is selected in <Multiply toolpath by axis> field.

Machining order has 3 types:

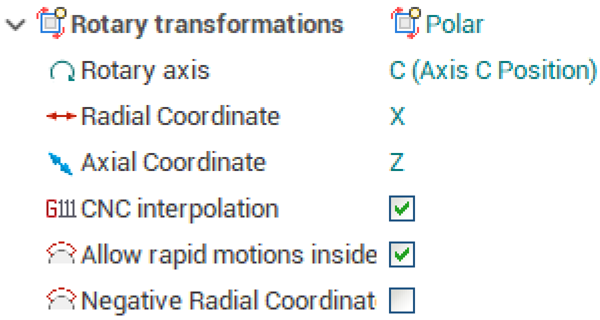
- <Consistenly> – repeating copies of a toolpath place along a circle, with the count and the angle step given in the appropriate field
- <Most distant> – repeating copies of a toolpath place along a circle, with the count and the angle step given in the appropriate field. Elements are sorted more distantly 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.

The following parameters can be set in multiply modes:

- <Multiply Step> - step axle-direction in units within the given axle;
- <Multiply Count> - quantity of block's repetitions;
- <**Steps**> - quantity of block's repetitions with own angle
- <**Angle1..N**> - angle of specific block's repetitions

<**Formalize as s**ubroutine> – the repeated block design as subprogram. Additionally you can set its name in the field. If <**Formalize as s**ubroutine> is disabled, then the repeated block over and over again insert in CLData with re-calculated coordinates.

<Rotary transformation> – 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.



The regime of converting is setting up in the field <Mode>:

- <Polar>;
- <Cylindrical>;
- <Off>.

For interpolation the following parameters are setting:

- The opportunity of executing by controller, setting with the help of the tick in the field <CNC interpolation>;
- <Tolerance> characterize the deviation of transformed trajectory from ideal in millimeters (inches);
- <Radial axis>;
- <Rotary axis>;
- <Axial axis>.

For cylindrical interpolation additionally set up the parameter <Cylinder radius>, on which the reamer of tool's movement is executed.

In addition to this method of copying, there is also Part Copies.

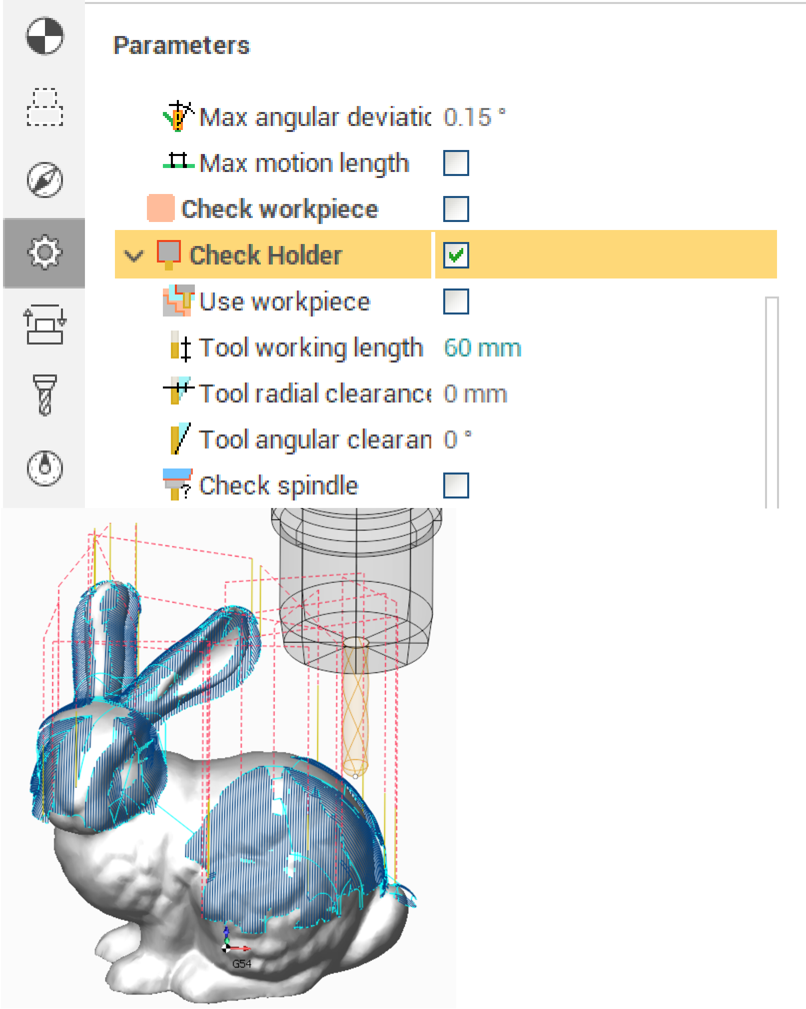
See also:

Mill machining

5.5.7.17 Check Holder in 3D operations

This option allows you to calculate the tool path concerning toolholder in 3D operations: Plane, Waterline and Drive. A check holder option can be used to avoid collisions with both part and workpiece. User can set additional radial and axial stock.

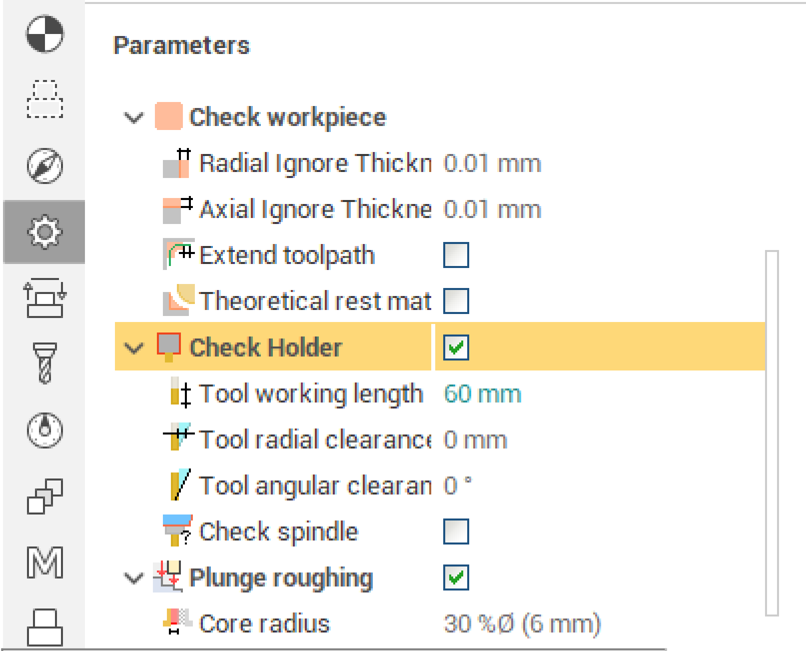
This option makes 3+2 machining easier. It is perfect for sculpture milling.



5.5.7.18 Check holder in Waterline roughing operation

This option allows you to calculate the tool path to avoid collisions of the tool holder with the workpiece in operation Roughing Waterline. Workpiece is not static: it updates dynamically while tool path calculation take into account upper layers.

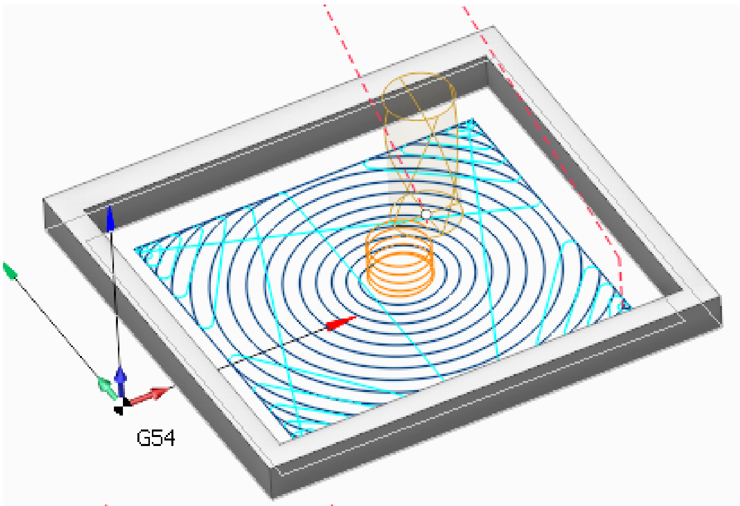
This option makes 3+2 machining easier. It is perfect for sculptures milling.

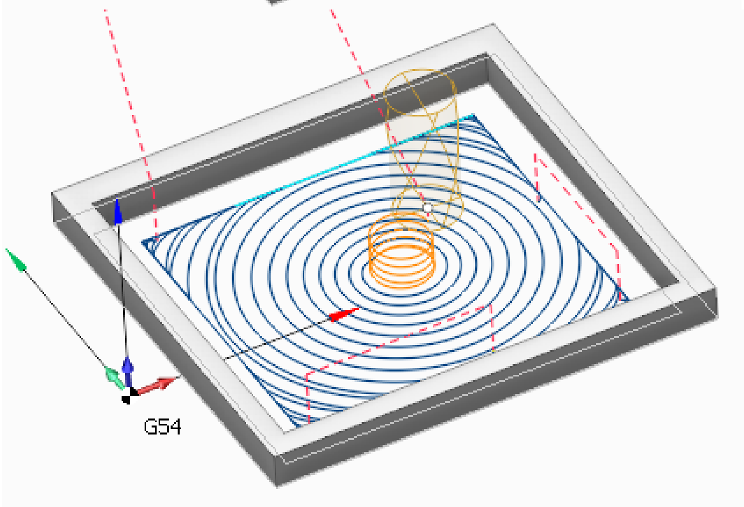


5.5.8 Adaptive SC

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.

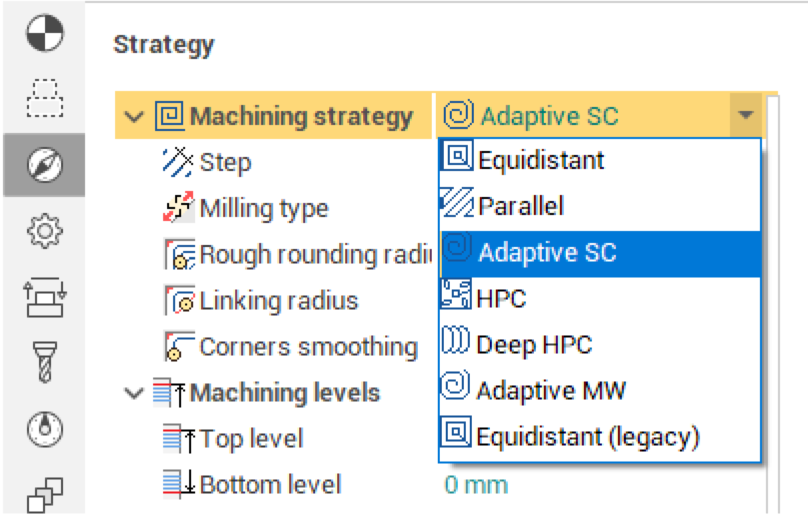




The strategy is available in the following operations:

- 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:



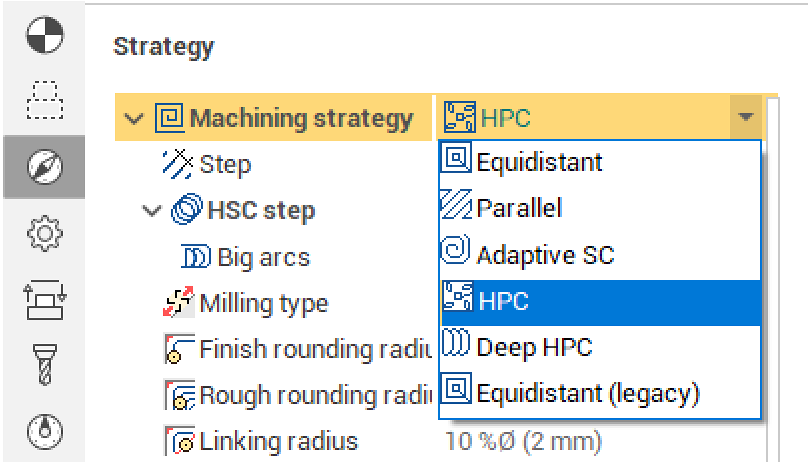
5.5.9 Pocketing strategies

The pocketing strategies are designed for the removing of material in the open and closed pockets.

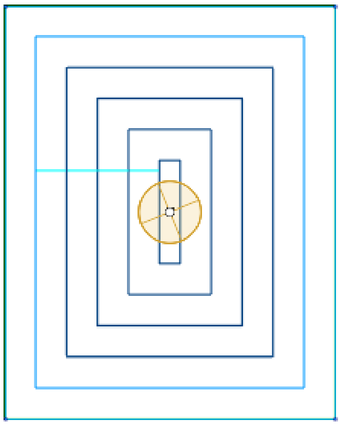
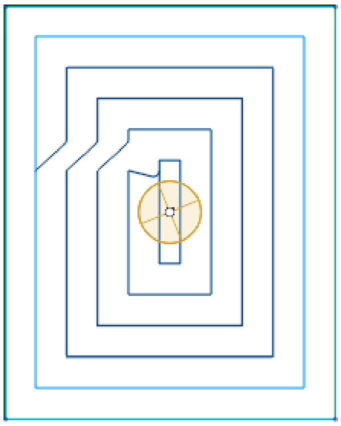
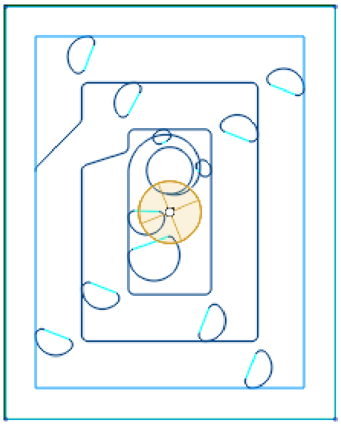
These strategies are available in the following operations:

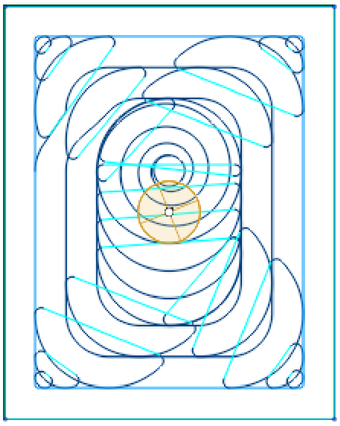
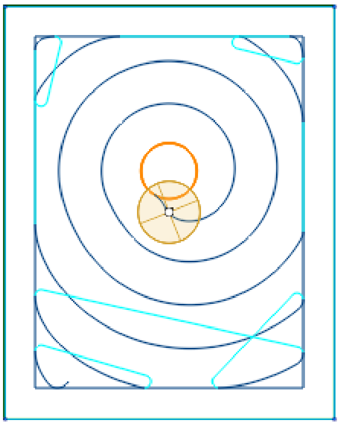
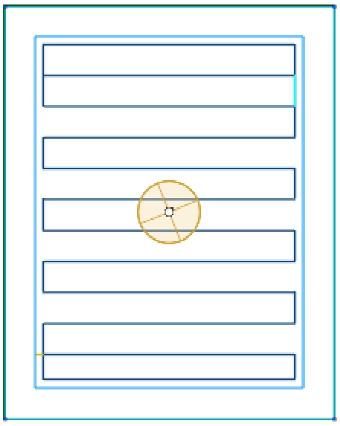
- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

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
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 SC 	**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

5.5.9.1 Features of Adaptive SC 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.

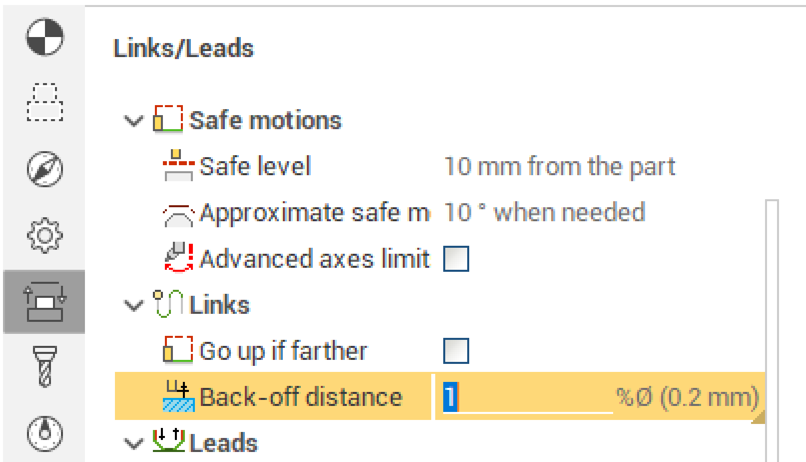
The material is removed in spiral-like fashion. There are no sharp corners in the toolpath. Smoothness of the toolpath is precisely controlled by the dedicated parameters for the roughing rounding radius, the finishing radius and the linking radius. Linking is done preferably in the working plane with 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.

5.5.9.2 How to choose the pocketing strategy

1. The choice number one is **Adaptive SC**. 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.

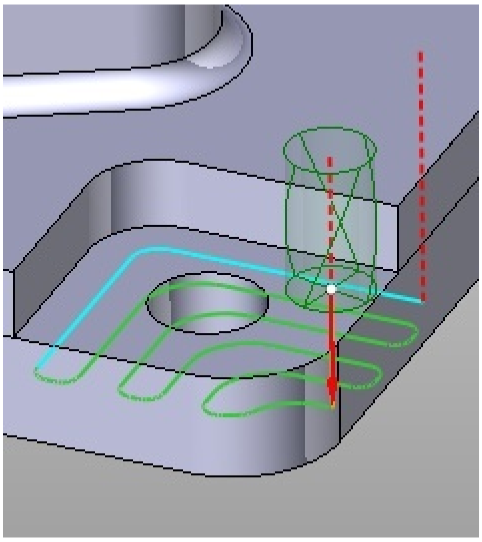
5.5.9.3 Tool path parameters

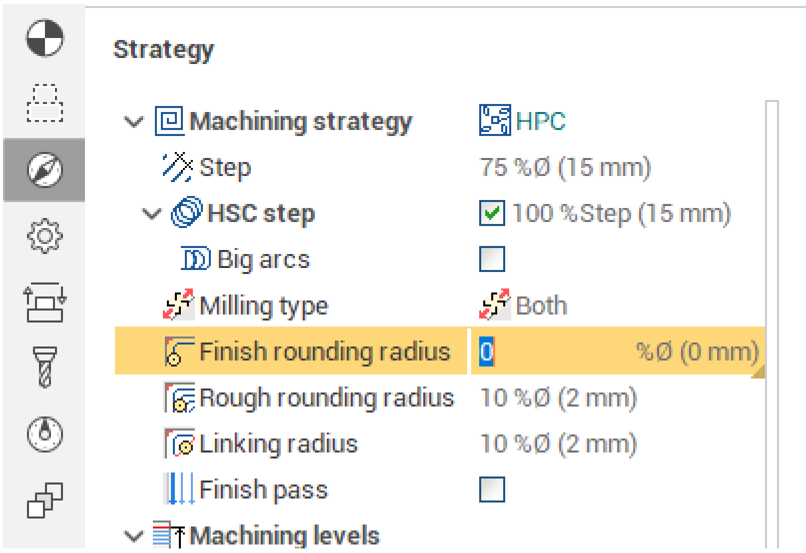
- Back-off distance parameter



The tool can be lifted above the already machined surface when it moves to the next trochoidal arc start position.

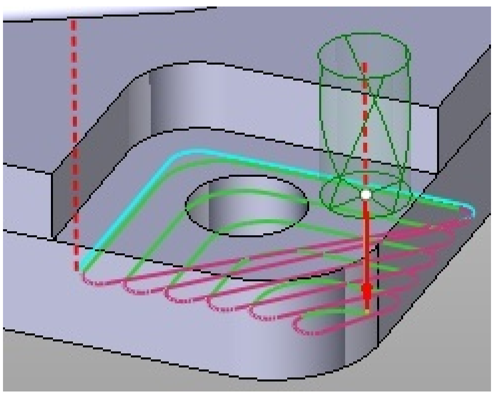
- Rounded links in zigzag mode





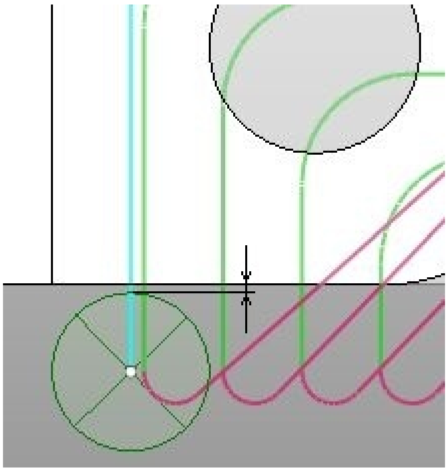
The ‘Finish rounding radius’, ‘Rough rounding radius’ and ‘Linking radius’ value is used for rounding of the links.

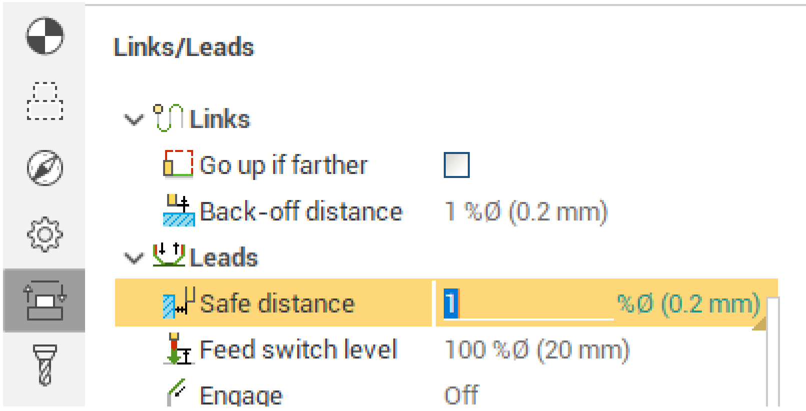
- Links on the same Z-level



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



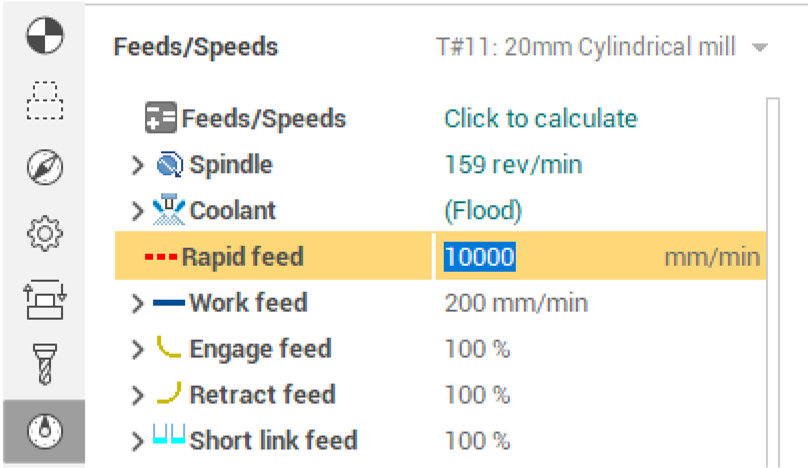


Safe distance is used to move the tool down/up from/to the safe surface.

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.

5.6 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.

SprutCAM X allows to design NC-programs for lathe machines with CNC.

See also:

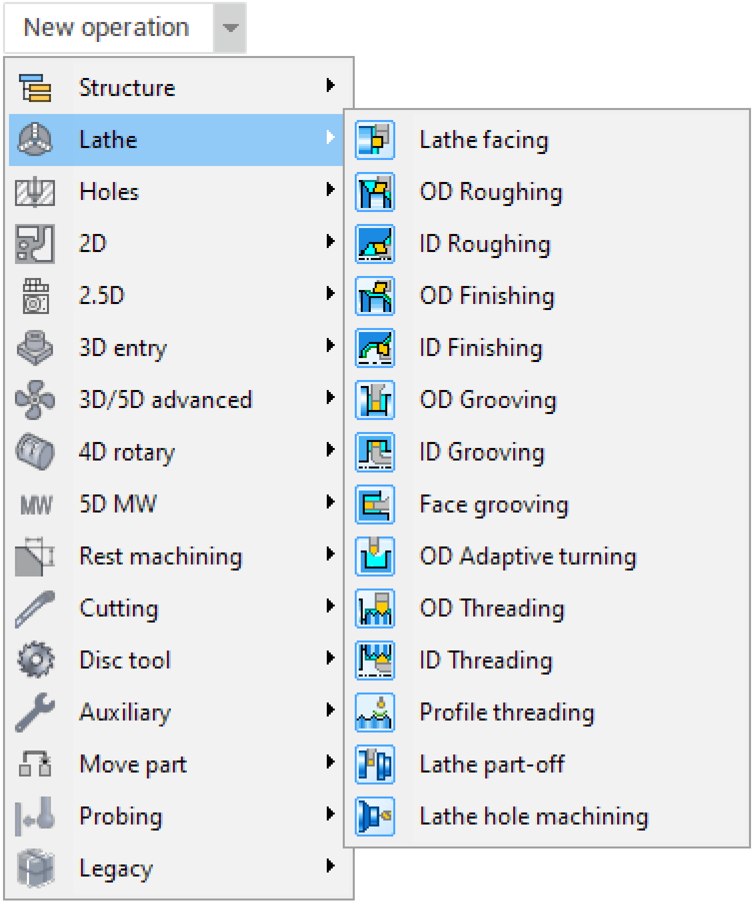
Types of lathe machining operations

Lathe machining operations

5.6.1 Types of lathe machining operations

SprutCAM X 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.



See also:

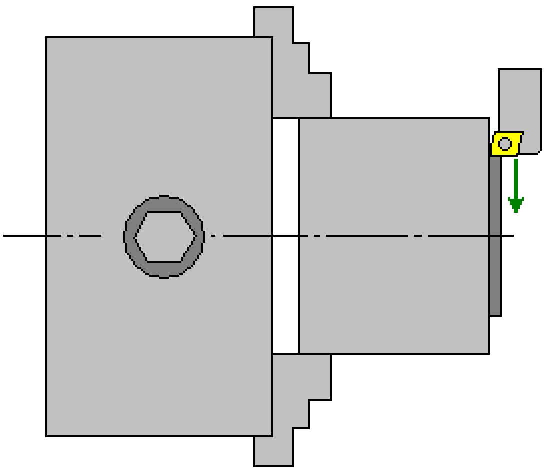
Lathe machining

Lathe machining operations

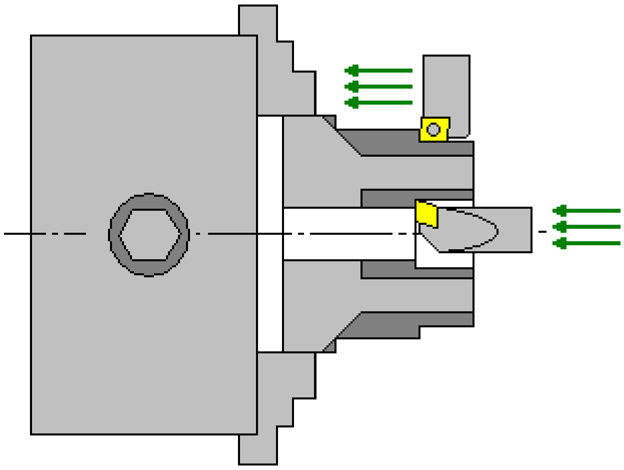
5.6.2 Lathe machining operations

In SprutCAM X lathe machining is grouped by operations:

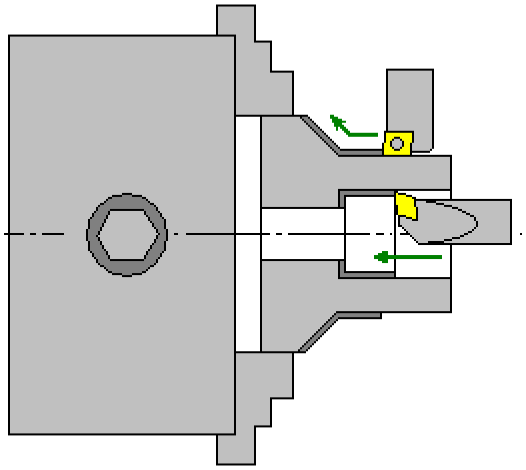
- <Lathe facing>



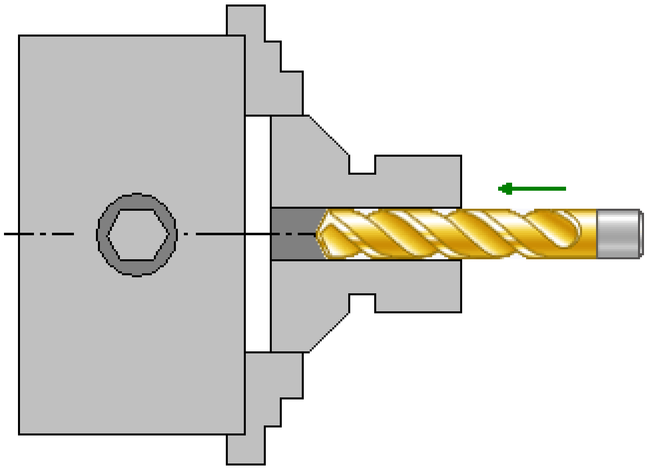
- <OD Roughing, ID Roughing>



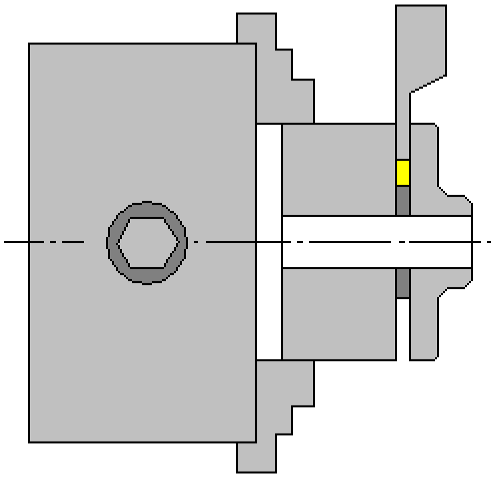
- <OD Finishing, ID Finishing>



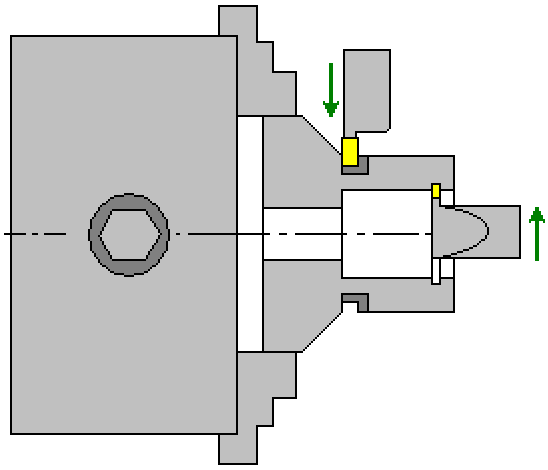
- <Lathe hole machining>



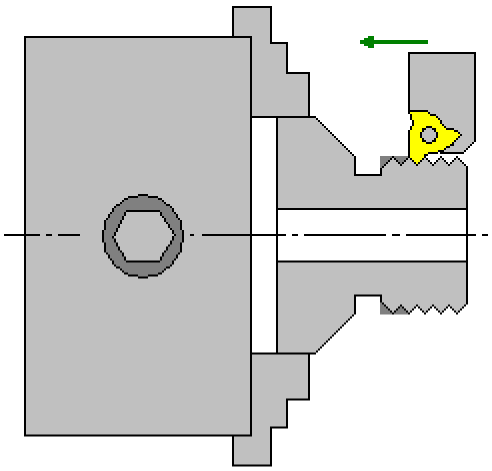
- <Lathe part-off>



- <OD Grooving, ID Grooving, Face grooving>



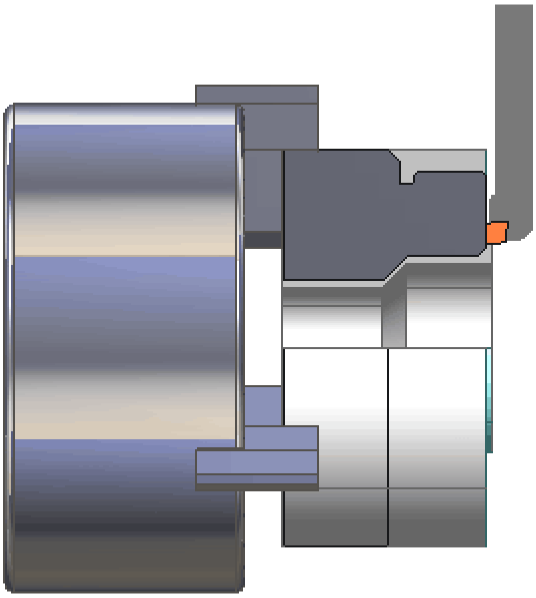
- <OD Threading, ID Threading>



All operations, which were inherited from the previous version moved to the Legacy group. See documentation for the previous version of SprutCAM to get more info about these operations.

See also:

5.6.2.1 Lathe facing operation



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.

Only one type of cycle allowed inside job assignment of this operation - Facing cycle. See the page of cycle parameters for detailed description of each strategy and Job assignment definition page for geometrical properties.

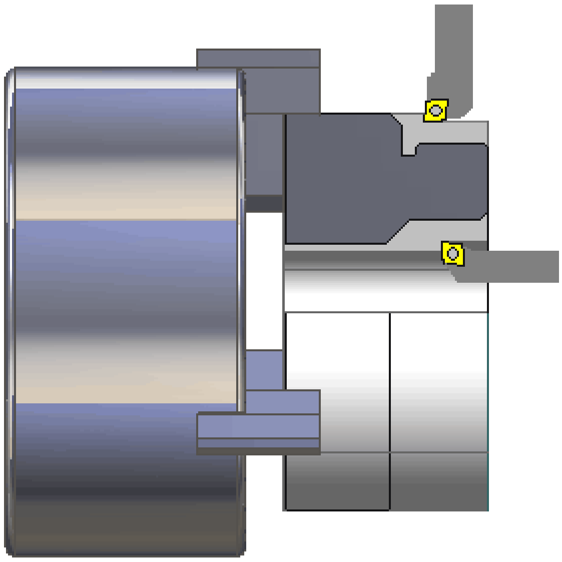
See also:

Lathe machining operations

Facing cycle

Job assignment

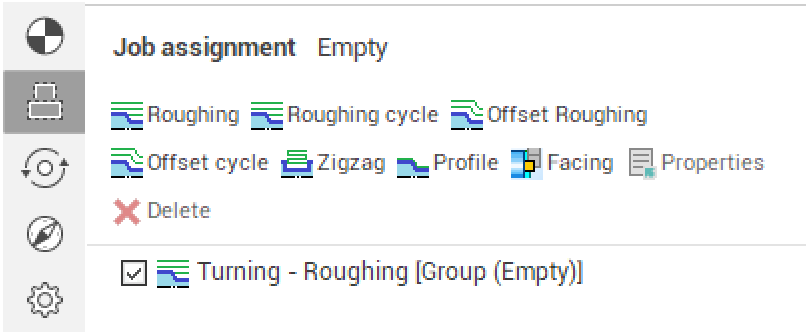
5.6.2.2 OD Roughing and ID Roughing operations



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.

To define the working zone just open Job assignment page, select geometrical primitives that you want to machine and click to the button with the strategy you want to use. After that you can drag start/end point of contour on the screen directly to reduce job zone.



The following strategies allowed for roughing operations:

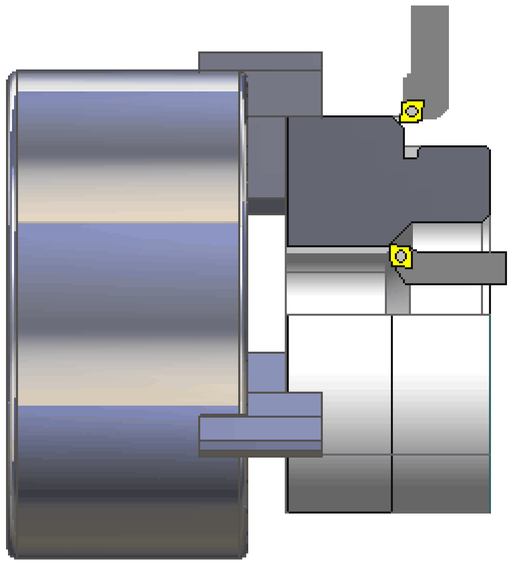
- Roughing;
- Roughing cycle;
- Offset roughing;
- Offset cycle;
- Zigzag;
- Profile;
- Facing.

See the page of cycle parameters for detailed description of each strategy and Job assignment definition page for geometrical properties.

See also:

Lathe machining operations

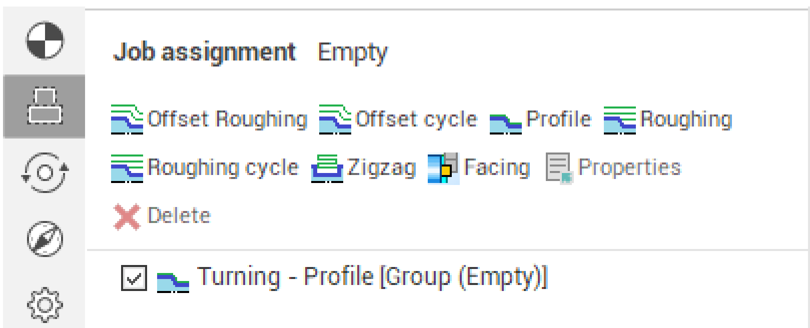
5.6.2.3 OD Finishing and ID Finishing operations



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.

Tool path can be generated without workpiece checking. This mode can be used after the preliminary rough machining or if the initial workpiece is near to the part.

To define the working zone just open Job assignment page, select geometrical primitives that you want to machine and click to the button with the strategy you want to use. After that you can drag start/end point of contour on the screen directly to reduce job zone.



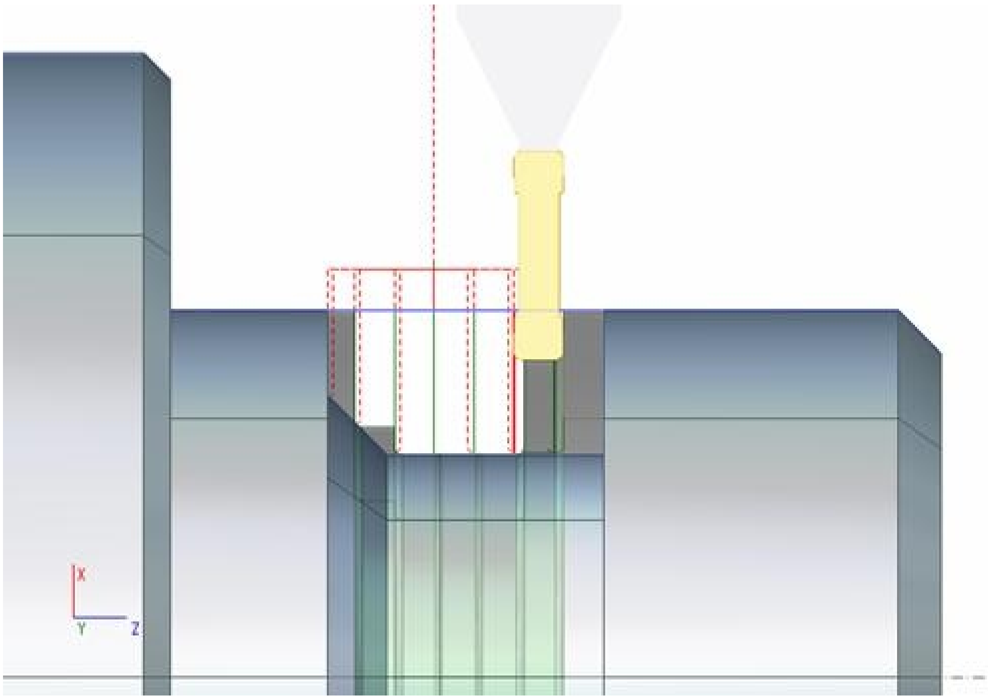
The following strategies allowed for finishing operations:

- Offset roughing;
- Offset cycle;
- Profile;
- Roughing;
- Roughing cycle;
- Zigzag;
- Facing.

See the page of cycle parameters for detailed description of each strategy and Job assignment definition page for geometrical properties.

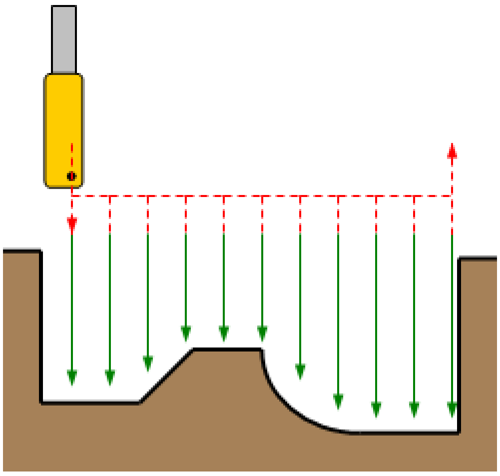
See also:

5.6.2.4 OD Grooving, ID Grooving and Face grooving operations

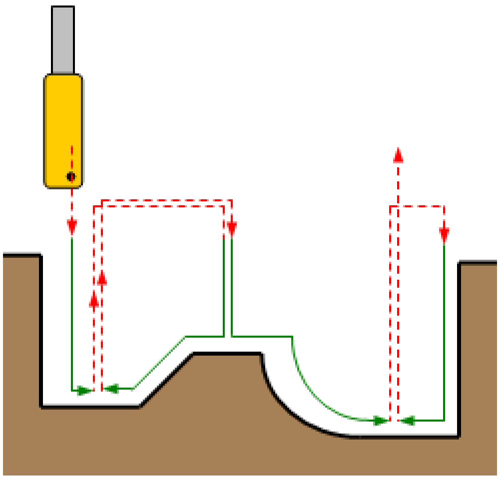


The lathe grooving operations are designed for the machining of the grooves or other zones that can not be made by other way.

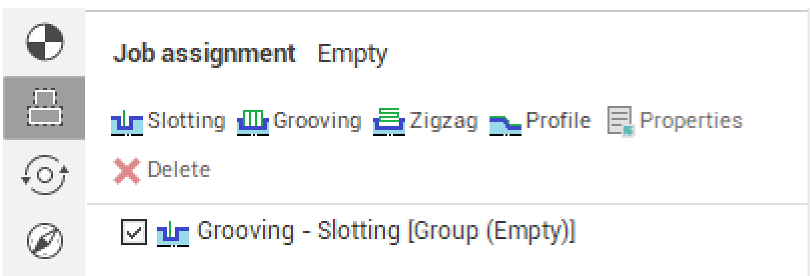
The predominant cutting direction of the groove tools is a radial motion inside part. The cutting by the left or right tool side must be excluded or minimized. The tool path of the grooving operation considers this specific feature of the tool. So the tool path is a series of the strokes that are parallel to the main machining direction. These rough strokes remove the sizeable volume of the workpiece.



The further workpiece shaping is performed by finishing strokes. The finishing strokes consider the specific tool feature also. These strokes remove the scallops after the rough strokes.



To define the working zone just open Job assignment page, select geometrical primitives that you want to machine and click to the button with the strategy you want to use. After that you can drag start/end point of contour on the screen directly to reduce job zone.



The following strategies allowed for grooving operations:

- Slotting;
- Grooving;
- Zigzag;

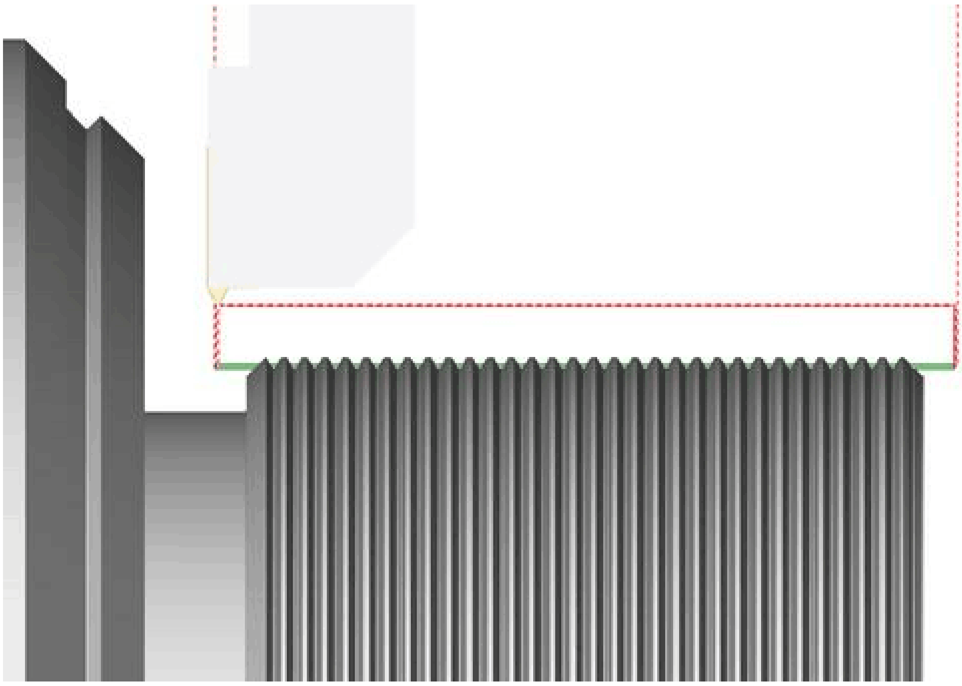
- Profile;

See the page of cycle parameters for detailed description of each strategy and Job assignment definition page for geometrical properties.

See also:

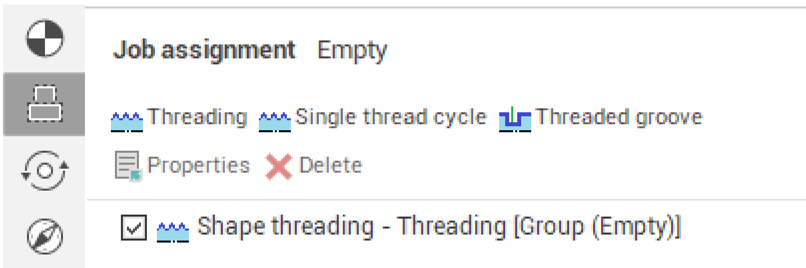
Lathe machining operations

5.6.2.5 OD, ID and Profile threading operations



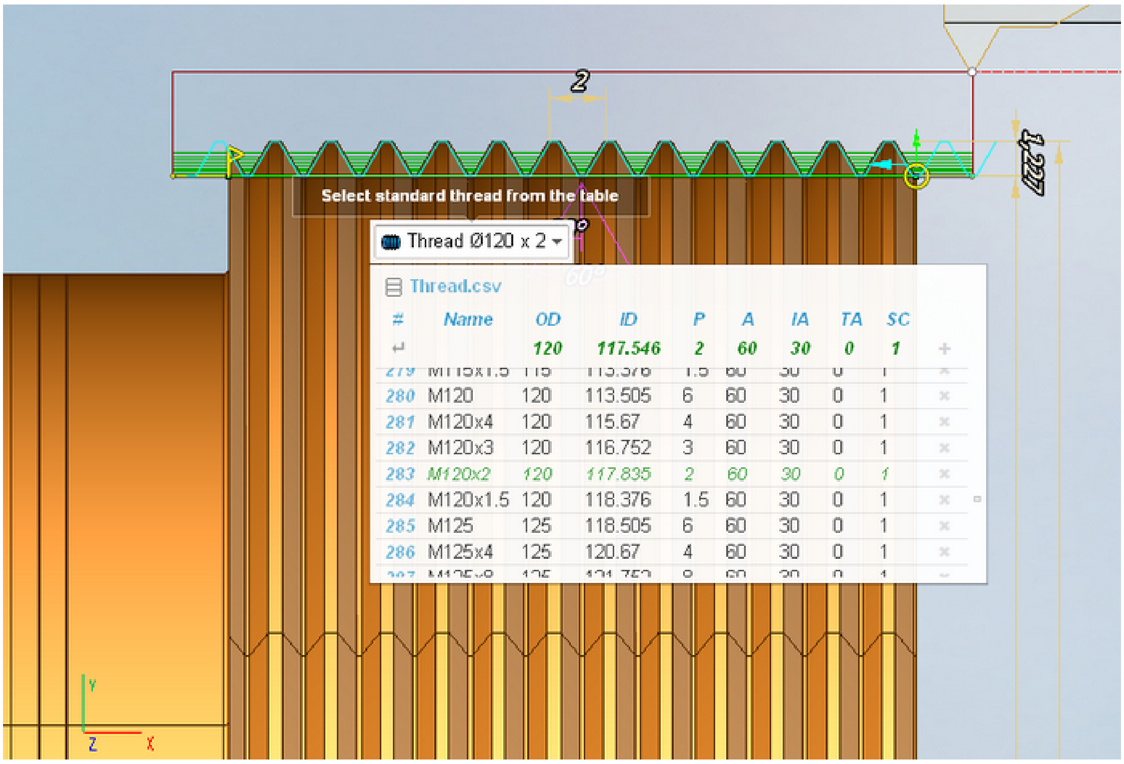
Lathe threading operations are designed for the threading by turn cutter or thread chaser. Helical surface forming is performed by the simultaneous spindle rotation and linear tool motion. Operation allows to thread as standard types of a thread (metric, inch, pipe threads etc.), and threads with a non-standard tooth profile depend on tool shape (diameters, step and angles can be difened manually). Inside the Profile threading operation you can also define an arbitrary threaded groove profile.

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.



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.



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 following strategies allowed for OD and ID threading operations:

- Threading;
- Single thread cycle;
- Thread cycle;

The following strategies allowed for the Profile threading operations:

- Threading for a free form thread;
- Single thread cycle for a free form thread;

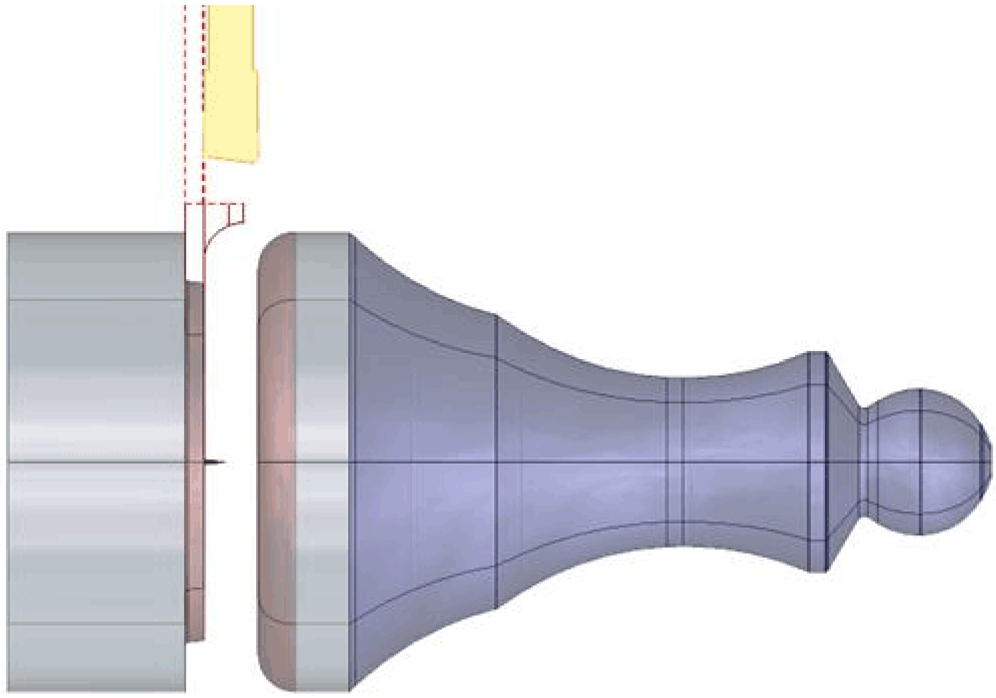
On the threads cycle's properties page can be assigned: thread name, depth, profile angles, number of starts, strategy to cut, number of passes etc. See the page of cycle parameters for detailed description of each strategy and Job assignment definition page for geometrical properties.

The simulation mode allows the checking of the thread shape. The cylindrical grooves are displayed instead the helical surface; because the simulation feature.

See also:

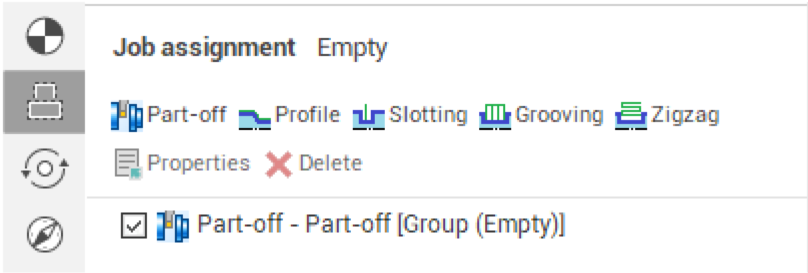
Lathe machining operations

5.6.2.6 Lathe part-off operation

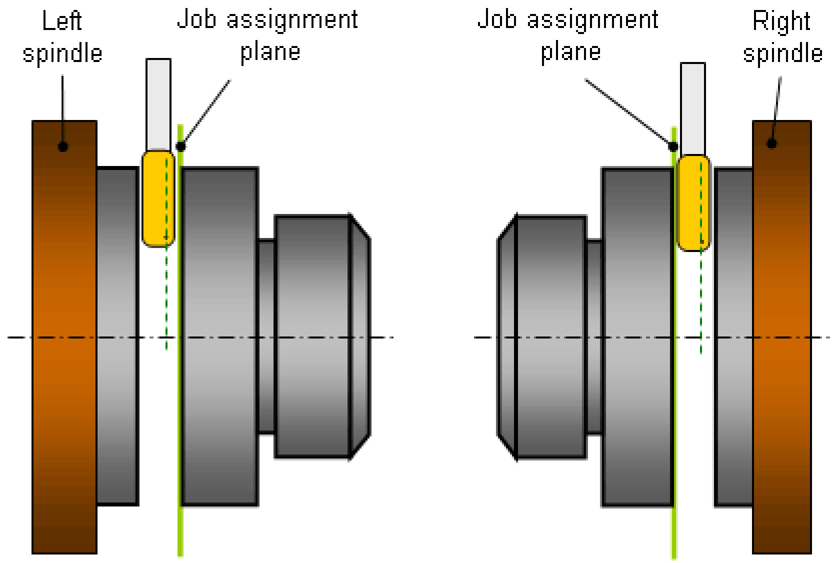


The lathe part-off operation is final operation in the revolutions part machining. It is designed to disjoint the ready machined part from the workpiece and shape the back side of a part.

The Job assignment tab for the operation allows to add Parting off cycle and also some additional machining cycles to be possible to prepare the workpiece before cutting off, for example make groove with the same tool.



The Part-off cycle automatically defines cutting profile at the maximal (left or right) coordinate of the part from highest to lowest point of the workpiece. Left or right side depend on machining side property that seen as green arrow next to the start profile point. Click it to change side.



There is the possibility to create the chamfer or rounding on the external diameter of the part and the special strategy for the chip breaking. This mode allows to set the peck length or the peck count. The delay parameter give the possibility to set the pause in the end of the pecks. The pause time can be assigned in seconds or in revolutions.

The following strategies allowed for part-off operation:

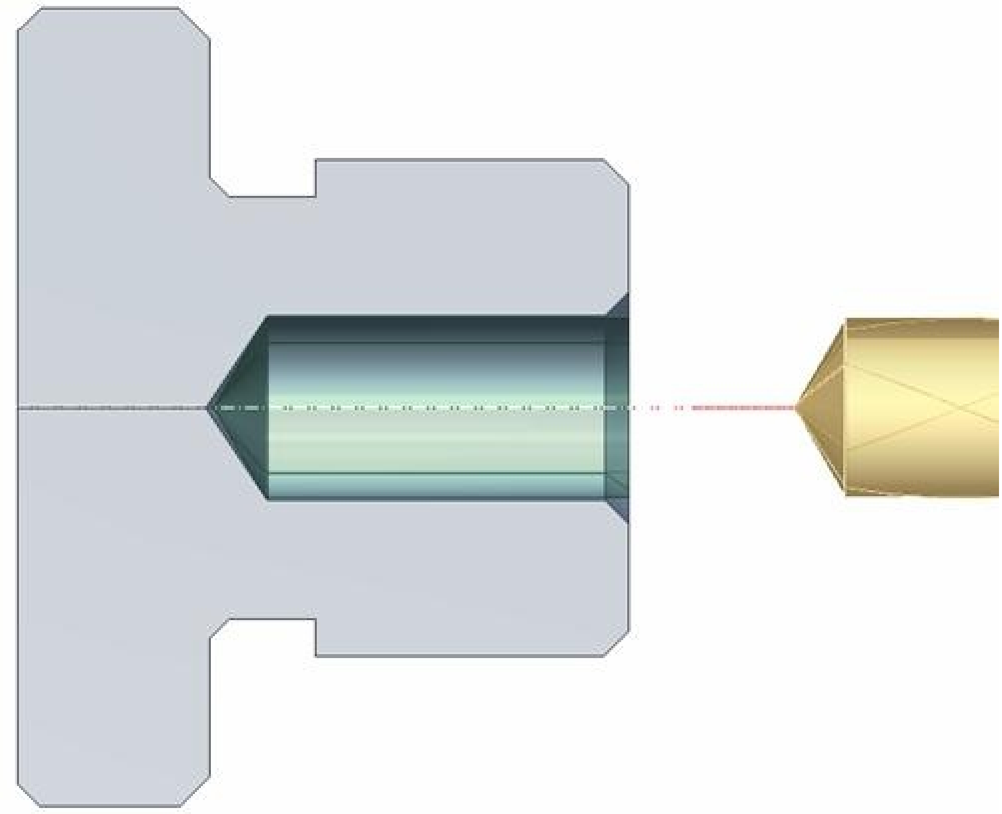
- Parting-off cycle;
- Profile;
- Slotting;
- Grooving;
- Zigzag;

See the page of cycle parameters for detailed description of each strategy and Job assignment definition page for geometrical properties.

See also:

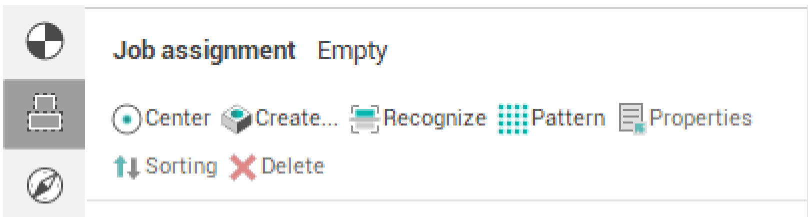
Lathe machining operations

5.6.2.7 Lathe hole machining operation



The operation is designed for the drilling, boring, centering and tapping of the axial hole by the fixed axial tool when the main rotational movement makes the workpiece. So the operation restricted to machine only axial holes.

The hole to machine with its top and bottom levels you can define in the Job assignment window. In the same window the user can access the function of automatic recognition of holes in the model.



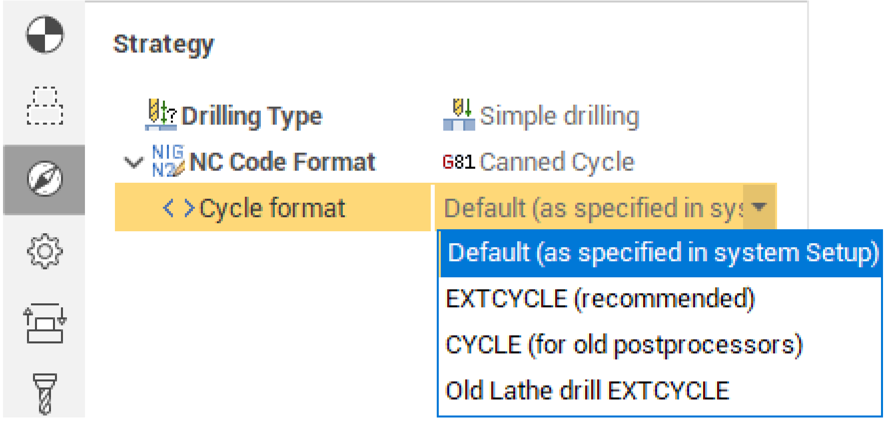
- Center — Create hole by center point
- Create... — Create hole by coordinates input
- Recognize — Automatically recognize holes in the part
- Pattern — Create holes array by pattern
- Properties — Properties of the selected items
- Delete — Delete selected items

The holes added in this window appears in graphical window too. The levels of each hole you can edit directly on the screen with mouse also.

Use operation parameters dialog that is accessed by the parameters button on the <Tool> tab to select the tool for hole machining.

Use <Feeds/Speeds> tab to setup cutting conditions: spindle revolution rate, cooling, feed rates for different motion modes(approach, retract, work feed and the like). Auxiliary transitions (non-cut transitions) are performed either with rapid feed rate or with work feed rate, this option is controlled by the <All non-cut feeds as rapid> check-box. Work feed rate motion for non-cut transitions is useful than machining non-orthogonal plane holes as some NC-systems control only the start and end positions of the tool when performing rapid motions.

Hole machining mode and other additional parameters are set in the properties inspector at the bottom left part of main window.



The dialog interface and parameter list are changing according to selected <Drilling type>.

NC code format defines the way of the g-code output:

- <Long Hang>. All motions are output as the elementary command (lines and circle arcs). Use this option for special cases then machine's CNC-system can't form canned cycle movements (for example, some CNC-systems do not support canned cycles at non-orthogonal planes).
- <Canned cycle>. The cycles is output. Every cycle contains a full set of motions to machine the hole within itself. The way of machining depends on the used CNC. See your CNC manual for the detailed information.

The <Safe distance> parameter defines dimension from the upper level of the hole to the point on the hole axis. The feedrate is switched in this point from the rapid to the work one. So this distance is used to avoid the collision on the rapid feed.

<Return distance> defines the <Return point> under the hole. It is the distance from the upper hole level to the point on the axis. This is the start and end point of the hole machining cycle.

For compatibility with older versions of postprocessors the system provides the ability to change the output format of the drilling cycle (when not expanded toolpath output method is used). In the properties inspector for the hole machining operation is a corresponding parameter <Cycle format>.

This parameter can have the following values.

- <Default (as specified in system Setup)>. The cycle format will be used, which is specified in the system settings. 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 processing routine is not implemented.

- <Old Lathe drill EXTCYCLE>. EXTCYCLE command will be used in the form as it was in old Lathe drilling operation.

See also:

Lathe machining operations

The ways of the holes machining

Hole machining operations realizes the wide range of the holes machining cycles. The cycle selection is performed on the <Strategy> page of the parameters dialog.

The next cycles are supported:

- Drilling cycle (G81, W5DDrill(481))
- Drilling cycle (G82, W5DFace(482))
- Drilling with chip removing cycle (G83, W5DChipRemoving(483))
- Drilling with chip breaking cycle (G73, W5DChipBreaking(473))
- Tapping cycle, tapping with chip breaking and removing (G84, W5DTap(484))
- Drilling cycle (G85, W5DBore5(485))
- Drilling cycle (G86, W5DBore6(486))
- Drilling cycle (G87, W5DBore7(487))
- Drilling cycle (G88, W5DBore8(488))
- Drilling cycle (G89, W5DBore9(489))
- Thread milling cycle (W5DThreadMill(490))
- Hole pocketing cycle (W5DHolePocketing(491))

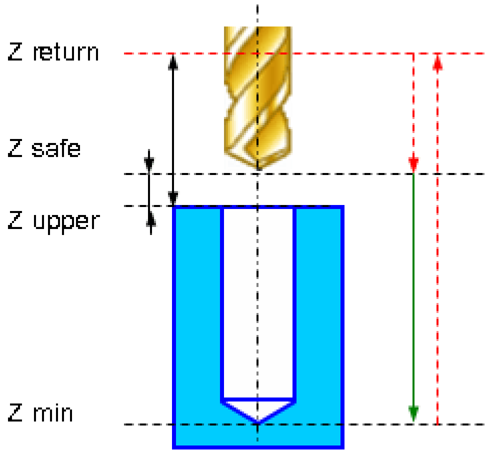
See also:

Types of machining operations

Hole machining operations

Drilling cycle (G81, W5DDrill(481))

Drilling cycle drills holes with rapid approach to the safe level and rapid retract the safe plane level.



Drilling cycle <G81> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.

- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- Rapid tool return to the <Z return> level.

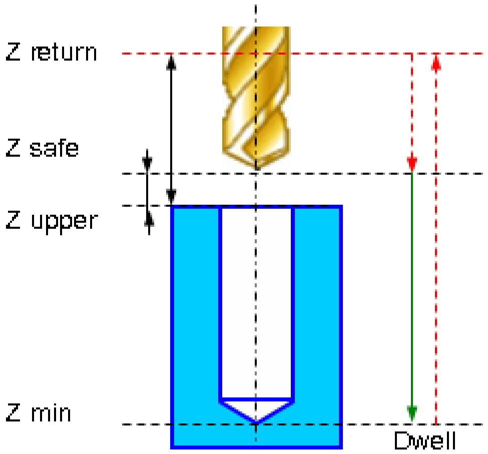
See also:

[Hole machining operation]

[The ways of the holes machining]

Drilling cycle (G82, W5DFace(482))

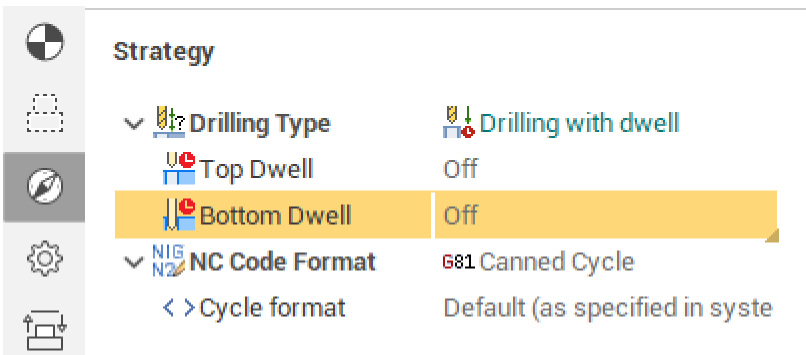
Drilling cycle drills holes with rapid approach to the safe level, dwell at hole bottom level and rapid retract the safe plane level.



Drilling cycle <G82> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- <Dwell> at the <Z min> level.
- Rapid tool return to the <Z return> level.

The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



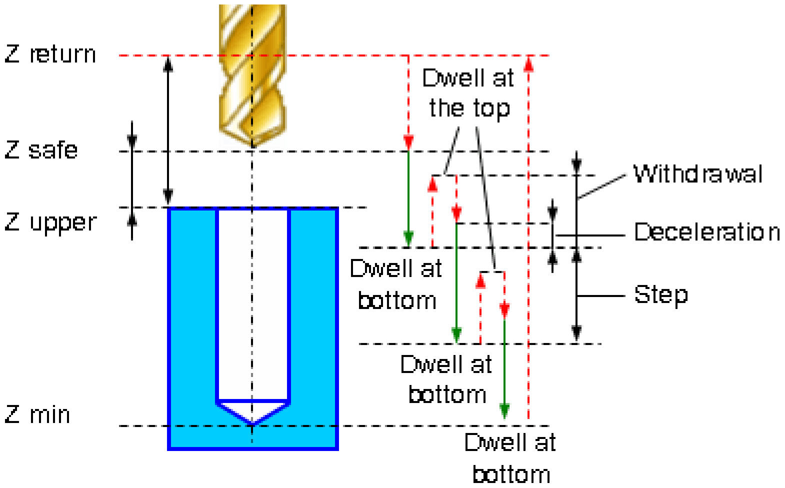
See also:

Hole machining operation

The ways of the holes machining

Drilling with chip breaking cycle (G73, W5DChipBreaking(473))

Drilling with chip breaking cycle performs tool approach to the hole center at the <Z return level>. When cyclic drilling is performed with tool retraction for chip breaking.



The cycle consists of:

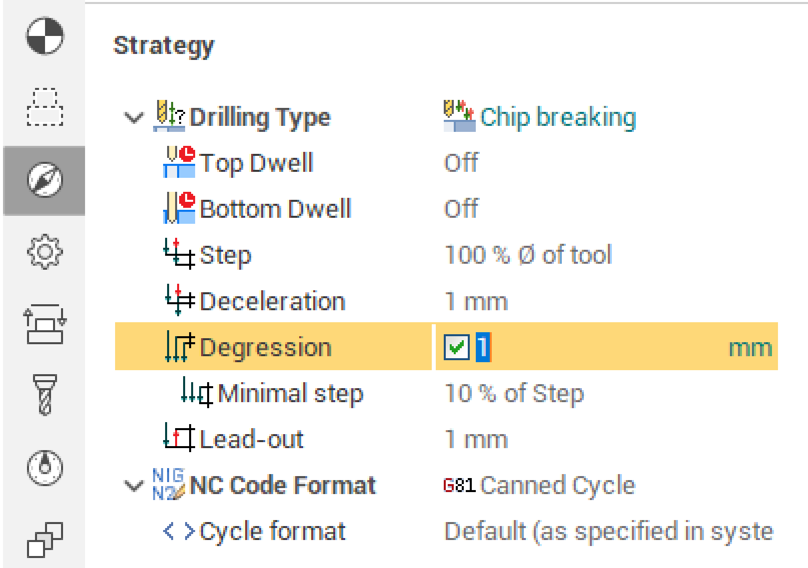
- Rapid approach to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate motion to the <Step depth S>.
- Dwell for the <Delay> at the bottom time.
- Rapid tool retraction for the <Withdrawal distance (Ld)>.
- Dwell for the <Dwell at the top> time.

- Rapid motion to the previous depth level, with a <Deceleration (Dcl)>.
- Work feedrate to the <Deceleration (Dcl)> with <Step (S)>.
- Dwell at for the <Dwell at the bottom> time.
- Repeat previous five iterations until the full hole depth is reached.
- Rapid tool return to the <Z retract> level.

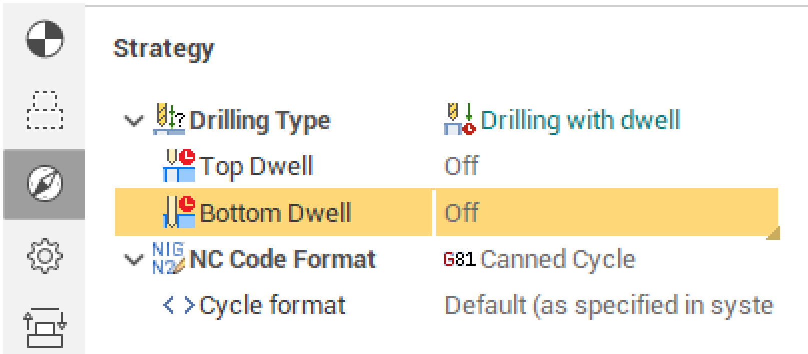
<Chip breaking parameters> panel defines the <Step>, <Deceleration> and <LeadOut>. The step can be specified by different ways:

- <Distance>. The step is equal to the input value.
- <Count>. The value defines the quantity of the tool pecks. The step is calculated as the hole depth divided into the peck count.
- <Percent>. The step is specified in the percent of the tool diameter.

If the <Depth degression> is checked then the depth of every following peck is reduced on the defined value, else the step is constant. The step reduction occurs until its value is not less than <Minimal step>. Minimal step is a percentage of the first step value.



The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



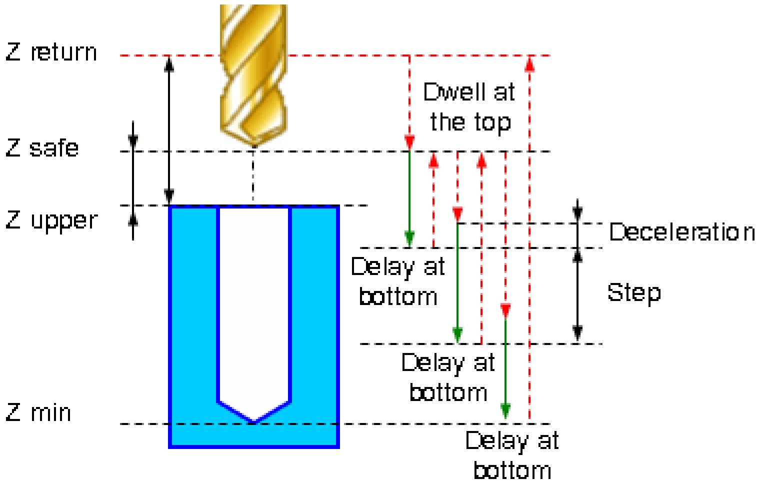
See also:

Hole machining operation

The ways of the holes machining

Drilling with chip removing cycle (G83, W5DChipRemoving(483))

Drilling with chip removing cycle performs tool motion to the hole center at the <Z return> level and consequent cyclic drill with tool retraction to the <Z safe> level.



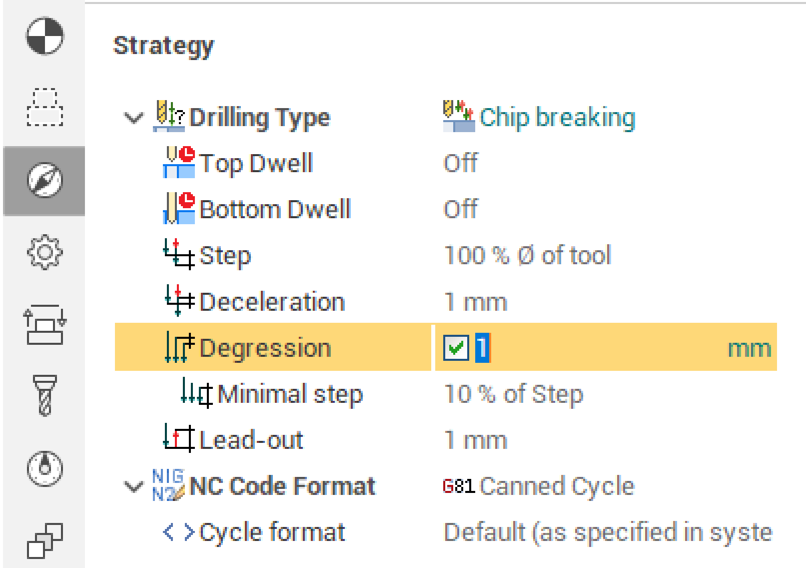
Cycle consists of:

- Rapid approach to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate motion to the <Step depth S>.
- Dwell for the <Delay at the bottom> time.
- Rapid return to <Z safe> level.
- Dwell for the <Dwell at the top> time.
- Rapid motion to the previous depth level, with a <Deceleration (Dcl)>.
- Work feedrate to the <Deceleration (Dcl)> with <Step (S)>.
- Dwell at for the <Dwell at the bottom> time.
- Repeat previous five iterations until the full hole depth is reached.
- Rapid tool return to the <Z retract> level.

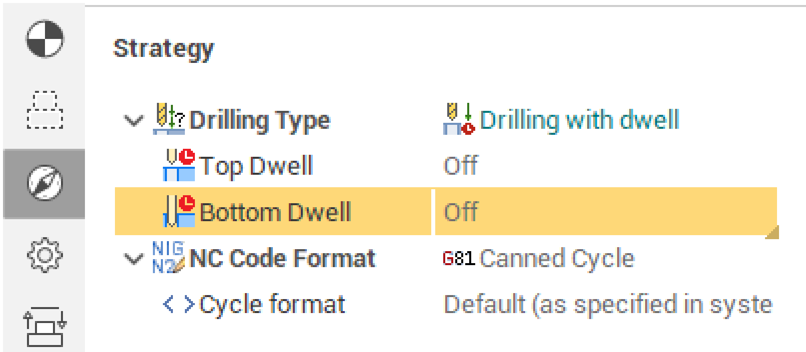
Chip breaking parameters panel defines the step and deceleration. The <Step> can be specified by different ways:

- <Distance>. The step is equal to the input value.
- <Count>. The value defines the quantity of the tool pecks. The step is calculated as the hole depth divided into the peck count.
- <Percent>. The step is specified in the percent of the tool diameter.

If the <Degression> is checked then the depth of every following peck is reduced on the defined value, else the step is constant. The step reduction occurs until its value is not less than <Minimal step>. Minimal step is a percentage of the first step value.



The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



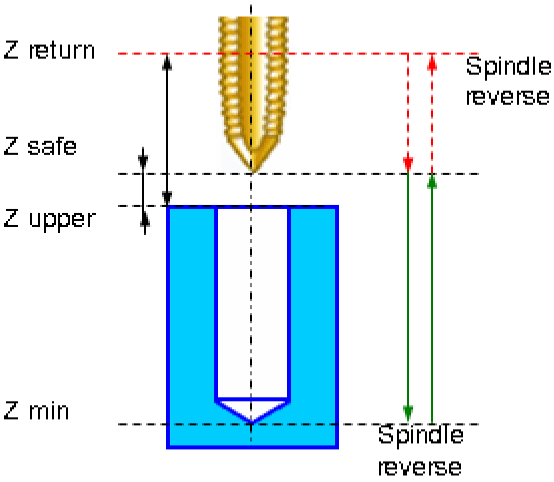
See also:

Hole machining operation

The ways of the holes machining

Tapping cycle, tapping with chip breaking and removing (G84, W5DTap(484))

Tapping cycle performs rapid approach to the <Z return> level, thread tapping with subsequent retraction at work feedrate with reverse spindle rotation.



<G84> tapping cycle includes:

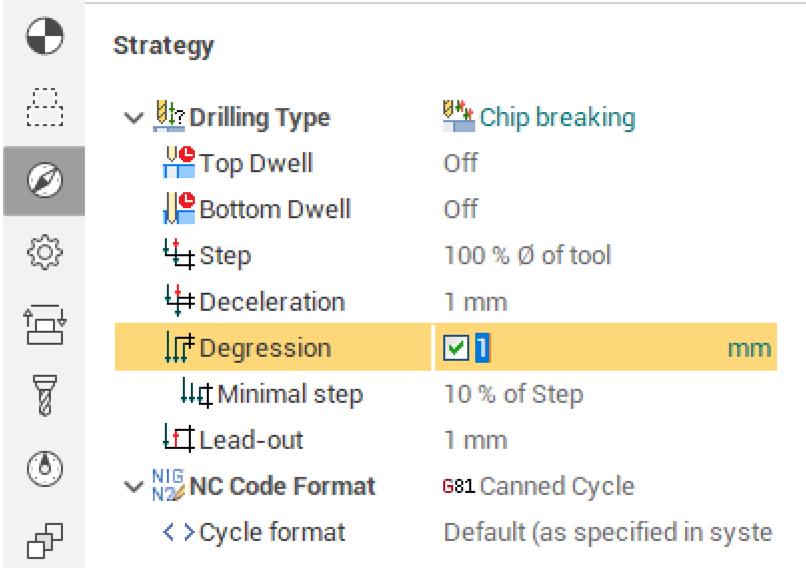
- Rapid approach to the hole center at the <Z return> level.
- Rapid travel to the <Z safe level>.
- Work feedrate motion to the <Z min> level and then <Spindle reverse>. If you select the tapping with chip removing or breaking strategy, the finish depth of the hole will be reached in several iterations.
- Work feed travel to the <Z safe> level.
- Rapid retract to the <Z return> level.
- Restore spindle rotation direction and speed.

The cycle parameters are defined in the <Thread parameters> panel. The <Thread pitch> defines the pitch in millimeters or inches. It depends on the current measurement units. The <Spindle position> is used for the multistart threads and defines the start <Spindle position> in degrees.

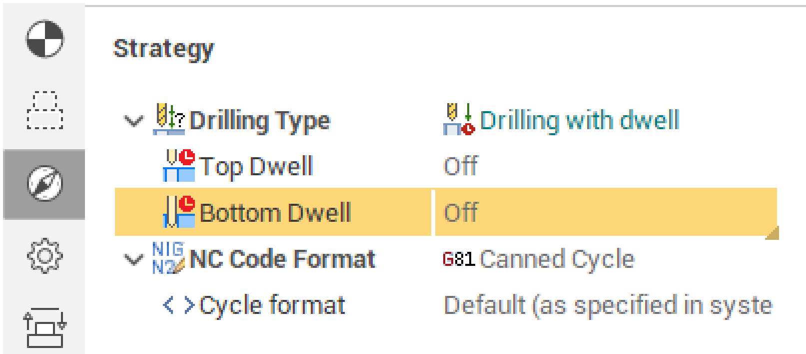
<Chip breaking parameters> panel defines the <Step>, <Deceleration> and <LeadOut>. The step can be specified by different ways:

- <Distance>. The step is equal to the input value.
- <Count>. The value defines the quantity of the tool pecks. The step is calculated as the hole depth divided into the peck count.
- <Percent>. The step is specified in the percent of the tool diameter.

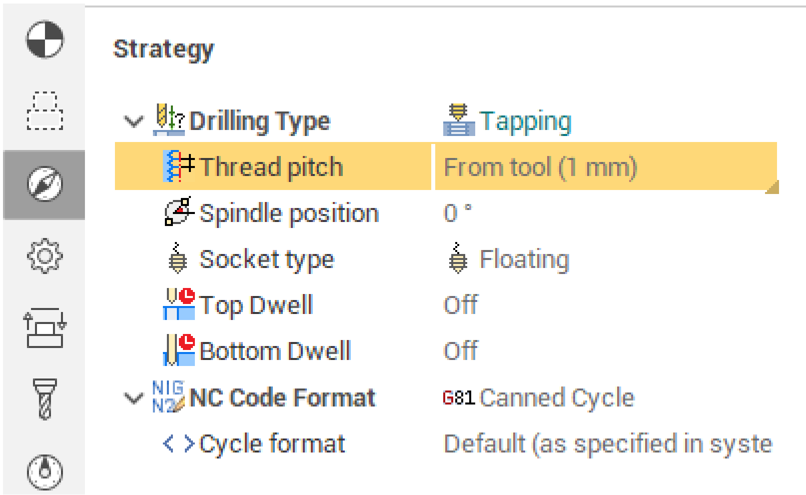
If the <Depth degression> is checked then the depth of every following peck is reduced on the defined value, else the step is constant. The step reduction occurs until its value is not less than <Minimal step>. Minimal step is a percentage of the first step value.



The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



Some numerical controls has different cycles for the different socket type. So the socket type can be defined as floating or fixed.



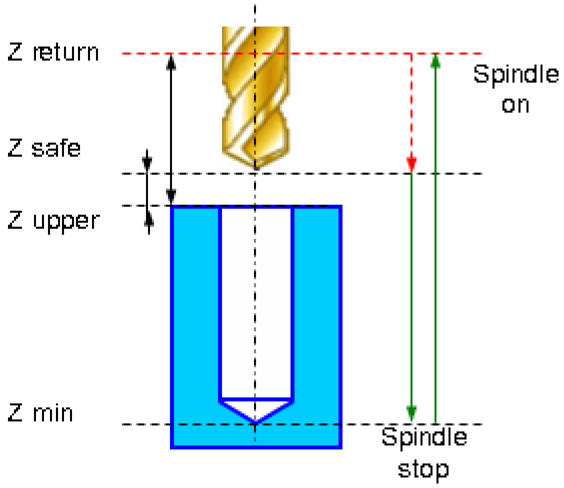
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G85, W5DBore5(485))

Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and work feedrate retract to the <Return> level.



Boring canned cycle <G85> consists of:

- Rapid travel to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the <Z min> level.
- <Spindle stop>.
- Work feedrate return to the <Z return> level.
- Restore spindle rotation direction and speed.

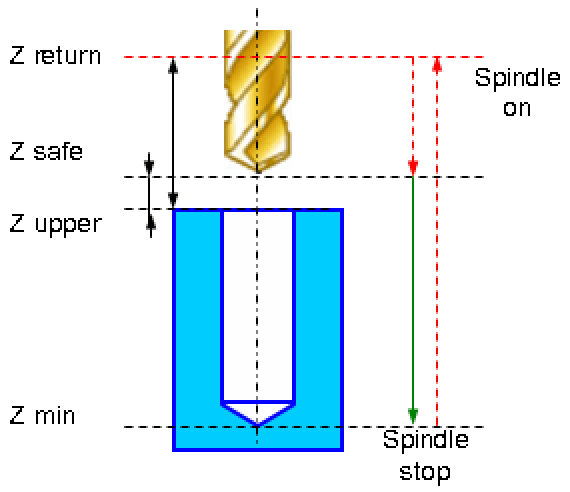
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G86, W5DBore6(486))

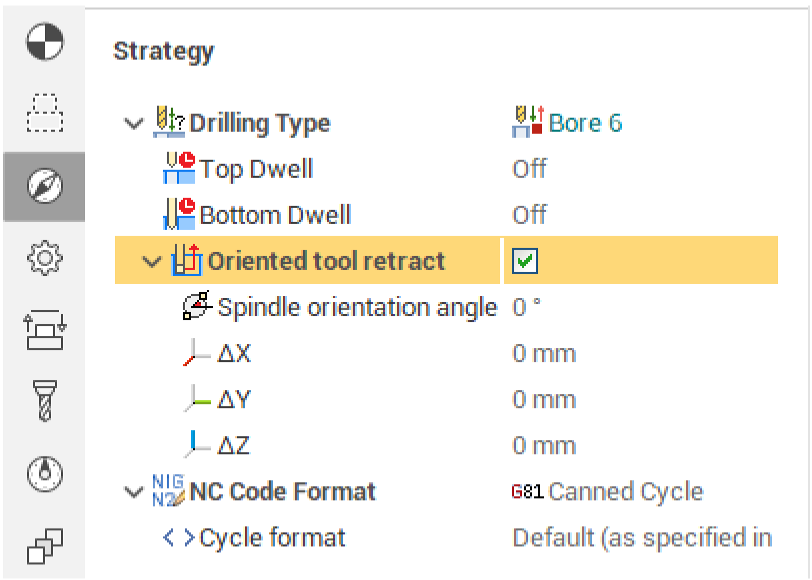
Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and rapid retract to the <Return> level.



Boring canned cycle <G86> consists of:

- Rapid travel to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the <Z min> level.
- <Spindle stop>. If <Oriented 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.
- Rapid return to the <Z return> level.
- Restore spindle rotation direction and speed.

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.



To edit the following parameters are available: the angle of the oriented spindle stop in degrees and the coordinates of tool offset after a stop.

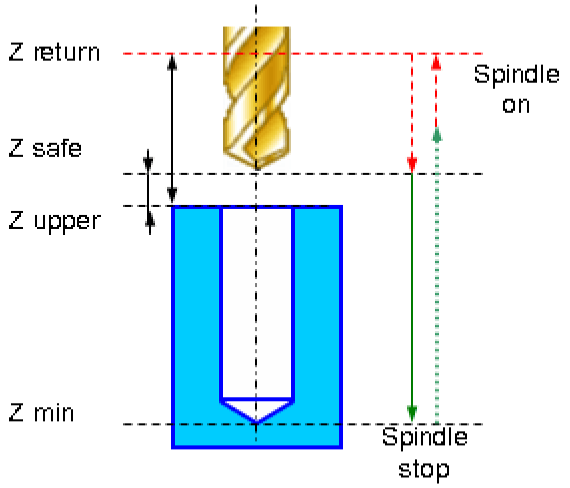
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G87, W5DBore7(487))

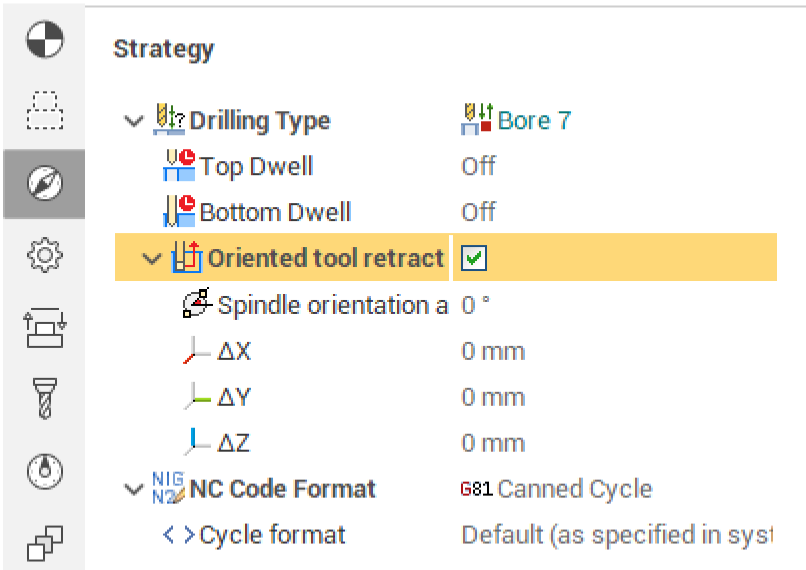
Boring cycle performs tool approach to the hole center, hole boring with stop at minimum level and manual retract to the <Return> level.



Boring canned cycle <G87> consists of:

- Rapid travel to the hole center at the <Z return> level.
- Rapid travel to the <Z safe> level.
- Work feedrate travel to the <Z min> level.
- <Spindle stop>. If <Oriented 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.
- Manual retract to the <Z return> level.
- Restore spindle rotation direction and speed.

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.



To edit the following parameters are available: the angle of the oriented spindle stop in degrees and the coordinates of tool offset after a stop.

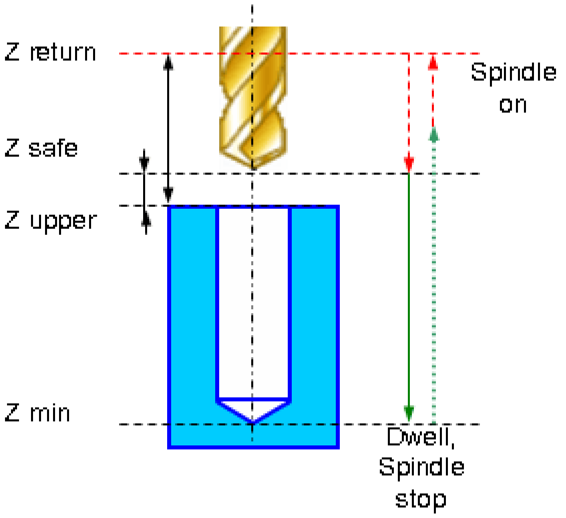
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G88, W5DBore8(488))

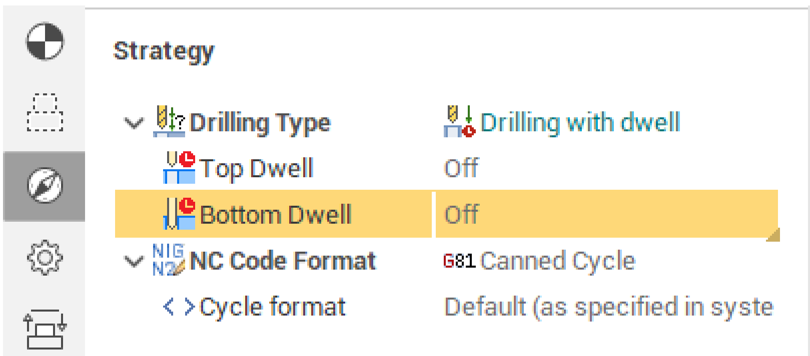
Drilling cycle type drills holes with rapid approach to the safe level, dwell at hole bottom level, spindle stop and manual retract to the safe plane level.



Drilling cycle <G88> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- <Dwell> at the <Z min> level.
- <Spindle stop>.
- Manual tool retract to the <Z return> level.
- Restore the spindle rotation direction and speed.

The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.



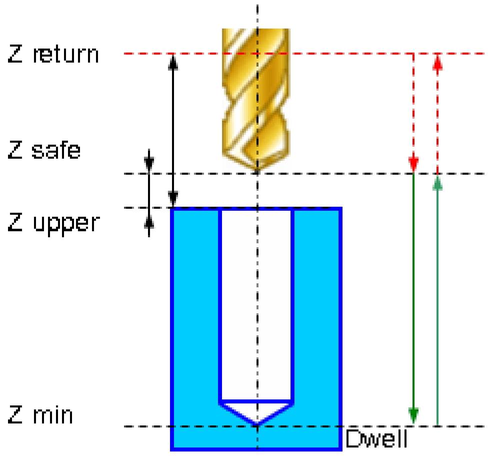
See also:

Hole machining operation

The ways of the holes machining

Drilling cycle (G89, W5DBore9(489))

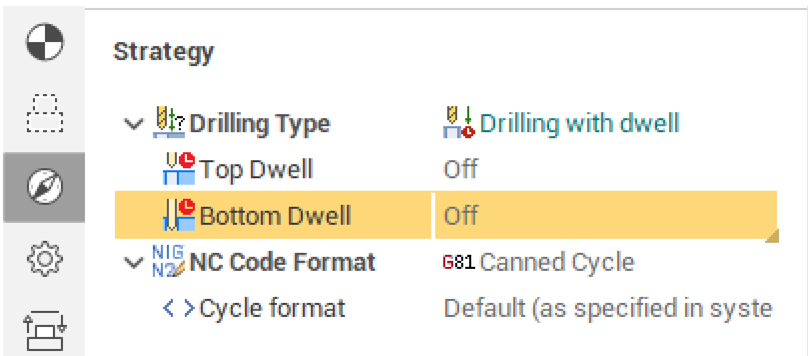
Boring cycle type bores holes with rapid approach to the safe level, dwell at hole bottom level, spindle stop and manual retract to the safe plane level.



Boring cycle <G89> consist of the following steps:

- Rapid tool motion to the hole center at the <Z return> level.
- Rapid descend to the <Z safe> level.
- Work feedrate motion to the <Z min> level.
- <Dwell> at the <Z min> level.
- Work feedrate return to the <Z safe> level.
- Restore the spindle rotation direction and speed.

The time of delay is defined on the <Dwell> panel in the field <Bottom dwell>. The time can be specified in seconds or in the numbers of the tool revolutions. In the last case the time is calculated automatically using the defined tool RPM. Delay is absent if the bottom dwell is off.

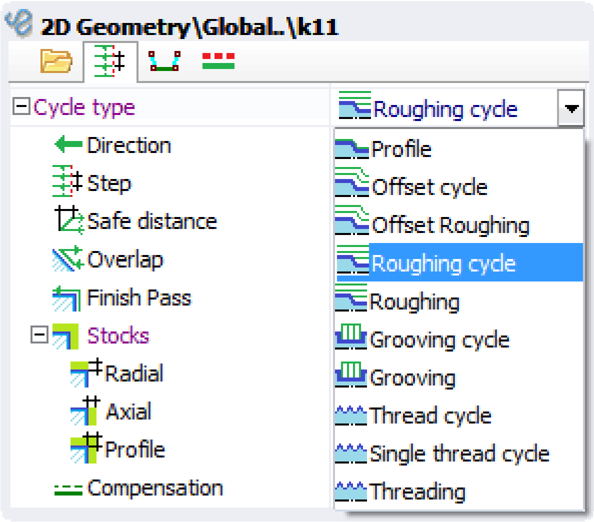


See also:

Hole machining operation

5.6.3 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.



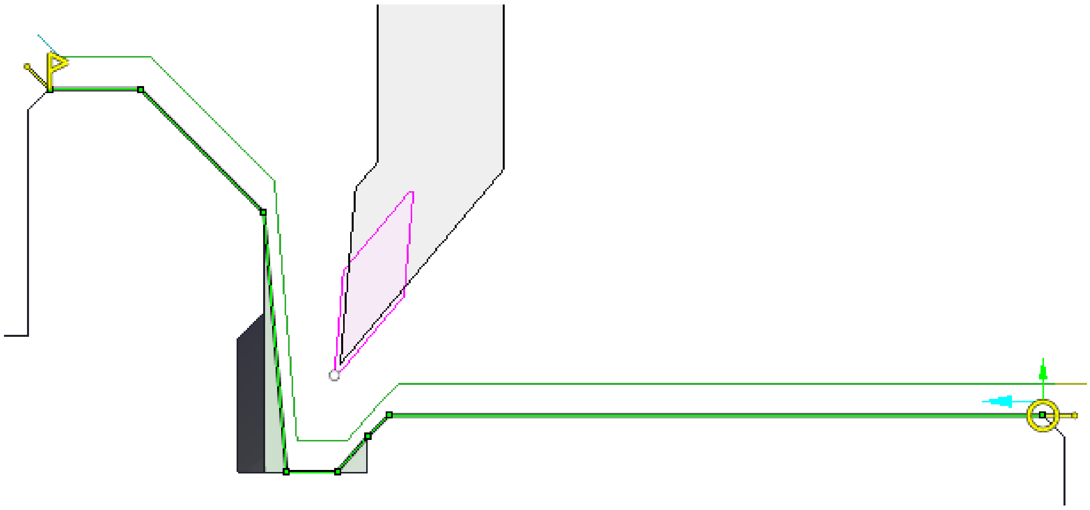
- In the "Profile" mode the defined profile is output into the tool path without any additions.
- Offset cycle generates the cycle ISO G73 based on the defined profile.
- Offset Roughing generates the same tool path like the ISO G73, but it is output in the expanded mode. This mode can check current 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).

See also:

Lathe machining

Lathe job assignment

5.6.3.1 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.

See also:

Lathe Machining

Lathe cycles

Job assignment item properties

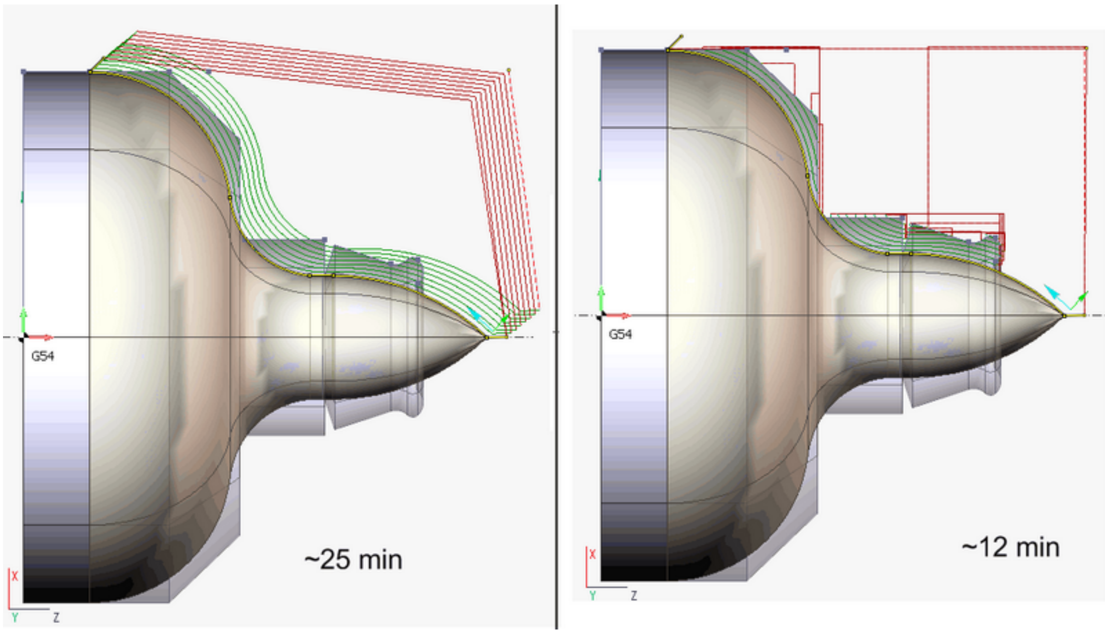
5.6.3.2 Offset Roughing

Offset Roughing generates the same tool path like the ISO G73, but it is output in the expanded mode.

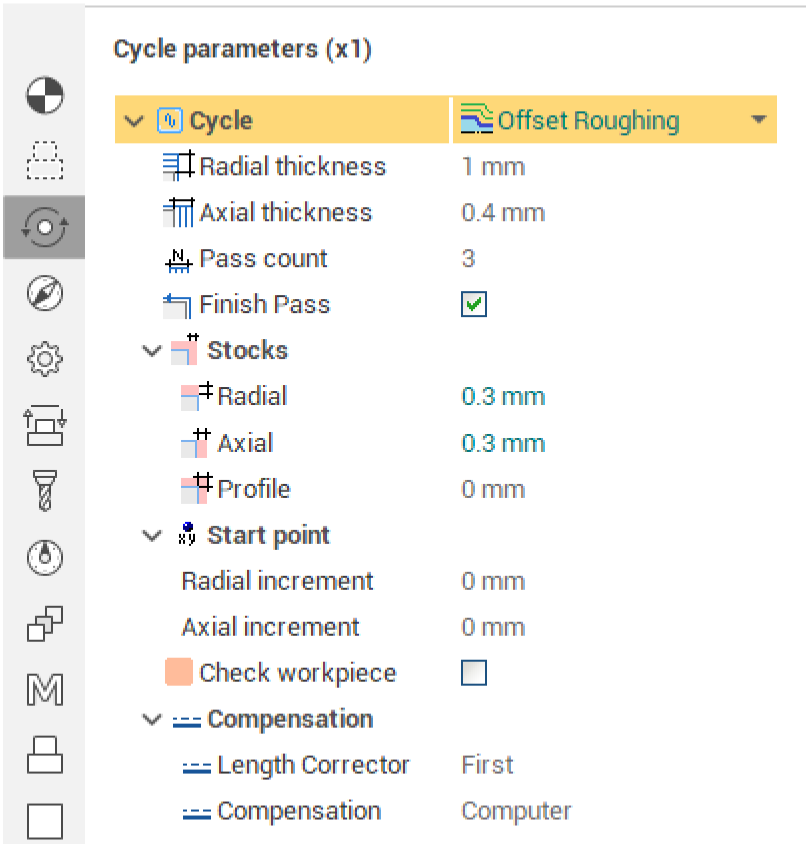
This mode can check current workpiece state.

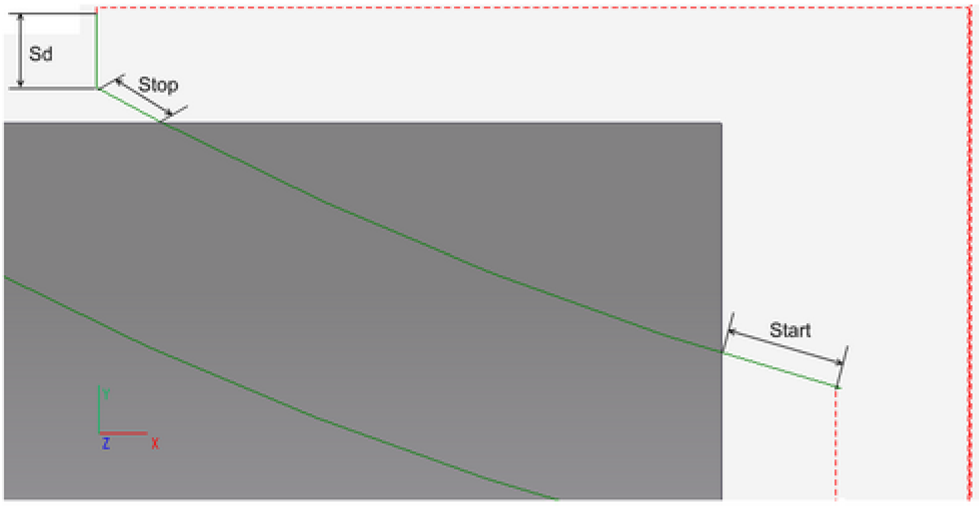
Checking the current state of the workpiece can reduce machining time. In the picture is represented the item. Mach

The use of algorithms based on the current state of the workpiece will reduce the processing time of this part by 50%



The workpiece checking parameters can be defined in the properties window.





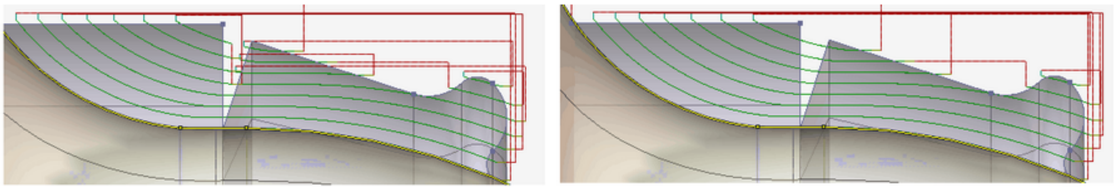
Sd — Safe distance

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

If the distance between two areas is less than **Ignore thickness** parameter, the system will treat them as one area.

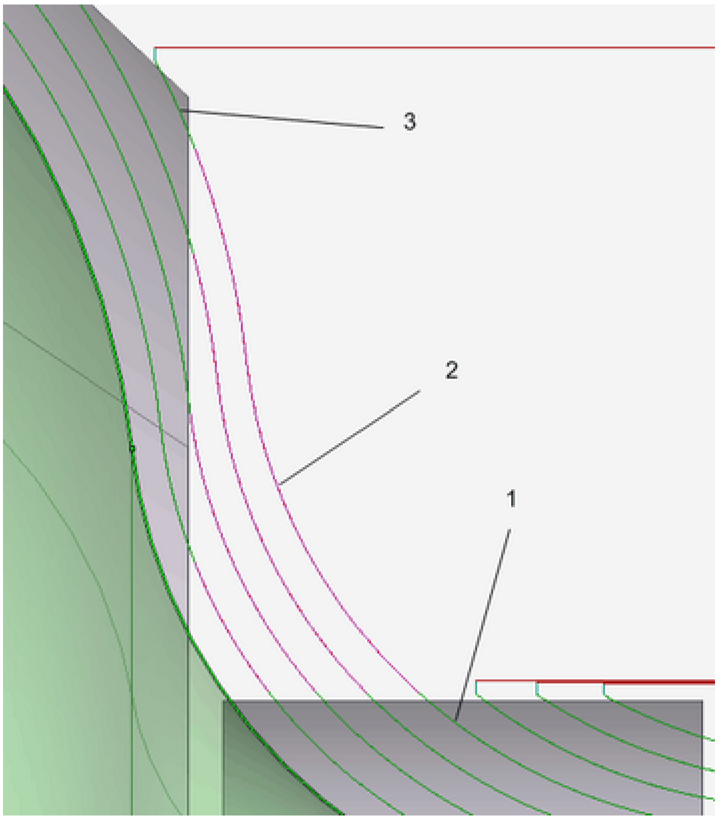


On the left side of the picture **Ignore thickness** parameter is less than the distance between the areas of machining,

On the right side of the picture **Ignore thickness** parameter more than the distance between areas of machining, th

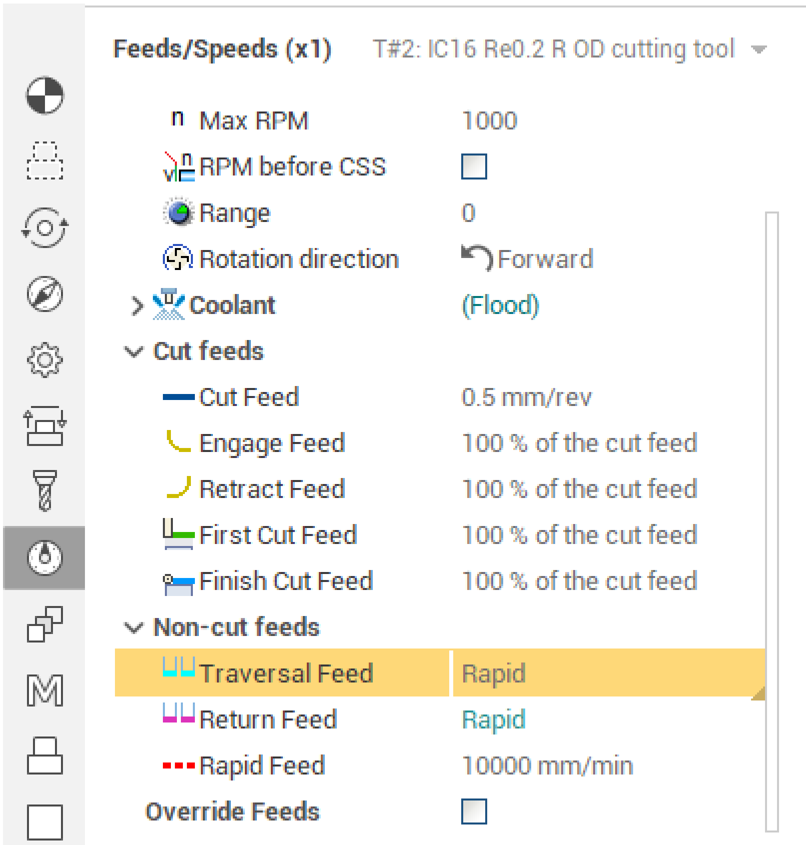
In some cases it is advisable to treat several areas as one, but move to the second area on the high feed with enable

Tool machines the first area (1) at the working feed, then moves to the second area (3) at traversal feed by the profil



The **Maximum traversal distance** specifies the maximum distance that the tool can pass on the traversal feed. If the distance bet

The system provides the ability to set the feed values for a particular profile.



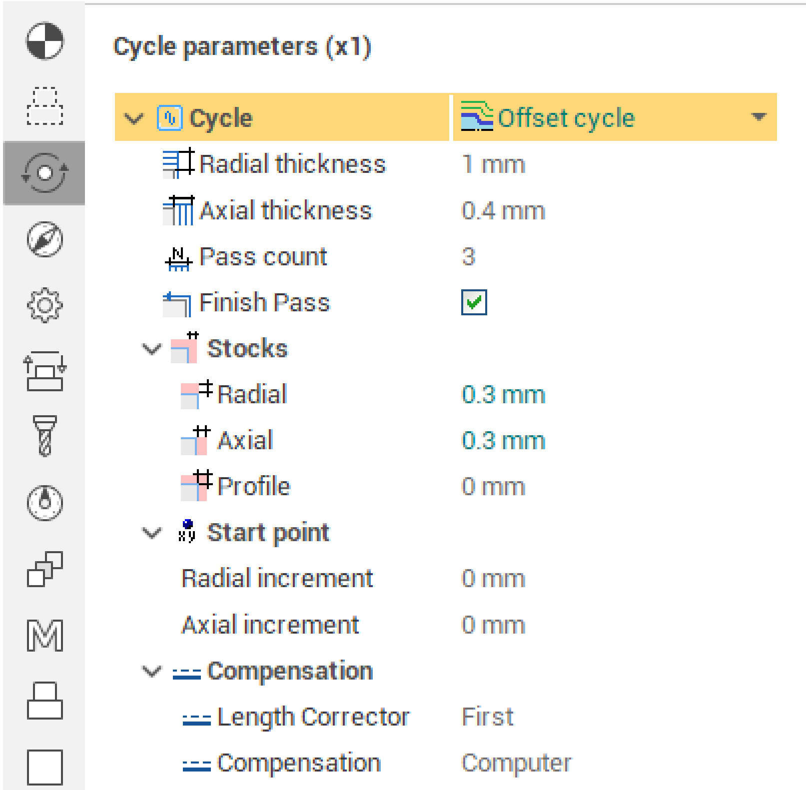
See also:

Lathe Machining

Lathe cycles

5.6.3.3 Offset cycle (ISO G73)

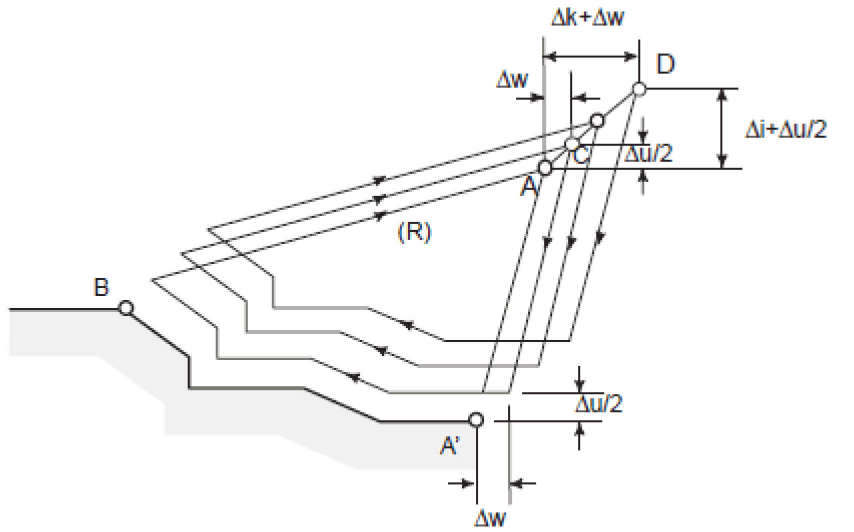
Offset cycle generates the canned cycle G73 based on the defined profile. The cycle parameters can be defined in the properties window.



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.

See also:

Lathe Machining

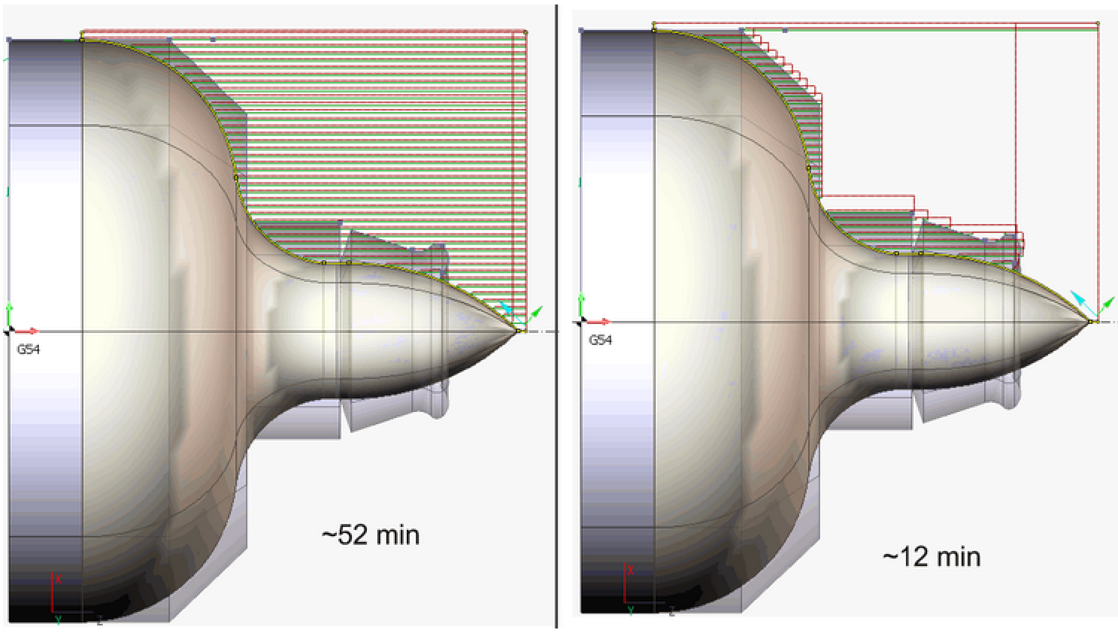
Lathe cycles

5.6.3.4 Roughing

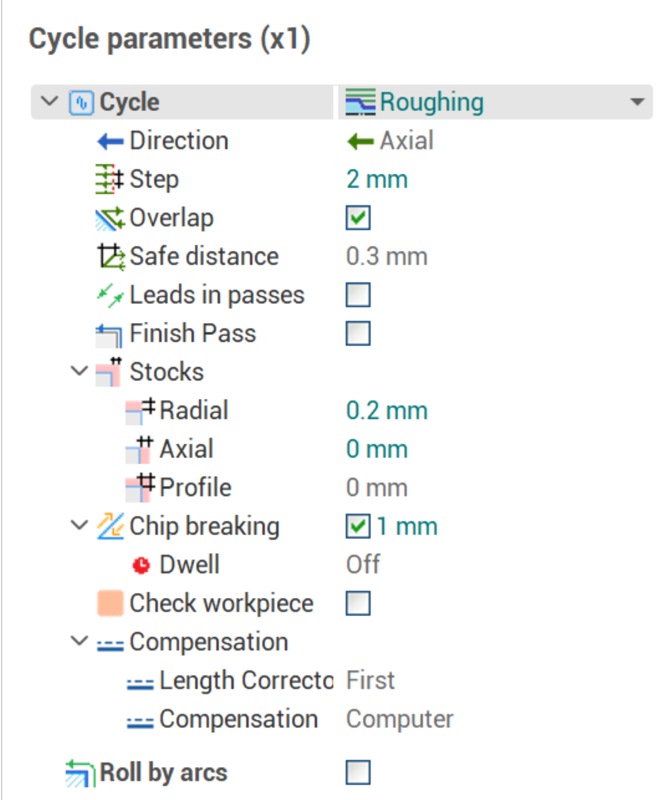
Roughing based on ISO G71/G72, but it is output in the expanded mode.

This mode can check current workpiece state.

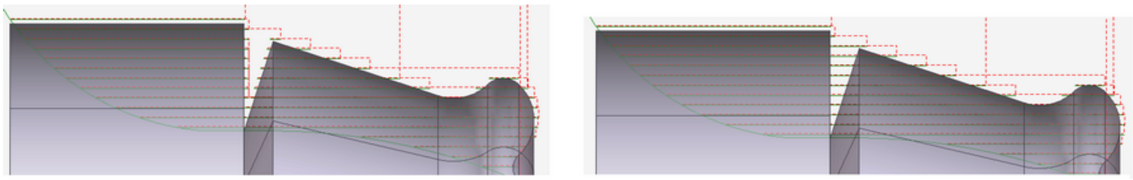
Checking the current state of workpiece can reduce machining time. In the picture is represented the item. Machining takes approximately 52 minutes. The use of algorithms based on the current state of workpiece will reduce the processing time of this part by 80%, excluding the tool passes through the air at the working feed.



The workpiece checking parameters can be defined in the properties window..

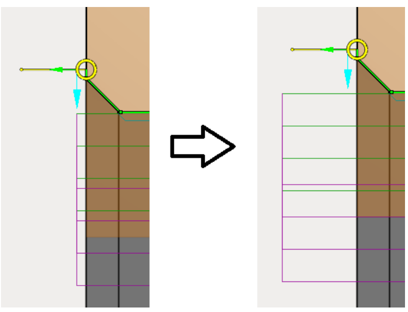


In some cases it is advisable to ignore the fact that tool passes through the air, if the distance of this pass is quite small. **Ignore thickness** parameter allows to specify the maximum distance of the pass through the air at the working feed. If the distance between two areas is less than **Ignore thickness** parameter, the system will treat them as one area.

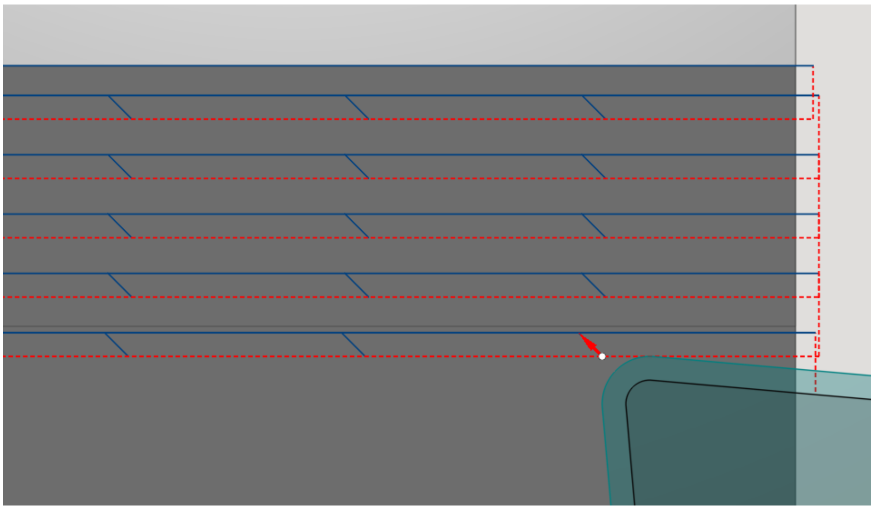


In the left side of the picture **Ignore thickness** parameter is less than the distance between the areas of machining, the system treats them separately. On the right side of the picture **Ignore thickness** parameter more than the distance between areas of machining, system treats them as one area.

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.



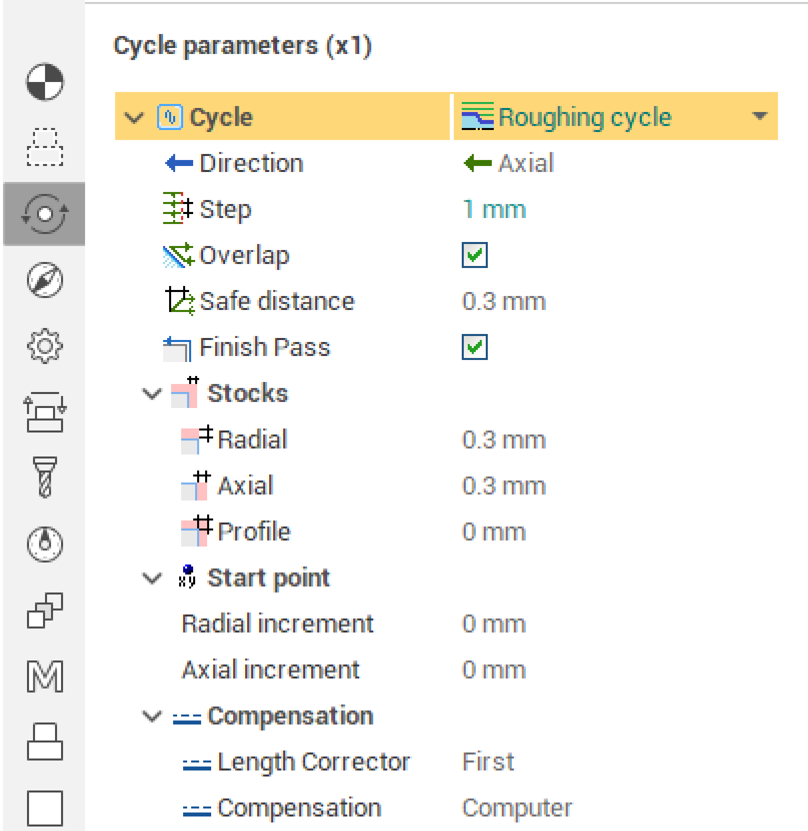
See also:

Lathe Machining

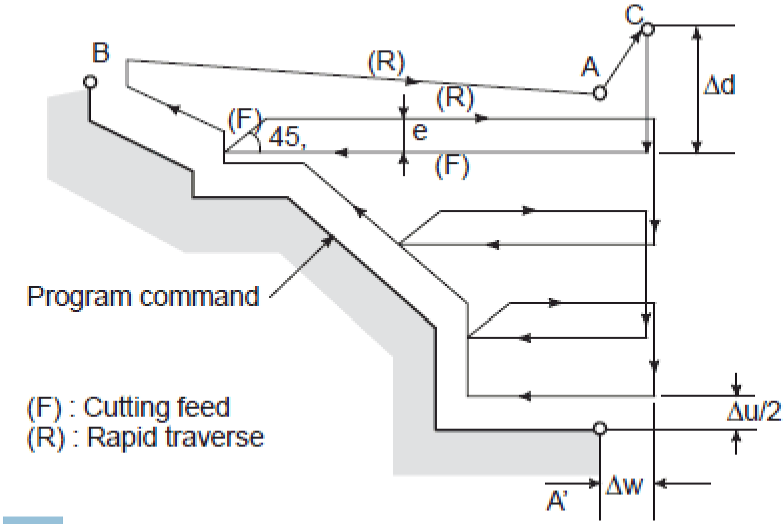
Lathe cycles

5.6.3.5 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____
N (nf)..........;

The move command between A and B is specified in the blocks from sequence number ns to 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.

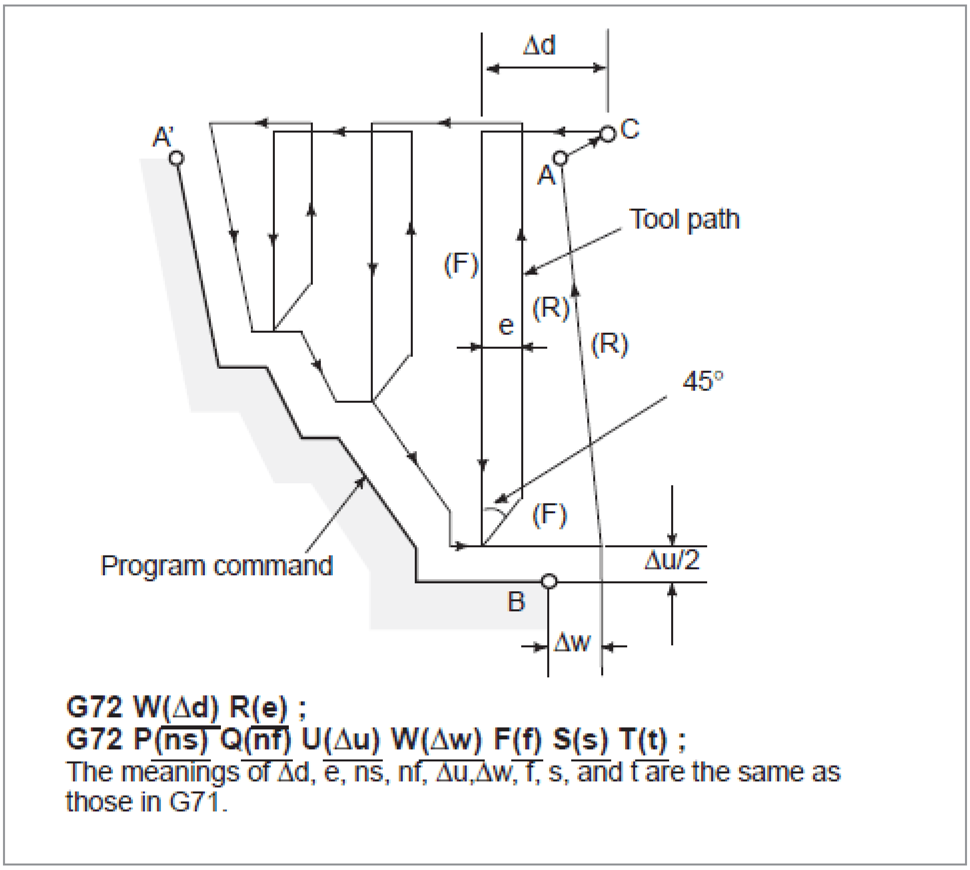
ns : Sequence number of the first block for the program of finishing shape.

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.



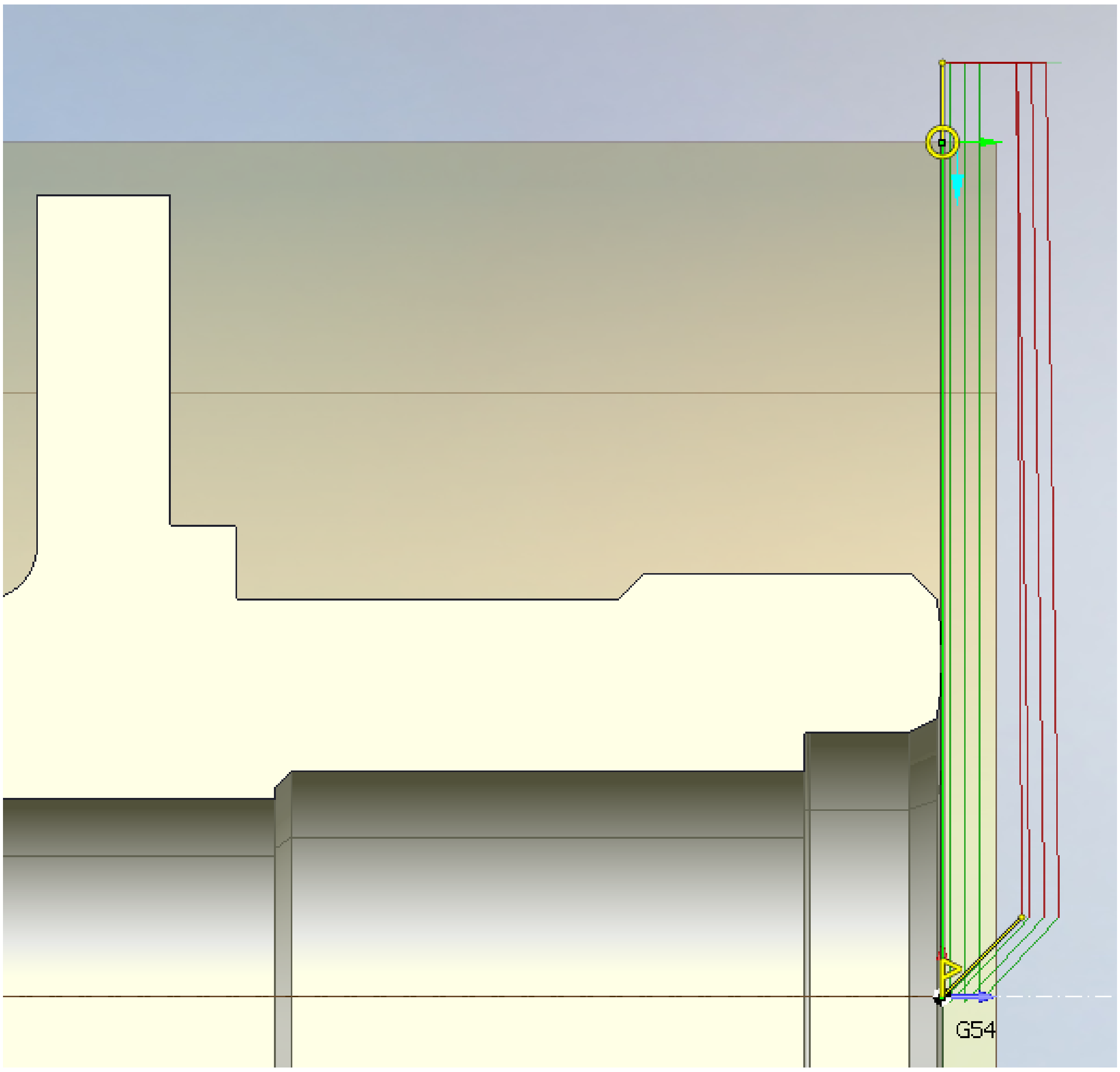
See also:

[Lathe Machining]

[Lathe cycles]

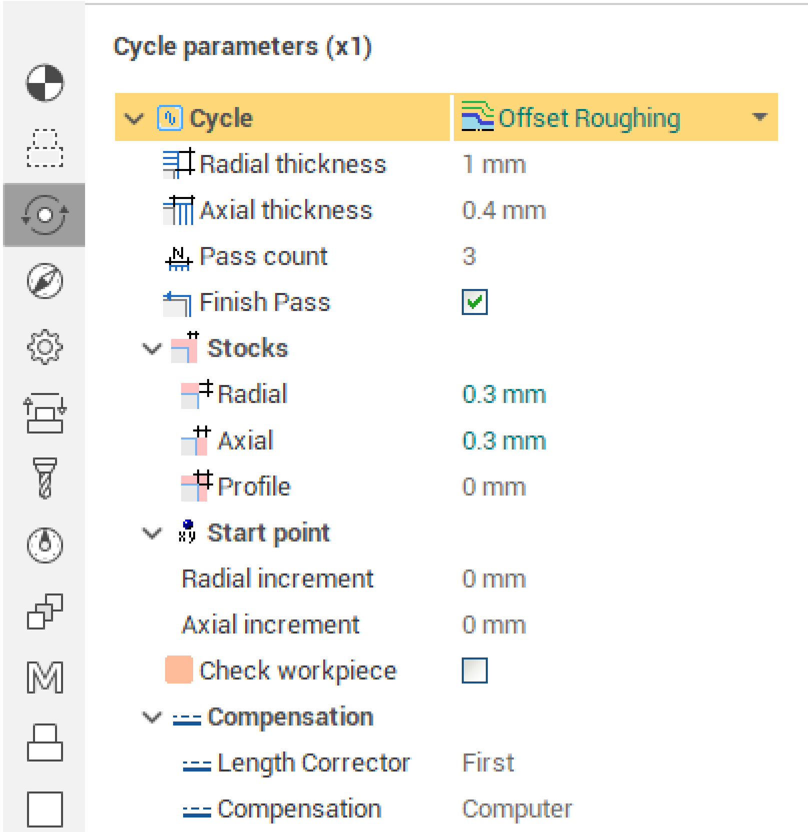
5.6.3.6 Facing

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.



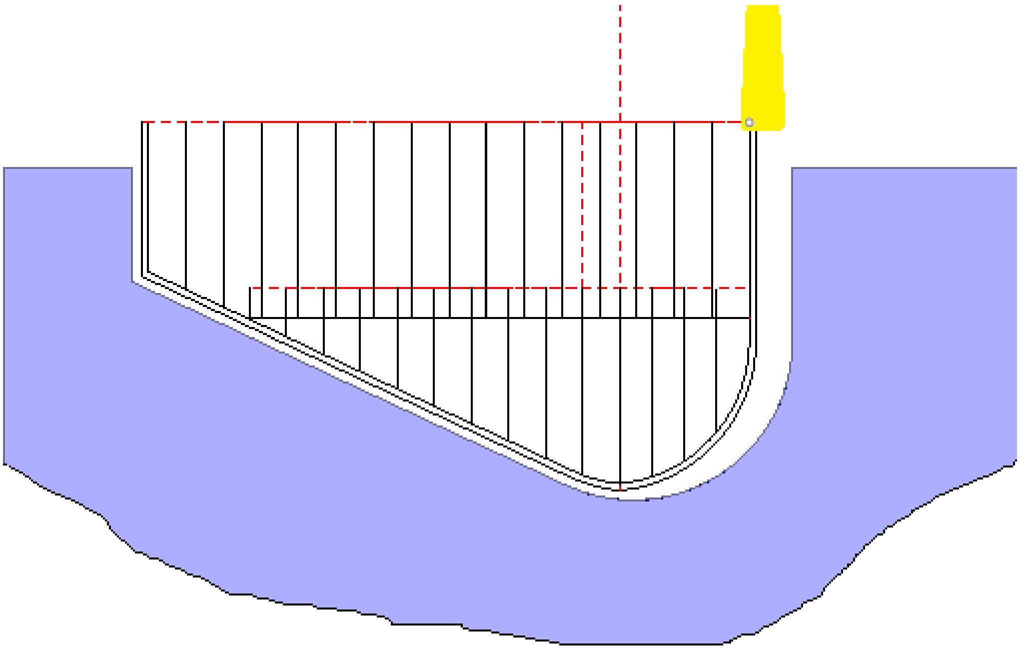
See also:

[Lathe Machining]

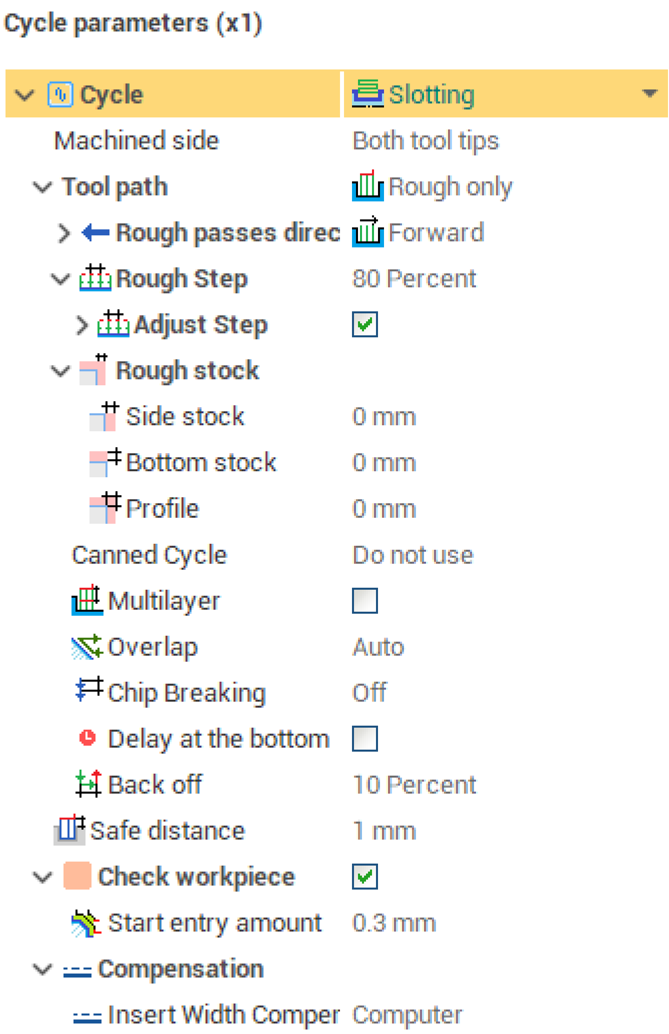
[Lathe cycles]

5.6.3.7 Grooving cycle

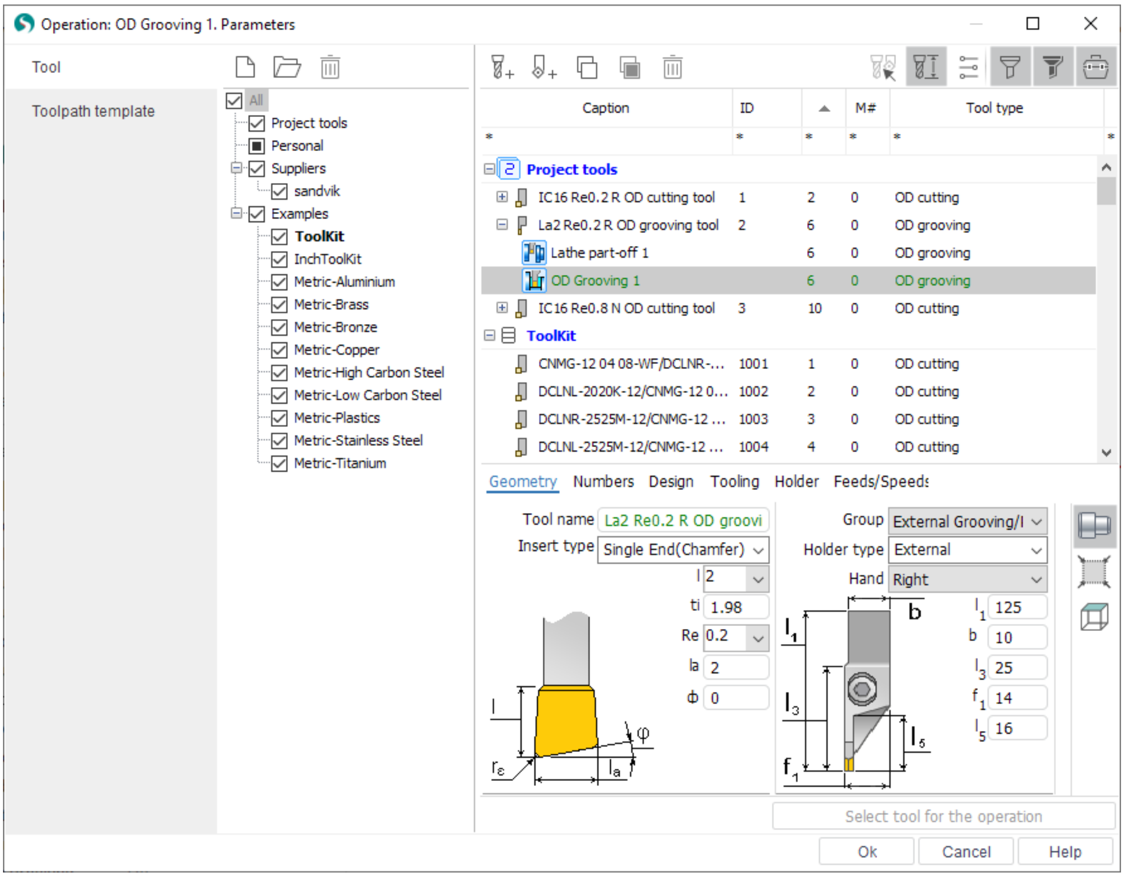
Advanced grooving element generates the complex tool path for the any kinds of the lathe grooves. It allows to machine the outer radial, inner radial, face or inclined grooves.



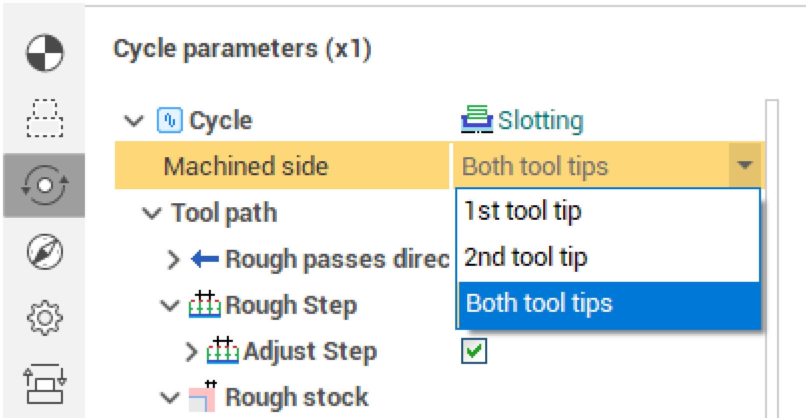
The cycle parameters can be defined in the properties window.



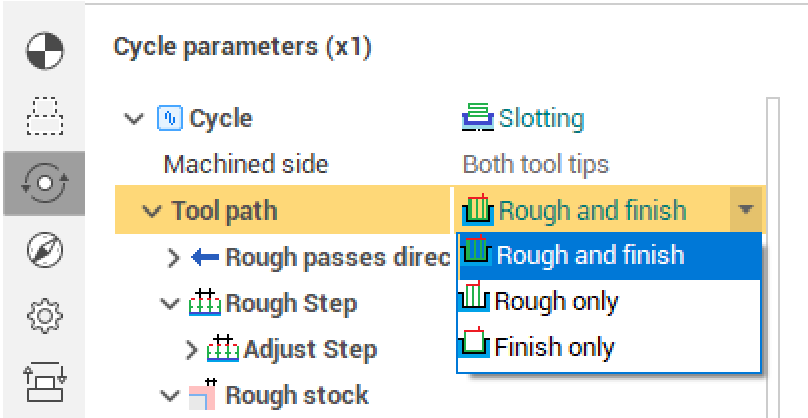
The groove cycle defines the bottom point of the groove contour and machines the both sides of the groove by the different tool tips. every side is machined from the end point of the profile to the bottom point. The tool tips are defined in the Tool dialog that is shown below.



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.

