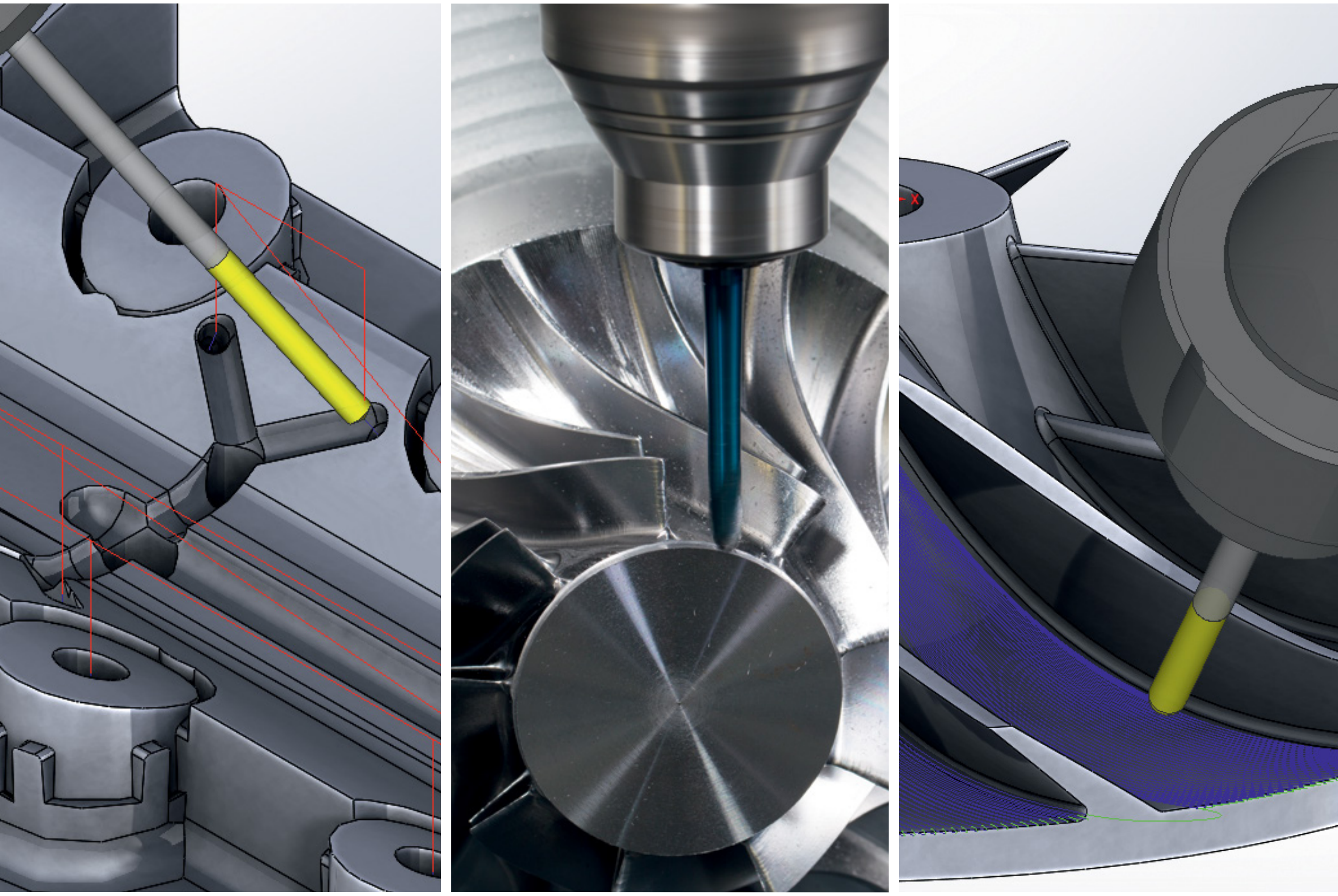
SolidCAM + SolidWorks

The complete integrated Manufacturing Solution

SolidCAM

iMachining – The Revolution in CAM!



SolidCAM User Guide: Simultaneous 5-Axis

iMachining 2D & 3D | 2.5D Milling | HSS | HSM | Indexial Multi-Sided | Simultaneous 5-Axis | Turning & Mill-Turn | Solid Probe

www.solidcam.com

SolidCAM 2013

Simultaneous 5-Axis Machining User Guide

Contents

1. Introduction

1.1 Adding a Sim. 5-Axis Operation 7
1.2 Sim. 5-Axis Operation dialog box 8
1.3 The stages of the Sim. 5-Axis Operation parameters definition 11

2. CoordSys

CoordSys page 14

3. Geometry

3.1 Geometry 16
  3.1.1 Drive surface definition 17
  3.1.2 Drive surface offset 18
  3.1.3 Curves definition 18
  3.1.4 Work type 19
  3.1.5 Edge curve 20
  3.1.6 Edge surface 20
  3.1.7 Lead curve 22
  3.1.8 Start and End edge curve 23
  3.1.9 Projection curves 26
3.2 Area 28
  3.2.1 Full, avoid cuts at exact edges 28
  3.2.2 Full, start and end at exact surface edges 29
  3.2.3 Limit cuts by one or two points 31
  3.2.4 Determined by number of cuts 31
  3.2.5 Extend/Trim 33
  3.2.6 Angle range 35
  3.2.7 2D Boundary 36
  3.2.8 Round corners 37

4. Tool

4.1 Tool definition 40
4.1.1 Spin definition 41
4.1.2 Feed definition 41
4.1.3 Rapid move parameters 43

5. Levels

5.1 Clearance area 46
5.1.1 Plane 46
5.1.2 Cylinder 48
5.1.3 Sphere 50
5.1.4 Keep initial orientation until distance 50
5.1.5 Arc fit 52
5.2 Machining levels 52
5.2.1 Retract distance 52
5.2.2 Safety distance 53
5.2.3 Rapid retract 53

6. Tool path parameters

6.1 Surface quality 56
6.1.1 Cut tolerance 56
6.1.2 Distance 57
6.1.3 Maximum Step over 57
6.1.4 Scallop 58
6.1.5 Surface edge merge distance 58
6.1.6 Apply outer sharp corners 59
6.1.7 Advanced options for surface quality 60
6.1.8 3D Tool compensation 62
6.2 Sorting 63
6.2.1 Cutting method 63
6.2.2 Direction of machining 66
6.2.3 Cut order 67
6.2.4 Machine by 69
6.2.5 Enforce closed contours 69

6.2.6 Flip step over ... 70
6.2.7 Start point ... 70
6.3 Tool contact point ... 72
6.4 Cutting side ... 74

7. Tool axis control

7.1 Output format ... 76
7.2 Interpolation ... 79
7.3 Tilting strategies (Tool axis direction) ... 80
7.3.1 Not to be tilted and stays normal to surface ... 80
7.3.2 Tilted relative to cutting direction ... 80
7.3.3 Tilted to surface normal by fixed angle ... 91
7.3.4 Tilted to axis by fixed angle ... 94
7.3.5 Rotated around axis ... 95
7.3.6 Tilted through point ... 96
7.3.7 Tilted through curve ... 97
7.3.8 Tilted through lines ... 102
7.3.9 Tilted from point away ... 103
7.3.10 Tilted from curve away ... 104
7.3.11 Tilted relative to impeller machining layer ... 108
7.4 Angle range ... 110

8. Link

8.1 Approach/Retract ... 114
8.1.1 First entry ... 114
8.1.2 Last exit ... 117
8.1.3 Home position ... 119
8.2 Links ... 120
8.2.1 Gaps along cut ... 120
8.2.2 Links between slices ... 123
8.2.3 Links between passes ... 126
8.3 Default Lead-In/Out ... 129
8.3.1 Type ... 129
8.3.2 Tool axis orientation ... 132
8.3.3 Approach/Retreat parameters (Use the...) ... 133
8.3.4 Height ... 135

8.3.5 Feed rate 135
8.3.6 Same as Lead-In 135

9. Gouge check

9.1 Gouge checking 138
9.1.1 Tool 138
9.1.2 Geometry 139
9.1.3 Strategy 140
9.2 Clearance data 155
9.2.1 Clearance 155
9.2.2 Remaining collisions 160

10. Roughing and More

10.1 Multi-passes 162
10.2 Plunging 165
10.3 Morph pocket 166
10.4 Depth cuts 168
10.5 Area roughing 171
10.6 Rotate and translate 174
10.7 Sorting 176
10.7.1 Reverse order of passes/tool path 176
10.7.2 Connect slices by shortest distance 177
10.8 Stock definition 178

11. Motion control

11.1 Angle pairs 183
11.2 Point interpolation 184
11.3 Machine limits 185
11.4 Pole handling 186
11.5 Move list writer 187
11.6 Tool repositioning 188

12. Misc. parameters

12.1 Message 191
12.2 Extra parameters 191

12.3 Smooth surface normals ... 191
12.4 Tool center based calculation ... 192

13. SWARF Machining

13.1 Adding a SWARF Machining Operation ... 195
13.2 Coordsys ... 196
13.3 Geometry ... 196
13.3.1 Synchronize with tilt lines ... 197
13.3.2 Synchronize with (upper/lower) curves ... 197
13.3.3 Synchronize with main direction ... 198
13.3.4 Automatic ... 198
13.3.5 Shortest distance ... 198
13.4 Tool path parameters ... 200
13.4.1 General parameters ... 200
13.4.2 Corners parameters ... 201
13.5 Tool axis control ... 203
13.6 Gouge check ... 205
13.7 Roughing and More ... 206
13.7.1 Pattern slices ... 206
13.7.2 Pattern layers ... 208
13.7.3 Sorting ... 209

14. Multiblade Machining

14.1 Adding a Multiblade Machining Operation ... 212
14.2 Geometry ... 215
14.2.1 Strategy ... 215
14.2.2 Part definition ... 216
14.2.3 Stock to leave on ... 216
14.2.4 Start offset ... 216
14.3 Tool ... 217
14.4 Levels ... 218
14.4.1 Clearance ... 218
14.4.2 Levels ... 219
14.5 Tool path parameters ... 220

14.5.1 Technology 220
14.5.2 Surface quality 226
14.5.3 Sorting 227
14.5.4 Edges 231
14.6 Tool axis control 233
14.7 Link 235
14.8 Gouge check 237
14.8.1 Gouging 237
14.8.2 Clearance data 237
14.9 Roughing and More 238
14.9.1 Stock 238
14.9.2 Rotation 239

15. Port Machining

15.1 Adding a Port Machining Operation 242
15.2 Geometry 244
15.3 Tool path parameters 245
15.3.1 Surface quality 246
15.3.2 Sorting 247
15.4 Tool axis control 250
15.5 Gouge check 251

16. Contour 5-Axis Machining

16.1 Adding a Contour 5-Axis Machining Operation 254
16.2 Geometry 255
16.3 Tool path parameters 256

17. Multiaxis Drilling Operation

17.1 CoordSys page 259
17.2 Geometry page 260
17.3 Tool page 262
17.4 Levels page 263
17.4.1 Clearance area 263
17.4.2 Levels 264
17.5 Technology page 268

17.5.1 Sorting ... 268
17.5.2 Sorting of cylindrical drilling patterns ... 269
17.5.3 Drill cycle type ... 276

18. Converting HSM to Sim. 5-Axis Milling

18.1 Source operation ... 279
18.2 Tool ... 280
18.3 Levels ... 281
18.4 Tool axis control ... 282
18.5 Gouge check ... 283
18.6 Motion control ... 285
18.7 Miscellaneous parameters ... 286

19. Machine simulation

19.1 Machine simulation user interface ... 289
19.1.1 Simulation menu ... 290
19.1.2 Simulation windows ... 302
19.1.3 Simulation toolbars ... 314
19.2 Machine simulation settings ... 316
19.2.1 Directory for Machine simulation definition ... 316
19.2.2 Tool path coordinates ... 316
19.2.3 Solid verification ... 317
19.2.4 Target loading ... 317
19.2.5 Environment ... 317
19.2.6 Enable collision control ... 318
19.2.7 Collision detection mode ... 319
19.2.8 Collision check in length-based mode ... 319
19.2.9 Tool path color scheme ... 320
19.2.10 Embedded move list ... 320
19.2.11 Background ... 320
19.2.12 File ... 322
19.2.13 Position ... 322

20. CNC-machine definition

20.1 CNC-machine definition 324
- 20.1.1 CNC-machine kinematic type 324
- 20.1.2 Spindle direction 326
- 20.1.3 Rotation axes direction 326
- 20.1.4 Rotation axes names 329
- 20.1.5 Rotation point 330
- 20.1.6 Translation axis limits 330
- 20.1.7 Rotation axis limits 331
- 20.1.8 Machine simulation name 332

20.2 CNC-machine model definition 333
- 20.2.1 Preparing a CNC-machine model 333
- 20.2.2 Understanding the structure of the CNC-machine 336
- 20.2.3 Reviewing the CNC-machine properties 338
- 20.2.4 Defining the CNC-machine housing 339
- 20.2.5 Defining the translational axis 340
- 20.2.6 Defining the tool 341
- 20.2.7 Defining the translational axis 342
- 20.2.8 Defining the rotational axis 343
- 20.2.9 Defining the magazine 345
- 20.2.10 Collision control 346
- 20.2.11 Defining the coordinate transformation 347
- 20.2.12 XML file structure 349

21. Exercises

Exercise #1: Eccentric shaft and cam machining 358

Exercise #2: Parallel to surface and Parallel to curve strategies 359

Exercise #3: Perpendicular to curve strategy 361

Exercise #4: Morph strategies 362

Exercise #5: Projection strategy 365

Document number: SC5AUG13001

Introduction

1

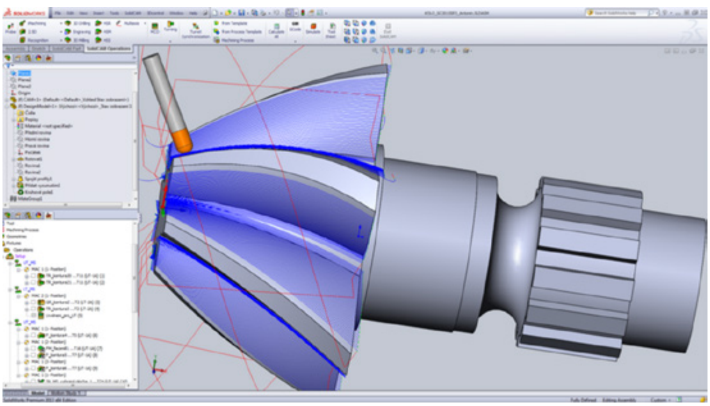
Welcome to the SolidCAM Sim. 5-Axis Machining module

Simultaneous 5-Axis machining is one of the most powerful modules of SolidCAM. It has the most advanced control over all the aspects of tool path and collision checking. SolidCAM utilizes all the advantages of Simultaneous 5-Axis machining and together with wide variety of Simultaneous 5-Axis cutting strategies, collision control, and machine simulation provides a solid base for your 5-Axis solution.

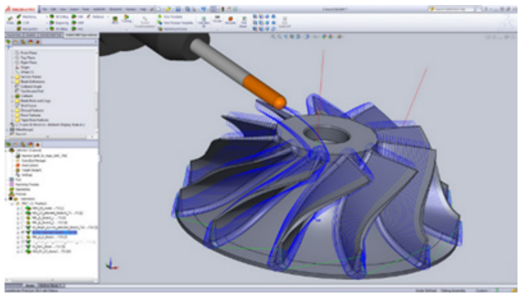
The 5-Axis module provides the following advantages:

- Wide variety of Simultaneous 5-Axis cutting strategies gets you and up cutting instantly
- Specific applications solutions for SWARF, Multi-Blade, Port, Contour 5-Axis, Multiaxis drilling, and Converting from HSM to Sim 5-Axis
- Flowline cutting produces a tool path that follows the natural shape of the component; particularly useful with blade type components
- Advanced tool tilting control and direct control on side tilting and lead/lag angles
- Multi-surface finish machining keeps the tool normal to the surface (or with specified lead and lag) to provide full control and smooth surface finish
- Multi-pass roughing techniques using multiple cut increments in the Z or XY planes with the option to trim to boundary
- Automatic gouge avoidance strategies that check each part of the tool and the holder
- Realistic full 3D machine simulation with comprehensive collision and axis limits checking

SolidCAM 5-Axis machining supports all 5-Axis machine tools including **Table/Table**, **Table/Head** and **Head/Head** gantry machines and the latest **Mill/Turn** machining centers.

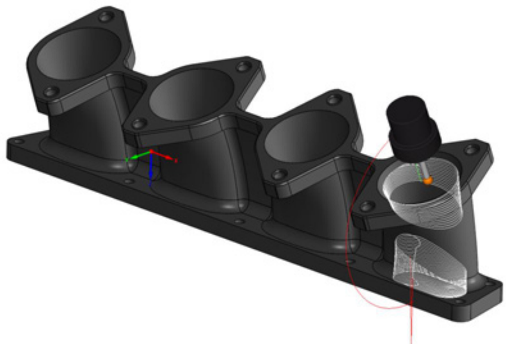


Multiblade Machining



The **Multiblade Machining** operation easily handles impellers and bladed disks, with multiple strategies to efficiently rough and finish each part of these complex shapes. All the blade surfaces are machined in a single operation. Various cutting strategies are supplied for roughing and finishing of the impeller parts. Multi-bladed parts are used in many industries. This operation is specifically designed to generate the necessary tool paths for the different multiblade configurations.

Port Machining



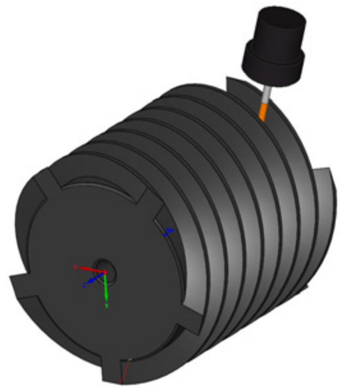
The **Port Machining** operation is an easy to use method for machining ports with tapered lollipop tool, and has collision checks for the entire tool (shank, arbor, and holder). You can choose to cut the top only, the bottom only, and specify how much stock to leave on the entire port. It uses 3-Axis machining as far into the port as possible, and then switches to 5-Axis motion. Smooth transitions are created where the tool paths meet at the middle of the port. It provides both roughing and finishing tool paths to make ports from castings or billet.

Contour 5-Axis Machining

The **Contour 5-Axis Machining** operation tilts the tool along a chained 3D profile drive curve, while aligning the tool axis according to the defined tilt lines, making it ideal for generating 5-Axis tool path for deburring and trimming.

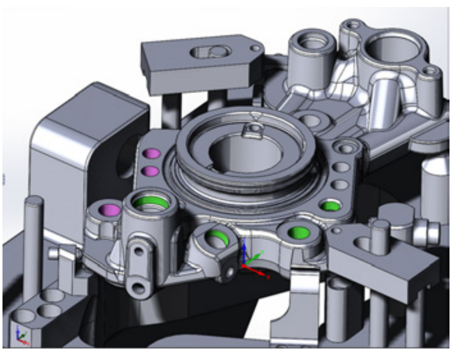
Swarf Machining

The **SWARF Machining** operation allows the side of the tool to be tilted over to machine the side wall at the correct angle. SWARF cutting uses the whole cutting length of the tool, resulting in better surface quality and shorter machining time.



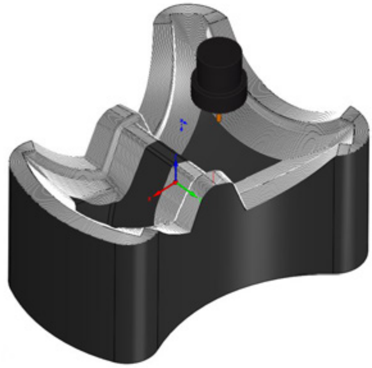
Multiaxis Drilling

The **Multiaxis Drilling** operation uses SolidCAM's automatic hole recognition and then performs drilling, tapping or boring cycles, at any hole direction easily and quickly. All the advanced linking, tilting and collision avoidance strategies available in other sim 5-Axis operations are also available in this operation, to provide full control of the generated Sim 5-Axis drilling tool path.



Converting HSM to Sim 5-Axis Milling

Converting HSM to Sim 5-Axis Milling operation converts HSM 3D tool paths to full 5-Axis machining, collision-protected tool paths. This maintains optimum contact point between the tool and the part and enables the use of shorter tools, for more stability and rigidity.



About this book

This book is intended for experienced SolidCAM users. If you are not familiar with the software, start with the lessons in the **Getting Started** Manual and then contact your reseller for information about SolidCAM training classes.

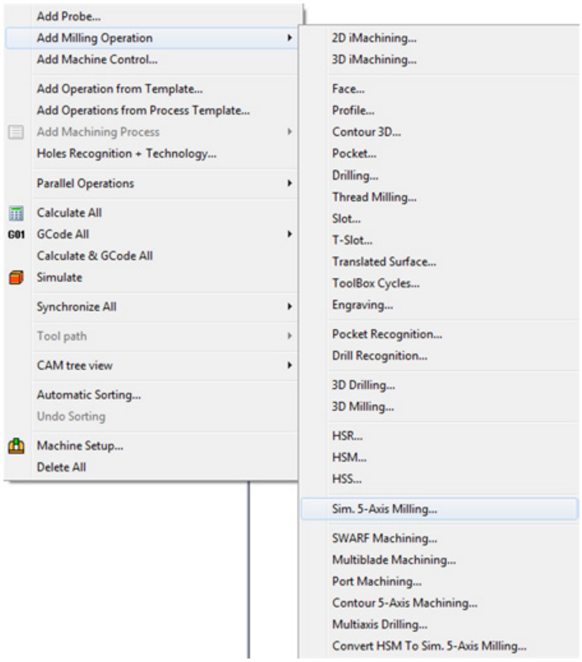
About the Exercises

The CAM-Parts used for this book are attached in a ZIP archive. Extract the content of the **Examples** archive into your hard drive. The SolidWorks files used for the exercises were prepared with SolidWorks 2013.

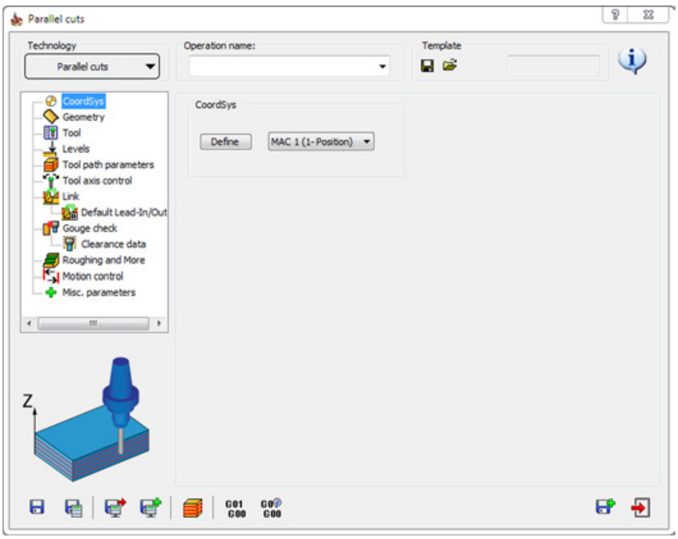
The contents of this book and exercises can be downloaded from the SolidCAM website **http://www.solidcam.com**.

1.1 Adding a Sim. 5-Axis Operation

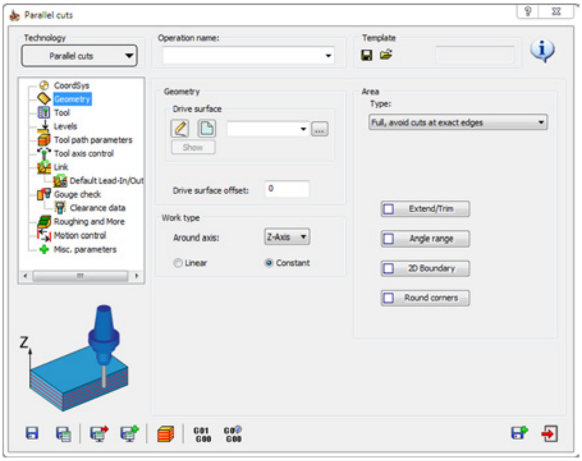
To add a **Sim. 5-Axis Operation** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose the **Sim. 5-Axis Milling** command from the **Add Milling Operation** submenu.



The default **Parallel cuts** dialog box is displayed.



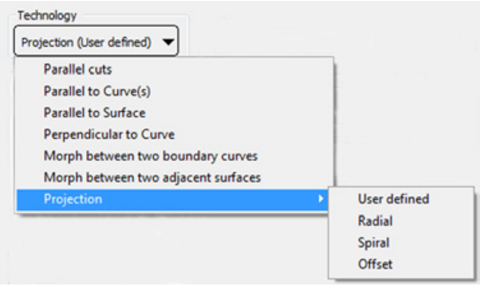
1.2 Sim. 5-Axis Operation dialog box



SolidCAM provides you with a number of Sim. 5-Axis operations designed for specific Sim. 5-Axis machining tasks. Each of these operations has a subset of parameters and options relevant for the chosen technology. Using the operations ensures quick programming of specific Sim. 5-Axis tasks..

Technology

This section enables you to define the type of the Sim. 5-Axis operation. SolidCAM provides you with the following types of the Sim. 5-Axis operation:



- **Parallel cuts**

  This strategy enables you to generate the tool path with cuts that are parallel to each other.

- **Parallel to Curve(s)**

  This strategy enables you to perform the machining along a lead curve. The generated cuts are parallel to each other.

- **Parallel to Surface**

  This strategy enables you to generate the tool path on the drive surface parallel to the specified check surface.

- **Perpendicular to Curve**

  This strategy enables you to generate the tool path orthogonal to a **Lead curve**.

- **Morph between two boundary curves**

  This strategy enables you to generate a morphed tool path between two leading curves. The generated tool path is evenly spread over the drive surface.

- **Morph between two adjacent surfaces**

  This strategy enables you to generate a morphed tool path on a drive surface enclosed by two check surfaces. The tool path is generated between the check surfaces and evenly spaced over the drive surface. This strategy can be used for the machining of impellers with twisted blades.

- **Projection**

  This strategy enables you to generate a tool path along a curve projected on the drive surface.

User defined

This strategy projects the curve selected in the **Projection curves** section down onto the drive surfaces.

Radial

This strategy projects a radial pattern on the surface. It can be particularly effective on circular shaped components and shallow areas.

Spiral

This strategy projects a spiral pattern on the surface.

Offset

This strategy projects the curve selected in the **Projection curves** section down onto the drive surfaces and creates offsets on the sides of the projection curve.

Parameter pages

The parameters of the Sim. 5-Axis operation are divided into a number of subgroups. The subgroups are displayed in a tree format on the left side of the Operation dialog box. When you click a subgroup name in the tree, the parameters of the selected subgroup appear on the right side of the dialog box.

- **CoordSys**

  Define the CoordSys position for the Sim. 5-Axis operation.

- **Geometry**

  Choose a geometry for machining and define the machining strategy and its parameters.

- **Tool**

  Choose a tool for the operation and define the related parameters such as feed and spin.

- **Levels**

  Define the Clearance area and the machining levels.

- **Tool path parameters**

  Define the machining parameters.

- **Tool axis control**

  Define the orientation of the tool axis during the Sim. 5-Axis machining.

- **Link**

  The **Link** and **Default Lead-In/Out** pages enable you to define how the Sim. 5-Axis cutting passes are linked to the complete tool path.

- **Gouge check**

  Avoid the tool gouging of the selected drive surfaces and check surfaces.

- **Roughing and More**

  Define the parameters of the Sim. 5-Axis roughing.

- **Motion control**

  Define the parameters related to the kinematics and special characteristics of the CNC-machine.

- **Misc. parameters**

  Define a number of miscellaneous parameters and options related to the Sim. 5-Axis tool path calculation.

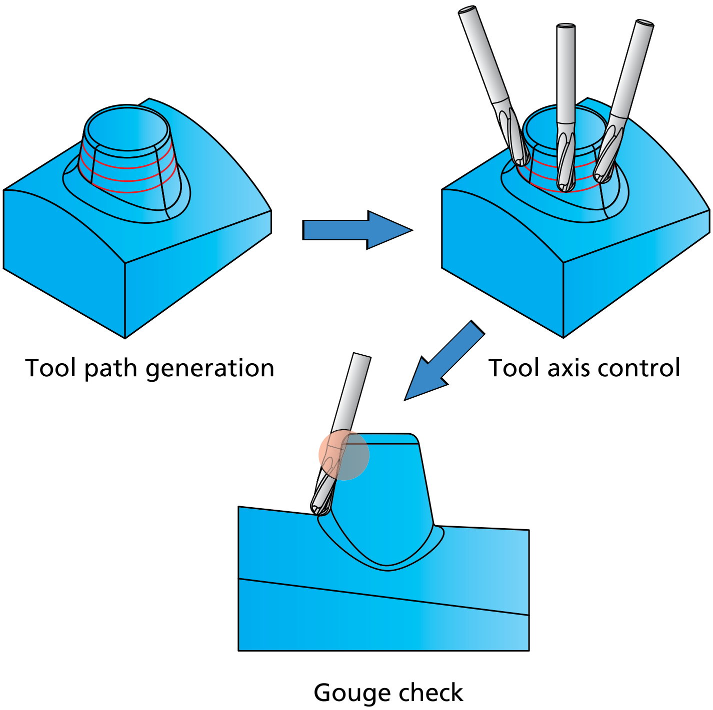
1.3 The stages of the Sim. 5-Axis Operation parameters definition

The operation definition is divided into three major stages:

1. **CoordSys**, **Geometry**, **Finish Parameters** and **Links** – generation of the tool path for the selected faces. Tool tilting and gouge checking are not performed at this stage.

2. **Tool axis control** – controlling the angle of the tool from the normal vector at every point along the tool path.

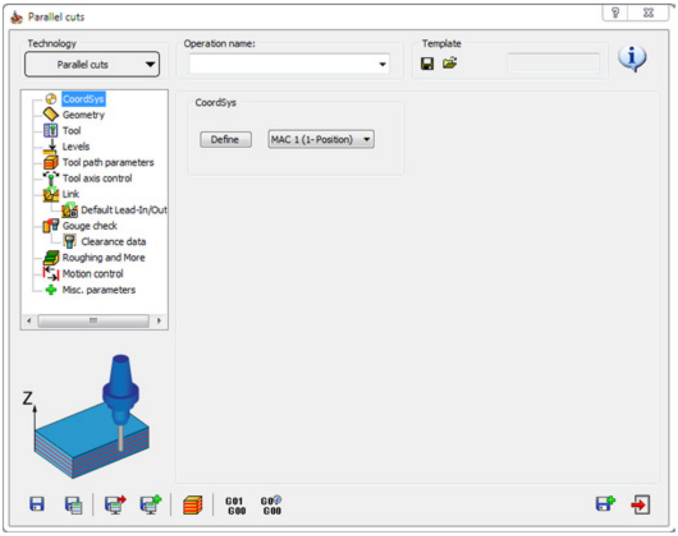
3. **Gouge check** – avoiding tool and holder collisions.

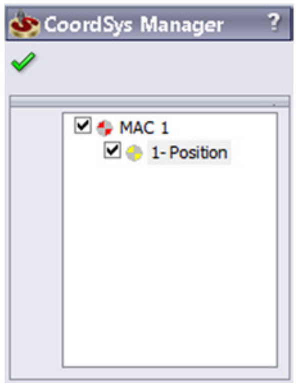


CoordSys

2

CoordSys page





On this page, you define the Coordinate System appropriate for the operation. Choose an existing Coordinate System from the list or click the **Define** button to define a new one. The **CoordSys Manager** dialog box is displayed. This dialog box enables you to define a new Coordinate System directly on the solid model.

When the Coordinate System is chosen for the operation, the model is rotated to the selected CoordSys orientation.

For more information on the Coordinate System definition, refer to the **SolidCAM Milling User Guide**.

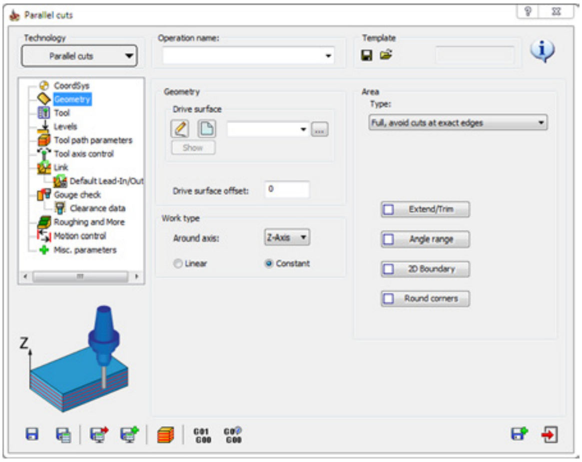
The CoordSys definition must be the first step in the operation definition process.

In a **Sim. 5-Axis** operation, you have to choose only the Machine Coordinate Systems. The Sim. 5-Axis tool path generated relative to the Machine Coordinate System contains the tool path positions and tool axis orientation at each tool path position. The tool path is generated in the 4/5-axes space relative to the Machine Coordinate System. The Machine Coordinate System is defined relative to the center of the rotation of the machine (CNC-machine origin).

Geometry

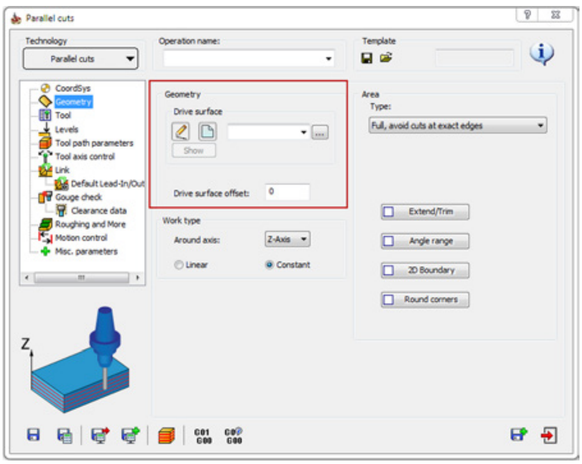
3

The **Geometry** page enables you to define the geometry and its parameters for machining.



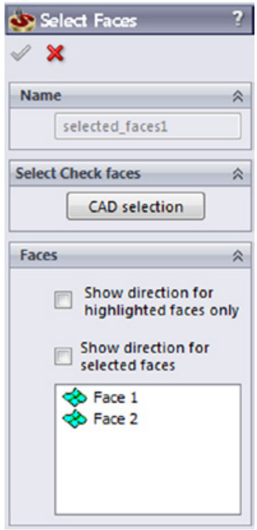
3.1 Geometry

This section enables you to define the geometry for the Sim. 5-Axis operation. All the machining strategies of the SolidCAM Sim. 5-Axis operation use a **Drive surface** geometry. For some strategies, additional geometries must be defined.



3.1.1 Drive surface definition

In the **Drive surface** section, choose the appropriate geometry from the list or define a new one by clicking the **New** icon ( ). The **Select Faces** dialog box is displayed. This dialog box enables you to select one or several faces of the SolidWorks model. Click on the appropriate model faces. The selected faces are highlighted.

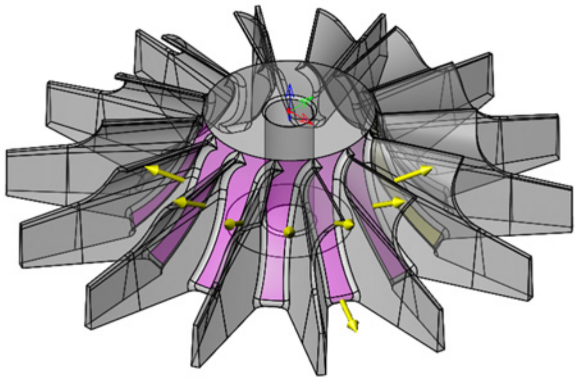


To remove selection, click on the selected face again or right-click the face name in the list (the face is highlighted) and choose the **Unselect** option from the menu.

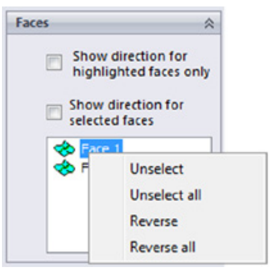
When transferring model files from one CAD system to another, the direction of some of the surface normals might be reversed. For this reason, SolidCAM provides you with the capability to display and edit the normals of model surfaces during the geometry selection.

The **Show direction for highlighted faces only** check box enables you to display the surface normals for the specific highlighted faces in the list.

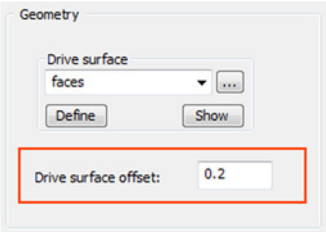
The **Show direction for selected faces** check box enables you to display the normals direction for all the faces in the list.



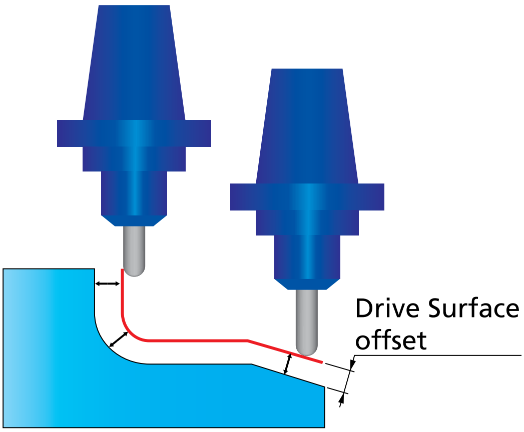
SolidCAM enables you to machine surfaces from the positive direction of the surface normal. Sometimes surfaces are not oriented correctly and you have to reverse their normal vectors. The **Reverse/Reverse All** command enables you to reverse the direction of the surface normal vectors.



3.1.2 Drive surface offset



The **Drive surface offset** parameter enables you to define a machining allowance for the drive surface. The machining is performed at the specified distance from the drive surface.

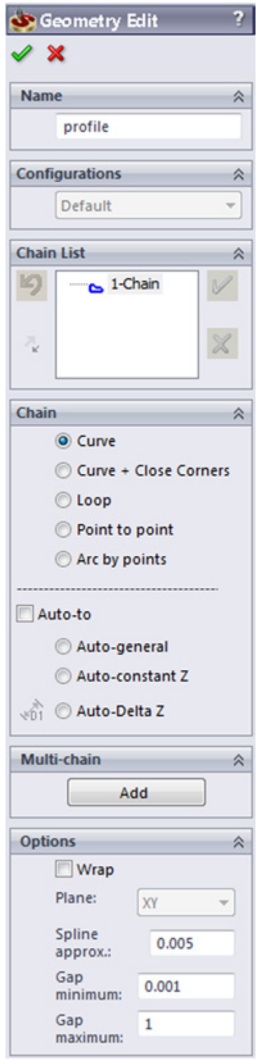


The offset is three-dimensional and expands the faces in every direction.

3.1.3 Curves definition

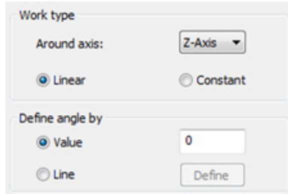
Some Sim. 5-Axis machining strategies use additional curve geometries for the tool path generation. SolidCAM enables you to define such geometries using the **Geometry Edit** dialog box.

For more information on the wireframe geometry selection, refer to the **SolidCAM Milling User Guide**.



3.1.4 Work type

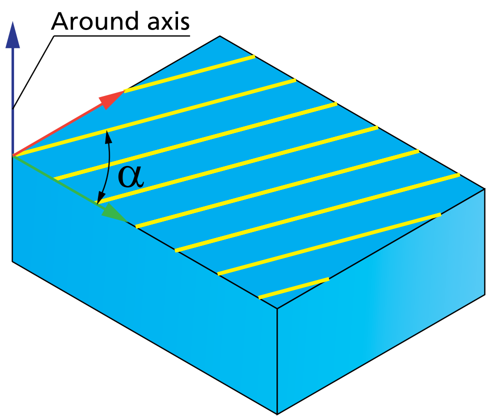
This section is available only for **Parallel cuts** strategy. It offers you two ways in which the tool path cuts can be performed:



- **Linear**

When this option is chosen, the tool path cuts are generated in the **Linear** manner, and the axis chosen from the **Around axis** list is normal to the plane of machining.

- When the **X-axis** is chosen from the **Around axis** list, the machining is performed in the YZ-plane;
- When the **Y-axis** is chosen from the **Around axis** list, the machining is performed in the ZX-plane;
- When the **Z-axis** is chosen from the **Around axis** list, the machining is performed in the XY-plane.

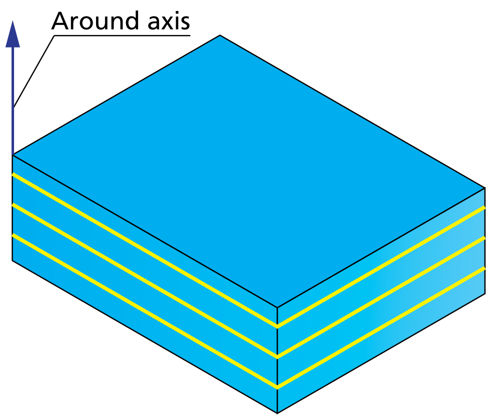


The **Define angle by** section enables you to define the angle of linear machining. The angle can be defined by entering its value in the edit box or by picking a line on the model.

This section is available only when **Linear** is chosen for **Work type**.

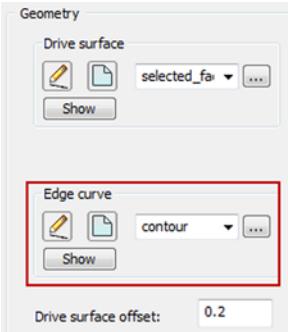
- **Constant**

When this option is chosen, the tool path cuts are generated in the **Constant Z** manner around the axis chosen from the **Around axis** list.



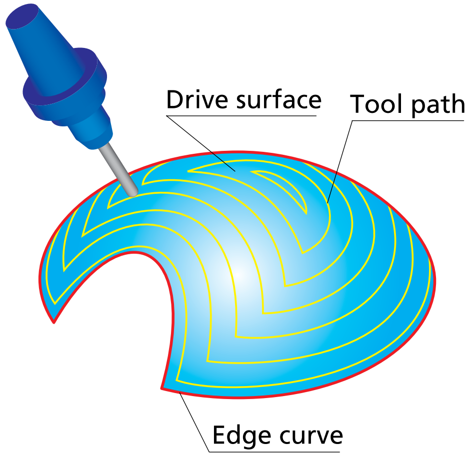
The **Around axis** list enables you to choose the axis (**X**, **Y** or **Z**) around which the tool path cuts will be generated.

