Numerical Control Machining

SPRUT–Technology

CONTENTS

INTRODUCTION TO SPRUTCAM

We thank you and congratulate you for choosing **SprutCAM**, our very powerful 32-bit Windows CAM system. This new generation system works directly with the imported CAD data of the initial model (including NURBS-representation) without any preliminary approximation or triangulation, and automatically calculates and develops a very efficient toolpath for machining the model, which can be graphically simulated to allow the user to see any potential problems with the type of machining parameters chosen. After the acceptance of this toolpath by the user, **SprutCAM** will generates an NC program using one of its extensive list of Posts. The system also allows users to develop Posts to their own specific requirements.

The model can be of various complexity whether two or three dimensional, and **SprutCAM** can generate a programme for **2-axis**, **2 ½-axis**, and **3-axis** machines, as well as allowing indexed **4-axis** machining. A vast choice of machining methods and strategies is available, and the level of accuracy can be set by the user. All this can run on any standard PC without requiring excessive power.

The most important features of the system can be highlighted as follows:

1. **Ease of Use**: - The system is very easy to use and is logically well organised 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 photorealistic 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** via IGES, DXF, STL, VRML, PostScript, Parasolid XT, 3dm, or SGM file format. Within **SprutCAM** 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 so that machining and restricted areas can be defined. Patterns and text for engraving or pocketing can also be created within the system.
4. **Very Sophisticated Machining cycles**: - The machining process can be set up very easily and is made up of a sequence of operations which are chosen by the user from a long list available; i.e. roughing, finishing, rest milling, hole drilling, engraving etc. Within each operation the user chooses the parameters that should be applied; i.e. water-line, 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. **Photo-realistic simulation**: - 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 video tape is being played back. He can choose the colour 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 colours.

6. **Postprocessor**: - Once the machining cycle is accepted the programme can automatically generate an NC programme to suit the user's machine or CNC system. Besides the long list of available Posts available and willingness of **Sprut Technology** to develop Posts to the user's requirements and request, it is also possible for the user to generate new posts or modify existing ones by using the inbuilt Invariant Processor.

With the powerful **SprutCAM** system the user can confidently undertake fast machining of very accurate parts, be they very complex 3D models, or simple engraving or pocketing, and in any material. Typically it can be used to machine punches, spark erosion electrodes, plastic moulds, machine parts, decorative elements, nameplates etc. Because it is truly Windows based and very easy and intuitive to use, 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.

BASE CONFIGURATIONS

The **SprutCAM** System ® is supplied in two basic configurations: **SprutCAM Expert** and **SprutCAM Machinist**. The configurations differ mainly in the availability of machining operations and number of supported CAD data import formats.

SprutCAM Expert

SprutCAM Expert - the most complete version, containing operations for the machining of 2D & 3D models including engraving. Designed for undertaking many tasks in **2**, **2 ½** and **3 axis** machining on CNC machines.

Importable formats:

- IGES
- DXF
- STL
- VRML
- PostScript
- Parasolid XT
- 3dm
- SGM

Machining operations:

- 2D-contour machining
- 3D-curve machining
- Engraving operation
- Pocketing
- Hole machining
- Waterline roughing operation
- Plane roughing operation
- Drive roughing operation
- Waterline finishing operation
- Plane finishing operation
- Drive finishing operation
- Plane finishing optimized operation (plane-plane)
- Complex finishing operation (plane-waterline)

- Combined finishing operation (waterline-drive)
- Operations for re-machining of rest material

SprutCAM Machinist

SprutCAM Machinist – version containing operations for 2D machining and engraving. Designed mainly for 2/2.5D machining on CNC machines.

Importable formats:

- IGES
- DXF
- PostScript
- 3dm

Machining operations:

- 2D-contour machining
- Engraving operation
- Pocketing
- Hole machining
- Operations for machining of 'rest' material
- 2.5D Pocketing
- 2.5D Walls machining
- 2.5D Chamfer machining

SYSTEM REQUIREMENTS

Disk space, needed for the system about 50 Mb for minimal setup and about 120 Mb for full.

Recommended RAM amount - not less than 64 Mb. Performance improves proportionately with additional RAM.

Display resolution - 1024x768 or more, 65535 colors.

Processor - Pentium III 600(MHz) or better.

The use of a video adapter with OpenGL significantly improves 3D visualization performance.

Windows operating system (Windows 95/98/ME/NT/2000/XP).

Note: *When using earlier versions of Windows 95, than OSR 2, it is necessary to check the presence in System folder of the OS catalogue dynamic libraries COMCTL32.DLL, GLU32.DLL, OPENGL32.DLL. If these libraries are missing, then copying them from later versions of Windows may be required.*

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 trajectory. Therefore, the higher specification of the computer, the faster the calculations will be performed.*

STANDARD PACKAGE

The standard **SprutCAM** ® package includes:

1. CD with **SprutCAM** ® system.

2. Documentation.

3. Electronic key to prevent unauthorized copying (depends on configuration and protection method).

4. License agreement.

5. Package box.

SYSTEM INSTALLATION AND LAUNCH

Install **SprutCAM** ® on the computer as follows.

1. Insert the CD into CD-ROM drive.

2. In Windows run SETUP.EXE from the CD. This can be done, for example, from the Start -> Run menu. In the 'Run' dialogue type :\SETUP.EXE (e.g. if your CD drive is D, then type D:\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:

- User name and organization;
- Folder, where the program shall be installed to (by default - C:\Program Files\Sprut Technology**SprutCAM****, where ** - version number);
- Choose default, minimal or custom installation (can exclude example project files, importable files, and other auxiliary files);
- Main menu folder, where the shortcuts for the executable system files will be placed (by default - **SprutCAM****, where ** - version number).

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

4. 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**** -> **SprutCAM**. Further, we recommend that a shortcut is created for **SprutCAM** on the Desktop or quick launch panel.

Note: *In some cases, when installing* ***SprutCAM*** *with electronic key under the Windows NT/2000 systems, the user may need to additionally install the electronic key driver. To do so, during the installation process in the Installation type window choose menu item Custom, and then in the next Select components window put checkmarks in the string Driver for key. The Key drivers will be saved to KeyDRV folder inside the* ***SprutCAM*** *folder. Installation of the key driver is necessary for the software to run. To install the key driver double click the file called Hdd32.exe located in the KeyDRV folder. Alternatively, this can also be done by selecting START - Programs -* ***SprutCAM***** - 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).

SYSTEM FILES

SprutCAM.exe – executable system module.

INP.exe – executable invariant postprocessor module.

SprutTutorial.exe – executable tutorial module.

***.stc** – system project files, contain geometrical model and machining operations. Calculated tool movement trajectories are stored with the same name but with mcd extension.

***.mcd** – connected with the project files of machining commands of operations, contain the calculated toolpaths. Can be used to generate NC programs by postprocessors. When opening a project file without the corresponding mcd-file, the machining operations reset automatically, and to obtain an NC program they must be recalculated.

CAD Files of type:

- *. Igs;
- *. Iges;
- *. Dxf;
- *. Stl;
- *. Vrl;
- *. Ps;
- *. Eps;
- *.x_t;
- *. 3dm;
- *. Sgm;

can be imported into **SprutCAM**.

***.spp** – postprocessors tuning files for the different CNC system (used by the **postprocessor** and the **invariant postprocessor**, generated by the **invariant postprocessor**).

***.ppp** – postprocessor tuning files of previous versions of **SprutCAM**. The old format is supported by the new version of **SprutCAM** too. **Postprocessors Generator** can reform the files into the ***.spp** format.

***.inp -** text version of the postprocessor files. (used and generated by the **invariant postprocessor**).

***.dsk** – 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 proper functioning of the program. Their modification or deletion can prevent the proper functioning of the program. Substitution of the files is needed only when upgrading the version and should be performed in strict accordance with the update information supplied.

TECHNICAL SUPPORT

Technical support of the software is carried out by the dealer or one of the representatives of SPRUT-Technology Inc.

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

Forum on our company WEB-page: www.sprut.ru

Technical support service: support@sprut.ru

Telephone "Hot line": (+7-8552) 59-94-10 or (+7-095) 263-69-70

Postal address:

JSC SPRUT-Technology,

P.O. 438,

Naberezhnye Chelny,

423812,

Russia

or

LLC Center SPRUT-T,

2nd Baumanskaya 5,

Moscow,

105005,

Russia

For frequent notification about updates of the current version, and release of new versions of **SprutCAM** ®, we recommend you contact your dealer or company representatives.

1 BRIEF AND TO THE POINT

1.1 DESCRIPTION OF SPRUTCAM

SprutCAM ® is a fully automatic system with many advanced functions.

The system creates all of the machining operations and replaces any manual composition of an NC program, command after command, or block after block, instead the user will only need to define what shape to do, and how to machine it. A user imports the model to be machined and defines the general requirements for the machining process, such as the height of the scallop, maximum cut angle, approach methods etc. And then, based upon the specified information the system calculates the optimal tool path within predefined limits and tolerance.

The procedure to obtain an NC program for a CNC machine usually consists of the following actions: import a model, create a sequence of machining operations, calculate them, and generate an 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 model. Thus, very few values need to be entered into the operation, before the toolpath is ready for calculation. Modification of the order of the machining operations and editing their parameters is possible at any stage of machining process calculation.

SprutCAM always observes the rule never to "**over-cut**" a model under any circumstances, whether it is a work pass, transition, approach, cutting or drilling. And it does not depend on the tool type, or on the type of machining or parameters entered. A user defines the machining method and the system generates an NC program that removes material from 'outside' of the model.

The interface of **SprutCAM** allows the user to alter any parameter in any order that is required. As the user selects different operations and parameters, the 'active graphics' in many of the dialogues update automatically. This allows the user to significantly reduce the time needed to get to know the system and the time spent reading the documentation.

SPRUT-Technology is always looking at how to improve their software products to make them convenient to use and boost your profits. Therefore, our technical support department will gladly answer any of your questions and be grateful for your suggestions and wishes.

1.2 FAST FAMILIARIZATION WITH THE SYSTEM

To get to know **SprutCAM** quickly, you will need to perform the following typical sequence of actions:

Set work mode to 3D model, by selecting the corresponding tab sheet on the main panel

.

Import model from graphical data exchange fileby, clicking the 

button and selecting the file you want in the window. You can also load a previously created project by clicking the  button on the main panel.

Set work mode to Machining by selecting the corresponding tab sheet on the main panel

.

Create a new operationby pressing the 

button and select the type of operation in the opened window. The newly created operation will be set as current and is ready for editing and execution.

Define the operation parameters. When creating an operation the system automatically sets default parameters, based on the type of operation and geometrical parameters of the model. In many cases any additional correction of parameters are not required, and the operation can be immediately executed, however the user can alter any parameter in the <Operation parameters> window which has tabs for: <**Tool**>, <**Feedrate**>, <**Toolpath**>, <**Parameters**> and <**Strategy**>.

Start operation execution by clicking the 

button. There is a process indicator at the bottom of the page

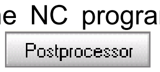
which shows how much of the calculation has already been completed both by the analogue bar on the right, as well as the percentage figure on the left. By clicking within the indicator area, you can stop the process.

The calculated machining operations can be photo-realistically simulated by selecting the Simulation tab on the main panel:

.

For each machining operation there is a list of all the individual movement instruction which, if opened during the simulation, shows each instruction being performed.

After simulation it is possible to change any of the parameters and recalculate the operation.

Generate the NC programme: Return to the machining panel and press the 

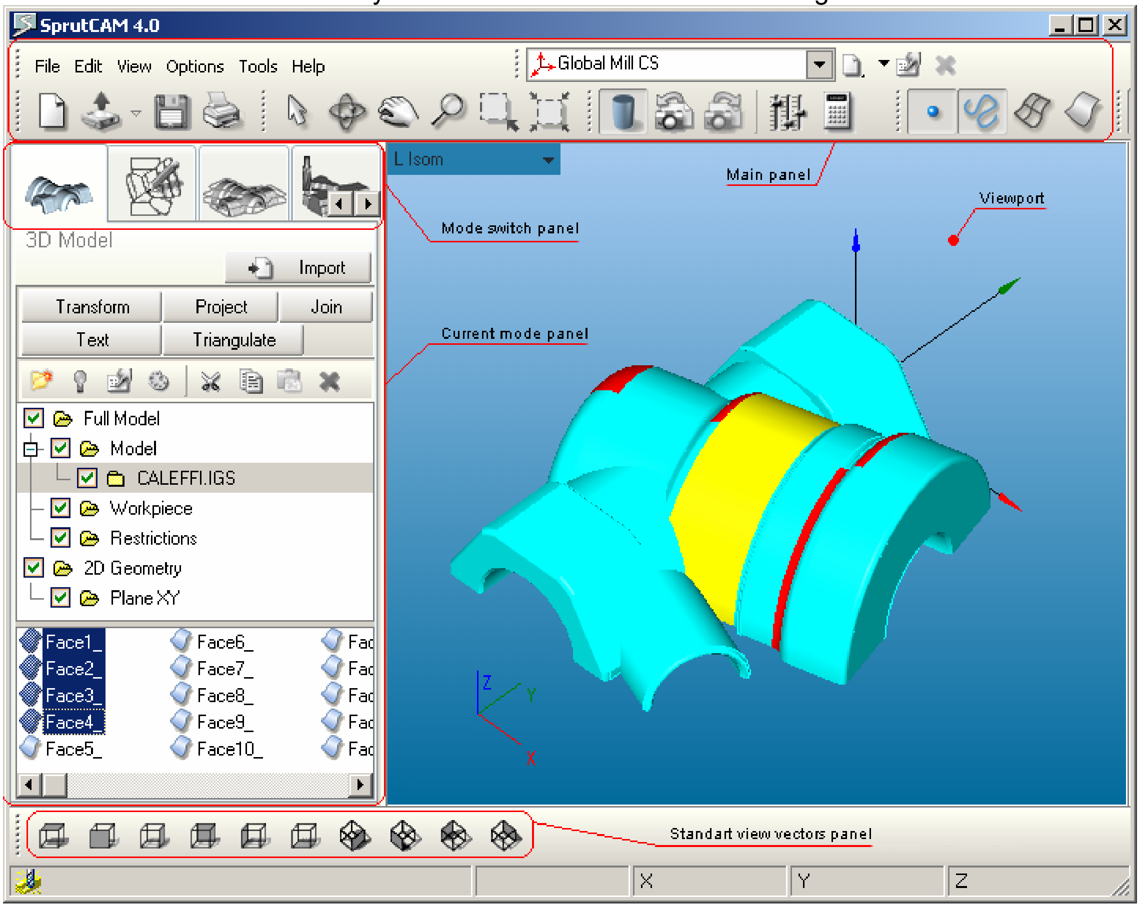
button at the bottom. In the opened

Postprocessor window, select the CNC system that you require the program to be written for (choose postprocessor file with <**spp**> extension) and select the < **Run** > button. The CNC code will be written to a file with the project name, and the 'file extension' set in the postprocessor.

2 GENERAL INFORMATION

2.1 SYSTEM'S MAIN WINDOW

The system's main window has the following view:



2.1.1 Main menu.

The Main menu consists of the following seven main items with the sub functions shown below. Almost all the items are duplicated on the main panel.

- <**File**>
  - <**New**> – closes the current project and initiates the system state anew. Deletes the current geometrical model and machining processes. Frees a part of the occupied computer memory. This function is also available from the Main Panel.
  - <**Open**> – loads an existing project from file. The menu item is duplicated on the Main Panel.
  - <**Reopen**> – loads a project from the list of the recently created ones.
  - <**Save**> –saves project under the current name. If the project has not been saved before, a new name will be requested. Function is also available from the Main Panel.
  - <**Save As**> – saves project under a new name.
  - <**Print**> - print 3D model.

- <**Processors generator**> – runs the application to generate postprocessor tuning files (Invariant postprocessor).
- <**Import**>
- <**Export in DXF**>
- <**Exit**> – quits the system. If the opened project is not saved, then a choice will be offered: either save the project or quit without it.

- <**Edit**>
  - <**Cut**> – removes the selected objects of the 3D model to the buffer. The menu item is duplicated in the model structure window and pop-up menu. This function is only available in the 3D model mode.
  - <**Copy**> - copies the selected objects of the 3D model to the buffer. This function is available both in the model structure window as well as from the context menu of the graphic window: onle when in the 3D model mode.
  - <**Paste**> – inserts the contents of the buffer into the active group. This function can be accessed from the model structure window and from the context menu of the graphic window, is only available in the 3D model mode.
  - <**Delete**> – deletes the selected objects of the 3D model. The function can be accessed from the context menu of the graphic window or by <Del> button.
  - <**Select all**> – selects all objects of the active group. The function can be accessed from the context menu of the graphic window or by button combination.

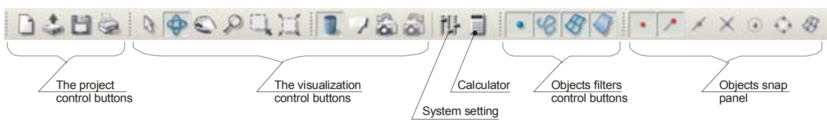
- <**View**>
  - <**Toolbars**> - shows/hides system panels, so as
    - <**Files**>;
    - <**View control**>;
    - <**Modes**>;
    - <**Filter**>;
    - <**Snaps**>;
    - <**Coordinate system**>;
    - <**View vectors**>;
    - <**Tools**>;
  - <**Model**> –switches the system to "3D Model" mode.
  - <**Geometry**> – switches the system to "2D Geometry" mode.
  - <**Operation**> – switches the system to "Machining" mode. Switching between the modes is also available from the tabsheets on the main panel.
  - <**Simulation**> – switches the system to "Simulation" mode. Switching between the modes is also available from the tabsheets on the main panel.

- <**Options**>
  - <**Edit**> - Customize dialog box allow you to choose which toolbars to display, and to add or delete buttons on the toolbar.
  - <**Load**> - opens the system setting window. The function can be accessed from the main panel.
  - <**Save as…**> - runs calculator. The menu item is duplicated on the main panel.

- <**Tools**>
    - <**Addin manager**> – opens the external applications window.
    - <**Calculator**> - runs calculator. The menu item is duplicated on the main panel.
    - <**Configure Tools**>
- <**Help**>
    - <**Contents**> – runs **SprutCAM** tutorial.
    - <**Help**> – loads WEB-page of JSC "Sprut-Technology". Server address: http://www.sprut.ru.
    - <**SprutCAM tutorial**> – runs **SprutCAM** tutorial.
    - <**SprutCAM forum**> – loads forum WEB-page of JSC "Sprut-Technology".
    - <Message **to a technical support**>
    - <**Check for update**>
    - <**On-line registration**> - allows user to register online on Sprut-Technology web-site, in order to be included in the Sprut-Technology news mailing list.
    - <**SprutCAM Page**>
    - <**SPRUT Technology home page**> – loads WEB-page of JSC "Sprut-Technology". Server address: http://www.sprut.ru.
    - <**Contact with Sprut Technology Inc**.> – prepares electronic mail for Sprut Technology technical support department. E-Mail:support@sprut.ru.
    - <**About…**> – information on **SprutCAM** ®.

2.1.2 Menu Bar

The Menu Bar is located in the upper part of the main window and provides access to the various tools, commands and functions such as the project management, visualization management, Cencel, undo, objects snap panel, calculator and menu tuning buttons are located.

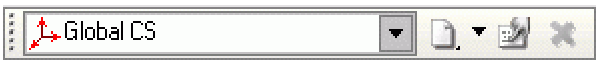


2.1.3 Coordinate system

To make the machining process generation more convenient the user may create any number of local coordinate systems. A coordinate system can be assigned as a parameter for any machining operation. The tool rotation axis will always be coincident with the Z axis of the defined coordinate systems, all geometrical parameters must be assigned relative to the defined coordinate systems, and the tool movement trajectory will be calculated in the same coordinate systems.

Coordinate systems management panel

Coordinate systems management panel is shown below



Active coordinate system – coordinate system, whose parameters are being used to display the model. All newly created operations, by default use the currently active coordinate systems.

- Selection of a coordinate system creation method.

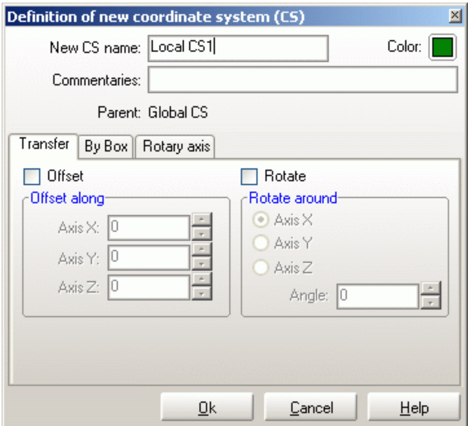
Creation of a coordinate system by dialogue window – Left clicking on this button will open the CS creation dialogue, within which you can left click and hold (or drag) a tab to obtain a sub menu for creating CS. as follows:

Coordinate system parameters window – Opening of the parameters window for the active coordinate system.

Deletion of a coordinate system – Deletion of the active coordinate system. All listed coordinate systems below this will rise by one level up. The global coordinate system cannot be deleted.

Creation of a coordinate system by use of the dialogue window.

In this method, the window shown below appears and is used to create a coordinate system by assigning the required parameters.



All transformations are performed relative to the active coordinate system. A newly created system can be displaced or rotated relative to the parent CS. If an additional rotation axis is used, then a coordinate system, that defines the position of the corresponding rotary axis, can be created. Upon clicking the cancel button, this operation can be discontinued and no new coordinate system are created nor is any changes made to the active one.

Interactive creation of a coordinate system by selection of an initial point and two leading vectors.

In this case the coordinate system is defined by the Zero point of coordinates and two leading vectors X and Y. When creating a CS by this method, first the Zero point must be assigned by moving the cursor on the screen to the desired point and if it is a valid point to be used as, the origin of the CS, then it is highlighted in green color. 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.

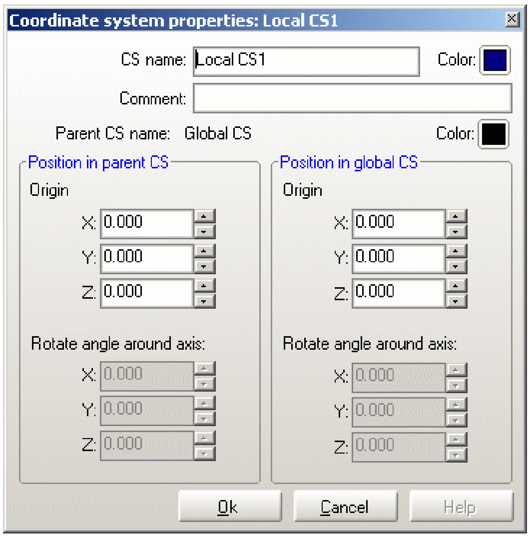
Interactive creation of a coordinate system by the current visualization plane.

 When creating a CS by this method, first the required view must beselected either by using the  button, or by using the standard view buttons at the bottom of the main screen. A Zero point must be assigned by moving the cursor to the desired point on the screen, and if it is a valid point that can be used as the origin of CS, it is highlighted in green color. The selection is confirmed by left clicking on it. After that, the newly created coordinate system is active.

Changing the existing coordinate system

To open the coordinate system parameters window press the  on the coordinate system panel.

The CS parameters window is shown below.



To move the selected CS, define the displacement value for the X,Y and Z axes. Displacement is performed relative to the parent coordinate system.

The name of the coordinate system can also be changed in this window, as well as its color and comment.

2.1.4 Standard view bar

The standard view bar is located at the bottom of the main window, and has six orthographic plane views: XY, ZY, -XY, -ZY, XZ, -XZ and four isometric views: -YXZ, -XZY, -YZX, XYZ. 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.

The standard view bar can be switched on/off from the menu item View -> Vector management. It is also dockable and can be located at any convenient place of the screen.

2.1.5 Objects filter

When choosing objects, they can be filtered by their type (point, mesh, surface).

Filter parameters can be set up by pressing the corresponding buttons on the Objects Filter Bar. By left-clicking the chosen button the corresponding filter is toggled on and off to: -

 – Allow/Restrict selection of points.

 – Allow/Restrict selection of curves.

 – Allow/Restrict selection of meshes.

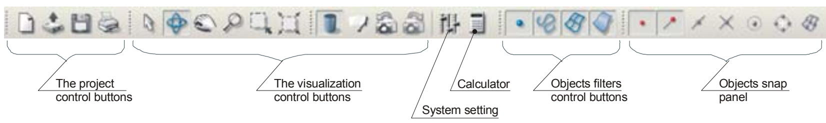
 – Allow/Restrict selection of surfaces.

The panel can be switched on/off from the menu item View -> Objects Filter. The panel is dockable and can be located at any convenient place of the screen.

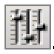
2.1.6 Process indicator

The process indicator runs when the system performs lengthy operations such as the geometrical model import and the toolpath calculation. Left clicking on the indicator cancels these operations only after being reconfirmed by the user.

2.1.7 Viewport and visualization control



The viewport is the biggest part of the main window. It displays a geometrical model, toolpath and 2D-geometrical elements.

User can create as many viewports as he needs. To add view port click in view caption panel (in the top left corner) and select "New view" in the opened popup. Active view can be closed via the same popup. Key mappings for the visualization control, background color, axes visualization parameters and other can be changed on the visualization tab of the set up window .

The user can control the graphical output by using the main panel buttons irrespective of which mode is selected. The following functions are available in the viewport (default key mappings):

Interactive rotate

Press and hold the right mouse button, and drag in the main visualization screen to rotate. Or use left mouse button with pressed . Drag up and down to rotate around the horizontal screen axis. Drag left and right to rotate around the vertical screen axis. Changing the view vector is also available by the view vector toolbar.

To activate this toolbar choose View -> Vector Management from the main menu. Press and hold X,Y or Z button and drag mouse to rotate around X,Y or Z axis of active coordinate system.

Interactive pan

Press and hold the middle mouse button, and drag in the main visualization screen to move graphic objects to a new location. Or use left mouse button with pressed .

Interactive zoom in and out

Press this button to set the zoom viewport mode. Press and hold the left mouse button, and drag vertically in the main visualization screen to dynamically zoom in and out. Horizontal mouse cursor movement does not affect imaging in any way in this mode.

Also you can use Ctrl + right mouse button or mouse wheel.

Window zoom

Zooms to display an area specified by two opposite corners of a rectangular window. Press this button to set the window zoom viewport mode. 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 cancelled.

Also you can use Alt+right mouse button.

Zoom extents

Press this button to set the view scale which will display all graphical objects in the current viewport. The current mode of the viewport will not be changed.

Undo

The button allows the user to save the current parameters of the active viewport (scale, visualization vector and etc.). The function is duplicated in the main menu.

Redo

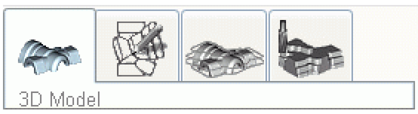
The button allows the user to restore the earlier saved parameters of the active viewport. The function is duplicated in the main menu.

Shade mode

Use this button to switch on/off the shade (render) mode. This mode is not available in the "2D Geometry" mode. The function is duplicated in the main menu.

2.1.8 Work modes

The main panel tab sheets are for the current 'work mode' management.



In **3D Model** 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) from the existing ones, and manage the object's visual properties.

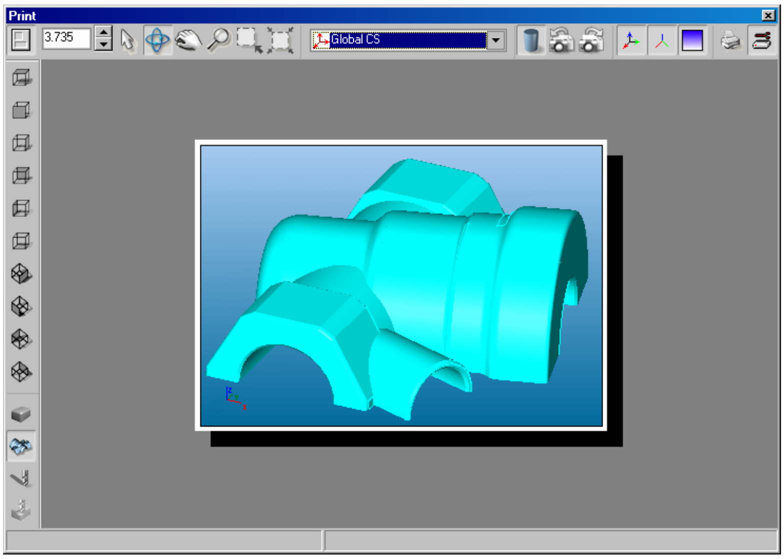
The built-in 2D Geometry  allows the user to create two-dimensional geometrical objects in arbitrary planes. The environment has powerful tools to construct parameterized geometrical models and the possibility to link them to the coordinates of a 3D model. The constructed 2D contours and points can be used when assigning parameters for subsequent machining operations.

In the Technology  mode, the user creates the machining processes, selecting from a list of machining operations of different types. Here also, the fine-tuning of all the machining operations parameters and the calculation of the tool movement trajectory can be performed. From this tab also, you can access the machining simulation subsystem to visualize the obtained toolpath; and the postprocessor, to generate NC programs.

In the Simulation  mode user has access to integrated machining simulation tools that allow the user to control dynamically the visualization parameters.

2.1.9 Printing

The objects that are displayed in the graphical view can be printed with the high resolution and in the required scale. The printing quality is defined by the printer setup. The set of the objects, view, scale and other are defined in the print dialog.



 - scale;

 - view control;

 - standard view;

Control buttons:

-  - show workpiece;
-  - show model;
-  - show tool path;

-  - show simulation result
-  - set up the printer
-  - print

Note: *The printing needs some time. The printing time depends on the quality and model complexity*

2.2 SYSTEM SETTINGS WINDOW

The system settings window can be opened by pressing the  button on the main panel or by choosing the Options -> Edit options items from the main menu.

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.

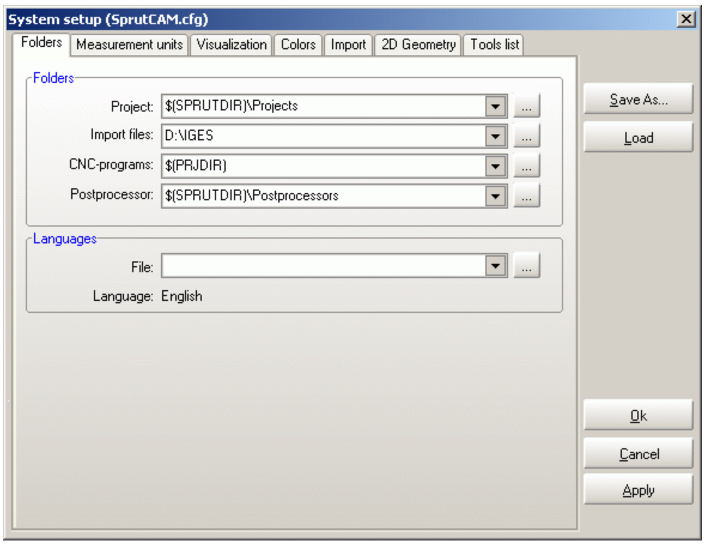
- The <**OK**> button quits this window and applies the settings to the current work session.
- The <**Cancel**> button quits this window and discards all the changes made.
- The <**Apply**> button quits this window, saving the settings into the current configuration file. The new parameters will be active in the current and future work sessions.
- The <**Save as**> button quits this window, saving the settings into the user-defined configuration file and make this file as current.
- The <**Load**> button loads the configuration file settings and make ones current. When next launching, user defined parameters will be loaded from this file.

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;
- 2D Geometry;
- Tools list;

2.2.1 Tab Folders



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

The Project directory is used for loading and saving projects.

The Import files path defines the directory, where by default the geometrical models (IGES, DXF etc.) files will be loaded from.

NC programs that have been created, by default, will be saved in the NC-programs directory.

Postprocessor tuning files, by default will be loaded from the Postprocessor folder.

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

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

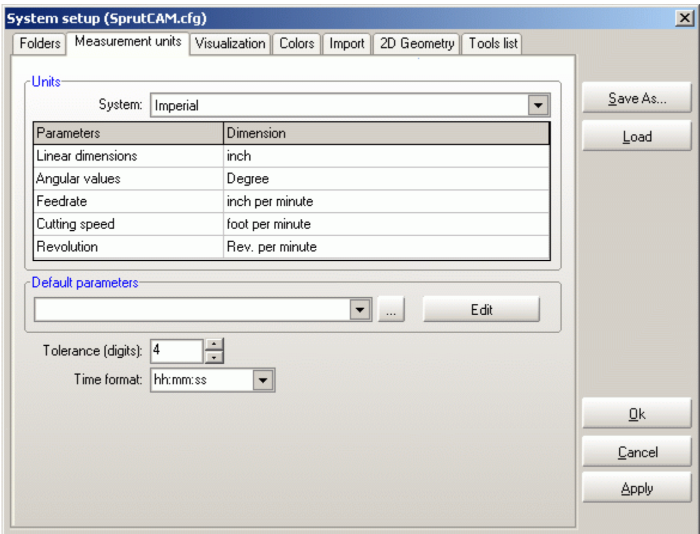
$(SPRUTDIR) - the directory from where **SprutCAM** was launched.

$(PRJDIR) – the directory defined in the Projects field.

When defining the real names of the directories used during the running of **SprutCAM**, 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.

2.2.2 Measurement units



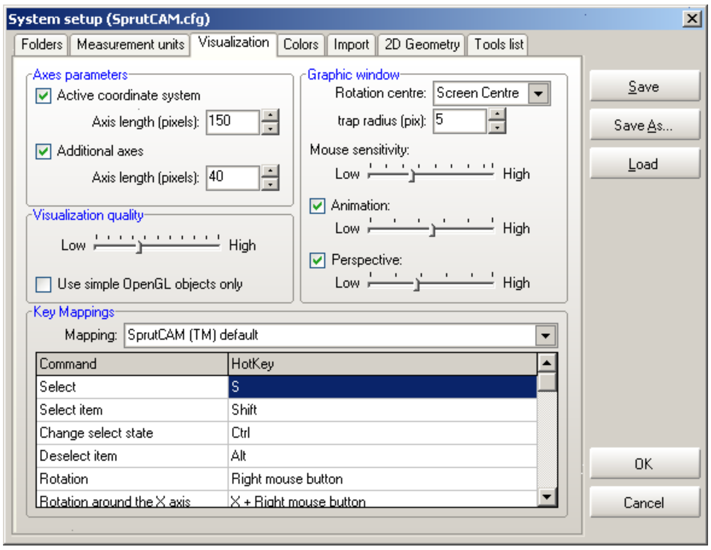
Allow define measurement units for the system.

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, all measurements of a model must be in millimeters.

Angular measurements are given in degrees with decimals.

Default parameters defines the file from which initial parameters will be taken while operation is creating.

2.2.3 Visualization



In the Axes parameters area user can control the axes drawing into view port.

In the Visualization quality area the user can assign the default visual quality of 3D objects. Moving the slider in the bottom part of the window will alter the visual accuracy. The accuracy is, the maximum deviation of sections, by which screen curves are approximated during drawing. The higher the value, the more computer memory resources will be used. The accuracy set will be applied to all the objects imported.

Note: *We do not recommend setting high accuracy on lower specification computers due to possible negative effects on speed.*

Key mappings area defines the hot keys for the visualization control. **SprutCAM** 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 Key mapping scheme only. Hot key is read only.

As default, option **Use simple OpenGL objects only is** switch off. If you have some problems with painting on your graphics accelerator try to tick "**Use simple OpenGL objects only**".

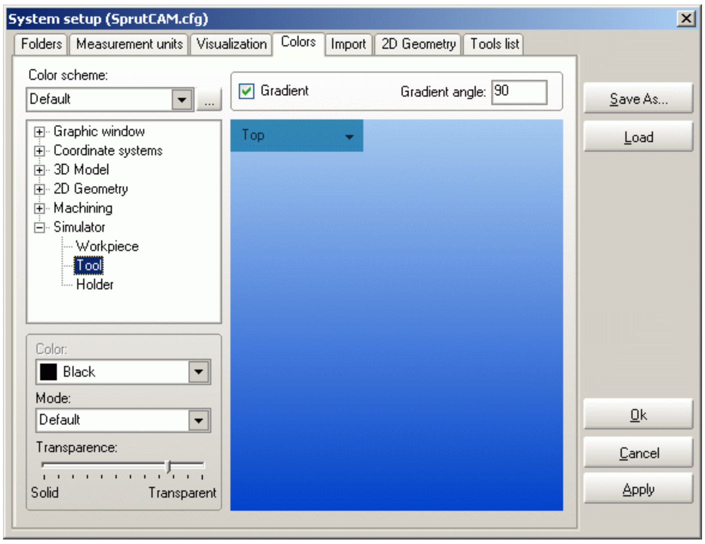
In the **Graphic window** area user can define the next parameters:

- **Rotation center** is a point of rotation center
- **Trap radius** is used for the objects selecting and highlighting.
- **Mouse sensitivity** defines the rotation and scale speed depending on mouse drag.
- **Animation** defines the rotation speed when rotation to the standard views is performed.
- **Perspective** - switch on/off the perspective mode of the graphic window.

Key mappings area defines the short keys for the visualization control.

2.2.4 Colors tab

Allows to define color scheme in the **SprutCAM**.



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.

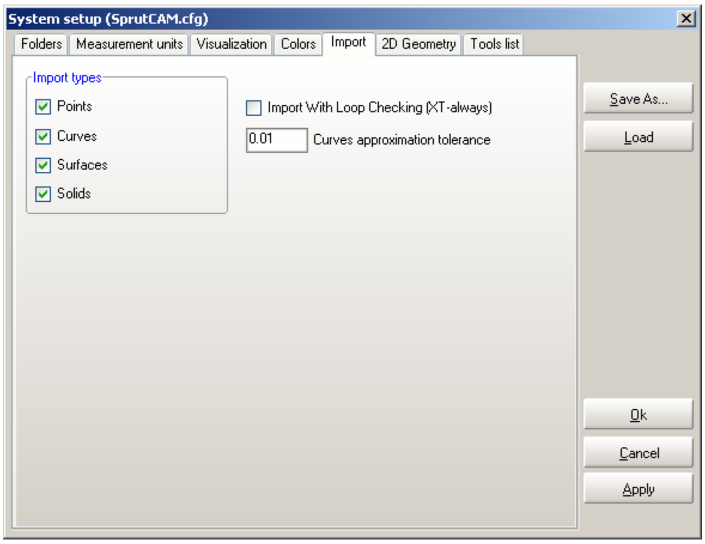
Under the color scheme, as tree-like structure, values of color of a separate element are shown. These elements are broken on groups: Graphic window, Coordinate system, 3D Model, 2D Geometry, Machining, Simulator. Having opened the necessary group, it is possible to edit values of color of any element 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 Transperence it is possible to install a transparency of a flowing element.

Check box Gradient it is possible to place Gradient drawing of a graphic window. Colors of a drawing are assigned in-group Graphic window: First gradient color and Second gradient color. In a case when gradient is switched - off, for a drawing of a graphic window Second gradient color is used.

In a field Gradient angle it is possible to set a gradient angle. The angle is set concerning a vertical axis.

2.2.5 Import tab

Here the general import parameters can be set.

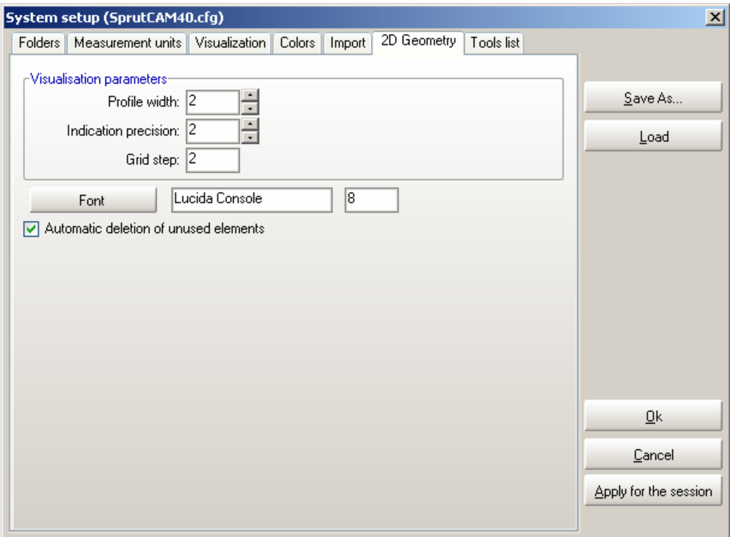


In the Import objects group, the user can define the types of geometrical objects, which can be imported from the geometrical data exchange (CAD) files. If the box opposite a specific type isn't selected, then the corresponding object will be ignored during the import process.

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

2.2.6 2D Geometry tab

Allows configuration for visual parameters for 2D geometry mode.



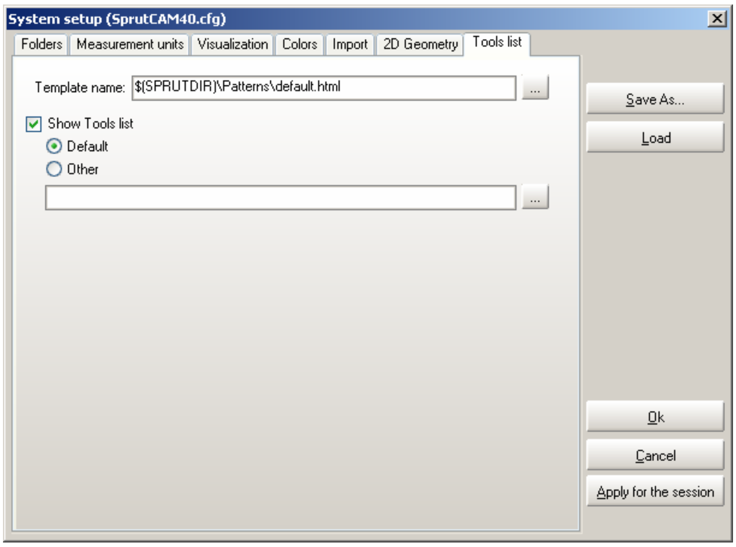
In the **Profile width** field, the line width is set, shown on the screen, in pixels.

In the **Indication precision** field, the number of digits after a comma of the current coordinates are set.

In the **Grid step** field, the distance between two nodes of a supplementary grid is set, measured in the current units of measurements.

To assign the font of the debugger window text press the **Font** button.

2.2.7 Tools list 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 with the system, from which the user can 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.

2.3 USABLE DATA EXCHANGE FILES

2.3.1 Project files

A project contains the systems status information at the moment of writing, and the parameters and status of the geometrical model and machining operations. The projects are saved in "*.stc" files. The project file contains all the necessary information for generating the tool movement and does not require the (separate) imported models files.

The calculated trajectories are saved in a separate 'linked' file of machining commands. The linked file has the same name as the project file, and extension *.MCD. The linked file is used for generation of NC programs by the postprocessor or the invariant postprocessor. The presence of the file itself is not obligatory. On the other hand, lack of the 'linked' file of machining commands, does not violate the project functionality. However, having loaded the project file, if there is no file linked to it, all operations will be rendered unexecuted, and in order to obtain all tool movement trajectories, the machining operations must be calculated again. The linked file of machining commands is created automatically when saving the project.

The menu item File of the main menu is designed for projects management. The functions for loading, saving and creating projects are duplicated as buttons on the main panel . Projects can also be opened by using the Drag and drop function, i.e. simply by dropping the project file (*.stc) onto the shortcut of the application.

2.3.2 Importable files

In **SprutCAM** 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), DXF (*.dxf), STL (*.stl), VRML (*.vrl), PostScript (*.ps, *.eps), or directly from Rhinoceros (*.3dm), Parasolid (x_t; xmt_txt) and SPRUT (*.sgm). The number of importable files depends on the system configuration and can be changed optionally.

The geometrical model of a machined part, workpiece, machining equipment can be prepared in any CAD/modeling system and imported into **SprutCAM** using any of the supported formats. **SprutCAM** 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** 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.

2.3.3 Postprocessor tuning files

Postprocessor tuning files to specific CNC system have the SPP (*.spp) extension. There is a unique tuning file for each CNC system. Tuning file contains all data regarding the CNC unit and compiled subprograms of machining programs processing. The file is required by the postprocessor for transformation of machining commands from the linked project file (*.mcd) into an NC program for this control. The files are used by the postprocessor and the invariant postprocessor and are created/altered by using the invariant postprocessor.

Editable text versions of the postprocessor for a CNC system have the same name as the corresponding postprocessor tuning file with an INP (*.inp) extension. The file contains the original texts/subprograms for the machining commands processing, and is used for making changes and generation of postprocessor tuning files. They are created and used by the invariant postprocessor.

2.3.4 NC program files

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

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 (*.spp). 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** is installed, to the CNC unit can be performed by any available method.

2.3.5 DXF export

The geometrical objects of **SprutCAM** 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**. The contours that were created in the built-in modeler are saved in the XY-plane.

3 GEOMETRICAL MODEL PREPARATION

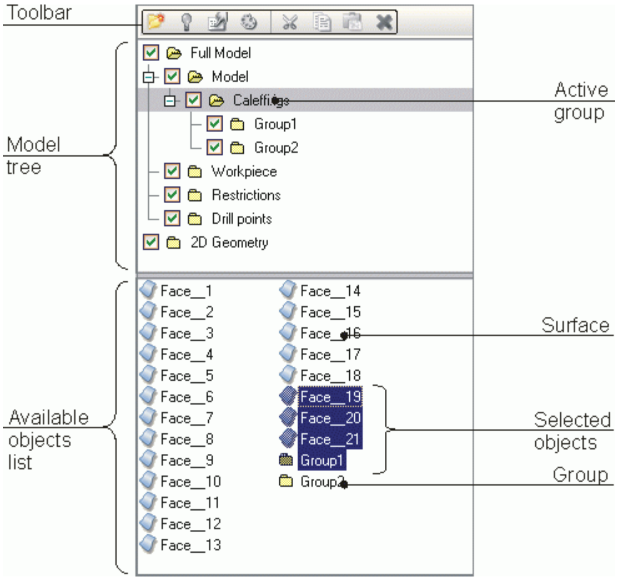
To choose the geometrical model preparation mode click onto the <3D Model> tab in the system's main window.

In the "3D Model" mode one can:

- import geometrical data (CAD) files;
- correct the structure of the geometrical model;
- perform spatial transformation of the geometrical objects;
- generate new elements from existing ones;
- 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. Activation of different functions can be performed 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 3D Model tab.

3.1 GEOMETRICAL MODEL STRUCTURE



A geometrical model in **SprutCAM** has a hierarchical structure. Working with the structure of the geometrical model is similar to the structure used in the Windows OS.

Objects that are geometrical (i.e. surface, mesh, curve, point) can be inside the structural objects (groups). These groups can contain any objects inside themselves, both geometrical objects, and groups. As in the Windows OS, files of different types are located inside folders, and these folders can contain both files and other folders. That is, groups with different 'levels' create the geometrical model structure.

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!*

The structure of the geometrical model is displayed in the geometrical model structure window, most functions for the correction of the model structure are found here as well. When creating a new project, the main group of the geometrical model is generated automatically. The model, is contained inside these predefined groups 2D Geometry, Model, Workpiece, Restrictions, Drill points.

In the 2D Geometry group, one can access the geometrical objects (contours and points), which were built in the 2D geometry mode. All 2D objects are grouped into folders with names of contours, where they were built. Objects in the 2D Geometry folder are automatically renewed when making any changes, whether it is construction, correction or deletion of geometrical elements or curves. Thus objects, which are located in the defined folder cannot be deleted, or transformed in the geometrical model preparation or technology creation modes. Only the visual properties can be altered. However, the objects from 2D Geometry group can be copied via the exchange buffer and inserted into another group. After that, the copied objects lose their connection with the 2D constructions environment and can be modified or deleted as any other geometrical object.

When creating any new operation the Model group is, by default, the model to be machined. Therefore, it is recommended to import and transfer into the model group those geometrical objects, which will be machined by most of the operations (i.e. the surface/solid model).

If when creating a machining operation the Workpiece group is not empty, then this group will be used by default as a workpiece. Otherwise - the workpiece will be chosen automatically, accordingly to the type of the operation and rules dictated by default. The freeform 'workpiece' model should be placed in the Workpiece group. If using simple-form workpieces (box etc.), this folder should be left empty.

The Restrictions group, when creating a new machining operation, is accepted as the restrictions of the operation. When necessary, it is recommended to put in there models of clamps, supports, vices and other machining equipment, and contours of the restricted areas, elements of the model which do not require machining in the present operation.

3.1.1 Geometrical objects types

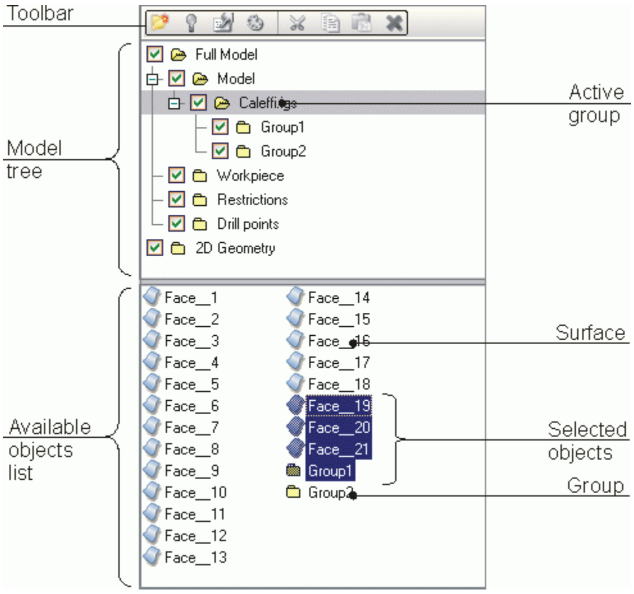
Faces, meshes, curves and points

Every geometrical object: Face, mesh, curve or point - is a whole element and cannot be divided into smaller parts. These are the objects that affect the path of the tool movement trajectory in machining operations, which are formed from them (the machined model, workpiece, restrictions etc). For more convenience, geometrical objects can be joined into groups.

During the import process, a unique name, containing the object type and serial number is given to every object. The name can be changed by the user in the geometrical model structure window.