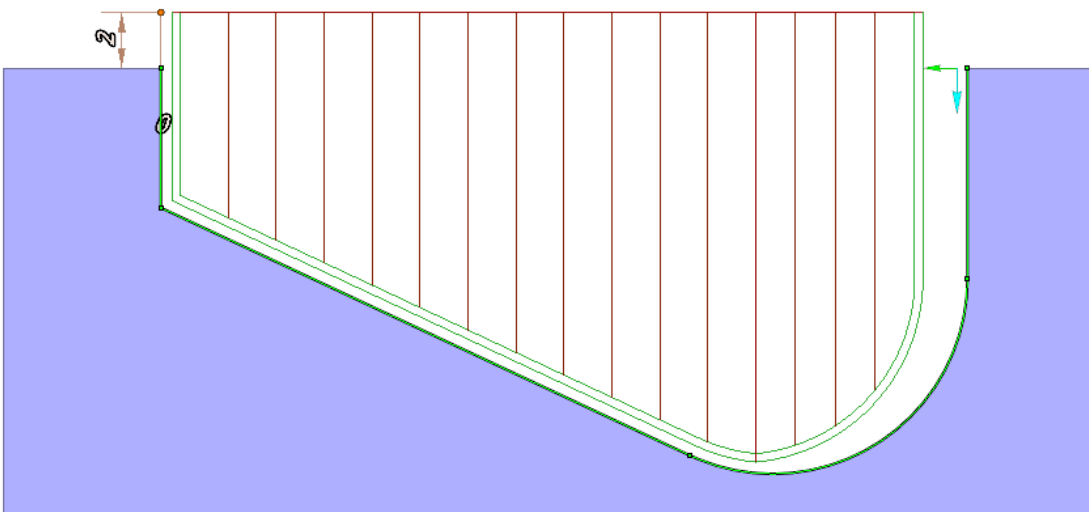


Advanced grooving cycle generates rough and finish passes. **Tool path** parameter defines what passes must be generated.



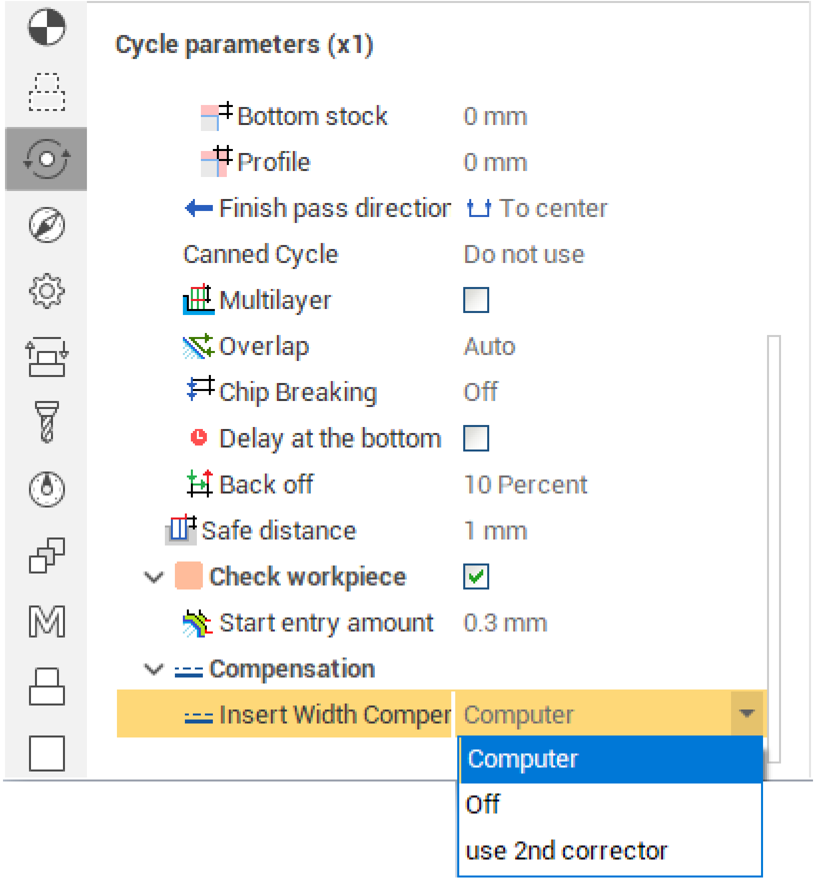
If **Finish only** is selected then finish pass only will be generated and the dialog will show only the parameters for the finish path. If **Rough only** is selected then the rough path will be generated and the parameter for the rough pass only is shown in the dialog. If **Rough and finish** is selected 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.

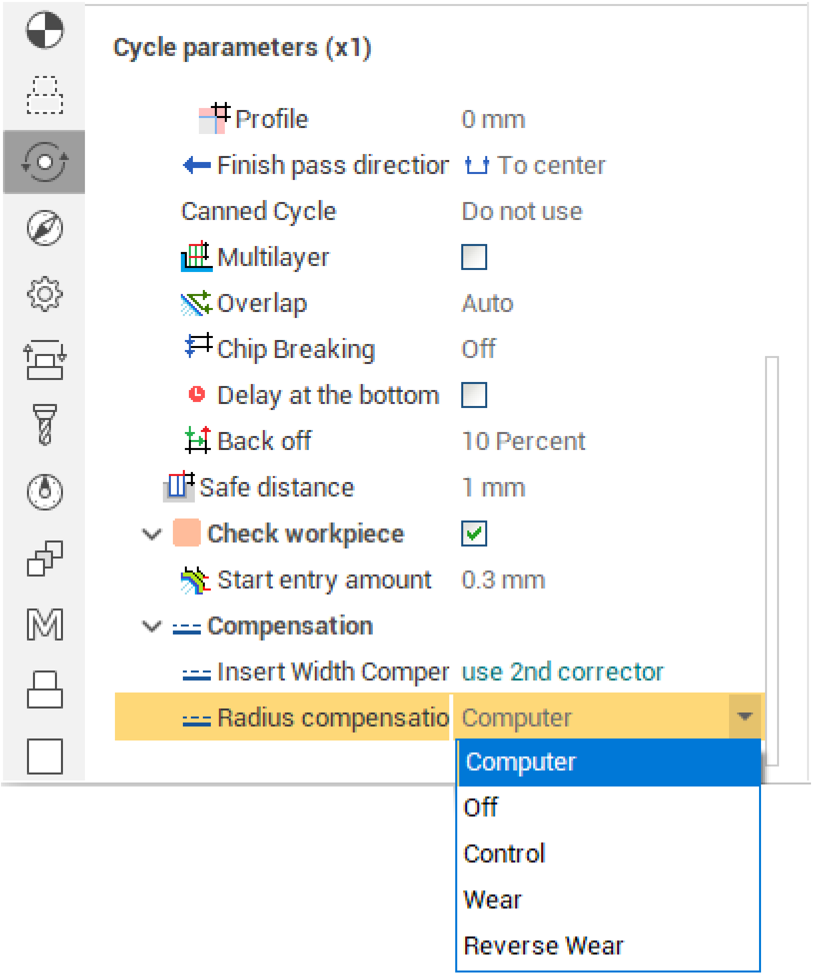


Computer mode generates the tool path for the first corrector only.

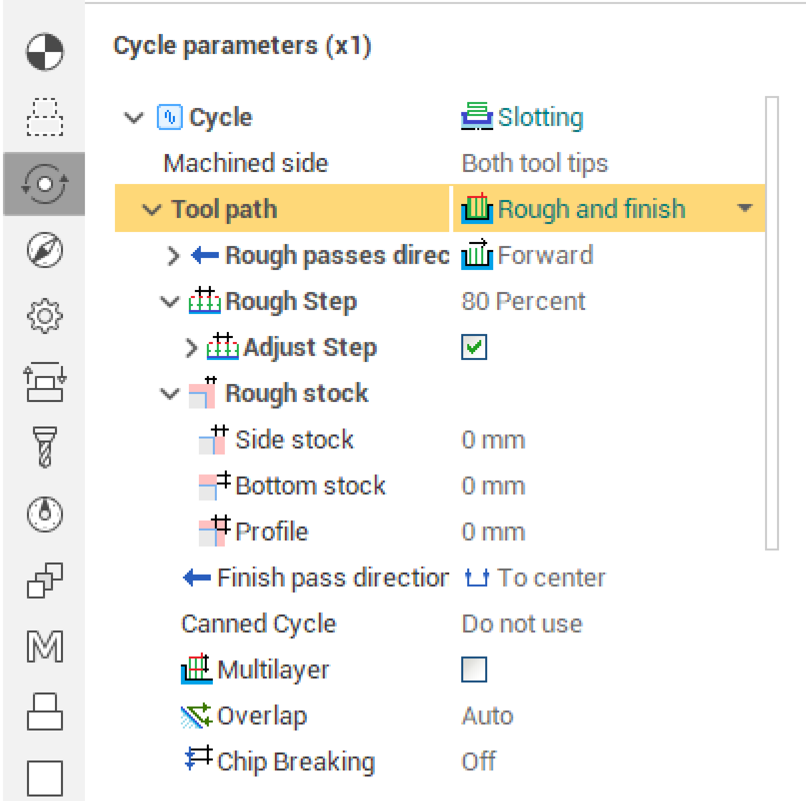
Off mode is not realized.

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 SprutCAM |X 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.

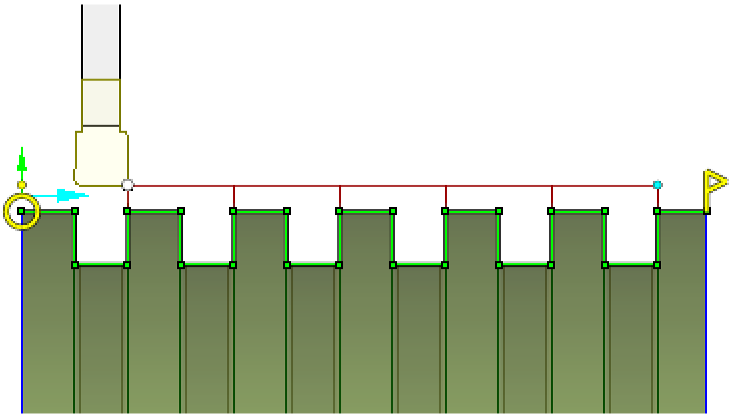
See also:

Lathe Machining

Lathe cycles

5.6.3.8 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.



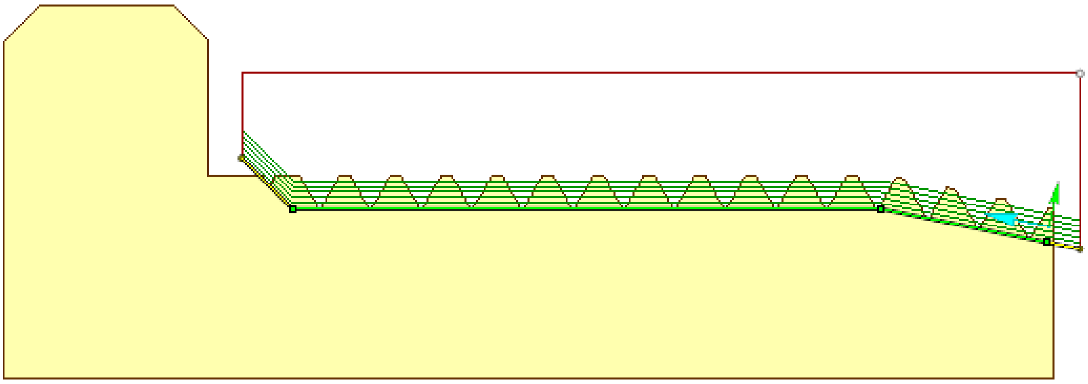
See also:

Lathe Machining

Lathe cycles

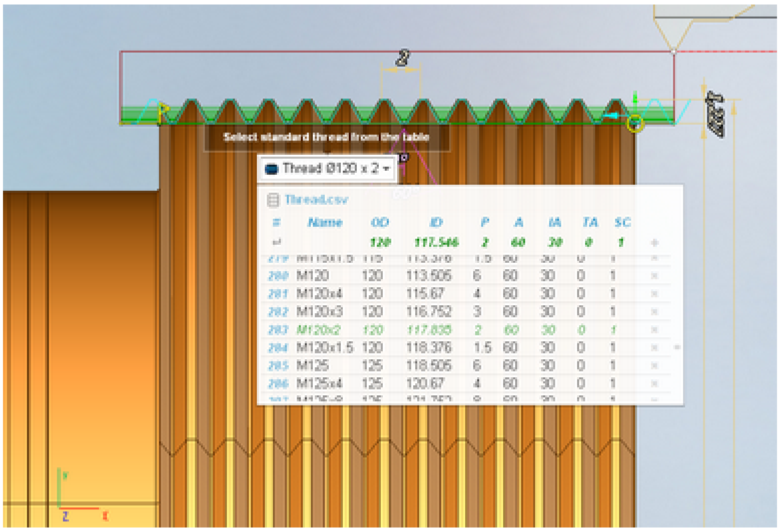
5.6.3.9 Thread cycles

Thread cycle allows to generate passes (one ore more) for creating thread with specified parameters.



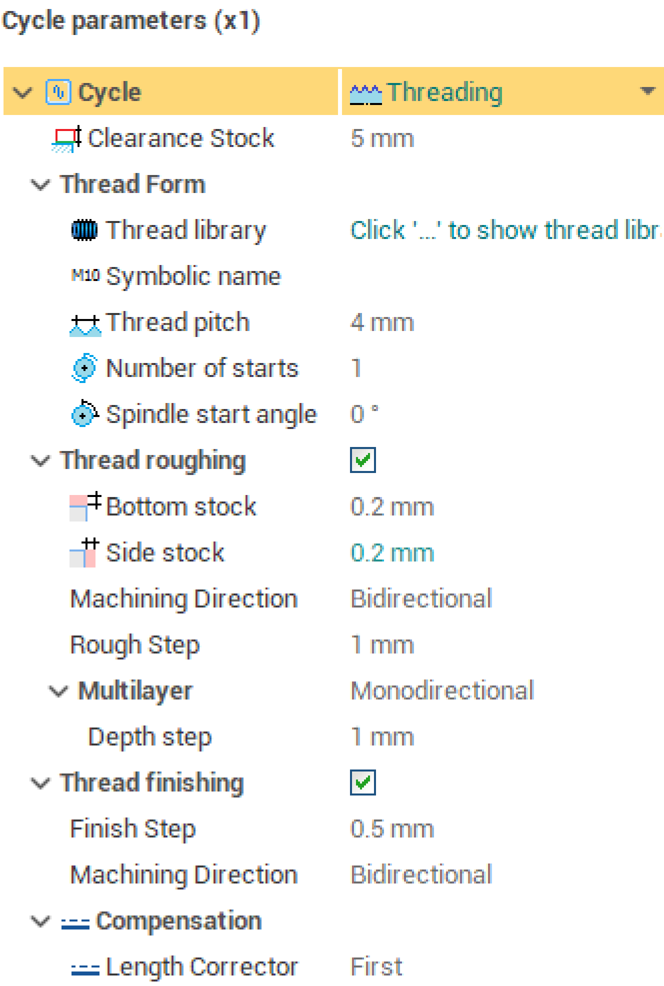
Threading area is specified by geometrical contour. Contour defines thread bottom diameter (inner diameter for external thread and outer diameter for inner thread). Second diameter is calculated by **Depth** parameter. Thread type - inner or outer - is specified by contour machining side (Perpendicular 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.

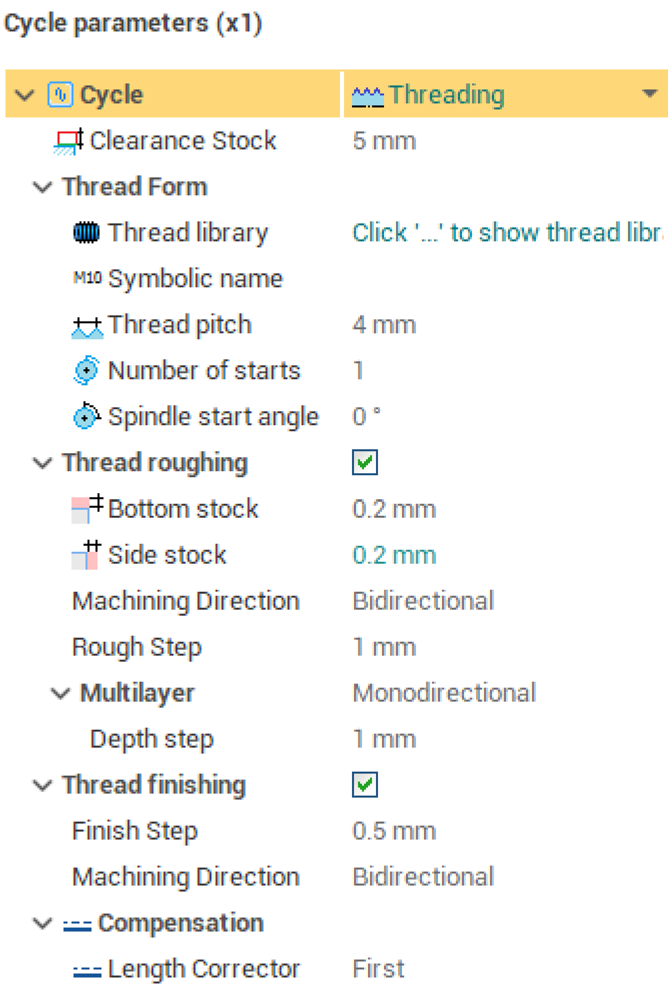
See also:

Lathe Machining

Lathe cycles

Standard form thread

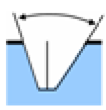
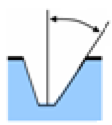
The Properties window for the job assignment item of the standard form threading operations has the following parameters.



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.

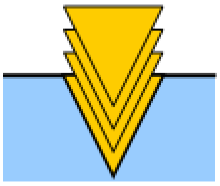
If plunge mode is **Flank** or **Alternate Flank**, then **Thread angle** and **Inclination angle** parameters defines angle of tool plunge at each pass.

Thread angle	Inclination angle
	

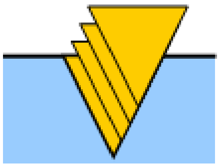
Parameters in **Sequence** group defines numbers of starts and plunge mode for each pass.

Value in **Sequence** combobox defines plunge mode. The following types of strategies are available:

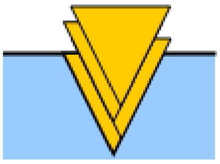
- **Radial**. The direction of plunge is perpendicular to the axis of rotation.



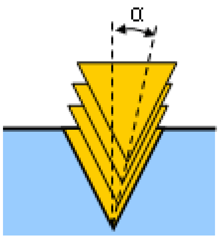
- **Flank**. The plunge is made along one side of the ledge.



- **Alternate flank**. Plunge is made alternately along the two lateral sides of the ledge.



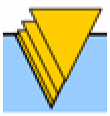
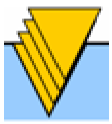
- **Modified flank**. Plunge with angle, specified at **Angle** parameter.



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.

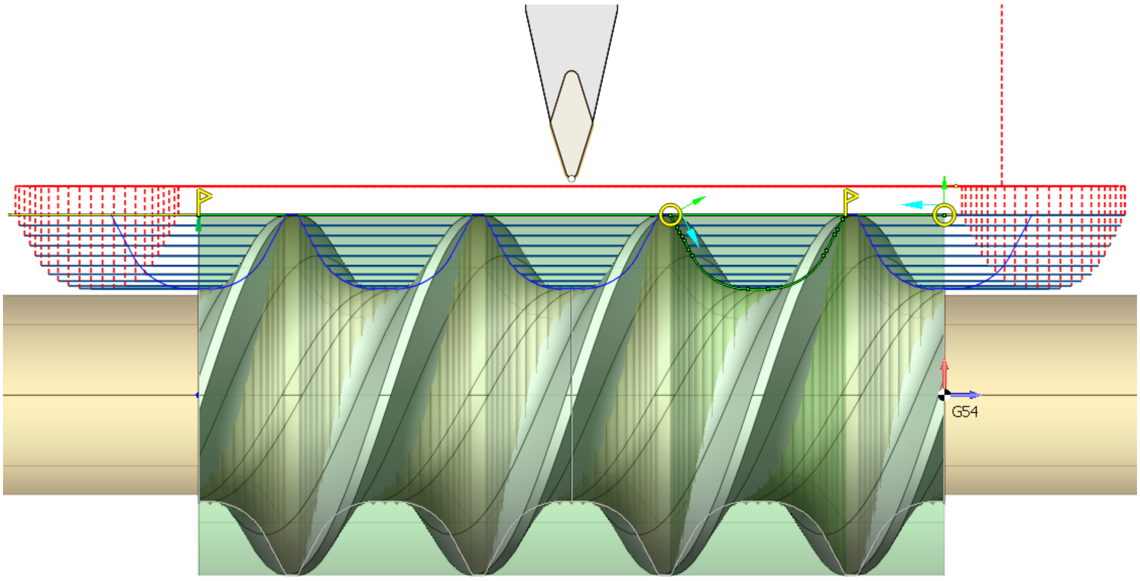
Equal area	Equal depth
	

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.

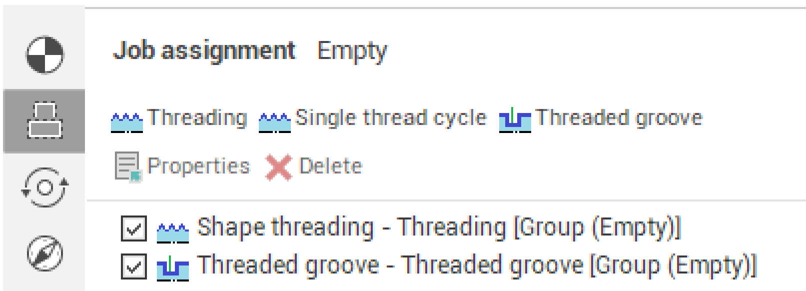
Free form thread

The profile threading operation allows you to make threads whose shape is different from the shape of the tool. This is achieved by removing material inside the entire threaded 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.

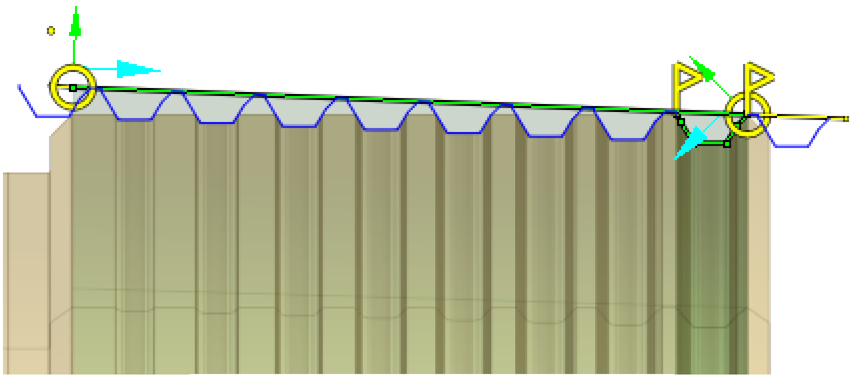


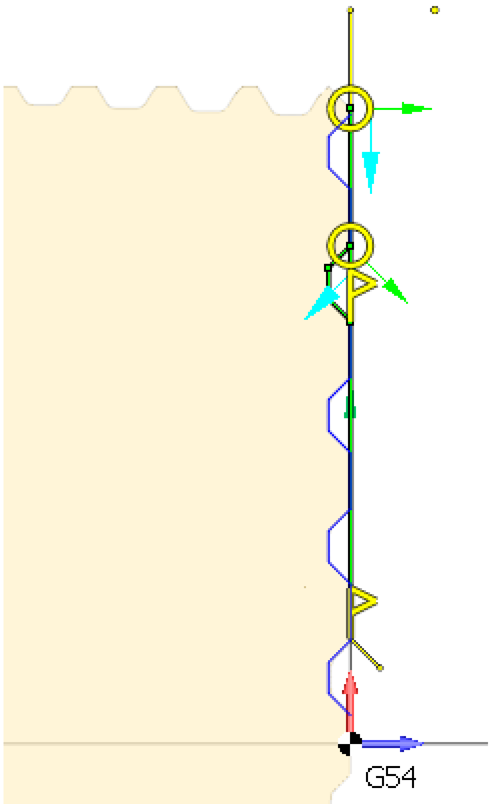
Thus, in contrast to the standard form threading operation, in the job assignment it is necessary to add two items at once.

- An item that determines the position of the beginning and end of the thread on the part: **Threading** or **Single thread cycle** (depend on desired output format).
- An item that defines the shape of the threaded groove - **Threaded groove**.



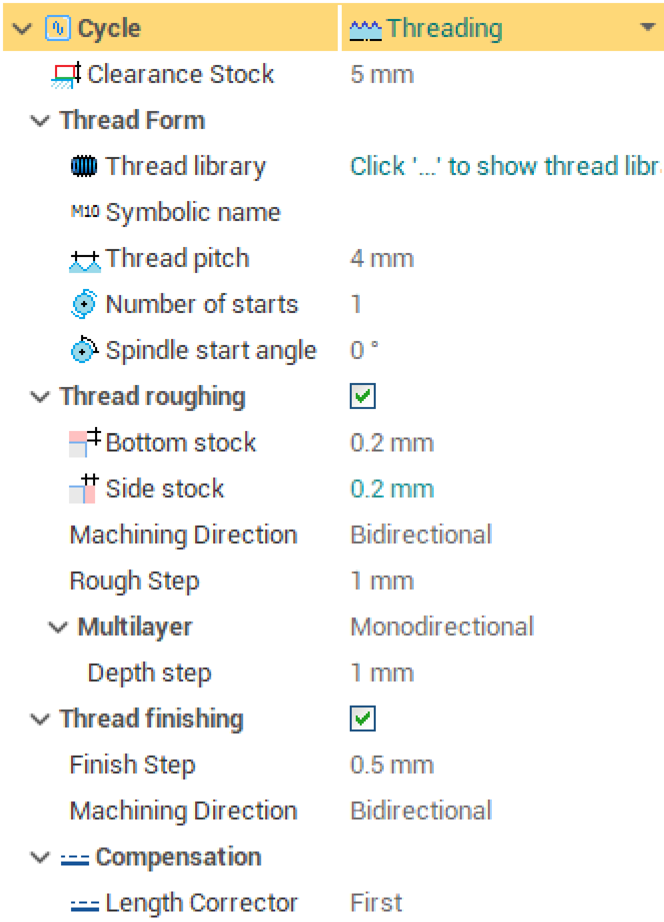
Depending on the orientation of the given profiles, you can get a cylindrical, conical or face thread (archimedean spiral).



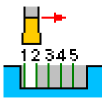


The Properties window for the job assignment item of the free form threading operations has the following parameters.

Cycle parameters (x1)



Thread roughing group. If it is enabled then successive passes along the longitudinal axis are performed with a given step.

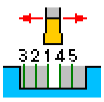


The **Bottom stock** and **Side stock** defines the stock values on the bottom and side walls respectively to leave for finishing.

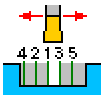
Rough step – step between two adjacent rough cuts along a thread. It can be set by total number of passes (the common width of a groove divided into the count of passes), or directly by the distance between passes.

The **Machining direction** parameter defines the order of rough cuts.

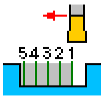
- Bidirectional.



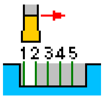
- Alternate.



- Forward.

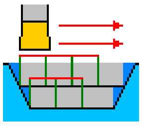


- Backward.

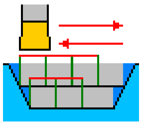


The **Multilayer** parameter allows to remove a rough material by several layers.

- The **Off** value correspond to the single-layer machining.
- **Monodirectional** – multilayer machining, direction of passes on each layer equally.

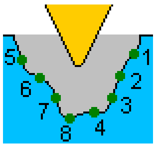


- **Zigzag** – multilayer machining, a root pass direction correspond the **Machining direction**, and the each following layer is opposite to the previous layer.



If multilayer is active then the **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.

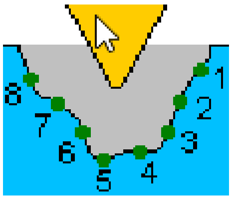
Thread finishing group. If it is enabled, then finish passes are allocated along a threaded groove profile is set in the job assignment.



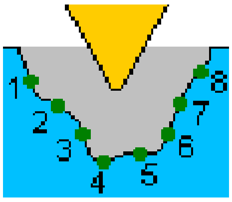
Finish step – step between the adjacent passes, calculated by the profile length. The step can be defined by count of passes or by distance directly.

The **Machining direction** parameter defines sequence of finish passes:

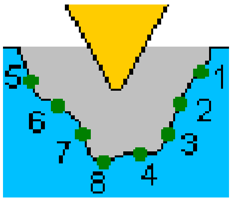
- Forward.



- Backward.



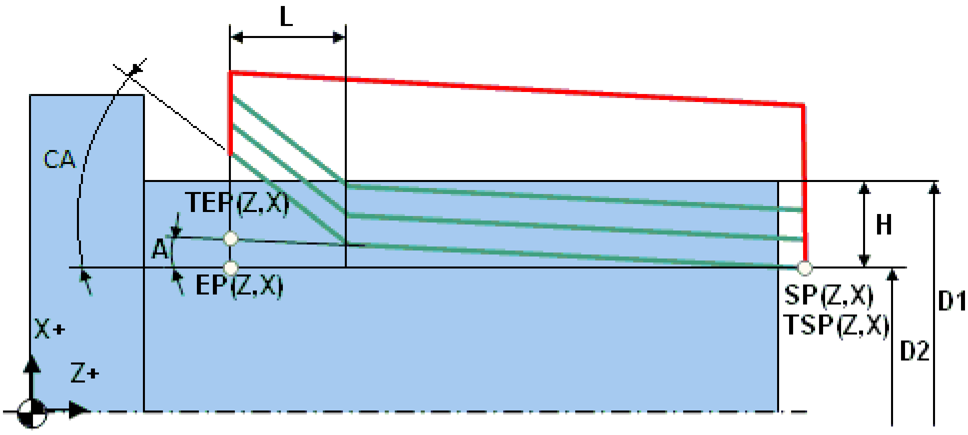
- Bidirectional.



Threading cycle types

The toolpath of threading operations can be generated using various output formats. We consider each of them separately.