3.1.5 Edge curve



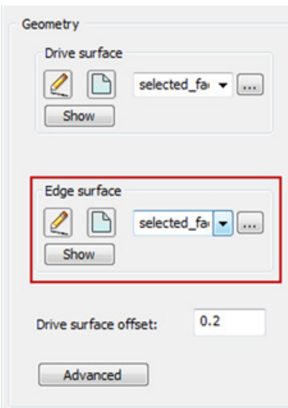
This section is available only for the **Parallel to curves** strategy.

The **Edge curve** section enables you to define lead curve for the operation using the **Geometry Edit** dialog box (see topic **3.1.3**).



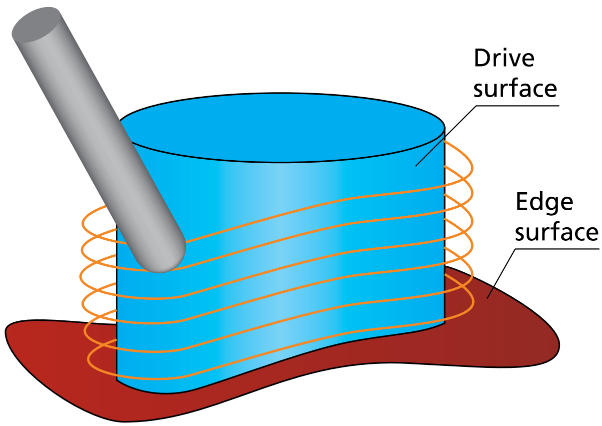
It is recommended to choose the Drive surface edge as the lead curve geometry to get better placement of the tool path.

3.1.6 Edge surface



This section is available only for **Parallel to surface** strategy. It enables you to generate the tool path on the **Drive surface** parallel to the specified check surface.

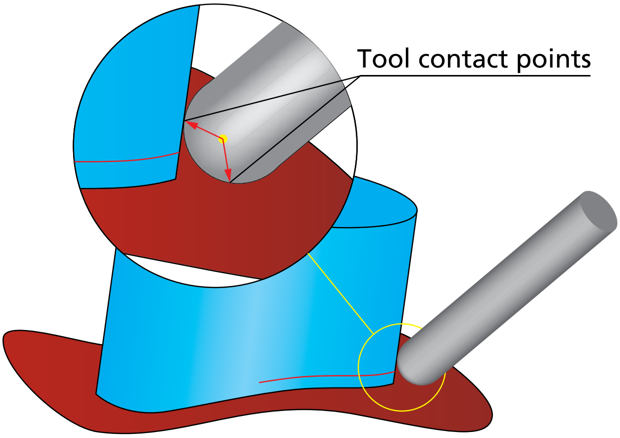
The **Edge surface** section enables you to define the check surfaces geometry for the tool path generation.



The drive and check surfaces have to be adjacent, i.e. they must have a common edge.

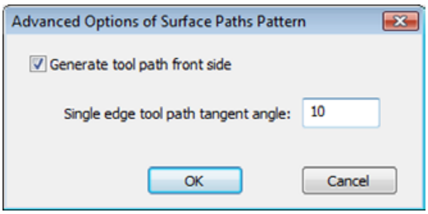
Depending on the defined **Tool tilting** (see topic **7.3**) it is recommended to activate the gouge checking (see chapter **9**), to make sure that the check surface will not be gouged.

When a ball-nosed tool is used with this strategy, it is recommended to use the **Tool center based calculation** option. With this option, the passes close to the check surface will be generated in such way that the tool is tangent to both the drive surface and the check surface. If the calculation is not based on the tool center, a wrong tool path is generated.

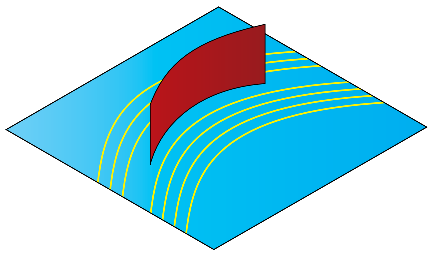


Advanced options

SolidCAM enables you to define a number of advanced options for the **Parallel to surface** strategy. Click the **Advanced** button to display the **Advanced Options of Surface Paths Pattern** dialog box.

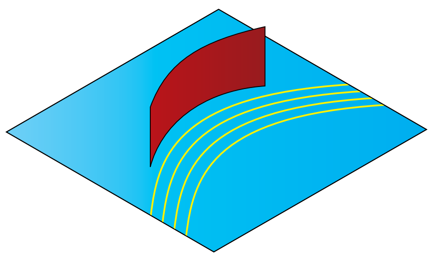


The **Generate tool path front side** option enables SolidCAM to take into account the normals of the defined check surface.

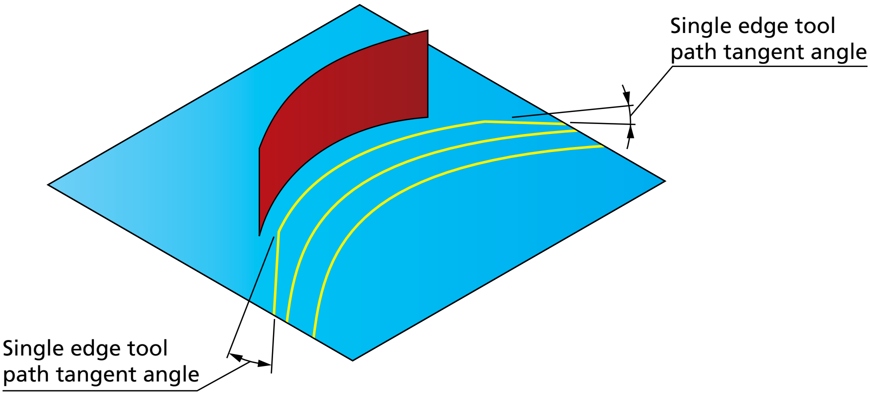


When this check box is not selected, the tool path is generated on the drive surface only from all the sides of the check surface.

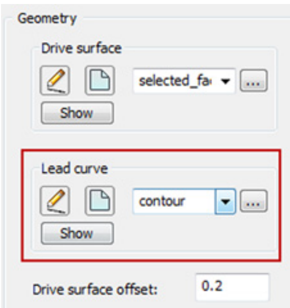
When this check box is selected, SolidCAM generates the tool path taking into account the direction of the check surface normals. The resulting tool path is located only at the front side of the check surface.



SolidCAM automatically extends the passes tangentially to the drive surface edges. Using the **Single edge tool path tangent angle** parameter you can change the extension direction. This option affects only the first pass (close to the check surface); all other passes are extended tangentially.



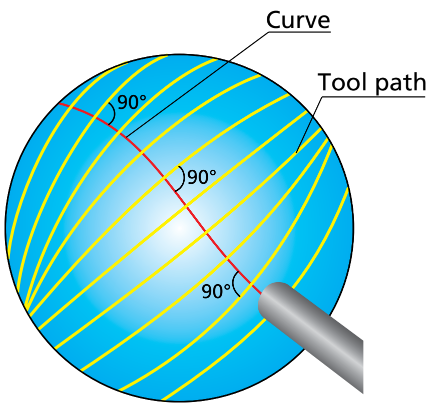
3.1.7 Lead curve

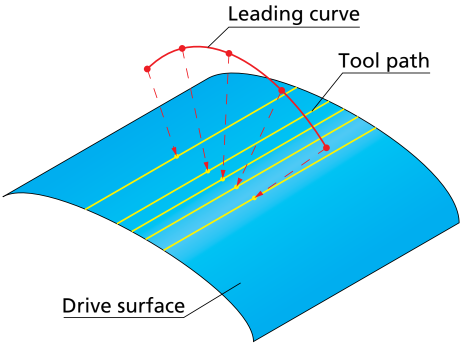


This section is available only for **Perpendicular to curve** strategy, which enables you to generate the tool path orthogonal to the **Lead curve** defined in the **Geometry** section.

Note that when the selected curve is not a straight line, the cuts are not parallel to each other.

The lead curve geometry does not have to be located on the surface. During the tool path calculation, SolidCAM generates in each point of the lead curve virtual points on the curve. The distance between these points is determined by the **Step over** value (see topic **6.1.3**). SolidCAM projects these points onto the drive surface; the direction of the projection is the normal vector of the curve at the virtual point. Where the normal vector intersects with the surface, a virtual surface point is generated. The passes are generated through these points, normal to the lead curve.

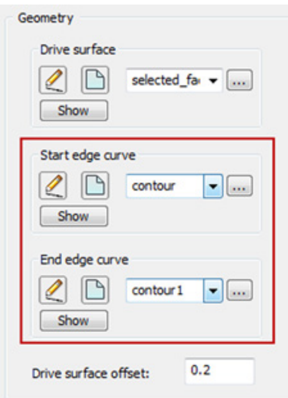




If the cuts cross each other at the edge of the surface, caused by an inappropriate lead curve, you will not get an acceptable result.

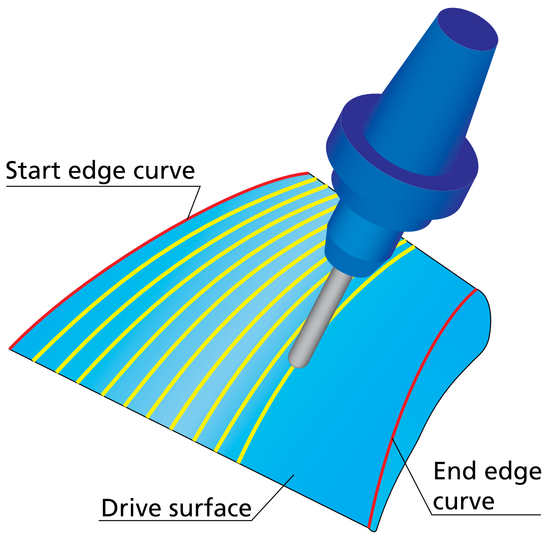
The lead curve must be located exactly on or above the drive surface. If the curve is not located above the surface, no tool path is generated. When only a part of the lead curve is located above the surface, only where the normal vector of the lead curve intersects with the drive surface a tool path is generated.

3.1.8 Start and End edge curve



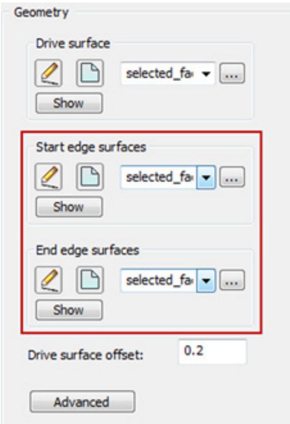
The **Start edge curve** and **End edge curve** sections are available for both **Morph between two boundary curves** and **Morph between two adjacent surfaces** strategies. The first strategy creates a morphed tool path between two leading curves. The generated tool path is evenly spread over the drive surface. The second strategy can be used for the machining of impellers with twisted blades.

These sections enable you to define the leading curves for the morphing using the **Geometry Edit** dialog box (see topic **3.1.3**).



It is recommended to choose the Drive surface edges as the lead curves geometry to get better morphing of the tool path.

Morph between two adjacent surfaces

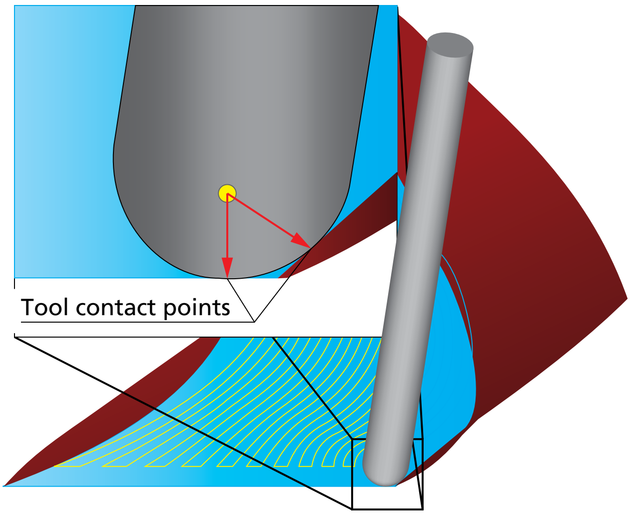


The **Start edge surfaces** and **End Edge surfaces** sections enable you to define the check surfaces geometry for the tool path generation.

The drive and check surfaces have to be adjacent, i.e. they must have a common edge.

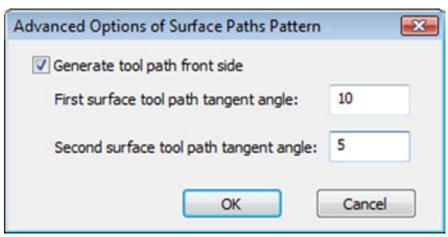
Depending on the defined **Tool tilting** (see topic **7.3**) it is recommended to activate the gouge checking (see chapter **9**), to make sure that the check surfaces will not be gouged.

When a ball-nosed tool is used with this strategy, it is recommended to use the **Tool center based calculation** option. With this option, the passes close to the check surfaces are generated in such way that the tool is tangent to both the drive surface and the check surface. If the calculation is not based on the tool center, a wrong tool path is generated.



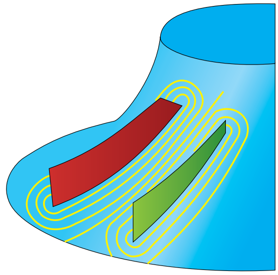
Advanced options

SolidCAM enables you to define a number of advanced options for the **Morph between two adjacent surfaces** strategy. Click the **Advanced** button to display the **Advanced Options of Surface Paths Pattern** dialog box.

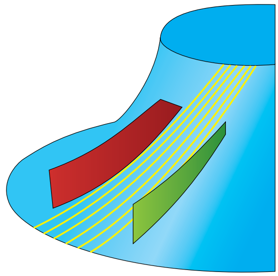


The **Generate tool path front side** option enables SolidCAM to take into account normals of the defined check surfaces.

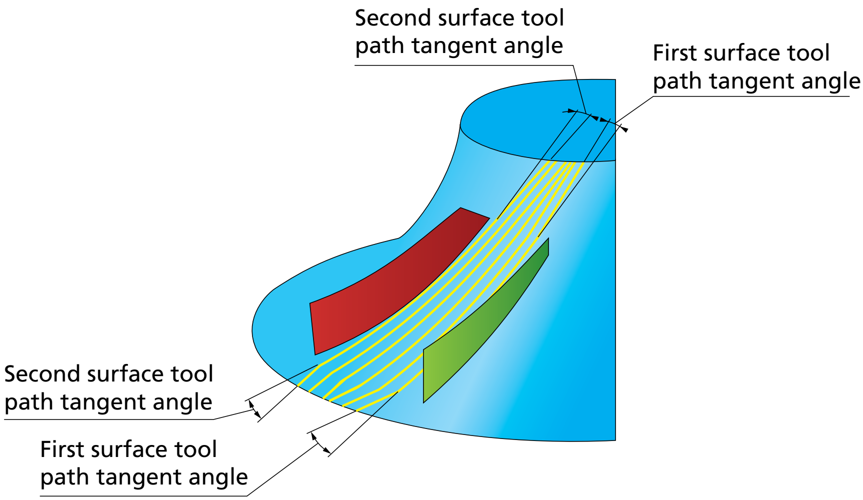
When this check box is not selected, the tool path is generated on the drive surface from all sides of the check surfaces.



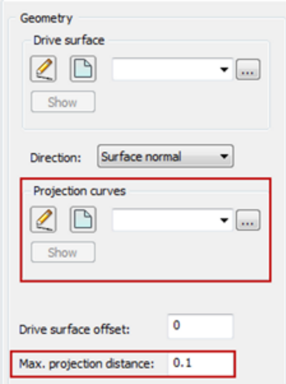
When this check box is selected, SolidCAM generates the tool path taking into account the direction of the check surfaces normals. The resulting tool path is located between the check surfaces only.



SolidCAM automatically extends the passes tangentially to the drive surface edges. Using the **First surface tool path tangent angle** and the **Second surface tool path tangent angle** parameters, you can change the extension direction. The direction can be changed for the first and last passes; all the internal passes are evenly morphed between them.

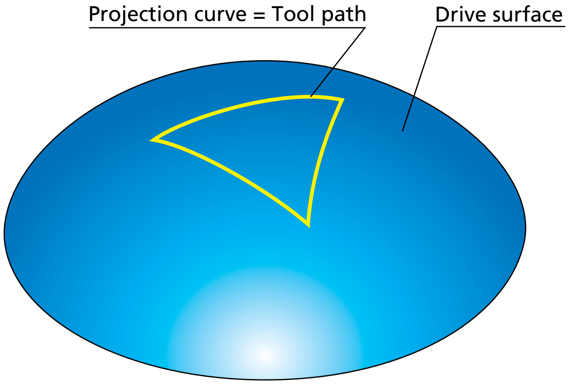


3.1.9 Projection curves



This section is available only for the **Projection** strategy that enables you to generate a single tool path along a curve. The **Projection curves** section enables you to define the curves for the tool path generation.

The option of **Projection curves** is available only in **User defined** and **Offset** technologies.



Max. projection distance

When the **Projection** tool path strategy is chosen, the system gets projection curves lying on the drive surfaces.

Due to tolerance issues in CAD systems, sometimes the curves do not lie exactly on the drive surfaces. This error can be compensated by the **Maximal projection distance** value. For example, if the value is 0.1 mm, it allows to have the projection curve 0.1 mm away from the drive surfaces.

Direction

This section enables you to specify to which direction the curves are projected.

X, Y, Z: The curve direction is defined as parallel to one of the Coordinate system axes.

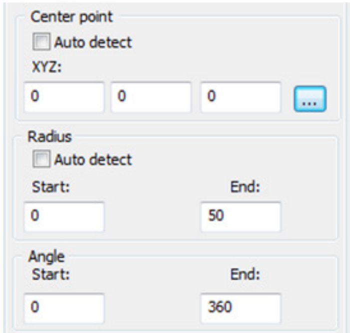
Line: The projection vector is defined by a line that you can pick directly on the solid model.

Surface normal: This option projects the curve into the normal direction of the surface below.

- If the curve/pattern lies exactly on the face, the tool path has the same shape and position as the curve.
- If the curve/pattern lies above the drive surfaces, the tool path changes. The tool path is built only on an interval between those normals that intersect the curve.

The projection has to lie within the **Max. projection distance** to the drive surface. Otherwise the curves cannot be considered for projecting.

Center point



This section enables you to define the position of a center point of the virtual circle that SolidCAM creates to generate radial/spiral passes.

The position can be defined manually by entering the coordinate values in the **XYZ** fields, picking the point directly on the solid model using the ... button, or automatically by selecting the **Auto detect** option.

Radius

- **Start:** This option enables you to create a virtual inner circle that starts the tool path from this inner circle so that the tool does not rub at the converging point where the radius is zero.
- **End:** This option enables you to to stop the tool path at the extreme end of the virtual outer circle.
- The **Auto detect** option enables you to detect the required maximum radius to automatically machine the surface.

Angle

This option defines an area based on two angles for the tool path calculation.

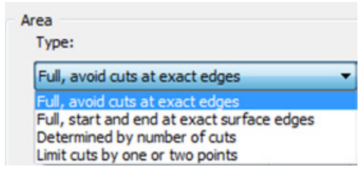
- **Start:** This parameters defines the starting angle of area definition for the tool path calculation.
- **End:** This parameter defines the ending angle of area definition for the tool path calculation.

The option of **Angle** is available only in the **Projection (Radial)** technology.

3.2 Area

The **Area** section enables you to define the cutting area on the drive surface.

The following options are available to define the area:

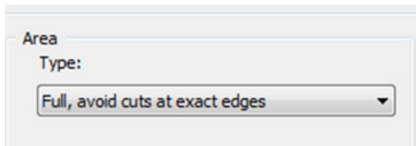


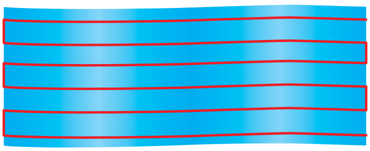
- **Full, avoid cuts at exact edges**
- **Full, start and end at exact surface edges**
- **Determined by number of cuts**
- **Limit cuts by one or two points**

All of these options are unavailable with the **Projection** strategy.

3.2.1 Full, avoid cuts at exact edges

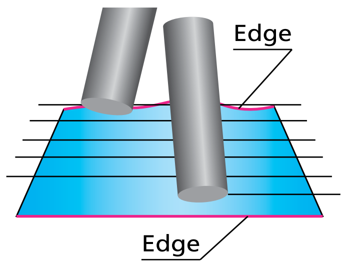
This option enables you to generate the tool path on the whole drive surface avoiding the drive surface edges. With this option, the minimal distance between the edge and the tool path is equal to half of the **Max. Step over**.



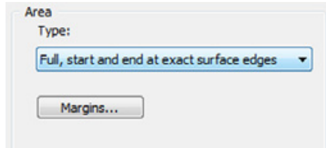


This option can be used when the boundary of the drive surfaces is not smooth and has gaps. The half of the **Max. Step over** offset from the surface edge enables you to compensate these defects of the surface. In case of large gaps, SolidCAM enables you to handle them using the **Gap along cuts** option (see topic **8.2.1**).

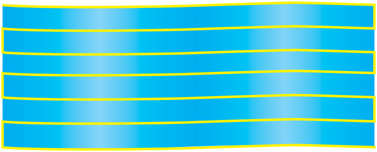
When the tool is oriented normally to the drive surface, make sure that the tool diameter is greater than half of the **Max. Step over**. Otherwise, unmachined areas are left at the drive surface edge. The image illustrates the use of this option. Note that the machining does not start at the exact edge of the surface. Therefore, the shape of the upper edge does not influence the tool path.



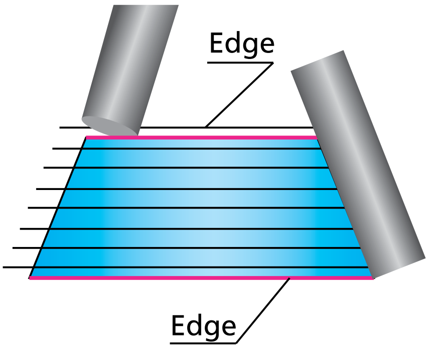
3.2.2 Full, start and end at exact surface edges



With this option, the tool path is generated on the whole surface starting and finishing exactly at the drive surface edges or at the nearest possible position.



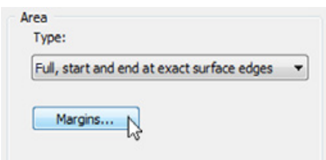
Make sure that the surface edges are perfectly trimmed. Gaps cause unnecessary air movements of the tool during the machining, therefore the **Full, avoid cuts at exact edges** option (see topic **3.2.1**) is preferable.



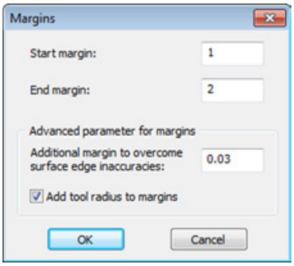
The number of cuts depends on the **Max. Step over** value. Since the first and the last cuts are located exactly on the drive surface edges, SolidCAM modifies the specified **Max. Step over** value (see topic **6.1.3**) to achieve equal distance between the cuts. The modified **Max. Step over** value used for the tool path calculation is smaller than the specified one.

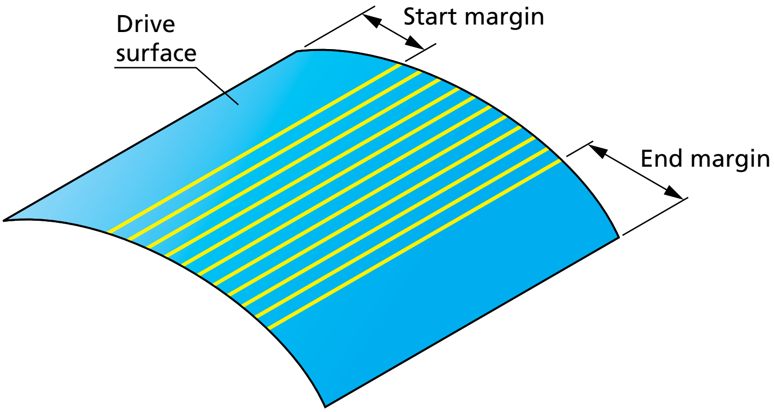
You can define margins for the tool path calculation when working with the following strategies: **Morph between two boundary curves**, **Parallel to curve**, **Parallel to surface**, and **Morph between two adjacent surfaces**.

Click the **Margins** button.



The **Margins** dialog box is displayed. This dialog box enables you to define a margin for the drive surface edges. The machining starts and finishes at the specified distances from the drive surface edges.





Advanced parameter for margins

Tool path strategies that use edge curves and surfaces sometimes encounter difficulties since CAD systems deliver the drive surfaces and the edge geometry (curves or surfaces) only within accuracy. If you would like to start the tool path exactly at the zero distance to the edge geometry, this is problematic, because the geometry can never be exactly aligned. To avoid this problem, SolidCAM provides you with the **Advanced parameter for margins** option.

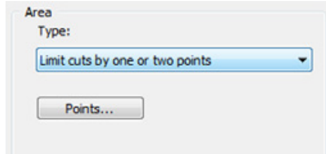
This option is available only for the **Morph between two boundary curves**, **Parallel to curve**, **Morph between two adjacent surfaces**, and **Parallel to surface** strategies, when the **Area type** is **Full, start and end on exact surface edges**.

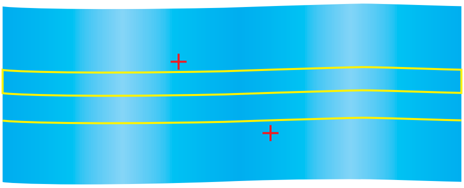
The **Additional margin to overcome surface edge inaccuracies** parameter enables you to compensate the inaccuracy of the CAD model edges. For example, to get the tool path at the 5 mm distance from the geometry, set the margin to 4.97 mm and the **Additional margin to overcome surface edge inaccuracies** to 0.03.

The **Add tool radius to margins** option enables you to expand the cutting area, which is defined by margins, by the tool radius distance.

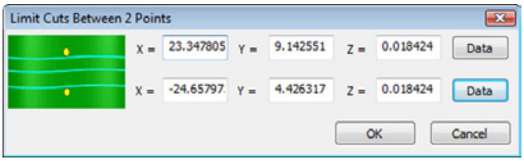
3.2.3 Limit cuts by one or two points

This option enables you to limit the tool path by one or two points.

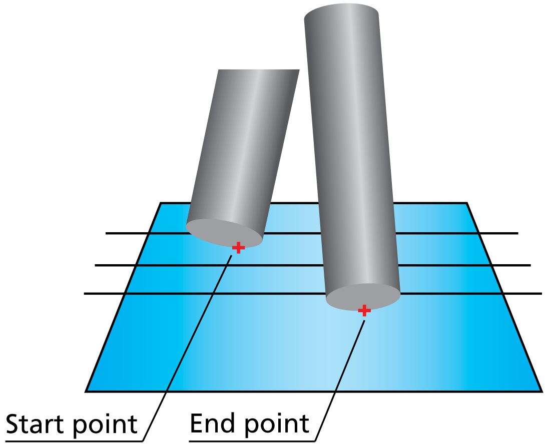




Click the **Points** button to define the limiting points with the **Limit Cuts Between 2 Points** dialog box. This dialog box enables you to enter the coordinates of the limiting points or define them directly on the solid model.

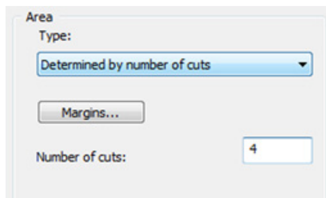


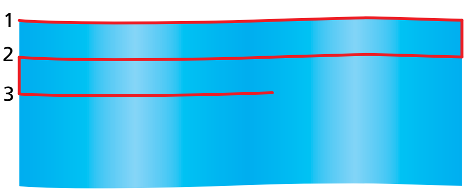
The defined limiting points must be located within the region of the cuts.



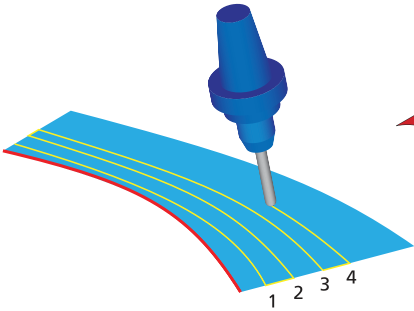
3.2.4 Determined by number of cuts

This option enables you to limit the tool path by a number of cuts. The **Number of cuts** parameter defines the number of cutting passes.

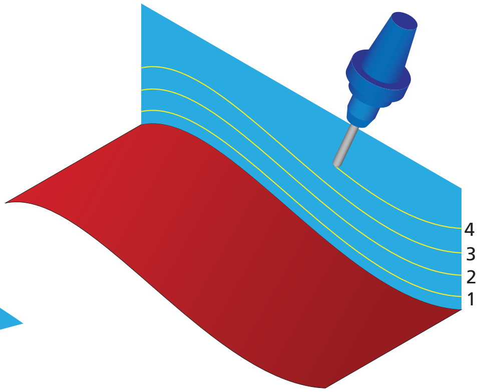




When the **Parallel to curve/Parallel to surface** strategy is chosen for the geometry definition, the **Determined by number of cuts** option generates the following tool path: the tool starts machining from the defined curve/surface and performs the number of cuts defined with the corresponding parameter.

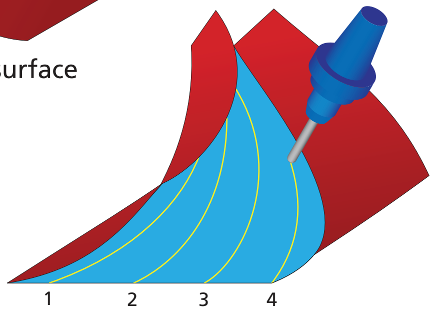


Parallel to curve

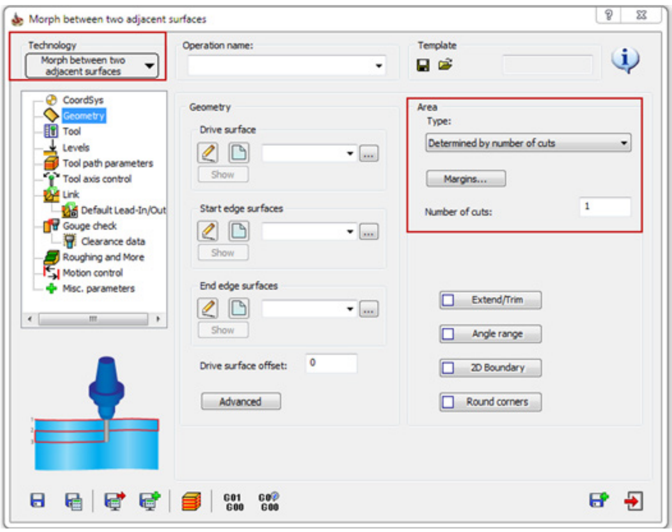


Parallel to surface

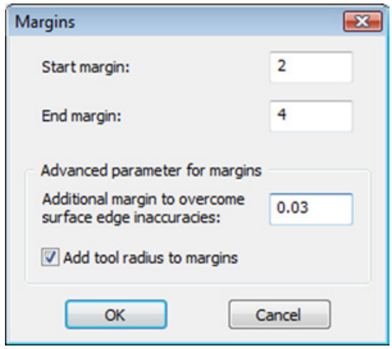
When the **Morph between two adjacent surfaces** strategy is chosen for the geometry definition, the area between the defined surfaces is divided by the number of cuts in such a manner that the first cut is performed at the **Start edge surface**, and the last cut at the **End edge surface**.



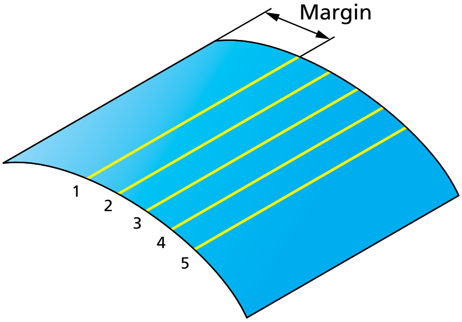
This option is available only for the **Parallel to curve**, **Parallel to surface**, and **Morph between two adjacent surfaces** technologies.



SolidCAM enables you to define a margin, shifting the first cut location. Click the **Margins** button to display the **Margins** dialog box.

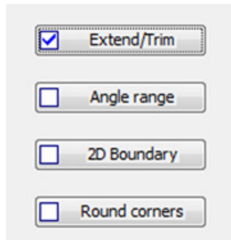


The location of the first cut is shifted by the distance specified by the **Start Margin** parameter.

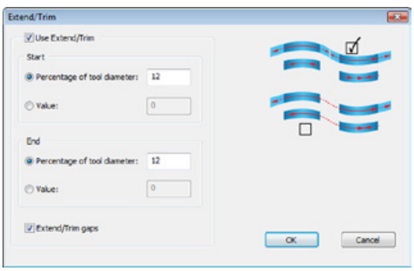


3.2.5 Extend/Trim

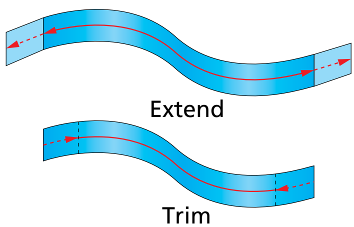
SolidCAM enables you to extend or trim the tool path tangentially along cuts. SolidCAM virtually extends or trims the drive surface tangentially and generates the tool path for it. In case of extending a tool path, the tool moves to the specified distance beyond the end of the surface. In case of tool path trimming, the tool stops at the specified distance before the surface boundary and moves to the next cut.



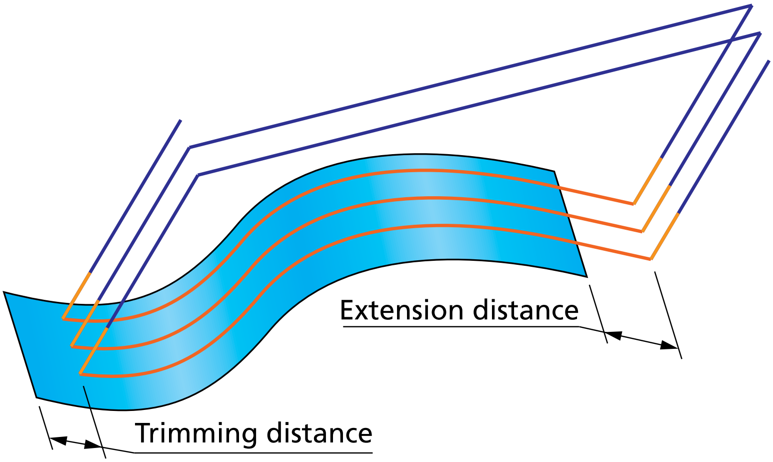
The **Extend/Trim** button displays the **Extend/Trim** dialog box. This dialog box enables you to define the extending/trimming distances.



- The **Start** option enables you to define the extending/trimming distance for the start of the cutting passes.
- The **End** option enables you to define the extending/trimming distance for the end of the cutting passes.



The distances can be defined either by values or by the percentage of the tool diameter. A positive value means extending of the passes; a negative value means trimming of the passes.

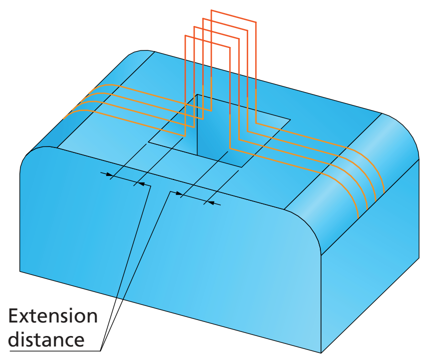


Note that with the **Zigzag** option (see topic **6.2.1**) the direction of the machining is changed for each cutting pass, so the start and the end of the passes are reversed for each pass. Therefore, to obtain the correct tool path, it is recommended to use the **One way** option. With this option, the start points of the passes are on one side of the drive surface and the end points are on the other side, providing you with the possibility of correct extending/trimming.

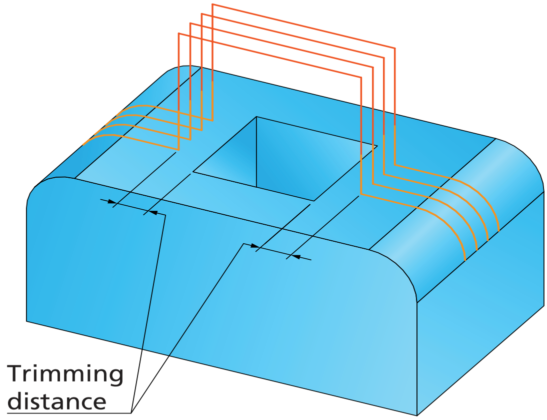
Extend/trim gaps

Selecting this check box enables you to apply the defined extending/trimming to all gaps detected along cutting passes during the tool path linking. In the gap area, the drive surface is virtually extended or trimmed tangentially by the distance specified in the **Start/End** sections. When the distance value is positive, the drive surface is extended; in case of a negative value it is trimmed.

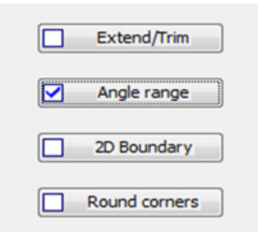
In case of extending applied to the detected gaps, the tool continues its move to the specified distance beyond the end of the surface, then performs linking in the gap area according to the parameters set in the **Gaps along cut** section of the **Link** page (see topic **8.2.1**), then continues the machining of the current cut at the specified distance before the second edge of the gap. As a result, the tool path is extended over the gap area at both sides.



When trimming is applied to the detected gaps, the tool stops at the specified distance before the gap edge, performs linking in the gap area according to the parameters set in the **Gaps along cut** section of the **Link** page, and continues the machining of the current cut at the specified distance after the second edge of the gap. As a result, the tool path is trimmed over the gap area at both sides.



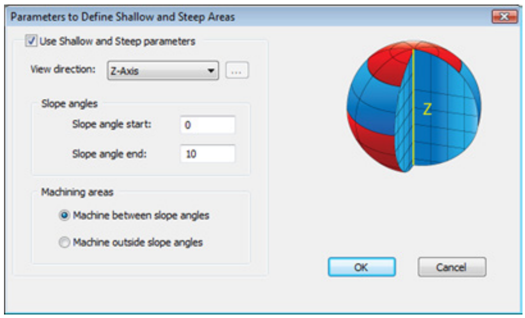
3.2.6 Angle range



SolidCAM enables you to define the cutting area by the surface inclination angle.

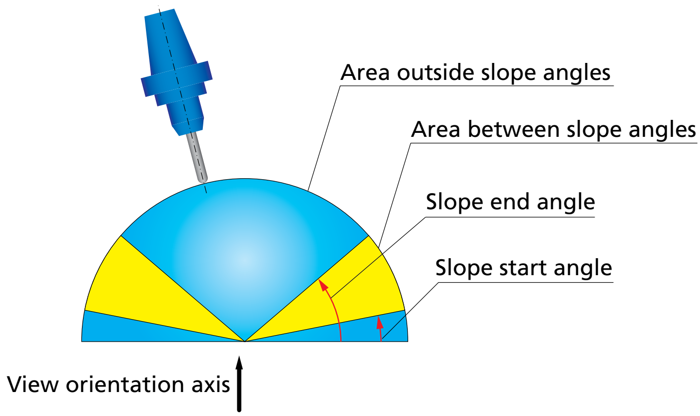
Click the **Angle range** button.

The **Parameters to Define Shallow and Steep Areas** dialog box is displayed. This dialog box enables you to define parameters determining the steep/shallow area to be machined.



View direction

SolidCAM enables you to define a vector from where the slope angle start and end are referenced. SolidCAM enables you to choose one of the Coordinate System axes (X-axis, Y-axis and Z-axis) or define a vector by an end point (the start point is automatically considered to be located in the Coordinate System origin).



Slope angles

The **Slope angle start** and **Slope angle end** parameters define the limit angles around the **View direction** vector.

Machining areas

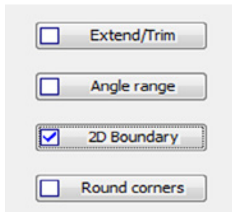
This option enables you to determine the area to be machined.

When the **Machine between slope angles** option is chosen, the machining is performed only at surfaces with inclination angles within the range defined by **Slope Start** and **Slope End** angles.

When the **Machine outside slope angles** option is chosen, the machining is performed only at surfaces with inclination angles outside the range defined by **Slope Start** and **Slope End** angles.

Note that the cutting area calculation is purely based on surface contact points. In other words, some portions of the surface geometry are virtually trimmed in order to split the part into shallow and steep regions.

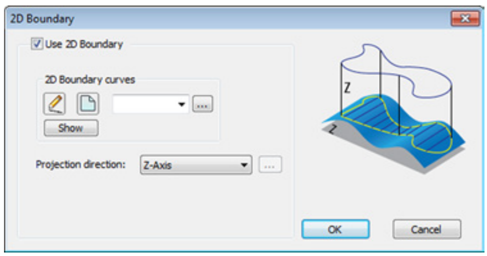
3.2.7 2D Boundary



SolidCAM provides you with a functionality to limit the machining to specific model areas. The machining limitation is performed by a planar boundary that is projected on the model. The projected boundary is "virtually" trimming the drive surfaces. All the contact points of the tool and drive surfaces are enclosed by this projected boundary.

Click the **2D Boundary** button.

The **2D Boundary** dialog box is displayed enabling you to define the boundaries.



2D Boundary curves

SolidCAM enables you to define a boundary based on a **Working area** geometry (closed loop of model edges as well as sketch entities).

For more information on the **Working area** geometry, refer to the **SolidCAM Milling User Guide**.