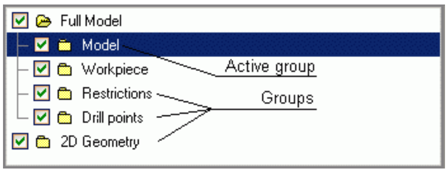
Group

Geometrical objects (face, mesh, curve or point), can be located inside the structure objects - groups. For their part, groups can contain any objects, both objects with geometrical nature and groups. Similar to Windows, files of different types are located inside folders, and the folders, consequently, can contain both files and other folders. Thus, groups with different enclosure levels form the geometrical model structure. The model is designed tree-like, with nodes, which can be either separate geometrical objects or groups of elements.



All modification operations on the selected objects can be performed on the separate geometrical objects or groups

Active group

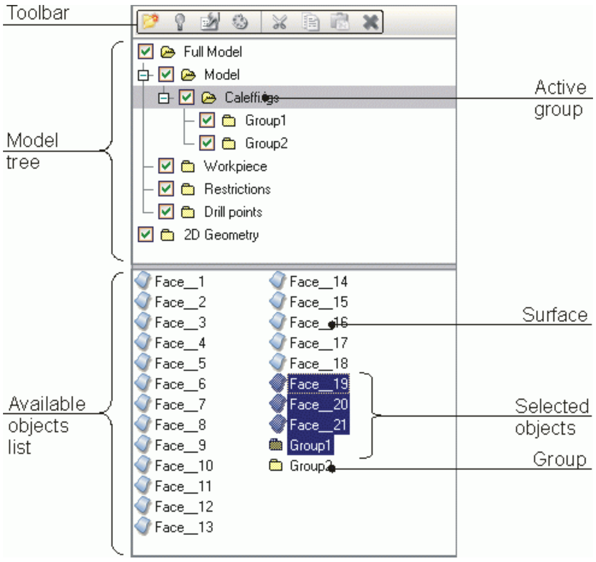


Only one group can be active at one time. Work (import, selection, transformations and etc.) is possible only with the objects located in the currently active group. The group, which is active is treated as a single object, and is considered indivisible. In order to work separately with elements located 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 group is shown on the main panel and in the model structure window. In the model structure window one can also find the list of geometrical and structural elements, inside the active group. 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 drop-down list in the main panel.

3.1.2 Geometrical model structure window

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



On the tool panel, you will find the following buttons:

 – New group creation. The group will be created inside the active one.

 – 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.

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 (in blue). When selecting an inactive group using the mouse or keyboard, the group becomes active and the list of available objects changes accordingly.

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. Double clicking on the group in the list of available objects, the selected group becomes active, and the list of available objects changes accordingly.

3.1.3 Object selection

In the system, one can select only the objects which are in the currently active group. Further, the group, which is located inside the active group, is perceived as a single object, and thus considered indivisible. In order to refer to elements lying inside a subgroup separately, one should open it i.e. activate. These rules are identical to the rules used when working with a file system. Working with a file or a folder is possible only after "entering" the folder containing it.

The currently active group is displayed in the main panel and in the model structure window. Also displayed in the model structure window, is the list of geometrical and structural elements in the Active Group. Selection of these objects can be performed both, from the list and the graphic window. Change of the active group is performed by defining the corresponding link in the model tree window or by selecting an option in the pop-up list in the main panel (right mouse click).

Selected objects can be highlighted in the graphic window by color. Selected objects are also highlighted by color in the geometrical model structure window.

All object modification operations are only performed on selected objects.

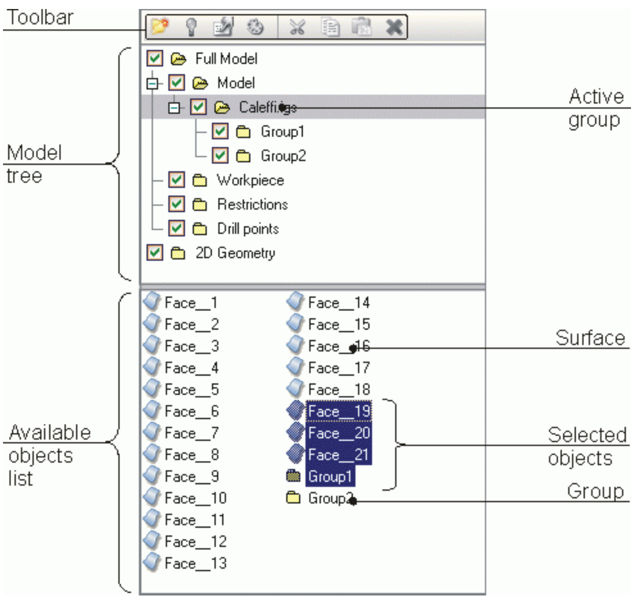
To select a part of the model or separate elements it is necessary to activate the group, which includes these geometrical objects, or groups. Then select the desired objects from the list of available objects in the graphic window. If selecting in the graphic window, the Select cursor  on the main panel must be switched on 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. For selection using the rectangular area 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 users to define objects singly. To select multiple objects, press and hold the Ctrl key. Doing this, the previously selected objects will remain selected, and the newly selected object will be selected. Note that, if an object that is already selected is selected again, it then becomes de-selected. Alternatively, by holding the Shift button, the selected object(s) will always be selected.

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.

3.1.4 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.

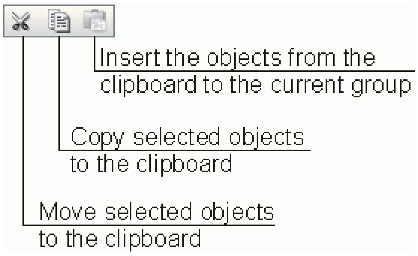
Predefined groups (Full model, 2D Geometry, Model, Workpiece, Restrictions, Drill points), and all objects inside 2D geometry group cannot be deleted or cut into the exchange buffer. The contents of the groups "2D Geometry" and "Drill points" cannot be modified using the exchange buffer. However copying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

Creating a new group

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

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).



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

The functions are duplicated in the main menu (Edit item) and 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.

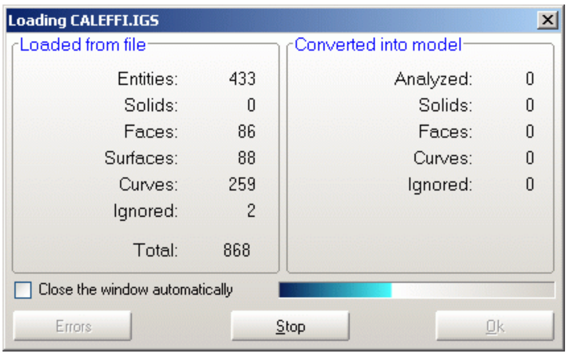
3.2 GEOMETRICAL OBJECTS IMPORT

Import of models from external CAD files is performed by clicking the 

button or by simply dragging the model file into the application's window.

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**.

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, accordingly, the process indicator runs twice. In the first stage, the system reads the file and analyses 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, limited surfaces (Faces) consist of an unlimited surface and several restricting curves. When reading the file, the limited surface itself 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 limited surface (Face).

Clicking the <Cancel> button during import will stop the loading process.

During the import process, the system analyses 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.*

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.

3.2.1 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.

Requirements for IGES-file

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

Types of importable objects

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

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

- Circular Arc type 100;
- Composite Curve type 102;
- Conic Arc type 104;
- Copious Data type 106;
- Plane type 108;

- Line type 110;
- Parametric Spline Curve type 112;
- Parametric Spline Surface type 114;
- Point type 116;
- Ruled Surface type 118;
- Surface of Revolution type 120;
- Tabulated Cylinder type 122;
- Transformation Matrix type 124;
- NURBS-curve Rational B-Spline Curve type 126;
- NURBS-surface Rational B-Spline Surface type 128;
- Offset Curve type 130;
- Offset Surface type 140;
- Boundary type 141;
- Curve on a Parametric Surface type 142;
- Bounded Surface type 143;
- Trimmed Surface type 144;

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

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

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

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

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

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

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

- If in the IGES export tuning menu there is an accuracy control, it should be set relatively high. This will allow the system to link surfaces forming the model more accurately, and consequently more accurately process it. It is not recommended to export the model with accuracy less than the required machining tolerance.

- It is advised to substitute objects types that cannot be imported by the system into those that can. For example, the draughtsman, working in the modeling program, may use a solid-sphere entity. Importing this into the system as an IGES-entity Sphere type 158 is impossible, but it is Ok 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.

3.2.2 Importing objects from DXF files

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

***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);

3.2.3 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** 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.*

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;

3.2.4 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).*

3.2.5 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.

Types of importable objects

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

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 is obligatory (#VRML V2.0 utf8), otherwise the file will be regarded as in an incorrect format.
- A packed file must first be unpacked.

3.2.6 Importing objects from 3dm files (Rhinoceros)

SprutCAM performs direct reading of project files from the Rhinoceros CAD system versions 1.0 - 3.0 (*.3dm).

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

3.2.7 Importing objects from SGM files (SPRUT)

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

3.2.8 Importing objects from Parasolid(TM) files

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

- Wireframe;
- Surface;
- solid bodies;

The data transmission format of Parasolid(TM) 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; xmt_txt; x_b; bmt_bin.

SprutCAM v.4.0 supports the Parasolid(TM) data transmission format up to 15.0 versions.

***Note**: Supports Parasolid(TM) format (BETA version).*

***Note**: **SprutCAM** supports only the text format of Parasolid(TM) XT. (Files with extensions x_t or xmt_txt)*

***Note**: **SprutCAM** currently does not support file partitions.*

***Note**: **SprutCAM** currently does not support some of types of geometry objects of Parasolid(TM) such as BLENDED_EDGE.*

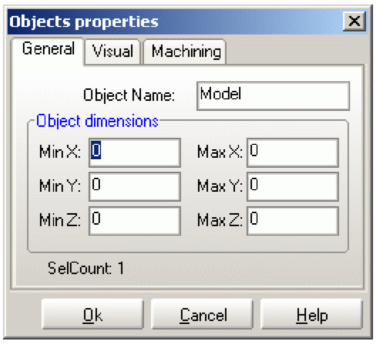
3.3 EDITING GEOMETRICAL MODE

For details regarding the structure of a geometrical (CAD) model, object types and object selection options, see the explanations in the chapter Geometrical model structure.

3.3.1 Geometrical objects properties

The Properties window is opened by pressing the button on the 3D model tab or from the context (right mouse click) 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 three tabsheets

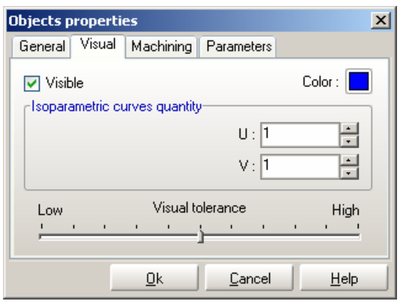
General Tab



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.

- <**Name**> – name of the selected object. If several names are selected, then the field will be empty.
- <**Box**> – overall dimensions of the selected object.

Visual Tab



Access to the visual properties of objects is duplicated on the visual properties tab: visibility and color. It is also possible to assign the number of isoparametric lines and the object's visual tolerance in the graphic window.

- <**Visible**> – if unchecked, then the selected object will not be displayed.
- <**Color**> – allows changing the color of the selected object.
- <**Isoparametric curves by U**> и <Isoparametric **curves by V**> – 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 these 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.

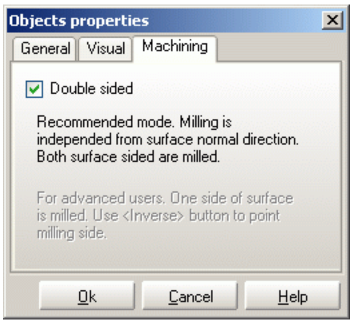
- <**Visual tolerance**> – allows the user to set the visual quality of 3D objects, or to find a compromise between satisfactory visualization quality and computer speed. The tolerance is adjusted using the slider bar control. Tolerance in this case is the maximum approximated deviation of sections used when drawing the curves on 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.

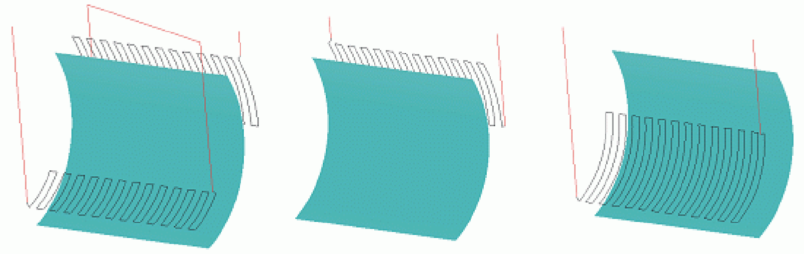
The altered visual properties will only be applied when the window is closed using the <Ok> button.

Machining Tab



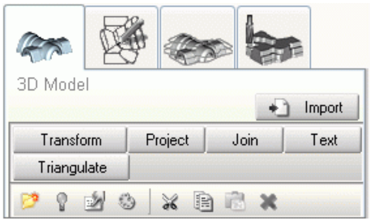
Double sided – allows the user to define the surface type. When loading a geometrical model all surfaces are set as "double sided". This means that surface machining will be performed independently from the normal vector direction - from both sides. Thus, the side of the surface being machined is defined 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)

3.3.2 Changing visual properties



Visual properties of objects can be also changed in the properties window.

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.

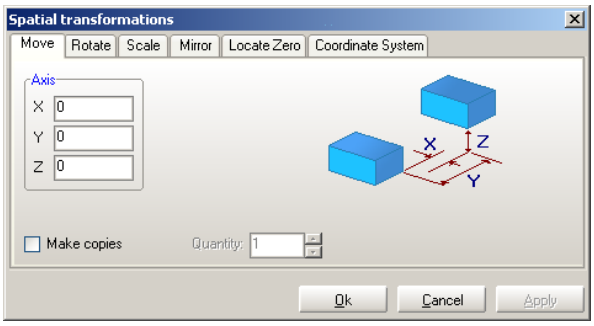
3.3.3 Delete

The button deletes the selected objects. If the objects to be deleted are used in a machining operation, then confirmation will be requested.

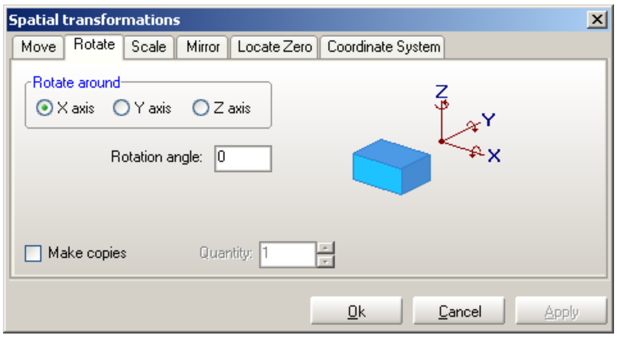
3.3.4 Spatial transformations

A wider range of transformation methods of selected geometrical objects is available in the Spatial transformations window. The window opens when the Transform button is pressed.

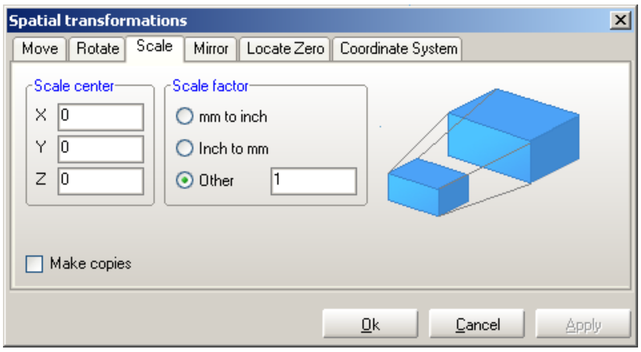
- On the "**Move**" page, the user can define the parallel transition of an object. If there is no checkmark in the field "Make copies", then the selected object will be transferred by the defined distance along each of respective axes. If the checkmark in that field is set on, then the selected object 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.



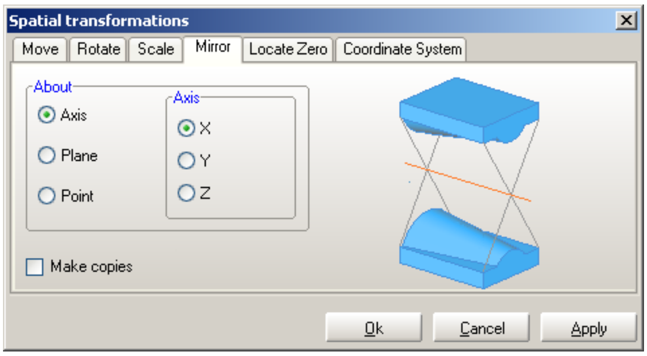
• On the "**Rotate**" page, the user can rotate selected objects round the selected axis to the defined angle. The angle is assigned in degrees. Working with copies is incremental, that is, every subsequent copy is obtained by rotation of the previous one around the defined axis to the defined angle.



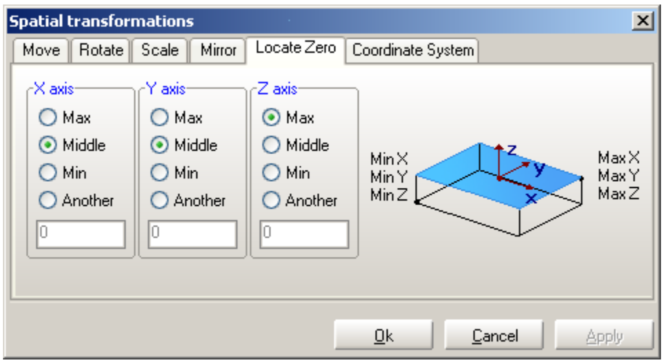
• On the "**Scale**" page, the user can enlarge or decrease selected objects. In the field "Center of scaling", the coordinates of the center point of scaling are defined. When assigning a coefficient of scaling of more than one, then objects will be enlarged. If a coefficient of scaling is less than one, then objects decrease in size accordingly.



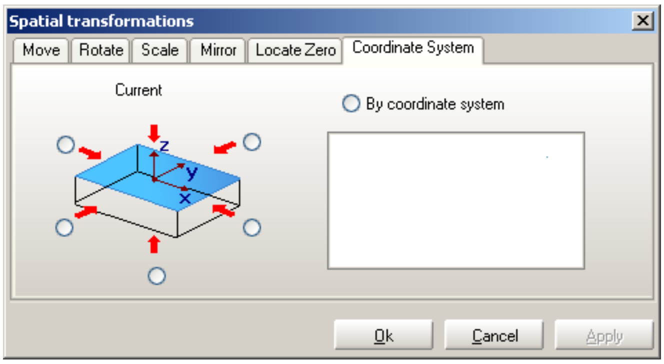
• On the "**Mirror**" page the user can obtain an object symmetrical to the selected one relative to an axis, plane or point. If a checkmark is set in the field "Make copies" objects will be copied.



- On the "**Locate Zero**" page the user can perform a parallel transition of an object according to its spatial dimensions.

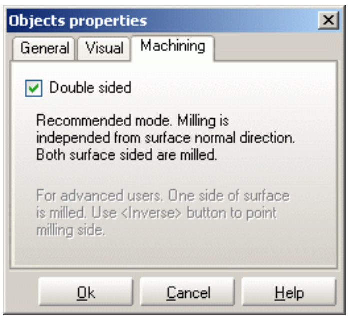


- On the "**Coordinate System**" page the user can turn an object, so that the selected edges are on the top.



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.

3.3.5 Inversion



For "non double-sided" surfaces, 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" surfaces, the direction of the normal is unimportant. Use of the inversion function does not affect the tool movement trajectory.

When loading a geometrical model all surfaces, by default are set double-sided.

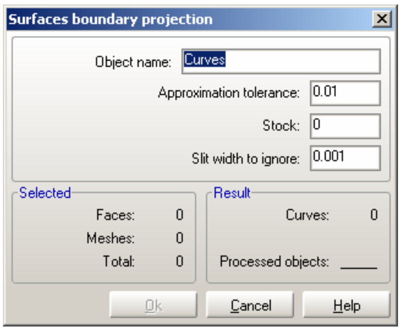
Alteration of the machining type of a surface ("double-sided"/"not double-sided") is performed in the surface properties window.

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

When setting the side to be machined, it is necessary to switch on the "show normals" mode with the objects selected (System settings window, 3D model tab).

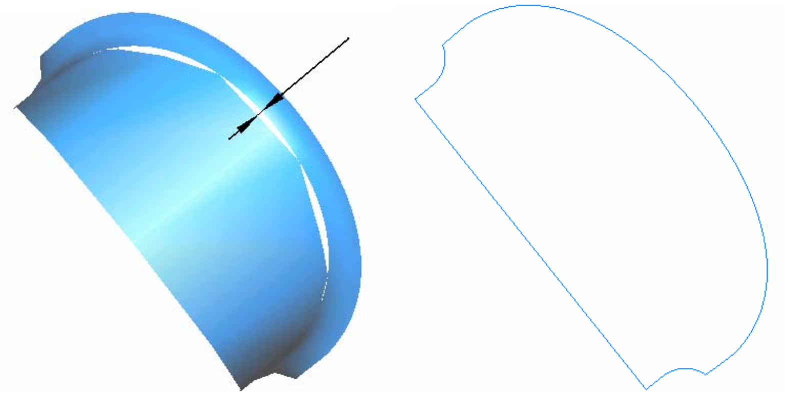
3.3.6 Outer borders projection

The Project button opens the Surface border projection window. The function is used for the construction of outer enveloping projections of the selected surfaces and meshes onto the XY 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 (folder) 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.

- <**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).
- <**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.

- <**Surfaces**> – number of surfaces selected.
- <**Meshes**> – number of meshes 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.
- <**Objects processed**> – total number of processed objects.

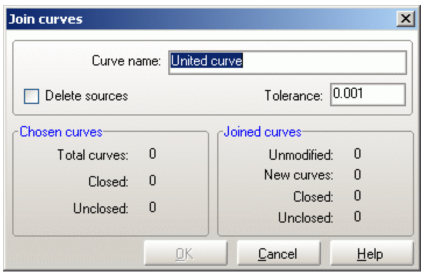
When changing any projection parameters, the values in the Obtained panel will automatically be recalculated.

If the results of the defined parameters are Ok, 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.

3.3.7 Curves joining

The 

button opens the Curves joining 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 joining. The curve joining function allows users to obtain a joined curve by linking neighboring curve sections.

- <**Curve name**> – name of the new curve. 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).

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.

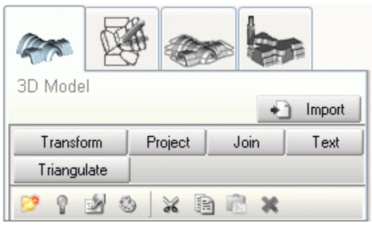
- <**Unmodified**> – number of curves left without modification.
- <**New curves**> – total number of new curves created.
- <**Closed**> – number of obtained closed curves.
- <**Unclosed**> – number of obtained unclosed curves.

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.

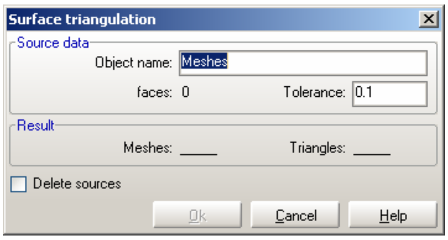
3.3.8 Surface triangulation



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.



- <**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.
- <**Delete source**> – a tick in this field means that when finishing the triangulation operation, all source objects will be deleted.
- <**Tolerance**> – maximum deviation of the resulting surface from the source one. By altering the tolerance value, one can achieve the desired surface tolerance.

The panel Selected shows the type and number of objects selected for triangulation.

- <**Faces**> – number of selected surfaces.

The panel Result 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.

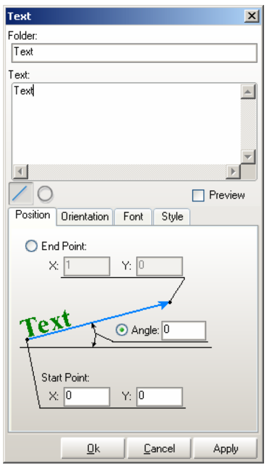
When changing the triangulation tolerance, the field values in the Result panel will automatically be recalculated.

If the triangulation results are Ok, 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.

3.3.9 Creating text

The text creation window can be opened either by pressing the  button on the 3D model panel, or by using the pop up menu in the graphic window or on the Model page.

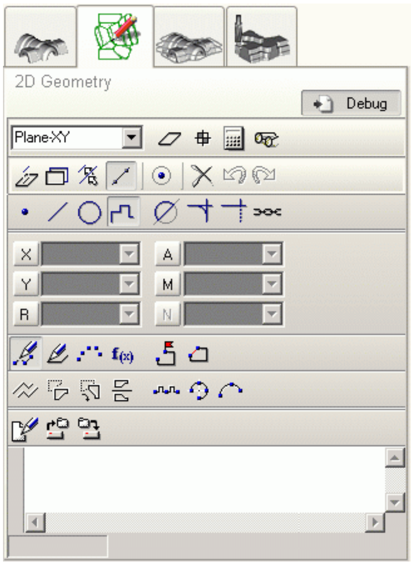


Text can be typed either along a line (default), or along a circle/arc. To change text font etc., press the Font button.

To preview the results press the « Preview » button. If all parameters assigned were correct, the text will be displayed in the graphic window. If required, the text parameters can be corrected. The above sequence of actions can be repeated until the desired result has been obtained.

Having assigned the text parameters, to create the text contours presses the « Ok »button. At folder with the name defined in the « Folder » field will be created in the model tree.

4 INTEGRATED ENVIRONMENT OF 2D GEOMETRY CONSTRUCTION



To enter the 2D geometrical constructions mode select the  in the main system window. The mode is designed for construction of auxiliary 2D geometrical objects; these can be located in different planes.

The constructed 2D contours can be used when assigning machining operation parameters (defining workpiece geometry, machining/restricting areas, leading curves in drive operations, the model for machining in engraving/curve machining operations). Points may be defined as the centers of holes to be machined or the coordinates of drill points.

To construct 2D geometrical objects the user can use an infinite number of planes. Planes can be created based on parts of a 3D model, view vector or an already created plane. Use of coincident planes is also allowed. Parametric links are automatically created during construction, between objects lying in the same plane. Objects lying in different planes are treated as independent.

All 2D geometrical objects in the system have two representations: graphical and lingual. There is an explicit connection between them. This means, that for every object displayed in the graphic window, there is a corresponding (text) string in the geometrical processor language and vise versa.

When defining an object interactively, the (text) string, which corresponds to it, will be created automatically. When deleting an object, the string that defines it, will be deleted as well. Editing the string will cause simultaneous alteration of the drawing and conversely, alteration of the drawing causes alteration of the string.

Information about geometrical objects in a drawing is kept as a program. It is self-sufficient and does not require additional information to restore the drawing. It helps saving geometrical constructions as a text file.

Example:

P12=X(0),Y(0) //Constructing a point in X=0, Y=0 coordinates

L11=P12,A(30) //Constructing a line via the point at an angle of 30 degrees

C11=P12,R(20) //Constructing a circle with the center at the earlier defined point and a radius of 20

There are many different ways to define objects in the geometrical postprocessor language.

A program reflects a sequence of constructions, all links between elements and numeric parameters, defined by the user. Their modification is possible at any time (both interactive and direct text editing), this makes the end-model qualitatively and quantitatively parameterized.

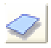
The 2D geometry mode is an interactive interface, which provides access to the functions of the geometrical processor. Use of the interactive interface allows the user to access all of the available functions without knowing the language syntax.

4.1 PLANE MANAGEMENT

All 2D geometrical constructions are performed in the currently active plane only. No construction can be made independently of a plane.

An unlimited number of arbitrarily located planes can be defined within the environment. Creation of a new plane is possible by either using a 3D model's points, by the current view or relative to an earlier defined point. Once created, a plane will contain a unique set of 2D geometrical objects, together with their construction sequence and parametric links. 2D objects created in other planes are considered independent.

Although the main purpose of planes is for the definition of 2D objects in 3D space, it is sometimes advised to use coinciding planes for the construction of geometrically unconnected geometry.

The plane management mode is selected using the  button.

This mode allows following operations on planes:

- current plane selection;
- parallel plane displacement along its normal;
- creation of a new plane by the current view vector;
- creation of a new plane by three points of a geometrical model;
- projection of points of a 3D model onto a plane;
- plane deletion.

4.1.1 Setting current plane

Every plane in 2D geometry has its own location in the global coordinate system, and can contain a multitude of 2D geometrical objects, together with a construction sequence and parametric links. Every plane has a unique name. The names of planes are displayed in the:

1. drop-down list.

7. geometrical model structure window of the 2D geometry environment.

When first started, one plane with the name "Plane XY" is created in the environment. That plane coincides with the plane XY of the global coordinate system; it also uses the same datum (X/Y/Z zero) point. Newly created planes are named "Plane” when created, where the serial number of the newly created plane is.

Choosing from the drop-down list of the existing planes performs selection of the required plane. Upon changing the plane, the corresponding view 'vector' will be made active in the graphic window, and all 2D objects previously created in that plane will be loaded in it. All subsequent constructions and changes will be performed in the active plane until the user changes the plane or quits the 2D construction mode.

To change the name of the current plane, simply edit the name of the plane on the tool panel. Only the name of the plane will be changed; its location in space and any 2D objects will be left unmodified.

The Z field shows the parallel Z distance from the global datum to the current plane. This value can be modified, which will move the plane parallel along the Z-axis. All elements previously defined in that plane will also be moved.

4.1.2 Creating a new plane

Initially the system automatically creates one plane, which coincides with plane XY of the global coordinate system. It is possible to create an unlimited number of planes (including coinciding ones), defined by the current view vector, by points from a 3D model or by copying a previously created plane.

To open the plane control panel presses the button. When the button is pressed, a new plane, which is parallel to the screen plane in the current view, will be created.

The button provides creation of a plane by using three points of a 3D model. The points used are ends of sections of the 3D wire model. To create a plane, pick three points by mouse. The first point will be the datum point of the coordinate system for the newly created plane, the second will define the positive direction of the X-axis, and the third one will define the Y-axis direction.

A point on the plane can be defined as a projection of the end of a 3D section of the wire model. Having created the projected points from the 3D model, a new plane can be created using them.

4.1.3 Deleting a plane

The current plane, together with the elements defined within it, can be deleted by pressing the button.

***Note** : Base XY plane can not be deleted.*

***Note**: In to be deleted plane, there may be elements, which are used in machining operations. When calculating such operations it will be necessary to either delete any links to the non-existent elements or create new geometrical elements.*

4.2 DEFINING GEOMETRICAL ELEMENTS

The construction of new 2D geometrical objects is performed in the currently active plane. No constructions can be made outside of any plane.

The plane contains a set of 2D geometrical elements, interconnected by a sequence of constructions and parametric links. When constructing elements, there will always be parametric links, which defines the construction method automatically made between the elements. This allows the changing of geometrical parameters of one element to automatically reconstruct all linked elements.

For example, if a point has been defined at an intersection of two lines, the alteration of the angle of one of the lines will automatically cause alteration of the point's coordinates. That will also consequently lead to the alteration of all geometrical elements using that point.

Geometrical objects, lying in different planes are considered independent.

Every plane has fixed location in the global coordinate system, which allows the location of 2D objects in 3D space using any appropriate method.

All 2D geometrical objects are represented in two ways: graphically and as a text description. There is an explicit connection between them. This means that for every object displayed in the graphic window there exists a corresponding (text) string in the program of the geometrical processor and vise versa.

Consequently, creation of objects is possible both interactively and conversationally. In both cases, the same geometrical objects will be created by any of the available defining methods. The user can use their preferred method to create, or to edit any element(s). When creating the object interactively, the program string that corresponds to it will be created automatically. Moreover, when using the conversational method, the object will be immediately displayed in the graphic window. Editing the program will cause simultaneous alteration of the drawing and conversely, interactive modification of the drawing leads to alteration of the program.

For interactive creation of an object, it is necessary to set its type and indicate the elements and parameters that define it. Because of interactive actions, the string corresponding to the conversational construction of the object will be created automatically. The interactive method of object assignment is most commonly used.

To define an object using the conversational method, it is necessary to add a string, which defines the new object, in the debug window. The conversational method is preferred when constructing a model that requires additional parametric links between the objects to be created.

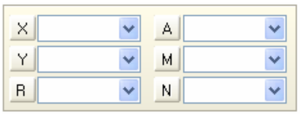
4.2.1 Object construction in interactive mode

To create a new element, its type should first be selected (point, line, circle, contour) and then the necessary data to create it. If there are several objects of the selected type that can be created by the users input(s), then the system will show all the possible variants and will wait for the required object to be selecting. The selected new object will be created.

For example, in order to construct a crossing point for two lines, the user should select on the panel the type of object Point and left mouse click on the first line in the graphic window, and then repeat this with the second line. The point will be constructed automatically where the two non parallel lines cross, and therefore the system will not request any additional information.

As another example, to construct a circle, that passes through a point and is tangent to two lines, it is necessary to select the type of element being defined Circle and in any order select in the graphic window a point and two lines. Then select from the offered variants the required circle by left mouse clicking on it. After this the defined circle will be constructed and will show up in the elements list. During construction of this circle a point and two lines were used. Alteration of the geometrical parameters of any of these elements will lead to automatic alteration of the circle.

When required, the numeric data for the element being defined should be entered in the appropriate fields in the panel.



X - Coordinate X;

Y - Coordinate Y;

A - Angle in decimal degrees;

R - Circle radius;

M - Distance, equidistant value, approximation tolerance when constructing splines;

N - Number of repetitions of operation (e.g. contour definition by multiple transitions with joining).

When defining objects, the system automatically alters the allowed types of parameters, which can be used for the object definition. The required values can generally be entered in any order.

Parameter values can be entered directly or, as a parameter of another object or a collection of parameters of other objects by means of the geometrical calculator. The entered numeric value for a parameter will be set. When entering values via the Geometrical calculator, an additional parametric link is created that automatically tracks all alterations of the base elements. Input of numeric values can be performed directly in the parameter values field. Activation of the Geometrical calculator for the input of additional parametric values can be made by pressing the button located next to the input field (X, Y, A, R, M or N).

For example, the radius of a circle can be assigned directly by entering a numeric value in the R field or as the distance from a point to a line using the geometrical calculator. In the first case, the circle radius will not be changed upon altering the other geometrical elements, and in the second, upon changing parameters of the line or the point, the circle radius will be automatically calculated as the current distance between the line and the point.

The  button is designed for the selection of separate modes for point or circle definition. If the button is not pressed, this means that the work being performed using the circumference of the circle, if pressed - with the circle center. The button status affects the definition modes as follows:

Defining point.

- Button is not pressed and a circle has been selected. It is presumed that the point being created lies on the selected circle.
- Button is pressed and a circle has been selected. It is presumed that the point being created is at the center of the selected circle.

Defining circle.

- Button is not pressed and a point has been selected. It is presumed that the circle being created passes through the selected point.
- Button is pressed and a point has been selected. It is presumed that the selected point is the center of the circle being created.
- Button is not pressed and a circle has been selected. The circle being created touches the selected circle.
- Button is pressed and a point has been selected. It is presumed that the center of the circle being created lies on the selected circle.

Once all the required data is entered, the new object will be created. If several variants of the new object are possible, then the system offers the user a choice of one or more variants.

The object definition process can be ended at any time by double-clicking the left mouse button within the work area. The object currently being defined will be created (if possible) in the current position.

Further modification of the finished constructions and viewing of the original text definitions is possible in the Debug window.

The button  is intended for changing the visualization modes. If this button is pressed only contours are displayed. When this button is unpressed then all types of geometric objects are displayed. For short viewing all geometric objects, it is possible to use the key.

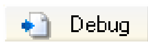
This panel  contains following functions:

 - Break the current construction;

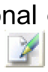
 - Undo

 - Redo

4.2.2 Objects definition in conversational mode

Definition of objects by the conversational method can be performed in the debug window, which opens when the 

button is pressed.

To input a conversational definition the user should select the editing mode by pressing the  button in the debug window. In the editing mode, any alteration of the program text in the language of the geometrical processor is possible. This includes input of a new object definition into the program, modification of the order of elements construction, and editing and deletion of definitions.

If definition of a geometrical element is performed that use parameters of previously constructed objects, and these objects are subsequently modified, the selected element will be changed automatically. If the conversational definition does not comply with the language of the geometrical processor syntax, or construction of the object is impossible using the defined parameters, then the string with the incorrect definition will be marked by a red point in the area to the left of the string.

Example:

- P11=X0,Y0 !a point defined in coordinate datum
- L11=P11,A30 !a line passing through the earlier defined point at an angle of 30 degrees
- P12=X10,Y20 !another point
- L12=P12,A(30*(sin(45))) !a new line using calculations
- P13=L11,L12 !a new point placed at the intersection of the previously defined lines

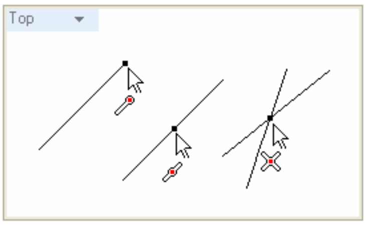
Most object definitions can be assigned interactively, which is normally more convenient. Nevertheless, definition of objects manually is advised when using specific assignment methods or when creating parametric models. The main feature of the conversational element creation method is the ability to use functions and variables when assigning values to numeric parameters.

4.2.3 Object Snaps

Any geometric constructions require the exact determination of the coordinates of the points, which are necessary to construct the geometric element. Exists two ways of the construction - by keyboard input and interactively with the use of a mouse. To indicate the exact position of the points by mouse are used Object Snaps to defined points of already constructed geometric objects.

The modes of snaps are located on the Snap panel.

When the snap is active, the mouse pointer movement near the snap point of the object makes the mouse pointer to jump in this point. Simultaneously near the pointer appears the icon indicating the snap type.

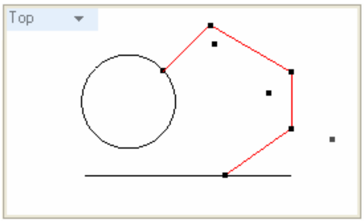


Simultaneously can be active several snap modes.

To change the snap is possible during the process of the construction of the geometric object.

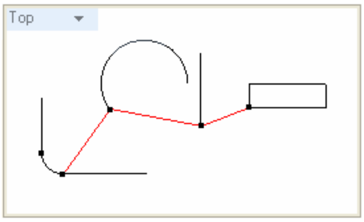
Point object snap

 The Point object snap constraints the mouse pointer to points



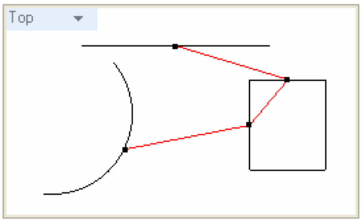
End object snap

 The End object snap constrains the mouse pointer to endpoints of contour, the endpoints of each segment that make up a contour.



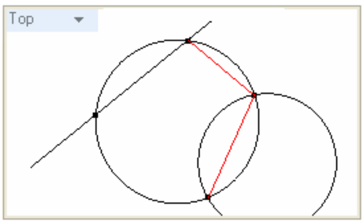
Middle object snap

 The Middle object snap constraints the mouse pointer to middle points of a cut, an arc or a contour segment.



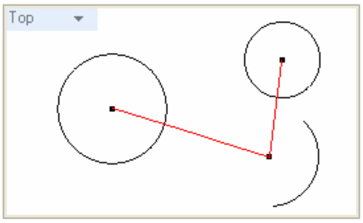
Intersection object snap

 The Intersection object snap constraints the mouse pointer to the intersection of two geometric objects.

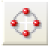


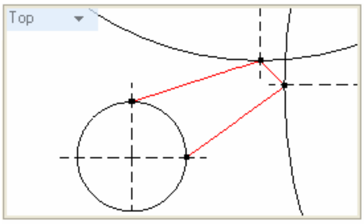
Center object snap

 The Center object snap constraints the mouse pointer to the center of a circle or an arc.



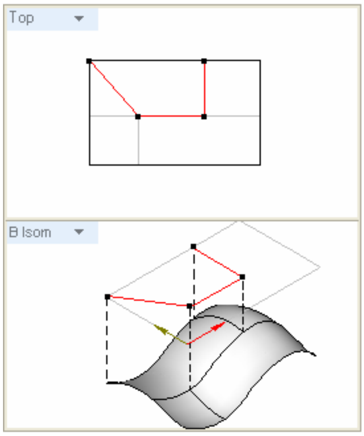
Quadrant object snap

 The Quadrant object snap constraints the mouse pointer to the quadrant of a circle or an arc.



Projection object snap to construction plane

 Projection object snap constraints the mouse pointer to the points that are knot points projection of surface edges of 3D model to construction plane



4.3 GEOMETRICAL CONSTRUCTIONS ON PLANE.

4.3.1 Point setting

For a Point setting, you should press the button, then, in any consequence, by mouse assign elements, which will determine the point, if necessary, enter X, Y and A parameters into proper parameter panel.

By mouse on the screen.

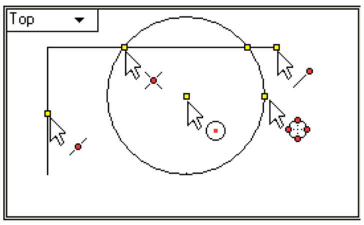
In order to enter a point directly you should click on the graphic window.

By X and Y coordinates.

Position of a point (X and Y coordinates) in the absolute coordinate system is entered on the 2D geometry parameter panel.

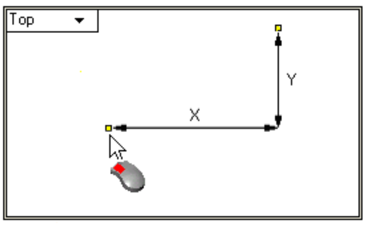
A Point setting on the geometric elements with a snap method.

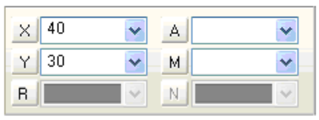
For the point setting put the pointer in the setting zone of the point on geometric elements. After that will appear the required snap image near the pointer, confirm the point setting by the left mouse button



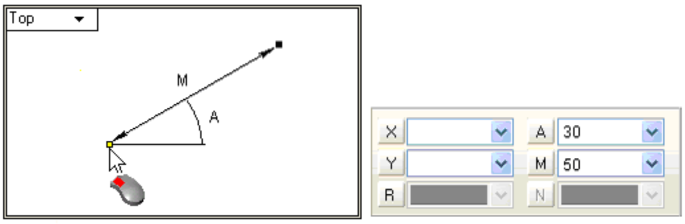
By offset from the defined point

By the pointer indicate the point and on the parameter panel set the bias value along X and Y



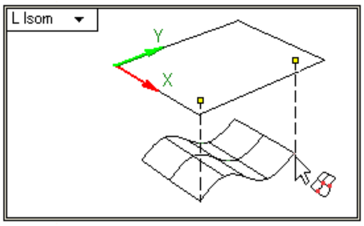


By the pointer indicate the point, on the parameter panel set the angle A, and distance M



By knot points of the model

For the point setting, indicate the pointer in the point setting zone on a model. After that will appear the required snap image near the pointer, confirm the point setting by the left mouse button.



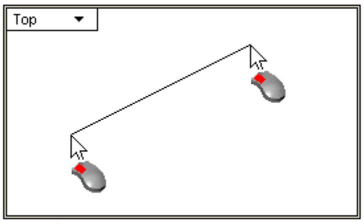
Comment*: The point is constructed as the point projection on a current plane It is possible to construct only projections of knot-points of a model The point construction is possible in any convenient for work views (Top; Isom and so on)*

4.3.2 Line setting

For a Line setting you should press the button. Then, in any consequence, by mouse select elements, which will determine the line. If there are several possible variants of Line setting, the system will reflect these variants and will wait for the right variant choice or, in case of need, enter appropriate parameter in the parameter panel.

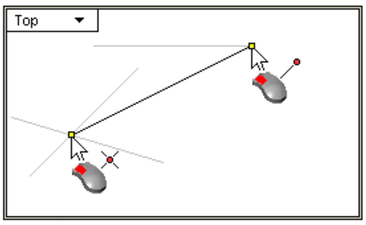
By two mouse clicks

- Indicate by the pointer the start position of the cut
- Indicate by the pointer the end position of the cut



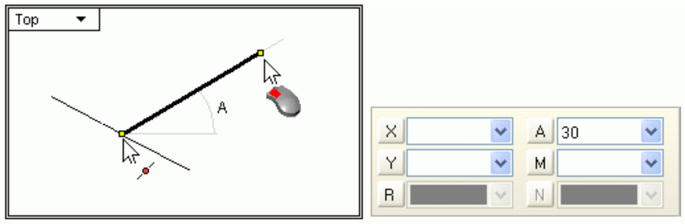
From one snap point to another snap point

- Using the snap system indicate the start point of the cut
- Using the snap system indicate the end point of the cut



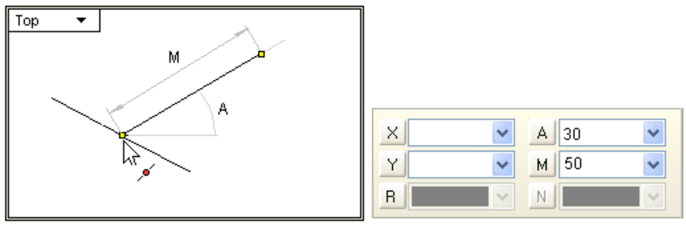
From the snap point at the required angle to position determined by the mouse click

- Using the snap system indicate the start point of the line
- Using the parameter panel set the slope angle of the line
- By the pointer indicate the position of the end point of the line



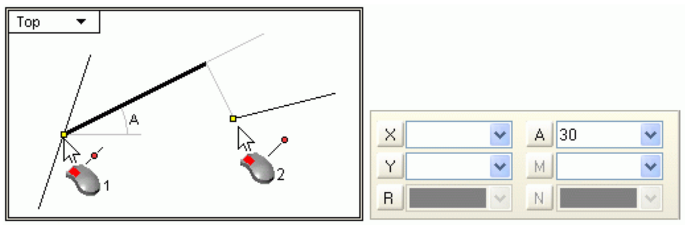
From the snap point at the required angle to the required distance

- Using the snap system indicate the start point of the line
- Using the parameter panel set the slope angle of the line and the distance to the end point of the line



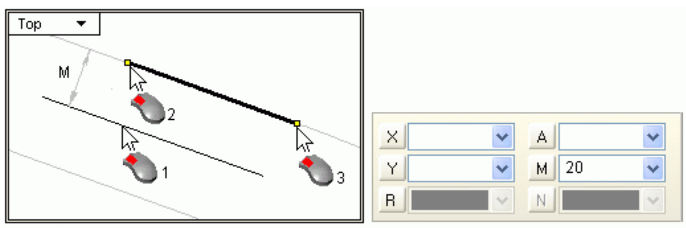
From the snap point under required angle to the snap point

- Using the snap system indicate the start point of the line
- Using the parameter panel set the slope angle of the line
- Using the snap system indicate the end point of the line



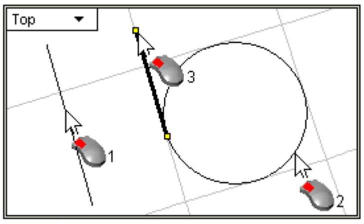
Line construction in parallel to the required line on the required distance

- Indicate the line that is to construct the parallel cut
- Using the parameter panel set the distance between the lines
- By the pointer indicate firstly the start position of the line, secondly the end position of the line



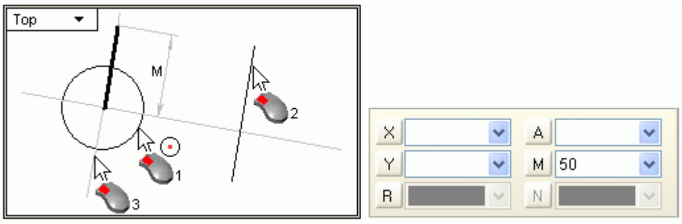
Parallel line construction through the snap point

- Indicate the line that will be paralleled to the required cut
- Indicate the circle
- By the pointer indicate firstly the end point of the line



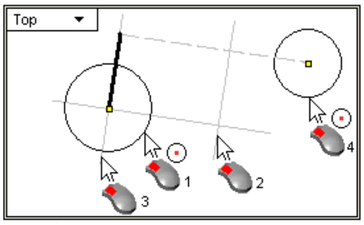
From the snap point of in parallel another line with required distance

- Using the snap system indicate the start position of the line
- Indicate the line that will be paralleled to the required cut
- Indicate the line on which will be the cut
- Using the parameter panel set the length of the cut



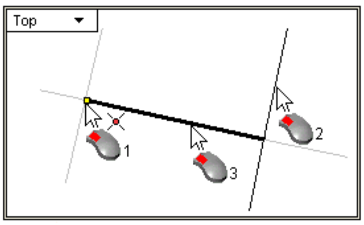
From the snap point in parallel another line to the snap point

- Using the snap system indicate the start position of the line
- Indicate the line parallel that is to construct the cut
- Indicate the line on which will be the cut
- Using the snap system indicate the position on the normal to which will be attached the end of the cut



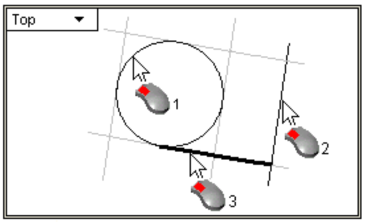
From the snap point in parallel another line

- Using the snap system indicate the start position of the line
- Indicate the line that will be perpendicular to the required cut
- Indicate the line on which will be the cut



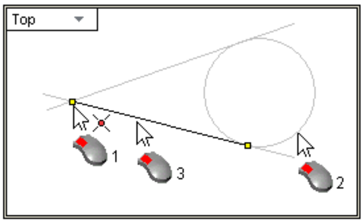
From the snap point tangential curve

- Using the snap system indicate the start position of the line
- Indicate the circle that will be tangential to the required cut
- Indicate the line on which will be the cut



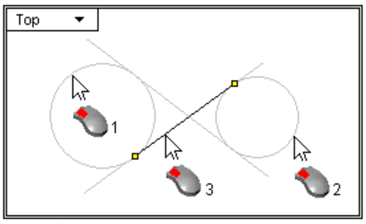
Tangential by curve from snap point.

- Indicate the circle (arc of the circle) that will be tangential to the required cut
- Indicate the line that will be perpendicular to the required cut
- Indicate the line on which will be the cut



Tangential from two curves.

- Indicate the first circle
- Indicate the second circle
- Indicate the line on which will be the cut



4.3.3 Circle setting

For a Circle setting you should press the button then, in any consequence, by mouse assign elements, which will determine the circle. If there are several possible variants of Circle setting, the system will reflect these variants and will wait for the right variant choice or, in case of need, enter appropriate parameter in parameter panel.