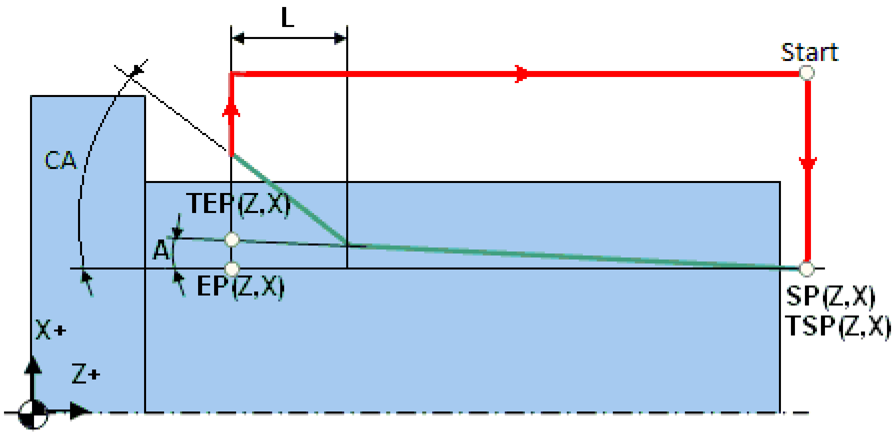
Multipass thread cycle (ISO G76) allows you to use a single frame of the NC-program to set all parameters necessary for machine to make a standard form thread. Required depth is reached automatically by generating several passes. Among the parameters of the cycle there are start and finish point coordinates, 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)
```

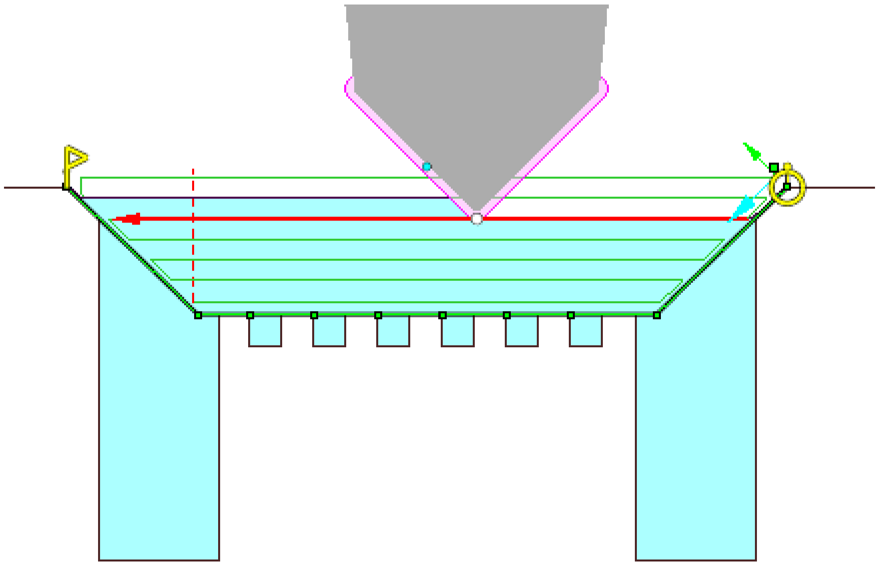
Advanced (expanded) thread machining is processed by using ISO G32/G33 (can be different in various machines). This command activates continuous cylindrical, taper or face threading mode with constant step. In this mode synchronization between tool movement and spindle rotation is enabled. All tool movements 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

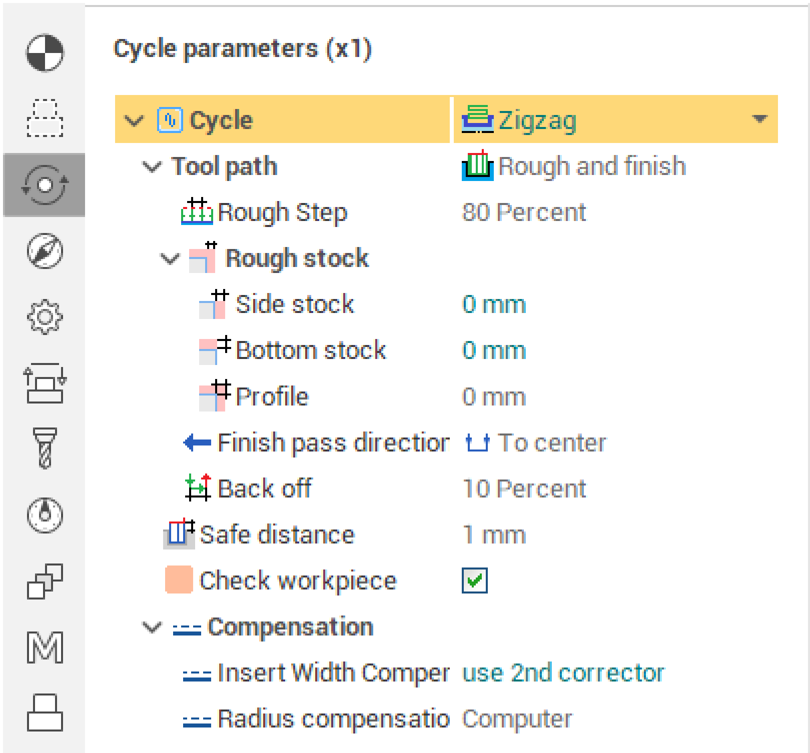
```
G00 X60.0 Z10.0 M08 (Approach to Start point)
G00 X29.4           (Approach to start of pass 1)
G32 Z-52.0 F2.0     (Threading synchronized with spindle)
G00 X60.0           (Return to Start)
Z10.0
X28.9               (Approach to start of pass 2)
G32 Z-52.0          (Threading synchronized with 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)
```

5.6.3.10 Zigzag cycle

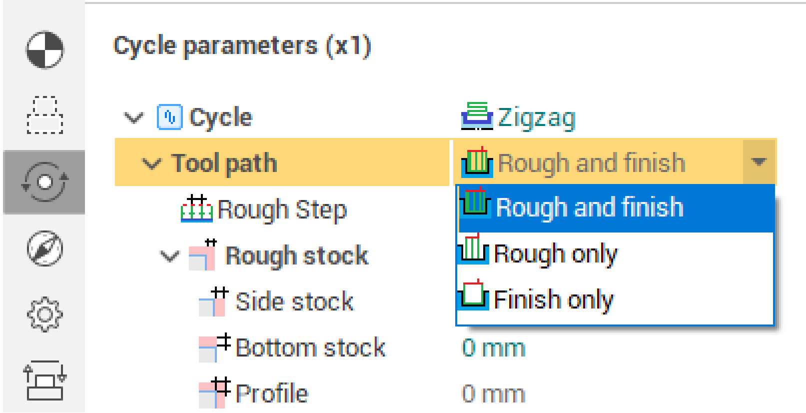


Zigzag lathe cycle generates toolpath useful for machining areas which are closed from both sides with neutral tool. The tool is lowered down layer by layer, and performs horizontal moves with changing direction for each layer.

Common parameter of the cycle



Zigzag cycle generates rough and finish passes. **Tool path** parameter defines what passes must be generated.

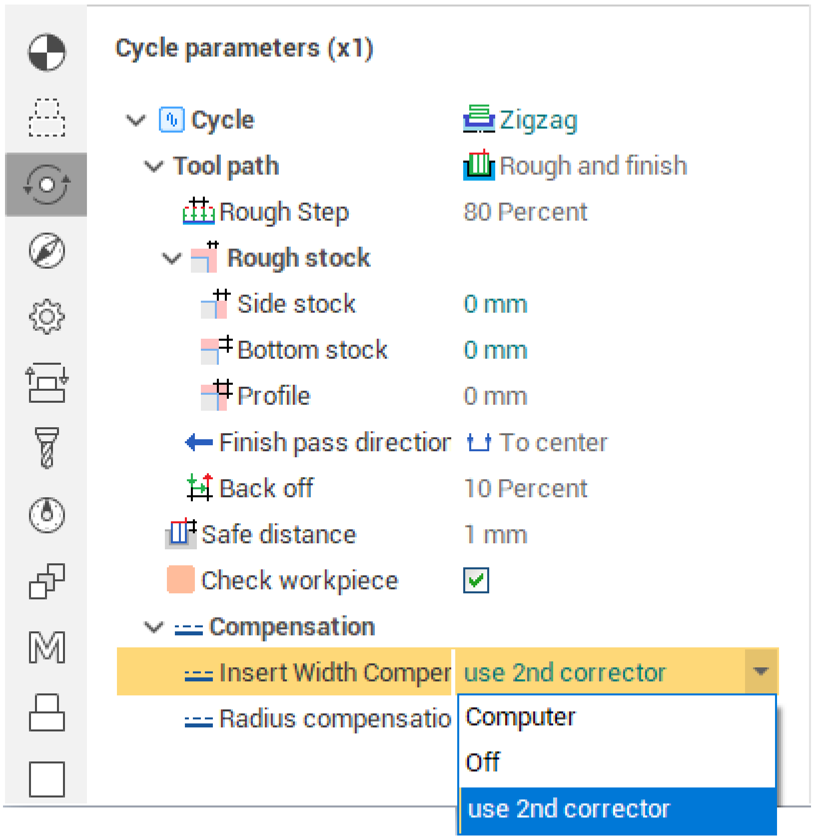


If **Finish only** is selected then finish pass only will be generated and the dialog will show only the parameters for the finish path. If **Rough only** is selected then the rough path will be generated and the parameter for the rough pass only is shown in the dialog. If **Rough and finish** is selected 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.

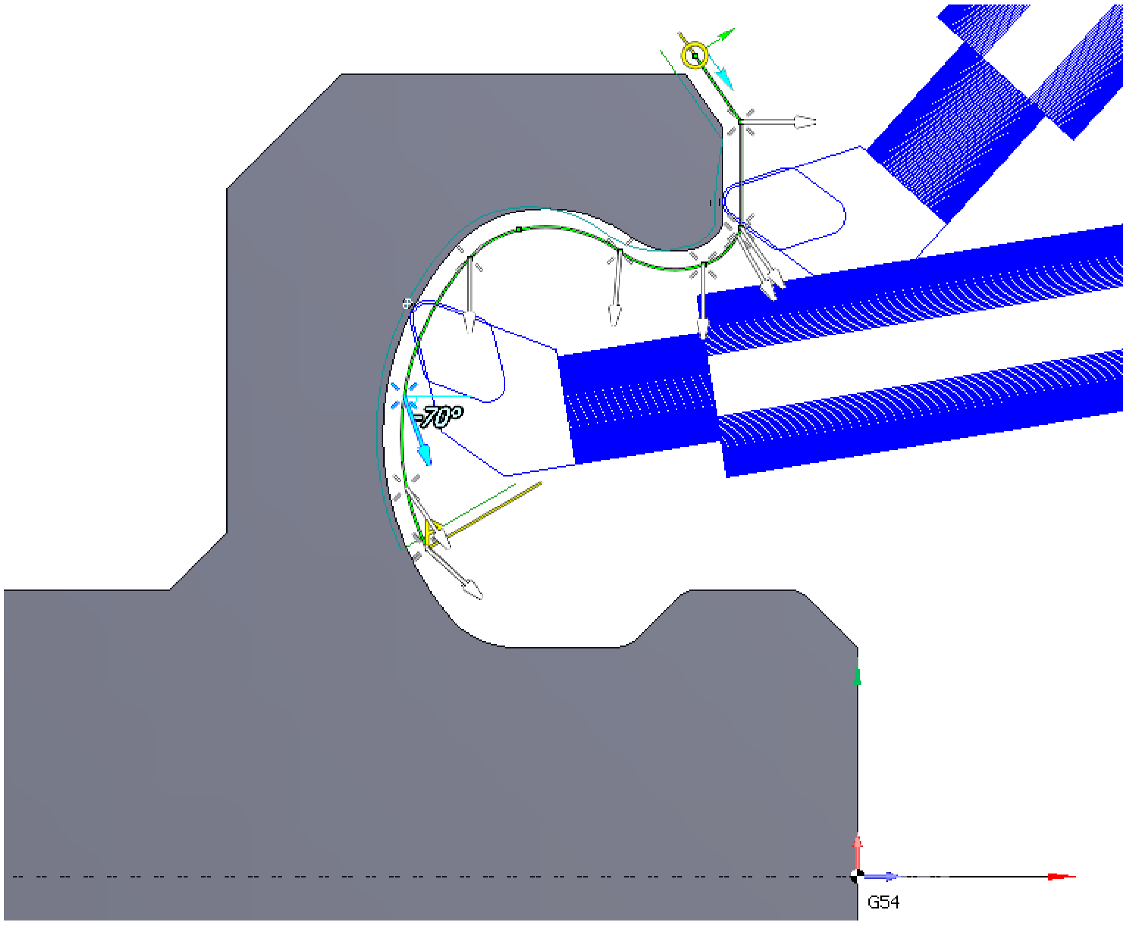
See also:

Lathe Machining

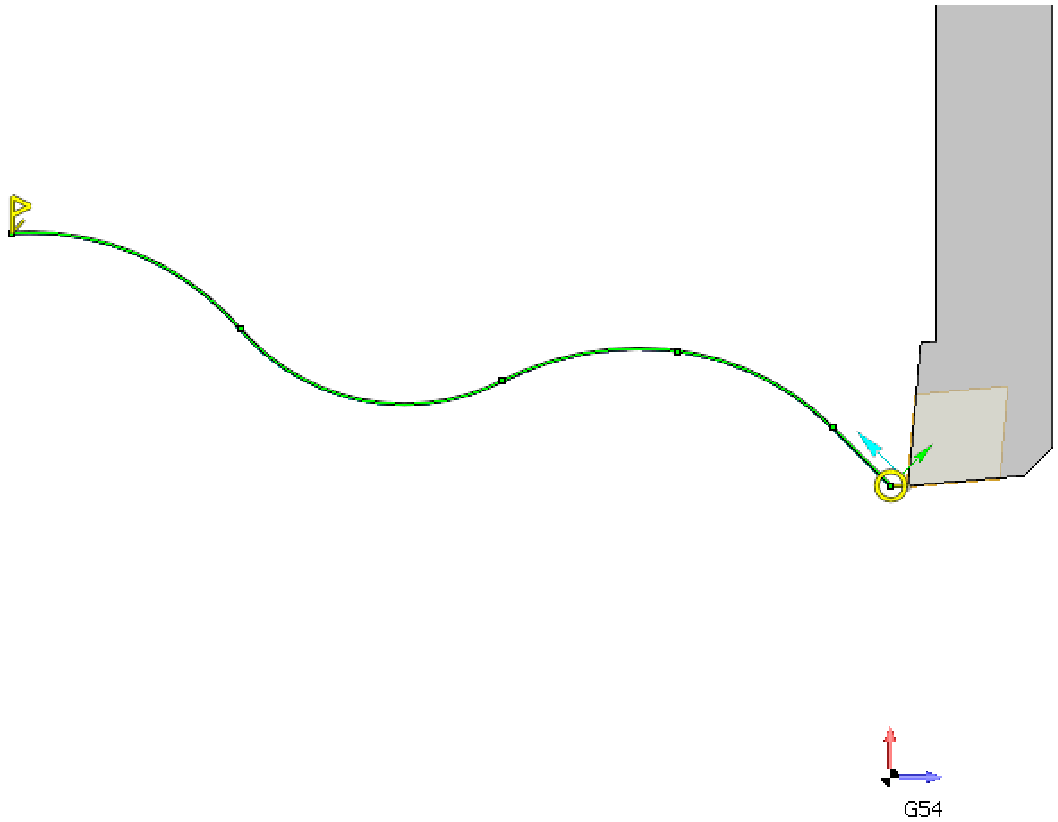
Lathe cycles

5.6.3.11 4-axis turning

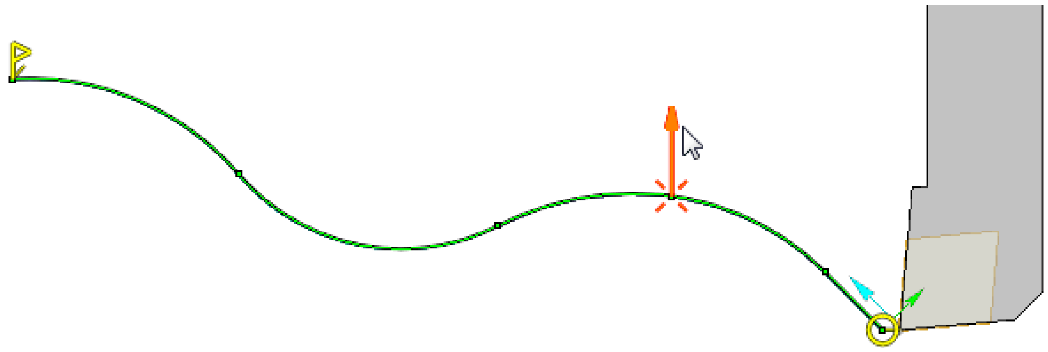
4-axis turning cycle allows to machine hard-to-reach areas of a part with just one operation by continuously changing of the tool inclination angle in the process of moving along a contour (using the 4-th axis of the machine, if available). This reduces the number of tool changes, thus increasing productivity. 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.



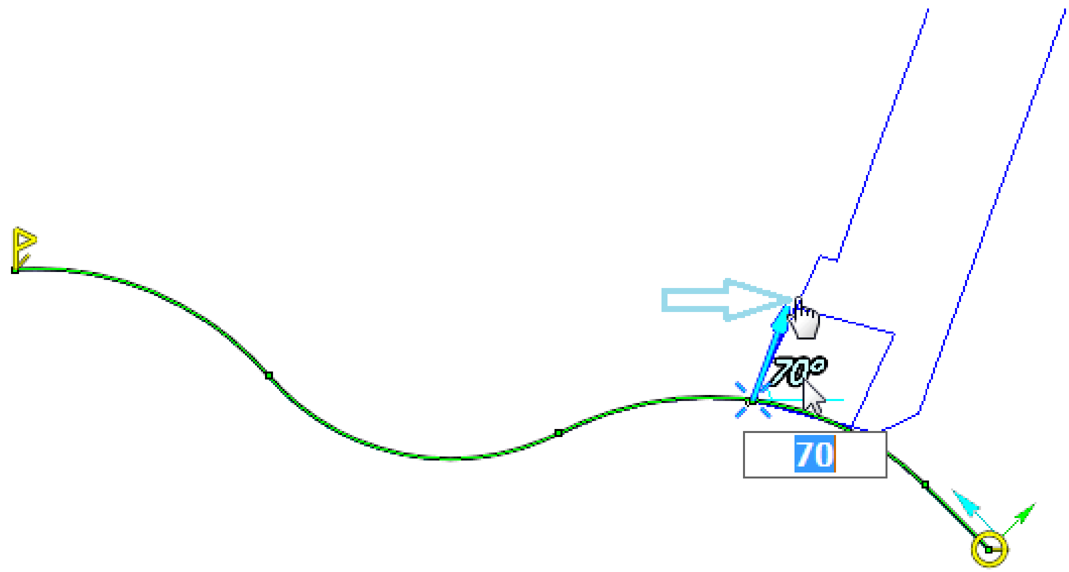
Almost all of the parameters for this type of machining can be specified interactively in a graphical window. Immediately after adding the "4-axis turning" item with some contour just the specified contour with start and end markers will appear. There will be no of additional visible tool inclination 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.



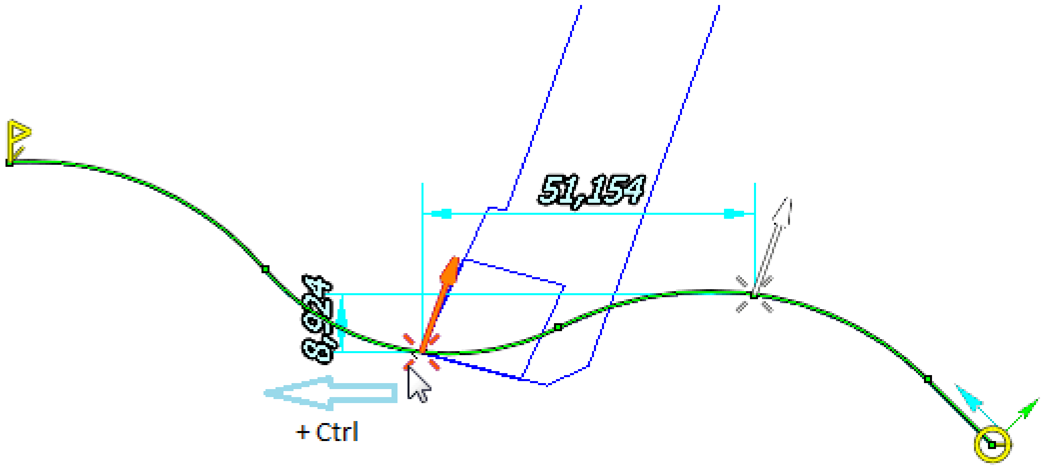
However, if you move the mouse cursor in the area of the nodal points of the contour then tool inclination vectors will be highlighted. By default, they are invisible, because the angle of the tool in these points is already corresponds to these vectors.



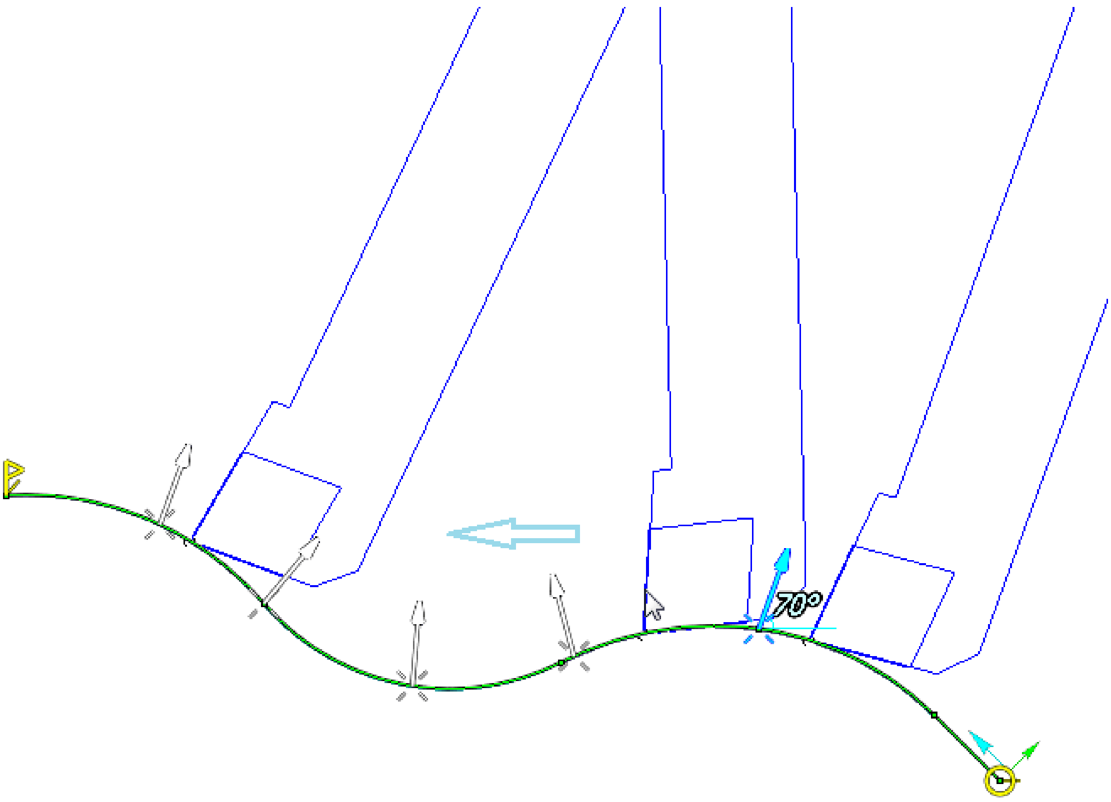
If you want to change the tool angle in the highlighted point, you can clutch and drag the vector. In this case the display will show angular dimension, which determines the angle of the tool relative to the axis of rotation, as well as the profile of the tool. Desired angle of the tool at a point can 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.

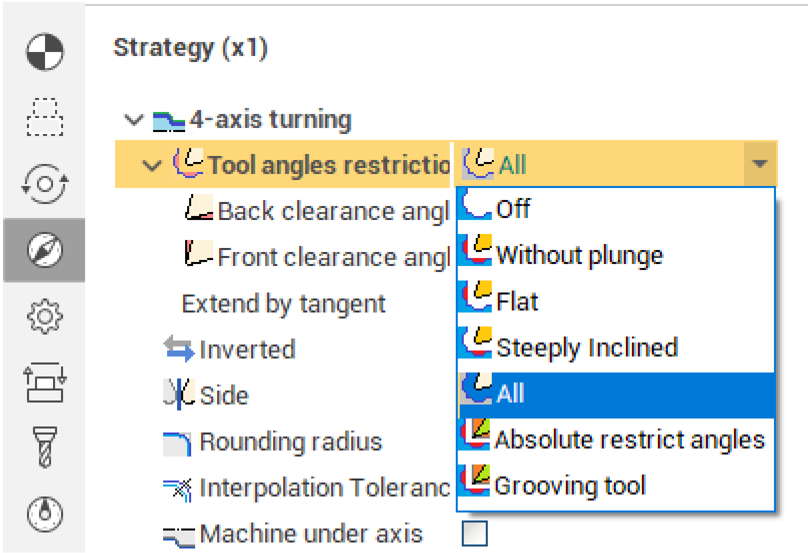


The slope between two given vectors vary continuously from the first vector to the second. It can be checked by clutching the tool profile, that appears when you select a vector, and drag it along the contour. The tool will move following the mouse pointer along the whole contour. The slope of the tool, 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.



See also:

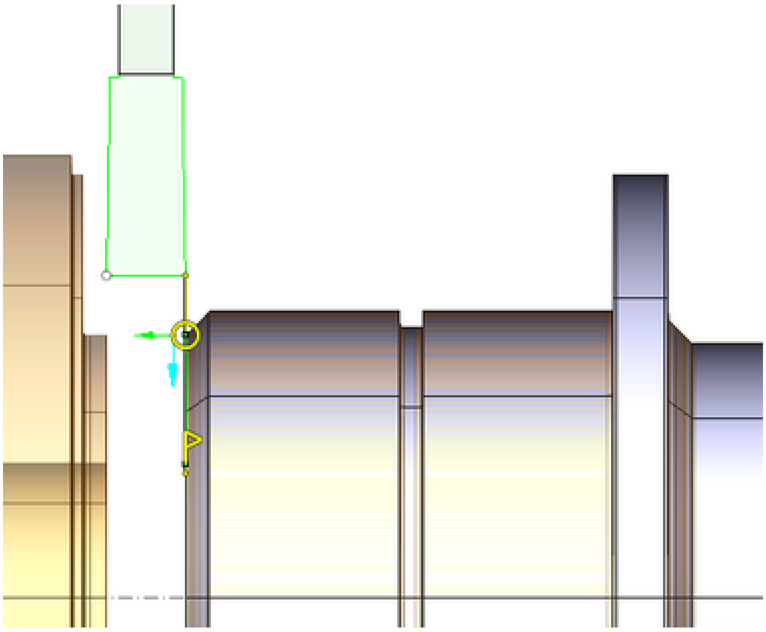
Lathe Machining

Job assignment element parameters

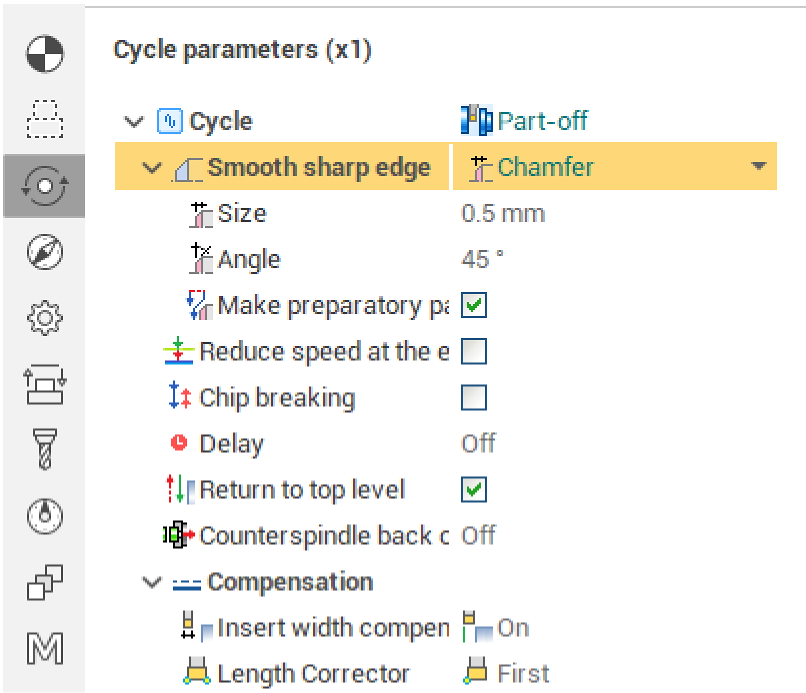
Contour editing modes

Lathe cycles

5.6.3.12 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.

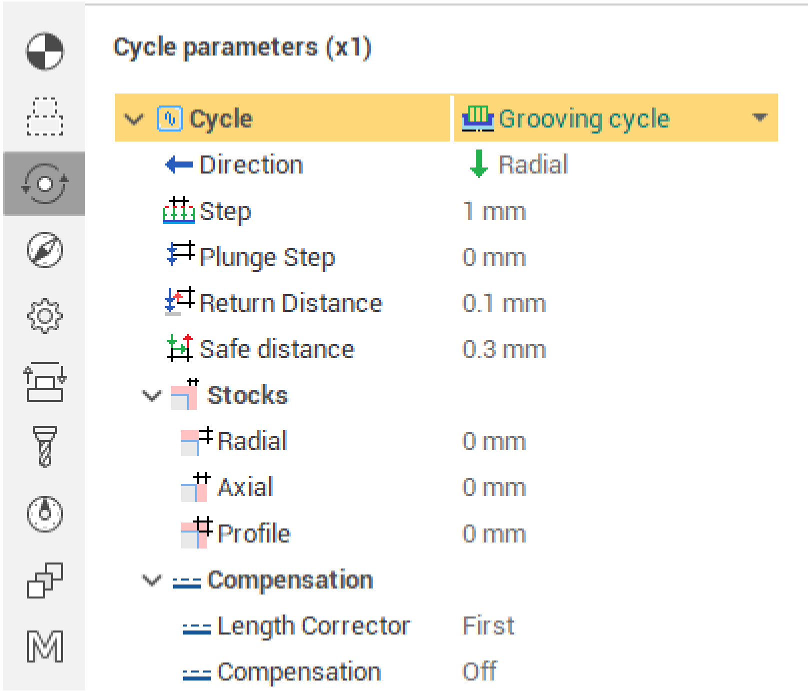
See also:

Lathe Machining

Lathe cycles

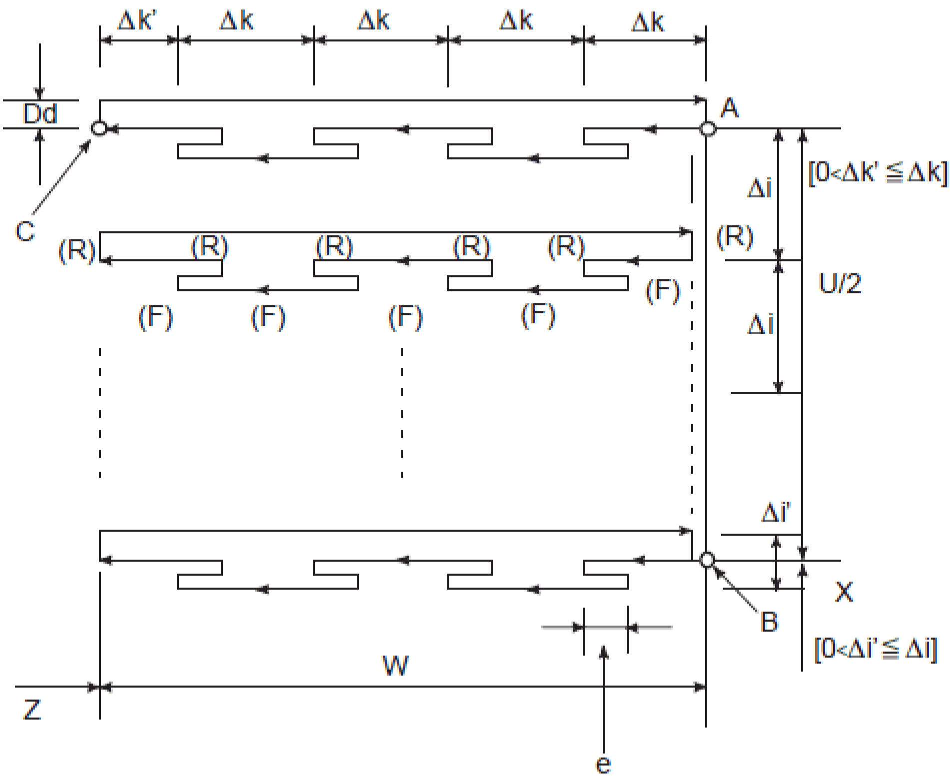
5.6.3.13 Peck groving/drillling cycle

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.

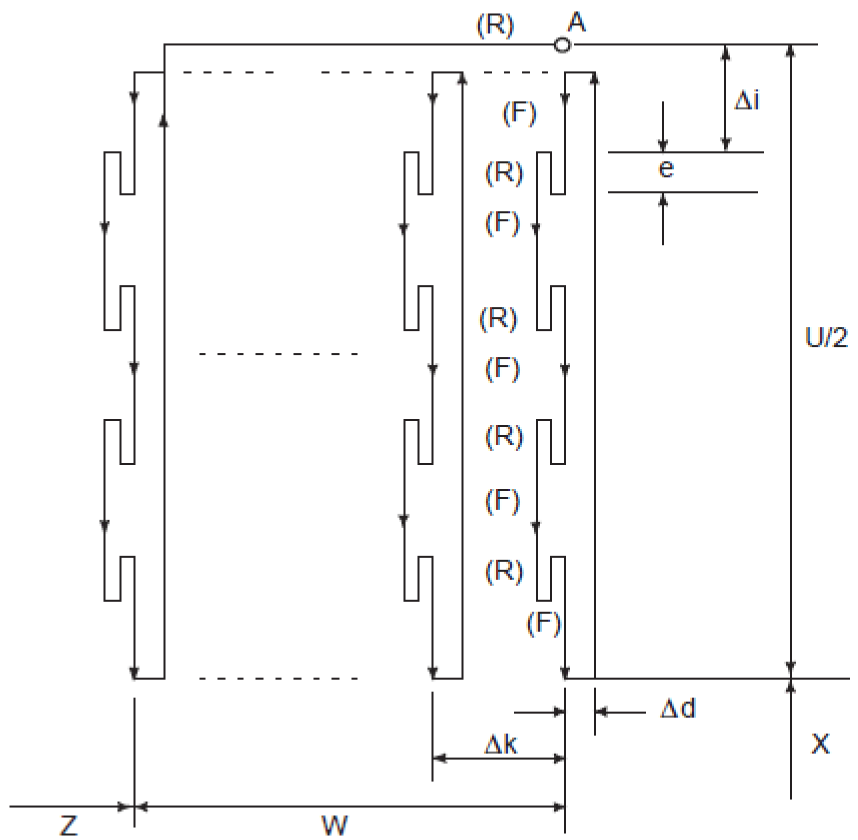
The following program generates the cutting path shown in the picture below. Chip breaking is possible in this cycle as shown below. if X(u) and P are omitted, operation only in the Z axis results, to be used for drilling.



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) ;

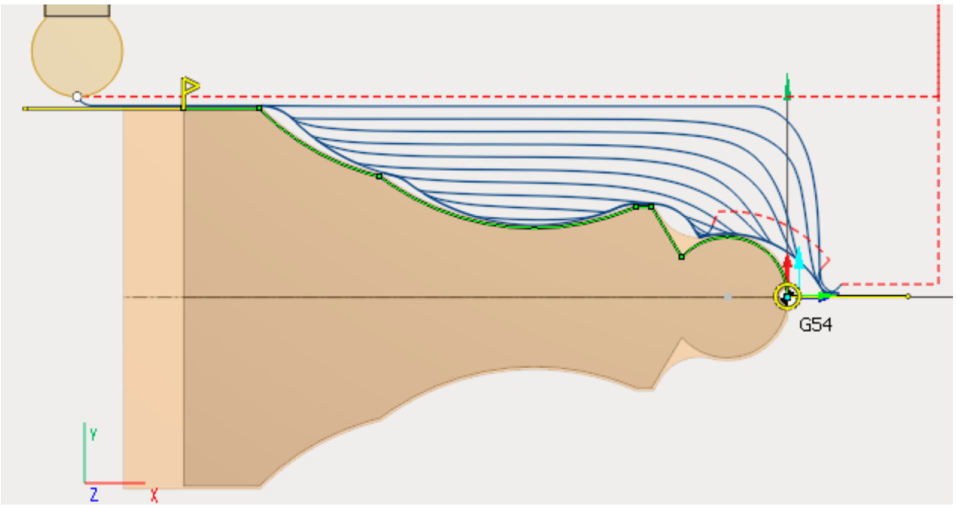
G75X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F(f) ;

See also:

[Lathe Machining]

[Lathe cycles]

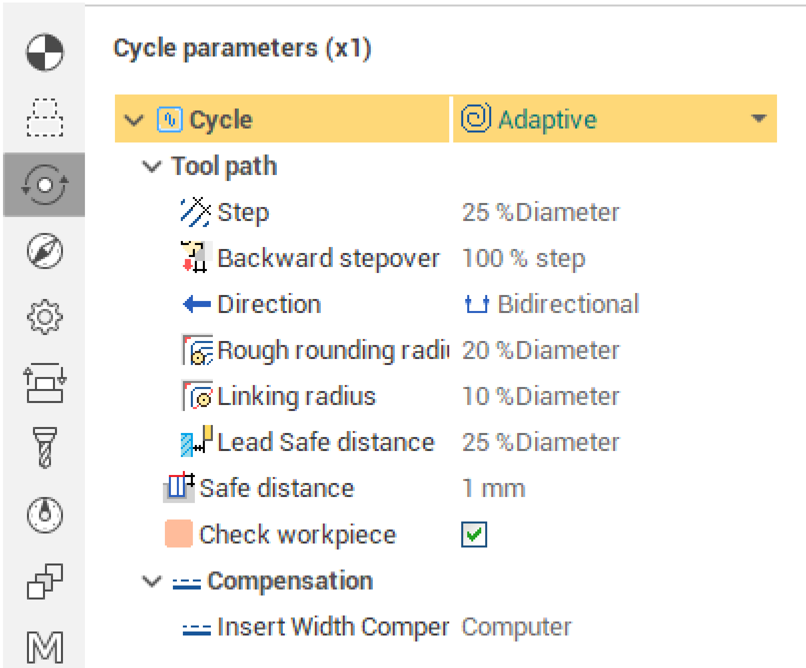
5.6.3.14 Adaptive roughing (turning)



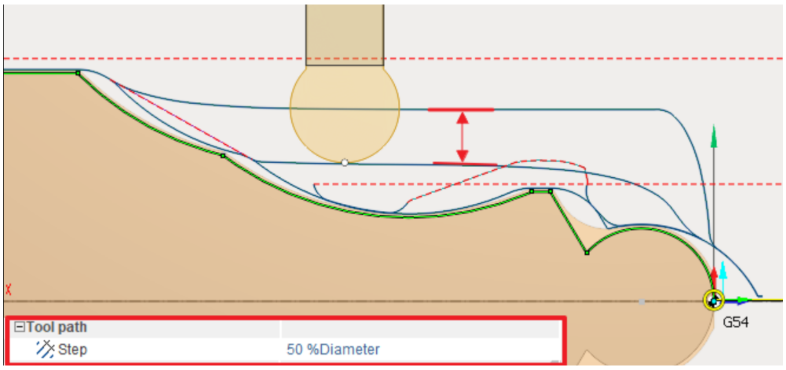
More than 2 times faster than traditional lathe grooving operation. This is achieved due to the high work speed and the optimized tool path. It uses tool with round insert.

This turning operation based on milling Adaptive SC operation.

Common parameter of the cycle:

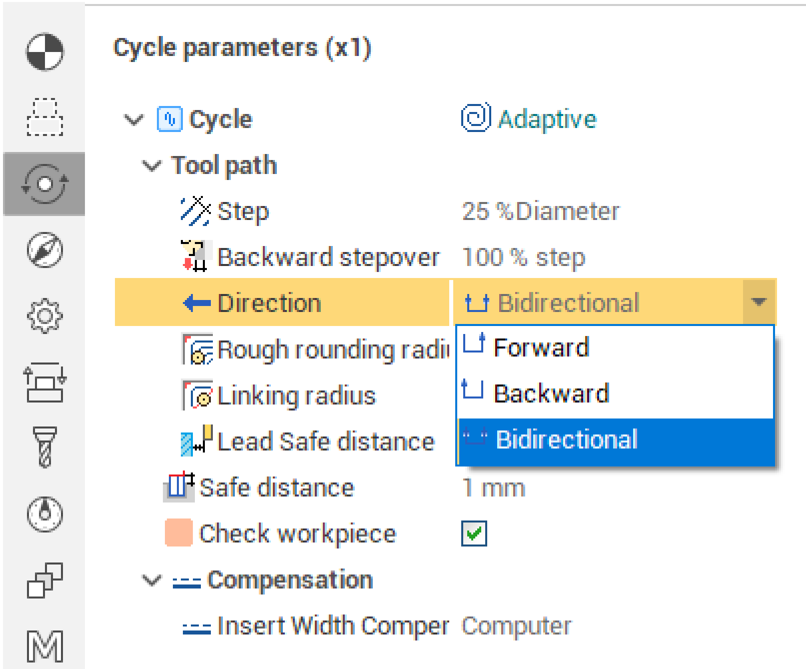


Step defines the distance between the passes. It can be defined in the percents of the tool width or in the units of the length.

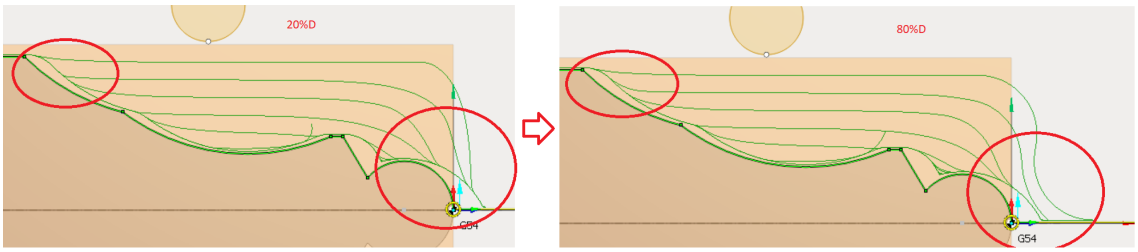


Backward stepover defines backward stepover in % of forward stepover (active if Bidirectional).

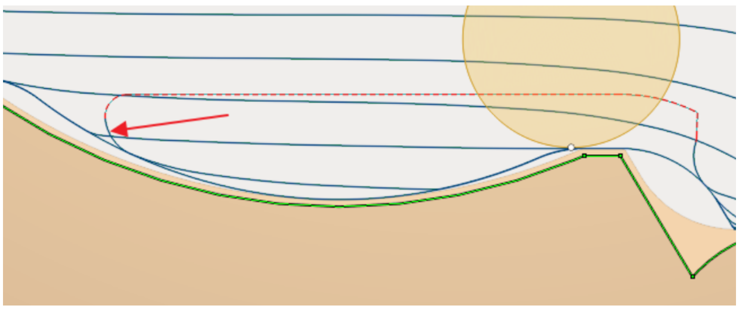
Direction defines direction of passes.



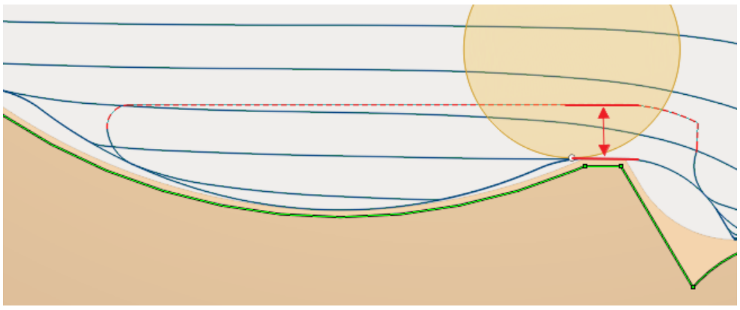
Rough rounding radius defines radius of cut in and cut out toolpath.



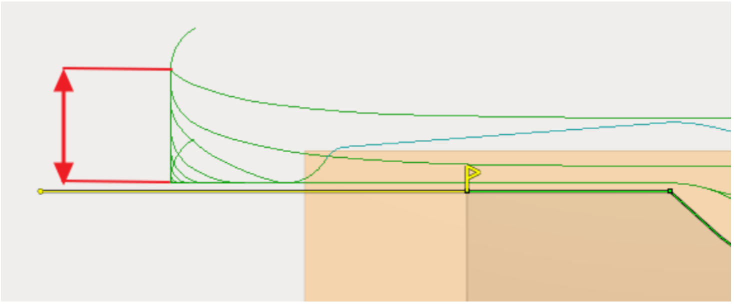
Linking radius



Lead Safe distance



Safe distance parameter defines the distance between the top groove level and the top curve point.

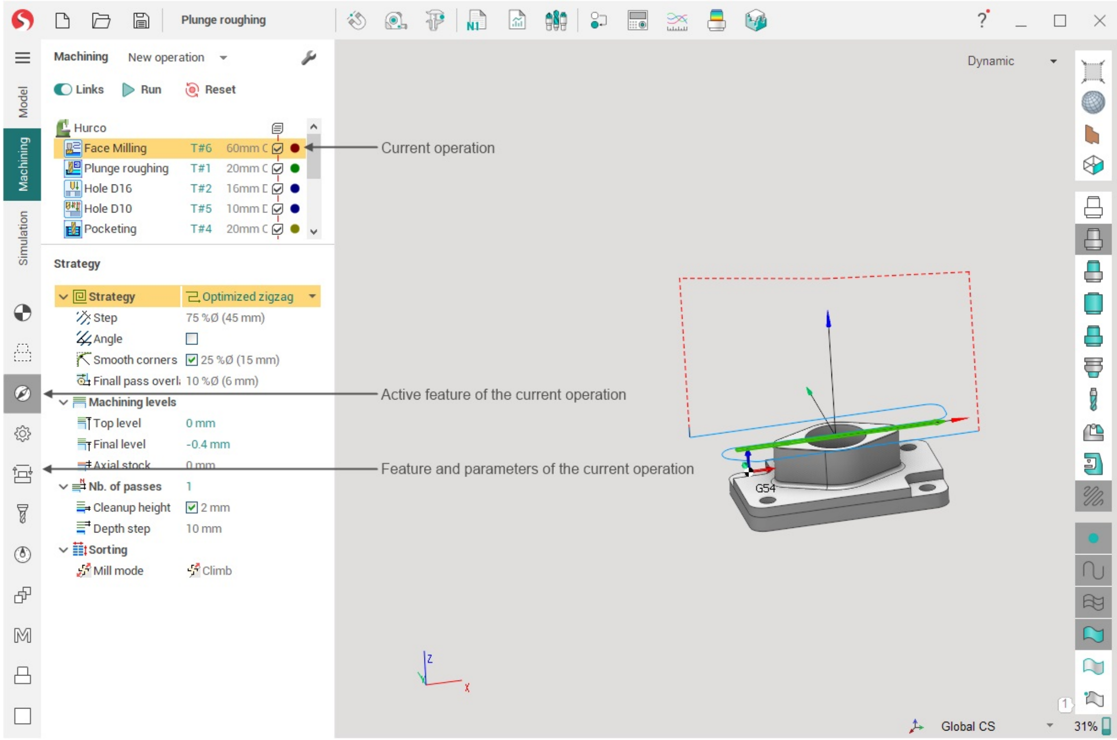


Although, when you turn on "check workpiece", the top level groove level will be selected automatically, and this option will not be active.

Check workpiece option removes all idle parts of the working motions that really do not cut the workpiece.

5.6.4 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.



See also:

Common principles of technology creation

Geometrical parameters of an operation

Defining part, workpiece and fixtures

Positioning of part at machine

Tool selection window

Tool change position

Operation local coordinate system

5 axes positioning

Approach and return rules

User operations

5.6.4.1 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.

See also:

Lathe machining

Lathe cycles

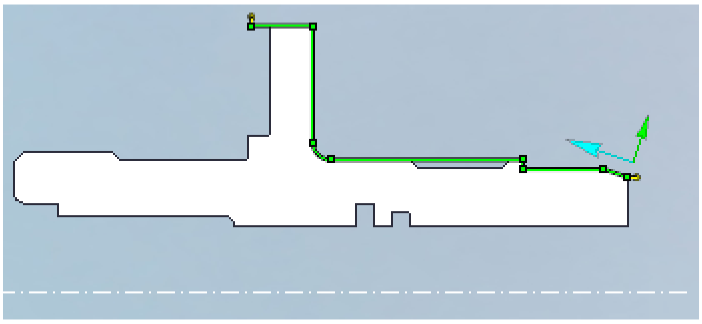
Job assignment item properties

Contour editing modes

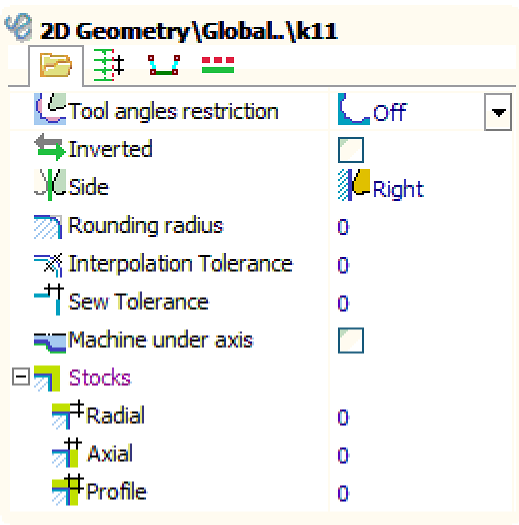
Job assignment element parameters

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.

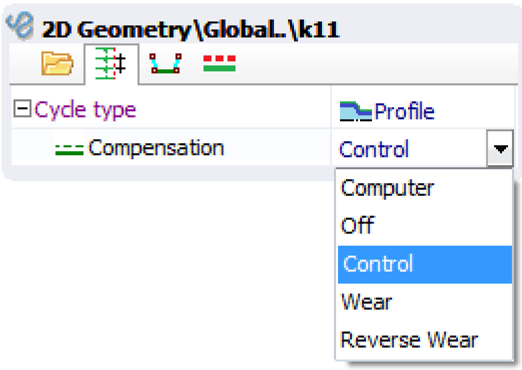
Interpolation tolerance allows to change contour by uniting set of its segments by arcs or lines. In this case, the resulting contour will be different from the original by not more than the specified value. This function is very convenient for splain curve processing.

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).

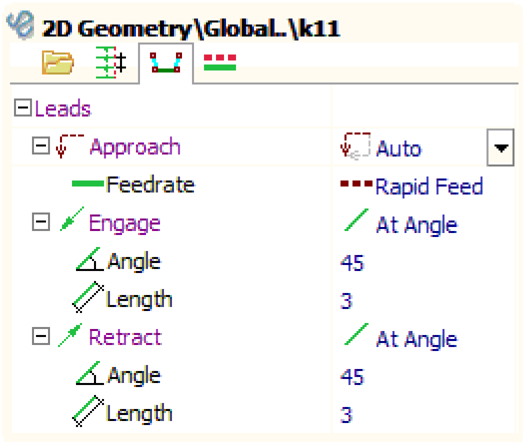
In the **Wear and Reverse wear** modes contours geometry transforms like in "**Computer**" mode. But correction commands are included before approach and retract frames too, like in the "**Control**" mode. Reverse wear mode differs from wear mode by correction direction.

"Machine under axis" parameter allows to create toolpath below rotation axis. So, all radial coordinates becomes negative.

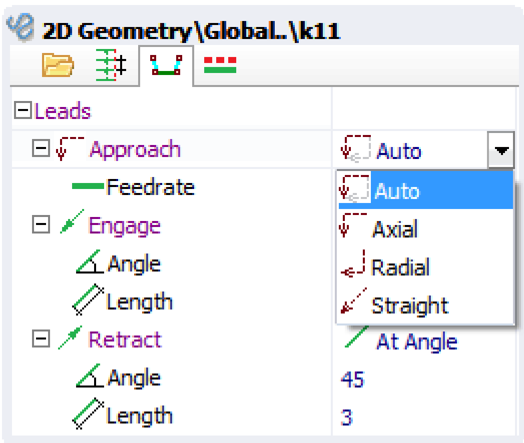
If "**Rounding radius**" parameter enabled and compensation mode is "Wear" or "Computer" then in outer corners of a contour the arcs with radius equal to tool nose radius will be formed. Sometimes it helps reduce the vibration of the machine from the abrupt change in direction of movement of the tool.

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.



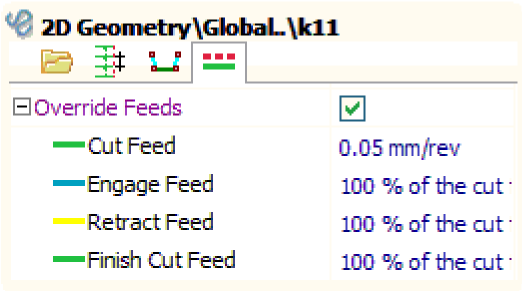
If there are several elements in operation transitions between them will be processed by rapid feed. The way of the approach to the first point of a contour from last point of the previous element is defined by an approach mode.



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.

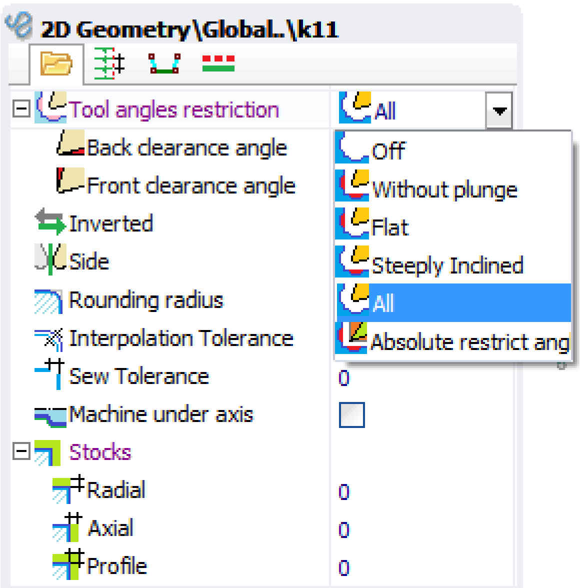


If "Override feeds" checkbox is not checked then contour feed is specified by operation parameters, else contour feed is specified in this window.

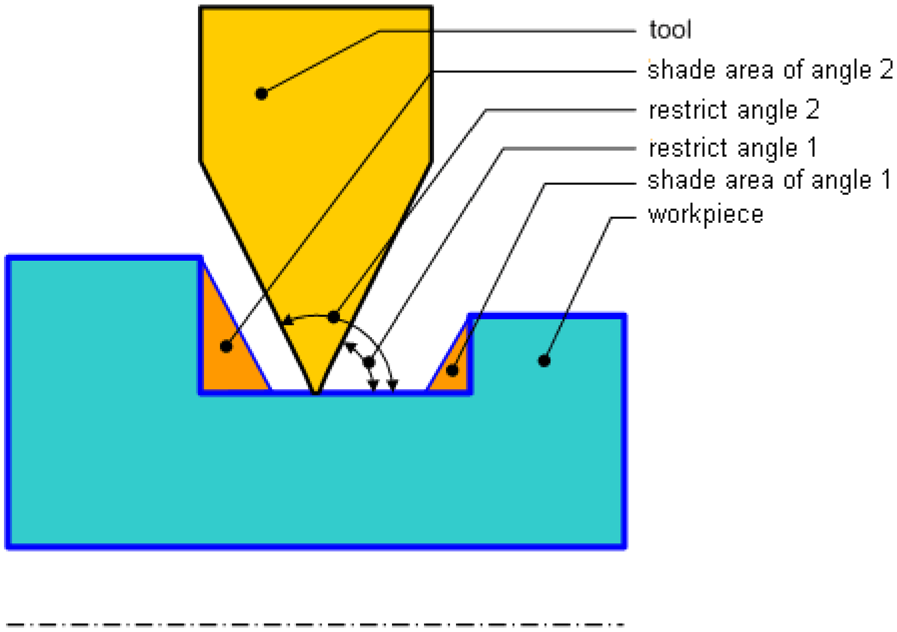
Tool angles restrictions

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.

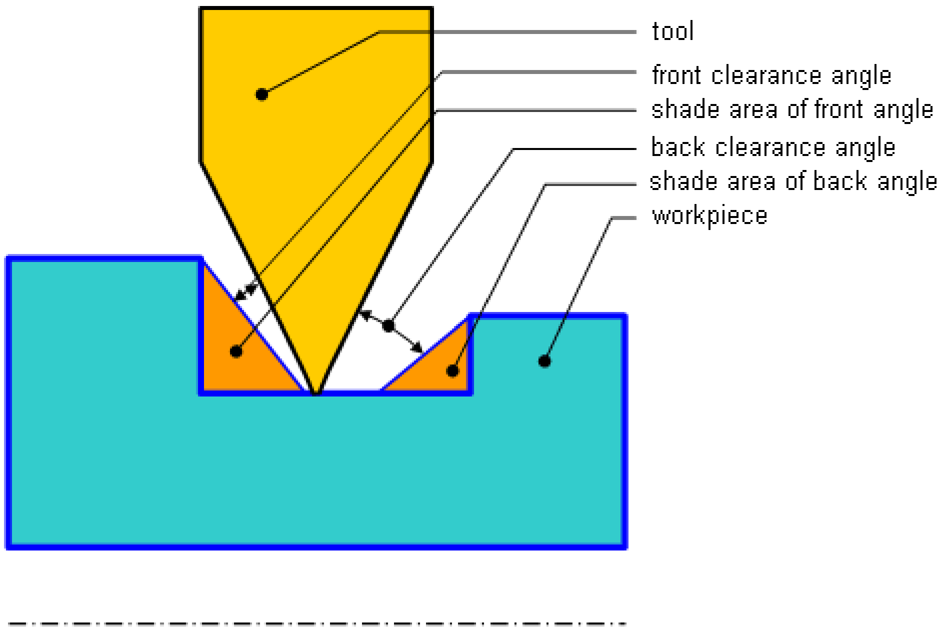


Absolute restrict angles mode allows to correct contour independently of tools real angles. These angles are measured from the horizontal line and are shown in the following figure.



"**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.

See also:

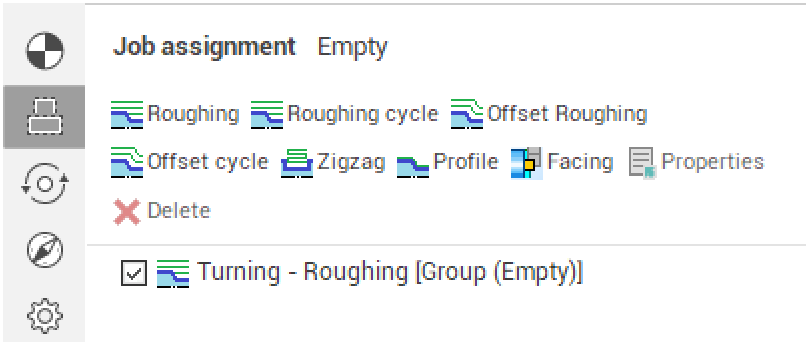
[Lathe machining](#)

[Job assignment](#)

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.

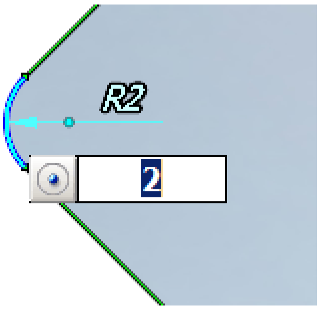
7. It is possible to undo and redo actions by clicking corresponding buttons at the panel.

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



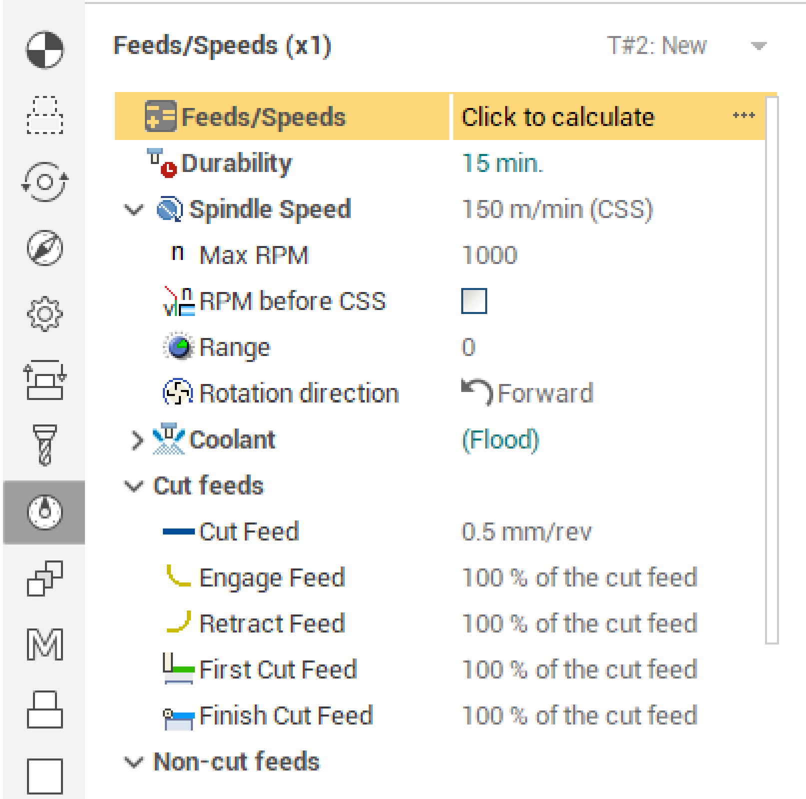
See also:

Lathe machining

Job assignment

5.6.4.2 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.

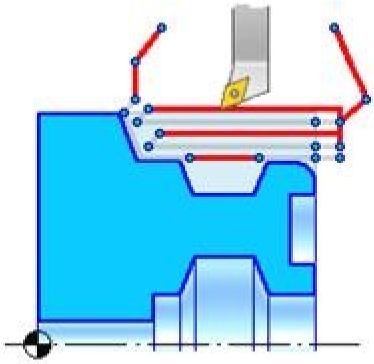
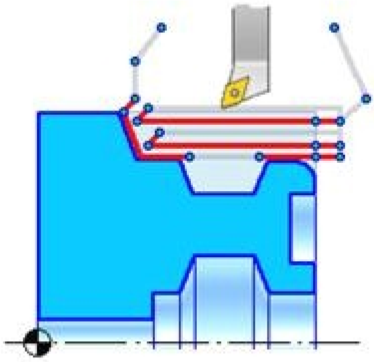
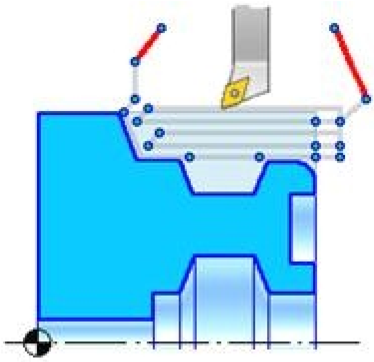
Range

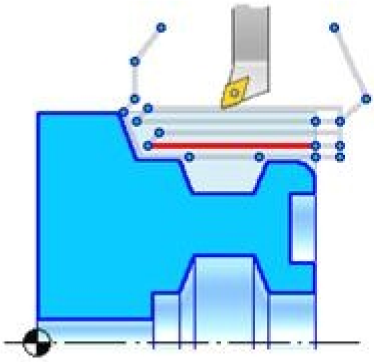
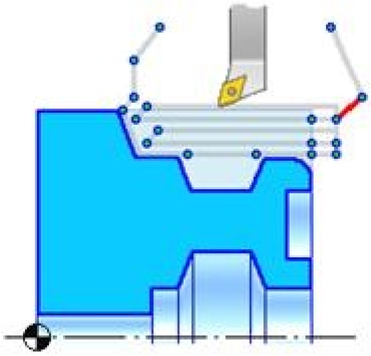
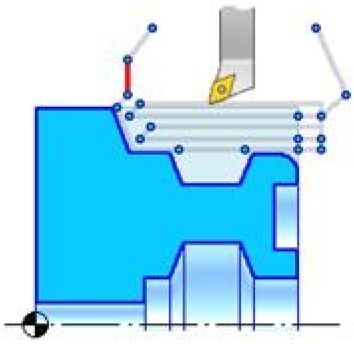
Range is the speeds range specific to the machine (with mechanical gear box).

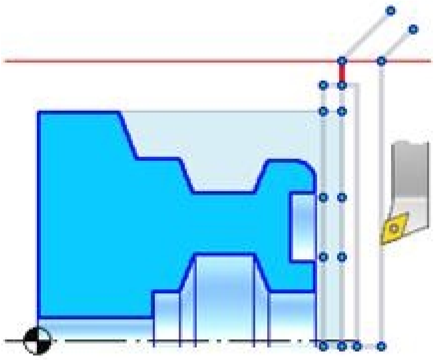
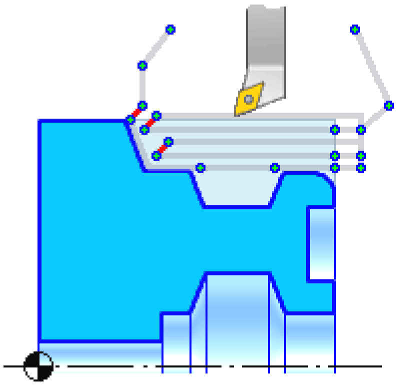
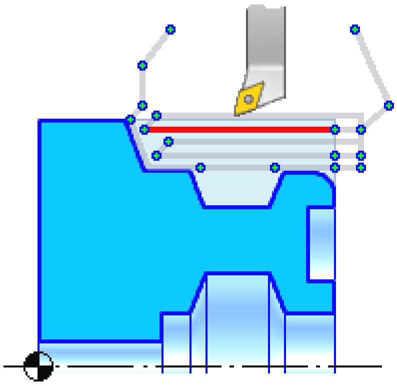
Feed

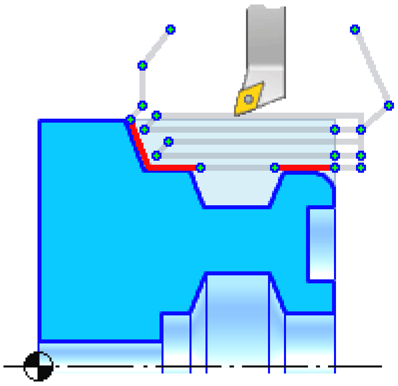
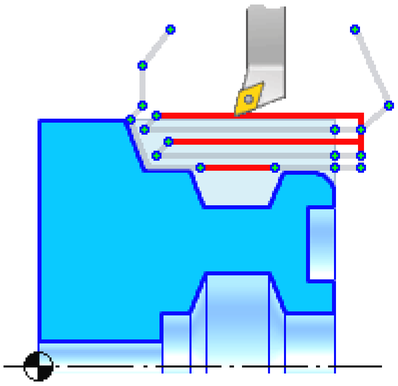
It is possible to set the feed for the different tool path segments. Depends on the operation type the tool path can contains the different segments, so the count of the feeds is changed.

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.*

See also:

Lathe machining

5.6.4.3 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 Job Assignment dialog window's <Inverse> field 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>.

See also:

Mill machining

Defining holes by coordinates

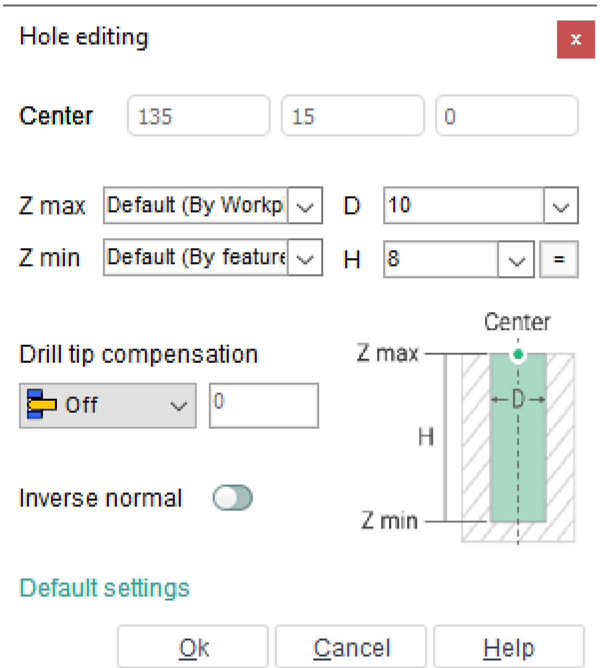
Defining holes by using a geometrical point object

Automatic hole recognition

Creating hole pattern

Defining holes by coordinates

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.



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.

<**Z max**> - top level mode:

- <By workpiece>;
- <By feature>;
- <Default> – parentheses will indicate the default value selected in the <**Default settings**> window;
- <Manual> – user set value manually.

<**Z min**> - bottom level mode:

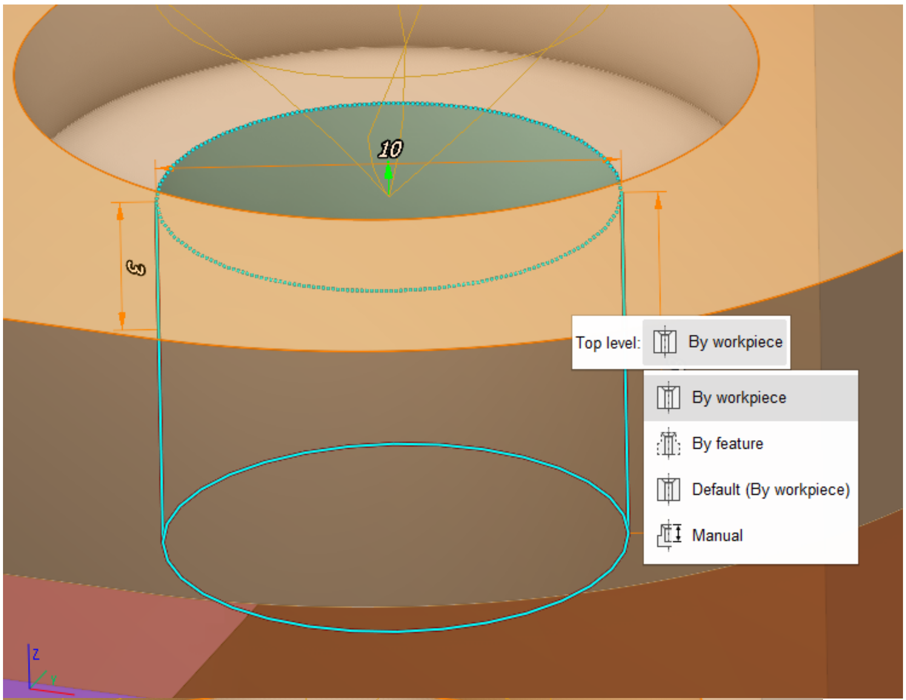
- <By feature>;
- <By model>;
- <Default> – parentheses will indicate the default value selected in the <**Default settings**> window;
- <Manual> – user set value manually.

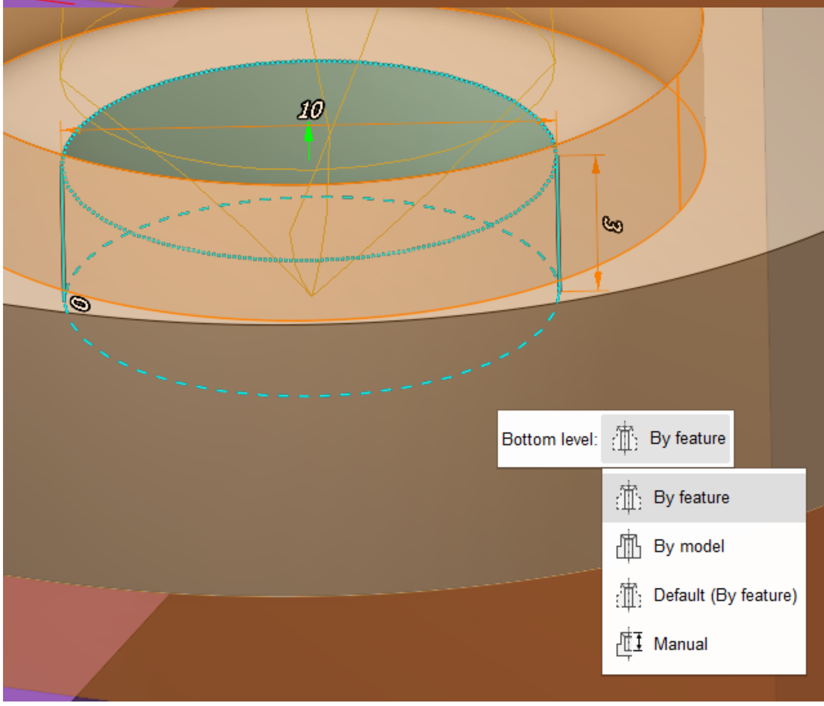
<Drill tip compensation> - choose the way the hole depth is specified:

- <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;
- <Auto> – hole depth is defined by the system based on whether the hole is blind or through.

<**Default settings**> button allows you to open window to set default values for <**Z max**> and <**Z min**>. 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:



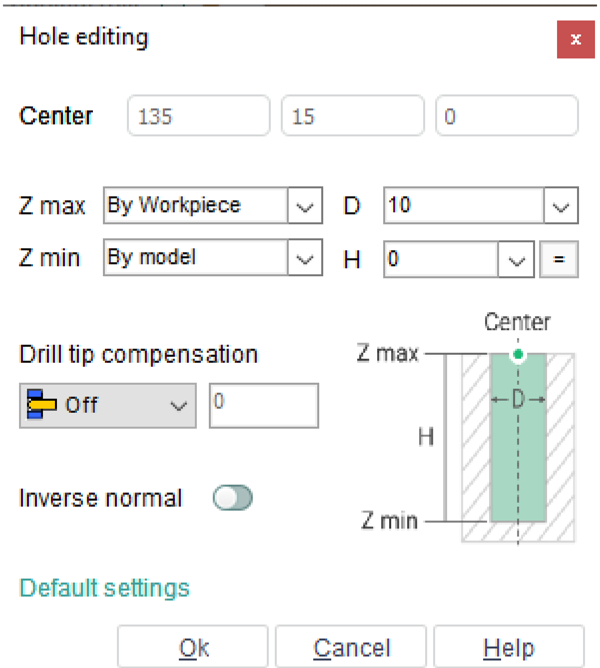


See also:

Job assignment for hole machining operation

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.



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.

See also:

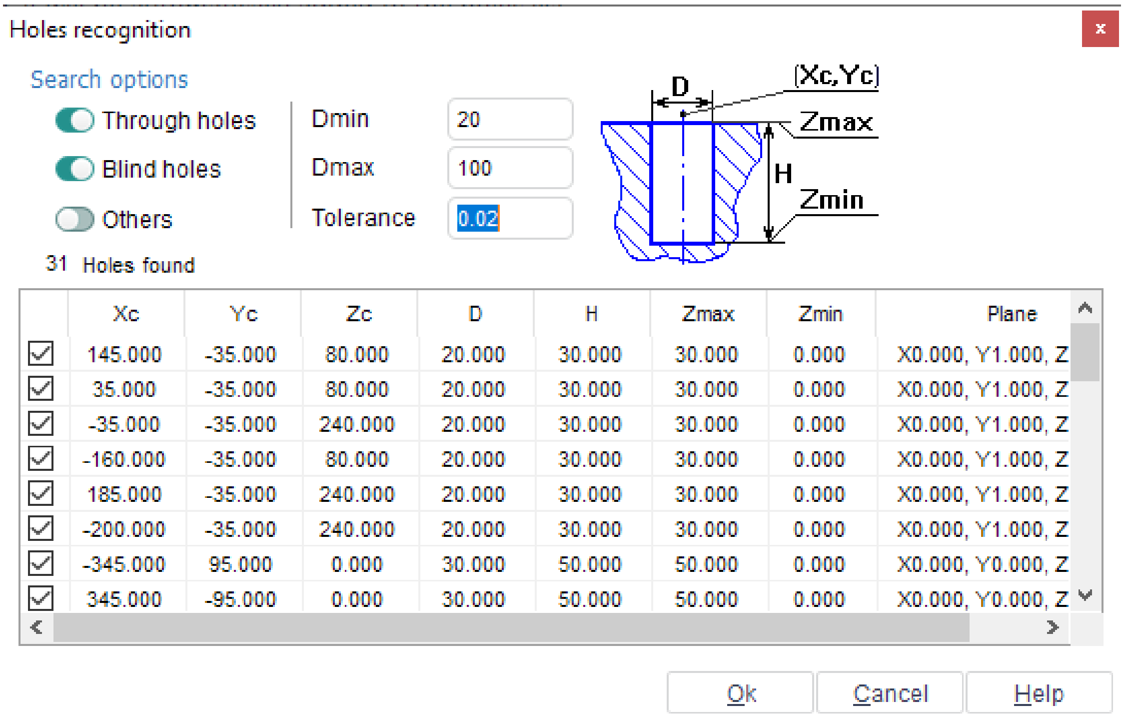
Job assignment for hole machining operation

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.

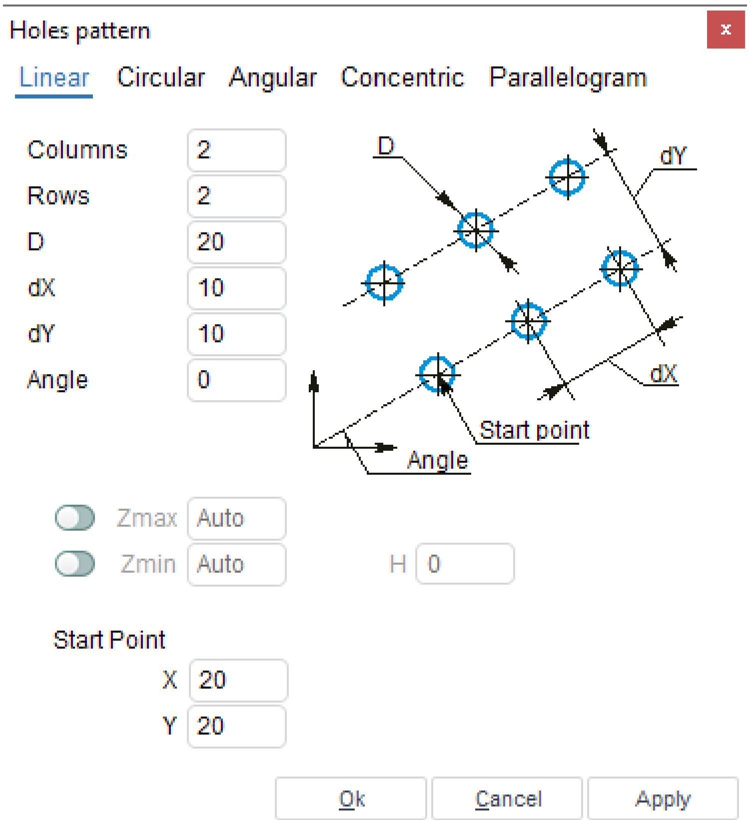
See also:

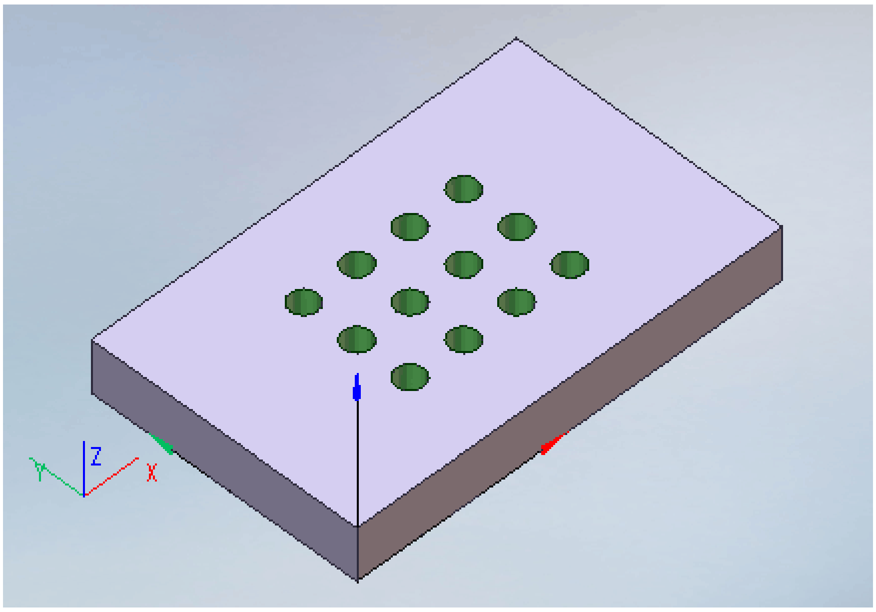
Job assignment for hole machining operation

Creating hole pattern

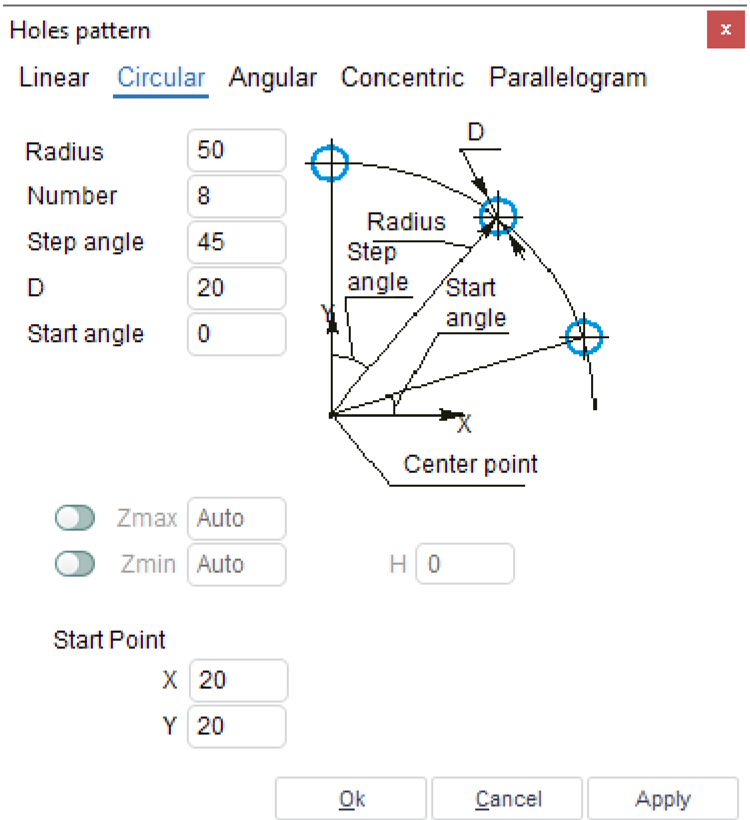
The system uses five types of pattern: <Linear>, <Circular>, <Angular>, <Concentric> and <Parallelogram>.

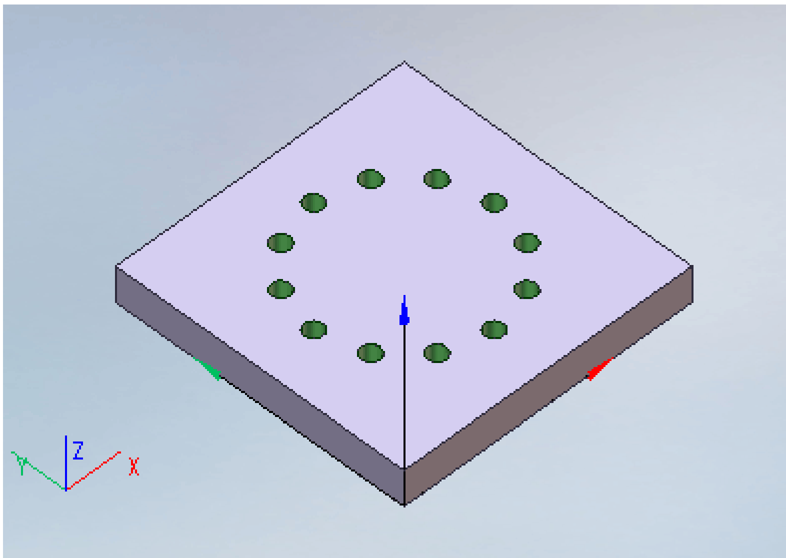
- On the <Linear> page user can create linear holes pattern:



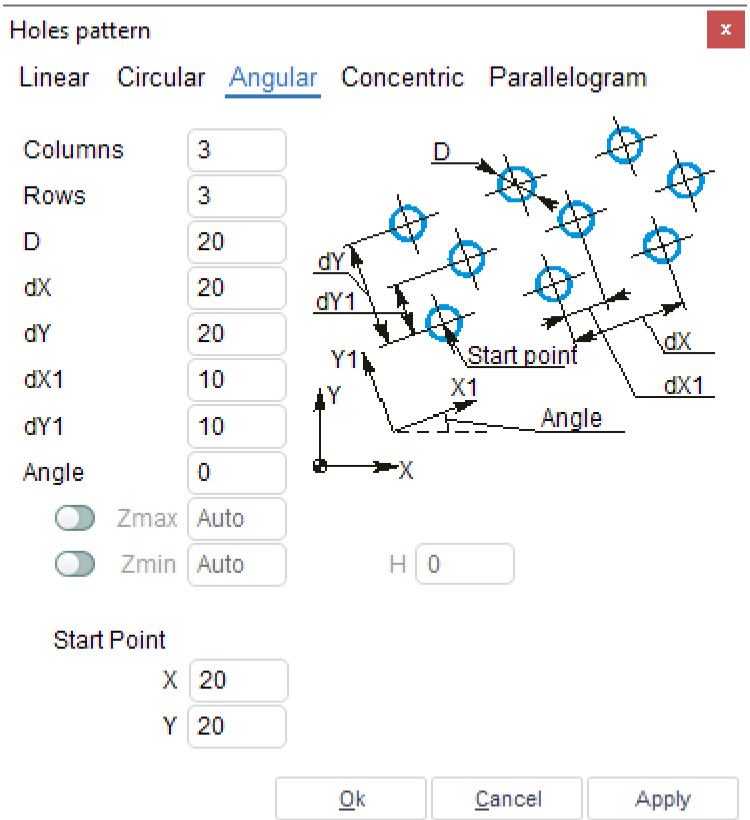


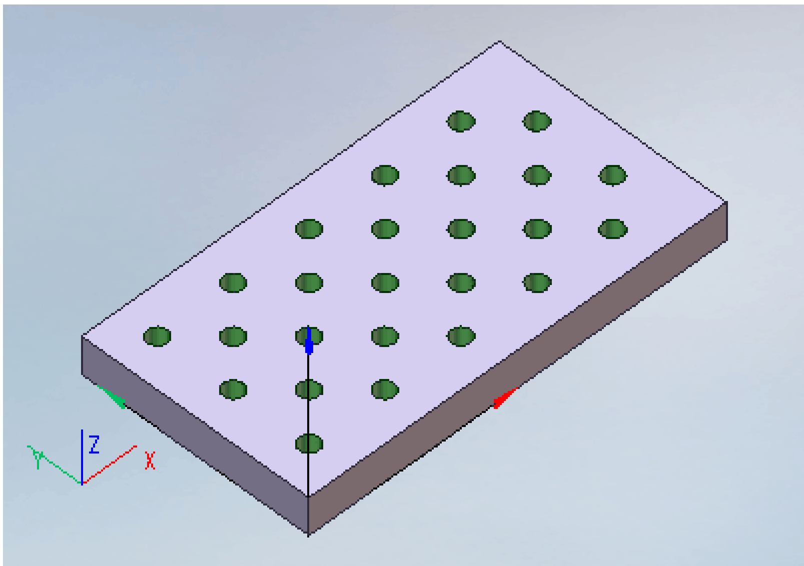
- On the <Circular> page user can create circular holes pattern:



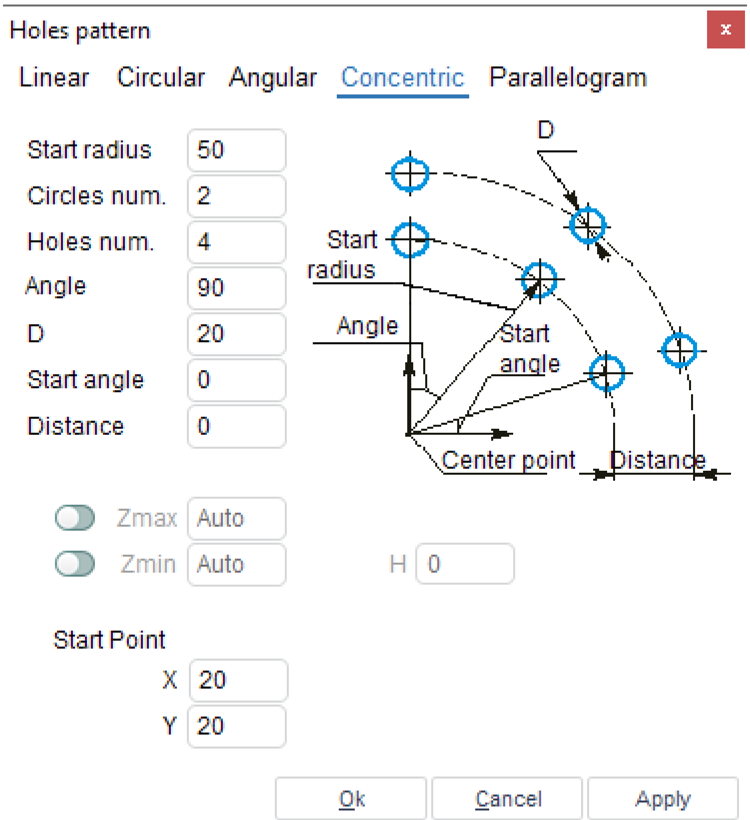


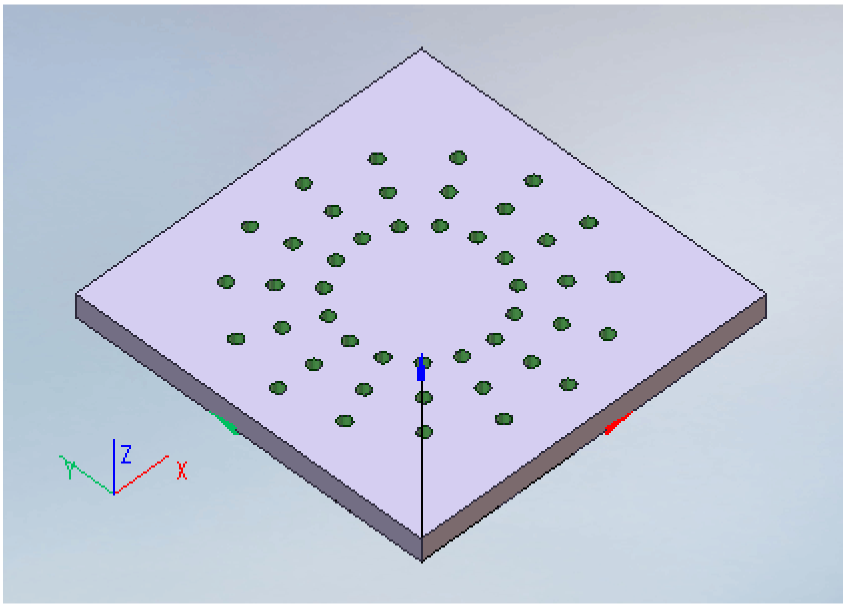
- On the <Angular> page user can create linear holes pattern:



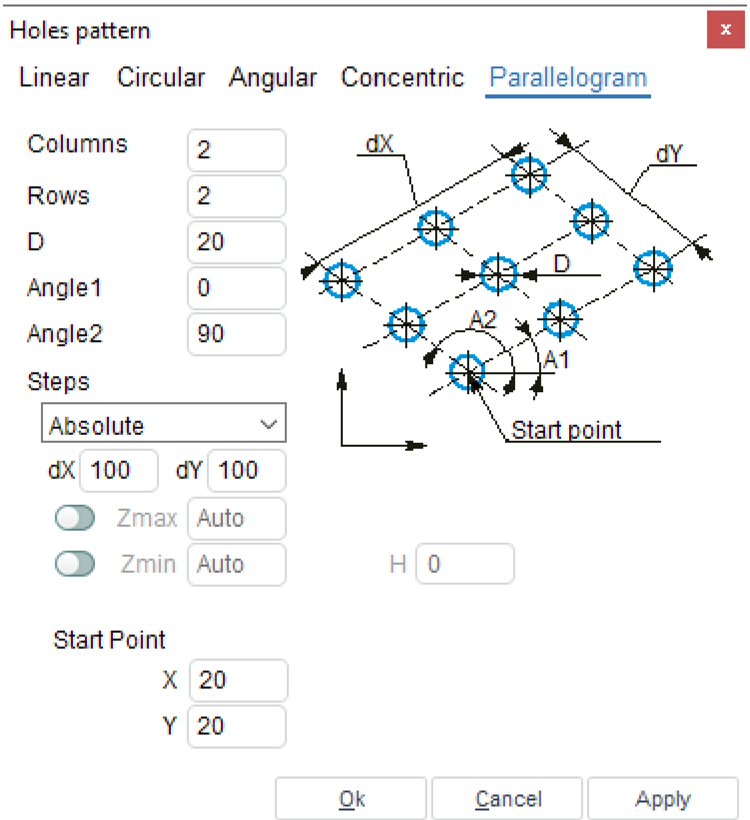


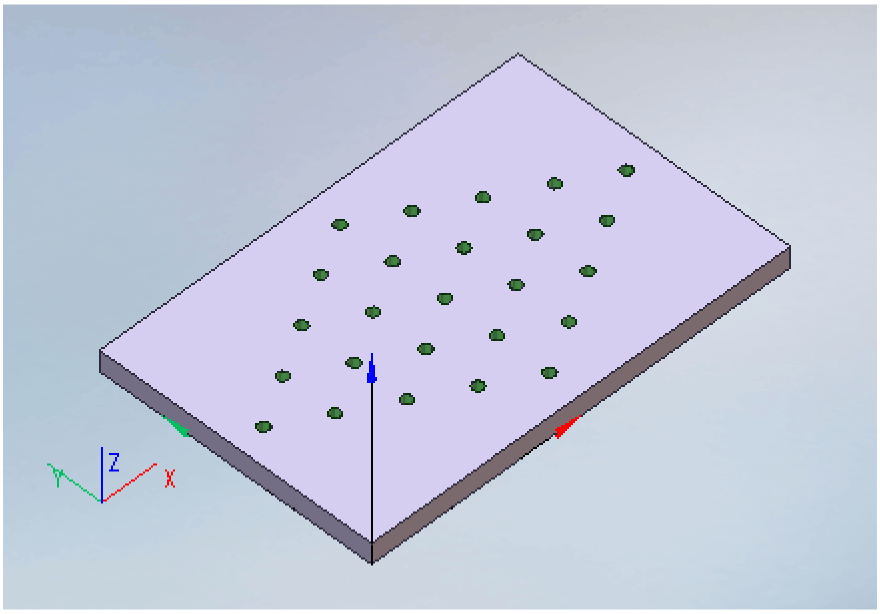
- On the <Concentric> page user can create concentric holes pattern:



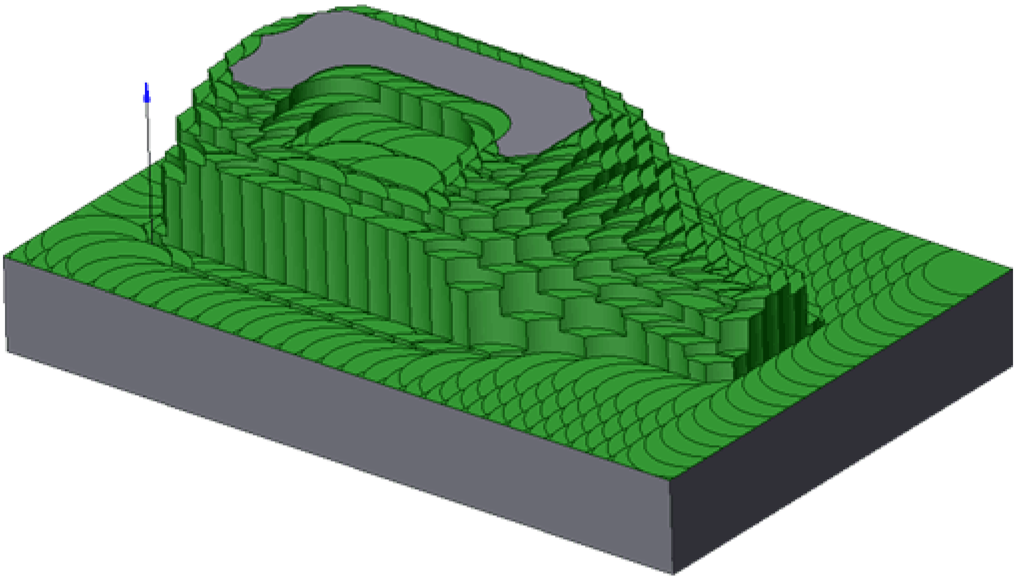


- On the <Parallelogram> page user can create parallelogram pattern:





Using the hole patterns together with automatic determination of hole levels allows someone perform roughing machining of the part by the axial plunging strategy.



See also:

Job assignment for hole machining operation

5.7 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. SprutCAM X 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 parts 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 operation 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.

1. Check an accuracy in setting numbers of tools. System doesn't control if in different operations various tools set on the same numbers.

2. 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.
3. Switch to control condition of cutting mode, check direction of spindle rotation, heat-removing and correctness of feeding values.
4. After any settings changing and recalculating make sure in absence of exclamation marks.

12. Generate a NC-program.

See also:

Lathe-milling machines types

Swiss lathes programming

Multi parts projects

Setting-up tooling

Positioning of part

The point of tool interchange

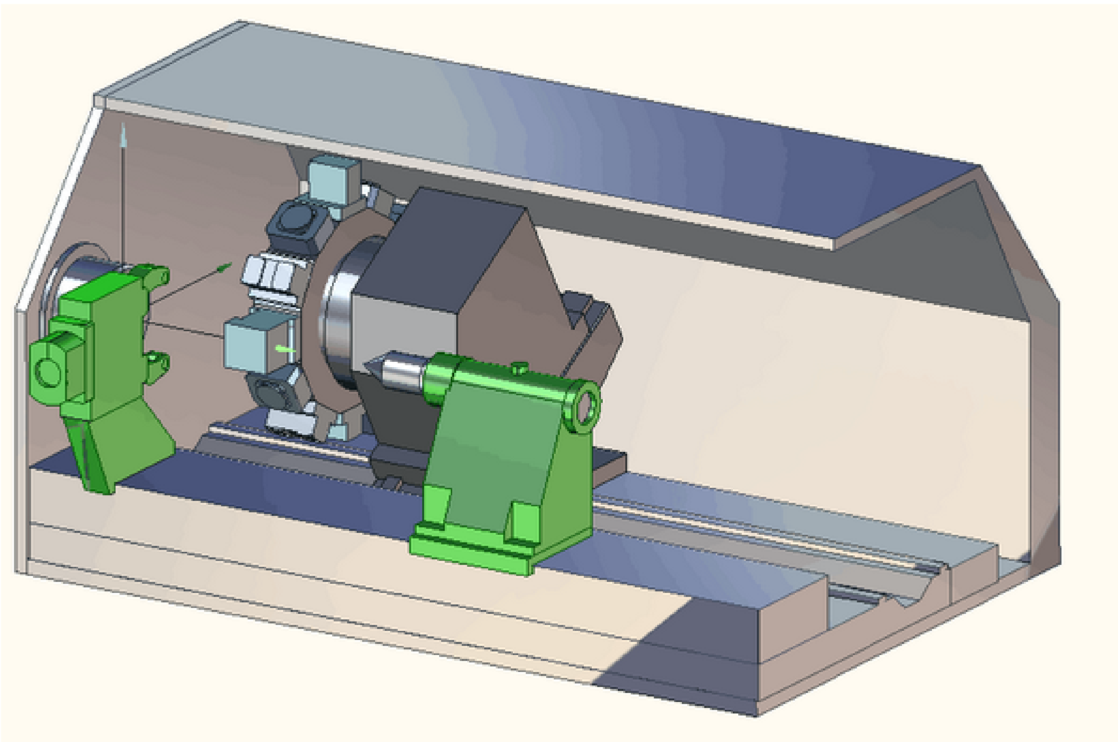
Positioning of tool

Obligatory testings before the final generation

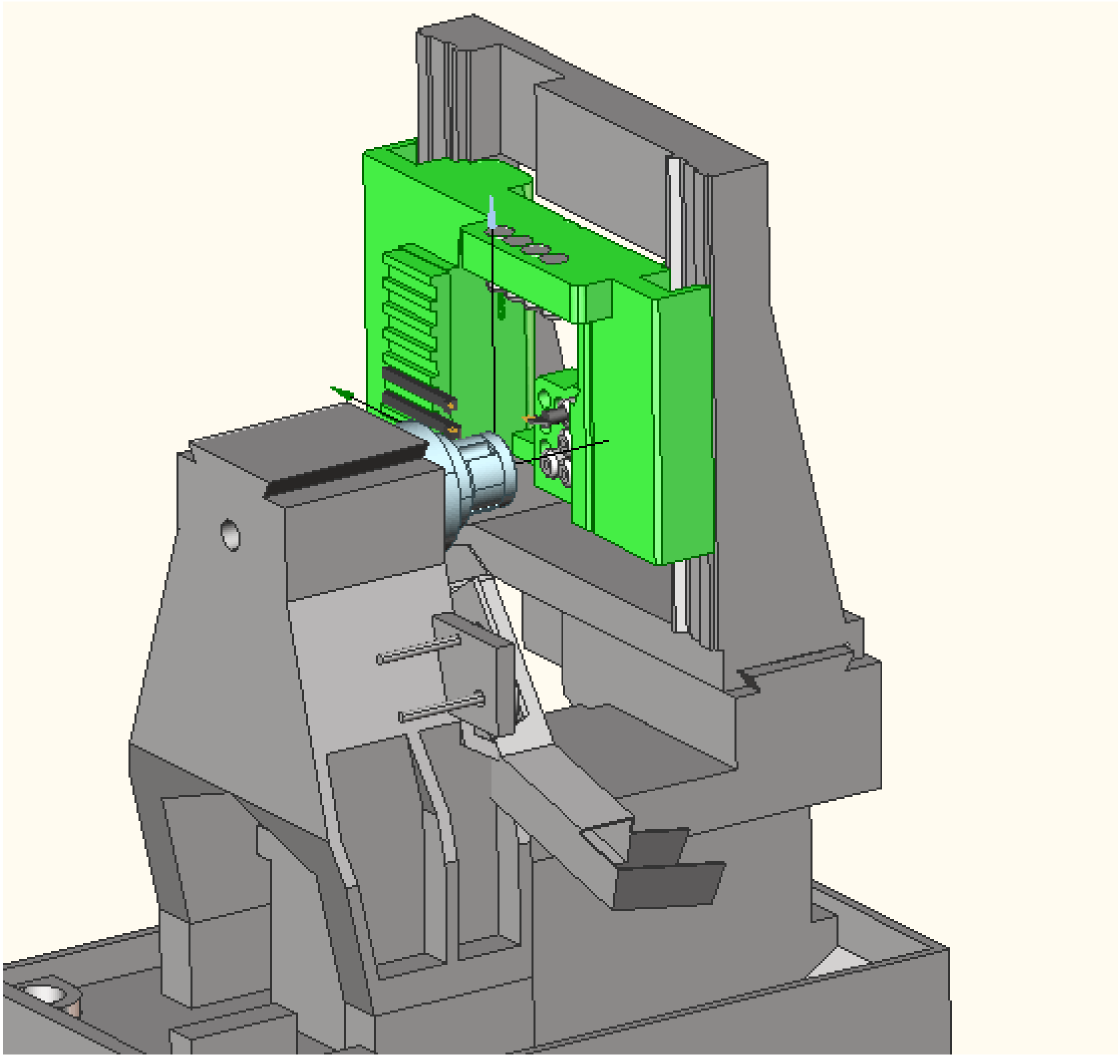
5.7.1 Lathe-milling machines types

SprutCAM X 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:



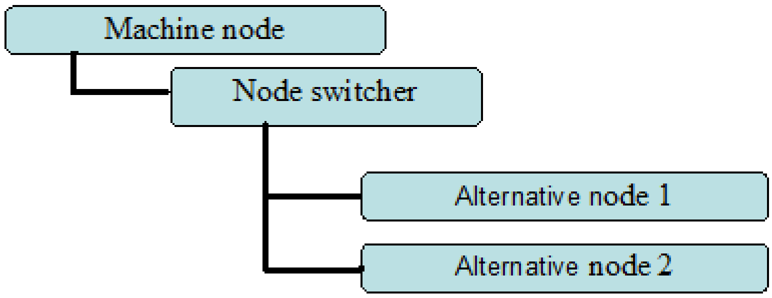
Using of machines kinematics allows to realize a visual inspection of collision all executing agency such as poppethead, collar plate, tool turrent and a tool prominent from it.

See also:

Treatment at lathe-milling processing centers

5.7.2 Setting-up tooling

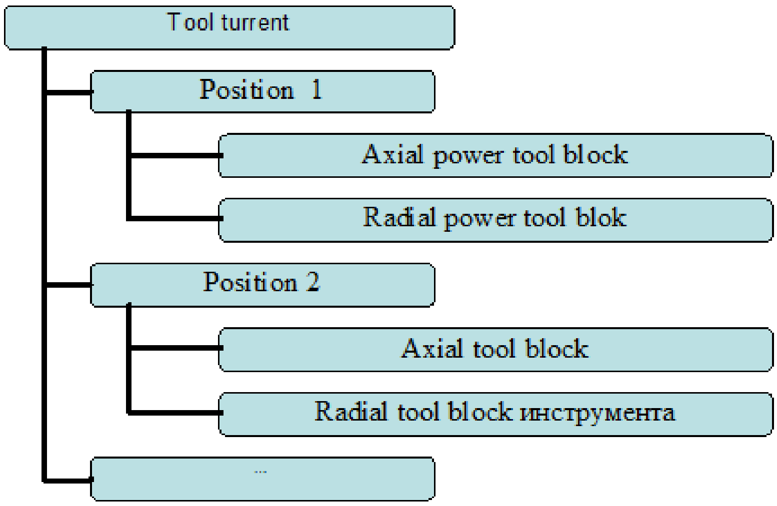
The kinematics of many machines may be appreciably changed by its changeover. For a possibility of quick and controlled machine changing in kinematics provided special node-switcher.



Node switcher at kinematic scheme of machine

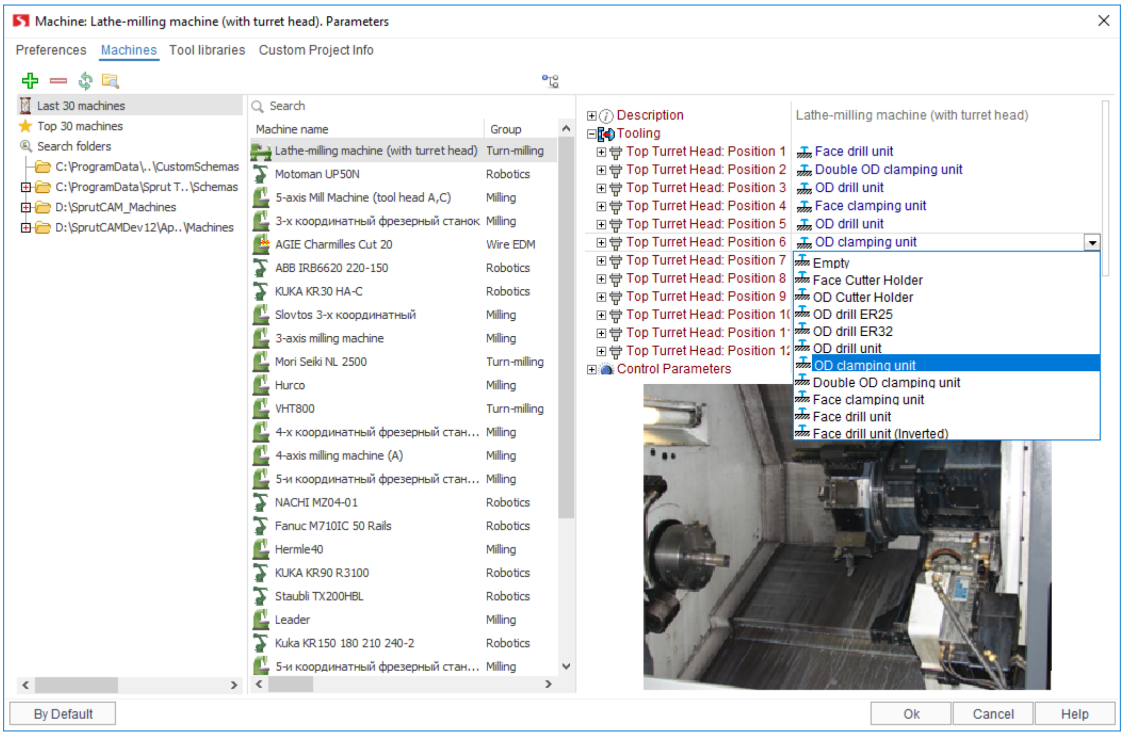
Node – switcher make possible to join to parent node of machine one from alternatives nodes. If in scheme presents a node-switcher, it shows in tree like drop-down list, from which one of alternatives node may be chosen.

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>



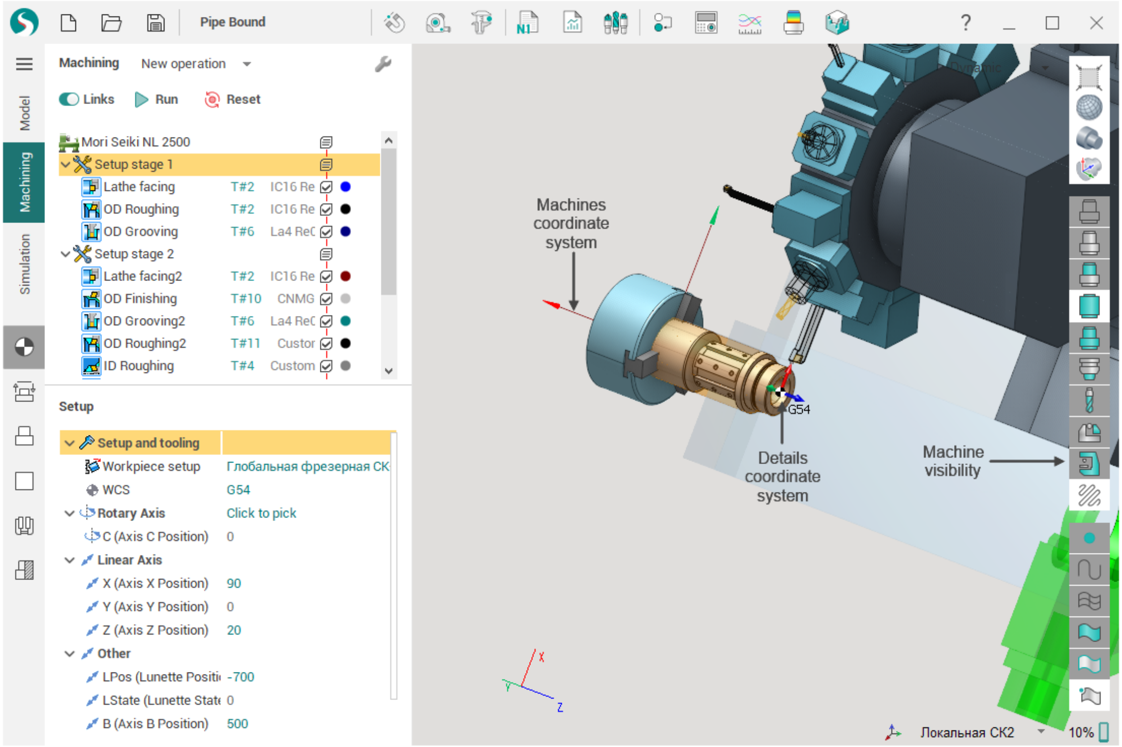
See also:

Treatment at lathe-milling processing centers

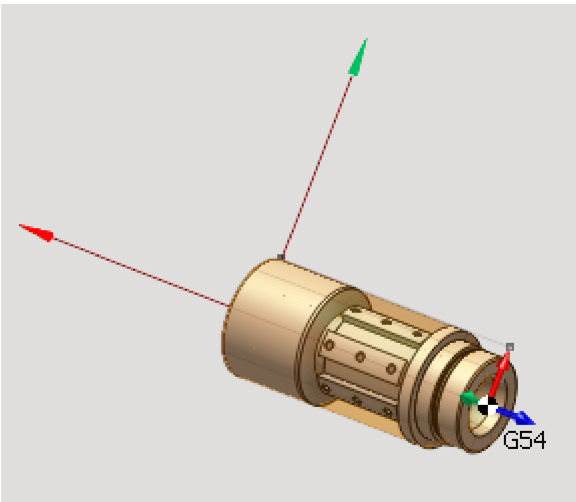
5.7.3 Positioning of part

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.

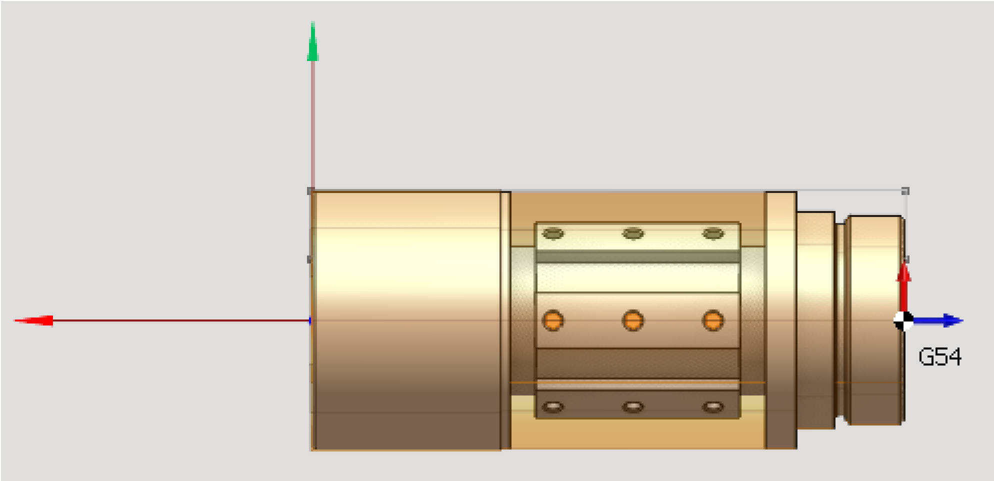
Necessary to discriminate part coordinate system and machine coordinate system. Detail coordinate system attached to detail and e.g. rotate with the detail when a spindle was rotated. Machine coordinate system is shown in left bottom corner of the screen when the machine is visible.



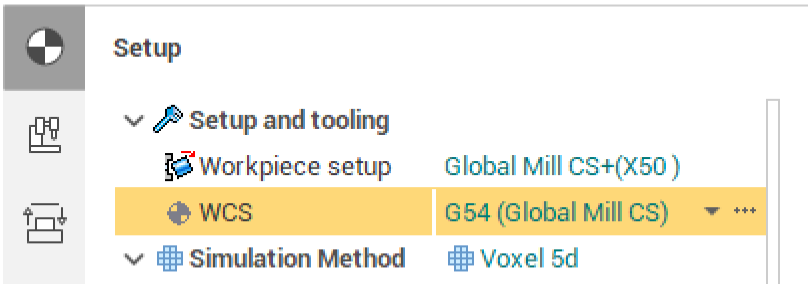
When visibility of machine is OFF auxiliary axles show a part coordinate system.



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.

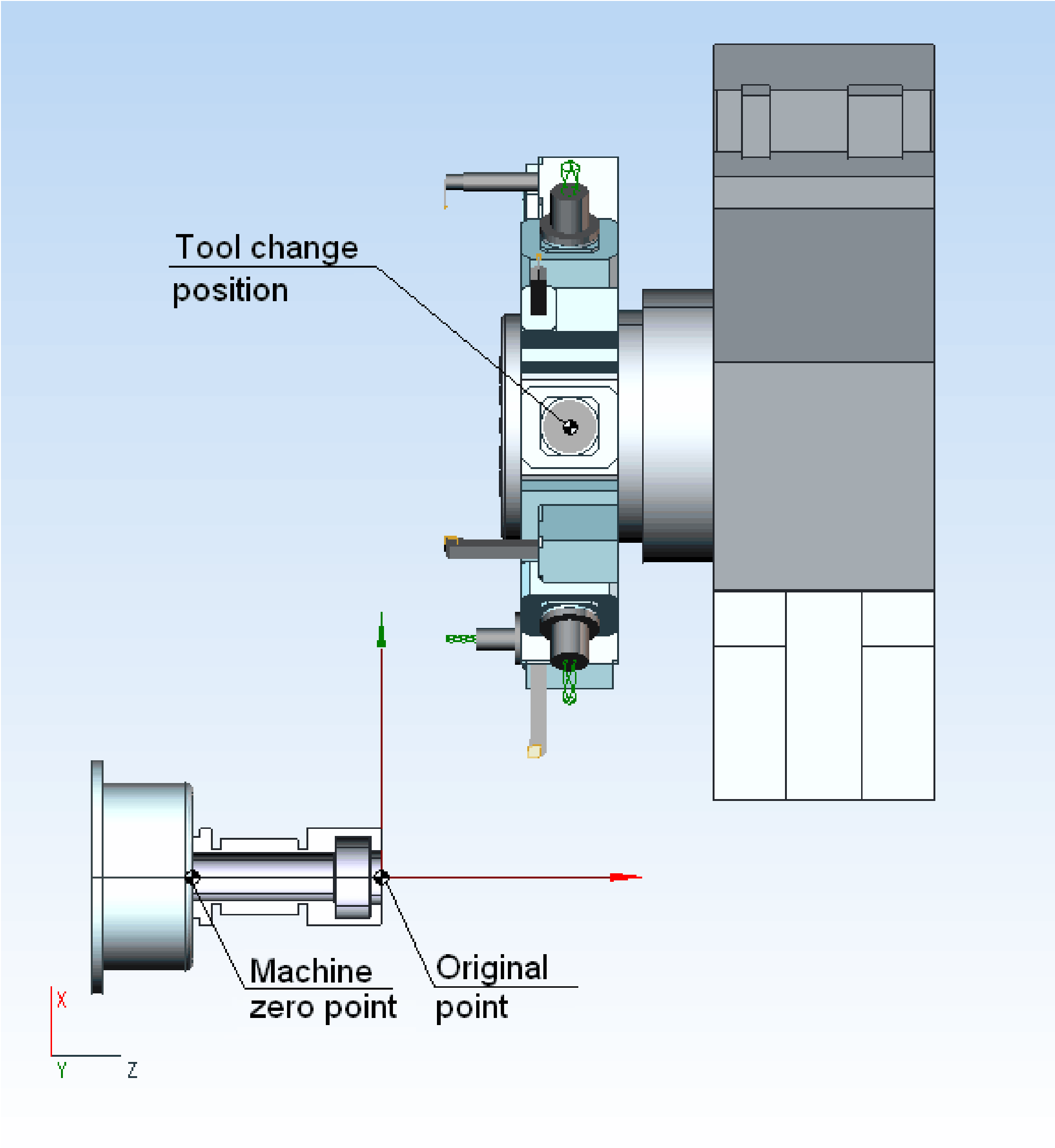


Machine zero point usually sets on a spindle nose. As result a detail will appear inside of machine. For a correct visualization necessary to set a detail zero point in machine coordinate system.

See also:

Treatment at lathe-milling processing centers

5.7.4 Tool change position



The coordinates of the point of tool interchange are counted off from the 0 of the machine. In SprutCAM X 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

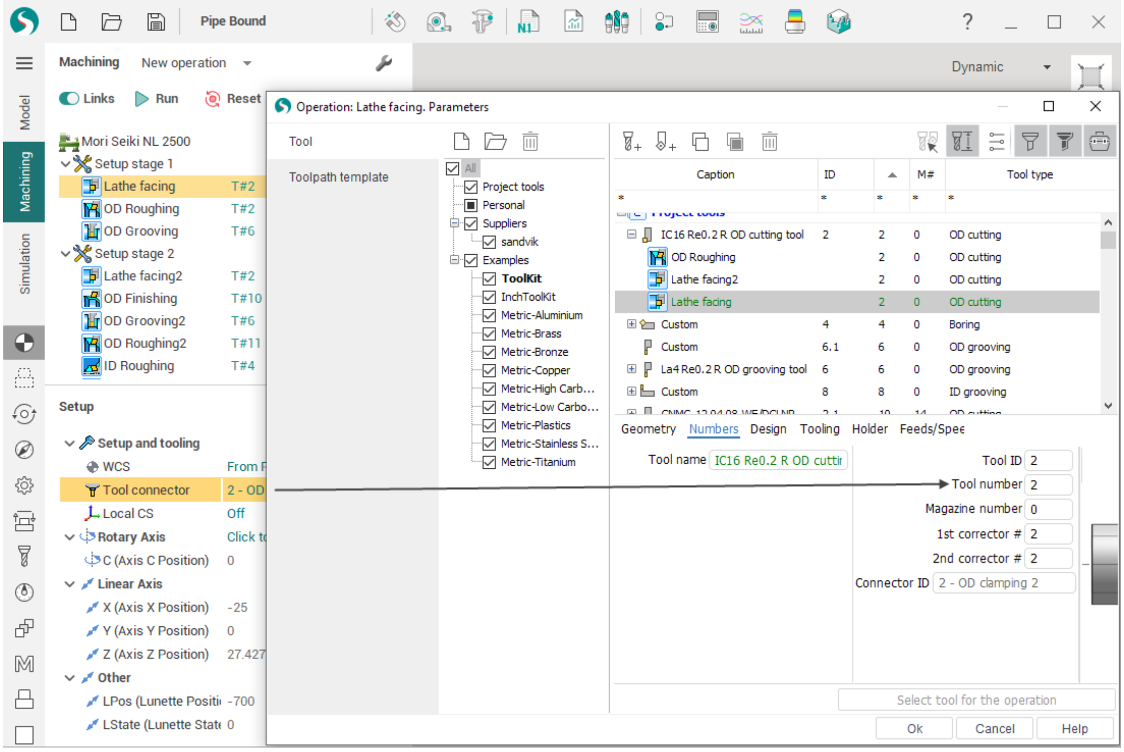
The following step is executing by the command **<GOHOME>** (G28), in other words it doesn't give coordinates.

See also:

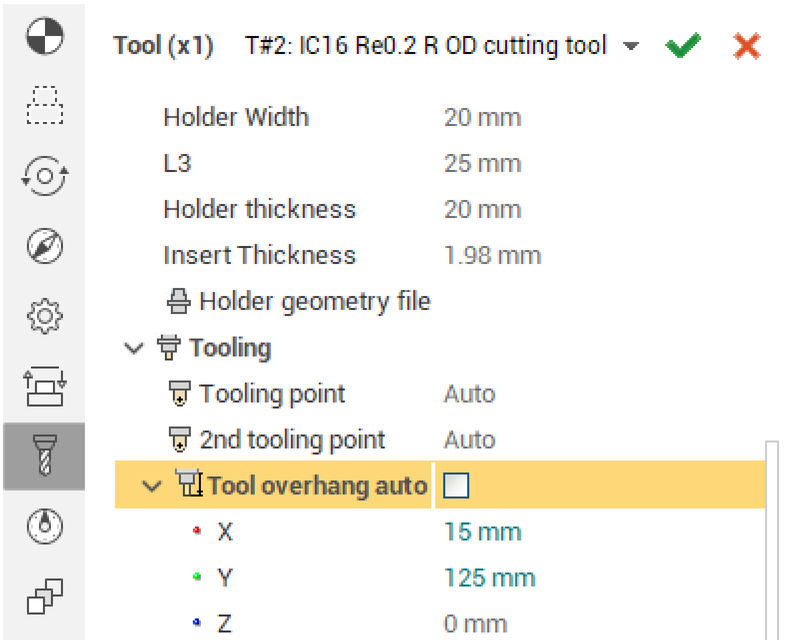
Treatment at lathe-milling processing centers

5.7.5 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.



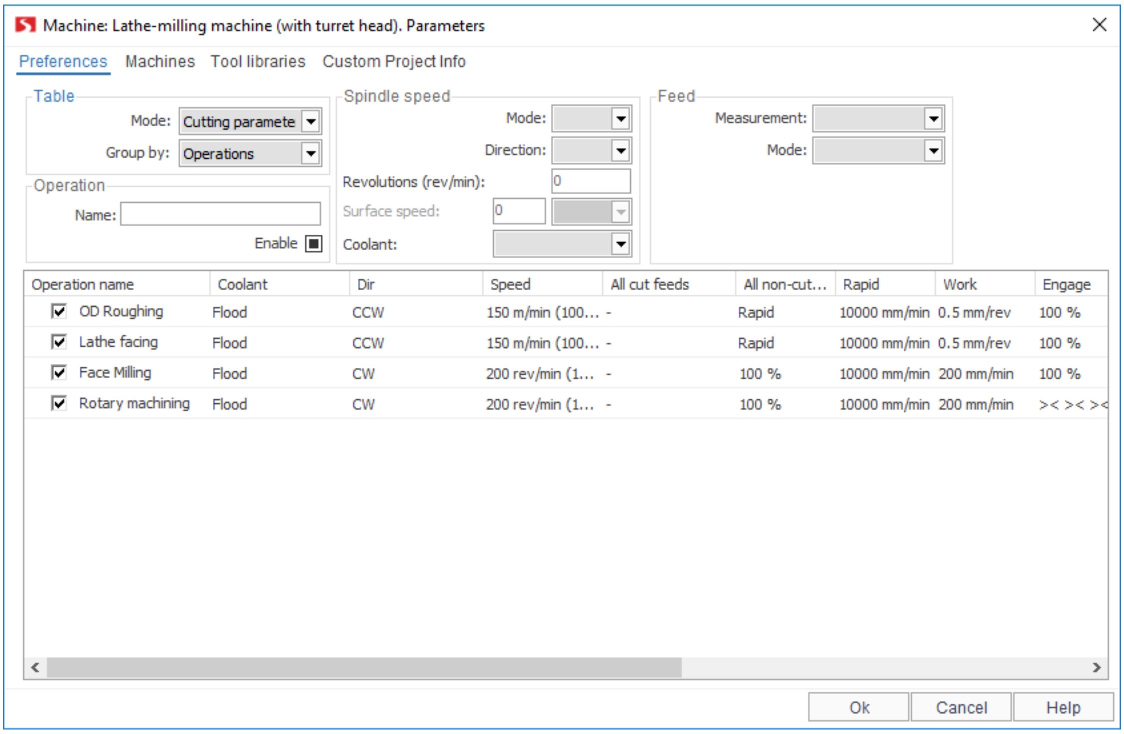
See also:

[Treatment at lathe-milling processing centers]

[The point of tool interchange]

5.7.6 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.

See also:

Treatment at lathe-milling processing centers

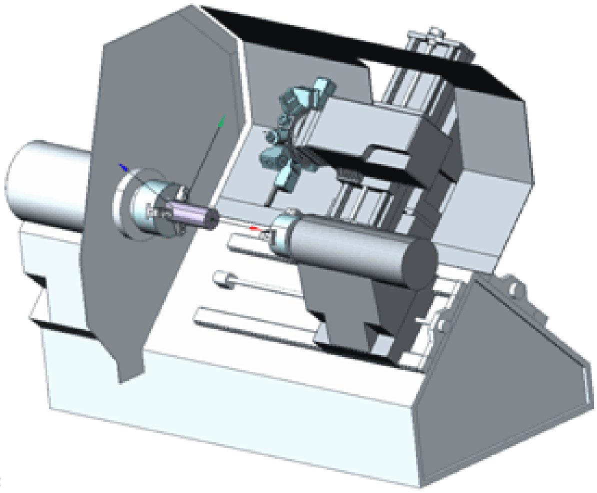
5.7.7 Counter spindle machining

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.

SprutCAM X can create NC-programs for counter spindle machining for both lathe and lathe-milling machining technologies.

For a more detailed explanation of SprutCAM X approach to the counter spindle machining please take a look at the following sample project of shaft machining.

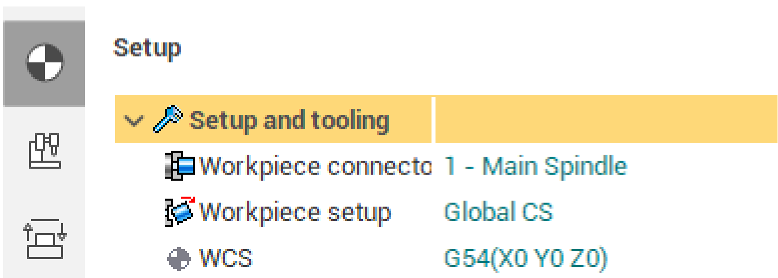
The machine is the <MaxTurn 65 with Counter Spindle>.



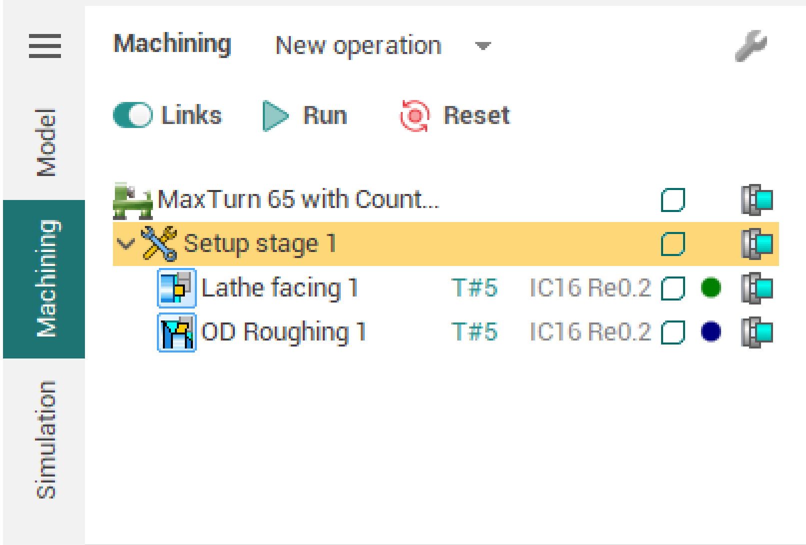
What is necessary to know before developing counter spindle machining process:

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.

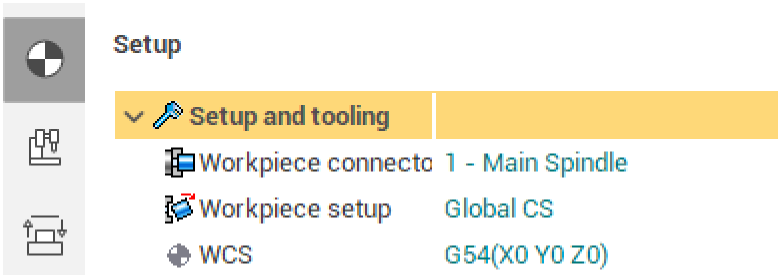


Please note: *To avoid mistaking active spindle in consequent operations specify <**From the previous operation**> value for the <**Current workpiece holder**> field.*

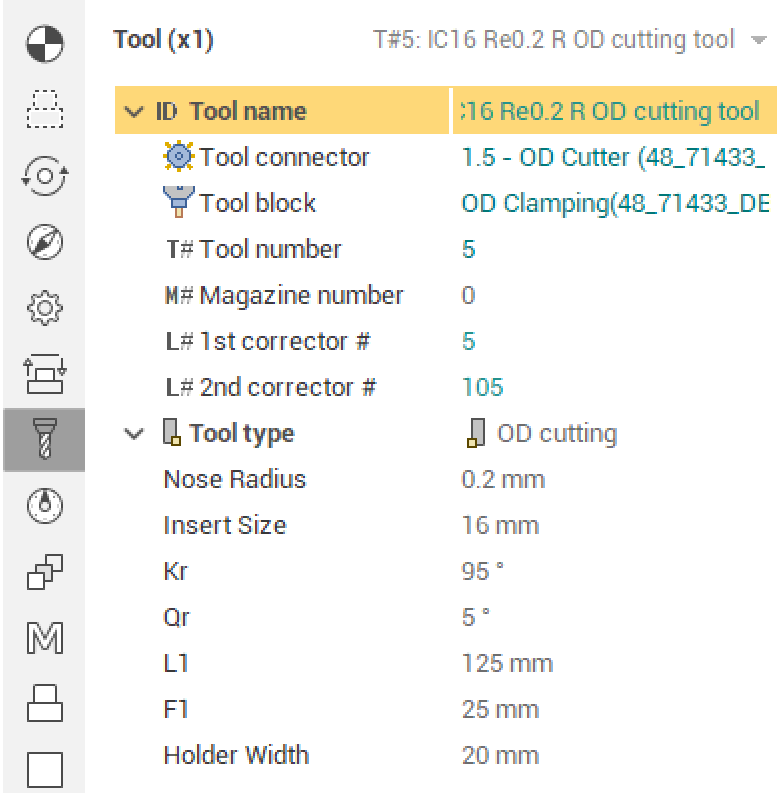
To refix workpiece from one spindle to another create a "Stage" or "Part" operation. For more information on the multi part projects see Multi parts projects*. To simulate the transfer of the part from the main to counter spindle you can use the* Turn take over *operation.*

After refixing create operations to machine the rest surfaces. Operations can be either lathe or lathe-milling.

In the **Setup Stage** or **Part** operation specify that machining is performed in another counter spindle. To do this specify, <2-Counter Spindle> in the <**Workpiece connector**> field, and specify new coordinate system in the <WCS> field.



Specify the appropriate hand of tool in the toolholder parameters on the <Tool> page.



SprutCAM X simulates both the machining operations and auxiliary operations so that user can control workpiece and machine movement and provide necessary modifications.

See also:

Multi parts projects

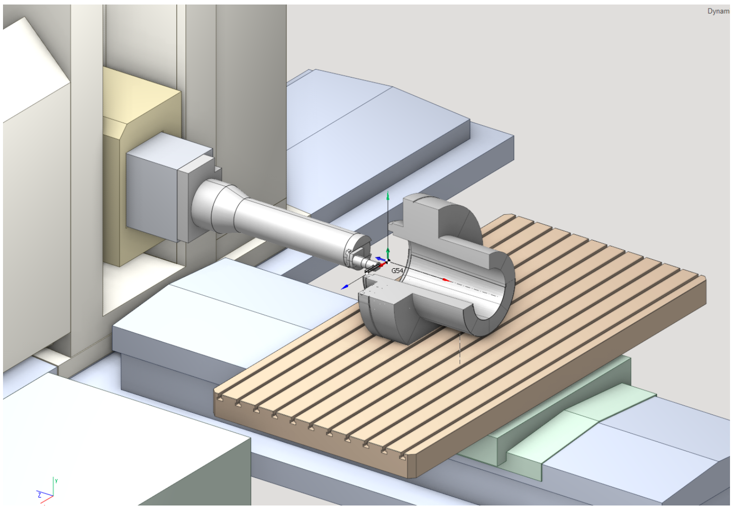
Swiss lathes programming

Turn take over

Sub spindle working

5.7.8 U-axis turning

5.7.8.1 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.

5.7.8.2 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

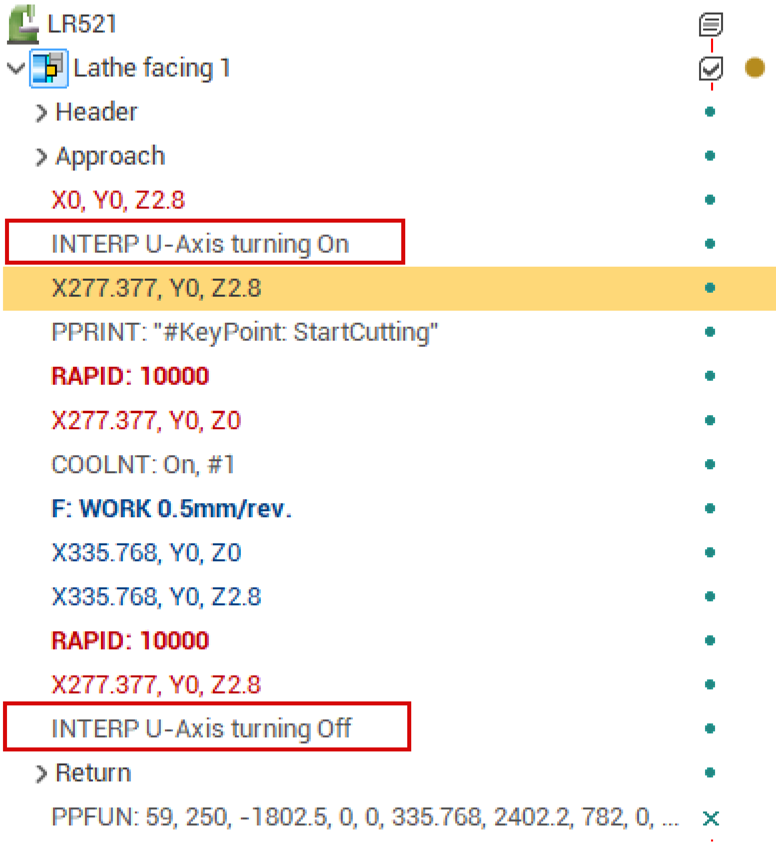
```
<TrevisanSubMachine>
   <!--turning-->
   <ToolNode>AxisU</ToolNode>
   <WrkNode>Workpiece1</WrkNode>
   <XAxisID>AxisX</XAxisID>
   <YAxisID>AxisY</YAxisID>
   <ZAxisID>AxisZ</ZAxisID>
   <ToolAxisID></ToolAxisID>
   <UAxisID>AxisU</UAxisID>
   <OriginG54BaseNode>AxisW</OriginG54BaseNode>
   <ApproachRule>G53 Z0; G53 B; LHPos;U;XY; G53 W;Z;</ApproachRule>
   <ReturnRule>Z10;XY </ReturnRule>
 </TrevisanSubMachine>
```

5.7.8.3 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. SprutCAM 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.



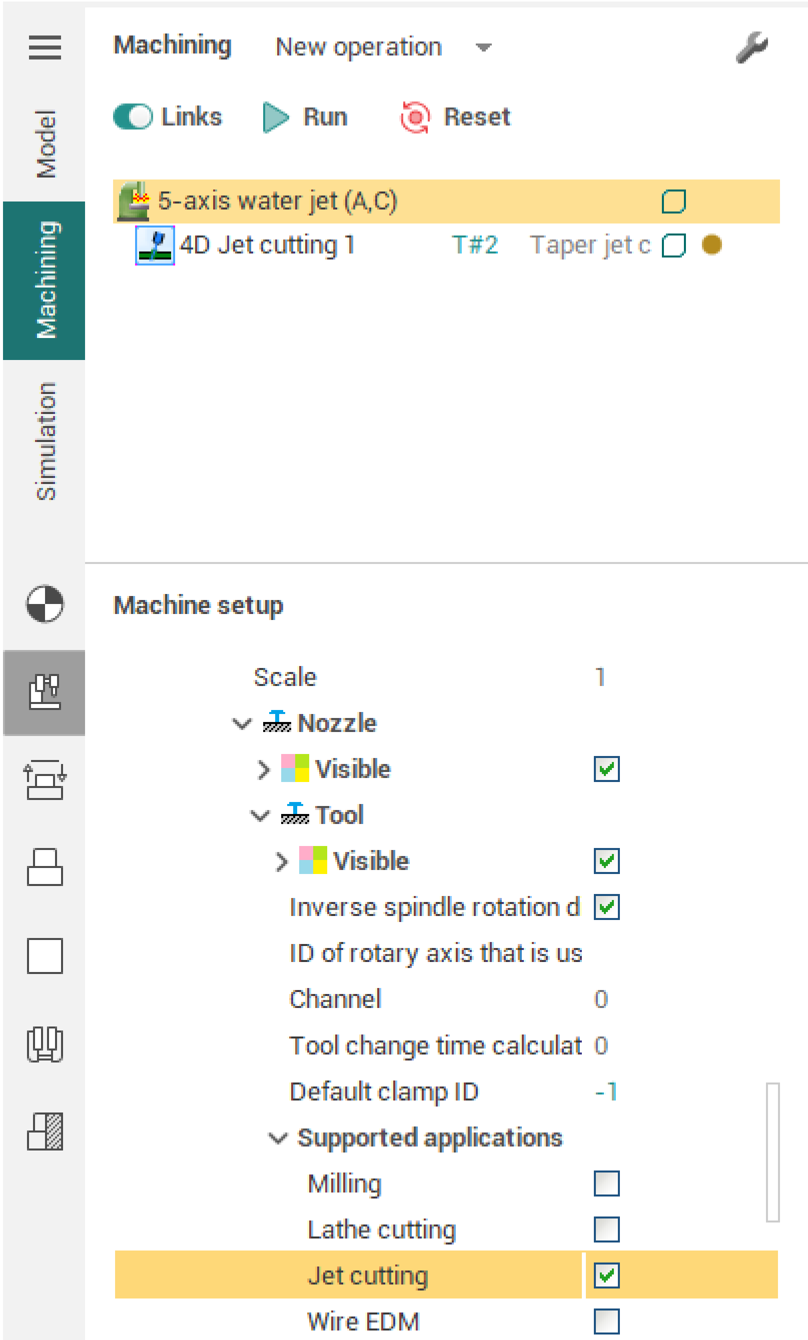
5.8 Machining on cutting machines

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.



See also:

[Jet cutting]

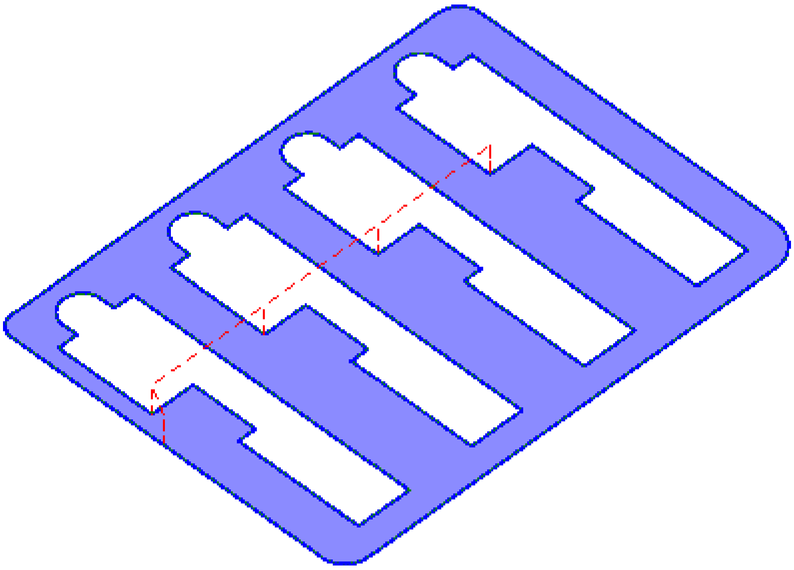
[Jet cutting job assignment]

[Jet cutting 4D operation]

Jet cutting 5D operation

Job assignment of Jet cutting 4D operation

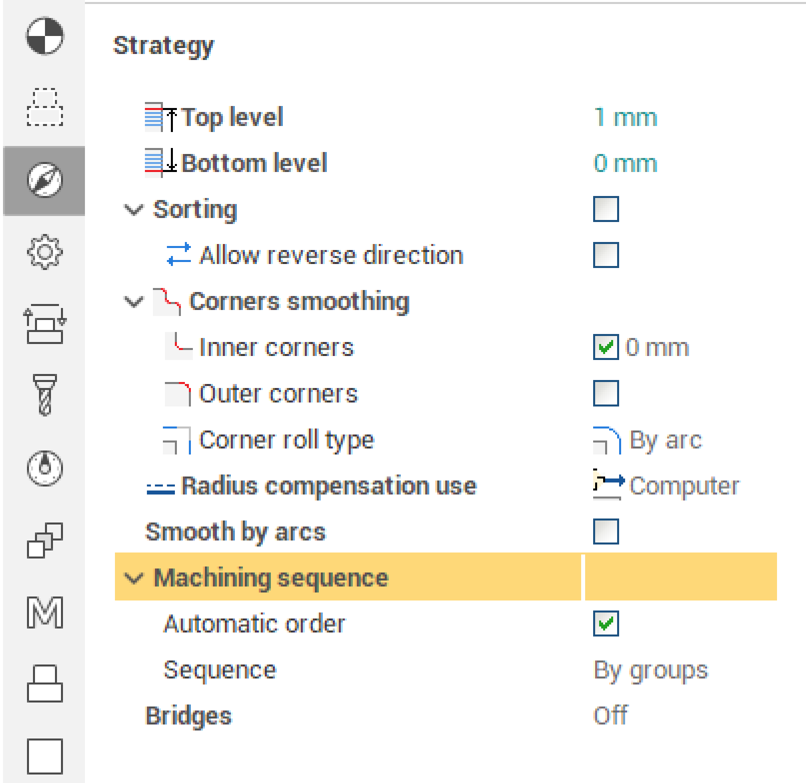
5.8.1 Jet cutting



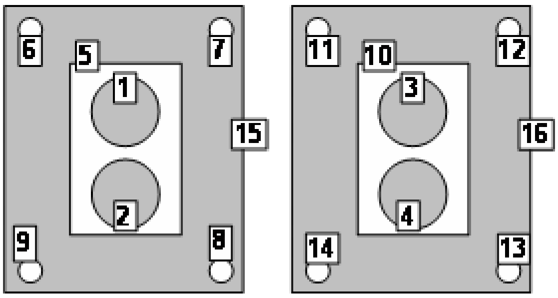
The operation is intended to carve the details from the sheet workpiece. The contours of the detail are defined by the curves projections.

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.

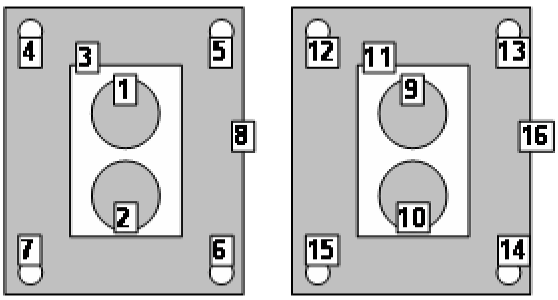
Use the panel below to define the machining order. The panel is located on the <Strategy> page of the parameters window.



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.

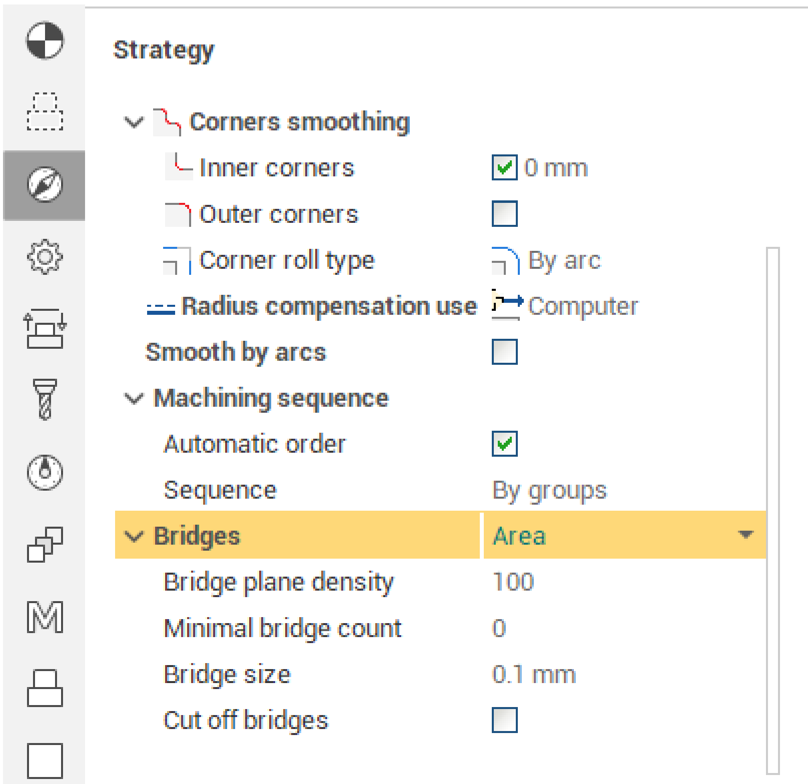
If the operation is performed using a local coordinate system or if using a swivel head then the system performs machining using the XY plane of the local coordinate system, and all work passes are consequently parallel to the XY plane of the local coordinate system.

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.

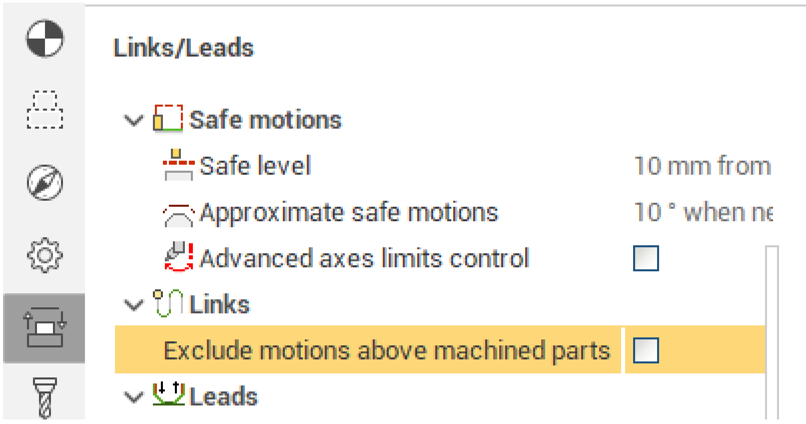
- <Count> – is defined concrete quantity of bridges. The equal quantity of bridges for all curves (for not closed curves a quantity of bridges on 1 less) will turn out.
- <Length (L)> – quantity of bridges is defined as length of a curve divided into the size specified in parameter <C> (for not closed 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.

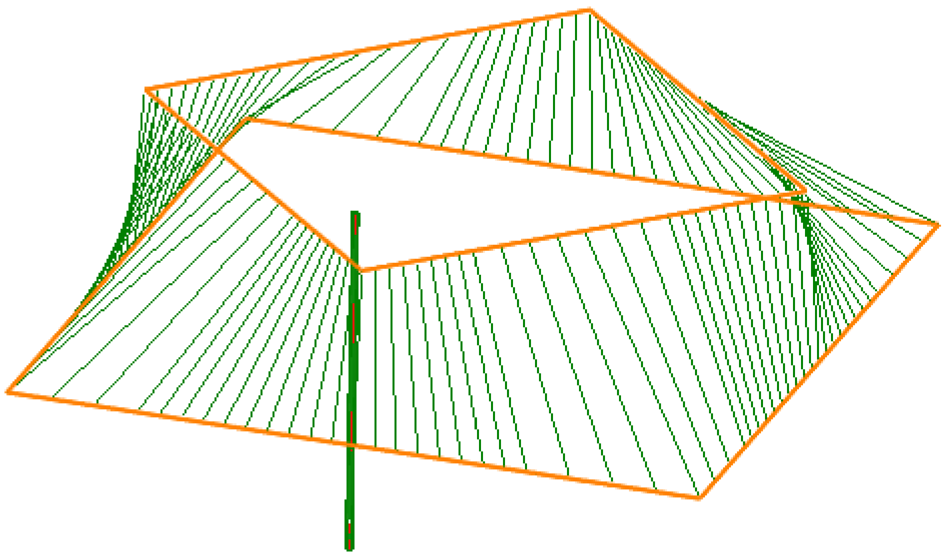


See also:

Processing on cut machines

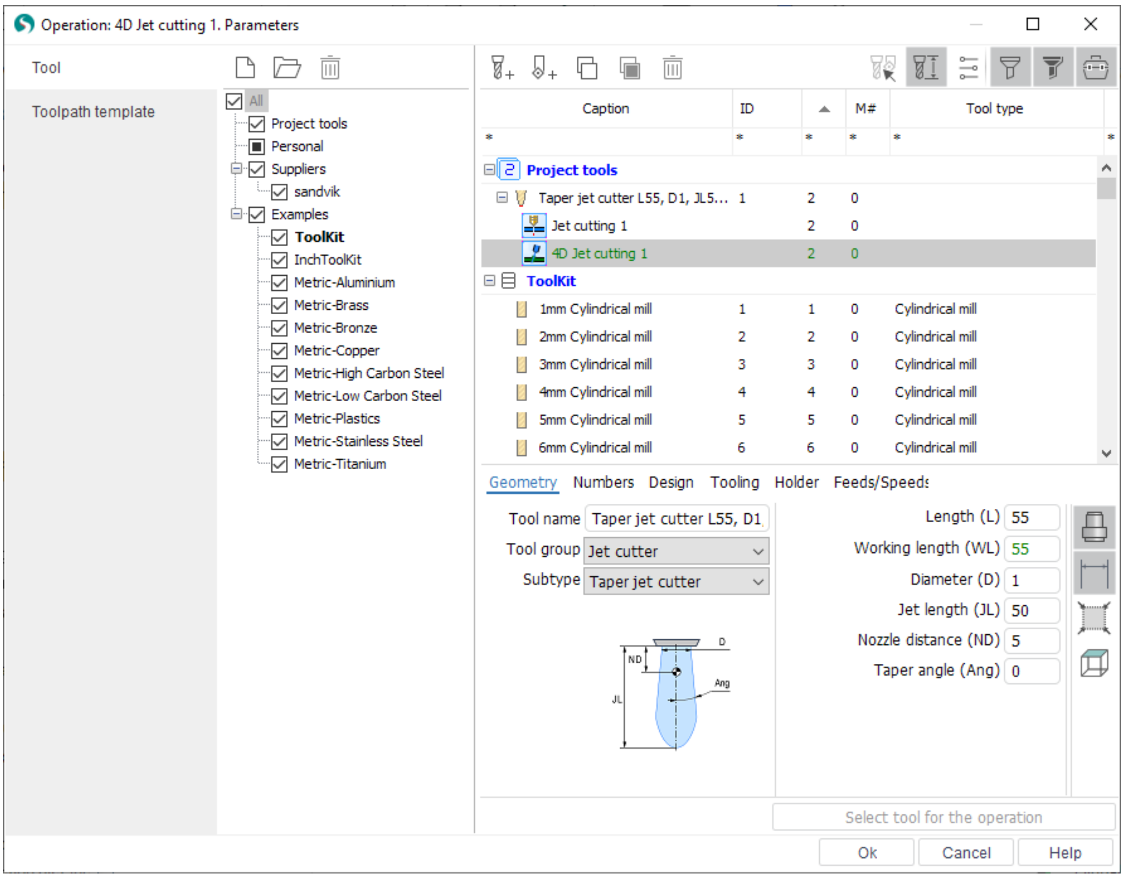
5.8.2 Jet cutting 4D

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.

See also:

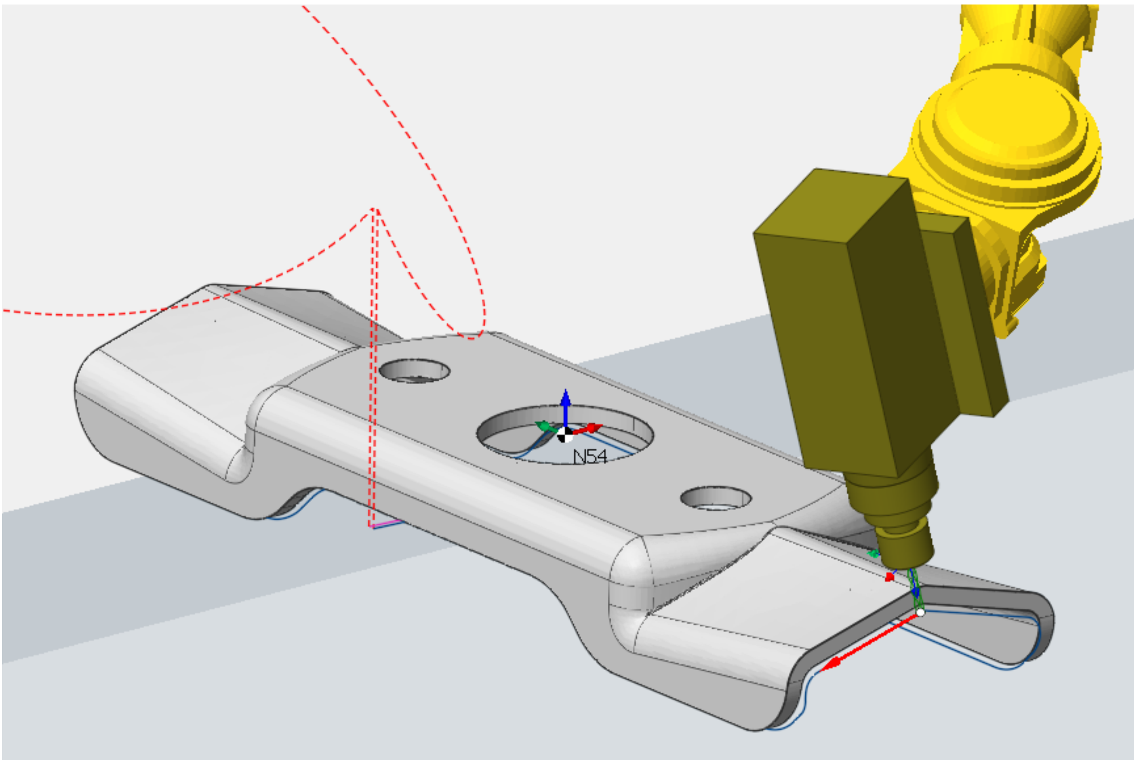
Processing on cut machines

Jet cutting 4D job assignment

Cutting conditions of Jet cutting 4D

Strategy of Jet cutting 4D

5.8.3 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.

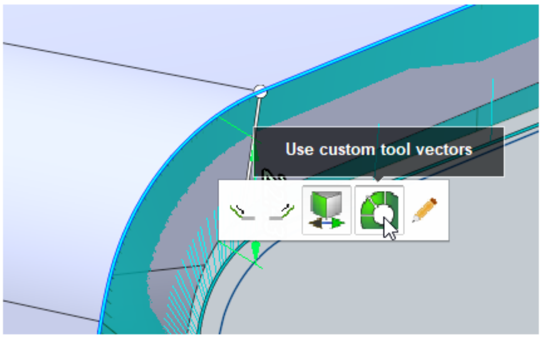
The operation requires at least 5 degrees of freedom. So active machine must have a minimum of three linear and two-three rotary axes. Very often the industrial robots are used for the cutting. If the machine schema doesn't support all degrees of freedom, then the generated tool path will be incorrect.

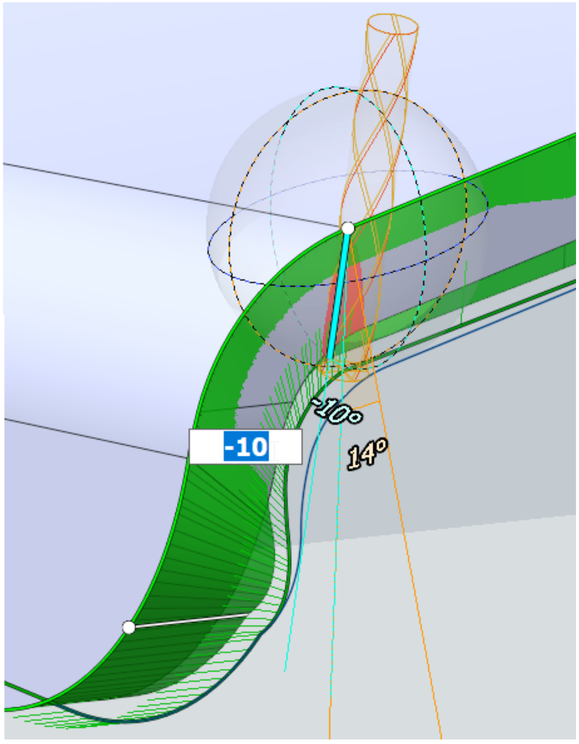
In the job assignment you should define the contours along which cutting will be performed:

- 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.

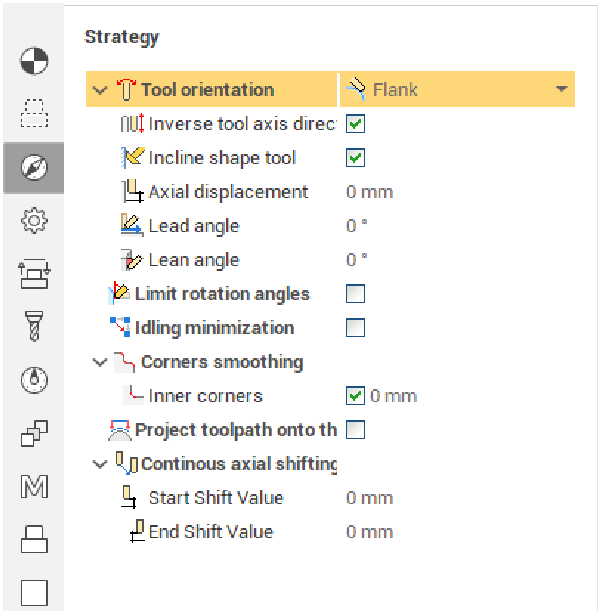
In the last two cases the tool orientation angles will be calculated from the nearest faces of the part.

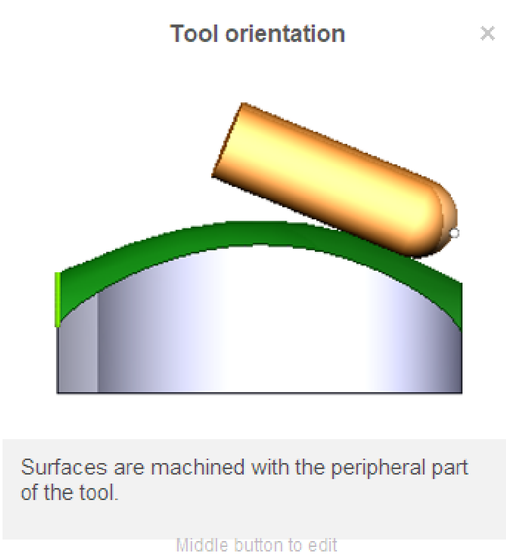
After the contours are set, you can switch to the custom vector editing mode, and adjust the exact tool angles at each of the curve points in the graphics window directly.





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.

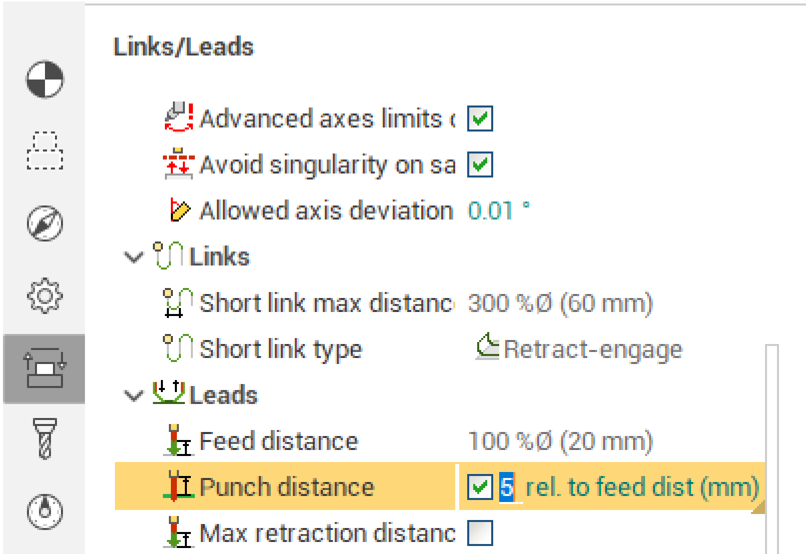




Also in Links/Leads tab you can use punch distance. This distance allows the tool to be retracted to the distance required for punching.

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.

5.8.4 Operations setup

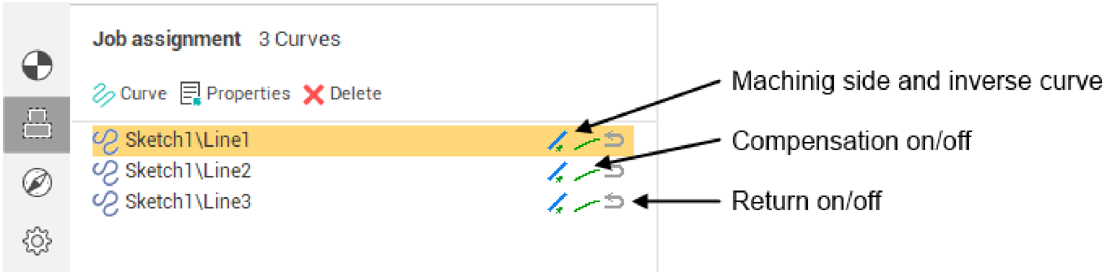
5.8.4.1 Jet cutting job assignment

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.

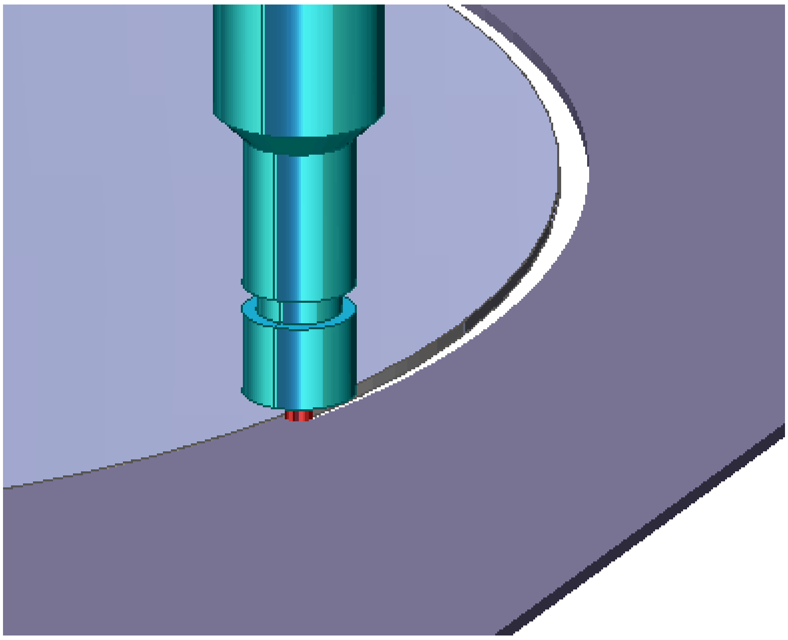


If toolpath passes <Along the curve> then NC-program will represent the curve itself. The tool radius compensation command is output into the program if the appropriate flag is checked. This is the simplest way of converting a custom CAD-defined curve to a CNC-code.

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.

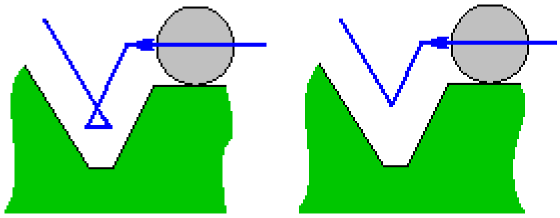
The Jet cutting operation's toolpath is the movement path of the central axis of the cutter.



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.

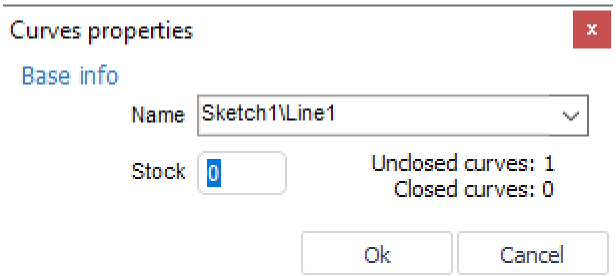
Modern CNC controls allow the programmer to use the actual component coordinates, and the operator enters the tool radius into the control. This allows the operator to for example, adjust for stock material for roughing/finishing operations. This can sometimes lead to situations where the offset toolpath 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.

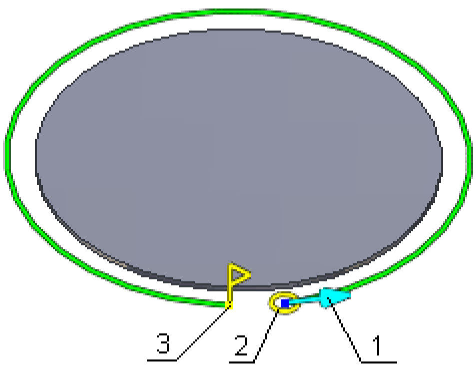


The start point can be input by coordinates or selected from the list. Along with geometry elements the list contains the following constants:

- <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).

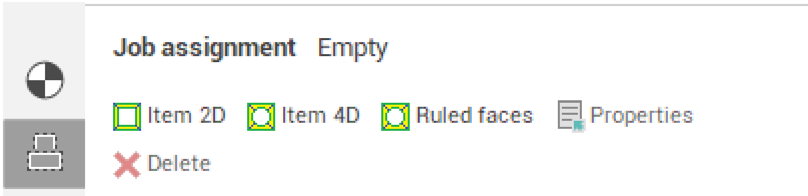
See also:

Processing on cut machines

5.8.4.2 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.

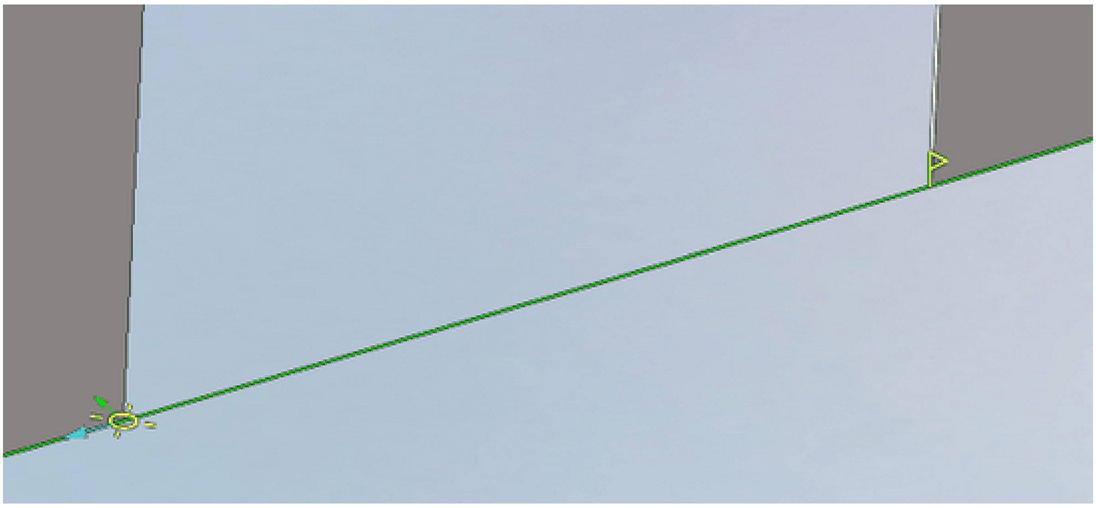
The following functions are available:

- <**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.

See also:

Wire EDM machining

2D job assignment item properties

4D job assignment item properties

Synchronization lines

Bridges

Multiselect feature

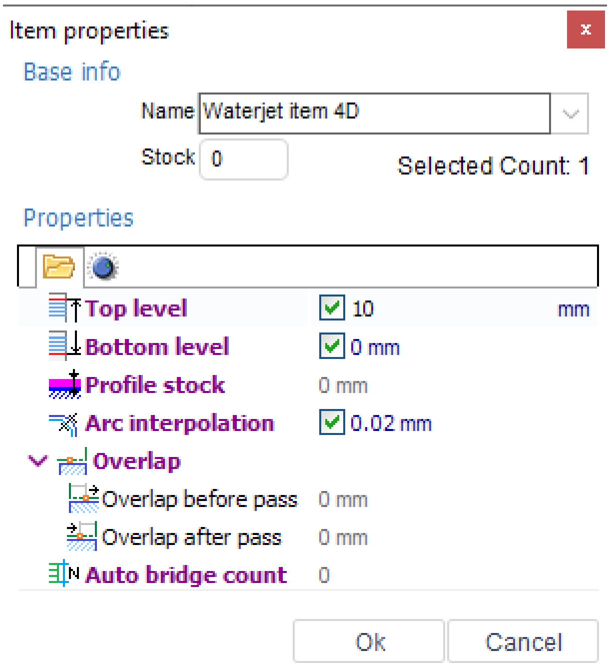
Wire EDM feature

2D job assignment item properties

Each element of the job assignment has a set of properties.

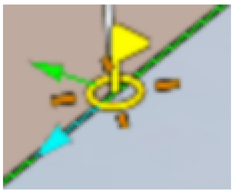
To view or edit the properties of 2D job assignment item select the item and click the <**Properties**> button, or double-click the item.

This is the item properties dialog:

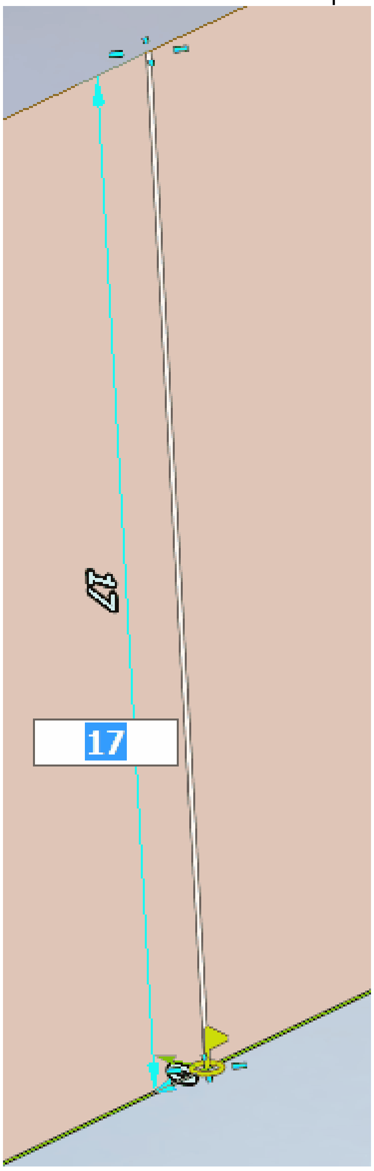


- <**Top level**> – plane for the top guide of the EDM-machine.To set the top guide level via the graphic window move the top level sign:

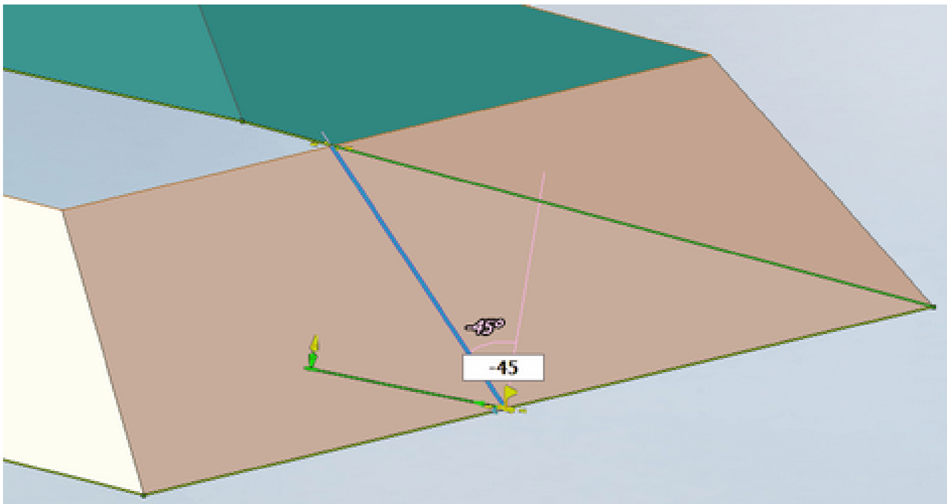
- <**Bottom level**> – plane for the bottom guide of the machine.To set the bottom guide level via the graphic window move the bottom level sign:



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:

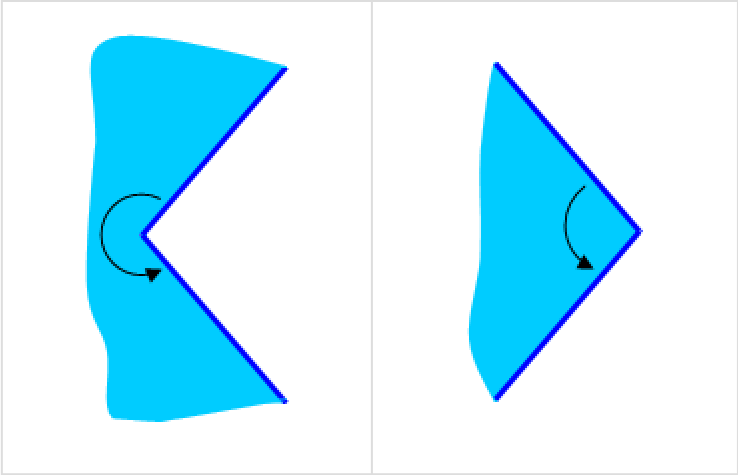
- <All passes> – taper will be applied for all passes of the contours.
- <Apply after pass> – taper will be applied after the pass number set in the <Pass #> field. Taper will be disabled for the N passes, passes starting with N+1 will have taper enabled.
- <Cancel after pass> – taper will be canceled 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. SprutCAM X 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: SprutCAM X simulation supports visualization of the wire path modified by the NC-controller corners rolling.

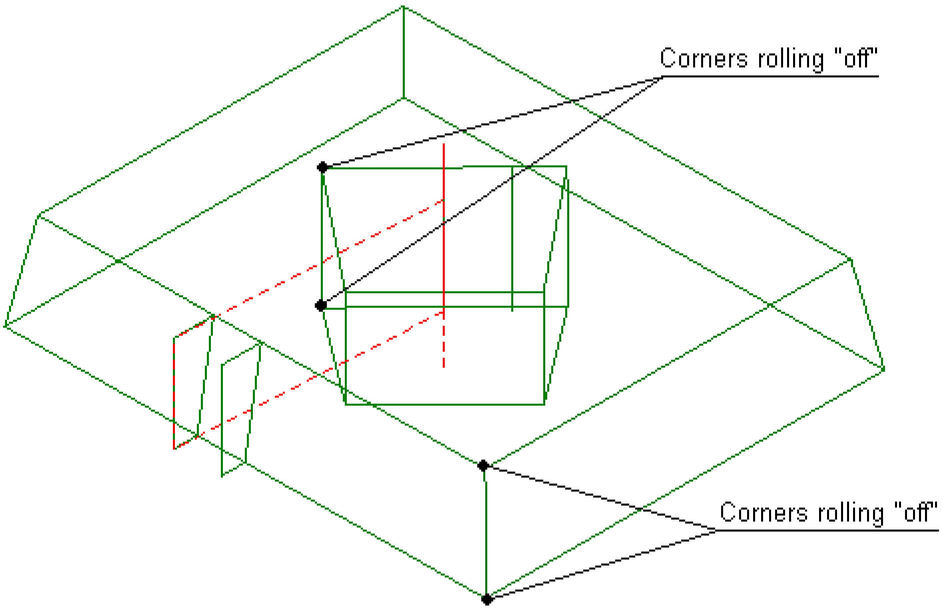
SprutCAM X supports various modes of corners rolling which can be set separately for inner(<Inner corners rolling>) and outer corners(<Outer corners rolling>). Whether the corner is inner or outer is determined by the angle value inside the part. Inner corner has the angle of 180° or more, outer has the angle less than 180°.

<Inner corner>	<Outer corner>

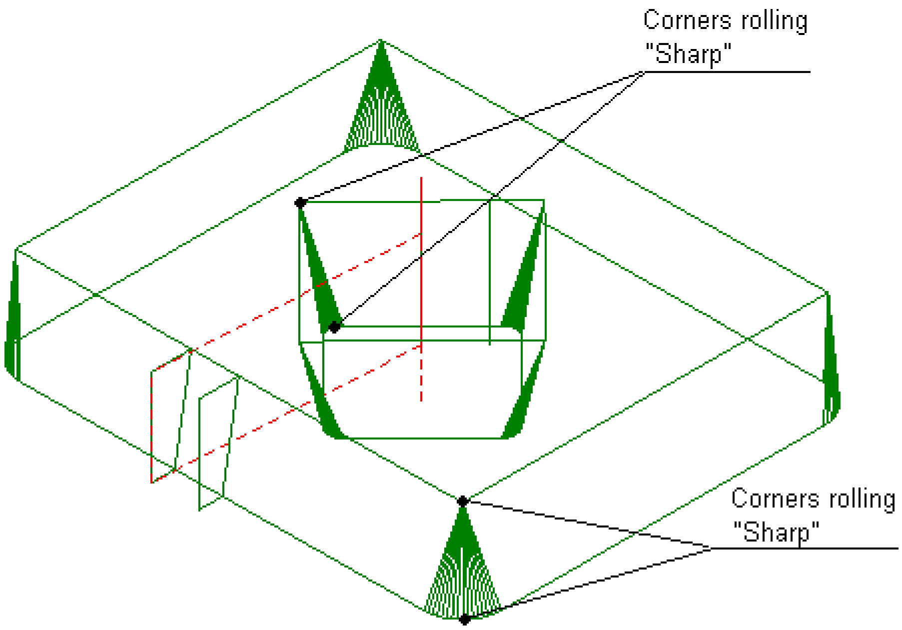


The following modes of rolling are supported:

- <Off> – in this mode corners rolling feature is disabled, rolling radii are not output into the NC-program.



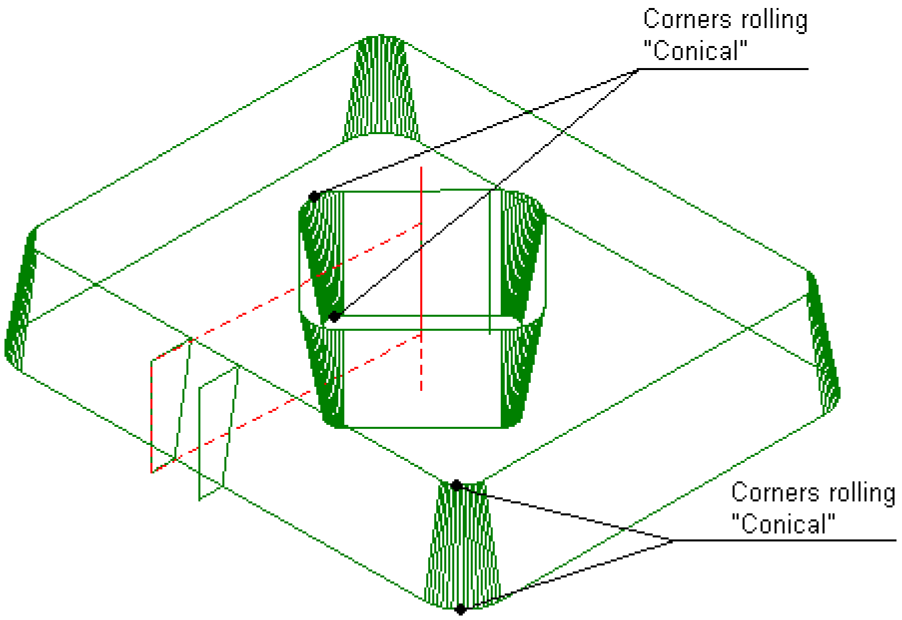
- <Sharp> – in this mode only the bottom contour corners are rolled, top level corners are not rolled. Therefore only the <Bottom radius> input field is enabled.



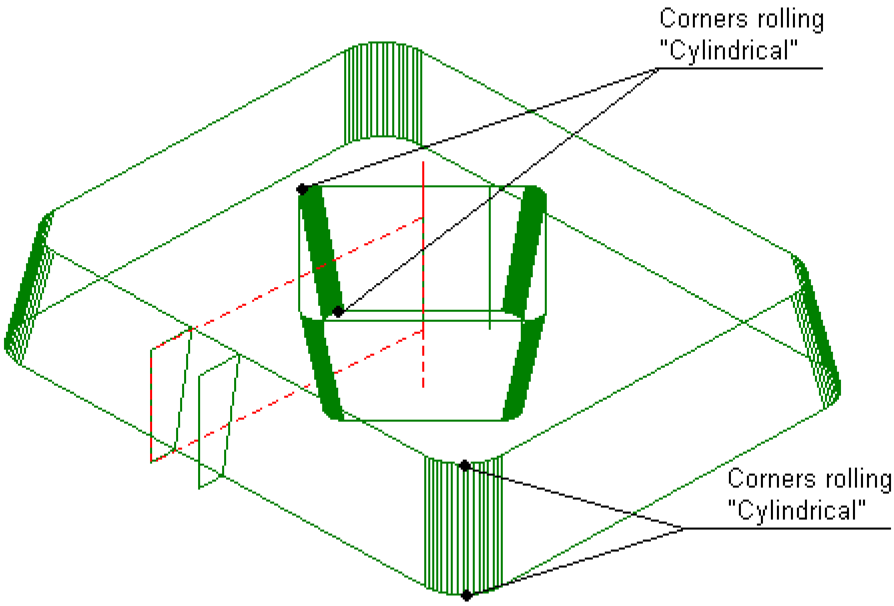
- <Conical> – in this mode the bottom contour corner rolling radius is set in the appropriate input field. Top contour rolling radius is defined a sum of bottom radius and a value depending on the taper angle and difference of top and bottom contour levels:

$$R_{top} = R_{bottom} \pm h \cdot tg\,\alpha$$

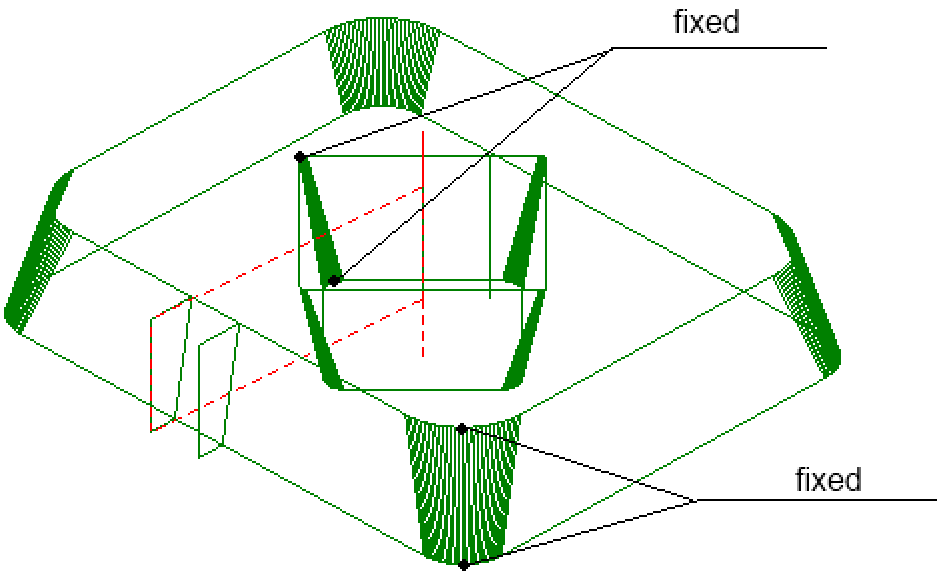
Thereby, a conical surface is machined on the part in the place of corner rolling.



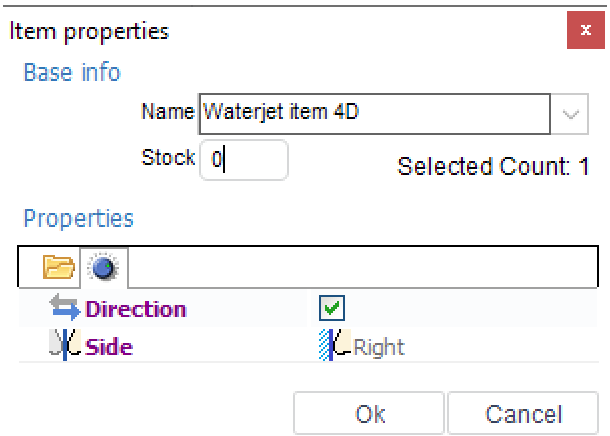
- <Cylindrical> – corner rolling radius is always equal for the top and bottom levels and is input in the <Bottom radius> field. Thereby, a cylindrical surface is machined in the place of corner rolling.



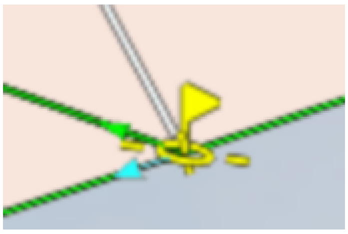
- <Fixed> – radii of rolling at top and bottom levels are defined independently in the respective fields and can be set to arbitrary positive values.



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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

4D job assignment item properties

Synchronization lines

Bridges

Multiselect feature

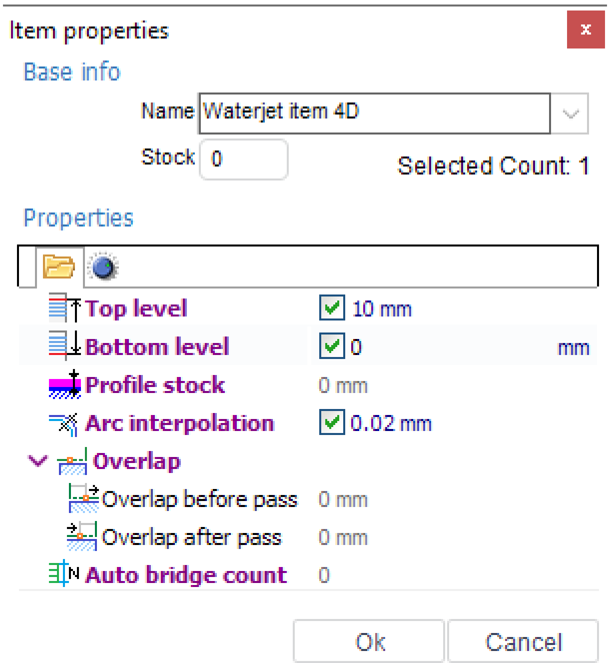
Wire EDM feature

4D job assignment item properties

Each element of the job assignment has a set of properties.

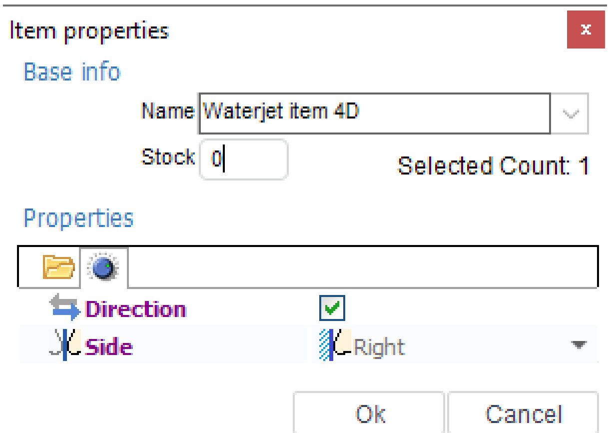
To view or edit the properties of 4D job assignment item select the item and click the <**Properties**> button, or double-click the item.

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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

Synchronization lines

Bridges

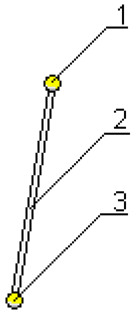
Multiselect feature

Wire EDM feature

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).

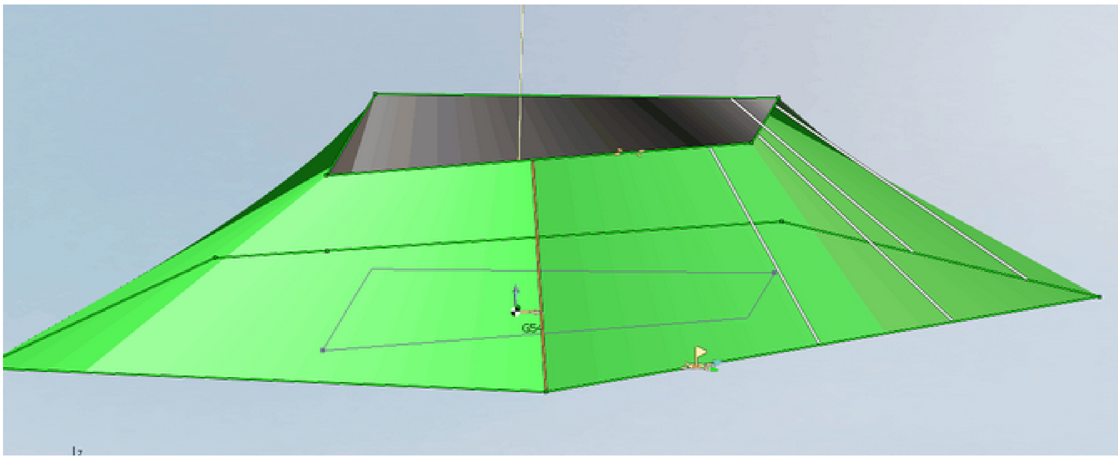


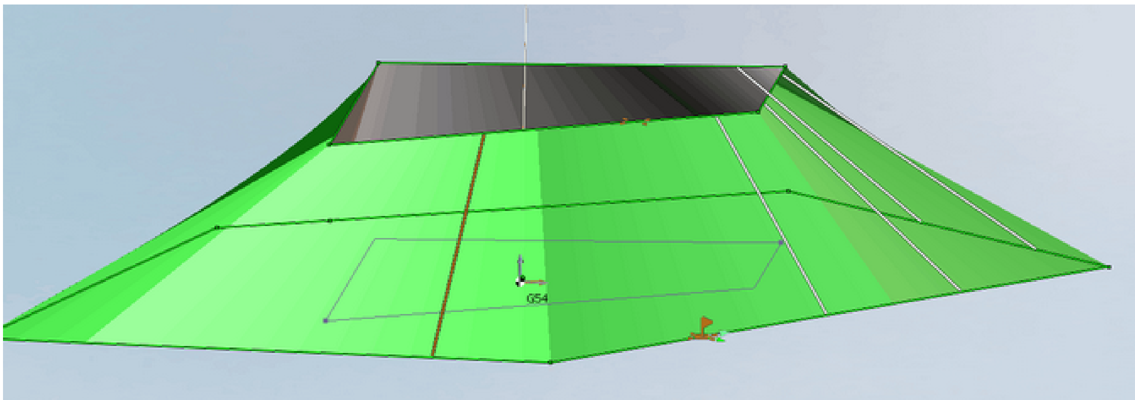
Synchronization lines can be copied and deleted in the graphic window.

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.





See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

Bridges

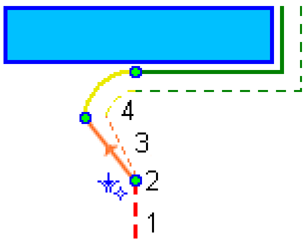
Multiselect feature

Wire EDM feature

Approaches/returns

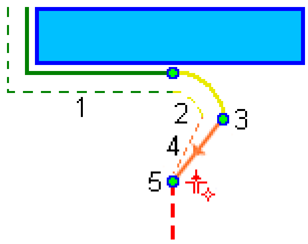
Lead-in and lead-out are the parts of the tool path, defined at the start and the end of each contour tool path. These are used for the correct machining at the start and end of a contour. These moves are used to enable various interpolation functions such as compensation, taper, multi axial interpolation, 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.

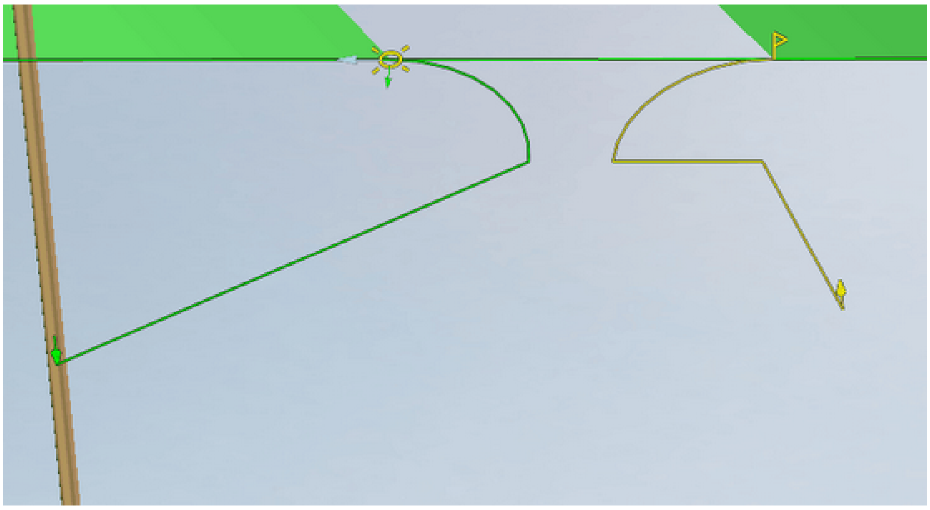
Retract from end point does the sequence in reverse:



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.

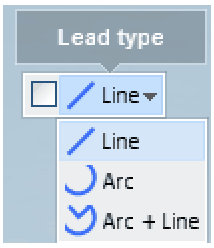
The lead-in/lead-out parameters are defined in the graphics window.

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.



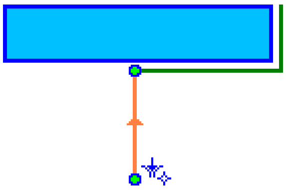
Approaches\returns markers are fully interactive. It is possible to move them and specify dimensions. Dimensions can be set as relative to other elements, as well as relative to the coordinate system.

It is possible to select lead mode in the pop-up panel:

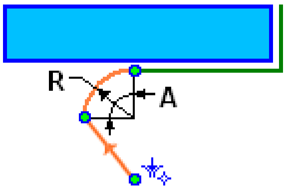


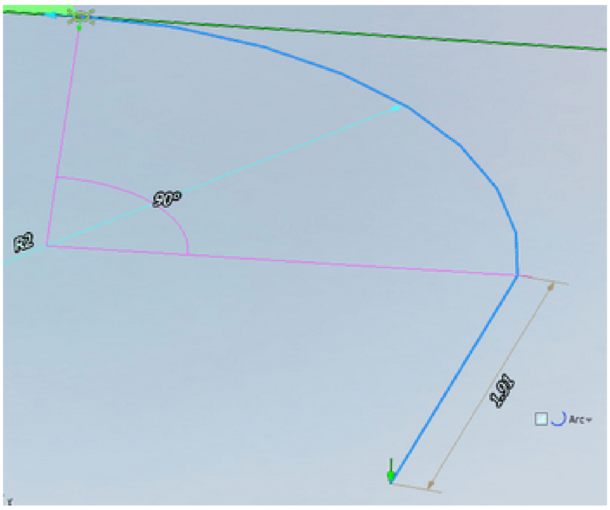
<Lead-type> – this panel is used to setup the lead type. There are several available lead types in the drop-down list:

- <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.

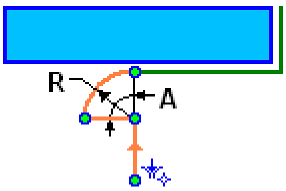


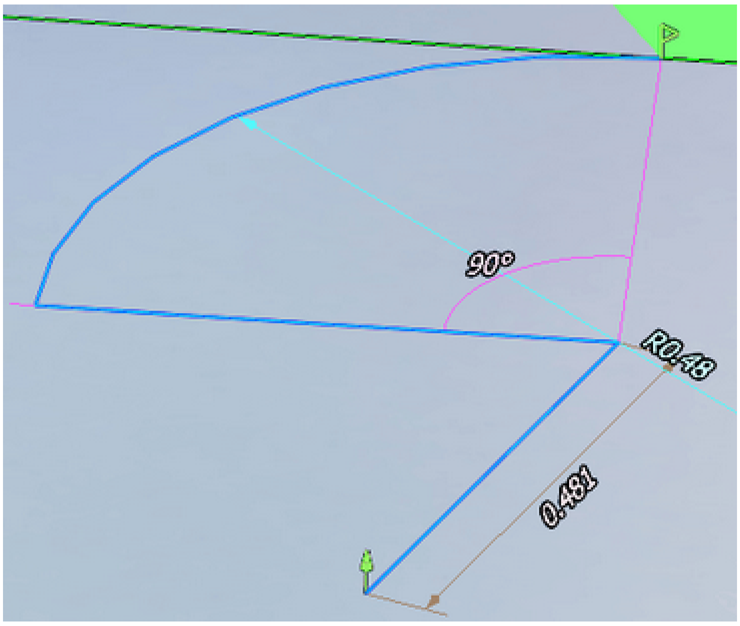
- <**A**rc> – lead-in has a linear motion from the wire load point to an arc start point. The arc move is tangent to the start of the machining contour.



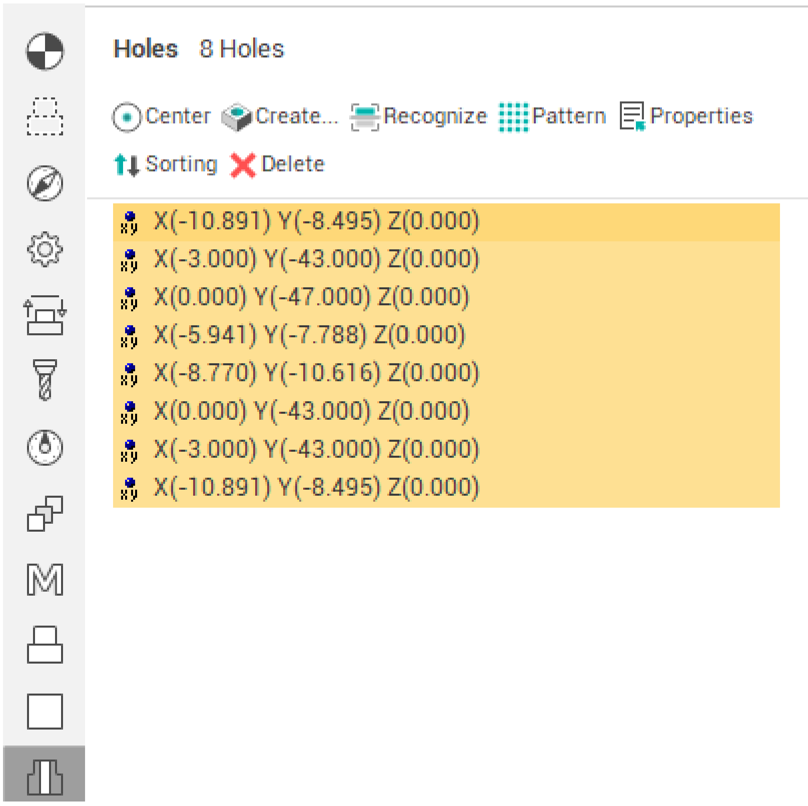


- <**L**ine and arc> – the first linear motion moves from the wire load point to the arc center point, then to the arc start point. Then the arc moves tangent to the start point on the contour.





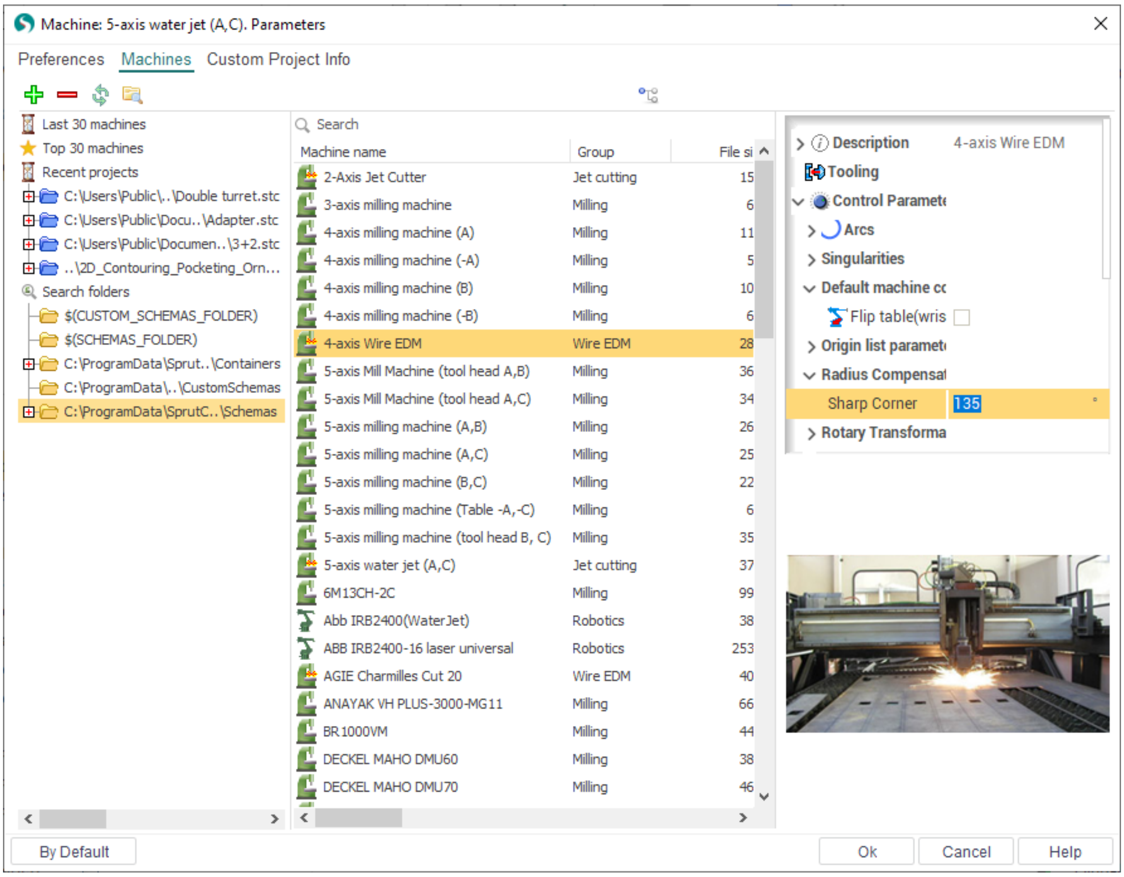
Attention: *All wire load or wire break points that are used when an operation is calculated, can be viewed on the <**Technology**> panel <**Holes**>. Also, you can export this list of points to use in another SprutCAM X project or other application. The export command is accessed from the* main menu *of SprutCAM X or from the context menu of hole list <**Export selected in DXF...**>.*



Wire radius compensation options

SprutCAM X 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. SprutCAM X 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. SprutCAM X have several options which can be used to 'tune' SprutCAM X'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 SprutCAM X simulation of radius compensation.

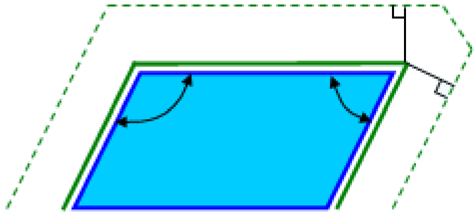


Use these properties:

- <Normal approach> – used for tuning the motion on approach and retract.

When <Normal approach> is on.	When <Normal approach> is off.
Start and end machining point stay on normal to contour.	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.



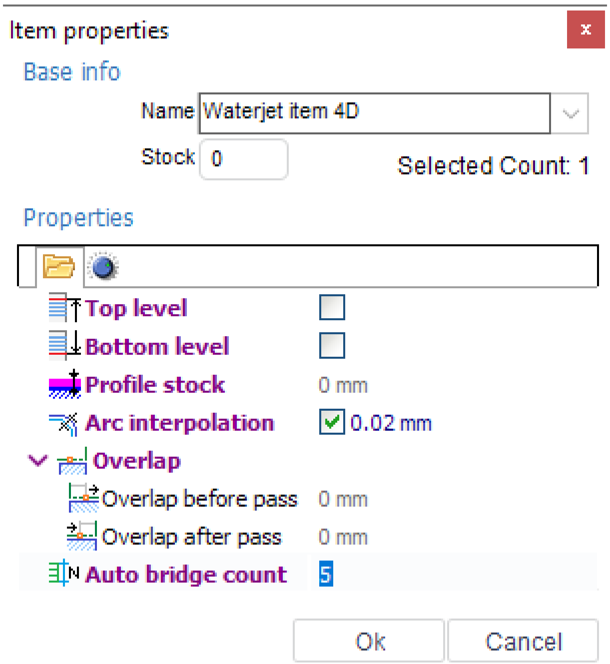
See also:

Wire EDM machining

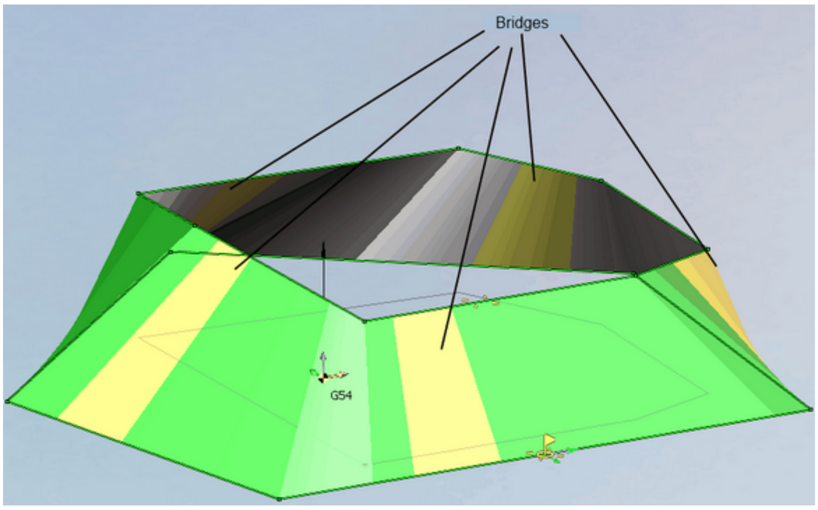
Bridges

<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.

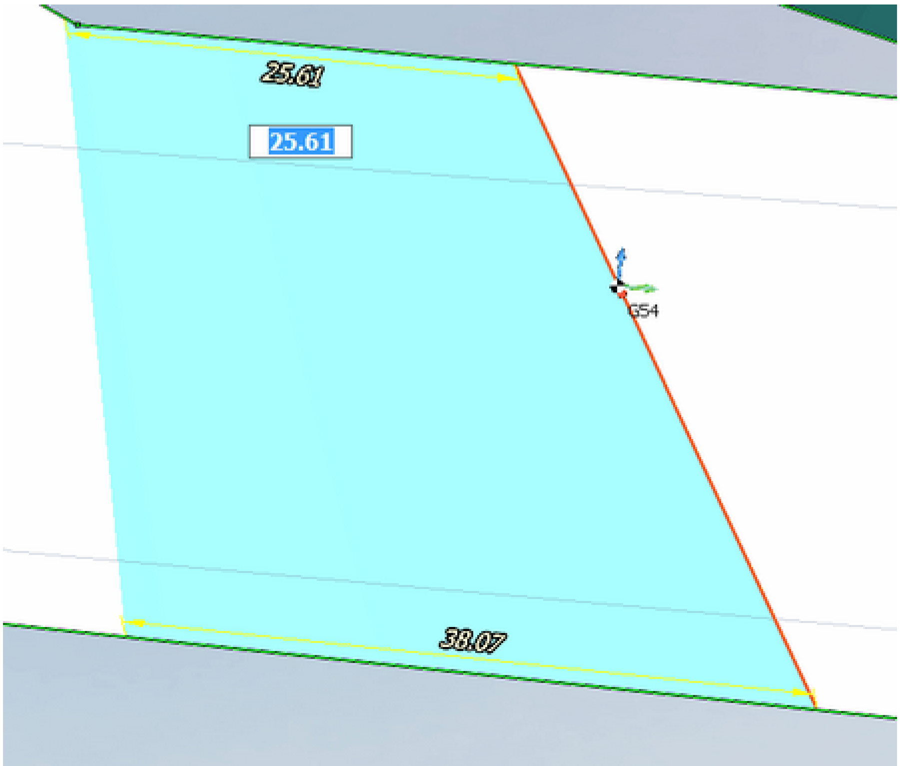


Auto bridges are placed at equal distance from each other. Change the parameters of bridges in the graphics window.



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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

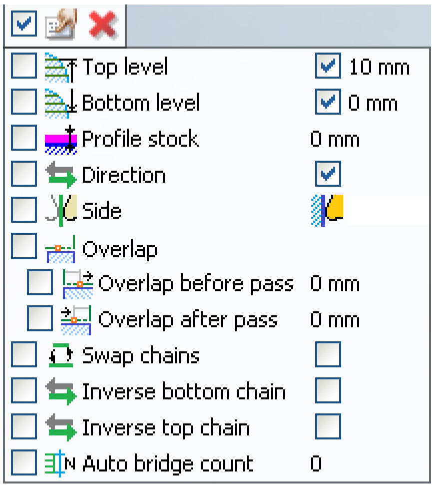
Synchronization lines

Multiselect feature

Wire EDM feature

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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

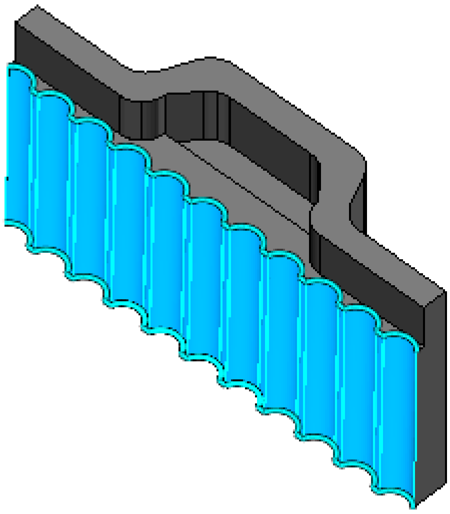
Synchronization lines

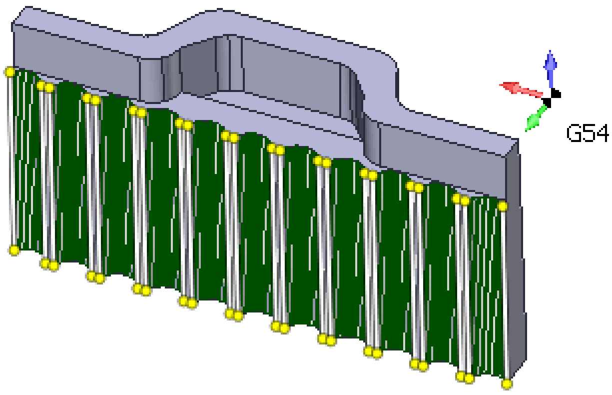
Bridges

Wire EDM feature

Wire EDM feature

Wire EDM feature is a chain of ruled surfaces with top and bottom edges lying in horizontal planes. SprutCAM X 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. SprutCAM X 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>. SprutCAM X will setup operation parameters according to the constructive element properties if the job assignment was empty.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

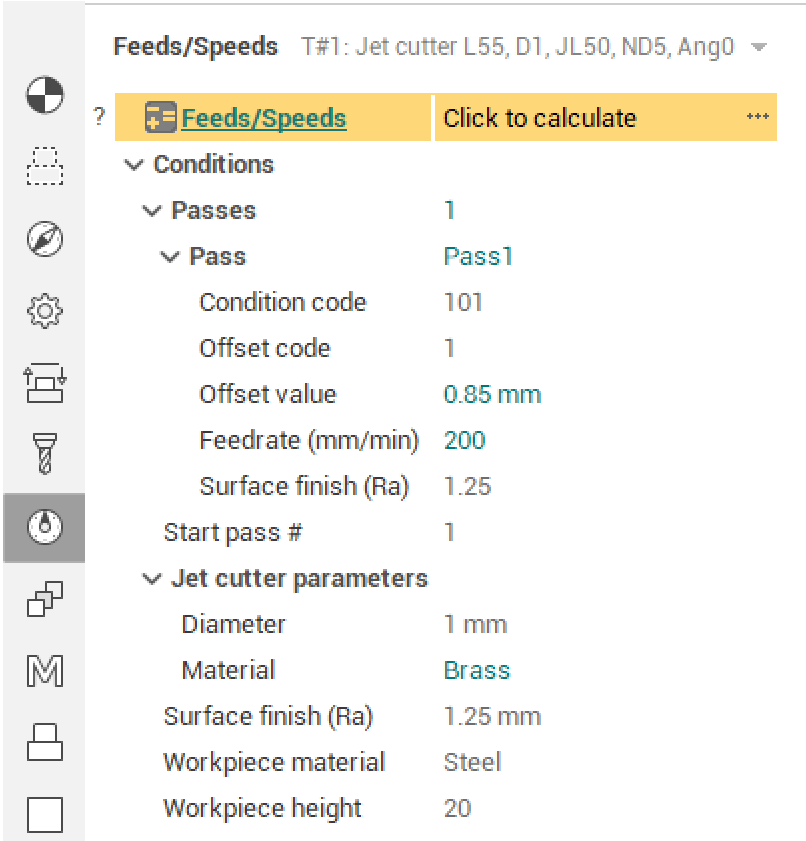
Synchronization lines

Bridges

Multiselect feature

5.8.4.3 Machining conditions of EDM and Jet cutting 4D operations

Defining of machining conditions is defined in operation parameters window on <Feeds/Speeds> page.



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.

An specific for every machine information about machining conditions can be filled and saved in special cutting parameters library. It is saved in single *.csv file. The current library file is shown on <Name> field in <Library> panel. From the list of this field can be selected one of the libraries from standard libraries folder of SprutCAM. The library can be assigned also from another place with help of standard file-dialogue window, that activated by the  button. In <Commentary> field is shown additional text information about selected library.

Machining technology library consists of so-called process technologies list. Every process technology contains following menu:

- <Technology> – 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.
- <Pass Parameters> – list of passes with process parameters for every pass.

The <Pass> term is one pass, that wire is does along the contour and following list of parameters is assigned for.

- <Pass name> – text description of the 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.

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

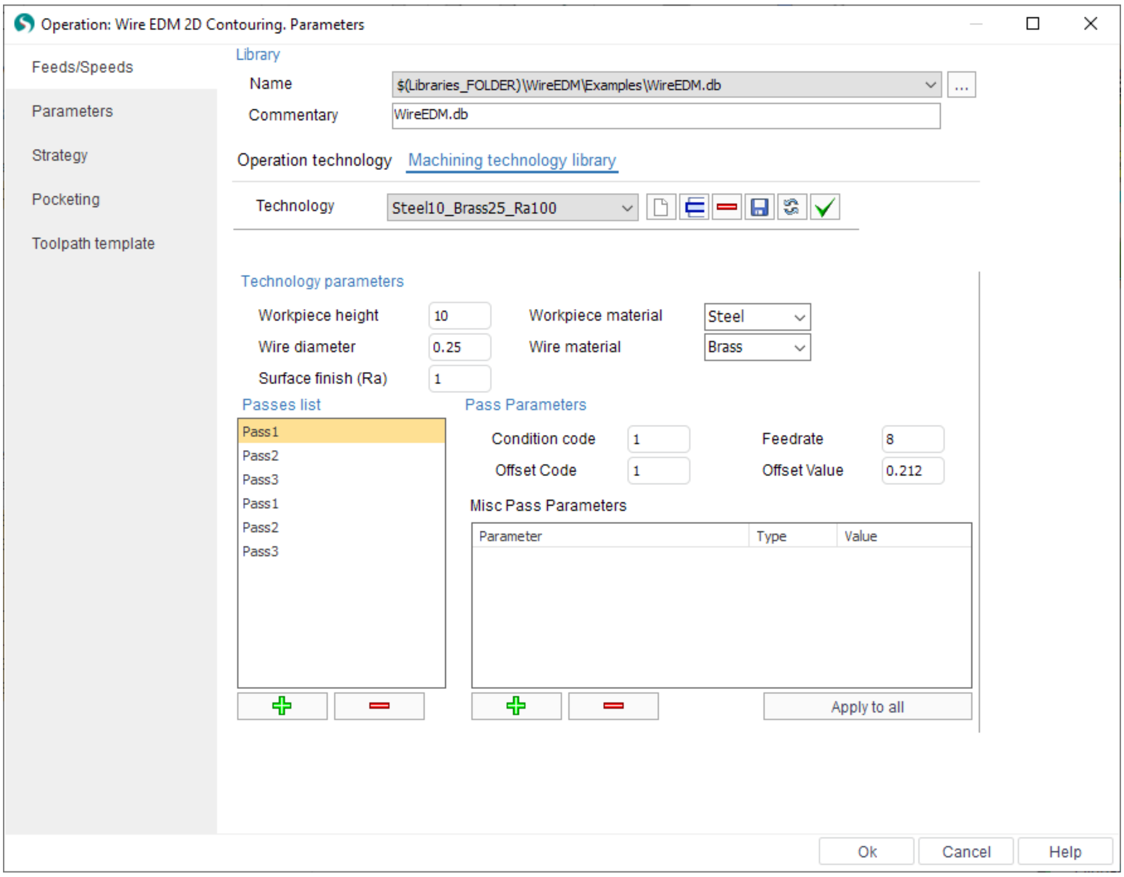
- Workpiece material: steel.
- Workpiece height: 40.
- Wire material: brass.
- Wire diameter: 0.5.
- Surface finish (Ra): 0.8.

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
...								

...

Working with conditions library does on the <Machining technology library> page:



At every instant only one active technology of library is edited. Its name is shown in the <Technology> field. In drop-down list of the field is shown list of all technologies, which library is contain. There is following functions to work with it:

- <New technology> – adds new technology to the list.
- <Rename technology> – renames active process technology.
- <Remove technology> – deletes active technology from the list.
- <Save library> – saves all changes to a library file.
- <Reload library> – loads information from a library file anew. All changes in technologies will be lost.
- <Select technology> – copies active technology parameters from the library to the operation technology.

Editing of current technology parameters does by changing values on the <Technology parameters> panel. The <Passes list> panel shows passes names, which active technology has. There is two functions available:

- <Add pass> – adds new pass to the list.
- <Delete pass> – removes selected pass from the list.

When one of the passes is selected, on the <Pass Parameters> panel its properties is shown. As stated above, in addition to basic parameters the pass has additional ones. Consisting of this parameters can be changed by user in a table on the <Misc Pass Parameters> panel. Input of names and values is performed by mouse click on corresponding cell. Below the table is three control buttons:

- <Add parameter> – adds a new line to the table of additional parameters of selected pass.
- <Remove parameter> – removes active line from additional parameters table.
- Apply to all – copies additional parameters table of selected pass to all passes of active technology.

Each wire EDM operation has its own process technology, that contains the same set of properties as a technology from machining technology library. When operation is calculating, it is follow technology from its parameters, but not from the library. Therefore, in order to apply active technology from the library, there is need to copy its parameters to the operation by pressing <Select technology> button. Properties of operation is shown on the <Operation technology> tab. They can to be edited even if the library is an empty. Library can be filled from this tab also by pressing <Save to library> button.

If there is filled library of processing conditions exists, then can be used quick search of machining technology with required parameters. After pressing the Find button system will analyze fields values from <Operation technology> tab and will look for closest technology in the library by using following algorithm. Among all technologies of library system will select those, that has workpiece material, wire material and diameter the same, as defined. Next, the system will look from founded for a technology, that will guarantee higher and closest surface roughness. At the same time system will take into account, that workpiece height must not to be less from defined and to be the closest. If there proper technology is exists, then system will go on <Machining technology library> tab and will activate this technology. Now need to press <Select technology> button to apply the technology. System will return to the <Operation technology> tab and new parameters can to be edited again to fit specific case of the machining.

On this tab is <Start pass #> parameter is exists. It allows to define number of the pass, from which machining will be begin. 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.

See also:

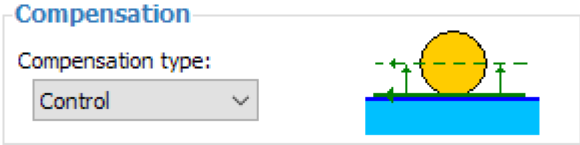
Wire EDM machining

5.8.4.4 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.



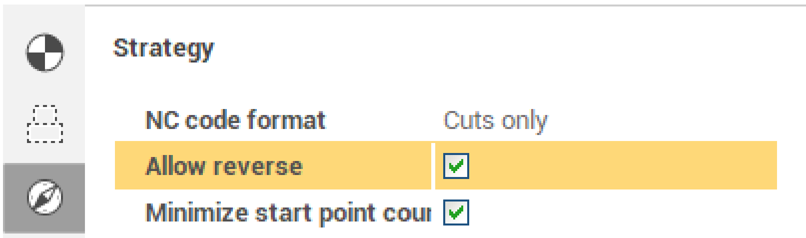
The following compensation types are available:

- <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.

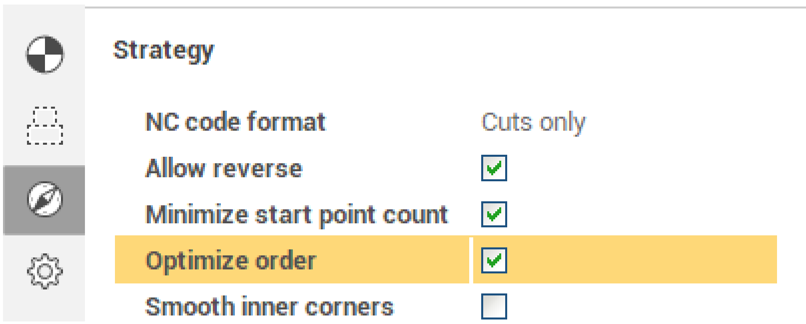
The value of compensation for each pass is defined as an <Offset value>, in the <Feeds/Speeds> tab, plus the stock value of the operation. For the compensation types <Computer>, <Both> and <Reverse Both> the value is used to construct an equidistant path, and for the <Control> type the value is entered 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.

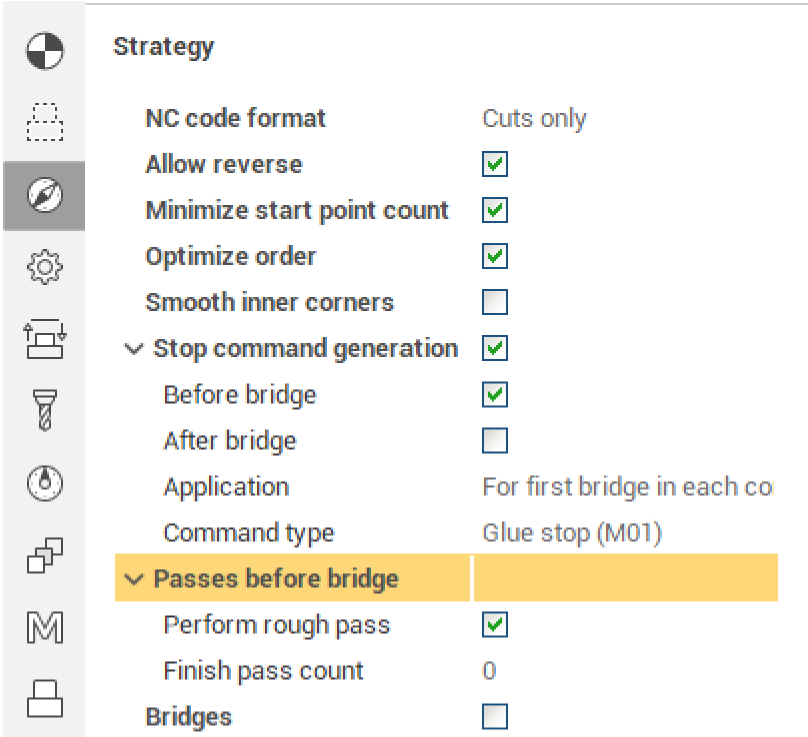
- <Reverse machining direction> – if you enable this option, the system will choose the direction for the pass, which provides the smallest length of the toolpath. If the option is disabled, then the direction of the pass will always correspond to that specified in the <Job assignment> for the contour.



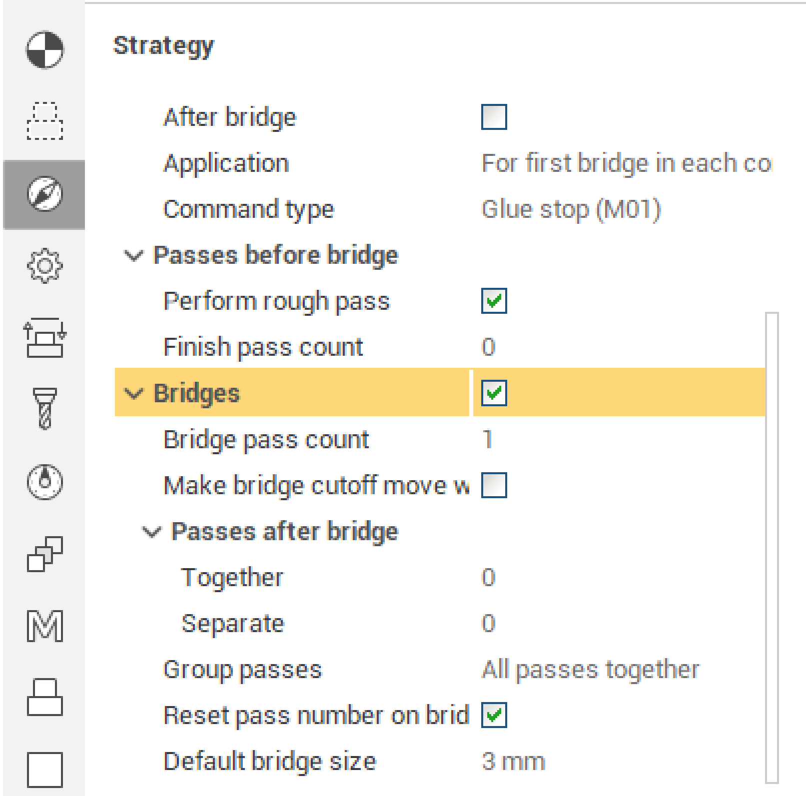
- <Optimize order> – this strategy determines the order of contours passes when the job assignment has more than one contour. The length of transitions between the contours will be minimal if the <Optimize order> option is enabled. If this option is disabled, then the order of the passes would be consistent 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.

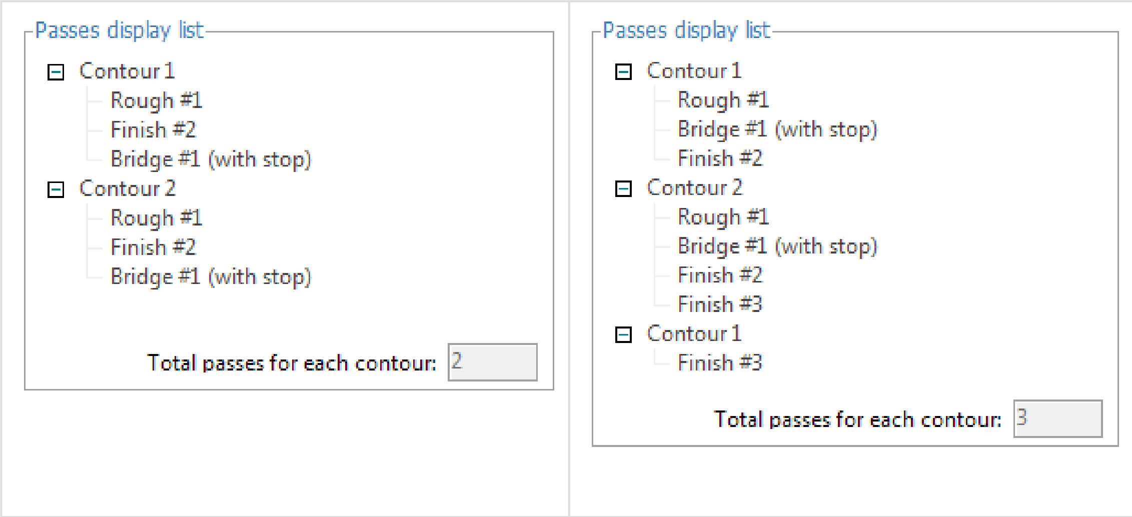


The <Bridge pass count> field sets the number of passes which will be performed for each bridge cutting on each contour. When the option <Reset pass number on bridge cuts> is enabled, then the count rates for the passes that define the cutting conditions for the bridge cutting moves are reset to the 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:

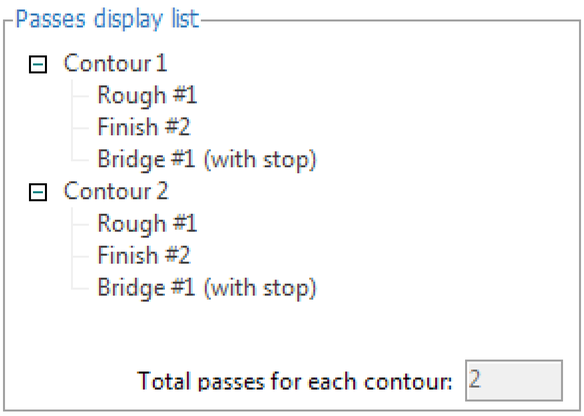


In the drop-down <Group passes> menu You can choose the way of grouping different types of passes for when working with several contours.The following options are available:

- <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>.

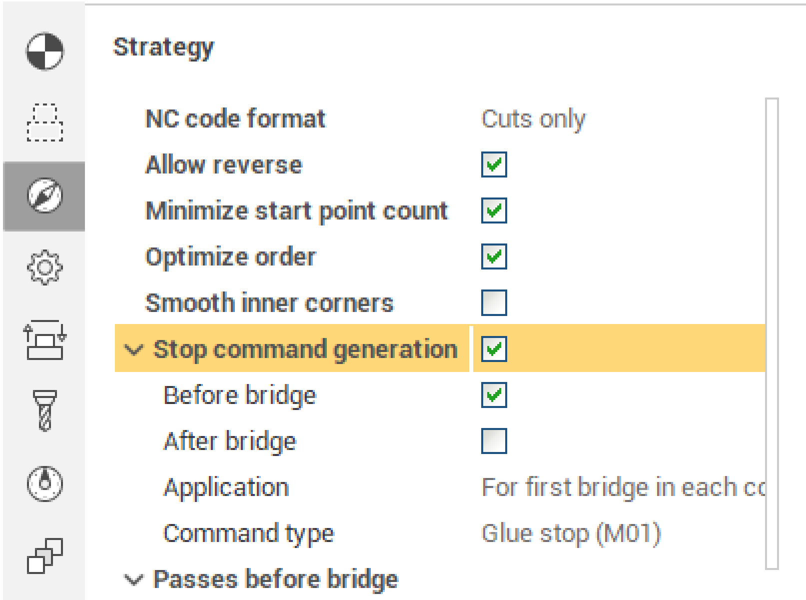
- <Passes display list>. On the <Strategy> tab for the Wire EDM operation there are many available options for contour machining that control the manner and the order of processing for the contours of the job assignment. To better understand the impact of a particular parameter on the machining sequence, 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.



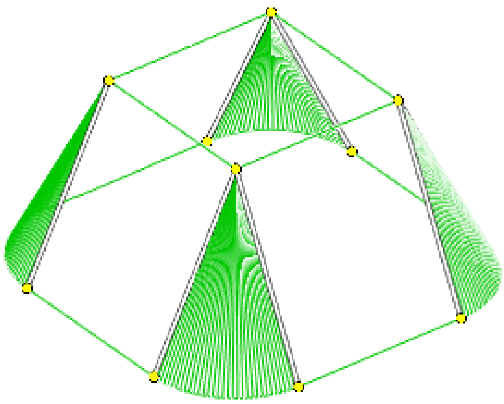
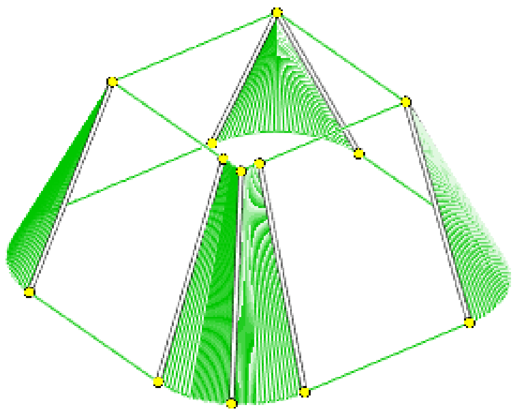
The drop-down list <Application> determines how the bridges should use the technological stops. The options include:

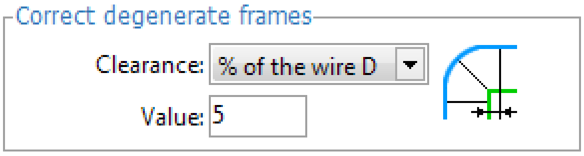
- <For every bridge> – stop command will be output for the bridge cutting move for every bridge specified in the operation.
- <For first bridge in each contour> – stop command will be output only for the first bridge of each contour.
- <For first bridge in operation> – stop command output only when cutting 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).

See also:

Wire EDM machining

5.9 Knife cutting

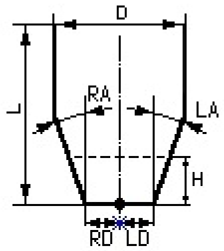
SprutCAM X has 2 special operations for the knife cutting:

- Knife cutting 2D
- Knife cutting 6D

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.

5.9.1 Cutting tool - "knife"



The knife dimensions are defined by 7 parameters:

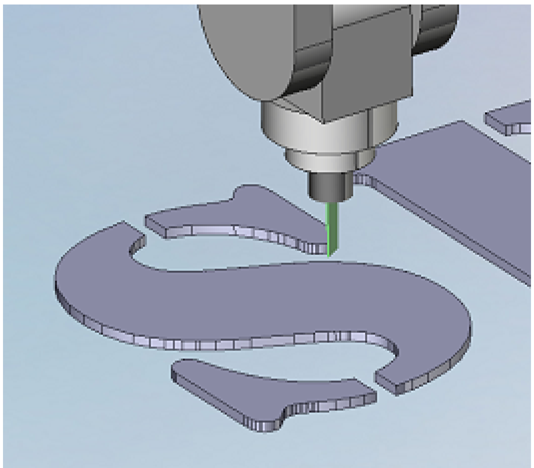
RD - distance from the tool tip to the right cutting edge
LD - distance from the tool tip to the left cutting edge
RA - inclination angle of the right cutting edge
LA - inclination angle of the left cutting edge
D - blade width
L - blade length
H- the depth of cutting by default. This value is constant 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.

See also:

KnifeCutting

5.9.2 Knife cutting 2D



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.

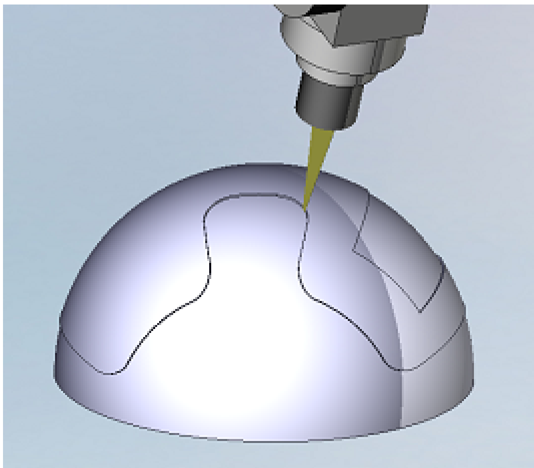
The knife usage adds the additional requirements for the machine. The machine has to have, except the Linear X,Y,Z-axes, the additional rotary axis that rotates the tool around. This axis must be defined in the machine schema. If it is absent, then the generated tool path will be incorrect.

The knife behavior in the sharp corners are defined by parameter "corner retraction".

See also:

KnifeCutting

5.9.3 Knife cutting 6D



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.

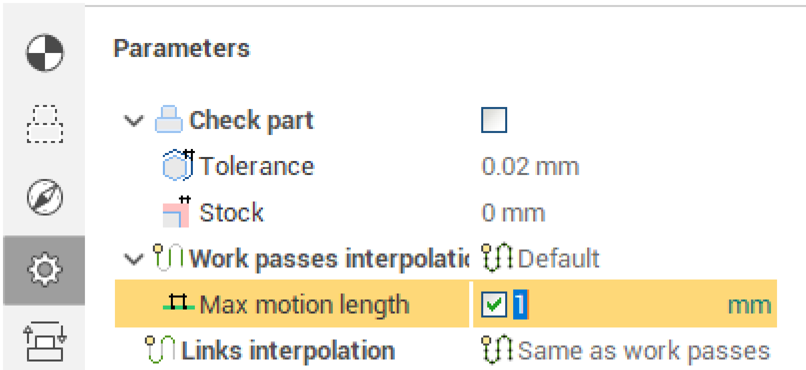
The knife behavior in the sharp corners are defined by parameter "corner retraction".

See also:

KnifeCutting

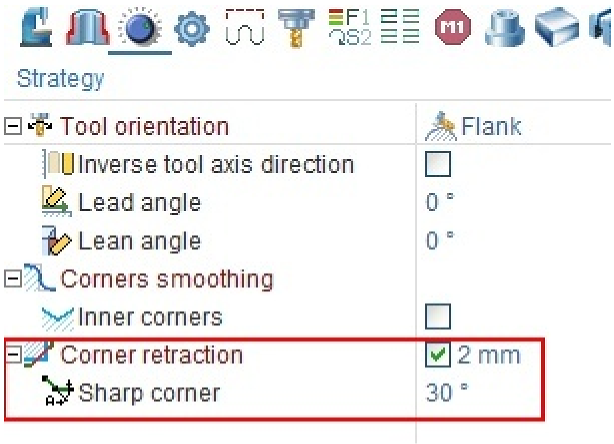
5.9.4 Knife corner retraction

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.

If you want to retract the knife in the corner, then enable the option "**corner retraction**" and define the retraction value. The retraction will be performed if the corner is sharper than defined value "**sharp corner**".



See also:

KnifeCutting

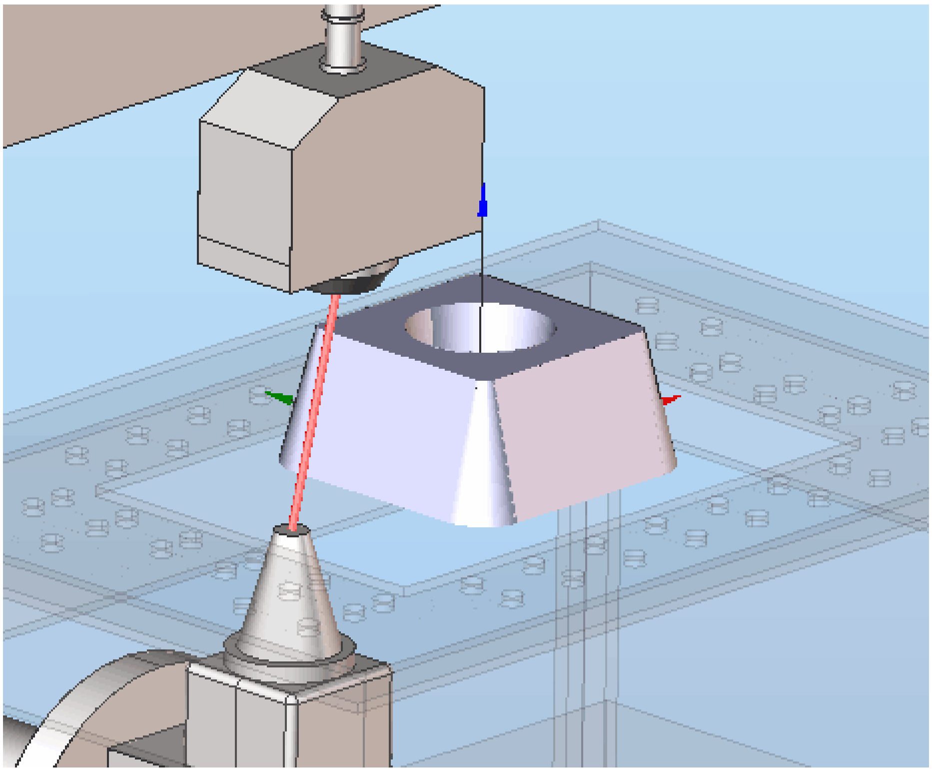
5.10 Wire EDM machining

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.



SprutCAM X 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. SprutCAM X 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. SprutCAM X 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.

See also:

Wire EDM 2d Contouring

Wire EDM 4d Contouring

Job assignment of wire EDM machining operations

Machining conditions of wire EDM machining operations

Approaches/returns of wire EDM machining operations

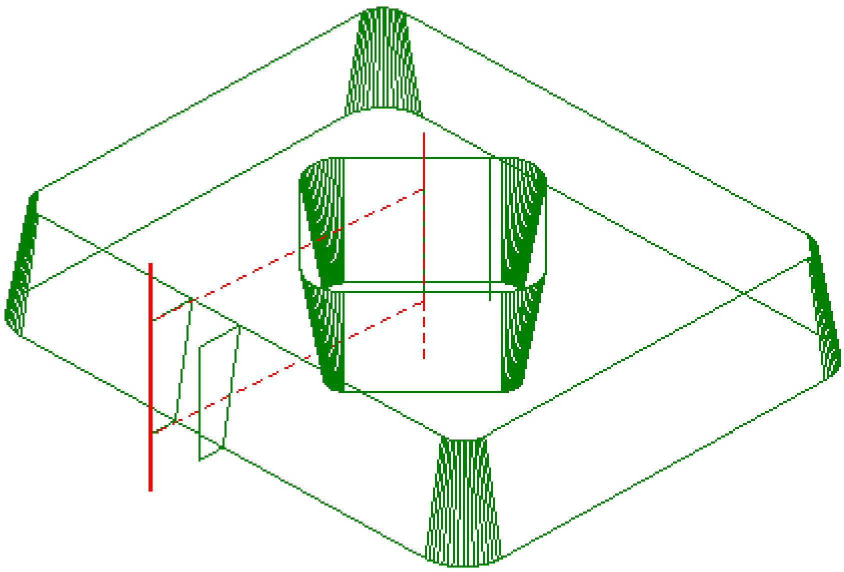
Strategy of wire EDM machining operations

5.10.1 Wire EDM machining operations

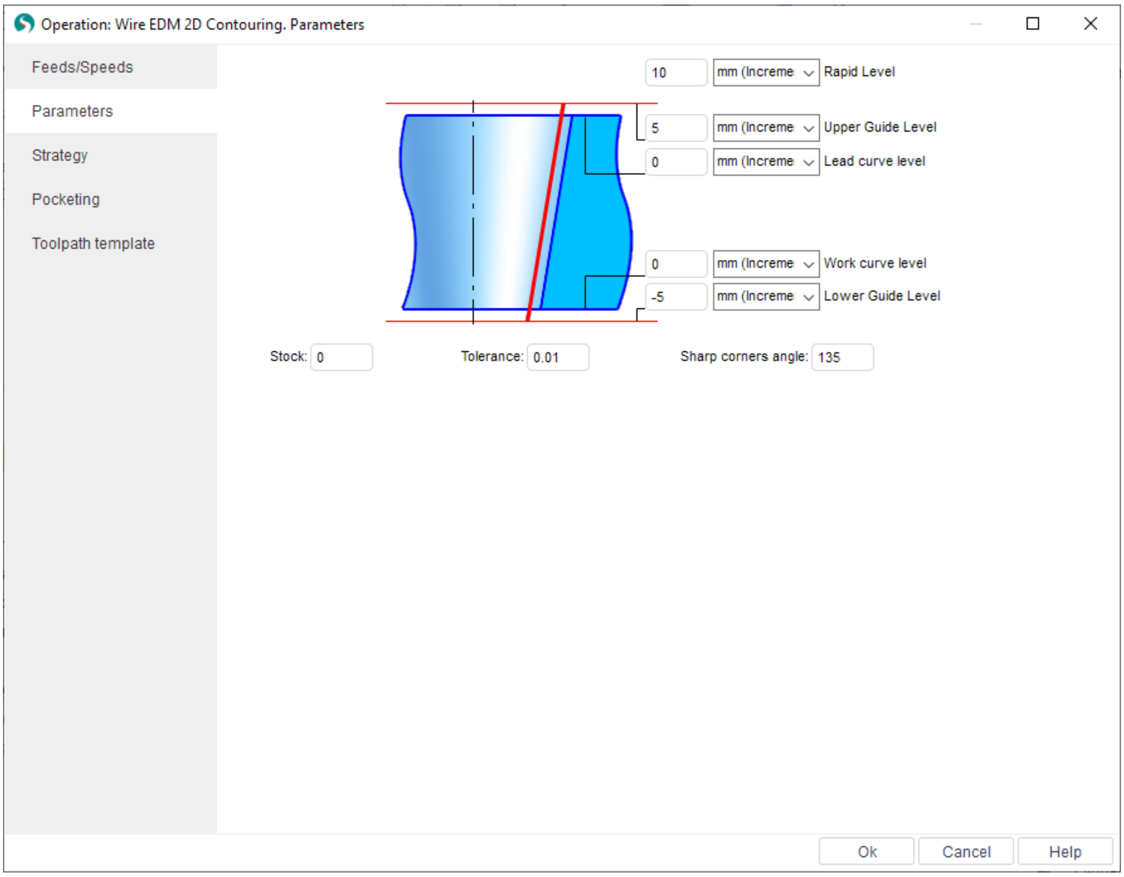
No content in this page. See child topics

5.10.1.1 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.



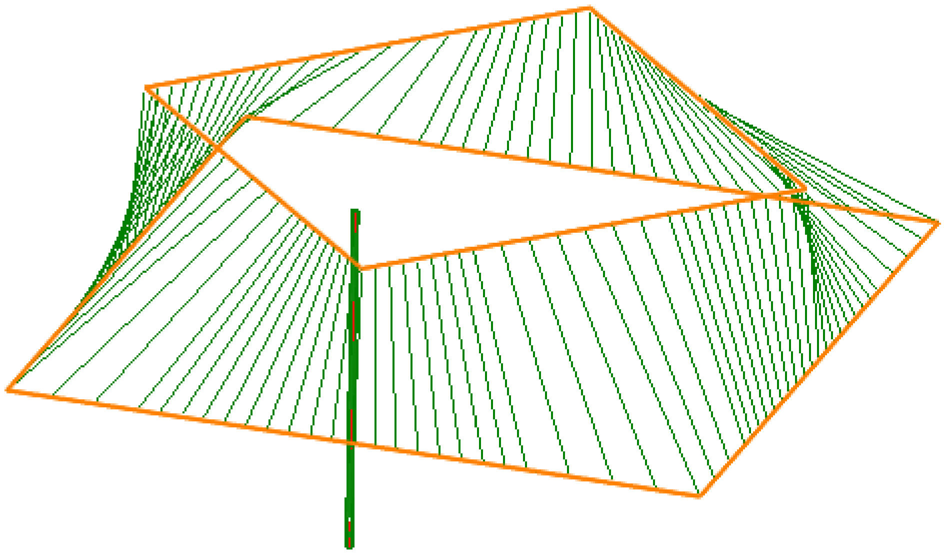
There are the following operation properties available:

- <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.

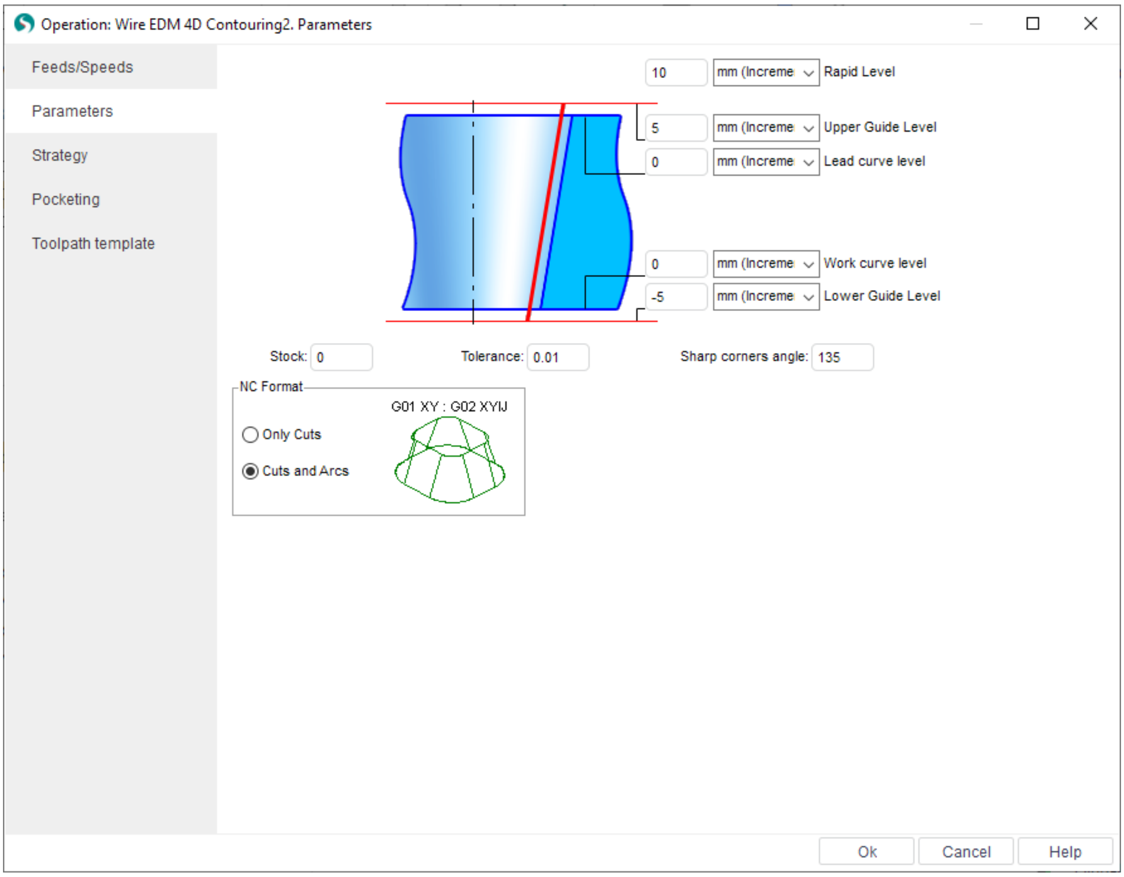
See also:

5.10.1.2 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.



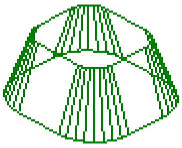
Specific options for every operation are defined in the operation parameters window on the <Parameters> page.



There is available following operation properties:

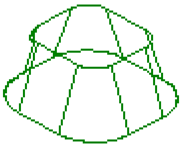
- <Working levels>:
- <Rapid Level> – defines the Z level where the rapid moves are performed. Its must be positioned above all working levels so that the rapid moves are performed at a height that is clear of fixtures etc.
- <Upper Guide Level> – defines the Z level for the upper guide of the wire EDM 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.
```

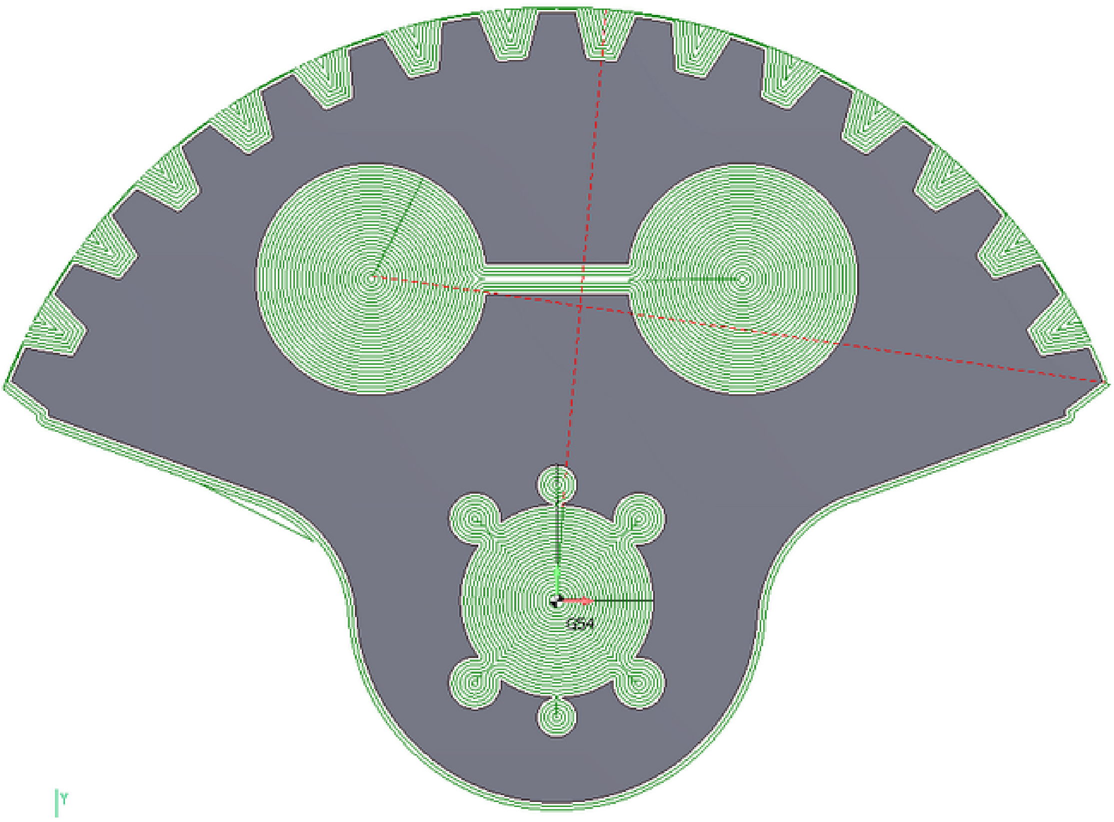


See also:

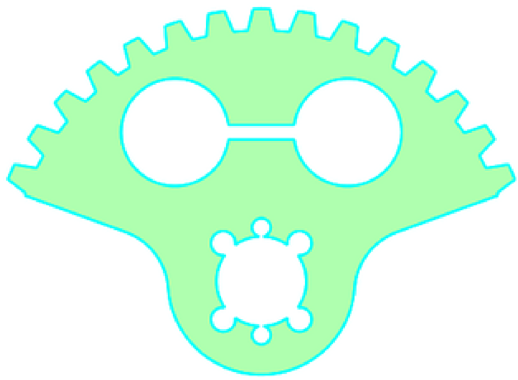
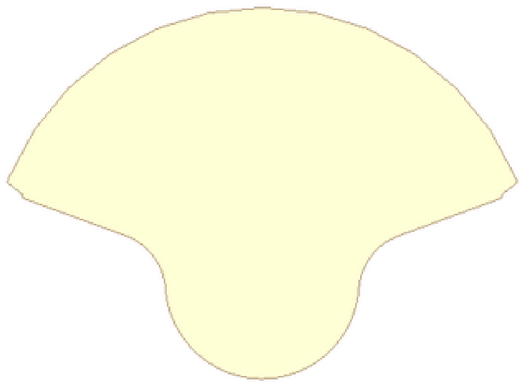
Wire EDM machining

5.10.1.3 Wire EDM coreless pocketing

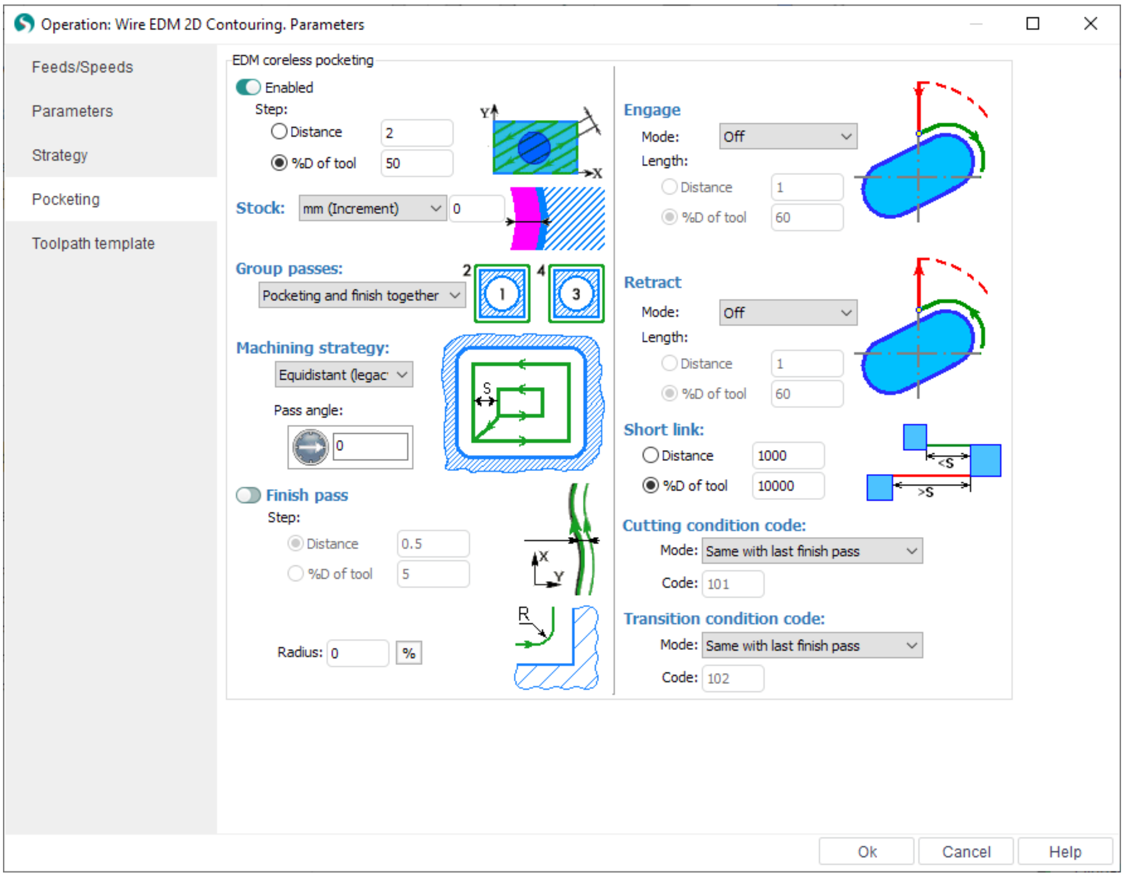
Wire EDM coreless pocketing is designed to remove material from areas without falling pieces of metal. The wire path usually begins in the pre-machined holes are located closer to the center of the workpiece. It consists of equidistant or zigzag passes, subsequently removed all of the material inside 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.