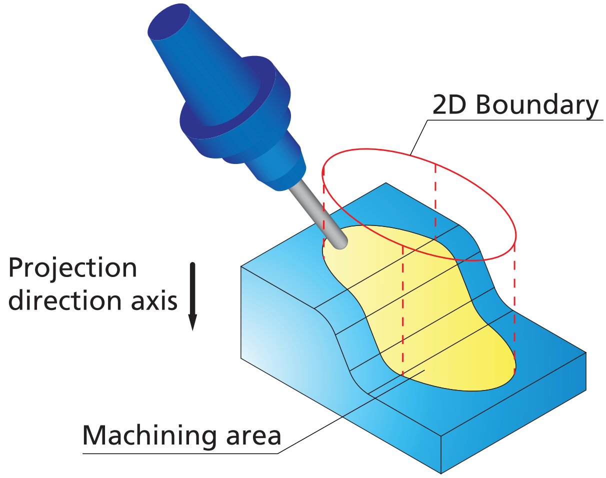
Both the **New** ( ) and **Edit** ( ) buttons display the **Geometry Edit** dialog box that enables you to define and edit the geometry.

The **Show** button enables you to display the already defined boundary directly on the solid model.

Projection direction

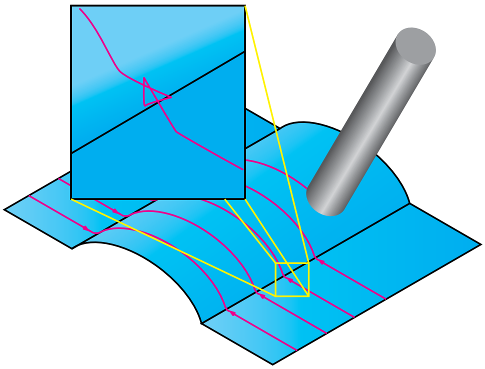
When a planar boundary is defined, SolidCAM automatically projects the geometry onto the solid model. The direction of the projection is defined by a vector.

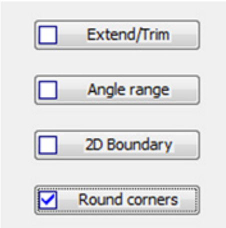
SolidCAM enables you to choose an axis of the Coordinate System as a projection direction vector or define a vector by an end point (the start point is automatically considered to be located in the Coordinate System origin).

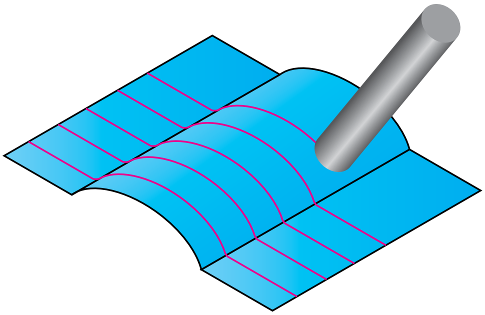


3.2.8 Round corners

In some cases, the Sim. 5-Axis tool path contains unnecessary fish tail movements in sharp corners or in small radius areas.





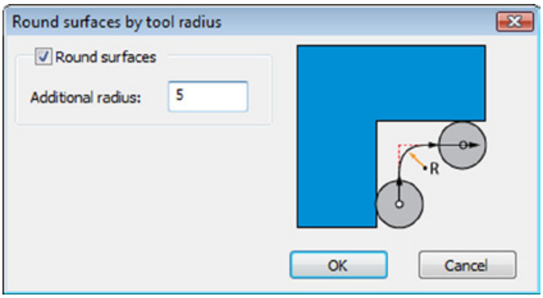


Using the **Round corners** option, you can avoid such movements and generate a smoother tool path.

Click the **Round corners** button.

The **Round surfaces by tool radius** dialog box is displayed enabling you to define the boundaries.

This option enables you to perform the rounding of the tool path. The rounding is performed in the direction of passes with a radius equal to the sum of the tool corner radius and the specified **Additional radius** value.

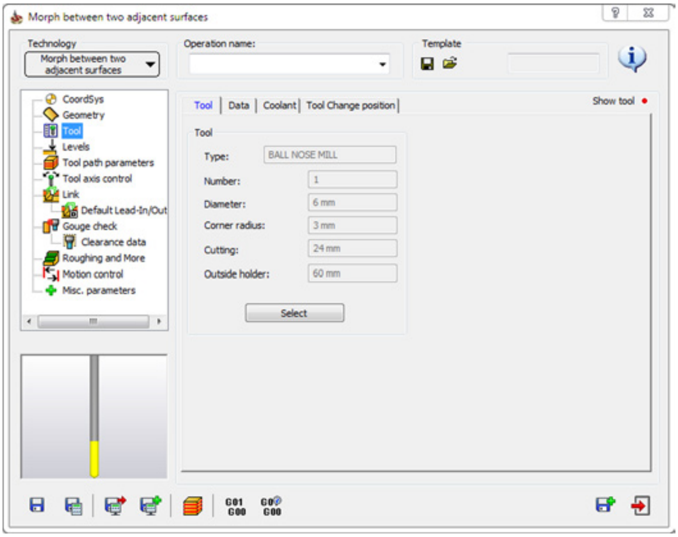


Tool

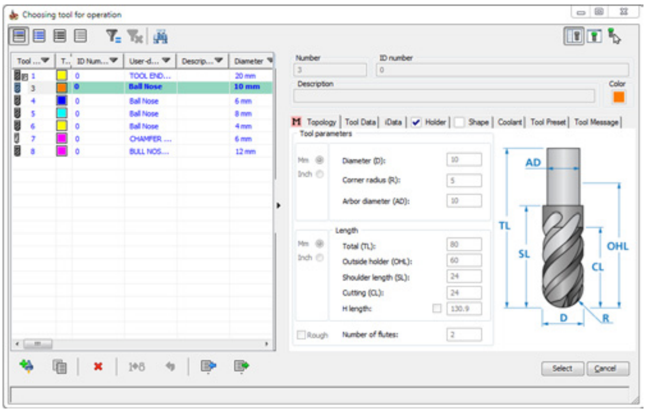
4

4.1 Tool definition

The **Tool** page enables you to choose a tool for the operation from the **Part Tool Table**.

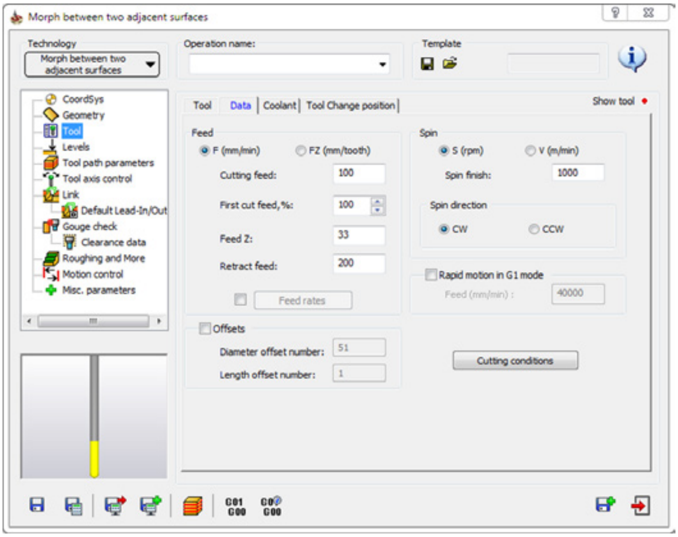


Click the **Select** button to display the **Part Tool Table** and choose a tool for the operation.



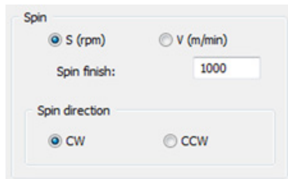
For more information on the tool definition, refer to the **SolidCAM Milling User Guide**.

Click the **Data** tab to display and define the **Spin** and **Feed** parameters.



4.1.1 Spin definition

The **Spin** section enables you to define the spinning speed of the tool. The spin value can be defined in two types of units: **S** and **V. S** is the default that signifies **Revolutions per Minute**. **V** signifies material cutting speed in **Meters per Minute** in the **Metric** system or in **Feet per Minute** in the **Inch** system; it is calculated according to the following formula:



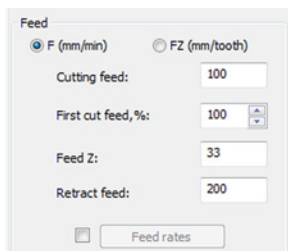
V = (S * PI * Tool Diameter) / 1000

Spin rate defines the normal spin rate used for rough milling. Spin finish defines the finish rate used for finish milling.

The **Spin direction** section enables you to choose between the clockwise (**CW**) or counterclockwise (**CCW**) direction

4.1.2 Feed definition

The **Feed** section enables you to define the tool feed for the Sim. 5-Axis operation. The feed value can be defined in two types of units: **F** and **FZ**. **F** is the default that signifies **Units per minute**. **FZ** signifies **Units per tooth** and is calculated according to the following formula:



FZ = F/(Number of Flutes * S)

The **F**/**FZ** buttons enable you to check the parameter values.

Cutting Feed

This field defines the cutting feed rate of the tool.

Feed Z

This field defines the feed of the tool movements from the safety position to the cutting depth.

For Tap tools, SolidCAM automatically calculates the **Feed Z** (**F** and **FZ**) value according to the following formulas:

F=Spin Rate * Pitch

FZ=Pitch

The calculated values are displayed in the **Feed Z** field. These values cannot be changed.

Retract Feed

This field defines the feed of the tool movements from the material to the retract level. The default value of the **Retract Feed** is calculated according to the following formula:

Retract Feed=Cutting Feed * 2

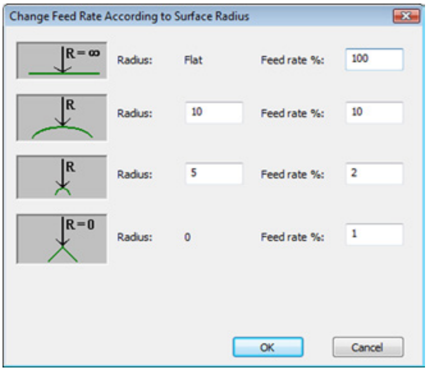
Feed Rates

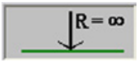
You can optimize the feed rate of different segments of the tool path. You define the machining feed rate, and SolidCAM modifies it according to the surface curvature. The surface curvature is calculated at each tool path position where the surface contact point of the tool is known.

This option is available only on single surfaces and cannot be used on the stitches between surfaces.

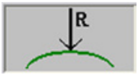
Select the check box and click the **Feed rates** button. The **Change Feed Rate According to Surface Radius** dialog box is displayed.

This dialog box enables you to define the feed rate according to the surface radius. The feed rate is defined as the percentage of the defined **Feed Rate** of the cutting movements.



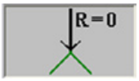


This section enables you to define the feed rate for the flat tool path segments where the curvature radius is infinite.



This section enables you to define the feed rate for the tool path segments of large radius (the radius value can be customized).

This section enables you to define the feed rate for the tool path segments of small radius (the radius value can be customized).



This section enables you to define the feed rate for the sharp corners of the tool path where the radius is equal to 0.

First cut feed percentage

In some machining cases, the tool load is not uniform along the tool path; the maximum tool load is reached along the first cutting pass. The **First cut feed, %** option enables you to change the feed rate at the first cutting path in order to optimize the cutting process. The feed rate is changed as a percentage of the defined feed rate (see topic **4.1.2**).

4.1.3 Rapid move parameters

Some 5-axis CNC-machines do not support synchronization between axis motors when the rapid movement (**G0**) is performed. The absence of synchronization causes the deviation between the calculated path and the real tool movements. SolidCAM enables you to avoid the problems described above by replacing all rapid movements (**G0**) with non-rapid ones using a particular feed rate.

This option enables you to control the use of rapid feed (**G0**). When the **Rapid motion in G1 mode** check box is not selected, the resulting GCode contains rapid movements (**G0**).

Example:

```
G0 X-2.942 Y75.567 Z24.402 A-88.436 B-26.482 M116
```

When the **Rapid motion in G1 mode** check box is selected, the resulting GCode does not contain **G0** commands. The rapid movements are performed using the feed rate defined by the **Rapid feed rate** parameter.

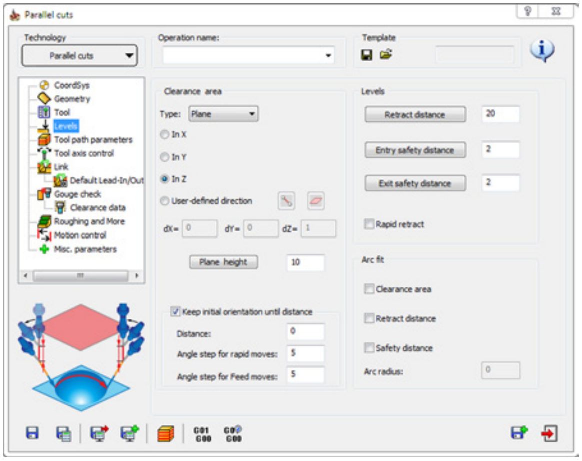
Example:

```
G1 X-2.942 Y75.567 Z24.402 A-88.436 B-26.482 F9998 M116
```

Levels

5

This page enables you to define the **Clearance area** and the machining levels.



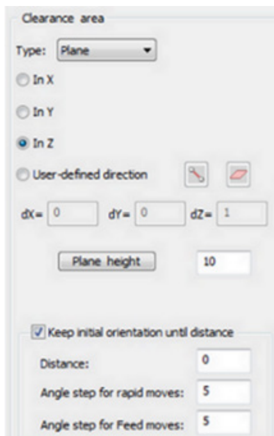
5.1 Clearance area

The **Clearance area** is the area where the tool movements can be performed safely without contacting the material. The tool movements in the **Clearance area** are performed with the rapid feed.

Depending on the drive surface or your machining strategy, you can choose different clearance area types:

- **Plane**
- **Cylinder**
- **Sphere**

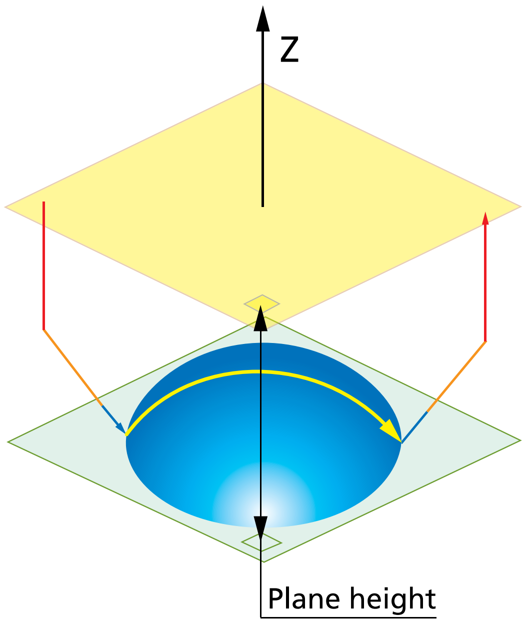
5.1.1 Plane



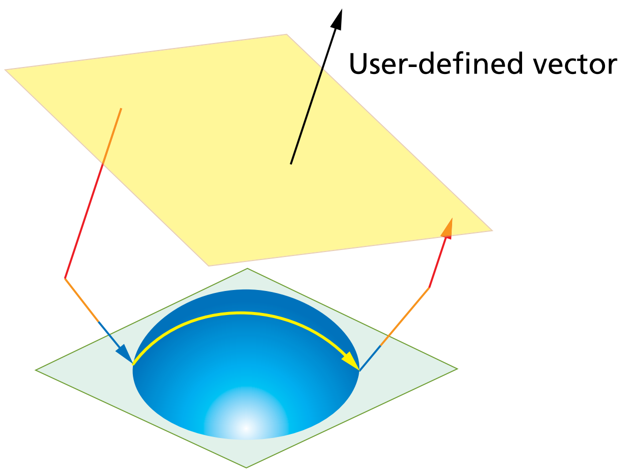
This option enables you to define the **Clearance area** by plane. The tool performs a retract movement to the **Clearance area** plane, and then a rapid movement in this plane.

The plane orientation is defined by a vector normal to the plane. With the **In X**, **In Y** and **In Z** options, SolidCAM enables you to define this vector as one of the Coordinate System axes (X, Y or Z).

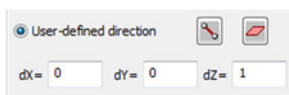
The **Plane height** parameter defines the distance between the appropriate Coordinate System plane and the **Clearance area** plane.



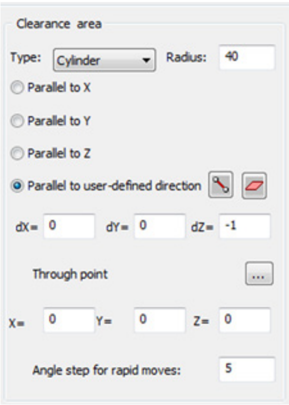
The **User-defined direction** option provides you with an additional capability to define the plane by an arbitrarily-oriented vector.



It enables you to define the direction vector by its coordinates (**dX**, **dY** and **dZ** parameters). Using the icons, you can pick the start and end points of the vector directly on the solid model, or select the entire face.

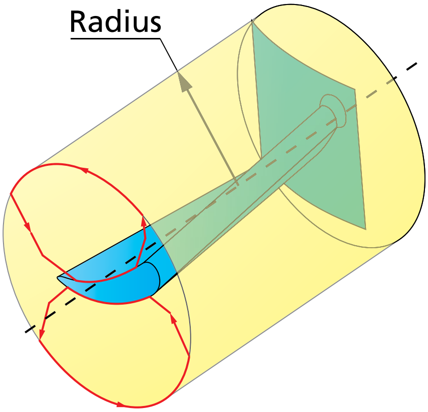


5.1.2 Cylinder

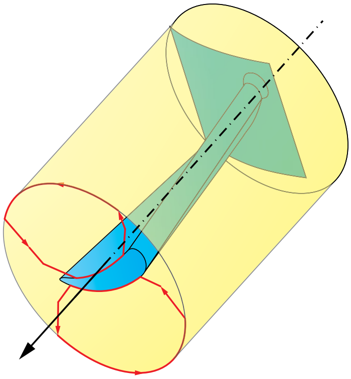


This option enables to define the **Clearance area** as a cylindrical surface enclosing the **Drive surface**. The tool performs a retract movement to the **Clearance** cylinder, and then a rapid movement along the cylinder surface.

The **Radius** parameter enables you to specify the cylinder radius.



The **Parallel to X**, **Parallel to Y** and **Parallel to Z** options enable you to define the cylinder axes only parallel to one of the Coordinate System axes (X, Y or Z).

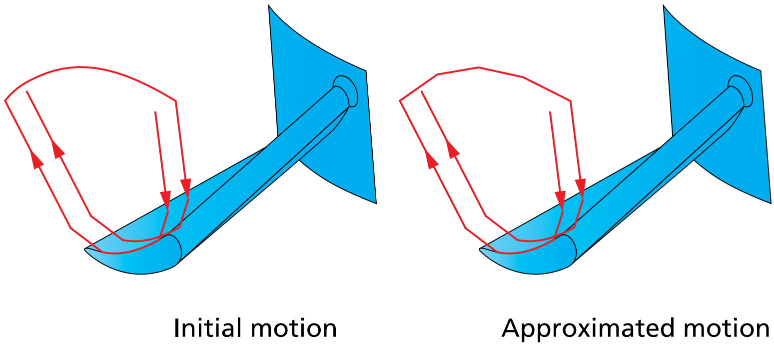


The **Parallel to user-defined direction** option provides you with an additional capability to define the cylinder axis by an arbitrarily oriented vector. It enables you to define the direction vector by its coordinates (**dX**, **dY** and **dZ** parameters). Using the icons, you can pick the start and end points of the vector directly on the solid model, or select the entire face.

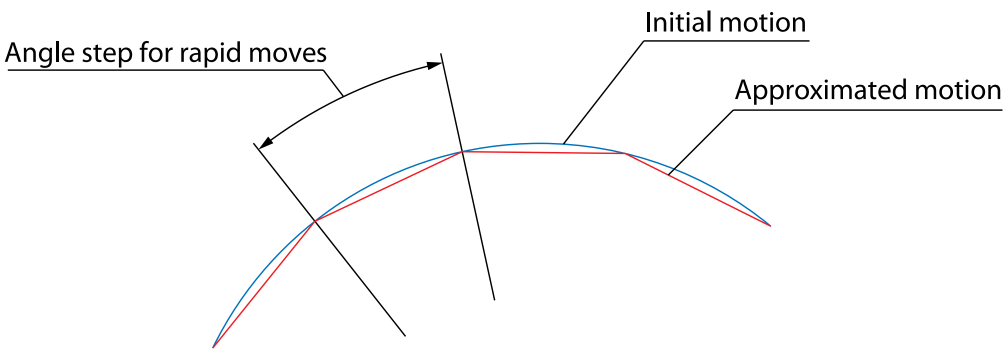
By default, the cylinder axis passes through the Coordinate System origin. You can specify the cylinder axis location either by clicking the **Through point** button or by defining the X, Y and Z coordinates of a point on the cylinder axis.

Angle step for rapid moves

SolidCAM enables you to approximate by lines the curved rapid movements performed at the spherical or cylindrical clearance area.



The approximation is controlled by the **Angle step for rapid moves** parameter illustrated below.



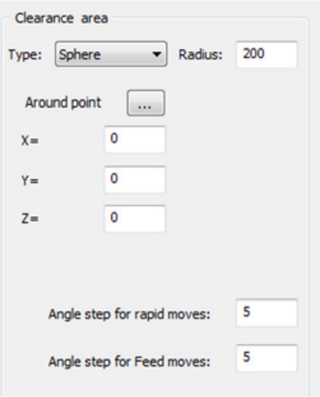
Small values of the **Angle step for rapid moves** parameter cause less deviation between the initial curve and the approximated path, but may cause some machines to slow down.

Using larger values of the **Angle step for rapid moves** parameter, you can reduce the number of approximation lines thus increasing the motion speed; however then collisions are possible, since the curved motion is simplified (when the **Angle step for rapid moves** is set to **90°** the circular movement is approximated to a square).

Angle step for feed moves

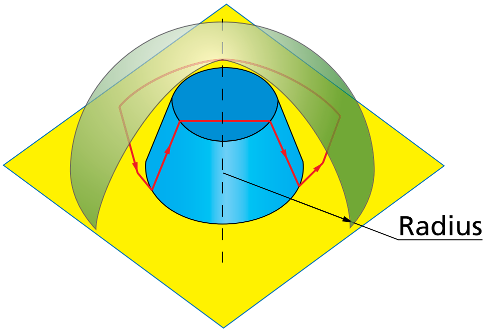
Similar to rapid movements, SolidCAM enables you to approximate by lines the curved feed movements.

5.1.3 Sphere



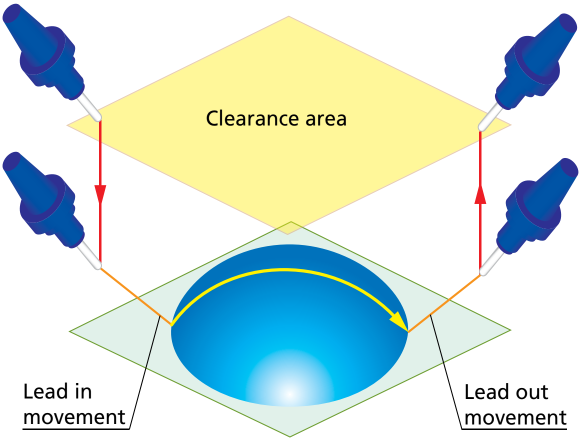
When this option is chosen, the **Clearance** area has a spherical shape; it should enclose your **Drive surface** (see topic **3.1.1**) geometry completely. The tool performs a retract movement to the **Clearance** sphere and then a rapid movement along the sphere surface.

The **Radius** parameter enables you to specify the sphere radius. By default, the sphere center is located at the Coordinate System origin. You can specify the sphere center location either by clicking the **Around point** button or by defining the X, Y and Z coordinates of the sphere center point.



5.1.4 Keep initial orientation until distance

This section enables you to alter the tool tilting for rapid movements in the **Clearance area**. When the tool descends from the **Clearance area** till the first cutting pass, it can be from the very beginning tilted according to the tool axis control parameters definition, or it can descend straight until a specified distance and then get tilted as required.

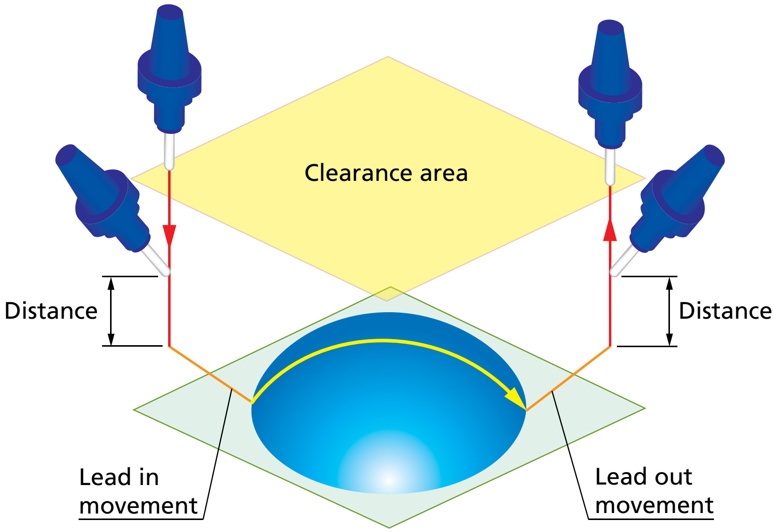


When the **Keep initial orientation until distance** check box is not selected, the tool descends from the **Clearance area**, tilted according to the tool axis control parameters definition.

After it has performed the cutting passes and the lead out movement, the tool returns to the **Clearance area**, tilted in the same manner

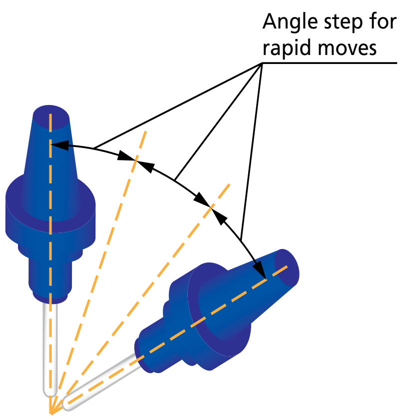
When the check box is selected, the tool descends not tilted, parallel to the vector normal to the plane, until it reaches the given distance to the beginning of the link movement, and then gets tilted as required. After it has performed the cutting passes and the lead out movement, the tool ascends up to the specified distance, gets adjusted to its initial angle, and returns to the **Clearance area** parallel to the vector normal to the plane. The **Distance** parameter defines the distance to/from the beginning of the link movement, at which the tool orientation changes.

The **Angle step for rapid moves** parameter defines the angle increments for the tool tilting.

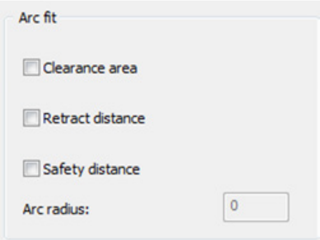


The **Angle step for feed moves** defines the angular step used for feed moves.

The **Keep initial orientation until distance** section is available only when the **Clearance area** is defined by plane.



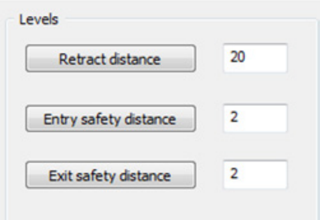
5.1.5 Arc fit



SolidCAM enables you to fit an arc to sharp angles. The arc can be fit at all the moves that go to the **Clearance area**, **Retract distance**, and **Safety distance**.

The fitted arc radius is determined by the **Arc radius** field.

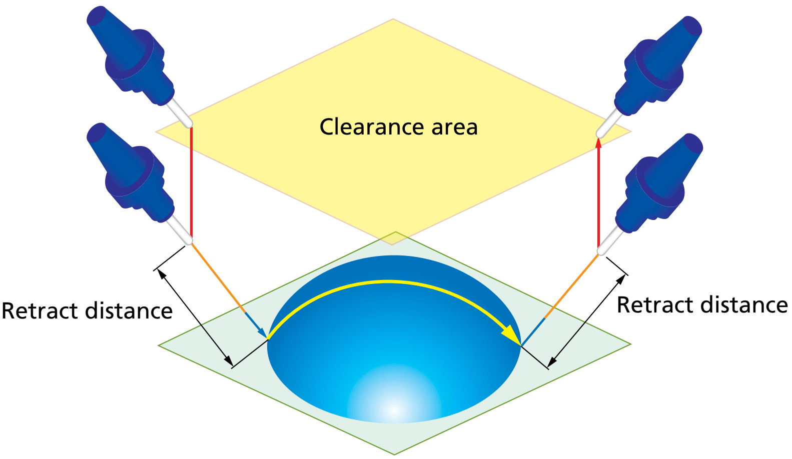
5.2 Machining levels



The **Levels** parameters enable you to define the **Retract** and **Safety distance** to approach and retract from the part.

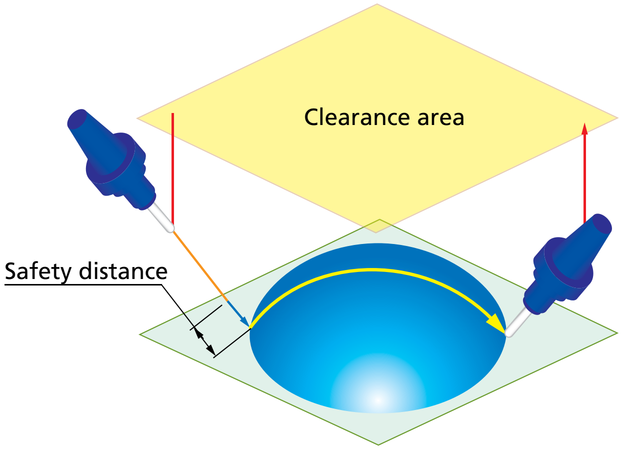
5.2.1 Retract distance

In the **Clearance** area, the tool turns to the final orientation for the first cut. After the rotation, the tool performs a rapid descent movement to the level specified by the **Retract distance** parameter.



5.2.2 Safety distance

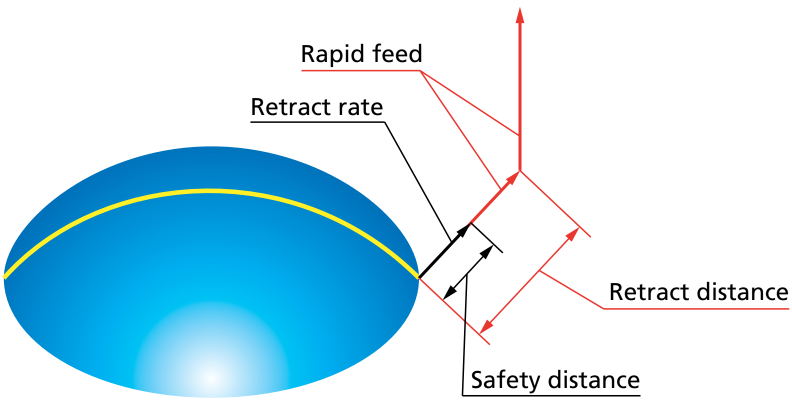
After the descent movement to the **Retract distance** level, the tool starts the approach movement to the material. The approach movement consists of two segments. The first segment is performed with rapid feed up to the **Entry safety** distance. From the **Entry safety distance** level, the approach movement is performed with the cutting feed. Upon retraction, the tool ascends to the **Exit safety distance**.



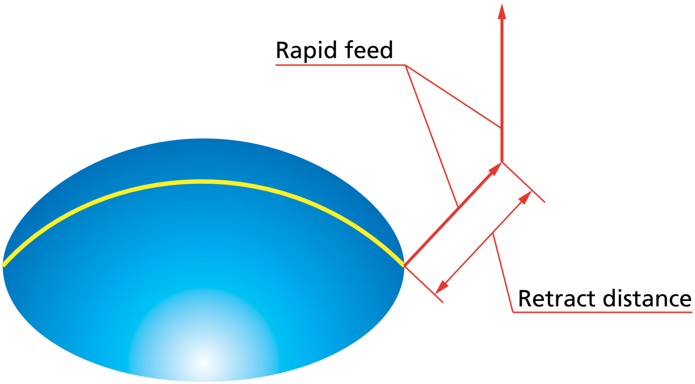
5.2.3 Rapid retract

This option enables you to perform the retract movement with rapid feed.

When this check box is not selected, the tool moves to the **Safety distance** with the feed defined as the **Retract Rate** parameter.



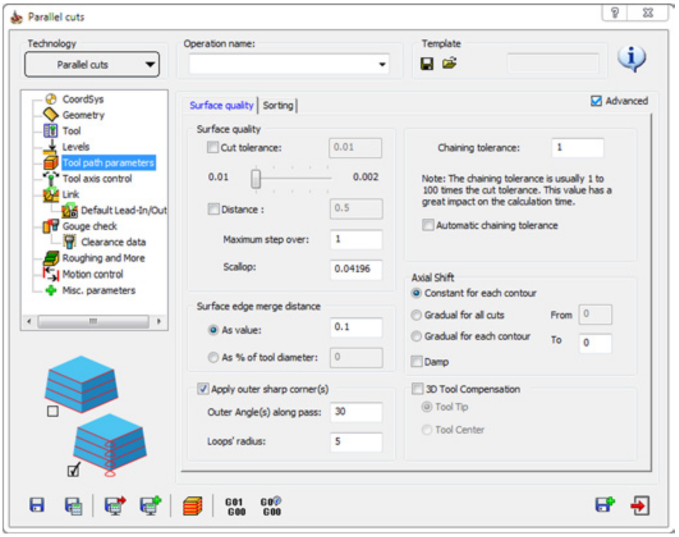
When this check box is selected, the retract movement is performed with rapid feed.



Tool path parameters

6

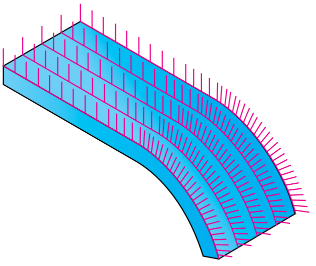
The **Tool path parameters** page enables you to define the parameters of finish machining.



6.1 Surface quality

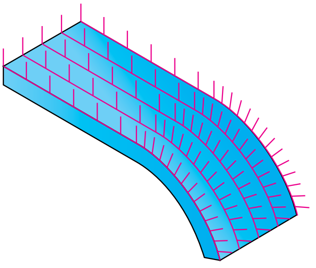
The **Surface quality** tab enables you to define the parameters that affect the surface finish quality.

6.1.1 Cut tolerance



The **Cut tolerance** parameter defines the tool path accuracy. The **Cut tolerance** parameter defines the chordal deviation between the machining surface and the tool path; the tool path can deviate from the surface in the range defined by the **Cut tolerance**.

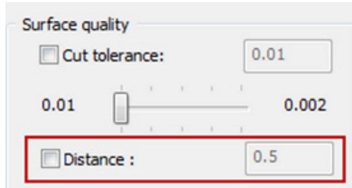
A smaller **Cut tolerance** value gives you more tool path points on the drive surface resulting in more accurately generated tool path. The result is a better surface quality, but the calculation time is increased.



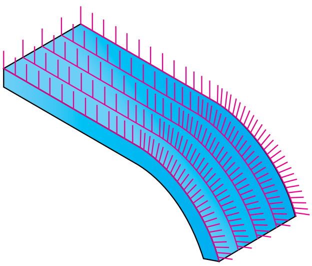
A greater **Cut tolerance** value generates less points on the tool path. After the machining, the surface finish quality is lower but the calculation is much faster.

You can type the value manually or adjust it using the slider.

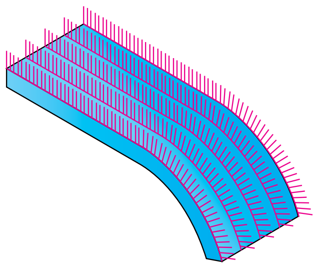
6.1.2 Distance



The **Cut tolerance** parameter defines the number of tool path points on a surface. The distance between these points is not constant and depends on the surface curvature: there are less points calculated on flat surfaces and more points on curved surfaces. The **Distance** parameter enables you to define the maximal distance between two consecutive tool path points. In other words, when the **Distance** option is used and the value is defined, SolidCAM generates tool path points at least at every specified distance.



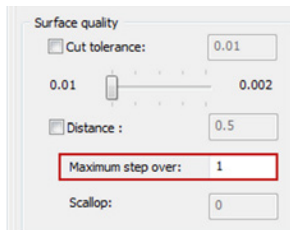
The **Distance** option is not used



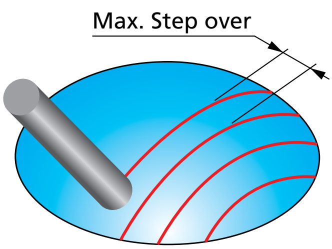
The **Distance** option is used

When the **Distance** option is not used, the number of tool path positions is determined by the **Cut tolerance** parameter and **Maximum angle step** parameter (see topic **7.2**).

6.1.3 Maximum Step over



This parameter defines the maximum distance between two consecutive cuts. The definition of the **Maximum Step over** parameter is different for each machining strategy:



- For the **Parallel cuts** strategy (see topic **3.1.4**), the **Maximum Step over** parameter defines the distance between the parallel planes.
- For the **Perpendicular to curve** strategy (see topic **3.1.7**), the **Maximum Step over** is measured along the curve, perpendicular to which the cutting planes are created.

- For the **Morph between two curves/surfaces** strategies (see topic **3.1.8**), the **Maximum Step over** defines the distance between two consecutive passes along the drive surface.

- For the **Parallel to curve/surface** strategies (see topics **3.1.6** and **3.1.7**), the **Maximum Step over** defines the distance between two consecutive passes along the drive surface.

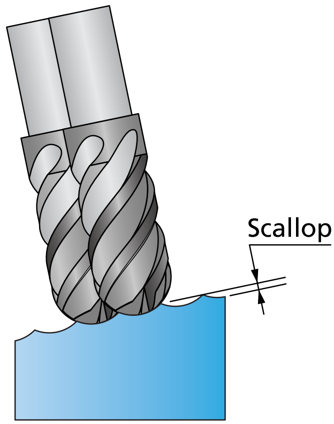
- For the **User defined** technology of the **Projection** strategy (see topic **3.1.9**), the **Maximum Step over** parameter is not relevant, because the projection curves can be chosen arbitrarily. For the **Spiral** and **Radial** technologies, the the **Maximum Step over** defines the distance between two consecutive passes.

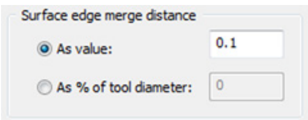
6.1.4 Scallop

The **Scallop** parameter enables you to define the cusp height of the machined surface.

The **Scallop** parameter is available only when a **Ball Nose Mill** tool is chosen for the operation.

The **Scallop** parameter corresponds to the **Maximum step over** parameter. When the **Scallop** is defined, SolidCAM automatically updates the **Maximum step over** value according to the chosen tool diameter and the **Scallop**; vise versa, when the **Maximum step over** is redefined, SolidCAM automatically recalculates the **Scallop** value.



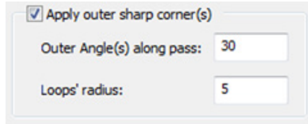


6.1.5 Surface edge merge distance

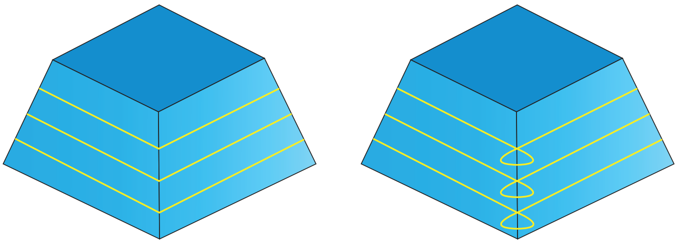
SolidCAM generates first tool paths for individual surfaces. Then they are merged together to form the complete tool path. The decision about merging is based on the **Surface edge merge distance** parameter. If all surface paths on a tool path slice are merged, SolidCAM checks if a closed surface path can be built by connecting the start to the end. This decision is made based on the **Surface edge merge distance** parameter meant to deal with minor gaps between surfaces edges.

The **Surface edge merge distance** parameter can be defined either as a numeric value (the **As value** option) or as a percentage of the tool diameter (the **As % of tool diameter** option). In both cases, this limit value must be greater than or equal to the **Cut tolerance** value.

6.1.6 Apply outer sharp corners

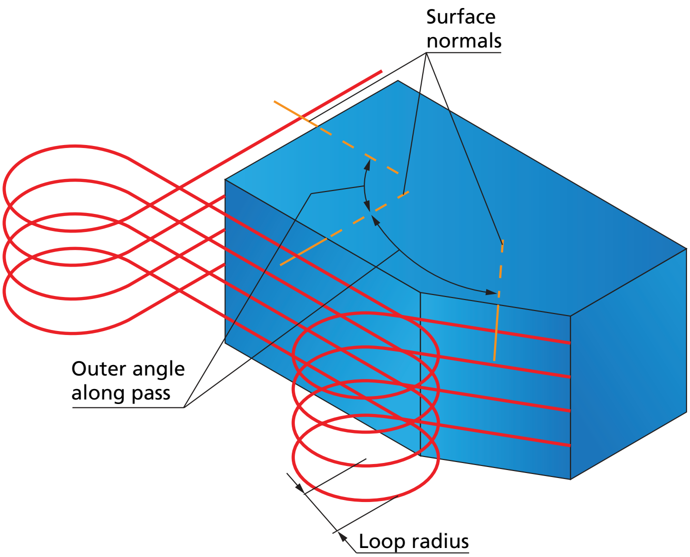


This option enables you to perform machining of adjacent outer edges of the model in such a manner that the sharpness of the corners is preserved. Instead of rolling around the edge that results in rounding of the corner, the tool path is extended for both edge surfaces, and the extensions are connected with a loop, resulting in an absolutely sharp machined corner.



When this check box is selected, you can define the parameters and conditions for corner looping.

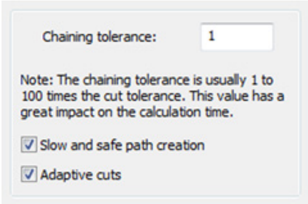
The **Outer angle along pass** parameter defines the maximal value of the angle between two normals to the surfaces of the corner to enable looping; for angles greater than defined, loops will not be performed.



The **Loops radius** value defines the radius of the loop to be performed.

Note that when the **Loops radius** is smaller than the radius of the tool, loops will not be performed.