Below you can see examples of a Circle setting

Center of the Circle setting

Center of the Circle setting see a point setting

Radius of the Circle setting

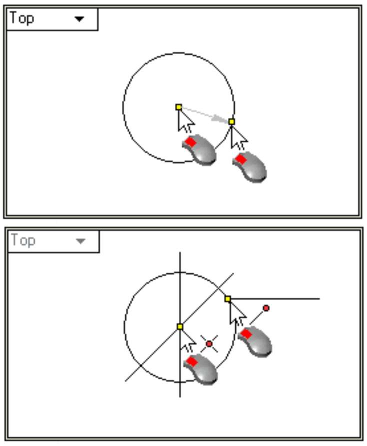
Radius you can set in two ways:

- By setting the radius on parameter panel
- Interactive point setting

Center-Radius

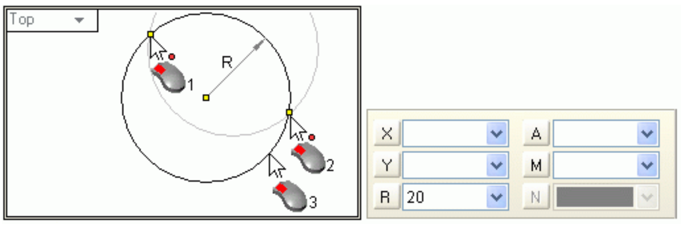
The button  should be active

- By the pointer on a screen or by setting the coordinates on the panel or using the snap system indicate the center of the circle
- By the pointer on a screen or by setting the coordinates on the panel or using the snap system indicate position of the point that will define the radius of the circle



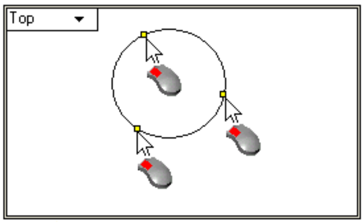
Two points - Radius

- By the pointer on a screen or using the snap system indicate the position of the first point
- By the pointer on a screen or using the snap system indicate the position of the second point
- In R-field on the parameter panel enter a radius value
- Among offered variants select the required circle



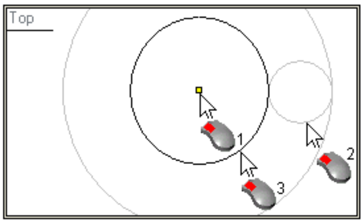
Three points

- By the pointer on a screen or by setting the coordinates on the panel or using the snap system indicate the first point on the circle
- By the pointer on a screen or by setting the coordinates on the panel or using the snap system indicate the second point on the circle
- By the pointer on a screen or by setting the coordinates on the panel or using the snap system indicate the third point on the circle



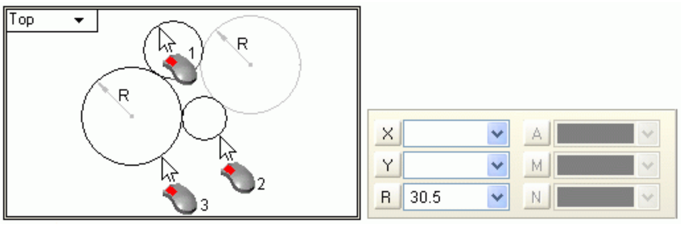
Center - Tangentially

- By the pointer on a screen or by setting the coordinates on the panel or using the snap system indicate the center of the circle
- Indicate the circle
- Among offered variants choose the required circle



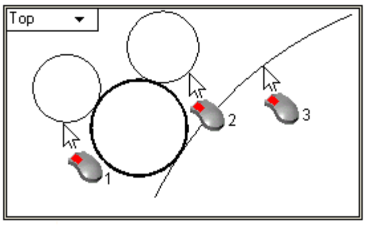
Tangentially two curves - radius

- Indicate the first circle
- Indicate the second circle
- In R-field on the parameter panel enter the radius value
- Among offered variants choose the required circle



Tangentially three curves

- Indicate the first circle
- Indicate the second circle
- Indicate the third circle
- Among offered variants choose the required circle



4.3.4 Trim/Extend

For fulfilling this operation, you should press the button 

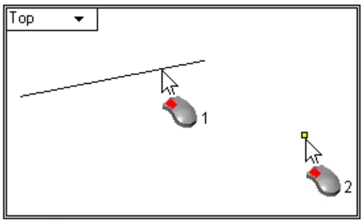
Curve extending (to the mouse position)

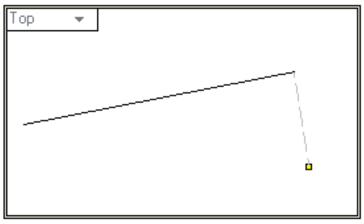
General sequencing:

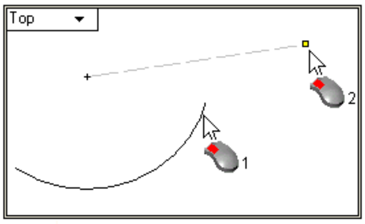
- Select a curve required to extend
- Indicate a point, you intend to extend the chosen curve

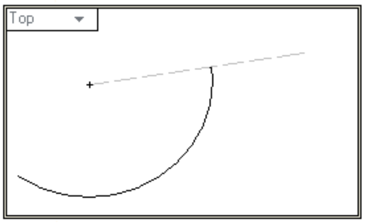
If the point is out of chosen curve, then it is formed a perpendicular to the curve, which is drawn from chosen fixed position of the cursor. This can be done this way:

Click on the screen; earlier prescribed point; and the any snap point, for example, point of intersection (Int), center of a circle (Cen), end of a curve (End) and so on.





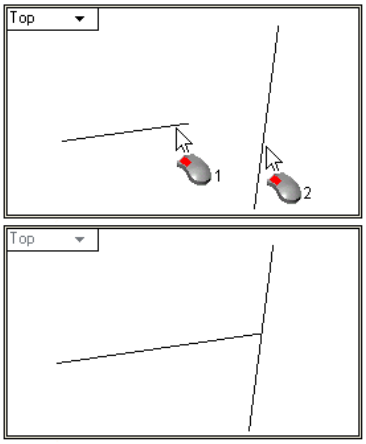




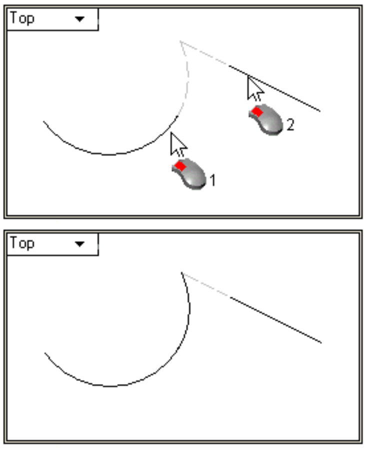
Extending till the crossing with the curve

Sequence of operations:

- Indicate the curve required to extend
- Indicate a curve, you intend to extend the chosen curve



If the curves obviously don't cross, the curve will be extended till the intersection of geometric elements formed by these curves.



4.3.5 Join

For fulfilling this operation, you should press the button 

Conditions of joining:

- Ends of joining curves should converge in one point.
- Curves, united to the outline, can't be used for constructing another outline.

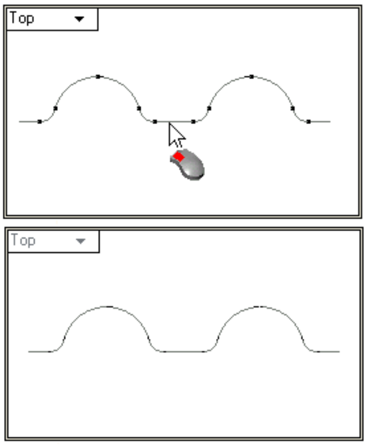
Joining from the indicated element to the first intersection is conducted automatically.

Then you should with the help of cursor indicate an element in the direction of which the joining will be continued.

To complete the joining the curve on the intersection you should press the  button. When you are on the final element, the joining will accomplish automatically.

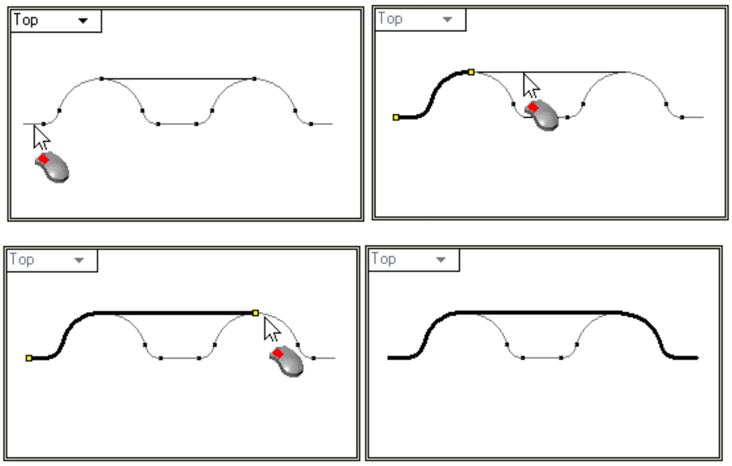
The joining of the group of objects that don't contain branches to the open curve

- Indicate any object from the group of the joined objects



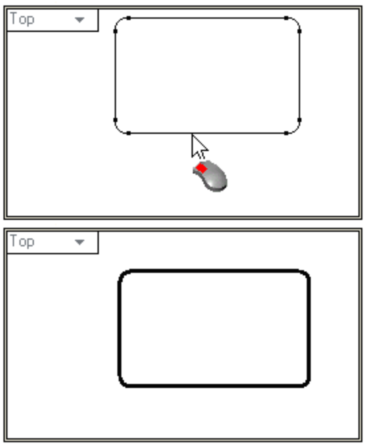
The joining of the group of objects that contain branches to the open curve

- Indicate the object that will be the start object in joining the curve. As a result will automatically occur the joining till the first intersection.
- Indicate the object that will define the direction of further joining



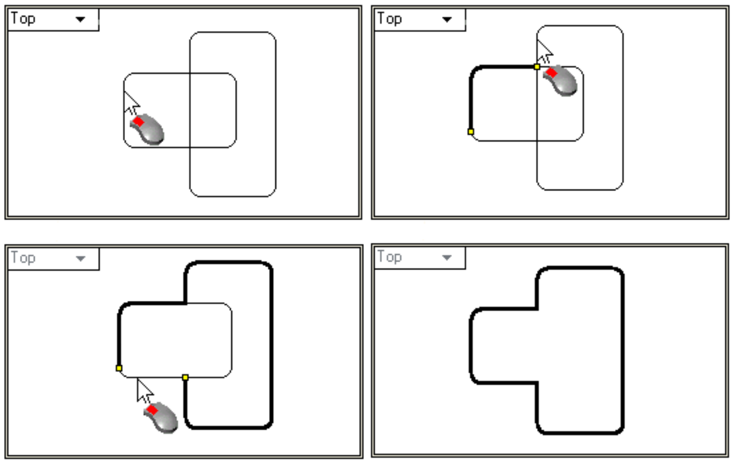
The joining of the group of objects that don't contain branches to the closed loop

- Indicate any object from the group of the joined objects



The joining of the group of objects that contain branches to the closed loop

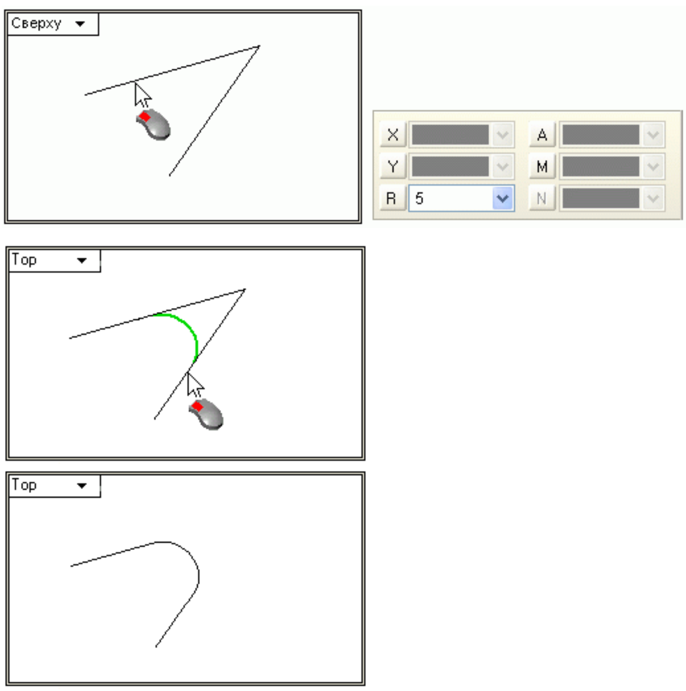
- Indicate the object that will be the start object in joining the curve. As a result will automatically occur the joining till the first intersection.
- Indicate the object that will define the direction of further joining



4.3.6 Fillet

For fulfilling this operation you should press the , by the cursor of the mouse click on the first curve, further on the second curve, then on the parameter panel you should determine the radius of the fillet. Frequently on the intersection of the curves, there are several variants of the fillet, in this case, the system offers to choose essential variant. These variants will be illuminated when you move the cursor of the mouse in curves intersection area. You need only to choose the right variant of fillet.

- Set the radius of the fillet on the parameter panel
- Indicate the first object
- Indicate the second object

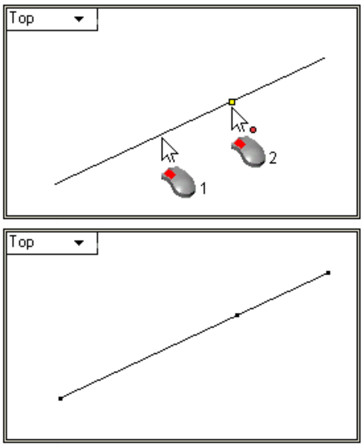


4.3.7 Split

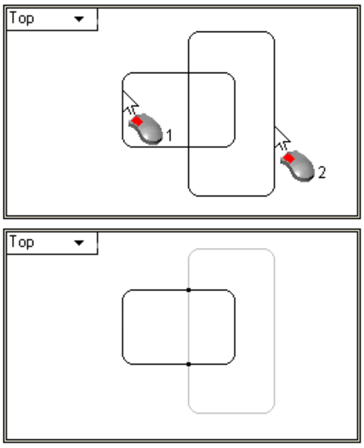
For fulfilling this operation you should press the button, by the cursor of the mouse click on the curve you have to split, then click on the object which is the border of partition.

Below you can see examples of curves splitting

The delimiter - the point on the curve



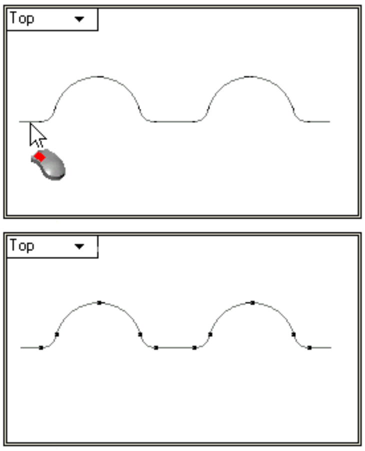
The delimiter - another curve



4.3.8 Explode

For fulfillment this operation you should press the button , by the cursor of the mouse click on the curve you need to explode on component elements.

On the picture, you can see an example of the curve exploding.

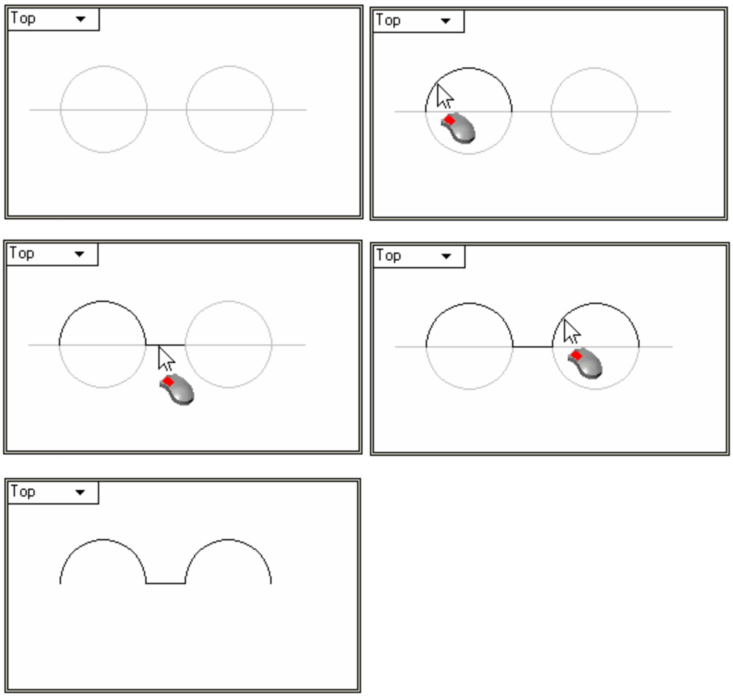


4.3.9 Fragment

This function executes the transformation of parts of geometric objects limited by other geometric objects to a contour.

For execution of this operation should activate the geometry layer. This can be done by pressing the button  or by holding the key CTRL. As a result, in a graphic window will image all geometric objects constructed earlier. Then press the button  and by mouse select the part of the geometric object that is necessary to convert this part to the contour. After transformation, the source geometric object remains without changes.

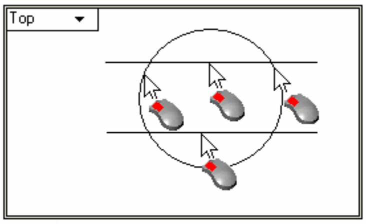
Below is shown the example of the transformation

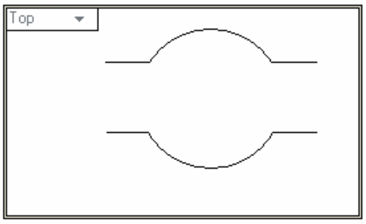


4.3.10 Clip/Trim

This operation is used when you need to clip the part of the curve (curve section).

For fulfilling this operation you should press the button . Then you need to click on the curve section you need to clip.





4.3.11 Contour construction modes

This panel  offers four different ways of constructing:

 - contour construction by the chain of geometric objects;

 - contour drawing;

 - contour construction on the set of points;

 - contour construction by mathematical expression.

The current construction mode defines available buttons on contour control panel.

For completing the contour construction press the button . The button  also completes the construction with closing the contour.

4.3.12 Contour construction by the chain of geometric objects

Construction by the chain of geometric objects mode is selected using the .

The components of the profile are created using previously defined elements (lines and circles). A profile must start and end at predefined 'points'. Selection of a point, during the construction process, ends the operation automatically.

To end the construction of a profile at the current point, press the  button. When the  button is pressed, a 'closing' element will be constructed, if it is required, and construction is finished.

During construction, it is possible to switch between the different profile assignment methods.

4.3.13 Contour drawing

Freeform drawing mode is selected using the  button. The elements of the profile are placed "by hand", the profile starts and ends at arbitrary points. The required type of profile element (line or arc) is defined by using the buttons.

To end construction of a profile at the current position, press the button. When the button is pressed, a 'closing' element will be created if required, and the construction is finished.

4.3.14 Contour construction on the set of points

Pressing the button activates the profile construction by point's mode.

When in this mode the management panel has two more buttons:

– profile construction methods.

Over the defined multitude of supporting points, the following can be constructed:

– polyline profile, supporting points are interconnected by line sections;

– spline profile, supporting points are approximated by splines;

When constructing a spline it is necessary to assign additional parameters in the input panel:

M – Approximation tolerance;

A – Angle of tangent at the start point;

A – Angle of tangent at the end point.

– Contour in which the creation points are connected by adjoining arcs.

In the input panel it is necessary to assign the parameter:

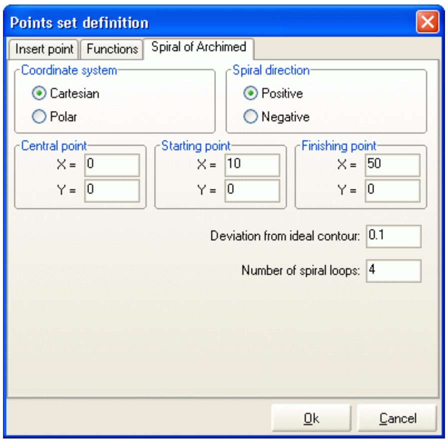
A – Angle of tangent of the first arc of the profile.

To end construction of the profile at the current point, press the button. When the button is pressed, a 'closing' element will be created if needed, and construction ends.

During construction, it is possible to change to a different profile creation method.

4.3.15 Archimedes spiral creation

For constructing the contour of an Archimedes spiral press the button f(x) and in opened window choose the Page Spiral of Archimed.

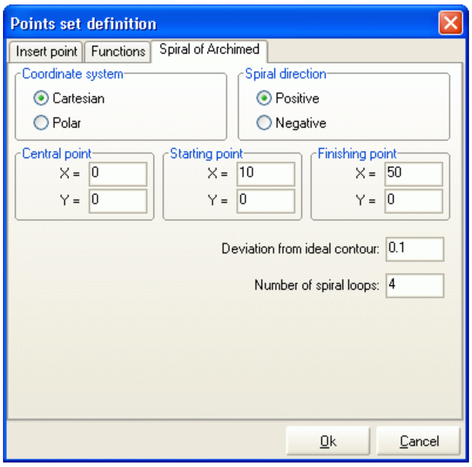


To create an Archimedes spiral it is necessary to define its start, end and central points. Tolerance of the contour means the distance between the points of the theoretical curve and the corresponding points of the constructed spiral. Contour-clockwise direction is the positive direction.

Having defined the parameters, press the button <Ok> to build the spiral or <Cancel> to cancel construction. If <Ok> is pressed, then a curve representing the Archimedes spiral will be constructed on the screen.

4.3.16 Contour construction on the list of points

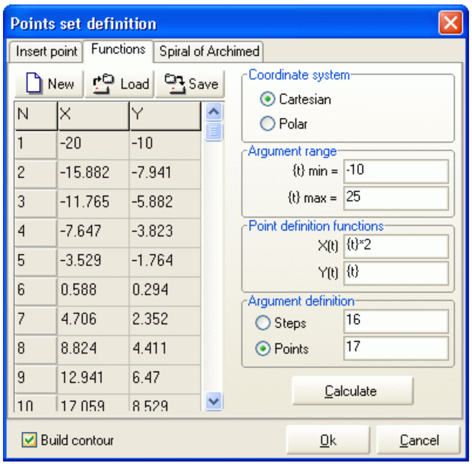
For constructing the contour on the list of points press the button f(x) and in opened window, choose the Page Spiral of Archimed



The point coordinates are set as shown on the picture. The points can be defined in absolute or incremental coordinate system.

4.3.17 Contour construction by mathematical expression

For constructing the contour on the list of points press the button f(x) and in opened window, choose the Page Functions

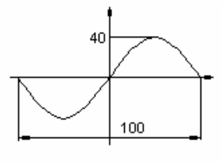


After defining, the parameters of a function press the button <Calculate> to receive the set of points. In the result of this operation will be filled the table of points.

For contour constructing set, a <Build contour> check on.

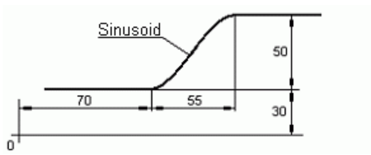
4.3.18 Constructing a sine wave profile (or a sinusoid).

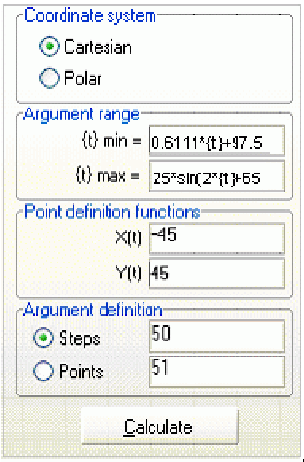
Let us see an example of construction of sine wave profile with the amplitude equal to 40mm and period equal to 100 mm.



It is known, that period of Y=Sin X function is equal to 180 degrees, where the length of the period is equal to 100 mm. Find the scale coefficient for the argument {t} 100/180=0.55555. Set the function expression X{t}=0.5555*{t}. In the expression Y{t}=40Sin(2*{t}) 40 - is the amplitude of the sinusoid, 2 - total number of semi periods of the sinusoid on the 100 mm section. On the sinusoid, it is possible to select any section, using the Argument range. Note that the range is assigned in degrees.

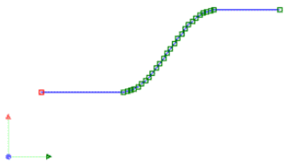
Let us see another example; below is a drawing of a contour, which contains a section that is represents by a part of a sinusoid.



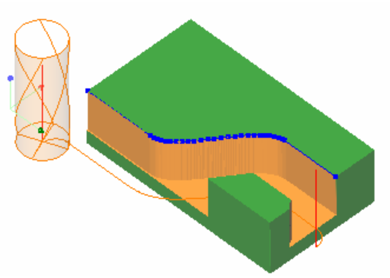


Required parameters of the sinusoid:
Period length - 110 mm
Amplitude - 25 mm
Require argument coefficient X{t} 110/180=0.6111.
From the drawing we define the offset coefficient:
for X{t} 70+55/2=97.5
for Y{t} 30+50/2=65
It is necessary to put the obtained values into the corresponding fields of the parameters window.
Calculate and as a result, the required profile is obtained.
Construct straight lines on the ends of the sinusoid and in the editing mode join the resulting three contours into a single one. The result is shown below.

Move to "Machining" mode and select the "2D contouring" operation and having assigned the required machining parameters, we can create the toolpath.

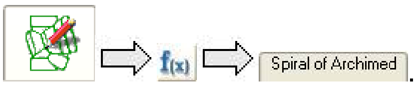


In the "Simulation" mode, the user will obtain the following machining result.



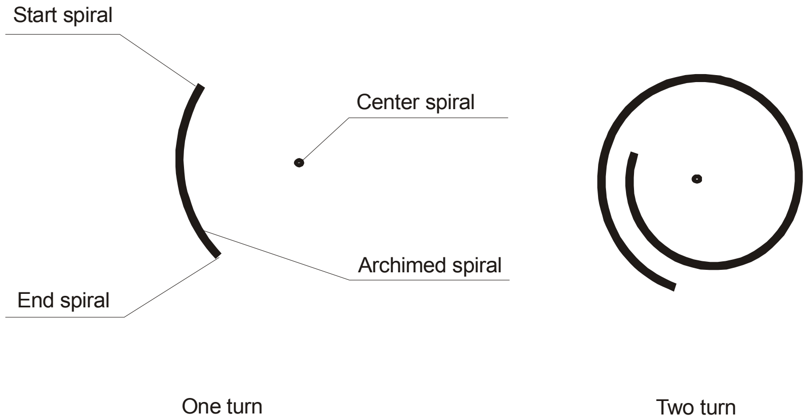
4.3.19 Constructing an Archimedes spiral based profile

Let us see a profile construction that contains an element of an Archimedes spiral,

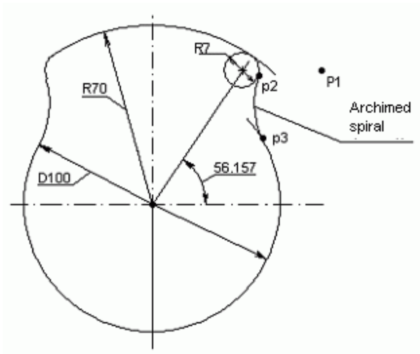


To construct an Archimedes spiral it is necessary to assign several parameters:

- The central point of the spiral
- The starting point of the spiral
- The finishing point of the spiral
- Number of loops of the spiral
- Deviation
- Spiral direction



Let us see an example of profile construction, one section of which is an Archimedes spiral



In the picture () there is a drawing of the profile cam. The section between points P2 and P3 is an Archimedes spiral, where P2 - is the starting point of the spiral, P3 - final, P1 - the spiral center.

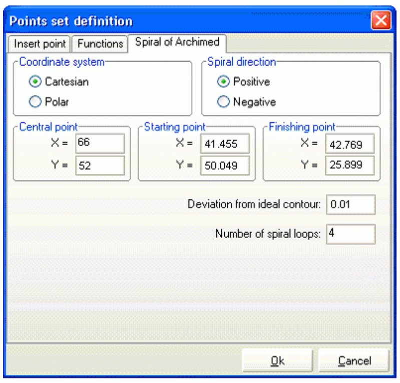
These points' coordinates were calculated by the designer using the rules of the cam required by the control mechanism.

P1 X=66; Y=52

P2 X=41.455; Y=50.049

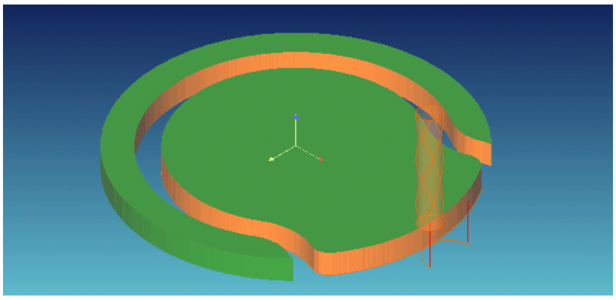
P3 X=42.769; Y=25.899

In the parameters assignment window enter the values as shown in the window (below). In the **SprutCAM**, graphic window there will be a display of the required spiral section. Next, construct the missing parts of the right hand side of the symmetrical cam. Using the construction method for symmetrical profiles, finish the construction of the left part of the cam and join all the areas into a single profile.



Move to "Machining" mode and select the "2D contouring" operation and enter the required machining parameters, create the toolpath.

In the "Simulation" mode, the user will obtain the following machining result.



4.3.20 Constructing ellipse based profile

Let us see an example of profile construction that contains ellipse elements.

Ellipse equations in the parametric form:

X{t}= a*sin{t}

Y{t}= b*cos{t}

Where a and b are the scale coefficients along the X and Y-axes (major & minor axes).

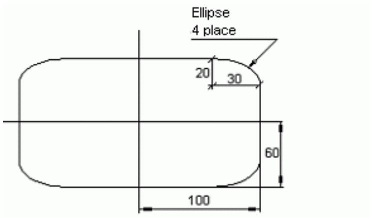
According to the drawing a = 30, b = 20

Argument ranges for the ellipse quarters:

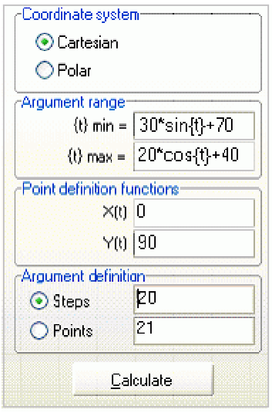
- Right top angle 0-90 degrees
- Right bottom 90-180
- Left bottom 180 -270
- Left top 270-360

Ellipse center offset along X and Y axes:

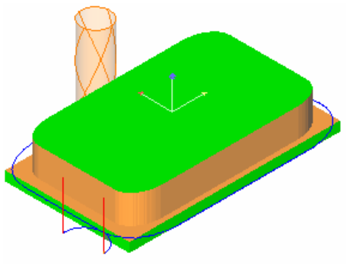
- Right top angle +70 +40.
- Right bottom +70 -40
- Left bottom -70 -40
- Left top -70 +40



In the parameters assignment window, (see pic.) in the corresponding windows there are parameter values for creation of an ellipse quarter, which is located in the right top corner of the profile. Complete the construction of the ellipses in the other corners of the figure. Next, construct profiles by using the two extreme points of the ellipses and join the 8 resulting profiles.



Move to "Machining" mode and select the "2D contouring" operation and having assigned the required machining parameters, we can create the toolpath. In the "Simulation" mode, the user will obtain the following machining result.



4.3.21 Operations with profiles

Press the  button to activate the profile operations mode.

The panel contains the following functions (in order):

- equidistant (offset) profile definition ;
- profile copying with move ;
- profile copying with rotation ;
- profile copying with mirroring ;
- multiple profile copying with move and joining ;
- multiple profile copying with rotation and joining ;

Number of copies for multiple copying operations can be defined in the field N.

4.3.22 Defining an equidistant (offset) profile

To construct an equidistant profile it is necessary to do the following:

- select the profile building mode by pressing the  button;
- select equidistant profile construction mode by pressing the  button;
- select the source profile;
- by clicking the mouse button within the working area select the point, through which the equidistant profile will pass, or in the M field define the distance from the source profile at which the new profile will be located;

When performing this operation, a new string defining the construction of the equidistant (offset) geometrical objects will be added to the geometry definitions (debug) program.

4.3.23 Profile copying with shifting

To construct a copy of a profile with parallel shifting it is required to do the following:

- select the profile building mode by pressing the button;
- select copying with shifting mode by pressing the button;
- select the source profile;
- The user can enter either the shift amount in the fields X and Y, or two points successively. If using points, the first point will be the start of the source coordinate system, the second - the start of the new (shifted) coordinate system.

When performing this operation, a new string defining the construction of the parallel geometrical objects will be added to the geometry definitions (debug) program.

4.3.24 Profile copying with rotating

To construct a profile by rotating it is required to perform the following:

- select the profile building mode by pressing the button;
- select profile construction by rotation mode by pressing the button;
- select the source profile;
- in the parameter assignment area A define the rotation angle.
- define the rotation center point.

When performing this operation, the Shift & Rotate function of the geometrical processor is used.

4.3.25 Profile copying with mirroring

To construct a mirror profile, do the following:

- select the profile building mode by pressing the button;
- select the construction of the mirror profile mode by pressing the button;
- select the source profile;
- Define the line, which will be the symmetry axis.

When performing this operation, the profile creation using axis symmetry function of geometrical processor will be added to the definitions program.

4.3.26 Multiple profiles copying with shifting and joining

Contour shifting will be performed so that the start point of every following copy will coincide with the last point of previous copy.

To construct a profile by shifting with joining, do the following:

- select the profile building mode by pressing the button;
- select profile construction by multiple shifting mode by pressing the button;
- select the source profile;
- Assign the number of copies in the parameter area N.

When performing this operation, the multiple shifting with joining function of the geometrical processor will be added to the definitions program.

4.3.27 Multiple profiles copying with rotation and joining

Profile rotation will be performed so that the start point of every subsequent copy will coincide with the last point of previous copy. For construction, it is necessary that the start and end profile points are at the same radial distance from the rotation center.

To construct a profile by rotation with joining, do the following:

- select the profile building mode by pressing the ;
- select profile construction by multiple rotation mode by pressing the ;
- select the source profile;
- Define the rotation center.
- Assign the number of copies in the parameter area N.

When performing this operation, the multiple rotations with joining function of the geometrical processor is used.

4.3.28 Viewing element parameters

The parameter element-viewing mode is activated by the button.

In the view mode, when the mouse pointer is moved over an object in the graphic view, the parameters of the selected object will be displayed in the corresponding fields of the input panel. The displayed parameters will depend on the type of selected objects.

Viewing the object parameters and checking dimensional values between elements is also possible in the geometrical calculator, which opens by pressing the button.

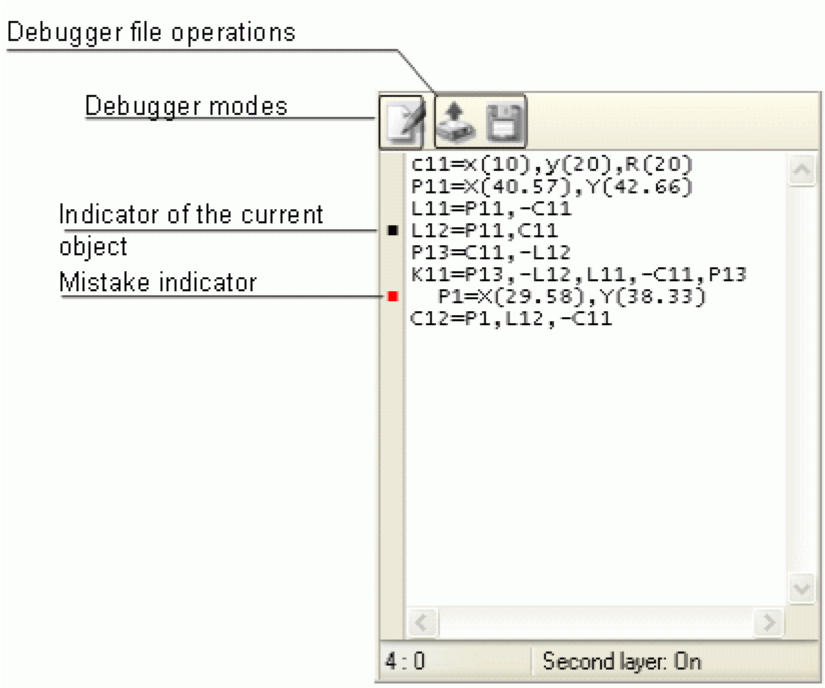
4.3.29 Geometric objects deleting

Selected by the mouse an active object will be deleted when pressing the <**DEL**> key.

4.4 ADDITIONAL FEATURES

4.4.1 Debug

To open the debug panel presses the Debug button.



The debug panel provides access to the conversational representation of all 2D object definitions, in the current plane. Simultaneous viewing of the graphic and conversational elements is possible and interactive editing of separate parameters.

Object definitions in the current plane are displayed as a sequential list of instructions on the screen using the geometrical processor language. Modification of the defining text of an element will lead to automatic regeneration of all objects constructed using the modified element.

All object definition methods have a corresponding conversational structure. This means that, an interactive creation of a 2D object will create a corresponding definition in the debug panel. Some definition methods may require auxiliary constructions. These auxiliary constructions are performed automatically; indices of the objects that are created are between 1 and 10. Auxiliary objects are not displayed in the graphic view.

The button saves the original definition text of all planes into a text file.

The button loads the previously saved file of original definitions.

- edit

4.4.2 Edit mode

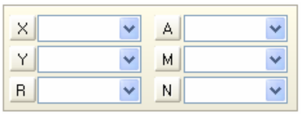
Edit mode is enabled by pressing the button in the debug panel.

Selection of an object using the left mouse button in the graphic view will enable the editing of its definition in the debug panel. Selection of a definition can also be performed directly from the debug panel.

There are several variations of how to edit the definition of an object.

The conversational definition of an object can be edited directly in the debug panel by selecting the required string by left mouse clicking on it. To identify which string relates to the required object, move the mouse over the object in the graphic area, and the corresponding string will have a marker next to it. Alteration of the defining text will lead to immediate reconstruction of any objects connected to the one being edited. Deletion or creation of new definitions in the geometrical processor language is possible.

Editing of one or all parameters of a 2D object is also possible by defining the type (or types) of the parameter to be edited (X, Y, R, A or M -respectively for coordinate X, coordinate Y, radius, angle or distance) by pressing the corresponding buttons on the Parameter value panel. When selected, its value from the current string will be placed into the parameter value panel. The value in this field can be edited using the keyboard. When edited, the definition string for the selected object will be updated, and all other objects that use this objects definition will be updated. All modifications will be displayed in the graphic view.



.

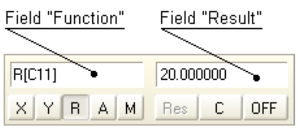
Editing of one or more parameters for an object can also be performed interactively by selecting the parameter buttons in the Parameter value panel required for modification (X, Y, R, A or M), and in the graphic view click the left mouse button on the object and holding it pressed drag the object (modify it). All changes will simultaneously occur in the graphic view, the definitions text and the parameter value panel. All objects that have definitions referring to the object that is being changed will also be updated.

4.4.3 Geometrical calculator

It is possible to obtain dimensional information about objects parameters or, their relationship with other objects by using interactive functions that return a numeric value. The Geometrical calculator gives access to these interactive functions. All these functions can be used in the conversational definition of an element instead of a numeric value. They can also be used in the mathematical expressions.

The geometrical calculator has two modes: the first one is for viewing parameters and correlations of the objects; the second one is for putting this result as the parameter into the definition of a new element.

To activate the Geometrical calculator in the parameter viewing mode press the  button on the tool panel. Parameters and correlations between objects can be viewed in the geometrical calculator window. When the <Clear> button is pressed, the fields Function and Result will be erased. To close the window presses the <Cancel>.



To view an element's parameter select the button for the required parameter in the geometrical calculator window (X, Y, M, A or R - respectively for coordinate X, coordinate Y, linear size, angle or radius) and left click on the object in the graphic view. For example, to determine the slope angle of a line, press the <A> button and in the graphic view left click the line. The objects string from the geometrical processor function will be displayed in the Function field, and a numeric value will be put into the Result field. Any of these strings can be copied using the exchange buffer into the definition sequence in the debug panel.

To view correlations between objects, press the button for the required parameter and point in the graphic view by the mouse at the required objects. For example, to check the distance from a point to a line, press the <M> button and using the mouse, left click the line and then the point (or vice versa) in the graphic view. The objects corresponding strings will be displayed in the Function and Result fields.

In order to use the result of the geometrical calculator as an object's parameter, when the new object is being created, and the parameter is required, select the required parameter name near the input are(X, Y, A, R, M or N), then define the required parameter/correlation between objects in the geometrical calculator window. When the definition is complete, close the calculator window using the <Ok>button. The obtained function will be used as a parameter of the element being created. This means that an additional parametric link has been imposed; further alteration of the basic elements will cause modification of this newly created object. If the window is closed using the <Close> button, no action will be taken.

For example, when defining a circle, the radius value may need to be set equal to the distance between two earlier defined points. To do so, during the circle creation press the <R> button next to the radius value input area  then select an appropriate calculator function, (<M> – distance between points function) and successively left mouse click on the required points. Closing the window by using the <Ok>button passes the defined parameter to the objects defining string.

All other geometrical calculator function can be used in a similar manner (coordinates X and Y, linear size, angle value, radius). Values can be calculated for:

- X coordinate of the point;
- Y coordinate or the point;
- distance between two points along the X axis;
- distance between two points along Y axis;
- aligned distance between two points;
- distance between two circles (edges);
- distance between a line and point/circle;
- line angle to X axis in degrees;
- angle of a perpendicular line;
- circle radius.

The following standard mathematical operations and functions can be used in the geometrical calculator:

- «+» - add;
- «-» - subtract;
- «*» - multiply;
- «/» - divide
- «^» - rising to a power.

It is necessary to remember that two arithmetical signs cannot be used together. As well as these operators, the following standard functions are also allowed in mathematical expressions:

- LOG (x) - decimal logarithm x;
- CTG (x) - cotangent x;
- ASIN(x) - arcsine x in the degrees;
- ACOS(x) - arccosine x in the degrees;
- SIN (x) - angle sine x in the degrees;
- COS (x) - angle cosine x in the degrees;
- ATG(x) - arctangent x in the degrees;
- TG (x) - angle tangent x in the degrees;
- SQRT (x)- square root from x;
- ABS (x) - absolute value x;
- SGN (x) - sign x;
- LN (x) - natural logarithm x,

Where x the function argument, which represents a number, a variable, a mathematical function or mathematical expression;

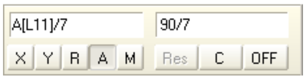
For example, consider a line drawn at an angle, and a point that is not 'part' of the line. The user may need to draw a second line, which passes through the point at an angle that is 1/7 of the angle of the first line.

The order of actions is following:

- Select line mode.
- Using your mouse, select the point in the graphic area.
- Press the <A> button next to the angle value input field 

. The geometrical calculator window will open.
- Select the angle function <A> on the geometrical calculator panel.
- Using the mouse in the graphic view, select the previously constructed angled line (for example, its angle is 90 degrees). The fields Function and Result of the geometrical calculator will show the following:

- Click your cursor in the Function field and modify it thus: A[L11] /
- The window will show the following:



Press the <Ok> button on the calculator panel or hit “Enter” on the keyboard. A second line will be created that passes through the point at an angle that is 1/7 of the original line. The modified string will be added to the conversational definition for the new line. Thus, an additional parametrical link between the lines has been created. If the angle of the first line is altered, the angle of the second line will be updated automatically.

4.4.4 Layers management

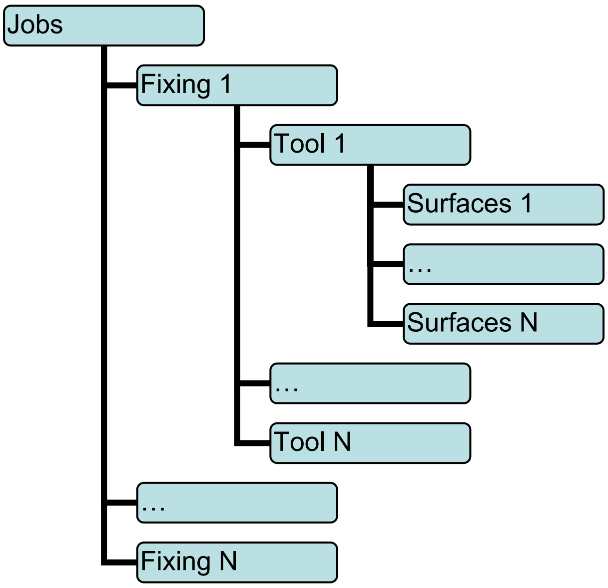
Every plane contains two layers. Press the  button to switch between the layers. It is recommended to use one of the layers for auxiliary constructions.

To move an element to another layer, hold the Shift key pressed, select the element with the mouse pointer and click the left mouse button.

5 CREATING MACHINING TECHNOLOGY

5.1 COMMON PRINCIPLES OF TECHNOLOGY CREATION

The process to machine a part on the milling machine with the CNC-controller has the hierarchical structure. In the **SprutCAM** the jobs tree node is the operation. The operations group can contain the other operations inside. The job list granularity can be arbitrary in any specific case. The example of the complex jobs tree is shown below.

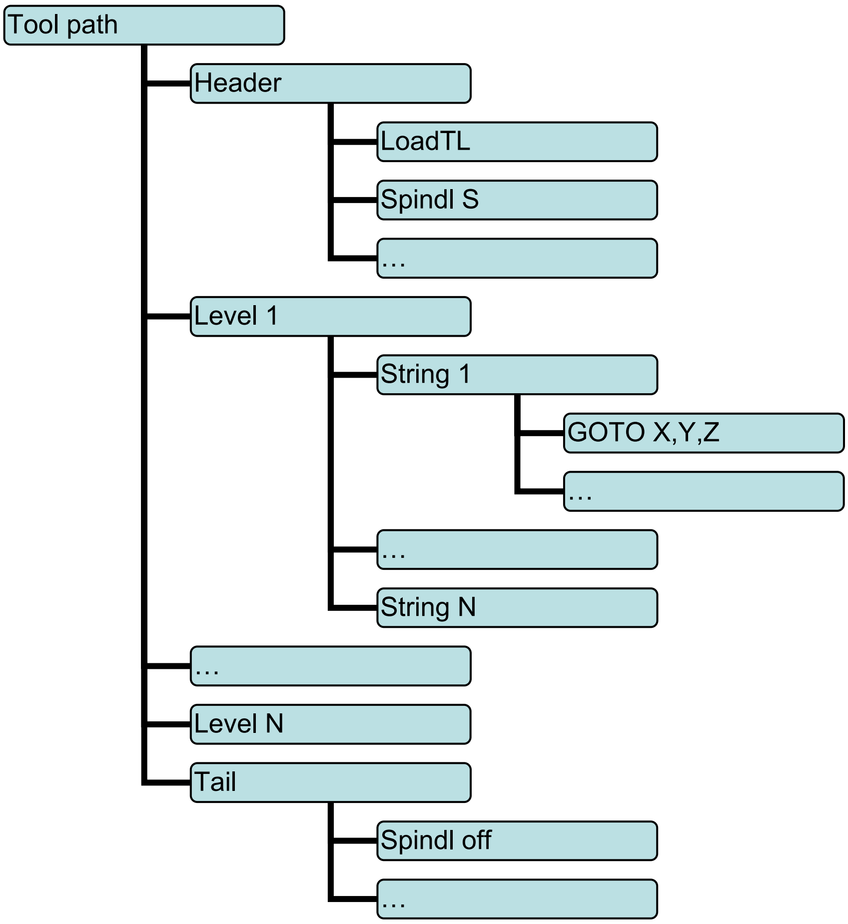


The fixing node is the actions sequence with the static workpiece position. The tool node is the actions sequence with the constant tool position. The surfaces node is the actions sequence for the concrete surfaces machining.

Any operation has the coordinate system, tool, machining and restriction surfaces etc. Therefore, the operation can include the fixing node, the tool node and the surfaces node simultaneously. It allows excluding the false complication in the simple cases. On the other hand, the job tree gives the possibility to describe the complex machining.

The task of user is the creating of the job tree to the surfaces node level. The further decomposition is performed automatically by the tool path calculation for the every operation. The tool path is the attribute of the operation. It depends on the operation type and parameters values. The operation type is defined then the operation is created. The operation type defines the strategy of the machining. The parameters values can be changed in any time. After the changing it is need to recalculate the tool path.

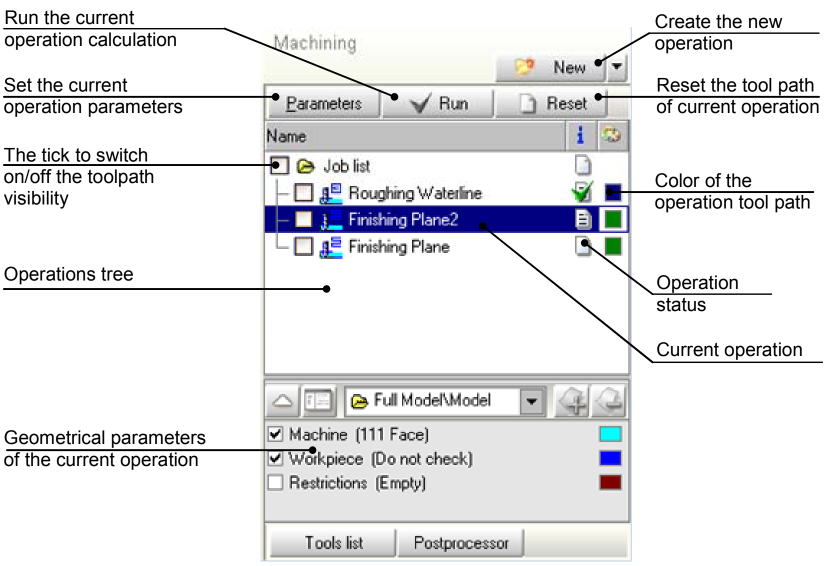
Actually, the tool path is a sequence of the technological commands in the **CLDATA** format. The tool path includes both the tool motion commands and the technological commands like the feed rate switch; spindle and coolant switch of/off etc. The tool path has the hierarchical structure like the jobs tree. Therefore, the elementary commands can be united into the groups. The type and the contents of the group depend on the operation type. For example, the structure of the tool path of the rough plane operation is shown below:



Automatically calculated tool path can be tested and edited in the simulation mode. After that, the tool path is output to the postprocessor to be transformed in NC program.