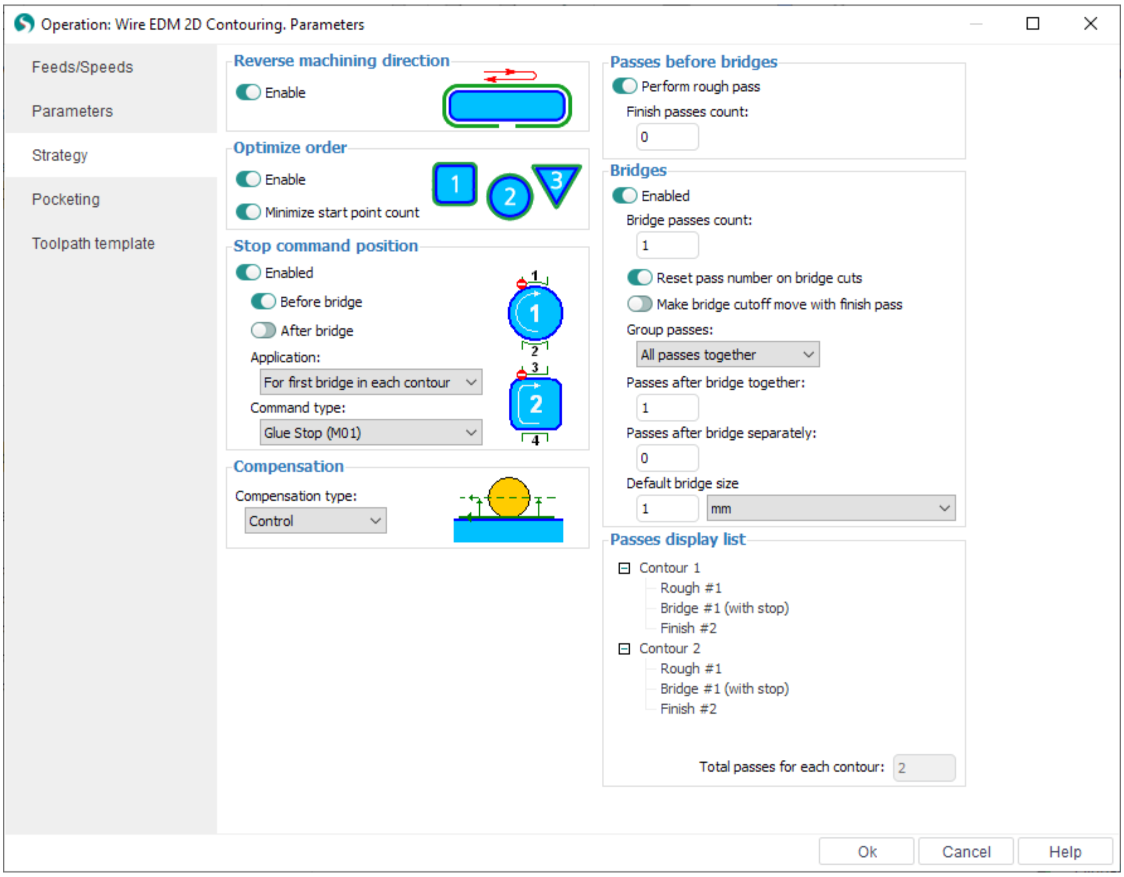
	
Job assignment	Workpiece

Wire EDM pocketing is not a separate operation of SprutCAM. 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.



See also:

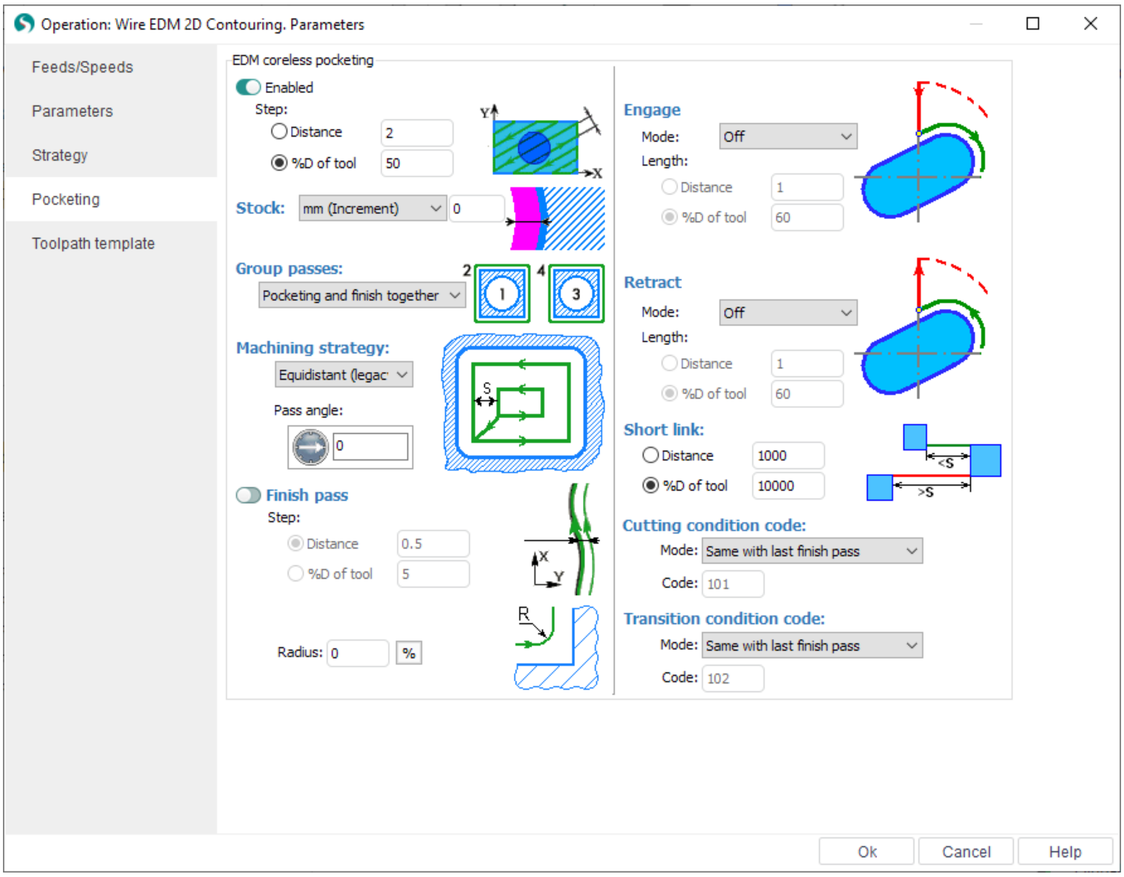
[Wire EDM machining]

[Pocketing parameters]

[Wire thread points]

Pocketing parameters

To enable a pocketing set the appropriate checkbox in the wire EDM operation parameters window on the Pocketing page.



<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.

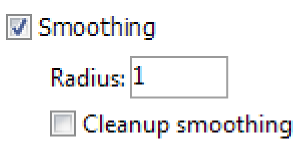
If you enable <Finish pass> a little additional stock will be left on the part. It will be removed by the last finishing pass, which always runs along the initial contour (at equidistant). This allows, for example, to remove the scallops after parallel roughing passes (if you specify a parallel machining strategy) and get higher quality of finished surface. This stock is relative to the base stock for finishing.

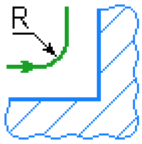
The <Stock> parameter allows you to specify the value of the material, which should be left to machine with finishing passes along the contour. Stock can also provide the necessary reserve to compensate the difference between the wire diameter and the size of the burned material. It can be defined in 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.

See also:

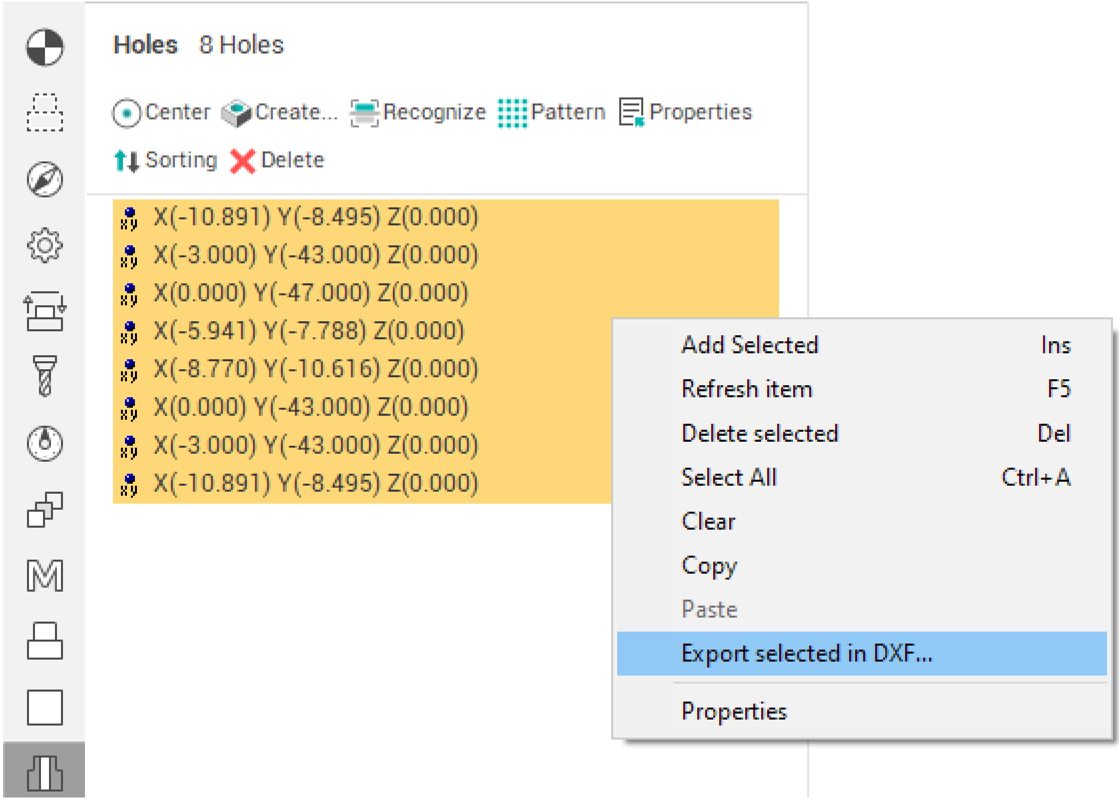
Wire EDM machining

Wire EDM coreless pocketing

Wire thread points

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.

See also:

Wire EDM machining

Wire EDM coreless pocketing

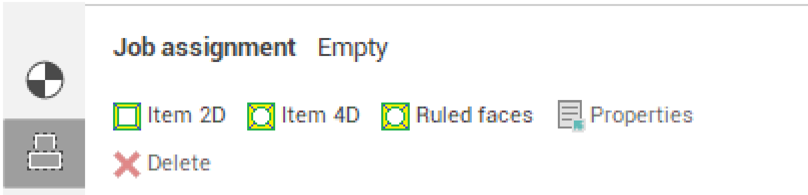
Pocketing parameters

5.10.2 Operations setup

5.10.2.1 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.

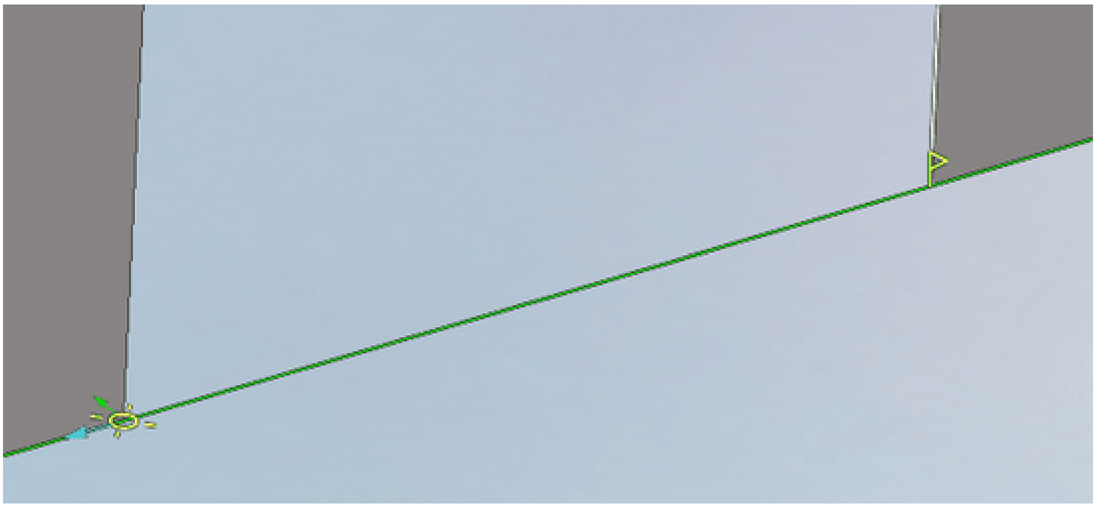
The following functions are available:

- <**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.

See also:

Wire EDM machining

2D job assignment item properties

4D job assignment item properties

Synchronization lines

Bridges

Multiselect feature

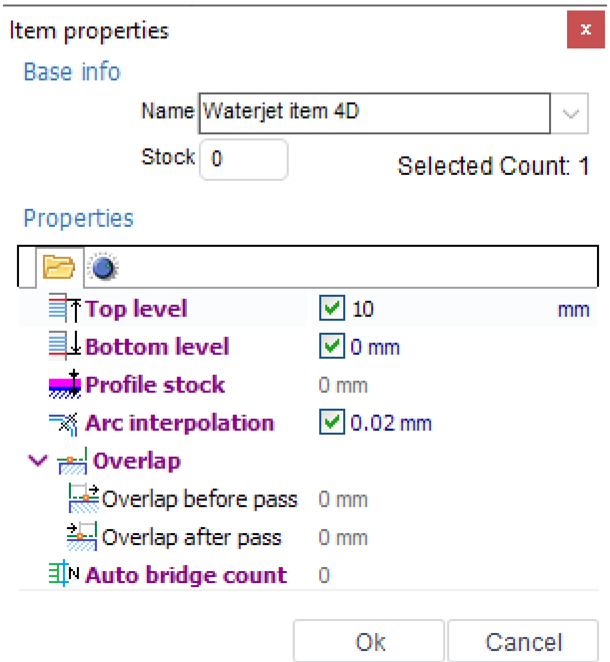
Wire EDM feature

2D job assignment item properties

Each element of the job assignment has a set of properties.

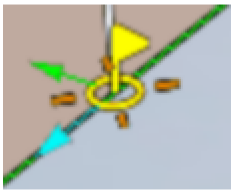
To view or edit the properties of 2D job assignment item select the item and click the <**Properties**> button, or double-click the item.

This is the item properties dialog:

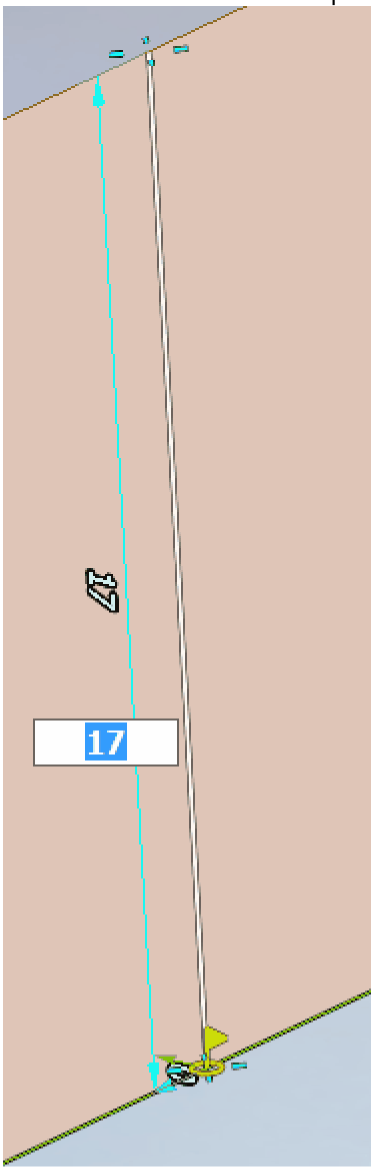


- <**Top level**> – plane for the top guide of the EDM-machine.To set the top guide level via the graphic window move the top level sign:

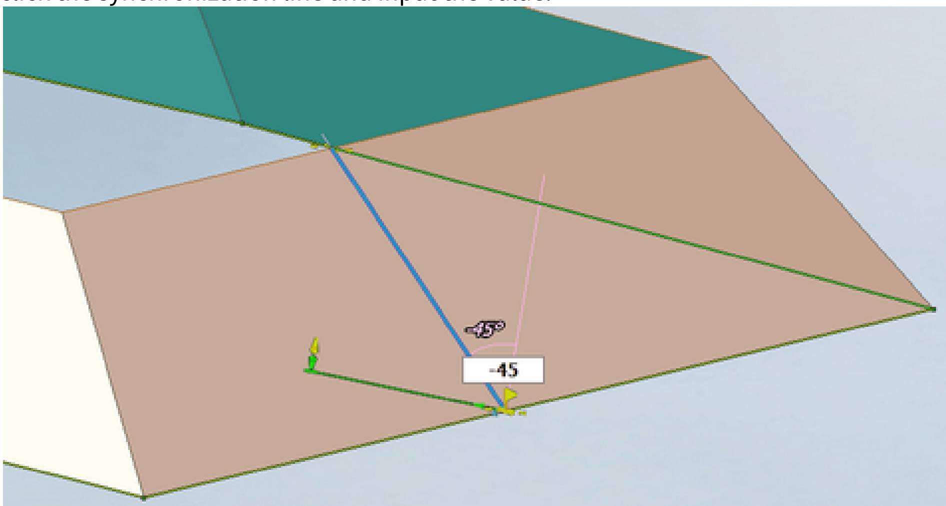
- <**Bottom level**> – plane for the bottom guide of the machine.To set the bottom guide level via the graphic window move the bottom level sign:



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:

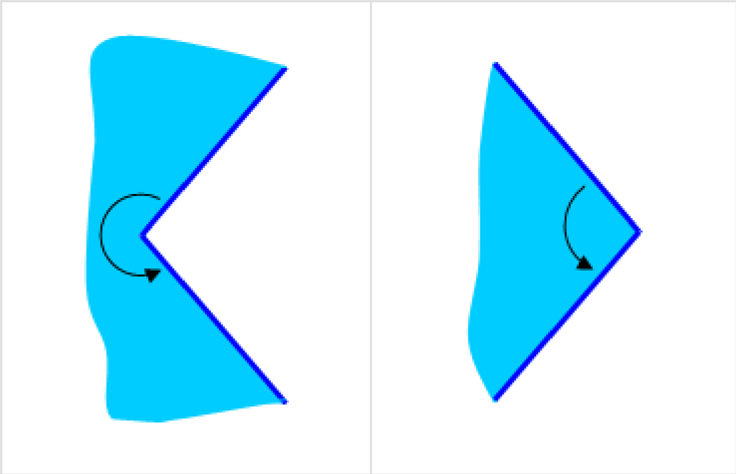
- <All passes> – taper will be applied for all passes of the contours.
- <Apply after pass> – taper will be applied after the pass number set in the <Pass #> field. Taper will be disabled for the N passes, passes starting with N+1 will have taper enabled.
- <Cancel after pass> – taper will be canceled 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. SprutCAM X 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: SprutCAM X simulation supports visualization of the wire path modified by the NC-controller corners rolling.

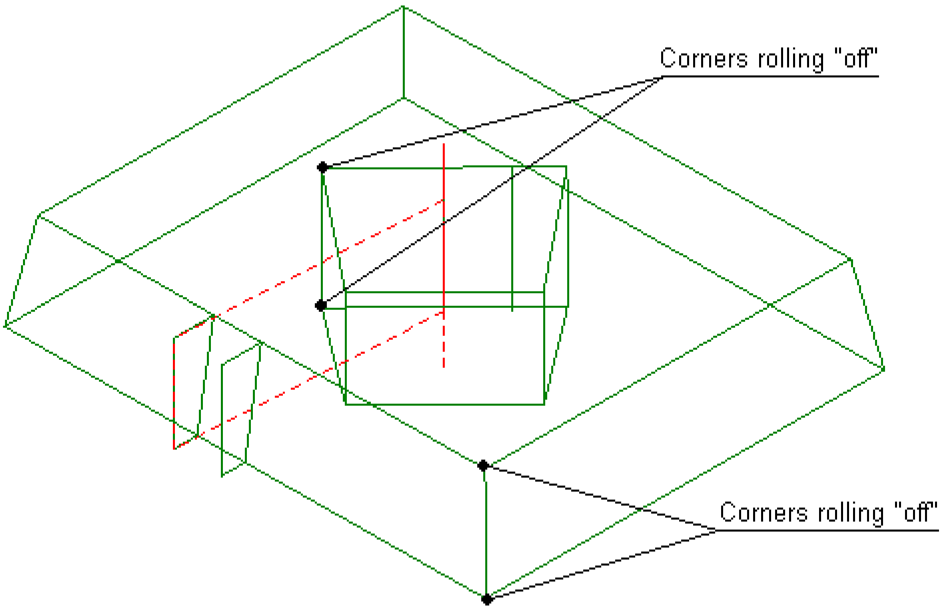
SprutCAM X supports various modes of corners rolling which can be set separately for inner(<Inner corners rolling>) and outer corners(<Outer corners rolling>). Whether the corner is inner or outer is determined by the angle value inside the part. Inner corner has the angle of 180° or more, outer has the angle less than 180°.

<Inner corner>	<Outer corner>

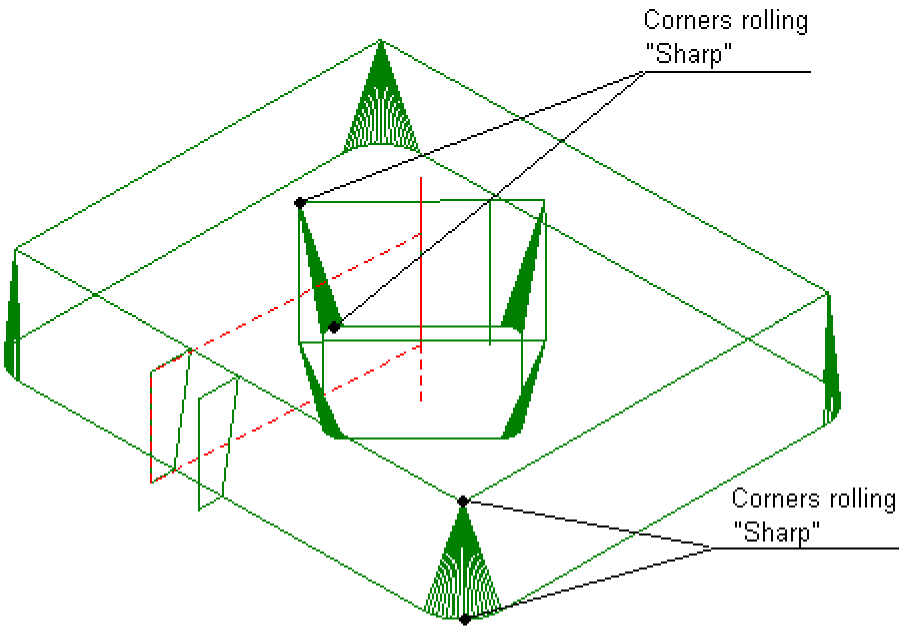


The following modes of rolling are supported:

- <Off> – in this mode corners rolling feature is disabled, rolling radii are not output into the NC-program.



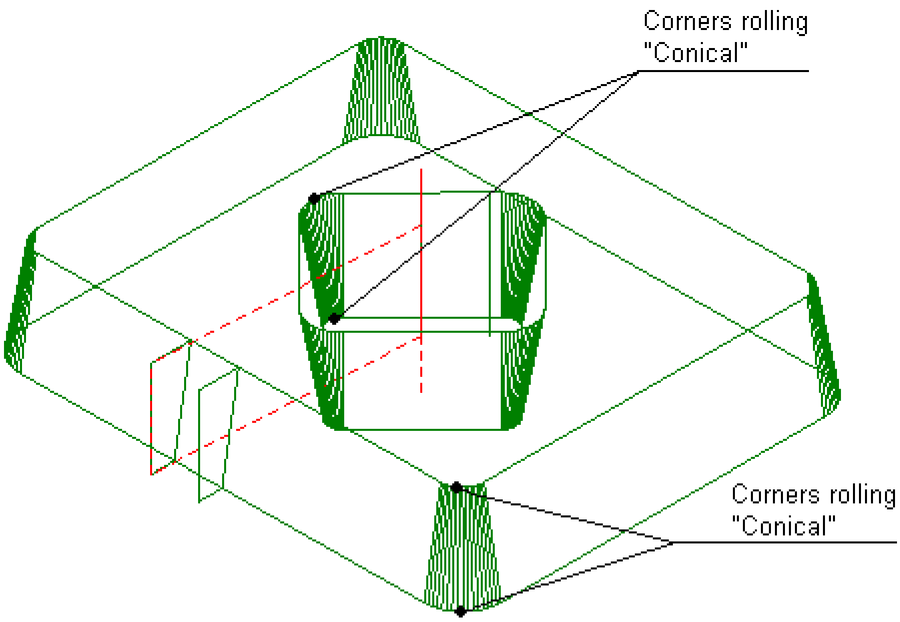
- <Sharp> – in this mode only the bottom contour corners are rolled, top level corners are not rolled. Therefore only the <Bottom radius> input field is enabled.



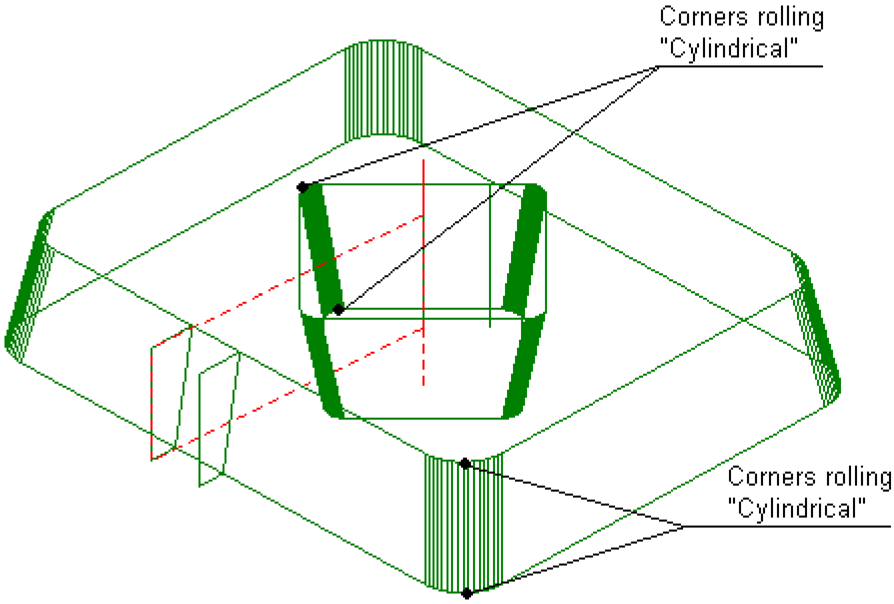
- <Conical> – in this mode the bottom contour corner rolling radius is set in the appropriate input field. Top contour rolling radius is defined a sum of bottom radius and a value depending on the taper angle and difference of top and bottom contour levels:

$$R_{top} = R_{bottom} \pm h \cdot tg\,\alpha$$

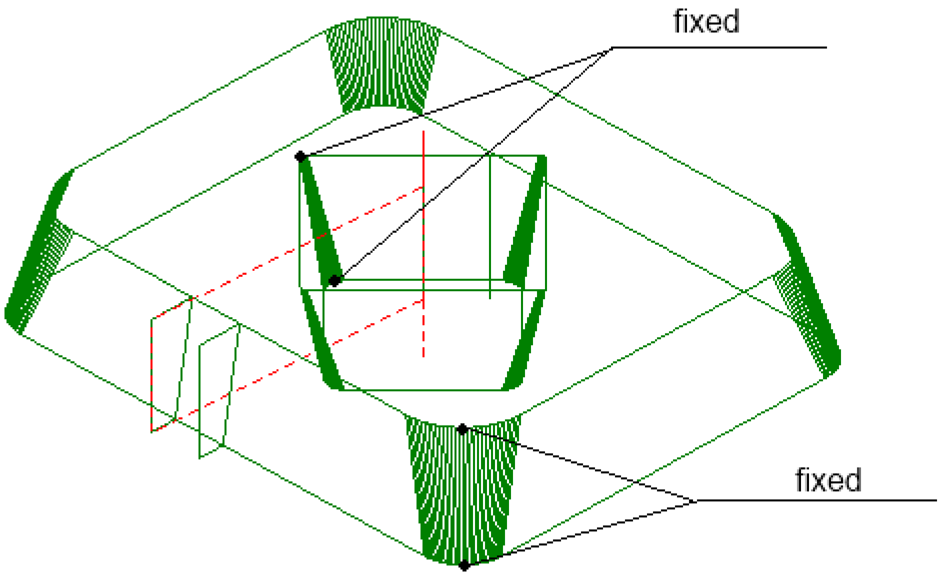
Thereby, a conical surface is machined on the part in the place of corner rolling.



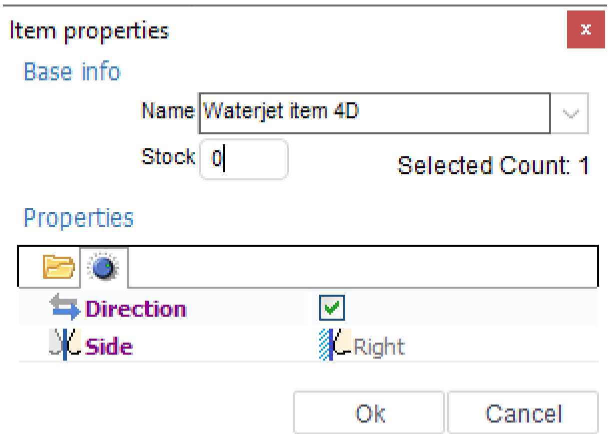
- <Cylindrical> – corner rolling radius is always equal for the top and bottom levels and is input in the <Bottom radius> field. Thereby, a cylindrical surface is machined in the place of corner rolling.



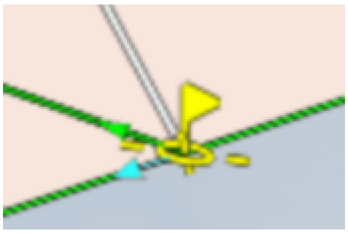
- <Fixed> – radii of rolling at top and bottom levels are defined independently in the respective fields and can be set to arbitrary positive values.



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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

4D job assignment item properties

Synchronization lines

Bridges

Multiselect feature

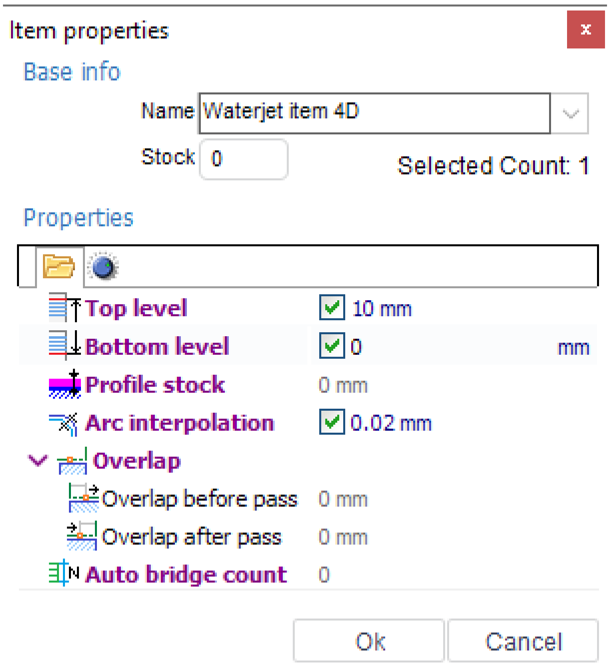
Wire EDM feature

4D job assignment item properties

Each element of the job assignment has a set of properties.

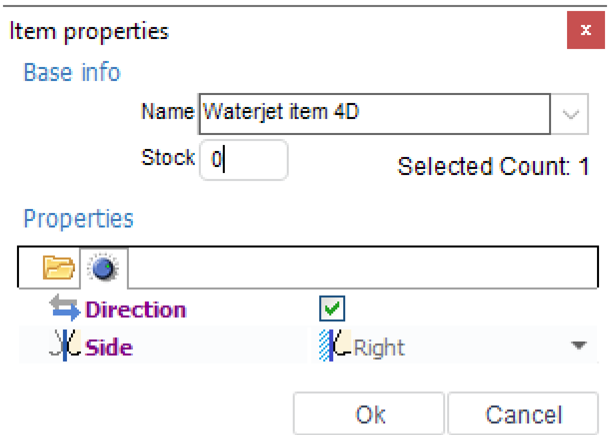
To view or edit the properties of 4D job assignment item select the item and click the <**Properties**> button, or double-click the item.

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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

Synchronization lines

Bridges

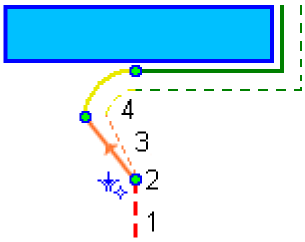
Multiselect feature

Wire EDM feature

Approaches/returns

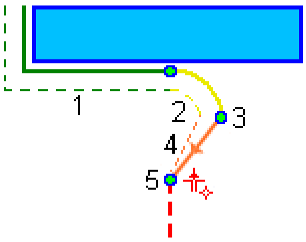
Lead-in and lead-out are the parts of the tool path, defined at the start and the end of each contour tool path. These are used for the correct machining at the start and end of a contour. These moves are used to enable various interpolation functions such as compensation, taper, multi axial interpolation, 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.

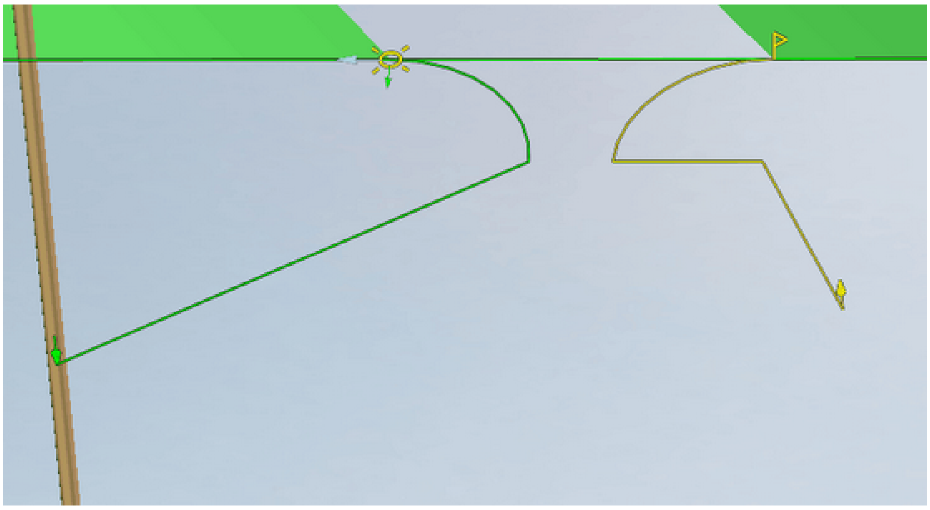
Retract from end point does the sequence in reverse:



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.

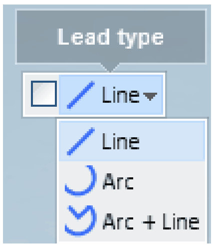
The lead-in/lead-out parameters are defined in the graphics window.

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.



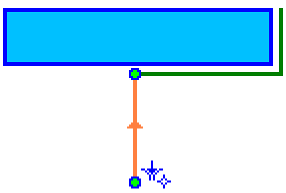
Approaches\returns markers are fully interactive. It is possible to move them and specify dimensions. Dimensions can be set as relative to other elements, as well as relative to the coordinate system.

It is possible to select lead mode in the pop-up panel:

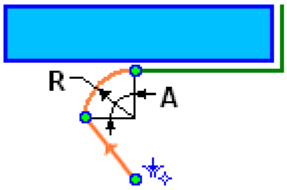


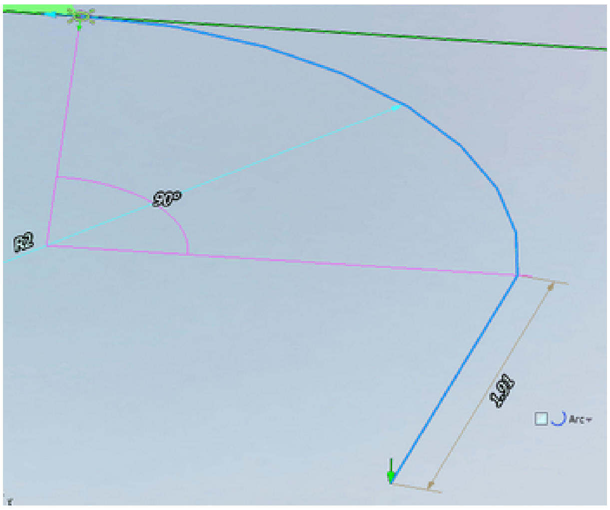
<Lead-type> – this panel is used to setup the lead type. There are several available lead types in the drop-down list:

- <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.

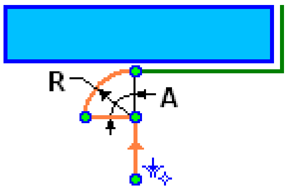


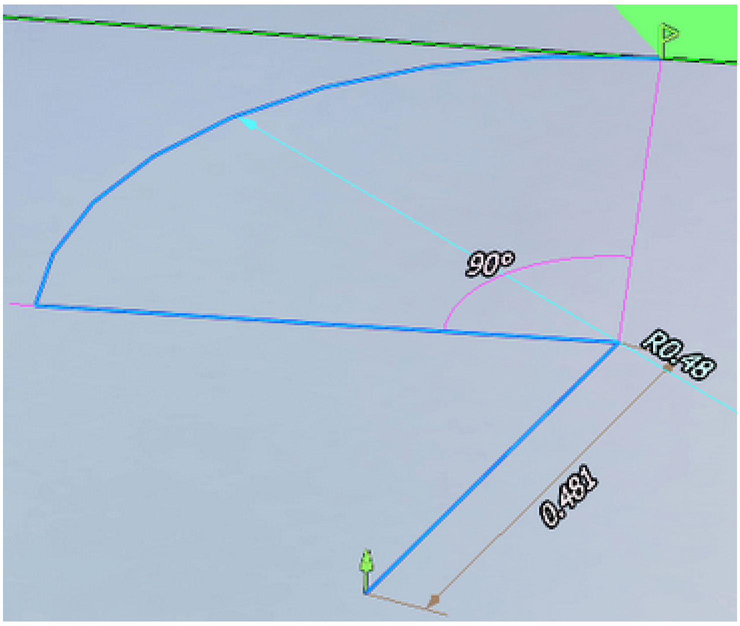
- <**A**rc> – lead-in has a linear motion from the wire load point to an arc start point. The arc move is tangent to the start of the machining contour.



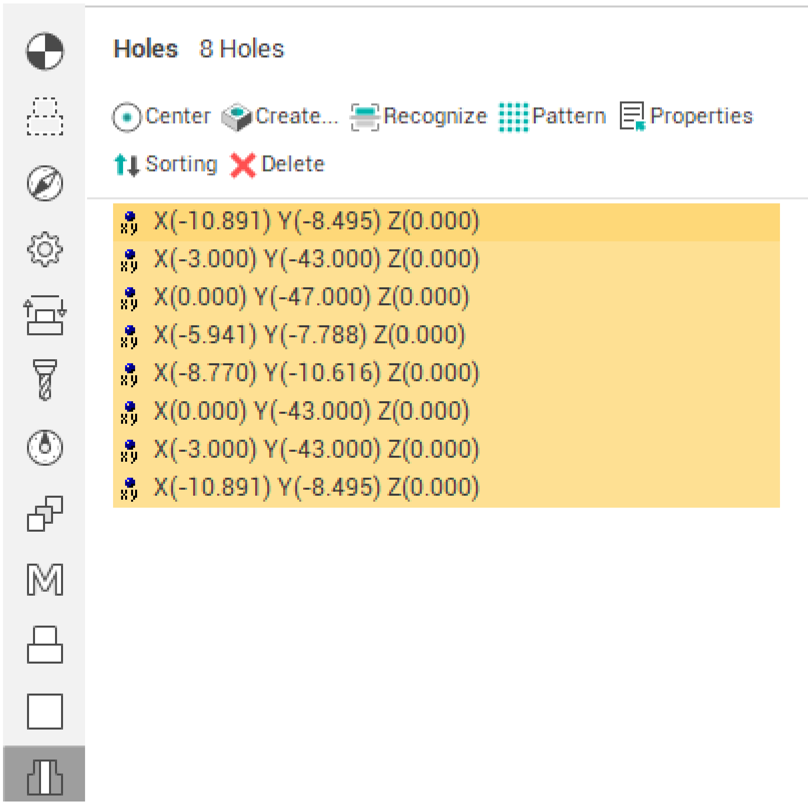


- <**L**ine and arc> – the first linear motion moves from the wire load point to the arc center point, then to the arc start point. Then the arc moves tangent to the start point on the contour.





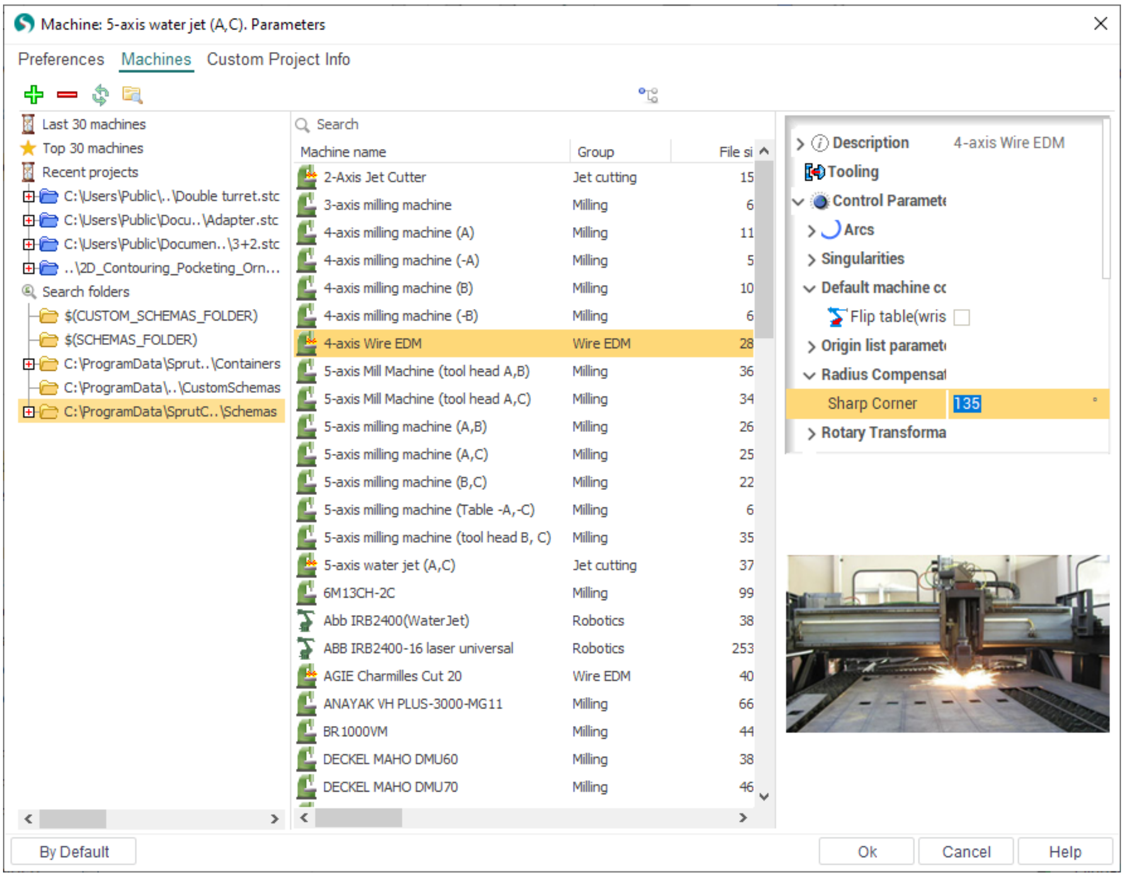
Attention: *All wire load or wire break points that are used when an operation is calculated, can be viewed on the <**Technology**> panel <**Holes**>. Also, you can export this list of points to use in another SprutCAM X project or other application. The export command is accessed from the* main menu *of SprutCAM X or from the context menu of hole list <**Export selected in DXF...**>.*



Wire radius compensation options

SprutCAM X 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. SprutCAM X 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. SprutCAM X have several options which can be used to 'tune' SprutCAM X'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 SprutCAM X simulation of radius compensation.

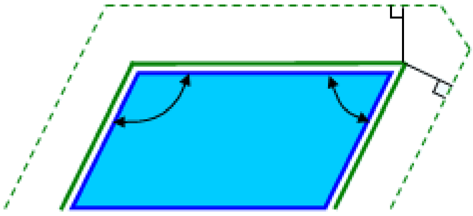


Use these properties:

- <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.



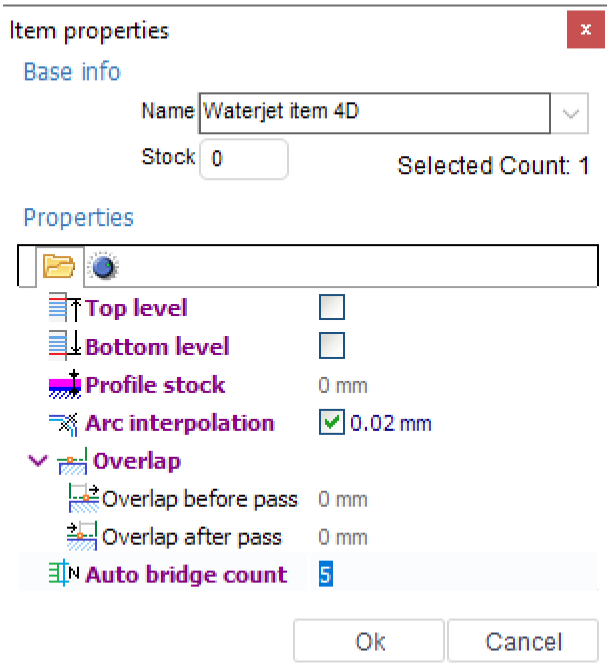
See also:

Wire EDM machining

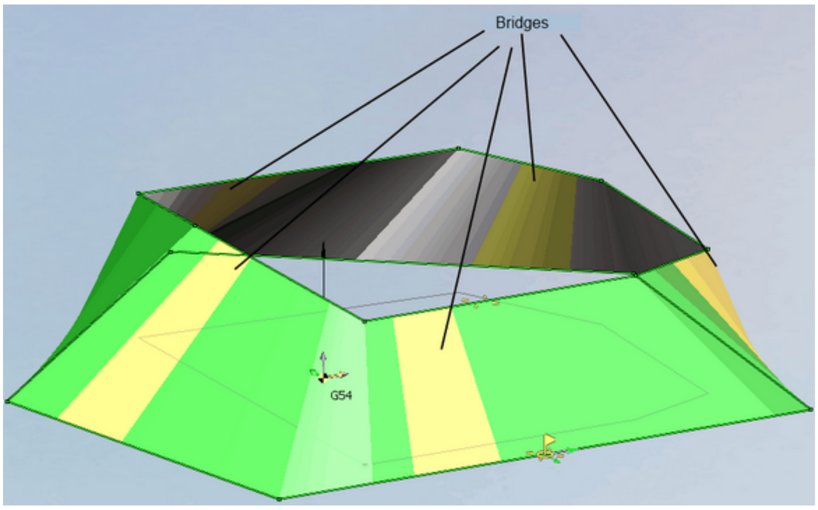
Bridges

<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.

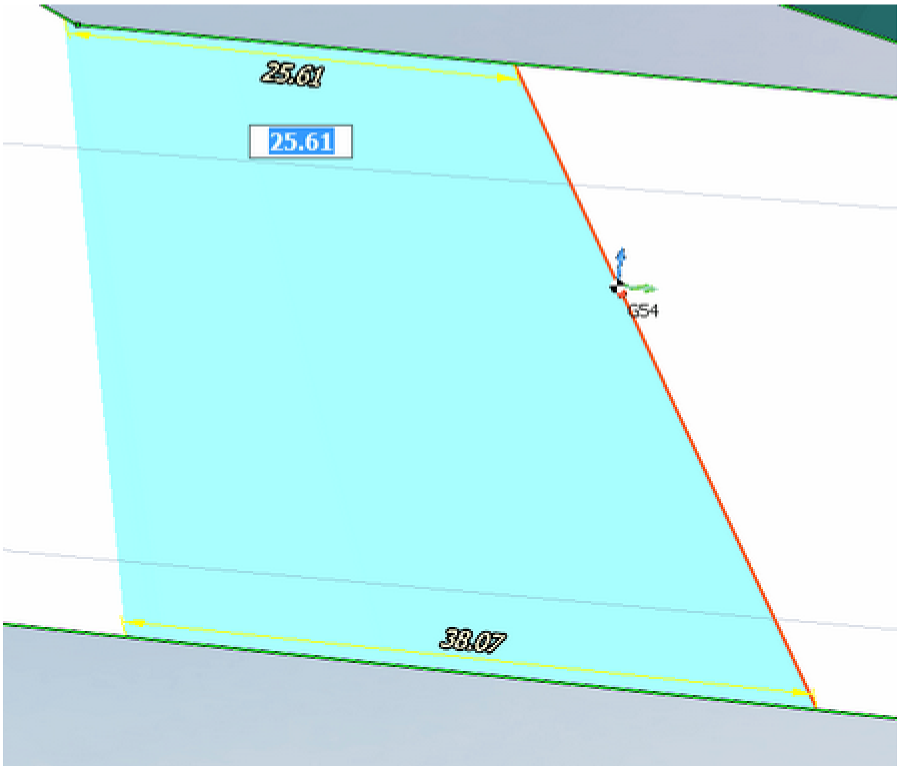


Auto bridges are placed at equal distance from each other. Change the parameters of bridges in the graphics window.



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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

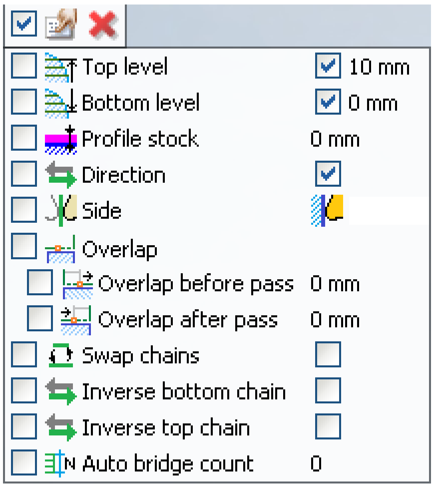
Synchronization lines

Multiselect feature

Wire EDM feature

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.

See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

Synchronization lines

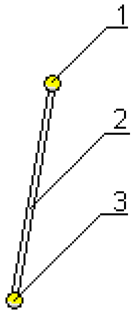
Bridges

Wire EDM feature

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).

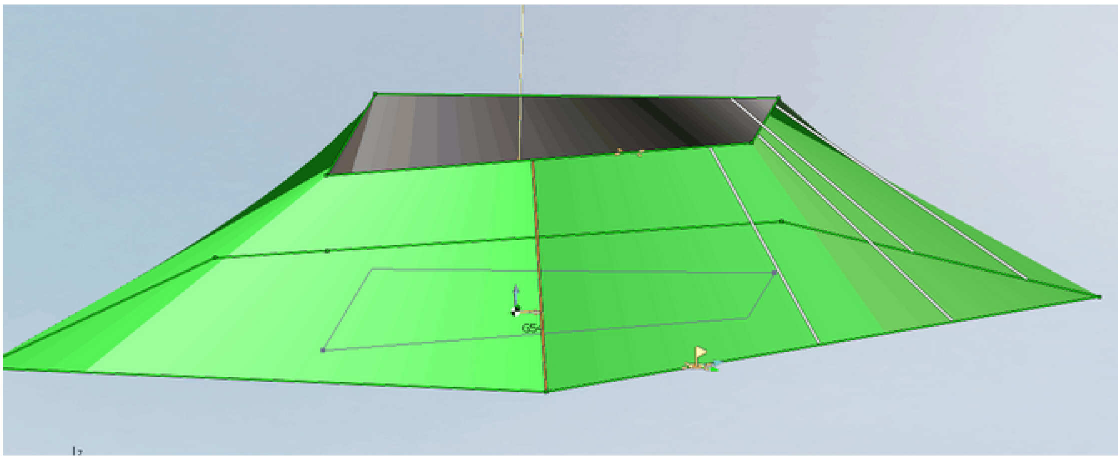


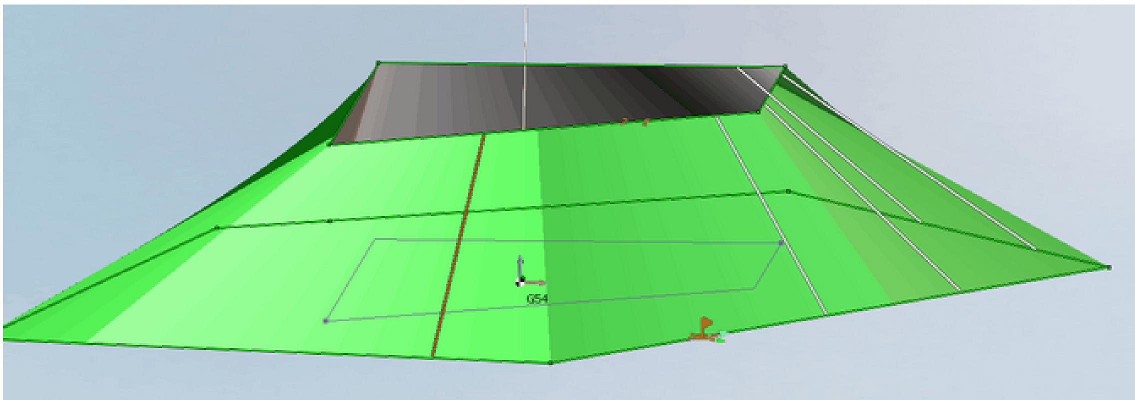
Synchronization lines can be copied and deleted in the graphic window.

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.





See also:

Wire EDM machining

Job assignment of wire EDM machining operations

2D job assignment item properties

4D job assignment item properties

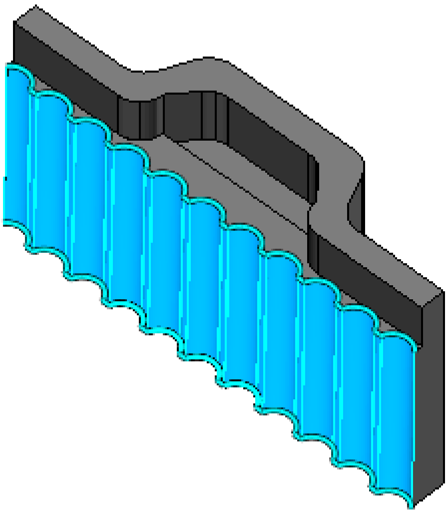
Bridges

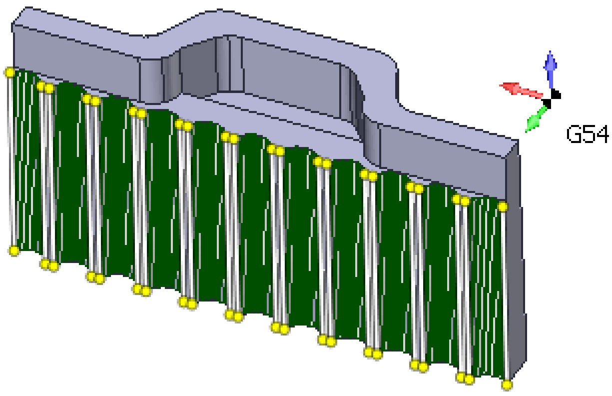
Multiselect feature

Wire EDM feature

Wire EDM feature

Wire EDM feature is a chain of ruled surfaces with top and bottom edges lying in horizontal planes. SprutCAM X 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. SprutCAM X 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>. SprutCAM X will setup operation parameters according to the constructive element properties if the job assignment was empty.

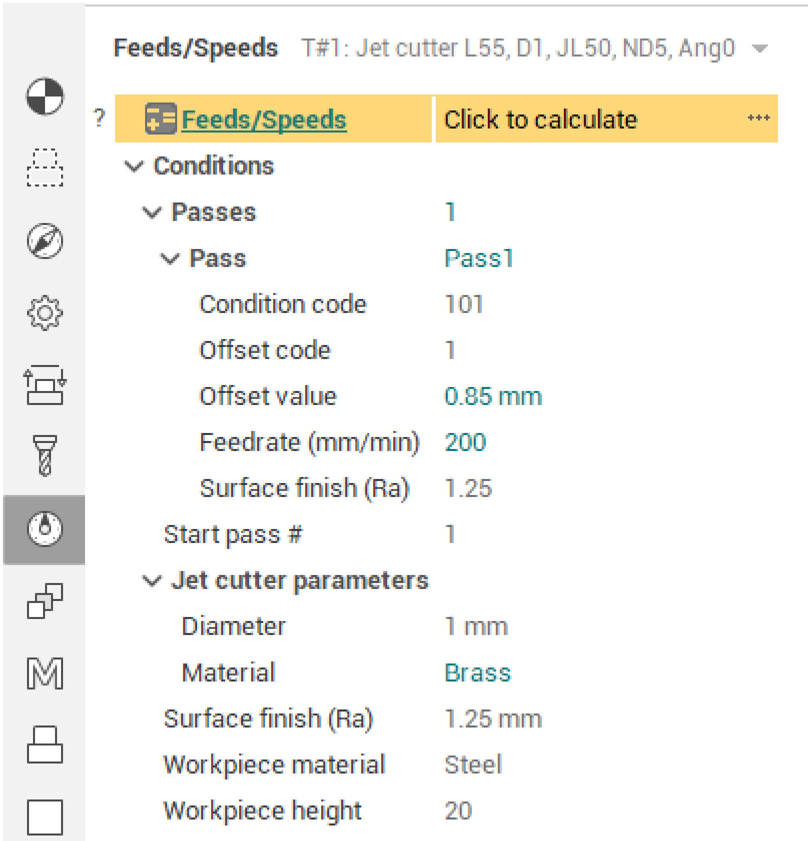
See also:

Wire EDM machining

Job assignment of wire EDM machining operations
2D job assignment item properties
4D job assignment item properties
Synchronization lines
Bridges
Multiselect feature

5.10.2.2 Machining conditions of EDM and Jet cutting 4D operations

Defining of machining conditions is defined in operation parameters window on <Feeds/Speeds> page.



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.

An specific for every machine information about machining conditions can be filled and saved in special cutting parameters library. It is saved in single *.csv file. The current library file is shown on <Name> field in <Library> panel. From the list of this field can be selected one of the libraries from standard libraries folder of SprutCAM. The library can be assigned also from another place with help of standard file-dialogue window, that activated by the ... button. In <Commentary> field is shown additional text information about selected library.

Machining technology library consists of so-called process technologies list. Every process technology contains following menu:

- <Technology> – 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.
- <Pass Parameters> – list of passes with process parameters for every pass.

The <Pass> term is one pass, that wire is does along the contour and following list of parameters is assigned for.

- <Pass name> – text description of the 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.

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

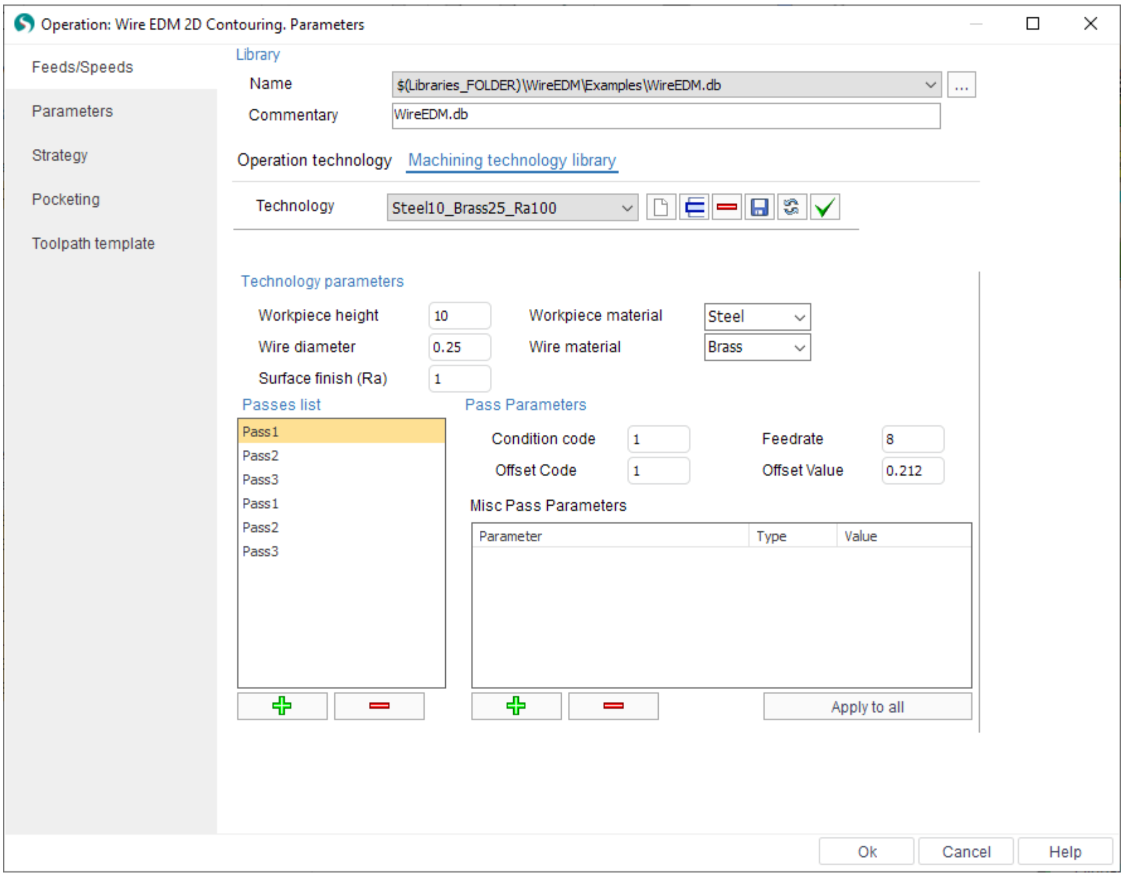
- Workpiece material: steel.
- Workpiece height: 40.
- Wire material: brass.
- Wire diameter: 0.5.
- Surface finish (Ra): 0.8.

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
...								

...

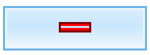
Working with conditions library does on the <Machining technology library> page:



At every instant only one active technology of library is edited. Its name is shown in the <Technology> field. In drop-down list of the field is shown list of all technologies, which library is contain. There is following functions to work with it:

-  <New technology> – adds new technology to the list.
-  <Rename technology> – renames active process technology.
-  <Remove technology> – deletes active technology from the list.
-  <Save library> – saves all changes to a library file.
-  <Reload library> – loads information from a library file anew. All changes in technologies will be lost.
-  <Select technology> – copies active technology parameters from the library to the operation technology.

Editing of current technology parameters does by changing values on the <Technology parameters> panel. The <Passes list> panel shows passes names, which active technology has. There is two functions available:

-  <Add pass> – adds new pass to the list.
-  <Delete pass> – removes selected pass from the list.

When one of the passes is selected, on the <Pass Parameters> panel its properties is shown. As stated above, in addition to basic parameters the pass has additional ones. Consisting of this parameters can be changed by user in a table on the <Misc Pass Parameters> panel. Input of names and values is performed by mouse click on corresponding cell. Below the table is three control buttons:

-  <Add parameter> – adds a new line to the table of additional parameters of selected pass.

- <Remove parameter> – removes active line from additional parameters table.
- Apply to all – copies additional parameters table of selected pass to all passes of active technology.

Each wire EDM operation has its own process technology, that contains the same set of properties as a technology from machining technology library. When operation is calculating, it is follow technology from its parameters, but not from the library. Therefore, in order to apply active technology from the library, there is need to copy its parameters to the operation by pressing <Select technology> button. Properties of operation is shown on the <Operation technology> tab. They can to be edited even if the library is an empty. Library can be filled from this tab also by pressing <Save to library> button.

If there is filled library of processing conditions exists, then can be used quick search of machining technology with required parameters. After pressing the Find button system will analyze fields values from <Operation technology> tab and will look for closest technology in the library by using following algorithm. Among all technologies of library system will select those, that has workpiece material, wire material and diameter the same, as defined. Next, the system will look from founded for a technology, that will guarantee higher and closest surface roughness. At the same time system will take into account, that workpiece height must not to be less from defined and to be the closest. If there proper technology is exists, then system will go on <Machining technology library> tab and will activate this technology. Now need to press <Select technology> button to apply the technology. System will return to the <Operation technology> tab and new parameters can to be edited again to fit specific case of the machining.

On this tab is <Start pass #> parameter is exists. It allows to define number of the pass, from which machining will be begin. 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.

See also:

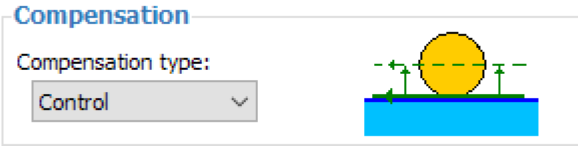
Wire EDM machining

5.10.2.3 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.



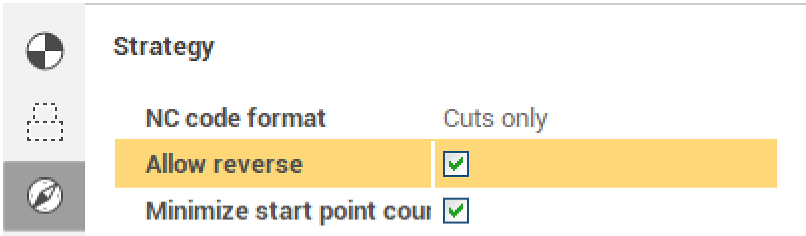
The following compensation types are available:

- <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.

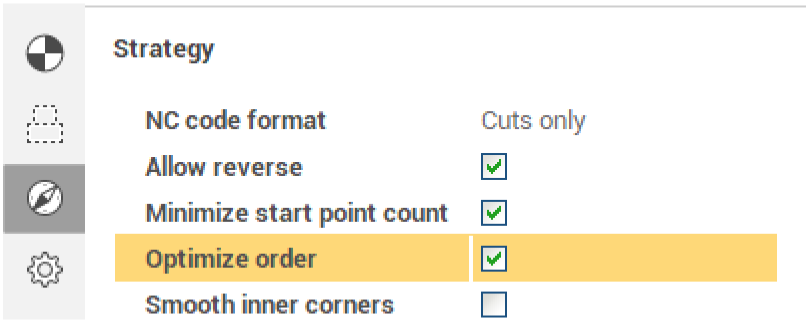
The value of compensation for each pass is defined as an <Offset value>, in the <Feeds/Speeds> tab, plus the stock value of the operation. For the compensation types <Computer>, <Both> and <Reverse Both> the value is used to construct an equidistant path, and for the <Control> type the value is entered 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.

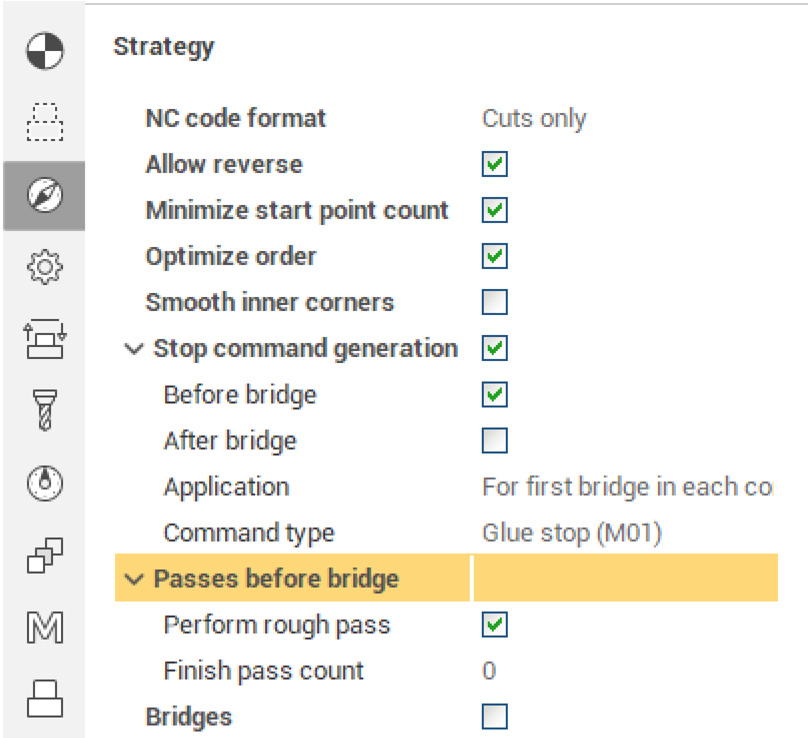
- <Reverse machining direction> – if you enable this option, the system will choose the direction for the pass, which provides the smallest length of the toolpath. If the option is disabled, then the direction of the pass will always correspond to that specified in the <Job assignment> for the contour.



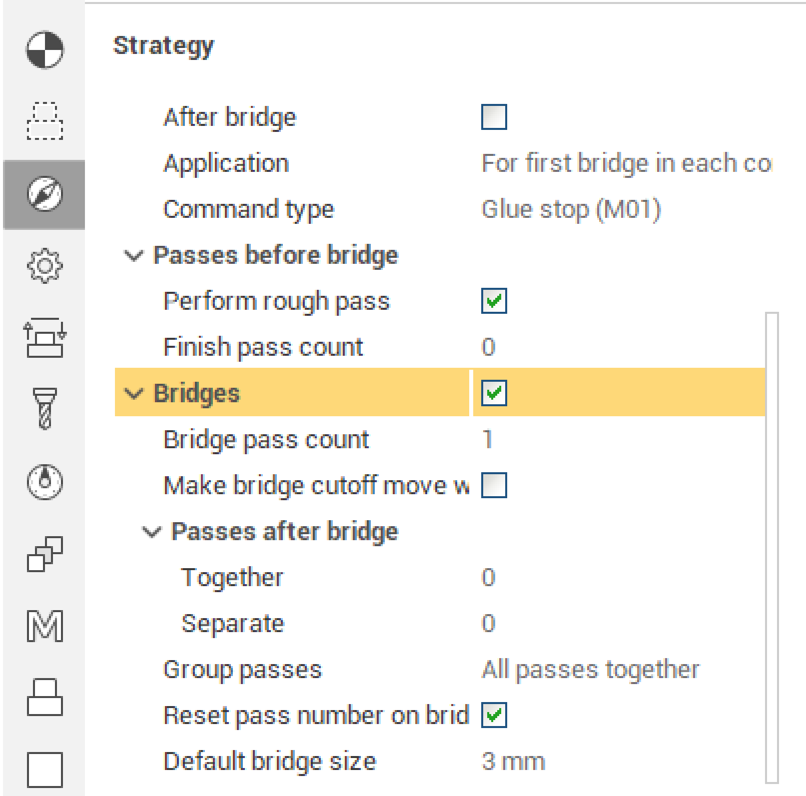
- <Optimize order> – this strategy determines the order of contours passes when the job assignment has more than one contour. The length of transitions between the contours will be minimal if the <Optimize order> option is enabled. If this option is disabled, then the order of the passes would be consistent 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.

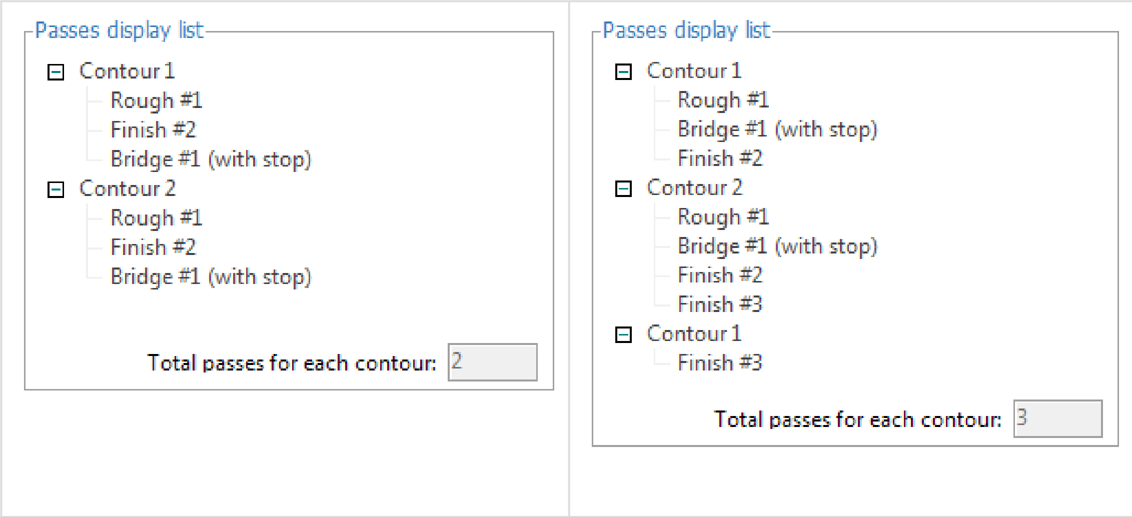


The <Bridge pass count> field sets the number of passes which will be performed for each bridge cutting on each contour. When the option <Reset pass number on bridge cuts> is enabled, then the count rates for the passes that define the cutting conditions for the bridge cutting moves are reset to the 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:

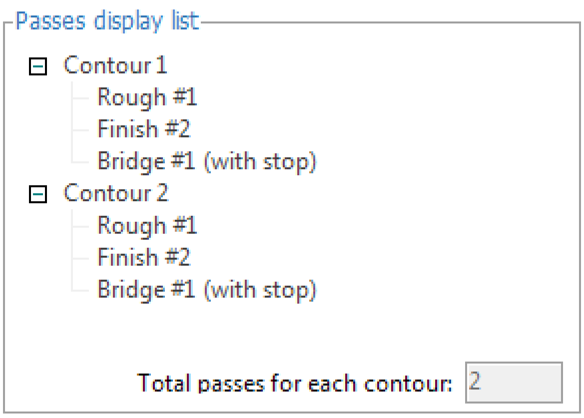


In the drop-down <Group passes> menu You can choose the way of grouping different types of passes for when working with several contours.The following options are available:

- <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>.

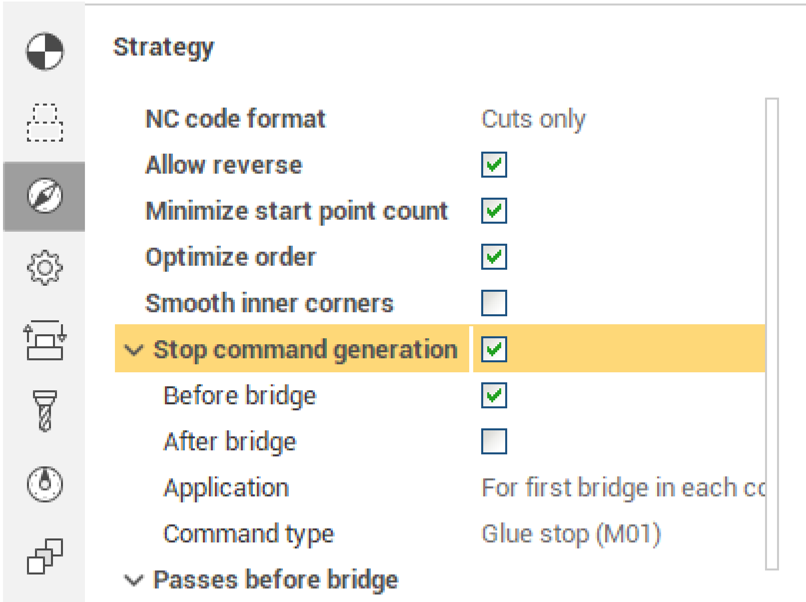
- <Passes display list>. On the <Strategy> tab for the Wire EDM operation there are many available options for contour machining that control the manner and the order of processing for the contours of the job assignment. To better understand the impact of a particular parameter on the machining sequence, 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.



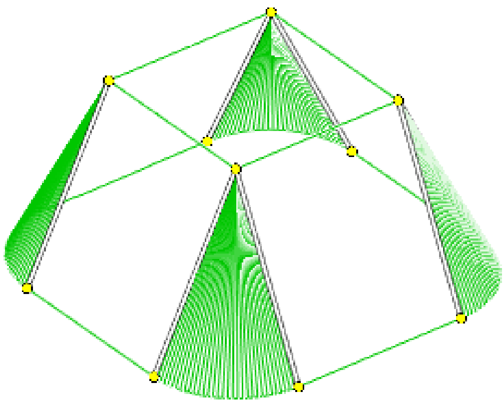
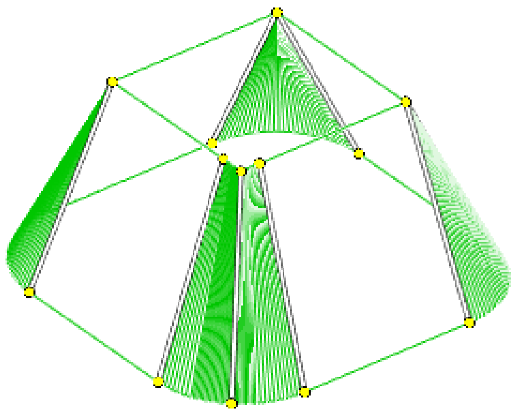
The drop-down list <Application> determines how the bridges should use the technological stops. The options include:

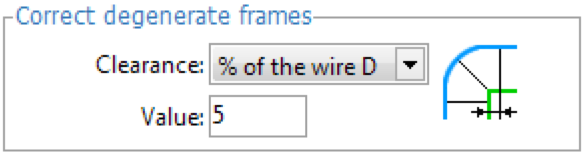
- <For every bridge> – stop command will be output for the bridge cutting move for every bridge specified in the operation.
- <For first bridge in each contour> – stop command will be output only for the first bridge of each contour.
- <For first bridge in operation> – stop command output only when cutting 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).

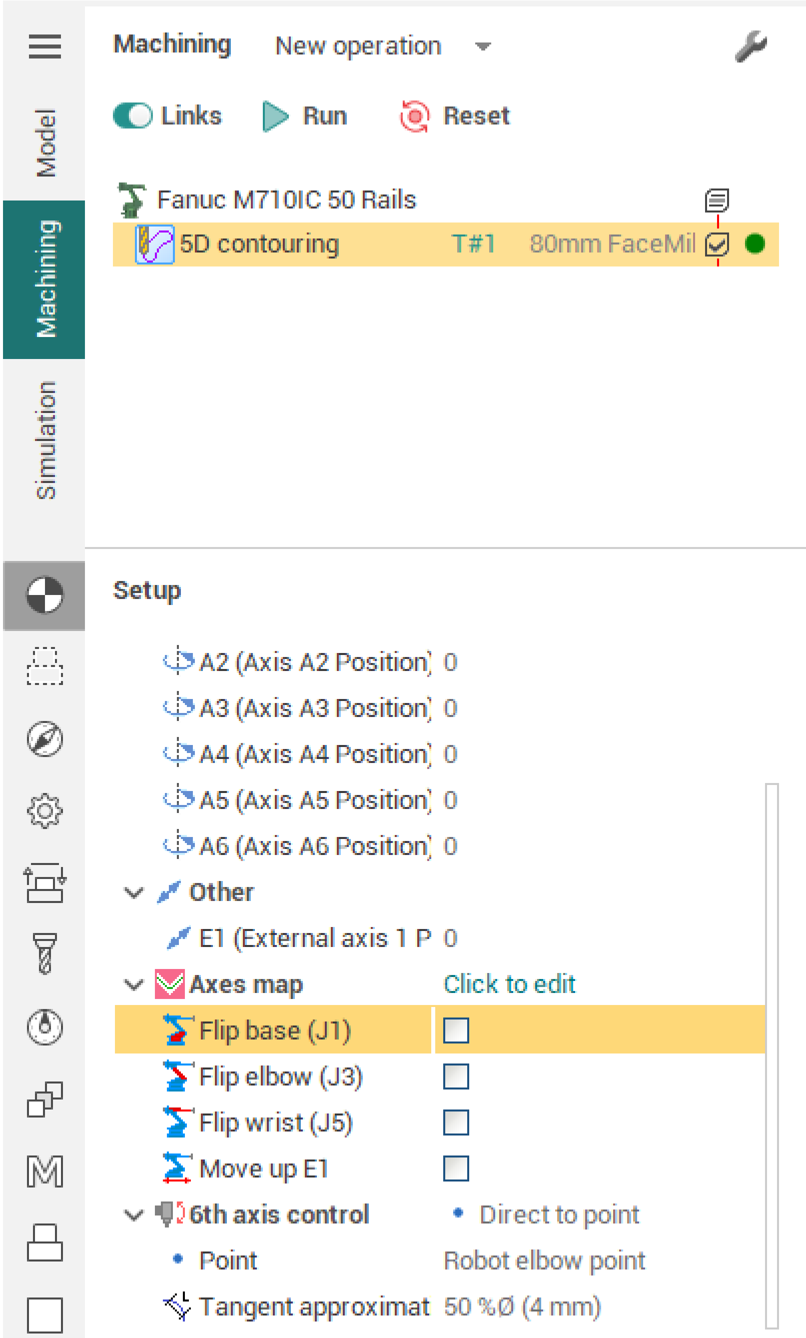
See also:

Wire EDM machining

5.11 Machining on industrial robots

SprutCAM X 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 SprutCAM X 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 SprutCAM X configurations:

- Robots
- Master
- Pro

5.11.1 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).

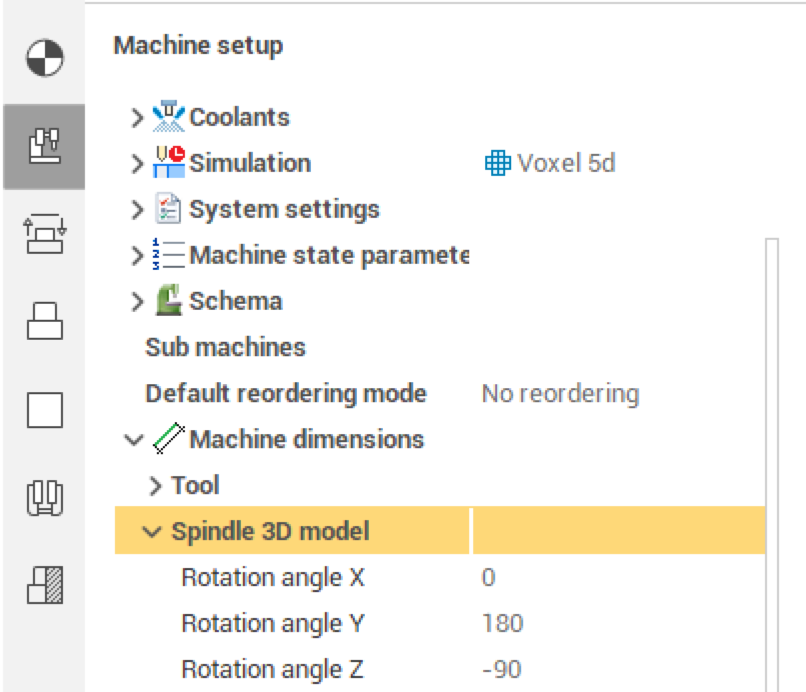
Defining the tool coordinate system

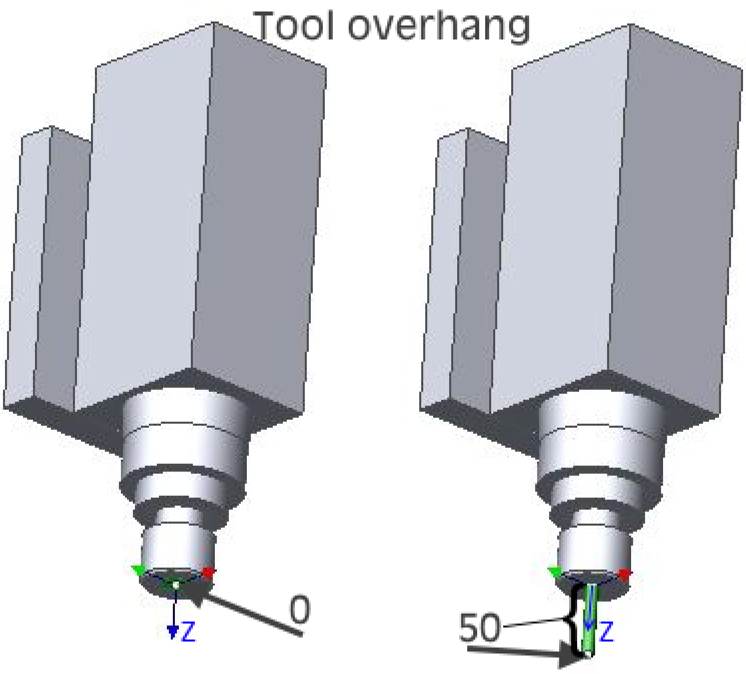
The tool coordinate system is defined by specifying the origin and the rotation angles of the tool CS relative to the base robot flange coordinate system, and the tool overhang.

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 SprutCAM X 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:

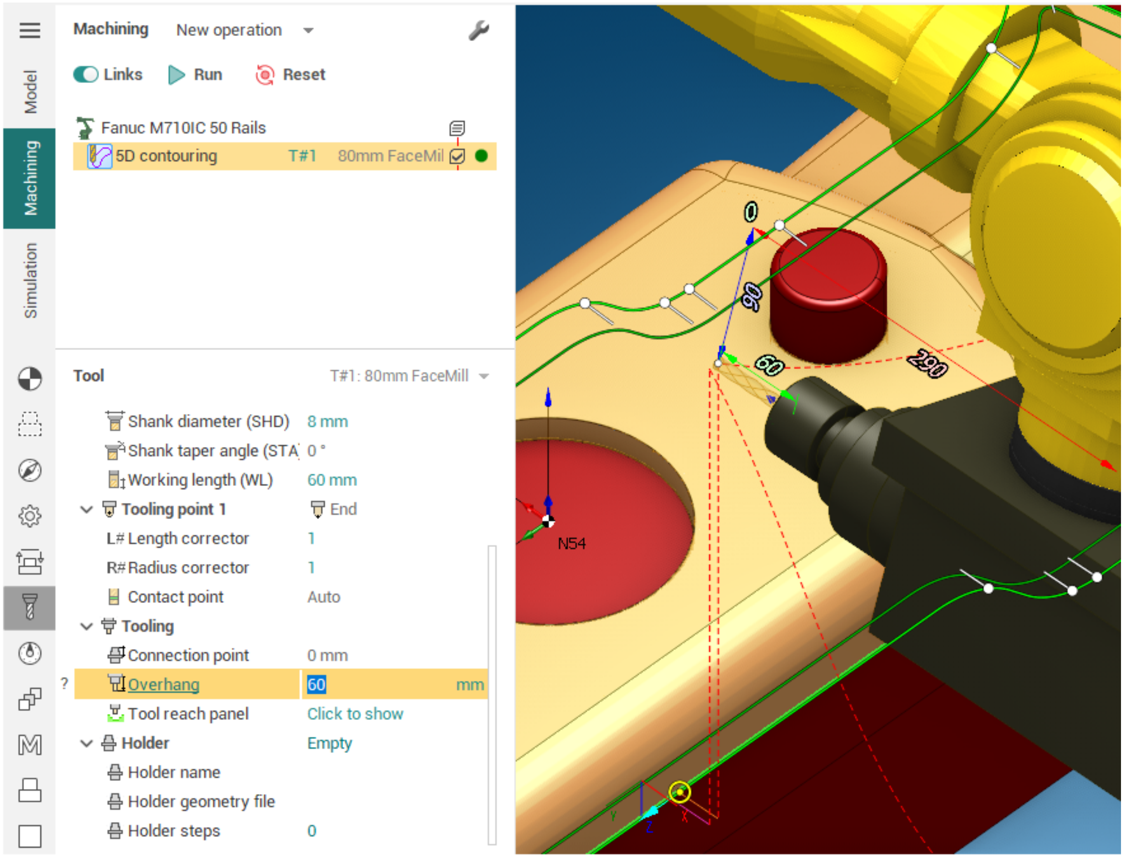
<X DefaultValue="134.83"/>).





2. Set the tool overhang for an operation

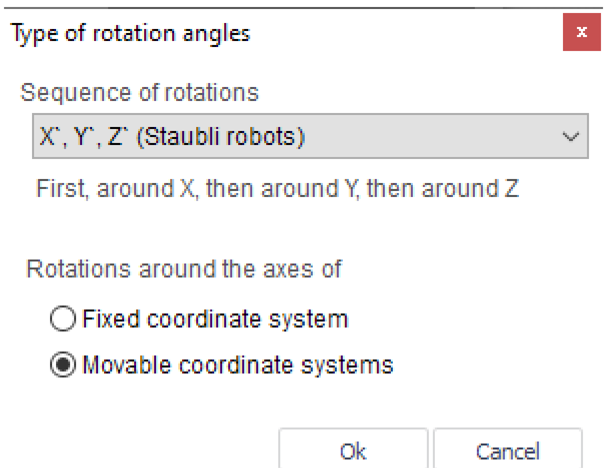
For every new tool you'll be using in the operations of the project you have to specify the tool overhang. To do so activate the <Tool> tab  in the operation parameters inspector and click on the <Overhang> parameter (As shown in the picture below). The tool overhang (the lime dimension) and the three 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 SprutCAM X 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.



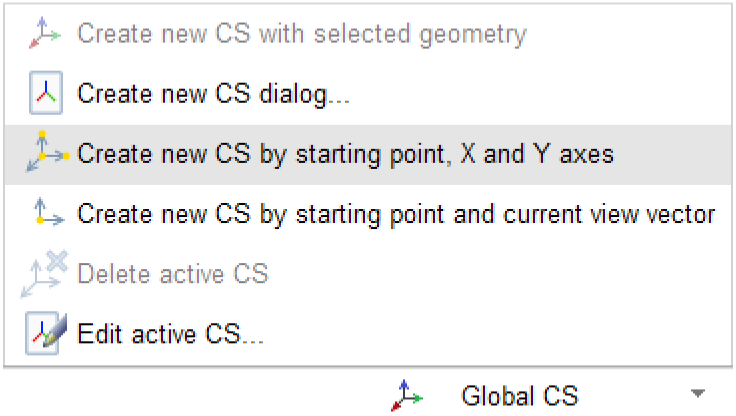
Defining the user coordinate system of the workpiece

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 SprutCAM X create a new coordinate system using exactly the same approach you used on the actual robot:

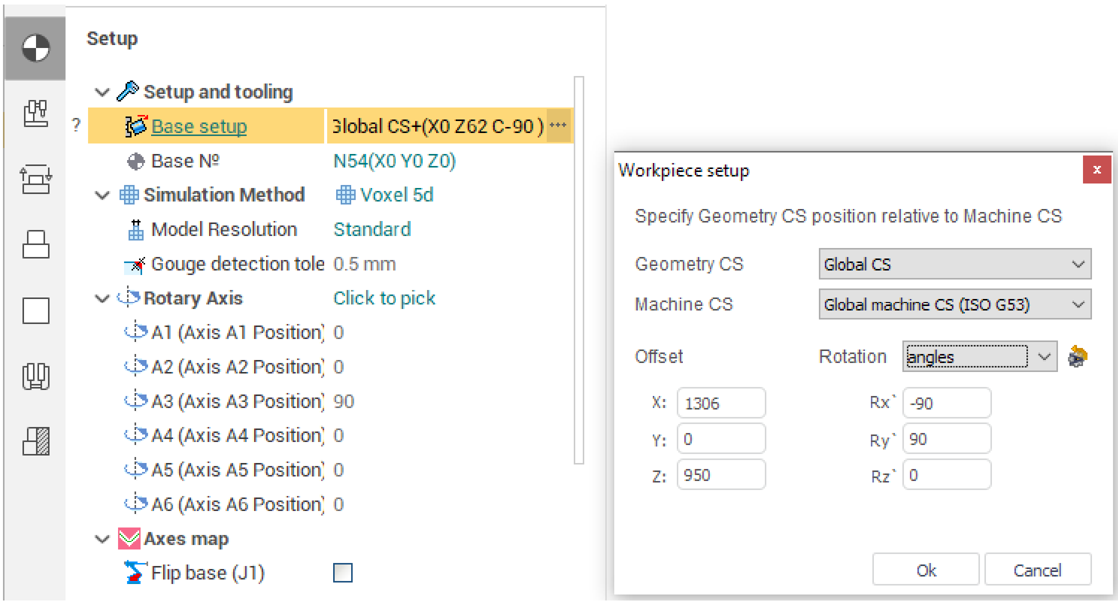
- Create a new coordinate system by selecting the <Creation of CS by setting starting point CS and direction of X and Y axes> command and click the three points on the model of the workpiece you have just been using when defining the user coordinate system on the real robot.

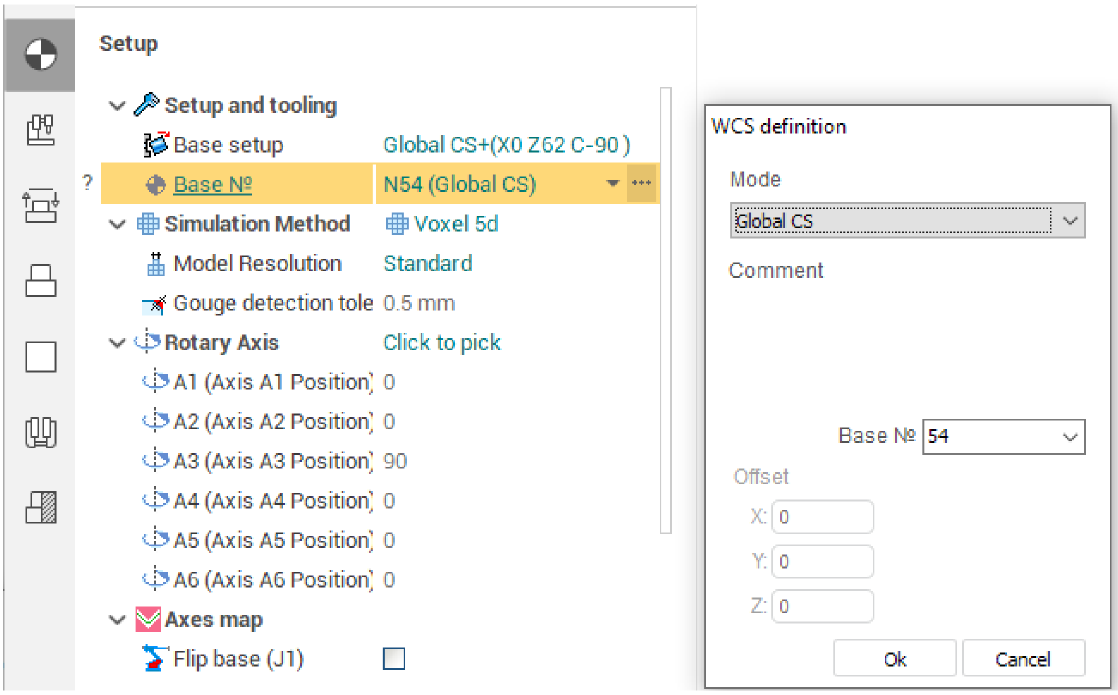


- In the <Workpiece setup> dialog select the newly created coordinate system as the <Geometrical CS>. In the boxes for the Translation and the Rotation enter the values XYZ ABC (WPR, RxRyRz, q1, q2, q3, q4) obtained on the robot.
- In the <Workpiece CS> dialog select the newly created coordinate system 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 SprutCAM X have to reflect the position of the real workpiece relative to the real robot.

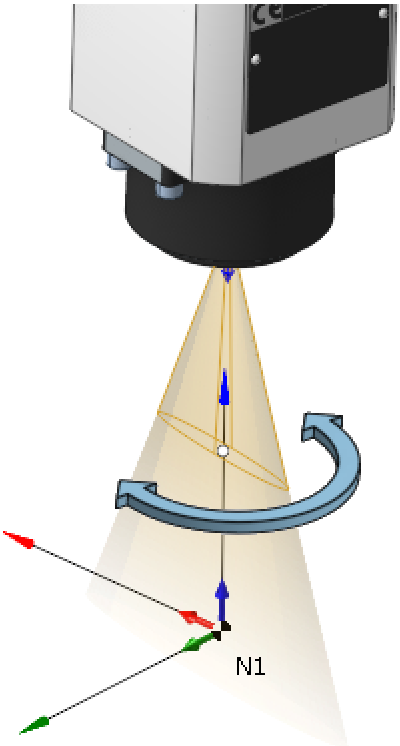
It is recommended to set up the user workpiece coordinate system in the root of the job tree (the very first item with the robot icon and caption in the tree) rather than in operations.





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 SprutCAM X as newly imported models of parts will be placed in more predictable positions relative to the robot.

5.11.2 Programming the robot's 6th axis

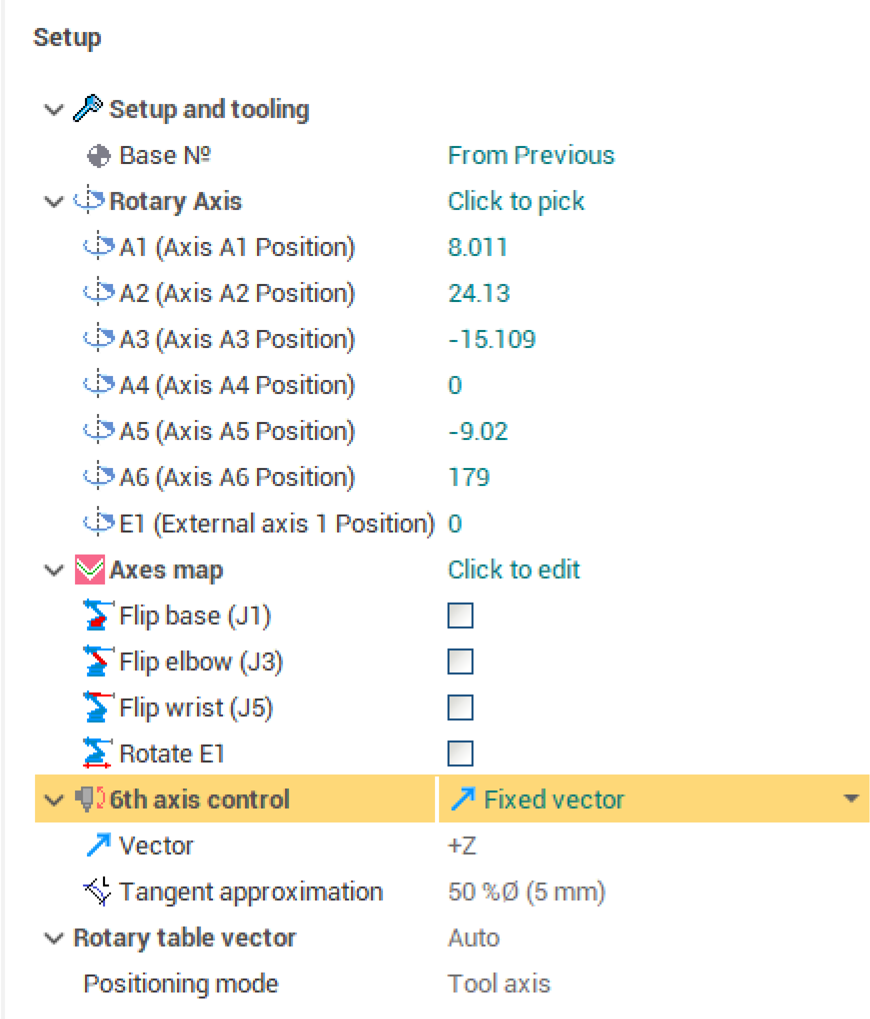


To position a rotating tool relative to the workpiece five degrees of freedom is enough, yet most of the robots have 6 degrees of freedom. The last 6th DOF is represented by an additional joint at the tool flange and is used to extend the robot's flexibility and the reach zone (by fixing the tool position 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 **SprutCAM X** there are two ways of programming the 6th axis:

- automatic,
- manual.

You can control it on the operation's properties inspector.

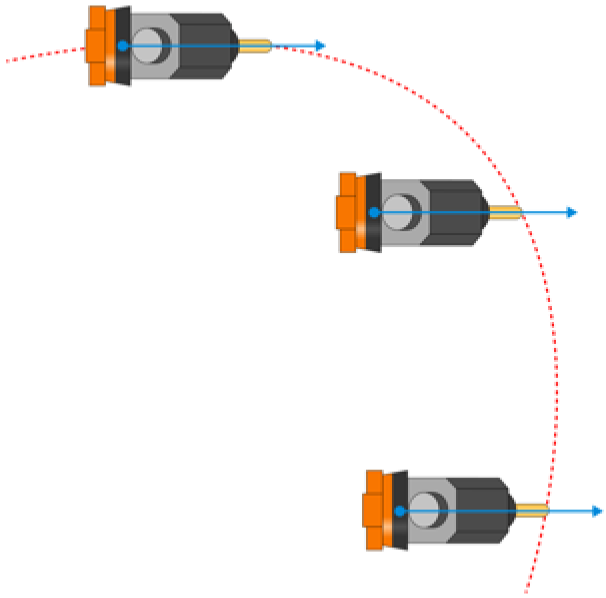


The **automatic way** is represented by the few modes of the **6th axis control** property.

- <**Fixed vector**>,
- <**Direct to point**>
- <**Toolpath**>.

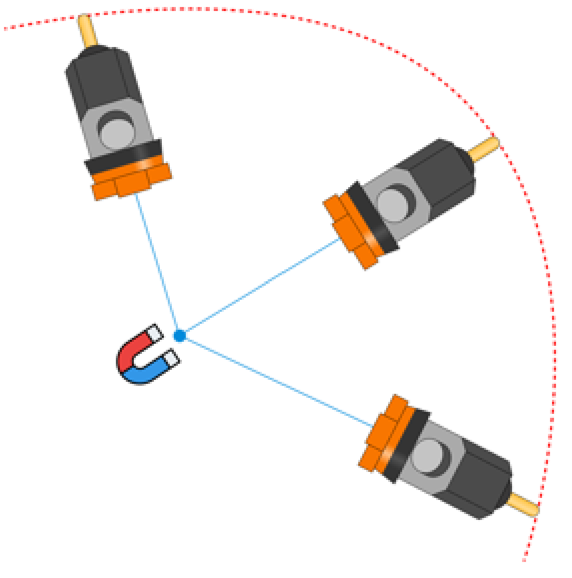
Fixed vector 6th axis control mode

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.



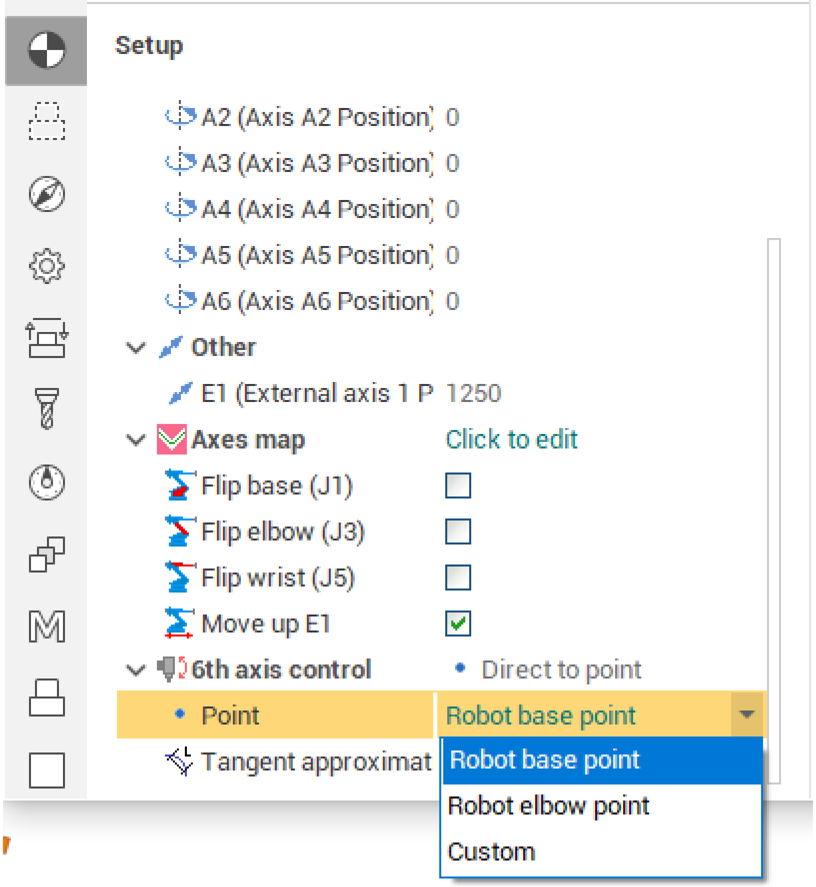
Direct to point 6th axis control mode

In this mode you specify a 3d point to which the robot's tool flange vector is directed during the machining.



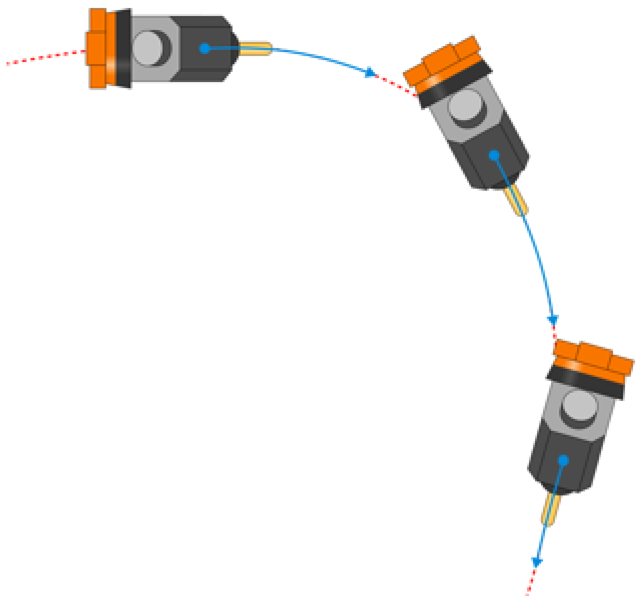
The point can be either

- the robot base point, or
- the robot elbow point,
- or a custom point.

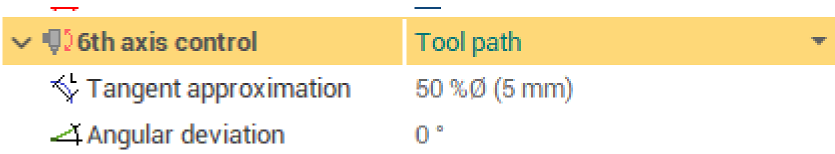


Toolpath 6th axis control mode

In this mode one axis of TCP is aligned with the toolpath tangent direction.

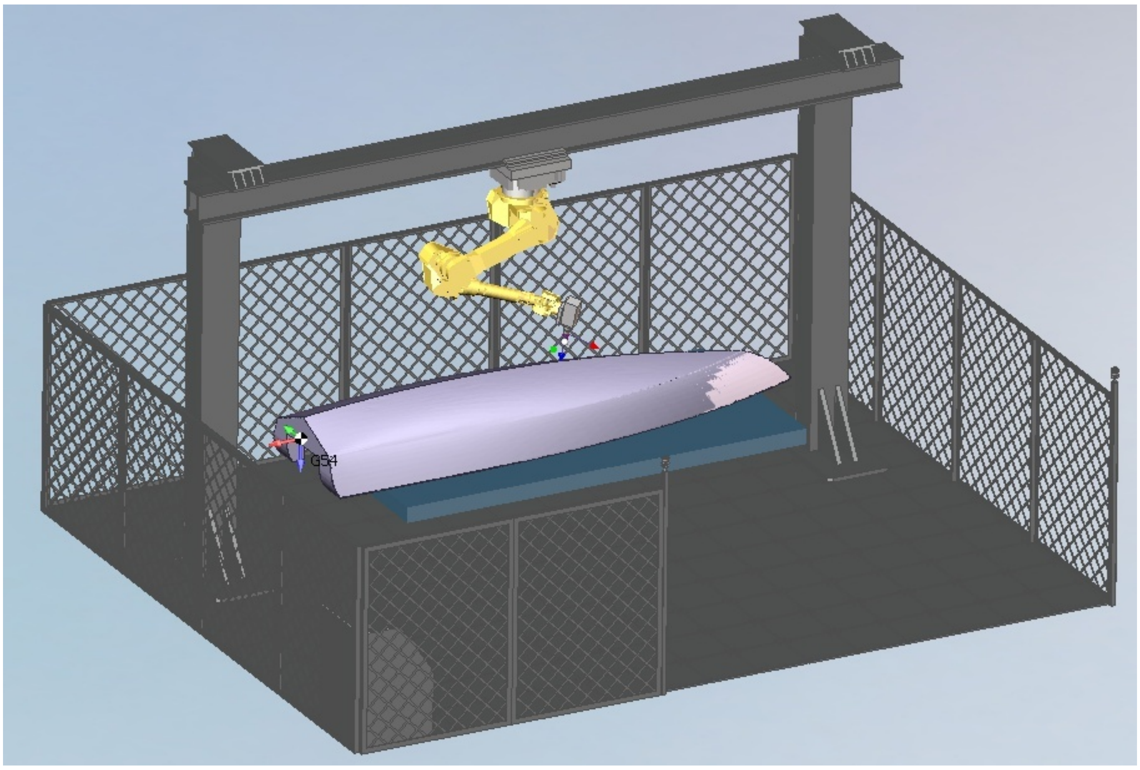


Optionally you can also define the **Tangent approximation tolerance** and the **constant angular deviation** relative to the tangent of toolpath at each point.



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.

5.11.3 Programming the rails position

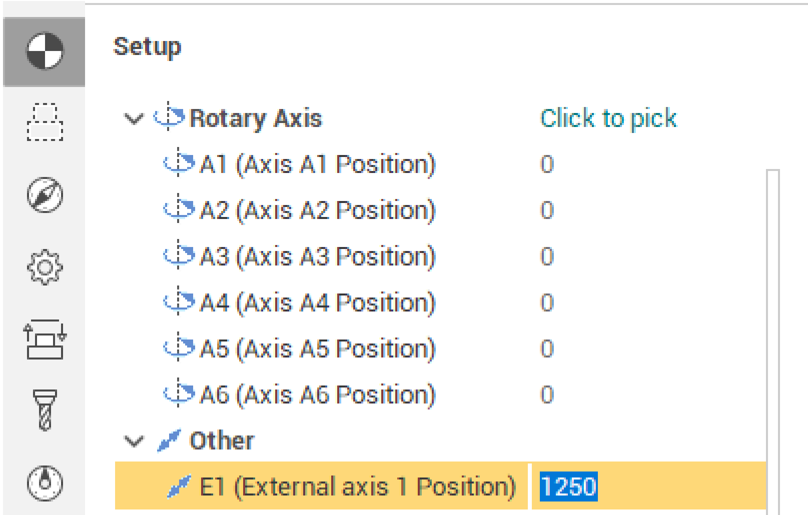


Often to increase the reach zone of an industrial robot, the robot is mounted on rails. SprutCAM X allows simultaneous control of up to 3 rail positioners.

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)



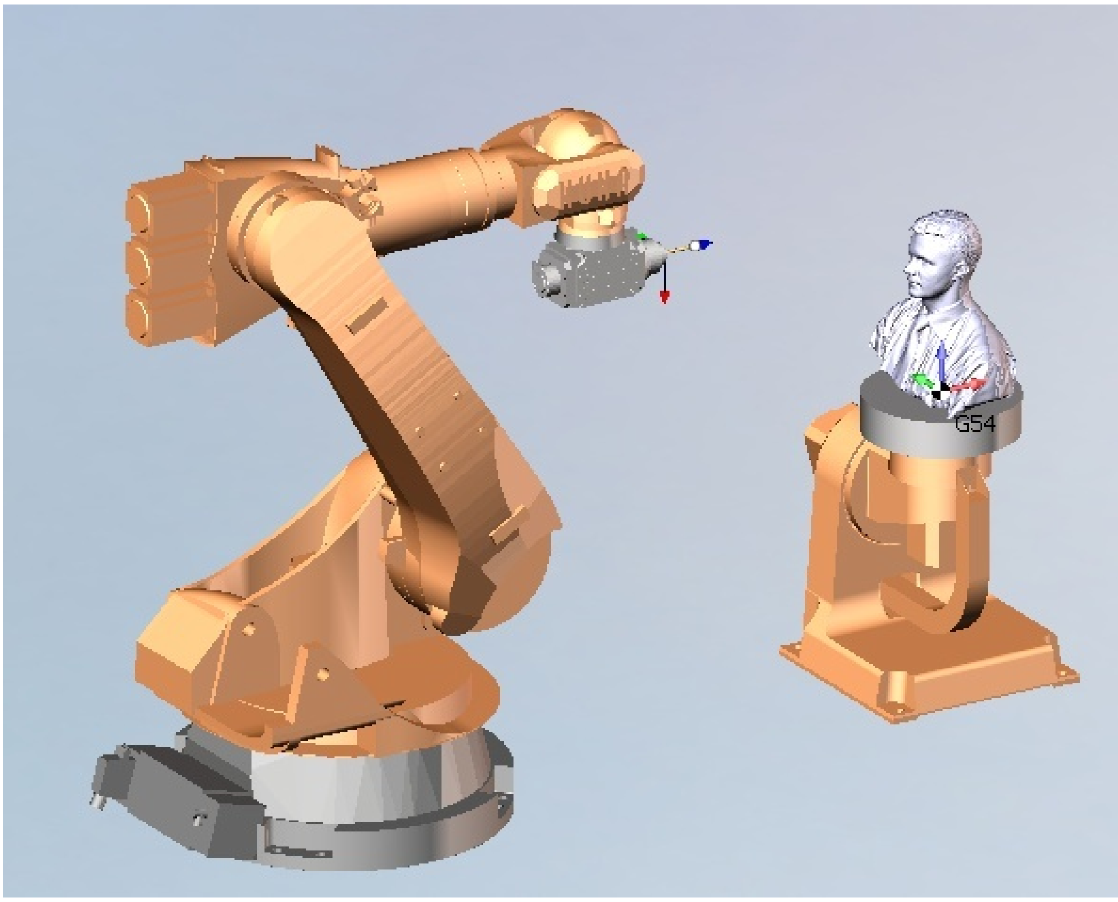
Move Up <the rail axis>

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).

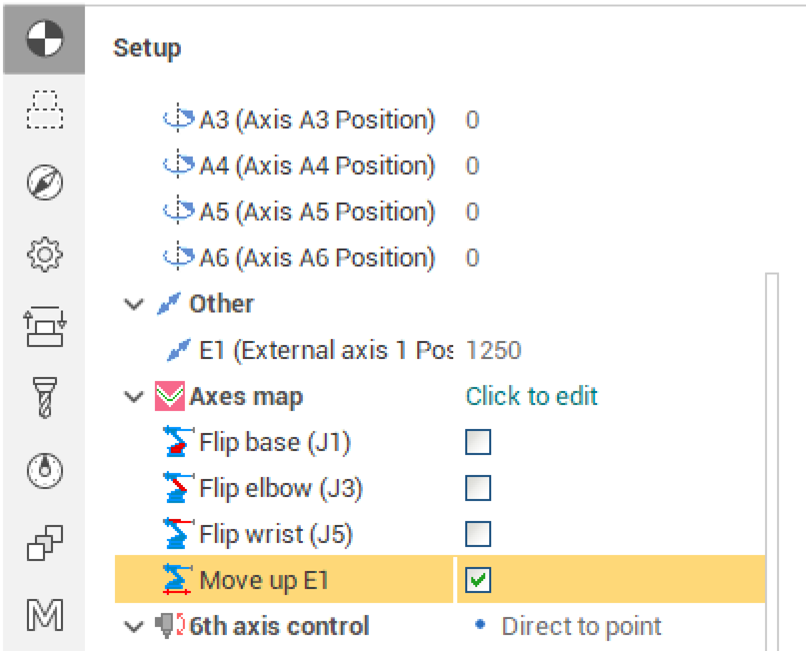
5.11.4 Programming the rotary table



A robotic cell can be equipped with a rotary or a tilting rotary table which have one and two additional rotary axes respectively.

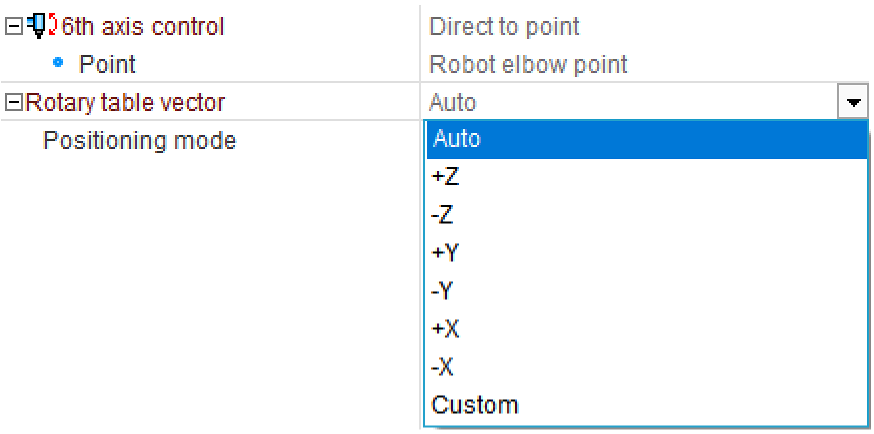
If the kinematic scheme of a robotic cell contains a rotary table, the following parameters become available in the operation setup.

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.

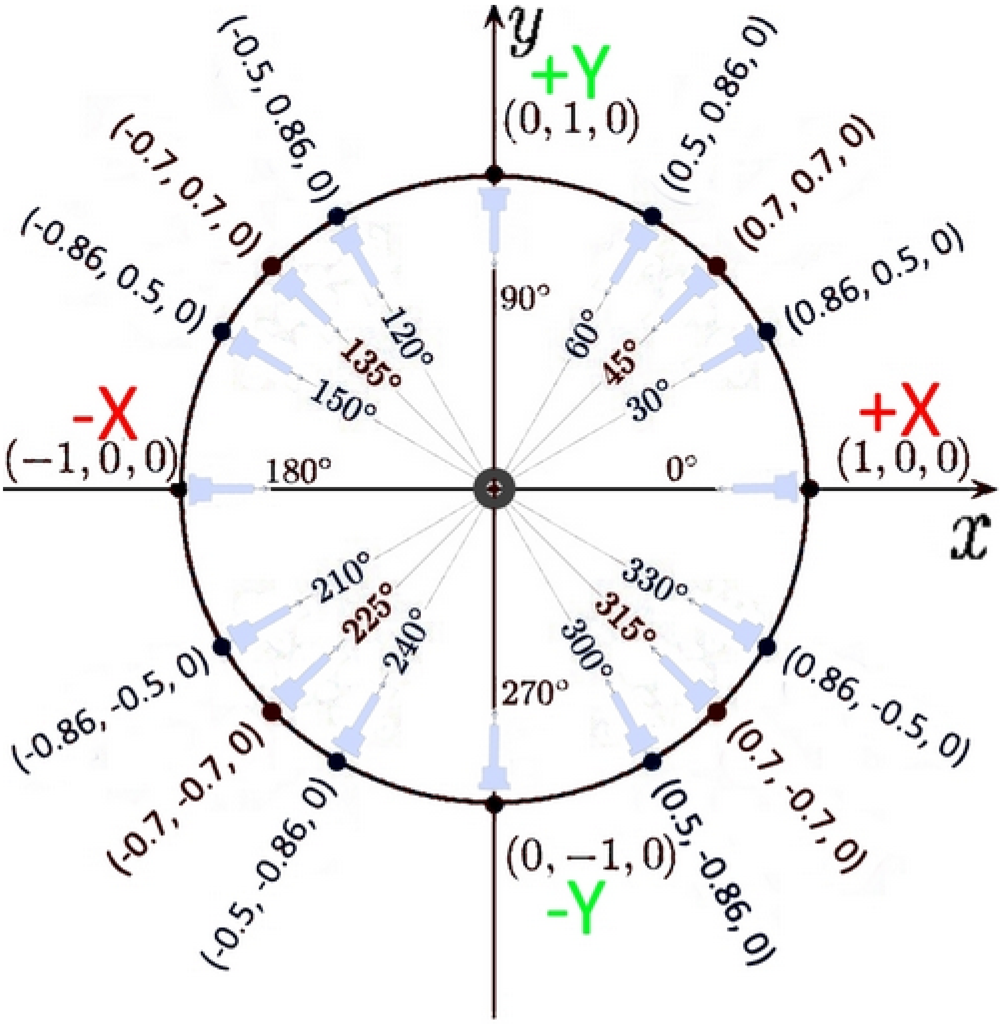


Rotate table parameters

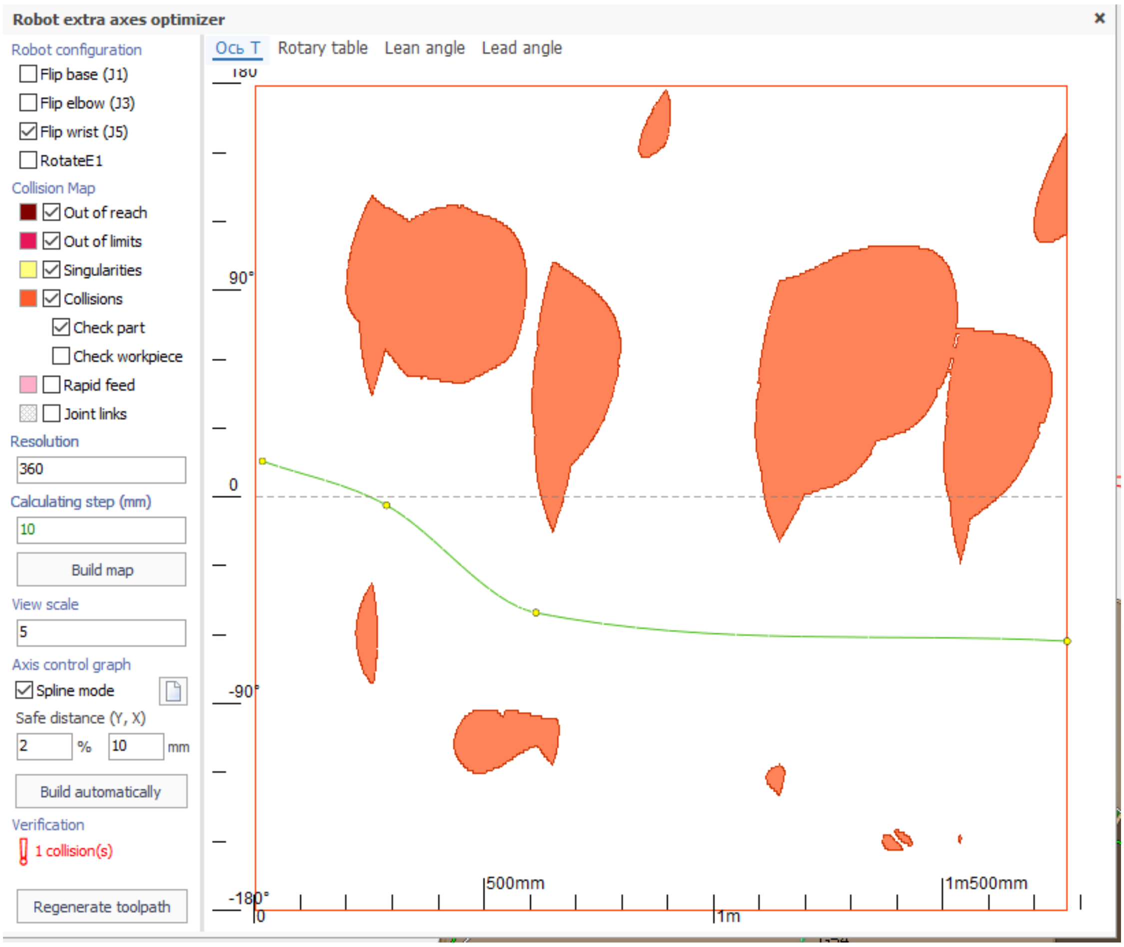
The automatic table rotation feature allows to rotate table during the machining in such a way that the tool always stays on one side of the table. By using the <Rotary table vector> you can set on which side of the table the tool will be positioned (the direction vector of the tool axis to the center of the table in the global coordinate system of the robot).



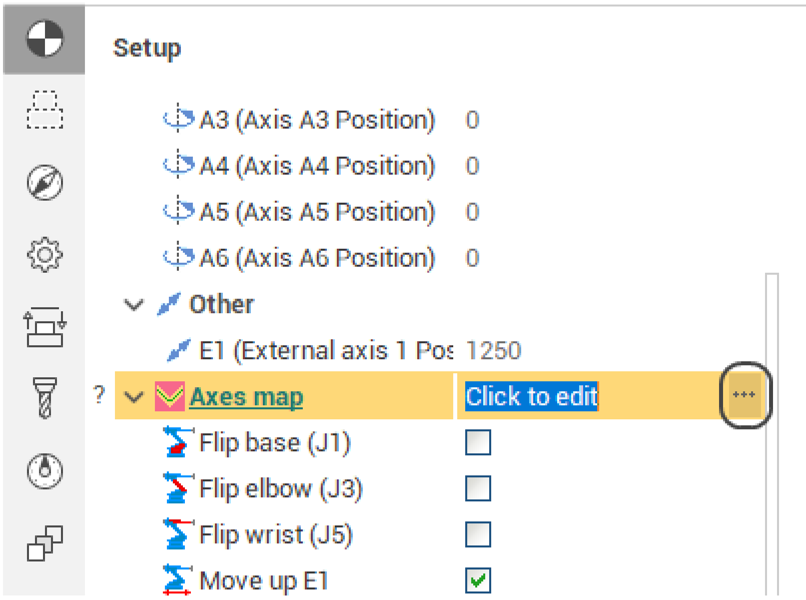
In the diagram below the examples of a vectors for different angles are shown.


y
+Y
(0, 1, 0)
(0.5, 0.86, 0)
(0.7, 0.7, 0)
(0.86, 0.5, 0)
+X
(1, 0, 0)
x
(0.86, -0.5, 0)
(0.7, -0.7, 0)
(0.5, -0.86, 0)
(0, −1, 0)
-Y
(-0.5, -0.86, 0)
(-0.7, -0.7, 0)
(-0.86, -0.5, 0)
-X
(−1, 0, 0)
(-0.86, 0.5, 0)
(-0.7, 0.7, 0)
(-0.5, 0.86, 0)
90°
60°
45°
30°
0°
330°
315°
300°
270°
240°
225°
210°
180°
150°
135°
120°

5.11.5 Robot axes map



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 feature is available in the following SprutCAM X configurations:

- Robots
- Master
- Pro

Remark: need additional "Advanced robots" module licence.

5.11.5.1 Available for optimizing axes/parameters

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

5.11.5.2 **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.

5.11.5.3 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.

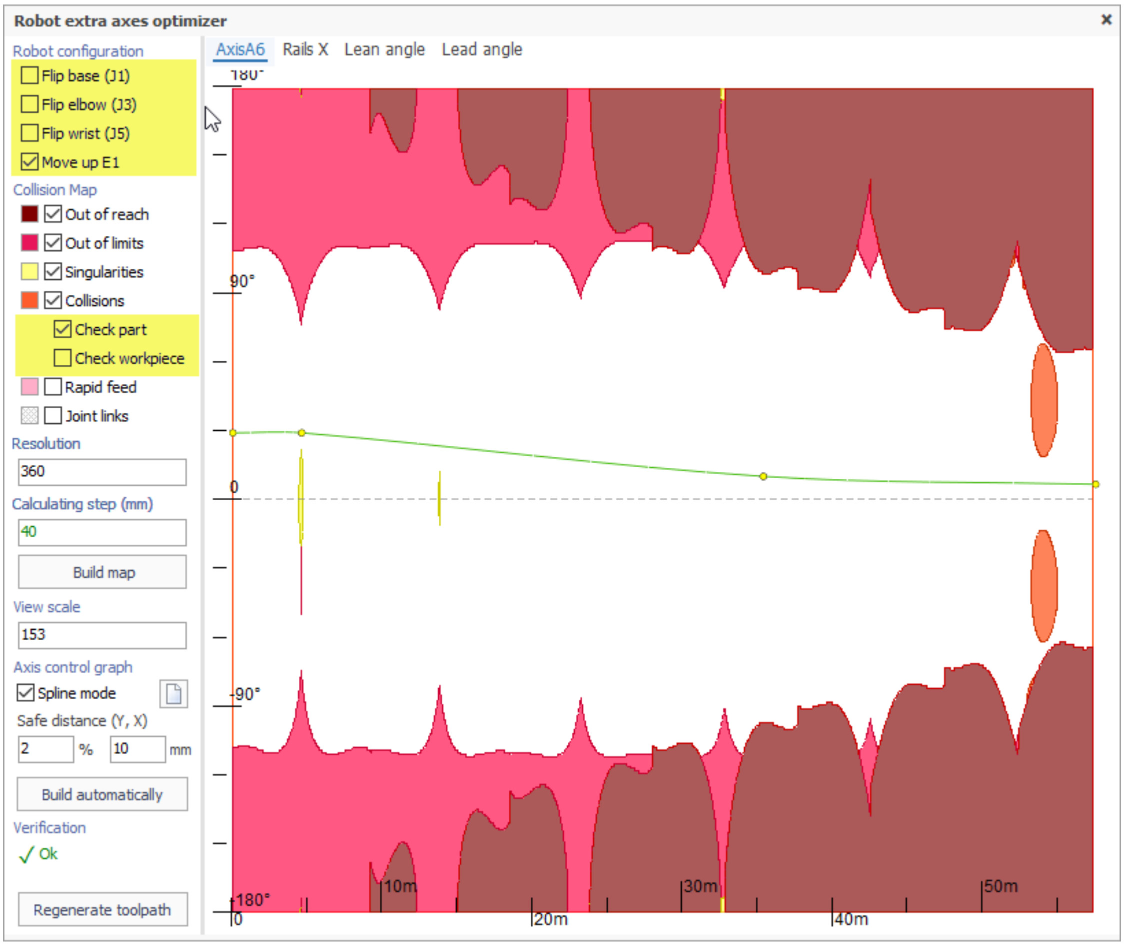
1. Out of reach zones (maroon) are zones which the robot can not reach.
2. Out of limits zones (purple) are zones which the robot can reach but some of the robot's axes go out of its' defined limits.
3. Singularity zones (yellow) are zones where the robot's joints move at extreme speeds.
4. Collision 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.

5.11.5.4 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.



5.11.5.5 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.

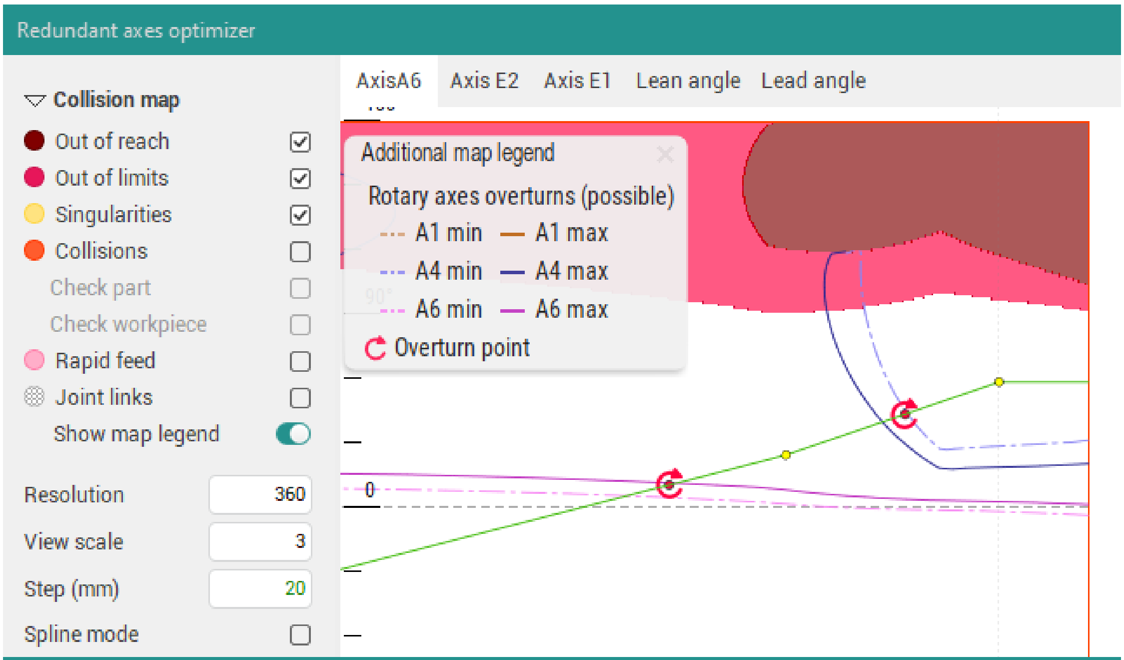
5.11.5.6 Rotary axes overturns detection and avoidance

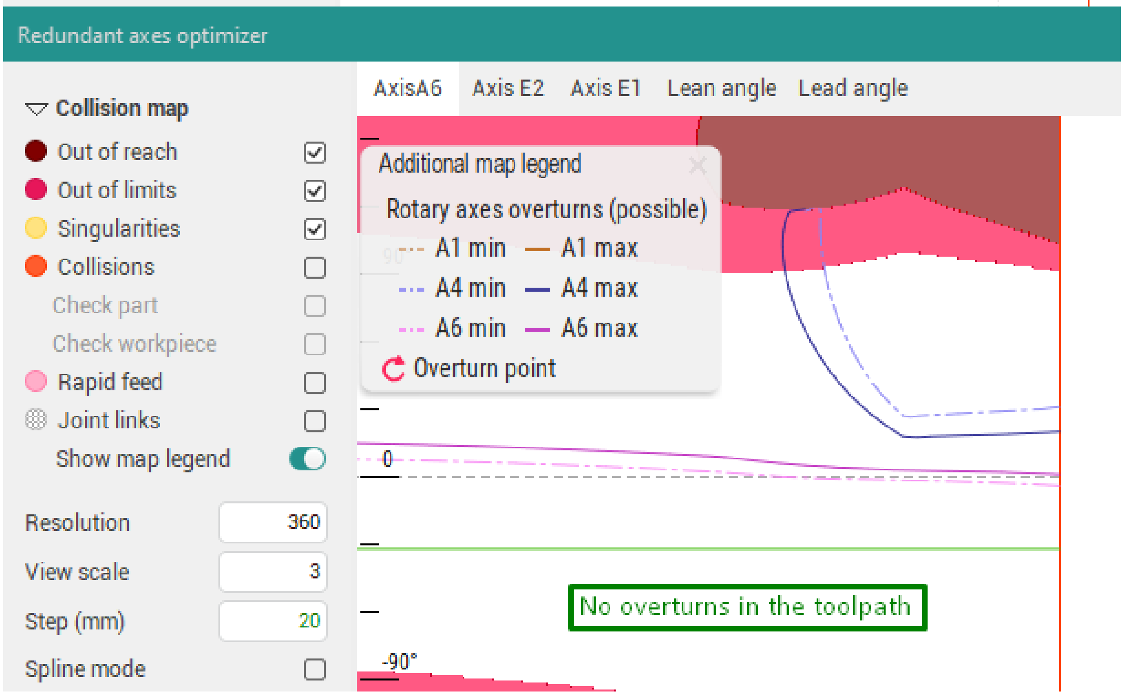
It is possible to use the redundant axis map to check for the potential problems in the toolpath caused by the rotary axes **overturns**. Overturns happen if the rotary axis reaches one of its limits and in order to continue machining it needs to do one full rotation (360°) forward or backward. In previous 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





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.