6.1.7 Advanced options for surface quality



The **Chaining tolerance** parameter defines the tolerance of the initial grid used for the tool path calculation. The recommended value is 1 to 10 times the **Cut tolerance**. In some cases, for simple untrimmed surfaces, the **Chaining tolerance** value can be defined up to 100 times the **Cut tolerance** and would increase the calculation speed significantly.

The surface contact paths are created while analyzing and slicing the surface patches. If due to slicing the tool path topology becomes very complex (for example, patches parallel to curve and surface are very large), sometimes the surface contact paths cannot be constructed safely.

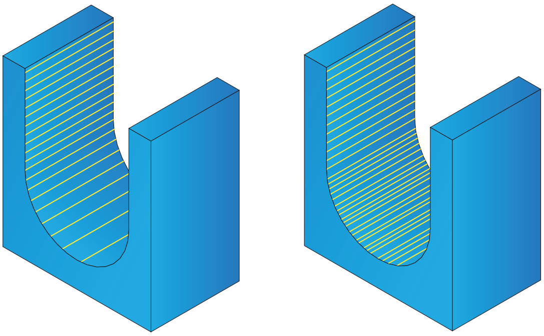
If the **Slow and safe path creation** check box is selected, a finer grid (based on the **Max. Step over** value) is applied for initial analysis of surface patches, thus delivering slow but safe results for surface contact points.

Adaptive cuts

This check box enables you to adjust the step over between tool path passes in an adaptive way, in order to ensure an acceptable distance between adjacent passes. This option is especially useful in machining of steep surfaces, molds, and U-shaped parts.

When this check box is not selected, the tool path passes can be distributed in such a manner that the distance between them is varying throughout the tool path. In certain cases, such distribution of passes may result in poor quality of surface machining.

When this check box is selected, additional lines can be inserted in the tool path if the distance between two adjacent passes is considered too large. As a result, the number of calculated cuts increases.



Adaptive cuts check box is not selected

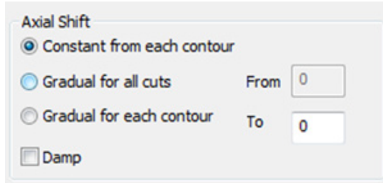
Adaptive cuts check box is selected

This option is not available for use with the **Parallel cuts**, **Perpendicular to curve** and **Projection** technologies.

Note that when this option is used, the calculation time increases.

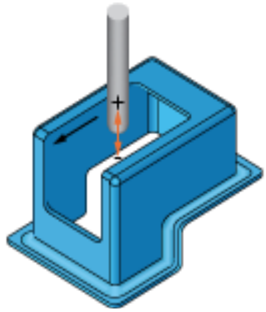
Axial Shift

SolidCAM enables you to offset the tool along the tool axis. You can choose one of the following options:



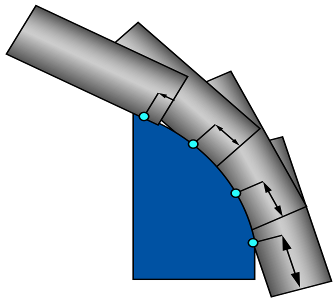
- **Constant for each contour**

  The offset value does not change along the axis. The value is set in the **To** field.



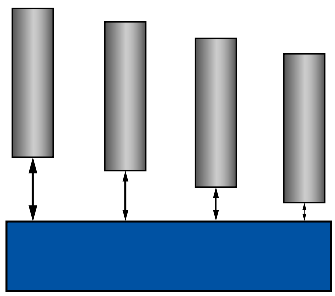
- **Gradual for all cuts**

  The contact point between tool and work piece gradually shifts along the axis with each consecutive cut. The offset increases from the value set in the **From** field up to the value set in the **To** field.



- **Gradual for each contour**

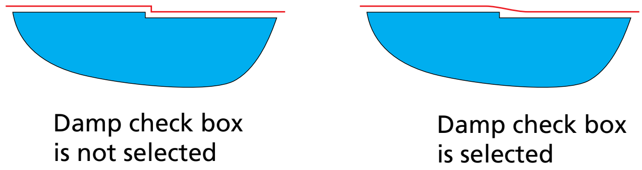
  The contact point between tool and work piece slides gradually for each cut. The offset increases from the value set in the **From** field up to the value set in the **To** field.



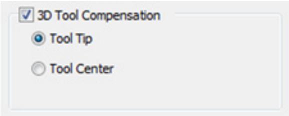
Damp

This **Damp** option enables you to smooth the tool path in such a manner that vertical jumps are avoided.

When this check box is not selected, the resulting tool path exactly follows the defined edge curve. When this check box is selected, the tool path is smoothed, and therefore does not contain sharp jumps.



6.1.8 3D Tool compensation

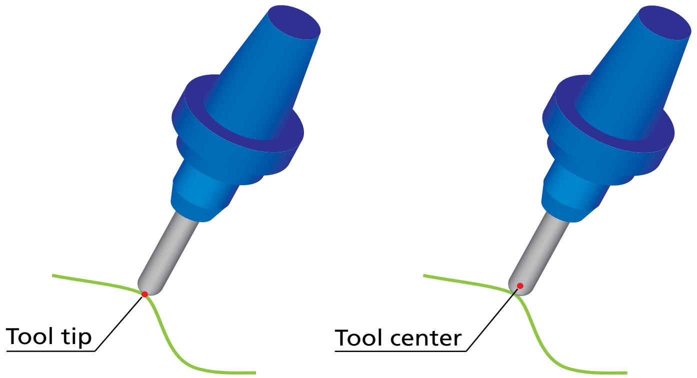


When this check box is selected, the tool compensation options of the CNC-controller are used in the GCode. The output tool path is recalculated according to the following formula:

C = T + R * N,

where **C** is a new coordinate of tool center, **T** is the coordinate of tool tip, **R** is the corner radius of the tool, and **N** is the tool vector.

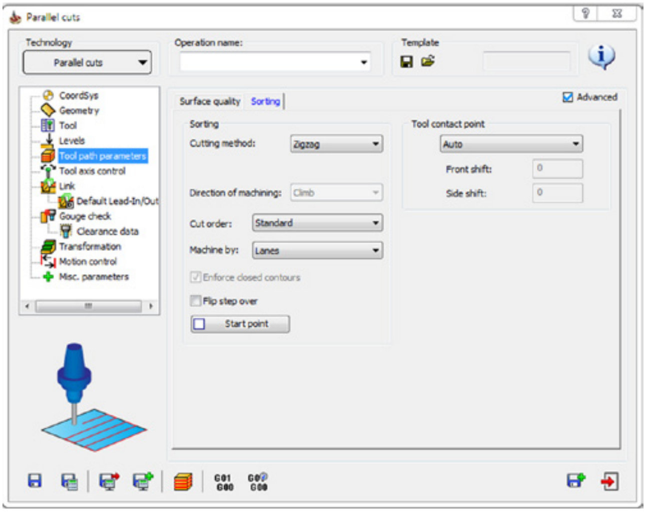
- When the **Tool Tip** option is chosen, the tool path is calculated according to the tool tip, and the *type_offset_3D:tool_tip* command is output to the GCode under *compensation_3d*.
- When the **Tool Center** option is chosen, the tool path is calculated according to the tool center and the *type_offset_3D:tool_center* command is output to the GCode under *compensation_3d*.



This option is available only when the tool of the **Ball nose mill** or **Bull nose mill** type is chosen for the operation.

6.2 Sorting

The **Sorting** tab enables you to define the order and direction of the cuts.

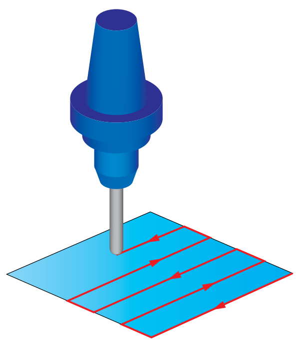


6.2.1 Cutting method

This option enables you to define how the cuts are connected. It has three choices: **Zigzag**, **One Way**, and **Spiral**.

Zigzag

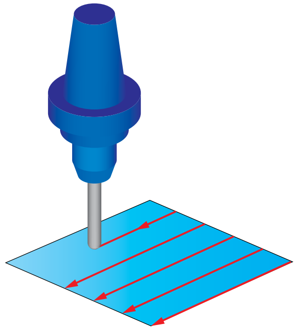
When the **Zigzag** option is chosen, the machining direction changes from cut to cut. The tool performs the machining of a cut in the specified direction, then moves to the next cut and machines it in the opposite direction.



Zigzag

One Way

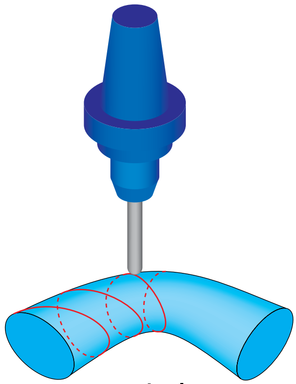
When the **One Way** option is chosen, all cuts are machined in the same direction. The tool performs the machining of a cut in the specified direction, then moves to the start of the next cut and machines it in the same direction.

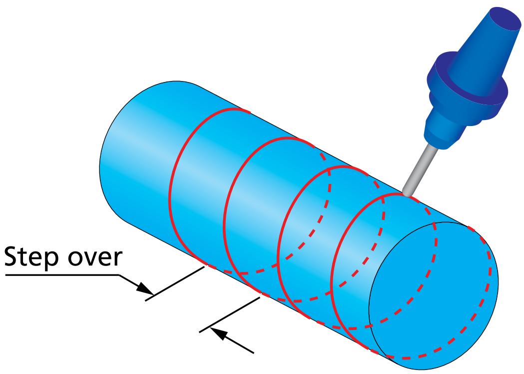


Spiral

With the **Spiral** option, SolidCAM generates a spiral tool path around the drive surface according to the chosen pattern.

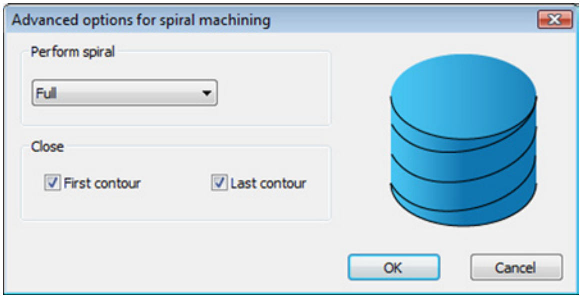
The spiral pitch is defined by the **Max. Step over** parameter.





This cutting method is available for use with all the strategies except for the **Projection** strategy.

Clicking the **Advanced** button displays the **Advanced options for spiral machining** dialog box.

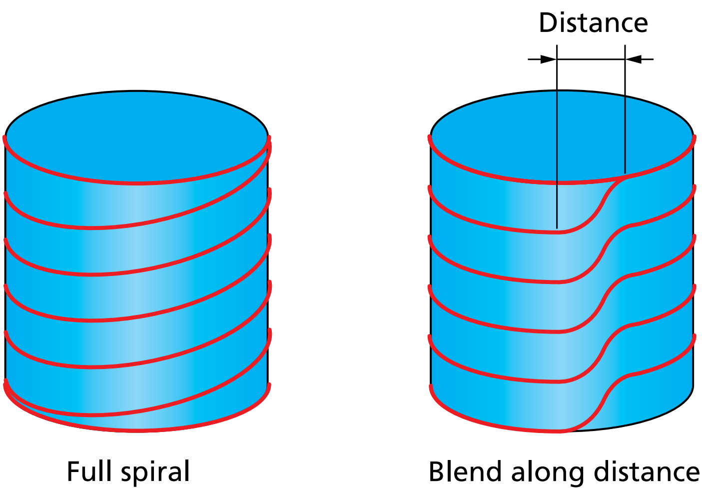


- **Perform spiral**

  When the **Full** option is chosen, the cuts are performed in constant spiral motions.

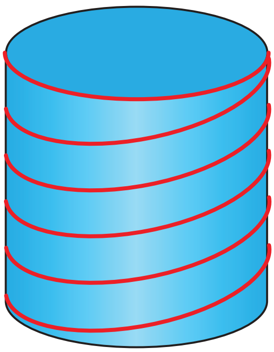
  When the **Blend along distance** option is chosen, every slice except for the last one is trimmed by a certain distance. The trimmed slice is connected with the following slice by a blend spline. The value in the corresponding edit box defines the trimming distance for the slice in case the **Blend along distance** option is chosen.

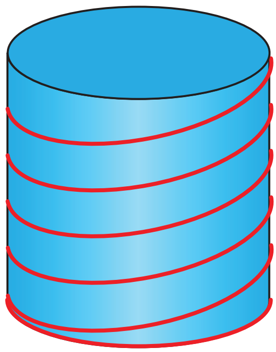
- **Close**



When the **First contour** check box is selected, machining of the first slice is performed in a closed contour. The spiral machining motions start with the second slice.

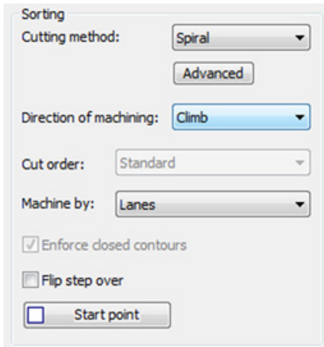
When this check box is not selected, the spiral machining motions start with the first slice. Likewise, when the **Last contour** check box is selected, machining of the last slice is performed in a closed contour.


First contour

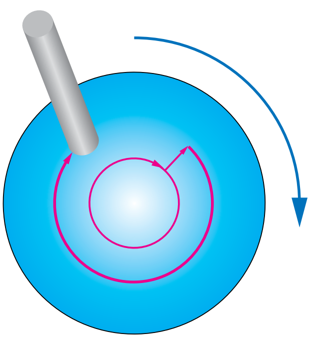

Last contour

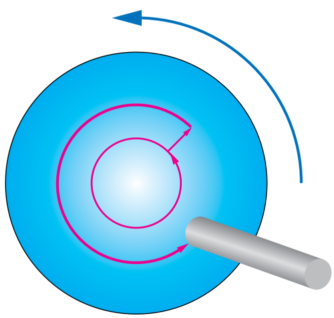
6.2.2 Direction of machining

When the **One Way** or **Spiral** options are chosen for **Cutting method**, SolidCAM enables you to define the following direction of cuts.

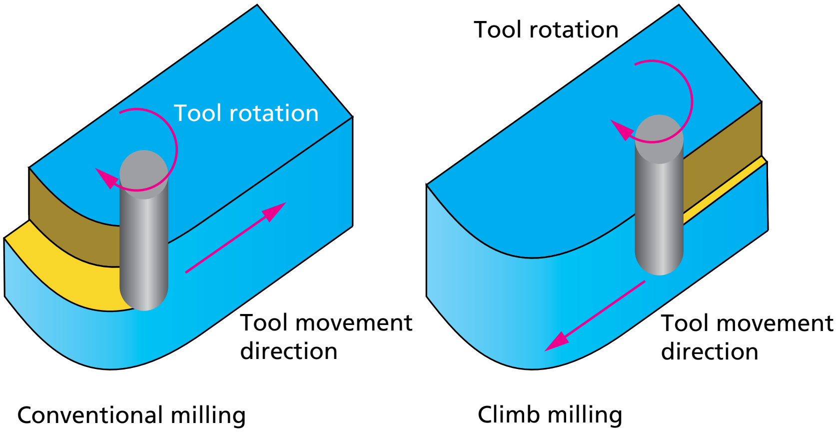


- The **CW** option enables you to perform the machining in the clockwise direction.
- The **CCW** option enables you to perform the machining in the counterclockwise direction.

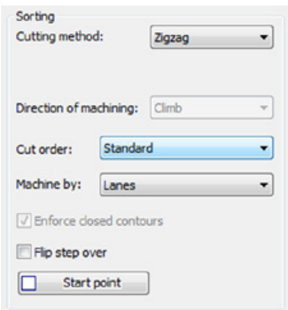

Clockwise


Counterclockwise

- The **Climb** option enables you to perform climb milling, which is preferred when milling heat treated alloys. Otherwise, chipping can result when milling hot rolled materials due to the hardened layer on the surface.
- The **Conventional** option enables you to perform the conventional milling, which is preferred for milling of castings or forgings with very rough surfaces.

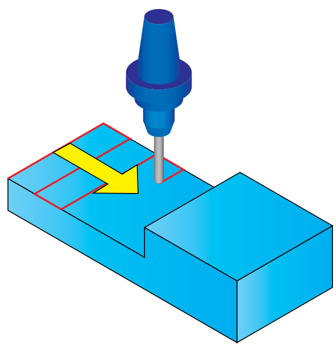


6.2.3 Cut order

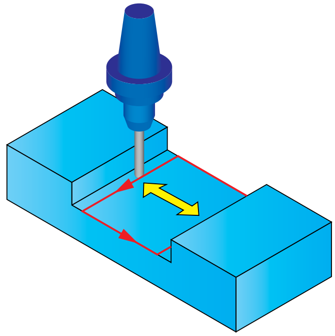


The **Cut order** option enables you to define the sequence of the cuts. The following options are available:

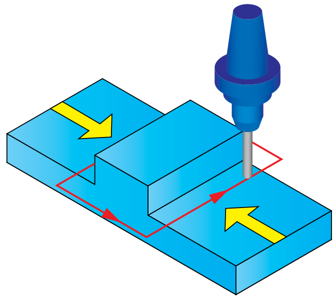
- With the **Standard** option, SolidCAM performs the machining from one side of the drive surface and continues to the other side.



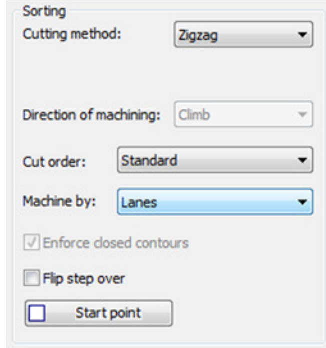
- With the **From center away** option, the machining starts from the center of the drive surface and continues outwards.



- With the **From outside to center** option, the machining starts from the drive surface edges and continues inwards.

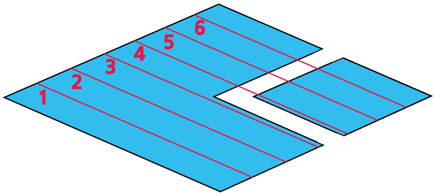


6.2.4 Machine by

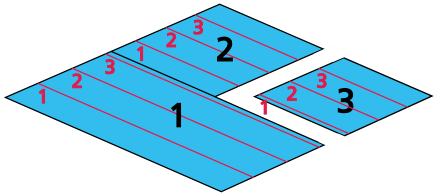


SolidCAM enables you to define the machining order for a Sim. 5-axis operation. The **Machine by** list enables you to choose the order of machining of certain areas; it defines whether the surface will be machined by **Lanes** or by **Regions**.

The generated tool path usually has a topology of multiple contours (lanes) on the drive surfaces. When the tool path is generated in many zones, it might be preferable to machine all the regions independently.

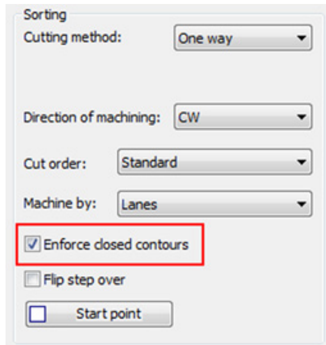


Lanes

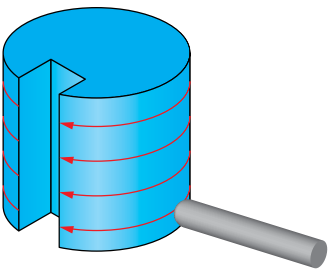


Regions

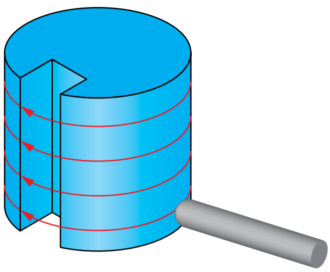
6.2.5 Enforce closed contours



When the geometry is not completely closed, the **Enforce closed contours** option enables you to close the geometry and perform the machining of closed contours.



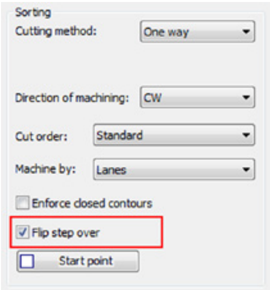
Enforced closed contours option is not selected



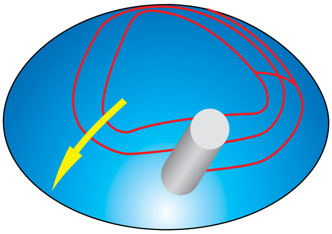
Enforced closed contours option is selected

This option is available only with the **CW** and **CCW** options chosen for **Direction of machining** (see topic **6.2.2**).

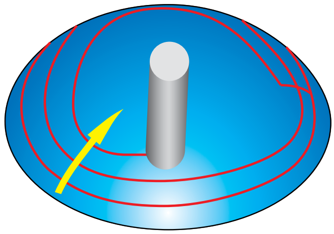
6.2.6 Flip step over



When the **Standard** option is chosen for the **Cut order**, the **Flip step over** parameter enables you to change the direction of the cuts.

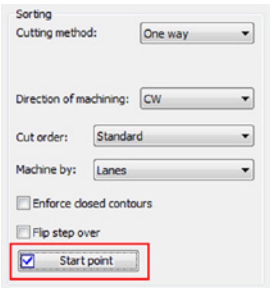


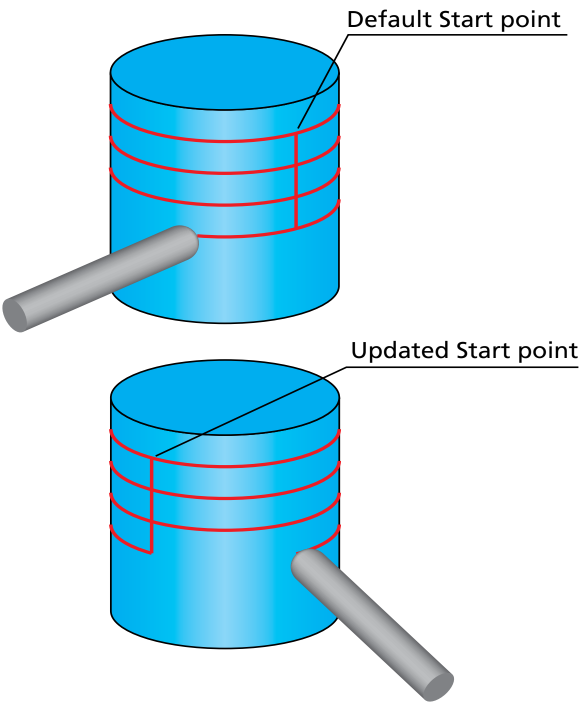
The machining starts from the top of the model.



With the **Flip Step over** option, the machining is reversed and starts from the edge.

6.2.7 Start point



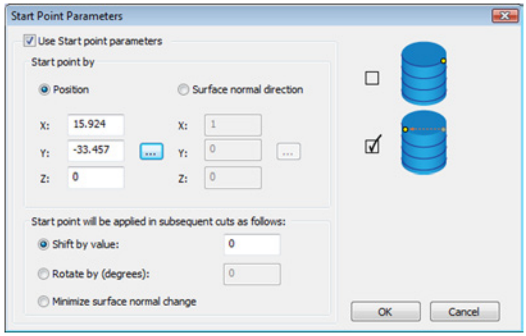


For closed contours, the **Start point** option enables you to define a new position of the start point of the first cut. The position is defined along a cut. The start points of the next cuts are determined automatically, taking into account the start point location, the cutting strategy and the **Rotate by** parameter.

Clicking the **Start point** button displays the **Start Point Parameters** dialog box.

Start point by

This section enables you to choose the method of start point definition.



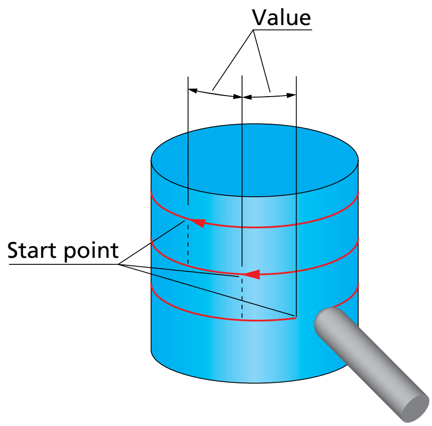
- **Position**

  This option enables you to define the start point by specifying the coordinates of the position. The coordinates of the selected point are displayed in the **X**, **Y** and **Z** edit boxes.

- **Surface normal direction**

  When this option is chosen, the start point is located at the intersection of the tool path with the vector specified by user-defined **XYZ** coordinates. Vector coordinates can be set by numeric values, or the direction can be picked on the model using the  button.

The selected start point is applied to the first cut. For the subsequent cuts, you can define the start point using one of the following options:

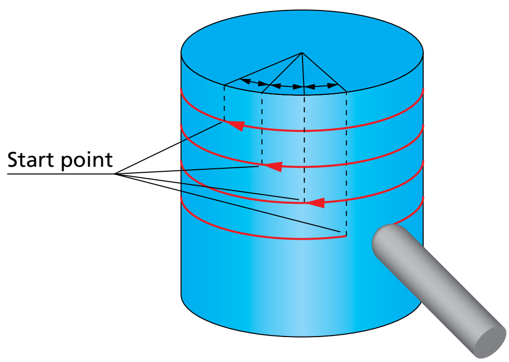


- **Shift by value**

  This option enables you to start the next cut at a specified distance from the previous start point. The distance defined in related edit box is measured along the path.

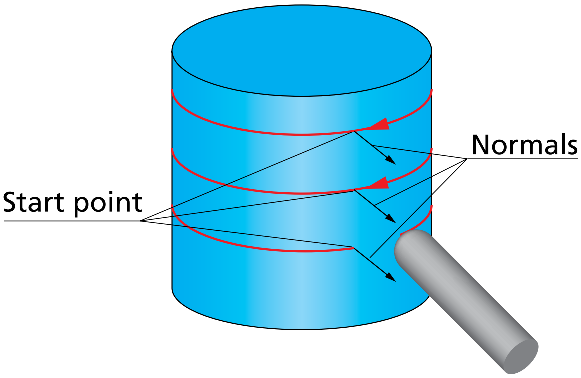
- **Rotate by**

  This option enables you to rotate the start position of the cuts relative to the start position of the first cut. The **Rotate by** value defines the rotating angle for the start position for subsequent cuts.



- **Minimize surface normal change**

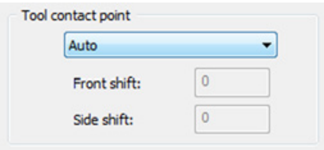
  With this option, SolidCAM automatically chooses the start points for passes in such way that the change of the direction between surface normals at the start points is minimal.



If the defined start point position is not located on the drive surface, SolidCAM automatically determines the closest point on the drive surface and uses it as the start point.

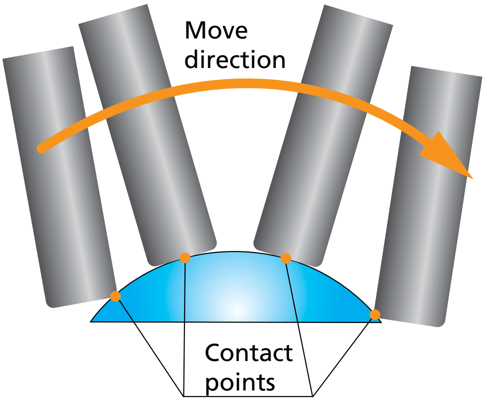
6.3 Tool contact point

The **Tool contact point** section enables you to define the point on the tool surface that contacts with the drive surfaces during the machining.



Auto

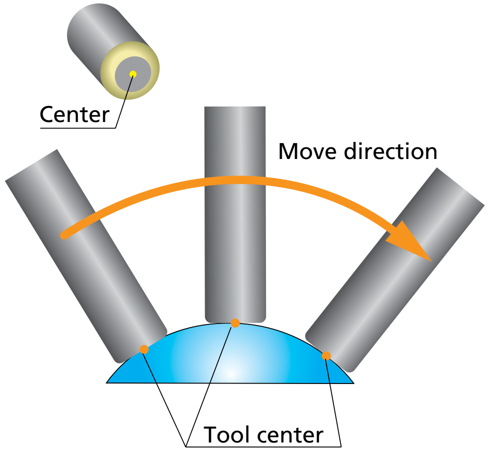
SolidCAM automatically defines the tool contact point. When the tool orientation changes in the process of machining, the tool remains in contact with the drive surface. The tool contact point moves from the tool tip to the tool center maintaining the tangency between the tool and the drive surface.



At Center

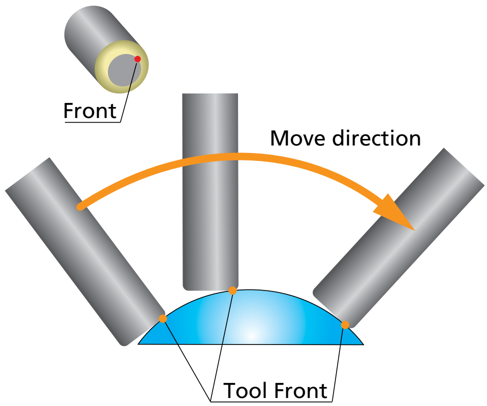
The tool contact point is located at the tool center (tool tip). If the tool orientation is changed according to the **Tilting options** (see topic **7.3**), the tool is tilted around the tool center point.

Note that with this option the tool is not located tangentially to the drive surface. Use the **Gouge checking** options (see chapter **9**) to avoid possible collisions.



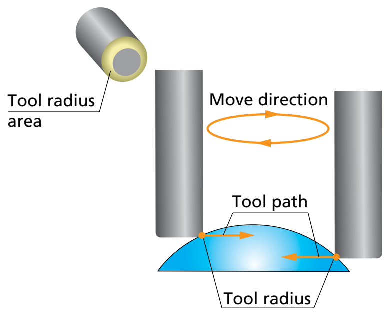
At Front

The tool contact point is located at the beginning of the tool corner radius (for bull nosed tools) in the direction of the tool movement. All changes to tool orientation are performed around the contact point and may cause gouges. It is recommended to use the **Gouge checking** options (see chapter **9**) to avoid possible gouges.



At Radius

The tool contact point is automatically determined at the tool corner radius area; the tool tip is not in contact with the drive surface.

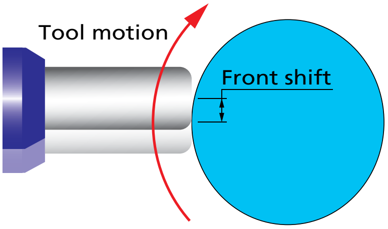


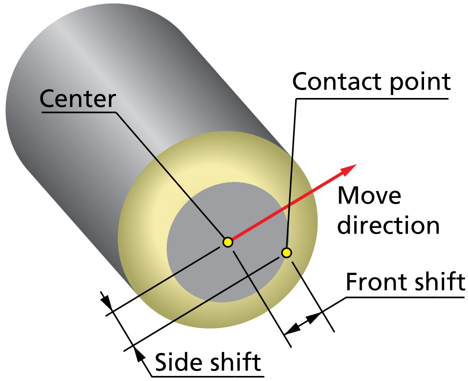
At user-defined point

SolidCAM enables you to define the contact point between tool and drive surface by the tool center shifting; the shifting is defined by two parameters:

- **Front shift**

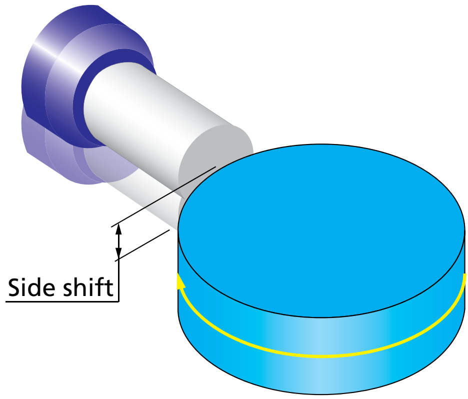
This parameter enables you to define the shift of the contact point in the tool motion direction. When a positive value is specified, the contact point moves from the tool center in the tool motion direction.





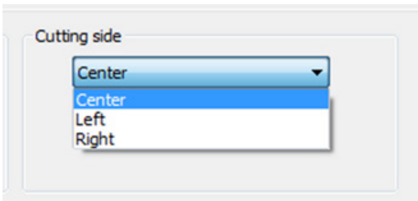
- **Side shift**

This parameter enables you to define the shift of the contact point in the direction perpendicular to that of the tool motion. When a positive value is specified, the tool center moves to the right side (relative to the tool motion direction) and the contact point moves from the tool center to the left side.



6.4 Cutting side

This section enables you to position the tool at the **Center**, **Left** or **Right** side of the tool path.

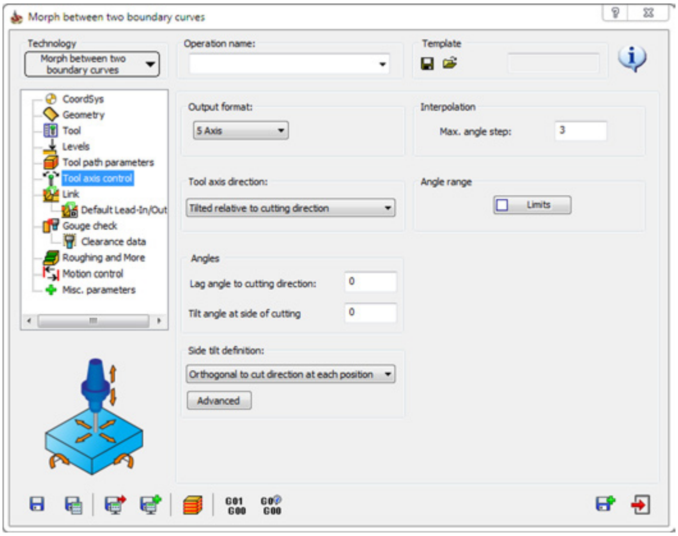


This option is available in **Projection** (**User defined** and **Offset**) strategies.

Tool axis control

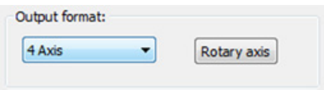
7

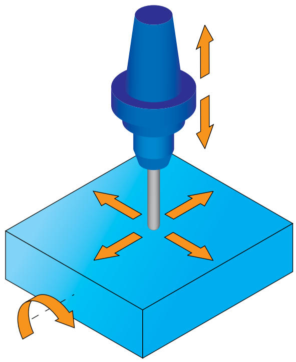
The **Tool axis control** page enables you to define the orientation of the tool axis during the Sim. 5-Axis machining.



7.1 Output format

This parameter enables you to choose the **Output format** of the current Sim. 5-Axis operation.

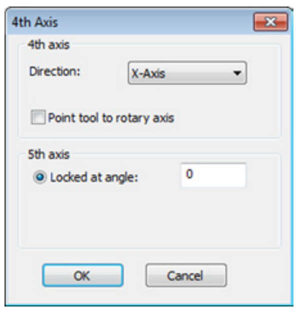




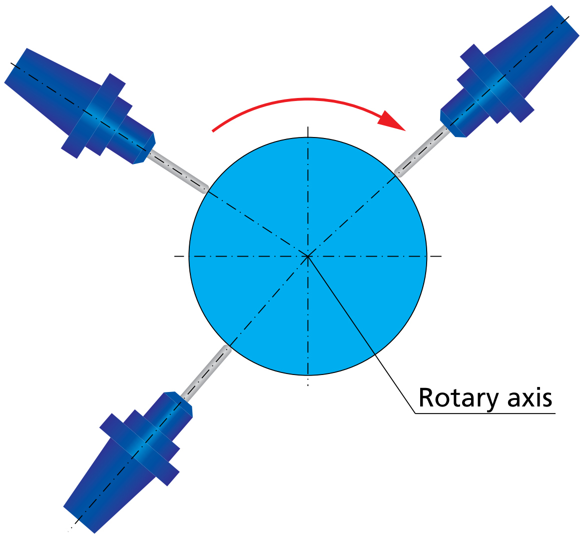
4 Axis

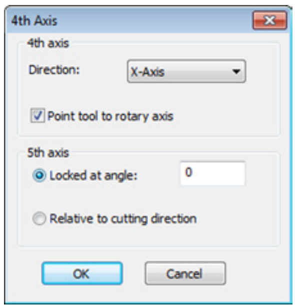
This output format is used for 4-axis finish operations such as turbine blade profiles and spiral parts. With this output format, SolidCAM generates a 4-axis GCode with tool tilting around the rotation axis. The tool is normal to the center line. The only tilt strategies available are those that support this type of tilting (4-axis). When this output format is chosen, SolidCAM enables you to define the rotary axis orientation.

Click the **Rotary axis** button to display the **4th Axis** dialog box. This dialog box enables you to choose the rotary axis (**X-**, **Y-**, **Z**-axis of the Coordinate System or another **User defined** axis).



The **Point tool to rotary axis** option enables you to define the tool axis orientation when the **4-Axis** output is chosen. With this option, the tool is oriented in such way that its axis intersects with the rotary axis. When this option is activated, all other options defining the tool axis orientation (tool tilting) are not available.



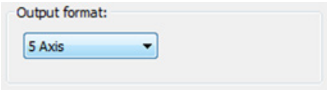


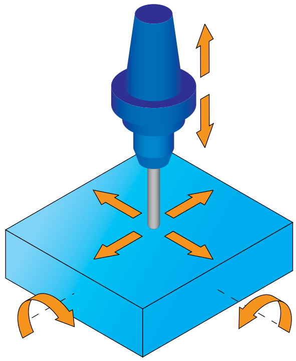
The GCode generated with the **4 Axis** output format is suitable for both 4-axis and 5-axis CNC-machines. When the 5-axis CNC-machine is used for the 4-axis operation, SolidCAM enables you to set the fifth axis to a specific angle and lock it in this orientation; the 4-axis machining is performed with the fixed fifth axis. In the **5th axis** section, the **Locked at angle** parameter enables you to specify the angle at which the fifth axis is locked. If the **Point tool to rotary axis** option is selected, you can either lock the 5th axis at the specified angle or make it **Relative to cutting direction** by selecting the corresponding option.

In most of 4-axis CNC-machines the tool axis direction (spindle direction) is always perpendicular to the rotary axis. Therefore, the **Locked at angle** parameter has to be set to 0 for this type of CNC-machines.

Some of the 4-axis CNC-machines have a spindle unit mounted with some fixed tilting angle to the rotary axis. In this case, the **Locked at angle** parameter must be set to the CNC-machine fixed tilting angle.

5 Axis



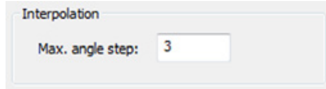


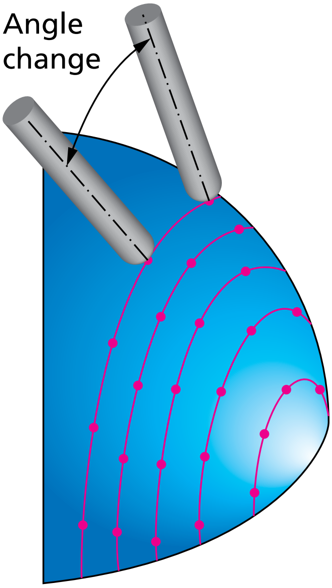
With this output format, simultaneous 5-axis output is performed. This output format is used for 5-axis finish and supports all types of 5-axis operations. You have complete control over all of the cutting parameters. The tool can be tilted to any possible direction supported by the machine. All the tilt strategies are available.

This output format of operation is available only for post-processors that support 5-axis machining.

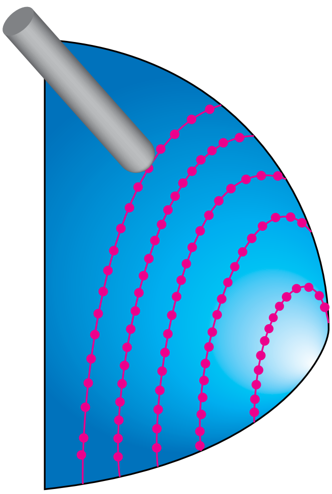
7.2 Interpolation

The **Max. Angle step** parameter enables you to define the maximal allowed angle change between the tool axes at two consecutive tool positions.



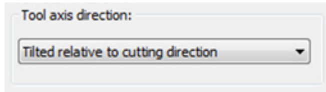


Decreasing the **Maximum Angle step** value causes SolidCAM to generate more tool path points.



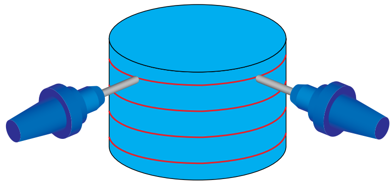
7.3 Tilting strategies (Tool axis direction)

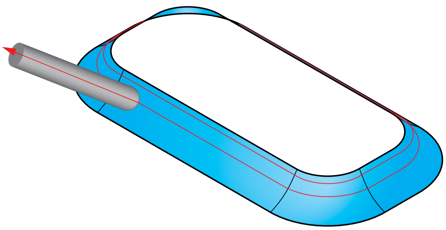
The **Tool axis direction** section enables you to choose the tool tilting strategy. The tool tilting strategies enable you to define the orientation of the tool axis during the machining relative to the surface normal.



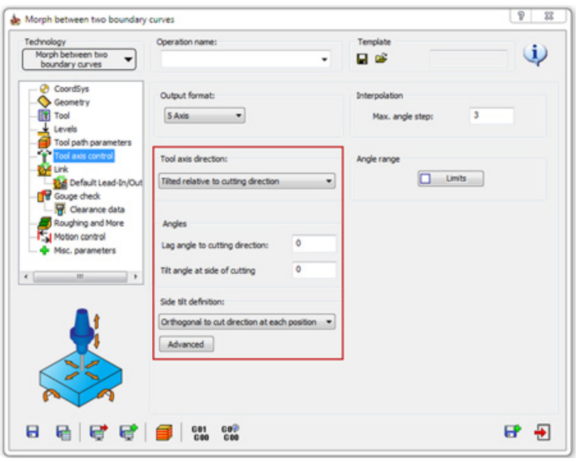
7.3.1 Not to be tilted and stays normal to surface

With this option, SolidCAM enables you to keep the tool axis direction coincident to the surface normal at the cutting position. In other words, the tool is always normal to the surfaces during the machining.