5.1.1 Job tree

Click the machining bookmark in the main window to start the job tree editing.



The top part of window contains the job tree. The bottom part is the geometrical parameters of the current operation.

- < 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 popup menu.

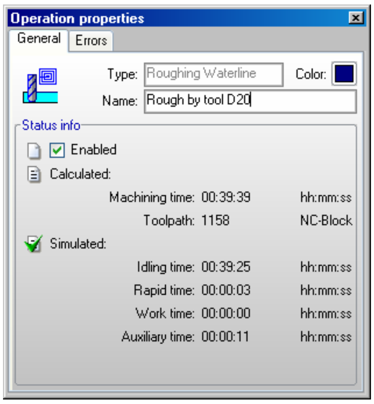
- Del – deletes the current operation;
- Ctrl+R – renames the current operation;
- Ctrl+X – cuts the current operation to the clipboard;
- Ctrl+C – copies the current operation to the clipboard;
- Ctrl+V – inserts the operation from the clipboard. Operation is added to the end of the current group.

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 click the right mouse button on the operation and choose the “enable” item from the popup 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). The operation group is calculated if all operations inside are calculated;
-  - Operation is calculated and simulated without error. 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.

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.

5.1.2 Standard job trees

In practice the operations sequence to machine the parts of the same type is coincided. For example, the machining of the tree-dimensional mold often can be executed by the sequence of three operations:

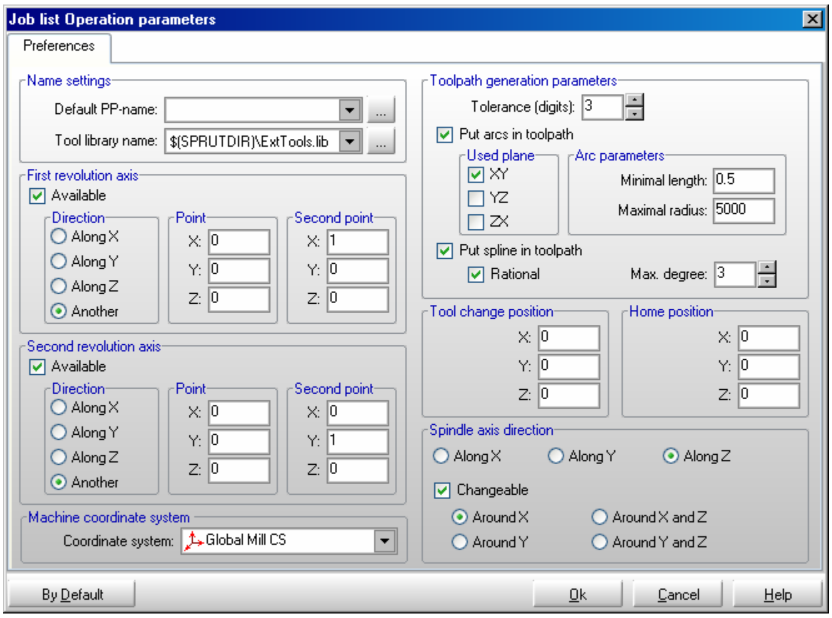
- Waterline Rough
- Complex finishing
- Complex rest areas re-machining.

If the numerous parameters of these operation including the tools, stocks, steps etc. are equal for the different mold models then it is expedient to use the standard job trees. To do it **SprutCAM** has the possibility to import the job tree or an operation from the project that was made earlier. These functions are accessible from the popup menu of machining mode.

- Export operation – saves the parameters of the current operation to the file (*.sto)
- Import parameters – fills the parameters of the current operation from the file that was saved earlier. (*.sto)
- Import operation – adds the operation or the job tree into the current job tree from the operation file (*.sto) or from the project (*.stc) correspondingly.

5.1.3 Parameters of the root operation

The machine characteristics are defined in the root operations of the job tree. These characteristics have an influence on the tool path calculation methods and on the parameters of all job tree operations. For the edit the machine characteristics if is necessary to select the root operation and to press the “Parameters” button.



«**Default PP-name**» defines the postprocessor file name that will be used as default for the NC program generation.

«**Tool library name**» defines the tools library file name. The library is used for the automatic tool assigning then the operation is created. In addition, the library suggests the tool for the user to be assigned from the parameters-tool dialog.

«**First revolution axis**» is enabled if the machine has a revolution device liked to CNC controller. It can be rotary head or rotary table. The location of the revolution axis is defined here.

«**Second revolution axis**» is enabled if the machine has the two revolution devices. The location of the second revolution axis is defined in the machine coordinate system with the zero position of the first revolution axis.

«**Machine coordinate system**» defines the coordinate system that is used to define the axes of the revolution devices.

«**Tool path generation parameters**» defines the restrictions that collide with the CNC controller. The coordinates in the **CLDATA** tool motion commands are rounded to the defined tolerance. The tool motion command will not be added to the tool path if the motion length is less then the defined tolerance.

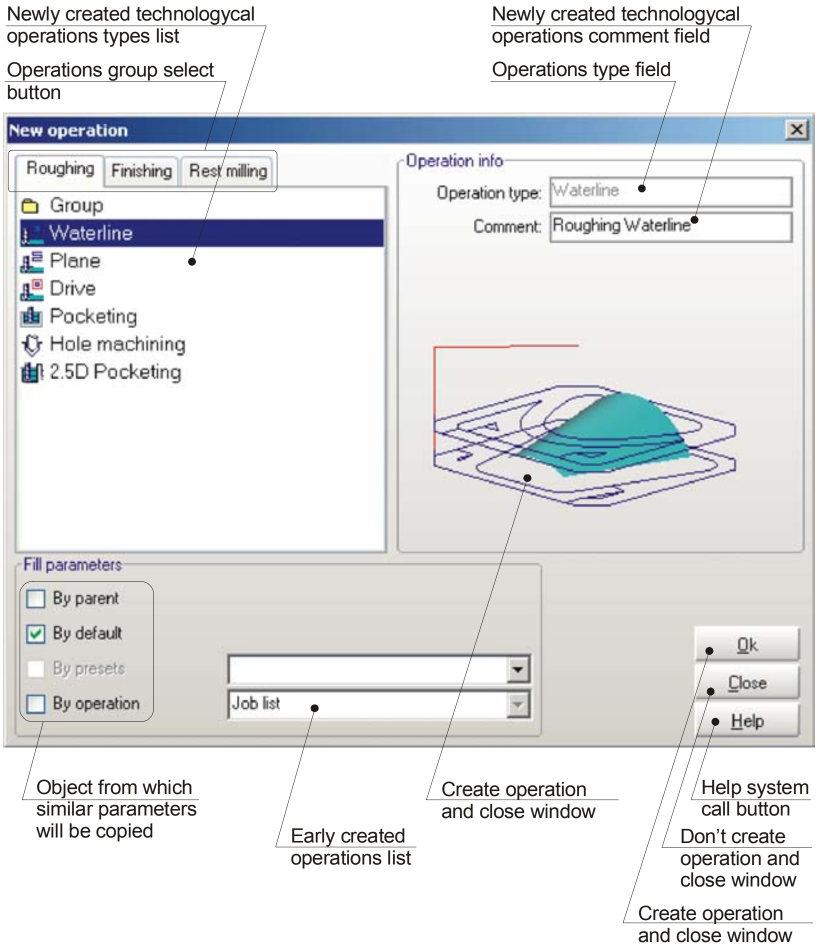
If "**put arcs in tool path**" is disabled then the linear motions only will be added to the tool path. If this mode is ticked then the circle arcs is output to the tool path. The arc must lie in the permissible plane. The arc length must be more then the minimal length. The arc radius must be less then the maximal radius. Else, the arc is approximated by the cuts with the operation tolerance.

Put splines in tool path can be ticked if the CNC-controller supports the spline interpolation in the **NURBS** format. If **Rational** is, ticked then both rational and polynomial splines are created. Else, the polynomial splines only are created. Max. degree defines the maximal degree of the spline that CNC-controller is support.

The fields "**Tool change position**", "**Home position**" and "**spindle axis direction**" have not the influence to the tool path in the current version of **SprutCAM**.

5.1.4 Creating new operation

The window for creation of a new machining operation is opened by pressing the New button.



In the Operation type area the user can select the type of the machining operation required. All operations are divided into three main groups: roughing, finishing and rest milling. When a group is selected, the available operations in that group are displayed. The number and the type of available operations depend on the configuration of **SprutCAM** being used.

The roughing operations enable removal of workpiece material, which lies outside the model being machined taking account of any restricting boundaries. Normally, the roughing operations are used for the removal of stock material where the shape and dimensions of the finished model differ greatly from the shape and dimensions of the workpiece.

The finishing operations perform machining of the final surfaces only, without workpiece material removal. When selecting the "complex" or "plane optimized" menu items, the system will create two adjacent operations with 'linked' parameters. 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 using a workpiece made of soft material.

The re-machining operations allow the user to perform machining only in those areas where material has been left after previous machining operations. The rest machining strategies are identical to other strategies; except that different default values are set. The roughing re-machining 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 allow the user 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. The operation of spatial transformation (offset) of the toolpath is also included in the rest milling operations group.

The type of operation can be selected from the list in the Operation type panel. When an operation is selected from the list, the text in the **Operation type panel** (right side of window) is updated. A default comment is added (Comment area) and the help graphic changes. The Comment can be edited by the user.

User may copy similar parameters from other object: parent operation, default parameters or other operation. Object for copying parameters may be selected from Fill parameters panel.

Closing the window using the <**Ok**> button creates a new operation of the selected type with parameters assigned. The parameters are assigned either by default, depending on the operation type and the geometry of the model being machined, or copied from an existing operation. This means that the new operation is ready to be calculated and does not require the input of many values. In most cases, it is a simple matter of adjusting a few parameters based on the particular model being machined. The newly created operation becomes the current operation, and is available for editing and execution.

Pressing the <**Cancel**> button cancels the operation.

Help - activates the help system.

5.1.5 Executing operation

Having created a new operation the system automatically assigns the values of all 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

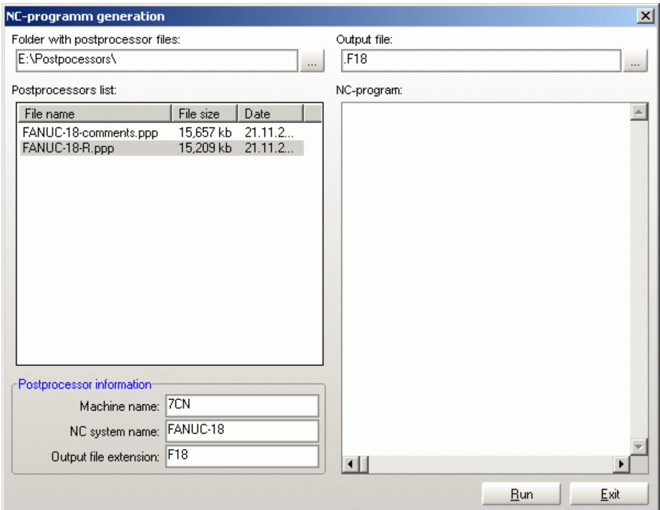
displays how far the current operation calculation has progressed. Clicking within the process indicator area will interrupt execution of the operation. The system will request confirmation of process interruption. If 'yes' is selected, the calculation will be cancelled, if 'no', the calculation will continue.

During the calculation process all visualization control buttons are enabled.

5.1.6 Generating the NC program

Generation of NC programs is performed by the postprocessor transforming the machining commands, which are stored in the connected project file (*.mcd), into the format of the selected CNC system. Selection of the CNC system is made by selecting the corresponding CNC tuning file (*.spp). The NC program will be output into a standard text file. Transfer of the NC program from the computer, where **SprutCAM** is installed, to the CNC machine can be made using any available method.

To run the postprocessor press the Postprocessor button. The window has the following view.



The required CNC system is selected using a postprocessor tuning file (*.spp). The name of the tuning file is displayed in the field Folder with postprocessor files. The fields Machine name and NC System name show the name of the milling machine and the CNC system, for which the selected tuning file is created. To change the postprocessor tuning file, press the ... button to the right of the Folder with processor files field, this opens the default folder of tuning files. The default folder that contains the postprocessor tuning files, and the definition of the default postprocessor to be used, can be defined in the system settings window.

The **Output file** field displays the name of the project assignment file, for which the NC program is being created. **SprutCAM** generates the assignment file with the name of the current project with an *.mcf extension. Transfer of the named set of machining commands files (*.mcd) and information about currently active operations (column On in the machining process 'list' window) is performed via this file. The assignment file can be changed to **SprutCAM** project file (*.stc) or to the machining commands file (*.mcd). NC code will be created using the defined file based on the enabled list of operations in the order that they appear. Changing of the assignment file is performed in the dialog window that opens by pressing the ... button

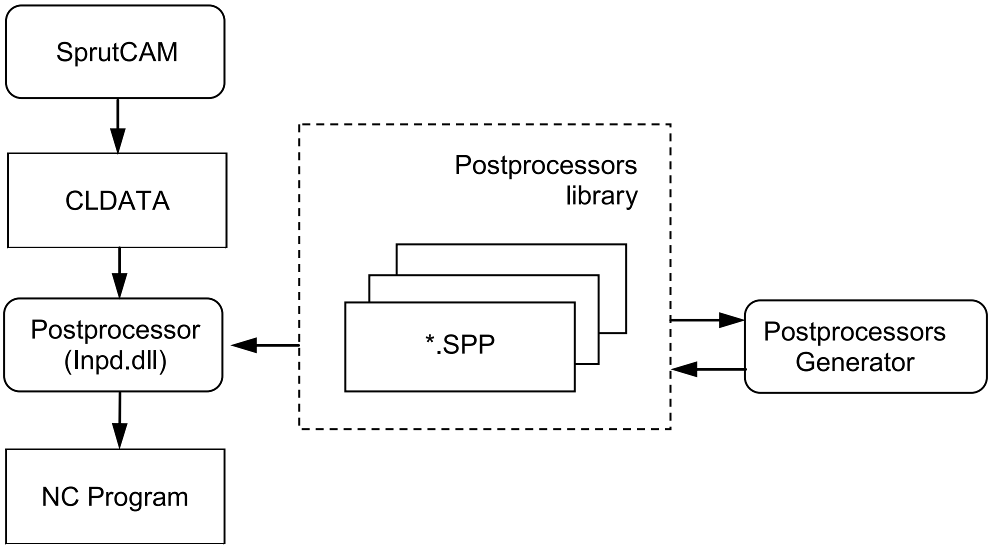
The NC program will be output to a text file with the name defined in the Output file field. The name of the output file can be entered either directly in the field or in the dialog window, which opens by pressing the ... system settings window.

When pressed, the <**Run**> button will start generating NC program for the selected CNC system. In the full version of **SprutCAM** 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.

One should note, that the postprocessor only creates NC code for all calculated machining operations that are 'enabled' at the time the postprocessor is run (column in the machining process list window).

Note: *If one needs to generate several separate NC programs for different operations, it is advised to first execute all the machining operations. And then, by placing ticks in different boxes in the machining process window, run the postprocessor to create the separate NC programs.*

The postprocessor system for **SprutCAM** is a separate standard module, and can, if required be run separately from **SprutCAM**. The name of the executable file is - SprutPP.exe. Postprocessor tuning files for CNC systems are created and edited using the invariant postprocessor (INP.exe).



5.1.7 Generating tools list

SprutCAM generates the tools list. This document contains the information for the machinist to machine the part. The tools list can be edited by the text editor and printed. The document is saved in the HTML format. The way to generate the tools list is defined in the setup window.

The tools list document has the next structure:

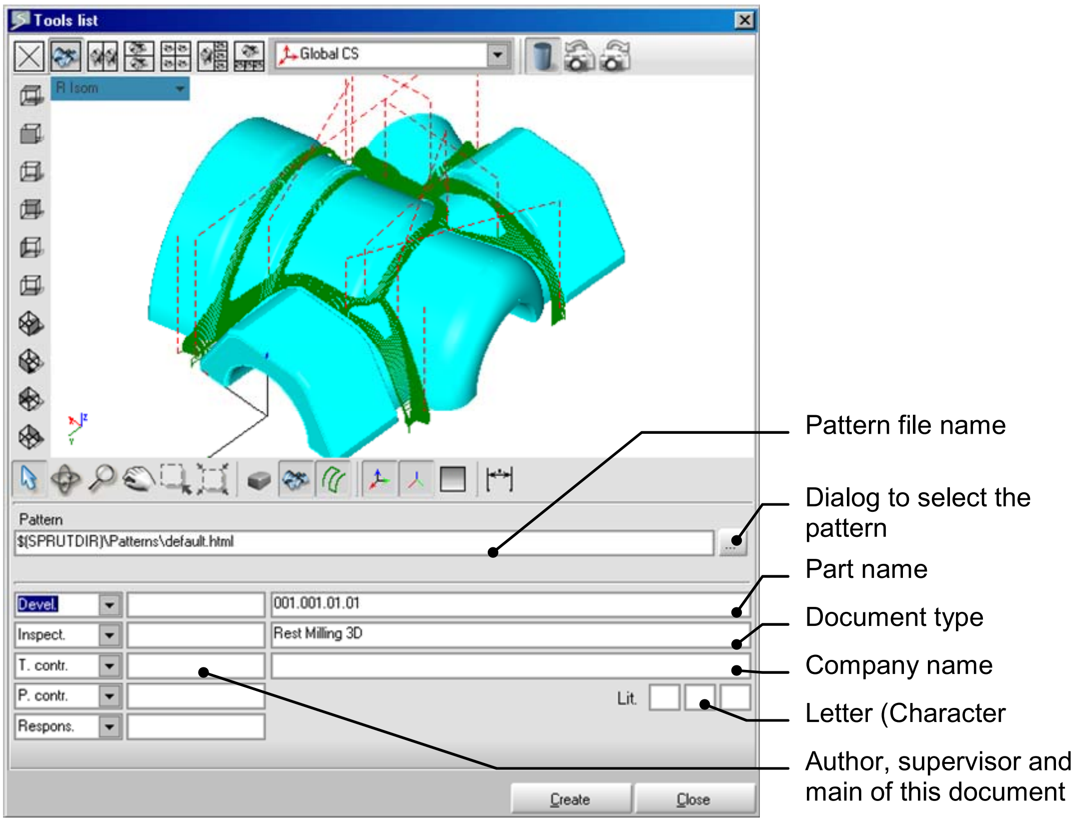
1. The drafts of the tool path the and model with the dimensions to set up the workpiece;

2. The operations list. Every operation has the own machining time and the total time is printed. The enabled and calculated operations only appear in the tools list.

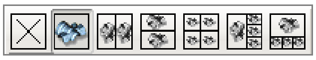
3. The table of the used tools

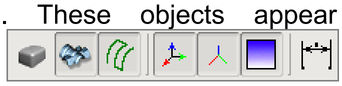
4. The table of the holes coordinates for the pre-drilling for the tool plunge.

Press the  button from the machining bookmark to open the Tool list generation window.



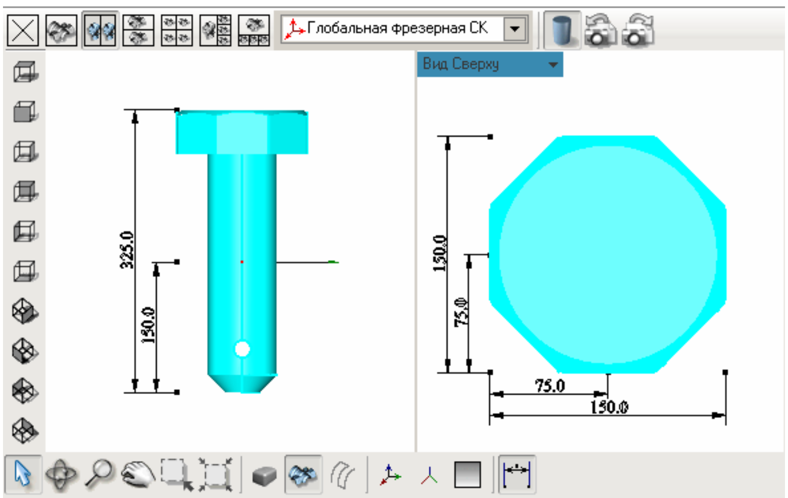
The next panel defines the drafts quantity and disposition:



The draft can contain the workpiece, source model, tool path, origin, axes and dimensions. These objects appear if press the corresponding buttons in  panel:

SprutCAM 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.

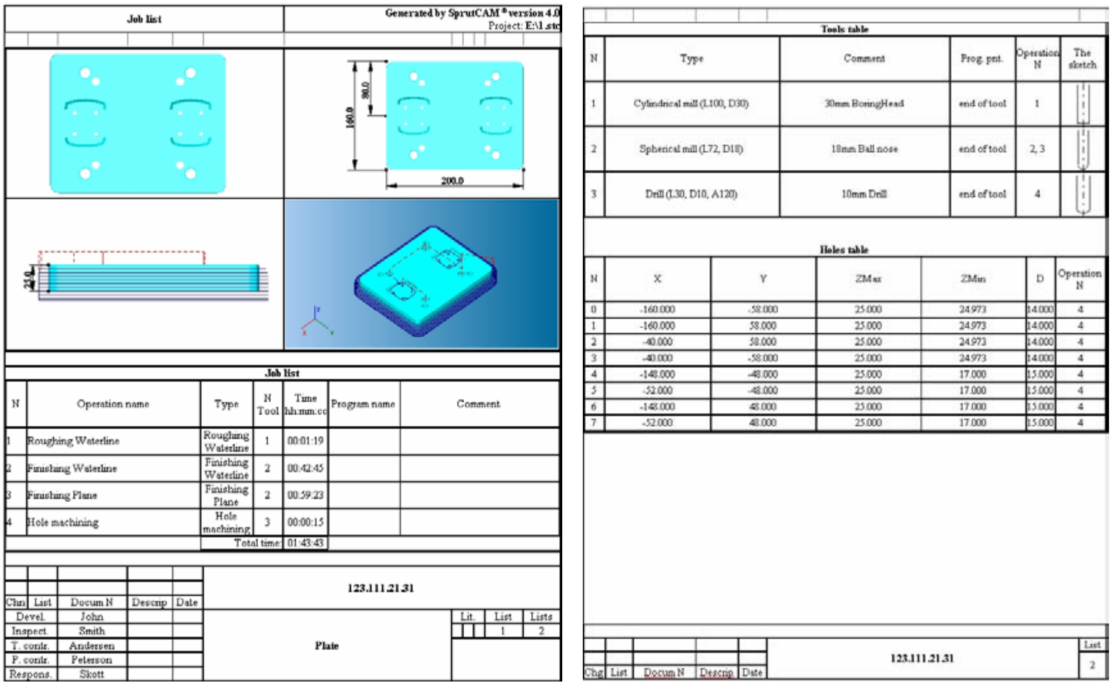
Dimensions example:



Fill the fields and click  to create the tools list. The editor to show the document is defined in the setup window.

Use  to close the dialog.

Example of result document show below:



Job list | **Generated by SprutCAM ® version 4.0** Project: E:\1.stc

Job list

N	Operation name	Type	N Tool	Time hh:mm:cc	Program name	Comment
1	Roughing Waterline	Roughing Waterline	1	00:01:19		
2	Finishing Waterline	Finishing Waterline	2	00:42:45		
3	Finishing Plane	Finishing Plane	2	00:59:23		
4	Hole machining	Hole machining	3	00:00:15		
		Total time:		01:43:43		

Chn	List	Docum N	Descrip	Date	123.111.21.31		
Devel.		John			**Plate**	Lit. / List 1 / Lists 2	
Inspect.		Smith					
T. contr.		Andersen					
P. contr.		Peterson					
Respons.		Skott					

Tools table

N	Type	Comment	Prog. pnt.	Operation N	The sketch
1	Cylindrical mill (L100, D30)	30mm BoringHead	end of tool	1	
2	Spherical mill (L72, D18)	18mm Ball nose	end of tool	2, 3	
3	Drill (L30, D10, A120)	10mm Drill	end of tool	4	

Holes table

N	X	Y	ZMax	ZMin	D	Operation N
0	-160.000	-58.000	25.000	24.973	14.000	4
1	-160.000	58.000	25.000	24.973	14.000	4
2	-40.000	58.000	25.000	24.973	14.000	4
3	-40.000	-58.000	25.000	24.973	14.000	4
4	-148.000	-48.000	25.000	17.000	15.000	4
5	-52.000	-48.000	25.000	17.000	15.000	4
6	-148.000	48.000	25.000	17.000	15.000	4
7	-52.000	48.000	25.000	17.000	15.000	4

Chg	List	Docum N	Descrip	Date	123.111.21.31	List 2

5.2 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 re-machining 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 model to be machined, operations can be divided into two groups:

- For the operations: area rest milling, engraving, 2D contouring и 3D curve milling the model to be machined is defined by a set of curves. All surface objects of the model are ignored.
- The model to be machined for all other operation types, the model is defined by surfaces. Solid bodies, surfaces and meshed objects can be used for defining the surface detail. All curves in the model definition list are ignored.

5.2.1 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 the stock 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.

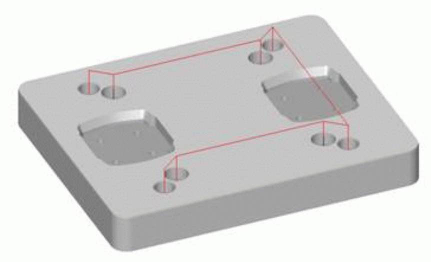
The list of all machining operations with a short description is shown below. The types of machining operations listed here are divided into the same groups that they appear, in the new operation creation window.

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.

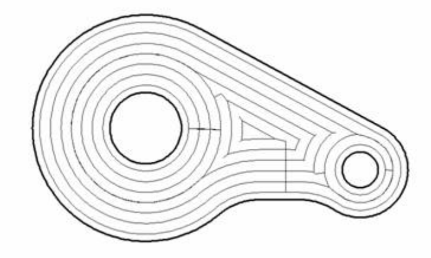
Roughing operations

Hole machining



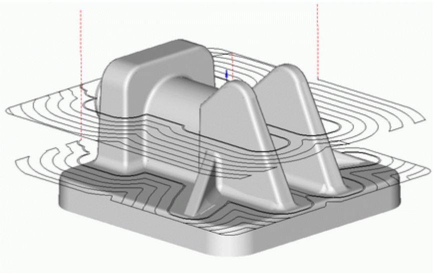
Creates a set of machining commands for holes. These include drilling, boring, centering or threading. The operation can be used both for hole machining and for preliminary drilling in tool plunge points in pocketing and waterline roughing operations.

Pocketing



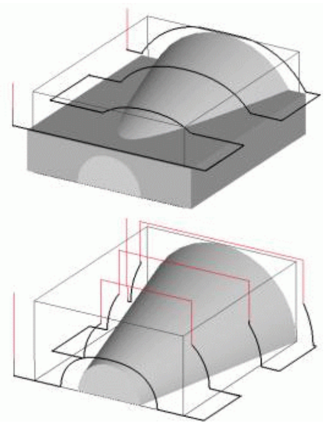
Waterline removal of material inside the defined area or pocket. The shape of the area for pocketing is formed from curves created on the horizontal (XY) plane. This operation is used for the 2 & 2.5D machining of pockets and isolated areas, and also for preliminary material removal before engraving (2D finishing) operations.

Waterline roughing operation



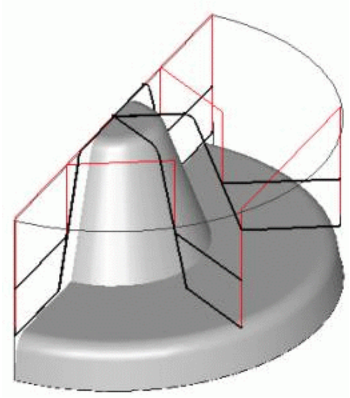
Waterline removal of stock material of a workpiece, which lies outside the 3D model. As in pocketing, the main part of the material is removed by the horizontal (XY) movements of the tool. The operation is often used for primary rough machining of complex models, which have considerable geometrical difference to the workpiece.

Plane roughing operation



Plane 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.

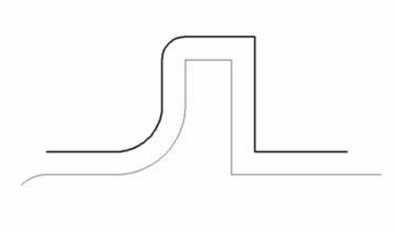
Drive roughing operation



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.

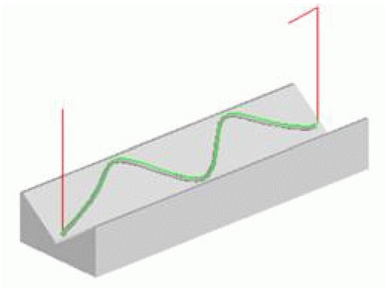
Finishing operations

2D contouring



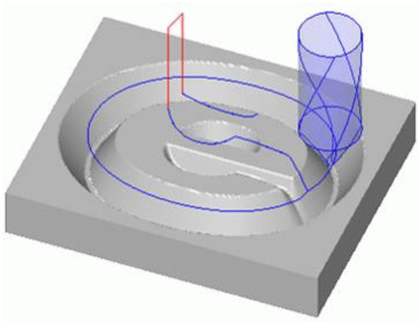
For the machining of horizontal contours or curves projected onto the horizontal plane. The horizontal movements of the tool are created based on the geometry being machined. The tool center or the tool edge can follow the contour. The operation is used for creating parts with vertical sides or for a machining pass with a constant Z depth etc.

3D curve milling



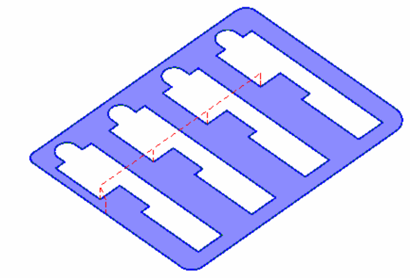
Generates a series of tool movements along freeform curves. The view of the trajectory in plane is similar to 2D contouring - tool movements are constructed with the tool center or edge passing along the contour. The Z coordinate at every point of the toolpath is calculated as a displacement based on 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.

Engraving operation



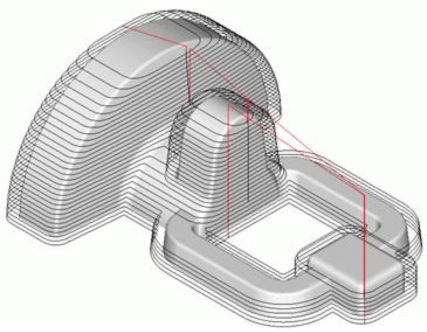
The operation is designed for the engraving of 2D geometry and inscriptions on flat areas. The image being engraved is formed from projections of curves onto the horizontal (XY) plane. Horizontal movements of the tool machine the main parts of the model's side edges. To create the sharp inner corners 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 sidewalls of pockets and for isolated areas during 2 & 2.5D machining.

Jet cutting operation



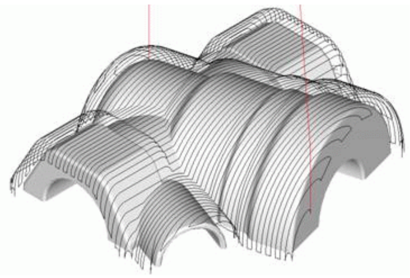
The operation is used to carve the parts from the sheet. The outer contours and the contour of the holes can be defined by any closed or unclosed curve. The carving is performed by the tool motion along the part contours. The holes are cut in first and the outer contour is cut later.

Waterline finishing operation



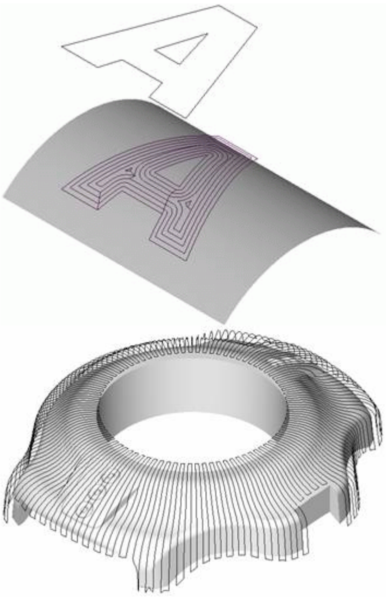
Waterline machining of surfaces of a volume model. Milling is performed by using horizontal movements of the tool. The operation gives a good result when machining models or their parts with their major surface areas that are close to the vertical. For machining of models of high complexity, it is recommended to use the waterline operation together with plane or drive.

Plane finishing operation



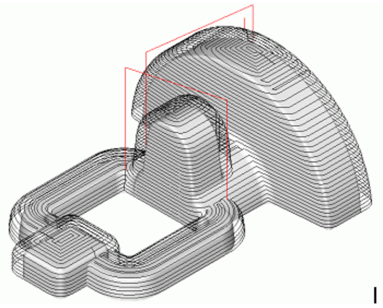
Plane machining of surfaces of a volume model. Passes lie in vertical-parallel planes. A good result can be achieved when machining flat areas and also areas close to the vertical that are perpendicular to the toolpath. Therefore, for machining of complex shaped models this operation is best used with the waterline or other plane operation, which has toolpaths perpendicular to the toolpath of the first operation.

Drive finishing operation



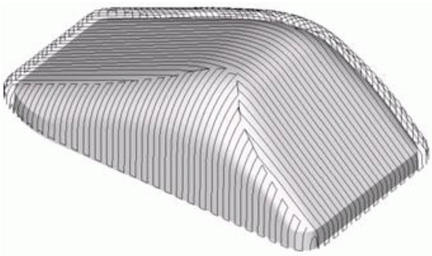
As in the plane operation, surface machining of a volume model is performed by separate strokes. Depending on operation parameters, the strokes lie either in vertical planes or vertical mathematical cylinders, the shape and location of which are defined by drive curves. The operation gives best results 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 operation



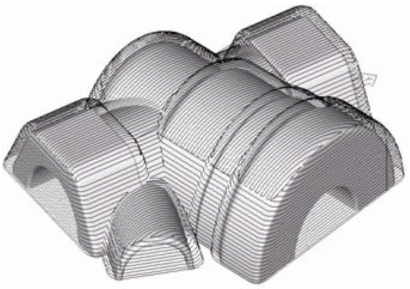
The toolpath for the surface machining of a volume model are formed in two stages. Firstly, the horizontal toolpaths (waterline), and then, for the remaining areas the toolpaths are created by using the rules for the drive operation. Because of this, both flat and steep areas are machined equally well. 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 operation



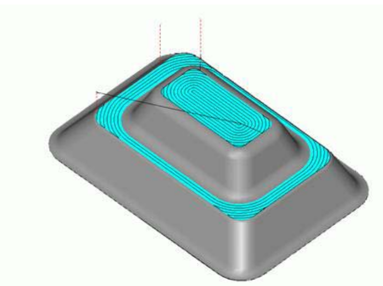
Two plane operations with mutually perpendicular toolpaths are created at a time for surface machining of a 3D detail. The default parameters of this operation are set so that every operation would machine only those surface areas of the model, where it can achieves an optimal result. This means that 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 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.

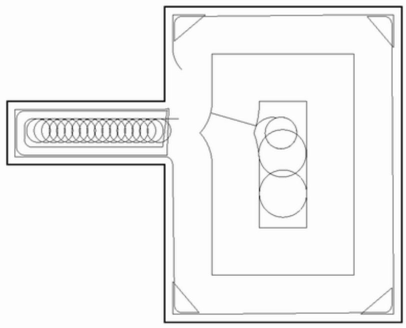
Flat land machining operation



The operation allows making a finish machining of flat horizontal surfaces of a part. The flat segments are recognized automatically. A tool trajectory consists of series of horizontal paths. All non-horizontal segments of model for machining are inspected to avoid gouges during machining.

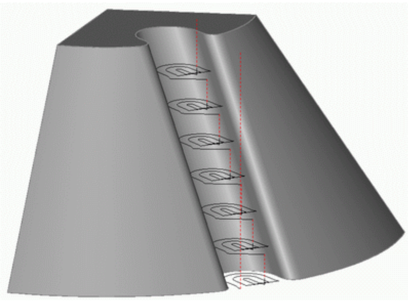
Rest milling operations

Area rest milling



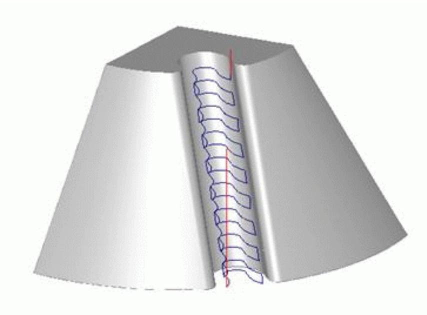
The area rest milling operation performs rest milling of remaining material. That is, it performs a waterline rest milling inside the defined area or pocket. The shape of the area is formed from curve projections on the horizontal (XY) plane. The operation is used for the re-machining of residual material using a tool of smaller diameter than the previous tool for 2 & 2.5D machining of pockets and isolated areas.

Waterline rest milling with clearance



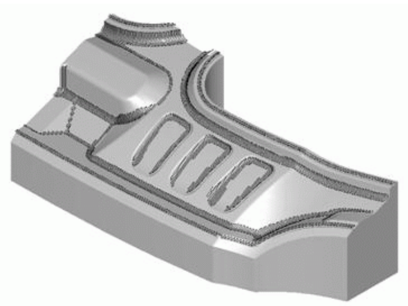
Re-machining of remaining material after a waterline roughing operation. The horizontal passes of the tool cut the remaining material left after previous machining operations that lie outside the 3D model. The most efficient use of the operation is re-machining using a tool with smaller diameter after roughing operations, i.e. as a semi-finishing operation.

Waterline 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 milling

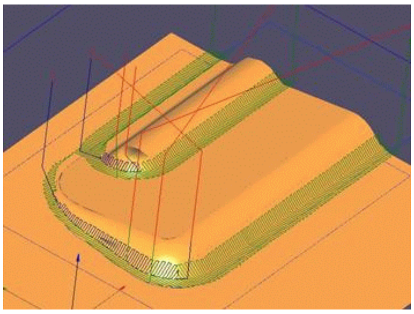


Rest milling of model surface using plane finishing operations. Model surface areas, insufficiently machined by the previous operations are milled using passes lying in vertical parallel planes. The operation is intended for use when re-machining slightly sloping areas and areas close to the vertical that are perpendicular (or close to) the toolpath.

Drive rest milling

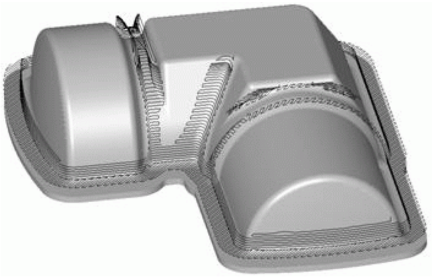
Rest milling of a surface model by drive finishing operations. A model’s surface areas, insufficiently machined by the previous operations are milled using passes lying either in vertical planes or mathematical cylinders. By default, the drive curve should be formed along the unfinished areas, which 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 milling



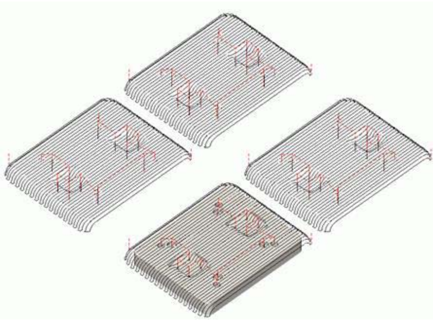
Rest milling of a surface model by optimized plane-finishing operation. Models surface areas, insufficiently machined by a previous operation are milled by using 2 plane operations with mutually perpendicular toolpaths. Because each operation only machines its optimal area, good results are achieved. Rest milling using plane-optimized operation is recommended for use where there are relatively large unfinished areas.

Complex rest 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 component parts. The operation allows to reduce time of calculation and debugging of an NC-program.

5.2.2 Operations group

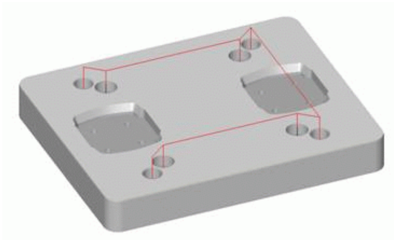
The group is intended for systematization of different operations with similar parameters. Creating a group does not result in any trajectory calculations, but parameters may be set for all operations in the group.

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).

5.2.3 Hole machining



The hole machining operations are designed for drilling, centering, boring, countersinking and tapping. 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 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.

Drill points can also be defined manually. 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 Model window. In the same window the user can access the function of automatic recognition of holes in the model. The machining sequence is defined by the order in the list, if transition optimization is disabled in the Strategy page.

The Z rapid height and drilling depth for every point can be assigned by the user or calculated automatically. When the Z rapid height is calculated automatically, it will be determined according to the workpiece model, and the drilling depth 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.

A transition between the holes being machined is performed via safe plane. Changing to the work feed is performed at a safe distance from the start of drilling. A safe distance for all holes can be assigned in the approach window. The bottom machining level limits the maximum depth of a hole.

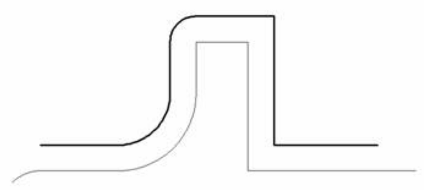
It is understood that the axis of the holes being machined and the tool rotation axis are parallel to the Z-axis. If an operation is performed in the local coordinate system or with a swivel head, then holes axes will be parallel to the Z-axis of the local coordinate system of the operation.

The required machining method is defined in the Strategy window. When machining holes, one of the following drilling cycles can be selected:

- Simple drilling (G81). Rapid approach to the safe distance, drilling and return to the safe plane.
- Face drilling (G82). Rapid approach to the safe distance, drilling, dwelling at the bottom spindle running and return to the safe plane.
- Deep drilling. Rapid approach to the safe distance, cyclic drilling with approach and retraction, following return to the safe plane.
- Drilling with chip break. Rapid approach to the safe distance, cyclic drilling with approach, dwelling in the bottom point and retraction, and return to the safe plane.
- Tapping(G84). Rapid approach to the safe distance, threading, rise on work feed with reverse spindle rotation and return to the safe plane.
- Bore type 5 (G85). Rapid approach to the safe distance, boring on work feed with spindle stopping at the bottom level, feed out to the safe plane.
- Bore type 6 (G86). Rapid approach to the safe distance, boring on work feed with spindle stopping at bottom level, rapid out to the safe plane.
- Bore type 7 (G87). Rapid approach to the safe distance, boring on work feed with spindle stopping at the minimum level and manual feed out to the safe plane.
- Bore type 8 (G88). Rapid approach to the safe distance, boring on work feed with a dwell at the bottom level, spindle stop and manual feed out to the safe plane.
- Bore type 9 (G89). Rapid approach to the safe distance, boring on work feed with dwell at bottom level, feed out to the safe plane.
- Hole machining by spiral. The method is designed for the machining of round holes, the diameter of which is bigger than the diameter of the tool. Machining is performed using helical tool movements (motion of the axes around a circle with simultaneous Z- movement). The diameter of the spiral is selected in accordance with the defined hole diameter and the tool size. Machining of every hole is performed in the following order: rapid approach to the safe distance, tool plunging by spiral at work feed, go around at the bottom level, and retract to the center of the hole and rapid to the safe plane.
- Hole pocketing. The method is designed for machining of round holes, the diameter of which is much bigger than the tool diameter. The material is removed from the hole in layers. The tool spiral plunges to each layer, and then widens the hole to the required diameter by moving round an Archimedes spiral with finishing mill pass round the hole. Machining of every hole is performed in the following order: Rapid

approach to the safe distance, waterline machining of the hole by cycles, consisting of plunge by spiral, hole widening by the Archimedes spiral and finishing pass around, and then retract to the center of the hole and rapid to the safe plane.

5.2.4 2D contouring



The operation is designed for machining along horizontal contours or curve projections on the horizontal (XY) plane.

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 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, with return.

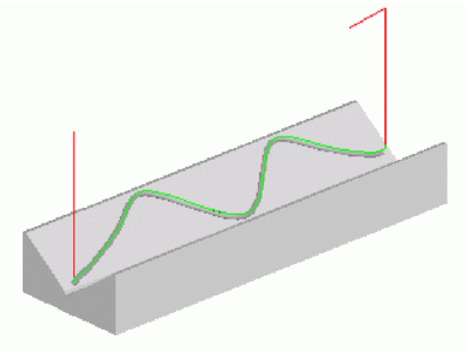
When setting the machining order By Contours each contour will be machined to full depth before moving to another contour. When setting By Depth, each contour will be machined at the current cutting depth, the tool will move to the next depth only after all contours are machined.

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.*

5.2.5 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.

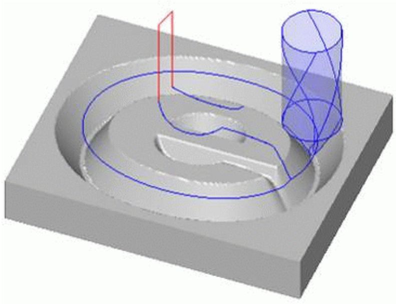
When setting the machining order By Contours, each contour will be machined to full depth before moving to another contour. When setting By Depth each contour will be machined at the current cutting depth, the tool will move to the next depth only after all contours are machined.

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.*

5.2.6 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 sidewalls 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).

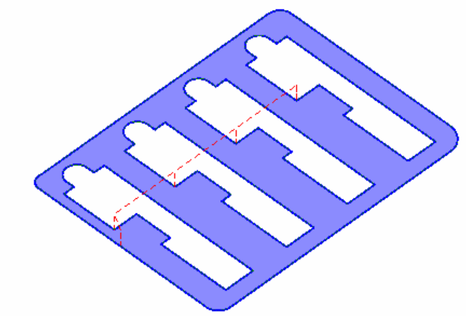
Only those areas, which lie within the workpiece or outside of restricted areas, will be added to the resulting toolpath. The workpiece, machining areas and restricted areas are defined by projections of closed curves. If the workpiece or restricted areas are not defined, then machining of the entire 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). Stepover between passes can be performed along a contour, with generation of intermediate approaches/retractions or via the safe plane.

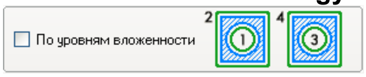
5.2.7 Jet cutting



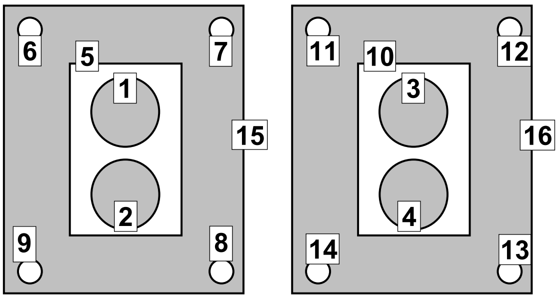
The operation is intended to carve the details from the sheet workpiece. The contours of the detail are defined by the curves projections.

The main difference from the 2D contouring is the machining order. 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.

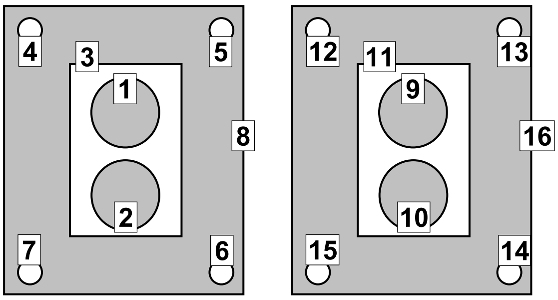
Use the panel below to define the machining order. The panel is located on the **Strategy** page of the parameters window.



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.

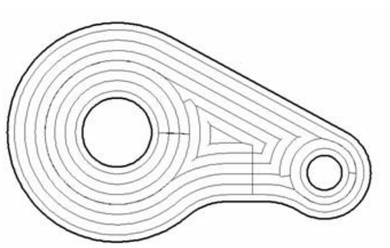
If the operation is performed using a local coordinate system or if using a swivel head then the system performs machining using the XY plane of the local coordinate system, and all work passes are consequently parallel to the XY plane of the local coordinate system.

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.

5.2.8 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.

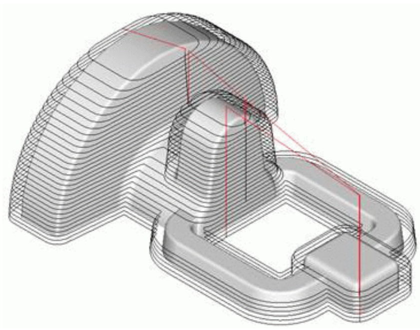
The operation performs removal of the entire material, which lies within the model being machined and outside any restricted areas or a workpiece. The workpiece, machining areas and restricted areas are defined by a projection of curves.

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.

5.2.9 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.

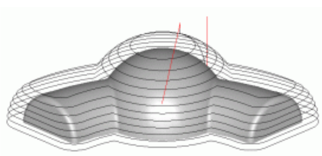
Machining of the model surface is performed using horizontal passes. The step between the passes of neighboring toolpaths can be either fixed or calculated according to the defined height of the scallop.

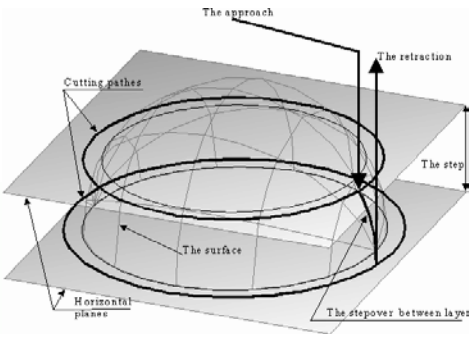
When using a local coordinate system or a rotary axis, the position of the model being machined will not change, the tool rotation axis is parallel to the Z axis of the local coordinate system, and all work passes are located parallel to the horizontal plane of the same system.

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.

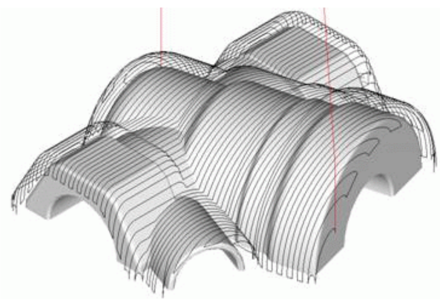
It is also possible to restrict machining of the areas of the restricting model and areas of edges rounding from the resulting toolpath.