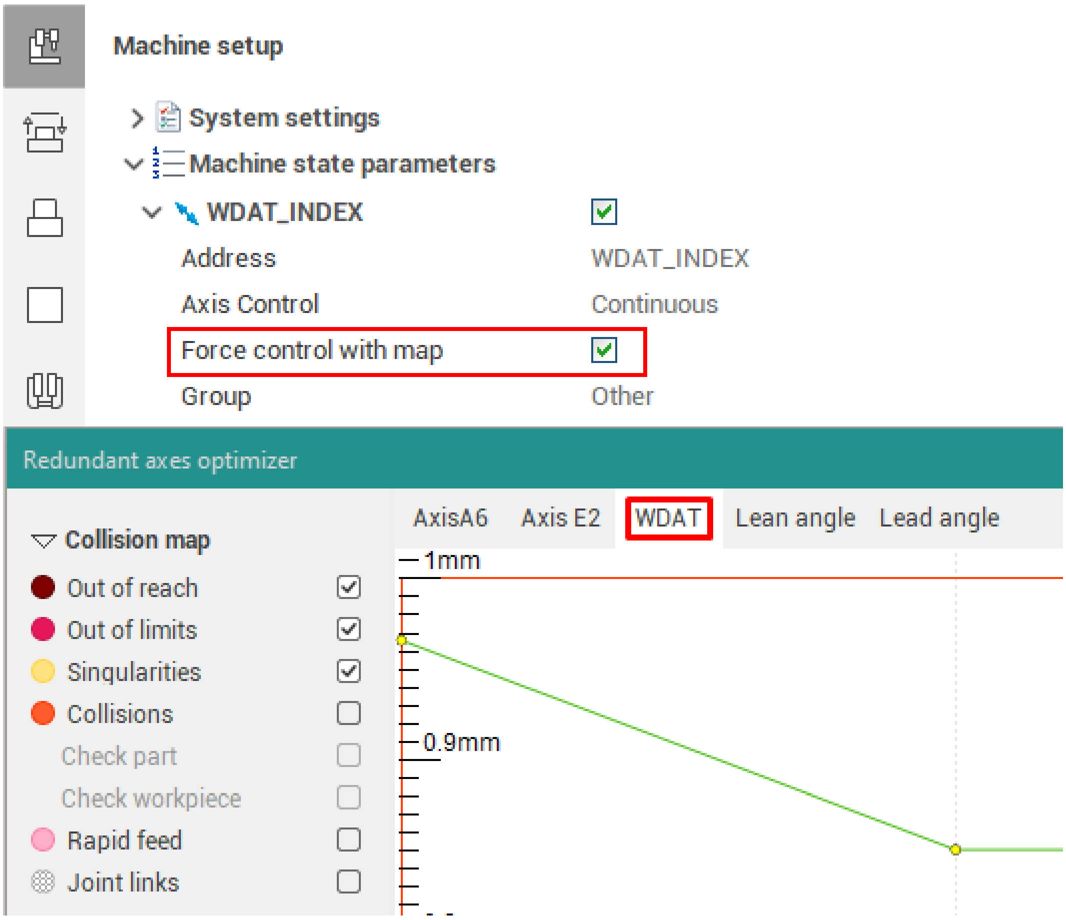
If the operation's toolpath contains overturns, you can try to **avoid** them by moving the spline so it doesn't intersect the given lines or the intersections are "fake" (the rotary axis value did not reach its limit yet in this point, but its value is equal to the axis min/max modulo 360°).

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.

5.11.5.7 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.

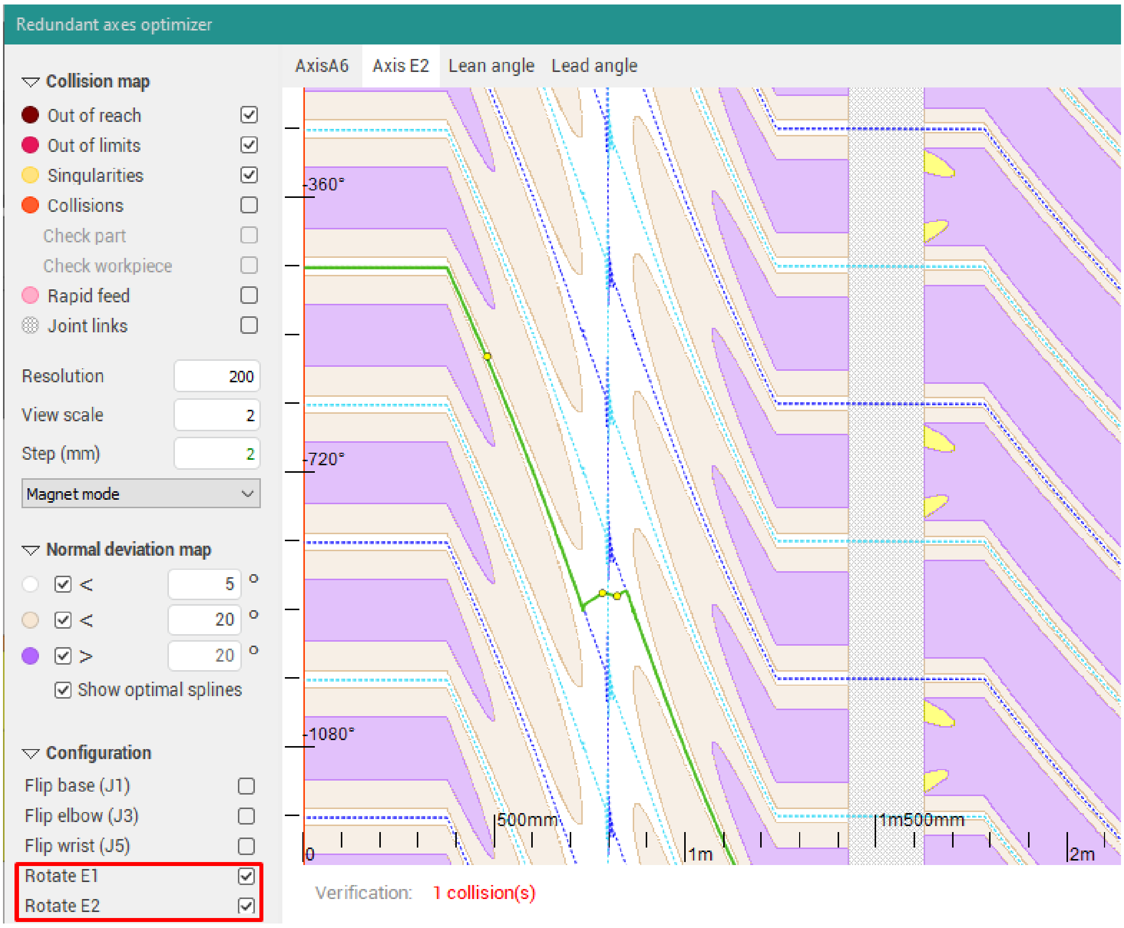
Enabling axis control with the map in xml

```
<SCType ID="WDATPOS" Caption="WDAT_INDEX" type="TMachineStateParameter">
        ...
        <ControlWithMap DefaultValue="True"/>
</SCType>
```

5.11.5.8 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.



5.11.5.9 Availability of the feature

The Robot Axes Map is available for robots as a part of the additional 'Robot +' module for SprutCAM X's configuration:

By default:

- Robot

As option:

- 5x Mill*
- Master*
- Pro*

* - the additional 'Robot +' license for support robots is needed.

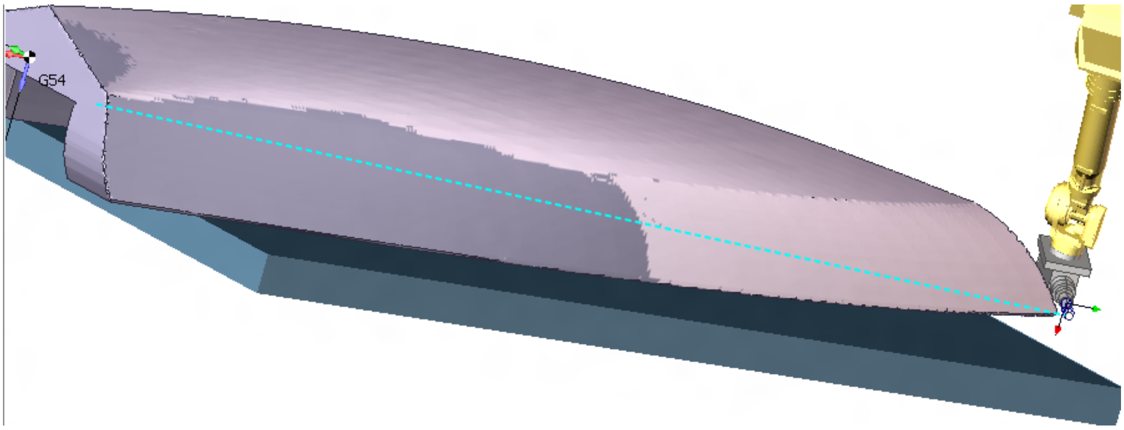
See also:

Axes map for 5-axis machines

5.11.6 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.

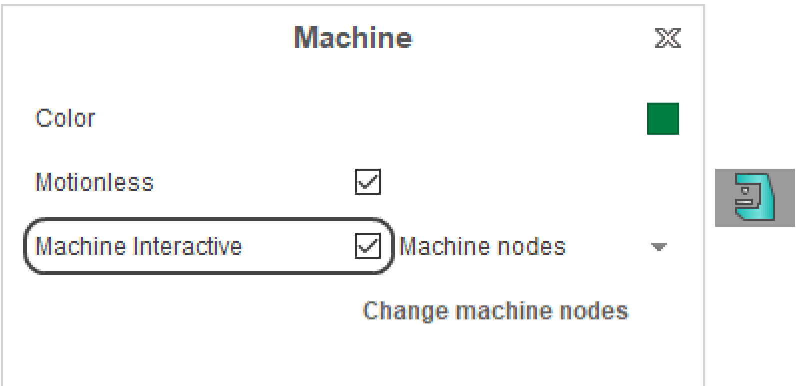
Let's take a look at the following example.



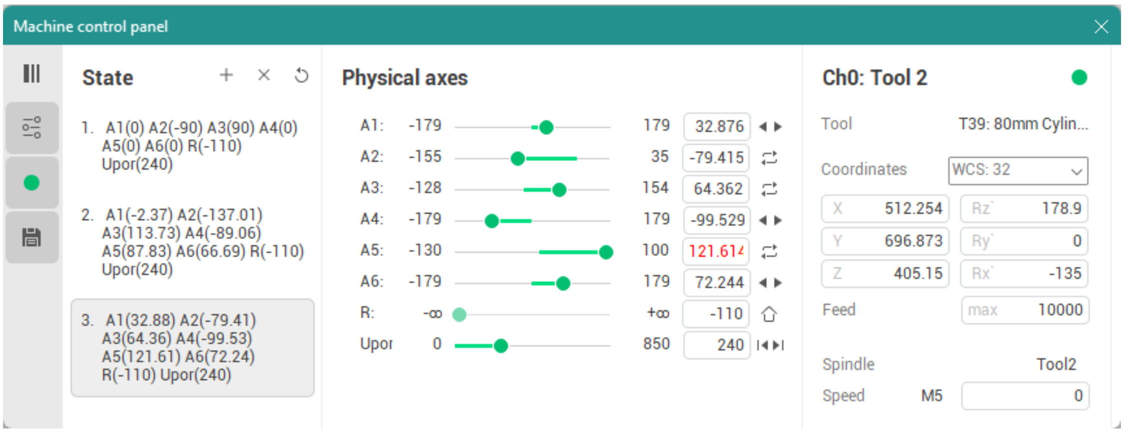
After the operations had been calculated, the simulation has revealed that the transition between two operations gouges the part. To modify the transition the list of intermediate points the robot will pass through should be formed.

The best way of doing it is by using the Machine control panel (click the  button at the main panel).

But first, make sure the <Interactive machine> mode is enabled.



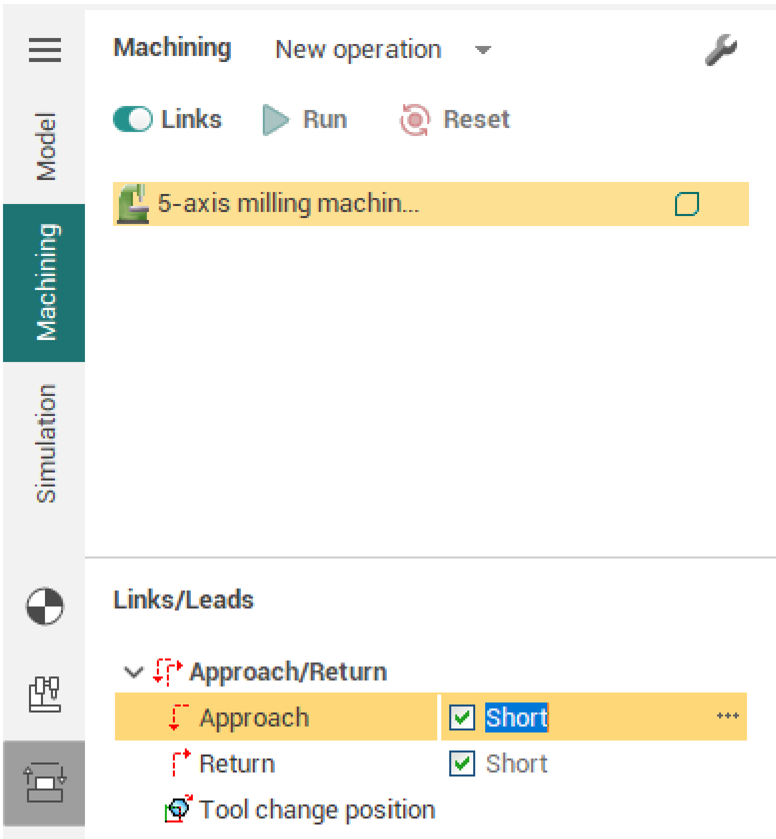
At the <Simulation> tab select the last command of the first operation (5D Contouring) as shown in the picture below. To define the first control point drag the tool to the desired position with the mouse.



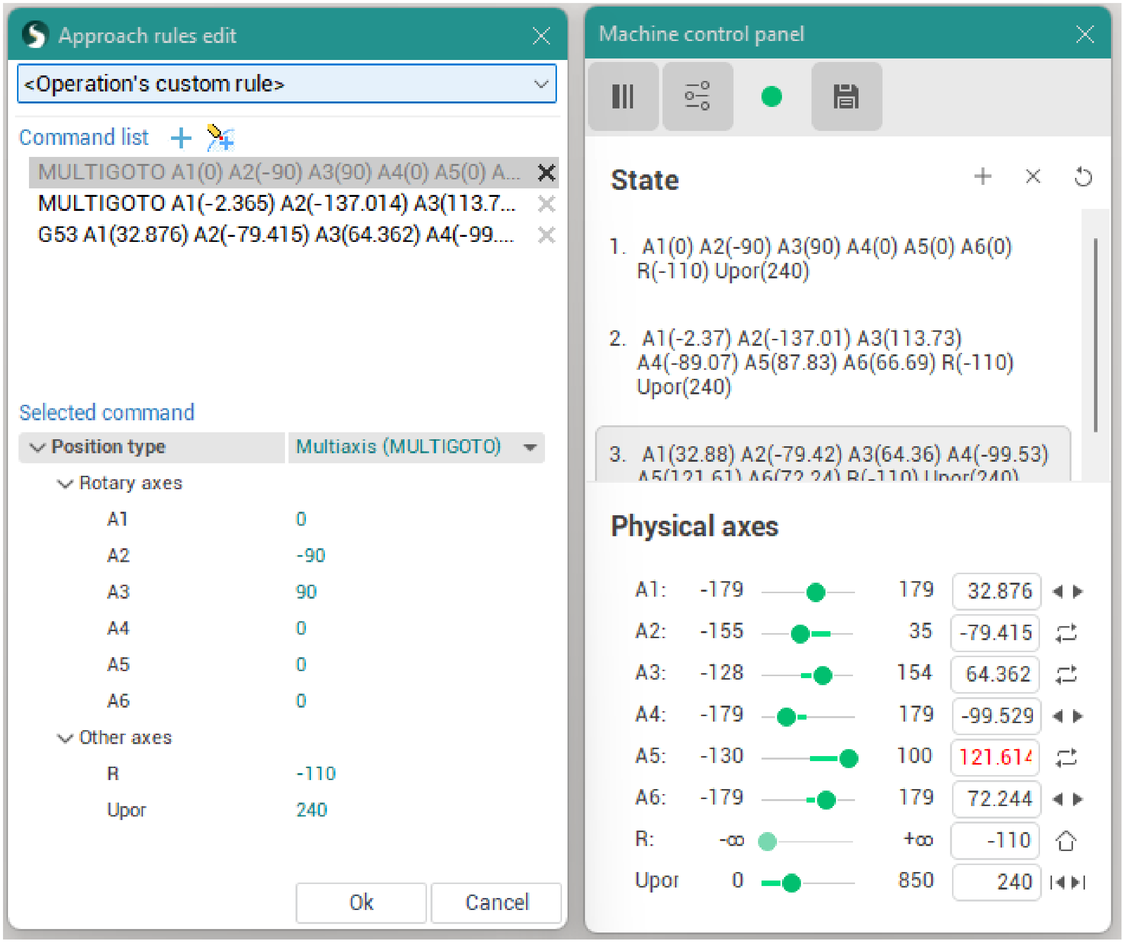
Then hit the <**States panel visibility**> button in the <Machine control panel> to save the position into the memory.

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.



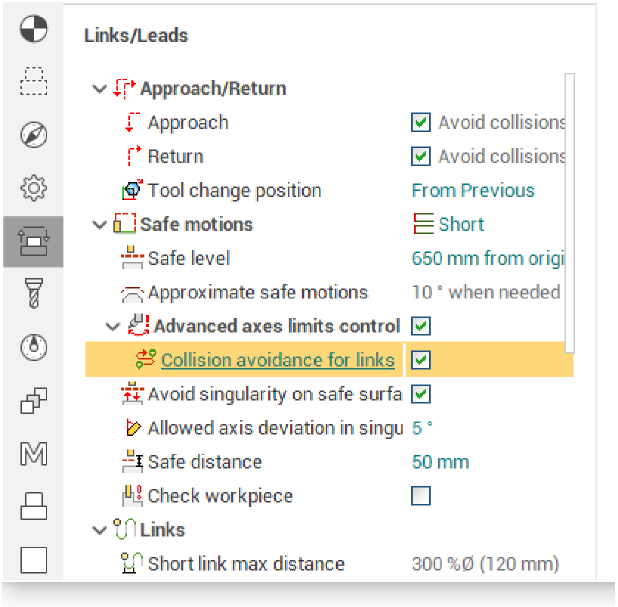
If exact values of axes are not important then control points can be added directly without formation of the list.

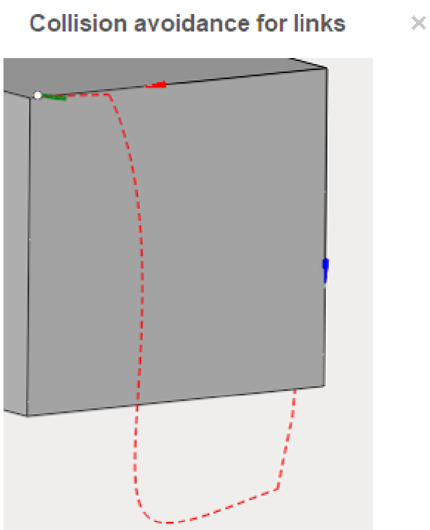
Similarly the approach sequence of the second operation (5d Contouring2) can be defined.

5.11.6.1 The planning of the links

SprutCAM X 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 "**Collision avoidance for links**" 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.





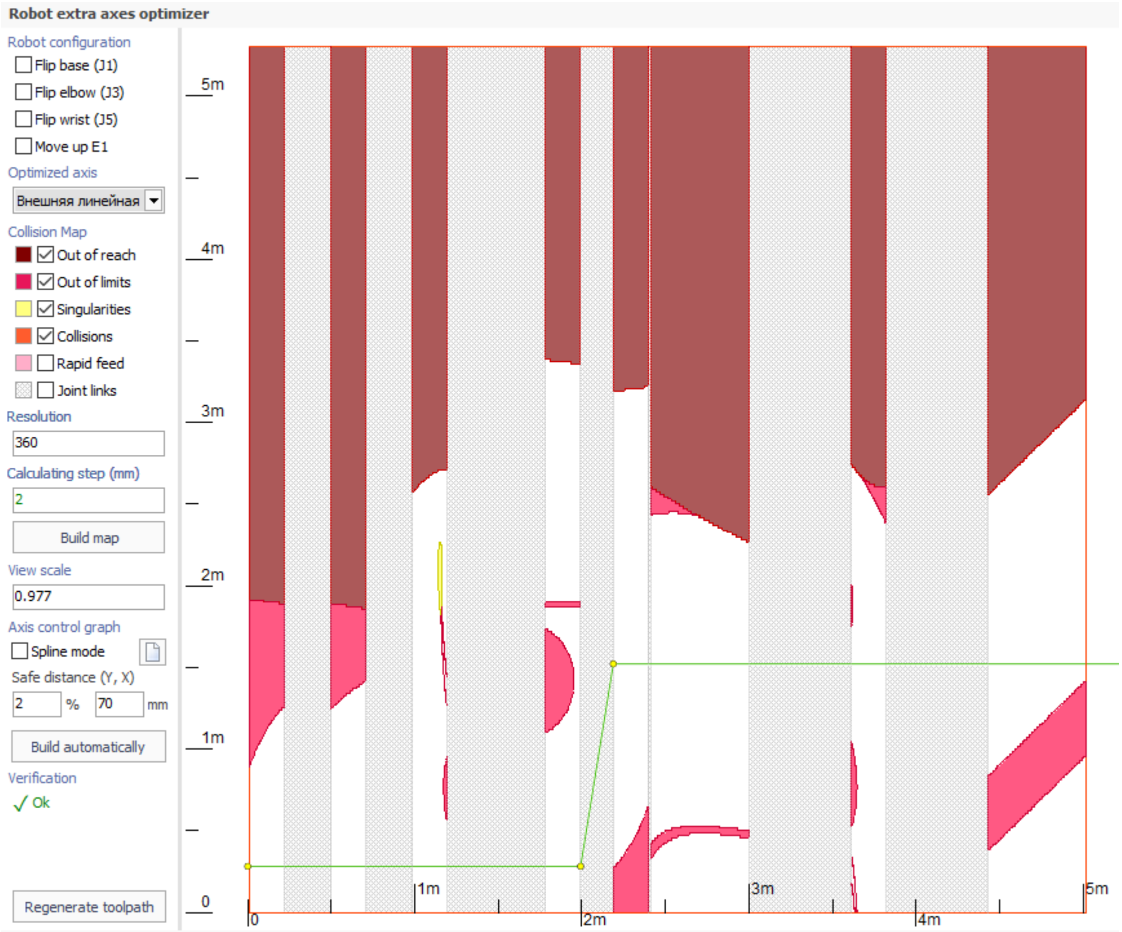
Supported operations

This option is available currently only for robots in the following operations (and their descendant operations, if not intentionally disabled):

- Morph operation
- Rotary operation
- 5d surfacing operation
- 5D contour and 6D contour operations
- Welding 5D and 6D operations
- 5D Meshing operation

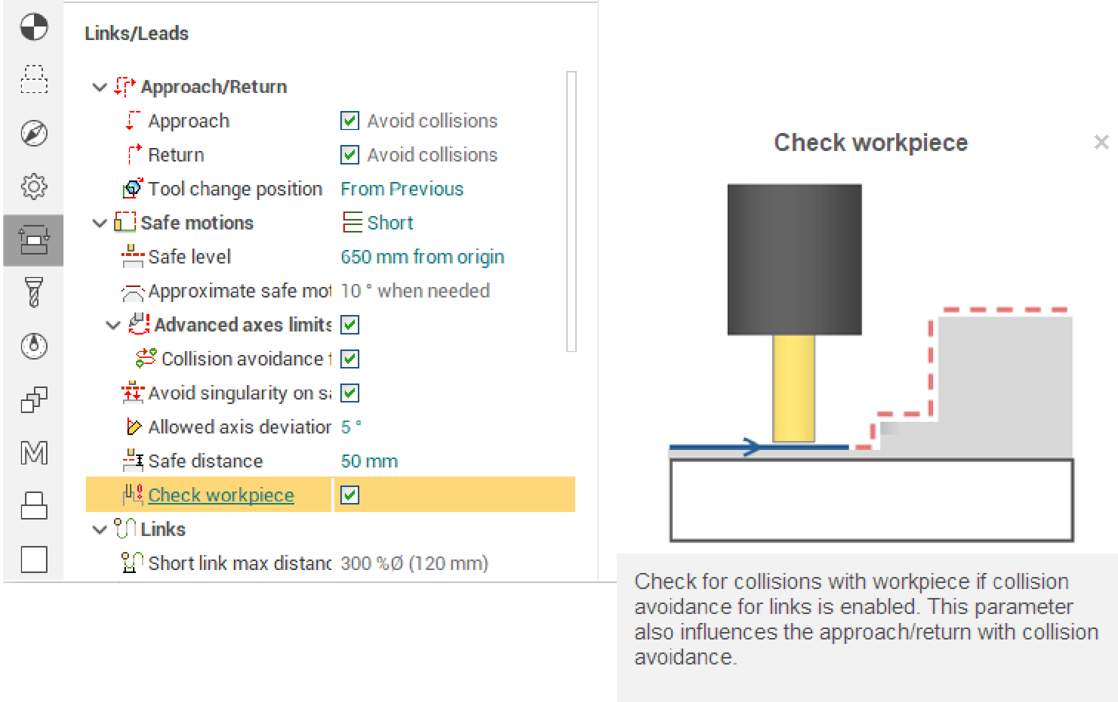
Link start and end points

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.



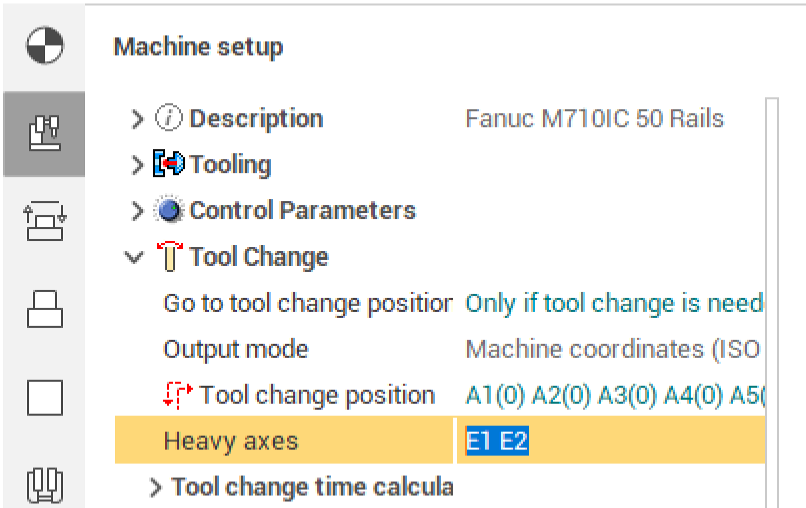
Workpiece collisions

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.



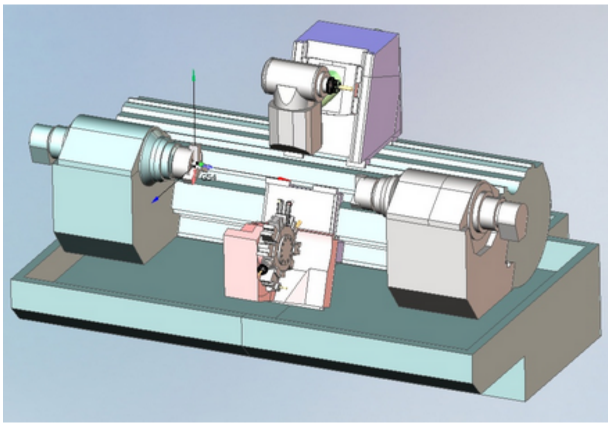
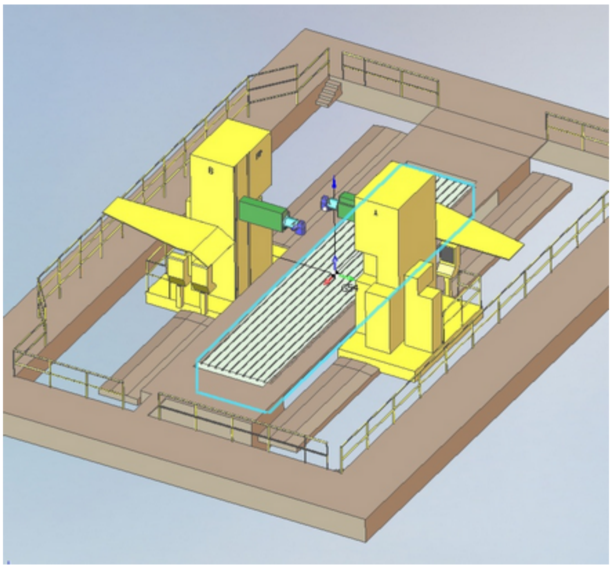
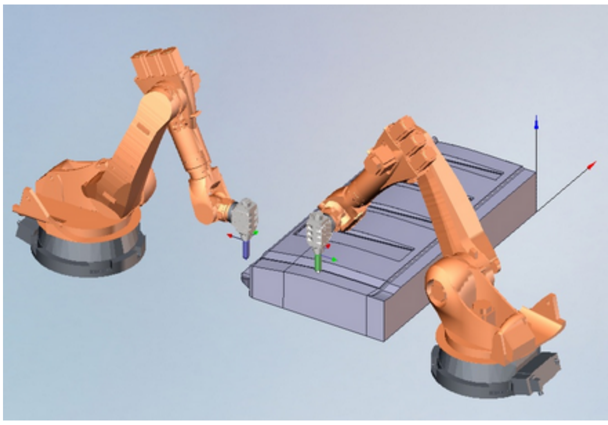
Exclude axes movement

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.



5.12 Multi Task Machining

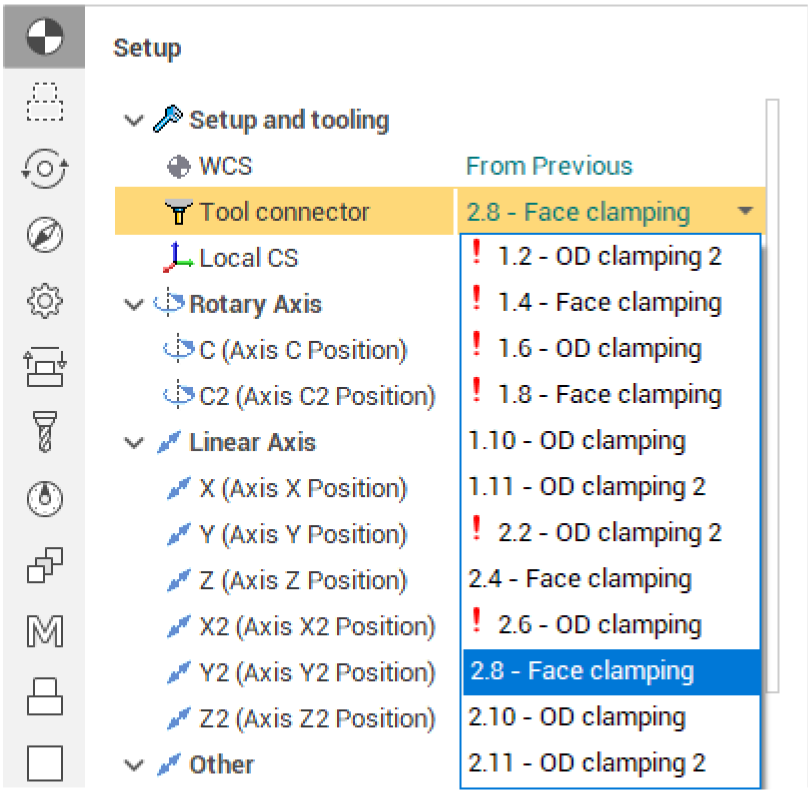
Multitask machines (MTM) allow to work by few tools simultaneously. MTM can be very different. For example, a multi-task turn-milling center has few spindles or/and turret heads inside one housing. Sometimes a multi-task machine is two or more identical machines that work under one workpiece.

multi-task turn-milling center	
multi-task milling machine	
two robots, working together	

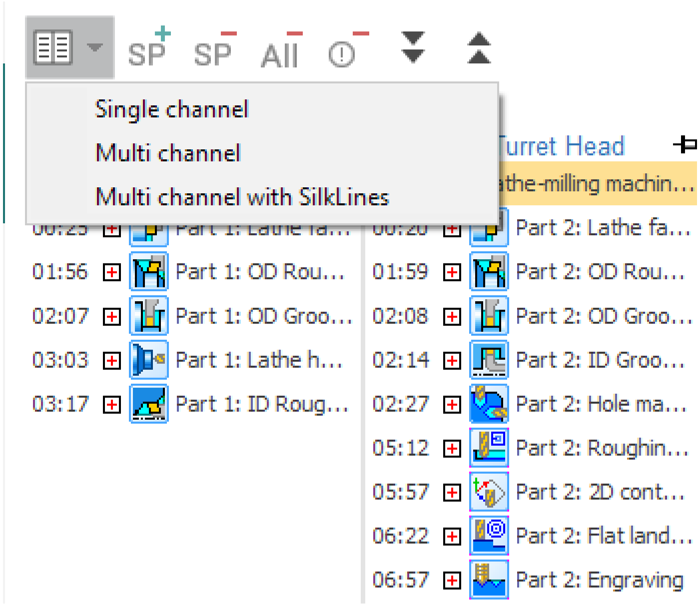
With multitask machining every tool is controlled by it's own program. This program is called as channel of control. So the multitask machine has more than one channel. Sometime the task of program of multitask machine can be presented as a creation of the separate unrelated projects for every channel. 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.

SprutCAM X 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 SprutCAM X. 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.

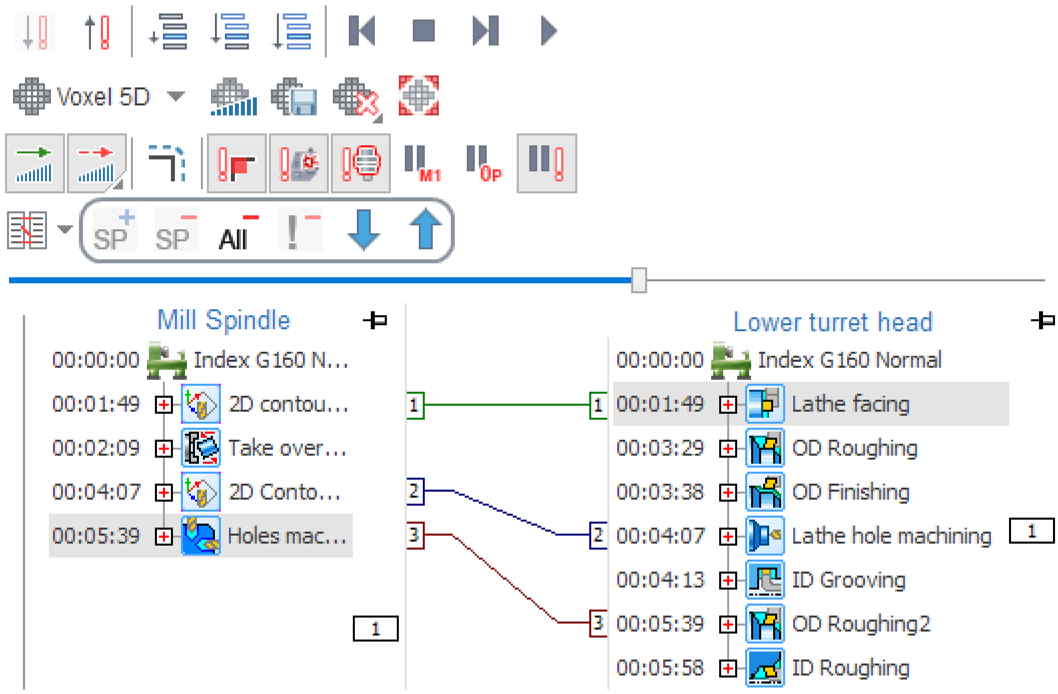
In **multichannel** mode the simulation is executed simultaneously in every channel like it will be executed on the real equipment. It this mode it's comfortable to check for the collisions with the taking care about motions in all channels. Detected collisions can be eliminated by the adding of sync points.

Multichannel mode with Silk lines is comfortable to add, remove and edit the sync points.

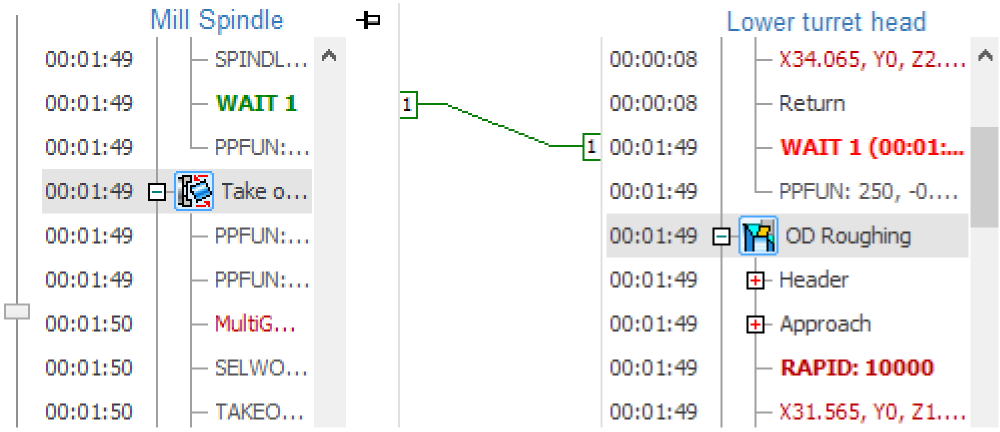
Sync points

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 SprutCAM X.



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

5.12.1 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

```
<SubMachines>
        <SCArray>
                <SubMachine>
                        <!--Main spindle-->
                        <ToolNode>AxisX</ToolNode>
                        <WrkNode>MainSpindle</WrkNode>
                        <XAxisID>AxisX</XAxisID>
                        <YAxisID>AxisY</YAxisID>
                        <ZAxisID>AxisZ</ZAxisID>
                        <ToolAxisID>AxisT</ToolAxisID>
                        <OriginG54BaseNode>Schema</OriginG54BaseNode>
                        <OriginG54>
                                <Rotation>
                                        <Convention>FixedXYZ</Convention>
                                        <R1>180</R1>
                                        <R2>0</R2>
                                        <R3>0</R3>
                                </Rotation>
                        </OriginG54>
                        <ApproachRule>Z(10);C;X;Z;</ApproachRule>
                        <ReturnRule>Z(10);X;Z</ReturnRule>
                </SubMachine>
                ...
        </SCArray>
</SubMachines>
```

- **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.

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 SprutCAM X.

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

SprutCAM X 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.

5.12.2 Swiss lathes programming

5.12.2.1 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. SprutCAM 16 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 ***..SprutCAM\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.

5.12.2.2 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.

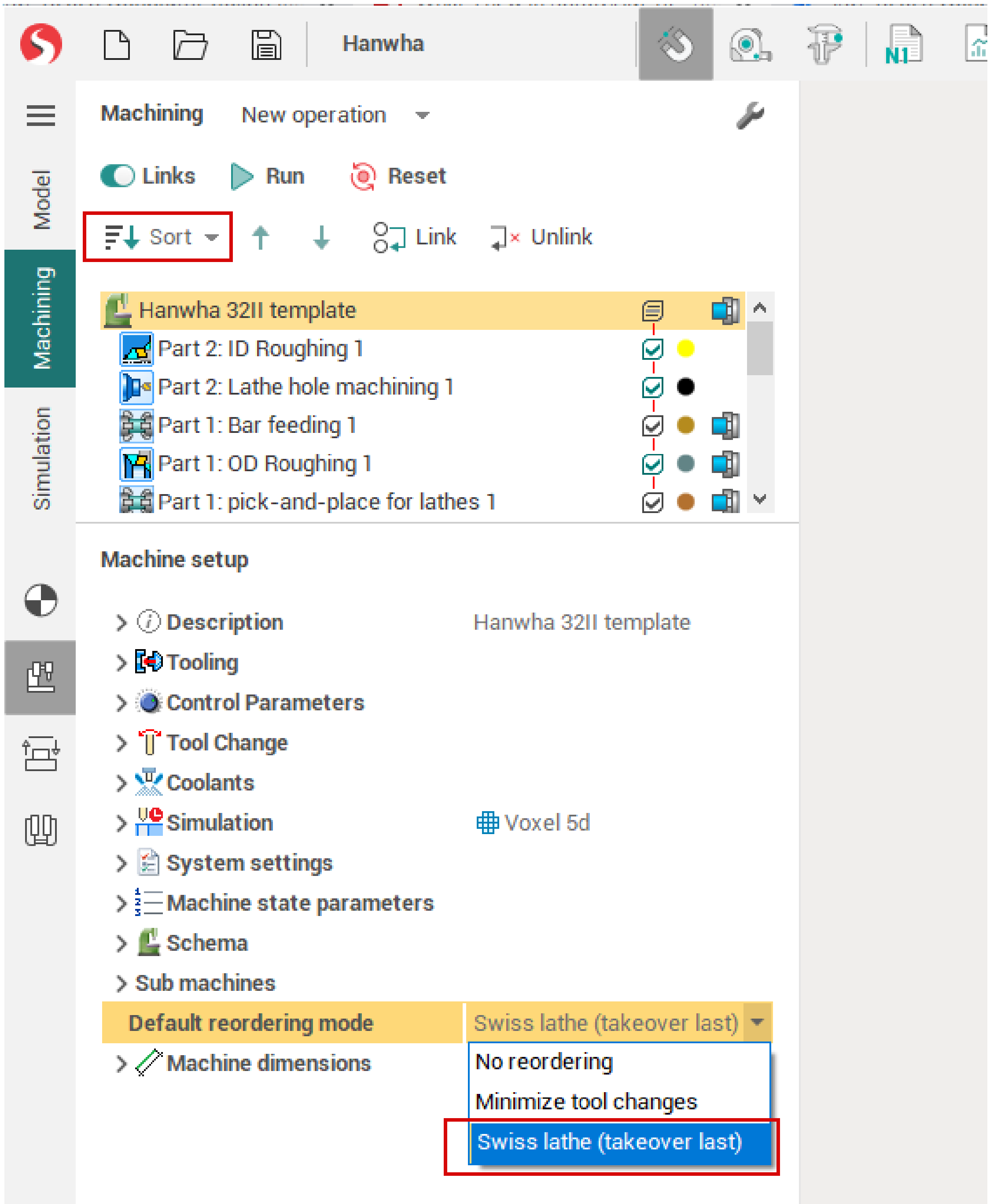
Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

https://www.youtube.com/watch?v=8wCs_BoxfIA

5.12.2.3 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.



See also:

Multi task machining

Multi parts projects

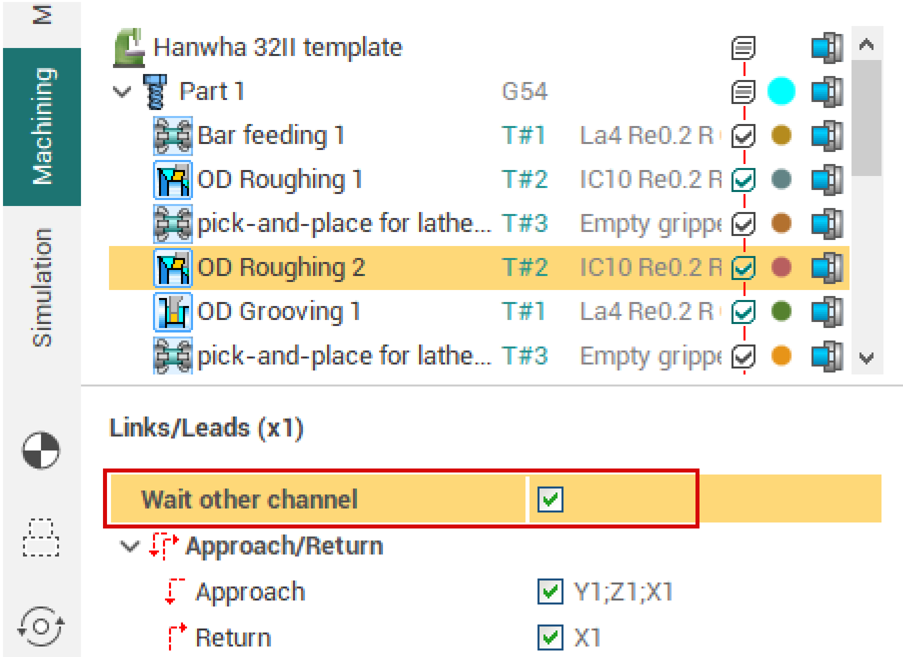
Turn take over

5.12.3 Automatic insertion of wait labels

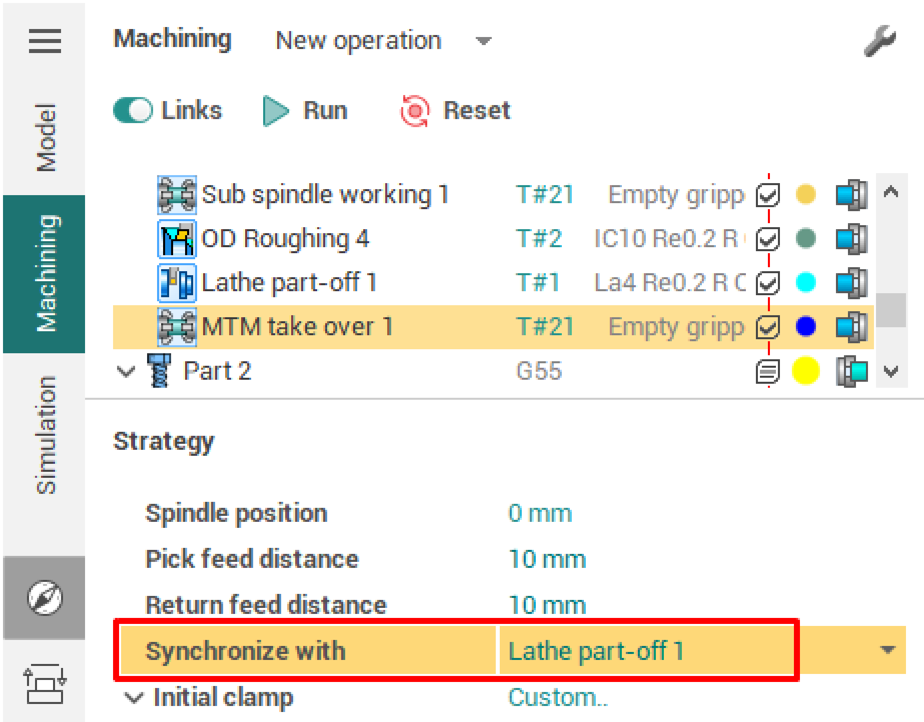
Before the insertion of wait labels, the operations must be reordered in the sequencing mode if the project contains more than one part. See the section of the multipart project for details.

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.

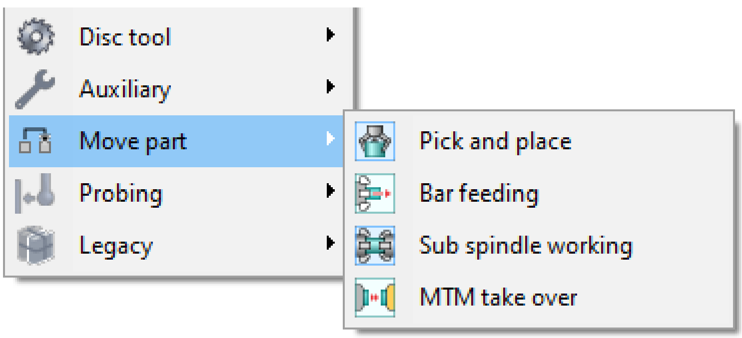


Another method for the automatic insert of the wait labels is using the Turn take over operation. It can be synchronized with the **part-off** operation.



5.13 Move part operations

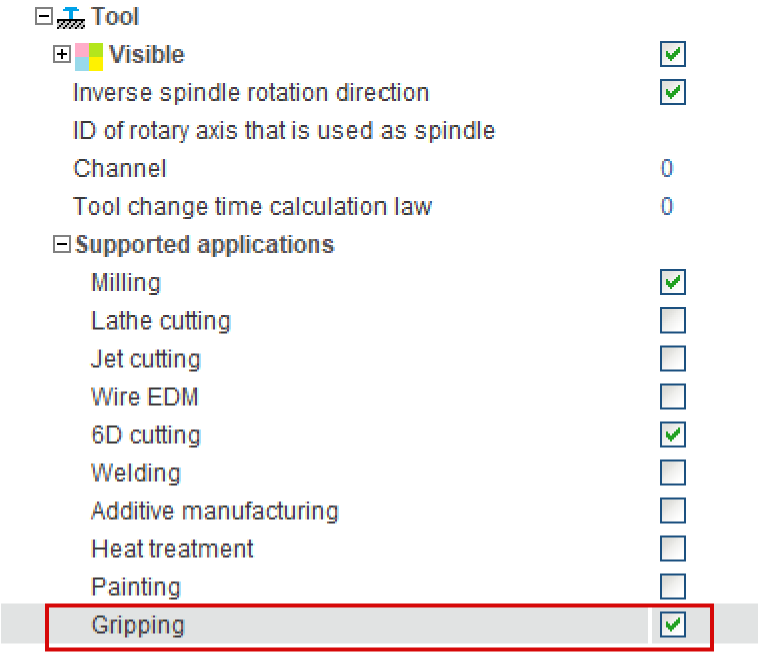
There is the separate group of the operations which can move the part using robot or swiss-type/lathe machine. It comprises **Pick and Place**, **MTM Takeover**, **Bar feeding** and **Sub spindle working** operations.



5.13.1 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.

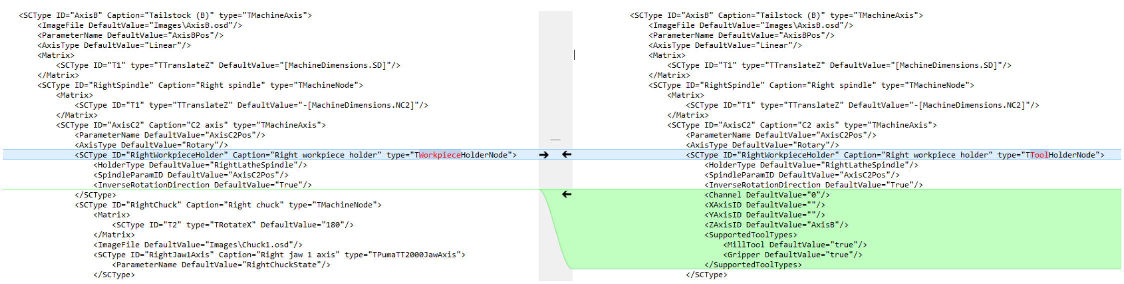


The fragment of the machine schema about the tool holder definiton is shown below. Note that the **Gripper** is enabled inside the **SupportedToolTypes.**

```
<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>
```

5.13.1.1 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 SprutCAM version 14 and earlier, then you need to modify the machine description. Below the differences between the old (left) and adapted (right) schemas is shown.



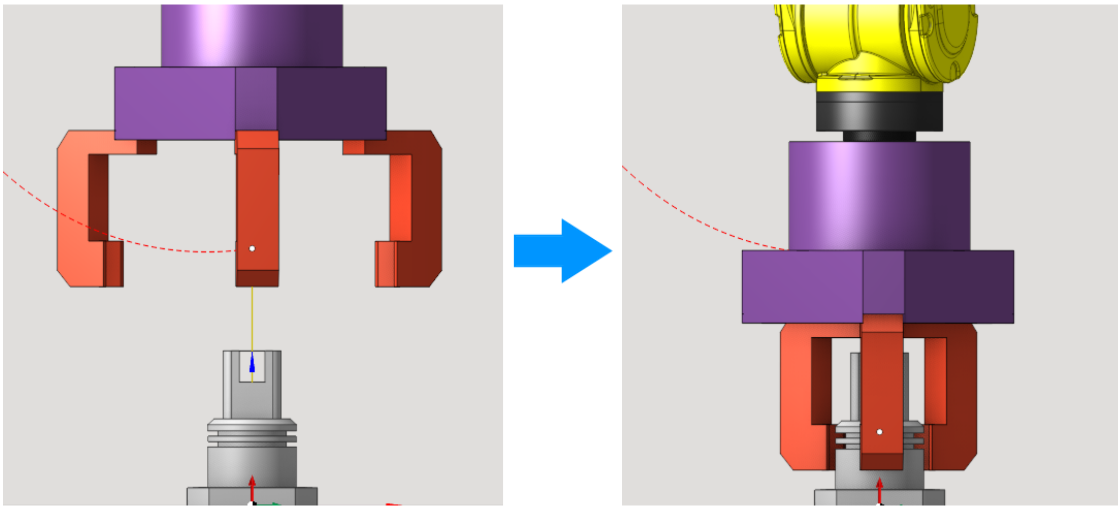
1. In the subspindle definition **TWorkpieceHolderNode** is replaced by **TToolHolderNode**.

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**

5.13.2 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.

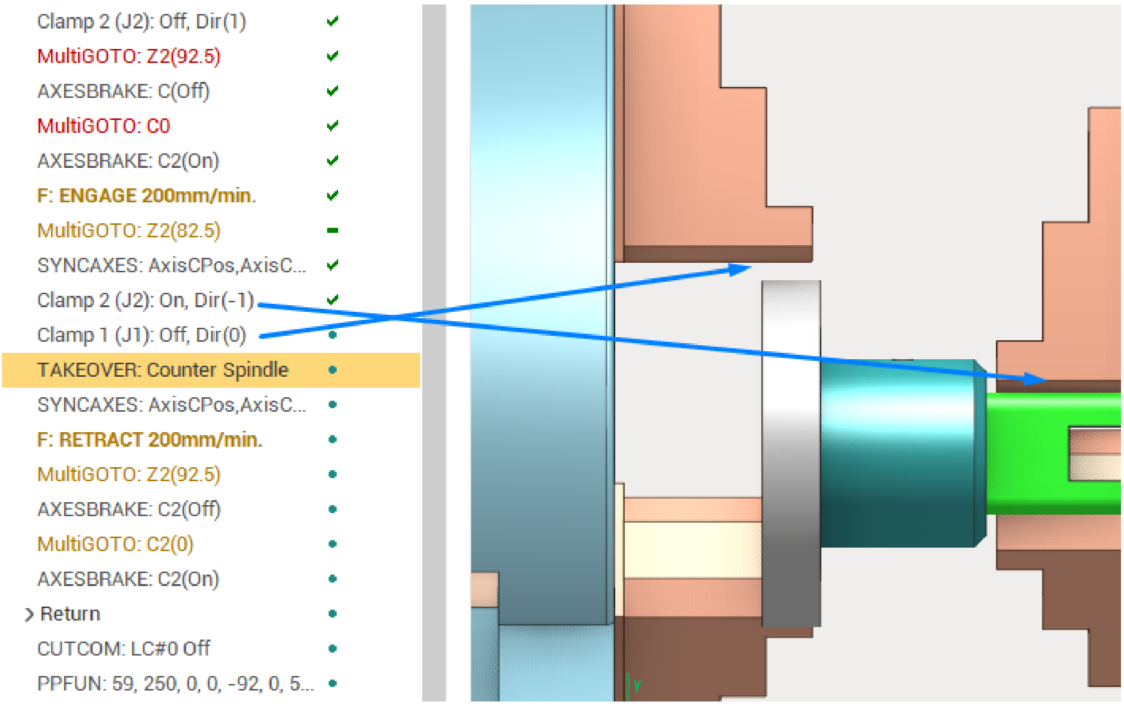


5.13.2.1 New CLData command

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).



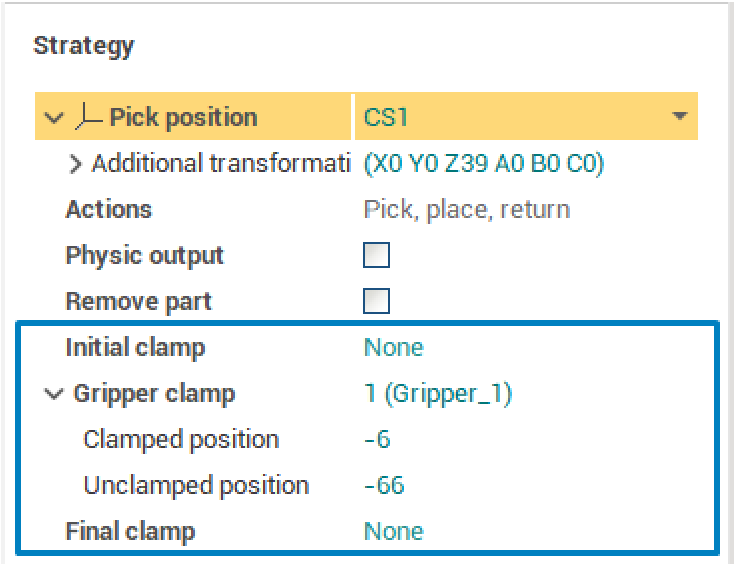
5.13.2.2 Automatic clamp control

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)

5.13.2.3 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.

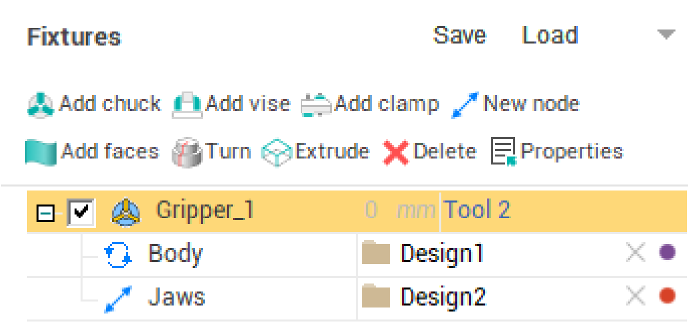
- **Clamp device** combo box allows to select the clamp which is used for the given move part stage. See the section below on how to make the clamp device available for selection in this list. This parameter can also be <**None**>, meaning that no command is outputted. The <**Custom**> enumeration item 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.

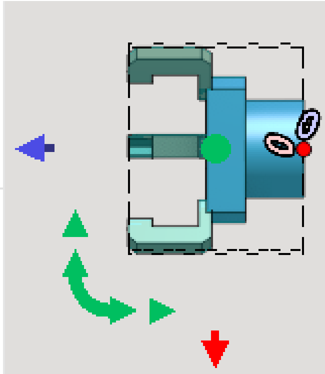
See the distributive projects with the "Move part" operations for an example on how to define the clamp parameters.

5.13.2.4 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.

1. **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 "SprutCAM Machine Maker" documentation.
2. **"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.

5.13.2.5 List of operations that support automatic clamp control

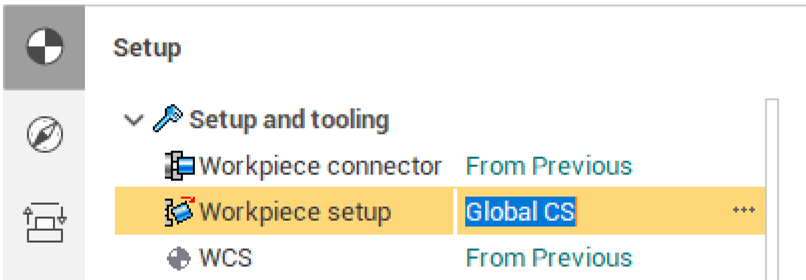
- Pick-and-place
- MTM take over
- Sub spindle working
- Bar feeding

5.13.3 Pick-and-place

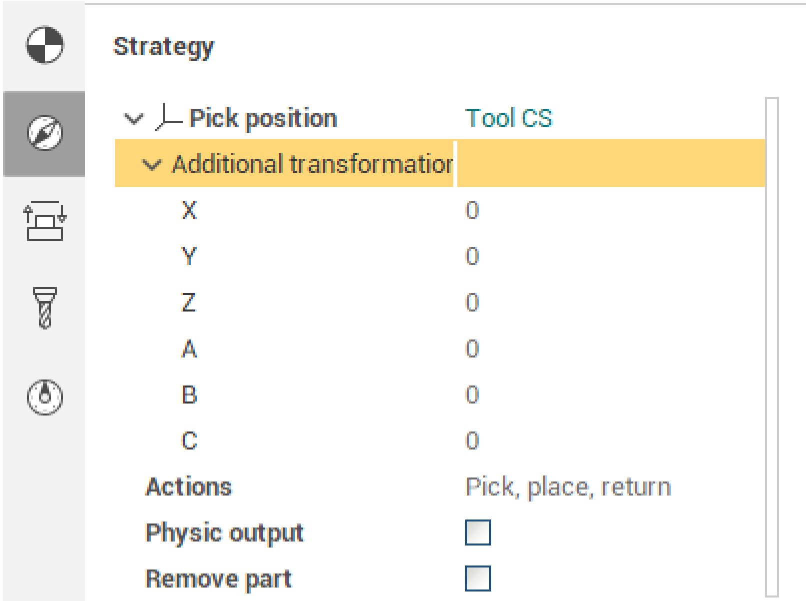
5.13.3.1 Tool path and parameters

"**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**.

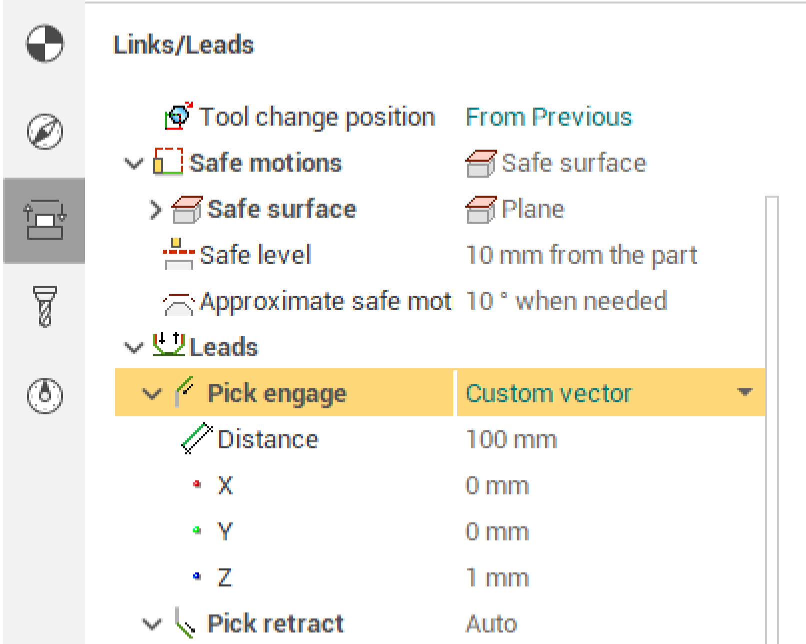


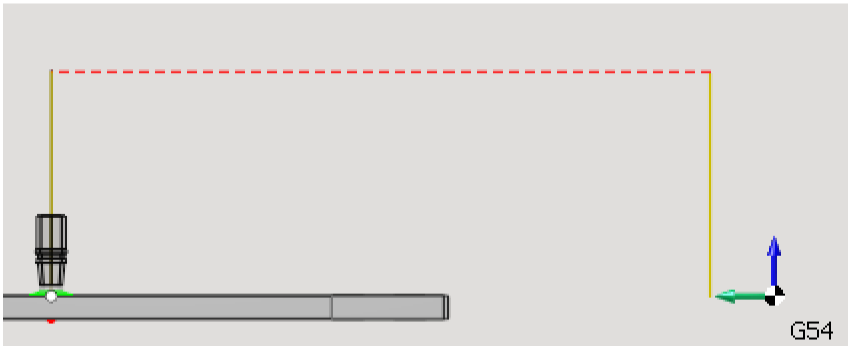
Tool path of pick and place operation has 3 main sections: pick, place, return.



1. **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.
2. **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.
3. **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.

5.13.3.2 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.

5.13.3.3 Creating a Pick and Place project tutorial

Video below demonstrates how to make the assembling projects.

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

https://www.youtube.com/watch?v=ldhIhf3v1nY

5.13.3.4 "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.

5.13.3.5 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.

See also:

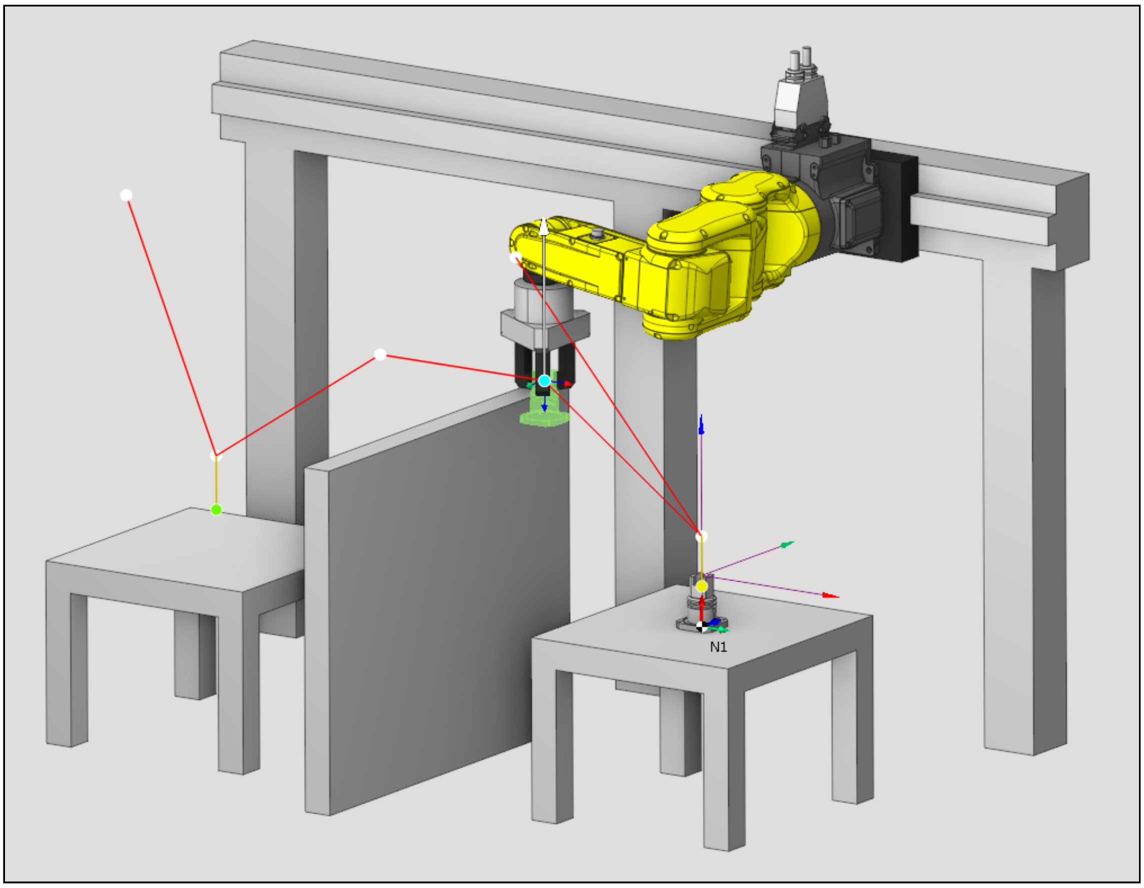
Move part operations

Machine requirements for part moving operations

Clamp devices control

Approach and return rules

5.13.3.6 Point Pick-and-Place



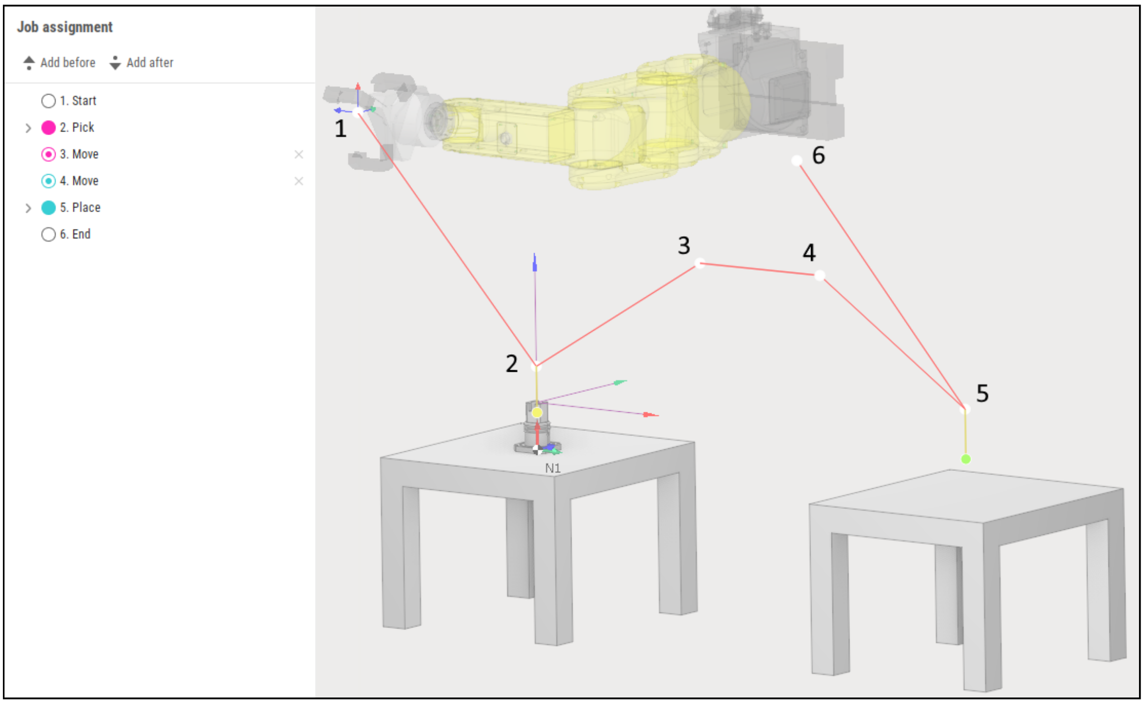
It's based on the Pick-and-place operation.

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.



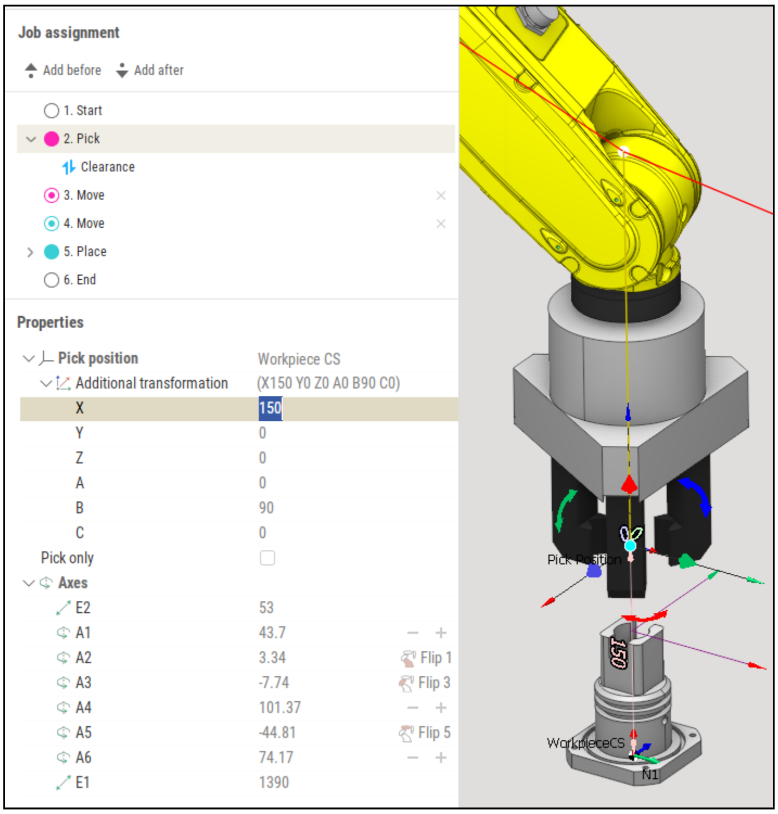
1. ◯ **Start point**

   It is read-only. This is the end point of the previous operation

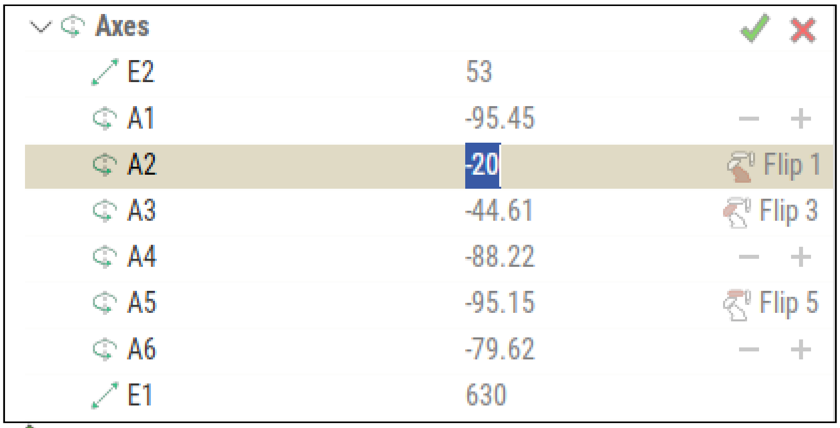
2. ● **Pick point**

   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:



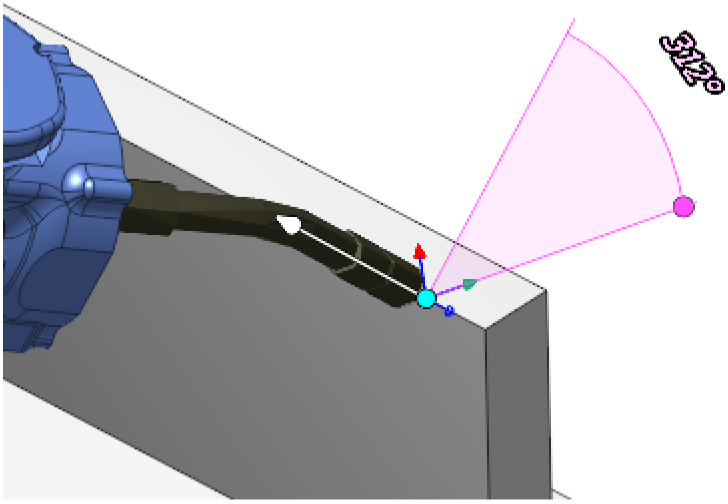
- moves the point to the tip of the machine
- returns previous values
- 360/+360
Flip - controls flips in the robot

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:

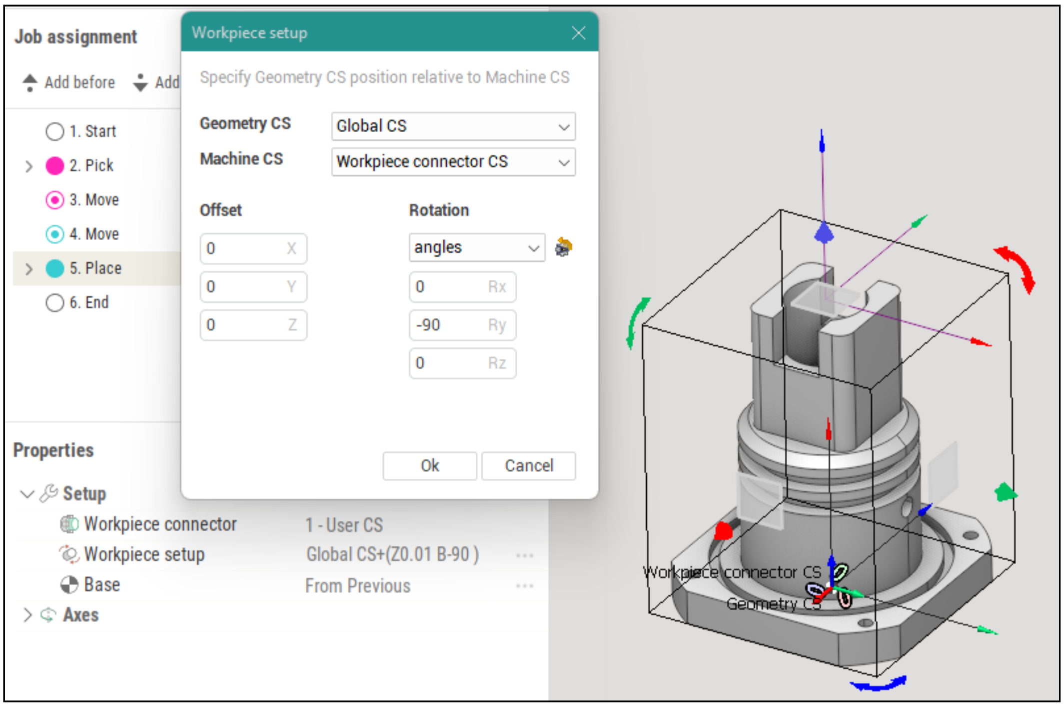
   Motion type - is set by what type to move to the point:
   - MultiGoto - Multi coordinate movement
   - PhysicGoto - Physical machine axes movement
   - Avoid collisions - Collision avoidance movement

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.

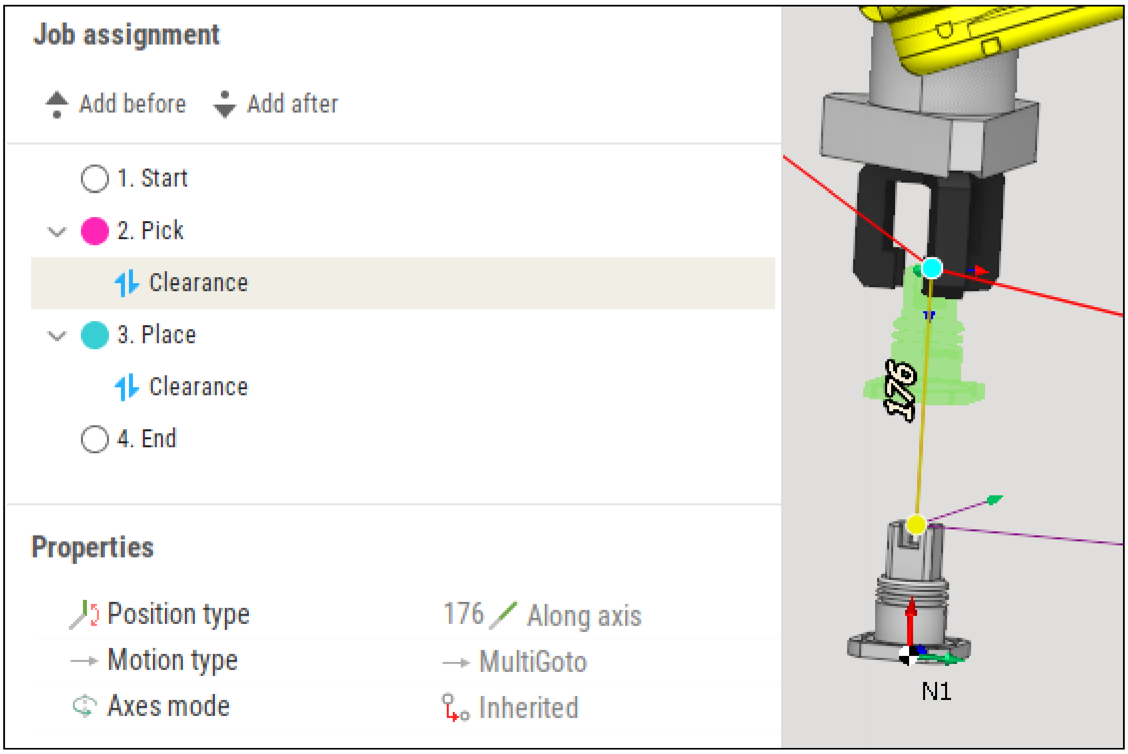


6. ◯ **End point**

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

See also:

Move part operations

Machine requirements for part moving operations

Clamp devices control

Approach and return rules

5.13.4 Turn take over

Turn take over is the special kind of pick and place operation designed to use in the MTM or swiss-lathe project template.

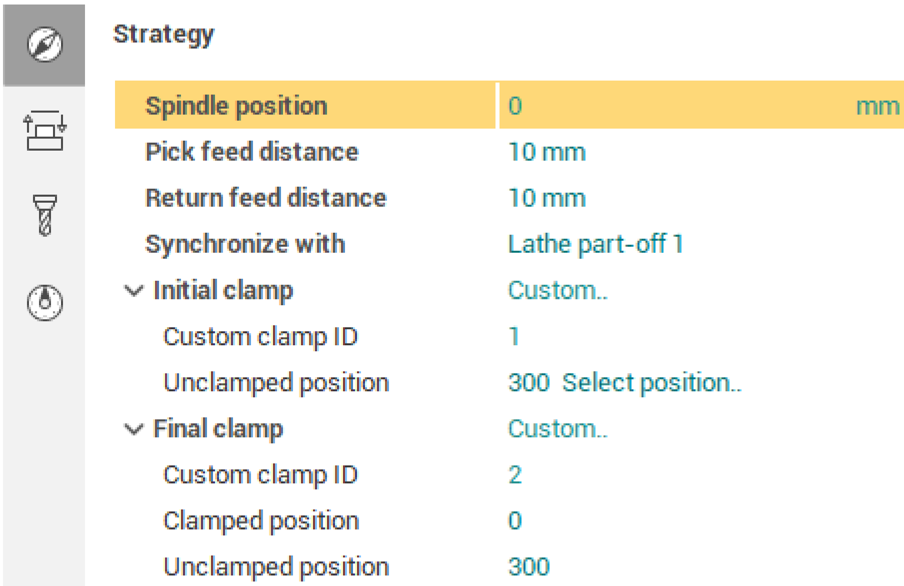
MTM projects and how to create it is shown in the video below.

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:

https://www.youtube.com/watch?v=iXFY8VliTBc

The main difference of the **Turn take over** operation from the simple **pick and place** 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.

5.13.4.1 Turn take over 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 feed distance

The length of the movement which is done using the engage feed before picking the part.

Return feed distance

The length of the movement which is done using the retract feed after picking the part.

Synchronize with

The <**Wait>** CLData command will be inserted which allows this operation to stop any machining until the specified operation is finished.

Clamp parameters

Select (optionally) the clamp devices which are used to hold the part during different stages of the Turn takeover. For more information about the clamp control parameters, see the Clamp devices control article.

5.13.4.2 Example project(s)

See the *Turn-Milling/Hanwha* and *Turn-Milling/Takeover* sample projects for an example of the part takeover between the spindles.

See also:

Move part operations

Pick and place

Swiss lathes programming

Machine requirements for part moving operations

Clamp devices control

Approach and return rules

5.13.5 Sub spindle working

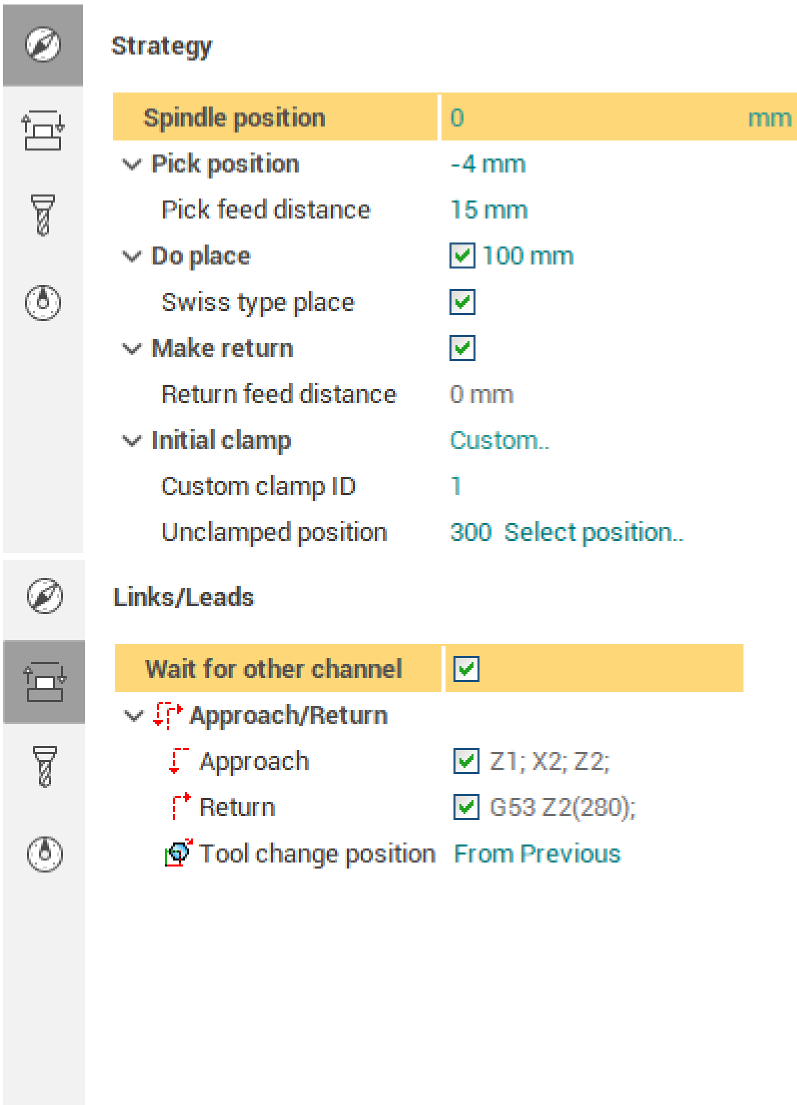
5.13.5.1 Operation overview

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.

5.13.5.2 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.

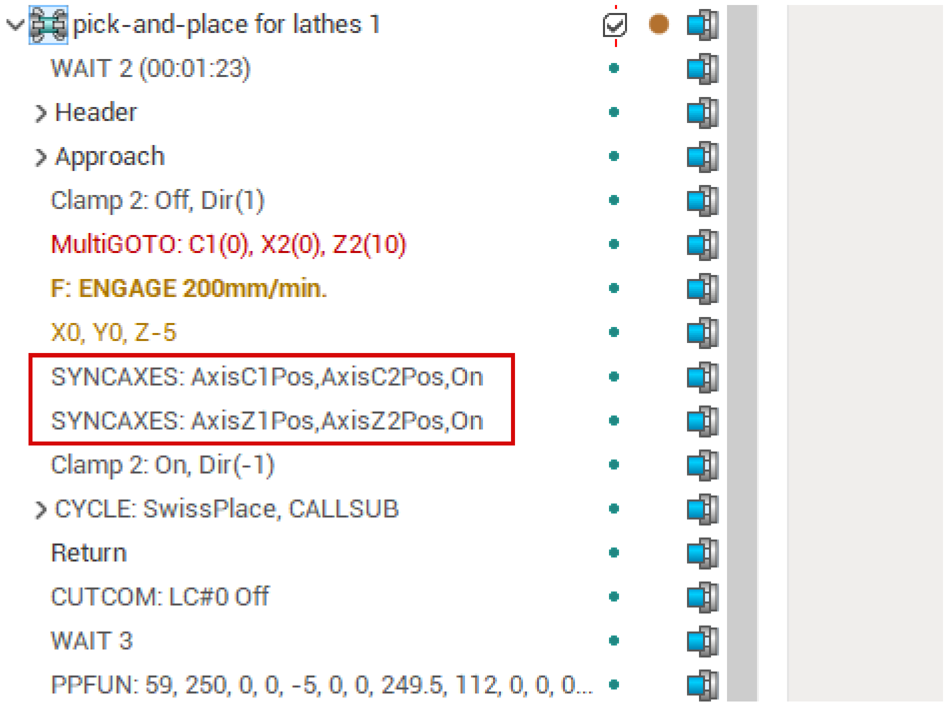
This parameter is visible only if there are at least 2 channels in the machine schema and the previous operation is from **another channel**.

Approach/Return parameters

The common (for the majority of the SprutCAM X operations) group of parameters, defining the approach and return sections of the operation. See the Approach and return rules documentation for more info.

5.13.5.3 Spindles synchronization

The **Sub spindle working** operation automatically generates New CL-data command for the **spindles synchronization.**



5.13.5.4 Example project(s)

See the *Turn-Milling/Hanwha* sample project for an example of lathe part machining with the spindles synchronization.

See also:

Move part operations

Swiss lathes programming

Machine requirements for part moving operations

Clamp devices control

5.13.6 Bar feeding

5.13.6.1 Operation overview

Any machining on the swiss lathe starts from the bar feeding. So the bar feeding operation must be the first in the operation list for the swiss lathes. On Hanwha swiss lathes it generates G300.

See the Creating a simple Bar feeding operation video for an example of using the operation.

5.13.6.2 Bar feeding parameters

Bar overhang

The axial distance between the spindle base point and the tool tip point.

Retract distance

The rebound of the limiter-tool before move it away.

Use canned cycle

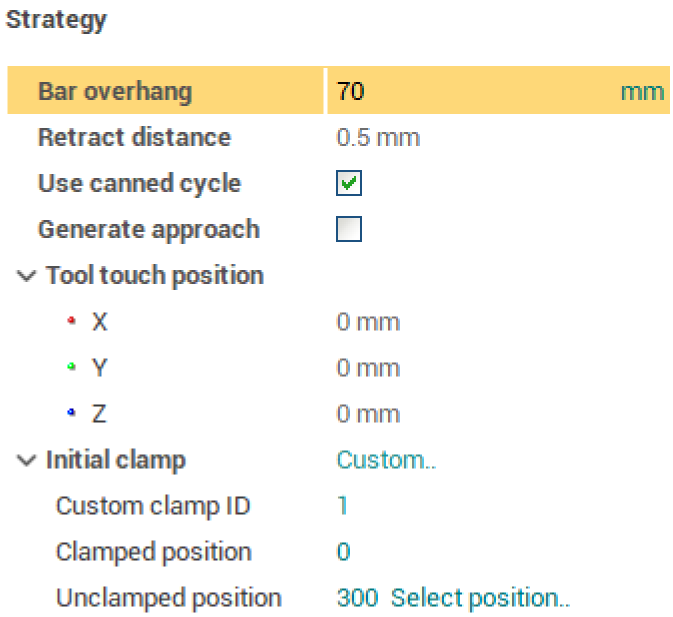
Whether the bar transferring process will be formalized as a cycle for the further analysis in the postprocessor or will be output as a sequence of elementary commands.

Generate approach

Disable this parameter if the tool is already located in the start point after cut-off, for example, in the case of the looped program.

Tool touch position

The touch position of the tool tip point in the workpiece coordinate system (G54).



Initial clamp

Select (optionally) the clamp device which initially holds the part and define the parameters for its usage. If enabled, special CLData commands will be generated for clamping/unclamping the part during the bar feeding. For more information about the clamp control parameters, see the Clamp devices control article.

5.13.6.3 Example project(s)

See the *Turn-Milling/Hanwha* sample project where the **Bar feeding** operation is used to initially position the part for further machining.

See also:

Move part operations

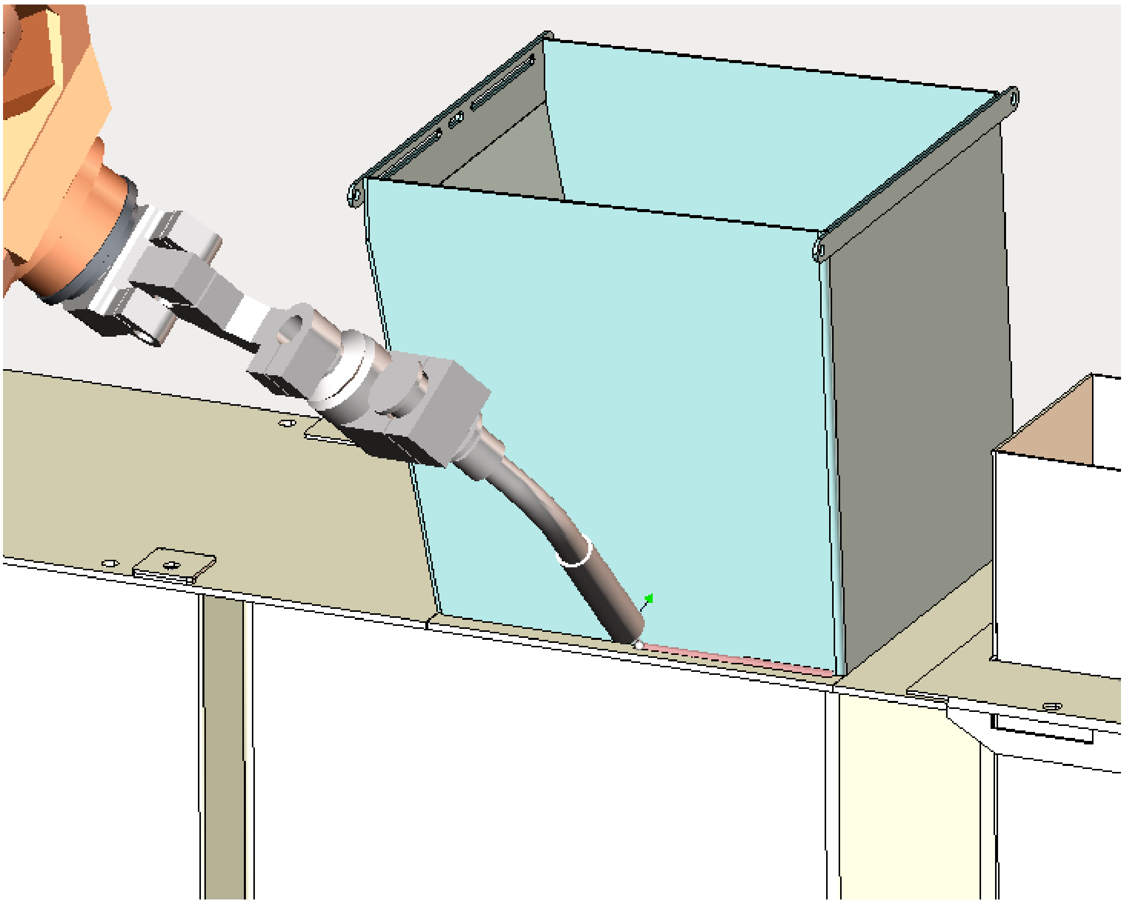
Swiss lathes programming

Machine requirements for part moving operations

Clamp devices control

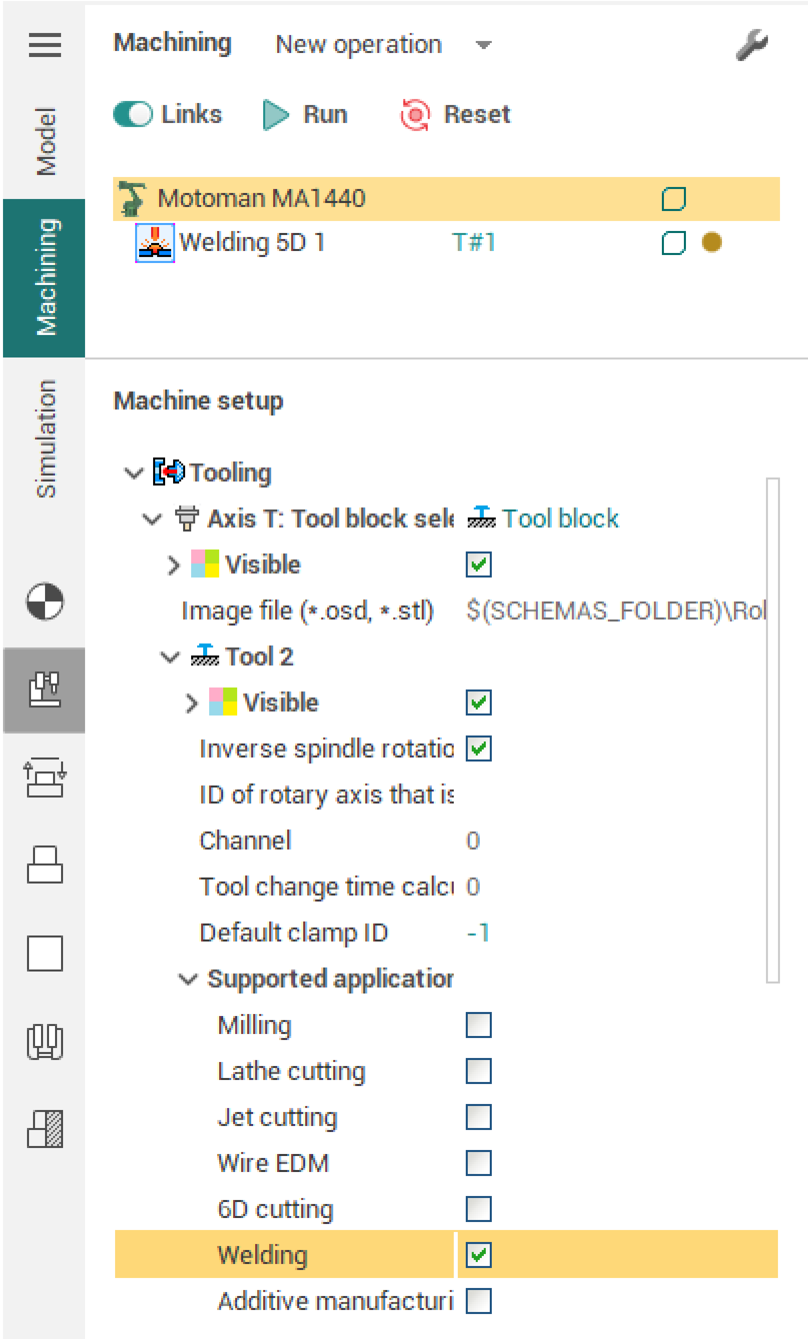
Approach and return rules

5.14 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>
```

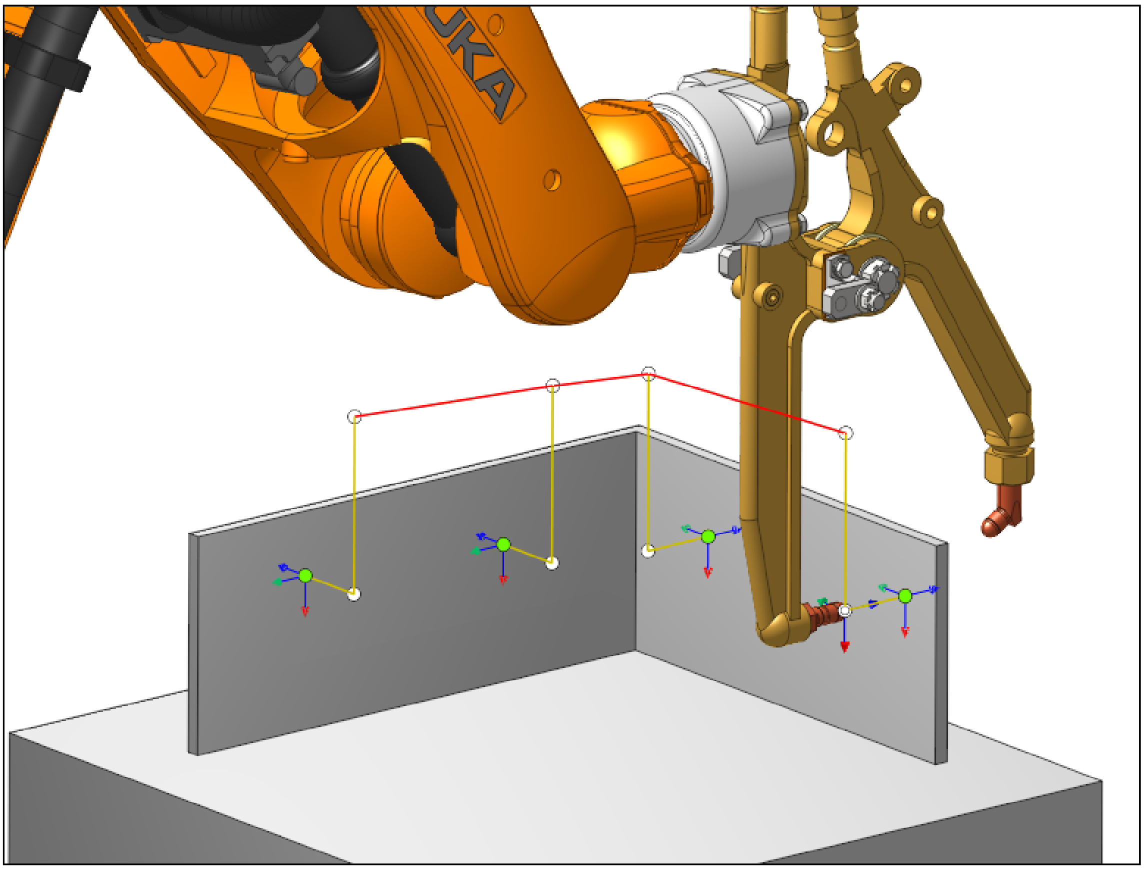
Setting up of operation to work with specific type of equipment can be made by writing in the postprocessor appropriate commands to control the equipment, or, if this is not enough, by the addition of a special operation on the basis of the Welding 5D operation, adapted to control specific equipment.

See also:

Welding

Operations which require adaptation

5.14.1 Point welding operation

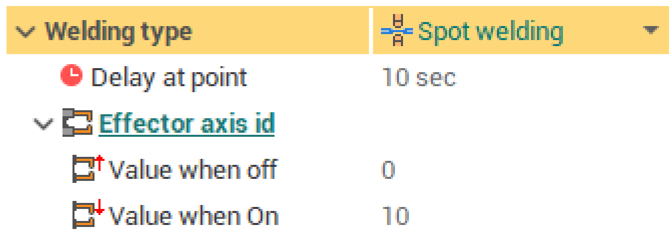


Operation can be used to **Tack weld** and **Spot weld.**

5.14.1.1 Job assignment

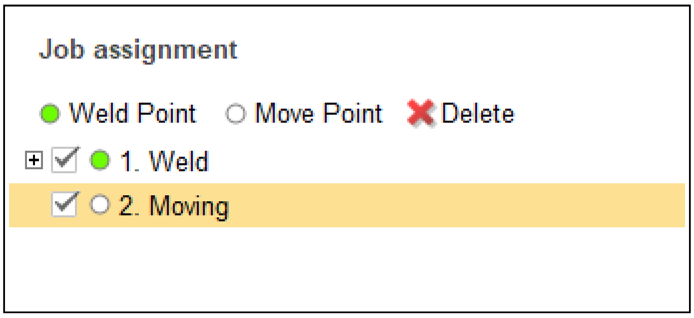
In the strategy tab, you need to select the type of welding:

Tack weld	**Welding type**: Tack welding Delay at point: 10 sec

Spot weld	

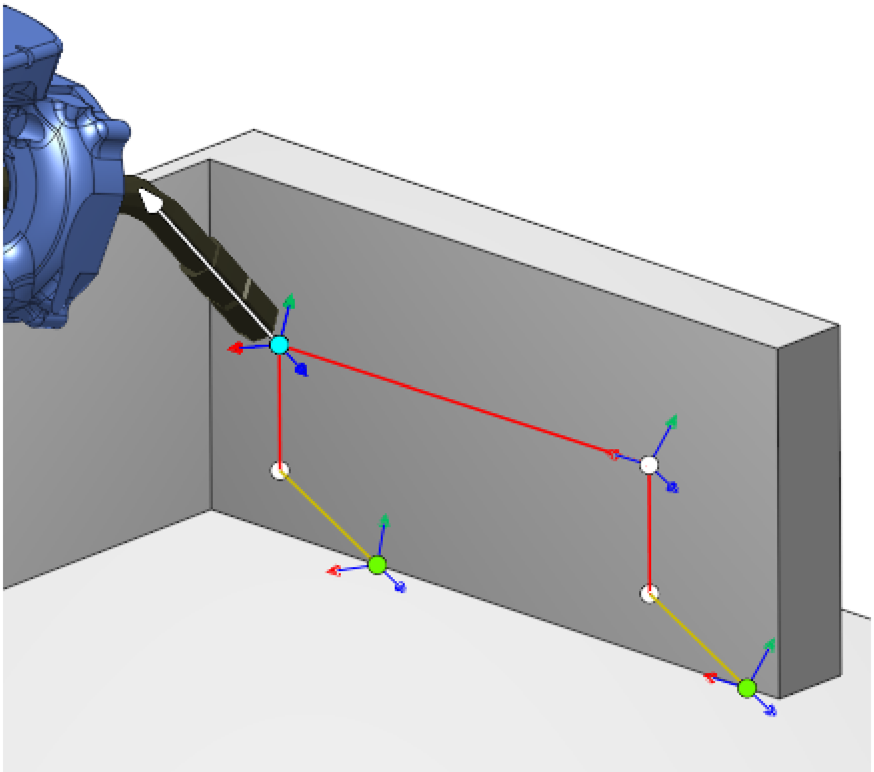
For both types the job assignment is the same.

The **job assignment** consists of two types of points:



1. Weld Point
2. Move Point

These points can be used to construct a chain of trajectories:

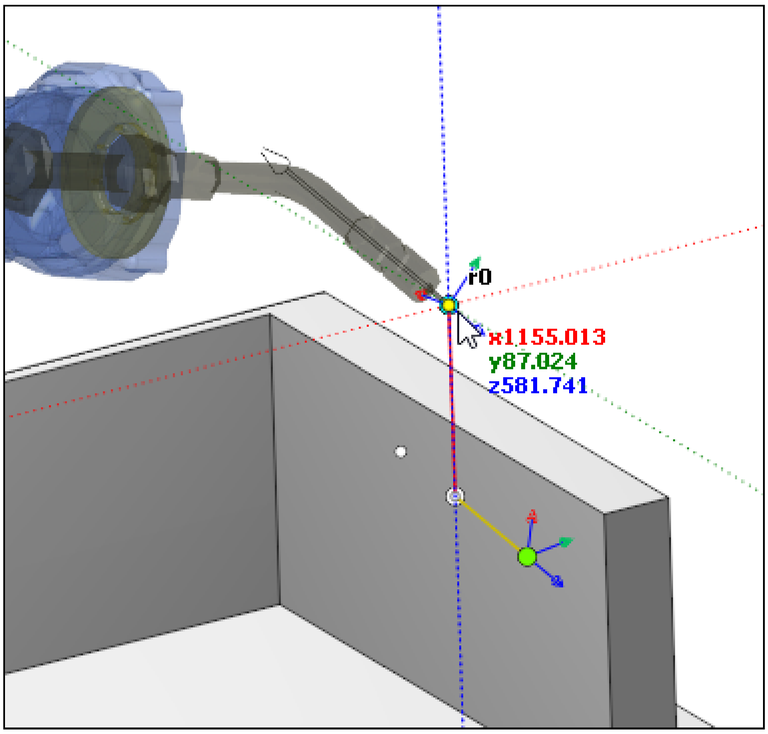


Drag points

These points can be moved by dragging.

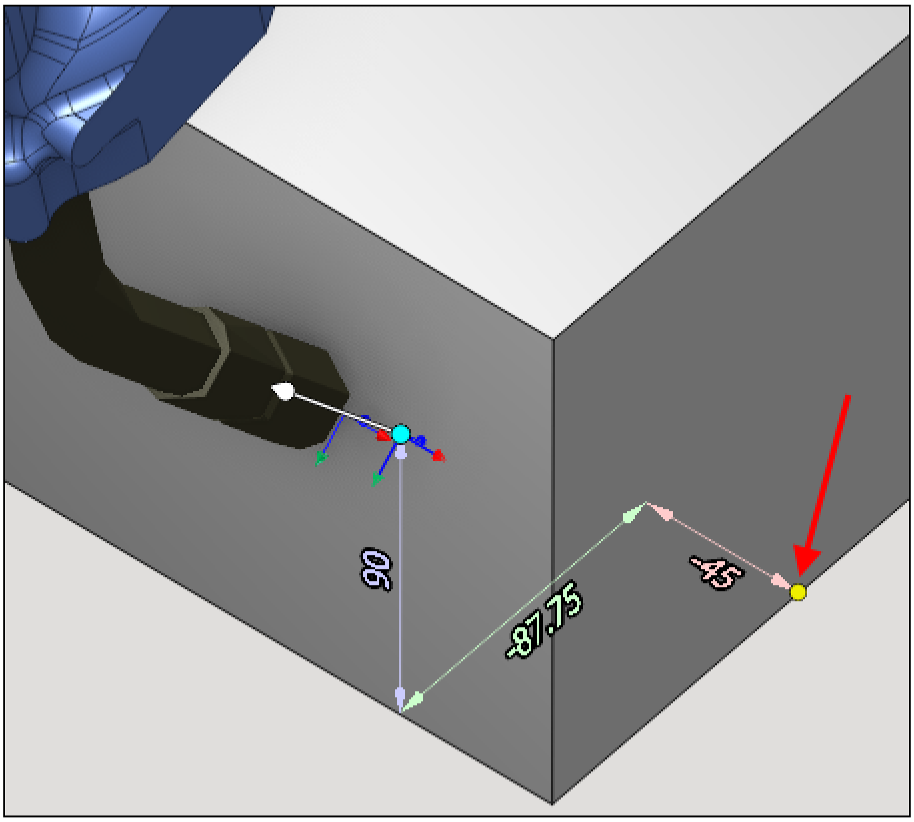
They attach to faces , curves , splines , edges , vertices . Tool axis rotates automatically when attached.

Move points can be easily moved using **Smart snap** :

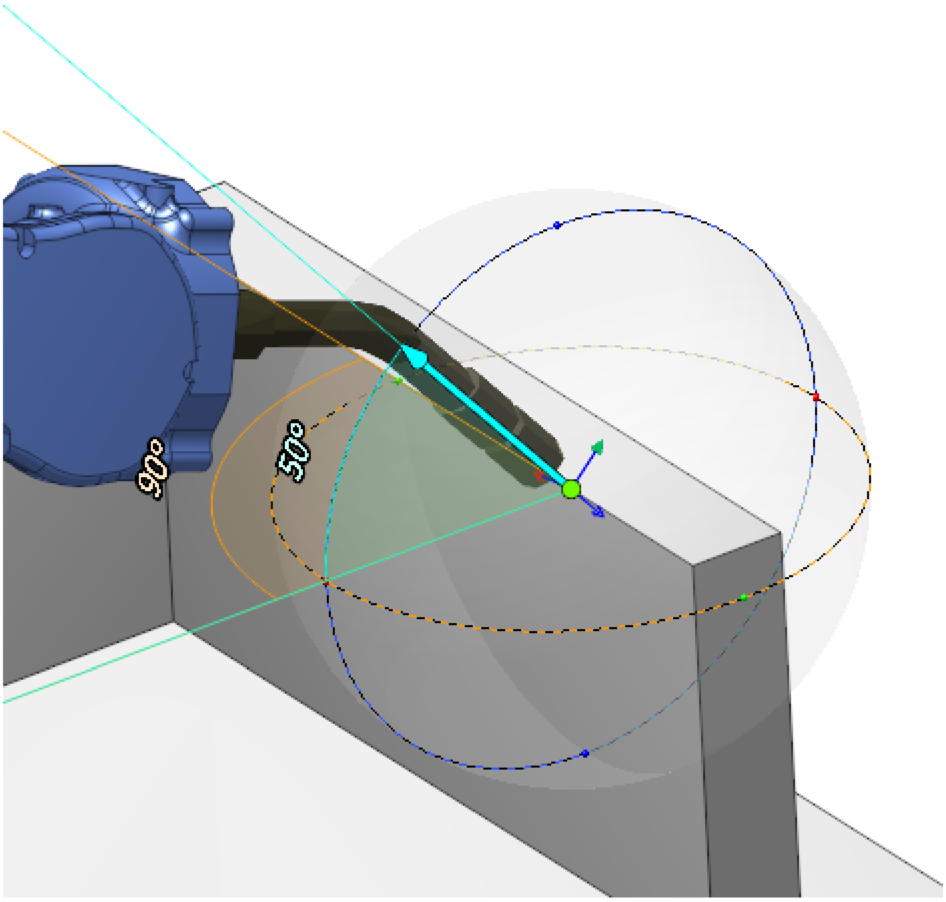


Position of the point can also be specified as an offset from the auxiliary yellow point.

To do this, first select the yellow point:



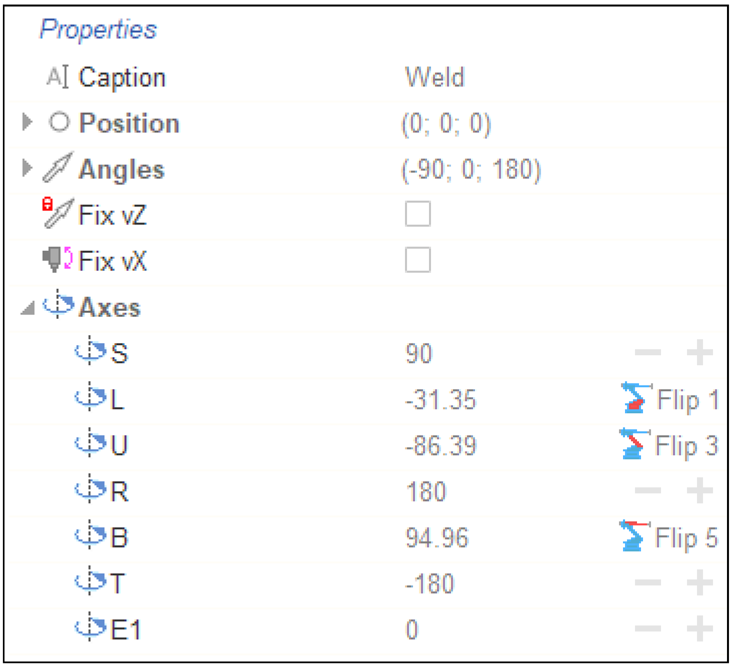
You can also rotate the axis vector by dragging the visual vector:



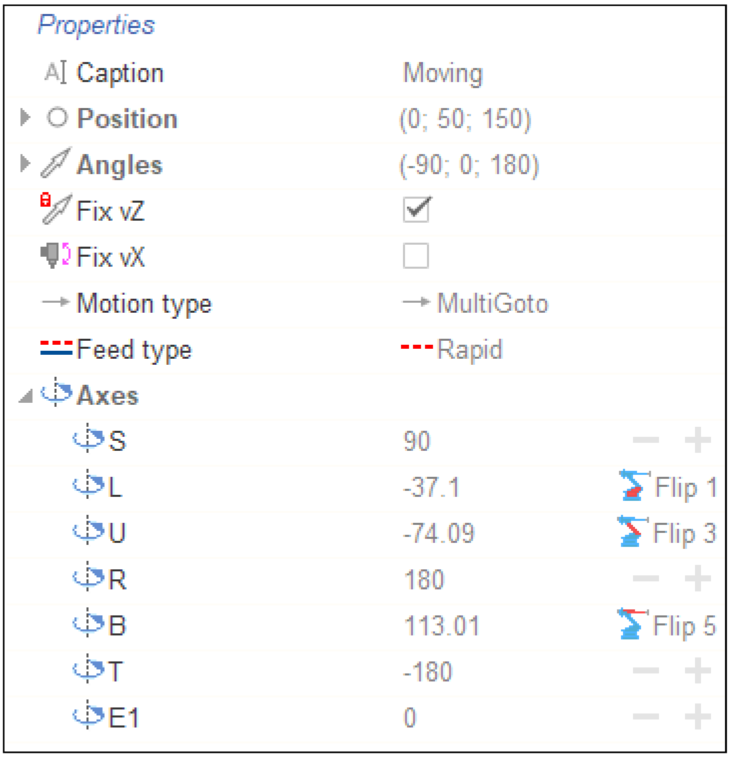
Point parameters

Points contains the following parameters:

Weld Point



Move Point



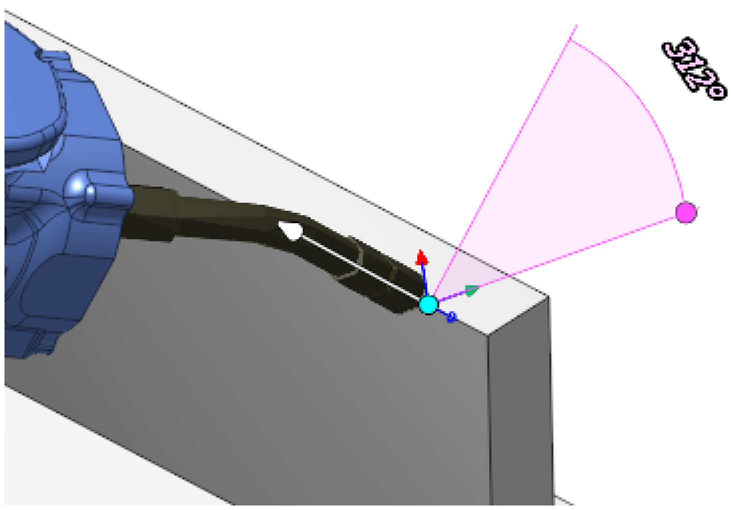
Caption - point name

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:

Axes		
S	90	− +
L	-37.1	Flip 1
U	-74.09	Flip 3
R	190	− +
B	113.01	Flip 5
T	-180	− +
E1	0	− +

- moves the point to the tip of the machine

- returns previous values

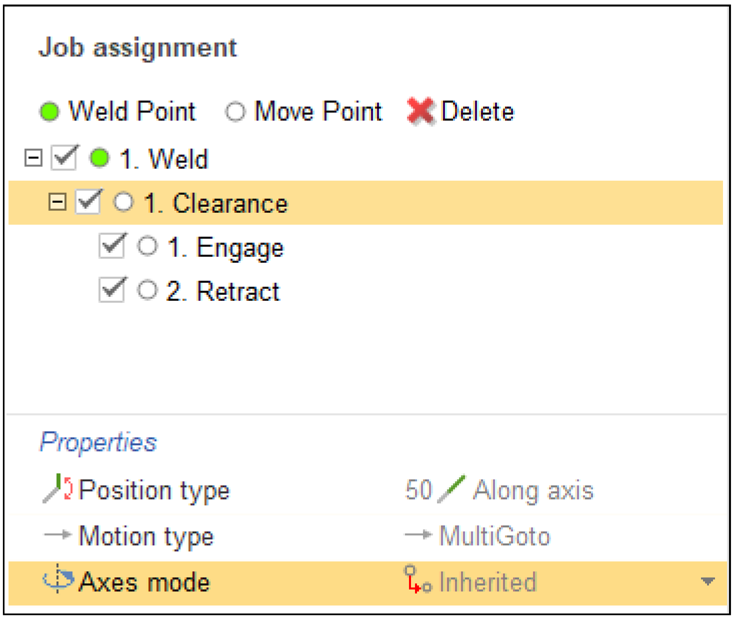
− + - 360/+360

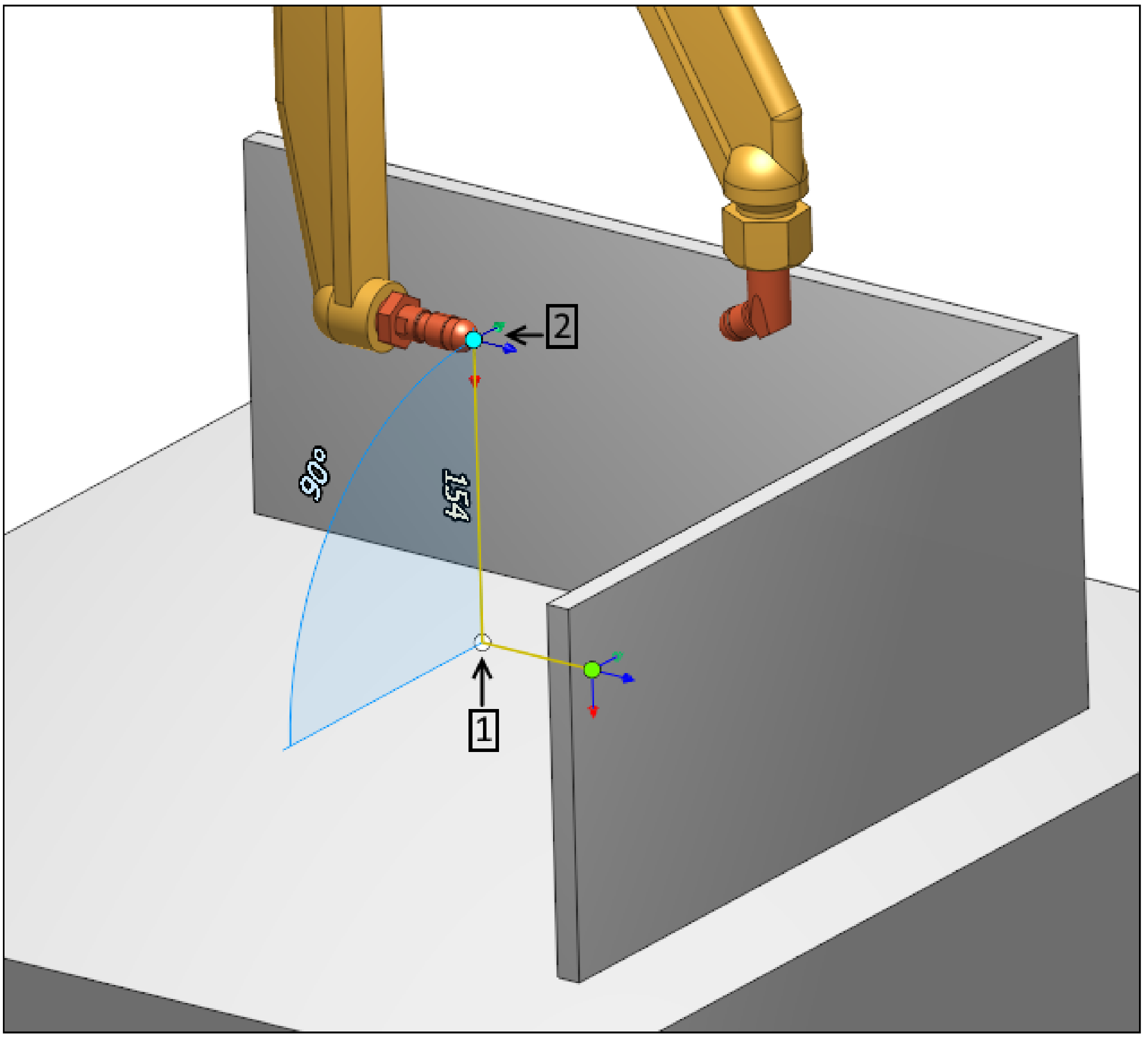
Flip - controls flips in the robot

Auxiliary Point

- **Weld point** also contains additional auxiliary points:

1. Clearence
2. Engage
3. Retract

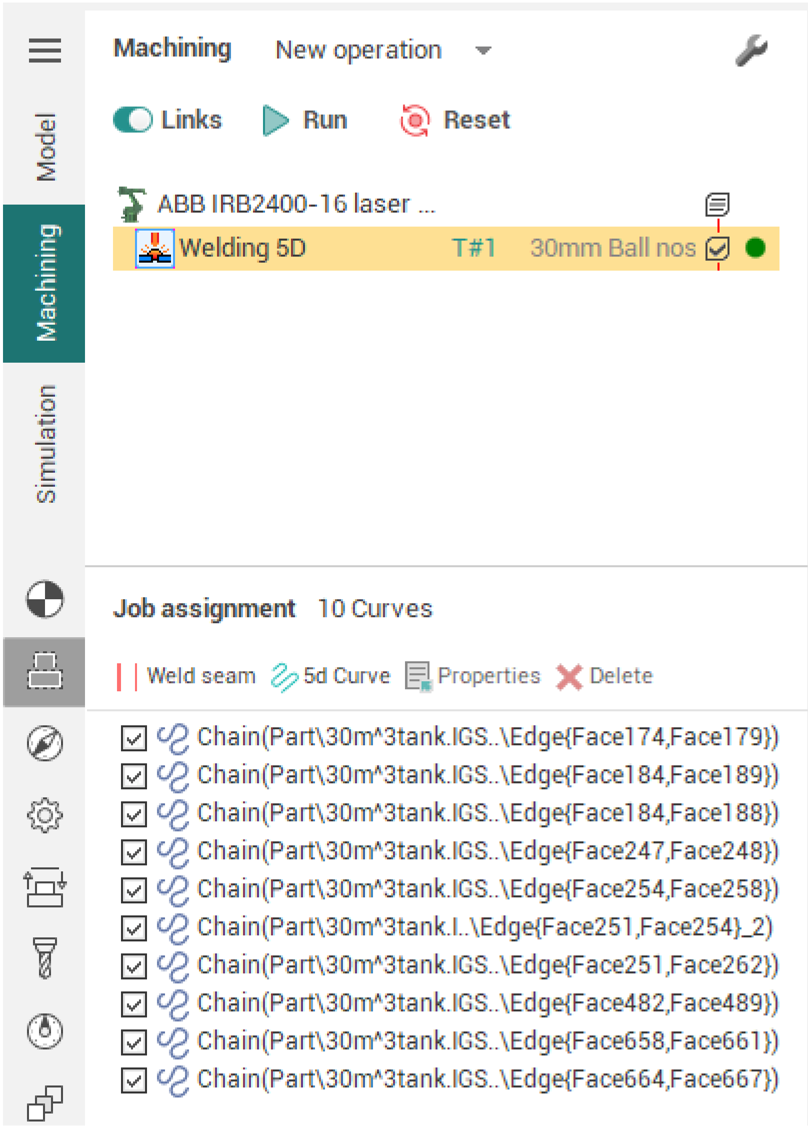




5.14.2 Welding 5D and 6D operations

Welding 5D operation 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.

Weld seam can be specified in the Job assignment window. The working process here is very similar to the curve definition in 5D Contouring operation.



It is enough to add edge between welded parts and the system automatically calculates the angles for each point of curve such way that the weld head is held as close as possible to the middle between the adjacent walls but to not collide with them. Then the curve with tool vectors will appear in the screen. You can grab any of the vectors and drag it to the desired direction, when suddenly the angle counted automatically by the system, you do not like. At the same time dimensions will shown on the screen, by clicking on which you can enter exact values of tilt angles of the vector. To change the direction along curve just click on the blue arrow at the start of the curve. You can also drag the start and end points of the curve, by holding the appropriate marker.

In the properties inspector of the operation such properties can be found.

- Lead and lean angles. Defines additional tool tilt to the side and along the curve for all curves specified in the job assignment.
- Idling minimization. It affects the order of curve uniting, if there is more than one curve. If it is not enabled, the curves will follow in the order as they specified. Otherwise, the order can be changed to reduce the idle movements.
- Arc interpolation. It reduces the number of frames in the NC code by combining short lines by arcs with a given accuracy. It is actual only if the machine supports spatial arcs (eg robots), because resulting arcs often lie in non-orthogonal planes.
- Extend (+)/Trim (-) passes. Allows to change (increase or reduce) the length of all defined curves without the need of start and end points dragging for each curve separately. By default has value of 50% of tool diameter (width of weld seam).
- Engage/retract. Allows to define additional segment at the start or the end of each curve.
- Safe motions, Safe surface and Links defines the type and sizes of links between curves.

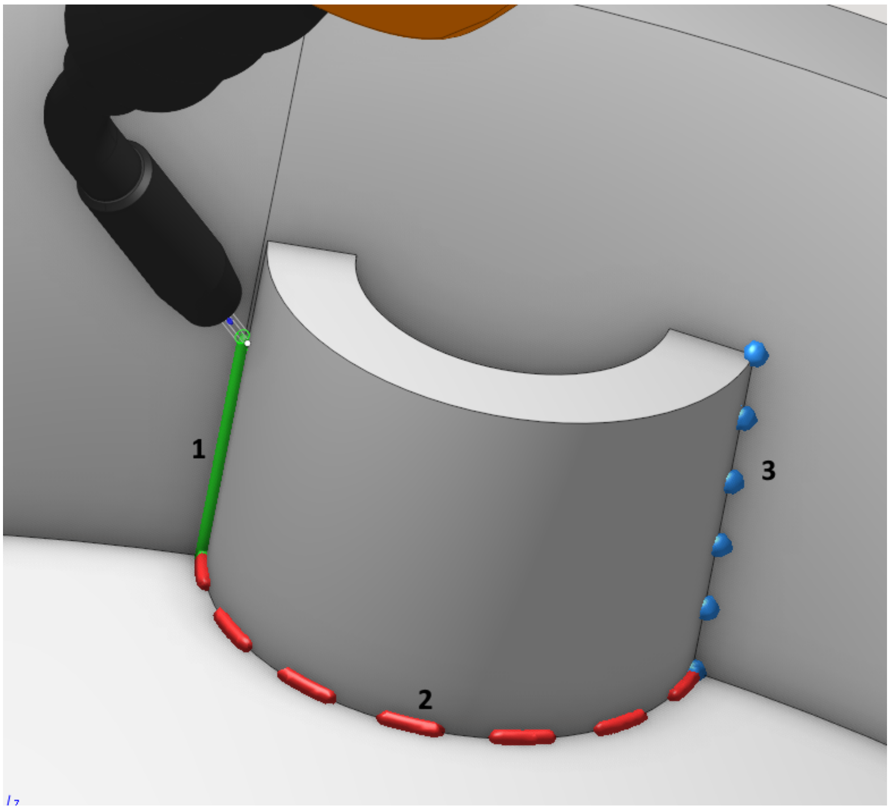
Then you can switch to the Simulation mode to see how the material is added to the place where the tip of the welding head is touching the workpiece. The thickness of the layer in simulation defined by the working length property of the tool, or if it is not set (more than the tool diameter), the thickness will equal to the tool diameter.

Welding 6d

In operation **Welding 6d**, the **Welding type** parameter is additionally available.

It supports the following welding process type types:

1. Seam welding
2. Stitch welding
3. Tack welding
4. Spot welding



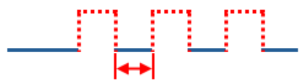
1 Seam welding

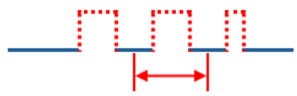
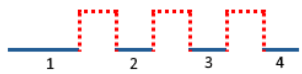
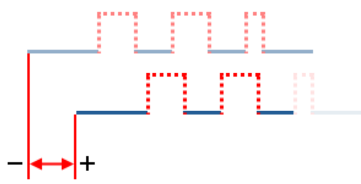
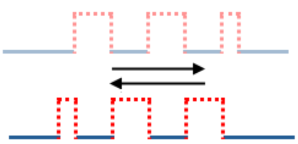
This is a continuous curved welding.

2 Stitch welding

It is intermittent. Contains the following options:

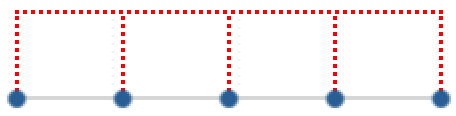
Stitch length	10 mm
Start stitch length	10 mm
End stitch length	10 mm
Spacing size	ISO spacing length
Spacing length	10 mm
Adjustment	from start
Additional shift	0 % Stitch (0 mm)
Invert stitches	☐
Retract distance	10 mm

Stitch length	
Start stitch length	
End stitch length	
Spasing size:	
ISO spacing length The size of the spacing itself is set	

	AWS center to center Sets the distance between the centers of strokes	
	Even (stitch count) The number of strokes is set, spacings are distributed evenly over the remaining length.	
Adjustment : Alignment of stitches along the contour.		
	from start	
	from center	
	from end	
	Auto resize spacing	
Additional shift Additional displacement of stitches along the contour.		
Invert stitches Invert the distribution of stitches along the contour		

Retract distance Distance of axial retraction from the contour at the transition between stitches	

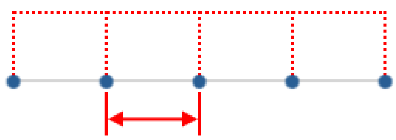
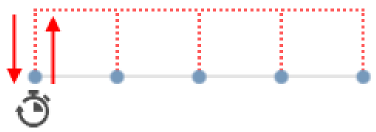
3 Tack welding



Tack welds are small and temporary welds used to hold parts together.

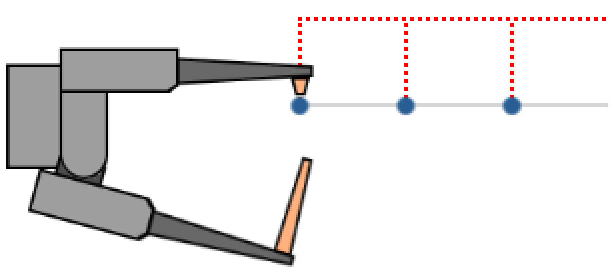
Contains the following options:

Points step	3 x (Count)
Delay at point	10 sec
Additional shift	0 mm
Invert stitches	☐
Retract distance	10 mm

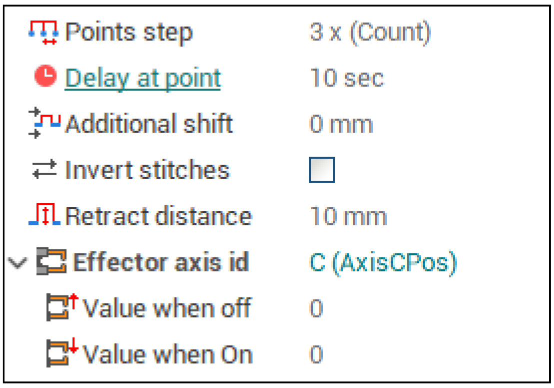
Points step. Step between points	
Delay at point. Dwell time at each point in seconds.	

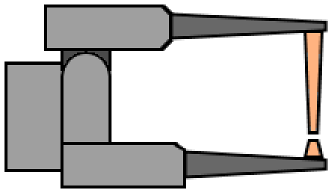
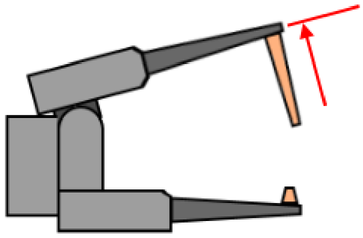
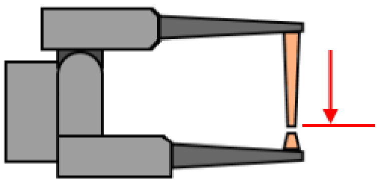
Other parameters are similar to Stitch welding

4 Spot welding



Contains the following options:

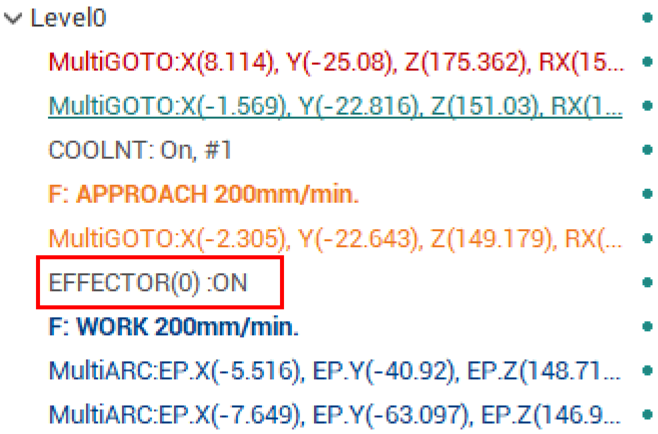


Effector axis id ID of the machine axis, which is the axis of the working body - welding clamp	
Value when off	
Value when On	

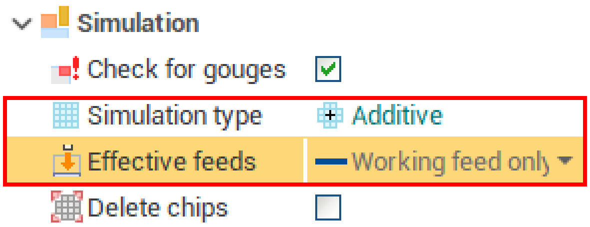
Other parameters are similar to Tack welding.

Effector On/Off command

This command turns welding on and off.



It is set if the **simulation type** is additive or paint according to the selected **Effective Feeds**. For additive operations and welding, the command is set to automatic.



See also:

Welding

Welding optional module

5.15 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.

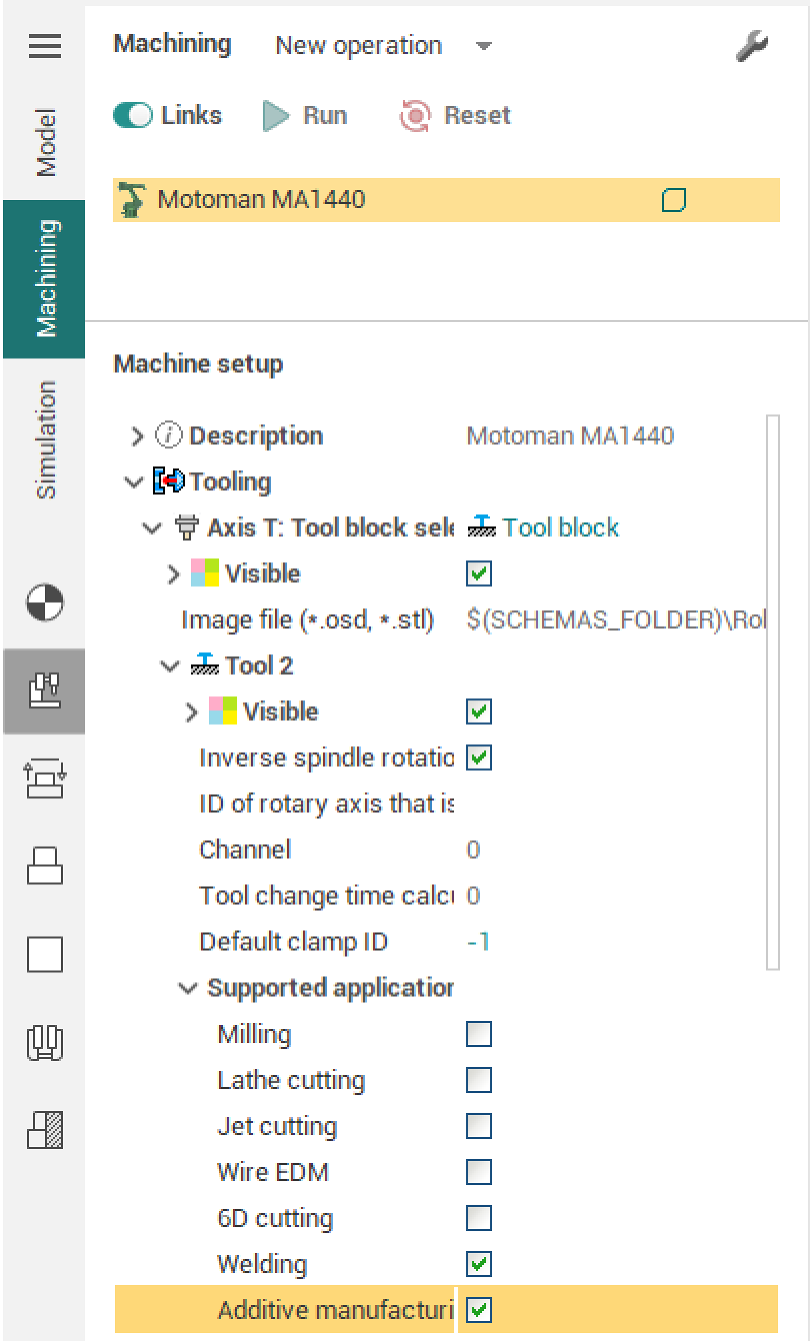
There are different kinds of cladding.

- 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.

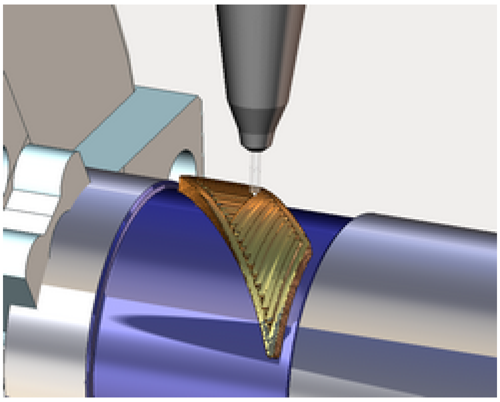
```
<SCType ID="AdditiveToolHolder" Caption="Additive tool holder" Type="TToolHolderNode">
  <SupportedToolTypes>
    <AdditiveTool DefaultValue="true"/>
  </SupportedToolTypes>
</SCType>
```

See also:

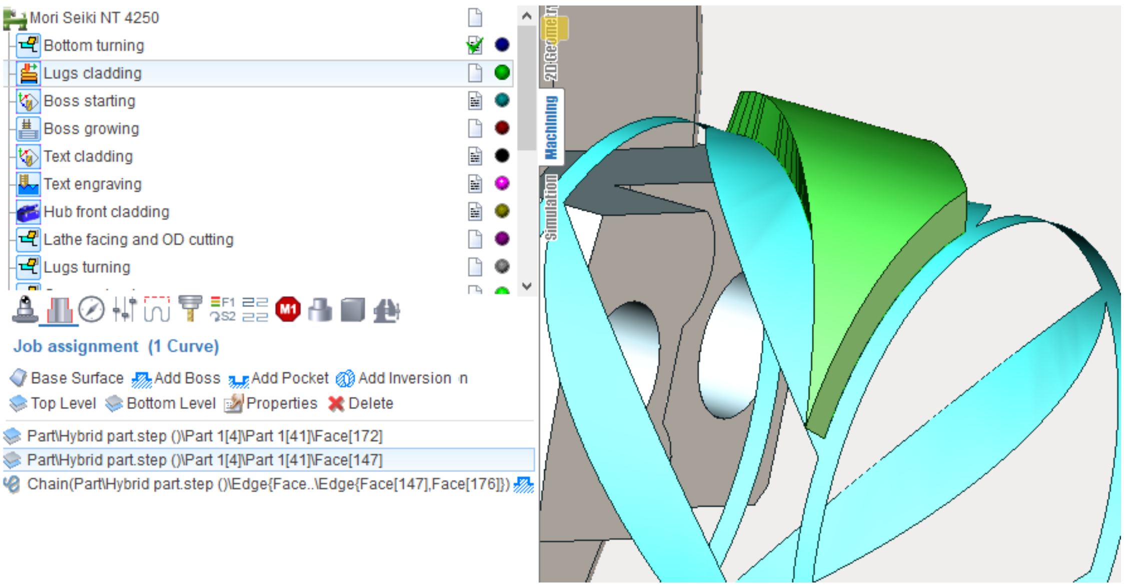
Cladding operation

Operations wich require adaptation

5.15.1 Area cladding operation



Area cladding operation designed to add a layer of material on a local piece of the part that can be limited by curves. The user interface and job assignment of the operation is similar to the Pocketing operation. The Job assignment window shown below.



The first task is a base surface definition 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.

In addition to the levels in the Properties inspector may be set the following parameters.

- Machining strategy. Parallel and Offset strategies now are realized.
- Angle of passes for parallel strategy. If you enable Swap angle between layers it will increase 90 degrees after each layer.
- Passes order: Outside to inside, Inside to outside, Zigzag. Defines the order of passes at level from outside to inside or conversely. If Zigzag selected then the order will change from level to level.
- Offset pass count for parallel strategy. Allows to enhance the quality of the outer surface. Should be more than zero if you need it.
- **Offset pass step** allows you to set the distance between offset passes. "Offset pass step" is available only when "Offset pass count" is more than 1. There is 3 value: 1) mm (entered value is calculated in mm); 2) %Ø (entered value is a percentage of the tool diameter; 3) % of Step (entered value is a percentage of **Step**).
- Step. Step-over between passes in the current system units (mm, inch), in percents of tool diameter (diameter of melting spot).
- Gap to prevent overlapping. In the case when it is making closed contour to prevent overlapping the layers to each other at the connection of the start and the end it is need to leave a gap approximately equal to the diameter of melting spot.
- Depth step. It determines how many layers of material it should make from the lower to the upper level. You can specify it in current system units (mm or inch), in percents of tool working length or directly specify the number of layers and the step will be calculated automatically.
- Initial depth. The depth of first level.
- Machining direction. Available variants: Zigzag, Forward, Backward. If you select Zigzag, the direction will inverted for each next string. Otherwise, all the passes will be executed strictly in a predetermined direction.
- Project toolpath onto the part. Allows to make cladding on the surfaces of complex geometric shapes. The initial toolpath formed on the base surface is projected onto the 3D model that is specified as the part of the operation.
- Tool axis orientation. Available variants: fixed, normal to surface 4D, normal to surface 5D, normal to base surface. Determines the law of the tool axis orientation calculation at the each point. If "fixed" selected, then orientation remain unchanged, same with the Z axis of operation's coordinate system. "Normal to surface 4D" - normal is taken from the part, but the slope will be considered along of one of the rotary axes only. "Normal to surface 5D" - tool axis coincides with the normal to the part. "Normal to base surface" - the normal orientation will taken from defined base surface.
- Safe level. Defines the level where long rapid motions it should perform.
- Short link max distance. Determines the distance above which the transition will be considered as long.
- Short link type, Long link type. Available variants: Straight, Via safe level. The way of link motions.
- Neighbor pass links on rapid. If it is disabled, then transition between neighbor passes will perform on working feed (with the supply of weldable material, activated burner, laser, etc.). When disabled, the link will on rapid feed, that is without melting the material.

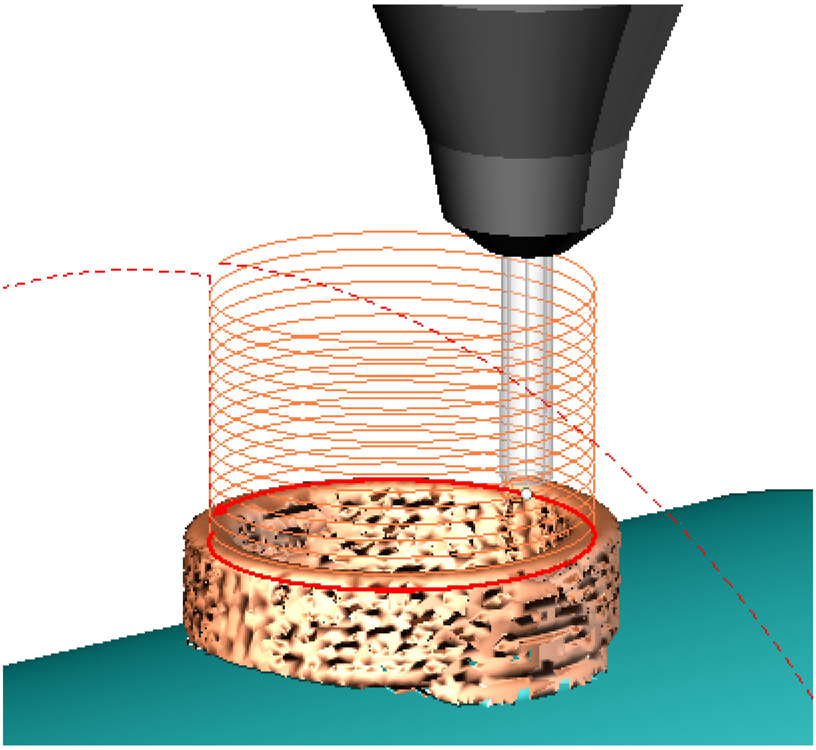
Then you can switch to the Simulation mode to see how the material is added to the place where the tip of the tool is touching the workpiece. The thickness of the layer in simulation defined by the working length property of the tool, or if it is not set (more than the tool diameter), the thickness will equal to the tool diameter.

See also:

Additive manufacturing

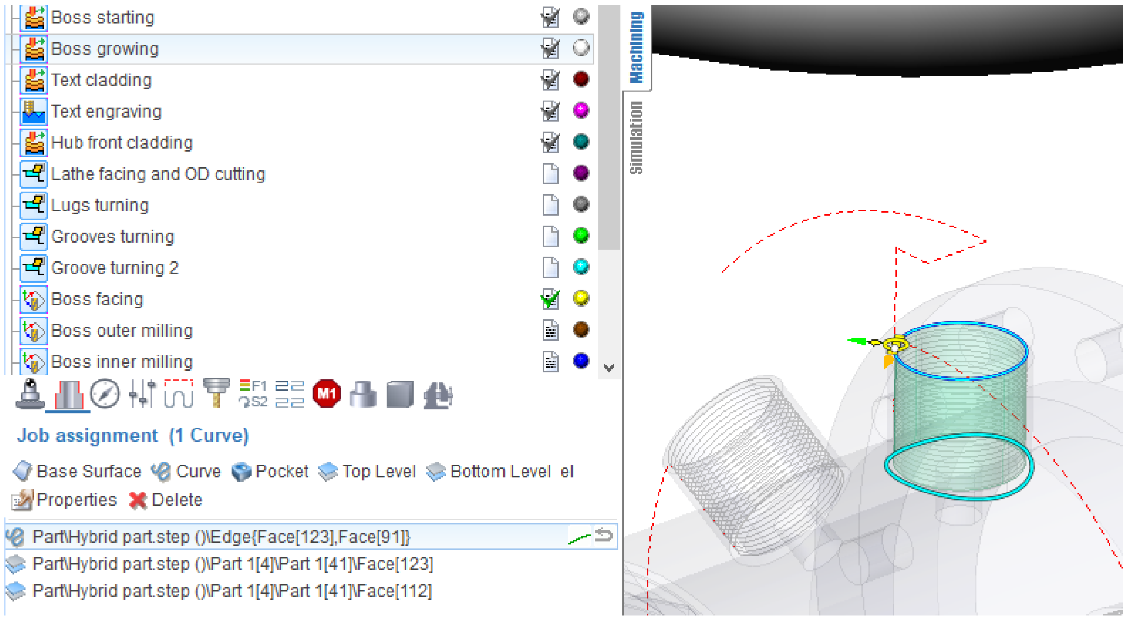
Cladding optional module

5.15.2 Curve cladding operation



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 Job assignment window shown below.



The first task is a base surface definition 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.

In addition to the levels in the Properties inspector may be set the following parameters.

- Depth step. It determines how many layers of material it should make from the lower to the upper level. You can specify it in current system units (mm or inch), in percents of tool working length or directly specify the number of layers and the step will be calculated automatically.
- Initial depth. The depth of first level.
- Rough passes at the level. Wall thickness and step should be defined if it is enabled.
- Step. Stepover between passes in the current system units (mm, inch), in percents of tool diameter (diameter of melting spot).
- Helical machining. If disabled toolpath will consist set of planar levels. If enabled then toolpath for each curve of job assignment will continuous spiral curve.
- Initial level of helical machining defines necessary or not to do starting planar pass before rising path.
- Idle moves minimization defines the order of transitions between curves. If disabled the curves will be machined in order that they placed in list. If enabled then minimization of intermediate transitions will performed and the order can vary.
- Allow reverse direction. It determines whether or not to change the direction of machining along the curve.
- Arc interpolation. It tries to interpolate toolpath by circular arcs with defined tolerance.
- Project toolpath onto the part. Allows to make cladding on the surfaces of complex geometric shapes. The initial toolpath formed on the base surface is projected onto the 3D model that is specified as the part of the operation.
- Tool axis orientation. Available variants: fixed, normal to surface 4D, normal to surface 5D, normal to base surface. Determines the law of the tool axis orientation calculation at the each point. If "fixed" selected, then orientation remain unchanged, same with the Z axis of operation's coordinate system. "Normal to surface 4D" - normal is taken from the part, but the slope will be considered along of one of the rotary axes only. "Normal to surface 5D" - tool axis

coincides with the normal to the part. "Normal to base surface" - the normal orientation will taken from defined base surface.
- Safe level. Defines the level where long rapid motions it should perform.
- Short link max distance. Determines the distance above which the transition will be considered as long.
- Short link type, Long link type. Available variants: Straight, Via safe level. The way of link motions.
- Neighbor pass links on rapid. If it is disabled, then transition between neighbor passes will perform on working feed (with the supply of weldable material, activated burner, laser, etc.). When disabled, the link will on rapid feed, that is without melting the material.

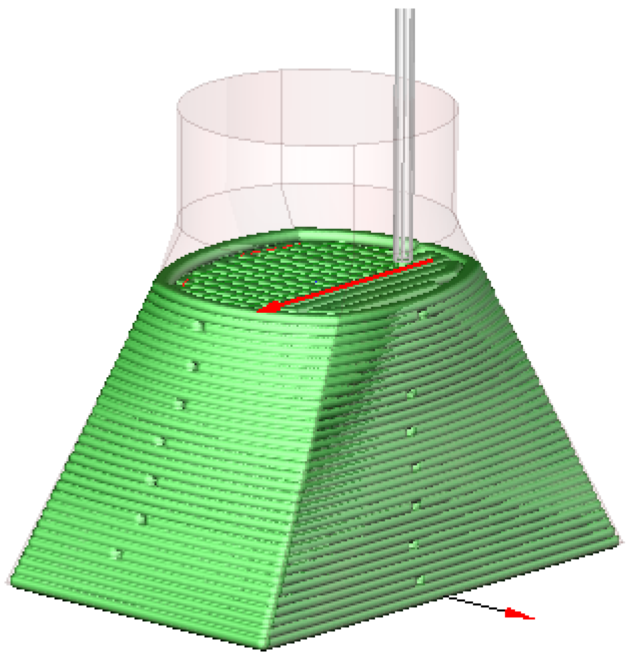
Then you can switch to the Simulation mode to see how the material is added to the place where the tip of the tool is touching the workpiece. The thickness of the layer in simulation defined by the working length property of the tool, or if it is not set (more than the tool diameter), the thickness will equal to the tool diameter.

See also:

Additive manufacturing

Cladding optional module

5.15.3 Cladding 3D operation



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.

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.

In addition to the levels in the Properties inspector may be set the following parameters.

- Machining strategy. Parallel and Offset strategies now are realized.
- Angle of passes for parallel strategy. If you enable Swap angle between layers it will increase 90 degrees after each layer.
- Passes order: Outside to inside, Inside to outside, Zigzag. Defines the order of passes at level from outside to inside or conversely. If Zigzag selected then the order will change from level to level.
- Offset pass count for parallel strategy. Allows to enhance the quality of the outer surface. Should be more than zero if you need it.
- **Offset pass step** allows you to set the distance between offset passes. "Offset pass step" is available only when "Offset pass count" is more than 1. There is 3 value: 1) mm (entered value is calculated in mm); 2) %Ø (entered value is a percentage of the tool diameter; 3) % of Step (entered value is a percentage of **Step**).
- Step. Stepover between passes in the current system units (mm, inch), in percents of tool diameter (diameter of melting spot).
- Gap to prevent overlapping. In the case when it is making closed contour to prevent overlapping the layers to each other at the connection of the start and the end it is need to leave a gap approximately equal to the diameter of melting spot.
- Depth step. It determines how many layers of material it should make from the lower to the upper level. You can specify it in current system units (mm or inch), in percents of tool working length or directly specify the number of layers and the step will be calculated automatically.
- Initial depth. The depth of first level.
- Machining direction. Available variants: Zigzag, Forward, Backward. If you select Zigzag, the direction will inverted for each next string. Otherwise, all the passes will be executed strictly in a predetermined direction.
- Safe level. Defines the level where long rapid motions it should perform.
- Short link max distance. Determines the distance above which the transition will be considered as long.
- Short link type, Long link type. Available variants: Straight, Via safe level. The way of link motions.
- Neighbor pass links on rapid. If it is disabled, then transition between neighbor passes will perform on working feed (with the supply of weldable material, activated burner, laser, etc.). When disabled, the link will on rapid feed, that is without melting the material.

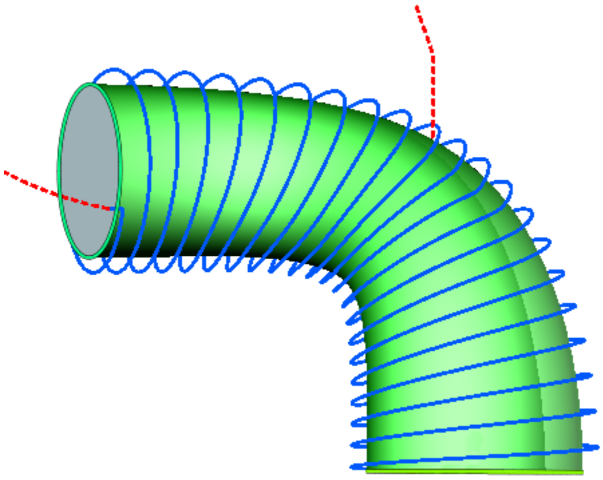
Then you can switch to the Simulation mode to see how the material is added to the place where the tip of the tool is touching the workpiece. The thickness of the layer in simulation defined by the working length property of the tool, or if it is not set (more than the tool diameter), the thickness will equal to the tool diameter.

See also:

Additive manufacturing

Cladding optional module

5.15.4 Cladding 5D operation



Operation "Cladding 5D" allows to increase material layer on the surface of a detail using strategies from "5d surfacing operation" operation, because this operation is based on it.

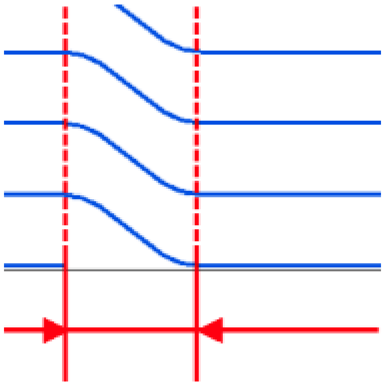
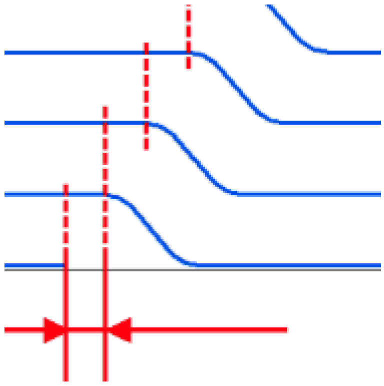
Two new strategies in addition to existing were added to the list:

- Spiral between two curves
- Spiral between two surfaces

To obtain spiral trajectories, preliminary passes are first generated, similar to the “Morphing” strategy from the “5d surfacing operation” operation. Then the chain of closed passages is transformed into spiral.

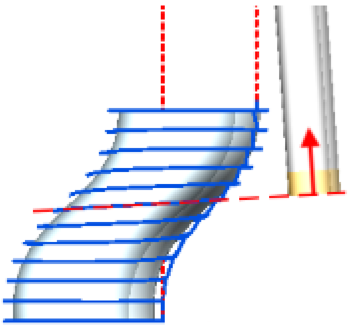
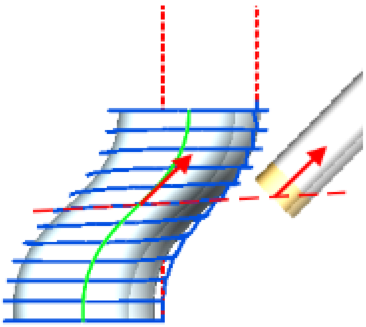
The thickness of the layer in simulation defined by the "Working length" property of the tool.

Added special parameters for toolpath modification to avoid several tool passes in one point. It allows better start/finish zones allocation and leads to better surface quality. These options are not available in spiral strategies:

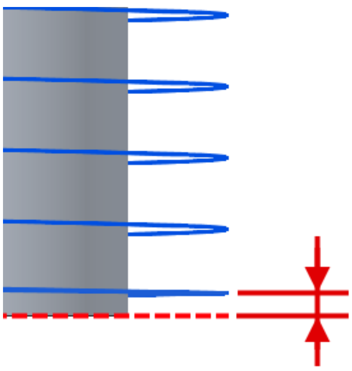
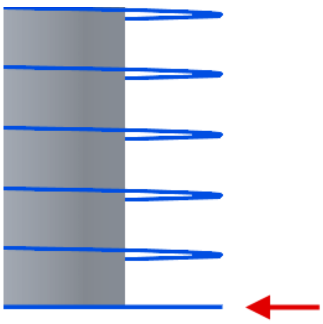
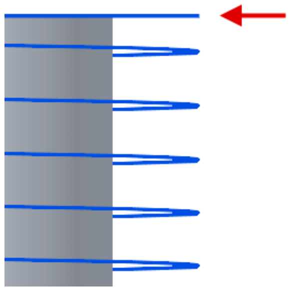
Gap for overlappnig avoidance	Start point offset(in the tab Links/Leads)
	
It allows to avoid the passage of the tool in the same place several times, as a result of which the height of layer can exceed the specified size. For this, the toolpath is shortened at the end of each curve by the specified value.	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.

In order to minimize altering tool normals during cladding, added strategies for Tool orientation: "Perpendicular to pass plane" and "Along curve", which set the normals relative to the median plane of the tool passes.

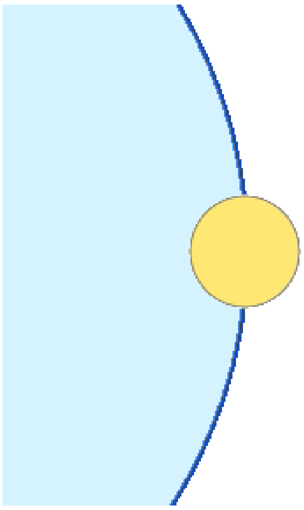
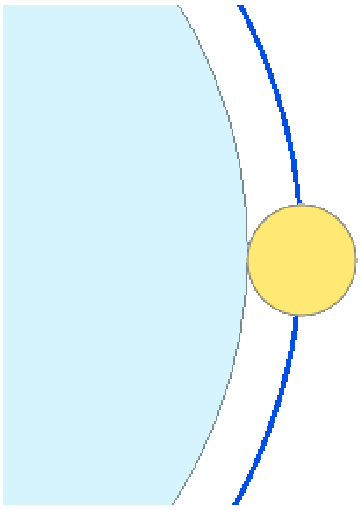
These strategies sets tool normal vector in reference to average pass plane.

Perpendicular to pass plane	Along curve
	
The tool orientation is set perpendicular to the middle plane of the pass	The tool orientation is set along the tangent to tilt curve in point where it intersects with the middle plane of the pass.

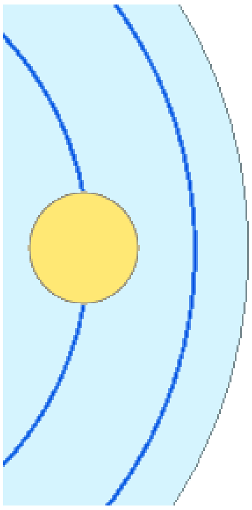
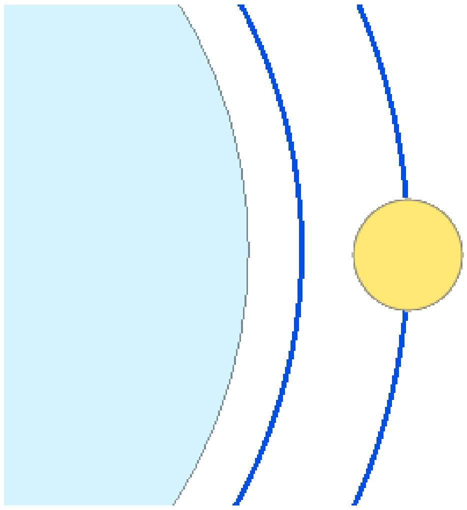
Additional passes allows to set specific machining conditions for initial and last layers: the height of cladding and the feed of the layer (on the feed page)

First step	First planar pass(for spiral strategies)	Final planar pass(for spiral strategies)
 The first layer will be executed with the specified step. It may be necessary, for example, to provide better adhesion to the substrate	 Before the ascending spiral, a complete pass will be made along the curve in the initial plane	 Spiral will be finish along the curve in the final plane

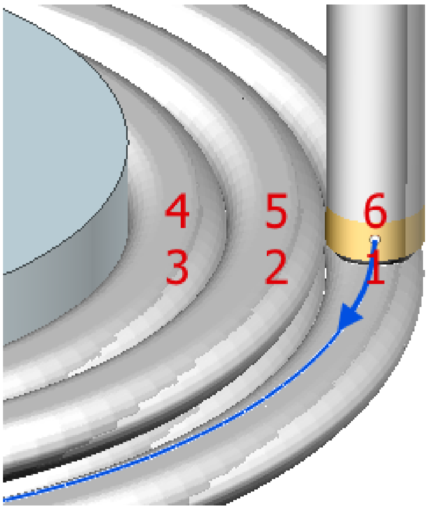
Offset job surfaces allows to disable the trajectory offset by the tool radius

 The trajectory will pass by the model	 Trajectory with considering the tool radius

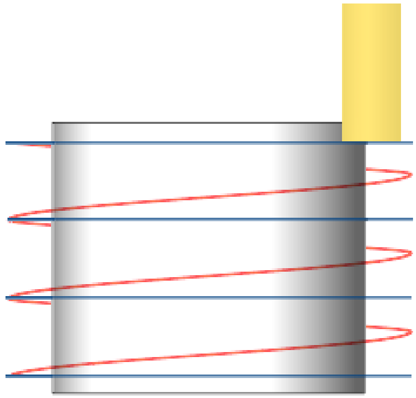
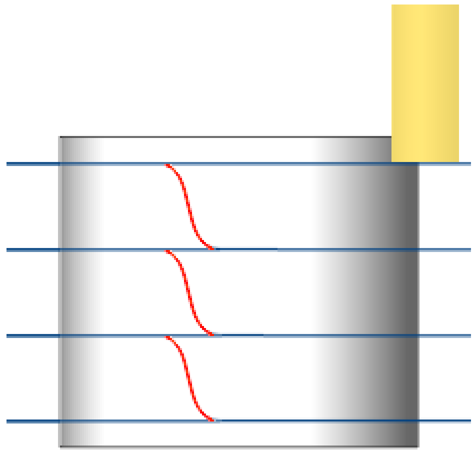
Cladding side allows to choose the side of cladding

Inside	Outside
 The toolpath is built inside the job assignment	 The toolpath is built outside of the job assignment

Added the following options to **Roughing passes:**

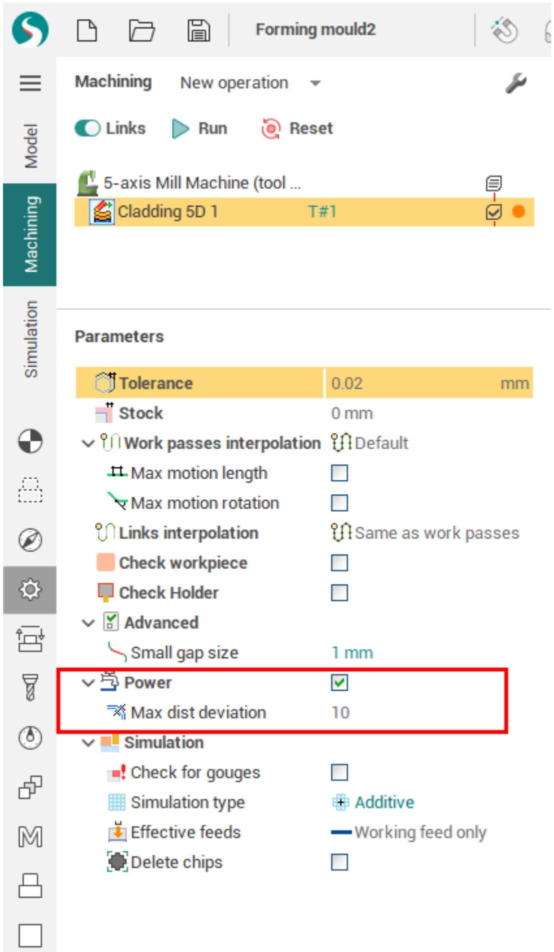
The alternation of the order of passes between the layers (Zigzag)
 It can help to get the toolpath without breaks, i.e. without having to switch off the feed of the material and transition on the rapid feed.

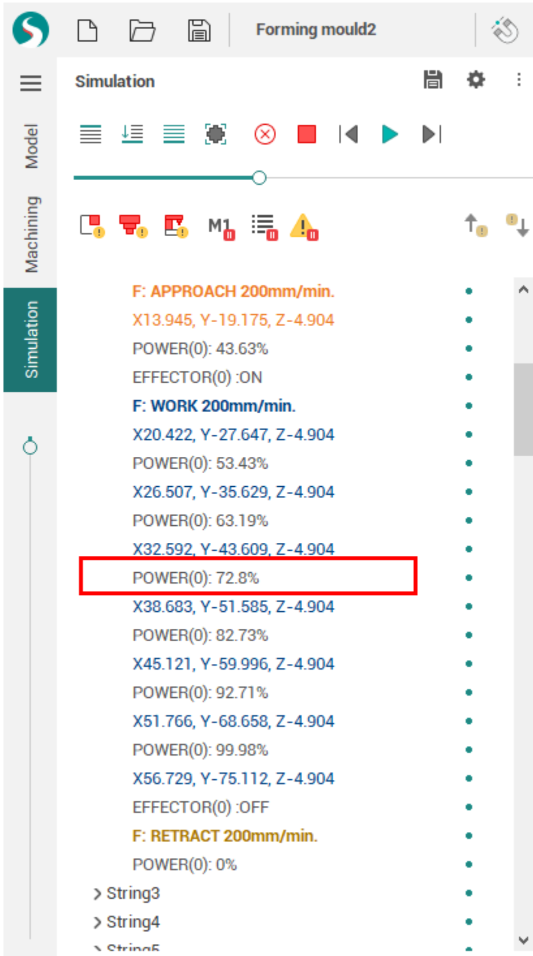
In spiral strategies for smoothing between layers in roughing passes, you can use the following options (only for Zigzag mode):

Spiral	Smooth
 Adds a spiral pass between the layers.	 Produces a smooth transition between layers. In order to avoid tool passing in the same place, the parameter **Smoothing distance** is used.

Option "**Power**"

This option allows you to change the speed of the extruder depending on the height of the layer.





See also:

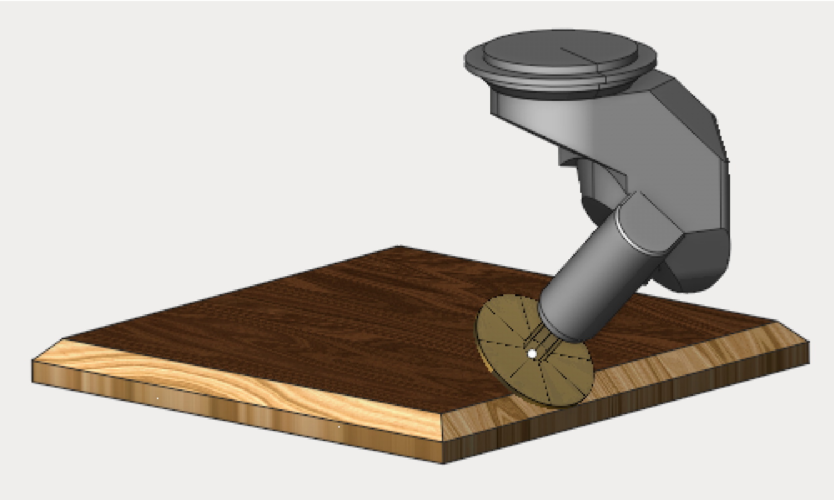
Additive manufacturing

Cladding optional module

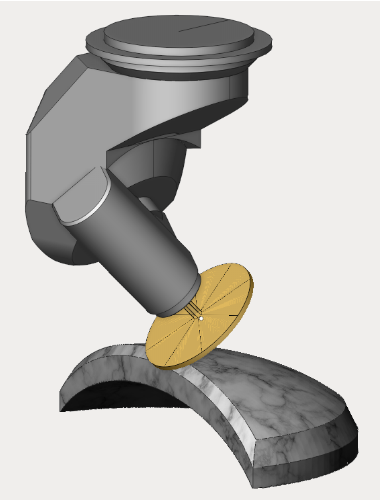
5.16 Disc tool machining

This group of operations is designed for sawing and machining materials such as wood, stone and similar with a disk tool.

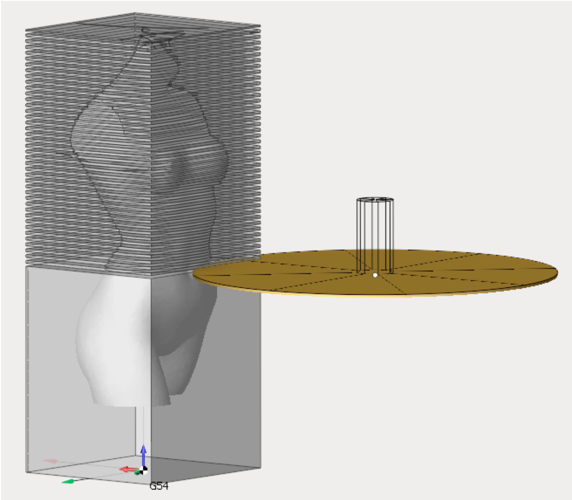
The **«Disc cutting 2D»** operation is primarily intended for sawing sheet materials, machining flat furniture facades, plates.



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.

5.16.1 **See also:**

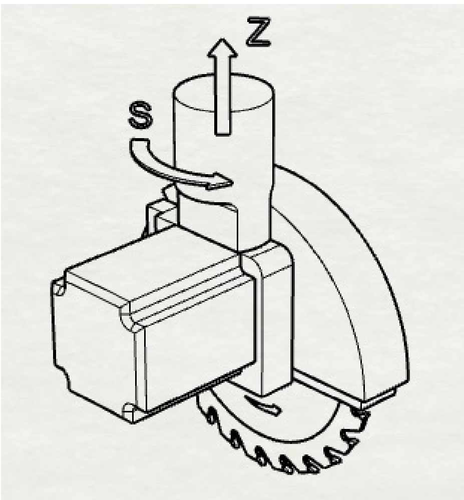
- Disc tool overview
- Disc roughing
- Disc cutting 2D
- Disc cutting 6D

5.16.2 Disc tool

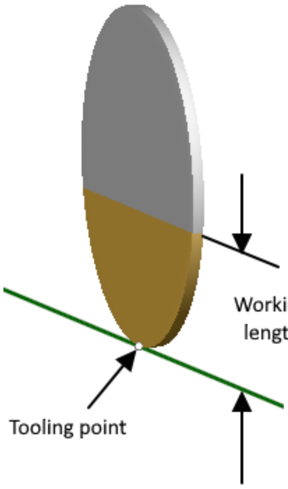
Two tool types are available for disc tool operations depending on the type of mounting.

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.

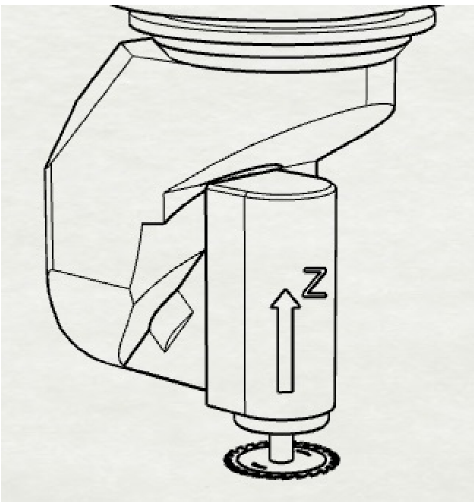


The tooling point of a saw blade is located on the edge of the saw unlike a milling tool, where this point is located on the tool end on its rotation axis. Saw blade type tools have also 'Working length' parameter which defines the height of saw's working area. The rest of the tool is assumed to be covered 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.

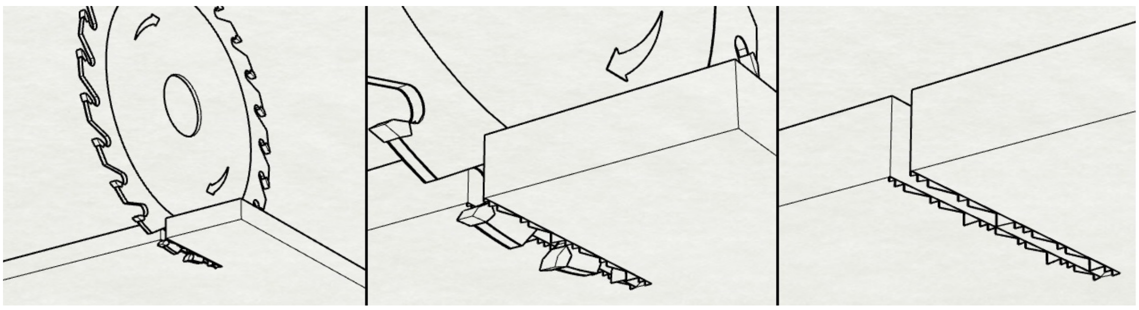


Milling Tool.

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).



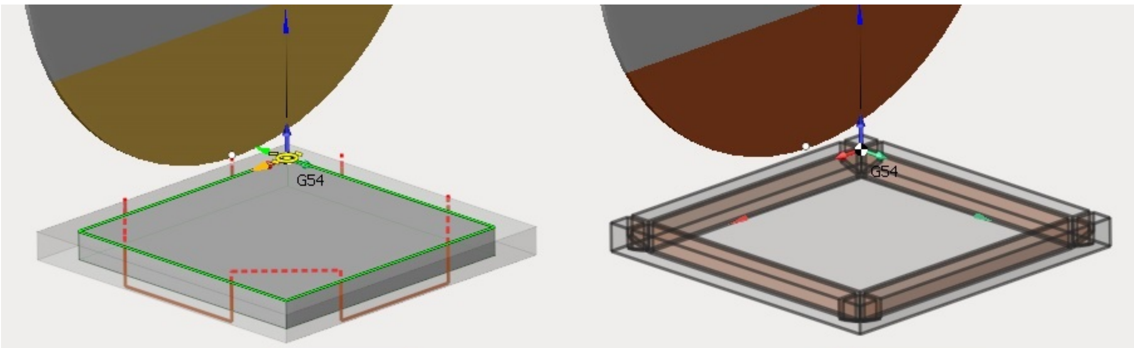
To avoid this situation (lets consider two pass variant) first saw pass is done without getting out of material on the panel's bottom level (as such ensuring that top edge contains no defects because panel's top level will be the zone where saw's teeth enter the material), while the second pass is done 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.

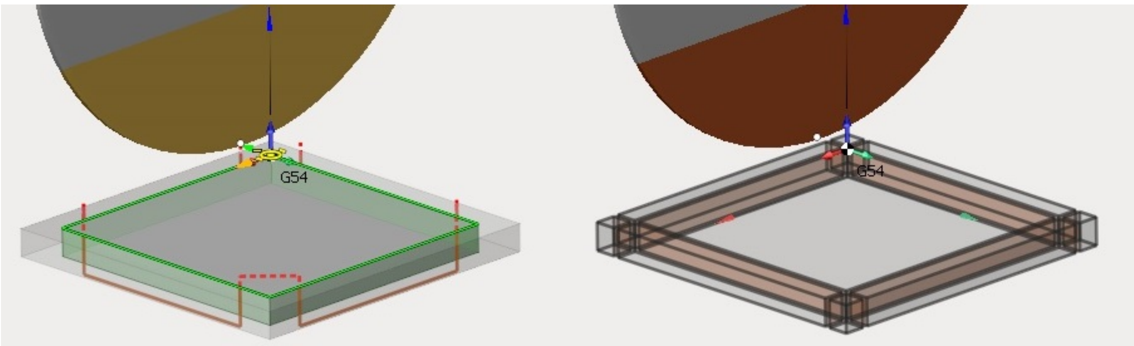
Outer corner overlap allows to increase the length of the contour along the faces for the specified amount. It can be necessary for removing workpiece chips along the faces of contour.

Lets consider an example. Suppose its needed to saw rectangular panel from the workpiece with stock.

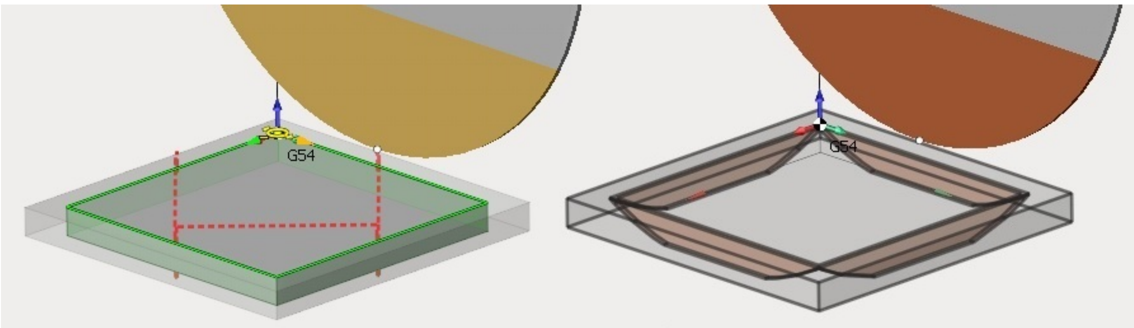
If the "Outer corner overlap" is not used, then material chips will be removed from part only after all faces are traversed by the tool, because the trajectory is formed according to the specified contour, not accounting for the workpiece stock.



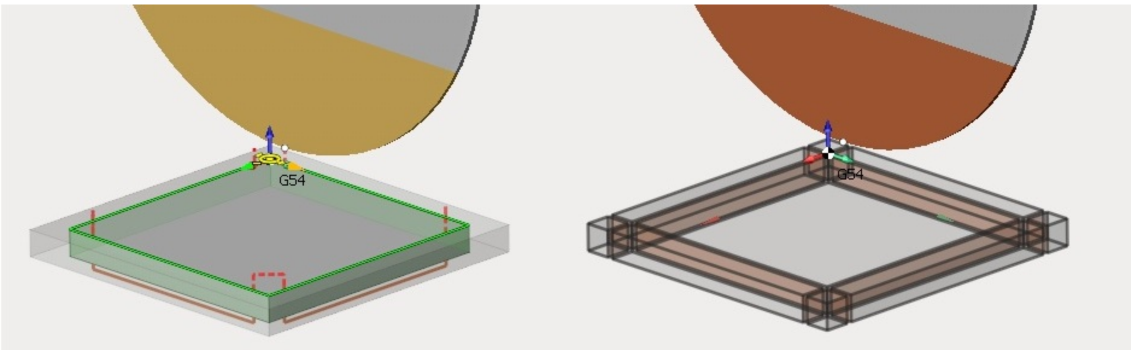
If overlap is used then material chips can be cut after a face is finished.



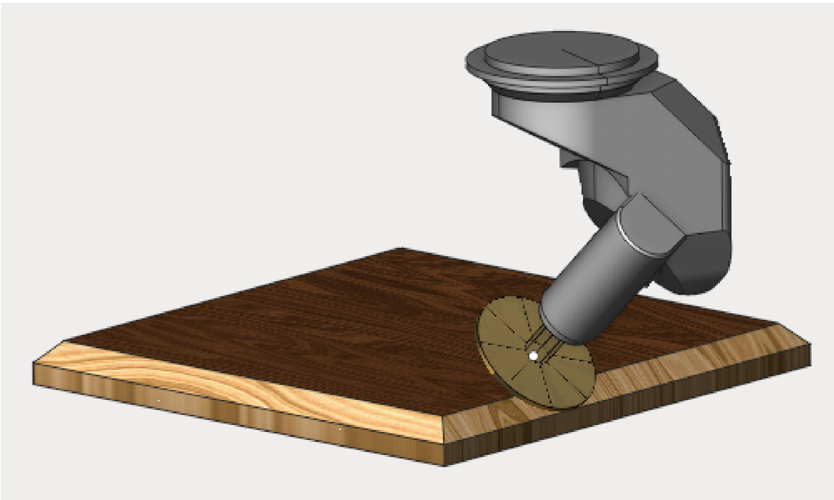
Another objective is during inner contour machining. Without changing overlap parameters toolpath trajectory will be limited by the saw profile touching the contour corner on workpiece top level.



Using overlap full contour of the part can be achieved on both top and bottom workpiece levels.



5.16.3 Disc cutting 2D



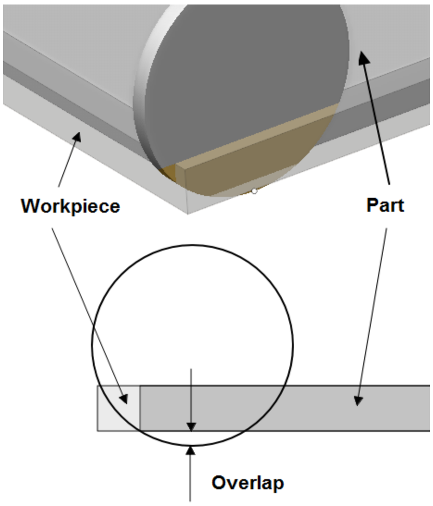
The operation is designed for the sawing of sheet materials. The most parameters of the **disc cutting 2D** are the same that **2D contouring** operation has. The job assignment can be defined by the planar curves (profiles) or edges of the 3D model.

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.

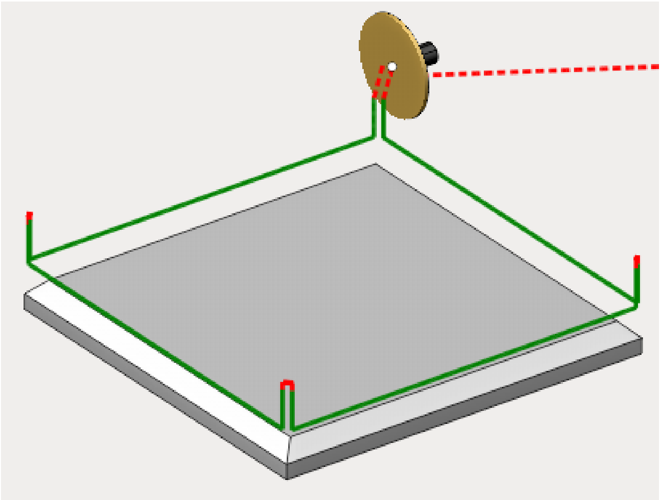
With the “Job assignment” tab open and given curves, you can also interactively select the direction of movement of the tool and its location relative to the curves (left/right) by left-clicking in the graphics window on the arrows on the curves.

Besides that the following parameters are available in the inspector

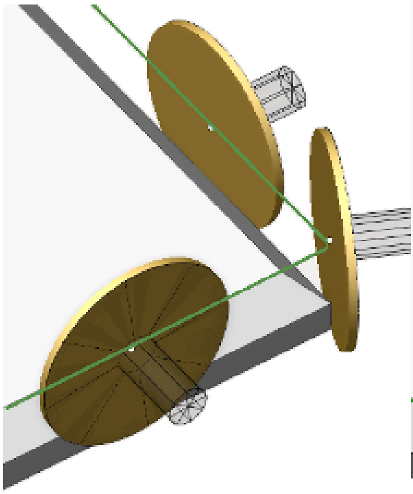
- **Overlap**. The parameter defines exceeding of the workpiece bottom level by the saw blade.

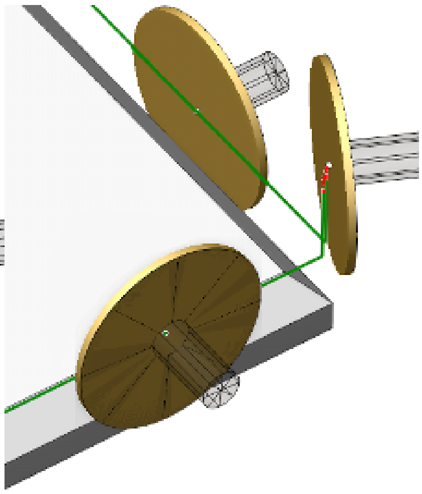


- **Overlaps in corners**. The parameter allows to specify the height on which the tool overturn will be done when changing side of the machined corner.

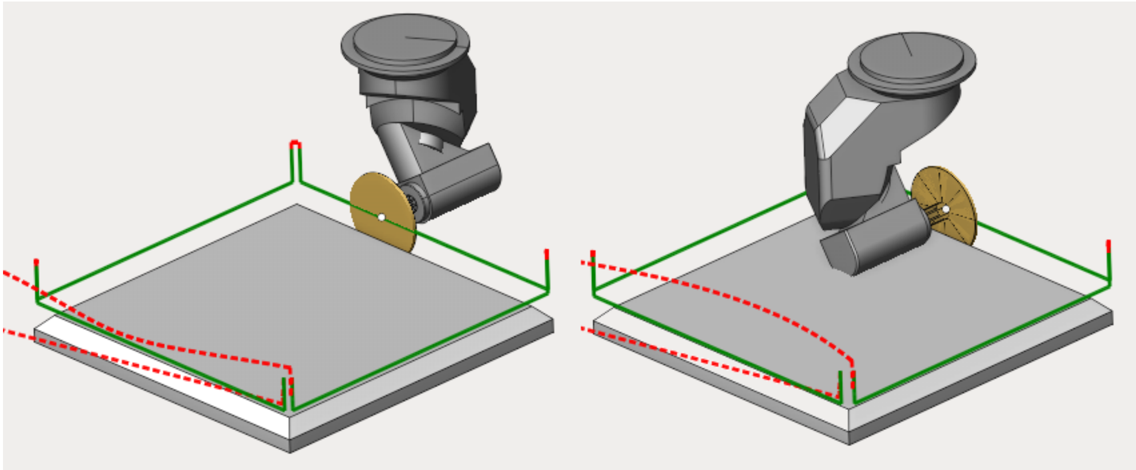


- **Sharp corners**. The parameter defines the angle limit for considering the corner sharp and during machining of which tool return is needed. On the image below is the passing the right angle (in the first case the sharp corner value is less than 90 degress, in the second one - greater).

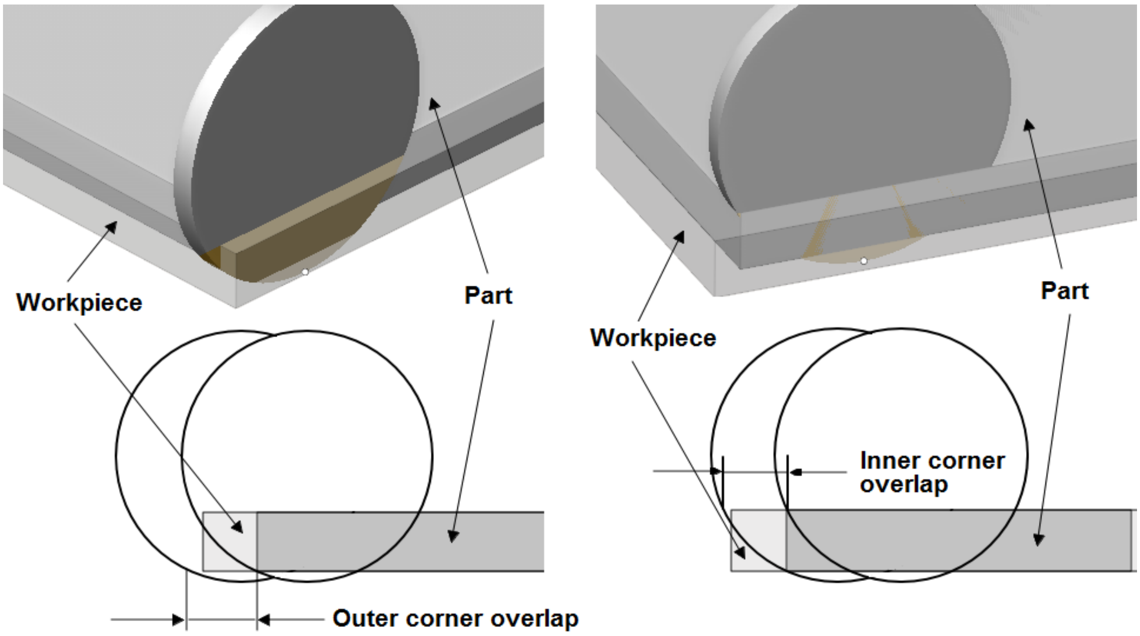




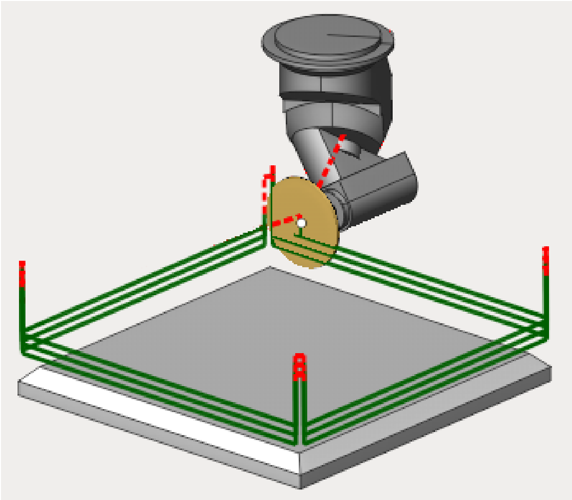
- **Sawing mode**. It defines the sawing type - climb, conventional, back or front.
  Climb/conventional - ensures the respective cutting mode. When changing the contour direction, the saw will overturn and start to machine the part with the opposite side.
  Back/front side - the saw always machines the part using the specified side, regardless of the direction of the contour.



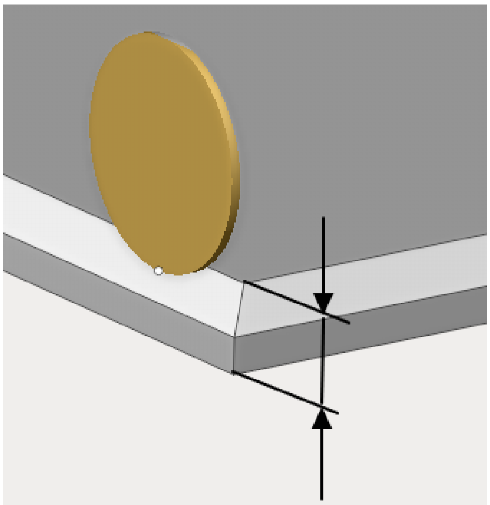
- **Overlaps**. Parameters specifying overlap value for outer and inner corners respectively.



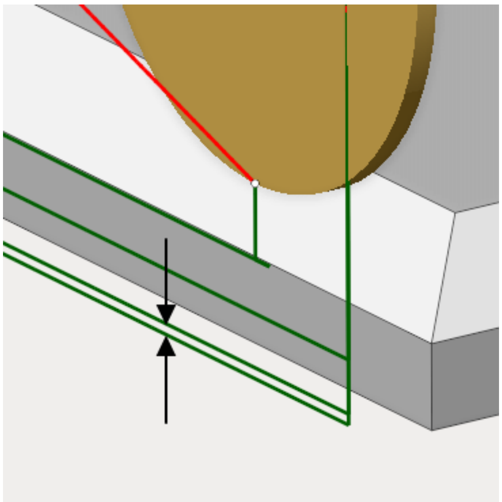
- **Layers (depth)**. Defines the number of tool passes. Can be specified as a number of passes or automatically calculated from top and bottom machining level. Can also be specified in mm or as a tool diameter percent.



- **Top and bottom part level** - define the height of the part to machine.

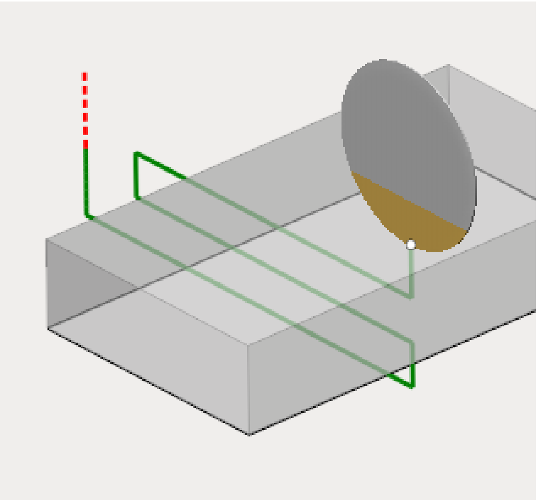


- **Z cleanup**. The parameter defines the stock value before last pass.

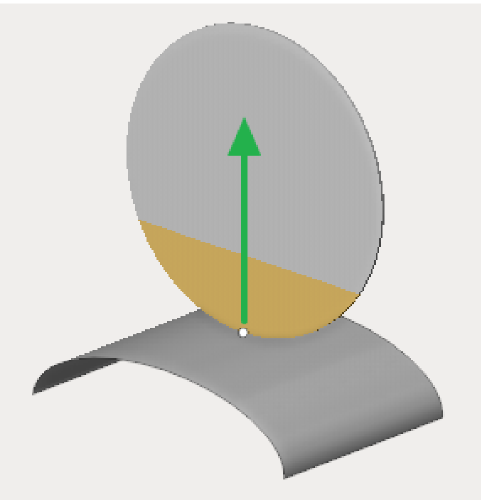


- **Sorting**. When machining by layers after performing the first pass along the first contour, a transition is made to the first pass along the next contour. The transition to the second pass is made after all contours have been processed. When machining by cavities, at first all passes along the first 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.

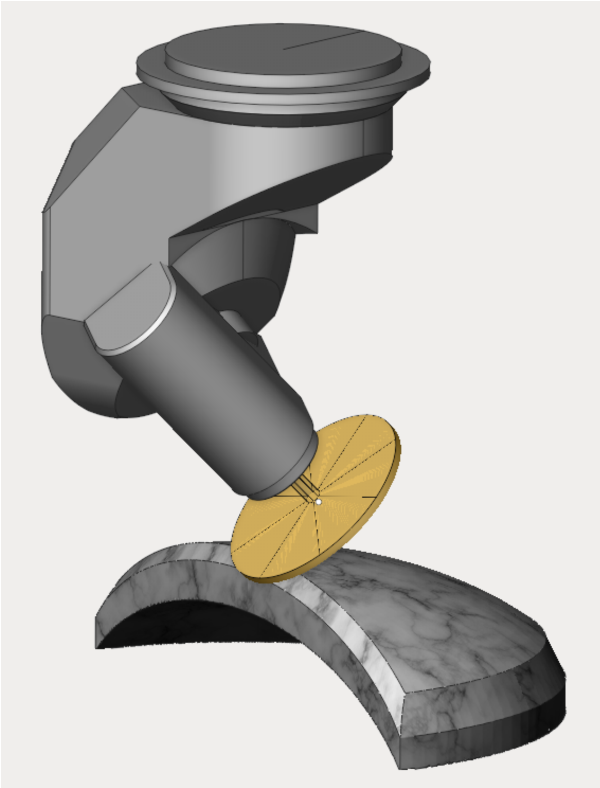


- **Tool axis orientation**. The parameter is used to determine the orientation of the tool axis on any part of the path. The direction of the axis can be fixed, normal to 4D and 5D surfaces, as well as normal to the base surface. It should be noted that, unlike standard milling cutter, the axis of the **Saw blade** type tool does not coincide with its rotation axis.



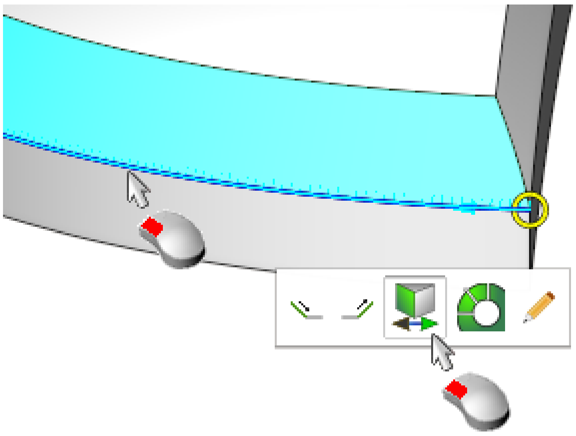
5.16.4 Disc cutting 6D

Operation "Disc cutting 6D" is designed for sawing both flat and volumetric parts.

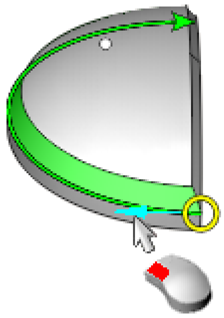


The interface for setting parameters and job zones is similar to the "5D Contour" operation.

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.

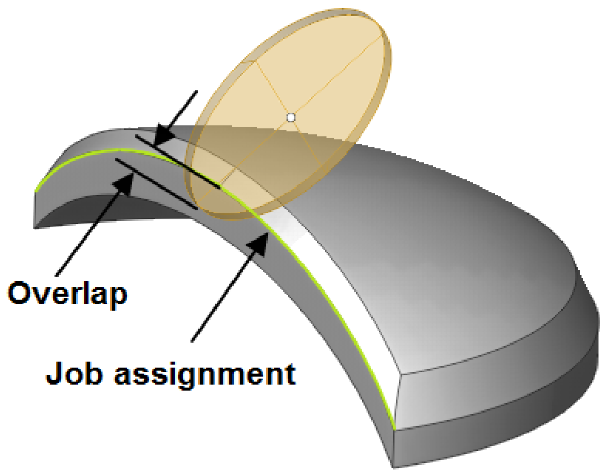


When the “Job assignment” tab is open and the feature was added, you can also interactively select the direction of the tool movement along the curve by left-clicking the arrow in the graphics window.

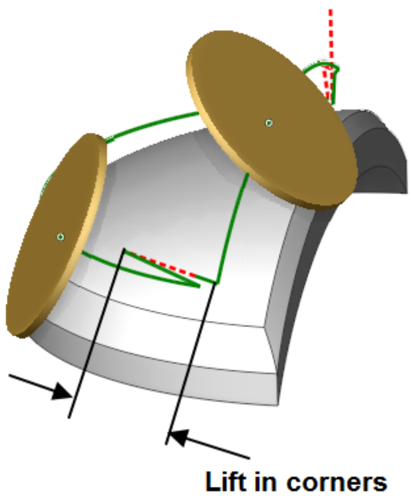


The following operation parameters are available in the inspector ("Strategy" tab).

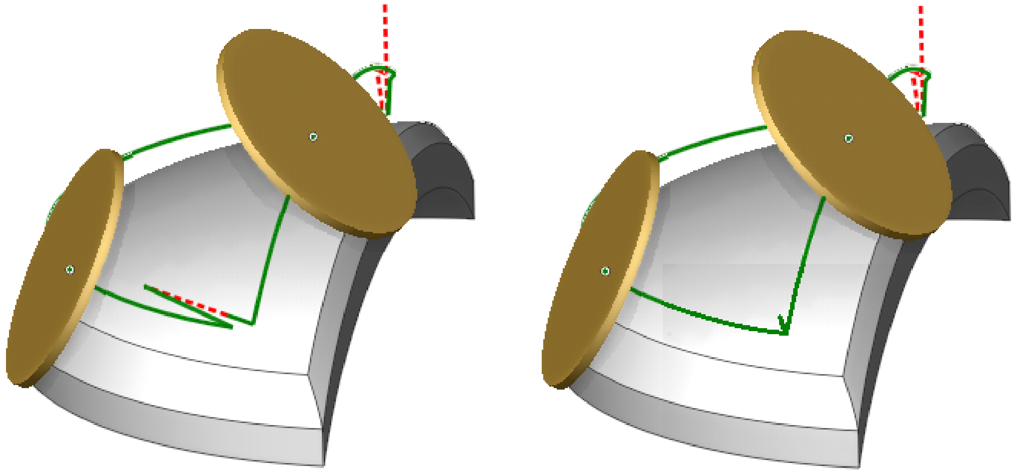
- **Overlap (deepening).** The parameter allows the saw in the end to move past the level defined in the job assignment (by an edge/curve).



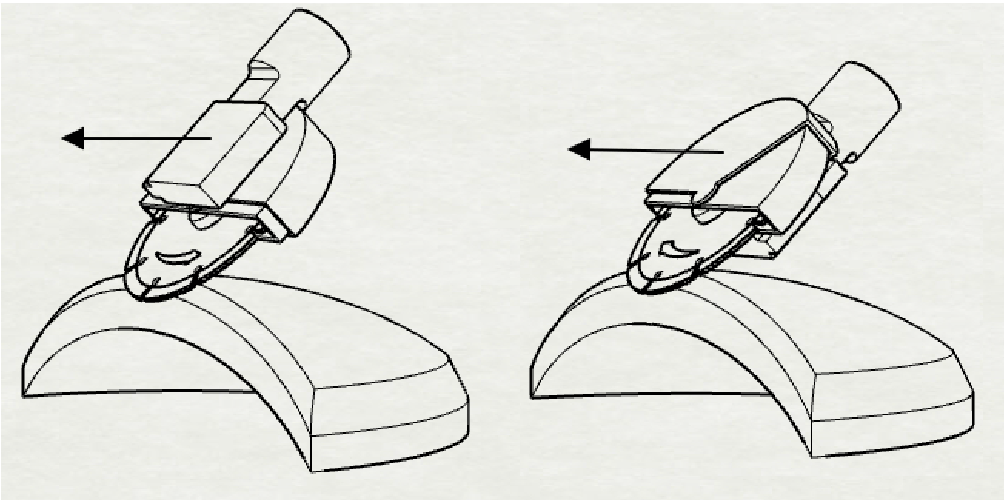
- **Lift in corners**. The parameter allows to specify the height where the tool overturn will be performed when changing sides of the machined corner.



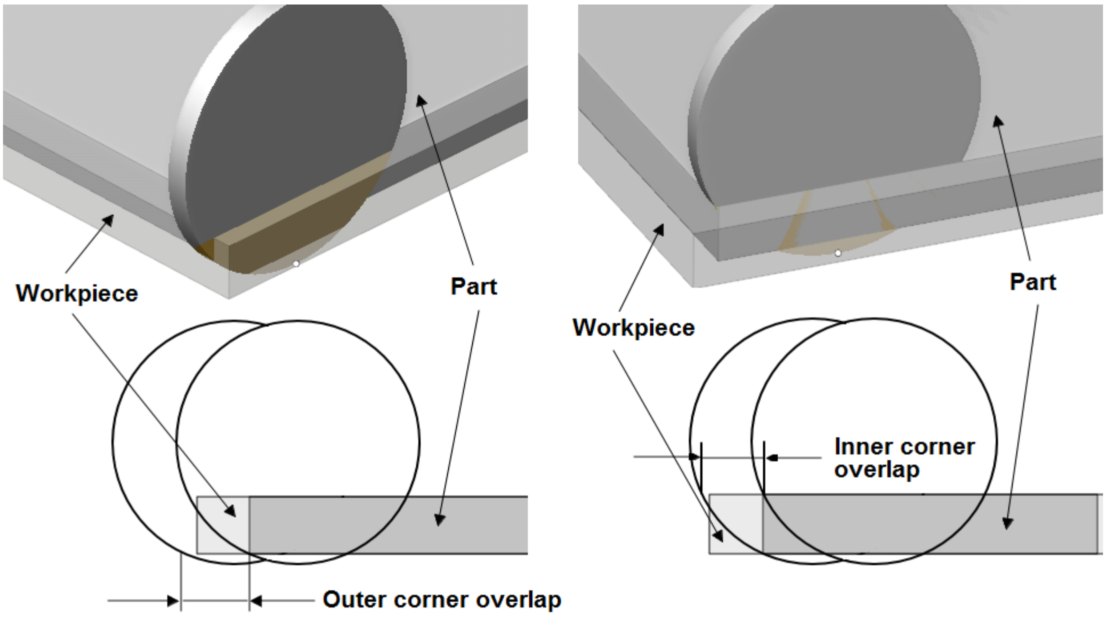
- **Sharp corner** defines the minimum value of the angle to consider a corner sharp, i.e. when tool overturn is necessary. On the example below two different trajectories around a corner are shown (in the first case sharp corner value is less than 90 degrees, in the second - more than 90).



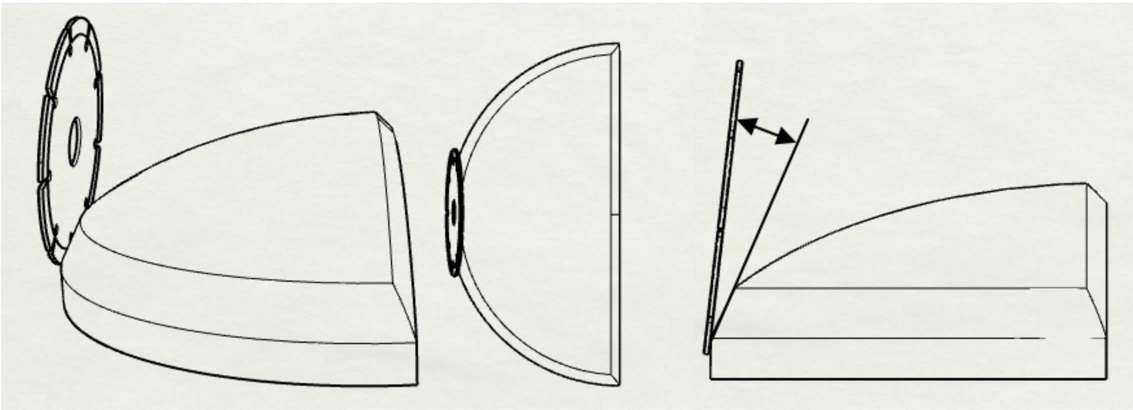
- **Sawing mode**. It defines the sawing type - climb, conventional, back or front.
  Climb/conventional - ensures the respective cutting mode. When changing the contour direction, the saw will overturn and start to machine the part with the opposite side.
  Back/front side - the saw always machines the part using the specified side, regardless of the direction of the contour.



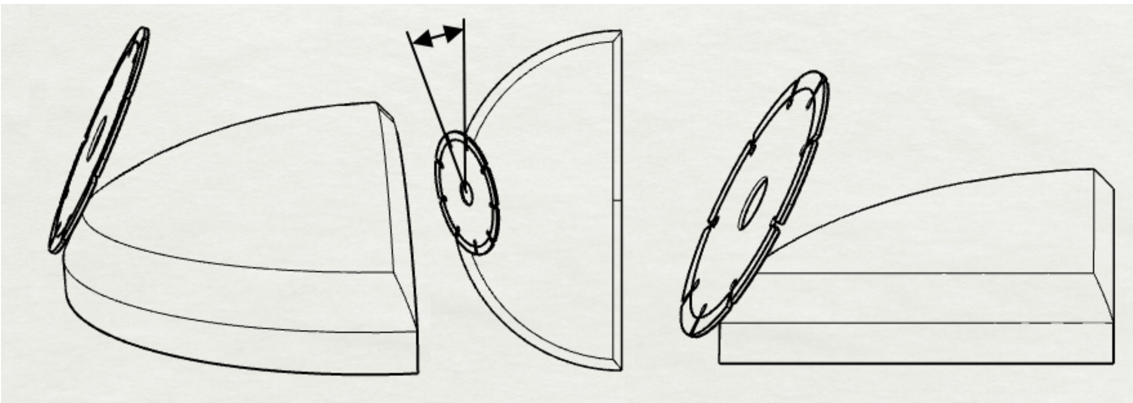
- **Overlaps** - parameters that determine the distance of overtravel for the outer and inner corners (outer and inner corner overlap respectively).



- **Lean angle**. The parameter defines the deviation angle of the saw's side surface from the machined face.

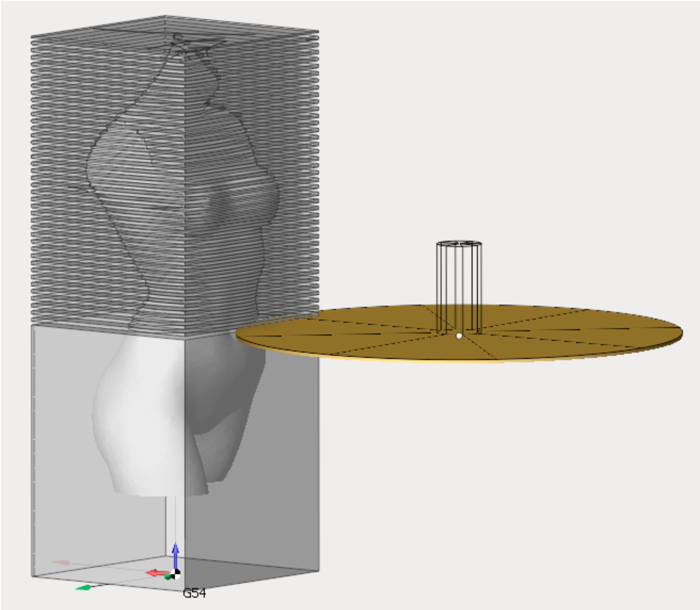


- **Lead angle**. The parameter defines an additonal angle between the surface of the saw and the tangent to the work contour at each trajectory point.



5.16.5 Disc roughing

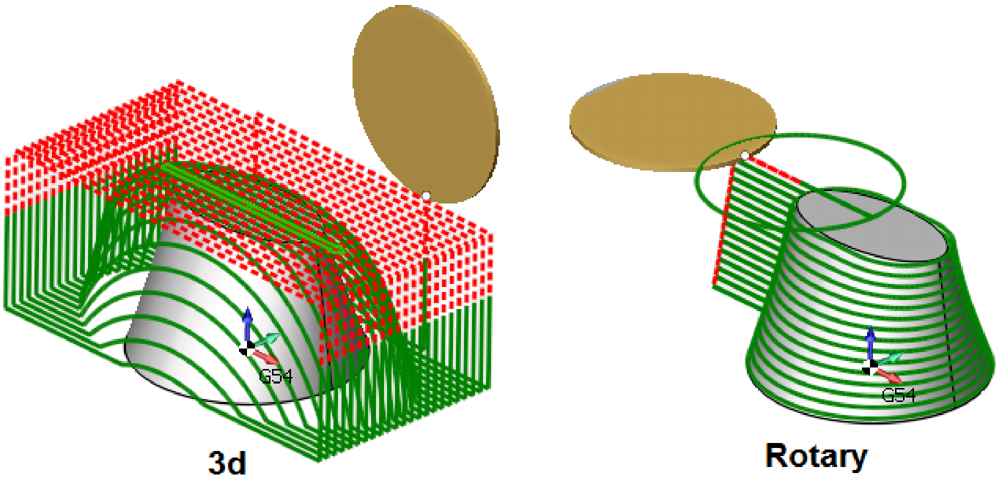
The Disc Roughing operation is designed to prepare stone material by making disc tool cuts in the material remove area and the following manual removal of thinned out material by spalling method, and after that usually finishing operation follows using the respective tool.



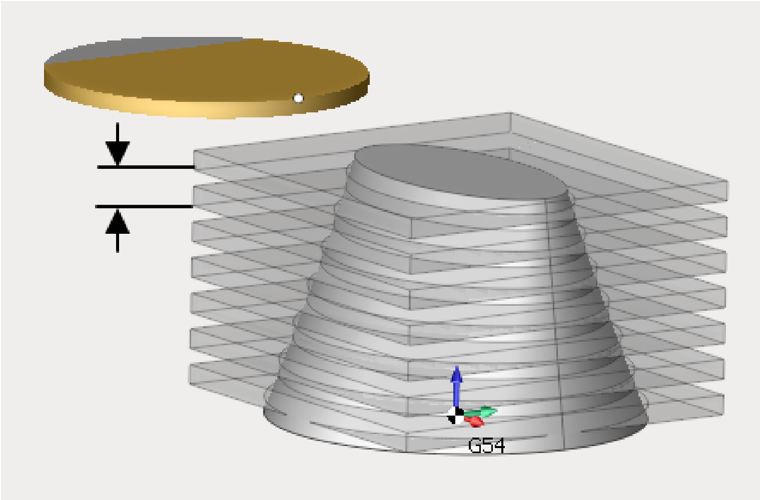
The imported geometry model is used automically as a job assignment. To limit top and bottom machining level you can select any geometrical item located on the required level and press the "Top level" or "Bottom level" button.

Besides that the following parameters are available in the inspector 'Strategy' tab.

- **Trajectory**. The parameter defines the way of trajectory forming - 3 dimensional (3D) or relative to axis (rotary).

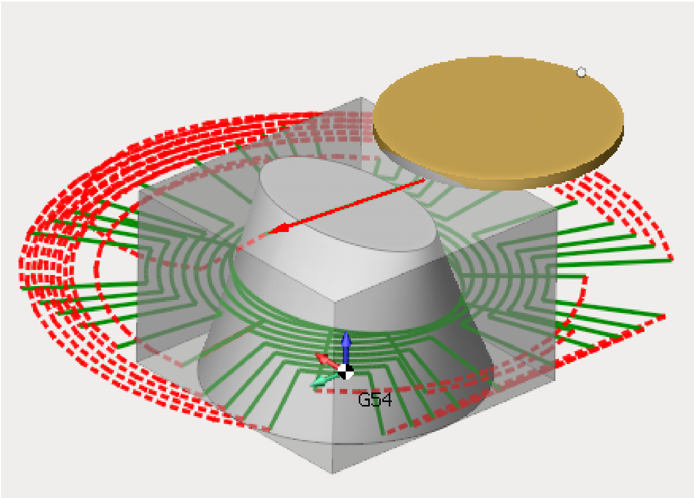


- **Step**. The parameter defines the step value between removed material layers.

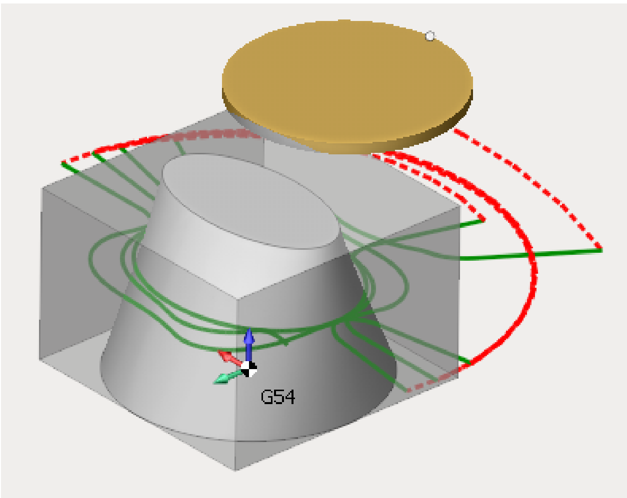


- **Strategy**. The parameter defines the strategy of toolpath forming.

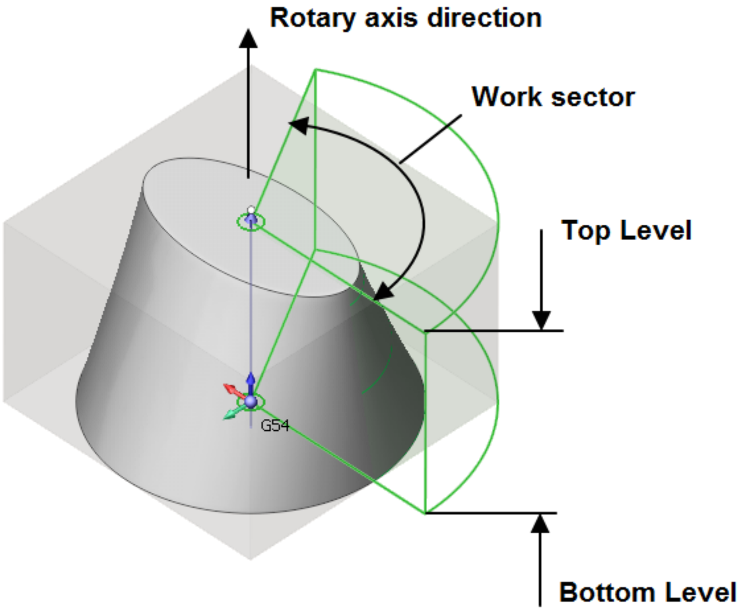
By layers. Constant distance between the tool passes.



Adaptive. In this strategy not exceeding given cut width is ensured. Material is removed by helics. The trajectory contains no sharp corners. The smoothness of trajectory is controlled by special parameters for rounding radii for roughing passes, finishing pass and transitions. Idling, if possible, is done in the working plane with a small additional gap. Plunging inside workpiece is done with tool preserving rolling technology.



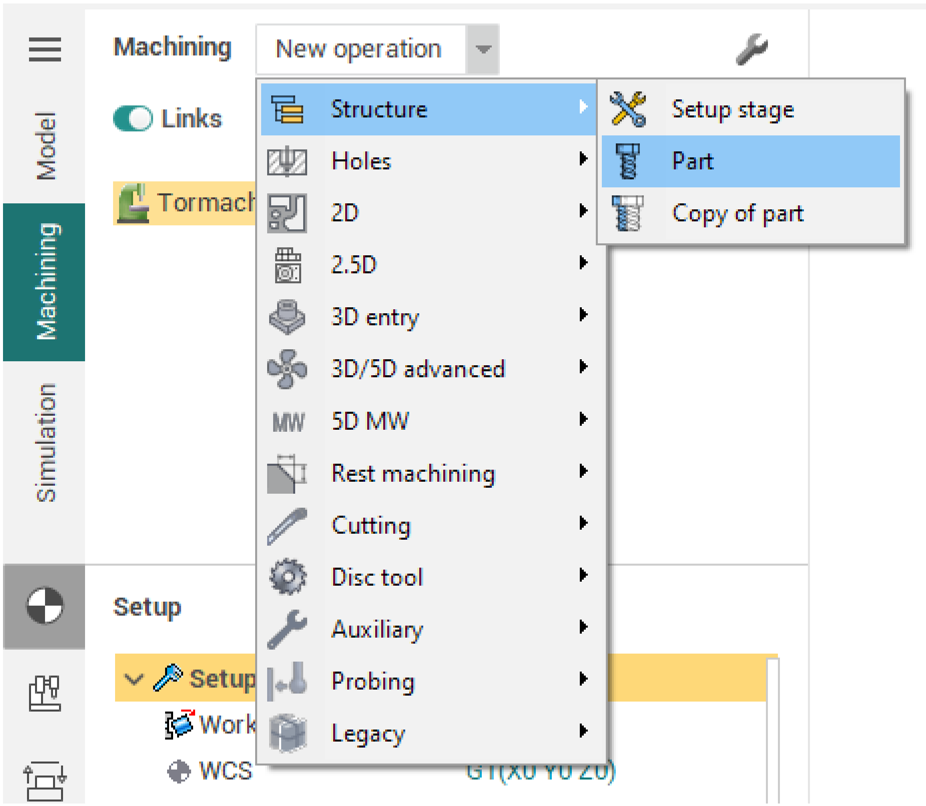
- **Job zone**. This group of parameters defines top/bottom machining level, machining direction and the machining sector.



5.17 Multi parts projects

5.17.1 Part as a group of operation

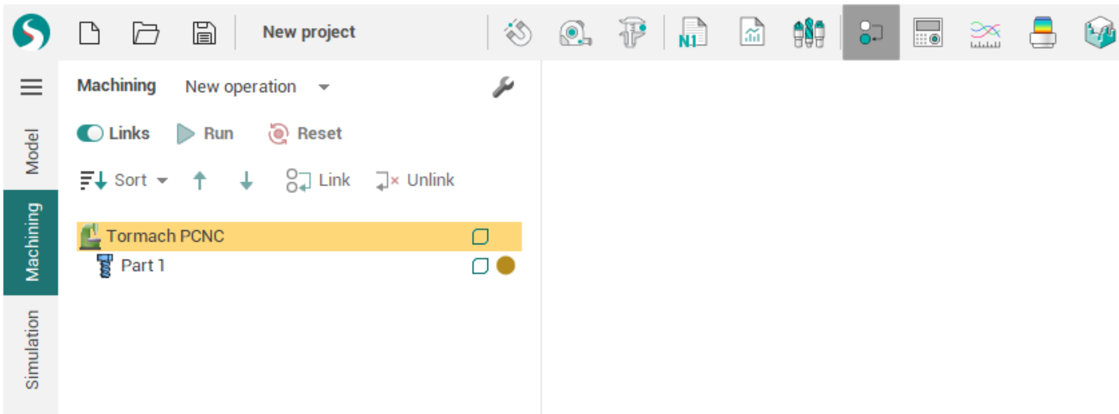
If you want to prepare the project with the several parts you need to create the special group of operation named as “part”.



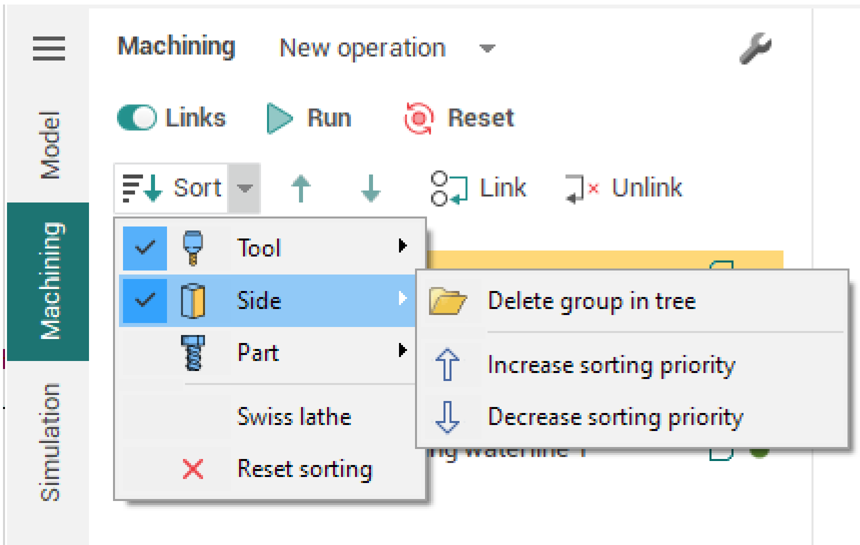
In the CAM tree the **part** group is always located inside the root (machine) or inside the **setup stage**, if it was created. If at least one **part** was created then all machining operations will be created inside the parts. All the operations inside the part group are working over one part. Usually for the part group it's neccesary to define the specific part geometry, workpiece, fixtures, setup location and origin. Operations inside will use these parameters. as the common parameters of the part.

5.17.2 Sequencing mode

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

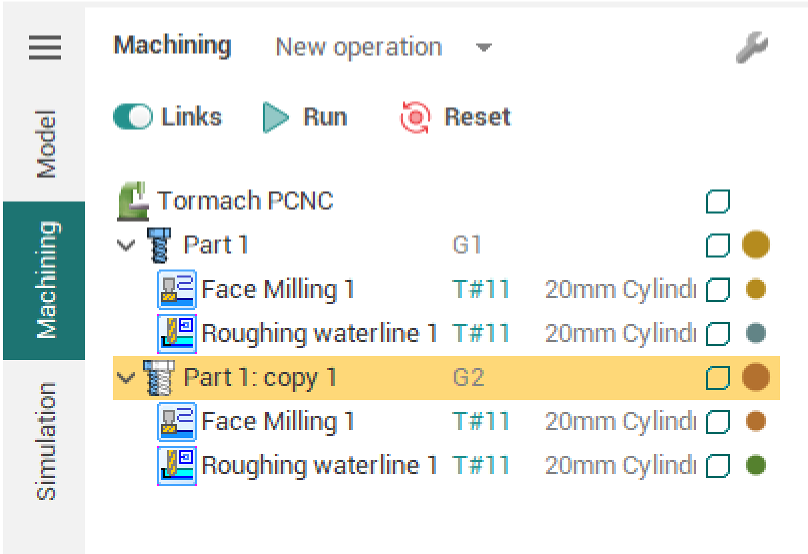


The links between the operations can not be calculated before the real machining order is set. Therefore the links calculation was separated from the main calculation process. Full workflow for the multipart projects contains the next steps:

1. Placements of the part in the work space of the machine
2. Design of the the machining process for every part. Calculation of the tool path without the links (approaches and returns).

3. Simulation of the process with the disabled sequencing mode, to check the correctness of the workpiece change etc.
4. Reordering of the operations in the **sequencing mode.**
5. Calculation of the links between the operations.
6. Simulation of the process in the sequencing mode to check the links and the tool changes between the operation.
7. Tool path generation

5.17.3 Part copies

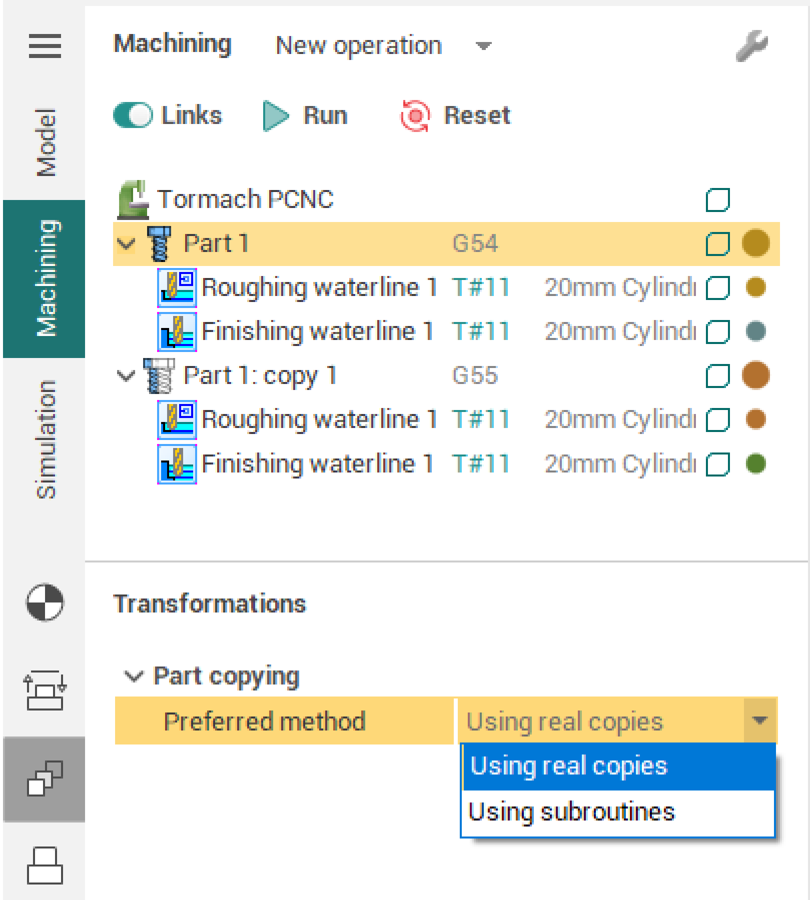


Copy of part is the special kind of simple part. It's designed to prepare the project with the several identical parts. Usually it can be done with the next steps:

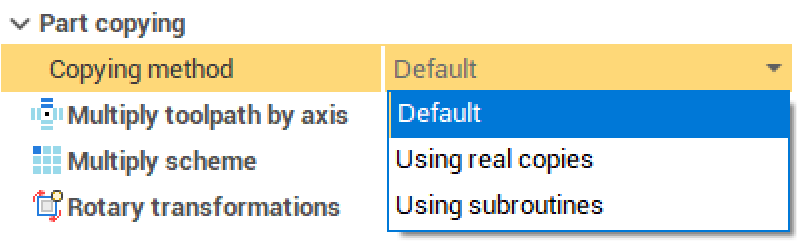
1. Create the prototype **part**. Add operations, that are neccesary for the machining.
2. Create **copy of part**. If the project contains more than one part then the copy of the current selected part will be used as prototype.
3. Assign the **workpiece setup** and **workpiece CS** for the copy.

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.



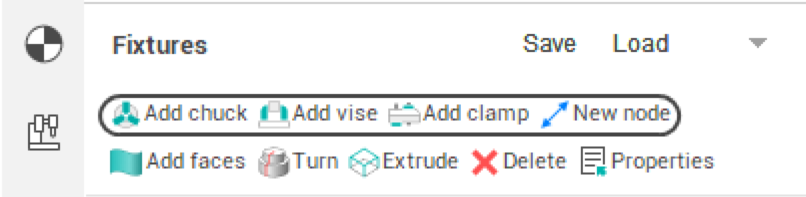
You can set individual copy parameters for each operation in the group.



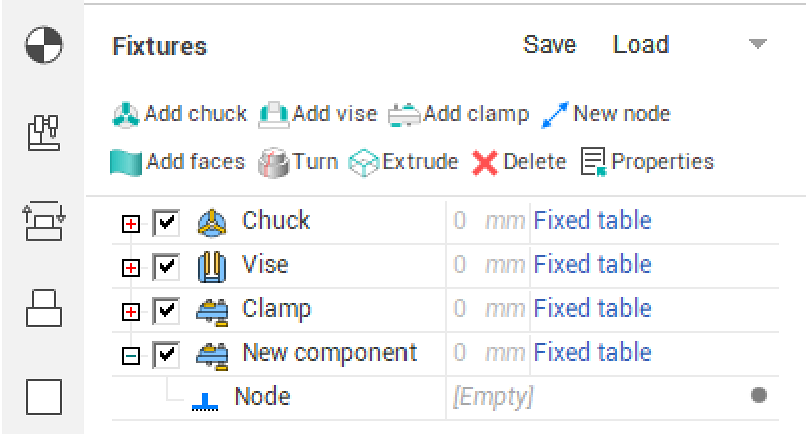
5.18 Fixtures

5.18.1 Creating a new fixtures

To create equipment, you should select an existing template or create your own version of equipment using the "New node" button.

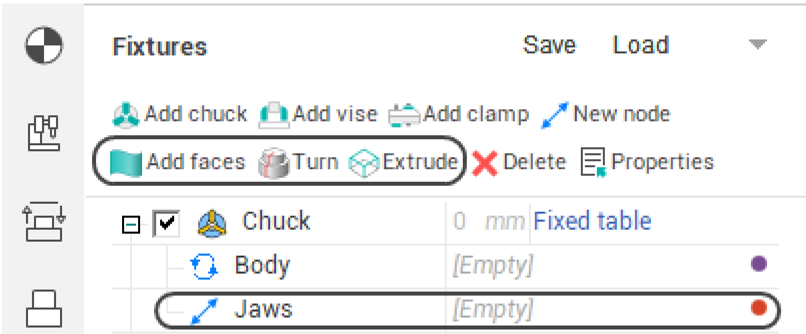


The newly created template and nodes are seen in the equipment tree.



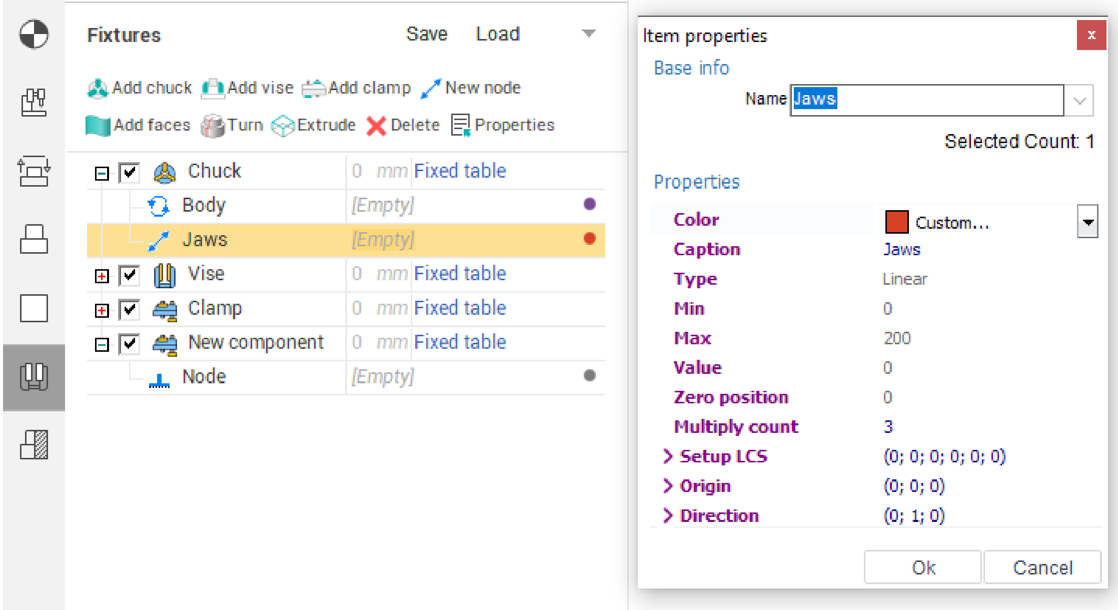
5.18.2 Geometry

To add geometry, select an equipment node and click one of the geometry buttons.

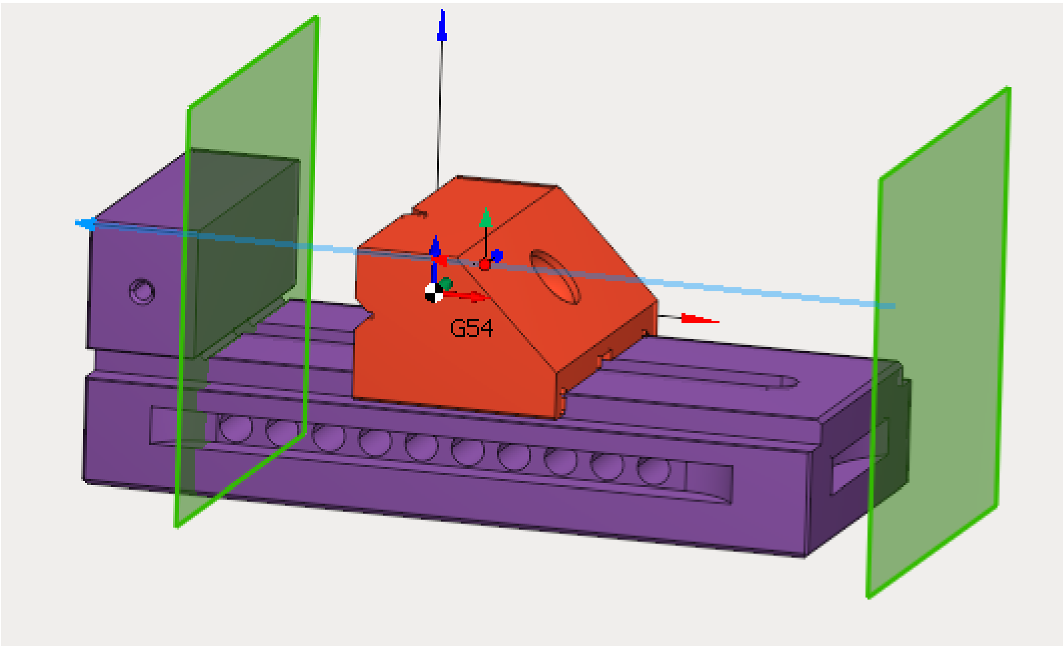


5.18.3 Configuring node parameters

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.

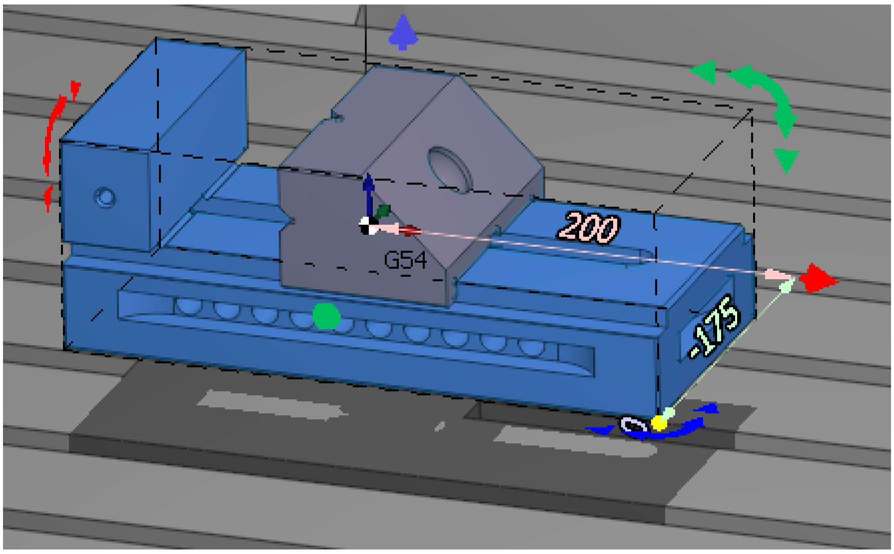


The main parameters of the node, such as the maximum (minimum) position, direction and rotation axis, can also be specified using visual objects.



5.18.4 Component setting

The position of the component is adjusted using a visual object.

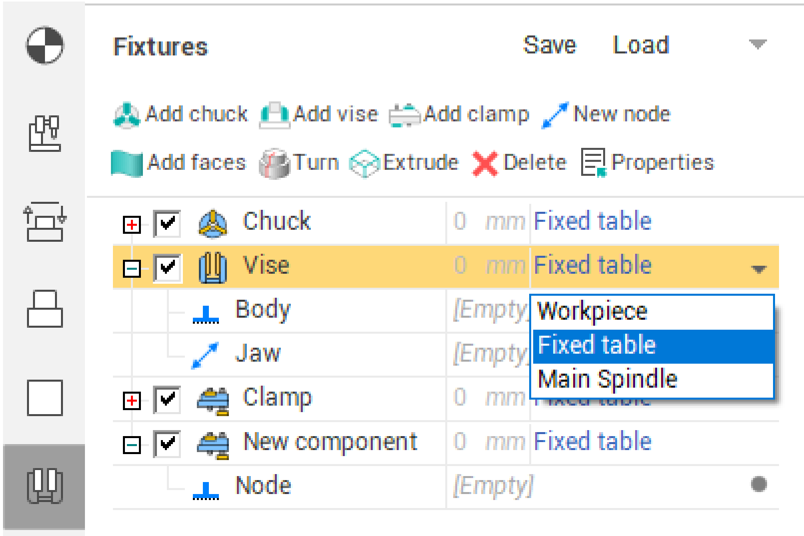


It can be invoked by double-clicking on the node or by one-click in component edit mode.  .

When you right-click on a visual object,  panel is showed up. It allows you to copy and delete components.

5.18.5 Snapping a coordinate systems

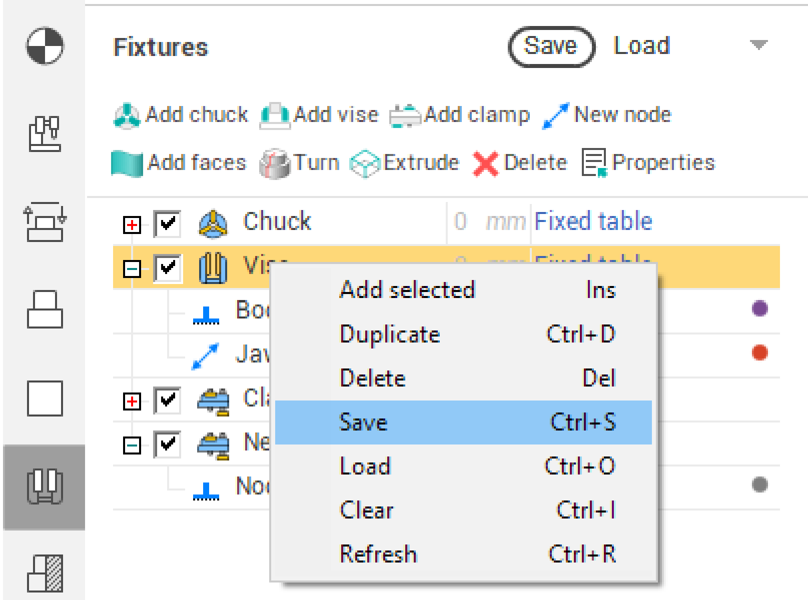
If your equipment is defined in a machine unit, then it is linked to a specific setup and moves with the machine.



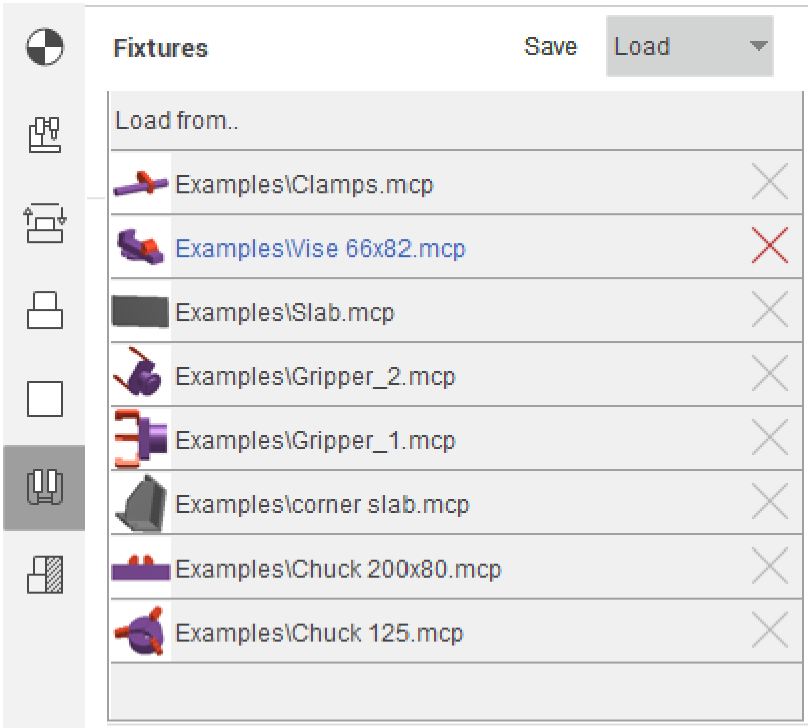
If it is in the operation node, then it is linked to the current setup and can be moved both with the machine tool and with the part.

5.18.6 Saving and loading

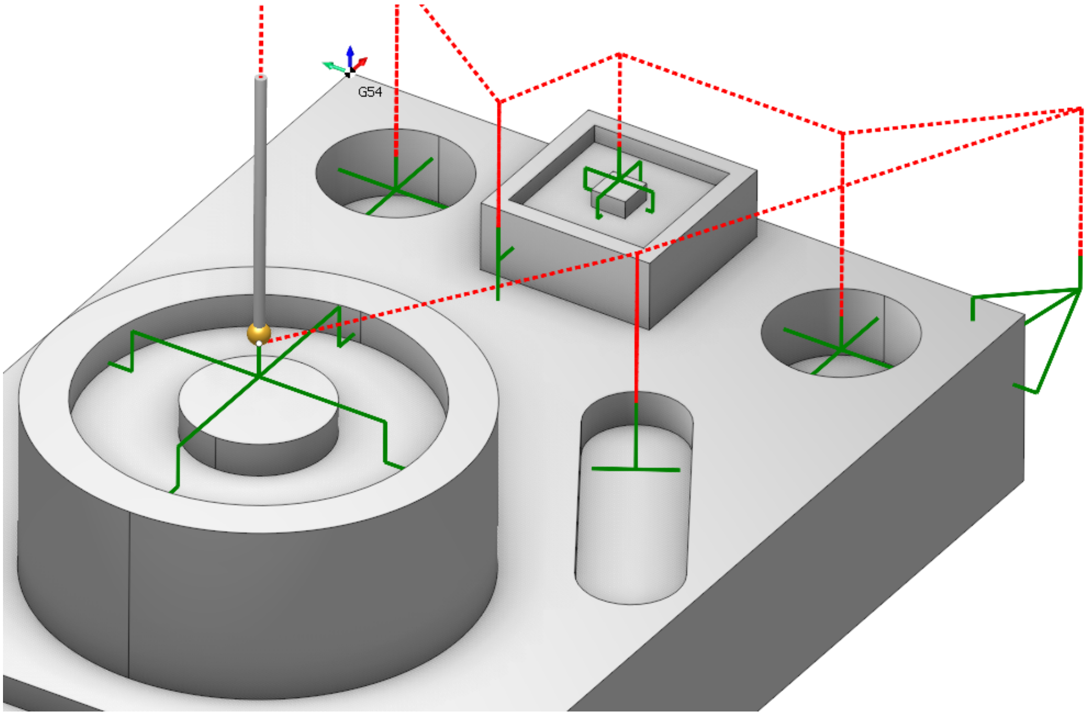
Equipment can be saved by right-clicking in the Component Tree or by clicking the Save button.



Hardware can be loaded by right-clicking in the Component Tree or in the drop-down panel of the Load button.



5.19 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:

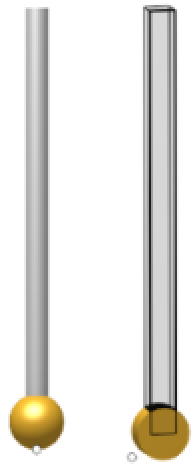
- control of the dimensions of critical surfaces of parts with output to the report;
- measurement of a part to compensate for inaccuracies in the location of the part and the geometric dimensions of real parts;
- measurement to detect inaccuracies in tool sizes;
- tool breakage control.

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).

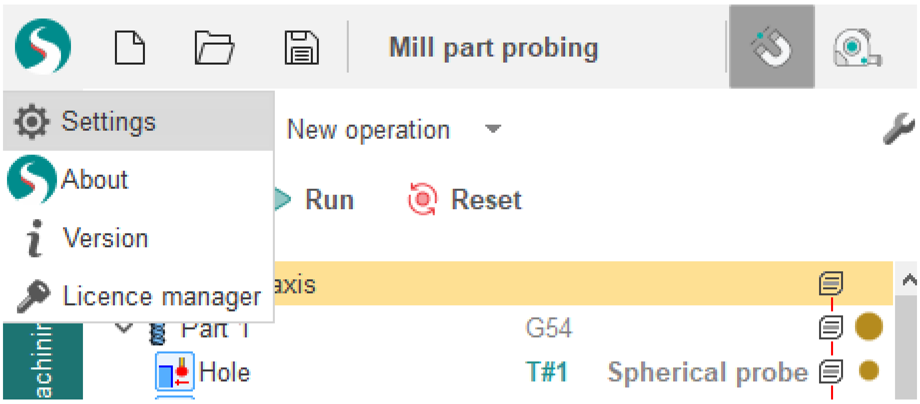
New group of Probing tools added: one tool for the axial case (tooling point at the axis) another for the turn case (tooling points can be at side).

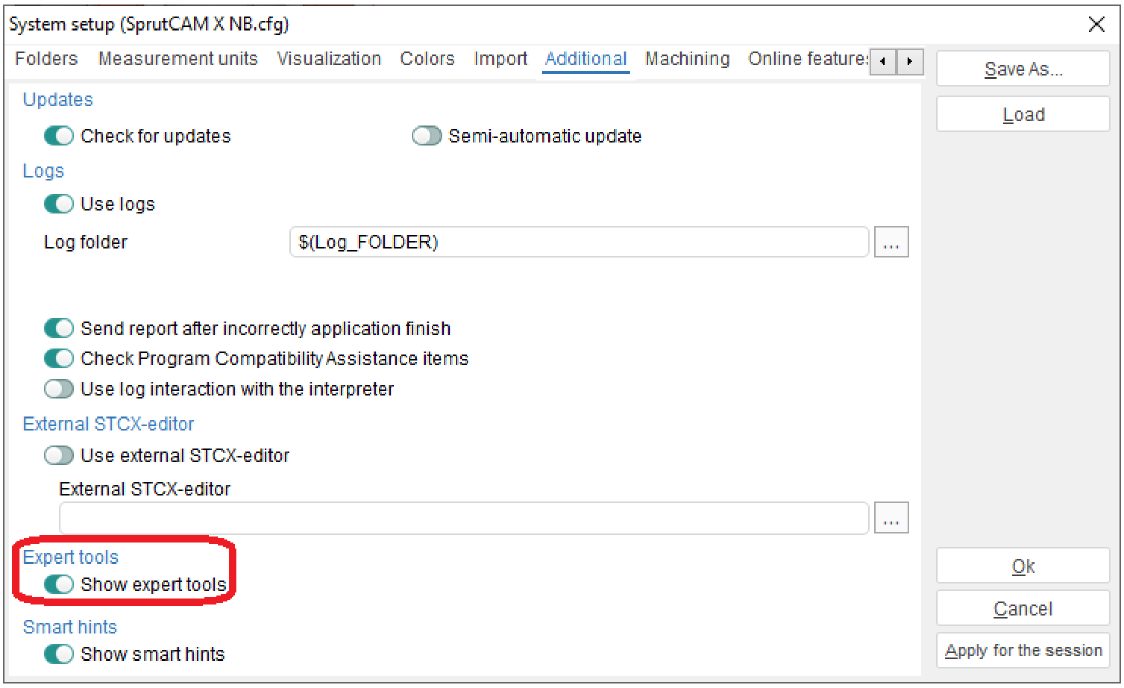


5.19.1 Creating own probing cycles (templates)

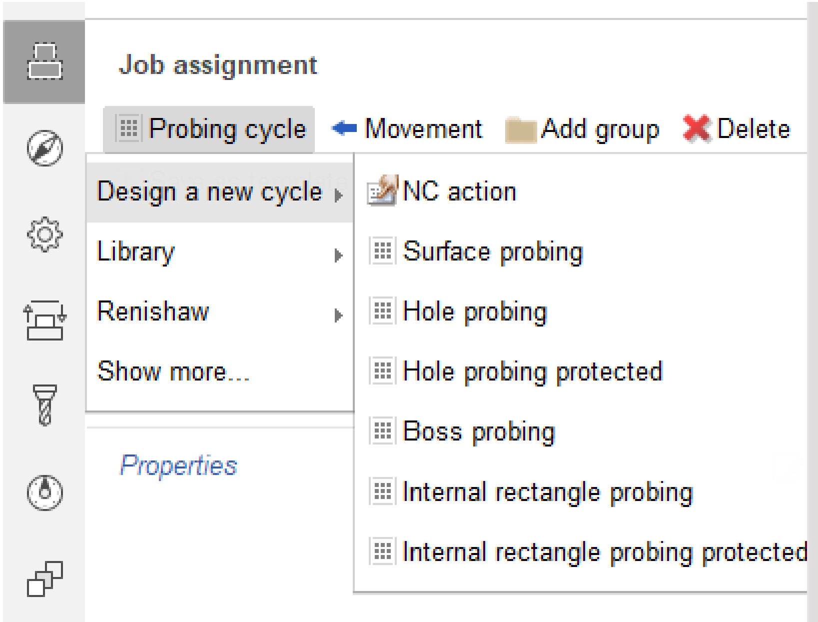
You can create your own probing cycles with individual properties for future use. New probing cycles may be united in library which is a separate file with the <**scpbl**> extension. This library can be shared with other users.

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.



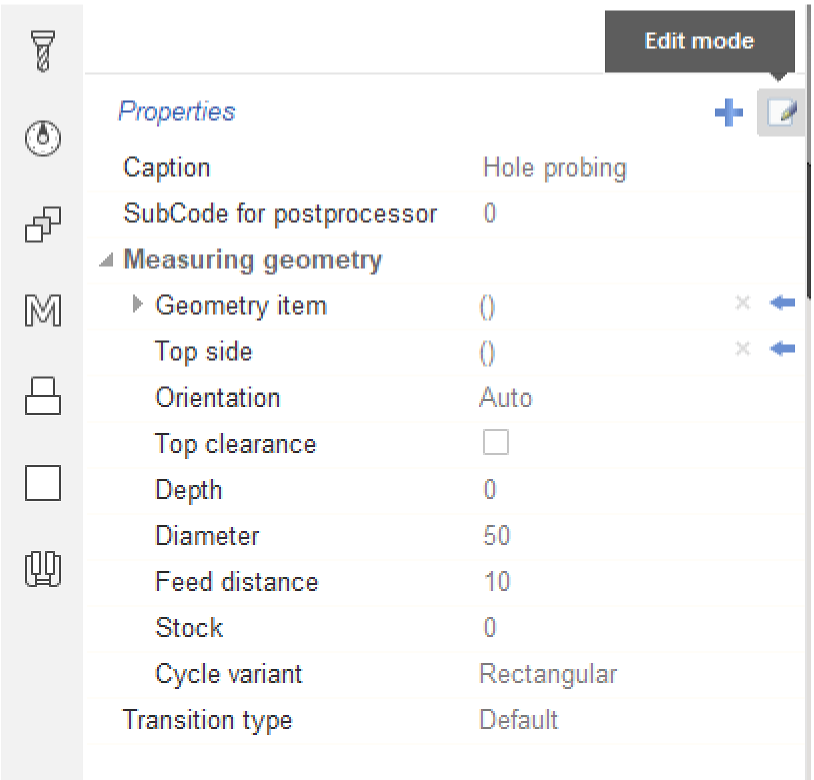


You can add one of empty cycles. Click <**Probing cycles**> in <**Job assignment**> tab of selected cycle. You find list of available cycles in <**Design a new cycle**> sub menu. There are 15 cycles and 1 additional element <**NC action**>. More information about this cycles is here.



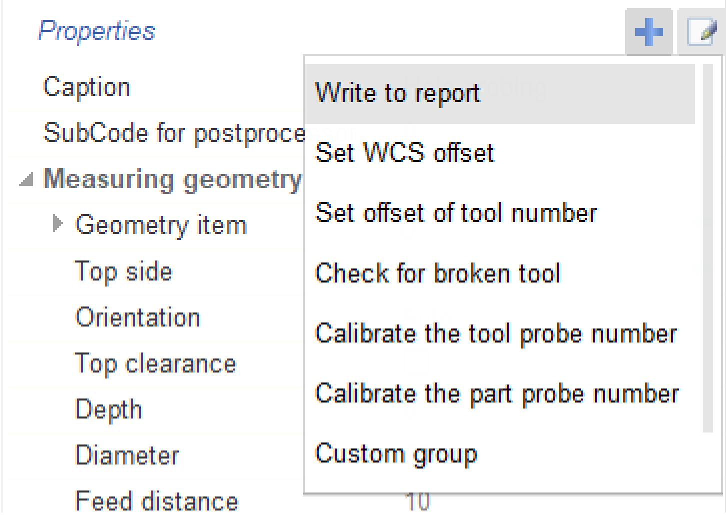
After adding the required cycle it is not necessary to set its geometric parameters, since these parameters will need to be set when using this cycle in the future. The main thing is to specify its properties that will be stored in the created template based on this cycle.

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 the 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>".



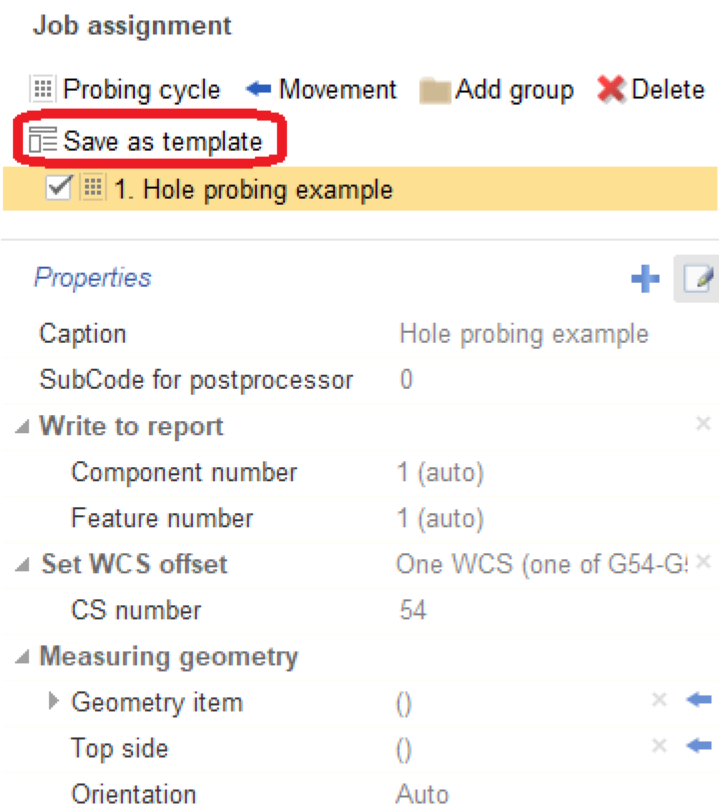
Click on <**Add custom property**> to add property from dropdown list. There are 8 properties with own parameters in list:

- <**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**> -calibrate 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**>.

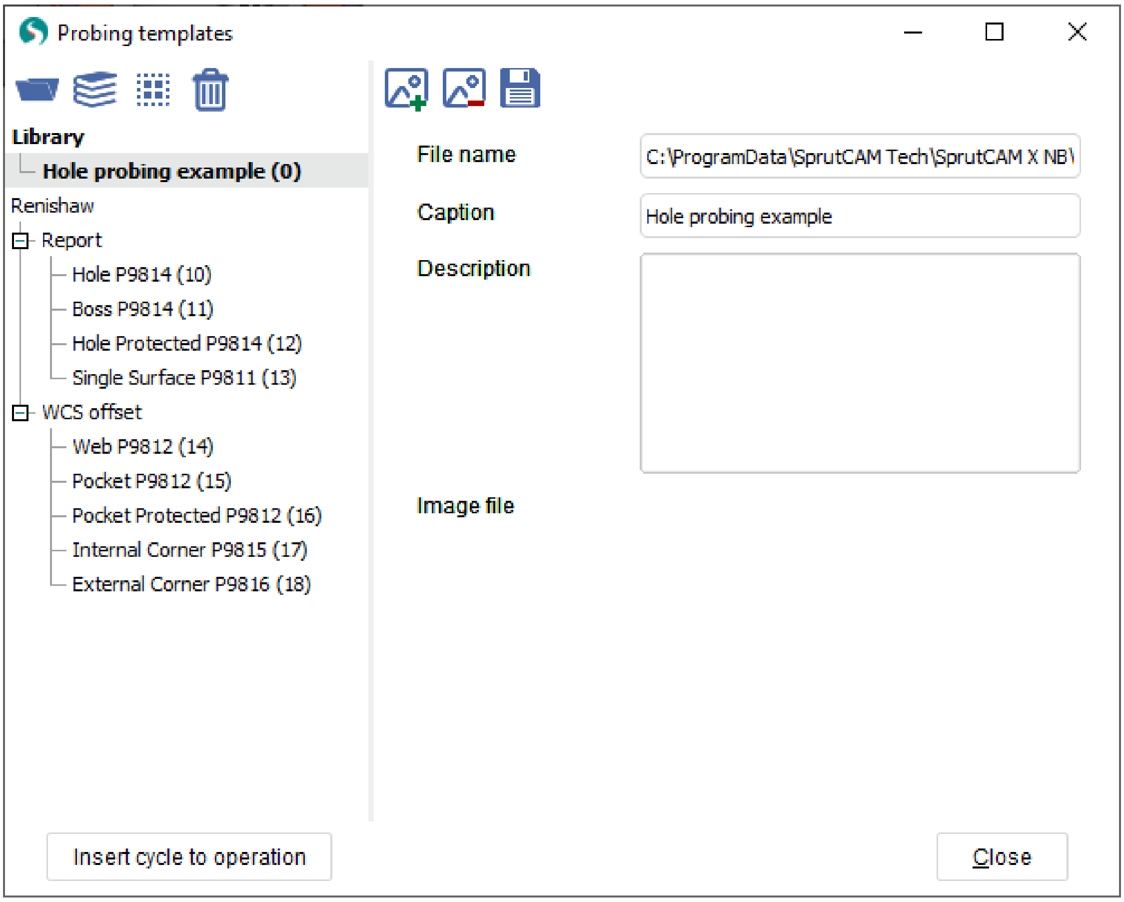


Access codes for parameters in CLData are described in postprocessor documentation "Probing cycle <WProbing> - Additional parameters".

After 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.



5.19.2 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.

5.19.2.1 Cycles with geometry

All cycles with geometry have some common parameters:

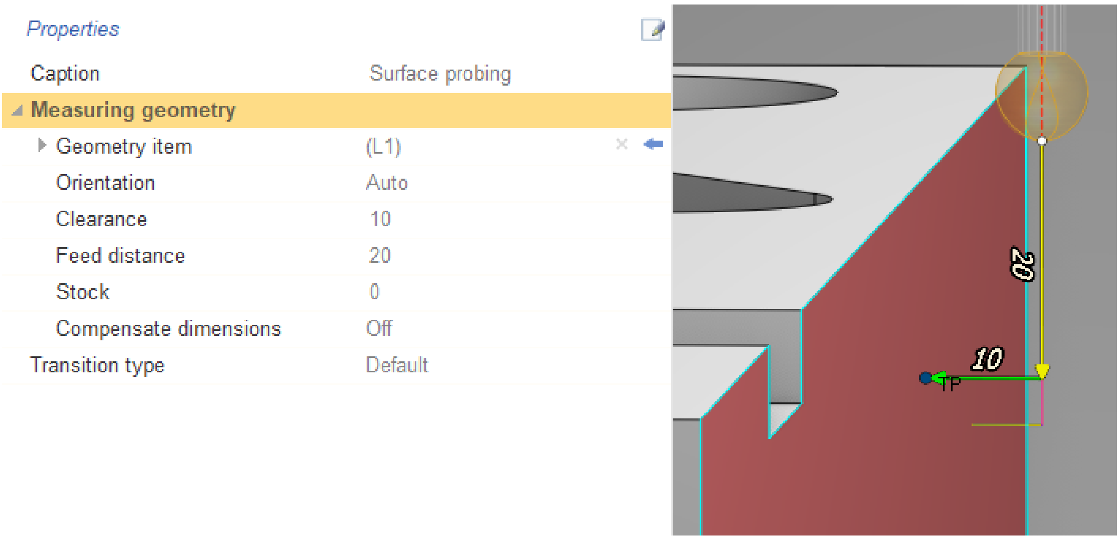
- <**Orientation**> - setting start orientation of tool in cycle. After adding cycle the orientation is set by <**Auto**>. You can switch on <**Manual**> and set orientation manually;
- <**Feed distance**> - distance of approach to start position of cycle and return from last position of cycle;
- <**Stock**> - 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 for 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:

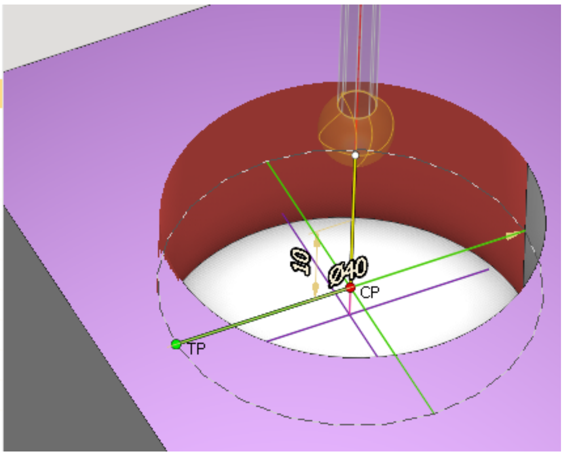
- <**Geometry item**> - measured surface. You can edit position and vector of the point manually. You need to expand <**Geometry item**> and set values in the corresponding fields: <**Target point**>, <**Target vector**>;
- <**Clearance**> - distance between approach point and touch point.

Access codes for parameters in CLData are described in postprocessor documentation "Probing cycle <WProbing> - Surface probing parameters".

Hole probing

Probing cycle designed for measuring bore.

Properties	
Caption	Hole
◢ Measuring geometry	
▸ Geometry item	(2)
▸ Top side	(1)
▸ Orientation	Manual
Top clearance	☐
Depth	10
Diameter	40
Feed distance	20
Stock	0
Cycle variant	Rectangular
Transition type	Default



Parameters of hole probing:

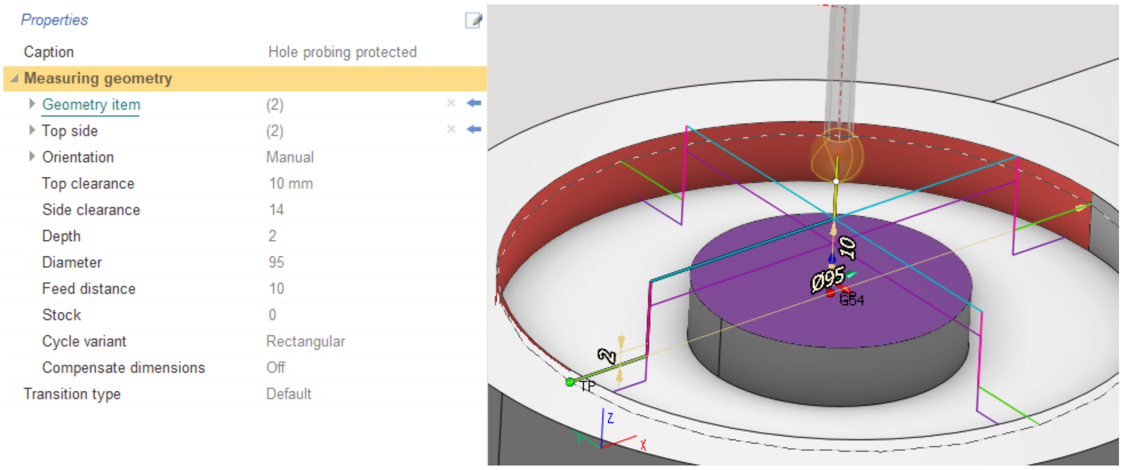
- <**Geometry item**> - measured surface of hole. You can edit position, vector and center of the point manually. You need to expand <**Geometry item**> and set values in the corresponding fields: <**Target point**>, <**Target vector**>, <**Center point**>;