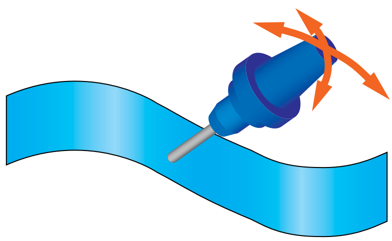




7.3.2 Tilted relative to cutting direction

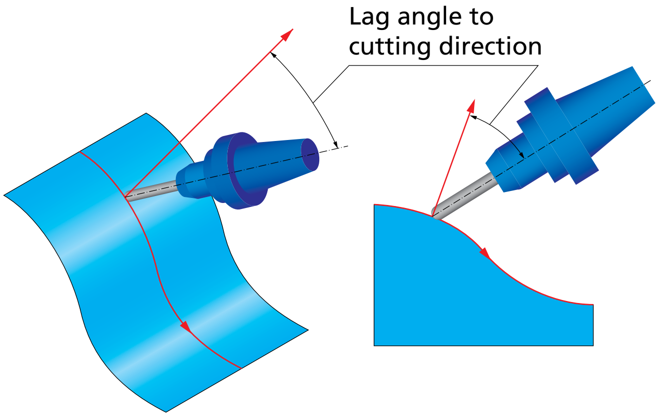


With this option, SolidCAM enables you to define the tool tilting relative to the cutting direction.

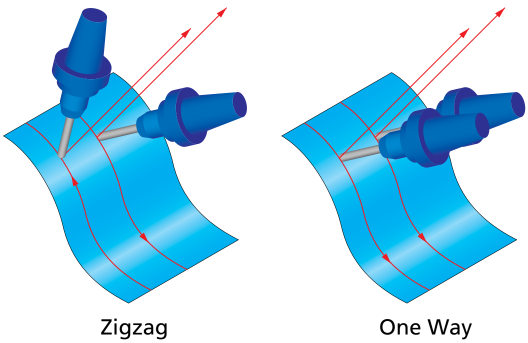


Angles

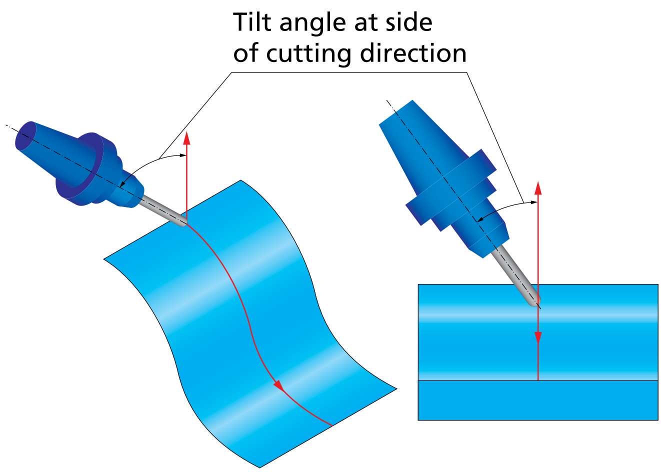
The **Lag angle to cutting direction** parameter enables you to define the tool tilting in the direction of the cutting pass. The **Lag angle to cutting direction** parameter is measured relative to surface normal.



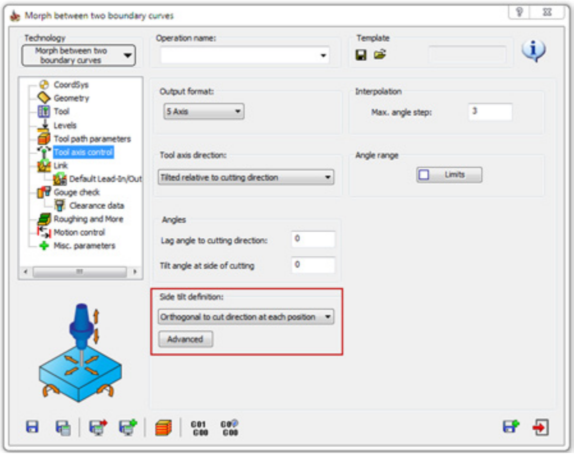
In case of **Zigzag** machining (see topic **6.2.1**), the cutting direction is changed from pass to pass. Therefore the tool tilting direction is changed according to the cutting direction. When the **One Way** cutting method is used, the tool tilting direction is the same.



The **Tilt angle at side of cutting** parameter enables you to define the tool inclination in the direction determined by **Side tilting** options. The **Tilt angle at side of cutting** parameter is measured relative to surface normal.



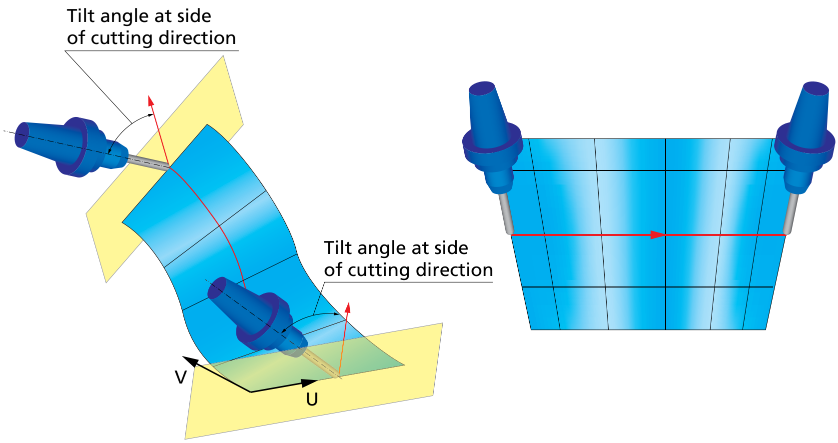
Side tilting options



SolidCAM enables you to choose the following options to define the direction of the side tilting:

- **Follow surface ISO Lines direction**

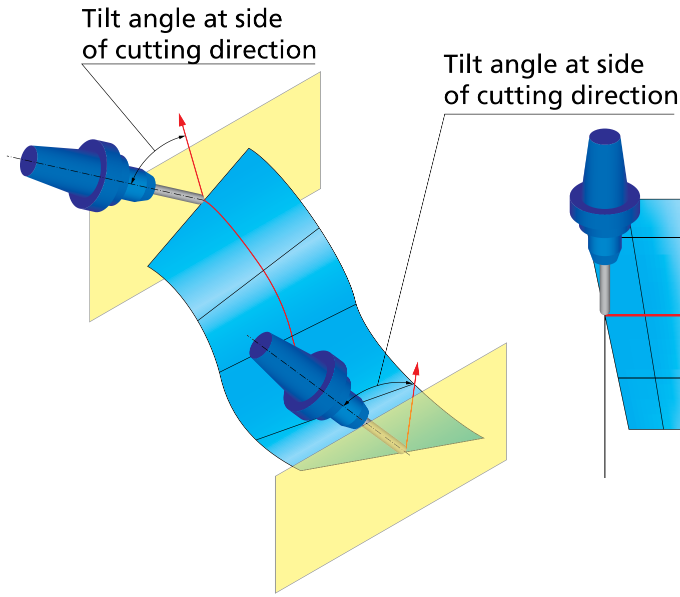
  The direction of the side tilting is chosen according to the direction of the U- and V-vectors of the drive surface.



Fanning (see **Advanced tilting parameters**) can be applied to avoid quick changes of tool orientation due to irregularities of the drive surface geometry.

- **Orthogonal to cut direction at each position**

  The plane of the side tilting is orthogonal to the tool path direction for each cutting position.



- **Orthogonal to cut direction at each contour**

  The direction of the side tilting is determined by an orthogonal line to a tool path segment. SolidCAM approximates the orthogonal vectors in all tool path positions of the segments according to the **Approximate** option:

  - **By one vector**: SolidCAM calculates a single orthogonal vector instead of all the tool path positions vectors.

  - **By two vectors**: SolidCAM calculates two orthogonal vectors instead of all the tool path positions vectors.

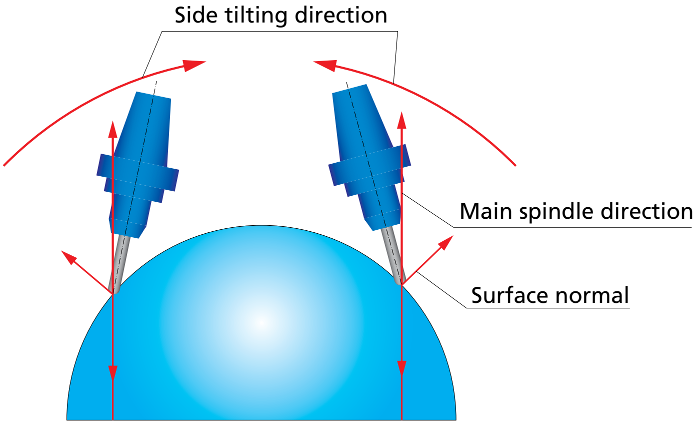
  - **Smooth**: SolidCAM calculates a number of orthogonal vectors to perform smooth changes in the side tilting direction.

  - **Smooth (Local)**: the orthogonal vectors are calculated by short segments around the tool path point.

This option is available for the **Tilted relative to impeller machining layer** and **Tilted relative to cutting direction** strategies, in the latter case only when the **Orthogonal to cut direction at each contour** option is chosen for **Side tilt definition**.

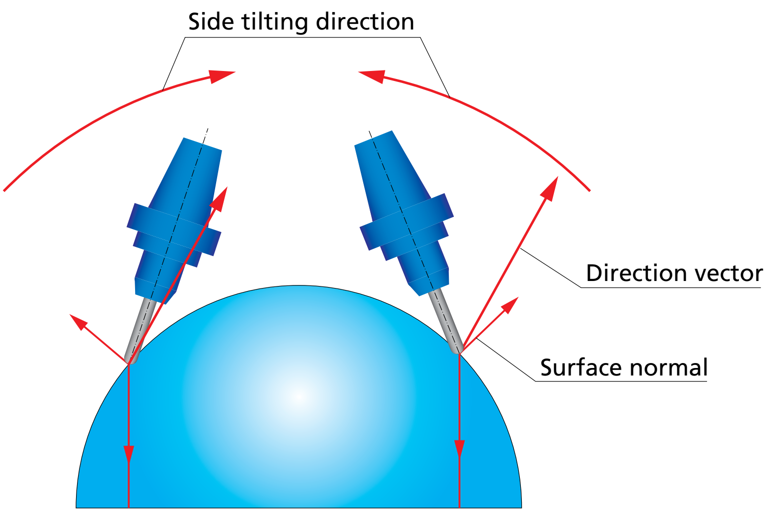
- **Use spindle main direction**

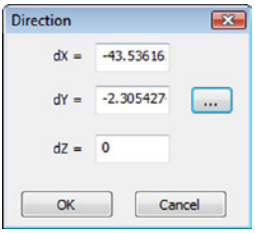
SolidCAM uses the spindle main direction vector definition as the reference for the side tilting direction. The side tilting is always performed in the direction defined by the spindle main direction vector.



- **Use user-defined direction**

SolidCAM enables you to specify the reference vector to determine the side tilting direction. The side tilting is always performed in the direction defined by user-defined vector.

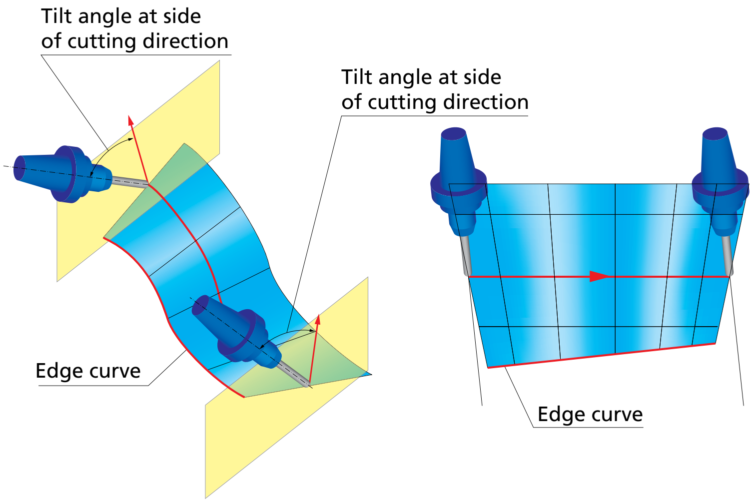




Click the **Data** button to display the **Direction** dialog box that enables you to specify the direction point for the vector. The vector starts from the Coordinate System origin and points to the specified location.

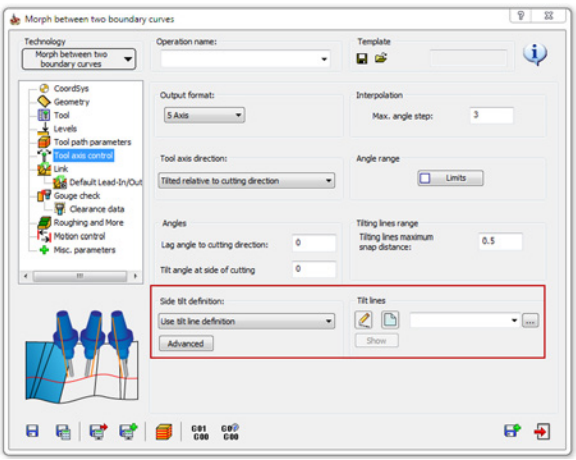
- **Orthogonal to edge curve**

With this option, the plane of the side tilting is orthogonal to the edge curve at each cutting position.

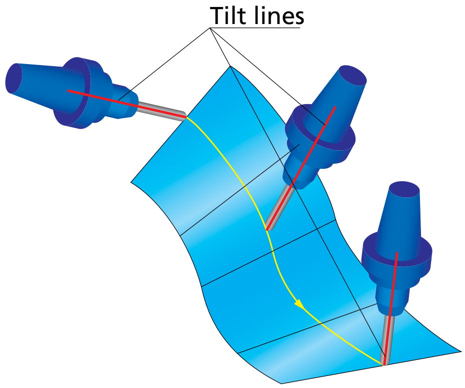


This option is available only with the **Parallel to curve** (see topic **3.1.4**) and **Morph between two boundary curves** (see topic **3.1.8**) strategies.

- **Use tilt line definition**



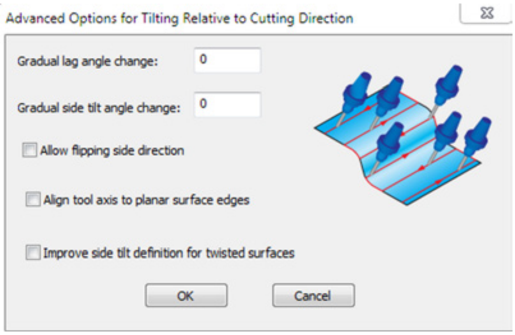
This option enables you to define the direction of the side tilting by a number of lines. The **Tilt lines** section enables you to choose the lines geometry from a list or define a new one with the button, displaying the **Geometry Edit** dialog box. The direction of the side tilting gradually changes passing through the defined tilt lines.



The **Tilting lines maximum snap distance** parameter defines the maximum distance between tilt line end points and the machining contour. When tilting is applied to a contour, only lines within this distance are used, while other lines that are far from the contour are ignored. Note that the tilt lines are snapped to the machining contour via the shortest distance from the line to the contour.

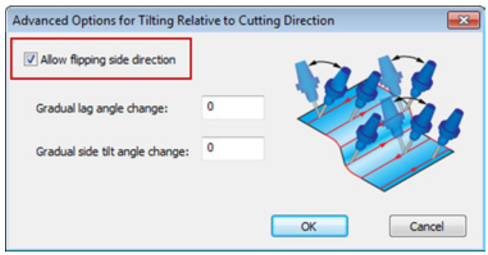
Advanced options for tilting relative to cutting direction

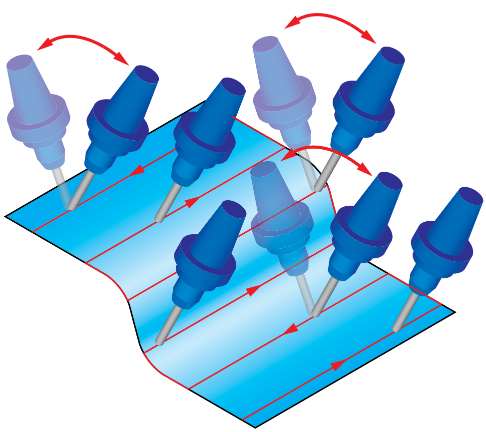
SolidCAM enables you to define a number of advanced parameters for the side tilting options. Click the **Advanced** button to display the relevant **Advanced Options for Tilting Relative to Cutting Direction** dialog box.



- **Allow flipping side direction**

  This option enables you to change the direction of the side tilting for the tool path generated with the **Zigzag** cutting method. When this check box is not selected, the direction of the side tilting is the same for all cutting passes. When this check box is selected, SolidCAM changes the side tilting direction to the opposite when the cutting pass direction is changed.

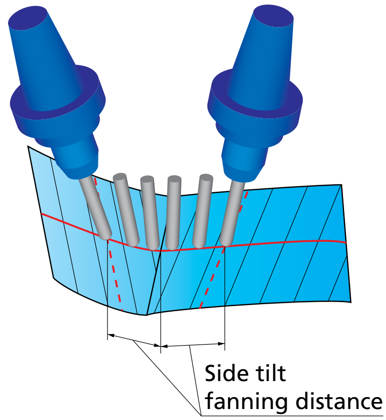




This check box is available when the **Follow surface ISO Lines direction** option is used for **Side tilt definition**.

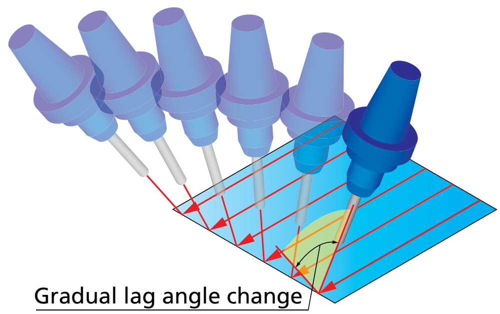
- **Side tilt fanning distance**

  This option is available only when the **Follow surface ISO Lines direction** option is used for the **Side tilt definition**. Using this option, SolidCAM enables you to control the side tilting direction at the intersection of two surfaces with different isometric directions. In such intersection areas, SolidCAM performs smooth transition of the side tool tilting taking into account the different direction of ISO vectors. The **Side tilt fanning distance** parameter defines the distances from the surface intersection where the transition of side tilting directions is started.



- **Gradual lag angle change**

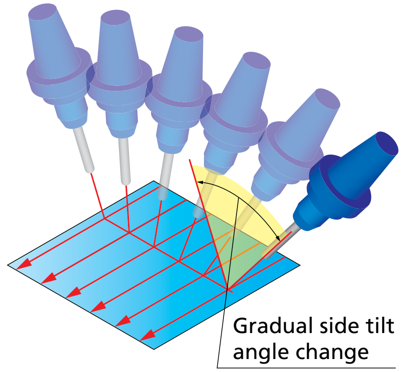
  SolidCAM enables you to change the lag angle gradually along the tool path. The lag angle is changed for each cutting pass; the final change of the lag angle at the end of the tool path is determined by the **Gradual lag angle change** value. For each cutting pass, the increment of the lag angle is equal to the **Gradual lag angle change** value divided by the number of cutting passes.



This option has no effect on the first cutting pass; the tool is tilted only with the lag angle value.

For example, the **Lag angle to cutting direction** parameter is set to 5°. The **Gradual lag angle change** value is set to 10°. In this case, the tool path is started with the lag angle of 5° and finished with the lag angle of 5°+10° = 15°. In the middle of the tool path, the lag angle is 5° + 0.5*10° = 10°.

- **Gradual side tilt angle change**



SolidCAM enables you to change the side tilt angle gradually along the tool path. The side tilt angle is changed for each cutting pass; the final change of the side tilt angle at the end of the tool path is determined by **Gradual side tilt angle change** value. For each cutting pass the increment of the side tilt angle is equal to the **Gradual side tilt angle change** value divided by the number of cutting passes.

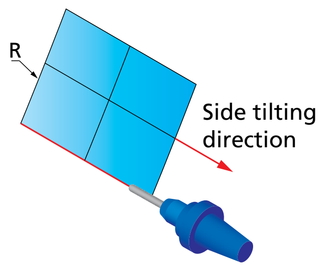
This option has no effect on the first cutting pass; the tool is tilted only with the side tilt angle value.

For example, the **Tilt angle at side of cutting direction** parameter is set to 5°. The **Gradual side tilt angle change** value is set to 10°. In this case, the tool path is started with the side tilt angle of 5° and finished with the side tilt angle of 5°+10° = 15°. In the middle of the tool path, the side tilt angle is 5° + 0.5*10° = 10°.

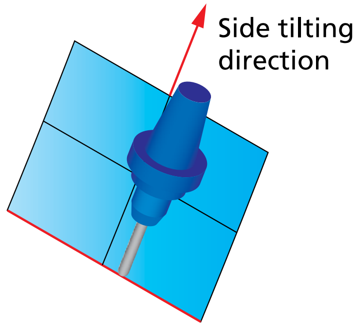
- **Ruled surface radius limit**

SolidCAM enables you to automatically recognize and machine ruled surfaces (a **Ruled surface** is a surface that can be swept out by moving a line in space) by the Swarf machining technology. With this technology, the machining is performed by the tool side that has a linear contact with the machined ruled surface. When the **Follow surface ISO Lines direction** option is used, SolidCAM automatically chooses the direction of straight lines (rulings) of the ruled surface as the direction of the side tilting.

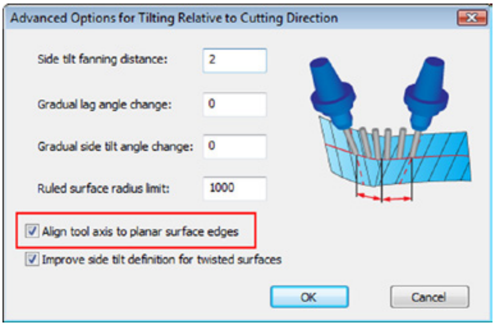
Sometimes the surfaces seem to be planar but actually have a curvature of large radius in one direction. SolidCAM considers these surfaces as ruled and uses rulings as the direction of side tilting causing wrong tool orientation. On the illustration, the horizontal isometric direction is defined by a straight line; the vertical isometric direction also seems to be straight but actually has a large radius of curvature. The surface is considered as ruled. The tool side tilting direction is chosen according to the direction of the horizontal ISO line, resulting in a wrong tool orientation (side tilting angle is 90°).



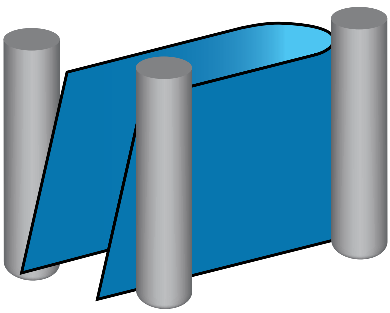
The **Ruled surface radius limit** parameter enables you to limit the maximum radius of curvature of ISO lines for a surface to be considered as ruled. All the curved ISO lines with a radius greater than the specified value are considered as straight lines. The face is not considered as ruled and machining is performed with a proper side tilting.



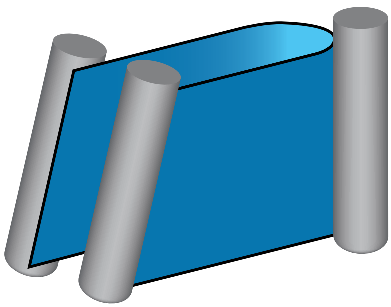
Align tool axis to planar surface edges



Although the tool axis will be aligned to the isometric direction, the orientation at the start and end of the blade will be aligned to its surface edge, even if this edge does not follow the isometric direction. This option can be used for impeller blade machining.

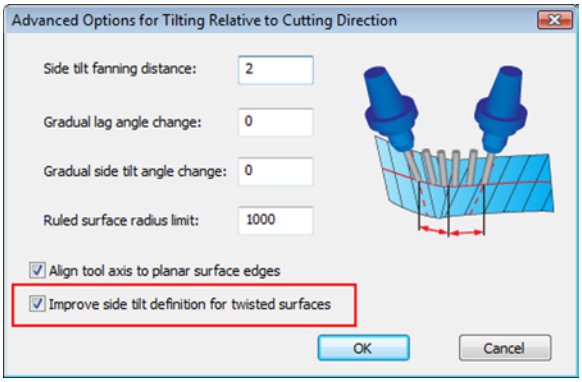


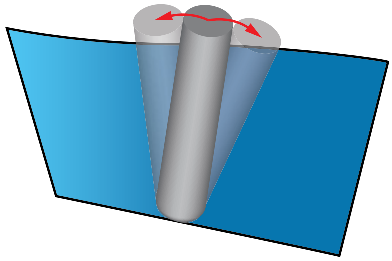
Align tool axis to planar surface edges check box is not selected



Align tool axis to planar surface edges check box is selected

Improve side tilt definition for twisted surfaces

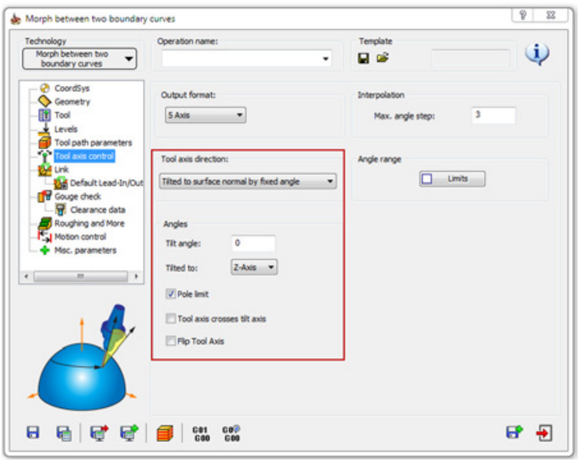




This option enables you to optimally adjust the tool tilting for swarf machining of twisted ruled surfaces. The idea is to get a contact line between the tool and the surface, which is nearly impossible when the surface is twisted.

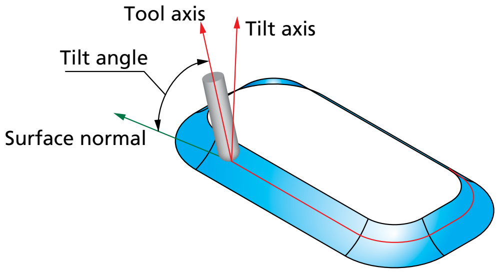
When the **Improve side tilt definition for twisted surfaces** check box is selected, the system determines an optimal surface alignment for the swarfed tool path to obtain better swarfing and better line contact between the tool and the surface. This check box is available for all the options used for **Side tilt definition**.

7.3.3 Tilted to surface normal by fixed angle



With this option, the tool is tilted in the tilting plane defined by a surface normal at the contact point and the specified tilt axis. The tilting is performed relative to the surface normal.

Angles



- **Tilt angle**

  This parameter defines the angle of the tool tilting from the surface normal in the tilting plane.

- **Tilted to**

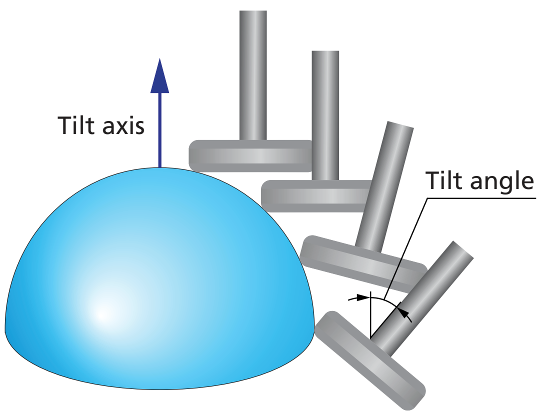
  This option enables you to define the direction of the tilt axis. SolidCAM enables you to choose one of the Coordinate System axes (X, Y or Z) or define the tilt axis by a line. When the **Line** option is chosen, click the **Data** button to pick the start and end points of the tilt axis line directly on the solid model.

Pole limit

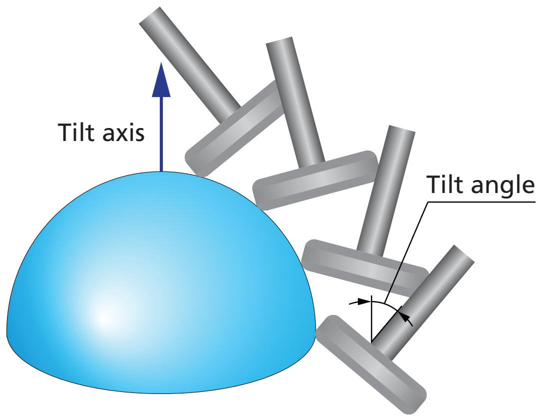
This option enables you to limit the tool tilting, occurring beyond the selected tilt axis to avoid pole problems that can cause heavy table rotations on the machine.

When this check box is selected, the tool will not be tilted beyond the selected tilt axis in any tool path point; it will be tilted only until the point when the tool axis is parallel to the defined tilt axis.

When this check box is not selected, the tool will overtilt the defined tilt axis.



Pole limit check box is selected

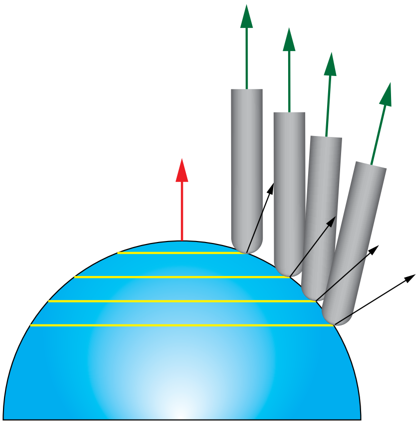


Pole limit check box is not selected

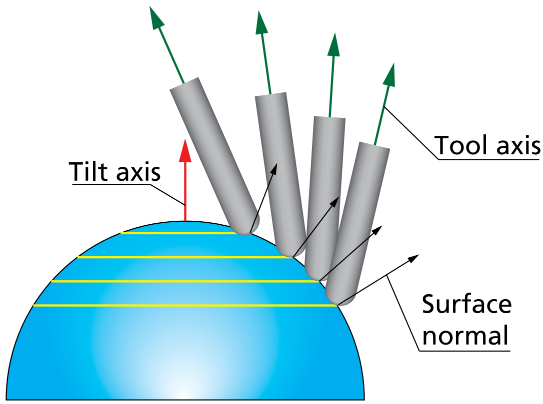
The **Pole limit** check box is available only for the **Tilted to surface normal by fixed angle** tool tilting strategy.

Tool axis crosses tilt axis

When this check box is not selected, SolidCAM checks the tool axis and tilt axis for intersections. In case of intersection, the tool axis is changed to be coincident with the tilt axis. In such manner, the tool axis cannot intersect with the tilt axis. When this check box is selected, SolidCAM does not check for the intersections; the tool axis can therefore cross the tilt axis.



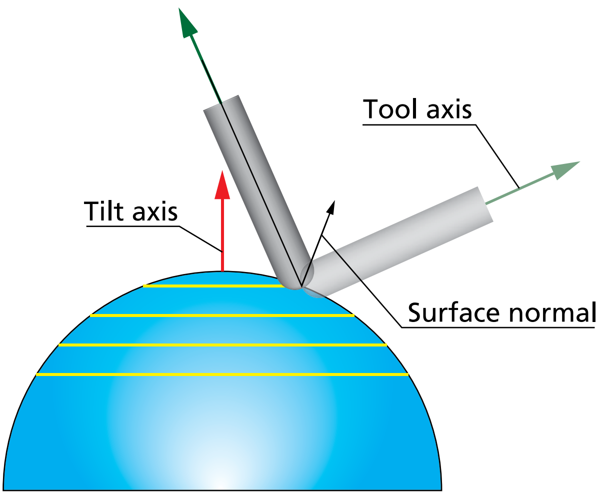
Tool axis crosses tilt axis option off



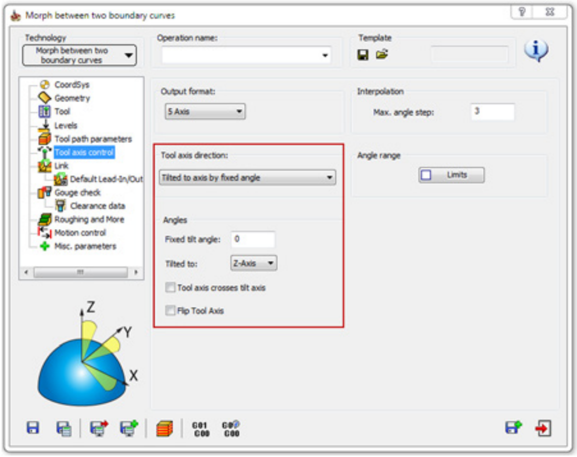
Tool axis crosses tilt axis option on

Flip Tool Axis

This option enables you to reverse the tool axis direction in the tilting plane relative to the surface normal.

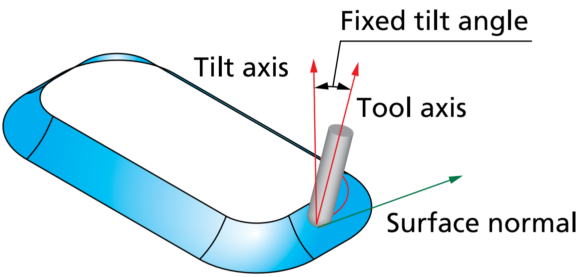


7.3.4 Tilted to axis by fixed angle



With this option, the tool is tilted in the tilting plane defined by the specified tilt axis and the surface normal at the contact point. The tilting is performed from the tilt axis.

Angles



- **Fixed tilt angle**

  This parameter defines the angle of the tool tilting from the tilt axis in the tilting plane.

- **Tilted to**

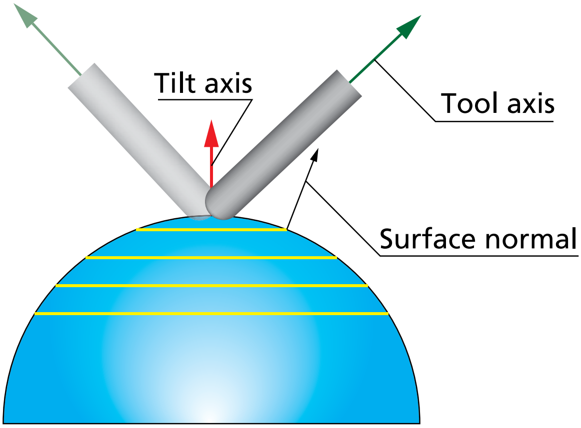
  This option enables you to define the direction of the tilt axis. SolidCAM enables you to choose one of the Coordinate System axes (X, Y or Z) or define the tilt axis by a line. When the **Line** option is chosen, click the **Data** button to pick the start and end points of the tilt axis line directly on the solid model.

Tool axis crosses tilt axis

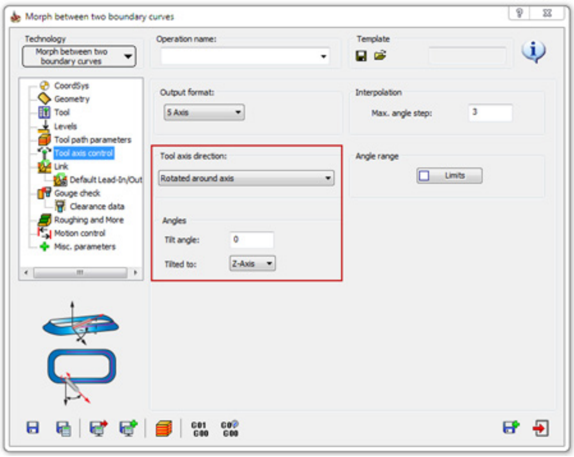
When this check box is not selected, SolidCAM checks the tool axis and tilt axis for intersections. In case of intersection, the tool axis is changed to be coincident with the tilt axis. In such manner, the tool axis cannot intersect with the tilt axis. When this check box is selected, SolidCAM does not check for the intersections; the tool axis can therefore cross the tilt axis.

Flip tool axis

This option enables you to reverse the tool axis direction in tilting plane relative to the surface normal.



7.3.5 Rotated around axis

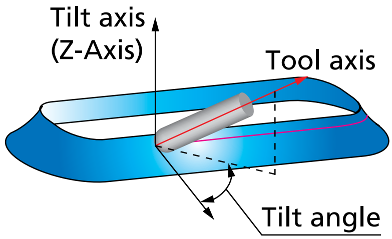


When this option is used, the tool is tilted from the surface normal direction around the chosen tilt axis.

Angles

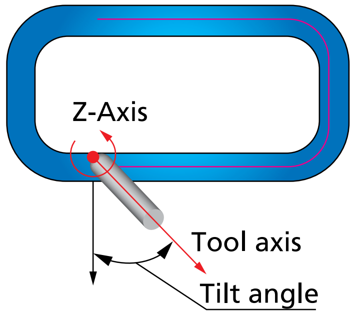
- **Tilt angle**

  This parameter defines the angle of the tool tilting from the surface normal around the tilt axis.

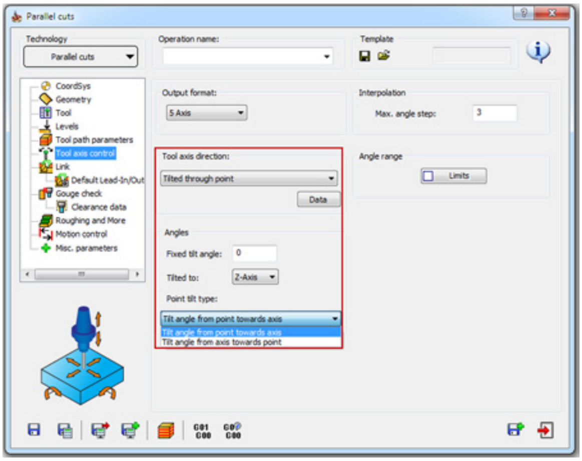


- **Tilted to**

  This option enables you to define the tilt axis. SolidCAM enables you to choose one of the Coordinate System axes (X, Y or Z) or define the tilt axis by a line. When the **Line** option is chosen, click the **Data** button to pick the start and end points of the tilt axis line directly on the solid model.

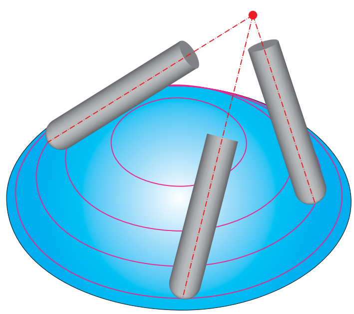


7.3.6 Tilted through point



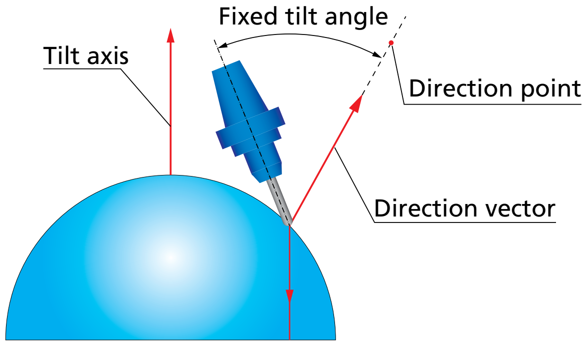
With this option, the tool is tilted through the specified point. The tool axis direction is defined by a vector from the contact point on the drive surface to the specified point.

Click the **Data** button to define the direction point on the solid model.



Angles

SolidCAM enables you to define an additional tilting from the calculated vector through the specified point towards the chosen tilt axis.



- **Fixed tilt angle**

  This parameter defines the angle of the additional tool tilting from the vector through the defined point towards the tilt axis.

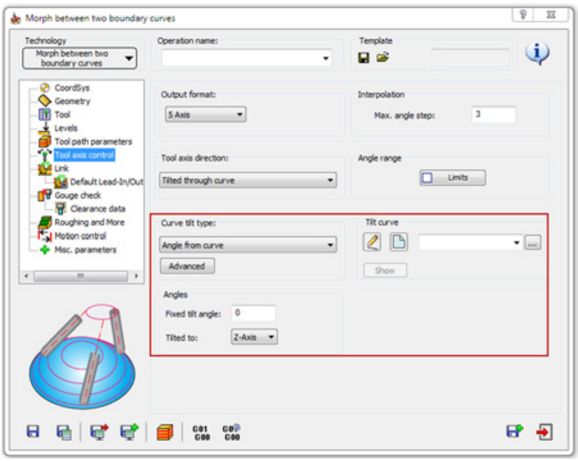
- **Tilted to**

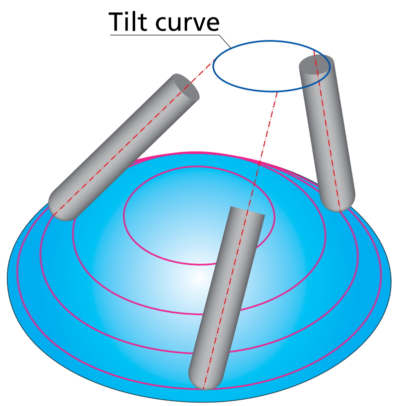
  This option enables you to define the tilt axis. SolidCAM enables you to choose one of the Coordinate System axes (X, Y or Z) or define the tilt axis by a line. When the **Line** option is chosen, click the **Data** button to pick the start and end points of the tilt axis line directly on the solid model.

- **Point tilt type**

  This option enables you to determine the direction from where the fixed tilt angle is measured: from the axis towards the point or from the point towards the axis.

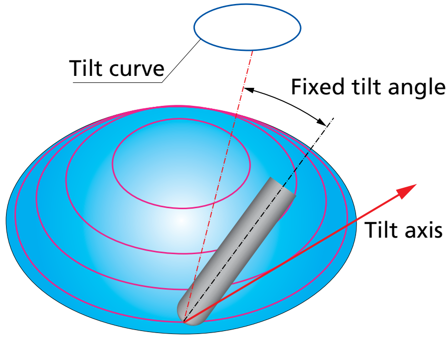
7.3.7 Tilted through curve





With this option, the tool axis intersects with the specified **Tilt curve**. The **Tilt Curve** section enables you to choose an existing profile geometry for tilt curve or define a new one with the button displaying the **Geometry Edit** dialog box.

Angles



SolidCAM enables you to define an additional tilting from the calculated vector through the specified tilt curve towards the chosen tilt axis.

- **Fixed tilt angle**

  This parameter defines the angle of the additional tool tilting from the vector through the defined curve towards the tilt axis.