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.





5.2.10 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 re-machining 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.

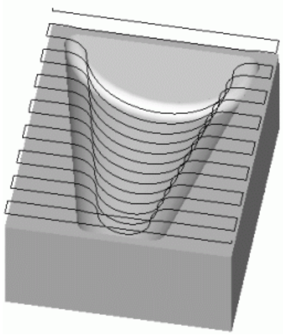
The areas of the model's surface being machined can be limited depending on the slope angle of the normal to the Z-axis. If for example, the user needs to machine flat areas with the slope angle of the normal to the Z-axis less than 45 degrees, then it is advised to set values of the minimum and maximum 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.

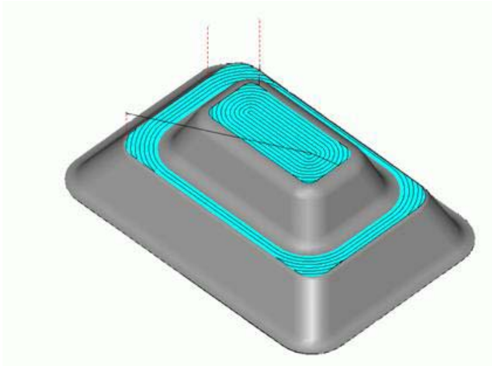
Note: *In order to avoid repeated machining of horizontal areas, the user should set the minimum value of the slope angle of the normal equal to 0 only for just one operation, for others - set it higher (e.g. 1 or 2).*

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.



5.2.11 Flat land machining operation



The operation is expedient to machine a model with horizontal flats. Machining consits 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.

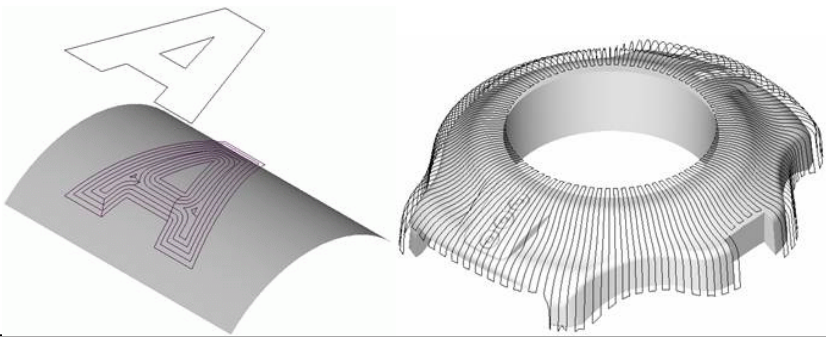
All horizontal segments will be recognized automatically during elements adding in the model for machining. At model preview, these segments are drawn by other color for clearness. All other surfaces of a machining model is inspected, as well as restricted model. The rule allows to avoid part gouges.

The milling type (climb or conventional) is available during a trajectory 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 trochoidal machining.

5.2.12 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 re-machining 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.

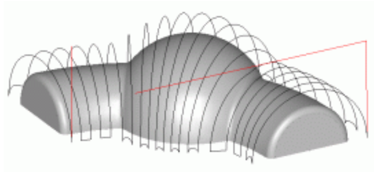
When using a local coordinate system or a rotary axis, the position of the model being machined will not change, the tool rotation axis is parallel to the Z-axis of the local coordinate system, and all work passes are located in planes or mathematical cylinders, perpendicular to the horizontal plane 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.

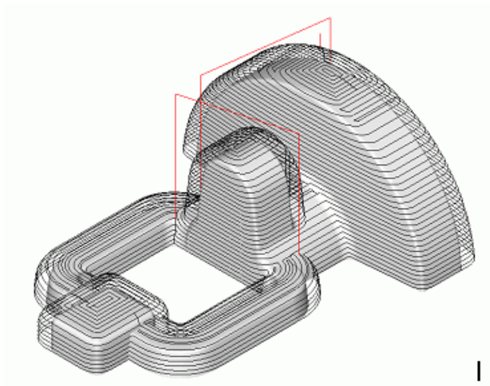
In order to machine only areas that have a small deviation from the normal to the plane of the work pass, it is advised to limit the frontal angle. For example, if one needs to perform machining of surface areas that are nearly perpendicular to the surface of a work pass, then it is advised to set a 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.



5.2.13 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.

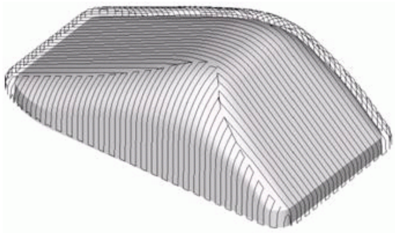
The toolpath for surface machining of a volume model is created in two stages. First, the horizontal tool paths are constructed (similar to the waterline operation), and then, by using the rules of the drive operation, toolpaths are created along a leading curve (leading curves in this case are the borders 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.

5.2.14 Optimized plane operation (plane-plane)



The optimized plane operation consists of two plane-finishing operations, whose passes lie in mutually perpendicular planes. The default parameters for these operations are filled in such a way that each operation only machines an optimal area of the model, this ensures efficient toolpath results. Owing to this, consistent machining quality on the entire surface of the detail can be achieved. Use of the optimized plane operation allows quality machining of models that have a complex surface shape, and also minimizes the machining time.

All parameters of the two plane operations are interconnected; this means that alteration of a parameter of one operation will be automatically applied to the parameter value of the other operation. An exception to this is the angle between the planes of the work passes and the X-axis - for this 'doubled' operation the value of the angle is set in such a way, that the planes of the work passes will remain perpendicular to one another. This requires that both operations be tuned separately. The default sets of parameters for the optimized plane operation(s) are identical to the plane finishing operation.

When creating a pair of plane operations, in order to achieve an optimal result, their parameters are filled in the following way. The angle between the planes of the work passes of the second operation and the X-axis is set to 90 degrees more than it is in the first one. The frontal angle is set to 45 degrees for both operations, so that any sloping area can be machined by one operation only. The minimum slope angle of the normal to the Z-axis, for the first operation is set equal to 0 degrees, and for the second - equal to 1 degree (i.e. horizontal areas will be machined by the first operation only). The maximum slope angle of the normal is set 90 degrees for both operations; this allows machining of all surfaces of a model.

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.

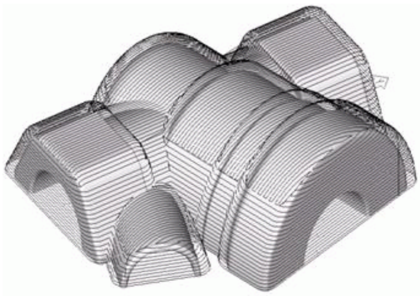
Local coordinate system or a rotary head, the position of the model being machined will not change, the tool rotation axis is parallel to the Z-axis of the local coordinate system, and all work passes are located in planes that are perpendicular to the horizontal plane of the local coordinate system.

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.*

5.2.15 Complex operation (waterline-plane)



The complex operation consists of two finishing operations: plane and waterline. Parameters of these operations are set in such a way, that any flat areas are machined by the plane operation and any areas that are close to the vertical - by the waterline. As a result, the system achieves consistently good machining quality over the entire surface of the model. The complex operation provides easier working conditions for the tool; this makes it possible to use longer tools with a smaller diameter. Use of the optimized plane operation allows the user to perform quality machining of models with a complex surface shape, and also minimizes the machining time.

Parameters for the operations are interconnected; this means that when a parameter is changed in one operation, the same value will be applied to an identical parameter of the other operation if one exists. This rule makes it necessary to adjust both operations separately. The default set of parameters for the plane and the waterline operations are identical to the normal set of parameters for these operations.

In order to achieve an optimal result when creating a pair of operations their parameters are filled in the following way. For the plane operation, the minimum and maximum slope angles from the normal to the Z-axis are set to 0 and 45 degrees for the plane operation and to 45 and 90 degrees for the waterline operation. This means that if an area of a surface has a slope less than 45 degrees, then it will be machined using the plane operation; otherwise the waterline operation will be applied.

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.

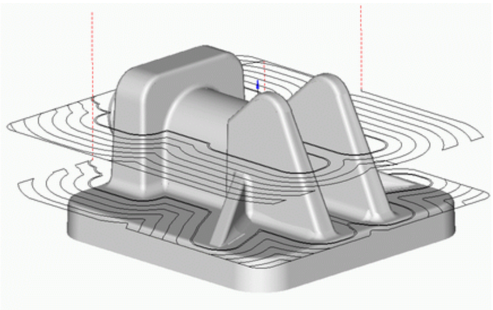
The work passes of the operation lie in two parallel vertical planes. The planes of different operations are perpendicular to each another. The positions of the planes are defined by the angle between these planes and the Z-axis. The step between the planes of neighboring work passes can be either fixed 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.

5.2.16 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.

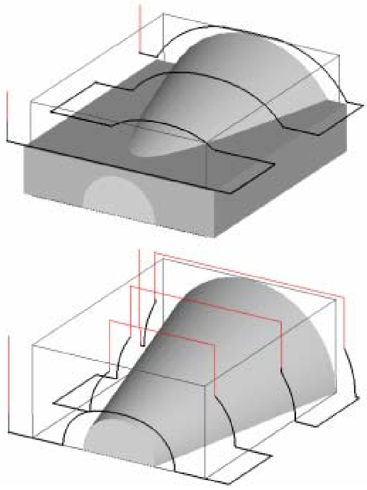
The workpiece can be assigned as a cube, cylinder, a mould with stock or prismatic form, as residual material after machining by previous operations, and also as a free-form geometrical model, consisting of solid bodies, surfaces, meshes and prisms whose bases are projections of closed curves. In the 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.

5.2.17 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.

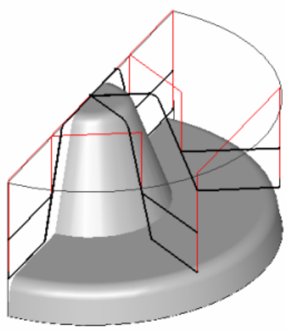
To limit pressure on the tool, the depth of material removed can be defined. If the depth of the material being removed from the workpiece exceeds the defined depth, then the material will be removed in several passes.

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.

5.2.18 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.

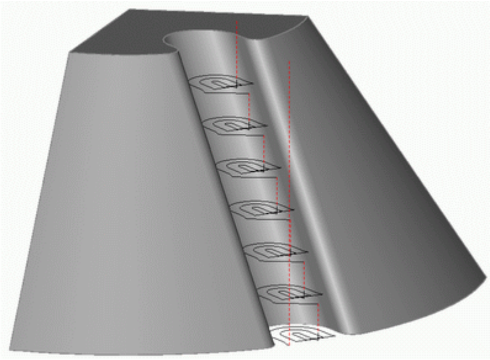
The operation performs removal of the entire material of the workpiece, which lies outside of the model being machined and outside the restricting model. As in the plane operation, separate paths are used to perform surface machining of the volume model. Depending on the operation parameters, the work 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 stepover 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.

5.2.19 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.

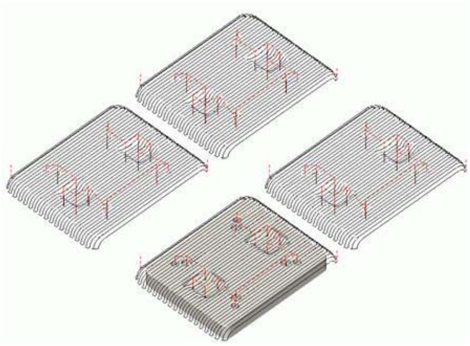
Rest milling of residual material is enabled in the system by selecting the workpiece as material that is left after all previous operations. Calculation of the workpiece and selection of unmachined areas is performed automatically by the system. This approach has some valuable benefits when compared 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. To get to know these refer to:

- Finishing waterline operation
- Finishing plane operation
- Finishing drive operation
- Finishing plane optimized operation
- Finishing complex operation

- List of types of machining operations

5.2.20 Multiply group



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 trajectory 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.
- **Round array**. Repeating patterns of a trajectory place along a circle, with the centre in point X and Y, radius R and the angle pitch given in the appropriate field.
- **Axis symmetry**. Repeating patterns of a trajectory symmetrical about the given axis. A point and an angle within a horizontal plane set the axis.
- **Point symmetry**. Repeating patterns of a trajectory 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 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.

5.2.21 Multi-coordinate machining (controlling rotary head/table)

Multi-coordinate machining can be performed in **SprutCAM** in two ways: by assignment of the angle of the rotary head or by assigning the coordinate system for an operation with the required rotary angles and transitions. In general, these methods differ only in the number of assignable parameters and commands that are written to the NC program to control the swivel head/table. When using a rotary head or table, the system will generate ROTABL commands only if the local coordinate system has a rotary angle position defined. If no rotary angle is defined, then the command will not be created.

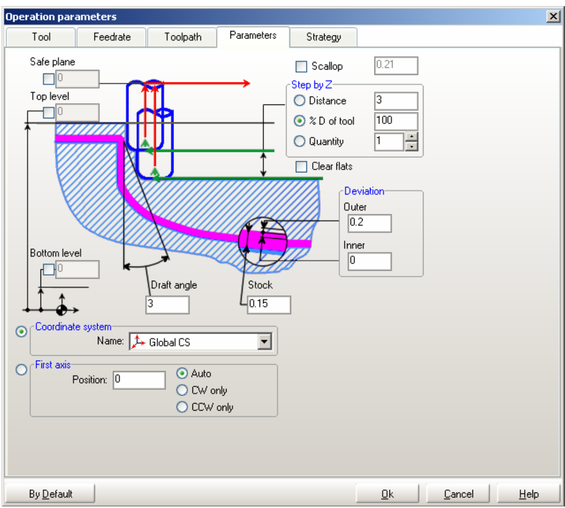
5.2.22 Setting the position for a swivel head

To control the fourth axis - positioning of a rotary head (table) it is firstly necessary to select **Use rotary axis** in the **system settings** window and to define the axis direction and a point it passes through. All values entered must be relative to the global coordinate system. These parameters will be common for all machining operations. The assigned parameters have to be correct during the operation calculation. If for any reason the parameters for the rotary head are changed, then the operations that use these parameters must be recalculated.

If Use rotary axis has been selected, then there will be a new Rotary axis position dialogue on the **Parameters page** in the <**Operation parameters**> window for an operation. In this dialogue the user can define the rotary angle for the head (in degrees) for the current operation. When a new position is entered for the rotary head, the system first searches for a corresponding coordinate system amongst the previously created ones. If a compatible coordinate system is found, then it is set as the coordinate system for the operation, if one isn’t found, then a new coordinate system will automatically be created which corresponds to the defined angle for the rotary head.

During operation calculation, before the toolpath is created, the system adds the **ROTABL** command with the user-defined angle (parameters window). If the command has been correctly processed in the postprocessor, then the corresponding commands for the rotary head (table) will be added to the NC program.

5.2.23 Assigning an operation's coordinate system



To define the coordinate system for multi-coordinate machining for use in a machining operation, one should first create the local coordinate system with regard to all the necessary transitions and rotations. Any local coordinate system can be defined as the coordinate system of an operation. If the position of a local coordinate system has been changed, then all operations that refer to it have to be recalculated. In the operation Parameters page, the user can select the required Coordinate system for the operation. The list of available coordinate systems is similar to the list of coordinate systems in the main window.

5.2.24 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.

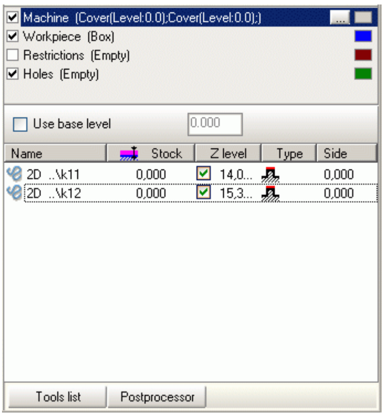
In the 2.5D operations, the system allows the user to visually create the geometry of a model by using flat profiles. If the parameters of a profile are changed, the displayed model is updated automatically. The operation processes the list, which consists of an arbitrary number of profiles and curve 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.

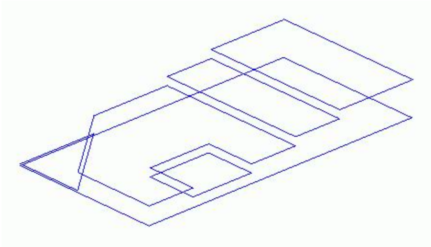
Machining is performed by a series of horizontal passes of the tool. The passes differ from each other only by the depth at which they are located. The number of such passes and their Z depth depend on the machining levels and the step defined on the Parameters page in the <Operation parameters> window.

In the same window, the user can set up the machining tolerance and the stock.

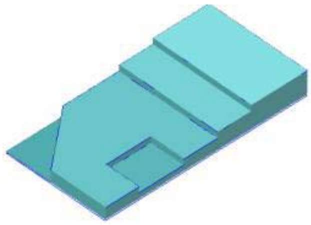
All 2.5D operations use 2D curves to define the model. Formation of the model is performed in the Model window, where one can assign parameters for either a group of elements or a single element.



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.

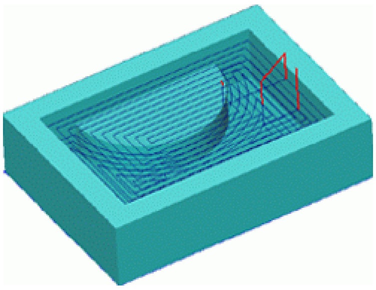


Curves that assign the model



Created model

5.2.25 2.5D pocketing operation



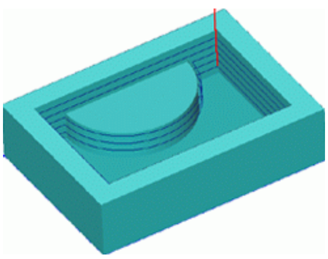
The operation is used for machining of pockets and islands, and for preliminary material removal.

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 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).

5.2.26 2.5D wall machining operation



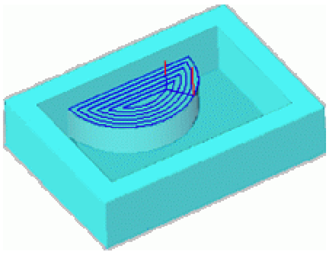
This operation is designed for machining the vertical walls of a models.

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.

5.2.27 2.5D cover machining operation



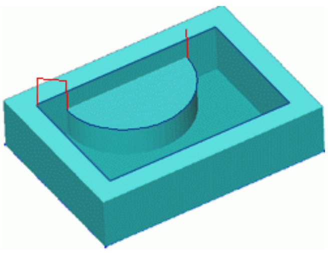
This operation is designed for the machining of horizontal areas of a model - these are known as "covers".

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.

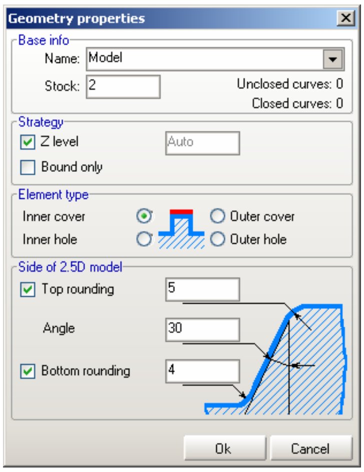
5.2.28 2.5D chamfer machining operation



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 double clicking a curve or point or by pressing the ... button on the <**Side**> field in the <**Model**> tab.



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.

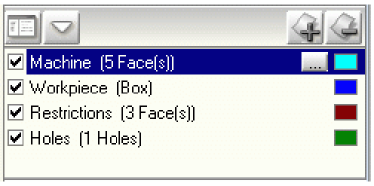
The same should be done for all elements that need to be machined in the operation. The sequence of actions can be performed either on one curve or on a group of curves, if the chamfer parameters for them are the same.

For the operation the user can also assign **milling type**, **corner Roll type**, **curves approximation** and **Step-over type**.

5.3 SETTING UP MACHINING OPERATIONS

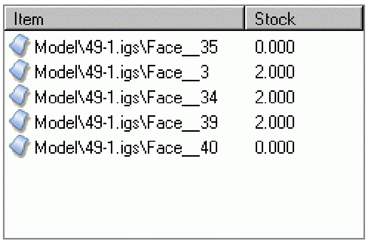
5.3.1 Geometrical parameters of operation

Every operation has a unique set of geometrical parameters. All types of operations have a model to be machined, a workpiece and a restricting area(s). In addition, the drive operations have a set of drive curves, and the drilling ones - a list of hole center coordinates, where drilling will be performed. Assignment of the listed parameters for the selected operation can be made in the bottom part of the "technology" page.



Every tab displays lines which identify a list of objects which form the model, a workpiece and a restricting model. If the current operation is a drive operation, then there will be an extra tabline to assign the list of drive curves. and if drilling - then a tab line to define the list of drill points. The check boxes near the lines allow to show/hide corresponding objects in the graphical window. An object included in a list is shown in the graphical window by the corresponding color. The colors, transparency and mode (shade/wire) of the list can be changes in the setup window. So then when a geometrical object is added to the list, it changes the its own color to the color of the list.

Detailed list of objects which form the selected model, workpiece and etc is displayed under the list of operation parameters or it can be open by the button.



Item	Stock
Model\49-1.igs\Face__35	0.000
Model\49-1.igs\Face__3	2.000
Model\49-1.igs\Face__34	2.000
Model\49-1.igs\Face__39	2.000
Model\49-1.igs\Face__40	0.000

Every item in the list represents the name of a geometrical object. The characteristics of the selected object(s), such as the type, additional stock etc. are displayed in the columns corresponding panel below the list. Number and types of columns depends on the type of a list and on the type of operation. If an object is renamed or deleted, then the reference to this now non-existent object in the list will be lost and will be marked with a question mark.

There are three ways to add an object to the list:

- Select an objects in the graphical window and click the button (see also **Object selection** and **Object Filter**)
- Open the list of all geometrical objects, either imported or constructed in the system by the button. Select objects in the opened list and click the button or drag selected objects to the model list.

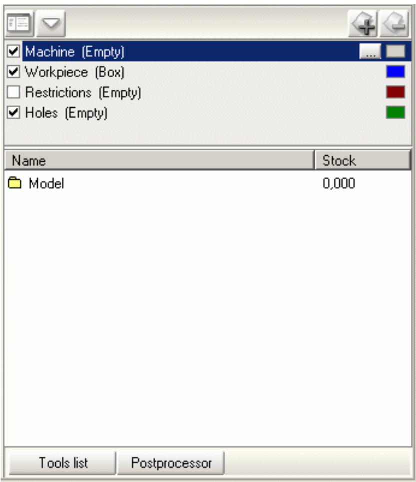
- Press and hold the "**A**" button on the keyboard (add) and click on object in the graphical window. After that the highlighted object is added to the list and changs color.

Similarly there are three ways to remove the object from the list:

- Select an object in the objects list and click
- Select an object in the graphical window and click
- Press and hold "**D**" button on the keyboard (delete) and click on object in the graphical window. After that the highlighted object is removed from the list and change the color.

5.3.2 Model for machining

All operation types require a model for machining. In the geometrical model structure a "Model" folder (group) is created automatically and cannot be deleted. The Model folder is created automatically at start-up, and is the default active folder, into which the model for machining is imported. When a new machining operation is created the Model folder is automatically added to the list for machine. This is set for all machining operations, and cannot be altered. The user can create a list of objects (single objects or folders) for machining.



A model for the **curve machining**, **engraving**, and the **pocketing** operations are created from curves. In all other operations, the model for machining is created from surfaces and meshes. Depending on the current type of operation, the filter allows the user to add references either to curves only or to surfaces and meshes.

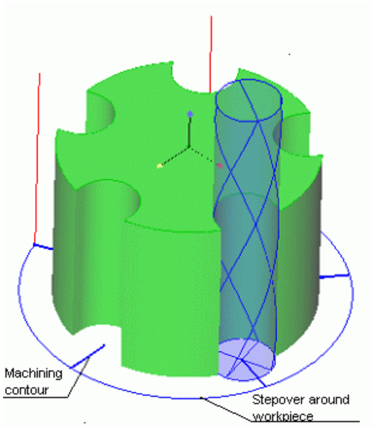
5.3.3 Assigning a model for 2D and 3D curve machining operations

The operations for machining 2D and 3D curves allow the user to create a toolpath from a selected curve. At a very basic level, using these operations, a curve can be transformed into an NC program. Curves that are to be used for creating a toolpath should be included into machine list for the operation. To define a circle from a point, add the point (circle center) on the Model page and assign it an additional stock equal to the radius of the circle.

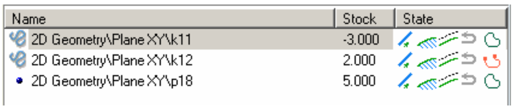
If several curves are selected for machining and idling minimization mode is deselected on the <**Strategy**> page in the <**Operation parameters**> window, then the order of their machining will correspond to the order in the list. To change the sequence of the geometrical objects in the list use the mouse dragging.

The machining direction for a selected curve can be reversed by a tick in the **Inversion** column. When selected, the tool will be displayed at the start of the curve with the curves direction indicated by an arrow.

The toolpath represents a displacement path from the assigned point of the profile being milled. It can be either the bottom center point of a cylindrical end mill, or the center point of a spherical end mill. Switch over is performed on the tool page. The machining trajectory level can be assigned on the <**Parameters**> panel in the <**Operation parameters**> window.

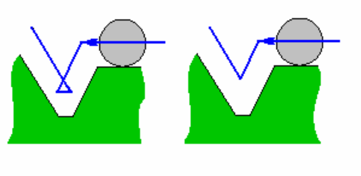


If a tool is required to return along an already machined contour then tick in the **With return** column and the tool will travel along the selected contour, and then travel back along the contour to the original point. The picture above shows an example of machining along a contour with return. The stepover from contour to contour is performed around the workpiece at the same level as the work contours.

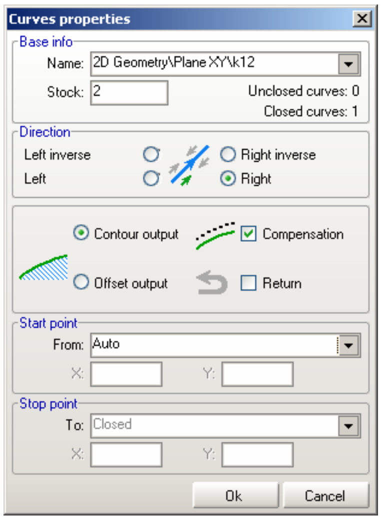


If a tick is placed in the Compens. column, and a compensation switch cut is selected in Toolpath approach and departure, then cutter compensation commands e.g. G41,G42,G40 and offset command e.g. D, will be added to the NC program. When Compens. is selected, then the Curve option should also be set.

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 **properties** window by double click on item or from the popup menu. This window contains the detailed information about item.

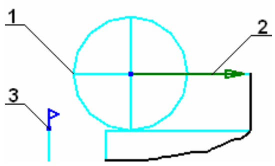


The start point can be input by coordinates or selected from the list. The list contains the providential constants:

- **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 arrear near the selected item.



1 – tool

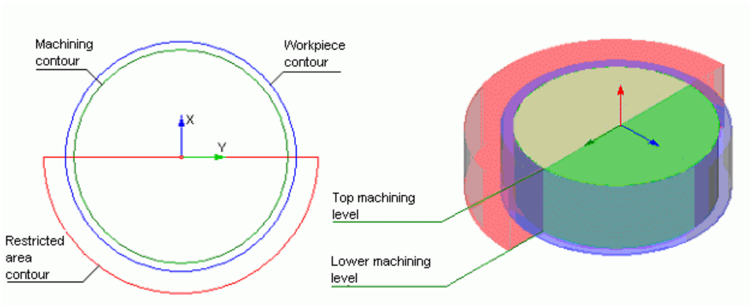
2 – machining direction arrow

3 –stop point

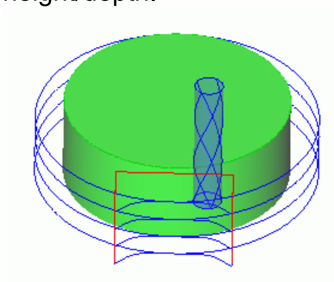
The start point is shown by the tool (1) and machining direction arrow (2). There are two kind of the stop point. - if the curve is closed. - if the curve is unclosed. While the points drugging

To define the start point click on the tool in the view, hold and move the mouse. The tool will move with the mouse. Click on the arrow to change the machining direction. The end point can be defined interactively too. Drad-and-drop the to change the stop point. If the stop point is under the knot point of the curve then the flag take the  form.

One of the features of 2D machining in **SprutCAM** is waterline machining along a defined contour. The depth of cut can be assigned either directly in millimeters, or as a percentage of the tool diameter, or by the required number of steps. The location in Z of a contour does not matter, the top and the bottom machining levels can be assigned on then <**Parameters**> panel in the <**Operation parameters**> window. Climb or conventional milling, and also whether an offset toolpath (left/right) is required can be selected on the Model page by selecting the options "Side left/right" and "Inversion" columns. The location of the tool can be displayed in the graphics area if the user tick the "Machine" line, And using this switch at the included button  it is possible to observe in the transparence mode workpiece in a restricted areas.



All profiles are defined as flat contours (see above left). The machining contour must not coincide with the workpiece contour. The other picture, above right, shows the (shaded) 3D image that will be displayed in the SprutCAM graphic window. The **Top Level** and **Bottom Level** values entered on the <**Parameters**> page define the height/depth.

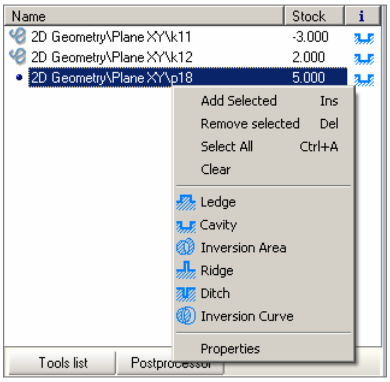


In the **Parameters window** on the "**Parameters**" tab, if the Z step-Quantity value is set to more than one, or the **Distance** value (depth of cut) is less than the distance between the **Top Level** and the **Bottom Level**, then the waterline-machining mode is entered automatically. When defining a specific number of cuts, the distance between the layers will be defined automatically. If the user defines a specific depth between passes that is not equally divisible between the top and bottom levels, then the system will automatically add an additional pass at the **bottom Level**.

2D waterline machining Strategy

All rules previously described for single pass (depth) machining are true for waterline machining. The step-down between passes (plunge), approach/retract moves, corner rolling and all other machining requirements for every pass will be performed as per single-level machining.

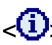
5.3.4 Creation of a model for machining with the engraving or pocketing operations



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.

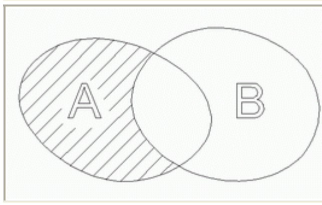
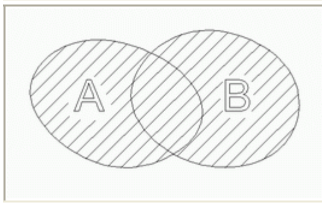
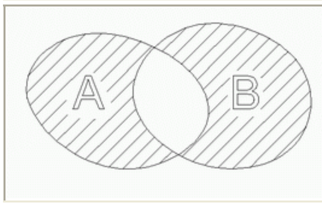
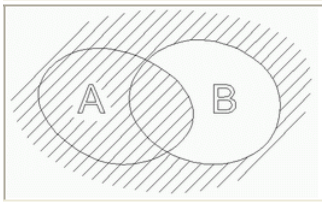
A "**Ledge**", "**Cavity**" or "**Inversion area**" can only be defined by closed curves. When an open curve is selected, then these types will be unavailable. If a closed contour consists of several fragments, then it must be joined into a single curve (see **Curve joining**). A "**Ridge**", "**Ditch**" 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 <> icon appears near the curve. The window is shown below is appear if the mouse cursor under this icon. The window is intended to view and edit the item parameters.

Property	Value
Stock	-3.000
Contour type	Cavity

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.

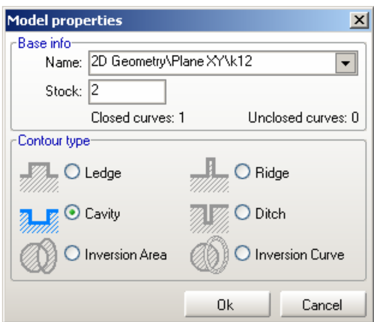
The results of area formation by two curves are shown in the pictures below. Model (solid) areas are shaded.

List contents	Resulting area
A – ledge B – cavity	
1. A – ledge 2. B – ledge	
1. A – ledge 2. 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.

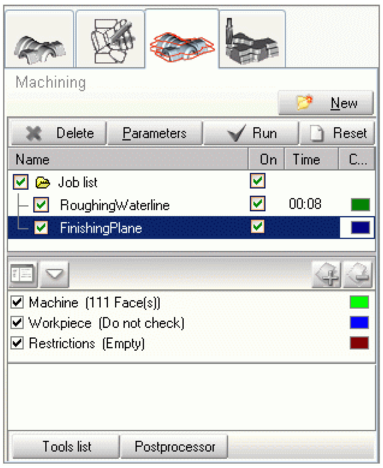
The order of the geometrical objects in the list can be changed by mouse dragging. The model being formed can be dynamically displayed in the graphical window.

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 popup 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.*

5.3.5 Defining the model for volume machining operations



Volume machining operations use a model for machining that is defined by surfaces and meshes. The following 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**.

The order that objects appear in the list does not matter and has no effect on the toolpath(s). For every geometrical object an additional stock can be defined in the "Stock" column. This is advised for use when separate surfaces of a model have to be machined with a different stock amount. If all surfaces have to be machined with an equal stock, then it is advised to use the stock option (parameters) for the operation. When calculating the toolpath, the defined additional stock will be summarized with the stock for the operation.

***Note:** In order to machine separate surfaces of a model, without cutting (gouging) the model, it is recommended to put these surfaces into the 'model' for machining list, and the whole model into the restricting list. By doing this, any surfaces located simultaneously in the 'model' and in the 'restricting' lists, will be machined, and will not be restricted.*

5.3.6 Defining a model 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 to visualize the 'depth' of the geometry. SprutCAM allows the construction of such models using 2D contours and automatically displays the volume model.

After the curve addition the <ⓘ> icon appears near the curve. If the mouse cursor is under the icon then the window is shown below is appears. The window is intended to view and edit the item parameters.

Property	Value
Stock	-3.000
Z level	0.000
Type	B

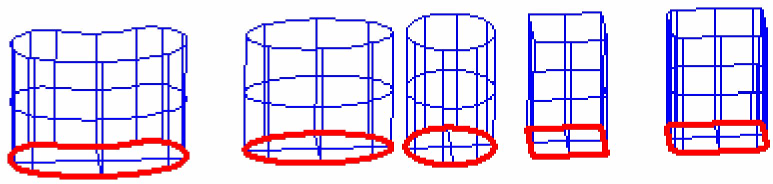
The window is closed automatically when the cursor leaves the icon.

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.

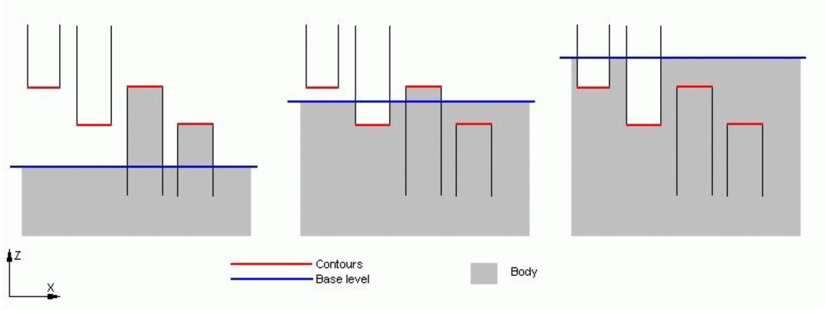
The '3D model' is constructed from 2D (flat) contours lying on different levels. There are two methods to add such areas: a 'cover' - "adds" material from the very bottom to the area level, and a 'hole' - removes the entire material from the top to the area level. This means that the 'side wall' for 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).

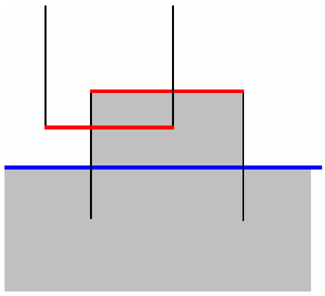


Of course, if hole-forms are located in empty space (or above the base level without other constructions), then they cannot press anything out, Likewise, cover-forms, which are located inside material (or below the base level without holes) cannot fill anything. Whereas, cover-forms located above the 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.

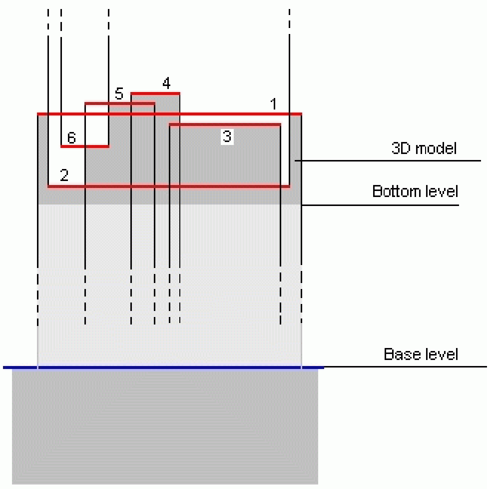
Left - Hole created first, then the cover. Right - Cover created first, then the hole.



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.

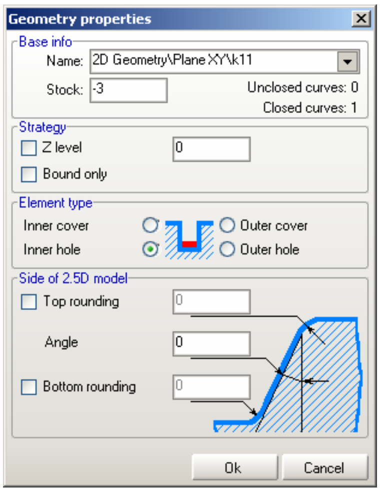
An example of such model is shown on the picture.



The Numbers define the sequence of actions:

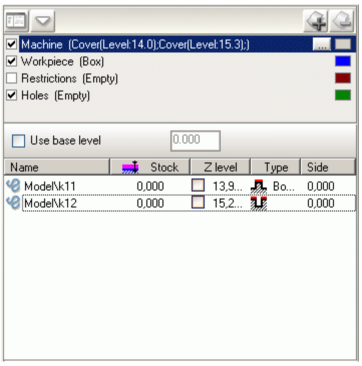
- creates a cover for the outer profile of the model;
- presses out a hole;
- creates a cover inside that hole;
- creates a cover higher than the level of the first cover;
- another cover, which intersects with the one created in article 4;
- presses out a hole in the last cover.

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 popup menu. The dialog is shown below:



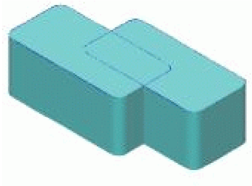
It is possible to set the wall shape in the window also. The angle defines the wall slope in degrees. The top and bottom fillets can be defined also.

Let's practice some techniques for the visual model construction.



In the "2D Geometry" mode construct two rectangular intersecting contours.
Switch to the "Machining" tab and select the "2.5D area pocketing" operation.
Open the "Model" window and add the two contours into the machining list of that operation. All constructions of a visual model are performed by using the commands and parameters available on the panel shown to the right. As was said above, any contour can form both a cover and a hole. In many cases the use of a base level is not required. The user can form the visual model by using an outer contour that forms the body, and adding or removing pockets (holes) or covers.

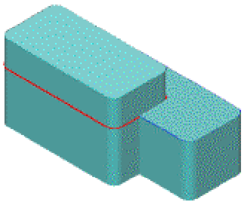
Activate the first contour, set the type to "Cover", and assign the contour level - to "0". If the contour level has not been defined, then it is considered, that its level coincides with the maximum Z coordinate of that contour.
Activate the second contour and set the same parameters as they are for the first one. The bottom level of the model is defined by the "Bottom level" parameter in the "Parameters" window.
Assign its value equal to -20.



Put a tick in the Preview box. The  button must be pressed.
There should be a model similar to that shown in this picture:
We can represent parameters and their conditions in a chart:

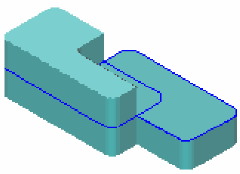
Contour 1	cover	level 0
Contour 2	cover	level 0

Thus, we have created one variant. Using both contours as covers, we have created a model. Both contours lie on the zero level, the bottom part of the model is at -20.



The next step - selects the first contour and in the "Level" field set the value equal to 10. As the result, the contour will be located on the level 10; the visual model will change accordingly.

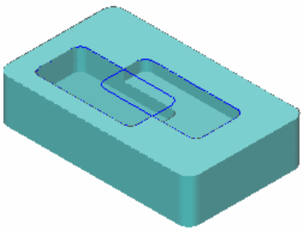
Contour 1	Cover	level 10
Contour 2	Cover	level 0



The same contour can be used several times.

Contour 1	Cover	level 10
Contour 2	hole	level 0
Contour 2	Cover	level 0

The result is shown on the picture
Add one more contour with number 3 to the list



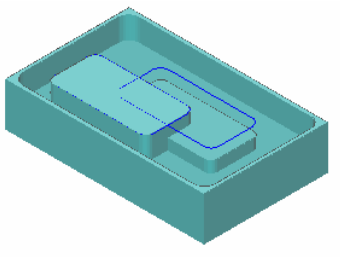
Contour 3	cover	level 5
Contour 1	hole	level 0
Contour 2	hole	level -10

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, contour1 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.

Examples for the use of Base level.



Activate the Base level mode

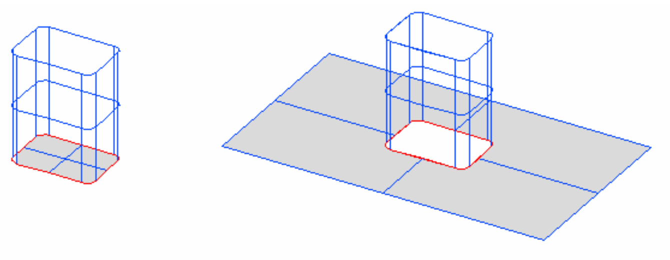
Make the base level 0.
Create a list of contours with these parameters:

Contour 3 hole level -10
Contour 1 cover level 0
Contour 2 hole level -5

The result should be similar to the one shown.

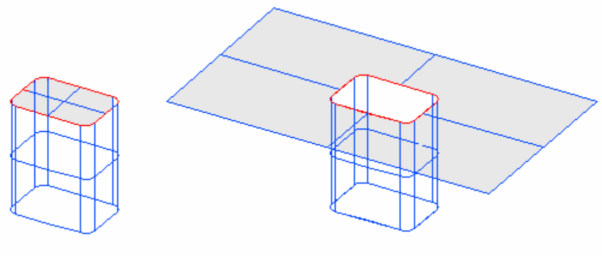
Model example with use "**Outside**" parameter.

Until now we have been dealing with pressing holes and pulling covers with the "Inside" parameter. Now we shall learn the application of the "Outside" parameter. Once again, if we make a comparison to the construction of sand figures, i.e. represent contours as shapes for working with sand, then, when we select the "Outside" parameter, we invert the work area of the tool. The pictures below show the different forms created by the same contour using the "Inside" (left) and "Outside" (right) modes.



Inside Outside

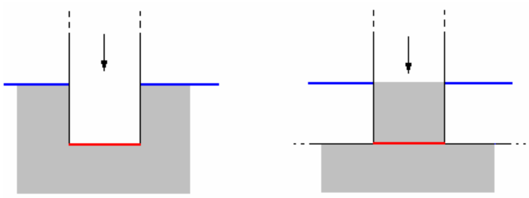
Forms for pressing out holes



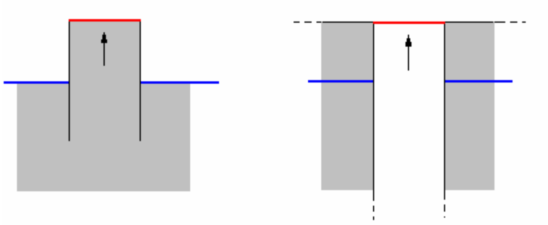
Inside Outside

Forms for pulling covers

See what happens when switching between modes

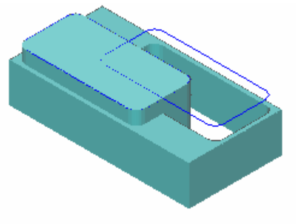


In the left picture there is the pressing of a hole by a contour set as "Inside", on the right the pressing of a hole by a contour set as "Outside".



Cover pulling is shown in this picture. Left is "Inside" parameter, right is "Outside".

Here is an example of using "Outside" parameter.



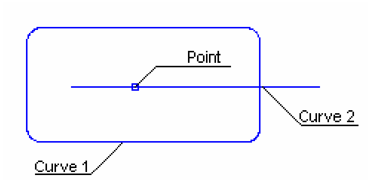
Create a list of contours with parameters defined as below.

Contour 1 Cover level 0

Contour 2 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.

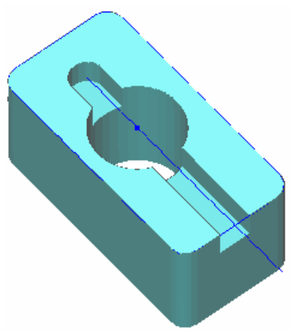
In 2D geometry create one closed contour, one open and one point, similar to the picture below:



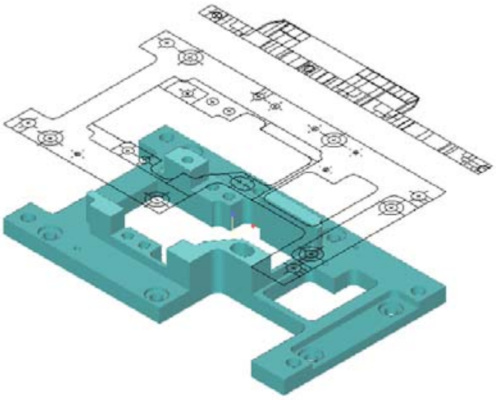
Create a list of elements with the parameters defined below:

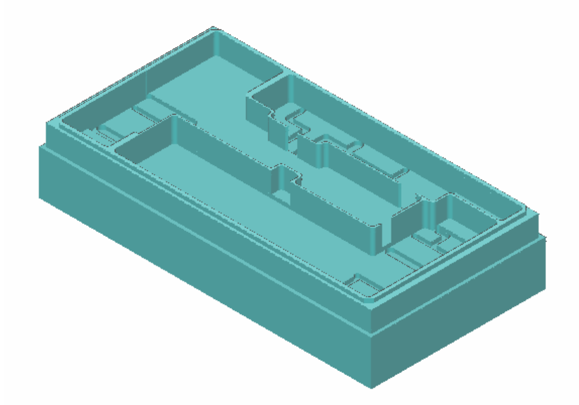
1. Contour 1 Cover level 0
2. Point 1 Hole level -20 additional stock 8
3. Contour 2 Hole level -5 additional stock 3

The result should be similar to the picture below:



Examples of visual model construction for 2.5D machining.

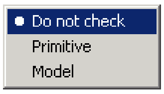




5.3.7 Assigning workpiece

A workpiece is an inherent parameter for all types of operations, both rough and finishing. In the roughing operations, the workpiece defines the initial shape of material from which will be machined by the tool. In the finishing operations, the workpiece model defines the areas, which require machining. To define a workpiece, select the **Workpiece** line in technological operation list. With **Preview** mode active, the workpiece being defined is highlighted in blue in the graphic window.

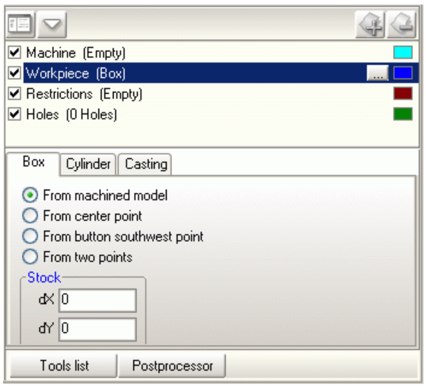
There are several ways of defining the workpiece geometry. The required method is selected from the drop-down list box.



- **Without workpiece**.

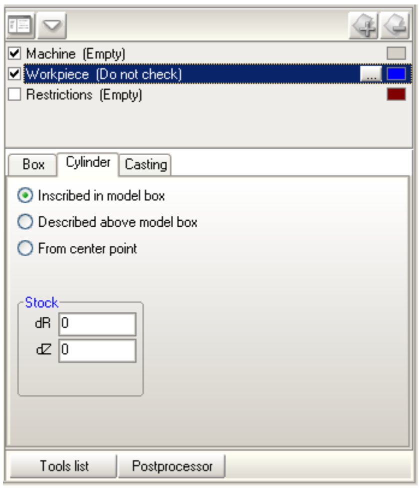
The "**Do not check**" mode is available only for finishing operations. Using a workpiece assignment method, the system performs machining of the entire model. Different results in roughing operations can be achieved by defining a workpiece that is bigger than the model for machining.

- **Box**.



The "**Box**" mode allows the user to define a workpiece as a cube. The edges of the cube are always parallel to the planes: **XY**, **XZ**, **YZ**. The dimensions and the size of the cube can be defined using several methods: When the from machined model option is selected, the coordinates for the corner points are calculated by the system so that the entire model for machining can fit inside the cube. An additional stock amount can also be added for the X and Y axes. When the from center point option is selected, the coordinates for the centre point of the cube and the overall dimensions are required. When defining the position and dimensions of the box using the from bottom southwest point option, the user should enter the coordinates of the corner point and the overall dimensions for the cube. When using the box from two points option, the coordinates of the diagonally opposite corners of the cube should be entered. In all cases, the size and the position of the workpiece in the Z axis is defined by the top and the bottom machining values entered in the **Operation parameters** window on the **Parameters** page.

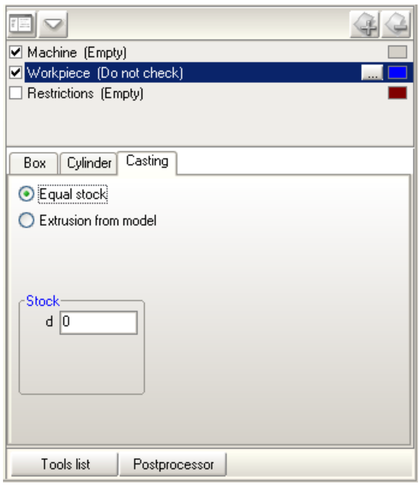
- **Cylinder**.