- <**Top side**> - determination of the top level from which we count the distance to the starting point of the cycle;
- <**Top clearance**> - if 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: The spindle turns before approaching each touch point.

Cycle variant	Angular
Start angle	0
Angular step	120
Step count	3

Access 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.



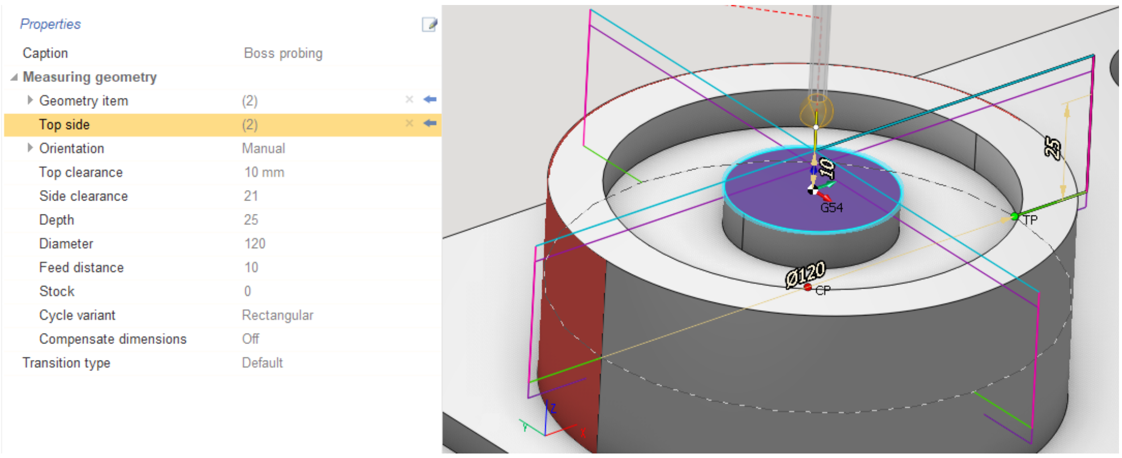
Parameters of hole probing protected:

- <**Geometry item**> - measured surface of hole with obstacle. You can edit position, vector and center of the point manually. You 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.

Access 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.

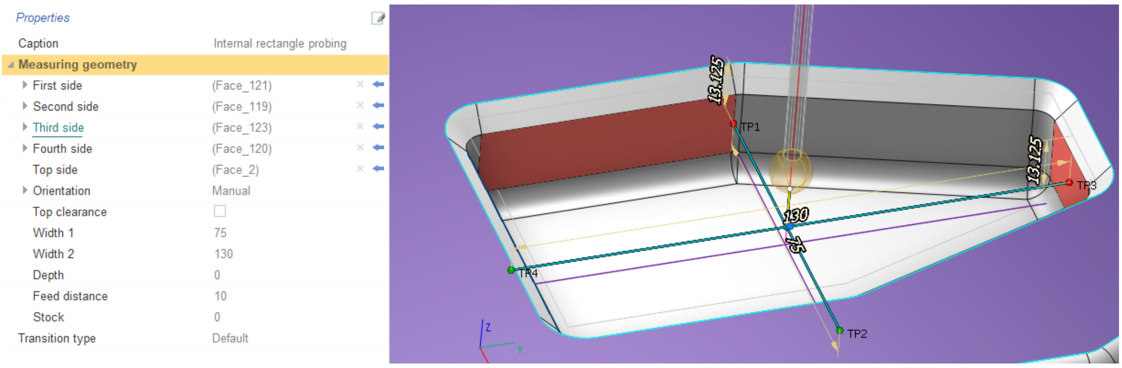


Parameters of boss probing are the same as in hole probing protected. The only difference is that <**Geometry item**> is the boss, external surface.

Access 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.



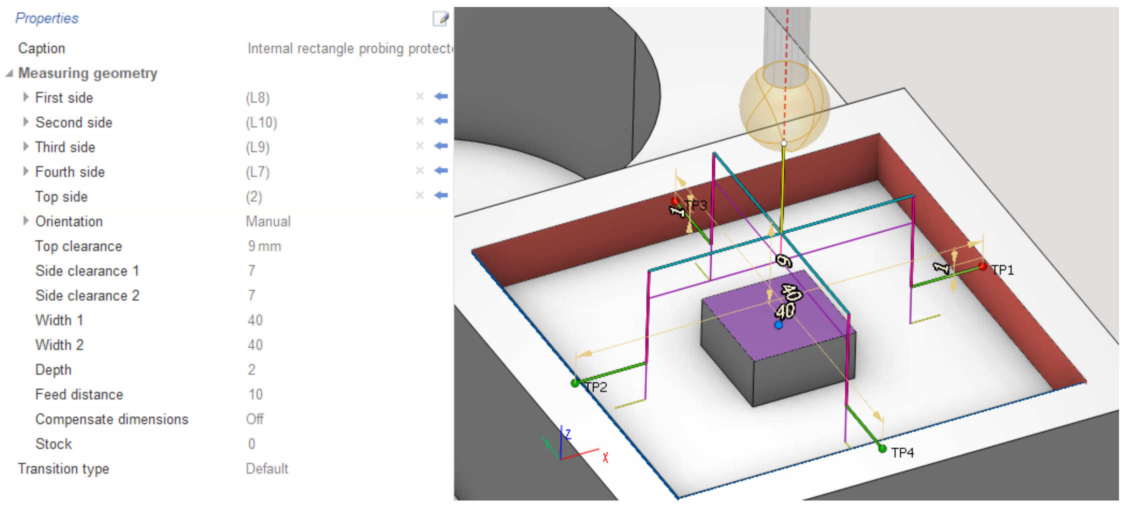
Parameters of internal rectangle probing:

- <**First side**> - the first touch point. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You need to expand <**Second side**> and set values in the corresponding fields: <**Target point 2**>, <**Target vector 2**>;
- <**Third side**> - the third touch point. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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**> - if 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.

Access 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.



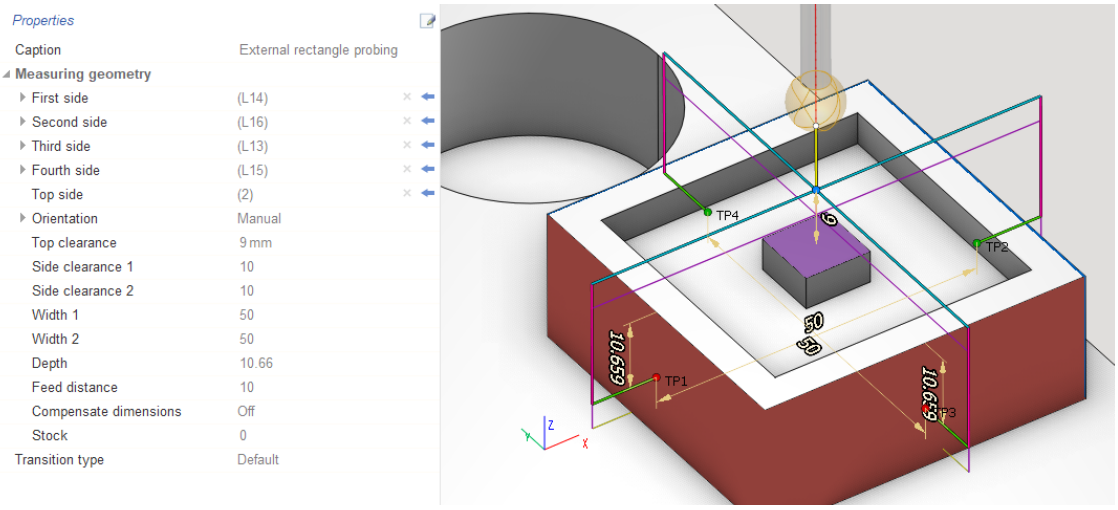
Parameters of this probing are the same as in internal rectangle probing. But it has two own parameters to avoid obstacles (<**Side clearance 1**>, <**Side clearance 2**>) and one parameter is changed (<**Top clearance**>):

- <**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.

Access 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.

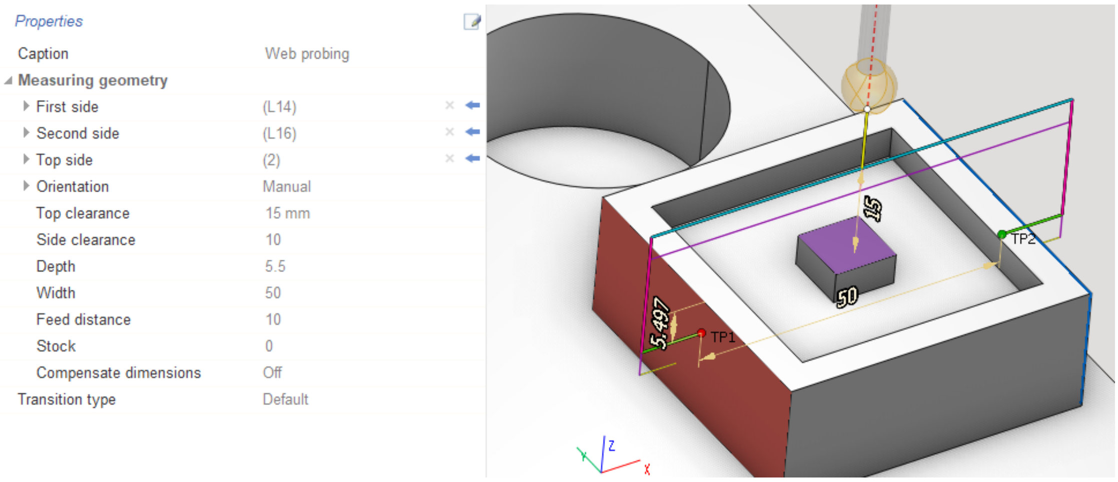


Parameters of extarnal rectangle probing are the same as in internal rectangle probing protected. The only difference is that touch sides are external surfaces.

Access 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.



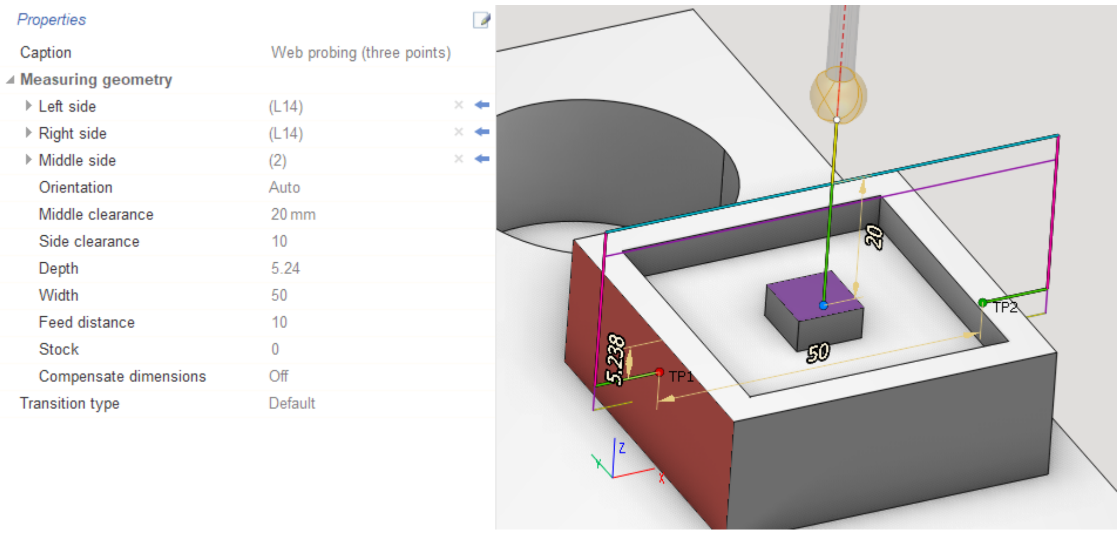
Parameters of web probing:

- <**First side**> - the first touch point. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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**>.

Access 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. The third side is between the first and second.



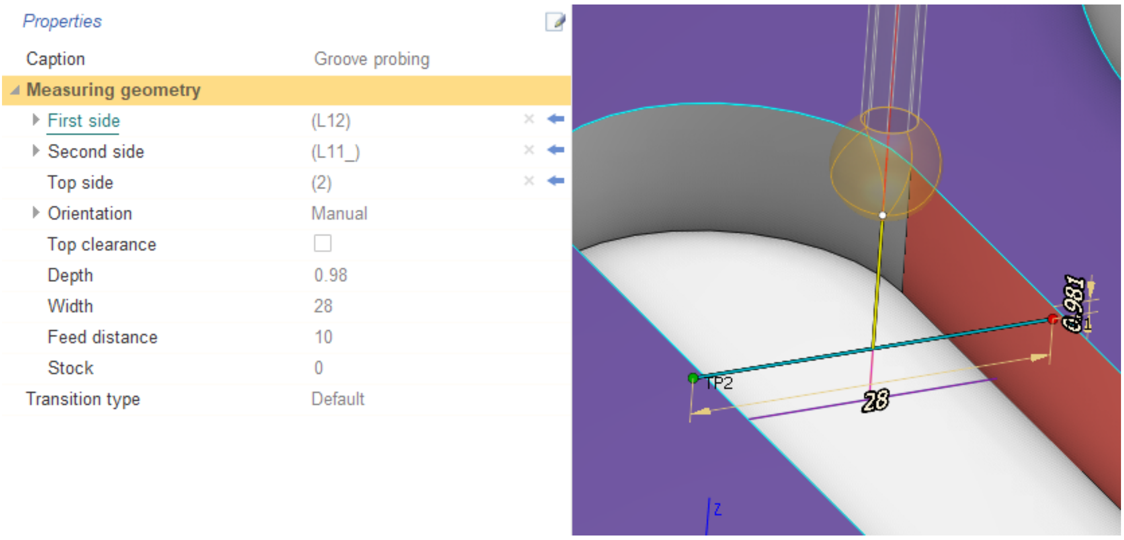
Parameters of web probing (three points):

- <**Left side**> - the first touch point. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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**>.

Access 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.



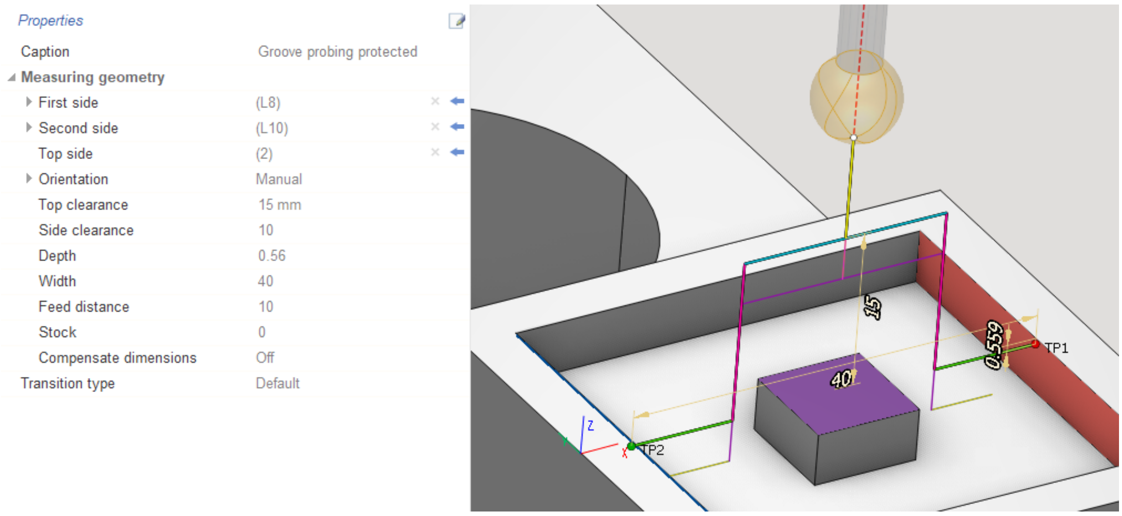
Parameters of groove probing:

- <**First side**> - the first touch point. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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**> - if 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**>.

Access 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.



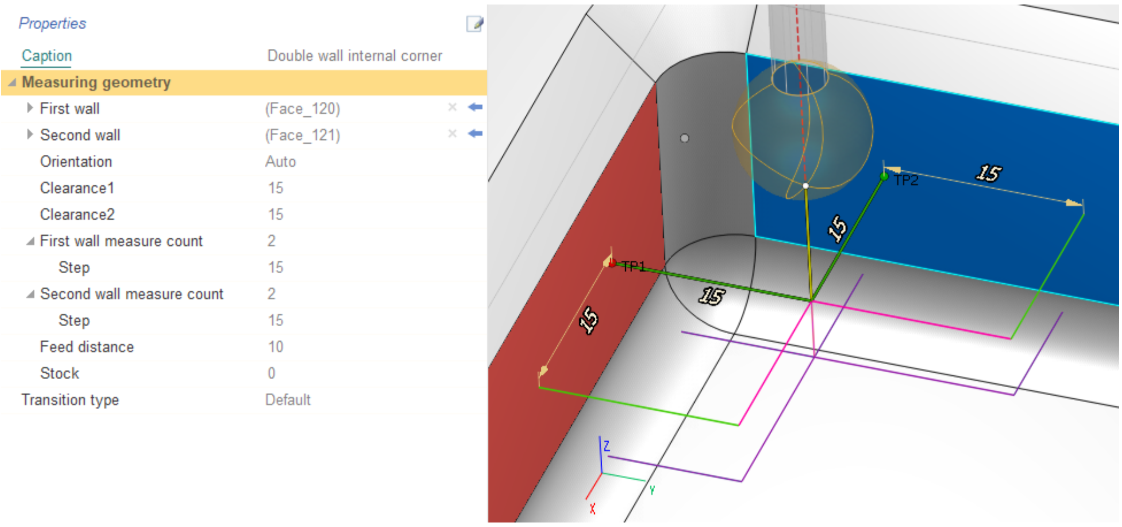
Parameters of this probing are the same as in groove probing. But it has one own parameters to avoid obstacles (<**Side clearance**>) and one parameter is changed (<**Top clearance**>):

- <**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.

Access 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:

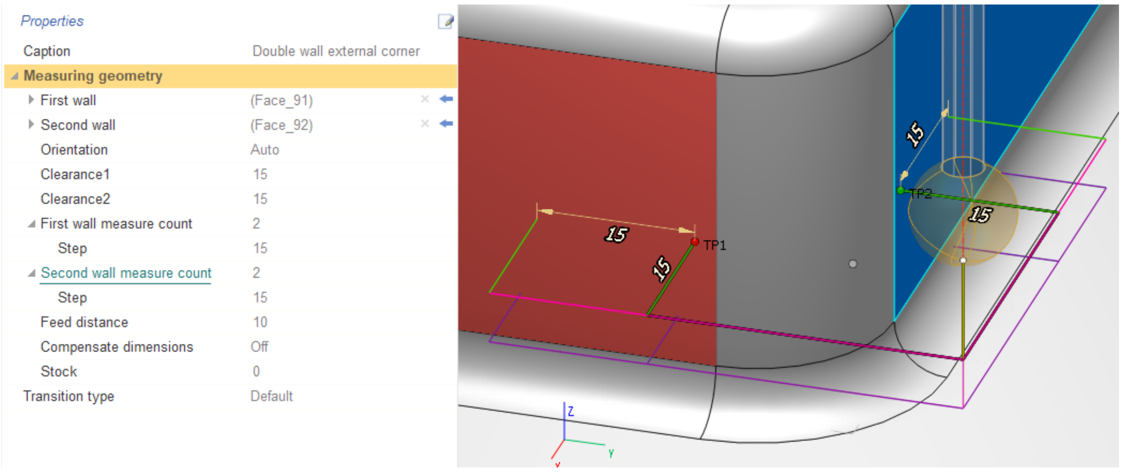
- <**First wall**> - the touch point of the first wall. You can edit position and vector of the point manually. You need to expand <**First wall**> and set values in the corresponding fields: <**Target point**>, <**Target vector**>;
- <**Second wall**> - the touch point of the second wall. You can edit position and vector of the point manually. You 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.

Access 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.

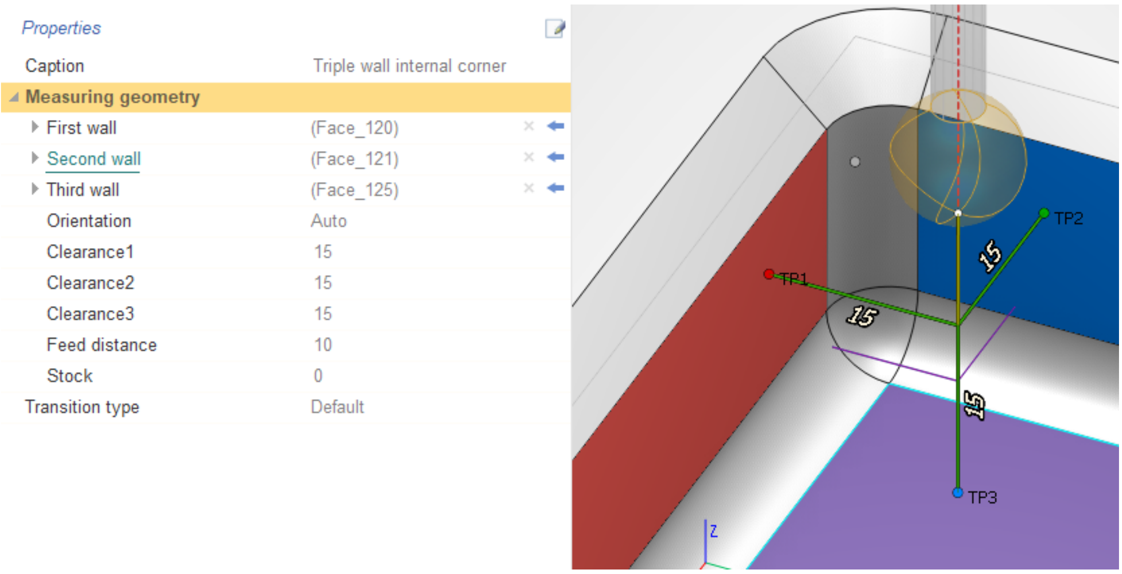


Parameters of this probing are the same as in double wall internal corner. The only difference is that touch walls are external surfaces.

Access 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.



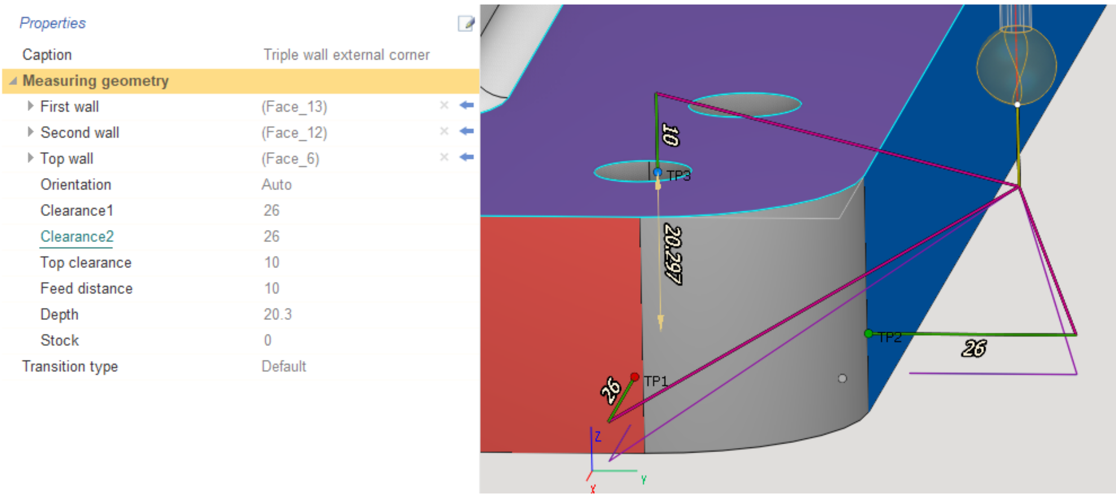
Parameters of triple wall internal corner:

- <**First wall**> - the touch point of the first wall. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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**>.

Access 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. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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. You can edit position and vector of the point manually. You 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.

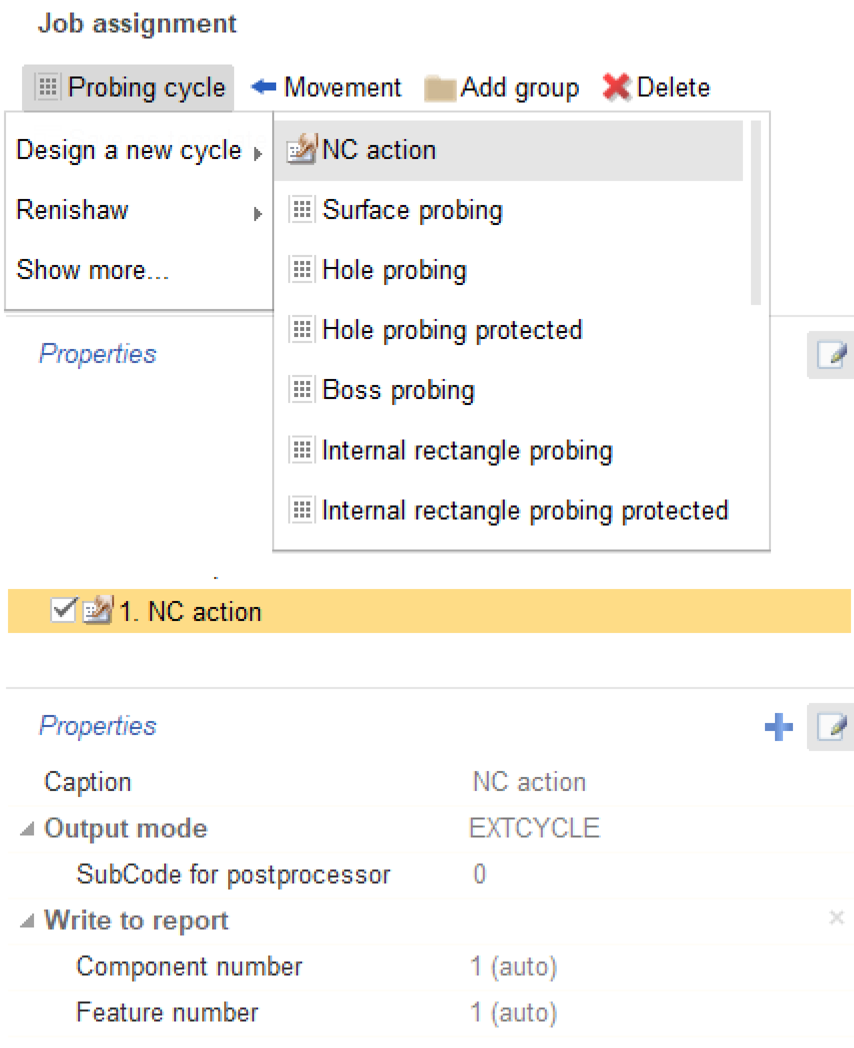
Access codes for parameters in CLData are described in postprocessor documentation "Probing cycle <WProbing> - Triple wall external corner probing parameters".

5.19.2.2 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.

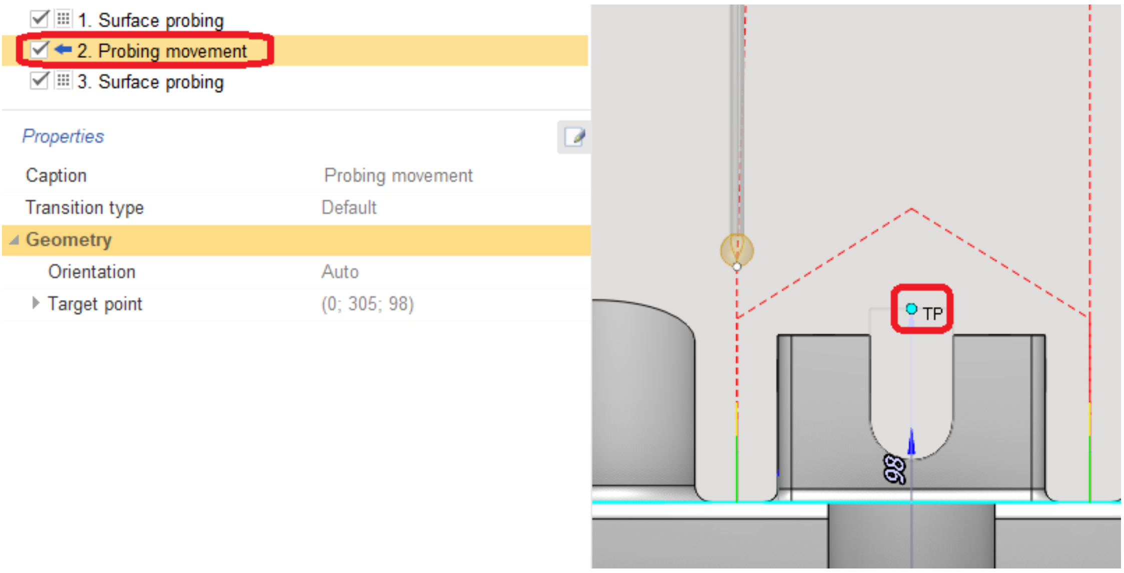
It does not generate any movements. However, it can contain parameters that need to be either directly displayed in the NC using the Insert command or as a cycle with an array of numeric parameters. Through the mechanism of custom properties, you can add any parameters there. Based on these parameters 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**>.



Access 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.



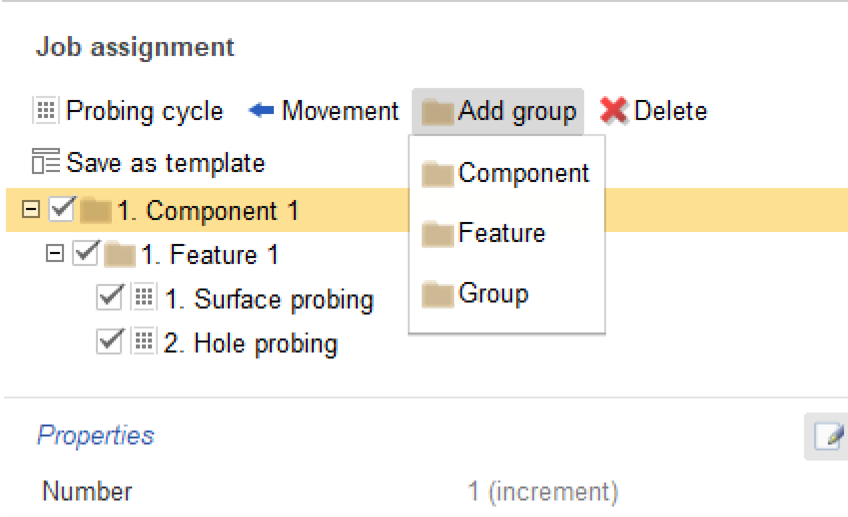
Elements to group

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:

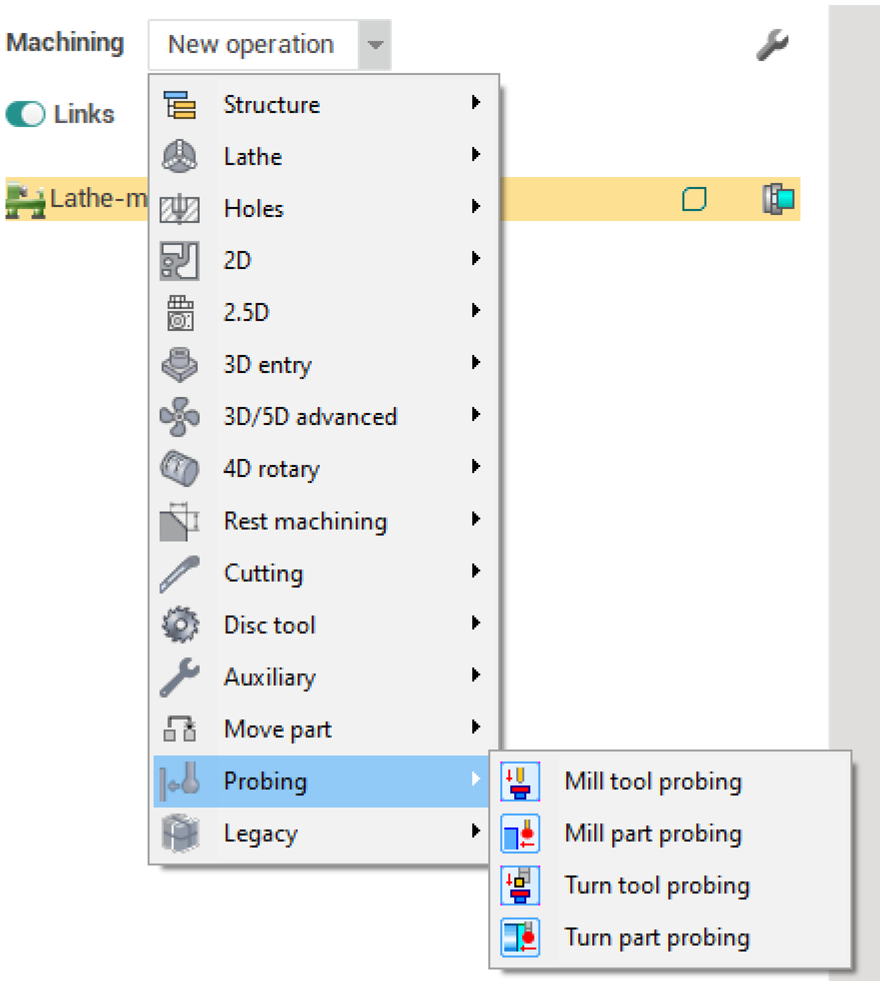
- <**Group**> - general grouping;
- <**Component**> - part to be measured;
- <**Feature**> - individual element or surface currently being measured.

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.



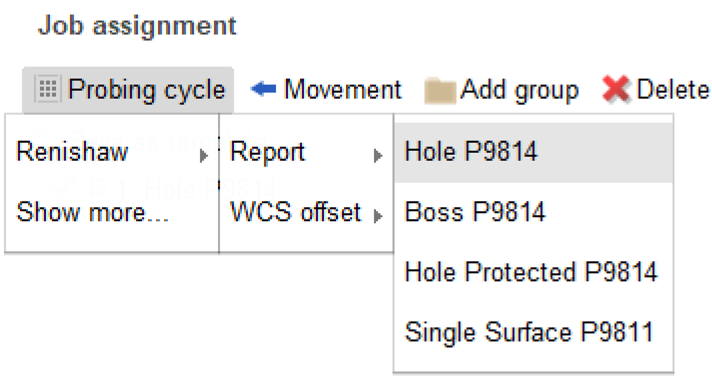
5.19.3 Use of prepared measuring cycles

You 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.



5.19.3.1 Part probing

Add 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.

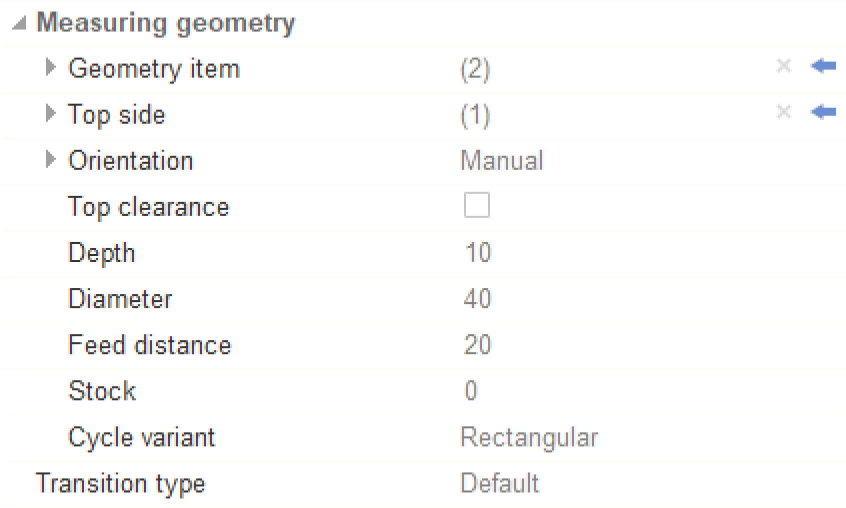


The example of working with cycles will be presented based on <**Hole probing**>.

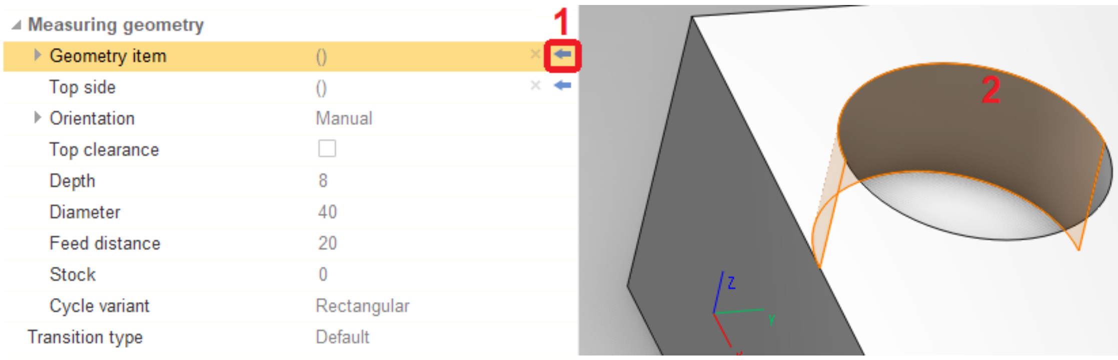
Add new probing cycle <**Hole P9814**> from default library **<Renishaw>,** group <**Report**>.

Properties of the added cycle will appear at the bottom of the <**Job assignment**>.

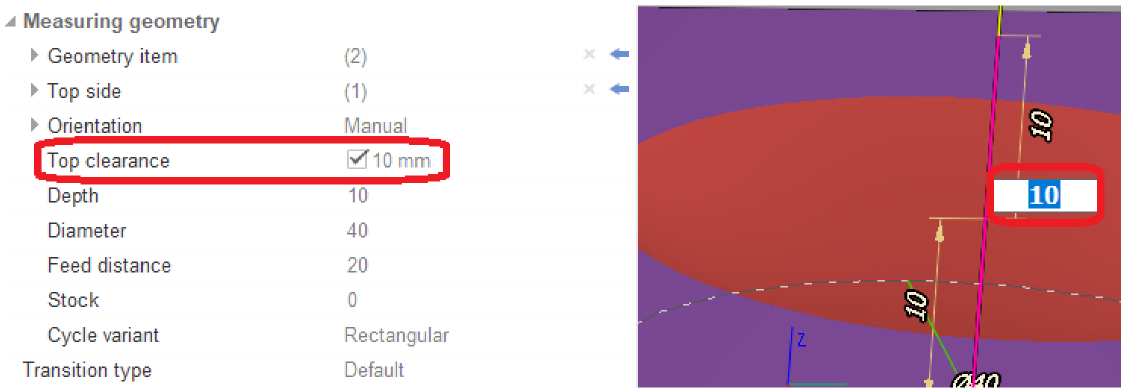
Use <**Measuring geometry**> to set parameters for cycle calculation.



Click on the arrow in the right part of the field and select a surface in the graphics window to add <**Geometry item**> or <**Top side**>. The selected surface is automatically added to the current field.



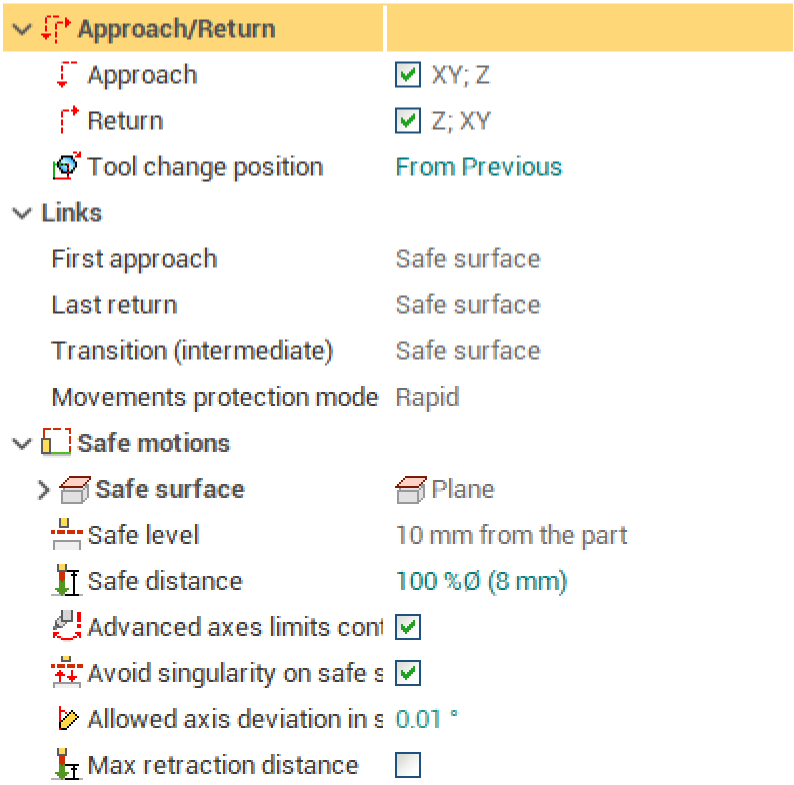
Other parameters are entered manually in the corresponding field. Some cycle parameters can be edited both in the inspector and in the graphics window.



5.19.3.2 Transitions

Go to the tab <**Links**> to set the parameters of transitions between probing cycles.

Links/Leads



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.

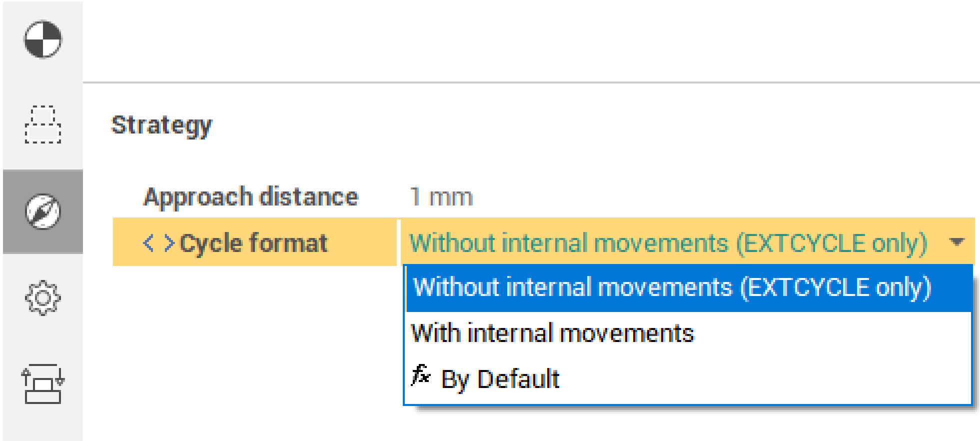
There is additional value for <**Transition (intermediate)**>: <**Orthogonal**> - if the start and end points of the transition are located at the same level in Z then go directly. If the start point is lower then at first go to the level of the end point then go directly. If the end point is lower then 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.

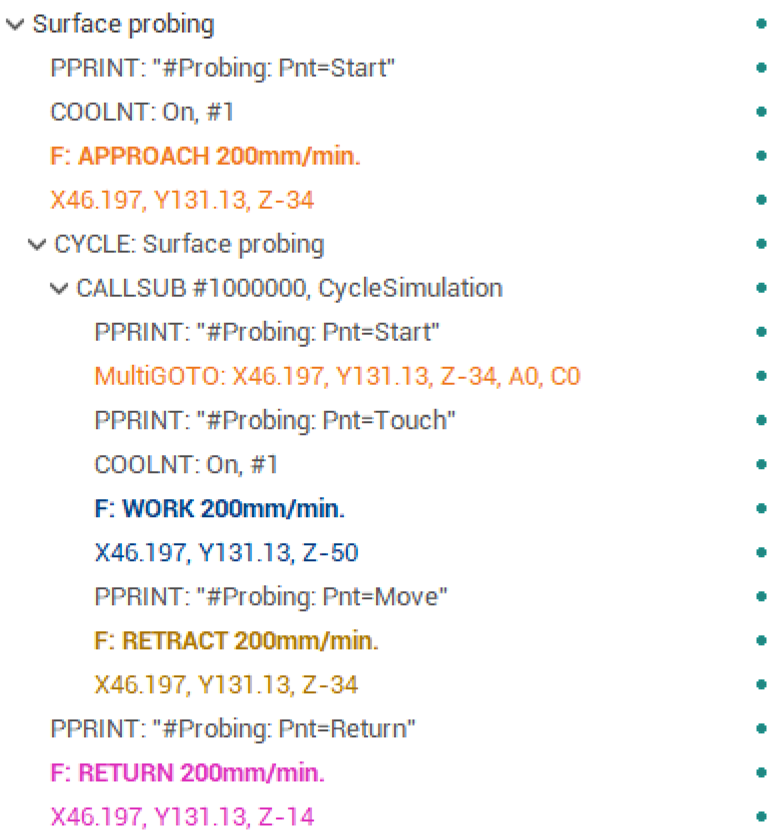
5.19.3.3 Cycle format

You can choose one of the options for outputting cycles in CLData. To do this, go to the <**Strategy**> tab.

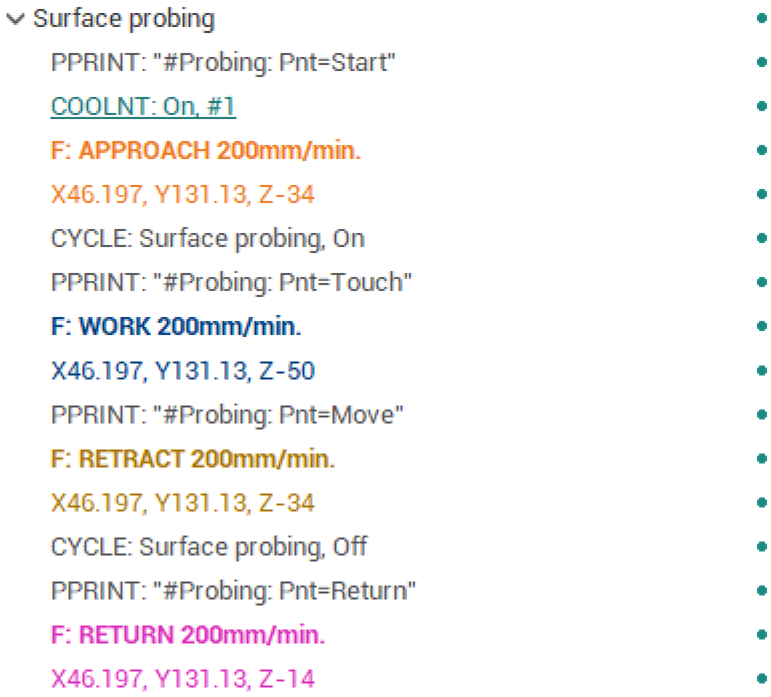


<**Cycle format**> has two values:

- <**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.

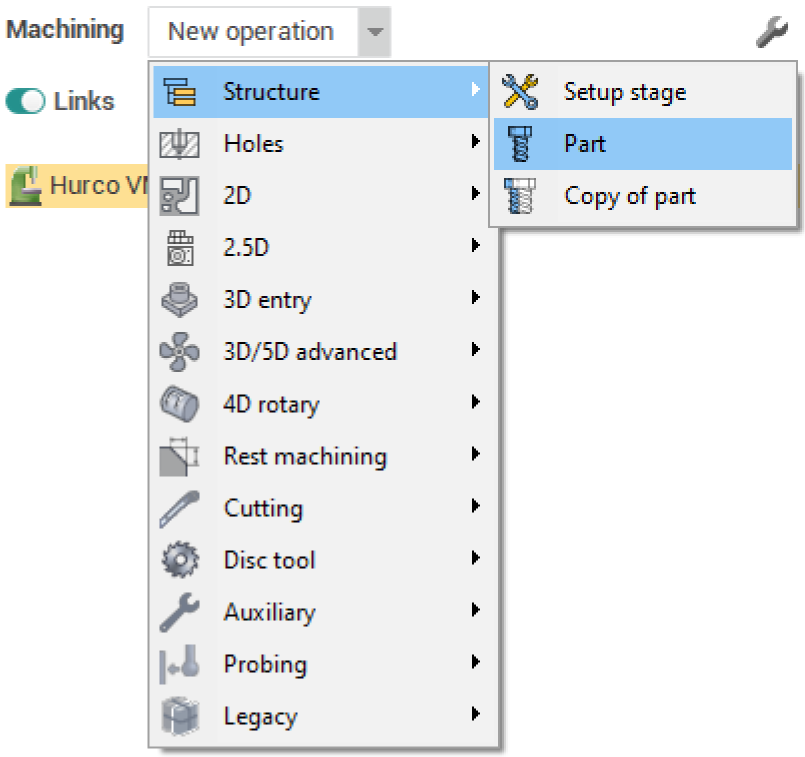


5.19.3.4 Tool probing

Setting parameters and transitions of cycles for the tool probing is exactly the same as for the part probing.

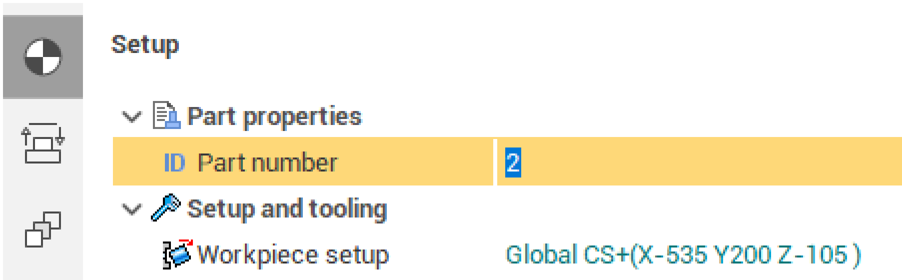
The probe can be not only a tool, but also a part which fix in a special place on the table. About such a probe, the tool is usually measured for its initial calibration or to check its integrity.

You need to add the <**Part**> group for correct work.



It is necessary to create separate measuring operations for each tool within this group.

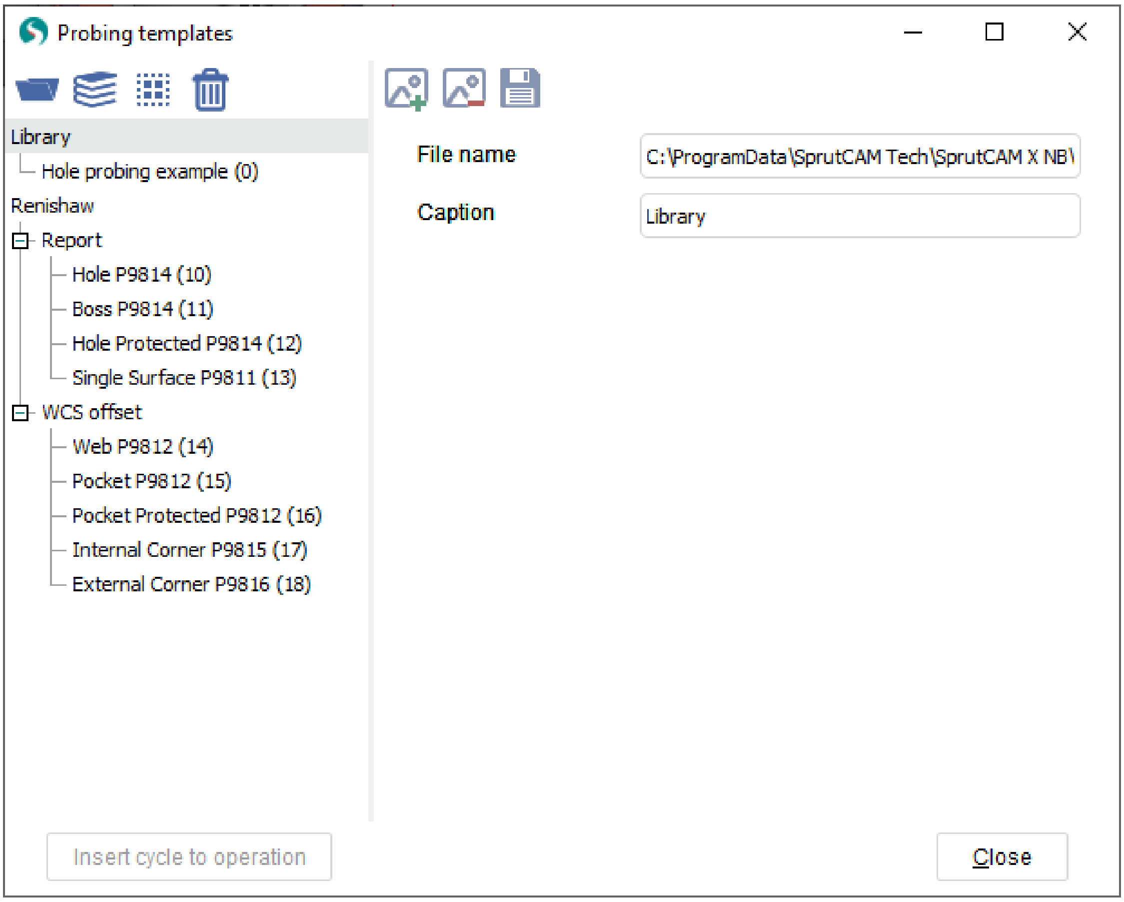
You need to switch to the <**Setup**> tab and set <**Part number**>. <**Part number**>can be used as probe ID.



The example of creating and working with tool probing operation can be found in the distribution project "Tool probing" in "Probing" folder.

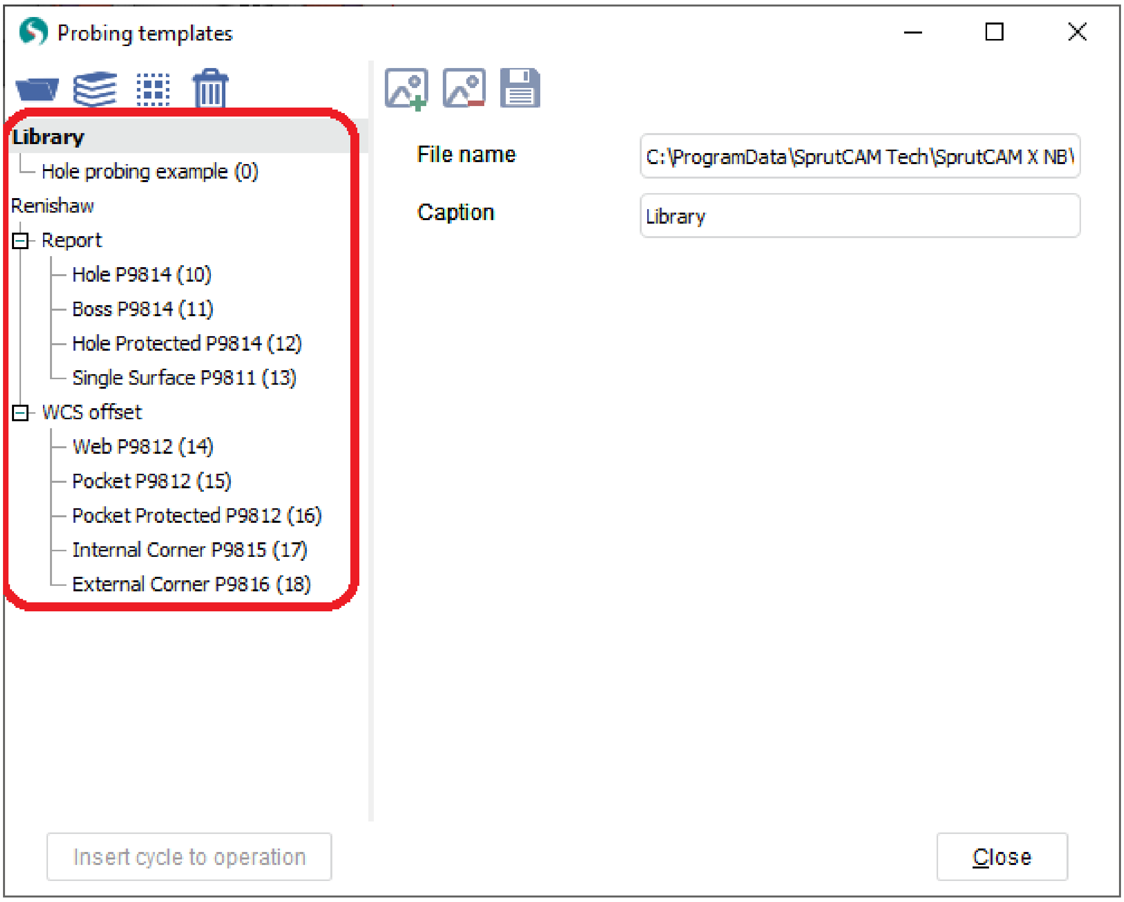
5.19.4 Probing templates window

The 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".



5.19.4.1 Library tree

On the left part of the window there is a tree with a list of libraries and cycle templates. All 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 .

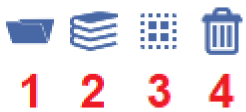
Elements are moved using drag'n'drop. You can move like this:

- 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.

5.19.4.2 Control elements

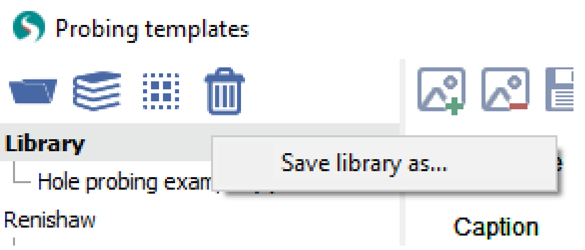


The tree element control panel is in the upper left part of the window. There are 4 controls on the panel:

1. <**Open library**> is button to add existing local libraries. After 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.

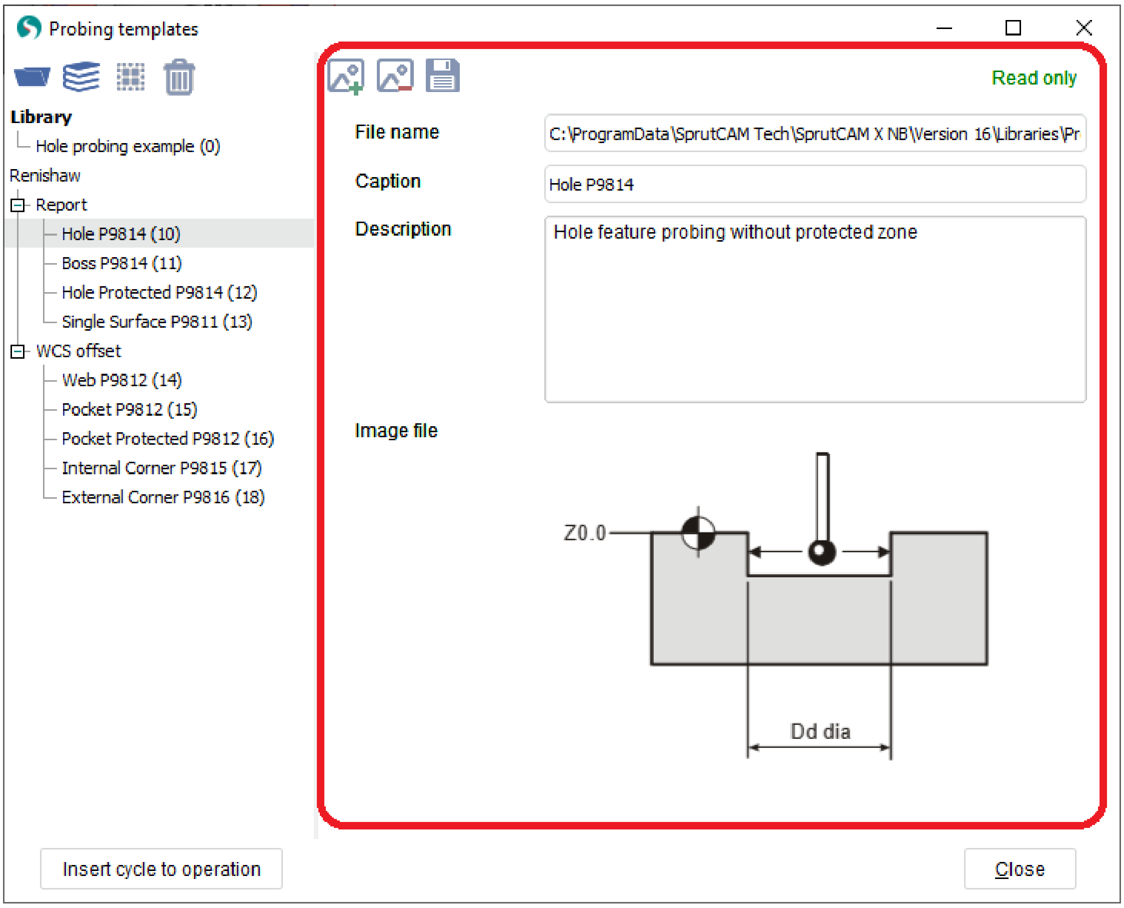
You can also make a copy of the library and save it to another location on your computer. To do this, select the required library and click the right mouse button. In the context menu that appears, click <**Save library as**>. After clicking on the button, a dialog box will open with the choice of the 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.



5.19.4.3 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.

Use the image control panel which is located on top of the parameters panel.

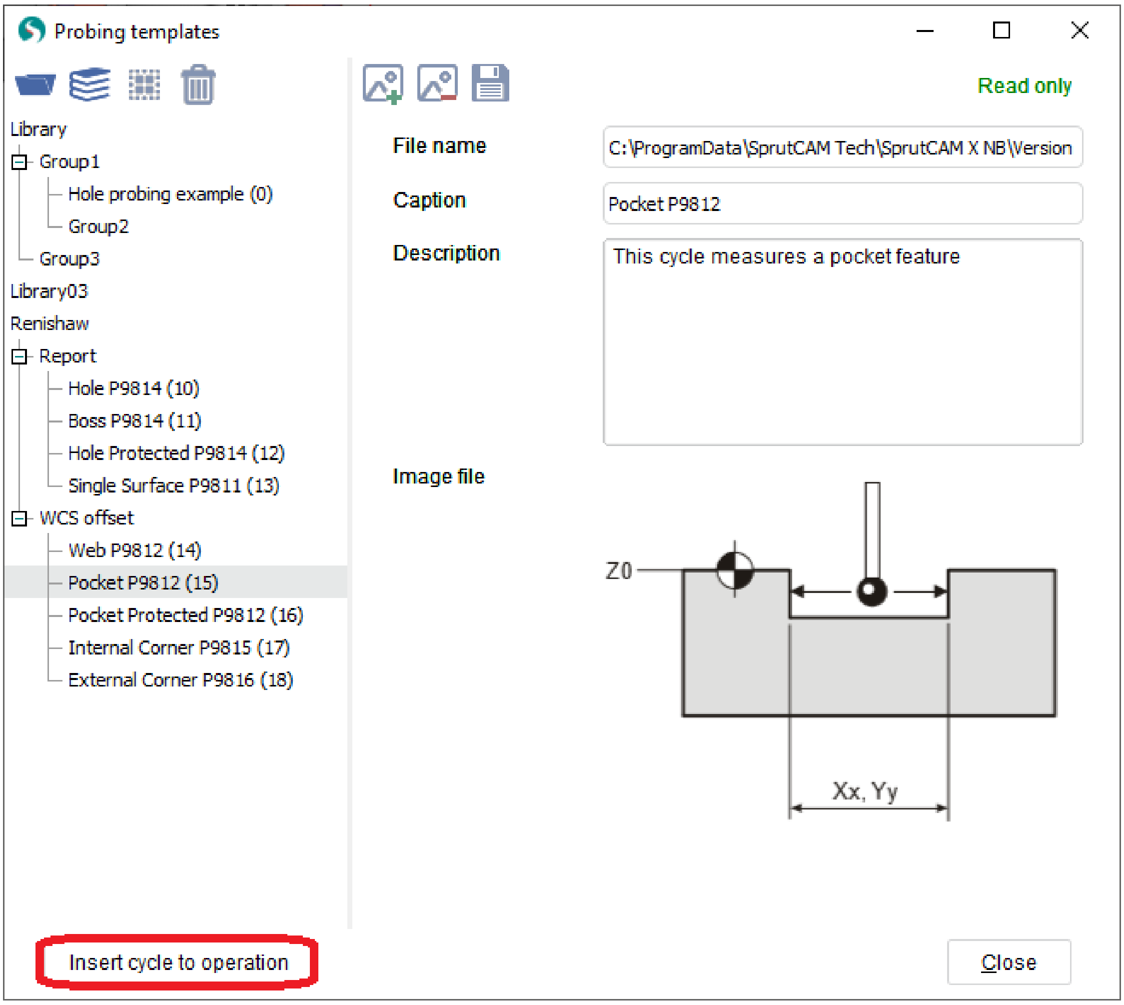
1. <**Load cycle image**> - adding an image for the selected cycle. When you click on this button, a dialog window opens for specifying the path to the image;
2. <**Remove cycle image**> - removing an image for the selected cycle;
3. <**Export image**> - save image of the selected cycle to an external file.

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.

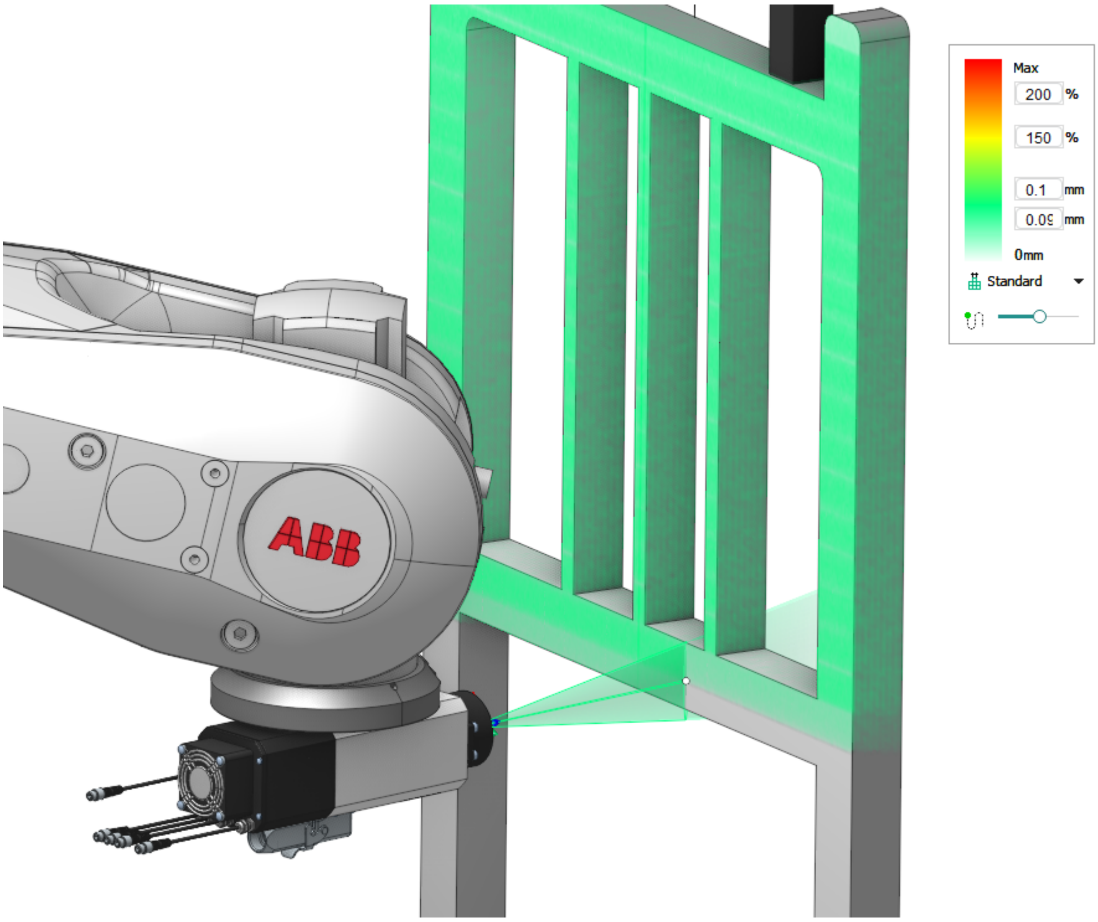
5.19.4.4 Saving and using cycles

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.


Probing templates
Read only
Library
Group1
Hole probing example (0)
Group2
Group3
Library03
Renishaw
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)
File name
C:\ProgramData\SprutCAM Tech\SprutCAM X NB\Version
Caption
Pocket P9812
Description
This cycle measures a pocket feature
Image file
Z0
Xx, Yy
Insert cycle to operation
Close

5.20 Spray painting

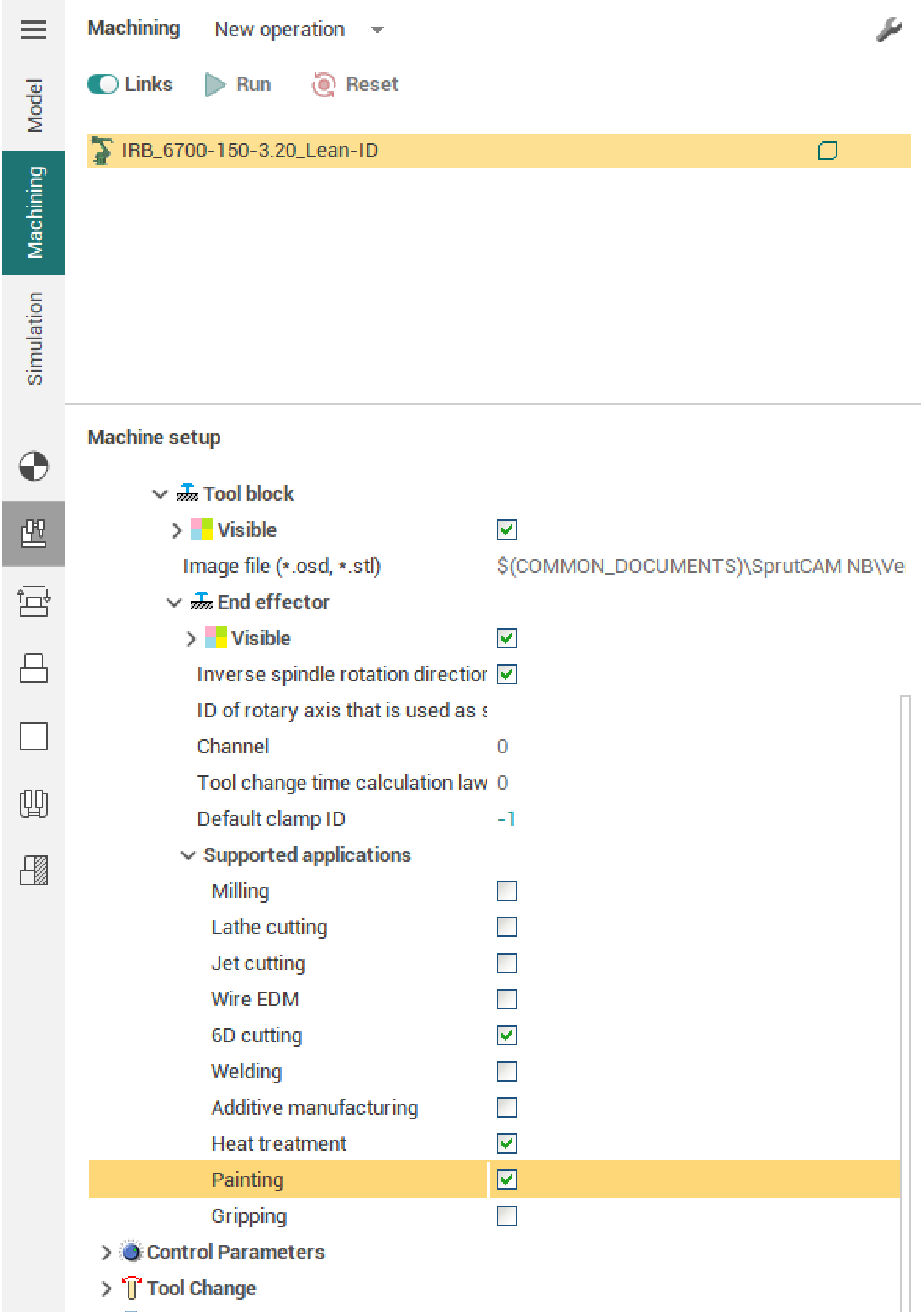


Spray painting is a painting technique in which a device sprays coating material (paint, ink, varnish, etc.) through the air onto a surface. In order to be able to process hard-to-reach places, such machining is often performed on equipment that has a large number of degrees of freedom, such as industrial robots.

The system supports programming and simulation of such machining.

5.20.1 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 "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).



5.20.2 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);

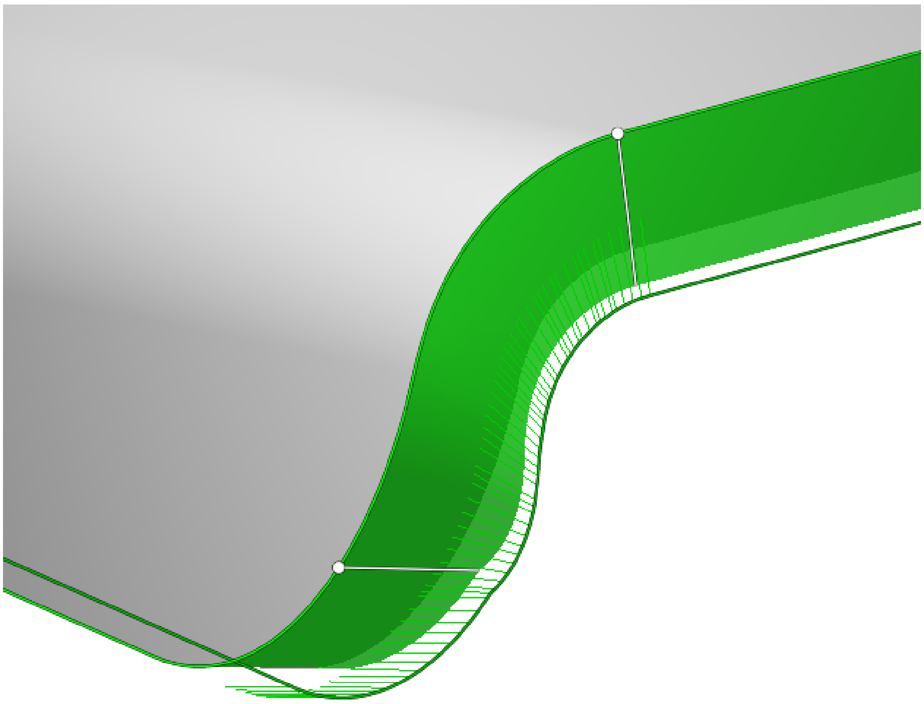
3) Morph spraying (base on a Morph operation);

4) Rotary spraying (based on a Rotary finishing operation).

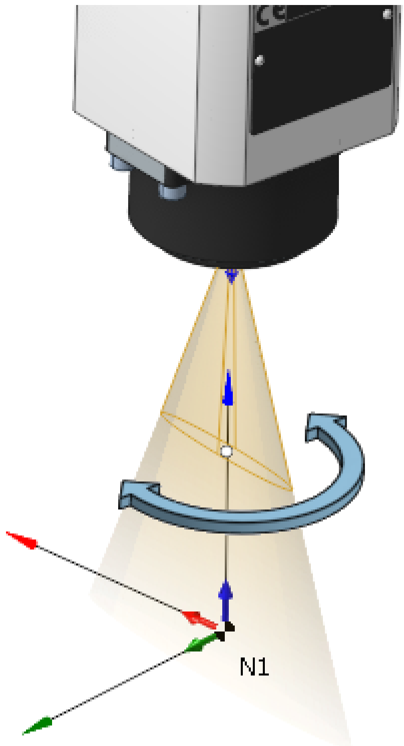
> ⓘ When you create any spraying operation, the system automatically chooses the spray tool for this operation on the "Tool" tab and sets the "Painting" simulation as a "Simulation type" on the Parameters tab by default.
> You can find information about spray tools in the section below.

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 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.



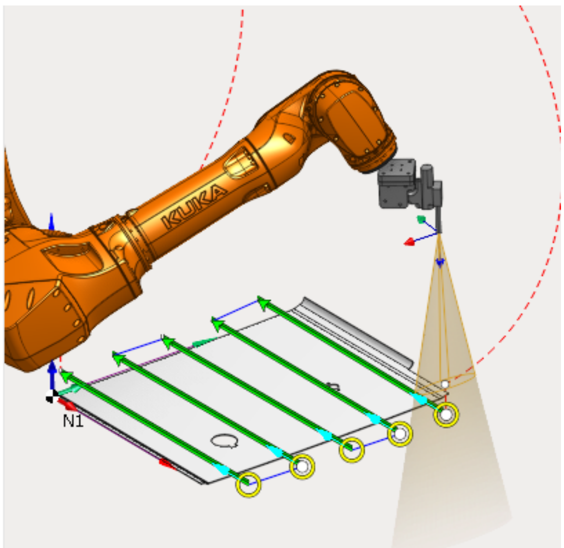
In order for the tool orientation to be preserved relative to the toolpath, you need to activate the **Toolpath** mode of the **6th axis control** property.

6th axis control	Tool path
Tangent approximation	50 %Ø (5 mm)
Angular deviation	0 °

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.

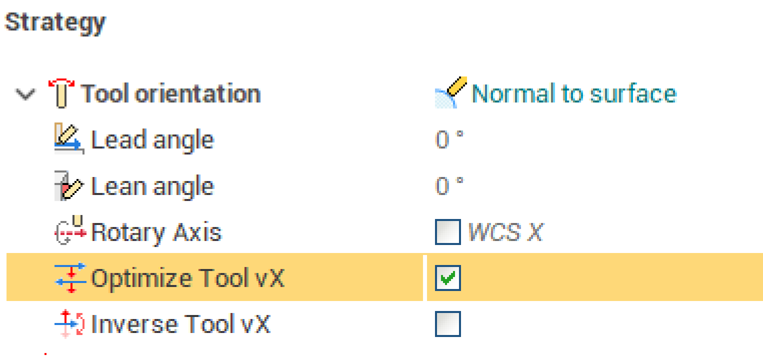
Strategy

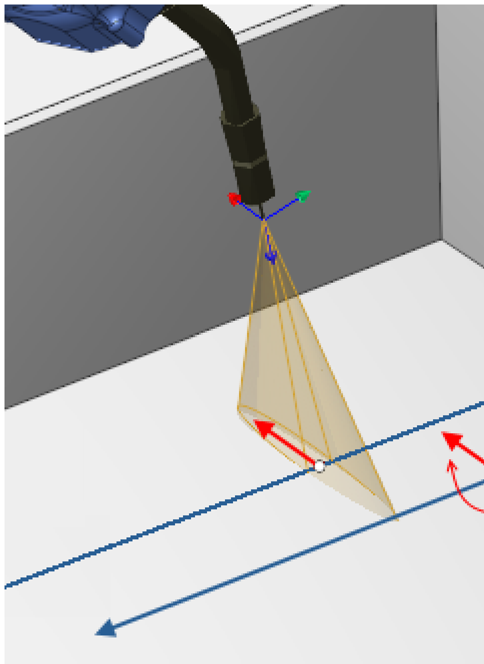
Tool orientation	Normal to surface
Lead angle	0 °
Lean angle	0 °
Rotary Axis	☐
Limit rotation angles	☐
Passes	1 x (Count)
Bottom level	0 mm
Z cleanup	☐
Sorting	By layers
Idling minimization	☐
Milling type	Both
To inverse odd curves	☑
Corners smoothing	
Inner corners	☑ 0 mm
Project toolpath onto the p	☐



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.

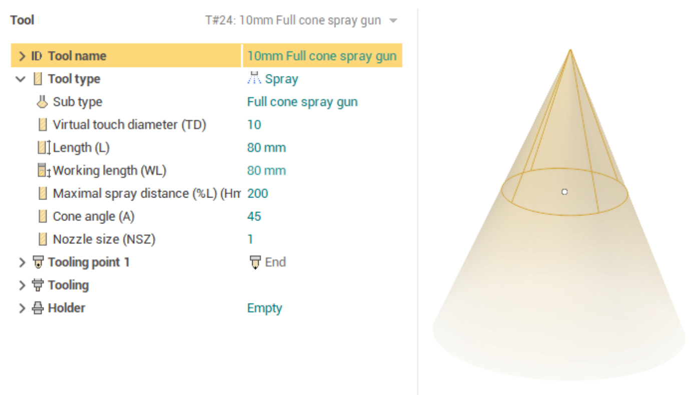




5.20.3 Spray tools

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.	

<table>
<tr><th>Tool type name</th><th>Picture</th></tr>
<tr><td>

Hollow cone spray gun.

It does not paint in the center, only a narrow strip along the edge works.

</td><td>

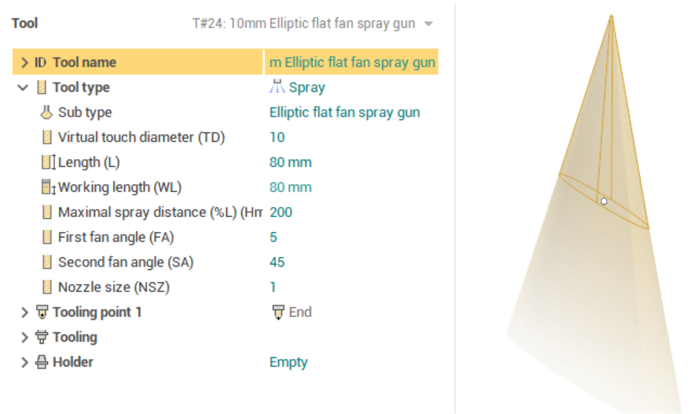
</td></tr>
<tr><td>

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.

</td><td>



</td></tr>
<tr><td>

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.

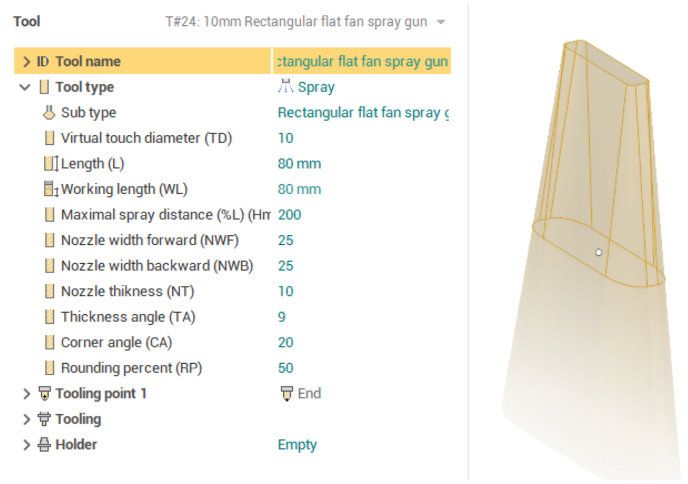
Use the **Mounting angle** property to rotate the main direction.

</td><td>

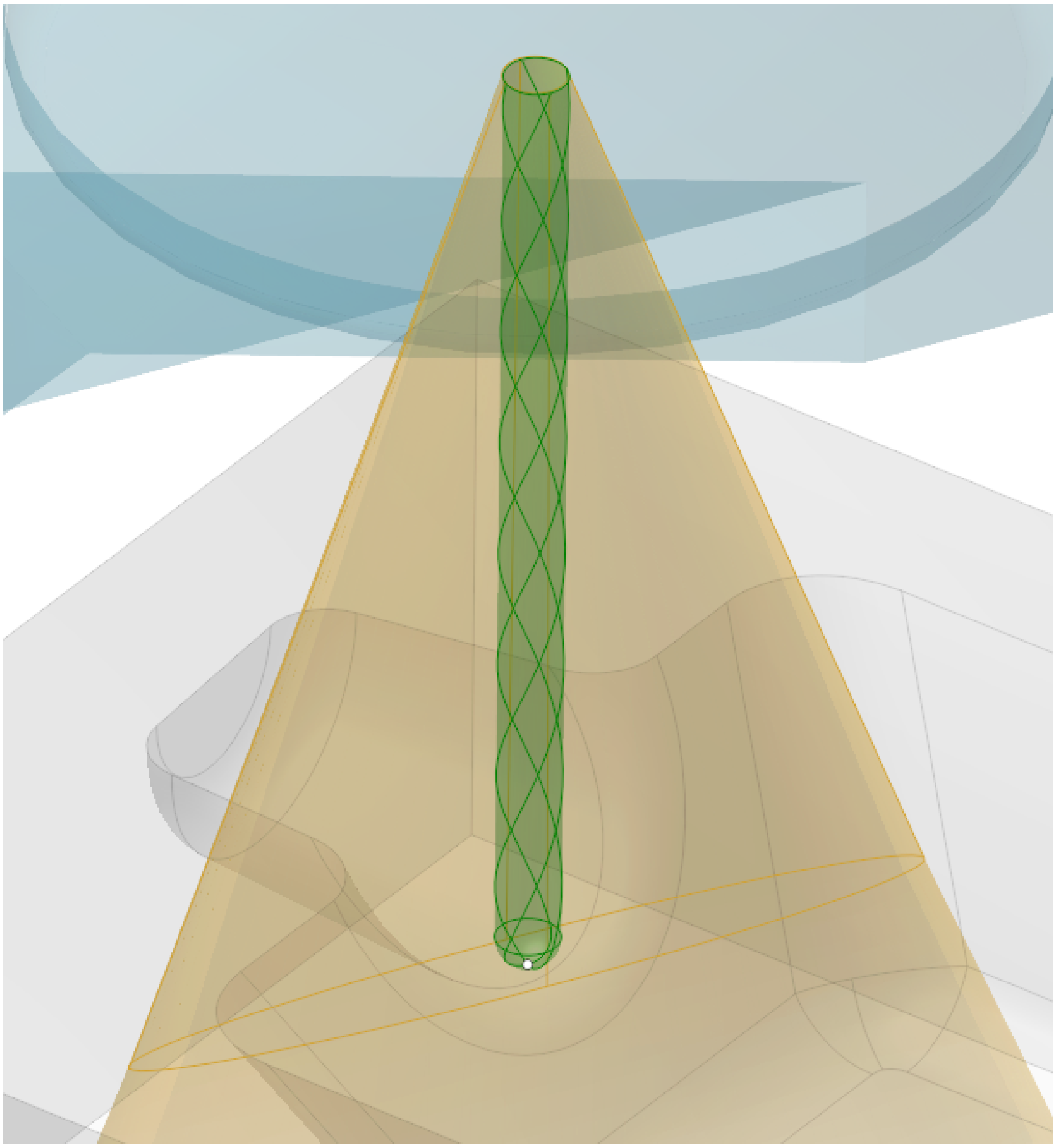
</td></tr>
</table>

<table>
<tr><th>Tool type name</th><th>Picture</th></tr>
<tr><td>Rectangular flat fan spray gun.

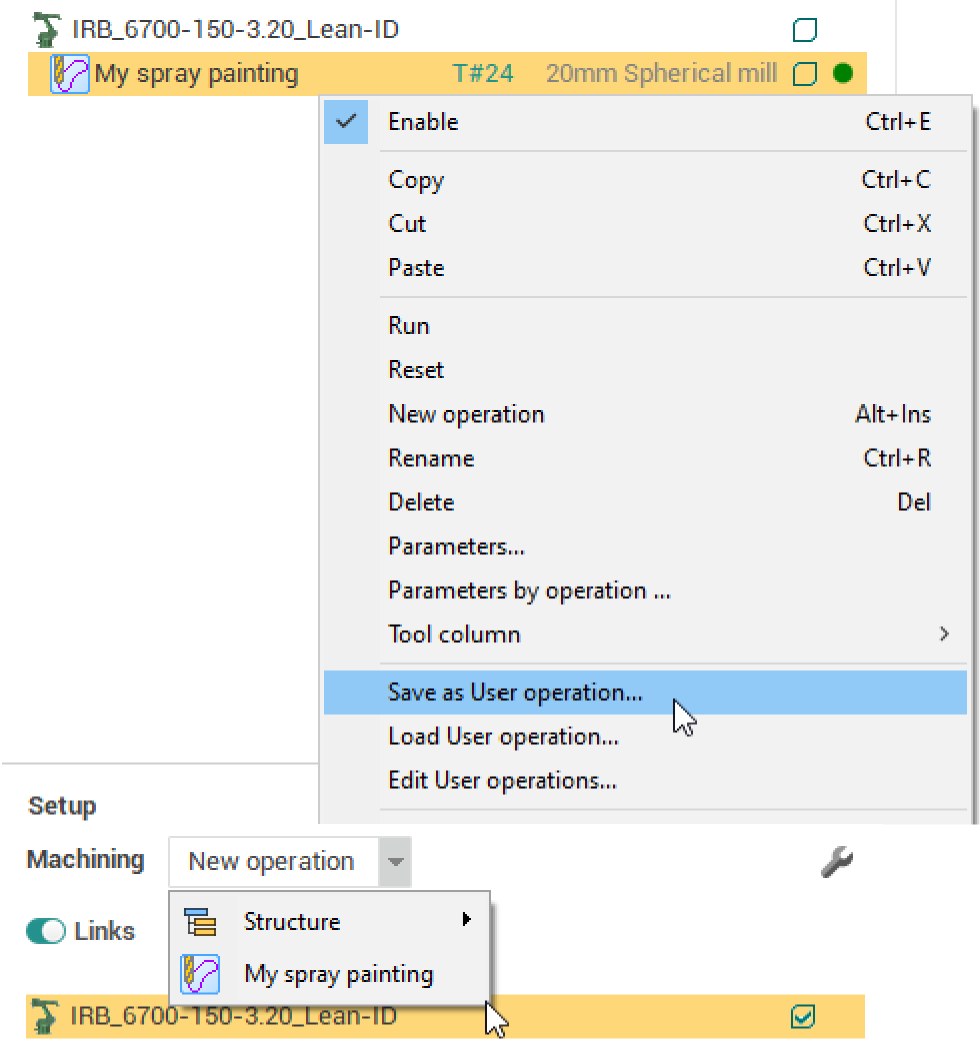
It's a versatile tool. By changing its properties, you can flexibly adjust the shape of the torch. Allows you to simulate the case of several nozzles lined up in a row.

Use the Mounting angle property to rotate the main direction.</td><td>
</td></tr>
</table>

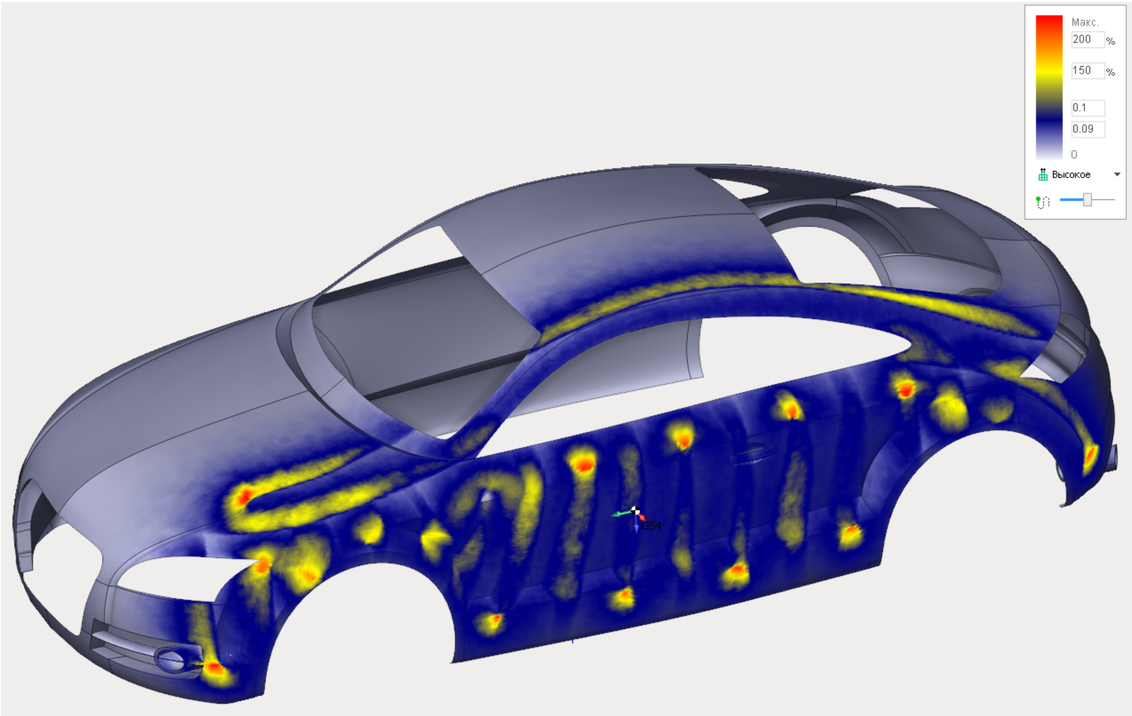
All spray tools contain the **Virtual touch diameter** property. It is a diameter of virtual spherical tool, like shown on the picture below. It is used in toolpath calculation to calculate tool-to-part contact. This makes it possible to use an existing universal strategies for toolpath calculation.



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 on this feature.



After the operation toolpath is ready, you can go to the **Simulation** tab to check the degree of coloring of the part surfaces. See Painting simulation section for more details.

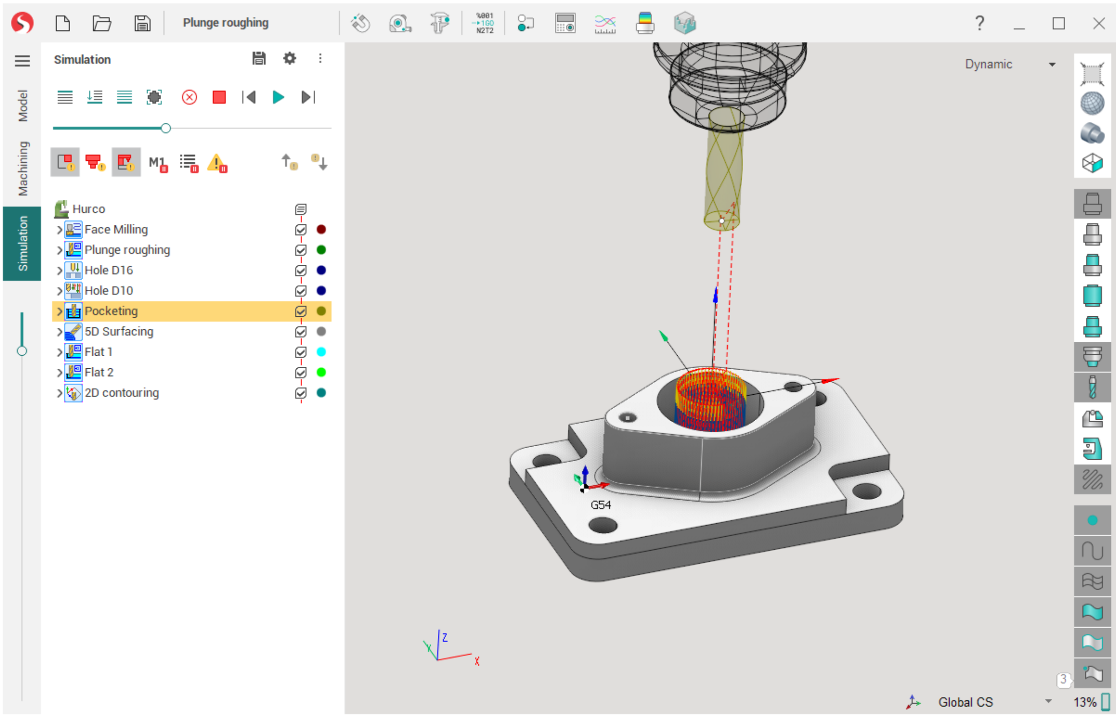

Макс.
200 %
150 %
0.1
0.09
0
Высокое
G54

6 Simulation

No content in this page. See child topics

6.1 Designation of the simulation mode

SprutCAM X has the embedded high quality multi-axis mill/turn machining simulation module. The module allows exact simulation and error checking of turning, threading, drilling and multi-axis milling.



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.

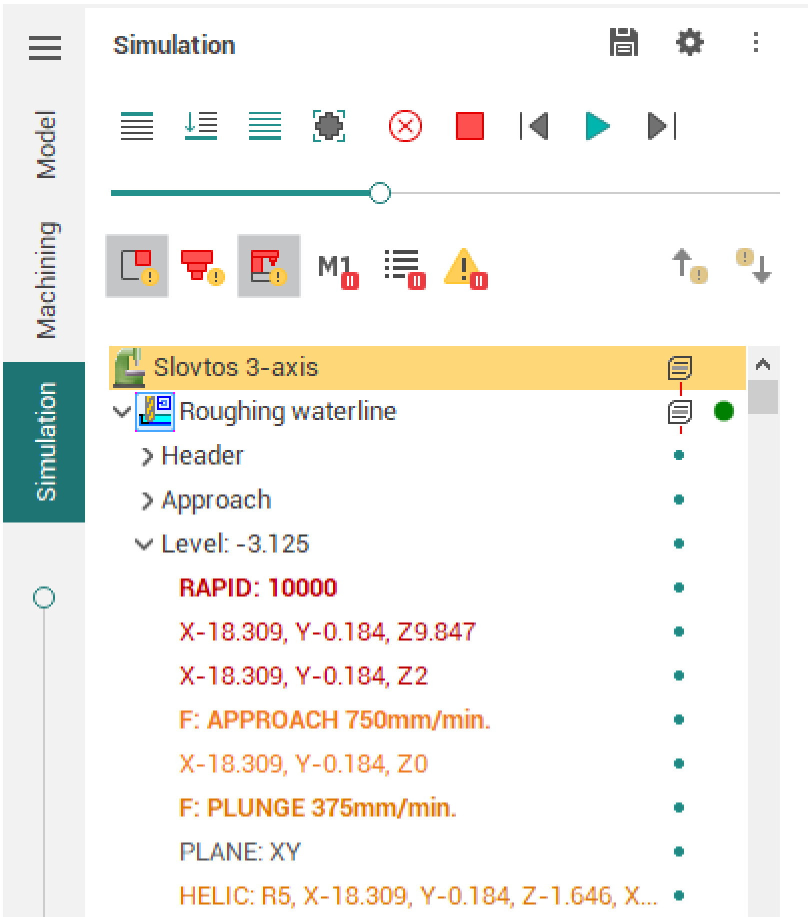
While simulating turn operations solids of revolution are used to represent a tool and a workpiece irrespective to the form of the source models. It is so to gain an effect of a revolving workpiece on a machine. If a mill operation follows a turn one, the true solid model is used for its simulation again.

6.2 Tool path motion

No content in this page. See child topics

6.2.1 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.

6.2.2 The list of the basic CL-data commands

The CLDATA command that is generated by SprutCAM is listed in the table. The full list of the CLDATA commands see in the part 3 of this manual <Postprocessors generator>.

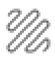
Command	Description
AXESBRAKE	Machine axes brakes control
CIRCLE	Motion by circle arc

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
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

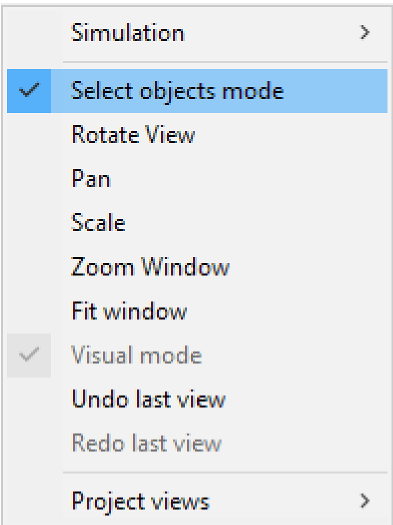
6.2.3 The selection of the CL-data commands from the graphical view

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.

6.2.4 Tool path editing

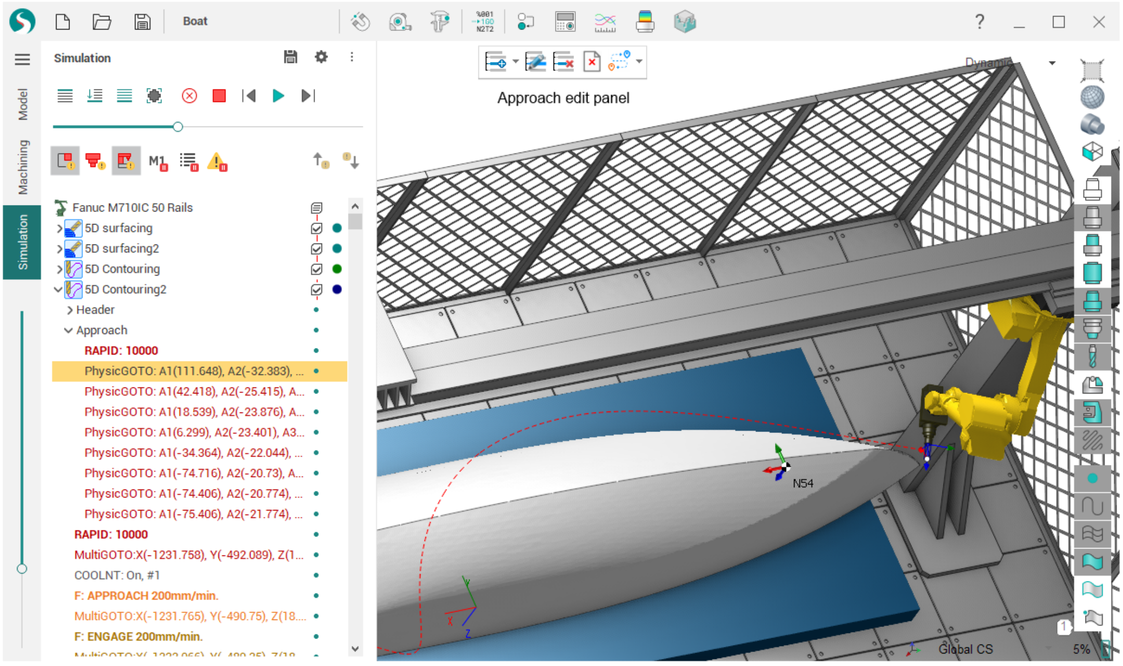
There is the possibility to edit the sequence of the CLDATA commands. The tool path re-calculation resets all changes.

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:

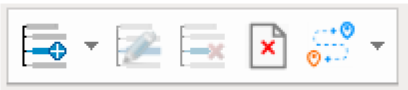
- [Del] – deletes the current node of the tool path tree;
- [Ctrl+X] – cuts the current node of the tool path tree into the clipboard;
- [Ctrl+C] – copy the current node of the tool path tree into the clipboard;
- [Ctrl+V] – inserts the command from the clipboard before the current node.

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

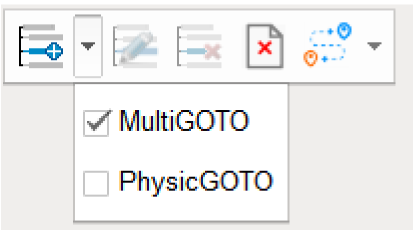


If the user selected the 'Approach' or 'Return' section inside tool path tree, additional panel becomes visible (see screenshot). On this panel the following buttons are available for more convenient editing of approach or return section (buttons are listed from left to right):

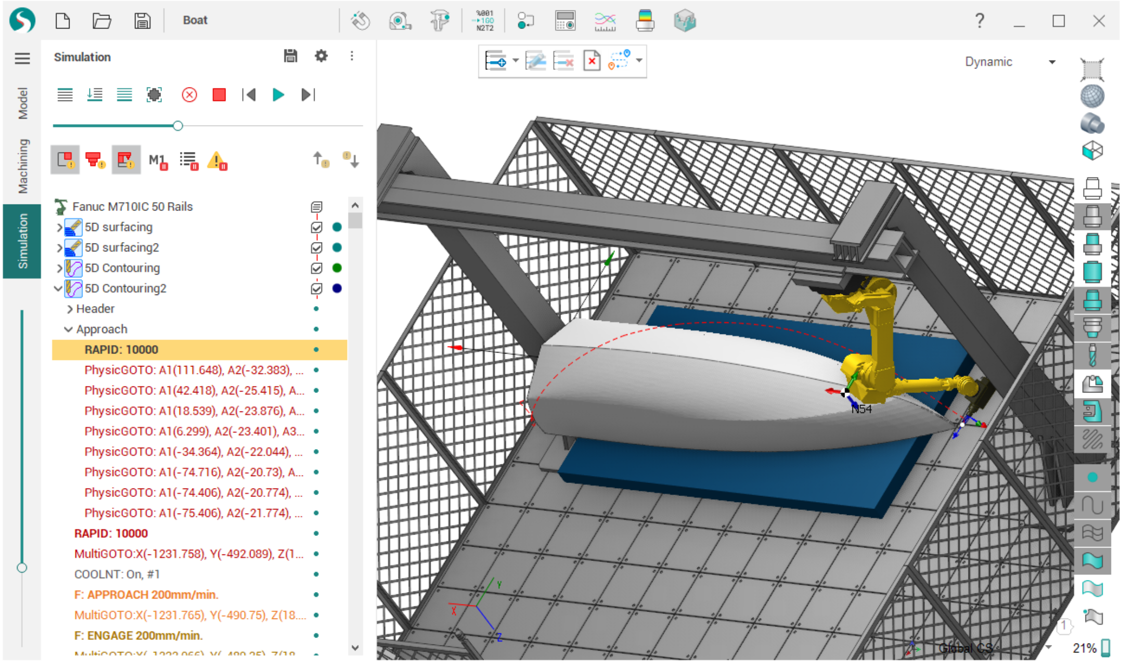


- **Insert current state** – this command inserts current machine state as a node of tool path tree. The state is inserted before current selected node of the tree.

Dropdown menu can be used to change type of inserted node (**PhysicGOTO** or **MultiGOTO**).



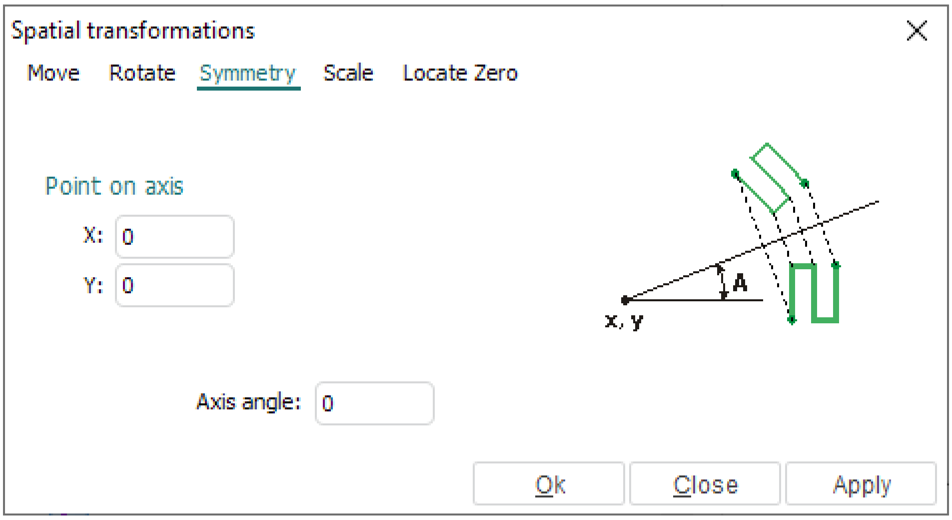
- **Edit current state** – press this button to start editing interactively current selected state of tool path tree. The coordinates of selected node are synchronized with current machine state. To end interactive editing process press this button again.
- **Delete current state** – deletes the current 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.

6.2.5 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.

6.3 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.

See also:

Tool motion controlling

Tool path errors detected by simulation

Feed rates optimization

Assigning workpiece parameters

Turbo simulation mode

Export simulation result as model

6.3.1 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.

See also:

Controlling simulation process

6.3.2 Tool path errors detected by simulation

Every node of the toolpath tree has the status. The status of the node is indicated by an icon:

- < > – Operation is disabled. It is not output to the NC program and not simulated;
- < > – Operation is not calculated (has no toolpath);
- < > – Operation is calculated (has the toolpath);
- < > – Operation is calculated and simulated without error;
- < > – Operation is simulated with errors (has the CLDATA commands with error);
- < > – 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;

- < > – CLDATA command is idle. The stock is not removed by this tool motion command.

During the simulation process, the influence of the CLDATA command on the workpiece is analyzed, and corresponding status is assigned to the command.

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 (appears if Check for gouges option in Operation Parameters inspector is enabled).
- — 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).

The type of error is described in the hint that appears after a delay under the CLDATA command with error.

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:

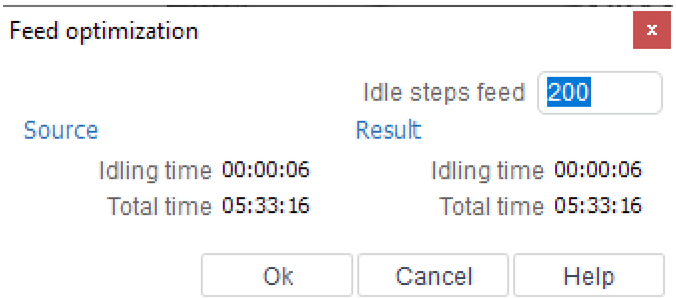
- <Go To Next error> – move the selection to the next error, marked node;
- <Go To Previous error> – move the selection to the previous error marked node.

See also:

Controlling simulation process

6.3.3 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.

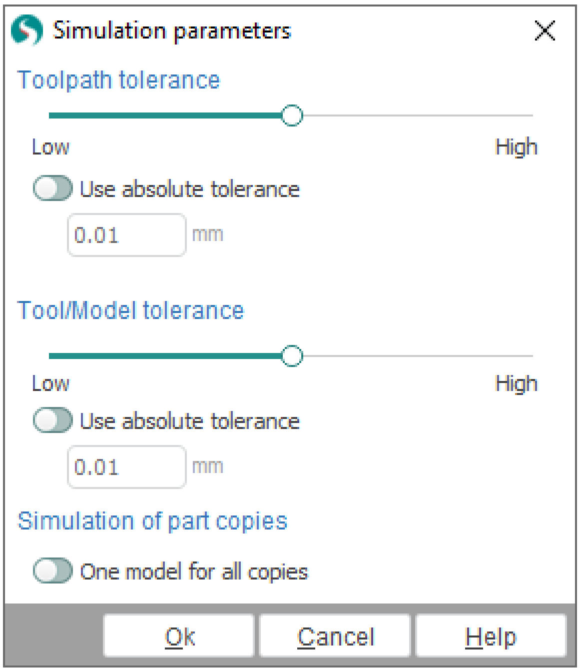
Note: *In an operation does not have the idle motions then the optimization does not give any effect. If the idle steps feed value is smaller then the real feed value then the optimization does not have the sense because of the machining time increases.*

See also:

Controlling simulation process

6.3.4 Assigning workpiece parameters

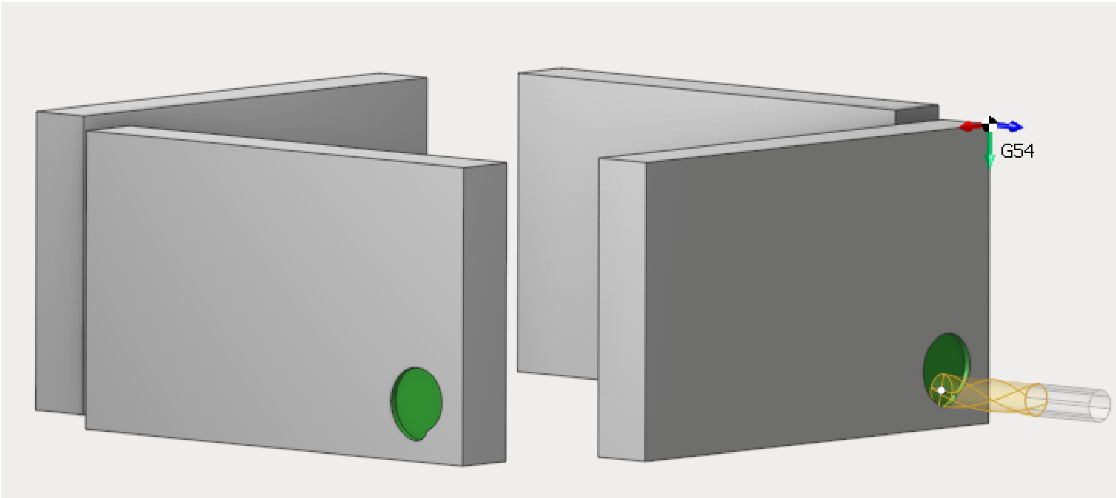
The simulation settings editing window opens when the button in the machining simulation window is pressed.



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*

See also:

Controlling simulation process

6.3.5 Turbo simulation mode

While simulating a 3d Milling project in SprutCAM X 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.

See also:

Controlling simulation process

6.3.6 Export simulation result as model

Click the in [the simulation mode](#) or <Export simulation result> in the [main menu](#) to save the simulation result that is on the screen to the file. Input the exported file name in the STL format. The file you will get can be used as a workpiece in the another SprutCAM X project or in the any software that can import STL file format.

See also:

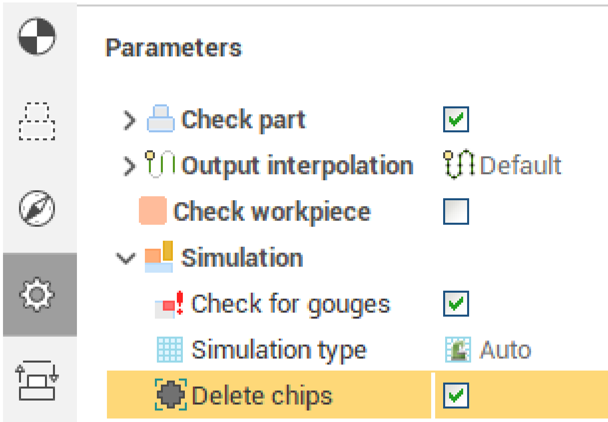
Controlling simulation process

6.3.7 Delete chips function

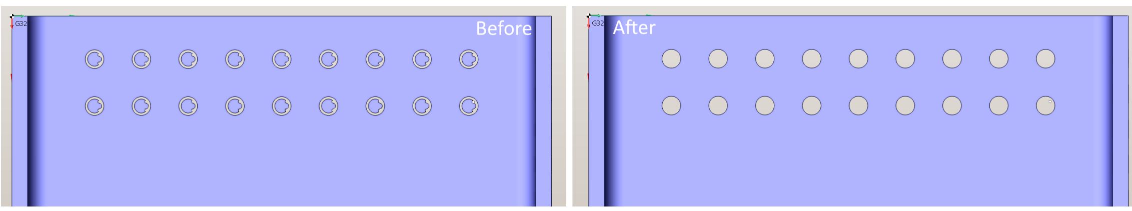
The function deletes chunks of the workpiece (or simply, chips) that do not contain the part geometry. (So, presence of the part is **crucial** for the **correctness** of the result).

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:



6.4 G-code based simulation

G-code based simulation allows considering features of the implementation of the postprocessor in the simulation processing. In this mode, the system automatically generates NC code for each operation while calculating. Controlling simulation is performed as Controlling simulation process.

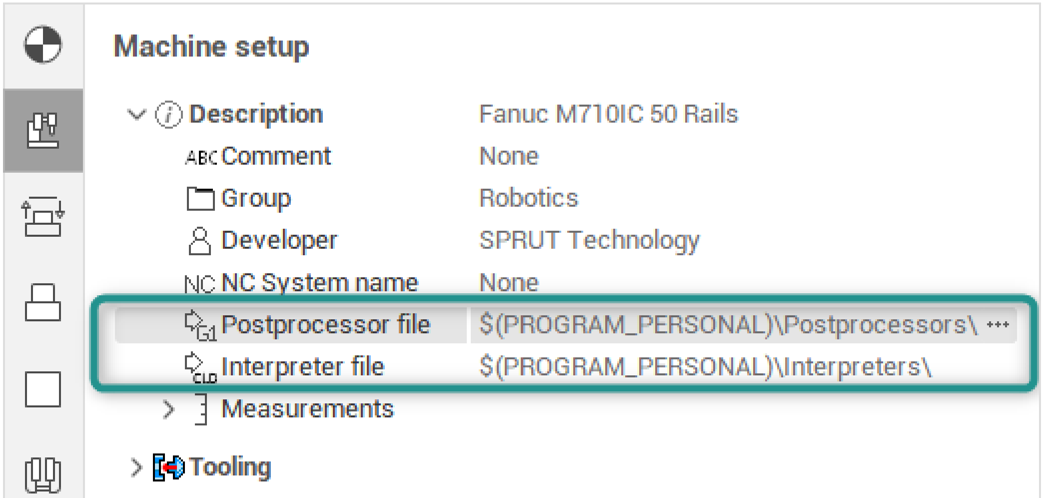
G-code based simulation function activation.

Activating of the function is carried out by pressing  button on the toolbar, with the active <Simulation> tab.

Note*: The button becomes unavailable if the* multitask machining *is used;*

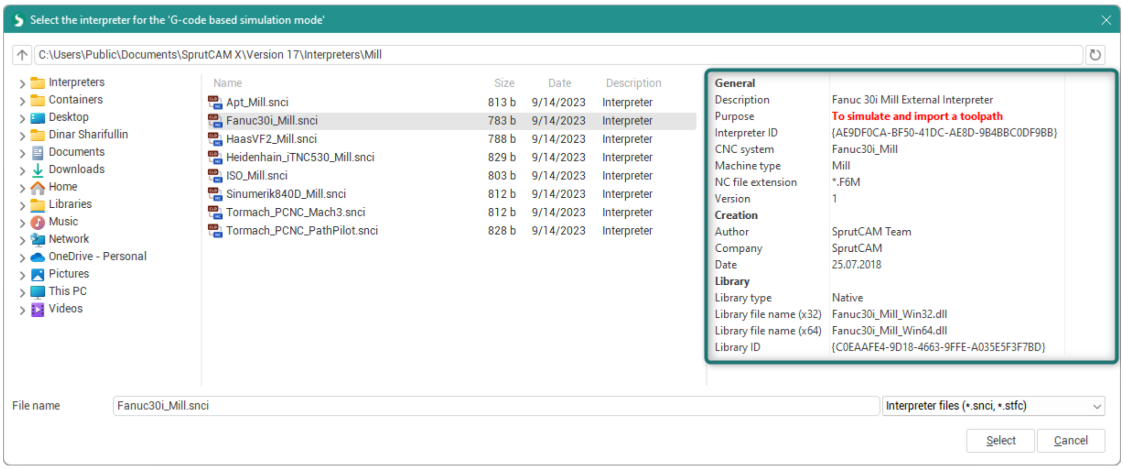
G-code based simulation parameters

G-code based simulation is formed on the basis on the selected postprocessor file and interpreter file specified on the <Machine setup> panel of the machine.

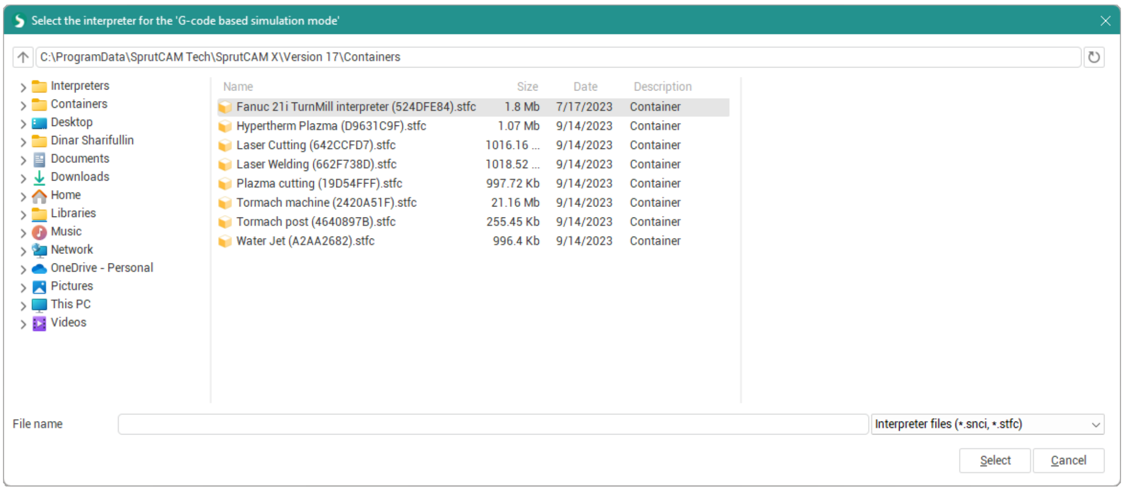


The interpreter is a CNC machine system settings file (* .snci), located in the directory $(PROGRAM_PERSONAL)\Interpreters.

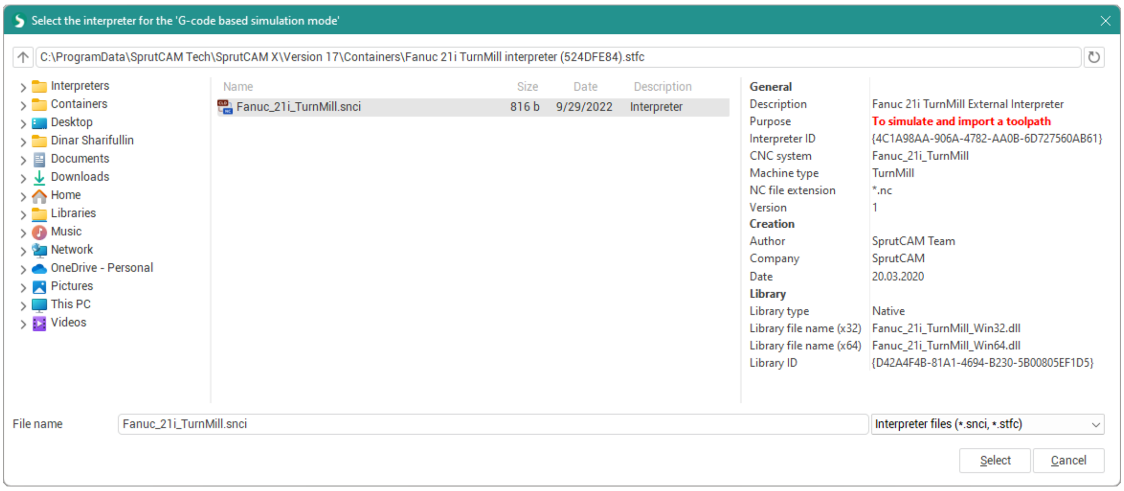
During the selection process, a preview of the interpreter information is available (description, purpose, CNC system, authors, etc.):



The ability to select an interpreter from the container is supported. To do this, get a container with an interpreter. Then open the interpreter's selection window and go to the container folder.



Enter the resulting container as a folder, select a simulation interpreter inside the container.



Setting the postprocessor and interpreter "default" in the kinematic scheme of the machine

You can specify the "default" name of the postprocessor file and interpreter file in the kinematic scheme of the machine. To do this, add the **SPPFile, SNCIFile** tags with links to the corresponding files to the machine XML file, and restart SprutCAM X.
Now, when selecting a machine in SprutCAM X, the postprocessor and the interpreter will be already set, and their values are obtained from the kinematic scheme. If necessary, from the SprutCAM X UI, you can override the postprocessor and interpreter values for the current project.
Example:
<SCType ID="Fanuc 30i" Caption=""Fanuc 30i" type=""Fanuc30i" Enabled="true">
<... other tags ... />
<**SPPFile** DefaultValue="$(PROGRAM_PERSONAL)\Postprocessors\Mill\Fanuc (30i)_Mill.sppx"/>
<**SNCIFile** DefaultValue="$(PROGRAM_PERSONAL)\Interpreters\Mill\Fanuc (30i)_Mill.snci"/>
<… other tags … />
</SCType>

Currently, interpreters of the following CNC systems are available for use:

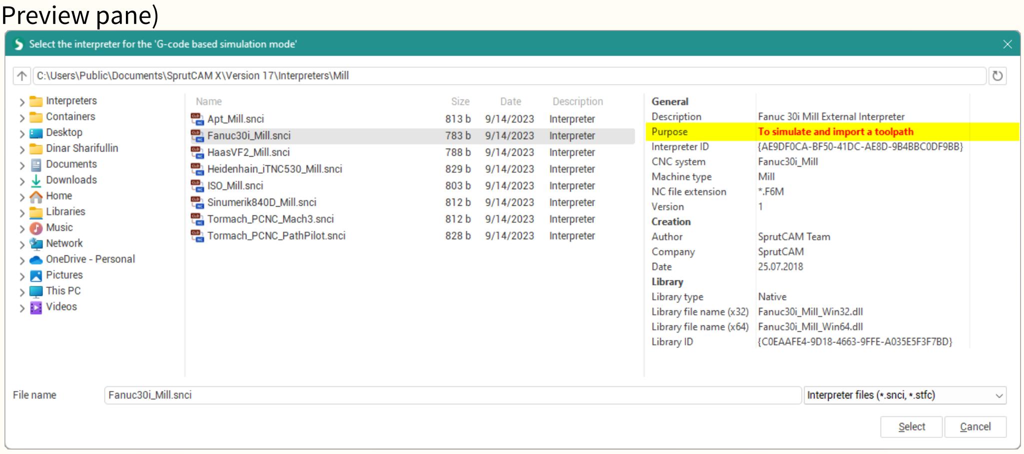
Machine group	CNC system	Comment	Note
Milling	APT	Import toolpath only	
	Apt_Simplify_3D	Import toolpath only	
	ISO	Import toolpath only	
	Global control	Import toolpath only	An additional license is required
	Fanuc 30i	To simulate and import a toolpath	
	Haas VF-2	To simulate and import a toolpath	
	Heidenhain iTNC 530	To simulate and import a toolpath	
	Mazatrol SmoothG	To simulate and import a toolpath	An additional license is required
	NC210	To simulate and import a toolpath	An additional license is required
	Sinumerik 840D	To simulate and import a toolpath	
	Tormach PCNC Mach3	To simulate and import a toolpath	
	Tormach PCNC PathPilot	To simulate and import a toolpath	
Turning	Mazatrol SmoothC	To simulate and import a toolpath	An additional license is required
Turn-milling	Fanuc 21i	To simulate and import a toolpath	An additional license is required
	NC220	To simulate and import a toolpath	An additional license is required
	Sinumerik 840D	To simulate and import a toolpath	
	Okuma OSP-P300	To simulate and import a toolpath	An additional license is required
Robot	Fanuc robot (R-30iB controller)	To simulate and import a toolpath	

Machine group	CNC system	Comment	Note
	Kuka robot	To simulate and import a toolpath	
	Motoman robot	To simulate and import a toolpath	
	ABB robot	To simulate and import a toolpath	
	Nachi robot (AW Format)	To simulate and import a toolpath	An additional license is required

Note: *All interpreters support command list generated by postprocessors in SprutCAM X distribution kit only.*

"Import toolpath only" interpreters are not supported matching line NC code - trajectory of tool movement.

When you select an interpreter, pay attention to its purpose (the **Purpose** field in the Preview pane)



The selected interpreter should be intended for simulation. Otherwise, the trajectory of the tool may be incorrect (shifted relative to the coordinate system of the workpiece, duplicated approaches/retracts, incorrect starting position, etc.).

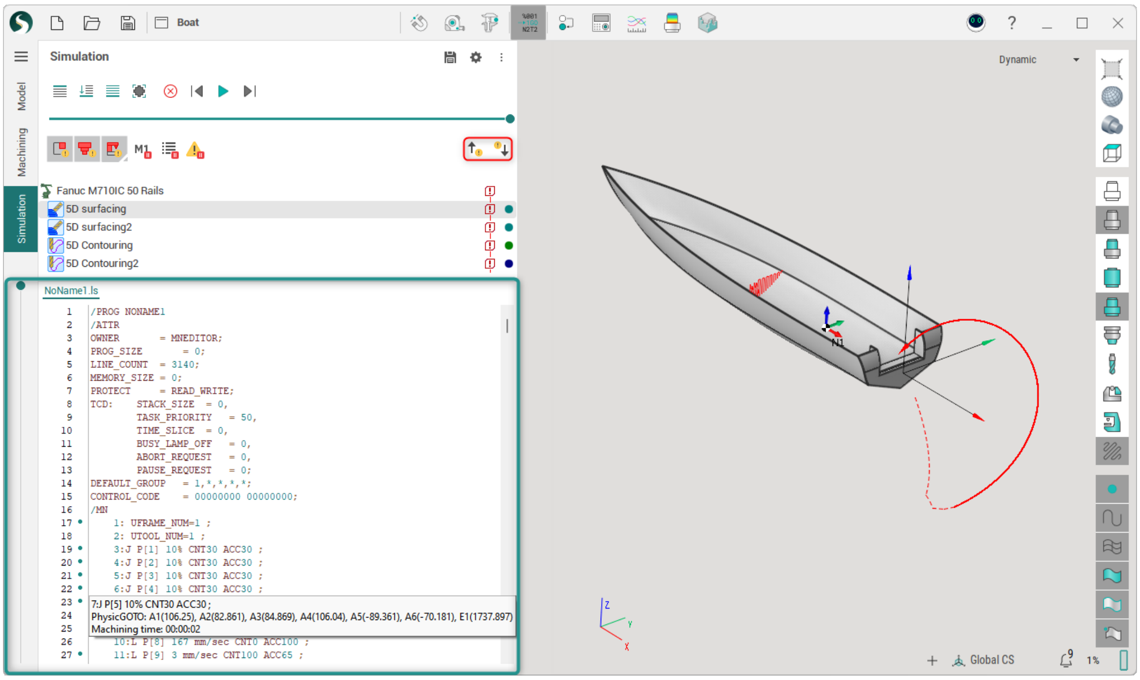
G-code based simulation features.

If the mode is enabled, then after the calculation of the toolpath SprutCAM X automatically generates a control program for CNC machine with pre-selected postprocessor settings file, perform the conversion of the NC code program text into the toolpath. The generated path will take into account the peculiarities of the implementation of the postprocessor. NC program text for the selected operation is displayed at the bottom of the <**Simulation**> mode page, immediately after the list of operations of the technological process.

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: [↓] or [↑].*

To the left of the NC program text, there is a gutter for displaying auxiliary information. 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.  and  buttons move the selection in the NC-code between the errors.



Support for third-party interpreters.

Supported third-party interpreters for modeling the text of the NC code. The file of interpreter settings (* .snci) should contain a link to the program library, which is used to interpret the NC code. The page Creating your own interpreter describes the process of creating of your own interpreter: settings file and application programming interface (API).

G-code based simulation demo video
Watch demo video

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:
https://www.youtube.com/watch?v=Wb_L3HkGF_I

See also:

Controlling simulation process

Selection of a machine and its parameters definition

Container manager

Creating your own interpreter

6.4.1 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 SprutCAM X, 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 SprutCAM X with the interpreter to obtain a secondary (restored) tool path is described in detail in the section G-code based program interpretation.

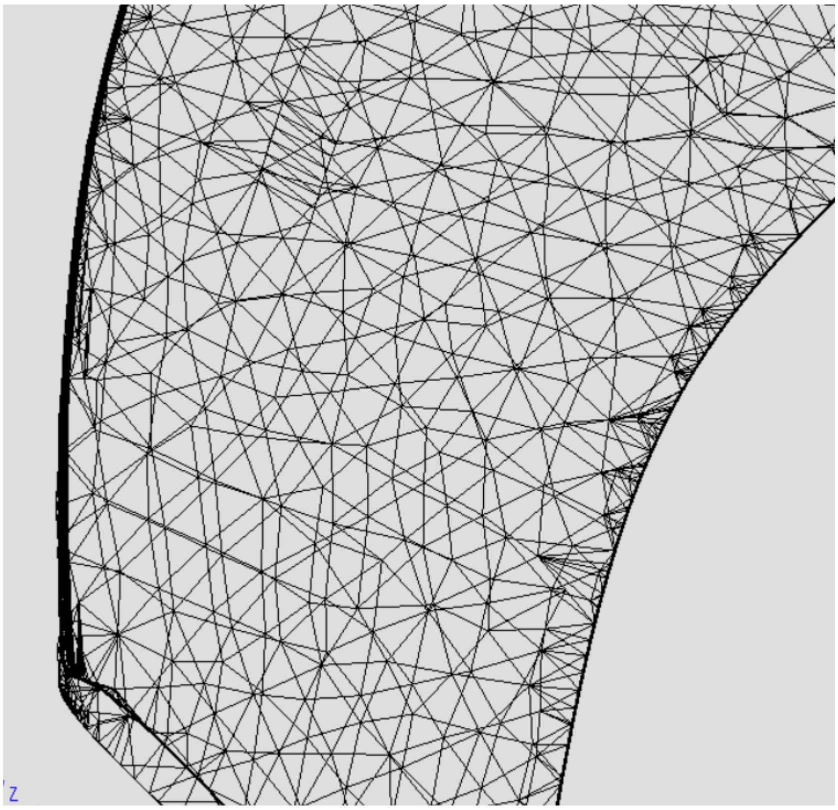
6.5 Painting simulation

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).

6.5.1 Recommendation for the model

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.



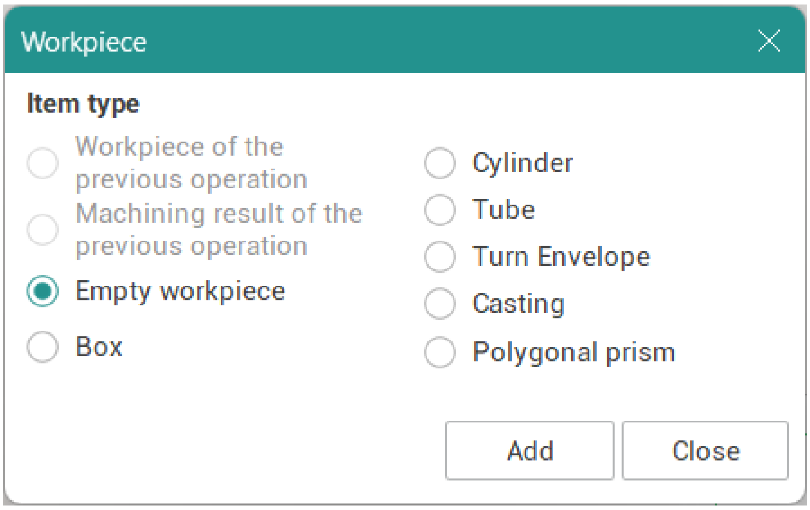
When you simplify your model, according to these recommendations, you will get much smoother and faster painting simulation.

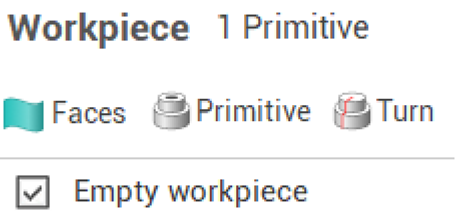
6.5.2 Preparing operation for painting

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 Simulation type | Painting in the operation parameters ;
- select the painting type Painting type .

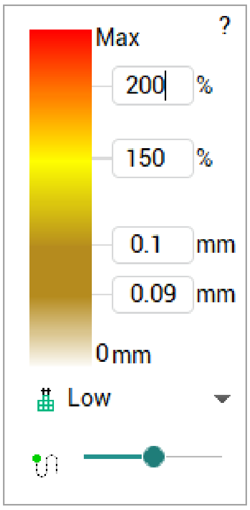
6.5.3 Painting types

There are two types of painting simulation in the system:

1. Simple 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.
2. Thickness measurement 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:
   a. set the work feed Work feed 10000 mm/min in operation feeds ;
   b. set the paint flow rate Paint flow rate 0.8 ml/s in operation parameters .
   After that, you can perform the calculation of the operation and go to the “Simulation” page.

6.5.4 Painting simulation form

On the **Simulation** page, when you select a painting operation in the operations tree, a form will appear that allows you to manage the painting simulation parameters.



The form displays a color scale, the lower color of which corresponds to the permissible thickness of the coating and is displayed by the color of the tool, while yellow and red colors are needed to display excessive thickness of paint overlay.

For permissible thickness, a range of values in mm can be specified. For excess thickness, the values are set in percent of the upper value of the permissible thickness (as in the example on the picture: 150% from 0.1 is 0.15).

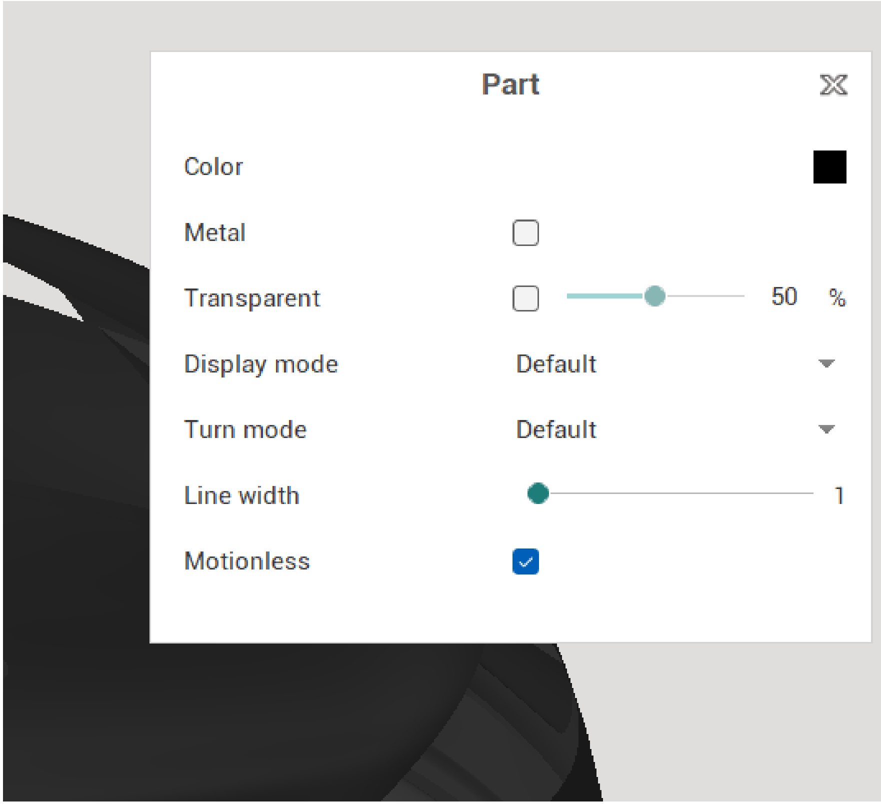
You can also change the resolution of the model, which will affect the accuracy and, accordingly, the performance. After changing the resolution, you should click "Reset" and run the simulation.

The last parameter on the form allows you to set the painting simulation step. With a large step accuracy decreases, but performance increases. This parameter, for example, can be used for quick results when high accuracy is not required.

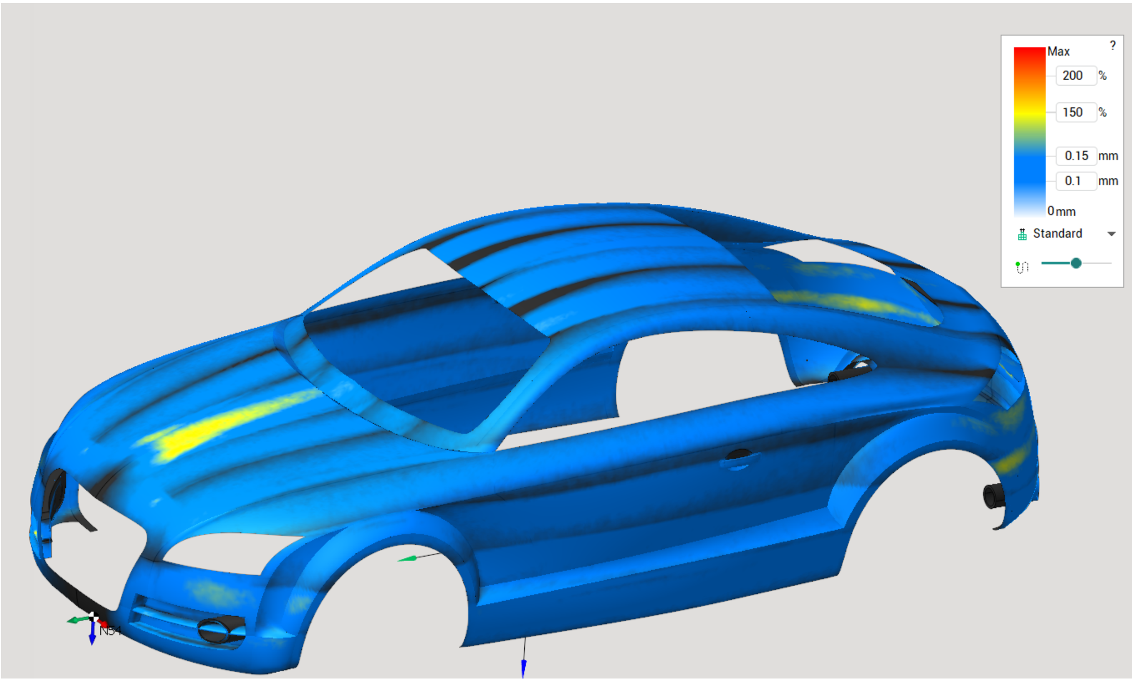
Note: The “Model resolution” parameter requires a reset. All other parameters have an impact on painting simulation immediately after the change.

6.5.5 Recommendation for the part color

We recommend you to set the **black** color for the part on Simulation page, because it's much better to control thickness of painting simulation result in a such way.

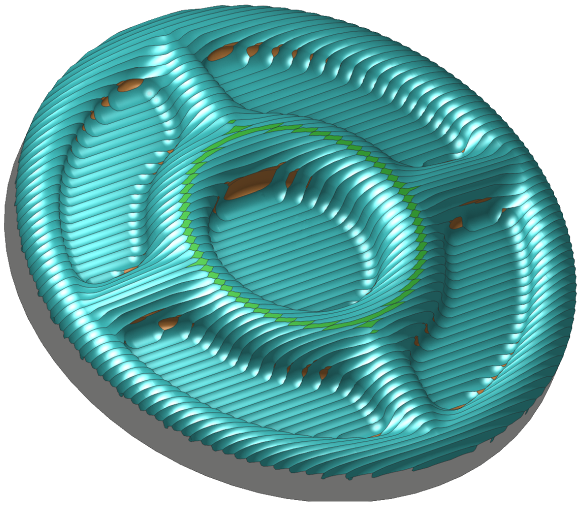


6.5.6 Example of painting simulation

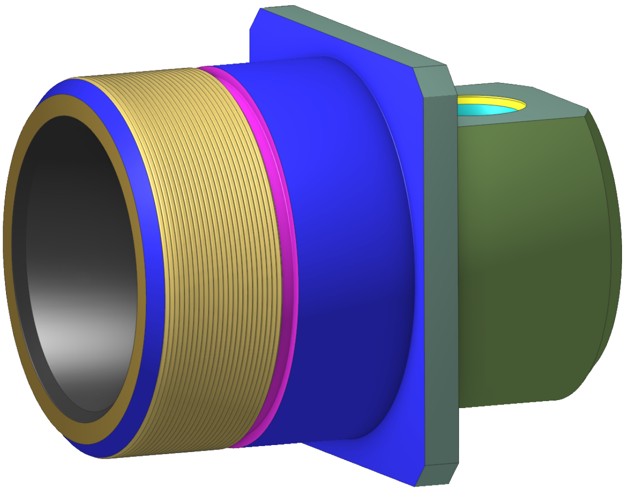


6.6 Solid Simulation

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.

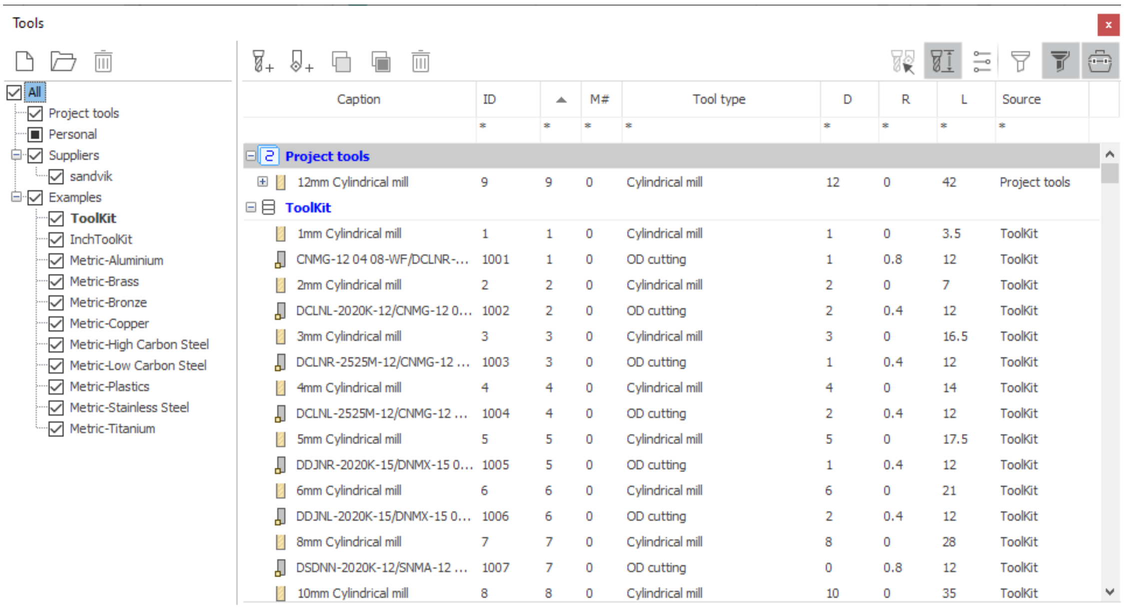


Notice: simulation of milling for 5-axis toolpaths works with limitations in beta version

7 Machining tool features

No content in this page. See child topics

7.1 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:

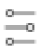
- when you click the  Tools button on the main toolbar in Machining mode;
- open the operation parameters window on the Tool tab;
- double clicking on the name in the tool column in the list of operations;
- clicking on the tool select button in the property inspector header.

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.

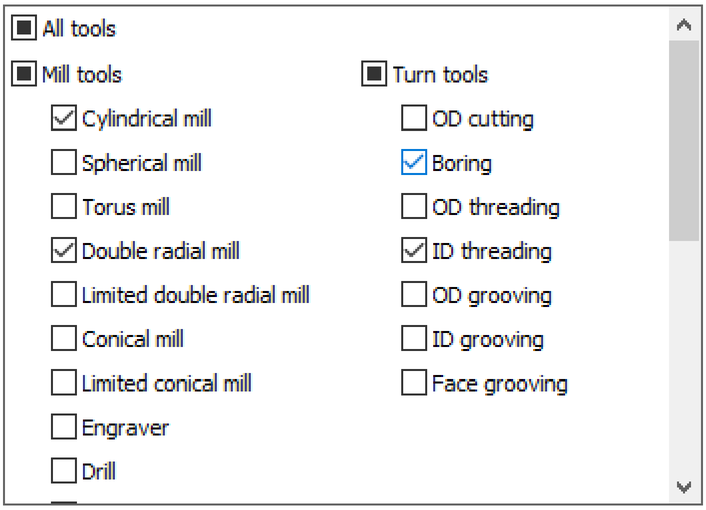
- — makes visible the tools used in project operations.
- — makes visible tools that are not used in any of the project operations.

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.



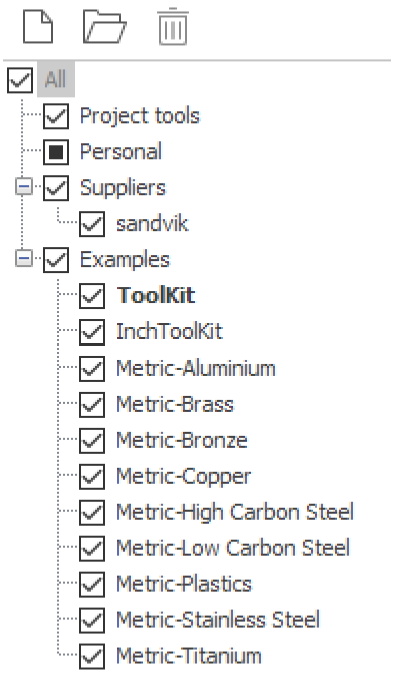
Sorting table rows is controlled by clicking on the desired column header.

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
    a. have the same tool number, magazine number and corrector number at the same time;
    b. 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.

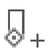
- **D** - for milling tools, the diameter of the tool; for turning tools, it either does not fill up or is filled with different parameters depending on the tool subtype.
- **R** - key tool radius (different depending on the specific type).
- **F** - working feed.
- **S** - spindle speed.
- **Direction** - 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.



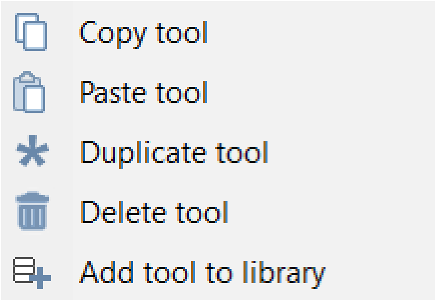
A more detailed description of the actions that can be performed on libraries using this panel can be found in the Managing libraries chapter.

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.

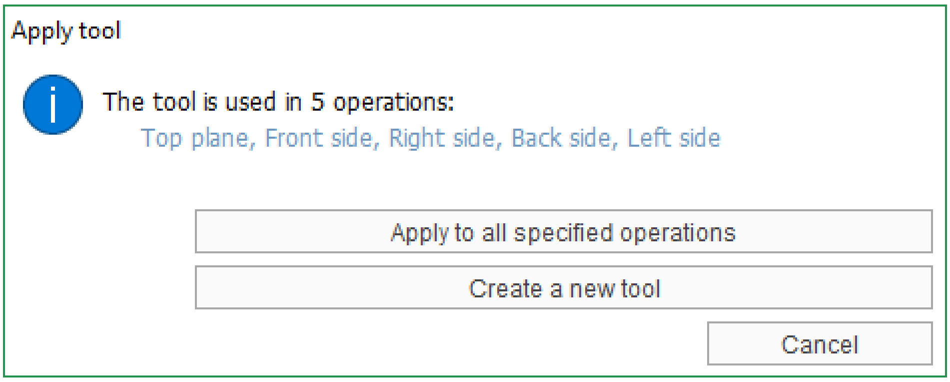
If a project tool is being edited,

- 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.

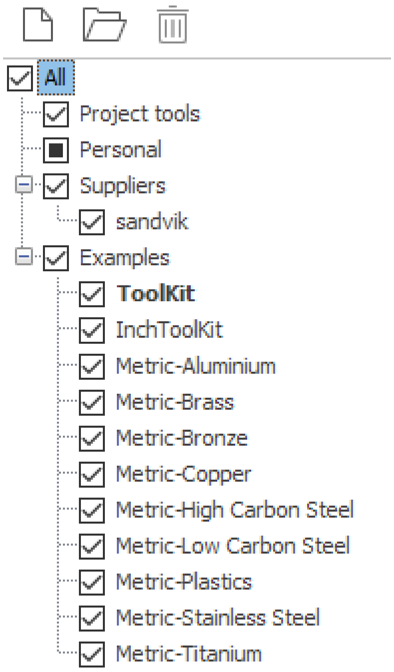
The Create a new tool button assigns a new ID to the tool of the current operation (it will be generated automatically) and creates a copy of this tool in the global list of project tools. Thus, the connection with copies of the tool in other operations is broken.

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.

7.1.1 Manage libraries

To manage tool libraries and project's tools, use the panel that is in the left part of the Tools window.



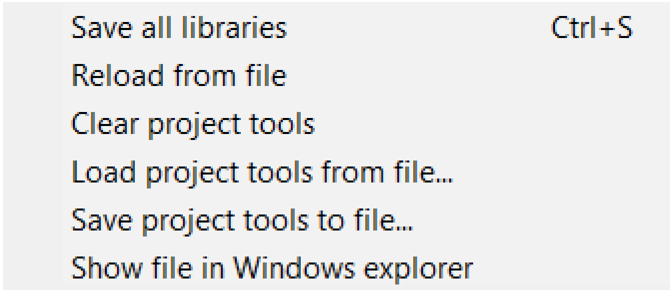
The panel displays all tool libraries connected to the system and divides them into several groups:

- **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.

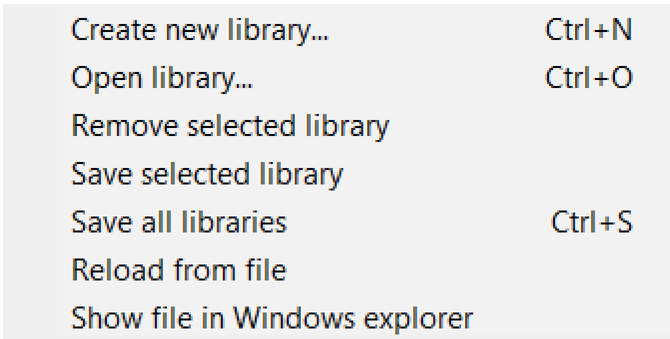
In the first place is always the **Project tools** item. This is a list of the tools of the current project, which is stored inside the stcp file, but can be saved in a separate text file (XML with *.tom file name extension). You can learn more about it in the Project tools chapter. The key difference 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).

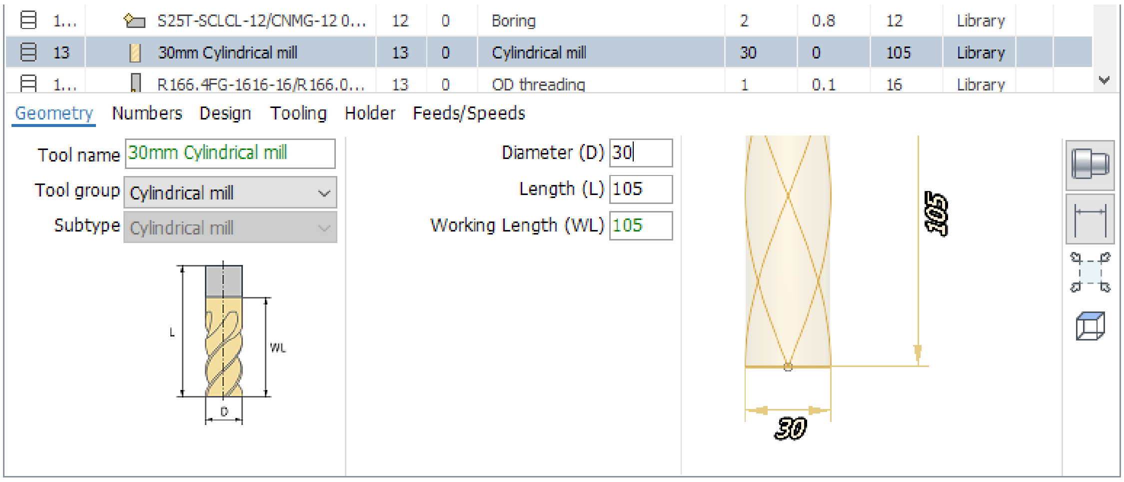


The following **actions** are available for the tool library from the popup menu and buttons on the top toolbar:

- 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).

7.1.2 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:

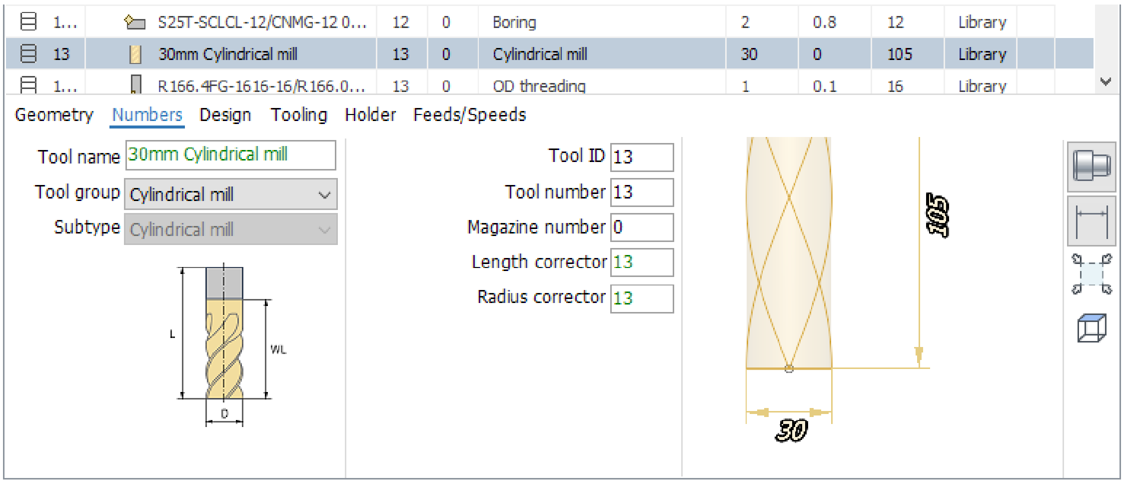
- Cylindrical mill;
- Spherical mill;
- Torus mill;
- Double radius mill;
- Limited double radius mill;
- Conical mill;
- Mill with negative radius;
- Limited conical mill;
- Engraver;
- Drill;
- Cutter;
- Tap;
- Thread mill;
- Center drill;
- Countersink;
- T-slot mill;
- Shaped mill;
- Knife;
- Saw 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.

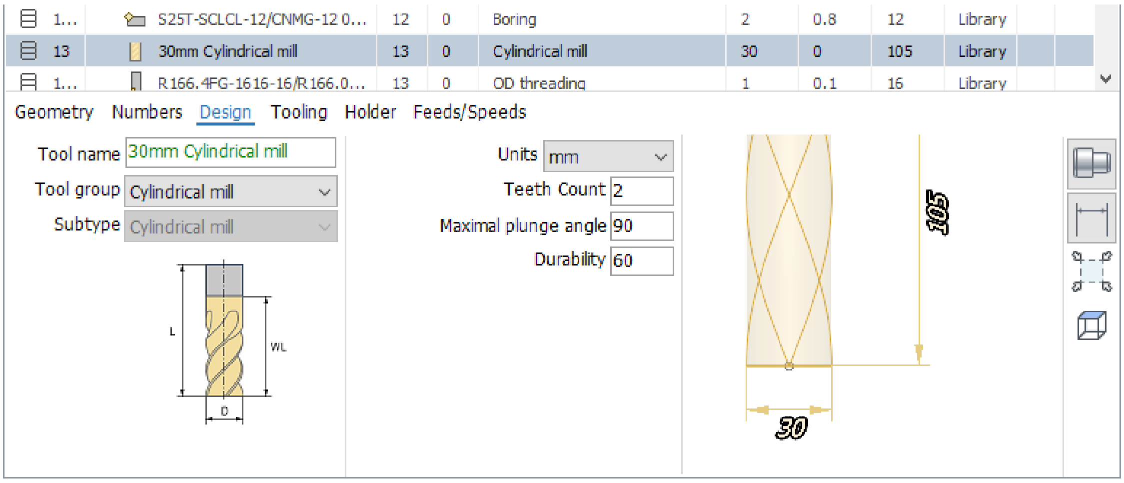
- ID is a unique tool identifier independent of its location on the machine.
- Tool number - the tool number on the machine. Usually corresponds to the position number in which the tool is fixed on the machine.
- The tool magazine number is used if there are more than one magazines on the machine.

- 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.



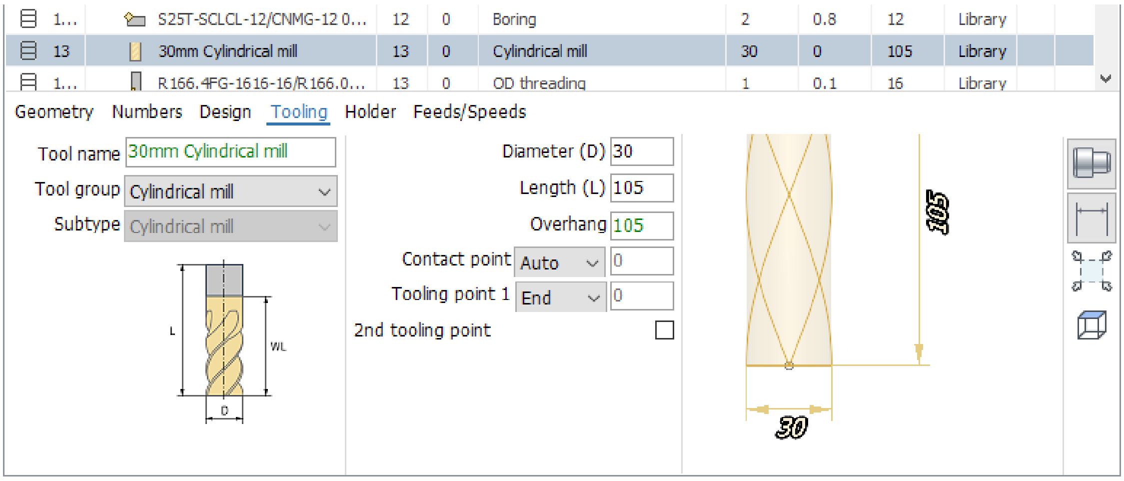
The following parameters are listed on the Design page.

- The number of teeth.
- Maximal plunge angle
- Tool units (mm or inch).
- Tool durability in minutes.



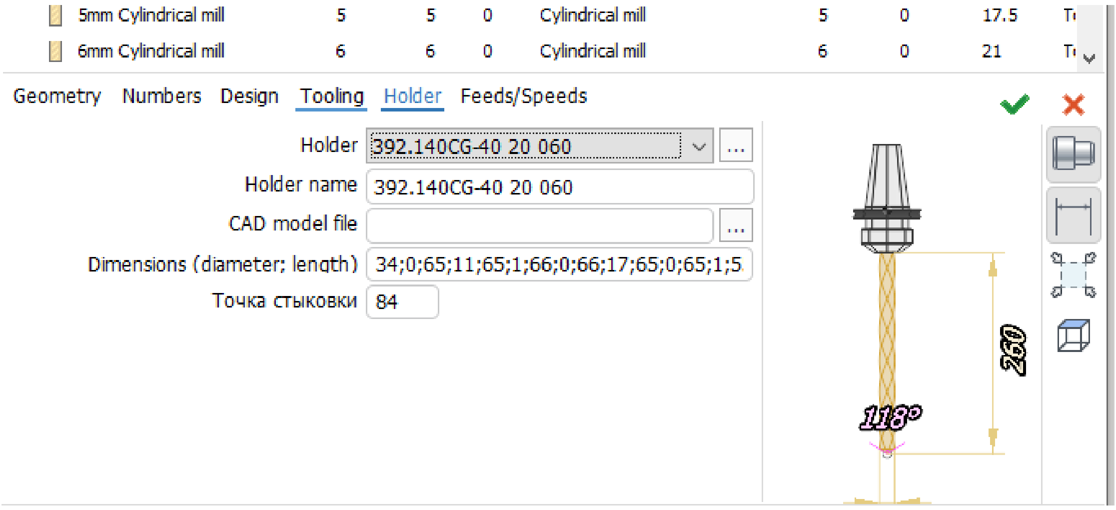
The Tooling tab contains such parameters.

- 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.



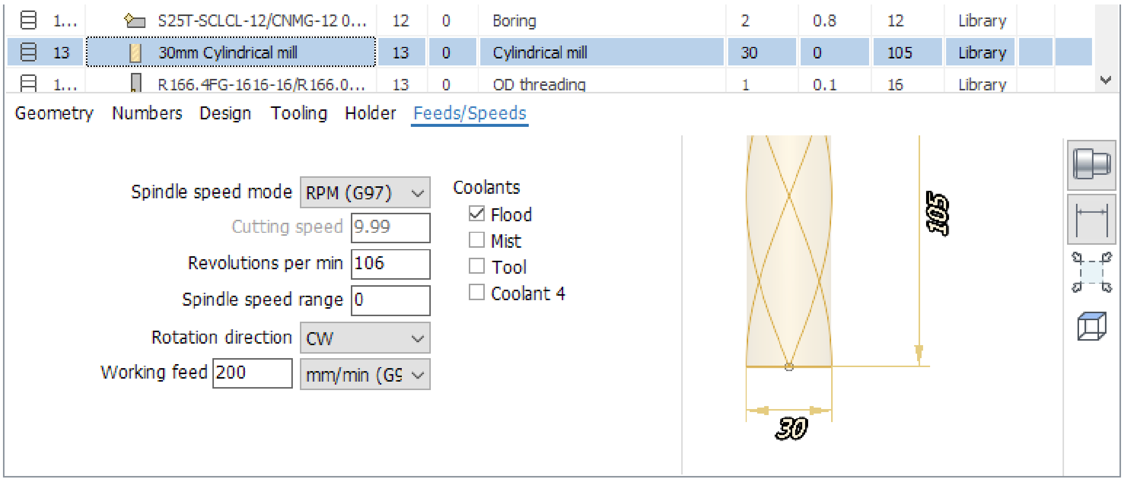
The Holder page allows you to set the parameters of the holder.

- 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.

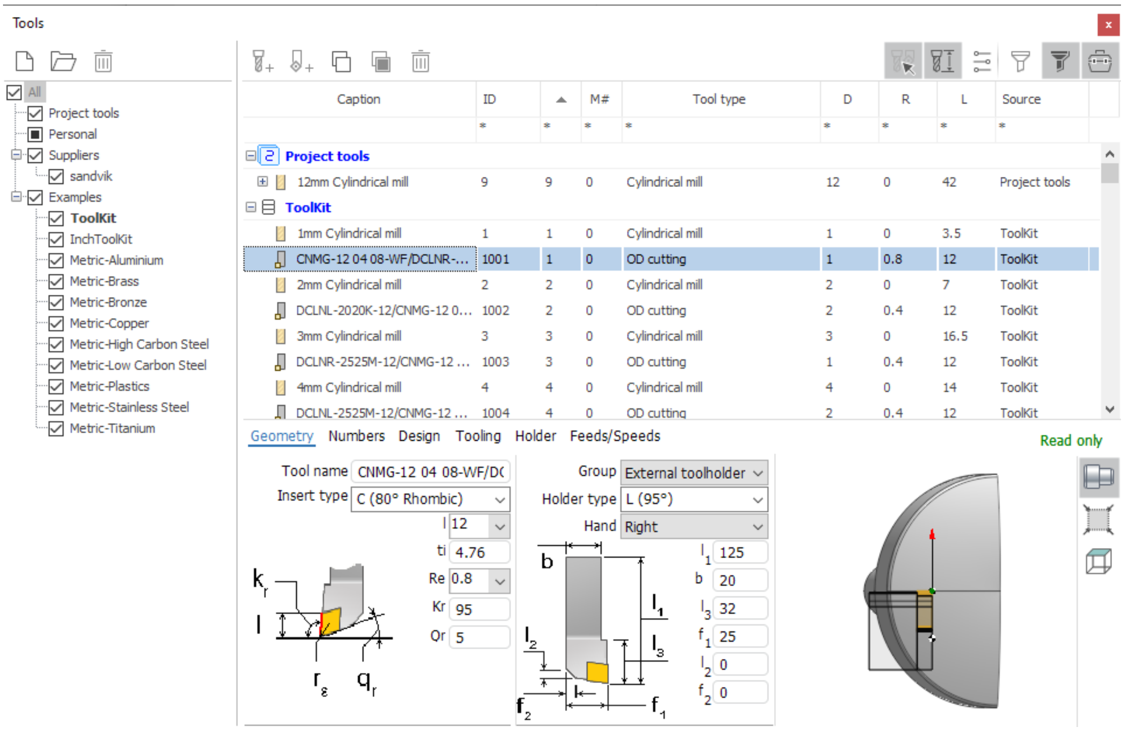


On the Feeds/Speeds tab you can edit such parameters as

- 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.



7.1.3 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.

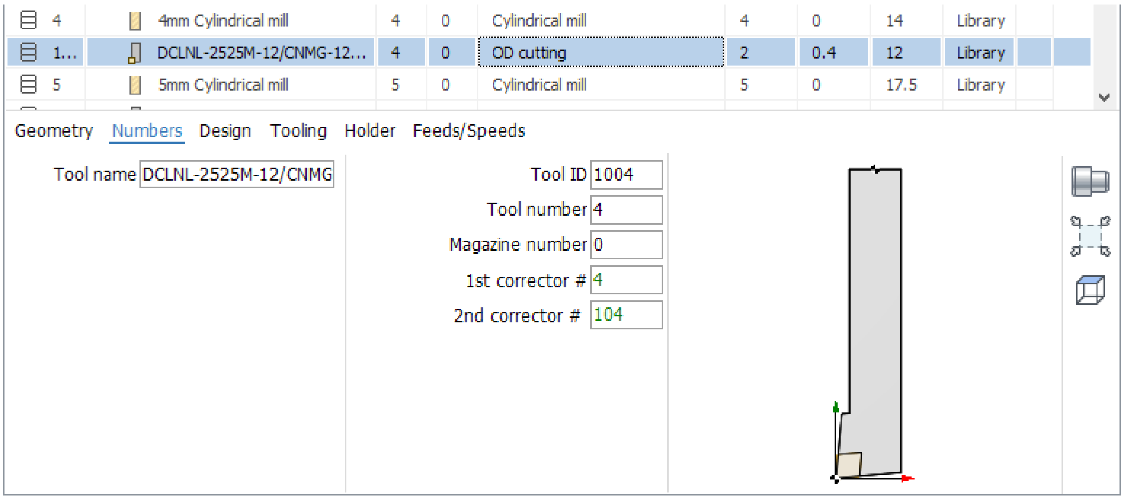
The Geometry tab displays the main geometric dimensions of the insert and holder, as well as the type of insert, type of holder, tool name. The composition of properties varies depending on the selected types.

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.

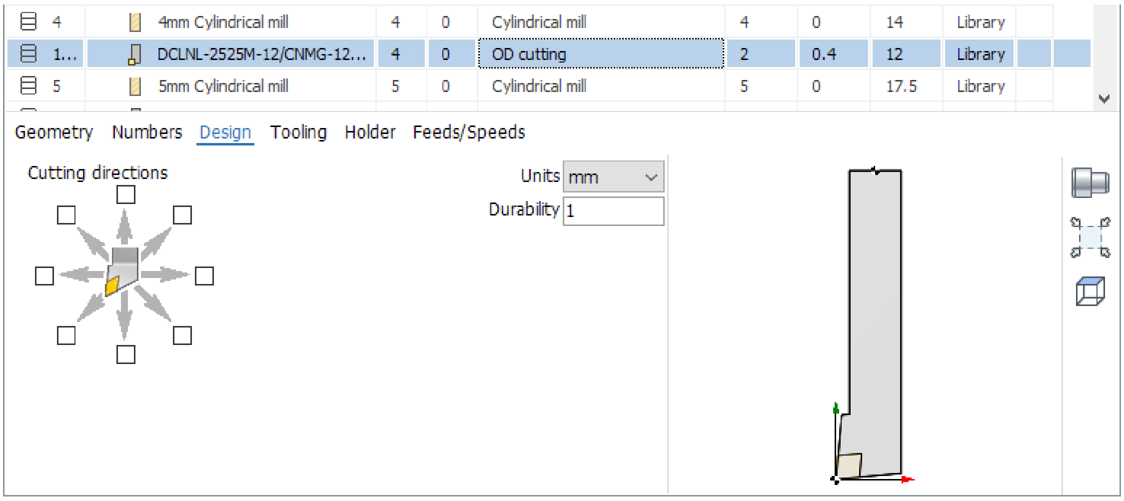
On the Numbers page, the tool identifier properties are displayed.

- 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.

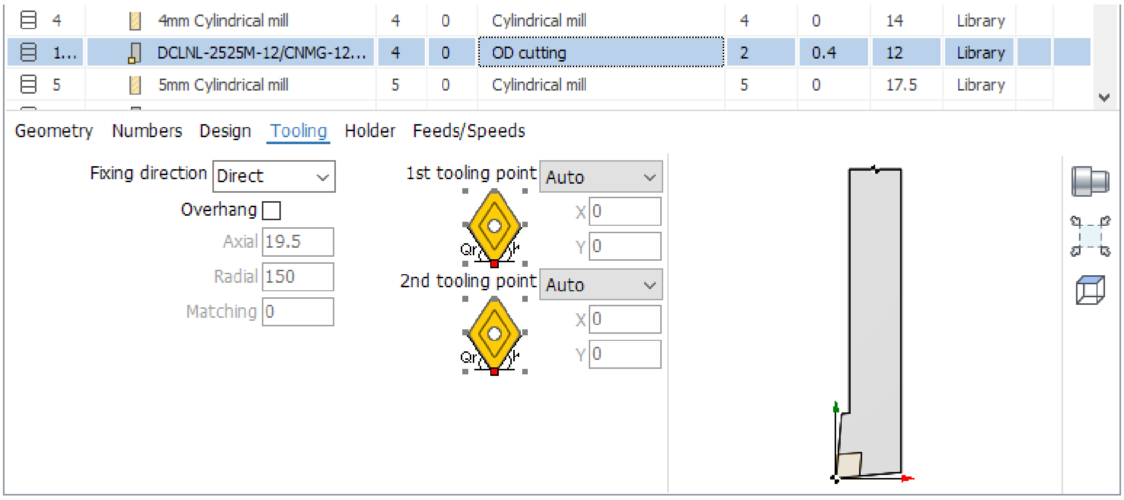
- Permissible cutting directions.
- Tool units (mm or inch).
- Tool durability in minutes.



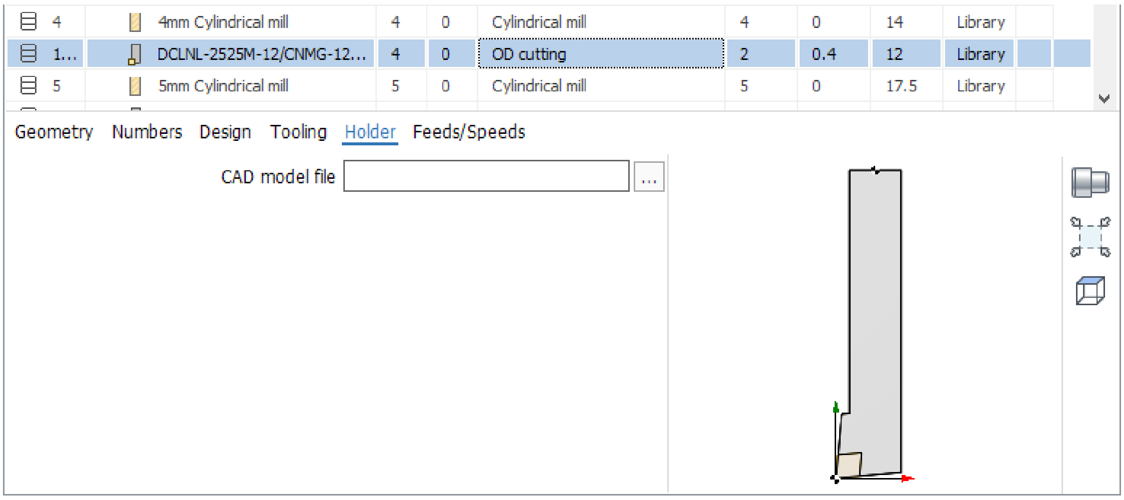
The Tooling tab contains such parameters.

- 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.

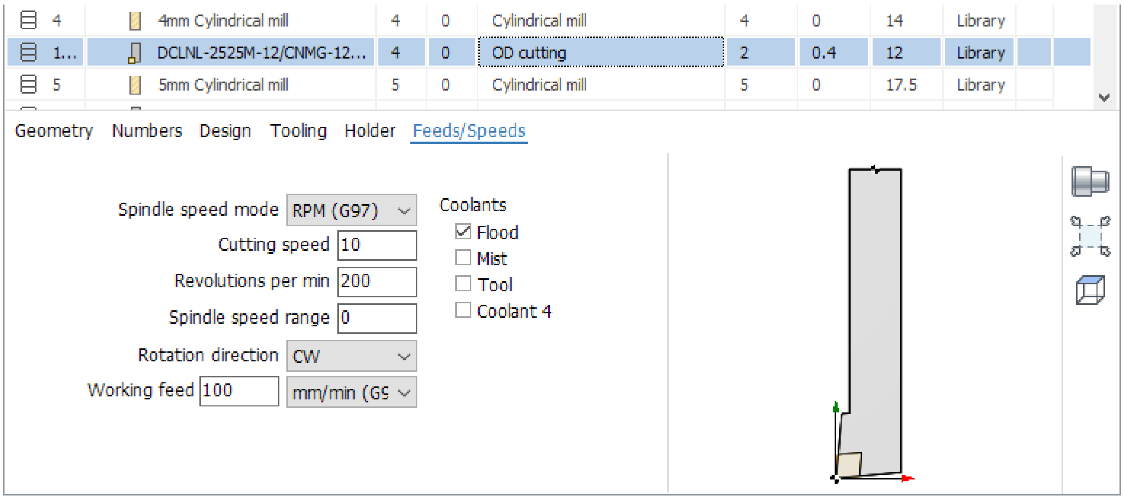


The Holder page allows you to specify the name of the CAD model file. The button opens a standard file selection window. You can specify files in the format *.stl and *.osd. Sets the 3D model of the holder for visualization and collision control during simulation.

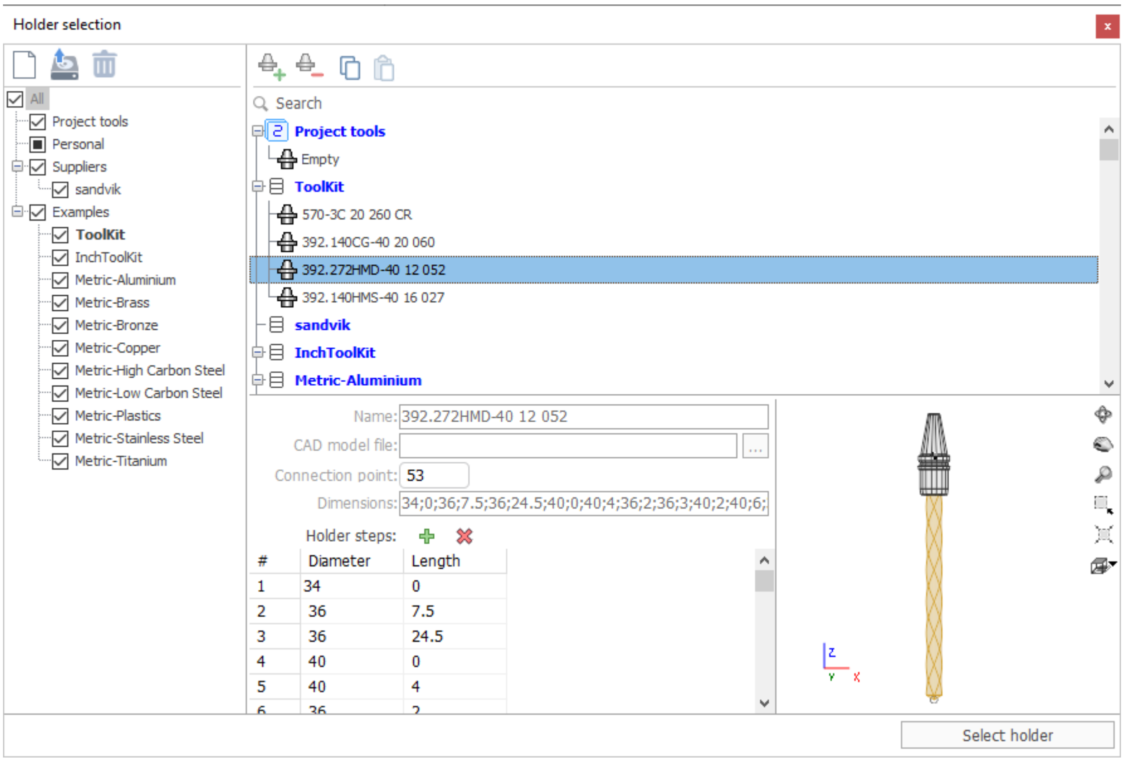


On the Feeds/Speeds tab you can edit such parameters as

- 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.



7.1.4 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.

7.2 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.

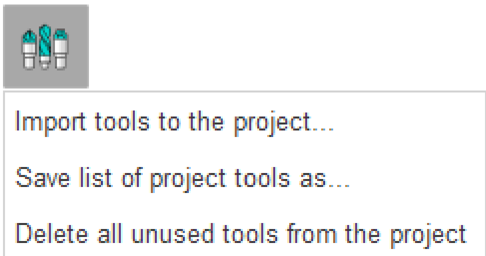
For each tool in the list is stored such data.

- **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.

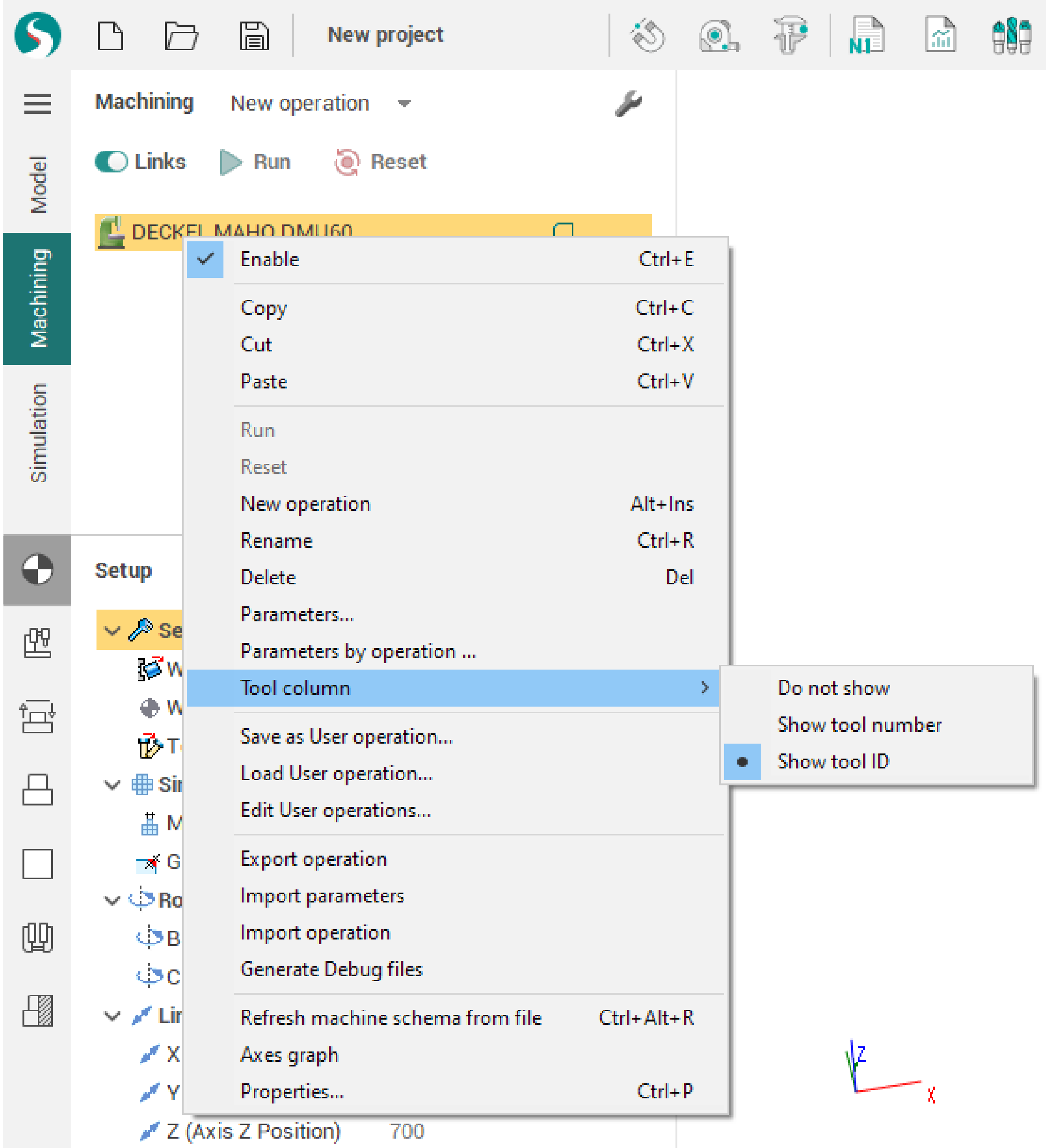
Load existing tool list... - loads a tool list from a file. A standard file selection dialog opens. The name of the loaded * .tom file is stored in the system settings. It also allows you to load tools from the tool library file (*.csv or *.db + *.properties formats) and add them to the current list. 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.

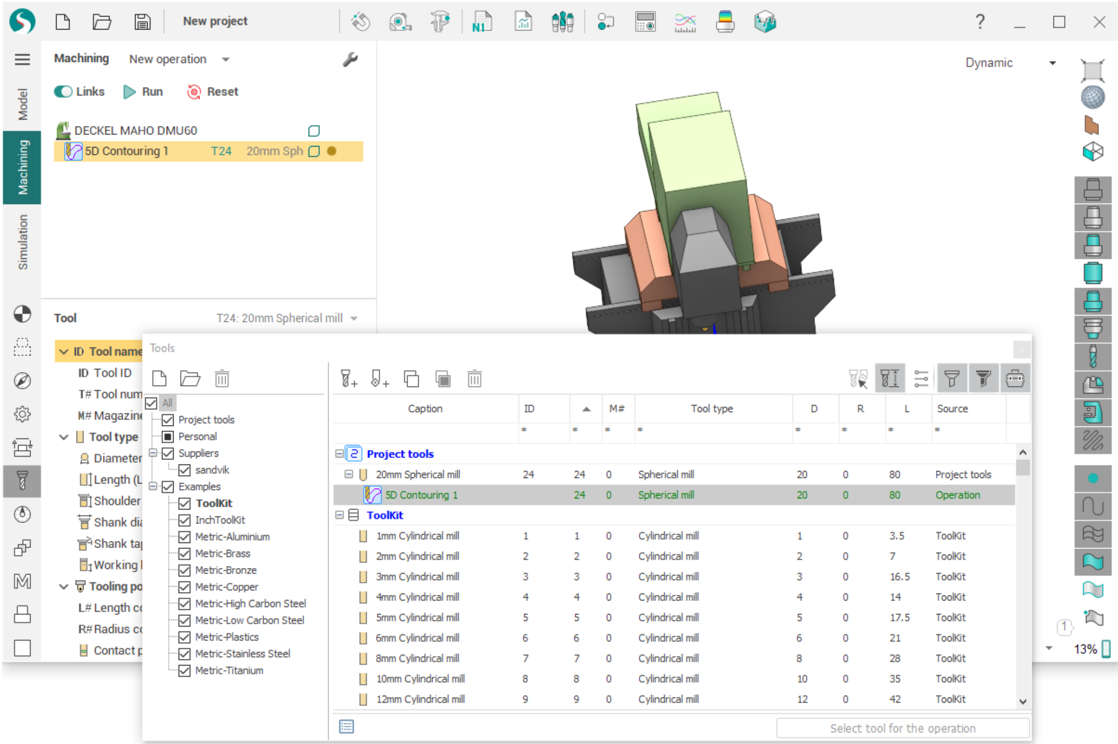
Delete all unused tools from the project - clears the tool list, deletes all unused tools and breaks the link to the file if the tools were previously saved to a file. The memorized file name for the current machine is deleted from the system settings. The new empty list of tools is automatically filled 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.


New project
Machining
New operation
Links
Run
Reset
Model
Machining
Simulation
DECKEL MAHO DMU60
Setup
Enable Ctrl+E
Copy Ctrl+C
Cut Ctrl+X
Paste Ctrl+V
Run
Reset
New operation Alt+Ins
Rename Ctrl+R
Delete Del
Parameters...
Parameters by operation ...
Tool column
Save as User operation...
Load User operation...
Edit User operations...
Export operation
Import parameters
Import operation
Generate Debug files
Refresh machine schema from file Ctrl+Alt+R
Axes graph
Properties... Ctrl+P
Do not show
Show tool number
Show tool ID
Z (Axis Z Position) 700



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**:
   a. **gray** - the operation tool does not differ from the tool in the list;
   b. **black** - the operation tool is different from the tool in the list;
2. **Font thickness** is used to indicate the difference between the tool and same tools inside another **operations** of the current project:
   a. **normal font** - the tool of this operation does not differ from the tools of other operations;
   b. **bold** - the tool of this operation is different from the tools of other operations.
3. The **tool identifier color** in the operations tree column is used to display the **correctness of tool numbers** in the project:
   a. **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;
   b. **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.
4. The **dashed underscore identifier** of the tool in the column is used to indicate **differences in machining conditions** (feeds and cutting speeds):
   a. **without underscore** - the machining conditions in this operation are **no different** from the conditions specified in the tool **in the global list of tools**;
   b. **with underscore** - the machining conditions in this operation **differ** from the conditions specified in the same tool **in the global list**.

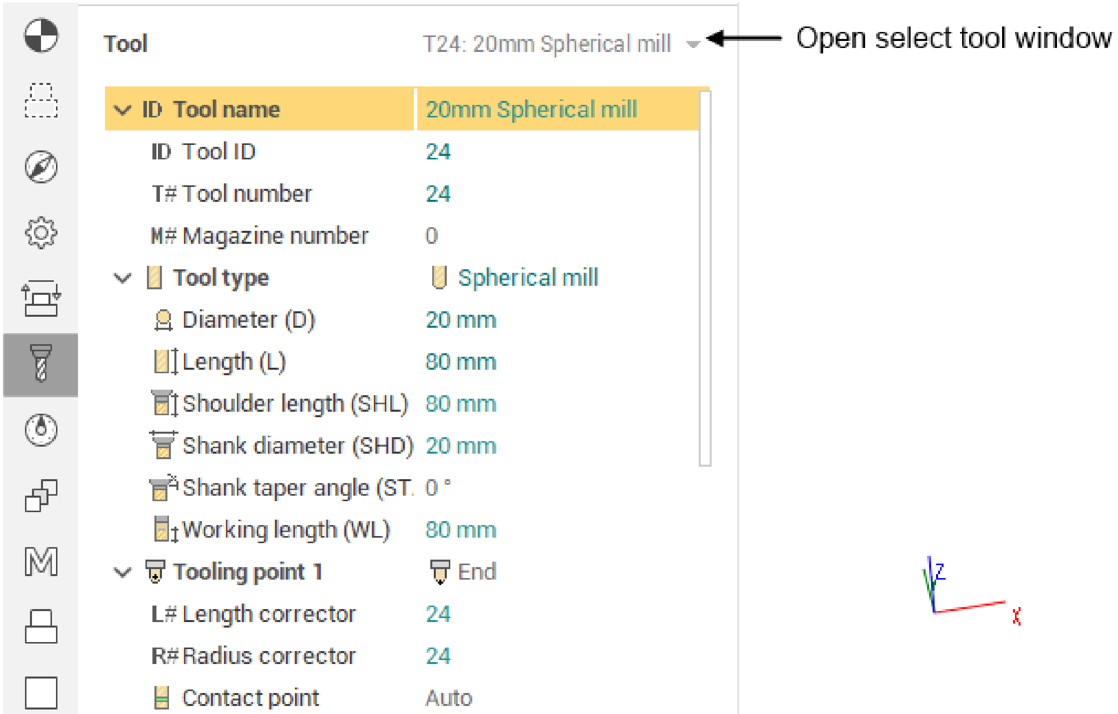
Thus, a gray tool with a normal font is completely “correct” (not different from any copy), and the most black and bold tool is most likely the most “wrong” (different from all other copies).

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.

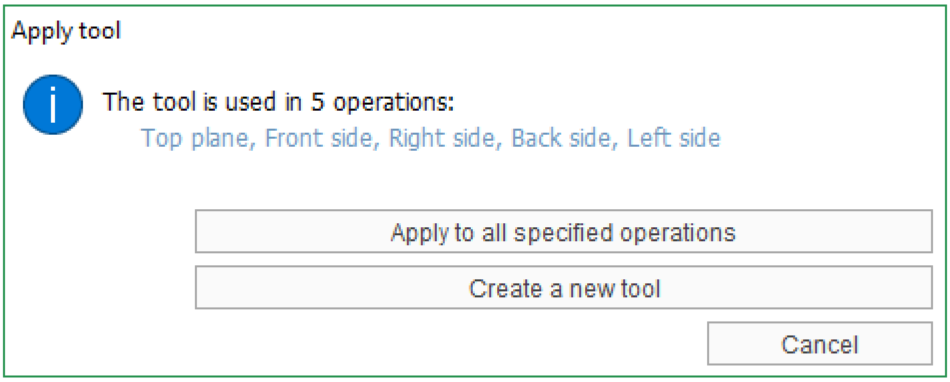
- **Front side**, **Right side**, **Back side** and **Left side** operations tool - **T#2: 80mm Cylindrical mill**. Drawn in gray, because it does not differ from the tool in the list, but is drawn in bold because it differs from the T#2 tool in the first "Top plane" operation.
- **Front holes** operation 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.

7.3 Creating shaped tools

Error: null

7.4 Holders (*.osd) window

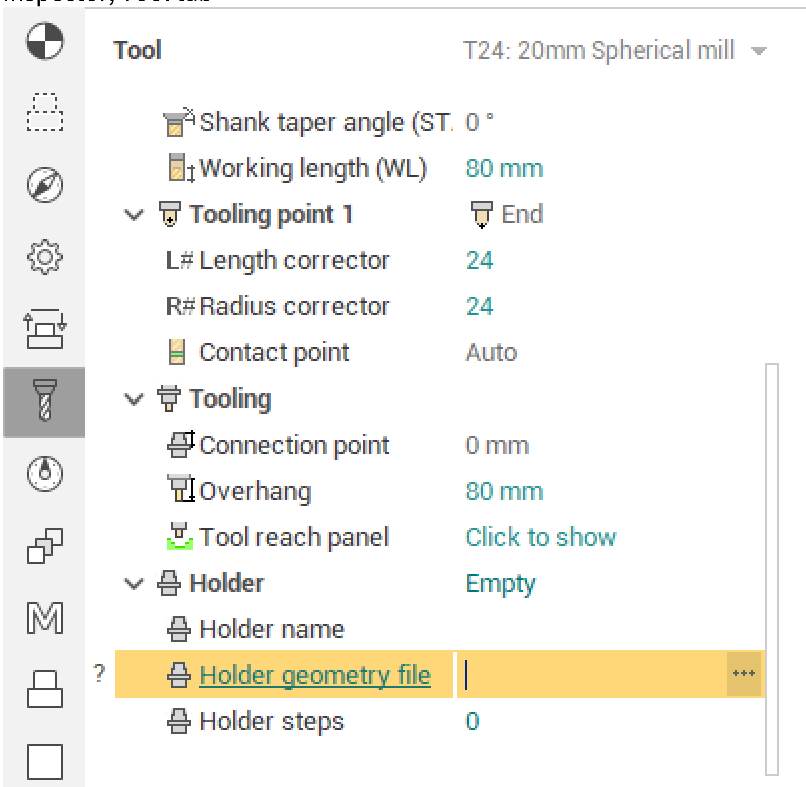
The window for opening and adding 3d models (*.osd) of holders is implemented.

It is possible to add previously prepared holder models from the computer to this window.

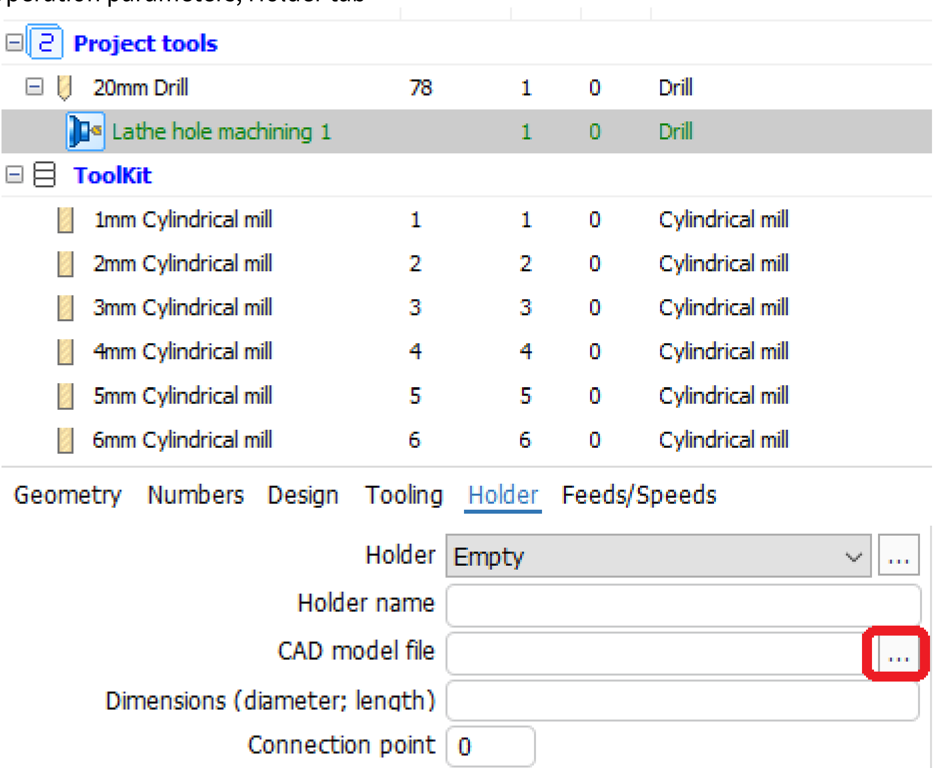
Previously added holders in this window are saved to the configuration file and will be loaded automatically.

This window can be caused from the following locations:

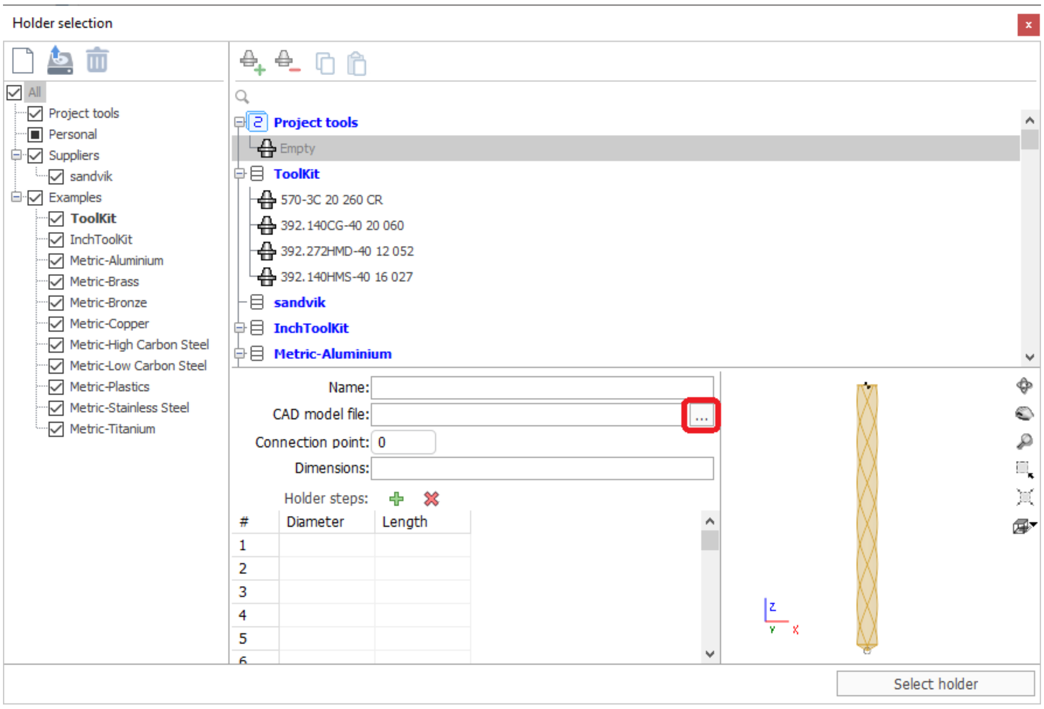
1. Inspector, Tool tab



2. Operation parameters, Holder tab



3. Holder selection window



7.5 Machining tools import API

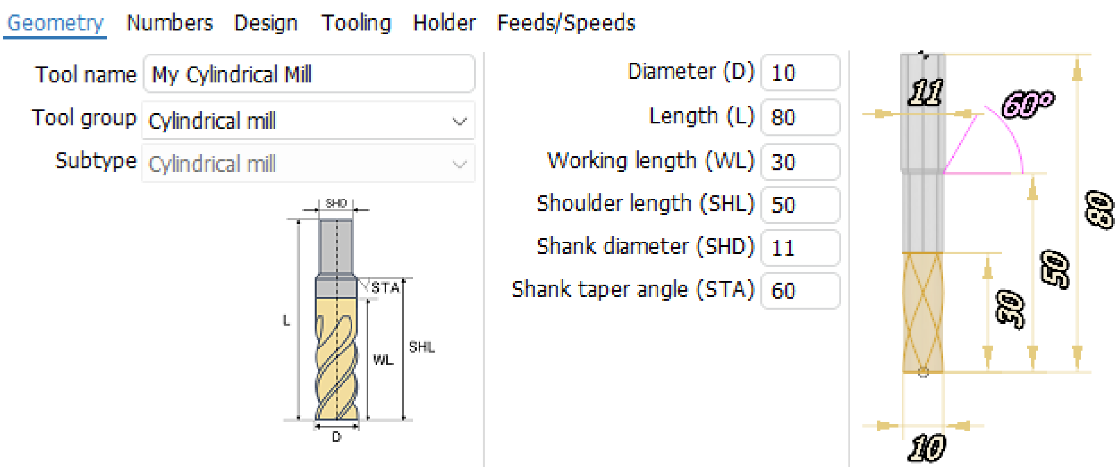
There is an Application programming interface (API) using which you can write your own program to import machining tools into the CAM system from any external system you want (like TDM/PDM/PLM systems). It allows you to create simple program (like shown on the picture below) inside which you can for 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.