- **Tilted to**

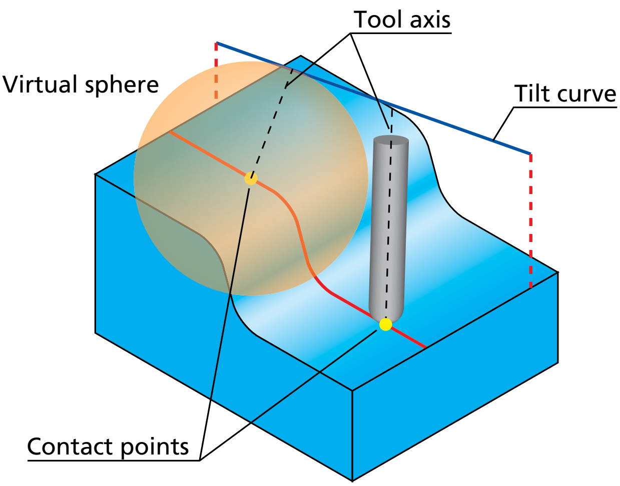
  This option enables you to define the direction of the tilt axis. SolidCAM enables you to choose one of the Coordinate System axes (X, Y or Z) or define the tilt axis by a line. When the **Line** option is chosen, click the **Data** button to pick the start and end points of the tilt axis line directly on the solid model.

Curve tilt type

The **Curve tilt type** options enable you to define how the end point of the tool axis vector is found on the **Tilt curve**. In other words, these options define how the tool axis is aligned relative to the **Tilt curve**.

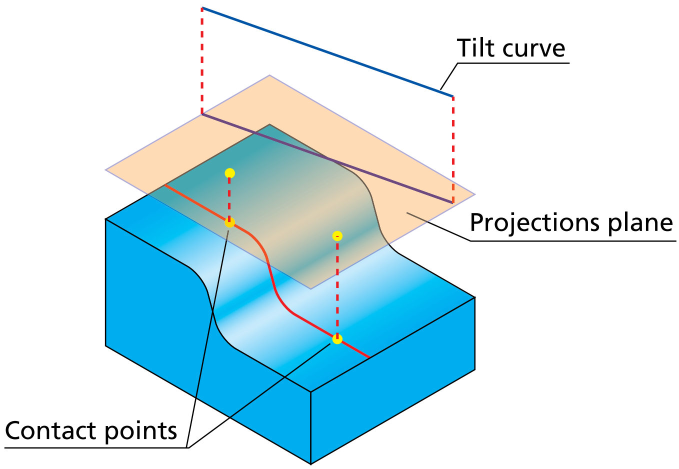
- **Closest point**

  The tool axis is defined by the direction vector that connects the contact point and the **Tilt curve** by the shortest distance. With this method, SolidCAM expands a virtual sphere from the contact point. The first intersection point between the sphere and the **Tilt curve** defines the shortest distance.

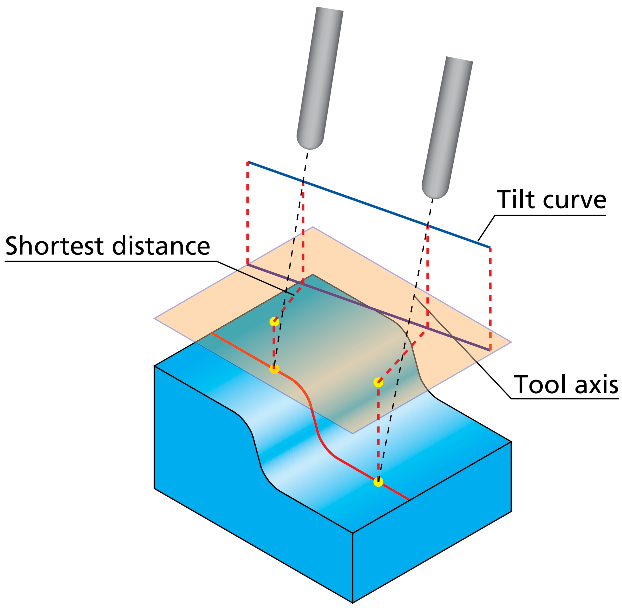


- **Angle from curve**

The direction of the tool axis is determined using the projections of contact points and the **Tilt curve** on a plane orthogonal to the main spindle direction. On the illustration, the tool contacts points and the **Tilt curve** are projected onto a plane parallel to XY-plane (the main spindle direction is the Z-axis).

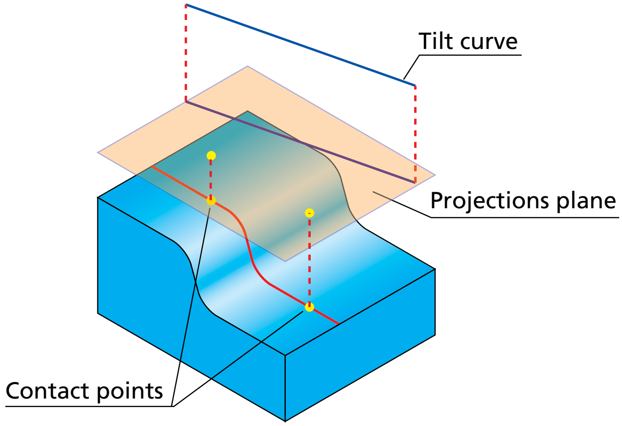


The projected contact points and the projection of the **Tilt curve** are connected by the shortest distance. The connecting points found on the projection are projected back onto the **Tilt curve** determining the direction of the tool axis.

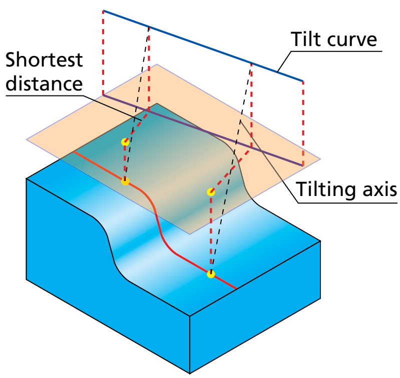


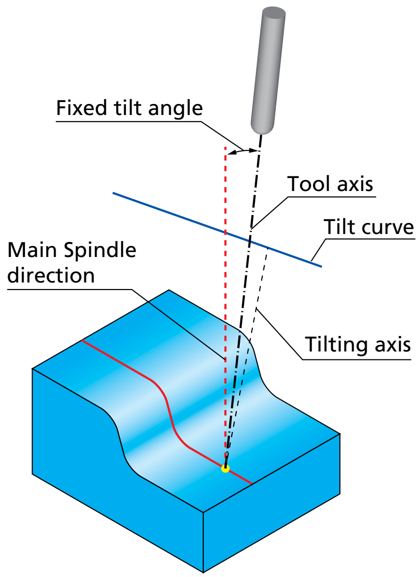
- **Angle from spindle, main direction**

  The tilting is performed from the main spindle axis at the specified **Fixed tilt angle** value towards the tilting axis. The orientation of the tilting axis is determined using the projections of contact points and the **Tilt curve** on a plane orthogonal to the main spindle direction (see the **Angle from curve** option). On the illustration, the tool contact points and the **Tilt curve** are projected onto a plane parallel to XY-plane (the main spindle direction is the Z-axis).



  The projected contact points and the projection of the **Tilt curve** are connected by the shortest distance. The connecting points found on the projection are projected back onto the **Tilt curve** determining the direction of the tilting axis.

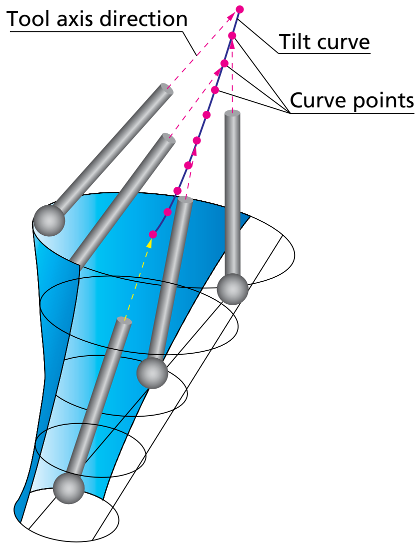




  The tool tilting is performed from the main spindle direction towards the defined tilting axis at the specified **Fixed tilt angle** value.

- **From start to end**

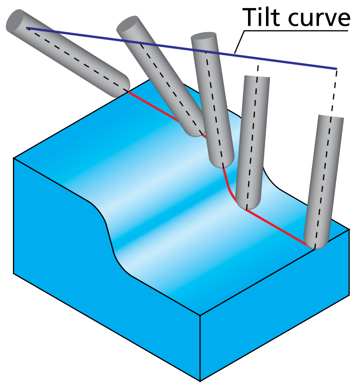
The **Tilt curve** is divided by a number of points equal to the number of cutting passes. The tool axis for each cutting pass is directed to the corresponding point on the **Tilt curve**.



Generally, the **Curve tilt type** option is used for tube milling, engine inlets machining and so forth. Usually, the tube milling is performed with constant-Z cuts. The number of the constant-Z cuts depends on the **Max. Step over** value. The **Tilt curve** is divided with a number of points equal to the number of constant-Z cuts. The tool axis for each such cut is directed to the corresponding point on the **Tilt curve**.

- **From start to end for each contour**

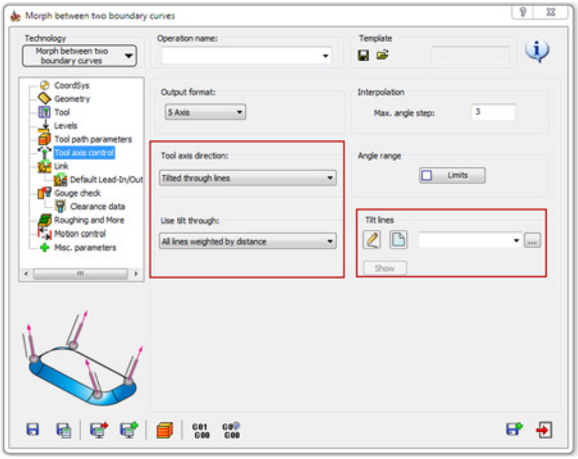
The **Tilt curve** is divided by a number of points equal to the number of tool path positions of the specific cutting pass. In certain tool path positions, the tool is tilted to the corresponding point on the **Tilt curve**. The tool tilting changes gradually from the start point of the **Tilt curve** to its end point. At the start point of a cutting pass, the tool axis is tilted to the start point of the **Tilt curve**. In the midpoint of the cutting pass, the tool axis is tilted to the midpoint of the **Tilt curve**. Accordingly, at the end of the cutting pass, the tool axis is tilted to the end point of the **Tilt curve**.



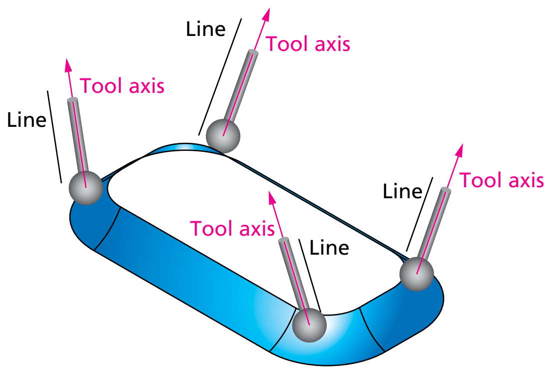
- **Automatic curve**

This is the only strategy where the curve is calculated automatically by the system for each contour and you do not have to input any tilting curve geometry. The automatically generated curve tries to dampen the tool motion by a user-defined dampening distance.

7.3.8 Tilted through lines



This option enables you to define the tool tilting by a number of lines. The tool axis is changed gradually along the tool path trying to pass through the defined lines.

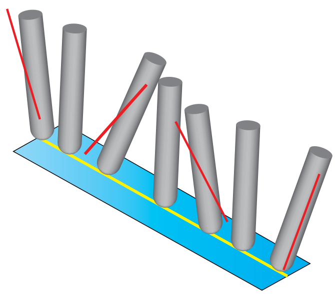


The **Tilt lines** section enables you to choose the lines geometry from the list or define a new one with the **Define** button, using the **Geometry Edit** dialog box. The direction of the side tilting gradually changes, passing the defined tilt lines.

The **Use tilt through** option enables you to define how the tool axis is approximated between the defined tilt lines. It has two choices:

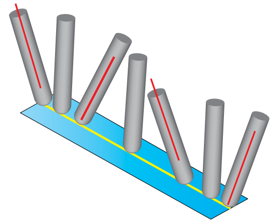
- **All lines weighted by distance**

  The direction of the tool axis is approximated between the tilt lines located close to the tool path, in order to perform smooth transition of the tilting between the tool path positions. In some cases the tilting of the approximated tool path does not coincide with the direction of tilt lines.

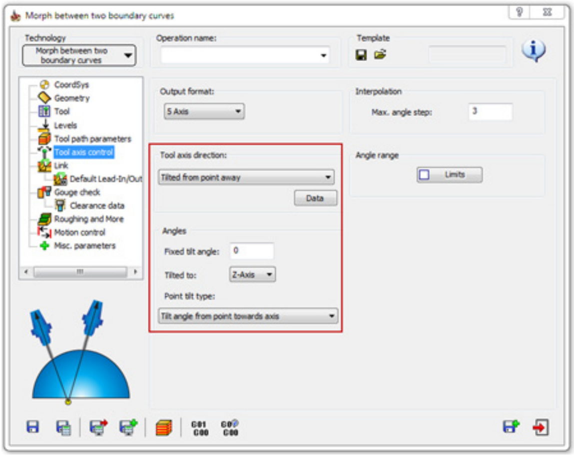


- **Always use closest two lines**

  The direction of the tool axis is approximated between two consecutive tilt lines along the tool path. In the resulting tool path, the tool axis coincides with the tilt lines at exact positions and smooth transition is performed between these positions. The maximum snap distance between tilt lines could also be defined in the **Tilting lines range** section.

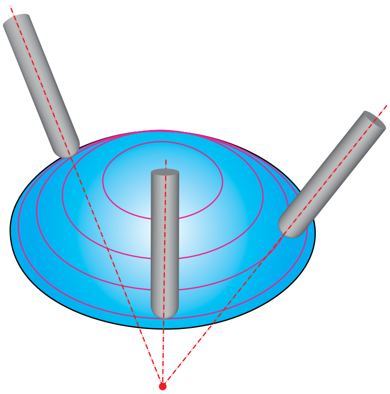


7.3.9 Tilted from point away



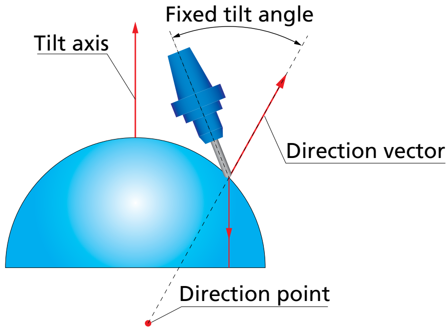
With this option, the tool axis is directed away from a specified point. This option is similar to the **Tilted through point** option. The tool axis direction is defined by a vector from the specified point to the tool contact point on the drive surface.

Click the **Data** button to define the direction point on the solid model.



The specified point has to be located under the drive surface.

Angles



SolidCAM enables you to define an additional tilting from the calculated vector through the specified point towards the chosen tilt axis.

- **Fixed tilt angle**

  This parameter defines the angle of the additional tool tilting from the vector through the defined point towards the tilt axis.

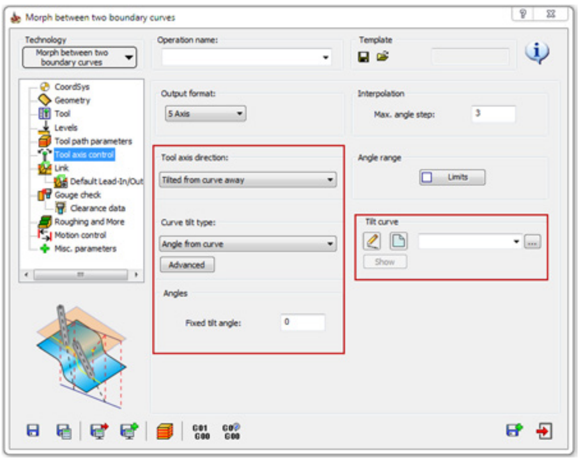
- **Tilted to**

  This option enables you to define the tilt axis. SolidCAM enables you to choose one of the Coordinate System axes (X, Y or Z) or define the tilt axis by a line. When the **Line** option is chosen, click the **Data** button to pick the start and end points of the tilt axis line directly on the solid model.

- **Point tilt type**

  This option enables you to determine the direction from where the fixed tilt angle is measured: from the axis towards the point or from the point towards the axis.

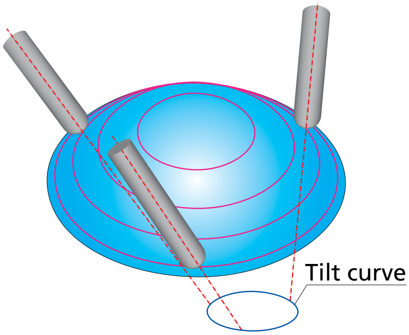
7.3.10 Tilted from curve away



With this option, the tool axis intersects with the specified **Tilt curve**, similar to the **Tilted through curve** option. The tool axis coincides with the vector directed from the point on the **Tilt curve** to the tool contact point.

The tilt curve has to be located under the drive surface.

The **Tilt Curve** section enables you to choose the existing profile geometry for the tilt curve or define a new one with the button using the **Geometry Edit** dialog box.

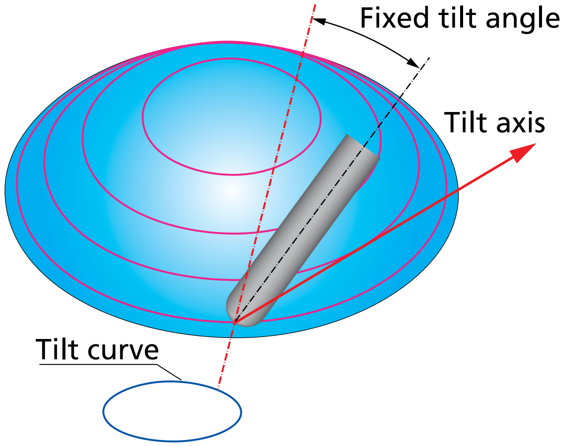


Angles

You can define an additional tilting from the calculated vector through the specified tilt curve towards the chosen tilt axis.

- **Fixed tilt angle**

  This parameter defines the angle of the additional tool tilting from the vector through the defined curve towards the tilt axis.

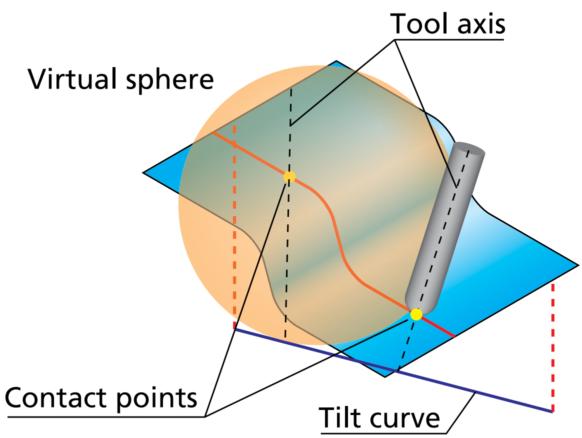


Curve tilt type

The **Curve tilt type** options enable you to define the way how the start point of the tool axis vector is found on the **Tilt curve**. In other words, these options define how the tool axis is aligned relative to the **Tilt curve**.

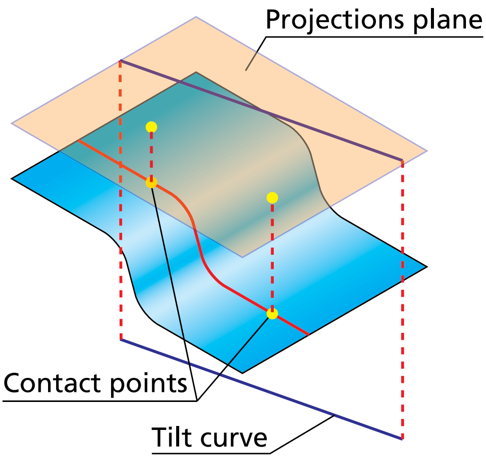
- **Closest point**

  The tool axis is defined by direction vector connecting the contact point and the **Tilt curve** by the shortest distance. With this method, SolidCAM expands a virtual sphere from the contact point. The first intersection point between the sphere and the **Tilt curve** defines the shortest distance.

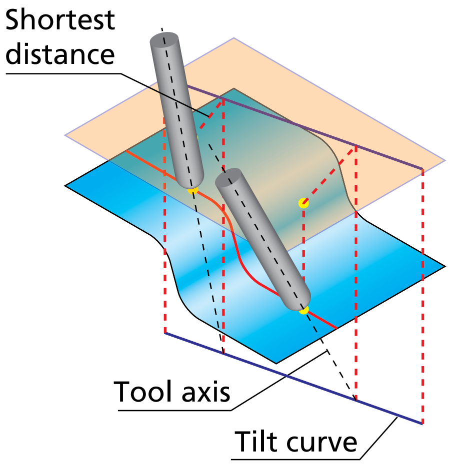


- **Angle from curve**

The direction of the tool axis is determined using the projections of contact points and the **Tilt curve** on a plane orthogonal to the main spindle direction. On the illustration, the tool contact points and the **Tilt curve** are projected onto a plane parallel to XY-plane (the main spindle direction is the Z-axis).

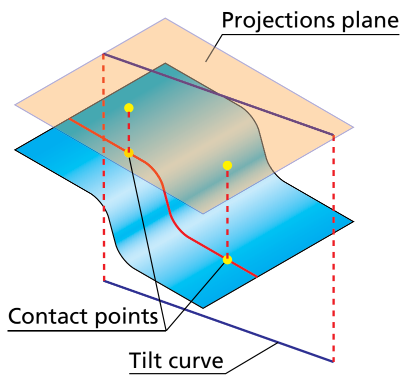


The projected contact points and the projection of the **Tilt curve** are connected by the shortest distance. The connecting points found on the projection are projected back onto the **Tilt curve** determining the direction of the tool axis.

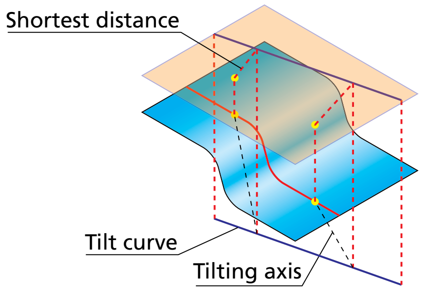


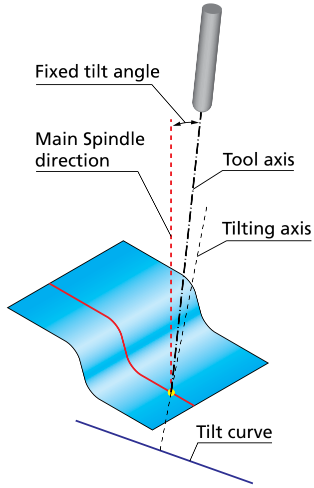
- **Angle from spindle, main direction**

The tilting is performed from the main spindle axis at the specified **Fixed tilt angle** value towards the tilting axis. The orientation of tilting axis is determined using the projections of contact points and the **Tilt curve** on a plane orthogonal to the main spindle direction (see the **Angle from curve** option). On the illustration, the tool contact points and the **Tilt curve** are projected onto a plane parallel to XY-plane (the main spindle direction is the Z-axis).



The projected contact points and the projection of the **Tilt curve** are connected by the shortest distance. The connecting points found on the projection are projected back onto the **Tilt curve** determining the direction of the tilting axis.

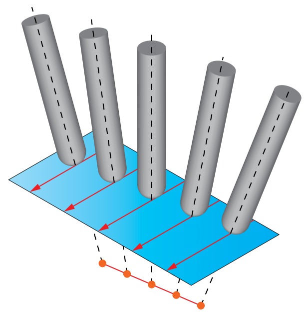




The tool tilting is performed from the main spindle direction towards the defined tilting axis at the specified **Fixed tilt angle** value.

- **From start to end**

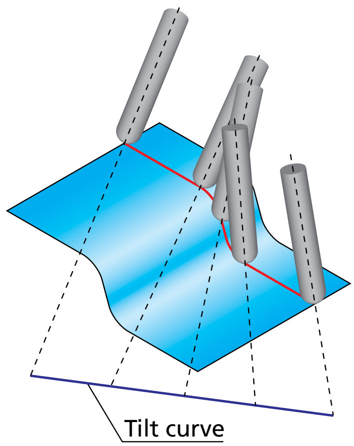
The **Tilt curve** is divided by a number of points equal to the number of cutting passes. The tool axis for each cutting pass is directed from the corresponding point on the **Tilt curve**.



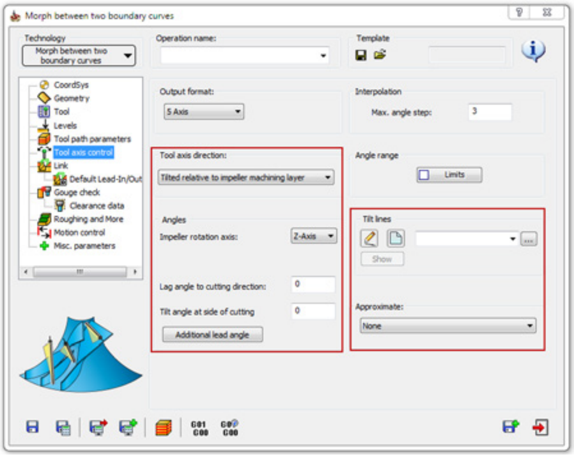
The **Tilt curve** is divided with a number of points equal to the number of cutting passes. The tool axis for each such cutting pass is directed from the corresponding point on the **Tilt curve**.

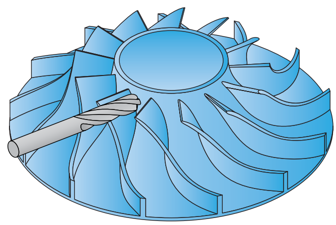
- **From start to end for each contour**

The **Tilt curve** is divided by a number of points equal to the number of tool path positions of the specific cutting pass. In certain tool path positions, the tool is tilted according to the vector from the corresponding point on the **Tilt curve**. The tool tilting changes gradually from the start point of the **Tilt curve** to its end point. At the start point of a cutting pass, the tool axis is tilted from the start point of the **Tilt curve**. At the midpoint of the cutting pass, the tool axis is tilted from the midpoint of the **Tilt curve**. Accordingly, at the end of the cutting pass, the tool axis is tilted from the end point of the **Tilt curve**.



7.3.11 Tilted relative to impeller machining layer

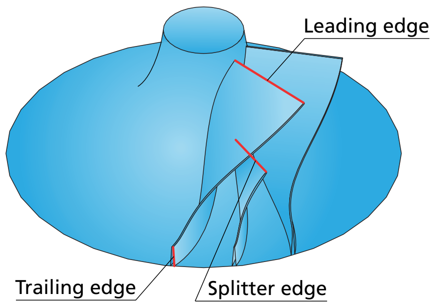




This tilting strategy is used for machining of impeller blades. Generally, the tool axis in this strategy is normal to the floor face of the impeller. The lag and side tilting can be adjusted by defining the corresponding parameters in the **Angles** section.

Such general tool tilting definition might cause gouges in certain types of impellers, especially in those with splitter blades, therefore additional local tilting definition might be required. Local tilting is defined by specifying the tilt lines and additional lead angles at the leading edge, splitter edge and trailing edge.

Tilt lines



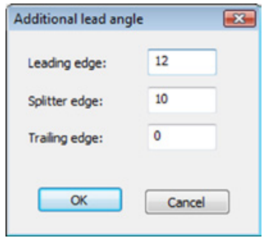
These lines are defined to apply additional tilting at the leading edge, splitter edge and trailing edge of the impeller blades. These lines must be located and oriented along these edges.

Angles

The **Lag angle to cutting direction** value enables you to define the tool tilting in the direction of the cutting pass. The **Lag angle to cutting direction** parameter is measured relative to surface normal.

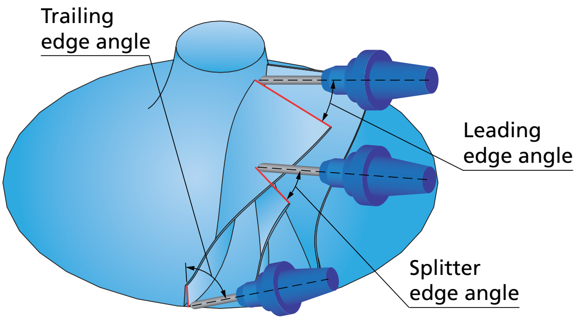
The **Tilt angle at side of cutting direction** value enables you to define the tool inclination in the direction determined by **Side tilting** options. The **Tilt angle at side of cutting direction** parameter is measured relative to surface normal.

Additional lead angle

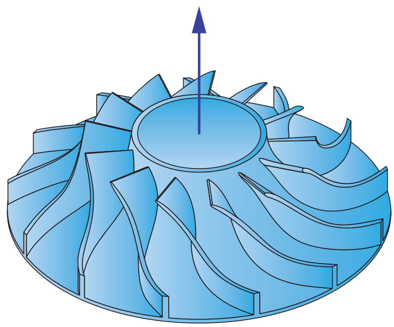


This button displays the **Additional lead angle** dialog box that enables you to define the additional lead angle values at the leading edge, splitter edge and trailing edge.

- **Leading edge**: this parameter defines the value of an additional tilt angle measured from the leading edge line.
- **Splitter edge**: this parameter defines the value of an additional tilt angle measured from the splitter edge line.
- **Trailing edge**: this parameter defines the value of an additional tilt angle measured from the trailing edge line.



Impeller rotation axis



This parameter enables you to define the rotation axis of the machined impeller part. The rotation axis can be represented by one of the Coordinate System axes (usually the Z-axis) or an arbitrary line defined by picking points on the solid model.

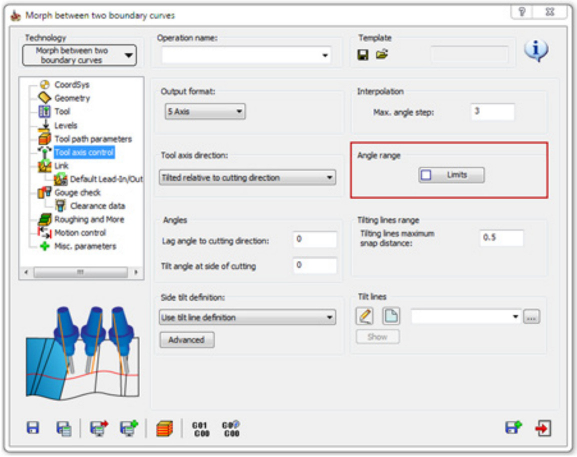
Approximate

This parameter enables you to calculate the direction vectors for tool path tilting by approximation.

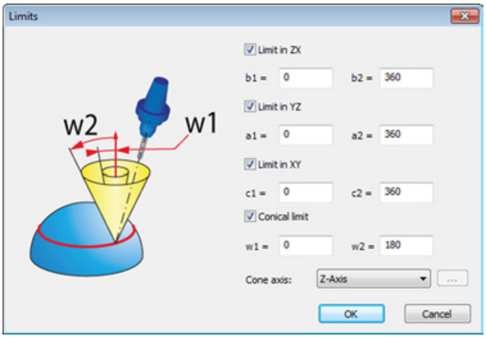
For more information, see topic **7.3.2**.

7.4 Angle range

SolidCAM enables you to limit the tool tilting along the tool path. Click the **Limits** button to define the angle range parameters.

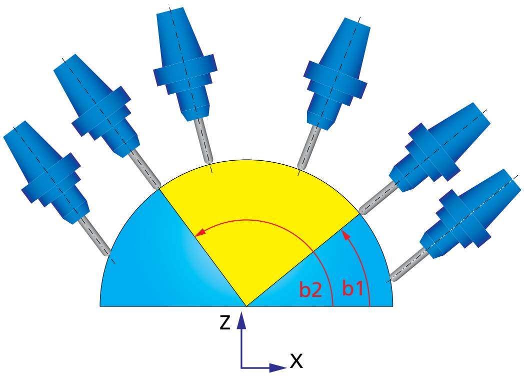


The **Limits** dialog box is displayed.



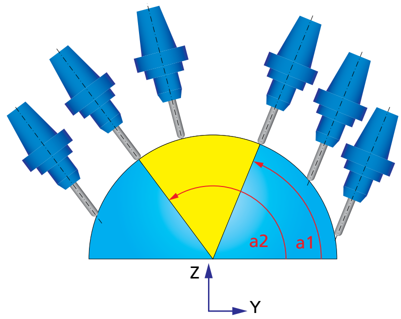
- **Limit in ZX**

  This option enables you to limit the tool tilting by projecting the tool axis on the ZX-plane of the current Coordinate System. The **b1** and **b2** parameters define the start and end angle of the limit.



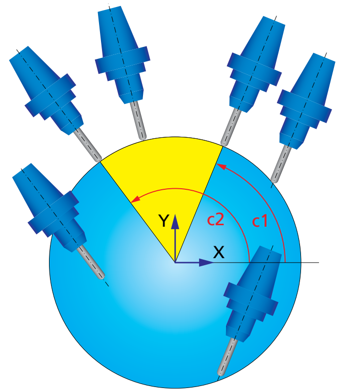
- **Limit in YZ**

This option enables you to limit the tool tilting by projecting the tool axis on the YZ-plane of the current Coordinate System. The **a1** and **a2** parameters define the start and end angle of the limit.



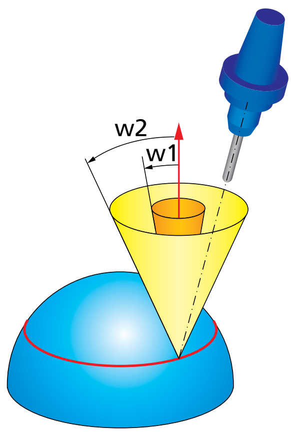
- **Limit in XY**

This option enables you to limit the tool tilting by projecting the tool axis on the XY-plane of the current Coordinate System. The **c1** and **c2** parameters define the start and end angle of the limit.



- **Conical limit**

SolidCAM enables you to limit the tool tilting between two angles starting from the normal vector to the tool path slice at the contact point. In other words, imagine two cones with different opening angles **w1** and **w2**; the cone vertex is located at the contact point. The tool axis direction is limited between these cones. The orientation of the cones depends on the **Cone axis** settings.



The **Cone axis** option enables you to define the direction of the limiting cones axis. You can choose either axis of the Coordinate System (X, Y or Z) or define the direction by a user-defined vector. If the tilting along the tool path is defined by a leading curve (e.g. the **Cuts along curve** option is used), you can choose the **Dynamically using leading curve** for the **Cone axis**. In this case, the tool tilting limiting cones are defined relative to a leading curve.

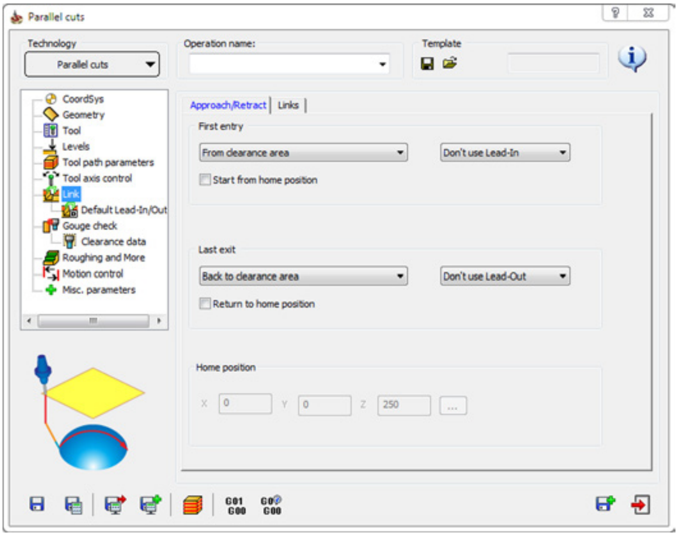
Link

8

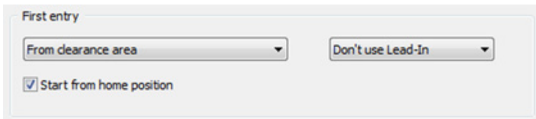
The **Link** page enables you to define the approach/retract of the tool and linking of Sim. 5-Axis passes into the complete tool path.

8.1 Approach/Retract

This tab enables you to define the parameters of tool approach and retract performed in the Sim. 5-Axis operation.



8.1.1 First entry

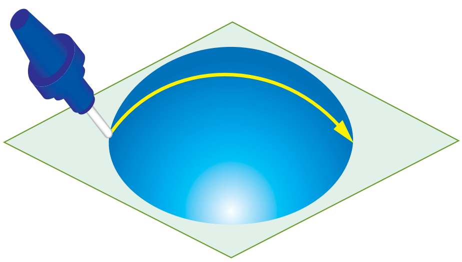


This section enables you to define the first approach of the tool to the cutting area.

SolidCAM enables you to specify the level from which the approach movement is started. The following options are available:

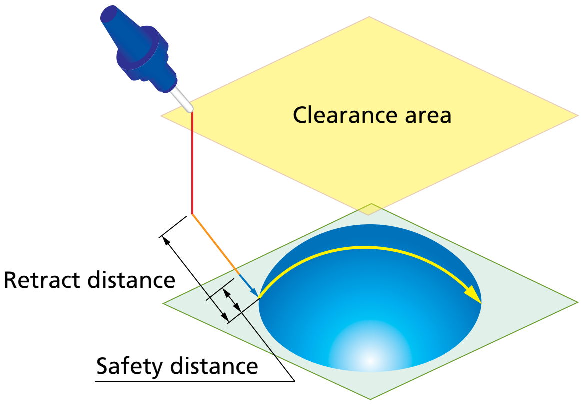
- **Direct**

  When this option is chosen, the machining is started directly in the first point of the tool path. No approach movement is performed.



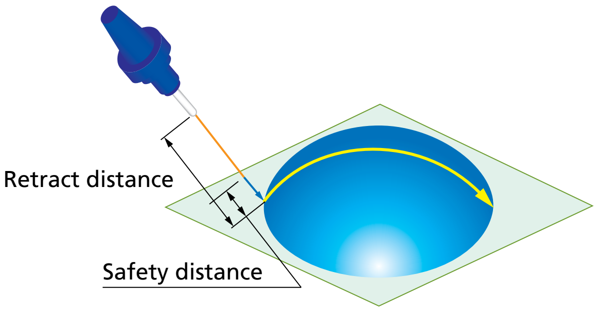
- **From clearance area**

  The approach movement is performed from the specified **Clearance area** (see topic **5.1**), through the **Retract distance** and **Safety distance** levels.



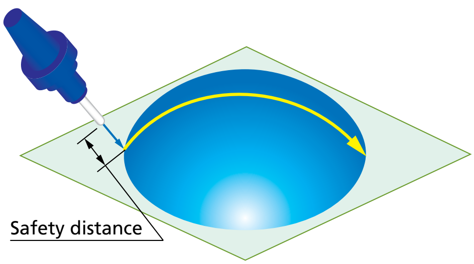
- **Use retract distance**

  The approach movement is performed from the **Retract distance** level.



- **Use safety distance**

  The approach movement is performed from the **Safety distance** level.



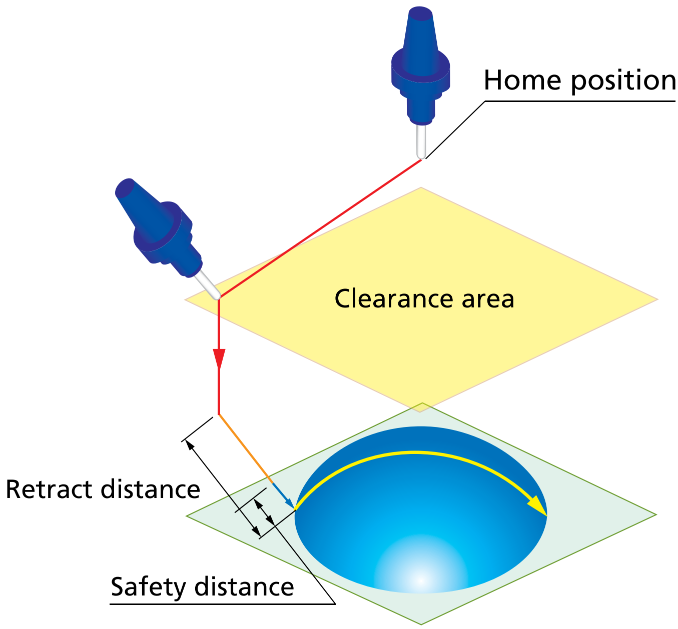
SolidCAM enables you to use the defined **Lead-In** options (see topic **8.3**) for the first entry definition.

Start from home position

SolidCAM enables you to define the **Home position** for tool path linking that can be applied to the first entry links. **Home position** is a point from which the first rapid movement of the tool starts during the approach.

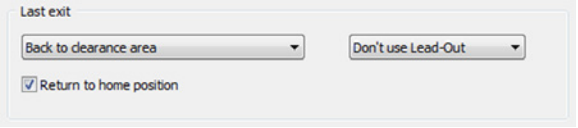
When the **Start from home position** check box is selected, the machining is performed as follows:

The tool is positioned at the specified **Home position**, with the tool axis parallel to the Z-axis of the Coordinate System. It then performs its initial rapid movement to the **Clearance area**/**Retract distance**/**Safety distance** level or to the start point of the first cutting pass (depending on the **First entry** setting), where it gets tilted according to the defined **Tool axis control** parameters. From that point it performs the approach movement to the drive surface (or directly starts machining the surface in case of the **Direct** option chosen for **First entry**).



8.1.2 Last exit

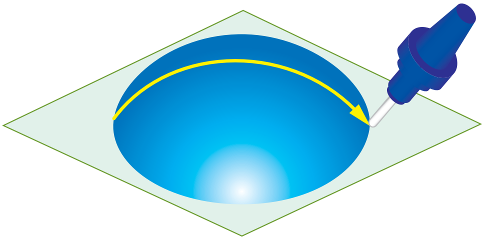
This section enables you to define the last retreat of the tool from the cutting area after the machining.