The "**Cylinder**" mode allows the user to define a cylindrical workpiece. The axis of the cylinder is always parallel to the tool axis. The height of the cylinder and its position along the Z axis is defined by the top and the bottom machining levels, that are entered in the **Operation parameters** window on the **Parameters** page. The diameter and coordinates of the cylinder axis can be defined by three methods:

- When defining a cylindrical workpiece by **Inscribed in model box** the coordinates for the center and the diameter are calculated such that the cylinder touches the inside edges of a theoretical box that contains the model.
- When defining a cylindrical workpiece using the **Described above model box** option, the coordinates for the center and the diameter are calculated so that the cylinder touches the outer corners of a theoretical box that contains the model.
- The **From center point** method allows the user to define the coordinates of the center and the diameter manually.

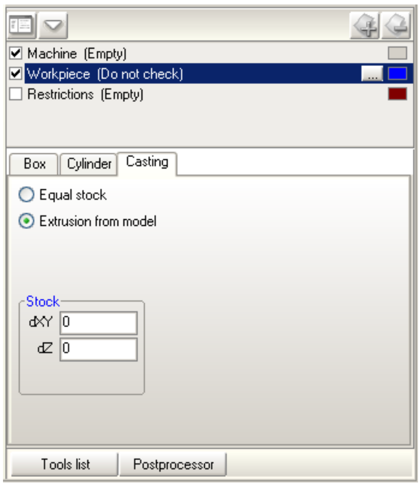
- **Casting**.



The "**Casting**" mode allows the user to assign a workpiece that is similar to the geometry of the model. This workpiece definition method is only available in volume machining operations.

- When defining a workpiece with **equal stock**, an equal stock amount is added to the model for machining, and this is used as the workpiece. The stock value is defined in the <**d**> field.
- When defining the casting as an **Extrusion from model**, a vertical extrusion of the outer edges of the model is used as the workpiece. The height of the extrusion is defined by the top and bottom machining levels entered in the parameters window. If a stock value is entered in the dXY field, the outer edges of the extrusion are offset by the defined distance.

- **The formation of a workpiece as an extrusion**.

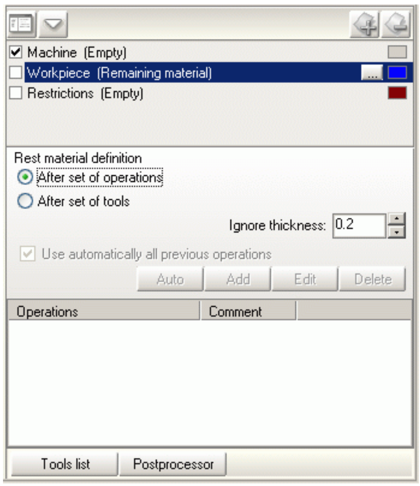


Is performed in a similar way to the model definition for the engraving and pocketing operations. The only difference is, that the curves which form the workpiece, have to be closed and form the outer or inner edges of the extrusion. The height of the extrusion is defined by the top and bottom machining levels, which are entered in the parameters window. Here, just as when defining a model for machining using curves, it is important to observe the order of the elements in the list. The outer border is similar to a ledge in a model for machining and inner - to the cavity. If the first element in the list is defined as an inner border, the workpiece model will be limited by a rectangle that contains all of the elements of the model. A workpiece defined by curves can be dynamically displayed in the graphic window. To activate this, the user should select the "Preview" option (shade mode must be active).

In the geometrical model section of the system there is a group called "Workpiece" that is created automatically and cannot be deleted. To use an imported model as a workpiece, it is recommended to place it in the "Workpiece" folder. The easiest way to do this is to make the "Workpiece" group active and then perform the import operation. When a new object is created/added to the list of objects that define the workpiece of an operation, a reference to the "Workpiece" group will be automatically added.

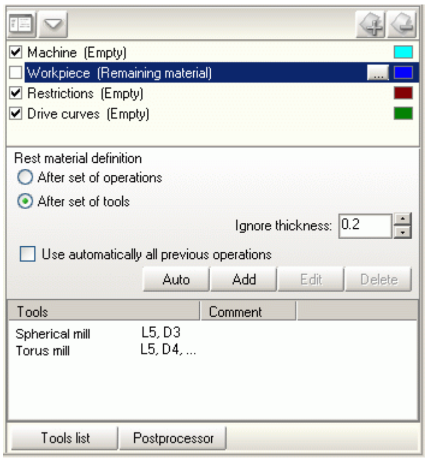
A default workpiece is created automatically, the type of which differs depending on the sort of operation selected. For roughing operations the default workpiece is set as a box from machined model. For the finishing operations the default is set as "without model". For the "rest milling" operations the default workpiece is set as remaining material.

- **Remaining material**.



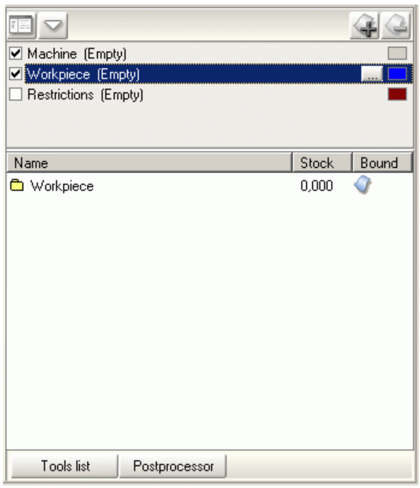
The "**Remaining material**" mode allows the user to define a workpiece as the material that remains after previous machining operations. A workpiece defined in this way, allows rest milling of residual material in different operations. When defining a workpiece by this method, before the toolpath calculation is started, the system analyses the cutter paths of all previous operations, after which it creates the geometrical model of the workpiece. In the rest above field the user can define what is best described as a 'stock' amount. If the remaining material protrudes outside of the model + stock amount, then these area's will be machined. Use of this parameter is required for avoiding unnecessary machining of insignificant area's left after a previous operation, e.g. scallop. Normally, the "**Rest above**" value should be more than the scallop value used for previous machining operations. The workpiece for the "**remaining material**" will not be dynamically displayed in the graphic window. To evaluate the shape of the workpiece, which will be created as a result of machining by previous operations, use the "**Simulation**" mode for those operations and from the resulting shape estimate the remaining material.

- **After set of tools**.



The system can perform rest milling of remaining material. The rest material can be calculated based on either material remaining after previous operations, or, as 'virtual' material remaining based on the tools in the tools list. The list is defined in the Model window on the Workpiece tab. The tools list can be automatically selected from the previous operations, or it can be manually altered in this view.

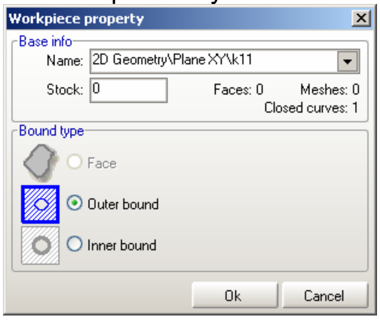
- **Geometrical model**.



The "**geometrical model**" mode allows the user to create a freeform workpiece from existing geometrical objects. In general, the freeform geometrical workpiece model can be created by a combination of elements of two types: extrusions, the basis of which is defined by a set of curve projections, and surface objects (solid bodies, surfaces and meshes). For the curve machining operations, engraving operations and pocketing operations, the workpiece can be defined by extruded curve projections only; solid bodies, surfaces and meshes are ignored.

Workpiece assignment by surfaces and meshes is performed in a similar manner to the definition of a model for volume machining operations. Use the and buttons to add or remove surfaces and meshes from the workpiece definition. The order that objects appear in the list is not important and does not affect the machining toolpath. An additional stock amount can be defined for every geometrical object or group of objects.

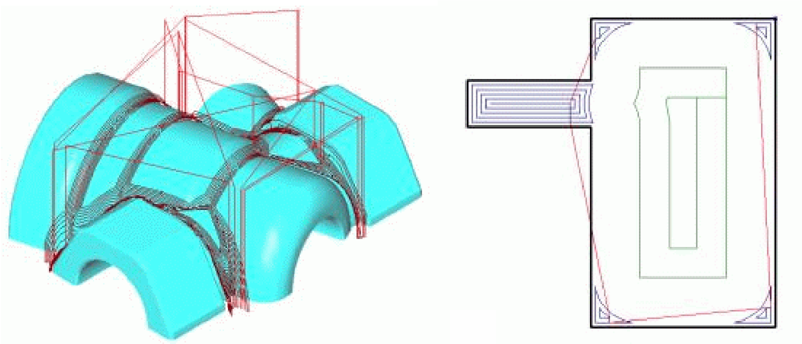
The item parameters are defined in the properties dialog. The dialog can be opened by double click on the item or from the popup menu.



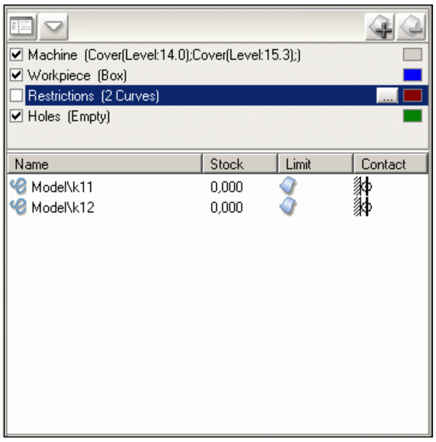
The window contains the detailed information about selected item.

***Note**: Additional stock is useful when assigning a workpiece as a casting or punch. To do so, use the model as the workpiece and add the required stock amount for the casting.*

Examples of rest milling after 'virtual' machining.



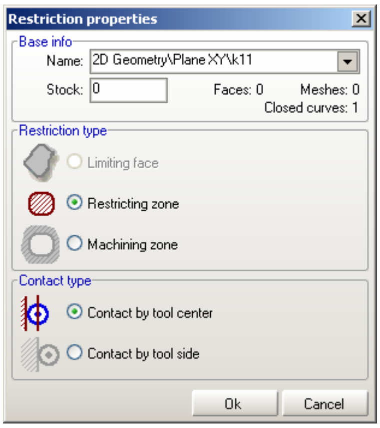
5.3.8 Assigning machining limitations



The restricting model is defined by a list of geometrical objects, which must not be machined. The restricting model should describe the geometry of equipment (vices/clamps etc.) that lie within the machining area, or describing the entire model when machining its separate elements.

To define the machining limitations as geometrical objects the user should open the **Model** window and select the **Restrictions** tab. For all types of operations, the restricting model can be defined by using closed planar (flat) curves. For machining operations that use surface/solid models the restricting model can also be defined by surfaces and meshes. If the restricting model is defined by using curves, then transitions over the restricted areas will be performed via the Safe plane.

The model item parameters are defined from the properties dialog also Double click on the item or select “properties” from the popup menu to open the dialog



5.3.9 Defining limitations using surfaces and meshes

The addition and removal of restricting meshes and surfaces is performed by using the  and  buttons in the Model - Restrictions window. The order that the objects appear in the list is not important and does not affect the toolpath. For every geometrical object the user can define an **Additional stock** which is an extra safe distance around the object(s).

Note: *If the same geometrical object is defined in both the model and restricting lists, then this object will be machined. This method allows a simple way for defining separate surfaces for machining whilst still considering the entire model. To do so, it is advised to put the entire model into the restrictions list, and the separate surfaces for machining added to the "Model" list.*

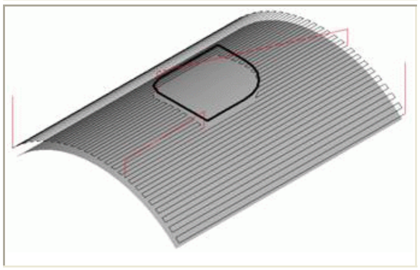
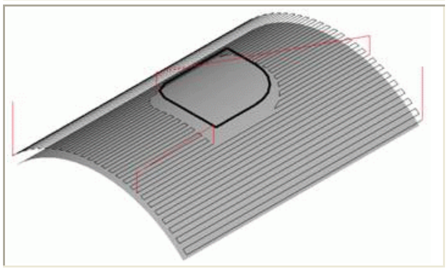
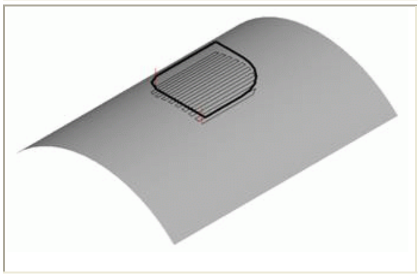
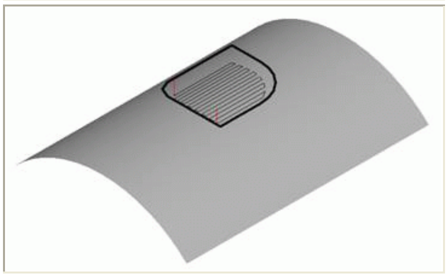
5.3.10 Assigning limitations using curve projections

Assigning limitations as closed curves. The restricting area is formed in a similar way to the model definition for the engraving and pocketing operations.

- **Restricting area** –the area, inside which machining is prohibited. The tool moves over it at the safe plane at rapid speed;
- **Machining area** – the area, outside which machining is prohibited.

The behavior of the tool when it contacts the restricting area can be altered thus:

- **Tool center** – the tool touches the area boundaries using its centre.;
- **Tool contour** – the tool touches the area boundaries with its edge and never goes beyond the area borders.

Curve type	Contact by tool center	Contact by tool contour
Restricted Area		
Machining Area		

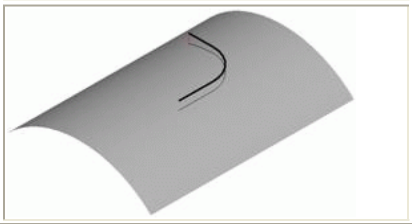
Any movement above restricted areas is performed via the safe plane only. The restricted area's defined by planar curves can be dynamically displayed in the graphic window. To activate this, put a tick in the **Preview** field in the «**Geometrical model structure window**»

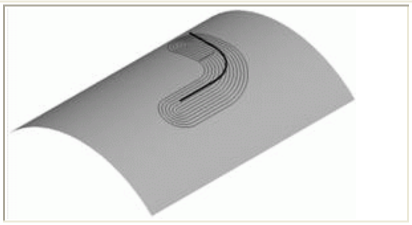
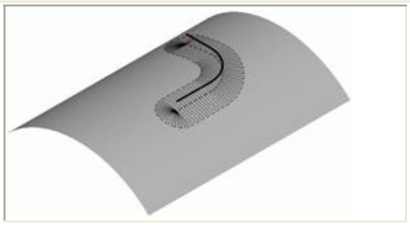
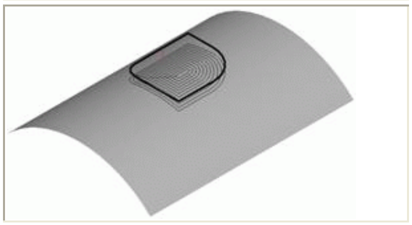
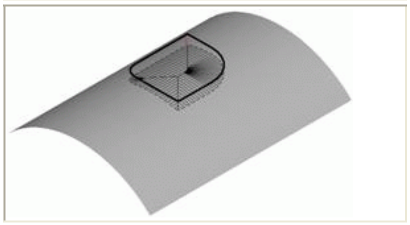
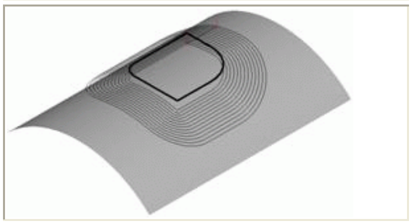
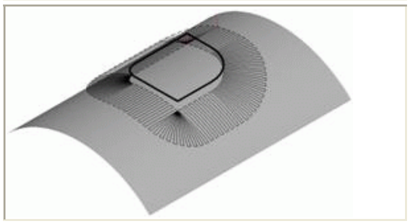
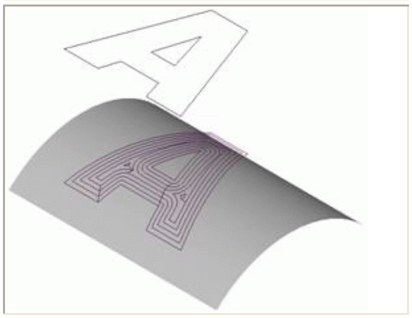
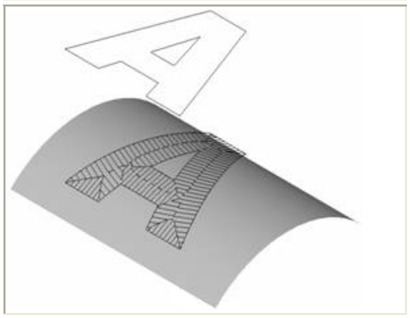
5.3.11 Assigning drive curves for drive operations

In the finish and rough drive operations the tool transition rule in a plane is defined by drive curves. To define the drive curves, the user should open the Model window and choose the drive curves tab. 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. To construct the area, all the rules defined in the "Creation of model for machining using the engraving and pocketing operations" should be followed.

The drive area formed by the curves can be dynamically displayed in green in the graphic window. To activate this mode, put a tick in the "Preview" box (shade mode must be active).

In the drive operations there are two main types of tool movement: 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	Trajectory when machining along the drive curve	Trajectory when machining across the drive curve
Ditch		Not recommended

Ridge		
Cavity		
Ledge		
Outer curve-cavity, Inner curve - ledge		

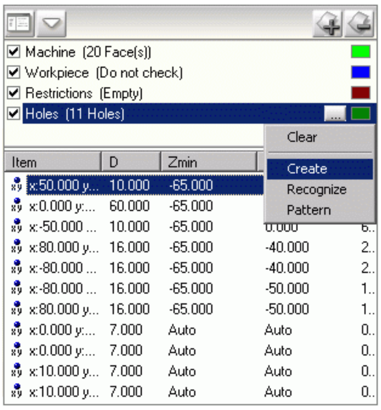
If no curve is defined on the drive curves tab, then the drive area will be calculated by the system. The default method of area calculation depends on the type of drive operation.

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.

5.3.12 Defining hole parameters



When defining the parameters for the waterline roughing operation, pocketing operation and 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.

In the waterline 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 versa, to use the holes obtained for tool plunging, the operation can be defined as a prototype for the waterline roughing or pocketing operations.

To create a hole press the <...> button and choose method of the creation hole: <**Create**>, <**Recognize**>, <**Pattern**>.

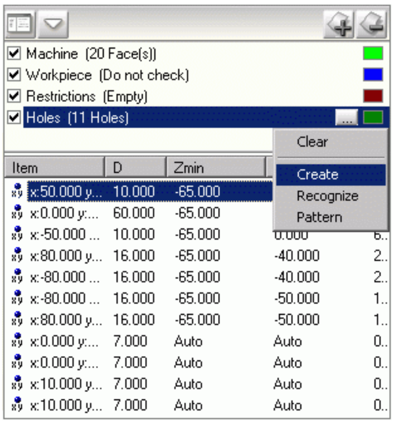
Each hole is defined by the coordinates for it's center, the diameter and also the value of the upper and bottom levels. There are two methods to define the center coordinates of holes: by coordinates or by a geometrical "point" object.

Regardless of the center definition method used, the depth of the hole is defined directly on the Model page. To define the top and the bottom levels, it is necessary to select the desired points from the list on the right and enter the Zmax and Zmin values.

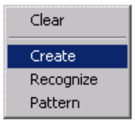
Zmax – defines the Z coordinate of the top of the hole. The coordinate can be defined either directly or calculated automatically. When calculating automatically, transition to work feed is performed using the safe distance from the workpiece.

Zmin – defines the Z coordinate of the bottom of the hole. The coordinate can be defined either directly or calculated automatically. When calculating automatically, the coordinate is taken from the model being machined.

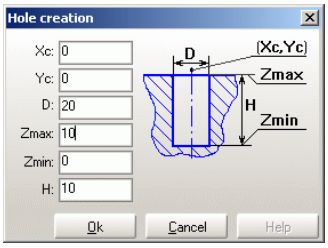
5.3.13 Defining holes by coordinates



To add a new hole to the list choose <**Create**> method of the creation hole.



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.

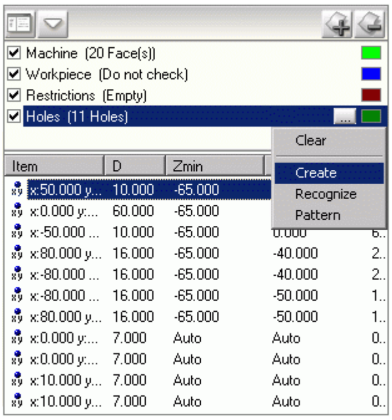
When hole's parameters are being altered, the changes are displayed in the graphic window.

5.3.14 Defining holes by using a geometrical "point" object

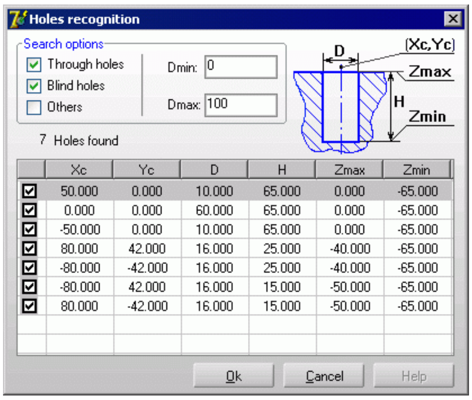
The user can add geometrical "point" objects to the holes list. A group of points can also be used. If a group contains objects of other types, then they will be ignored. The addition and removal of points is performed by using the  and  buttons, or by dragging and dropping into the list with the mouse. Every point defines the X and Y coordinates for the hole center.

5.3.15 Automatic hole recognition

To start a recognition choose <**Recognize**> method.



It is possible for the system to search for holes that already exist in a model. It can find the center position, diameter and depth (where possible). The search option is found on the <**Holes Tab**> of the model for machining window. 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.

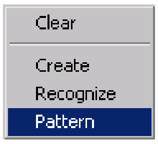
Others – holes, for which only the center coordinates, can be defined but not the diameter and/or the depth of the hole. Such holes might have a variable diameter e.g. with facets, or just be curves.

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.

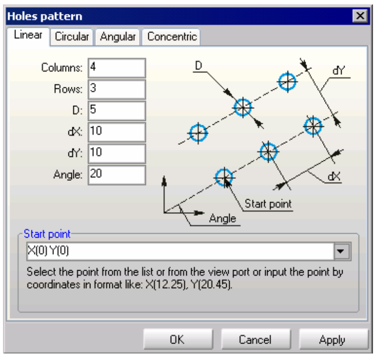
5.3.16 Create's hole pattern.

To add a hole pattern to the list choose this method of hole creation.

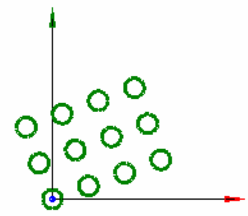


The system uses four types of pattern: linear, circular, angular and concentric.

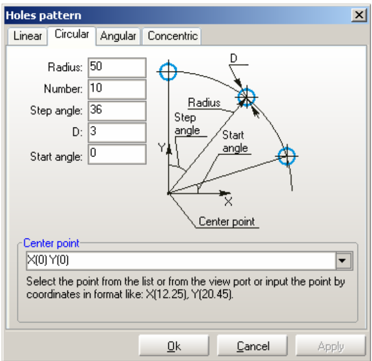
- On the "**Linear**" page user can create linear holes pattern:



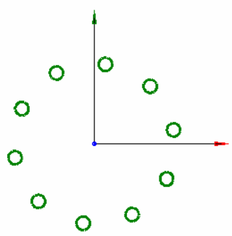
With the above given parameters the result will be twelve holes:



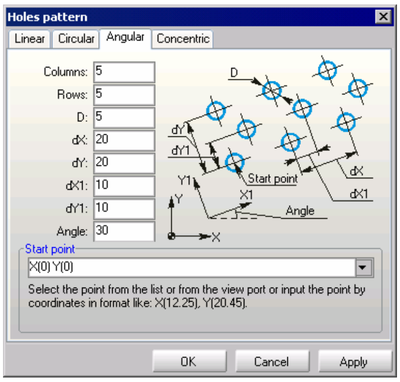
- On the "**Circular**" page user can create circular holes pattern:



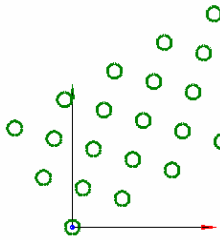
With the given parameters the result will be ten holes:



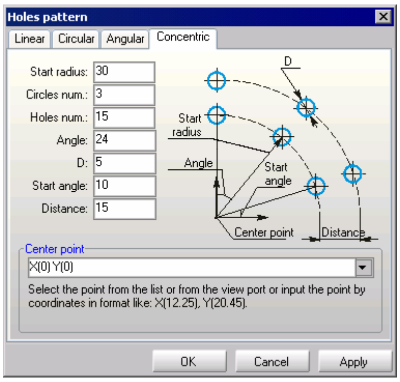
- On the "**Angular**" page user can create linear holes pattern:



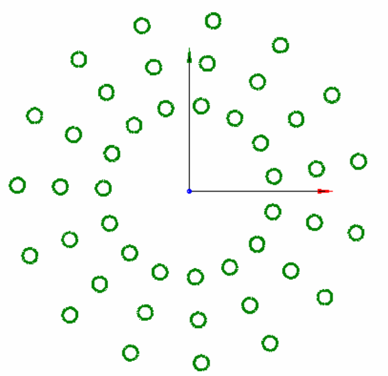
With the given parameters, the result will be twenty-five holes:



- On the "**Concentric**" page user can create concentric holes pattern:



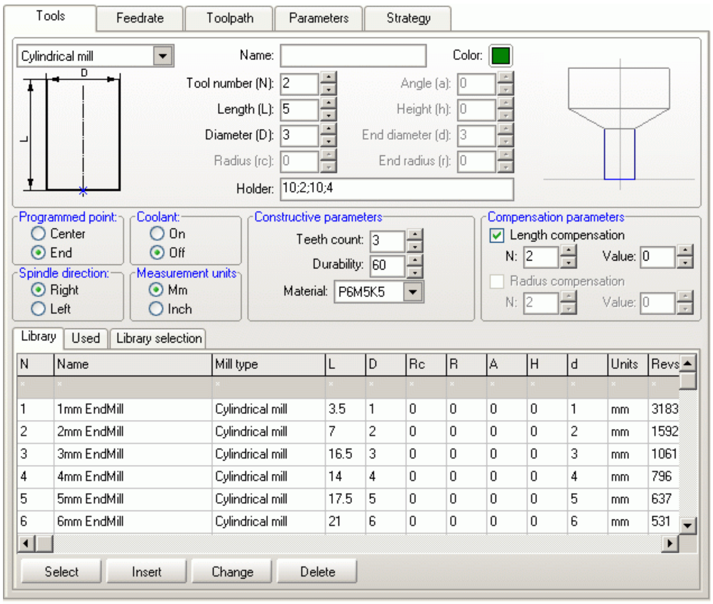
With the given parameters, the result will be forty-five holes:



5.3.17 Defining the tool for the current operation

A machining operation in the system uses one tool. One of the key parameters for an operation is the cutting tool that is to be used. In the milling operations available in the **SprutCAM** system, the cutting tool must be an axial tool, i.e.: end mills, drills or engraver.

The definition of the geometrical parameters for a tool and other machining parameters connected to its use can be performed in the <Operation parameters> window on the <Tools> page. The window opens by pressing the <Parameters> button.



There are 10 different types of tool which can be defined using a range of different geometrical parameters:

- Cylindrical mill: length, diameter;
- Spherical mill: length, diameter;
- Torus mill: length, diameter, rounding radius;
- 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;
- Mill with negative radius : length, diameter, rounding radius of the peak;
- 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.

If the value of a parameter entered makes it impossible to build the tool, then this parameter will be highlighted in red.

There are two graphics that display the current tool, the standard view and the real view. The standard view shows the chosen tool type with the key dimensions. The real view shows the tool in proportion using the current geometrical parameters and the type of the tool.

The field defines the tool number that will be entered into the NC program. If a previously used tool is required for a new operation, then this can be selected from the list found by selecting the 'Used' tab at the bottom of the tool page.

Machining and other parameters:

- **Measurement units**. of the tool parameters - millimeters and inches.

- **Programmed point**. Indicates from which point on the cutter (center or end) that the NC program will be calculated.
- **Spindle Direction**. Defines the spindle rotation direction as left (clockwise) or right (countrclockwise).
- **Color**. Indicates which color will be used for displaying the toolpath for the current operation and also the color of the tool during machining simulation.
- **Teeth number**. Defines the number of teeth for a tool. The parameter can be used for cutting speed calculation.
- **Material**. Indicates the material the tool is made of. The parameter can be used for cutting speed calculation.
- **Durability**. Defines the allowed working time for the tool in minutes. When the specified cutting time is reached an appropriate command will be added to the **CLDATA** intermediate language. The postprocessor, if the function is enabled, can output either a tool change command or add a note to the NC program.
- **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.
- **Length compensation**. When tool length compensation activation is required at the beginning of an NC program, the system inserts the required commands for this. The length-offset number for the tool being used is entered into the field "N", this will be output in the NC program. The compensation length value is for information only and is not output in the NC program. Tool length compensation is cancelled at the end of the program.
- **Radius compensation**.. Radius compensation can only be activated for the 2D and 3D curve machining operations. When machining a curve on the right or left, the appropriate cutter radius compensation command will be added to the NC program. The radius offset register number should be placed in the field "N", this will be the number used in the NC program. The compensation value is used for toolpath calculation and for machining simulation. The toolpath is calculated as an offset toolpath, which is calculated as the difference between the tool radius and the compensation value. If the offset toolpath calculated for the NC program needs to be the same as the contour itself, then the compensation value should be equal to the tool radius. By doing this the actual contour coordinates will be used in the NC program, but the simulation in **SprutCAM** will correctly display what will occur on the milling machine.

Once the parameters for a tool have been entered, they can be saved into the tool library for use in future operations. The tool library is displayed in the lower part of the tool page.

Search options are entered in the first row of the corresponding column. A search can be performed both, by a parameter value and by a mask. To define a mask the following commands can be used:

***** - (asterisk) - an arbitrary string.

? - (question) - an arbitrary symbol.

- - (dash) - interval.

, - (comma) - enumeration.

The interval and enumeration commands can be used for numeric parameter searches only.

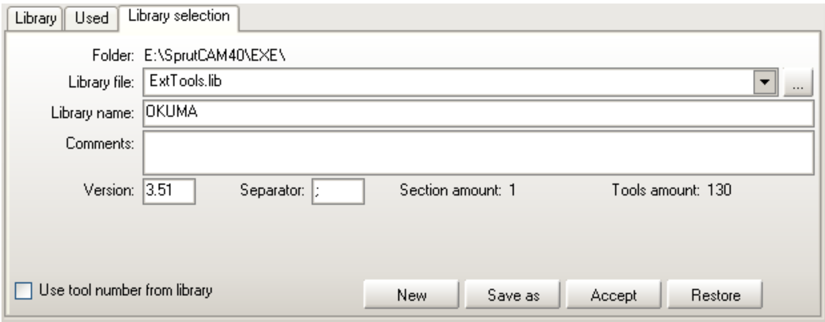
Examples:

- Find all tools with a name that contains the phrase "mill".
- In the "Name" field, type in the mask: *mill*
- Find all tools with diameter from 10 to 20 inclusive
- In the "D" field type in the mask: 10-20
- Find all tools with diameter 15, 23, 47 mm.
- In the "D" field type in the mask: 15,23,47

A mask can be defined using either separate commands, or, a combination of commands. To use the selected tool, press the <**Select**> button.

5.3.18 Library selection

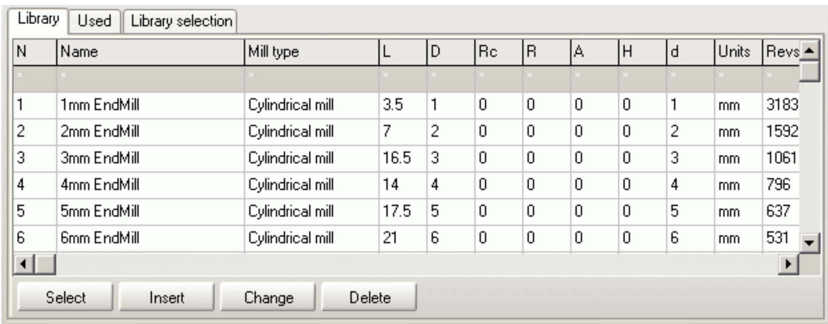
The Library selection tab allows the user to select a tool library, and define parameters of the current library.



Here the user can define another library or create a new one. To load the selected library, press the "Accept" button. To restore any altered parameters, press the "Restore" button. Pressing the "Save as" button saves the current library, and closes the window.

The list of all available tool libraries can be displayed using the "Library file" dropdown menu. Several tool libraries can be used, when it is necessary to create programs for several different milling machines, each of which may have it’s own individual tool magazine.

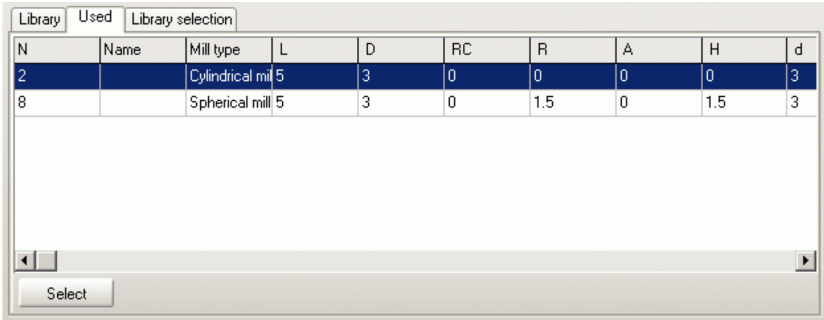
5.3.19 Page Tools library



A list of tools that are currently available for use is displayed on the <Library> tab. To use the parameters for a particular tool in the library, select the tool from the list and press the <Select> button. To add a new tool to the library, set the required parameters for the tool in the upper part of the window, and then press the insert button. To replace the selected tool in the library with a new one, using the parameters entered in the upper part of the window; press the <Change> button. To delete the selected tool from the library, press the <Delete> button.

Note: *The tool number in the library does not refer to the tool number in the NC program!*

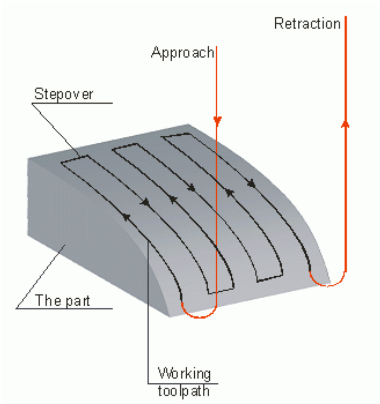
5.3.20 Page Using



N	Name	Mill type	L	D	RC	R	A	H	d
2		Cylindrical mil	5	3	0	0	0	0	3
8		Spherical mill	5	3	0	1.5	0	1.5	3

The Used tab displays all tools used in the current machining project.

5.3.21 Approach, retraction, and plunge methods



Approach and retraction

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 cancelled at the start of the retraction move.

The Approach of a tool is performed as follows:

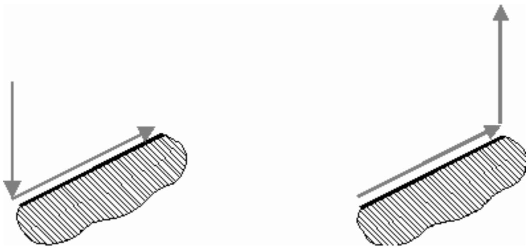
- Tool approach to the plunge point at the safe plane for the operation.
- The Z-axis rapids to the safe level or safe distance before beginning the approach move (depending on settings). The safe level is measured from the Z top level in the current coordinate system. The safe distance is the Z distance 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:

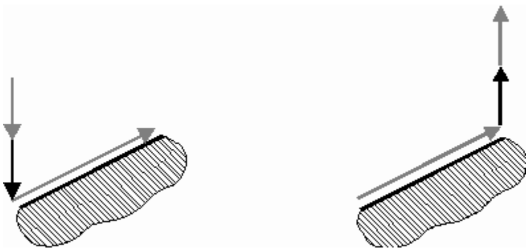
- End of the work pass.
- Retraction of the defined type at the work feed.
- Vertical rising of the tool to the safe plane at rapid feed.

In the system there are the following approach/retraction methods available:

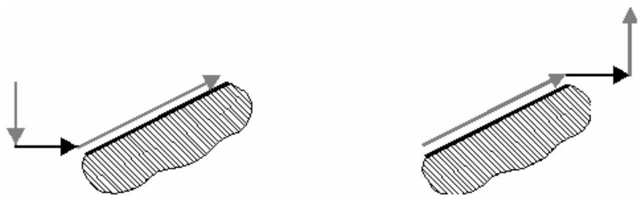
- **Without approach** The approach and retraction trajectory parts is not generated.



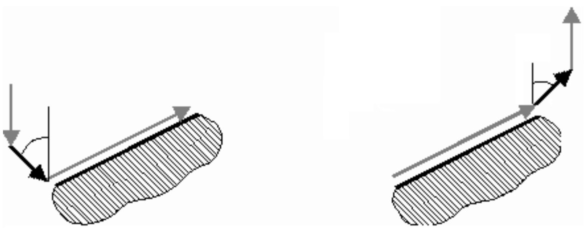
- **Vertical** The approach is performed vertically to the first point of the work pass. The retraction - vertical from the last point of the work pass.



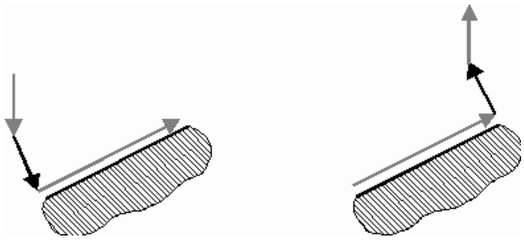
- **Horizontal** The approach is performed horizontal to the first point of the work pass. The retraction - horizontal from the last point of the work pass.



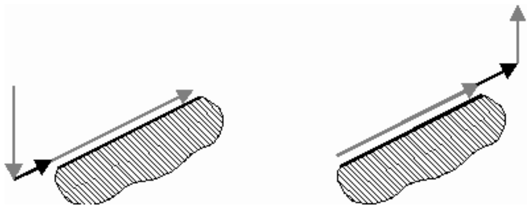
- **Angle to Z-axis** The approach is performed by angle to Z-axis to the first point of the work pass. The retraction - by angle to Z-axis from the last point of the work pass.



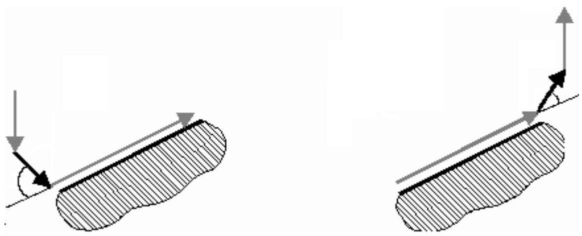
- **By normal** The approach is performed along the normal to surface at the first point of the work pass, the retraction - from the last point



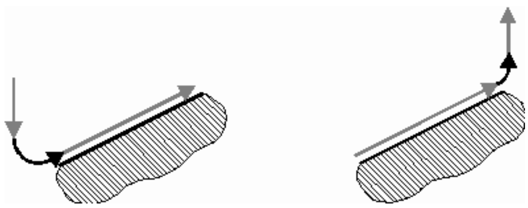
- **By tangent** The approach is performed tangentially to the first machining point and the retraction tangentially from the last point



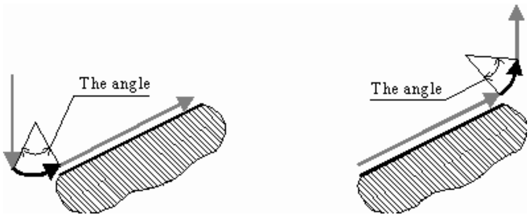
- **Angle to tangent** The approach is performed by angle to tangent to the first machining point, and the retraction by angle to tangent from the last point



- **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.

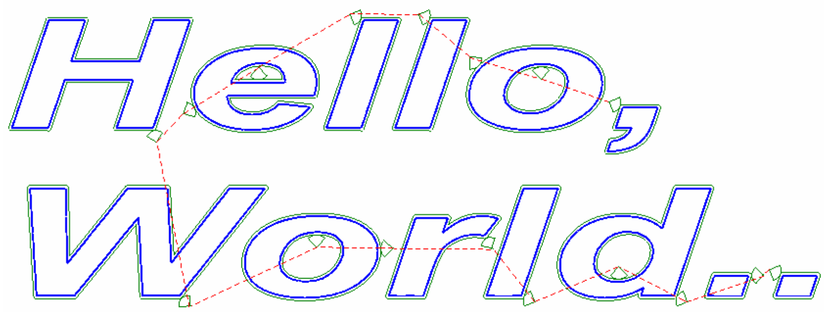


The definitions of the geometrical parameters for approach moves are made in the "Toolpath" page in the <**Operation parameters**> window.

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.

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.



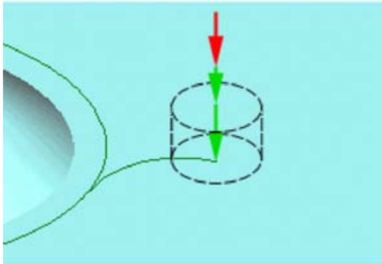
Plunge methods.

When it is impossible to approach the machining area from the outside, the system automatically generates a plunge move 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:

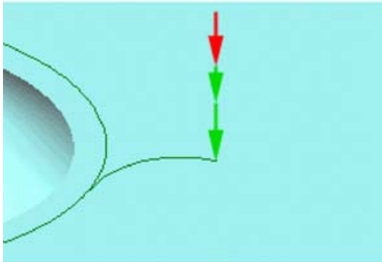
- Tool approach to the XY plunge point at the safe plane of the operation.
- Traverse vertically at rapid feed to the safe level or the safe distance before cutting begins (depending on settings). The safe level is the Z top level of the current coordinate system. The safe distance is the distance to the start of the defined plunge.
- Vertical descent at the approach feed to 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:

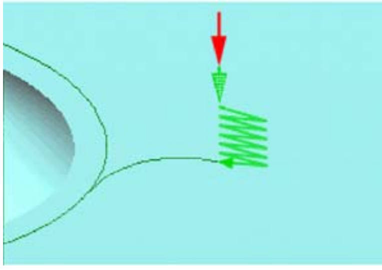
- Through drill point. If it is required for the tool to plunge through a pre-drilled hole, the system scans for an appropriate hole based on the center coordinates, the depth and diameter. If a suitable hole is found, then ramping will be performed at its center. Otherwise, the system will automatically generate an appropriate hole and add it to the list. In the "model" window the user can define the holes, where tool plunging is desired (refer to "Assigning hole parameters").



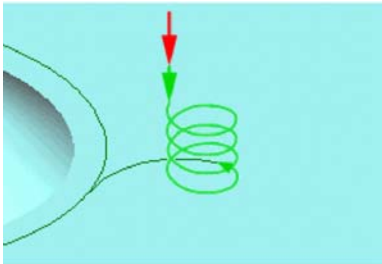
- **Axial**. Performed along the vertical straight to the first point of approach.



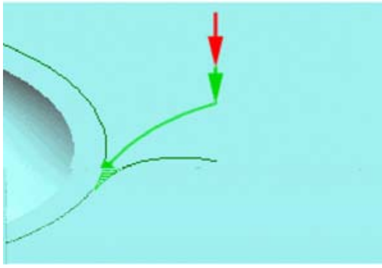
- **Zigzag**. The tool performs reciprocal movements along a straight section, connected to the first approach point. The length of the section is a user-defined option.



- **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.

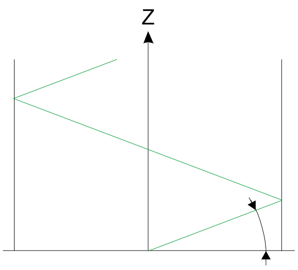


- **Along approach curve**. The tool plunges along the approach curve. The plunge move is completed at the end of the approach move.

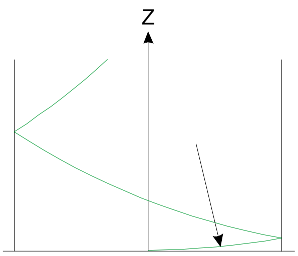


For the last three plunge types (zigzag, by spiral and along approach curve) the system applies the selected Z movement rule. Two motion types are available: angular and radial:

- **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.

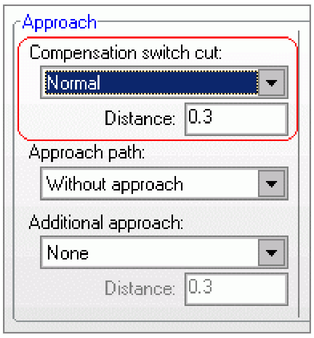


- **Radial**. The vertical tool movement is performed according to the sinusoidal rule, where the depth variation speed at the last point is equal to zero. The parameter is defined by an arc radius, the center of which is located parallel to the Z-axis from the starting point of the toolpath.

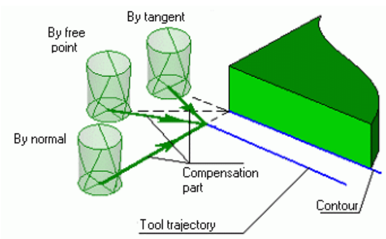


Compound approaches and retractions

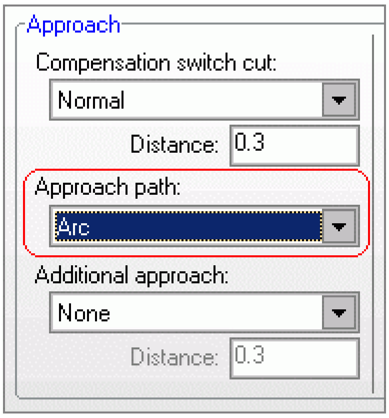
An approach move is performed in the same plane as the contour being machined. An approach move is a part of the machining toolpath, which is added to the starting point of the contour being machined. It can consist of three areas - trajectory inclusion area, the approach area itself and the area that represents an add-on to the contour being machined. Using these three methods, the user can obtain an optimal approach trajectory in every case.



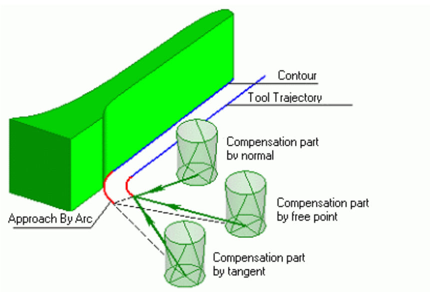
Activation of tool radius compensation.
The activation and cancellation of cutter radius compensation can only be performed on linear tool movements, as either a tangent or normal (perpendicular) move to the next toolpath move, or from an arbitrary point. Immediately following the compensation activation 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.



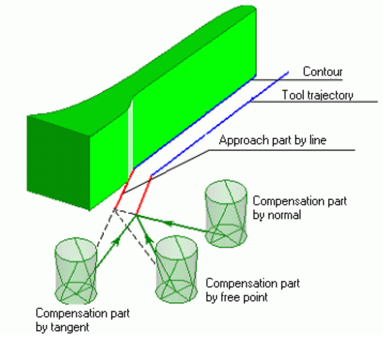
The compensation activation area is formed by the Milling unit's drive system; the older units form a simple linear transition, more modern CNCs can create a trajectory with control over tool contact with the workpiece.
The picture to the right shows the available methods for the additional moves required 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.



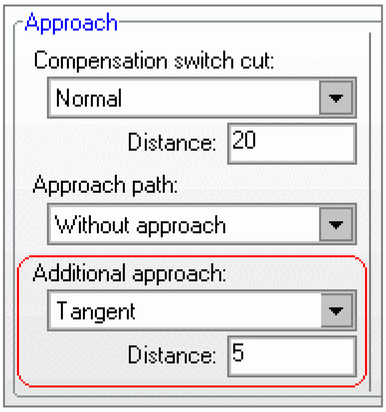
Approach.
Rather than plunge the tool into the workpiece at the start point on a contour, it is possible to add an approach move. The choices are none, tangent, normal and from start point. This approach path would connect directly to the contour itself, or, if an 'Additional approach' move is selected 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 parameters for the different approach strategies can be entered using the fields on the right of the Toolpath page.



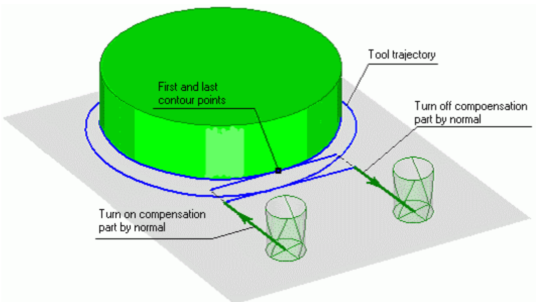
The picture above shows an arc approach move. The end of the arc touches the models contour. Actually, the approach area forms part of the machining contour. The question of whether to use an approach move or not has to be decided by the user. It depends on the specific conditions and requirements of the model being machined.



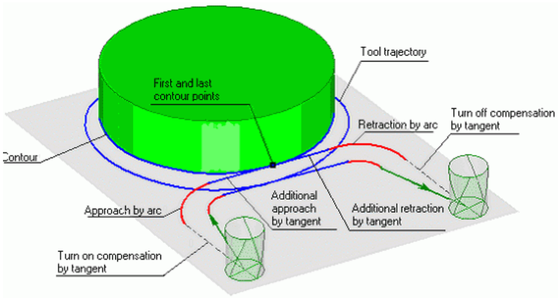
The picture to the left shows an approach-using angle to tangent. This also uses a compensation switch cut as in the example above.



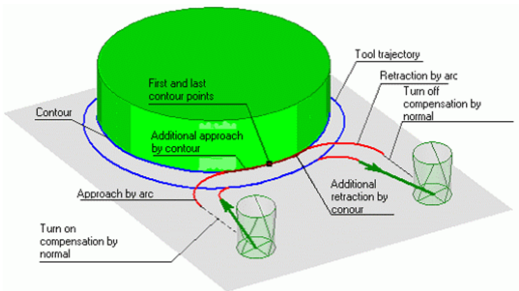
Contour extension area.
It might be required on occasions, to extend either the start of a contour, or the end, or both. To achieve this in **SprutCAM** we use the 'Additional approach' and/or 'Additional retraction' options. An additional approach is added at the start of a contour, additional retraction 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.