```
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();
```



8 Scripts in SprutCAM X

SprutCAM Xcan be extended and customized by scripting.

SprutCAM Xuse Sprut4 script engine.

There are free ways to use scripts in SprutCAM X.

- 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 SprutCAM X.

See also:

Application Programming Interface

Brief Sprut4 description

Scripts IDE

Scripted operation

Operation with scripted toolpath commands

Scripted SprutCAM X launch

8.1 Brief Sprut4 description

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 IDE documentation

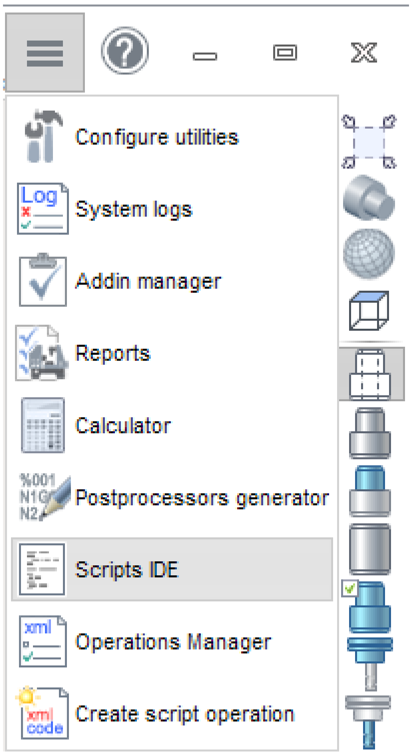
See also:

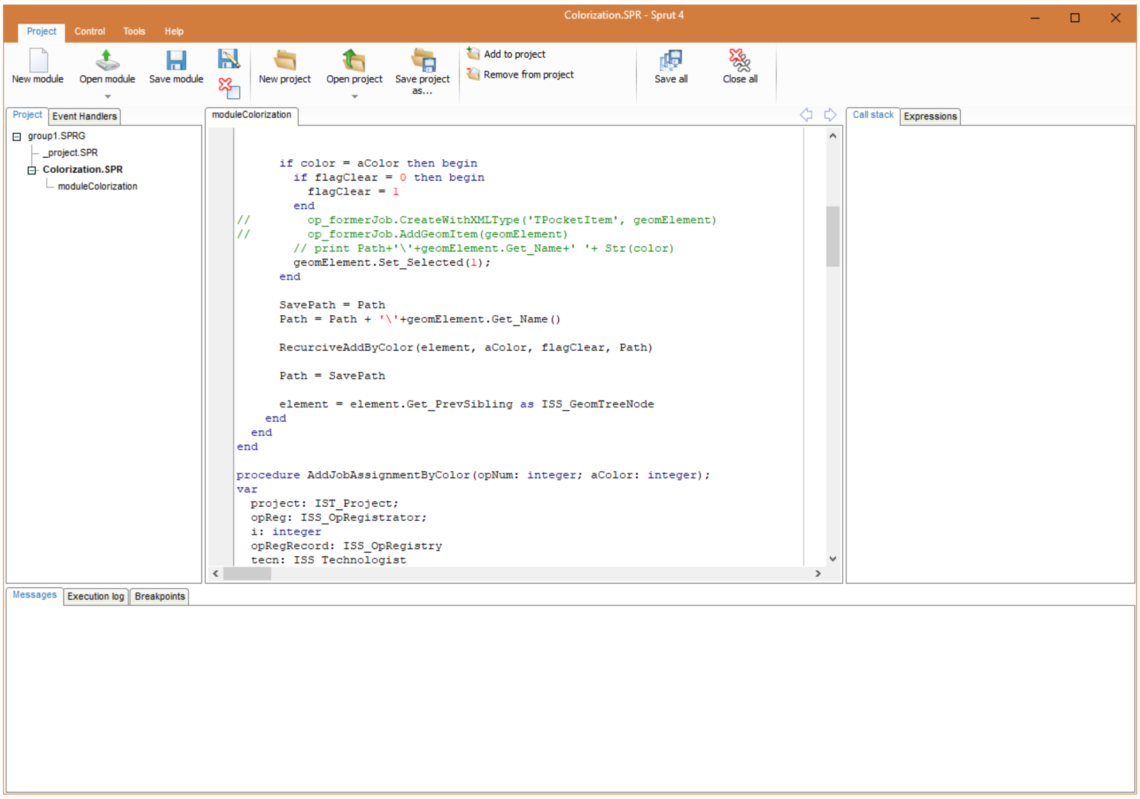
Scripts IDE

Application Programming Interface

8.2 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 Sprut4 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.

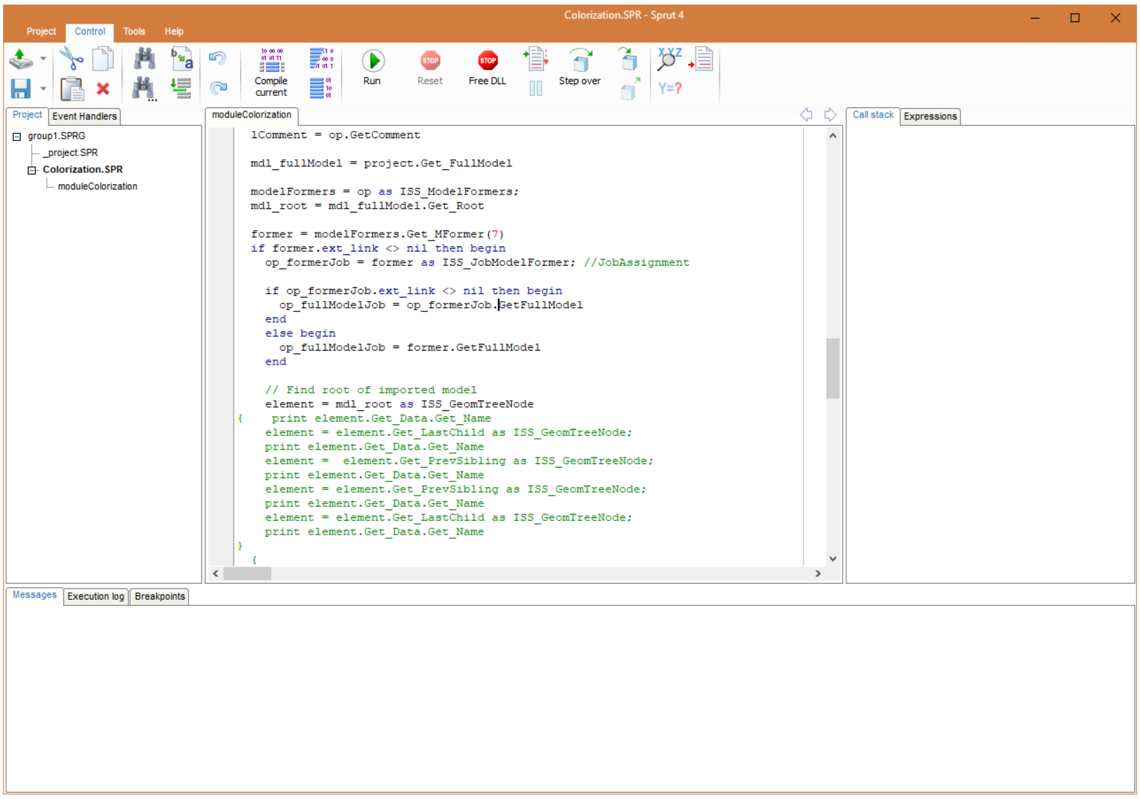
"Objects" panel holds objects that may have event handlers. List of available events for the selected object are shown in the "Events" panel below.

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 SprutCAM X 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 Sprut4 routine (function or procedure), in which the user can describe on the programming language execution of some actions when the event occurs. In SprutCAM X 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.

See also:

Application Programming Interface

Brief Sprut4 description

8.3 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 Sprut4 language can be found in SprutCAM X API documentation

See also:

Scripts IDE

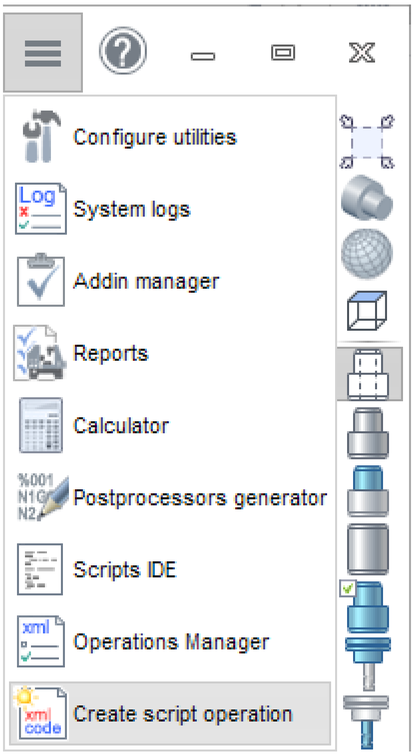
Brief Sprut4 description

8.4 Scripted operation

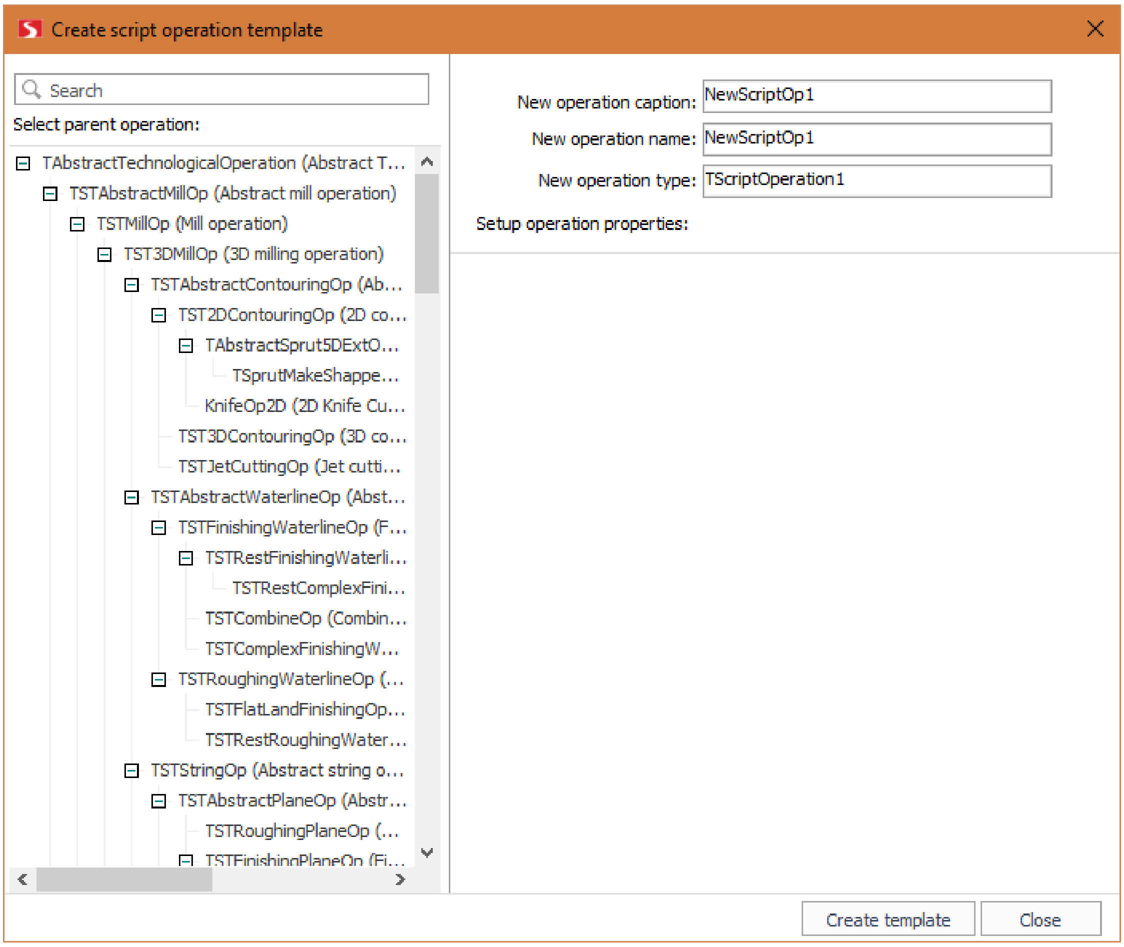
Script operation can have OnCreateMethod, ChangePropertyMethod and MakeWorkPathMethod methods defined.

- OnCreateMethod is a sprut4 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 Sprut4 project.

Also you can manually create operation descriptor xml-file and include it into SprutCAM X xml-configuration.

See also:

Application Programming Interface

Brief Sprut4 description

Scripts IDE

8.5 Operation with scripted events

Operation with scripted toolpath commands is created from a regular SprutCAM X 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 SprutCAM X processes that object in toolpath. See Scripts IDE for more information.

See also:

Application Programming Interface

Brief Sprut4 description

Scripts IDE

8.6 Scripted SprutCAM X launch

Supply Sprut4-script as command line parameter for SprutCAM X launch. SprutCAM X will execute that script right after system initialization is complete. Script can import geometry and create operations.

See also:

Application Programming Interface

[Brief Sprut4 description](#)

[Scripts IDE](#)

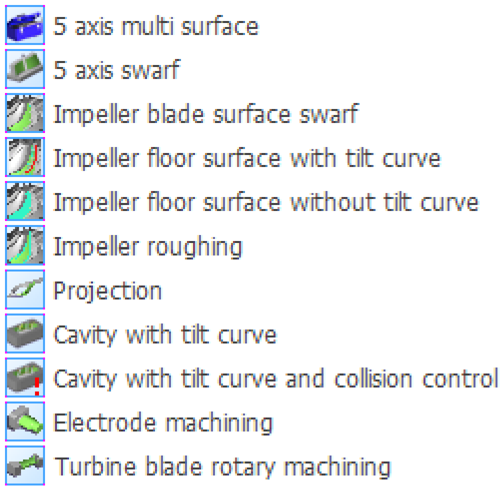
9 SprutCAM X's licensed modules

SprutCAM X's licensed modules — it is features, which can be turned on in one of the existing configurations when additional license is present.

9.1 5D MW - advanced 5 axis milling module

This module is useful for creating tool paths for 3, 4, and 5 axis machines, and industrial robots (the last only for configuration 'Robots + 5D MW').

There are operations:



This module is available for the next configuration .

See also:

5 axis multi surface

5 axis swarf

Impeller blade surface swarf finishing

Impeller floor surface with tilt curve

Impeller floor surface without tilt curve

Impeller roughing

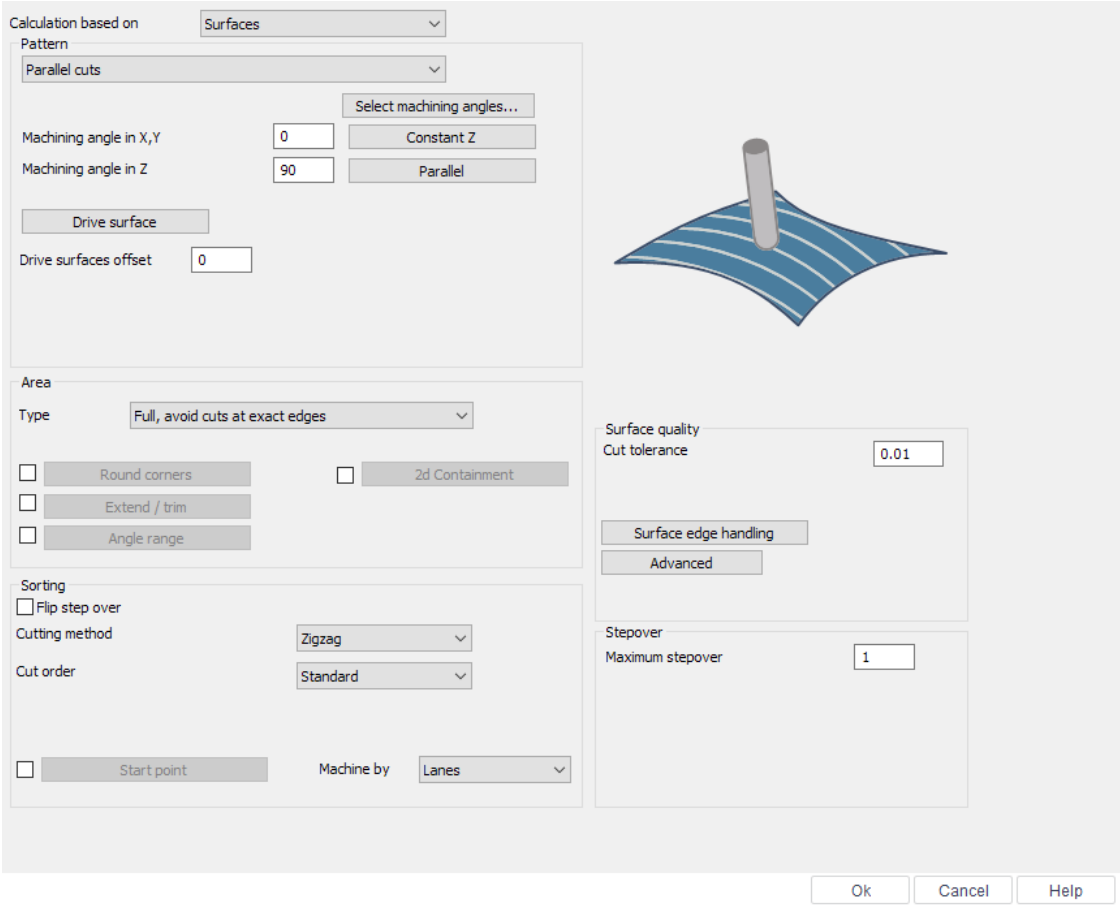
Cavity with tilt curve

Turbine blade rotary machining

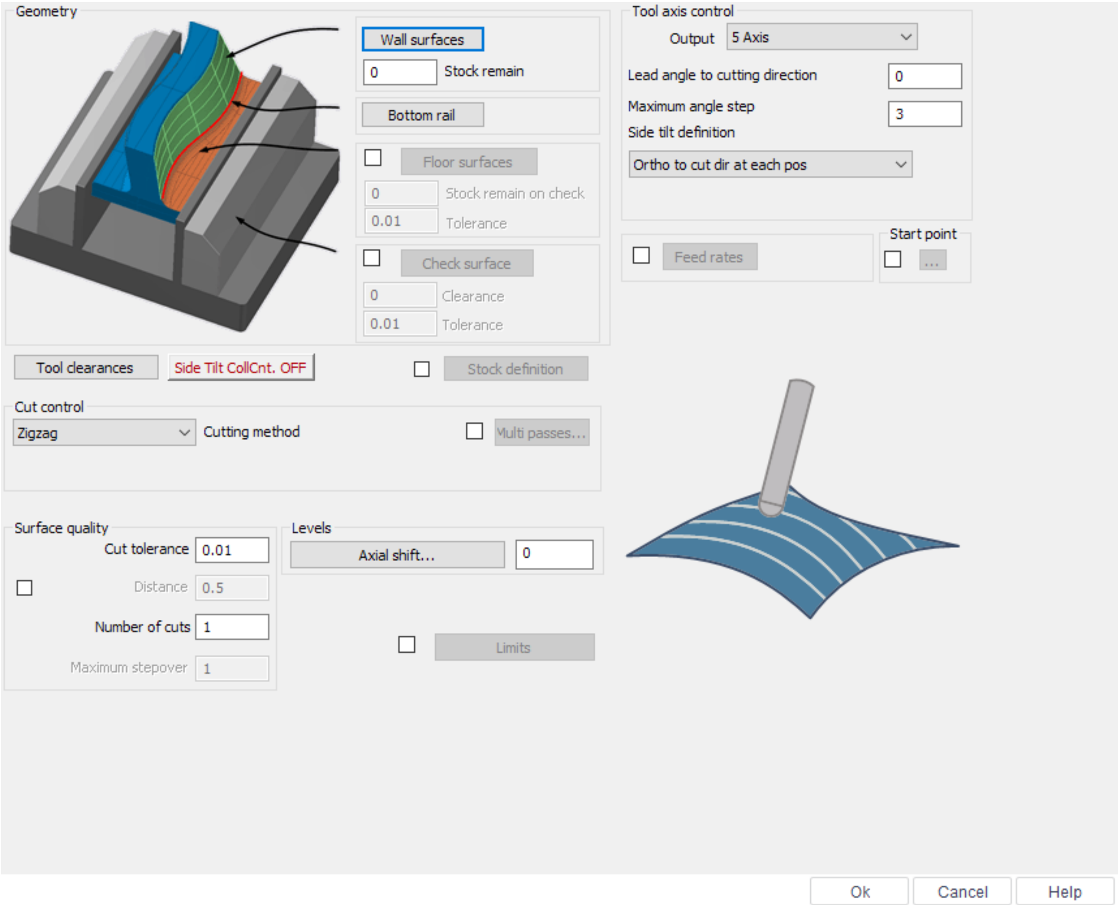
Electrode machining

Projection

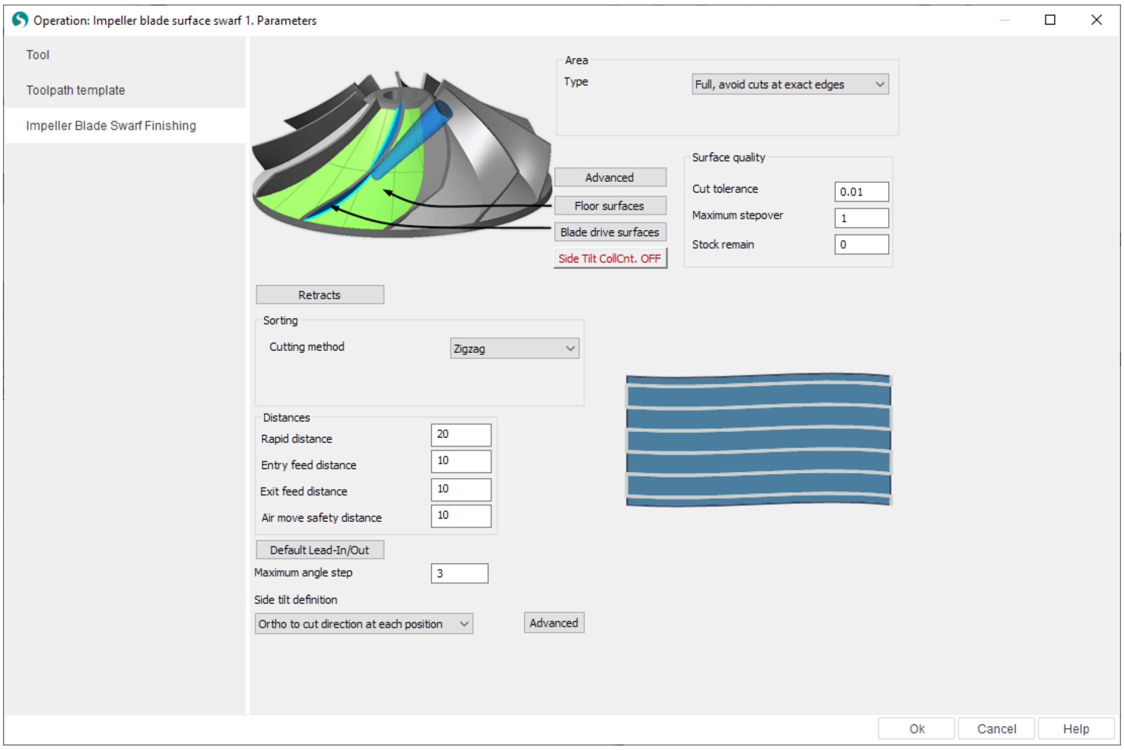
9.1.1 5 axis multi surface



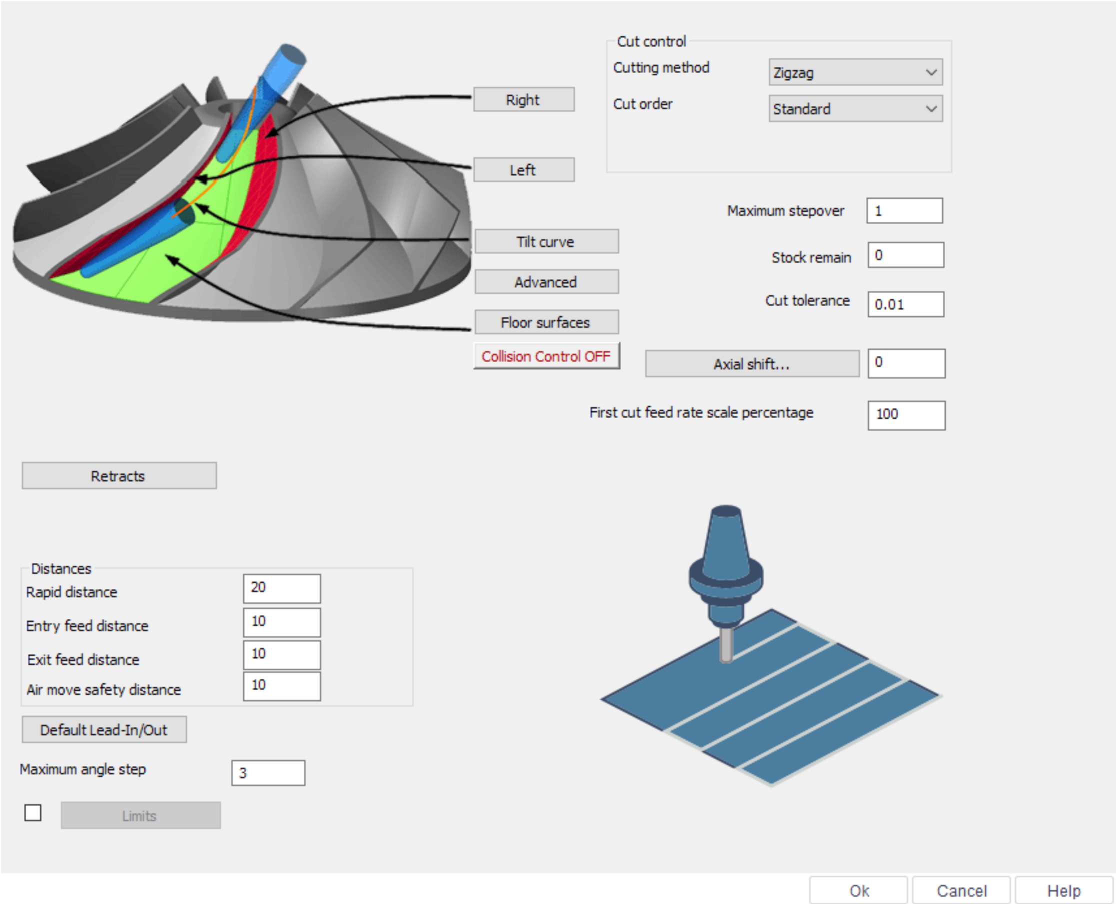
9.1.2 5 axis swarf



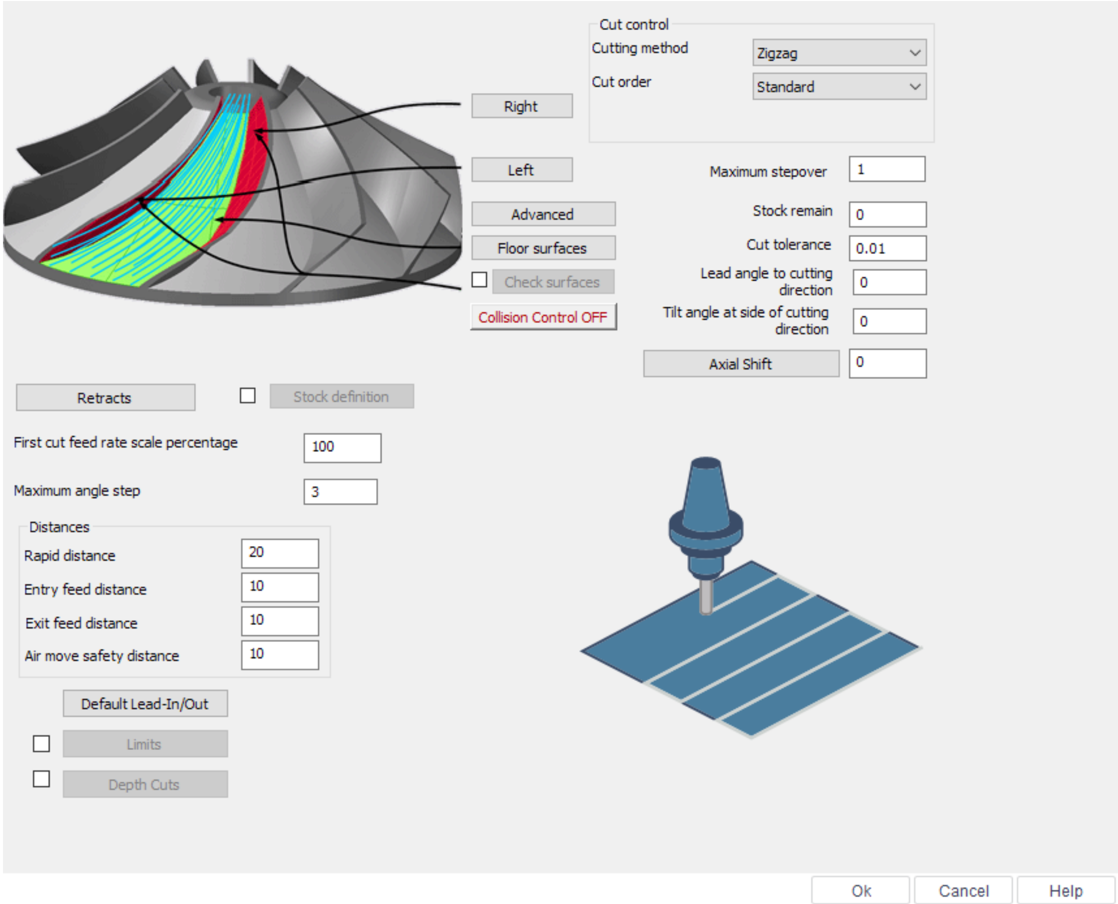
9.1.3 Impeller blade surface swarf finishing



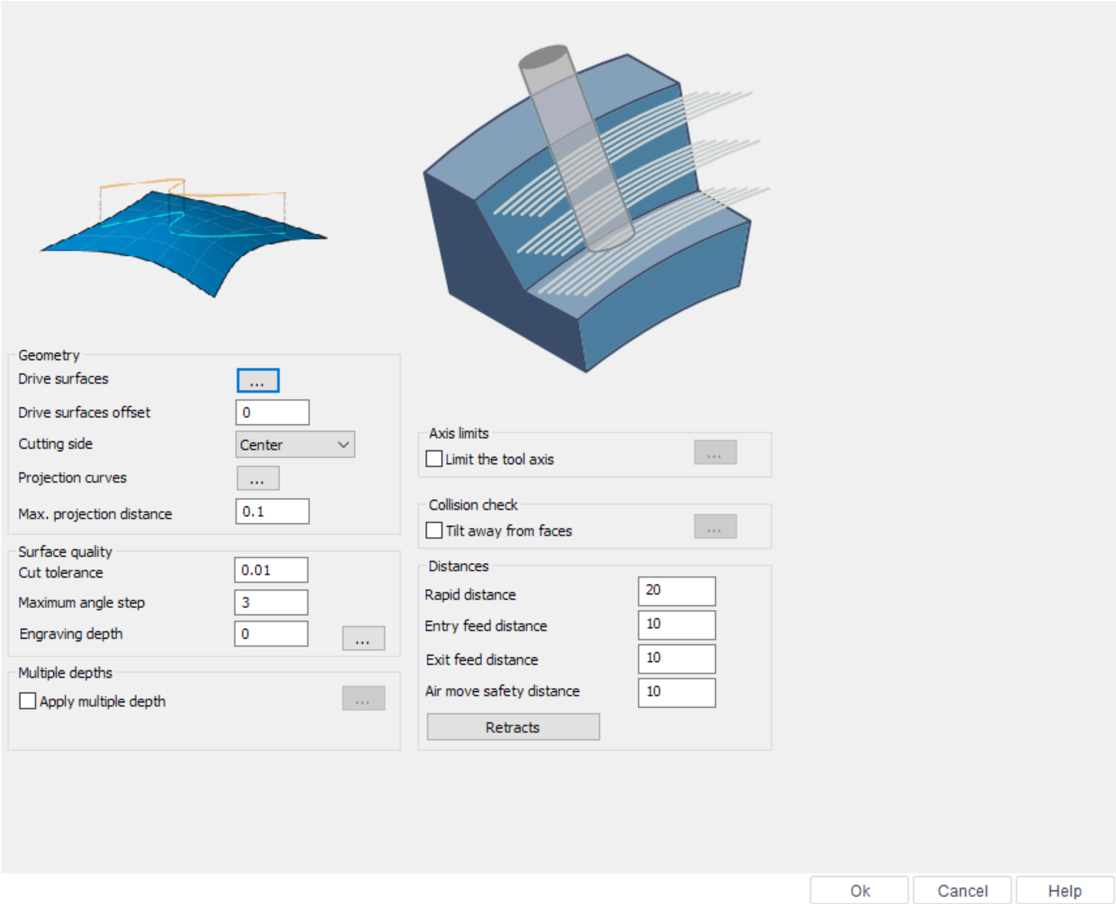
9.1.4 Impeller floor surface with tilt curve



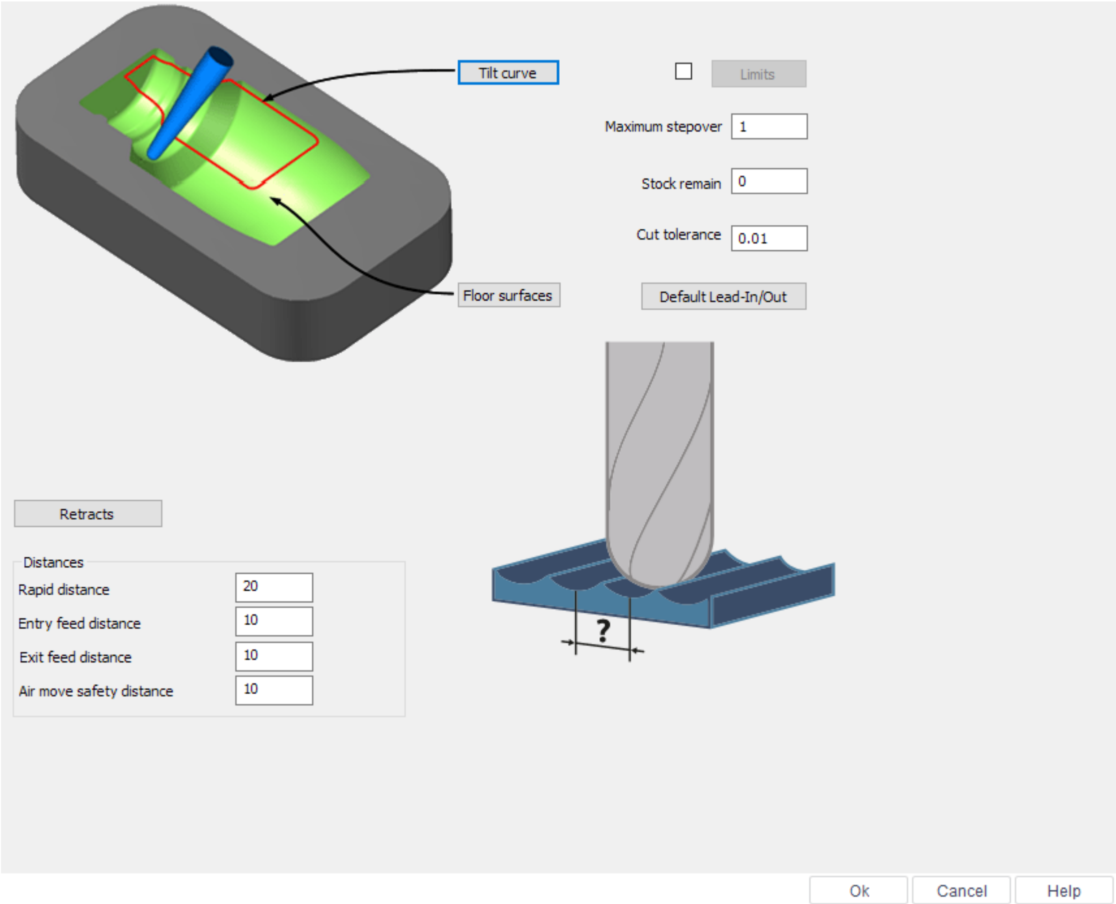
9.1.5 Impeller roughing



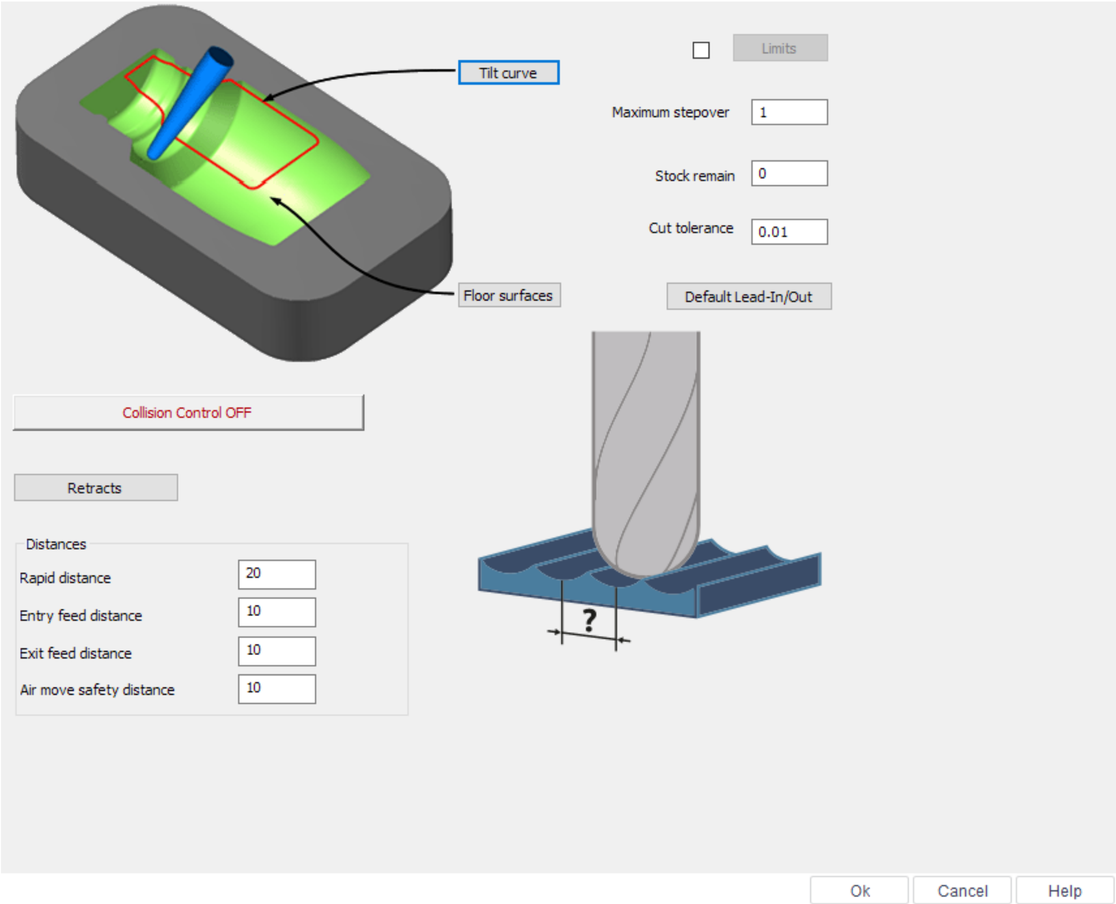
9.1.6 Projection



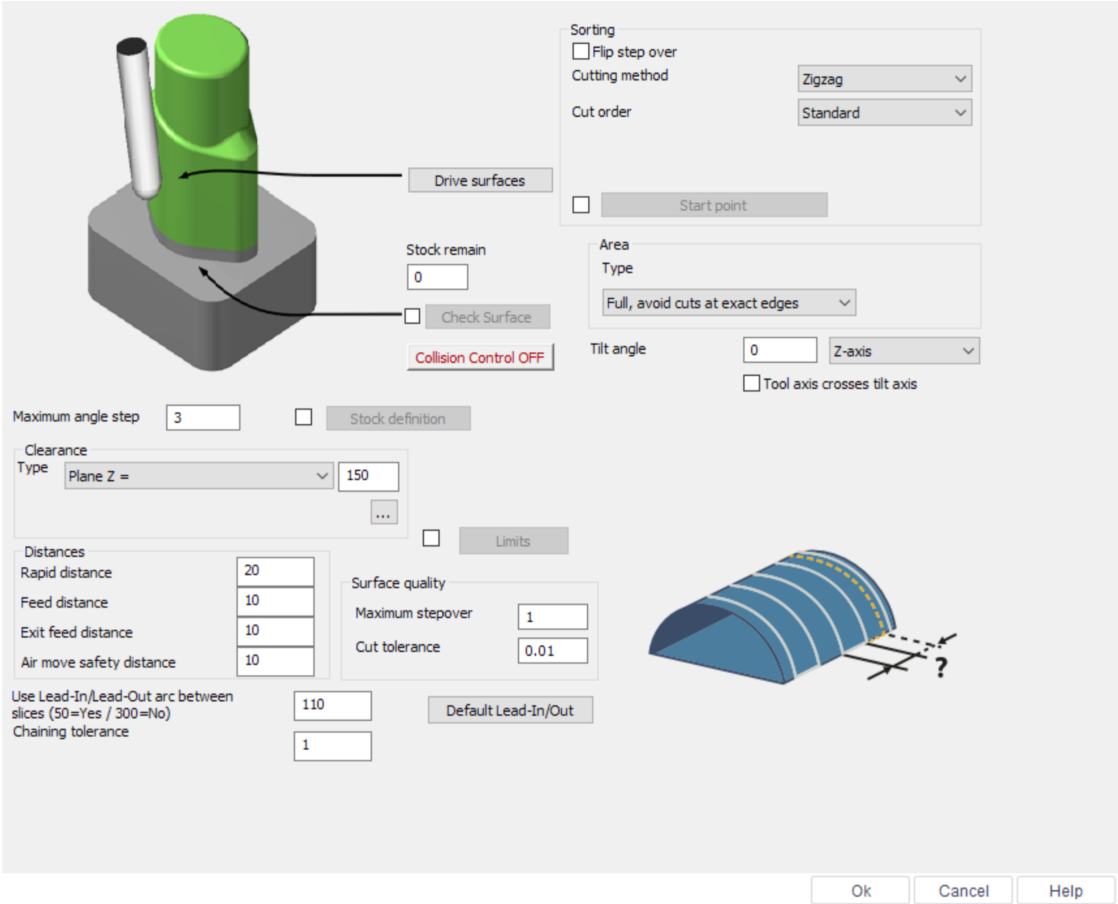
9.1.7 Cavity with tilt curve



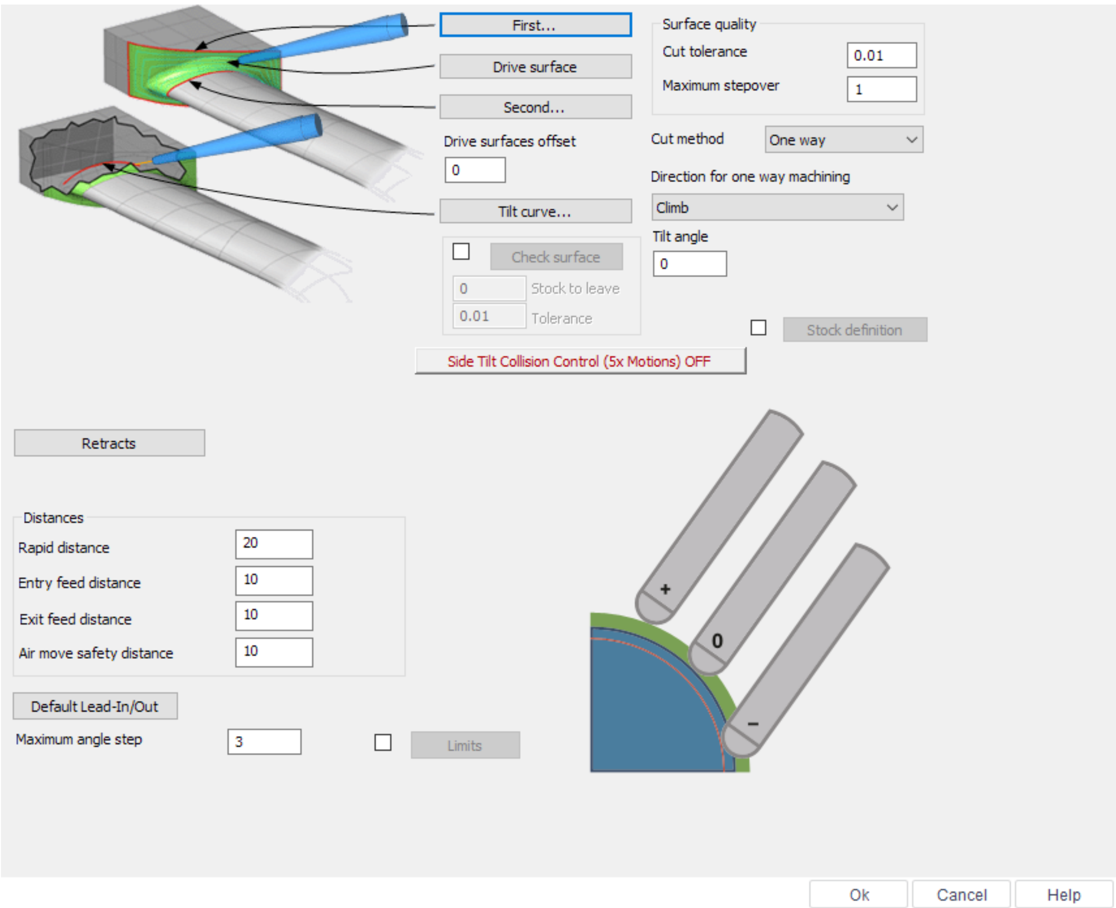
9.1.8 Cavity with tilt curve and collision control



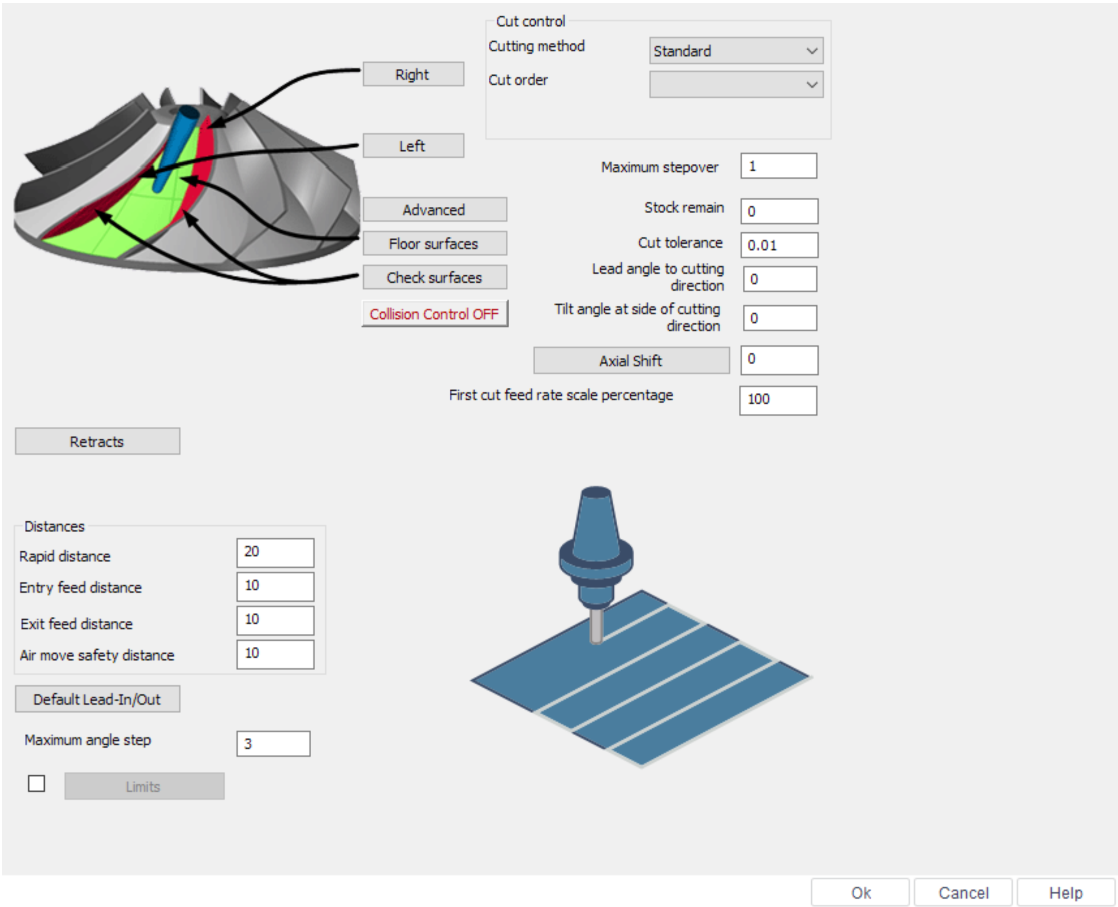
9.1.9 Electrode machining



9.1.10 Turbine blade rotary machining



9.1.11 Impeller floor surface without tilt curve



9.2 Adaptive SC

Module for high-speed milling.

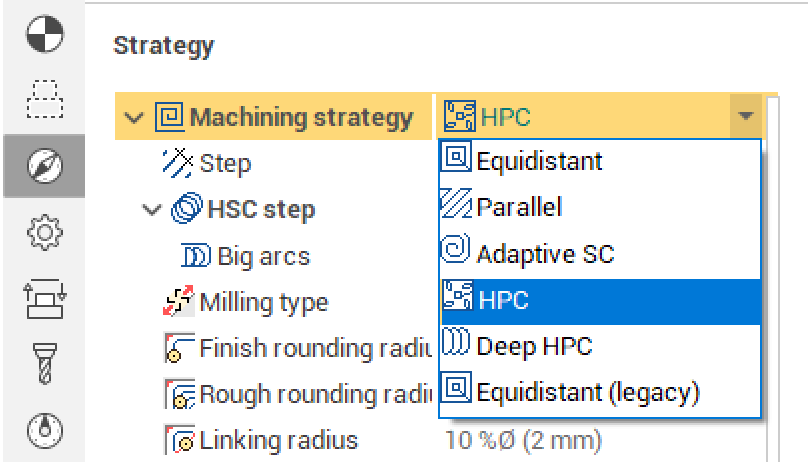
> **Note:**
> Need an additional license.

The pocketing strategies are designed for the removing of material in the open and closed pockets.

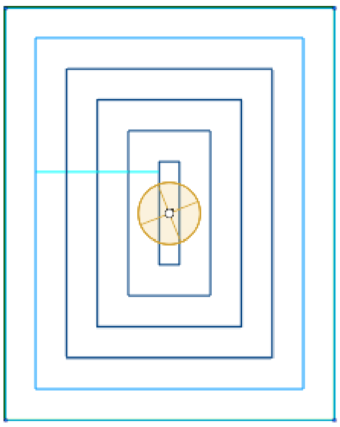
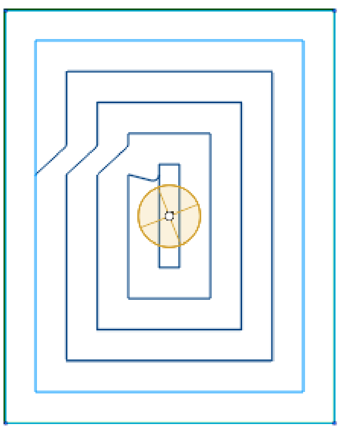
These strategies are available in the following operations:

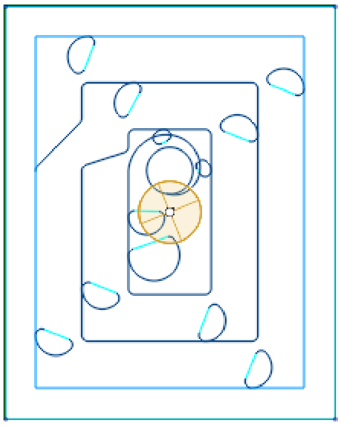
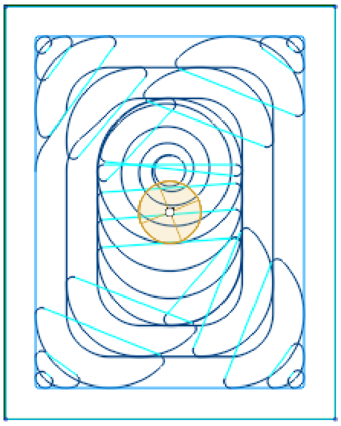
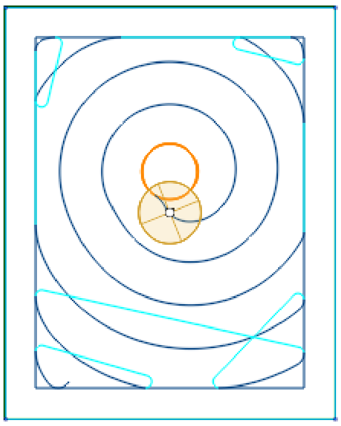
- Rough waterline
- Pocketing
- Pocketing 2.5D
- Flat land finishing

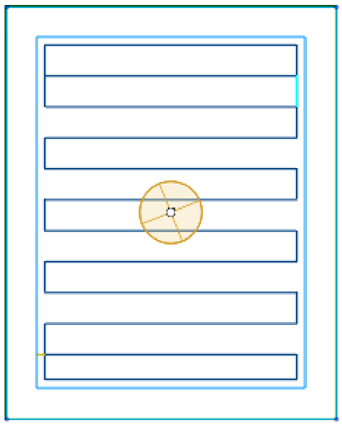
Seven strategies are available to select in the Machining strategy drop-down:



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.
Adaptive SC 	**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

9.2.1 Features of Adaptive SC 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.

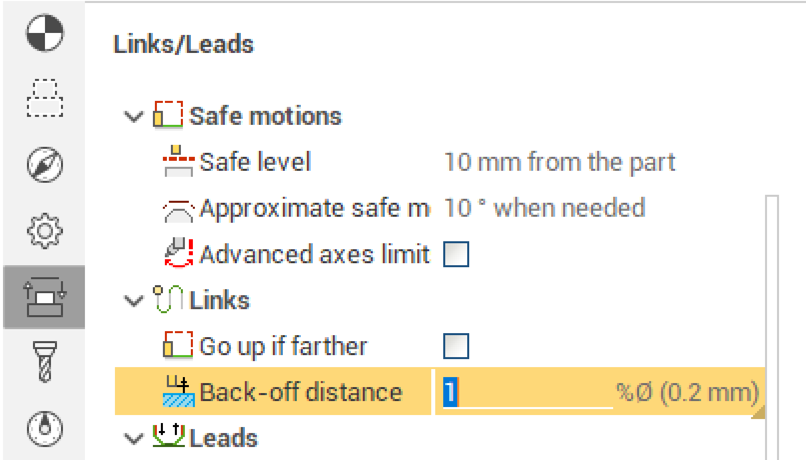
The material is removed in spiral-like fashion. There are no sharp corners in the toolpath. Smoothness of the toolpath is precisely controlled by the dedicated parameters for the roughing rounding radius, the finishing radius and the linking radius. Linking is done preferably in the working plane with 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.

9.2.2 How to choose the pocketing strategy

1. The choice number one is **Adaptive SC**. 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.

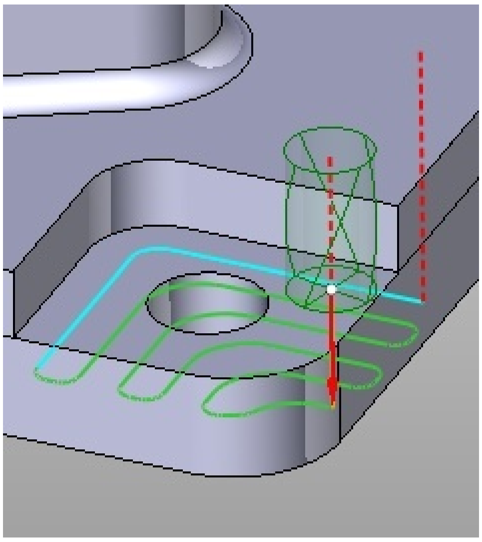
9.2.3 Tool path parameters

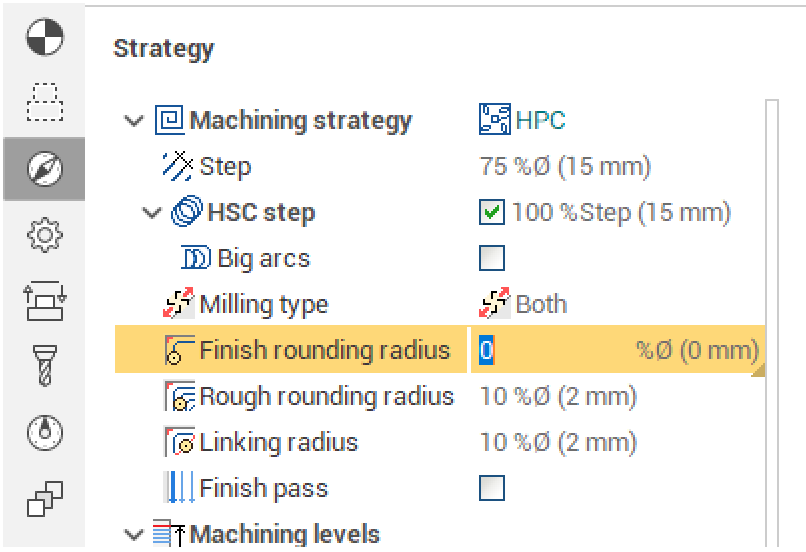
- ### 9.2.3.1 Back-off distance parameter



The tool can be lifted above the already machined surface when it moves to the next trochoidal arc start position.

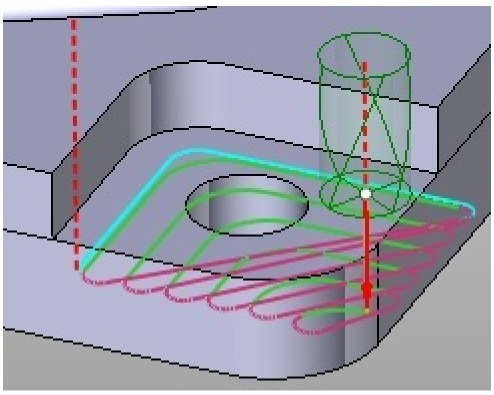
- 9.2.3.2 Rounded links in zigzag mode





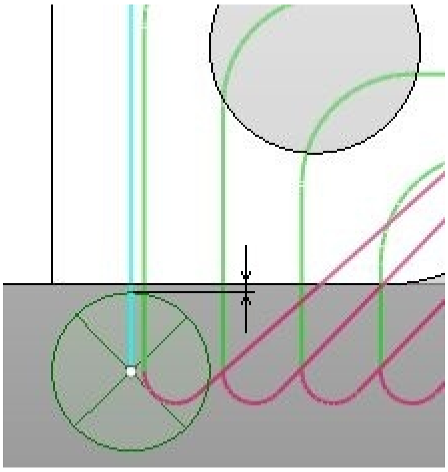
The ‘Finish rounding radius’, ‘Rough rounding radius’ and ‘Linking radius’ value is used for rounding of the links.

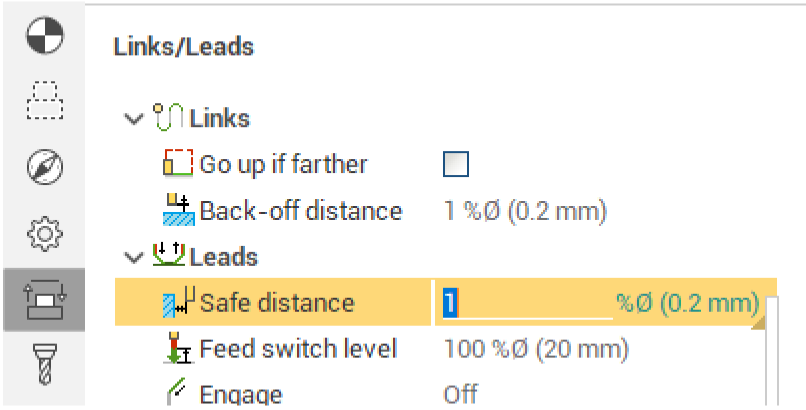
- 9.2.3.3 Links on the same Z-level



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.

- ### 9.2.3.4 Safe distance



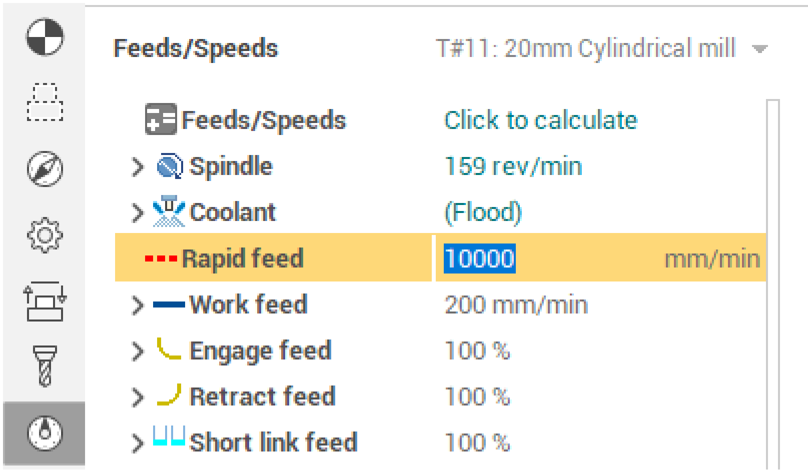


Safe distance is used to move the tool down/up from/to the safe surface.

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.

- 9.2.3.5 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.

See also:

Pocketing strategies

SprutCAM X features matrix

9.3 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.

See also:

Bypass inaccessible positions and singularities

SprutCAM X features matrix

9.4 Robot Mill

Including the ability to use milling operations in the 'Robot' configuration.

This module is available for configuration.

> **Note:**
> Need an additional license.

See also:

SprutCAM X features matrix

9.5 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 SprutCAM's >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.

9.6 Operations

Some of SprutCAM X operations may be flexible added into various software configurations by getting a special license. They are grouped into modules listed later in this section.

9.6.1 Sawing

Sawing operation allows calculating a toolpath for cutting with a circular saw. Available as an option.

> **Note:**
> Operation must be licensed separately.

See also:

Sawing operation

Configurations of SprutCAM

9.6.2 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.

See also:

5D Contouring operation

Configurations of SprutCAM

9.6.3 Welding

This operation is available as option of SprutCAM X.

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.

See also:

Welding_43062

Welding 5D and 6D operations

Configurations of SprutCAM

9.6.4 Cladding

The optional module of SprutCAM X 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.

See also:

Additive manufacturing

Area cladding operation

Configurations of SprutCAM X

9.6.5 Knife

Two separate operations:

- knife 2D
- knife 6D

These operations licensed separately.

9.6.6 Multiblade Basic

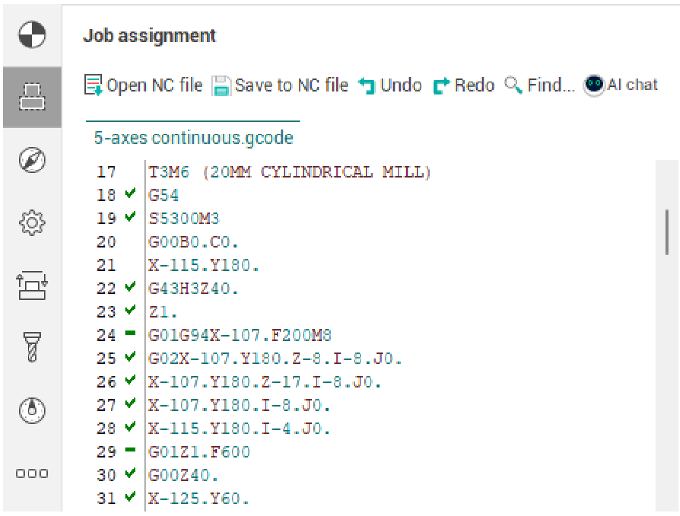
It includes 5 axis operation powered by Module Works for turbine impeller's milling.

See also:

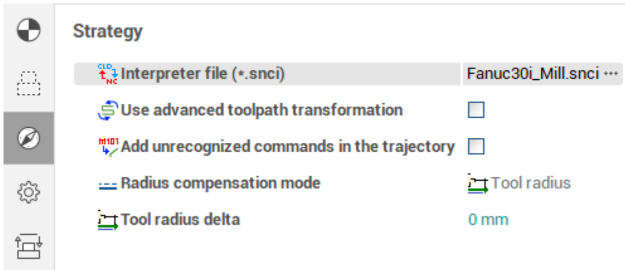
SprutCAM X features matrix

9.6.7 G-code based operation, G-code based lathe operation

Set NC program	Select the interpreter



...



< **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.

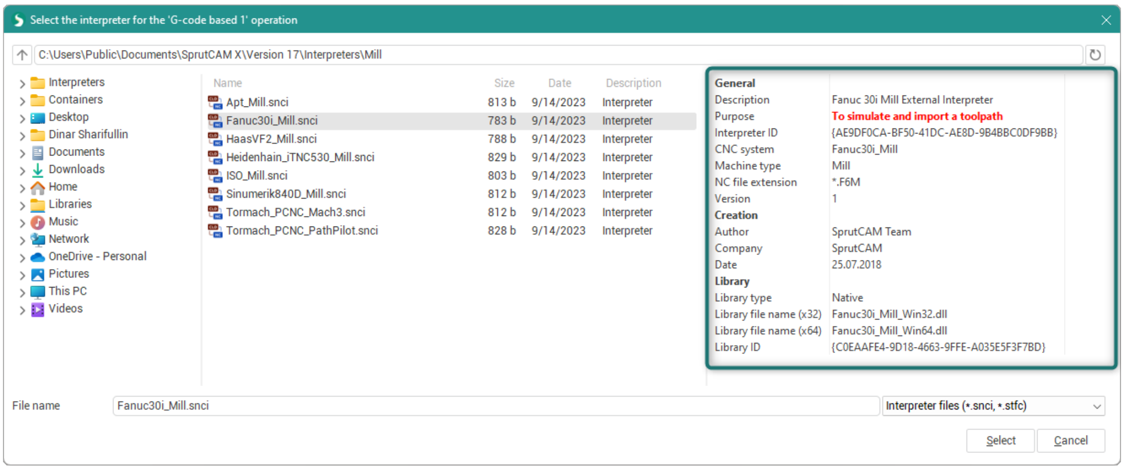
The toolpath is formed on the basis of the following operation parameters:

- **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.

Currently, interpreters of the following CNC systems are available for use:

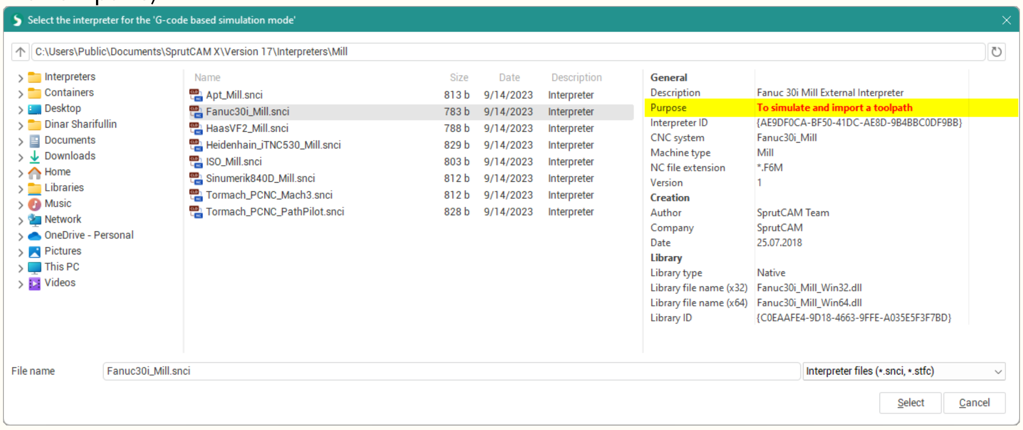
Machine group	CNC system	Comment	Note
Milling	APT	Import toolpath only	
	Apt_Simplify_3D	Import toolpath only	
	ISO	Import toolpath only	
	Global control	Import toolpath only	An additional license is required
	Fanuc 30i	To simulate and import a toolpath	
	Haas VF-2	To simulate and import a toolpath	
	Heidenhain iTNC 530	To simulate and import a toolpath	
	Mazatrol SmoothG	To simulate and import a toolpath	An additional license is required
	NC210	To simulate and import a toolpath	An additional license is required

Machine group	CNC system	Comment	Note
	Sinumerik 840D	To simulate and import a toolpath	
	Tormach PCNC Mach3	To simulate and import a toolpath	
	Tormach PCNC PathPilot	To simulate and import a toolpath	
Turning	Mazatrol SmoothC	To simulate and import a toolpath	An additional license is required
Turn-milling	Fanuc 21i	To simulate and import a toolpath	An additional license is required
	NC220	To simulate and import a toolpath	An additional license is required
	Sinumerik 840D	To simulate and import a toolpath	
	Okuma OSP-P300	To simulate and import a toolpath	An additional license is required
Robot	Fanuc robot (R-30iB controller)	To simulate and import a toolpath	
	Kuka robot	To simulate and import a toolpath	
	Motoman robot	To simulate and import a toolpath	
	ABB robot	To simulate and import a toolpath	
	Nachi robot (AW Format)	To simulate and import a toolpath	An additional license is required

Note: *All interpreters support command list generated by postprocessors in SprutCAM X distribution kit only.*

"Import toolpath only" interpreters are not supported matching line NC code - trajectory of tool movement.

When you select an interpreter, pay attention to its purpose (the **Purpose** field in the Preview pane)



The selected interpreter should be intended for simulation. Otherwise, the trajectory of the tool may be incorrect (shifted relative to the coordinate system of the workpiece, duplicated approaches/retracts, incorrect starting position, etc.).

< **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.

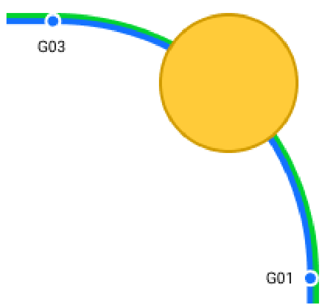
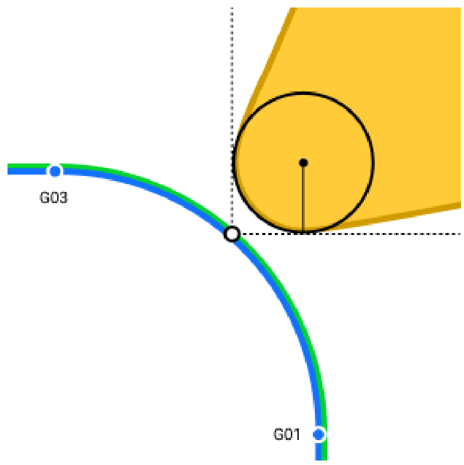
< **Step of physic movements dividing** > - available only when using **Use advanced toolpath transformation** (see above). In this mode, at the first stage, the tool path is converted into a geometric curve. At the next stage, machine movements are rasterized with the step specified in the current parameter 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.

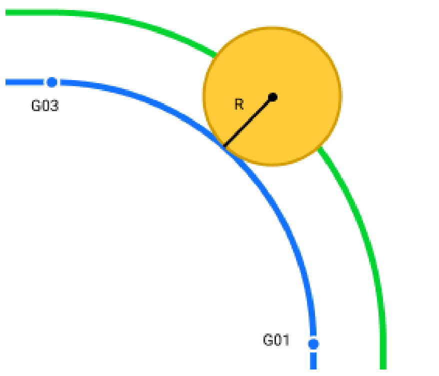
< **Add unrecognized commands in the trajectory** > - this parameter adds all commands of the control program not recognized by the interpreter to the toolpath. Unrecognized commands are added to the tool path description as a technology command parameter < **INSERT** > before the recognized commands.

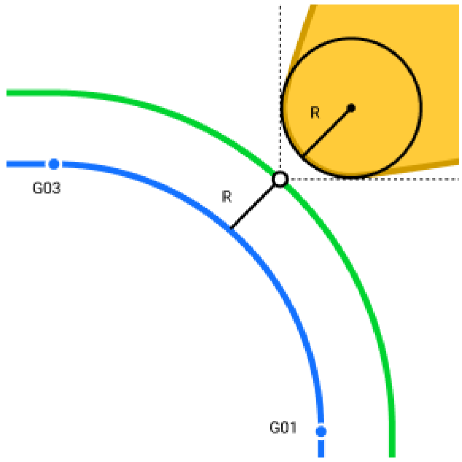
Example of how the parameter "Add unrecognized commands in the trajectory" works

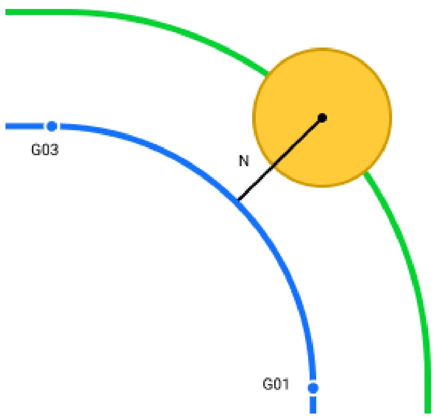
NC program	This parameter is disabled	This parameter is enabled
M107 G0 F3600 X13.885 Y55.321 Z0.15 G1 F3000 X14.618 Y55.174 E0.09324 G1 X15.622 Y55.102 E0.21879 G1 X17.918 Y55.102 E0.50516 G1 X18.665 Y55.142 E0.59846 G1 X19.403 Y55.261 E0.6917 G1 X19.926 Y55.396 E0.75907 G1 X20.67 Y55.619 E0.85594 Commands **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

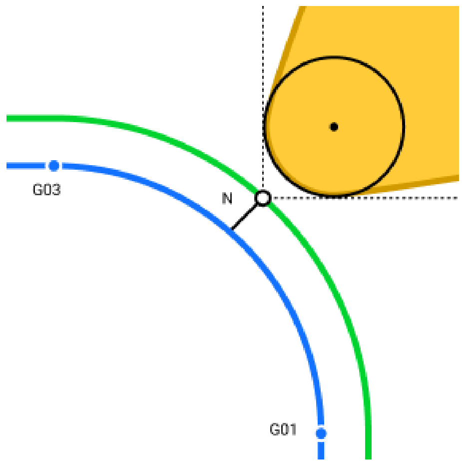
< **Radius compensation mode** > - Ability to disable or set an arbitrary compensation value on the tool radius.

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**

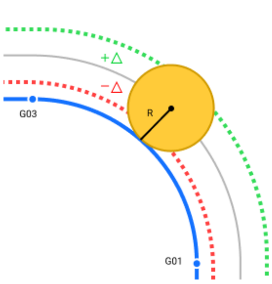
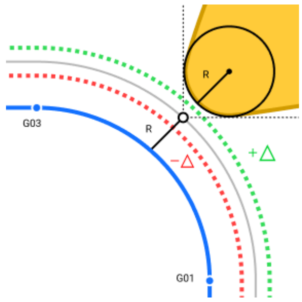
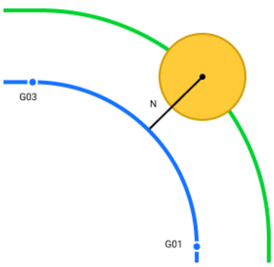
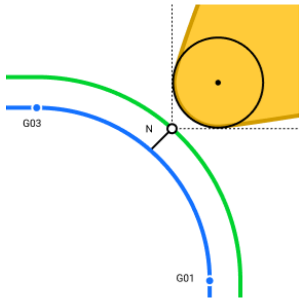
<table>
<tr><th>Parameter value</th><th>Milling</th><th>Turning</th></tr>
<tr><td>Tool radius</td><td>

Radius compensation is equal to the tool radius</td><td>

Cutter Radius compensation is equal to the tool nose radius</td></tr>
<tr><td>Custom value</td><td>

Radius compensation is determined by the specified arbitrary value .</td><td>

Cutter radius compensation is determined by the specified arbitrary value .</td></tr>
</table>

Parameter	Milling	Turning	Note
< **Tool radius delta** >	 *Additional tool offset.* **The total offset** is calculated as the sum of the *tool radius* and the value of the *additional offset.*	 *Additional tool offset.* **The total offset** is calculated as the sum of the *tool nose radius* and the value of the *additional offset.*	The setting is only available when using the <Tool radius> value set as the **radius compensation mode** (see above)
< **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

Sorry, the widget is not supported in this export.
But you can reach it using the following URL:
https://www.youtube.com/watch?v=XpM6z6IXpKk

See also:

9.6.7.1 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:

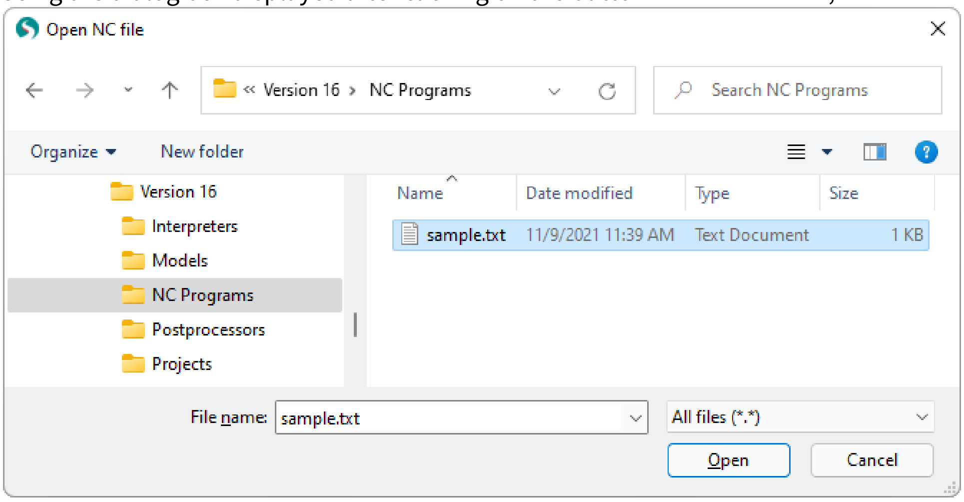
1. **Written manually** directly in the built-in text editor:

   > **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
   ```

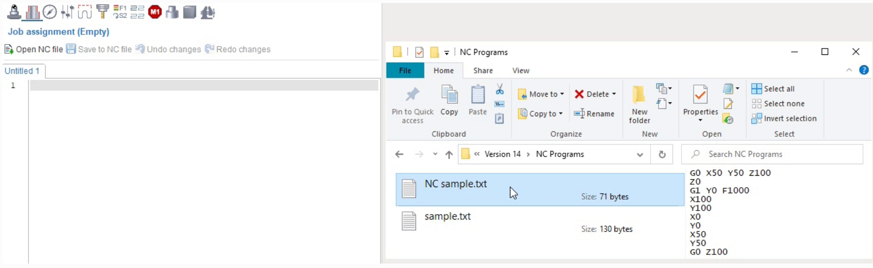
2. **Loaded from an external file** and edited, if necessary. File opening is done:

   a. Using the dialog box displayed after clicking on the button Open NC file;

   b. 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

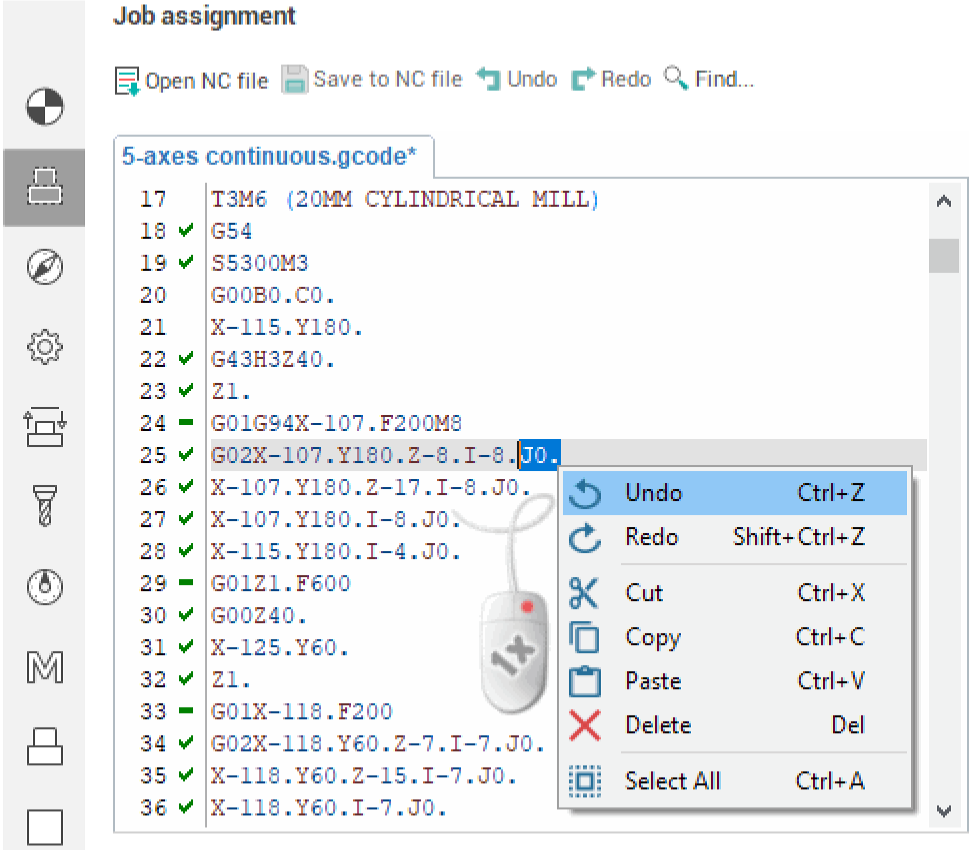


By default, the title of the operation parameters panel is < **Job assignment** Empty >, and the text editor tab name is < Untitled # > for new projects or < Project_Name # > for previously saved projects. The < # > symbol indicates the tab sequence number. After saving the text of the NC program to 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 SprutCAM X 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.

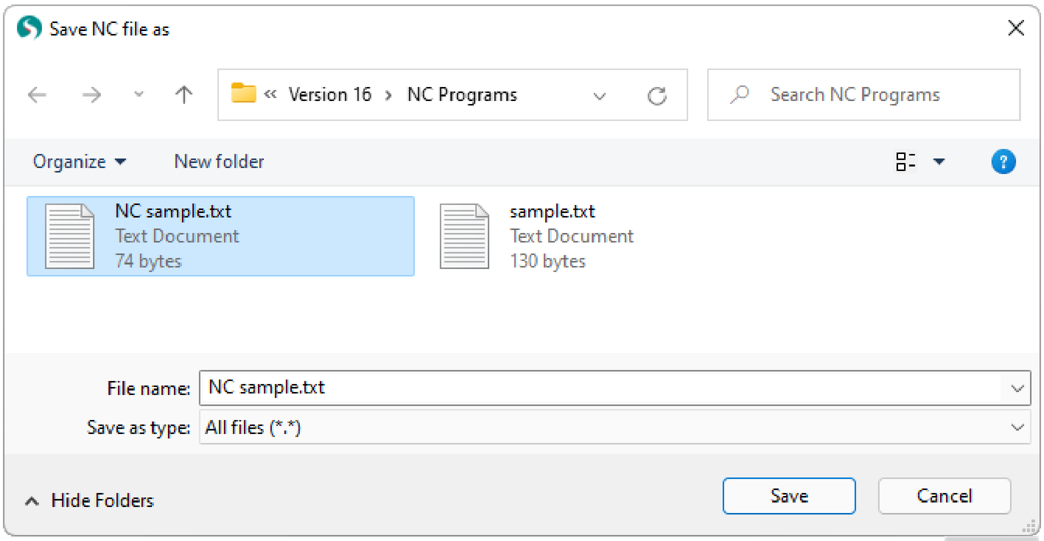


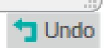
Displaying the name of the tab in bold and also the < * > symbol at the end of its name during editing indicates that there is a change between the last text state at the moment of opening or saving the NC file and its current representation on the screen.

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 . 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.



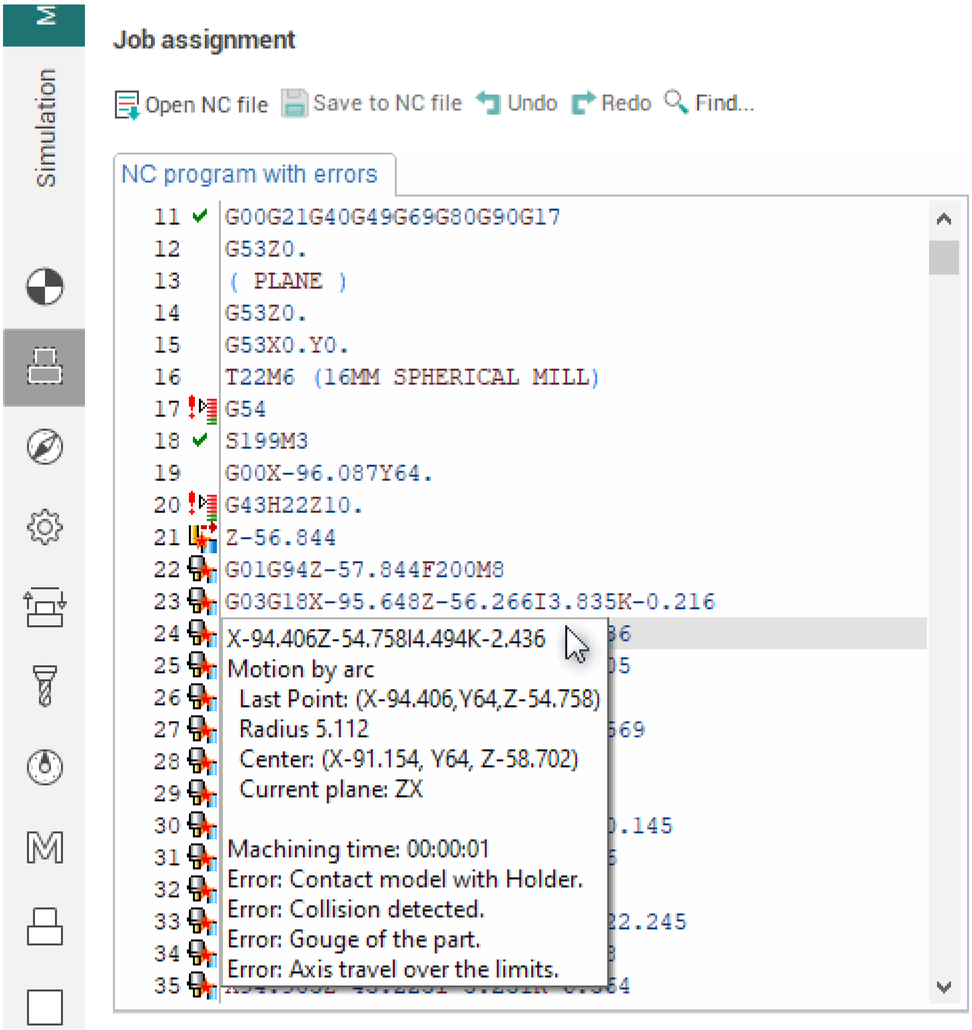
- All changes made since the last state savings will be canceled by pressing the button 

 or using the hot key <**Ctrl + Z**>.

Note: *If the changes made are accidentally canceled, they can be returned by pressing the button* 

 *or using the hotkey <**Shift + Ctrl + Z**>.*

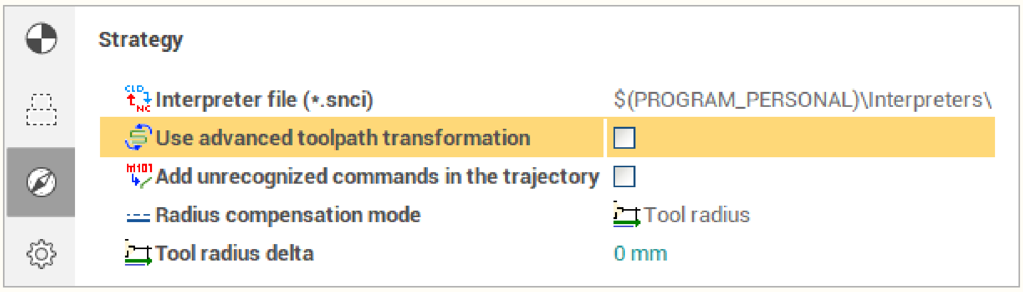
After a slight delay of the mouse pointer over the line with the NC program text, a popup hint shows a description of the associated nodes of the trajectory tree (CLData commands).



***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: [↓] or [↑].*

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).

See also:

G-code based operation / G-code based lathe operation

Keyboard shortcuts for working with NC text

Tool path errors detected by simulation

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

Key strokes	Perform action
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
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

Key strokes	Perform action
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
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

See also:

G-code based milling operation

Job assignment for G-code based operation, G-code based lathe operation

9.7 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.

9.7.1 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.

See also:

Operations which requires adaptation

Configurations of SprutCAM X

9.7.2 Welding

The following operations are possible

- «Laser welding»;
- «Arc welding»;
- Another kind of welding.

Note: option must me licensed separately.

See also:

Operations which requires adaptation

Welding

Welding 5D and 6D operations

Welding option

Configurations of SprutCAM X

9.7.3 Cladding

The following operations are possible

- Laser cladding
- Arc weld cladding
- Another kind of additive technologies

Note: option must me licensed separately.

See also:

Operations which requires adaptation

Configurations of SprutCAM X

9.7.4 Jet Cutting

The following operations are possible

- «Laser cutting»
- «Hydro cutting»
- «Plasma cutting»

Note: option must me licensed separately

See also:

Operations which requires adaptation

Configurations of SprutCAM X

9.8 Teamcenter Integration Module

The description of the Teamcenter integration module is located at this link Teamcenter PLM Integration Module.

10 Appendix

No content in this page. See child topics

10.1 Operations matrix

Configurations Operations	Express	Wire EDM	Lathe	Cutting	2.5 Mill	3x Mill Entry	3x Mill Advanced	Rotary	5x Mill
Operations for 2/2.5D milling									
2D contouring	✓			✓	✓	✓	✓	✓	✓
Engraving					✓	✓	✓	✓	✓
2.5D pocketing					✓	✓	✓	✓	✓
2.5D wall machining					✓	✓	✓	✓	✓
2.5D cover machining					✓	✓	✓	✓	✓
2.5D chamfer machining					✓	✓	✓	✓	✓
Pocketing	✓				✓	✓	✓	✓	✓
Hole machining	✓			✓	✓	✓	✓	✓	✓
Operations for 3D milling + indexed rotary axes									
3D contouring						✓	✓	✓	✓
Flat land machining	✓				✓	✓	✓	✓	✓
Waterline roughing operation						✓	✓	✓	✓
Plane roughing operation						✓	✓	✓	✓
Waterline finishing operation						✓	✓	✓	✓
Plane finishing operation						✓	✓	✓	✓
Optimized plane operation							✓	✓	✓

Configurations Operations	Express	Wire EDM	Lathe	Cutting	2.5 Mill	3x Mill Entry	3x Mill Advanced	Rotary	5x Mill
Complex operation (waterline-plane)							✓	✓	✓
Morph operation							✓	✓	✓
3D Helical operation							✓	✓	✓
Scallop operation							✓	✓	✓
Face milling					✓	✓	✓	✓	✓
FBM						✓	✓	✓	✓
Operations for 4-axes and 5-axes milling									
4D contouring operation								✓	✓
4D surfacing								✓	✓
Rotary waterline								✓	✓
Rotary roughing								✓	✓
Rotary finishing								✓	✓
Morph 4D								✓	✓
5D contouring									✓
5D surfacing							✓		✓
5D by meshes									✓
Rest machining									
Corners cleanup							✓	✓	✓
Pencil							✓	✓	✓
Chamfering					✓	✓	✓	✓	✓

Configurations **Operations**	**Express**	**Wire EDM**	**Lathe**	**Cutting**	**2.5 Mill**	**3x Mill Entry**	**3x Mill Advanced**	**Rotary**	**5x Mill**
Lathe									
OD roughing, ID roughing operations			✓		○	○	○	○	○
OD finishing, ID finishing operations			✓		○	○	○	○	○
Lathe facing operation			✓		○	○	○	○	○
Lathe hole machining operation			✓		○	○	○	○	○
Lathe part-off operation			✓		○	○	○	○	○
OD grooving, ID grooving, Face grooving operations			✓		○	○	○	○	○
OD threading, ID threading, Profile threading operations			✓		○	○	○	○	○
Cut machining									
Jet cutting				✓	✓	✓	✓	✓	✓
Jet cutting 4D				✓					✓
Jet cutting 5D									✓
Wire EDM									
2D contouring		✓			○	○	○	○	○
4D contouring		✓			○	○	○	○	○
Cladding									
Area cladding						○	○	○	○
Curve cladding						○	○	○	○

Configurations Operations	Express	Wire EDM	Lathe	Cutting	2.5 Mill	3x Mill Entry	3x Mill Advanced	Rotary	5x Mill
Cladding 3D						○	○	○	○
Cladding 5D									○
Welding									
Welding									
Knife cutting									
Knife cutting 2D				○		○	○	○	○
Knife cutting 6D				○		○	○	○	○
Disc tool									
Disc roughing				○	○	○	○	○	○
Disc cutting 2D				○	○	○	○	○	○
Disc cutting 6D				○	○	○	○	○	○
Auxiliary operations									
Group	✓	✓	✓	✓	✓	✓	✓	✓	✓
Auxiliary	✓	✓	✓	✓	✓	✓	✓	✓	✓
G-code based, G-code based lathe	○			○	○	○	○	○	○

✓ — available

○ — option (need additional license)

10.2 SprutCAM X features matrix

Configuration **Types of machining**	**Express**	**Wire EDM**	**Lathe**	**Cutting**	**2.5x Mill**	**3x Mill Entry**	**3x Mill Advanced**	**Rotary**	**5x Mill**
3+2					✓	✓	✓	✓	✓
Turn XZCY					○	○	○	○	○
Turn XZCYB								○	○
WireEDM					○	○	○	○	○
Additional advanced modules									
Features									
Adaptive SC						○	○	○	○
G-code based simulation	✓	✓	✓	✓	✓	✓	✓	✓	✓
Teamcenter integration		○	○	○	○	○	○	○	○
Robot** +				○					○
Multiblade Basic									○
Multichannel					○	○	○	○	○
Virtual PC	○	○	○	○	○	○	○	○	○
Block internet activity	○	○	○	○	○	○	○	○	○
Block all export	○	○	○	○	○	○	○	○	○
Block MachineMaker	○	○	○	○	○	○	○	○	○
Block NC output	○	○	○	○	○	○	○	○	○
Operations									
5D MW									○
G-code based operations	○		○	○	○	○	○	○	○
Additive 5D									○
Additive 3D						○	○	○	○

Configuration Types of machining	Express	Wire EDM	Lathe	Cutting	2.5x Mill	3x Mill Entry	3x Mill Advanced	Rotary	5x Mill
Disc cutting 2D				○	○	○	○	○	○
Disc cutting 6D									○
Disc Roughing							○	○	○
Knife cutting 2D				○		○	○	○	○
Knife cutting 6D				○		○	○	○	○
Welding									○
Painting									○
MultiAxis cutting				○					✓

✓ — available

○ — option (need additional license)

** — Feature used supporting robots in milling configurations.

10.3 List of interpreters

Currently, interpreters of the following CNC systems are available for use:

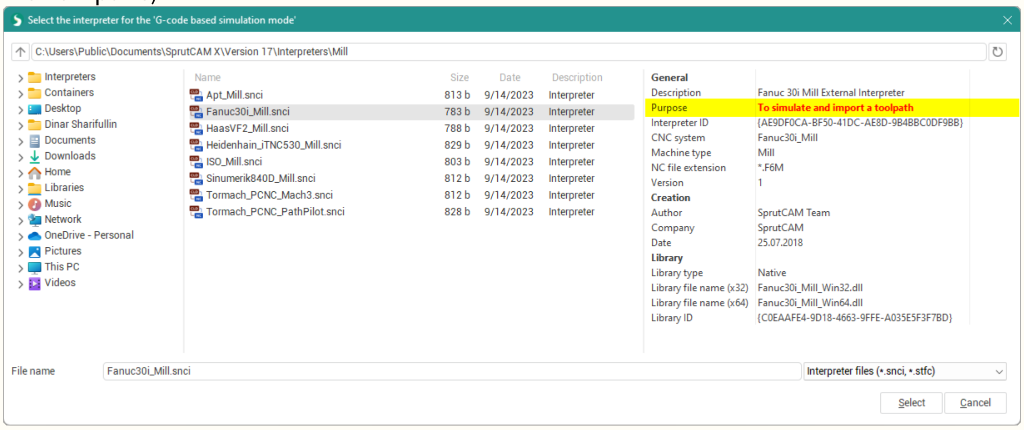
Machine group	CNC system	Comment	Note
Milling	APT	Import toolpath only	
	Apt_Simplify_3D	Import toolpath only	
	ISO	Import toolpath only	
	Global control	Import toolpath only	An additional license is required
	Fanuc 30i	To simulate and import a toolpath	
	Haas VF-2	To simulate and import a toolpath	
	Heidenhain iTNC 530	To simulate and import a toolpath	
	Mazatrol SmoothG	To simulate and import a toolpath	An additional license is required

Machine group	CNC system	Comment	Note
	NC210	To simulate and import a toolpath	An additional license is required
	Sinumerik 840D	To simulate and import a toolpath	
	Tormach PCNC Mach3	To simulate and import a toolpath	
	Tormach PCNC PathPilot	To simulate and import a toolpath	
Turning	Mazatrol SmoothC	To simulate and import a toolpath	An additional license is required
Turn-milling	Fanuc 21i	To simulate and import a toolpath	An additional license is required
	NC220	To simulate and import a toolpath	An additional license is required
	Sinumerik 840D	To simulate and import a toolpath	
	Okuma OSP-P300	To simulate and import a toolpath	An additional license is required
Robot	Fanuc robot (R-30iB controller)	To simulate and import a toolpath	
	Kuka robot	To simulate and import a toolpath	
	Motoman robot	To simulate and import a toolpath	
	ABB robot	To simulate and import a toolpath	
	Nachi robot (AW Format)	To simulate and import a toolpath	An additional license is required

Note: *All interpreters support command list generated by postprocessors in SprutCAM X distribution kit only.*

"Import toolpath only" interpreters are not supported matching line NC code - trajectory of tool movement.

When you select an interpreter, pay attention to its purpose (the **Purpose** field in the Preview pane)



The selected interpreter should be intended for simulation. Otherwise, the trajectory of the tool may be incorrect (shifted relative to the coordinate system of the workpiece, duplicated approaches/retracts, incorrect starting position, etc.).