SolidCAM enables you to specify the level to which the retreat movement is performed. The following options are available:

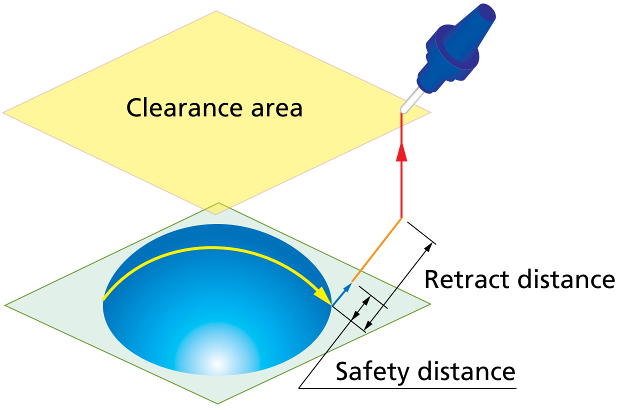
- **Direct**

  When this option is chosen, the machining is finished directly in the last point of the tool path. No retreat movement is performed.



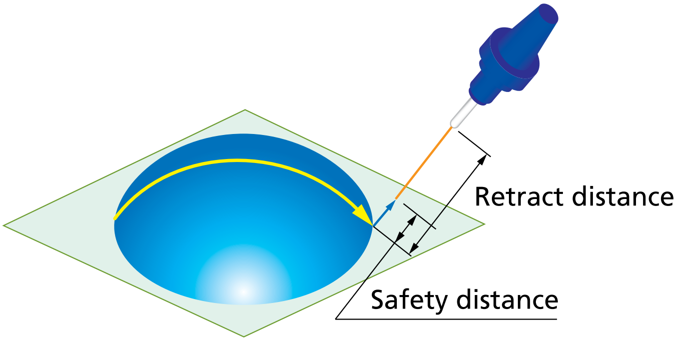
- **Back to clearance area**

  The retreat movement is performed back to the specified **Clearance area** (see topic **5.1**), through the **Retract distance** and **Safety distance** levels.



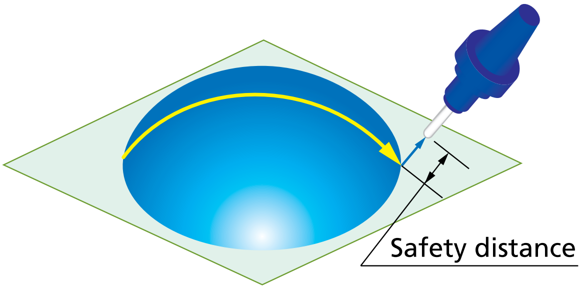
- **Use retract distance**

  The retreat movement is performed to the **Retract distance** level.



- **Use safety distance**

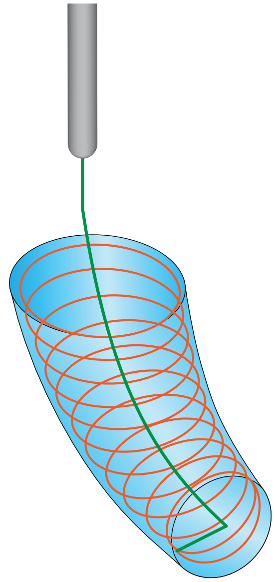
  The retreat movement is performed to the **Safety distance** level.



- **Back to clearance through tube center**

  This option is useful in case of port and tubular parts machining. The idea is to enable the tool to exit from the machined part through the tube center outwards.

  When this option is chosen, the tool retreats from the interior of the machined tubular part through its center up to the clearance area level.

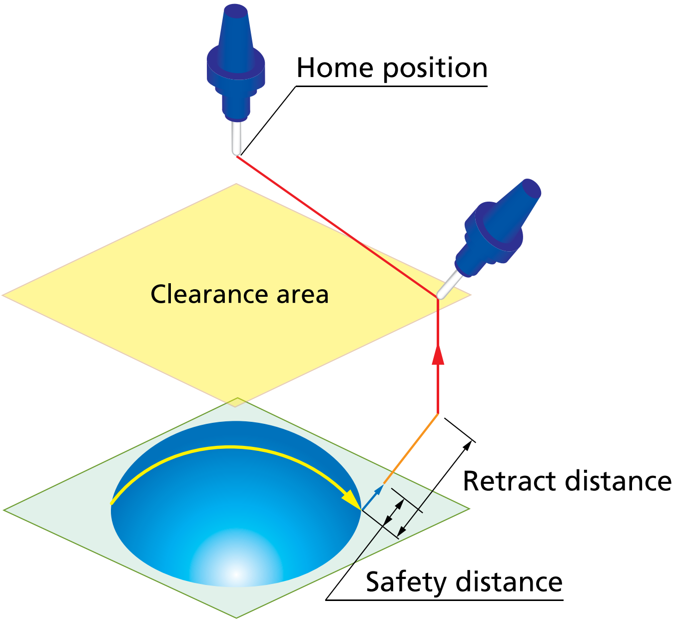


SolidCAM enables you to use the defined **Lead-Out** options (see topic **8.3**) for the last exit definition.

Return to home position

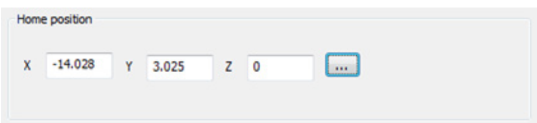
SolidCAM enables you to define the **Home position** for tool path linking that can be applied to the last exit links. **Home position** is a point to which the tool eventually returns after the retreat.

When the **Return to home position** check box is selected, the machining is performed as follows: after the last cutting pass, the tool returns to the **Clearance area**/**Retract distance**/**Safety distance** level (depending on the **Last exit** setting) or directly to the **Home position** (in case of the **Direct** option chosen for **Last exit**).



8.1.3 Home position

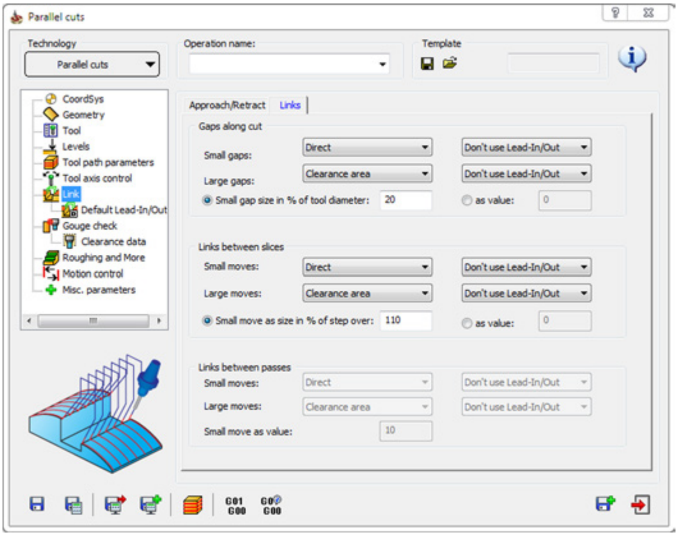
This section enables you to define the coordinates of the home position.



This section is available when the **Start from home position** and/or the **Return to home position** check box is selected.

8.2 Links

This tab enables you to define the parameters of tool path linking performed in the Sim. 5-Axis operation.



8.2.1 Gaps along cut

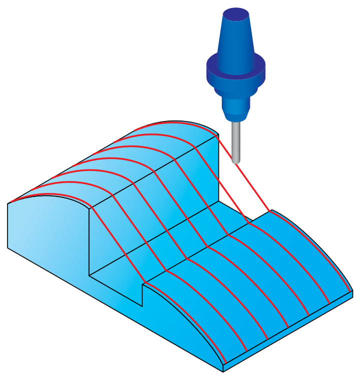
During the tool path linking, SolidCAM detects gaps along the cutting passes. The **Gaps along cut** section enables you to define how the tool moves in such gap areas.

SolidCAM enables you to define different ways of movements in gap areas of different sizes. Depending on the size of the gap, it is possible to choose two different options for large and small gap areas. The maximum size for gaps to be considered as small can be specified either as a percentage of the tool diameter using the **Small gap size in % of tool diameter** parameter or by a **Value**.

The following types of movements are available, both for small gaps and for large gaps:

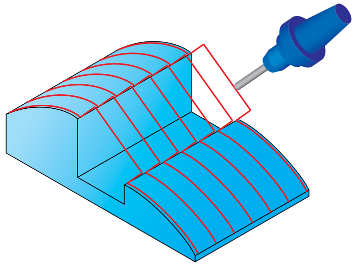
- **Direct**

  The tool moves in the shortest way to the other side of the gap, without any retracting movements. The tool path in the gap area is performed by a straight line; the tool moves at the cutting feed rate.



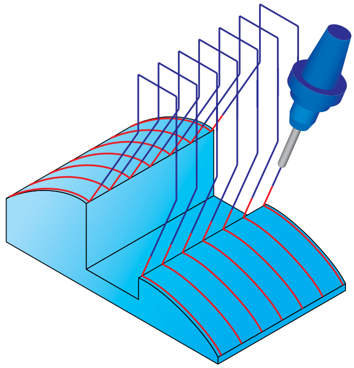
- **Safety distance**

When the gap area is detected, the tool performs a retract movement in the tool axis direction to the **Safety distance**. From this point the tool moves directly to the **Safety distance** of the next segment of the pass. All the tool movements are performed with a cutting feed rate.



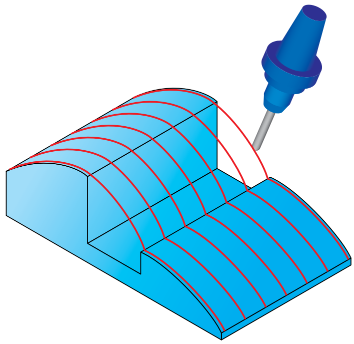
- **Clearance area**

When the gap area is detected, a link movement between the pass segments is performed through the **Clearance area**. All the movements above the **Safety distance** are performed with the rapid feed rate. All the movements below the **Safety distance** are performed with the specified cutting feed rate.



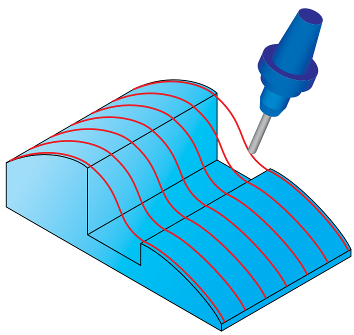
- **Follow surfaces**

In the gap area, the tool follows the drive surface geometry. Along the connection movement, SolidCAM tries to maintain tangency between the pass segments; when it is not possible, SolidCAM maintains tangency only for the first pass segment.

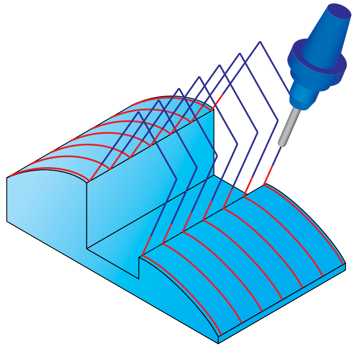


- **Blend spline**

SolidCAM connects the pass segments with a spline tangential to both segments.



- **Safety distance and rapid**

  When the gap area is detected, the tool performs a retract movement in the tool axis direction to the **Retract distance**. From this point the tool moves directly to the **Retract distance** of the next segment of the pass. All the tool movements below the **Safety distance** are performed with a cutting feed rate. All the movements above the **Safety distance** are performed with the rapid feed rate.

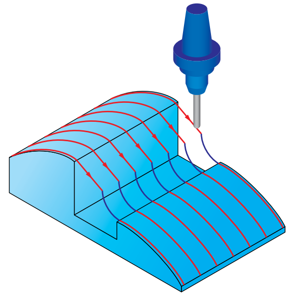
- **Follow Stock**

  When a gap area is detected, the tool performs a retract movement to the stock upper level and makes a link on the upper level height.

SolidCAM enables you to use pre-defined **Lead-In/Lead-Out** strategies (see topic **8.3**) to perform the movements between segments of a pass divided by a gap.

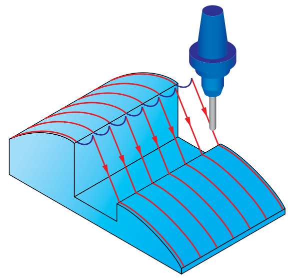
- **Use Lead-In**

  SolidCAM performs the approach movement to the drive surface after the gap using the specified **Lead-In** options.

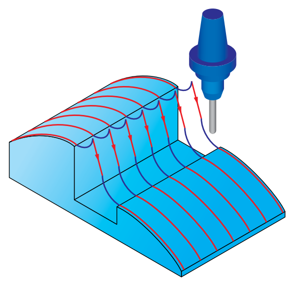
- **Use Lead-Out**

  When a gap is detected, SolidCAM performs the retreat movement using the specified **Lead-Out** options.

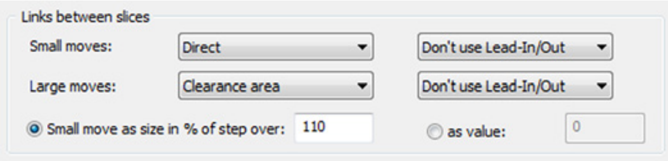
- **Use Lead-In/Out**

  SolidCAM performs the approach movement to the drive surface after the gap using the specified **Lead-In** options and performs the retreat movement using the specified **Lead-Out** options.

8.2.2 Links between slices

This section enables you to define how the tool moves between cutting passes.

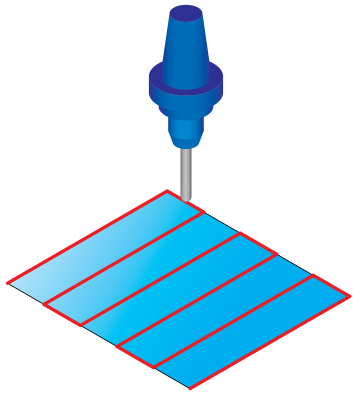


SolidCAM enables you to define different ways of movements between passes spaced with different **Maximum Step over** (see topic **6.1.3**). Depending on the **Maximum Step over** value, it is possible to choose two different options for large and small movements between passes. The maximum size for movement to be considered as small can be specified either as a percentage of the **Maximum Step over** using the **Small move size in % of step over** parameter or by a **Value**.

The following types of movements are available both for small and large movements:

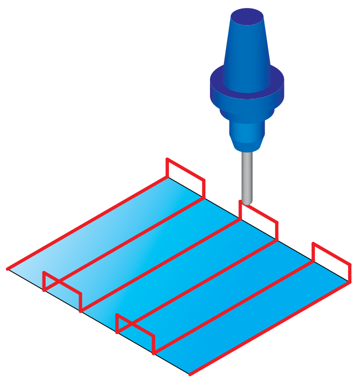
- **Direct**

  The tool moves in the shortest way to the next pass, without any retracting movements. The linking tool path is performed by a straight line; the tool moves at the cutting feed rate.



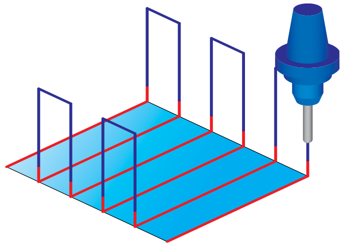
- **Safety distance**

  The tool performs a retract movement in the tool axis direction to the **Safety distance**. From this point the tool moves directly to the **Safety distance** of the next pass. All the tool movements are performed with a cutting feed rate.



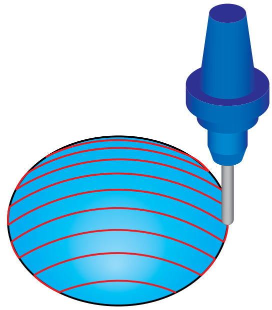
- **Clearance area**

  A link movement between passes is performed through the **Clearance area**. All the movements above the **Safety distance** are performed with the rapid feed rate. All the movements below the **Safety distance** are performed with the specified cutting feed rate.



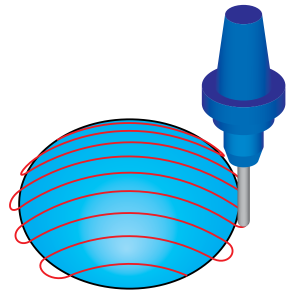
- **Follow surfaces**

  The tool follows the drive surface geometry.



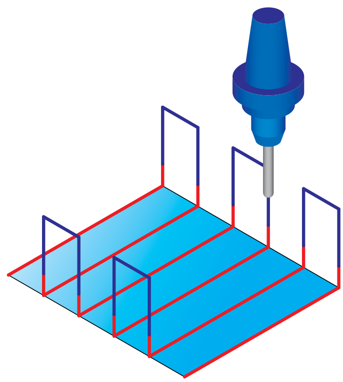
- **Blend spline**

  SolidCAM connects the passes with a spline tangential to both passes.



- **Safety distance and rapid**

  The tool performs a retract movement in the tool axis direction to the **Retract distance**. From this point the tool moves directly to the **Retract distance** of the next pass. All the tool movements below the **Safety distance** are performed with a cutting feed rate. All the movements above the **Safety distance** are performed with the rapid feed rate.



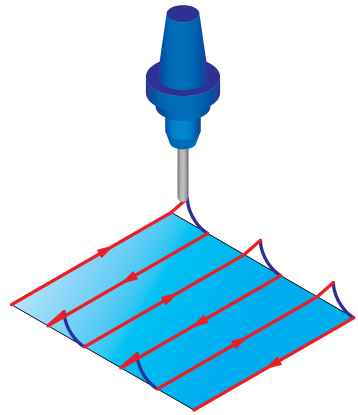
- **Follow Stock**

  The tool performs a retract movement to the stock upper level and makes a link on the upper level height.

SolidCAM enables you to use pre-defined **Lead-In/Lead-Out** strategies (see topic **8.3**) to perform the movements between passes.

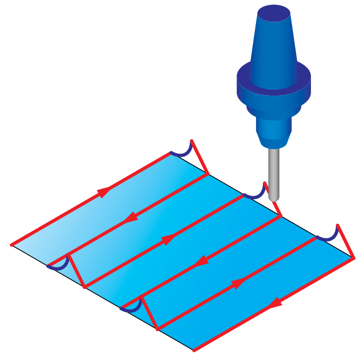
- **Use Lead-In**

  SolidCAM performs the approach movement to the drive surface after the gap using the specified **Lead-In** options.



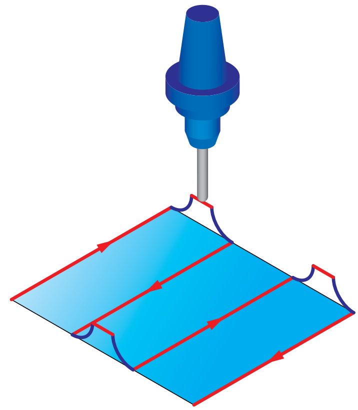
- **Use Lead-Out**

  When a gap is detected, SolidCAM performs the retreat movement using the specified **Lead-Out** options.

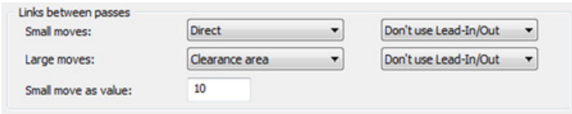


- **Use Lead-In/Out**

  SolidCAM performs the approach movement to the drive surface after the gap using the specified **Lead-In** options and performs the retreat movement using the specified **Lead-Out** options.



8.2.3 Links between passes



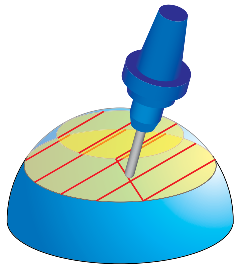
When the **Multi-passes** option or the **Depth cuts** option is used, SolidCAM performs the machining at several cutting levels. The **Links between passes** option enables you to define how the tool moves between these levels.

SolidCAM enables you to choose two different linking options for large and small movements between cutting levels. The maximum size of movements to be considered as small can be specified by a **Value**.

The following **Link types** are available:

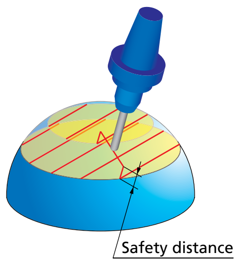
- **Direct**

  After the level machining, the tool moves directly to the first pass of the next level.



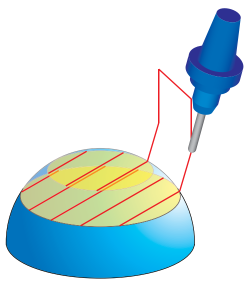
- **Safety distance**

  After the level machining, the tool retracts to the **Safety distance** and then directly moves to the **Safety distance** of the next level.

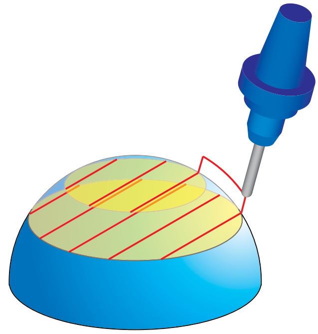


- **Clearance area**

  After the level machining, the tool retracts to the **Clearance area** and then performs the approach movement to the next level.

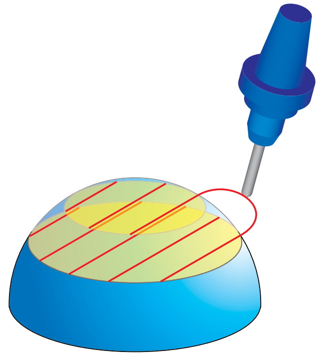


- **Follow surfaces**



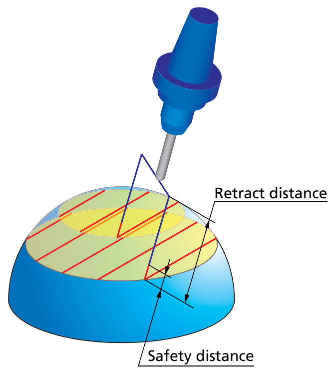
After the level machining, the tool retracts to the **Safety distance** and then performs an approach movement to the **Safety distance** at the next level. During the movement, the tool follows the drive surface geometry at the offset specified by the **Safety distance** parameter.

- **Blend spline**



The tool movement connecting the cutting levels is performed by a spline tangential to the cutting passes.

- **Safety distance and rapid**



After the level machining, the tool retreats to the **Retract distance** and then directly moves to the **Retract distance** at the next level.

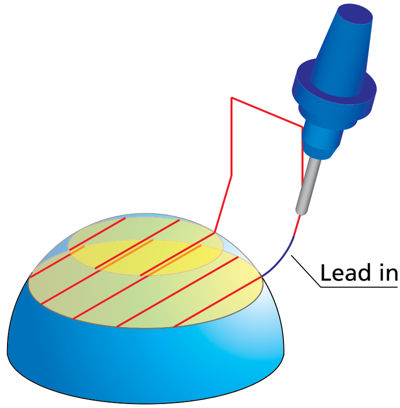
- **Follow Stock**

The tool retracts to the stock upper level and makes a link on this height when connecting tool paths between slices.

SolidCAM enables you to use pre-defined **Lead-In/Lead-Out** strategies (see topic **8.3**) to perform the movements between cutting levels.

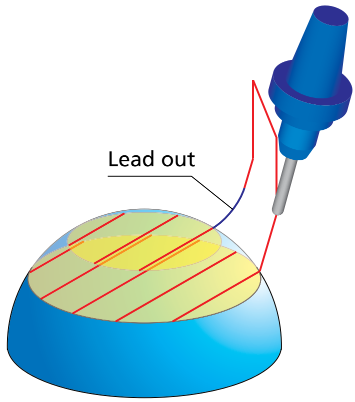
- **Use Lead-In**

  The approach movement is performed to the cutting level using the specified **Lead-In** options.



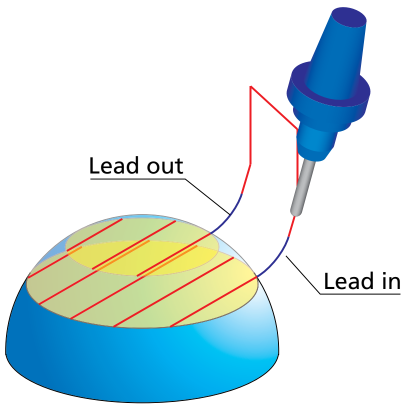
- **Use Lead-Out**

  The retreat movement is performed from the cutting level using the specified **Lead-Out** options.



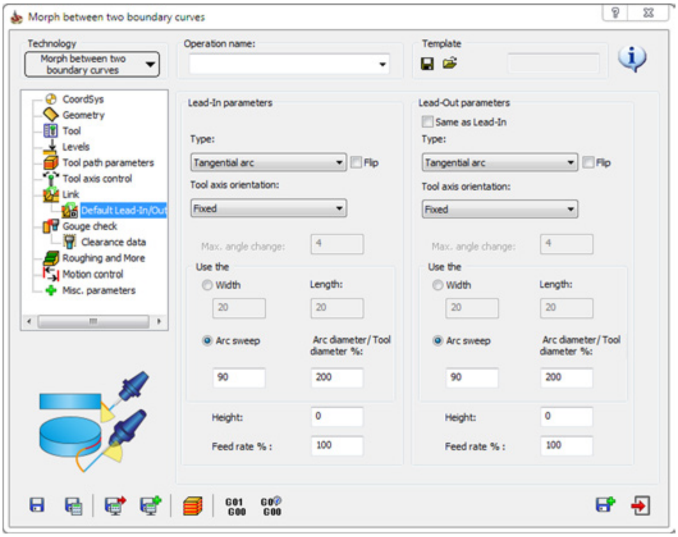
- **Use Lead-In/Out**

  The approach movement is performed to the cutting level using the specified **Lead-In** options, and the retreat movement is performed from the cutting level using the specified **Lead-Out** options.



8.3 Default Lead-In/Out

The **Lead-In/Out parameters** sections of the **Default Lead-In/Out** page enable you to define the parameters used for the approach/retreat movements.

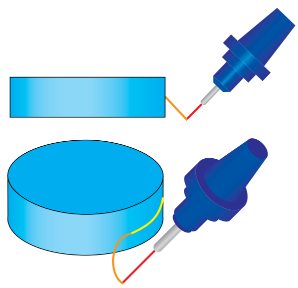


8.3.1 Type

SolidCAM enables you to choose one of the following types for the approach/retreat movements:

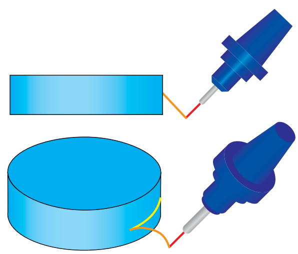
- **Tangential arc**

  The approach/retreat movement is performed with an arc tangential to the drive surface. The approach/retreat arc is generated in the plane orthogonal to the tool axis. Therefore, the arc orientation also depends on the tool tilting (see topic **7.3**). In the illustration, the tool tilting angle is 45°. Therefore, the arc orientation is also 45° to the drive surface. Setting the tilting angle to 0° enables you to perform an arc approach/retreat movement in a vertical plane. When the tilting angle is 90°, the approach/retreat arc is generated in a horizontal plane.



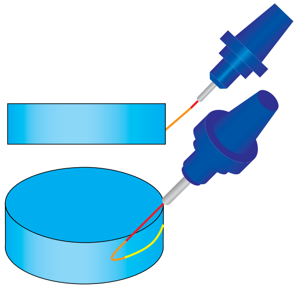
- **Reverse tangential arc**

  The approach/retreat movement is performed with an arc tangential to the drive surface, like the **Tangential arc** option, but with the direction of the approach/retreat arc reversed. The approach/retreat arc is generated in the plane orthogonal to the tool axis.



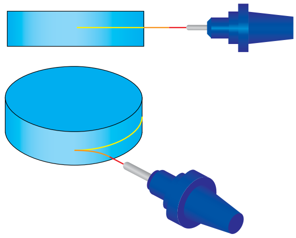
- **Vertical tangential arc**

  SolidCAM performs the approach/retreat movement with an arc tangential to the drive surface located in the plane of the tool axis. Note that when the side tilting options (see topic **7.3**) are used, the plane of vertical tangential arc is not changed.



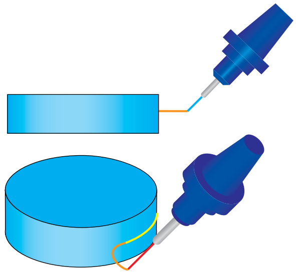
- **Reverse vertical tangential arc**

  SolidCAM performs the approach/retreat movement with an arc tangential to the drive surface located in the plane of the tool axis, similar to the **Vertical tangential arc** option, but the direction of the approach/retreat arc is reversed.



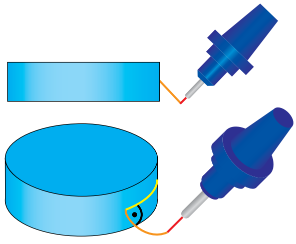
- **Horizontal tangential arc**

  SolidCAM performs the approach/retreat movement with an arc tangential to the drive surface located in the plane of the cutting movement. The arc orientation is independent from the tool orientation.



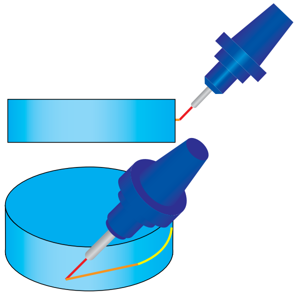
- **Orthogonal arc**

  SolidCAM performs the approach/retreat movement with an arc orthogonal to the cutting pass located in the plane perpendicular to the tool axis. The approach/retreat arc orientation also depends on the tool tilting (see topic **7.3**).



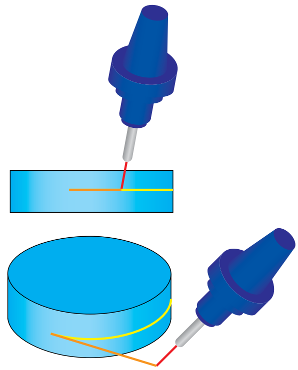
- **Tangential line**

  SolidCAM performs the approach/retreat movement with a line tangential to the cutting pass.



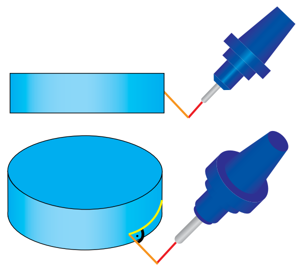
- **Reverse tangential line**

  With this option SolidCAM performs the approach/retreat movement with a line tangential to the cutting pass like the Tangential line option, but the direction of the approach/retreat line is reversed.



- **Orthogonal line**

  SolidCAM performs the approach/retreat movement with a line orthogonal to the cutting pass.



- **Position line**

  With this option SolidCAM uses start point and direction as input and interpolates the tool positions from the line to the first and last point in the contour. The line that positions the tool is defined as **Direction**.

- **Vertical profile ramp**

  With this option SolidCAM allows the lead to follow the tool path contour shape till it reaches a specified length and height. The ramp direction is along the cutting direction.

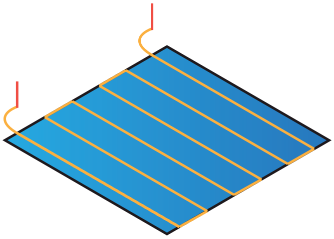
- **Reverse vertical profile ramp**

  With this option SolidCAM allows the lead to follow the tool path contour shape till it reaches the specified length and height. The ramp direction is opposite to the cutting direction.

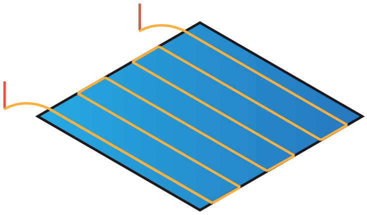
Flip

This option is useful when regular approach/retreat by arc is obstructed by part or fixture geometry and therefore will result in a collision between the tool and the machined workpiece. The defined approach/retreat arc can be flipped to ensure unobstructed access to the drive surface (and/or unobstructed exit in the end of machining).

When this check box is selected, the lead move is flipped to the opposite side from its initial orientation, as if it was mirrored relative to the tool path.



Flip check box is not selected



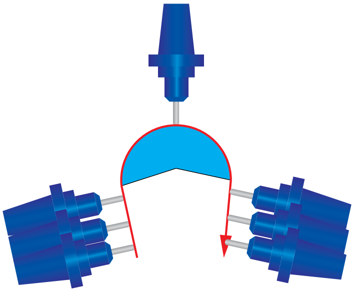
Flip check box is selected

8.3.2 Tool axis orientation

SolidCAM enables you to control the tool orientation during the approach/retreat movement.

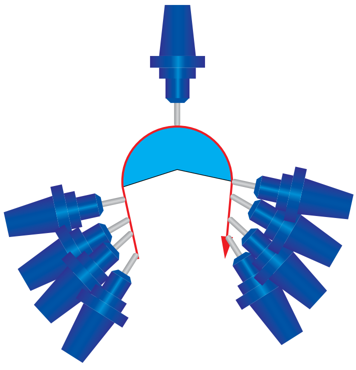
- **Fixed**

  The tool orientation is constant during the approach/retreat movement. The orientation changes at the start/end point of the cutting pass.

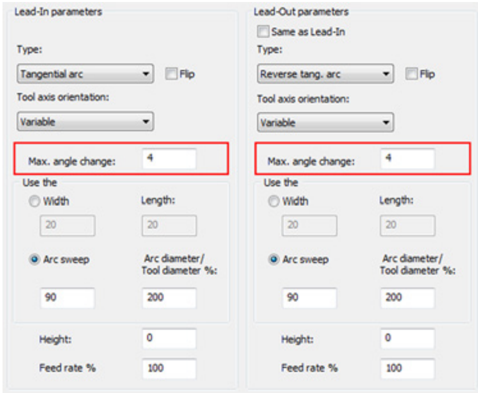


- **Variable**

  The tool axis orientation continuously changes during the approach/retreat movement. This option enables you to avoid marks on the part surface caused by the tool rotation at the start/end point of the cutting pass.



In case of large macro moves, the total change of tool axis orientation might be great. Therefore, SolidCAM enables you to limit the total change of tool axis orientation along a macro move with the **Max. angle change** parameter.



- **Tilted**

  This option maintains the main tilting strategy used in the **Tool axis control** page. If the main tilting strategy is set to **Tilted through point**, the lead-in/out uses the same strategy.

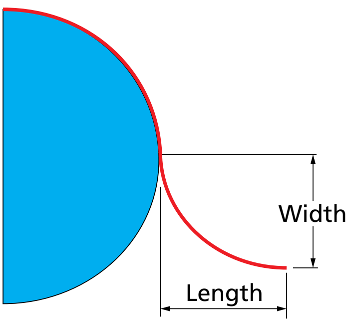
8.3.3 Approach/Retreat parameters (Use the...)

This section enables you to define the dimensions of the approach/retreat arc or line.

Width/Length

In this section, SolidCAM enables you to define the approach/retreat arc or line by the width and length parameters.

The **Width** and **Length** parameters define the bounding rectangle surrounding the arc. The defined arc has a 90° sweep.



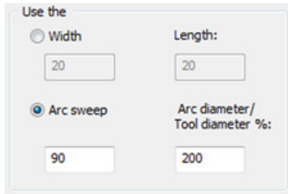
Fillet

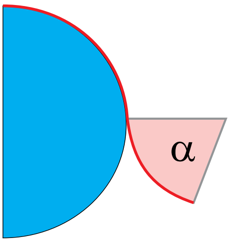
In this section, SolidCAM enables you to create a fillet with a specified radius between the line and the contour.

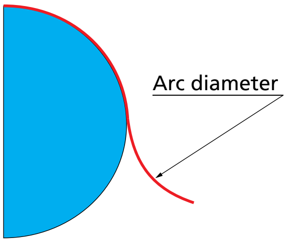
This option is available only if the chosen approach/retreat type is **Orthogonal line**, **Tangential line**, **Reverse tang. line**, **Vertical profile ramp**, and **Reverse vertical profile ramp**.

Arc sweep/Arc diameter

SolidCAM enables you to specify the diameter of the approach/retreat arc using the percentage of the **Arc diameter** to the **Tool diameter**. For example, when a tool with the diameter of 10 mm is used and the percentage parameter is set to 200%, the resulting approach/retreat arc diameter is 20 mm.



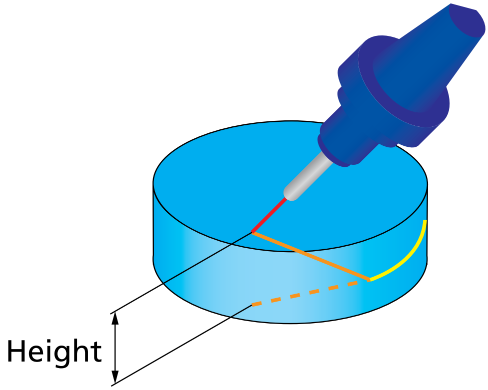




The **Arc Sweep** parameter enables you to define the angle of the approach/retreat arc segment.

8.3.4 Height

This parameter enables you to define the incremental height of the first point of the approach/retreat movement. When the approach movement is performed with a line, the **Height** parameter enables you to perform inclined ramp approach. When the approach movement is performed with an arc, the **Height** parameter enables you to perform a helical approach.

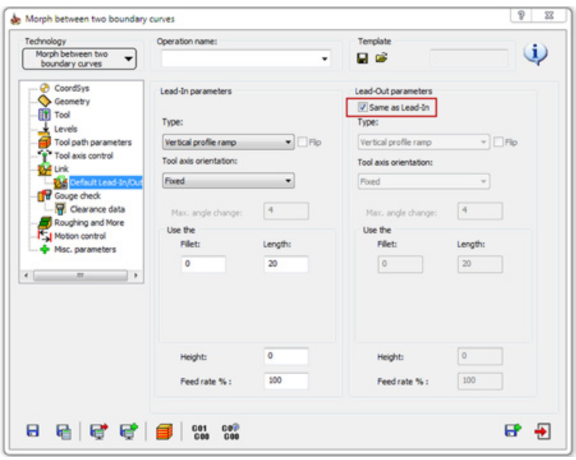


8.3.5 Feed rate

This parameter enables you to define the feed rate for the approach/retreat movement.

8.3.6 Same as Lead-In

The **Same as Lead-In** option enables you to use the defined **Lead-In** strategy for the **Lead-Out** definition.



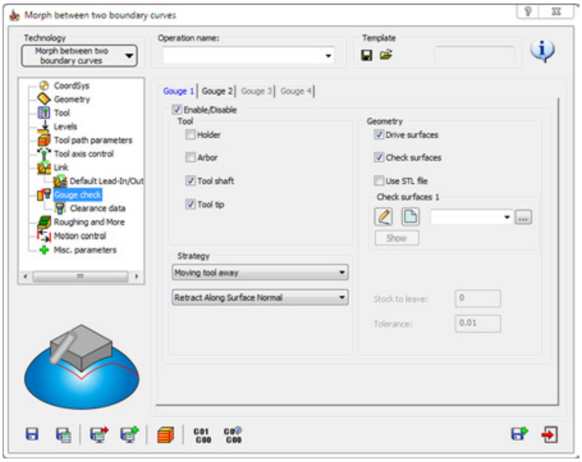
Gouge check

9

The **Gouge check** page enables you to automatically detect and avoid the possible collisions between the tool (and the tool holder) and the workpiece.

9.1 Gouge checking

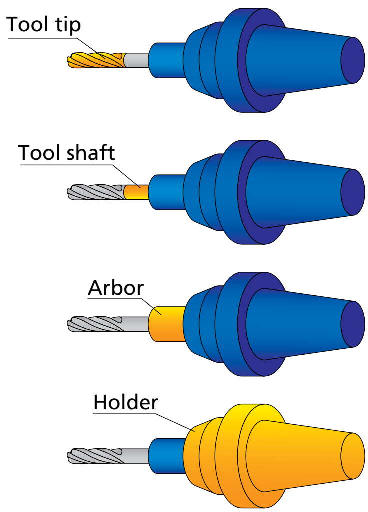
SolidCAM enables you to define four different sets of gouge checking parameters. In each set you have to choose components of the tool holding system and model faces to check the possible collisions between them. You also have to define the strategy how to avoid the possible collisions. Combining these sets, SolidCAM enables you to choose different strategies for avoiding the different types of possible collisions.



Select the **Enable/Disable** check box to activate a set of gouge checking parameters.

9.1.1 Tool

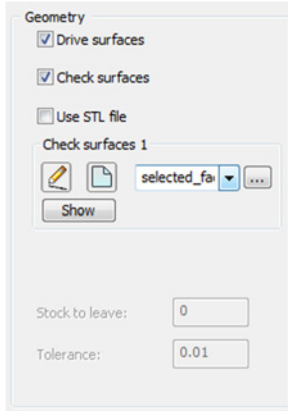
This section enables you to choose both tool and tool holder components to perform the gouge check for them.



SolidCAM enables you to choose the following part of the tool and tool holding system to perform the gouge check:

- Holder Geometry
- Arbor (shank)
- Tool Shaft (between flute length and arbor)
- Tool Tip (the flute length)

9.1.2 Geometry



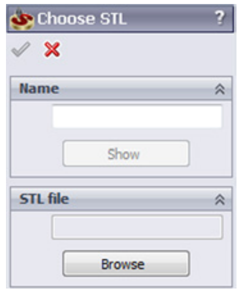
The **Geometry** section enables you to choose the model faces for which the gouge checking is performed.

Drive surfaces

When this option is chosen, SolidCAM performs the gouge checking for the **Drive surfaces** (see topic **3.1.1**), avoiding the possible collisions.

Check surfaces

This option enables you to choose a number of non-drive surfaces on the model as the Check surfaces and perform the gouge checking for them.



The **Check surfaces** section enables you either to choose the Check surfaces geometry from the list or define a new one with the button displaying the **Select Faces** dialog box.

When the **Use STL file** check box is activated, the **Check surfaces** group enables you to choose a check surfaces geometry from an STL file. The **Define** button enables you to display the **Choose STL** dialog box.

The **Browse** button in the dialog box enables you to choose the necessary STL file. The full name (including the path) of the chosen STL file is displayed in the **STL file** edit box.

When only the **Check surfaces** option is chosen, SolidCAM enables you to define two additional parameters:

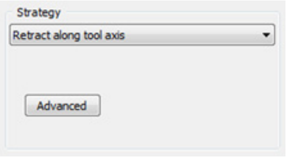
- **Stock to leave**. This parameter enables you to define an allowance for the Check surfaces. The tool cannot come closer to the Check surface than the specified value. For example, if the **Stock to leave** value is set to 1, SolidCAM checks that the tool is kept away from Check surfaces by 1 mm.