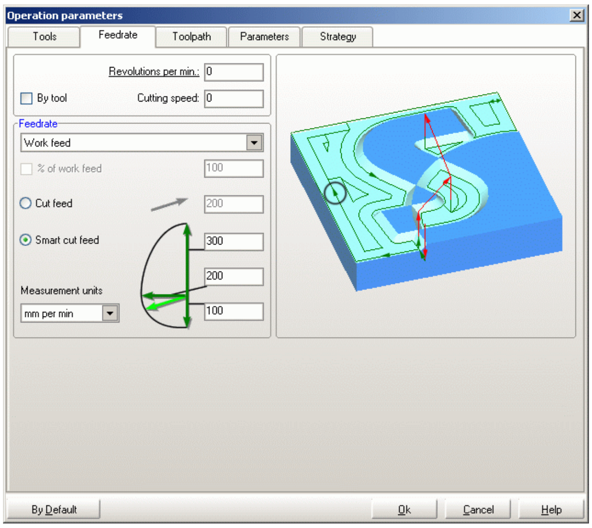
The picture above shows a similar situation to the previous example, but with approach and retraction moves as well.



This example has the along contour option selected, the "Additional approach" and "Additional retraction" options are set to "Along curve".

5.3.22 Cutting modes

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.

The user can also define the feedrates 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.
- **Approach feed** defines the feed at which the approach move of a tool. By default, the approach feed is equal to 50% of the work feed.
- **Retract feed** defines the feed at which the retraction from the object being machined. By default, the feed is equal to the work feed.
- **Plunge feed** defines the feed at which the system performs a vertical (Z) plunge to the next machining level. By default, it is equal to 50% of the work feed.
- **Feed to next** defines the feed that is applied when the tool moves to the next toolpath. By default, it is equal to 50% of the work feed.
- **Retrace feed** defines the feed that is used if the tool moves over an already machined area. By default, it is equal to 300% of the work feed.
- **First pass feed** defines the feed that is used for the initial cut of the tool in the workpiece. By default, it is equal to 50% of the work feed.
- **Finish pass feed** for roughing operations this defines the feed used when cutting along a surface. By default, it is equal to 50% of the work feed.
- **Measurement units**. A feed can be assigned either in millimeters per minute or in millimeters per revolution of the spindle.

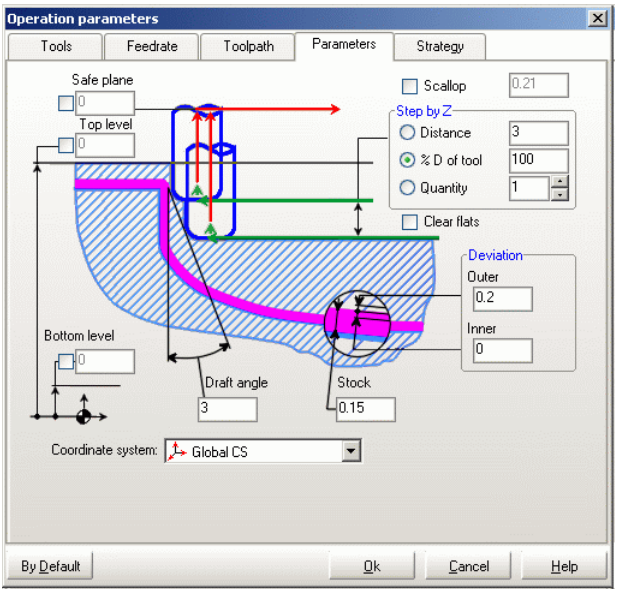
If the <**Cut feed**> option is selected, then the feed value can be defined in the appropriate field. Using this option means that the feed does not depend on the tool movement direction.

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 feedrates 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.

5.3.23 Assigning operation parameters

The main parameters for the current operation can be defined in the <**Operation parameters**> window. It opens when the <**Parameters**> button is pressed. 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:

- **Coordinate system** 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** 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, 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.
- **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.

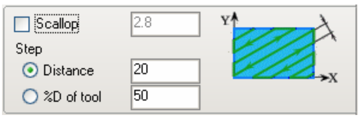
5.3.24 Defining the machining strategy

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.

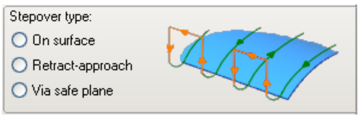
The following parameters can be defined on the page:

- **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.

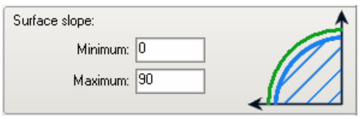


- **Stepover type** defines the toolpath when moving from one machining pass to the next.

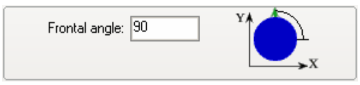


- **Roll type** defines the necessity of avoiding peaks and edges of the model when machining using the volume machining finishing operations.

- **Surface slope** in the volume machining finishing operations limits the surfaces being machined depending on their slope angle.

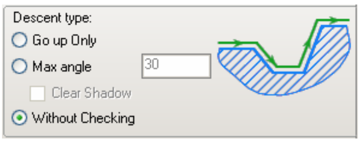


- **Frontal angle** in the plane and drive finishing operations limits the surfaces being machined depending on the angle between the cutting tools face in the cut direction, and the models surface(s).



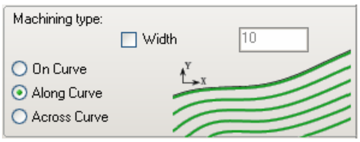
- **Angle** assigns the cut angle direction for the plane operations.

- **Descent type** defines the tool descent strategy for the plane and drive finishing operations.



- **Machining direction** during the finish machining by depths defines the machining sequence, either upwards or downwards.

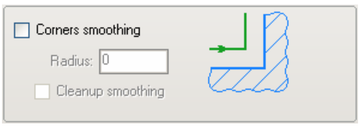
- **Machining type** in the drive operations defines the strategy for the work passes formed either along or across the drive curves.



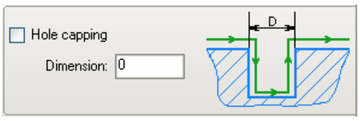
- **Upward only** in the plane and drive finishing operations restricts tool movements on the model to the upward (Z+) direction.

- **Trochoidal machining** allows to reduce the NC data much in comparison with the cycloid 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.



- **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.

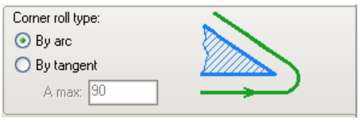


- The **allow 3D toolpath** mode allows the system to create a 3D toolpath for material removal in all areas that are not accessible for machining on the current level (e.g. inner corners).

- **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.

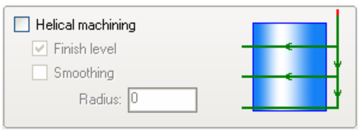
- **Machining order** defines the machining sequence in the curve machining operations (by contours, by depth).

- **Corner roll type** defines the outer corner angle bypassing method in the curve machining operations.

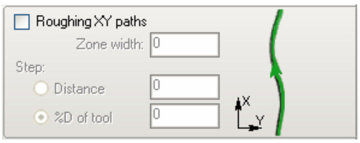


- With "**Idling minimization**" active, the total distance that the tool moves for machining all selected curves, is kept to the minimum. Otherwise, the machining will be performed according to the order defined on the Model page.

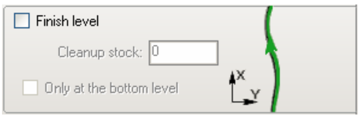
- **Helical machining** allows the user to obtain a spiral-like toolpath when machining a 2D curve.



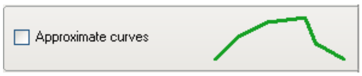
- **Roughing XY paths** allows the use of several XY passes to remove the stock material instead of a single pass. This can improve the surface finish by reducing the load on the tool.



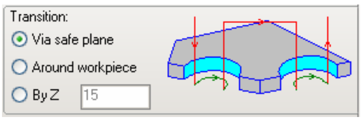
- **XY cleanup** allows the user to obtain a higher quality surface finish by leaving only a small user defined stock for an automatic finish cut.



- **Approximate curve** approximate trajectory by arcs.



- **Transition** allow select stepover method for curve machining operation.

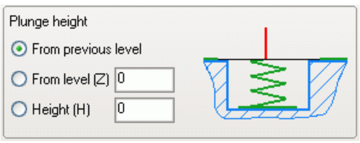


The strategy of flat land machining on axis Z.

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.

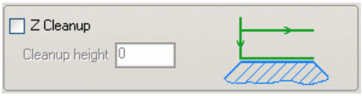
- **Machine By Layer** The material may be deleted for some passes. For this purpose, it is necessary to switch on Machine by Layer.

- **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.



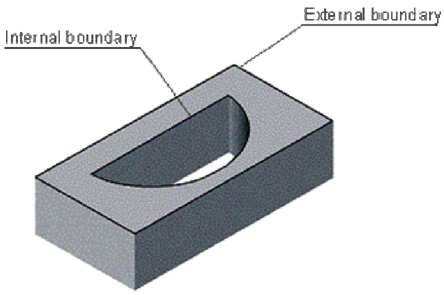
On the <**Strategy**> page in the <**Operation parameters**> window for the hole machining operation the user can define the cycle type for hole machining (drilling, boring, tapping etc.) and its parameters. The set of parameters depends on the cycle type. **Points for Drilling** can be defined in the **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.

5.4 BASIC MACHINING DEFINITION

5.4.1 Model to be machined

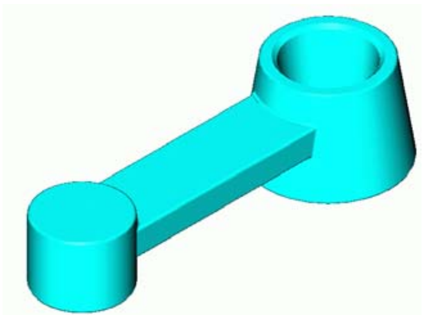
A model to be machined - is a set of geometrical elements that form the model, or parts of it that the user wants to obtain after machining. Depending on the operation type, the model being machined can be defined by different methods.

For the curve machining operations (2D contouring, 3D curve milling), a machining model 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 trajectory will be created with regard to the user-defined options, e.g. tool offset left/right/centre 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 centre of the curve then any stock values are ignored, and no offset toolpath is calculated.



In the engraving and pocketing operations, the machining model 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.

Note: *To help to visualize the model creation from curves at the top machining level, it is recommended to activate the dynamic preview of the model by selecting the 'Preview' option on the operations Model page.*



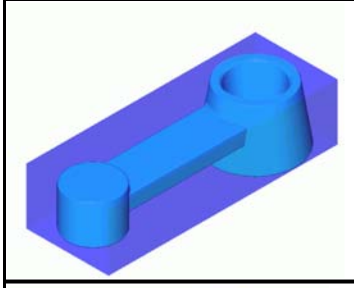
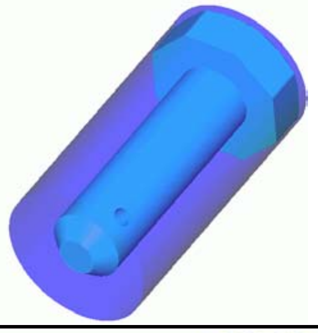
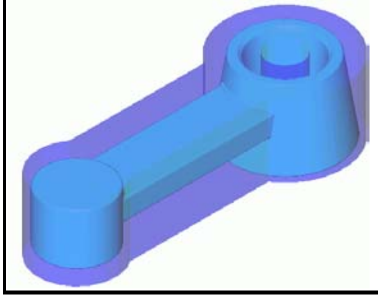
For most volume machining operations, the model to be machined is defined by a set of solid bodies, surfaces and meshed objects. The objects can be present in any combination and be grouped in an arbitrary way. All curves that are in the machining model will be ignored. When defining the definition machining model, every geometrical object or its group can have an additional stock amount defined, which will be added to the operation stock. The tool trajectory is calculated to remove the workpiece material, which lies outside the definition machining model, considering any stock amounts. This means that the tool will never cut the solids, surfaces or meshes that make up the definition machining model.

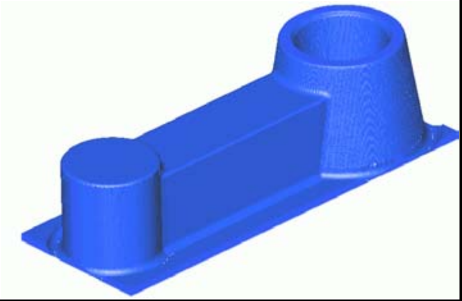
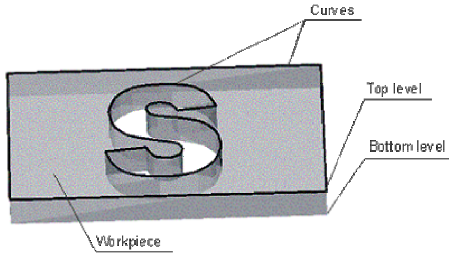
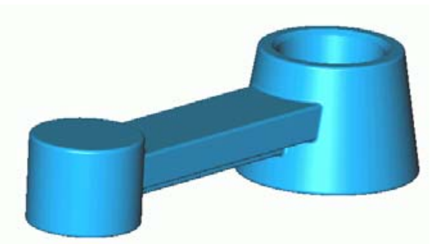
5.4.2 Workpiece

A workpiece model 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.

A workpiece can be assigned by the following methods:

Workpiece	Description
	In the finishing operations the user can select the "without workpiece form" mode. With this workpiece method defined, machining of the entire model will be performed. Similar results in finishing operations can be achieved by defining a workpiece with a size that is bigger than the model for machining.
	A workpiece shaped like a cube can be defined relative to the dimensions of the model for machining, or by direct entry of the coordinates of the cube.
	A workpiece shaped like a cylinder with the rotation axis parallel to the Z-axis. The parameters of the cylinder can be defined relative to the dimensions of the model for machining, or by direct entry of the coordinates of the cylinder.
	A casting can be defined in two ways: either as a model with an equal stock amount all over, or as an extrusion, the outline of which was obtained by a projection of the model onto the XY plane.

	A workpiece as the material that is left after machining by previous operations. This method for defining the workpiece allows the performing of rest milling residual material left by previous operations.
 	A free-form workpiece can be defined by a set of geometrical objects of different types with the ability to define an individual stock for every object.

In general, a geometrical model of a free-form workpiece can be created by the simultaneous combination of two types of elements: extrusions formed by a set of curve projections, and surface objects (solid bodies, surfaces and meshes). For the curve machining operations, (e.g. engraving and pocketing) the workpiece can only be defined as extrusions of curves; solid bodies, surfaces and meshes will be ignored. The base for an extruded workpiece is an area created from projections of closed curves onto the horizontal plane, taking into consideration any additional stock and method of inclusion into the area for every curve or group. Also, the order that the curves and groups are entered into the list affects the workpiece model. The closed curves can define an outer, or an inner border (hole) with any additional stock. The objects are added one after the other to the resulting area using the Boolean operations. Consequently, the order that the objects appear in the list considerably affects the look of the resulting area. A workpiece created by (extruded) curves has the same height over its entire area, i.e. between the upper and the lower machining levels.

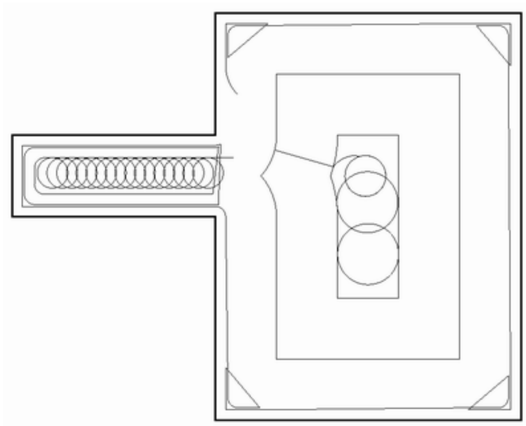
Note: *To make it easier to visualize an extruded workpiece it is recommended to turn on the 'Preview' option on the Model page of the operation.*

For most volume machining operations, a workpiece can also be defined by a set of solid bodies, surfaces and meshed objects. The objects can be present in any combination and be grouped in an arbitrary way. When creating a workpiece, for every geometrical object or their group an additional stock can be applied. The workpiece is formed from objects, offset from the assigned ones by the value of the additional stock. Use of the additional stock allows the user to; for example, obtain a similar workpiece as when defining a casting with equal stock. To do so, set the workpiece model as the model that is being machined and add an additional stock amount that is equal to the value of the foundry stock.

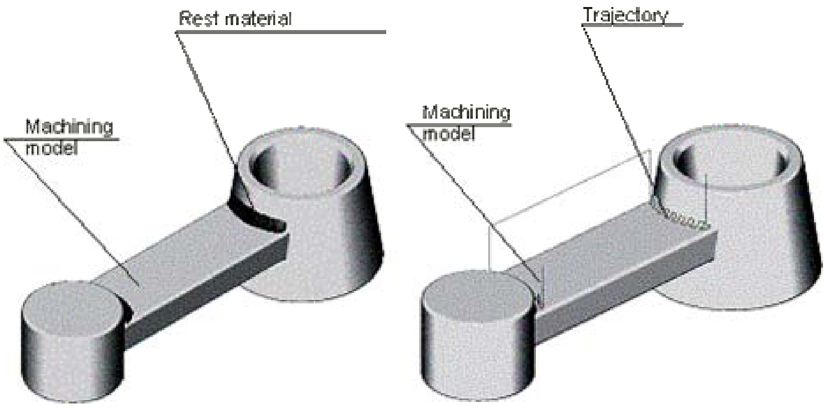
5.4.3 Rest milling of remaining material

Rest milling of remaining material in the system is achieved by defining the workpiece as the material left after machining by all previous operations. The calculation of the workpiece and of the unmachined areas is performed automatically by the system. This approach has some considerable benefits when compared to the usual method of "machine after a tool of certain size". Firstly, the system can correctly consider the remaining material even if previous operations do not coincide with each other and the rest milling operation. And secondly, it considers the machining by all previous operations with different tool types and with all peculiarities of the calculated toolpaths.

The roughing 'rest milling' operations perform removal of the entire available residual material. The finish 'rest milling' operations machine the surface of the model only in those areas that are unmachined. Application of the rest milling operation allows the user to considerably reduce the machining time for complex shaped models without extra work for the user.



To obtain an optimum toolpath during rest milling, it is possible to define the maximum value of the amount of unmachined material that can be ignored. It is recommended to set the value 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.



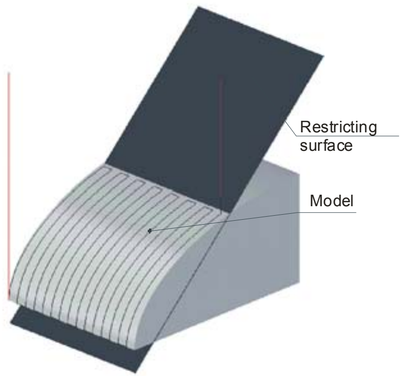
5.4.4 Machining limitations

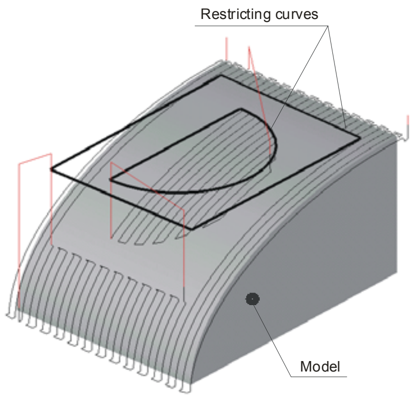
In the system there is the possibility to use area limitations, which will be considered when calculating a tool path. In other words, the user can define areas of 3D space, which the tool cannot enter during machining. Using these limitations (restrictions) the user can define for example the position and the dimensions of machining fixtures/clamps. They can also be used to define areas of the model that must not be machined, or alternatively, select parts of the model for machining.

All tool movements, irrespective of the operation type, can only be performed outside the borders of the limiting (restricting) model. 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.

In general, a restricting model can be defined by the simultaneous combination of two elementary object types: 2D areas parallel to XY (restricted areas and machining areas) and surface objects (solid bodies, surfaces and meshes). For the curve machining operations e.g. engraving and pocketing, 2D areas only can define the restricting model; any solid bodies, surfaces and meshes will be ignored.

2D areas are formed from projections of closed curves onto the horizontal plane, considering any additional stock and the method of inclusion in the area for every curve or group, and also the order of the curves (groups) in the restrictions list. Projections of closed curves can define either a restricted area or a machining area. A 'restricting' area does not allow machining inside the contour, and a 'machining' area outside the contour. Any of the variants can define the limitation to the tool centre or to its edge. Consequently, the order of objects in the list considerably affects the look of the resulting area. The resulting restricting area that is created from the restricted areas and machining areas, acts on any level below the safe plane. This means that all movements in the restricted areas can only be performed via the safe plane.





Note: *To aid the creation of restricting areas, it is recommended to activate the dynamic preview mode by selecting it in the model definition window. Machining can be performed only outside of the borders of the displayed extrusion based on the restriction type.*

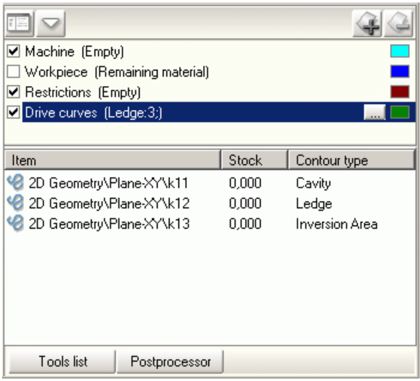
For most volume machining operations, solid bodies, surfaces and meshed objects can be added to the restricting model. Objects can be present in any combination and be grouped in an arbitrary way. An additional stock can be defined for every geometrical object or their group. By using the additional stock option of the restricting model, equidistant offset objects can be created by the user. The offset amount is equal to the value of the additional stock. Use of the additional stock in the restricting model makes it possible, for example, to leave a guaranteed stock for surfaces that are not machined by the current operation, but will be finish machined by subsequent ones.

If the same surface object is present in the restricting and the machined models, then it will be considered as a (to be) machined object. This means that to machine only selected surfaces, and leave others, it is recommended to add the whole model as the restricting model, and those surfaces that are to be machined add as the 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.

5.4.5 Drive curves



In the drive operations, the drive area defines the toolpath in plane (XY), and the Z coordinate is calculated based on the condition that the tool touches the model being machined.

The drive area is created form curves projected onto the horizontal plane, considering any additional stock and also the method of inclusion into the area for every curve or group, and also the order of the curves (groups) in the resulting area. A projection of any curve can define a ridge, ditch or inversion curve. The thickness is defined by the value of the additional stock. Projections of closed curves can also define a ledge, cavity or inversion area, also considering any additional stock. The objects are added one by one into the resulting area using the Boolean operations. Thus, the order that the objects appear in the list considerably affects the look of the resulting area. The toolpath construction method inside the defined area is dictated by the operation strategy.

Note: *To aid the creation of drive areas, it is recommended to activate the dynamic preview mode by selecting it in the model definition window. Machining can be performed only outside of the borders of the displayed extrusion based on the drive. Machining can be performed only outside the borders of the displayed extrusion based on the type of drive area.*

If a drive area is not defined, then the system will automatically create the drive area as a projection of the outer edge of the model being machined.

5.4.6 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.*

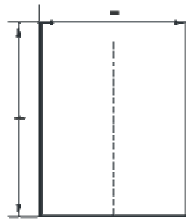
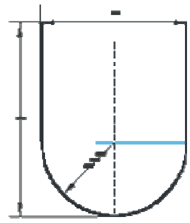
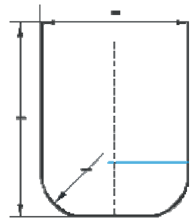
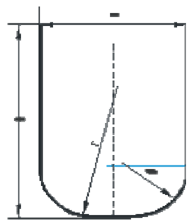
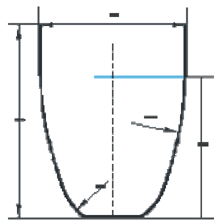
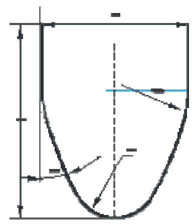
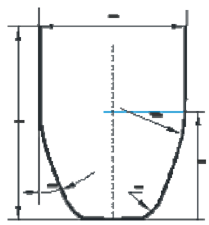
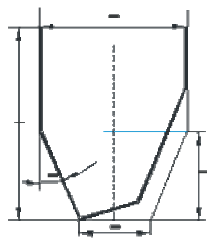
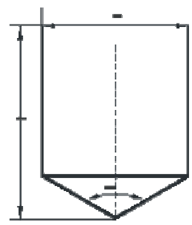
Points from the unified list can be defined as parameters for the hole machining operation.

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.

5.4.7 Tool

Calculation of machining trajectory 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:

Table: Supported tool types.

Cylindrical mill: length, diameter;	Spherical mill: length, diameter;	Torus mill: length, diameter, rounding radius;
 Cylindrical mill	 Spherical mill	 Torus mill
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;
 Double radial mill	 Limited double radial mill	 Conical mill
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;
 Limited conical mill	 Engraver	 Drill

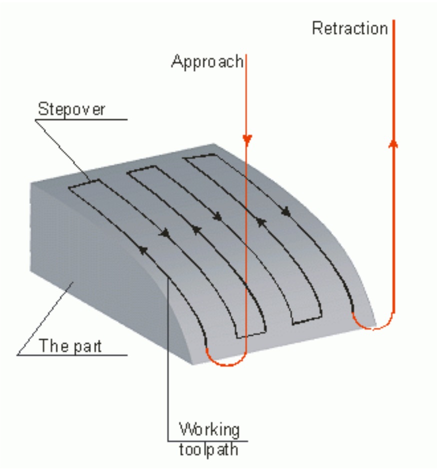
Besides the parameters that describe the shape of the mill, the user can also define for the tool:

- 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.

5.4.8 Tool movement trajectory areas



Approach

Approach means the path the tool takes to arrive to the first point of a work pass. The approach always belongs to the imaginary surface that contains the work pass. This means that in the waterline operations feeds they are performed in the horizontal planes, in plane operations - vertical, in drive operations - vertical mathematical cylinders defined by appropriate curves and etc.

The set of available approach methods depends on the operation type. The approach method can be defined in the approach and retraction modes window.

Retraction

Retraction means a sequence of tool transitions, which provides its smooth feed away from the last point of a work pass. The retraction, just as the approach, always belongs to the imaginary surface that contains the work pass. That means that in the waterline operations lead-outs are performed in the horizontal planes, in the plane ones - in vertical, in drive - in vertical mathematical cylinders, assigned by request curves etc.

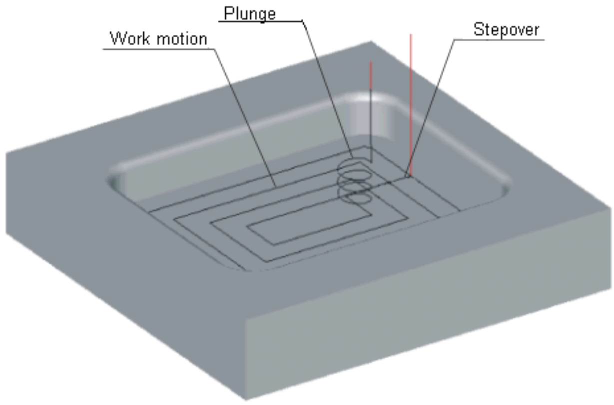
The set of available retraction methods depends on the operation type. Retraction method can be defined on the Toolpath page.

Plunge

A tool plunge normally means a sequence of tool moves within the workpiece body, which allows lowering 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 is impossible.

The plunge type can be assigned for operations that perform waterline stock removal (waterline roughing and pocketing operations). In other operations, plunge by special methods is not performed. The plunge method is defined on the Toolpath page.



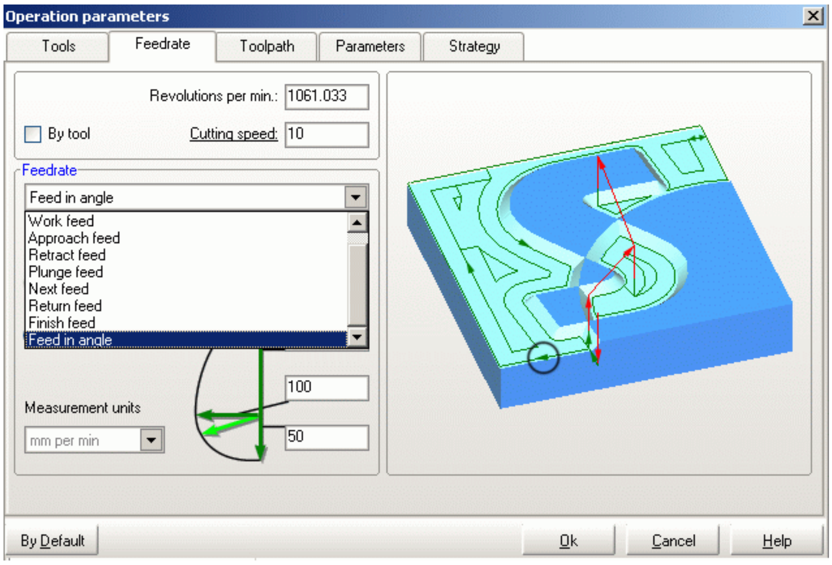
Work pass

A work pass is an elementary tool movement trajectory when machining the model surface. For the plane operations, the work pass is a single stroke, for the waterline operations - an elementary horizontal trajectory section etc.

Stepover

Stepover is an elementary trajectory section between work passes. For the plane operations it will be an X/Y axis stepover, and for the waterline - Z-axis. A stepover depending on the selected machining type can be performed either touching the model, or via the safe plane, or by the distance needed for approach and retraction. Stepover cannot contain more than one fragment of approach and retraction.

5.4.9 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 stepover 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 trajectory of the earlier completed stepover 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.

Feed value can be either permanent or calculated, depending on the slope angle of every elementary trajectory 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 trajectory 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** 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.

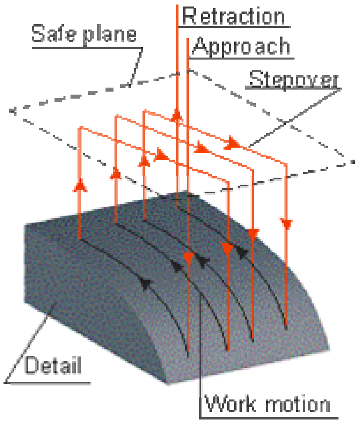
Rapid feed can be assigned by the permanent value only. The work feed can be either permanent or variable; its real value can be assigned manually or calculated automatically by the cutting modes calculation module. All other feeds are assigned either analogously, or in percents from the work feed. When 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.

5.4.10 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.



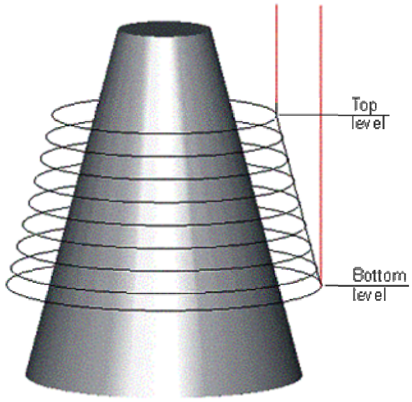
5.4.11 Top and bottom machining levels

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.



5.4.12 Tolerance

Machining tolerance is assigned in the system by the maximum deviations of the approximated tool movement trajectory 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).

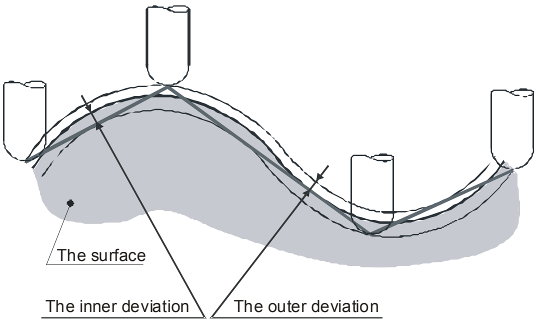
For deviation from the detail, the positive direction is outwards from the detail surface (inwards to the tool), and negative for deviation into the detail - (outwards from the tool). Thus, the machining tolerance is equal to the sum of deviations outside and inside the detail. One should note that increasing 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 trajectory.

The sum of the deviations must be more than zero, otherwise it will be impossible to construct an approximated trajectory.

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 stock. 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** page.



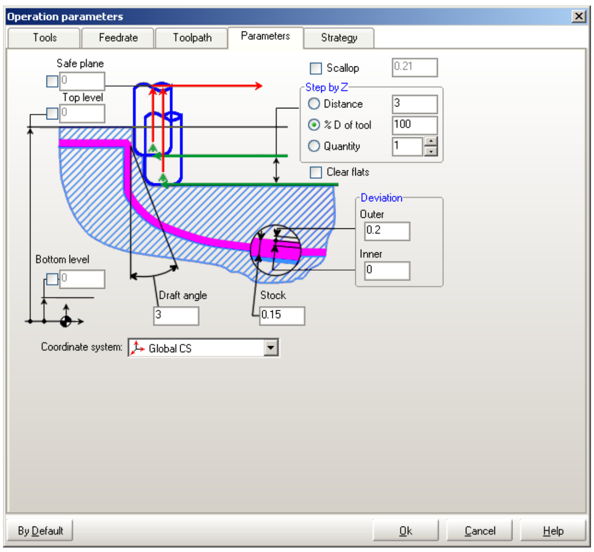
5.4.13 Stock

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.

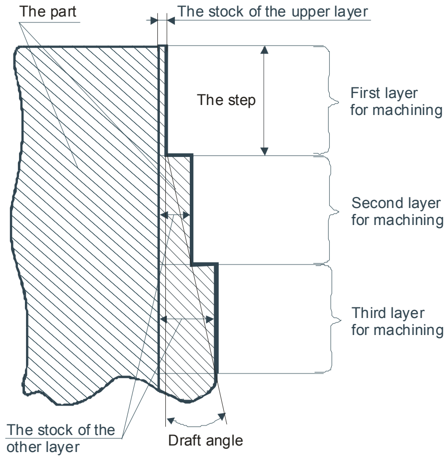


5.4.14 Draft angle

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 draft 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 draft 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.

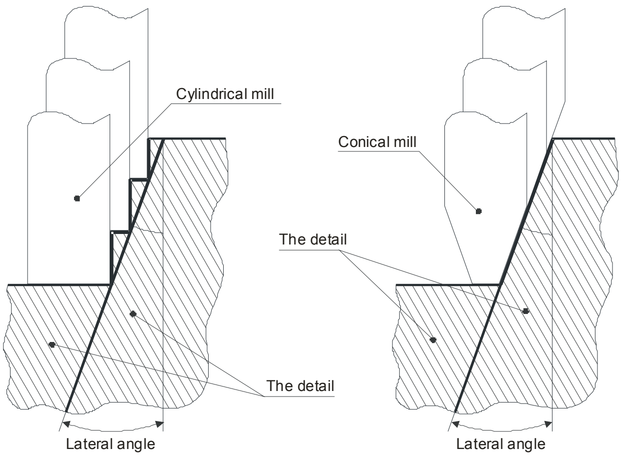
The draft angle value cannot be negative, nor can it be more than or equal to 90 degrees. Setting the draft angle too high will lead to a large layer of unmachined material at the lower layers.

The draft angle can be assigned on the Parameters page.



5.4.15 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.

5.4.16 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.

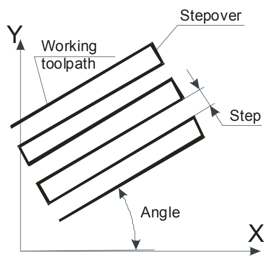
For more convenience, the step value can be assigned by several methods.

- **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.

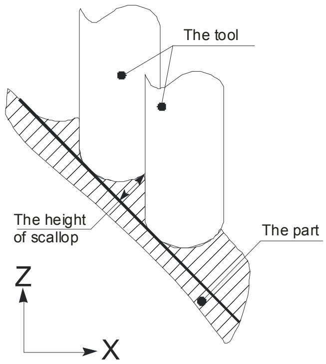
Irrespective of the assignment method, the real step value must be more than zero.

The step in the vertical direction can be assigned in the Parameters page, and in the horizontal - on the Strategy page.



5.4.17 Selecting step by scallop height

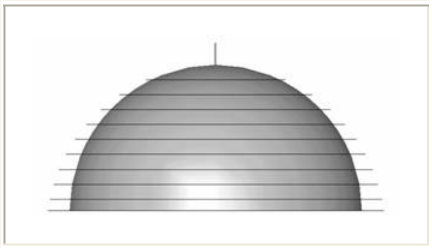
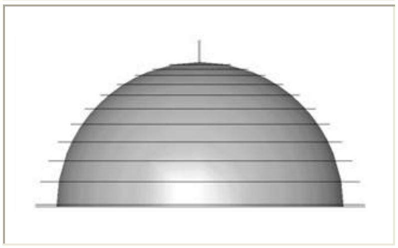
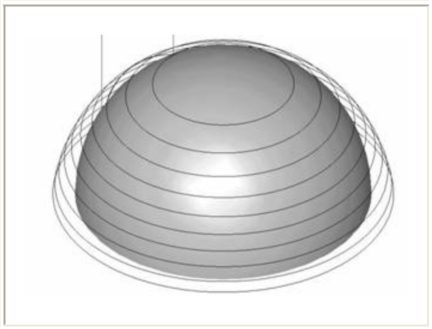
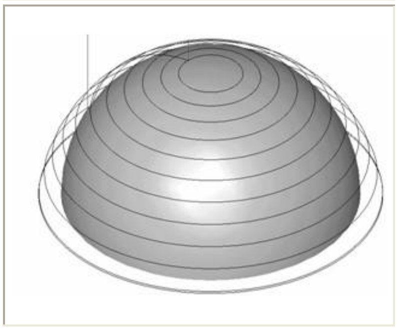
When machining surfaces, whose radius of curvature does not coincide with the profile radius of the tool, then there appears a scallop of residual material between the neighboring passes. Its height depends on the surface shape of the model being machined, the type and size of the tool and the distance 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.

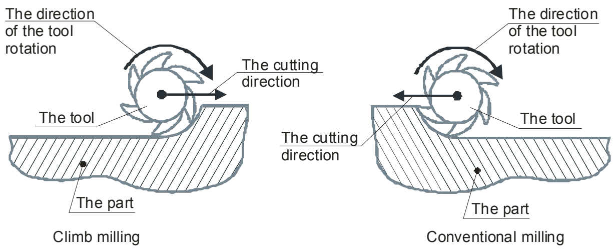
The maximum height of the scallop can be assigned by a positive value only. It is not advised to set the height of the scallop less than the machining tolerance.

The scallop value at step in the vertical direction can be assigned in the Parameters page, and in the horizontal - on the Strategy page.

5.4.18 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.

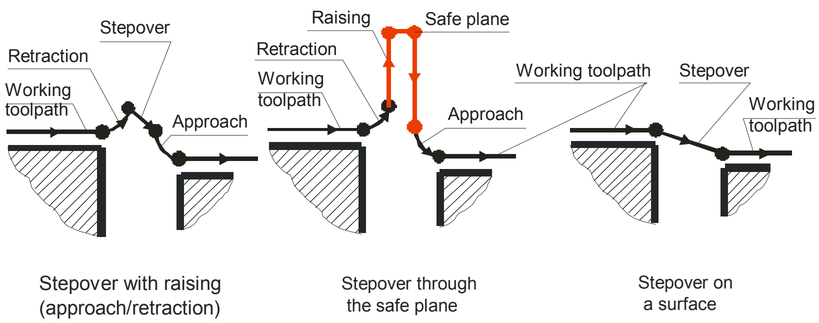
5.4.19 Stepover method

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 trajectory 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 stepover from one work pass to another are performed by the following methods:

- **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.
- **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.
- **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 stepover 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.

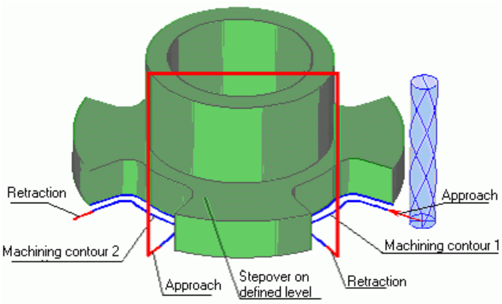
In the curve machining operations, **SprutCAM** gives the user the following stepover methods:

- via safe plane.

* around the workpiece and at the defined Z height
* by Z.
* 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 stepover 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.

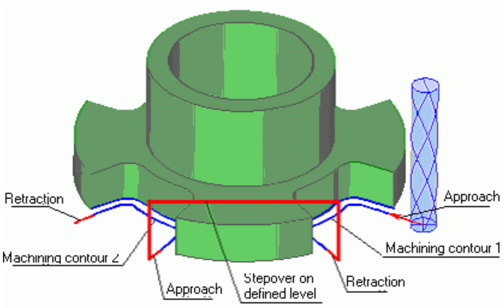
SprutCAM in the 2D machining mode allows the user to work either using or not using a workpiece. If a workpiece is not used, then the stepovers between work contours will be performed either by the safe plane, or at the defined Z level.

Using a workpiece will provide a safer work mode because the system will automatically create a stepover trajectory with control of tool collision with the workpiece, also in this case, **SprutCAM** will give the user more possibilities to optimize the machining trajectory.

The workpiece can be defined on the Model page. For a complete description on how to create a workpiece, refer Workpiece.

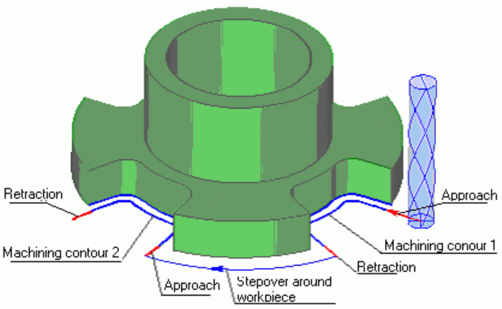
By Z

The plane level for transitions can be assigned on the machining <Strategy> page relative to the zero of the current coordinate system.



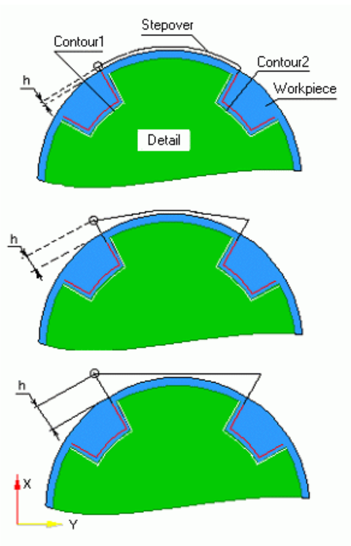
Around workpiece

The trajectory 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 stepover trajectory depends on the workpiece shape and will be formed as a curve around the workpiece area that is found. The shape of the stepover curve is also affected by the distance from the workpiece to retraction and approach points to contours.

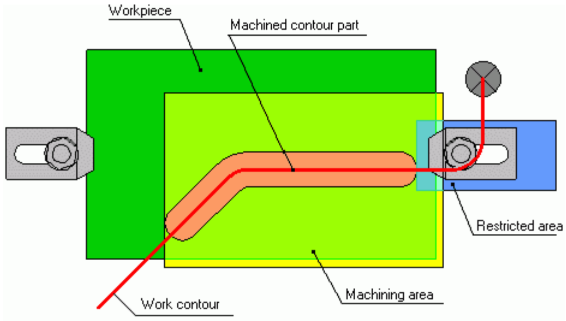
An example of trajectory alteration, depending on the value of that figure (h) is shown in these pictures. The trajectory can change its look from a straight line to the curve that repeats the shape of the workpiece as shown on the picture. Concave areas of the workpiece pass along the shortest curve, convex - along the rounding one.

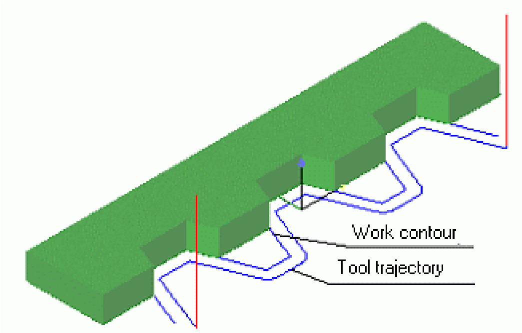


The stepover curve is constructed touching the edge of the tool against the workpiece, therefore in practice, when assigning a workpiece; the user should add an additional stock to the workpiece on the Model page.

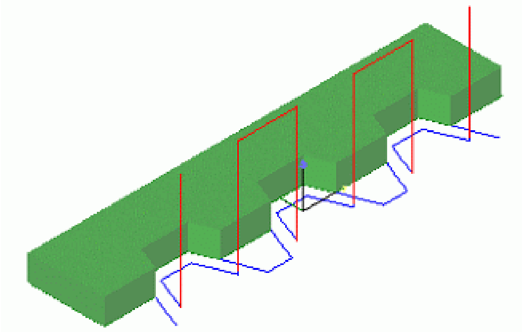
In order to obtain the desired stepover curve the user should use different values of h for the approach and retraction points, and also alter the workpiece profile.

One should pay special attention to the case where the contour has areas that go beyond the workpiece area. In **SprutCAM** there is a rule - if a workpiece has been defined, the machining trajectory will be formed within the workpiece area only. If machining and restricted areas have also been defined, then the trajectory will be formed within the workpiece area, inside the available machining areas and outside the restricted areas (see pic.).

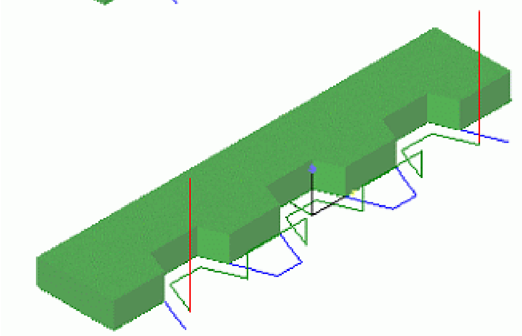




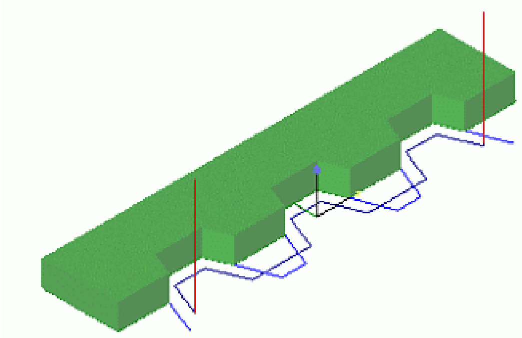
In the picture (left) the trajectory is formed regardless of a workpiece, using the activation and deactivation area of compensation by tangent. As can be seen in the picture the trajectory is formed equidistant to the work contour.



With a workpiece (bar) defined. The trajectory 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 stepovers. This operation uses stepover at the defined level.



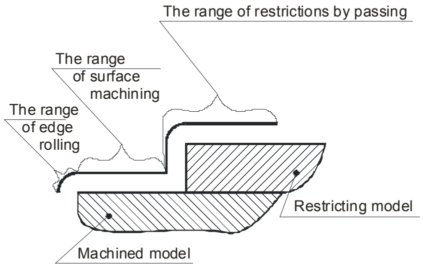
In this operation, the system uses stepover round the workpiece. One should note that the compensation activation and deactivation blocks will automatically be added to the corresponding areas of the work contour.

5.4.20 Roll type

In the finishing operations, when machining a detail surface it is possible to mark out the following trajectory sections:

- machining area of form-creating surfaces;
- edges rounding area between these surfaces;

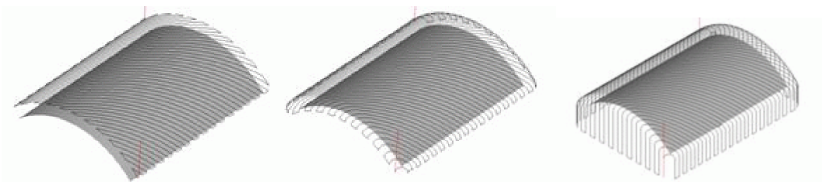
- bypass area of restricting model.



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 trajectory decreases and consequently, decreases the machining time on a milling unit. Defining the roll type as surfaces only activates this tool movement trajectory creation method.

In the with edges mode, the resulting trajectory 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 trajectory. Besides this, in the plane and drive operations all the vertical trajectory areas will be added.



Surfaces only With edges With restricting model

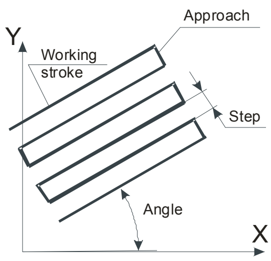
The roll type can be defined in the Strategy page.

5.4.21 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.

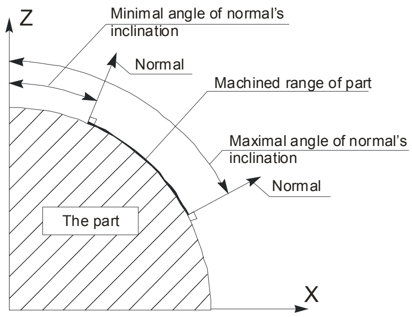
The angle of work passes for plane operations can be assigned in the Strategy page.



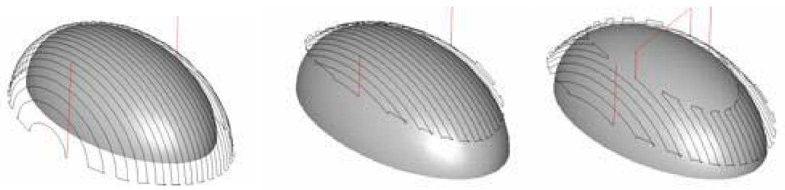
5.4.22 Maximum slope angle of normal

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).

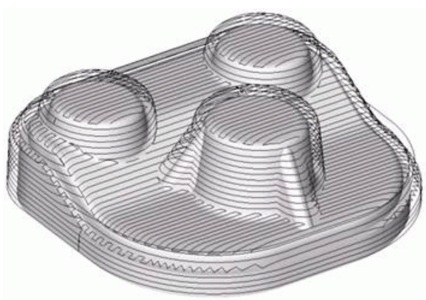


It is understood that the plane machining method is optimal when milling surfaces that are closer to the horizontal, and the waterline machining gives best results when machining surfaces that are closer to the vertical. Use of the maximum slope angle of the normal allows the user to machine the horizontal 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 trajectory 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.



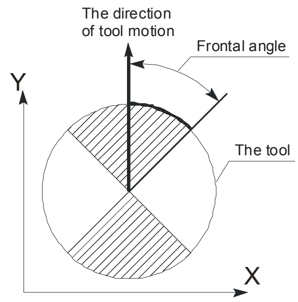
The maximum slope angle of the normal can also be used for milling horizontal areas using a cylindrical tool or for machining surface areas with a slope angle equal to the side angle of the tool e.g. conical mill.

The maximum angle of the normal for the finishing operations can be assigned on the Strategy page.

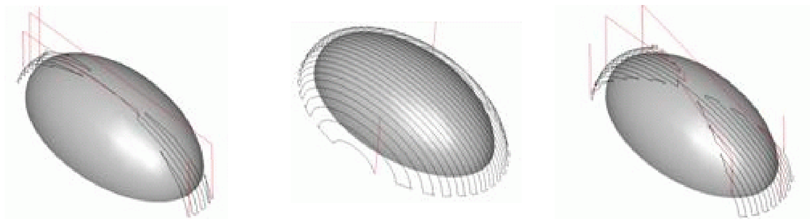
5.4.23 Frontal angle

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 trajectory 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.

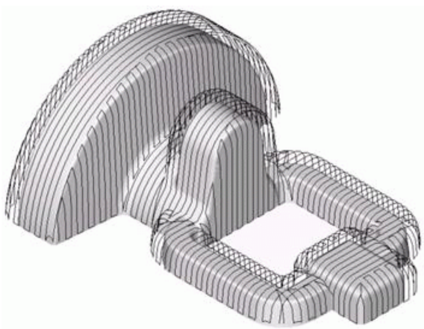


In the picture, there is a spherical mill - viewed from above. An area of the work pass will be included in the resulting trajectory only if the angle between projections onto the horizontal plane of the tool movement direction and the normal vector to the surface at the contact point is less than defined. For the spherical mill, the following will also be true: any area of the pass will be included in the resulting trajectory 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).

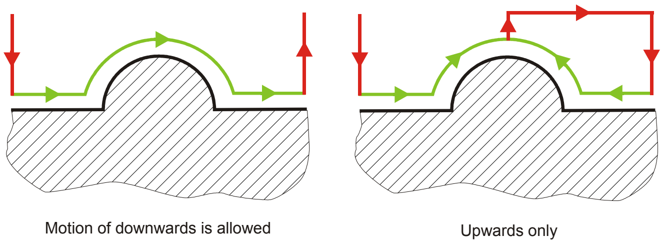


The frontal angle can be within the limits of 0 degrees (machine only perpendicular to the movement direction) to 90 degrees (without limitation). For two plane operations that are mutually perpendicular, the optimal value for the frontal angle will be equal to 45 degrees.

The frontal angle for the plane finishing and drive operations can be defined in the Strategy page.

5.4.24 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.

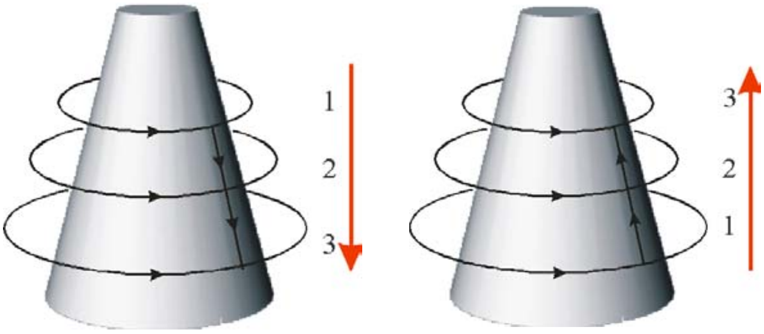
The machining upwards only mode for the plane finishing and drive operations can be defined in the Strategy page.

5.4.25 Machining direction

In the waterline finishing and combined operations it is possible to define the priority direction of work passes joining in the resulting trajectory. 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.



5.4.26 Machining methods in drive operations

In the drive operations, the view of the tool trajectory 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

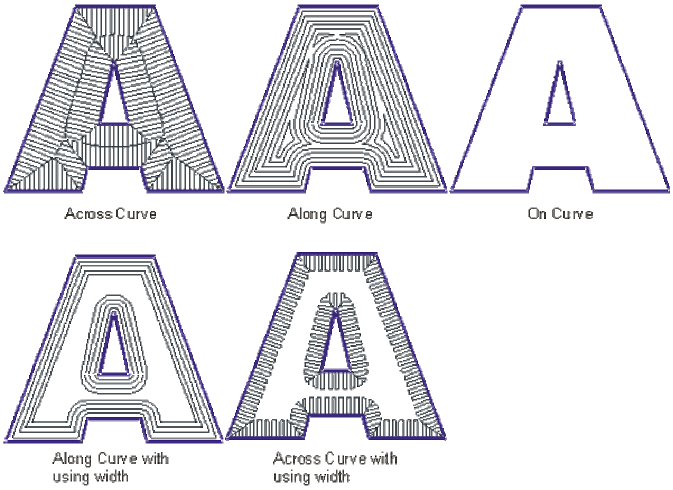
along the curves, which are equidistant to the area borders. The step between the neighboring passes is equal to the defined machining step.

- **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.

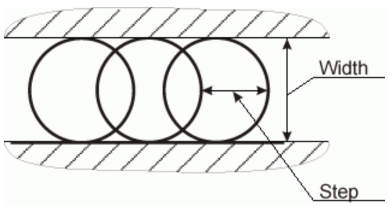
The view of the trajectories with different combinations of parameters set.



The machining method for the drive operations can be assigned in the Strategy page.

5.4.27 Trochoidal machining

Trochoid 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 cycloid and at the same time secure the tool.



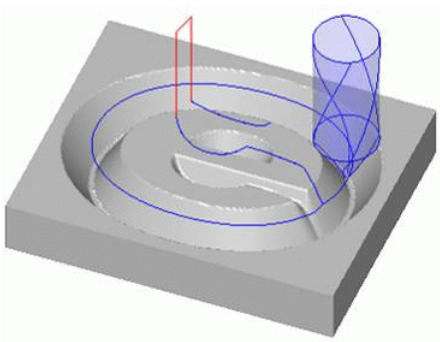
The width and the step of the trochoidal path depends on the trochoid step mode. **SprutCAM** gives 4 modes:

- **Do not use trochoid**. 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.

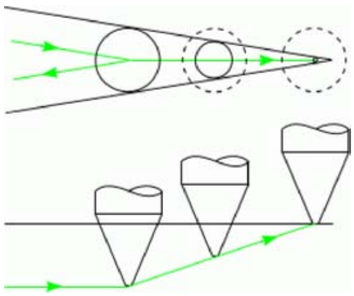
- **For islands removing only**. The mode generates the minimal quantity of the additional circles. The circles diameter is minimal and enough to remove the islands. The trochoidal step is equal to the tool diameter.
- **With pocket step**. The mode guarantees the uniform tool load. The radius of the additional circles cannot be less than the smooth radius. The step of the trochoid is less or equal to the pocket step. The mode is recommended for high speed machining.
- **Reduce on the cleanup**. The trochoidal tool path is generated similarly to the third mode but the trochoidal step on the cleanup is equal to the cleanup step to reduce the tool vibration.

5.4.28 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 trajectory 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.

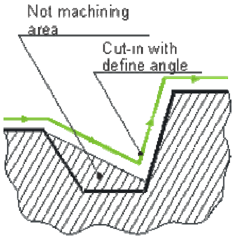
The 3D corner machining function for the engraving and pocketing operations can be activated on the Strategy page.

5.4.29 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.

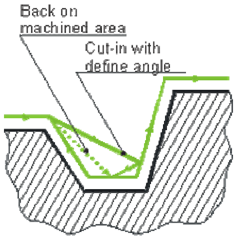
One of the following descent types can be used:

- **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.



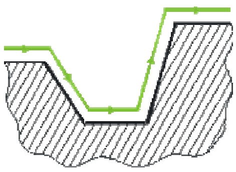
Control cut-in angle
without re-machining shadow areas

- **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).



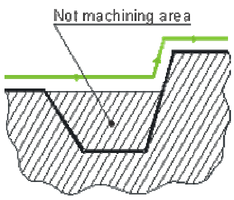
Control cut-in angle with
re-machining shadow areas

- **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

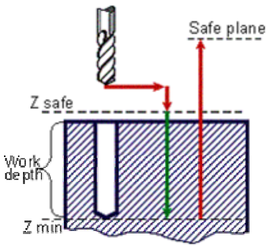
The descent type for the plane roughing and drive operations can be assigned in the Strategy page.

5.4.30 Hole machining cycles

Machining cycles for the hole machining operations can be assigned in the Strategy page.

Simple drilling (G81)

Performs drilling of a hole with a rapid approach to the safe distance and subsequent retraction to the safe plane.

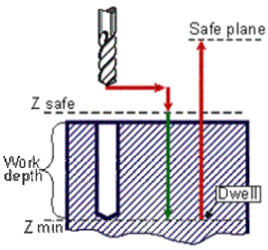


Drilling cycle of the G81 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to Z min.
- Rapid retraction of the tool to the safe plane.

Face drilling (G82)

Performs drilling of a hole with the rapid approach to the safe distance and with dwell upon reaching the Z min level with spindle running and subsequent retraction to the safe plane. Dwell time is assigned as seconds in the Dwell time field.

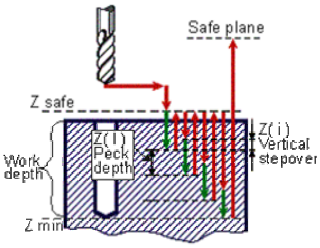


Drilling cycle of the G82 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to Z min.
- tool dwell.
- Rapid retraction of the tool to the safe plane.

Deep drilling

Performs tool approach to the safe distance and following cycled drilling according to the defined values of peck depth ZI and withdrawal value Zi, which can be assigned in the Peck depth (ZI) and Vertical stepover (Zi) fields.

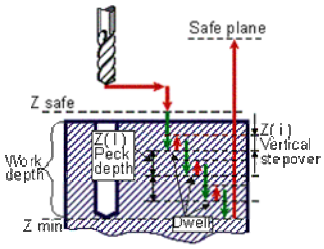


The cycle includes the following:

- Rapid feed of the tool to Z safe.
- work pass to the peck depth.
- Rapid retraction of the tool to Z safe.
- Rapid pass to the ZI - Zi distance.
- Work pass to the ZI + Zi distance.
- Rapid retraction of the tool to Z safe.
- Rapid pass to the 2*ZI - Zi distance.
- work pass to the ZI + Zi distance.
- Rapid retraction of the tool to Z safe.
- Steps 7 to 9 repeated till the required depth has been reached.
- Rapid retraction of the tool to the safe plane.

Drilling with chip break

Performs tool approach to the safe distance and following cycled drilling regarding the defined values of peck depth ZI, withdrawal value Zi and dwell, which can be assigned in the Peck depth, Vertical stepover and Dwell time fields

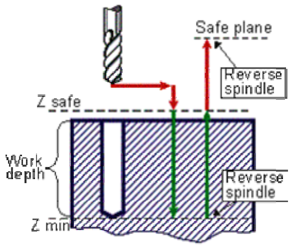


The cycle includes the following:

- Rapid feed of the tool to Z safe.
- Work pass to the peck depth (ZI).
- Retraction to the withdrawal value (Zi) and dwell.
- Work pass to the ZI + Zi distance.
- Retraction to the (Zi) value and dwell.
- work pass to the ZI + Zi distance.
- Retraction to the (Zi) value and dwell.
- Steps 6 to 7 repeated till the required depth has been reached.
- Rapid retraction of the tool to the safe plane.

Tap drilling (G84)

Performs tool approach to the safe distance, threading (tapping) with following return at the work feed with reverse spindle rotation.

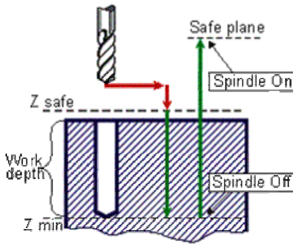


Drilling cycle of the G84 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to the Z min distance.
- Spindle reverse and retraction on the work pass to the safe plane.
- Restoration of the initial spindle rotation speed and direction

Bore type 5 (G85)

Performs tool approach to the safe distance, boring at the work feed with spindle stopping at the minimal level and retraction at the work feed to the safe plane.

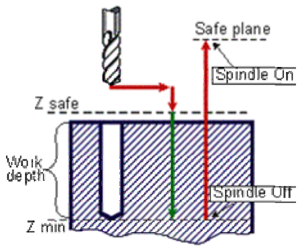


Drilling cycle of the G85 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to the Z min distance.
- Spindle stop.
- Retraction on the work feed to the safe plane.
- Restoration of the initial spindle rotation speed and direction.

Bore type 6 (G86)

Performs tool approach to the safe distance, boring at the work feed with spindle stopping at the minimal level and retraction at the rapid feed to the safe plane.

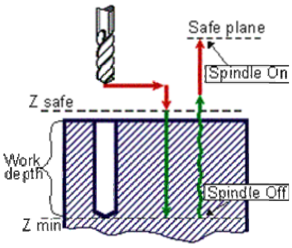


Drilling cycle of the G86 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to the Z min distance.
- Spindle stop.
- Rapid retraction of the tool to the safe plane.
- Restoration of the initial direction and spindle rotation frequency.

Bore type 7 (G87)

Performs tool approach to the safe distance, boring at the work feed with spindle stopping at the minimal level and manual retraction to the safe plane.

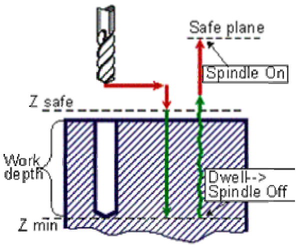


Drilling cycle of the G87 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to the Z min distance.
- Spindle stop.
- Manual retraction to the safe plane.
- Restoration of the initial spindle rotation speed and direction.

Bore type 8 (G88)

Performs tool approach to the safe distance, boring at the work feed with dwell and following spindle stopping at the minimal level and then manual retraction to the safe plane.

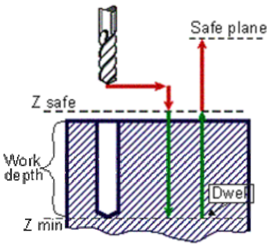


Drilling cycle of the G88 type includes the following:

- Rapid feed of the tool to Z safe.
- work pass of the tool to the Z min distance.
- Dwell.
- Spindle stop.
- Manual retraction to the safe plane.
- Restoration of the and spindle rotation speed and direction.

Bore type 9 (G89)

Performs tool approach to the safe distance, boring at the work feed with dwell and following retraction at the work feed to the safe plane.



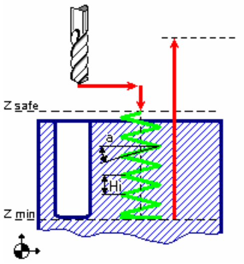
Drilling cycle of the G89 type includes the following:

- Rapid feed of the tool to Z safe.
- Work pass of the tool to the Z min distance.

- Dwell.
- Retraction on the work feed to the safe plane.

Hole machining by spiral

This method is designed for the machining of round holes, with a diameter larger than the diameter of the tool, but less than 1.5 times the diameter. The hole machining is performed using spiral tool passes (movement of the rotation axis round a circle with gradual tool lowering). The diameter of the spiral is selected according to the defined diameter of the hole and tool size.

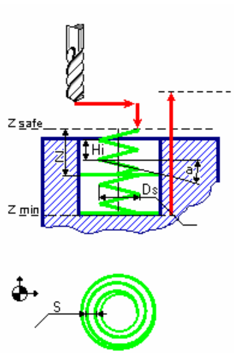


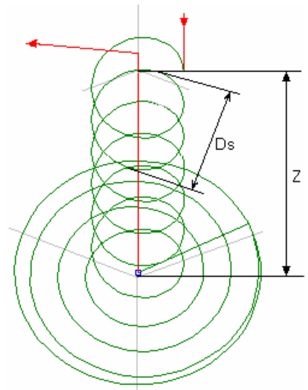
Spiral machining includes the following:

- Rapid feed of the tool to Z safe.
- Spiral work pass of the tool to the Z min depth. The spiral diameter is equal to the difference between the hole diameter and the tool diameter, and the tool lowering speed is defined by the plunge angle (a) or spiral step (Hi).
- Pass round a circle at the bottom level. Whose diameter is equal to the spiral diameter.
- Retraction to the hole center.
- Rising on the rapid feed to the safe plane

Hole pocketing

The method is designed for machining of round holes, with a diameter much bigger than the tool diameter. Pocketing is performed using layers. The tool cuts-in by spiral to every layer and then widens the hole to the required diameter by moving along the Archimedes spiral with clean pass of the mill round the circle. The tool movement trajectory by spiral will be performed using the circular approximation.





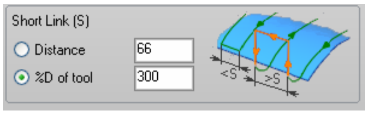
Hole pocketing includes the following:

- Rapid feed of the tool to Z safe.

- Cut-in by spiral to the plunge depth Zl. The spiral diameter Ds is assigned as a percentage of the tool diameter, and the tool lowering speed is defined by the plunge angle (a) or spiral step (Hi).
- Movement along the Archimedes spiral on that level with the S step, before the tool rotation axis comes to the circle with diameter equal to the difference between the hole diameter and the tool diameter.
- Clean pass round the circle of the defined diameter without changing the level.
- Steps 2 to 4 repeated till the required hole depth has been reached, with stepover to the start point of the next plunge without changing the level.
- Retraction to the hole center.
- Rising on the rapid feed to the safe plane.

5.4.31 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.

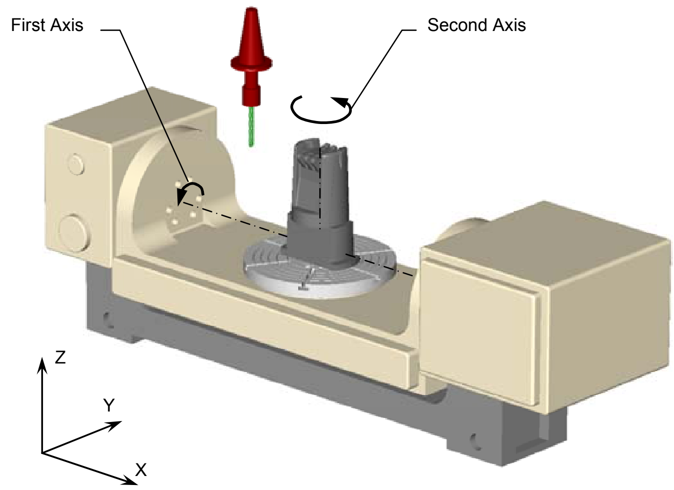


5.4.32 Rotary axes

SprutCAM enables to make NC-programs for indexed four- and five-axes milling (3+1 and 3+2 axes milling). So at first rotary axes are positioned for an operation and next the three-axes milling starts in process.

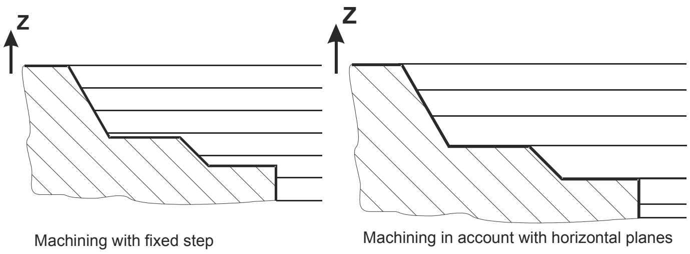
Availability and location of rotary axes of the machine must be described in the Parameters window of Job list (the root node of the operations tree). According with the machine configuration the rotary axes can be unavailable (for a three-axes milling machine), alone axis is available (for a four-axes machine) or two axes are available (five-axes machine). The axes placement and they direction are described in the main coordinate system of the machine. There is a rule for five-axes machines: the first rotary axis turns a table relative to the frame and the second axis revolve a chuck on the table. Parameters of the second rotary axis must be set for the initial position of the first axis.

The rotary axes position for each of operations is assigned in the Parameters window. A local coordinate system is automatically created for every new position of rotary axes. It is possible to create manually the local coordinate system by the axes position before. Next, this one can be used too to fill the axes position value for an operation in the same Parameters window.



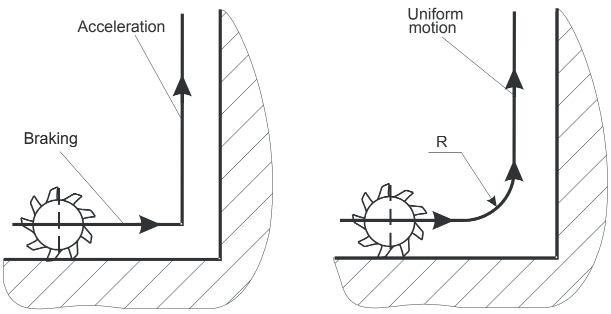
5.4.33 Machining horizontal planes (Clear flats)

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.

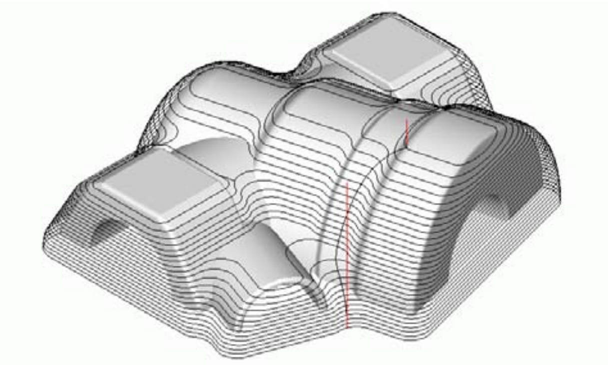


In the waterline machining, the tool passage levels are defined by the step value, which assigns the thickness of the material being removed. Upon activating the clear flats function, the step becomes variable. If on the layer being machined there are horizontal planes, then the value of the step being 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.

5.4.34 Corners smoothing



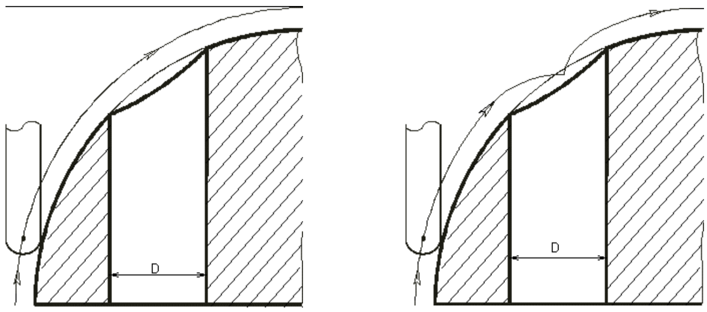
When there is a sudden change of the tool movement direction, the milling unit performs deceleration before starting the turn, and then accelerates again. This fact can lead to vibrations and high tool and milling machine wear. The problem can be solved if the trajectory has very few or no breaks. For this reason, in the system there is the trajectory smoothing function at the defined radius for machining inner corners of the model.



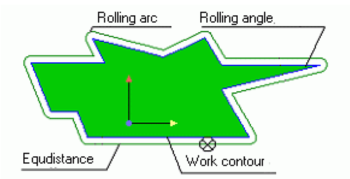
An example of machining with corner smoothing.

5.4.35 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.

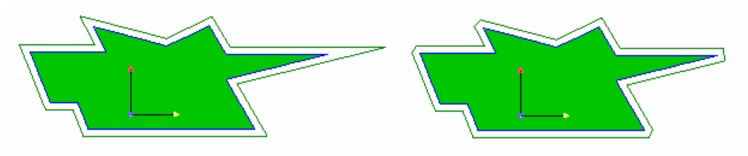


5.4.36 External corner roll types



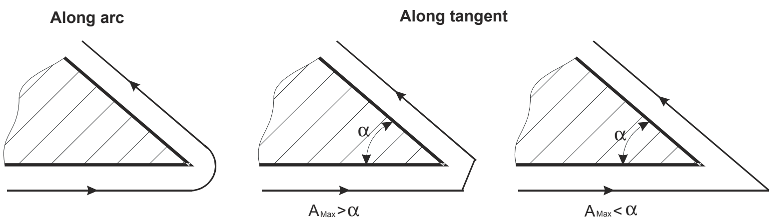
In **SprutCAM** there are two types of external contour corner rolling - by arc and by tangent. Corner rolling is possible only when **SprutCAM** creates an offset toolpath. Selection of the roll type can be made on the <Strategy> page. Select the Corner roll type area by choosing either the By arc or By tangent mode. When rolling by arc, an arc whose radius is equal to the offset value will be inserted into the trajectory corner.

The By tangent mode in its part allows performing two types of corner rolling - without rolling and rolling by tangent. The work contour can contain any type of corners, including very sharp ones. Going around a sharp corner without rolling is not recommended; not rolling the tool in this case will create a longer toolpath. The By tangent roll type is applied to get the tool around a corner the shortest way. The way this condition is applied is based on the angle of the corner being rolled. If the By tangent option on the <Strategy> page is selected, and the Amax angle value is set, then the tool will roll by tangent all corners sharper than the Amax value, and corners bigger than Amax along the shortest distance, using a straight line.



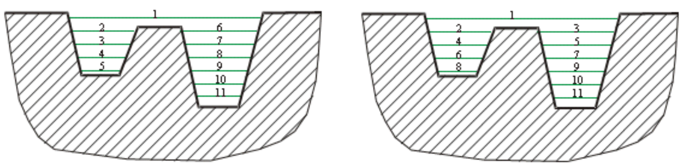
In contour machining operations, the left picture, Amax = 1° i.e. rolling will be activate on angles the size of which is less than one degree. The operations in the right picture Amax = 90°, corner rolling will be performed on angles less than ninety degrees accordingly.

Roll types



5.4.37 Machining order (by depth or by contours)

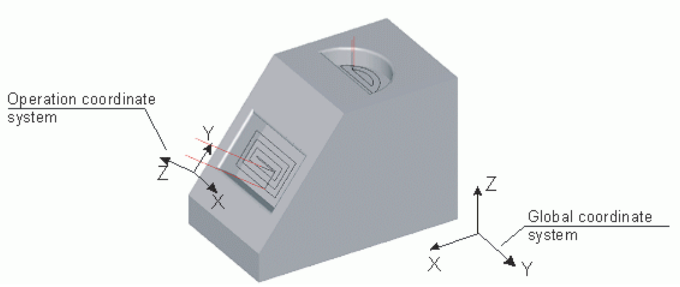
When machining multiple contours in several passes, two options exist for controlling the order of depth machining. When machining by depth, after completion of the first pass along the first contour the system then performs execution of the first pass along the next contour. The execution of the second 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

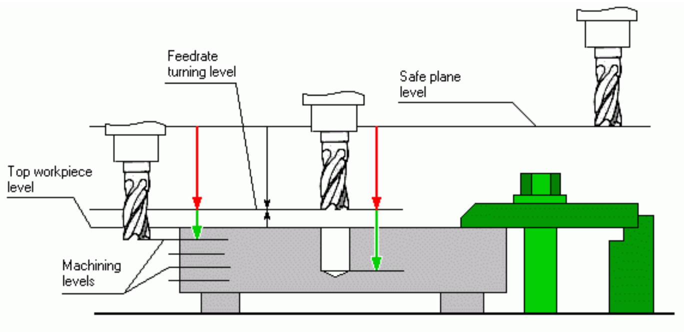
5.4.38 Operation coordinate system

In the system it is possible to program machining for several workpiece setups. All operations for one workpiece setup have to have an identical coordinate system. The operation coordinate system defines the coordinate system of the workpiece and can be setup on the <Parameters> page in the <Operation parameters> window of the current operation. Any previously created coordinate systems can be used as the operation coordinate system (see how to assign coordinate system refer to "Coordinate system").



5.4.39 Tool plunge

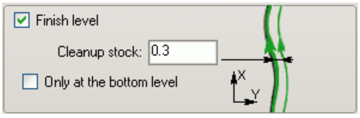
Tool plunge is a movement of a tool from the plane level (depth) of tool stepover along the Z-axis from the top machining level, which can be assigned in the parameters assignment window. If the stepover plane level coincides with the top machining level, for example, during stepover around the workpiece, 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 Toolpath window.

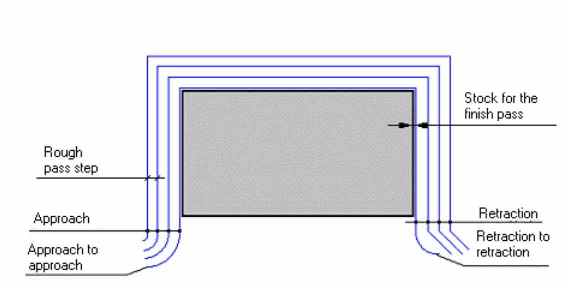
5.4.40 Assigning a finish pass in the XY plane

A finish pass for closed and open curves in 2D machining operations can be set up in the Strategy window.



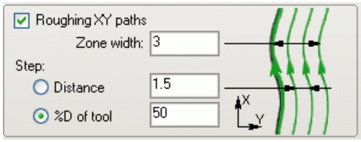
When activating this option, the additional stock will be left on the model, which will be removed by the last finishing pass. It will allow the user to obtain higher quality surface finish.

The value of the stock can be defined as the absolute value or as a percentage of the tool diameter.

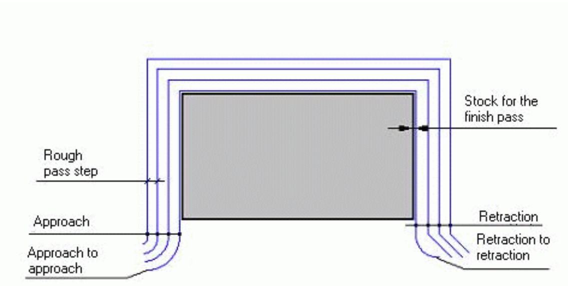


5.4.41 Assigning a rough pass in the XY plane

Roughing passes for closed and open curves in 2D machining operations can be set up in the Strategy window.

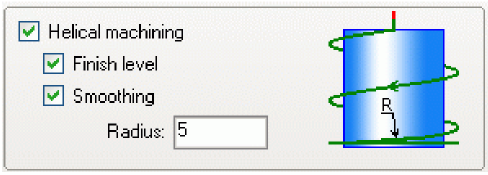


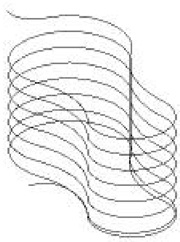
When activating this option, the stock from the model will not be removed in one go; instead, it will be done according to the assigned parameters. The value of the stock can be defined as the absolute value or as a percentage of the tool diameter.



5.4.42 Helical machining

When machining 2D curves it is possible to obtain a spiral-like trajectory. 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.



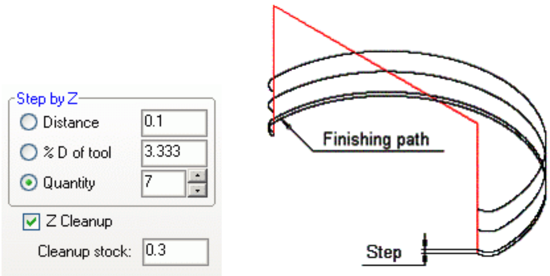


The number of spiral loops depends on the Z step assigned in the <**Parameters**> page. If a tick is placed in the "**Clean pass**" box, the system will generate a horizontal trajectory area for bottom machining.

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.

5.4.43 Z cleanup

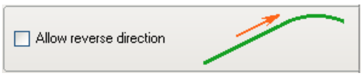
In 2D and 2.5D machining operations, it is possible to activate the clean pass mode in depth, e.g. at the bottom level. This is assigned on the <Parameters> page in the <Operation parameters> window. All previous passes will be automatically allocated equally along the Z axis.



5.4.44 The allow 3D toolpath

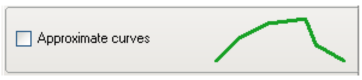
The allow 3D toolpath mode allows the system to create a 3D toolpath for material removal in all areas that are not accessible for machining on the current level (e.g. inner corners).

5.4.45 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.

5.4.46 Approximate curve



Approximate curve allow approximate trajectory by arcs.

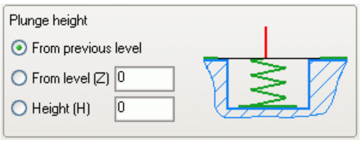
5.4.47 Idling minimization

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.

5.4.48 Machine By Layer

The material may be deleted for some passes. For this purpose, it is necessary to switch on Machine by Layer. 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.

5.4.49 Plunge Height



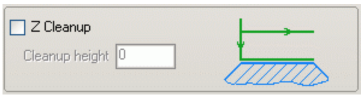
The height of the layer is set in the Plunge Height panel; in particular, it defines a value of level Z on which the given type of plunge will join.

- 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.

5.4.50 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.

5.4.51 Z cleanup



For the definition of width machined material layer on finish pass, it is necessary to switch on an Z cleanup option and to enter a stock value on finish pass.

6. MACHINING SIMULATION

6.1 DESIGNATION OF THE SIMULATION MODE

SprutCAM has the embedded simulator with the dynamical and realistic visualization.

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 show the rest areas the thickness of that is more then the defined value;
- 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.

6.2 THE OBJECTS OF THE SIMULATION MODE AND THE CONTROL OF ITS VISUALIZATION

The graphical view in the simulation mode shows 5 objects. The next buttons allows to show, to hide and to change the visualization mode of these objects:

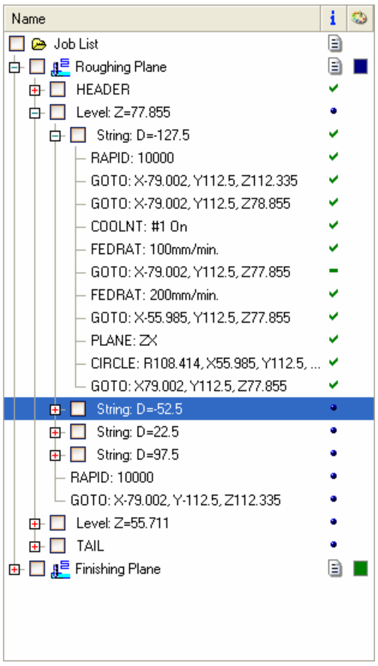
-  - shows/hides the source model in the graphical view;
-  - shows/hides the tool path of the calculated operations in the graphical view. The tool path of the selected operation is visible always. The visibility of the other operations depends on the ticks near to the corresponding tool path node.
-  - shows/hides the tool of the current operation in the graphical view;
-  - shows/hides the tool holder. This button will not take the effect if the tool is invisible or if the holder on the current operation is not defined;
-  - shows/hides the simulated part. If the simulated part is invisible while the block-by-block simulation then the simulation speed grows.
-  - defines the mode of the simulated part representation. If the button is down then the zone of the simulated part has the color of the tool that machined this zone. Else the simulated part is monochrome.
-  - shows the dialog to define the color of the simulated part.

6.3 THE TOOL PATH

6.3.1 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 is 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.



6.3.2 The list of the basic CL-data commands

The **CLDATA** command that is generated by **SprutCAM** is listed in the Table 1. The full list of the **CLDATA** commands see in the part 3 of this manual “Postprocessors generator”.

Table 1

Word	Value
CIRCLE	Motion by circle arc
COEFF	Coefficients of the spline
COMMENT	Comment
COOLNT	The coolant Switch on/off
CUTCOM	The tool radius and tool length compensation
CYCLE	Holes machining cycles
DELAY	Delay
FEDRAT	Fed rate
FINI	The end of the commands list
GOTO	The linear motion
LOADTL	The tool loading
MOVNRB	The start of the spline curve
PARTNO	The part number

PLANE	The work plane (XY/YZ/ZX)
PPFUN	Postprocessor function
RAPID	Rapid motion
ROTABL	Rotary axis
SPINDL	The spindle Switch on/off
Header	The group of elementary commands in the operation begin
Tail	The group of elementary commands in the operation end
Level	The group of elementary commands that move the tool in the fixed Z-level
String	The logical group of elementary commands
Block	The group of elementary commands that consists the logical block of tool path
Offset	The group of elementary commands that consists the motions on one offset curve

6.3.3 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 move the tool. The first group is the motions commands and it contains the GOTO and CIRCLE commands. The all other commands are related to the second group.

If the tool path is visible (button is down), then each motion command will have the equivalent curve fragment in the graphical view. All motion commands define the tool trajectory curve. There is the <> near the every node of the tool path tree, that contains other nodes inside. The selected node is shown on the screen independently of the tick state. To show the tool path outside of the selected node it is necessary to tick near the required node.

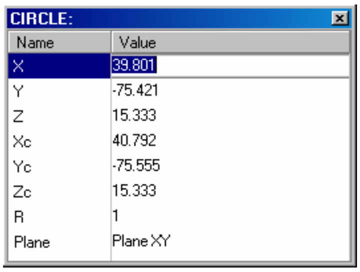
Click the button before the selection of the required tool path fragment. The fragments inside or the selected node is highlighted then the mouse is moved. It is necessary to click the left mouse button on the highlighted object to select it. The tool path outside of the selected node is shown transparently and cannot be selected. Press and hold “Shift” key and double click in the graphical view to select the parent of the selected node.

6.3.4 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.

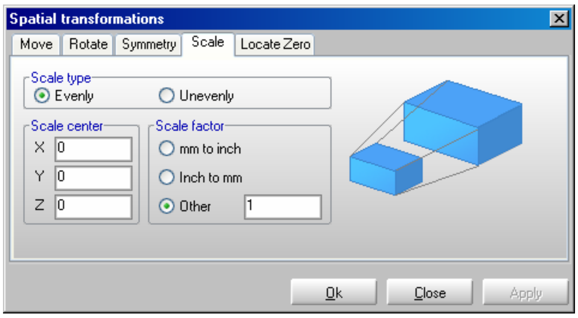
Double click on the node or select “edit” item from the context menu to edit the parameters of the selected **CLDATA** command. The opened dialog is not modal i.e. it is not need to close the dialog to edit other node. The dialog caption is a type if the edited node. The list of the parameters depends on the node type.


CIRCLE:

Name	Value
X	39.801
Y	-75.421
Z	15.333
Xc	40.792
Yc	-75.555
Zc	15.333
R	1
Plane	Plane XY

6.3.5 Tool path spatial transformations

Click the right mouse button on the operation and select the “Tool path transformation…” item from the context menu. The opened dialog is an analogue of the model transformation dialog.



The type of the spatial transformation is selected by the bookmark:

1. Move bookmark allows to define the linear move the tool path by three axis;
2. Rotate bookmark allows to rotate the tool path around Z axis;
3. Symmetry bookmark allows to make the symmetry transformation relative any vertical plane;
4. Symmetry bookmark allows to perform evenly and unevenly scaling of tool path
5. Locate zero bookmark allows to move the tool path using the dimensions of toolpath.

6.4 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.

To drop the simulation results, one should press the button. When pressed, the workpiece will be reinitialized according to the parameters defined in the Simulation settings window.

The model being machined can be displayed using the following methods:

Cutting part – shows the model being machined, the image is very similar to the machining results obtained on the milling machine. This method is set by default and is most frequently used.

Rest areas – visualizes only the unmachined material with a layer thickness that exceeds the value in the Show rest: field of the Simulation settings window. The method is recommended for estimation of the location and the amount of residual material, and to estimate the thickness of the layer of the unmachined material.

Gouges – shows only the detail gouge areas. Gouges are formed, for example, in areas of machining with a negative stock.

Note: *If the 'Gouge' and 'Rest areas' options are used with a low detail tolerance, this may give incorrect results on surface areas that are close to vertical. In this case, it is recommended to magnify the problem area in the graphic window and wait for more precise results, or to increase the detail tolerance in the Simulation settings window.*

6.4.1 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, after the last command run and the simulation model formed. 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.

- - runs the back simulation from the current node to the start of the list. The screen updating 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 is stopped after the next node is arrived;
- -runs the step-by-step simulation from the current node to the end of 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 process stopped then the sibling node is arrived. The screen update is performed once after simulation;
- - runs the fast simulation from the first operation until the current node. The screen update is performed once after simulation;
- - runs the fast simulation of the full operations list. All commands run independently of the current node. The screen update is performed once after simulation;

Just below there is a tool movement method selection switch.

If **By blocks** is selected, then the tool will move block-by-block, with drawing at intermediate points only.

When the **Smooth** method is selected, the tool will move smoothly with constant speed. In this mode, the 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.

Changing the method and movement speed is possible during the simulation process.

6.4.2 Tool path errors detected by simulation

Every node of the tool path tree has the status. The status of the node is indicated by icon:

- < > - Operation is disabled. It is not output to the NC program and not simulated;
- < > - Operation is not calculated (has no the 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 is disabled. It is not output to the NC program and not simulated;
- < > - **CLDATA** commands is 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.

While the simulation process the influence of the **CLDATA** command on the workpiece is, analyzed and corresponding status is assigned to the command. Three types of errors are checked:

Статья I. Stock removing on the rapid feed is forbidden. If the stock is removed and the feed rate is rapid then command is marked as error;

6. Tool holder collision. If tool holder touch the workpiece when the **CLDATA** command is marked as error;

7. Impossible to calculate the compensation. Tool radius compensation is performed while the simulation. The compensation value is defined in the operation parameter window on the tool bookmark. If the compensation cannot be performed for the **CLDATA** command then the motion is marked as error.

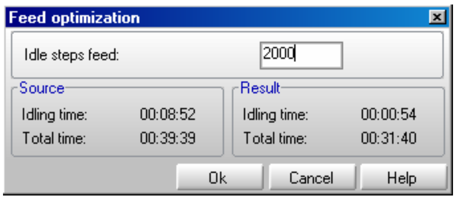
The type of the error is described in the hint that appears after delay under the **CLDATA** command with error.

Button defines the action then the error is detected. If 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:

- **Next error** (Ctrl+N) – move the selection to the next error marked node;
- **Previous error** (Ctrl+P) – move the selection to the previous error marked node.

6.4.3 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:

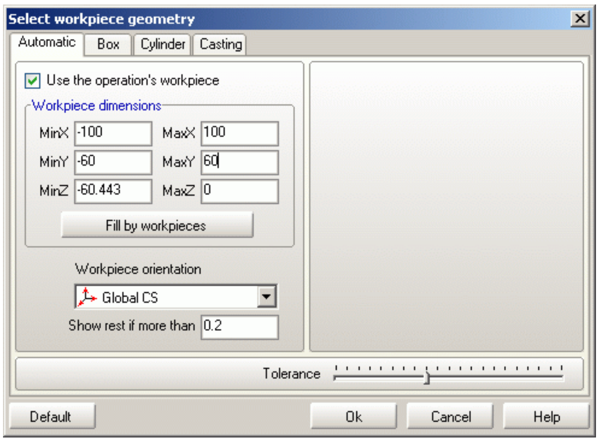


On the left hand, side of window the total machining time and the time idle motion is displayed. On the right hand side, the obtained with the defined feed rate value the total and idle time is displayed. After "**OK**" the new "**fedrat**" commands is inserted into the **CLDATA** sequence to change 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.*

6.4.4 Assigning workpiece parameters

The simulation settings editing window opens when the button in the machining simulation window is pressed.



In the window, the user can assign parameters that will be used. The first time the simulation subsystem is run, the parameters will be calculated automatically. This means that for the workpiece dimensions, the system will take the aggregate dimensions of the models being machined and the workpiece for all operations of a single machining process. By default, the system will consider the forms of operation workpieces. The workpiece for simulation is obtained by summarizing the workpiece of all operations of a machining process.

When the **Use the operation's workpiece** option is selected, the workpiece model will be created by summarizing all the available workpieces in the machining process. This means that if in every operation of the machining process, a different workpiece model has been assigned, and then in the simulation, an aggregate model will be displayed. If there is no tick in the box, then for simulation the system will use an extrusion with the dimensions defined in the Workpiece dimensions fields.

In the **Workpiece dimensions** fields the user can define the workpiece dimensions for simulation. If the defined dimensions are less than the aggregate dimensions of workpieces of operations and there is the tick in the Use the operation's workpiece box, then in simulation will act only on that part of the aggregate workpiece that does not exceed the defined dimensions. Such workpiece assignment method allows more precise machining simulation of a small area with an economical use of the computer resources. If the Fill by workpieces button is pressed, the workpiece dimensions will take the aggregate dimensions of the models and workpieces being machined for all operations of the machining process.

The workpiece can be oriented relative to any of the existing coordinate systems.

The position of the **Detail tolerance** slide, defines the relative visualization tolerance with which the detail being machined will be obtained after machining simulation. The higher the detail tolerance, the more accurate the visual model will be; the simulation however will require more computer resources.

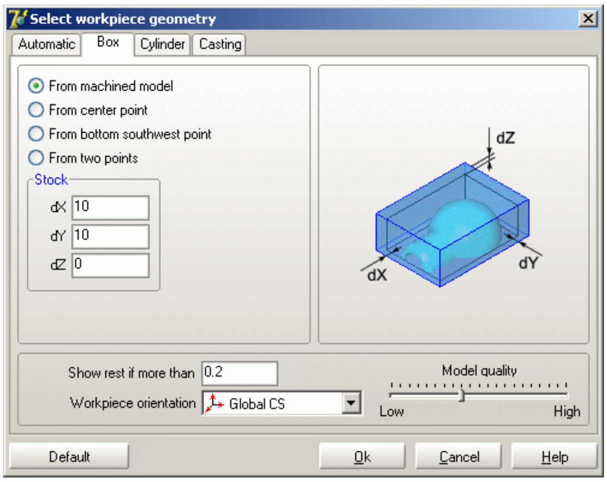
Note: *Assigning high tolerance is not recommended on slower computers.*

The **Show rest if more than** parameter allows the user to assign the layer depth value of unmachined areas (according with the chosen measurement units). When visualizing residual material or gouges, only those areas with a material layer thickness that exceeds the defined value will be displayed in the graphic window.

When the "Defaults" button is pressed, the dimensions of the workpiece will be calculated as the aggregate dimensions on the details being machined and workpieces of all operations of a machining process, and the system will put the tick in the Use the operation's workpiece box, and position the Detail tolerance slide to 1/3 from the maximum allowed.

To close the window and save all changes press the "Ok" button. If the window is closed by using the "Cancel" button, no changes will be applied.

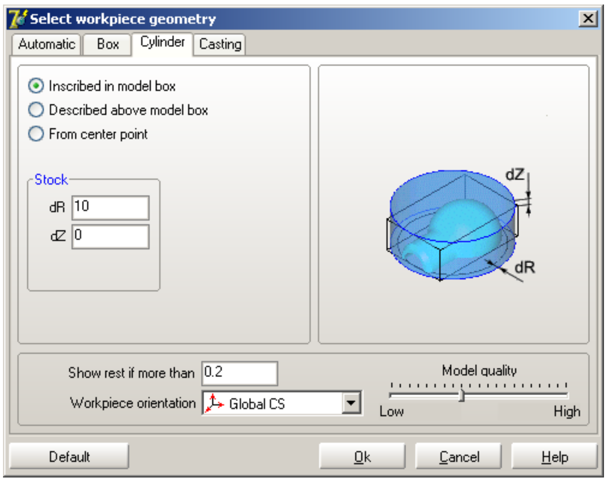
Assigning workpiece by box.



The "**Box**" mode allows the user to define a workpiece as a cube. The edges of the parallelepiped are always parallel to the planes: XY, XZ, and YZ. The dimensions and the size of the cube can be defined using several methods:

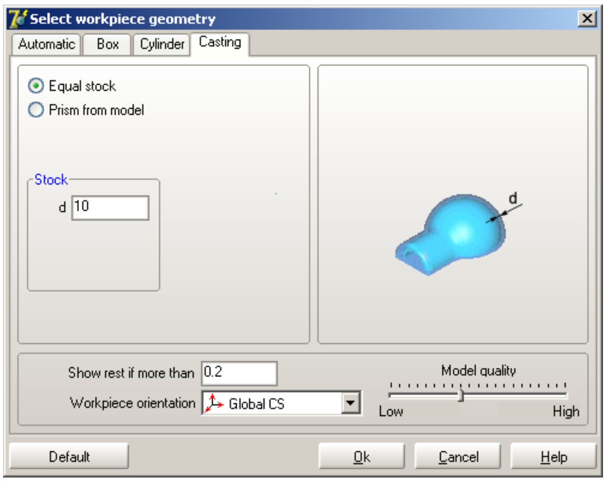
- When the **From machined model**, option is selected, the coordinates for the corner points are calculated by the system so that the entire model for machining can fit inside the cube. An additional stock amount can also be added for the X and Y-axes.
- When the **from center point** option is selected, the coordinates for the centre point of the cube and the overall dimensions are required.
- When defining the position and dimensions of the box using **the from bottom southwest point** option, the user should enter the coordinates of the corner point and the overall dimensions for the cube.
- When using the box **from two point's** option, the coordinates of the diagonally opposite corners of the cube should be entered. In all cases, the size and the position of the workpiece in the Z-axis is defined by the top and the bottom machining values entered in the parameters window.

Assigning workpiece by cylinder.



The "Cylinder" mode allows the user to define a cylindrical workpiece. The axis of the cylinder is always parallel to the tool axis. The height of the cylinder and its position along the Z-axis is defined by the top and the bottom machining levels that are entered on the Parameters page in the operation parameters window. The diameter and coordinates of the cylinder axis can be defined by three methods: When defining a cylindrical workpiece by inscribed in model box the coordinates for the center and the diameter are calculated such that the cylinder touches the inside edges of a theoretical box that contains the model. When defining a cylindrical workpiece using the described above model box option, the coordinates for the center and the diameter are calculated so that the cylinder touches the outer corners of a theoretical box that contains the model. The From center point method allows to define the coordinates of the center and the diameter manually.

Assigning workpiece by casting.



The "Casting" mode allows assigning a workpiece that is similar to the geometry of the model. This workpiece definition method is only available in volume machining operations. When defining a workpiece with equal stock, an equal stock amount is added to the model for machining, and this is used as the workpiece. The stock value is defined in the d field. When defining the casting as an Extrusion from model, a vertical extrusion of the outer edges of the model is used as the workpiece. The height of the extrusion is defined by the top and bottom machining levels entered in the parameters window. If a stock value is entered in the dXY field, the outer edges of the extrusion are offset by the defined distance.