This parameter is used only in case when only the **Check surfaces** option is chosen for the gouge checking (the **Drive surfaces** option is not chosen). In case when both **Drive surface** and **Check surface** options are turned on, SolidCAM uses **Drive surface offset** value (see topic **3.1.2**) to define the machining allowance for both drive and check surfaces.

- **Tolerance**. This parameter enables you to define the accuracy of the gouge checking for the **Check surfaces**. The value defines the chordal deviation between the tool path and the Check surfaces.

This parameter is used only in case when only the **Check surfaces** option is chosen for the gouge checking (the **Drive surfaces** option is not chosen). In case when both **Drive surface** and **Check surface** options are turned on, SolidCAM uses **Cut Tolerance** value to define the gouge checking tolerance for both drive and check surfaces.

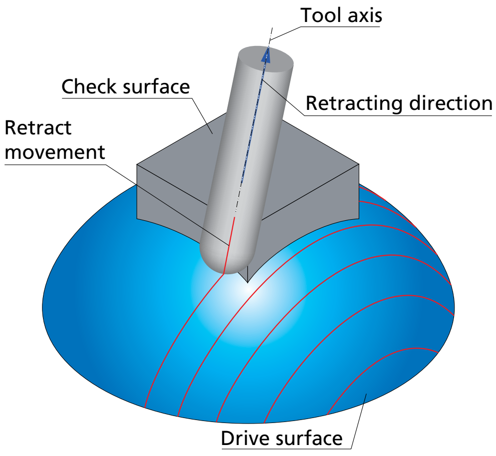
9.1.3 Strategy

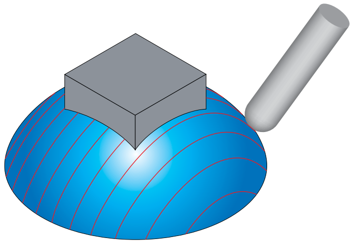


SolidCAM provides you with a number of strategies enabling you to avoid possible gouges.

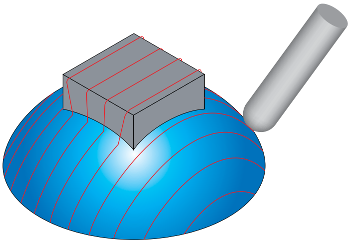
Retract along tool axis

When this option is chosen, SolidCAM enables you to avoid the possible collisions by retracting the tool in the direction of the tool axis. When a possible collision is detected, the tool performs a retract movement in the tool axis direction at the automatically calculated distance and then "flows" around check faces avoiding the gouge. The initial gouging tool path is substituted with a new one free of gouges.





Initial tool path gouging the check surfaces

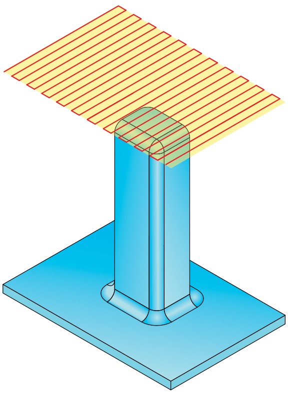


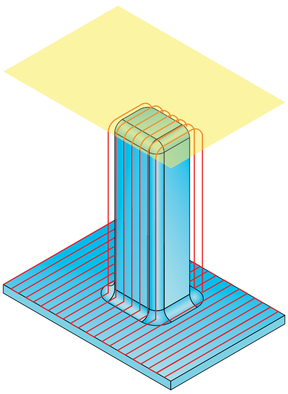
Updated tool path not gouging the check surfaces

SolidCAM provides you with advanced parameters for projection of the tool path from the drive surface plane onto the model to be machined.

In certain machining cases, it is convenient to define the required tool path on a flat surface, which facilitates the definition of parameters such as distance between cuts, machining angle that is set in the XY-plane, etc. Then this tool path can be projected onto a 3D model that needs to be machined.

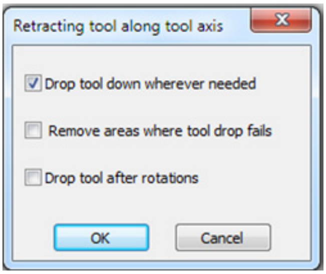
Consider the example of the following part: all of its surfaces need to be machined in simple parallel cuts. Instead of selecting all of its faces as drive faces to process the tool path on, you can create a new flat surface, on which the tool path will be defined. This surface will be defined as the drive surface, to which the actual tool path will be applied. You can easily check the resulting tool path on the flat surface, edit if necessary, and then to project it onto the faces of the model to be machined. Note that these faces will be defined as check surfaces.





To project the obtained tool path on the check surfaces, click the **Advanced** button.

The **Retracting tool along tool axis** dialog box is displayed. This dialog box enables you to define the parameters of tool path projection.



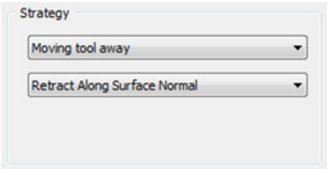
The **Drop tool down wherever needed** check box enables you to activate the projection of the tool path on the required 3D geometry. When you select this check box, the tool path applied to the drive surface is projected onto the defined drive surfaces.

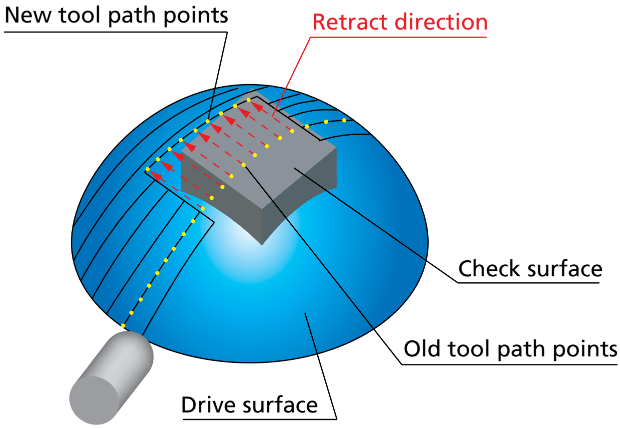
The **Remove areas where tool drop fails** check box enables you to exclude the areas, where the projection cannot be performed, from the tool path. When this check box is selected, the positions where the projection failed are removed.

The **Drop tool after rotations** check box enables you to activate the projection of the tool on the surface after rotating or transforming the tool path.

This option is available only if **Rotate&Translate** is used in **Roughing and More**.

Moving tool away



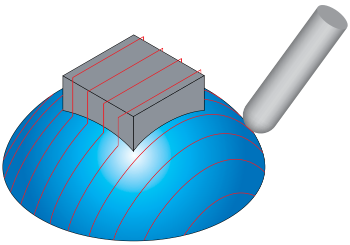


This option enables you to move the tool path positions, in which possible gouging is detected, in the specified retract direction away from the check surfaces. The new tool path positions form the updated tool path free of gouges.

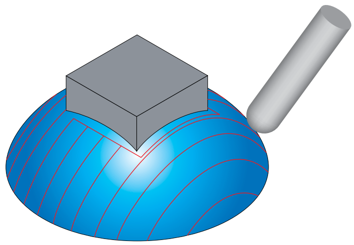
SolidCAM enables you to choose the following options to define the retract direction:

- **Retract in X, -X, Y, -Y, Z, -Z**

  The retract movement is performed along the chosen axis.



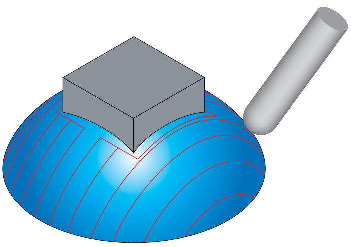
Retract tool in Z



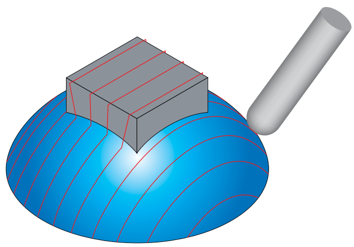
Retract tool in X

- **Retract in ZX, YZ, XY plane**

  The retract movement is performed in the chosen plane. The retract movement is performed in the direction, defined by the projection of the drive surface normal vector on the chosen plane.

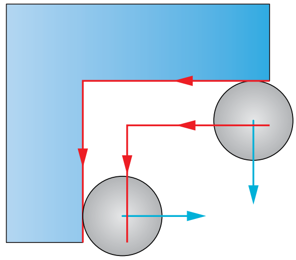


Retract tool in XY plane



Retract tool in ZX plane

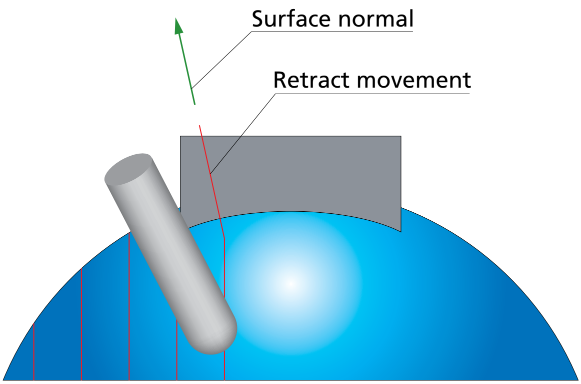
- **Retract-Optimized in ZX, YZ, XY plane**



With this option, the retract movement is performed in the chosen plane, similar to the **Retract tool in XY, ZX, YZ plane** options; the differences are in the direction of the retract movements in the chosen plane. The contact points, at which collisions are detected, are projected on the chosen plane and connected into a contour. This contour is offset outwards by a distance equal to the sum of the tool radius and the **Stock to leave** values. This option enables you to perform the retract movements in optimal directions, generating the shortest tool path.

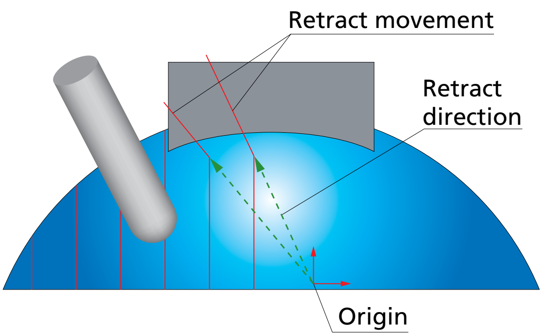
- **Retract Along Surface Normal**

The retract movement is performed in the direction of the drive surface normal at the contact point.



- **Retract Away from Origin**

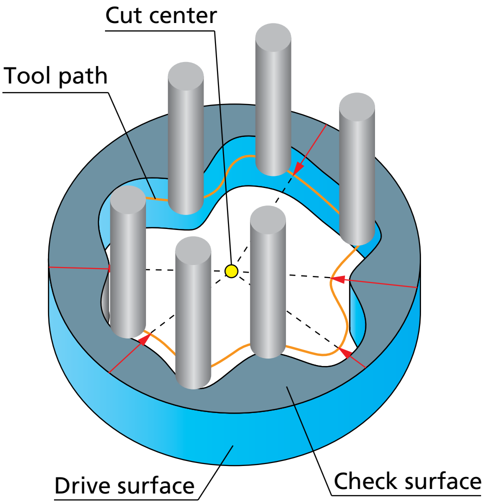
The retract movement is performed in the direction of the vector from the Coordinate System origin to the tool contact point.



- **Retract Towards Cut Center**

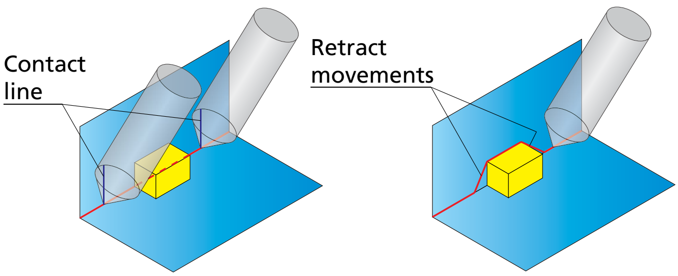
The retract movement is performed in the direction of the center of the cutting pass. This option is useful for tube milling.

On the illustration, the machining of the drive surface is performed with the parallel Z cutting passes. When the gouging of the check surface occurs, the retract movement is performed to the cutting pass center, avoiding the gouge.



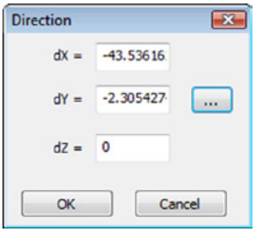
- **Retract Along Tool Contact Line**

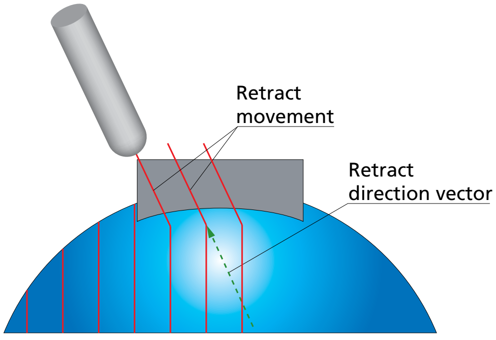
With this option, the retract movements are performed along the contact line between the tool and the drive surface.



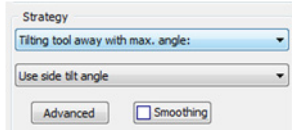
- **Retract at user-defined direction**

This option enables you to define the direction of the retract movements by a vector. The **Direction** dialog box (available with the [...] button) enables you to define the direction vector by its coordinates (**dX**, **dY** and **dZ** parameters). Using the [...] button, SolidCAM enables you to pick the start and end points of the vector directly on the solid model.





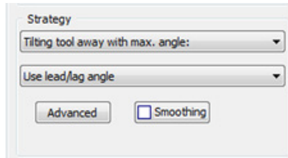
Tilting tool away with max. angle



The **Tilting tool away with max. angle** option enables you to avoid the possible collisions by the tool tilting.

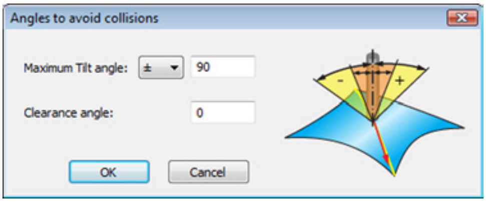
SolidCAM enables you to choose the following options to define the direction of the tool tilting:

Use lead/lag angle



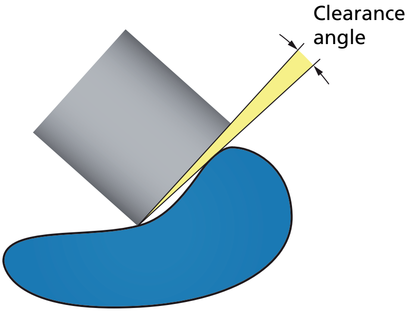
With this option, the tool tilting is performed in the cutting direction.

The **Advanced** button enables you to choose the tilting direction and specify the range for the tilting angle in the **Angles to avoid collisions** dialog box.

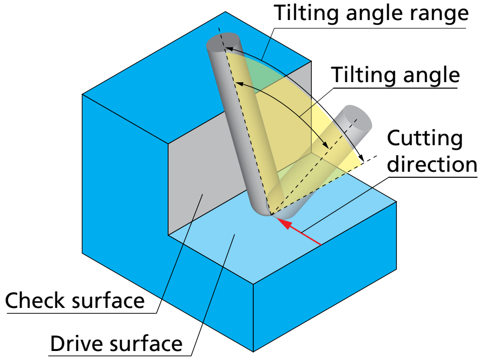


When the positive direction (+) is chosen for **Maximum tilt angle**, the tool can tilt in the positive cutting direction at an automatically chosen angle. The angle is within the range from 0 to the specified value. When the negative direction (-) is chosen, the tool tilting is performed in the negative cutting direction. When plus-minus direction (±) is chosen, SolidCAM automatically chooses either positive or negative direction of the tilting and performs it at an angle from the specified range.

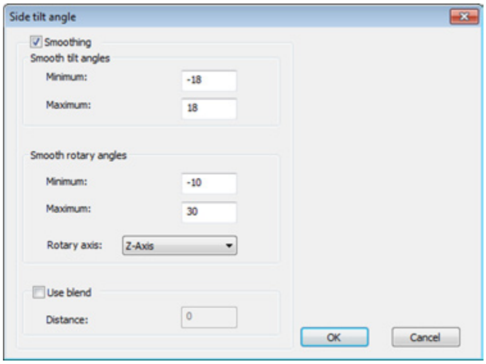
The **Clearance angle** protects the tool tip flat end back side against collisions with the drive face.



On the illustration, possible collision is detected and avoided by tool tilting. The tool tilting is performed in the negative cutting direction. The tilting angle is chosen automatically from the specified range.



The **Smoothing** button enables you to smooth the tool path in its cutting direction, as well as the side tilt angle in the **Side tilt angle** dialog box.



The smoothing is defined according to a rotary axis and can be applied to the following:

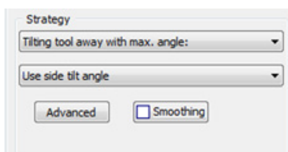
- To the tilt angles away and towards the specified axis. You have to define minimum and maximum angles that provide the range of freedom of the smoothing which should be used to smooth out the tool axis orientations away and towards the rotary axis.
- To the angles which tilt around that axis. You have to define minimum and maximum angles that provide the range of freedom of the smoothing which should be used to smooth out the tool axis orientations around the rotary axis.

First, select an appropriate rotary axis. The minimum and maximum values of the tilt and rotary angles depend on how much freedom you would like to allow.

The **Use blend** section enables you to define the **Distance** between the collision point and the point where the tool tilting starts.

Use side tilt angle

With this option, the tool tilting is performed in the side direction, relative to the cutting direction.

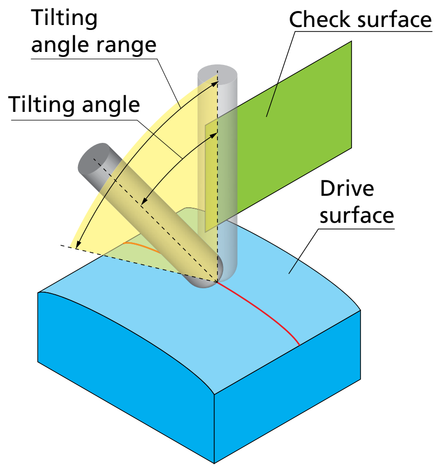


The **Advanced** button enables you to choose the tilting direction and specify the range for the tilting angle in the **Angles to avoid collisions** dialog box.

When the positive direction (+) is chosen for **Maximum tilt angle**, the tool tilting is performed at an automatically chosen angle to the right side relative to the cutting direction. The angle is located in a range from 0 to the specified value. When the negative direction (-) is chosen, the tool tilting is performed to the left side relative to the cutting direction. When plus-minus direction (±) is defined, SolidCAM automatically chooses the direction of the side tilting and performs it at an angle from the specified range.

The **Clearance angle** protects the tool tip flat end back side against collisions with the drive face.

On the illustration, the possible collision of the tool with the check surface is detected and avoided by the side tilting. The tool tilting is performed to the left side relative to the cutting direction. The tilting angle is chosen automatically from the specified range.



The **Smoothing** button enables you to smooth the tool path in its cutting direction, as well as the side tilt angle in the **Side tilt angle** dialog box.

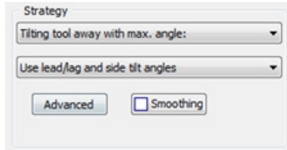
The smoothing is defined according to a rotary axis and can be applied to the following:

- To the tilt angles away and towards the specified axis. You have to define minimum and maximum angles that provide the range of freedom of the smoothing which should be used to smooth out the tool axis orientations away and towards the rotary axis.
- To the angles which tilt around that axis. You have to define minimum and maximum angles that provide the range of freedom of the smoothing which should be used to smooth out the tool axis orientations around the rotary axis.

First, select an appropriate rotary axis. Then define the minimum and maximum values of the tilt and rotary angles that depend on how much freedom you would like to allow.

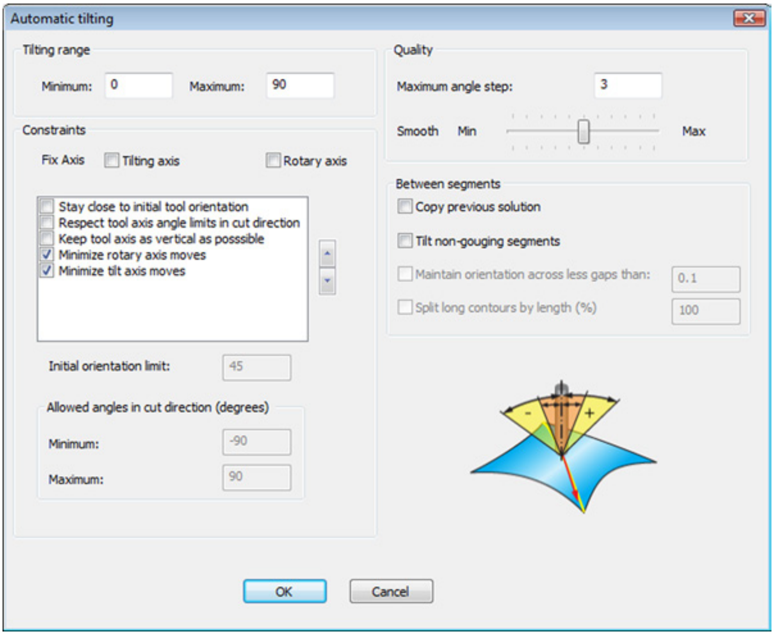
The **Use blend** section enables you to define the **Distance** between the collision point and the point where the tool tilting starts.

Use lead/lag and side tilt angles



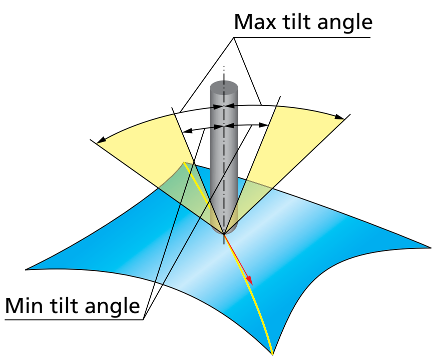
With this option, SolidCAM enables you to use the combination of lead/lag tilting and side tilting to avoid possible collisions.

The **Advanced** button displays the **Automatic tilting** dialog box. This dialog box enables you to define parameters of the tilting in the cutting direction and the side tilting.



Tilting range

This section enables you to control the side tilting by defining the minimum (**Min. tilt angle**) and maximum (**Max. tilt angle**) angles of the side tilting.



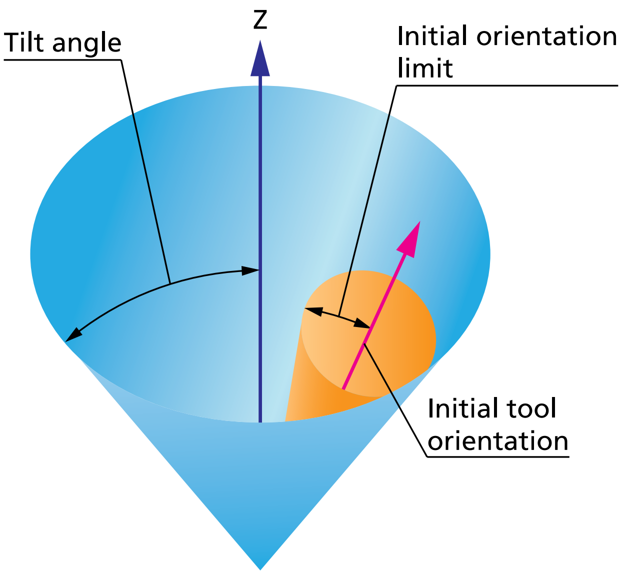
Constraint

The **Fix axis** option enables you to fix the CNC-machine axes (**Tilt axis**, **Rotary axis**) during the tilting movement used to prevent collisions.

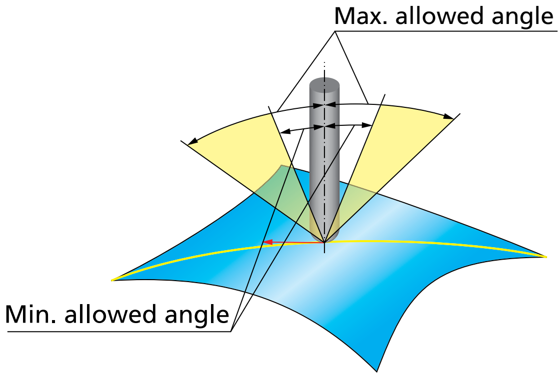
The check boxes section enables you to define the optimization preferences and set their priority. The preferences are ordered in the list from the highest priority (top of the list) to the lowest priority (bottom of the list). The **Up** and **Down** buttons enable you to change the order according to your preferences. Select the required option in the list and click the **Up** or **Down** button to change its position.

- **Stay close to initial tool orientation**. This option enables you to perform an optimal tool tilting with a minimum angular deviation from the initial tool orientation.
- **Respect tool axis angle limits in cut direction**. With this option, SolidCAM tries to keep the tool orientation in the cutting direction in the specified limits.
- **Keep tool axis as vertical as possible**. With this option, SolidCAM tries to find a gouge avoidance tilting solution with the tool orientation as close to vertical as possible.
- **Minimize rotary axis moves**. This option enables you to minimize the rotary axis movements.
- **Minimize tilt axis moves**. This option enables you to minimize the tilting axis movements.

The **Initial orientation limit** option defines the angular deviation from the initial tool orientation.

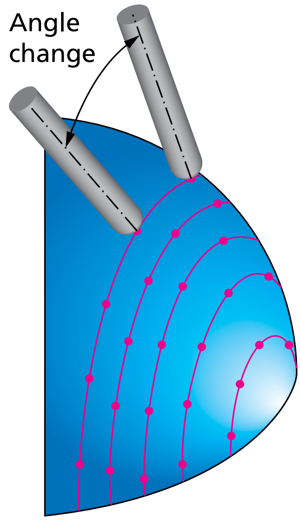


The **Allowed angles in cut direction** section enables you to control the tool tilting in the cutting direction. SolidCAM automatically chooses the necessary tilting angle in the range specified by the **Minimum** and **Maximum angle** values.



Quality

The **Max. angle step** parameter defines the maximal allowed angle change between the tool axes at two consecutive tool positions.



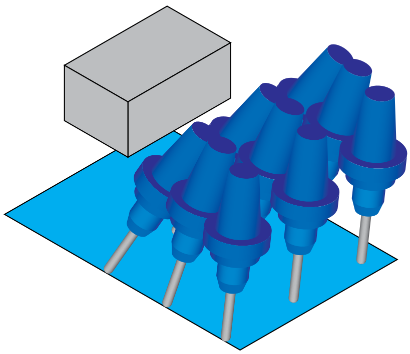
Smooth

This slider enables you to perform re-optimization of the tilting movement to prevent collisions. During the re-optimization, SolidCAM smooths the tilting movements.

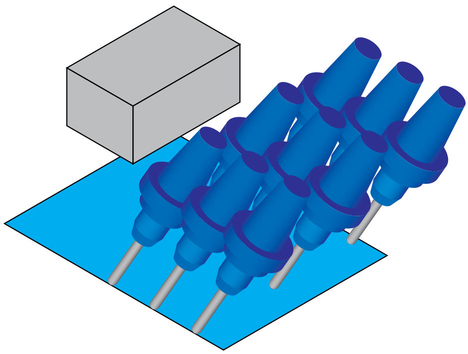
Between segments

- **Copy previous solution**

  SolidCAM tries to use for the actual cutting pass the gouge avoidance tilting solution used for the previous cutting pass.



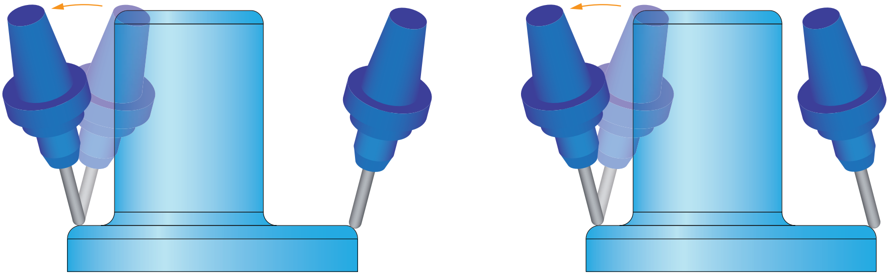
The check box is not selected



The check box is selected

- **Tilt non-gouging segments**

When this check box is selected, the tool tilting will be maintained in all non-gouging segments of the tool path.

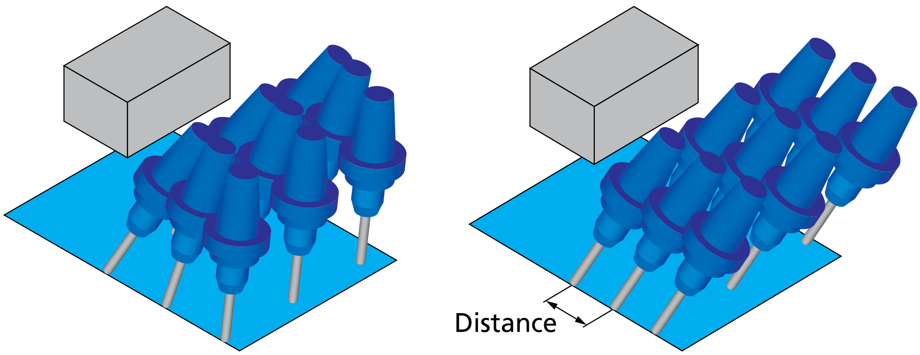


The check box is not selected | The check box is selected

When the **Tilt non-gouging segments** check box is not selected, the **Maintain orientation across less gaps than** and **Split long contours by length** check boxes are not available.

- **Maintain orientation across less gaps than**

SolidCAM enables you to use the same tilting for two successive cutting passes when the distance between them is smaller than the specified value.



The check box is not selected | The check box is selected

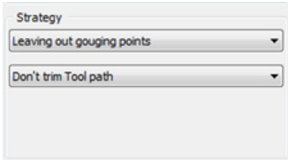
- **Split long contours by length**

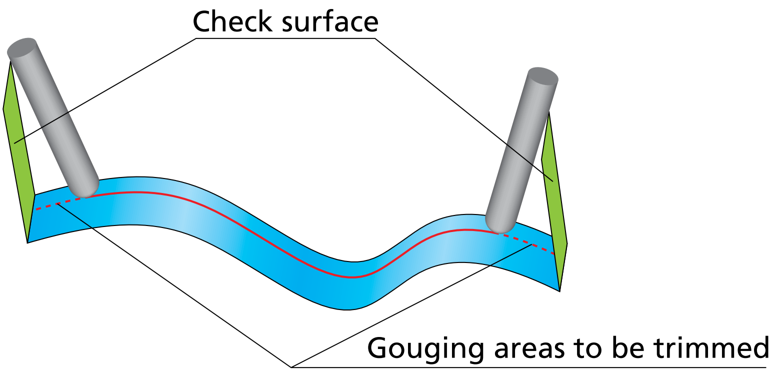
  Generally, SolidCAM tries to find a solution to avoid possible gouges for a complete pass. In case of long passes it is very difficult to find a solution that will be relevant along the whole pass length. To simplify this task, SolidCAM provides you with the **Split long contours by length** option that is enabled when you avoid the possible collisions using tilting in the cutting direction and side tilting. It enables you to split the original contour into multiple contours, with the length defined as percentage of the whole contour length. When the initial passes are split into a number of smaller passes, SolidCAM calculates the solution for avoiding collision for each pass.

  This option is available only if the **Maintain orientation across less gaps than** check box is selected.

Leaving out gouging points

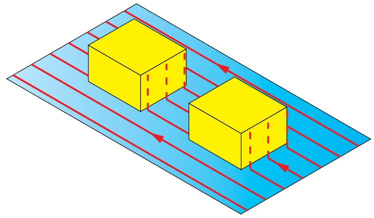
When this strategy is used, SolidCAM trims the segments of the tool path where the collisions are detected. The updated by trimming tool path does not contain gouges.



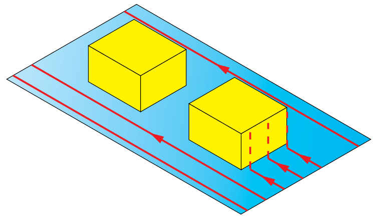


The following trimming options are available:

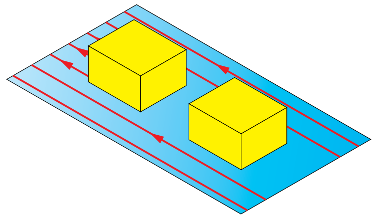
- **Do not trim tool path**. With this option, only the colliding segments of the tool path are trimmed out.



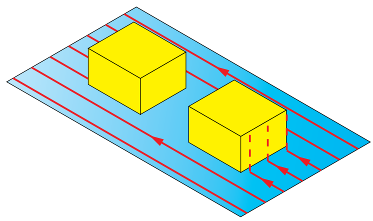
- **Trim tool path after first collision**. With this option, SolidCAM trims the whole cutting pass, after the first detected collision.



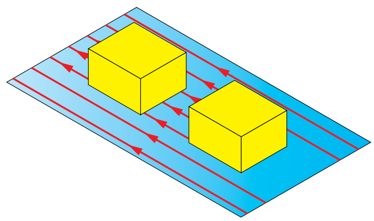
- **Trim tool path before last collision**. With this option, SolidCAM trims the whole cutting pass, before the last detected collision.



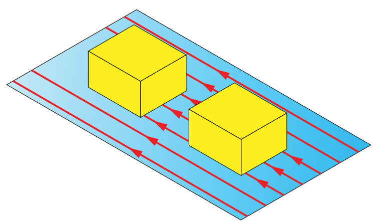
- **Trim tool path between first and last collision**. With this option, SolidCAM trims the cutting path between the first and last detected collisions.



- **Trim tool path before first collision**. When a collision is detected, the tool path is trimmed in such a manner that the portion of the current cut from the beginning till the first collision is removed from the tool path.

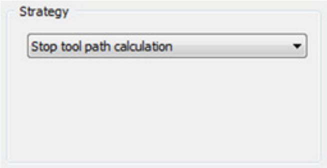


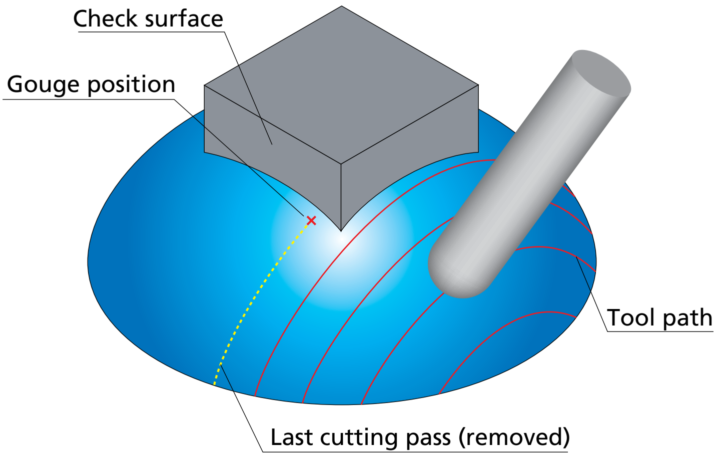
- **Trim tool path after last collision**. When a collision is detected, the tool path is trimmed in such a manner that the portion of the current cut from the last collision till the end of the cut is removed from the tool path.



Stop tool path calculation

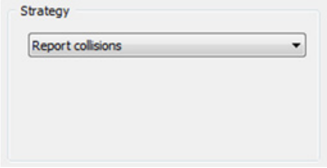
When this option is chosen, the tool path is generated until the position where the first gouge occurs. At this point the tool path calculation is stopped. The last cutting pass (where the gouge is detected) is not included into the operation tool path. You have to edit the machining parameters and calculate the tool path again to avoid the gouge.

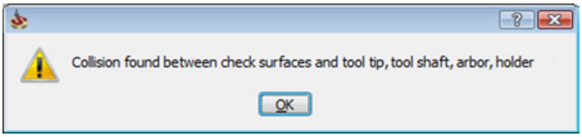




Report collisions

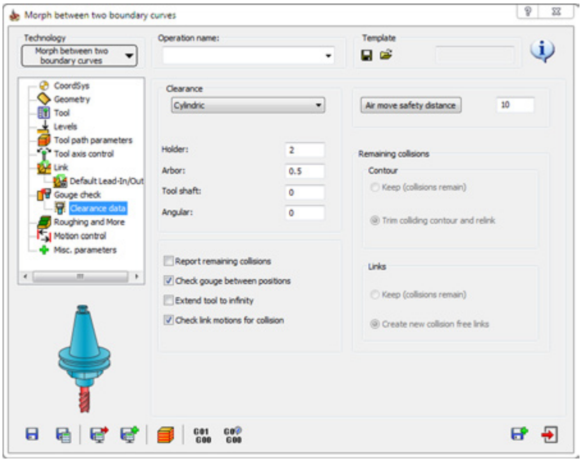
With the **Report collisions** option, SolidCAM checks only for collision between the tool and the check faces, without trying to avoid the collision; a warning message is displayed.





Using the simulation, you can check the collision areas and choose the appropriate method to avoid gouging.

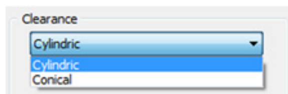
9.2 Clearance data



The **Clearance data** page enables you to define the clearance offsets for arbor and tool holder in order to get a guaranteed clearance gap between arbor, tool holder and workpiece.

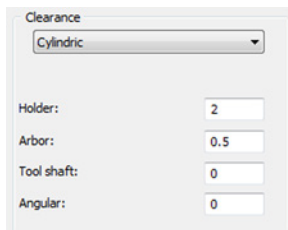
9.2.1 Clearance

SolidCAM enables you to choose either **Cylindric** or **Conical** shape of the tool holder, arbor and tool shaft clearance.

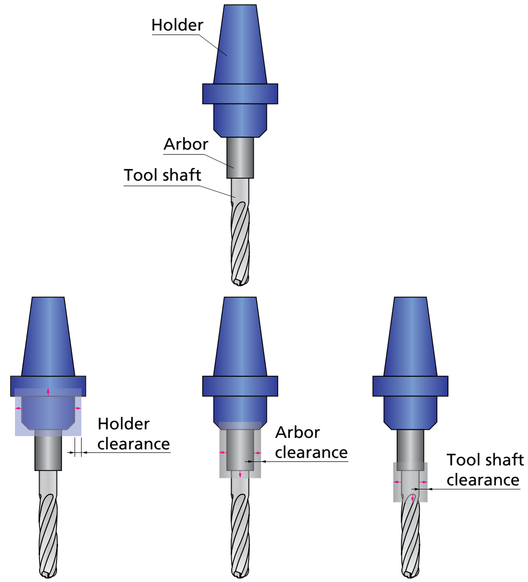


Cylindric clearance

The **Holder** parameter defines the offset applied to the tool holder cylinder from all sides. The **Arbor** defines the offset applied to the arbor cylinder from all sides. The **Tool Shaft** parameter defines the offset applied to the tool shaft cylinder from all sides. The **Angular** parameter defines the angular offset applied to the tool.

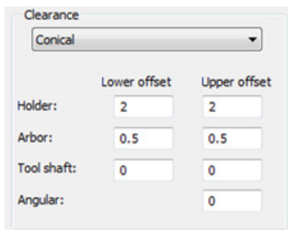


Generally, an arbor is the tool extension located between the tool shaft and its holder. Lollipop and Slot End Mills do not have tool shaft, the cylindrical connection between tool and holder is considered as arbor.



Conical clearance

Conical clearance is applied similar to the cylindrical one being defined with **Upper** and **Lower offset** values.

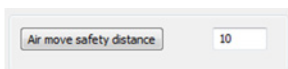


Angular conical clearance is applied between the tool and collision surface. It is spanned between the contact point of the tool, the drive surface, and the collision point.

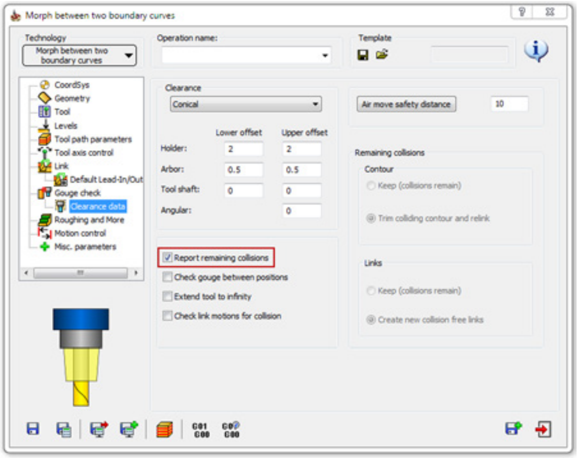
The **Upper offset** value has to be greater than the **Lower offset** value.

Air move safety distance

This parameter enables you to define the minimal distance between the clearance area and the **Drive surface** (see topic **3.1.1**).



Report remaining collisions



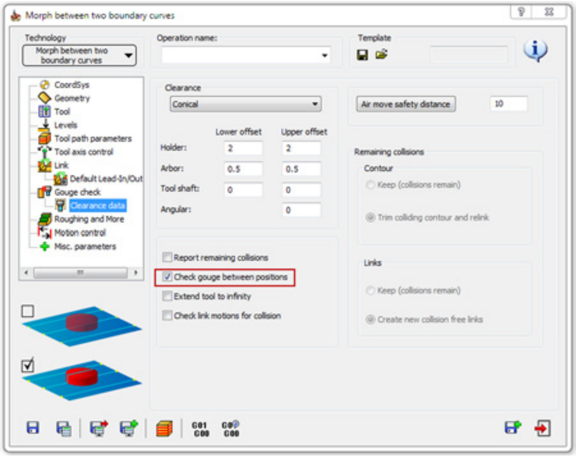
This option enables you to generate a report about possible collisions that remain in the tool path after gouge checking. When this option is turned on, SolidCAM checks the tool path using the tolerance two times greater than the specified value (see topic **9.1.2**) to detect collisions.

You can turn off the collision checking between the tool path positions. In such case the tool path calculation is accelerated, but the possibility of remaining collisions is present. The **Report remaining collisions** option is helpful to notify about possible collisions in the resulting tool path.

In case of projection or trimming 5-Axis operations used together with the **Report remaining collisions** option, SolidCAM notifies you about collisions. The reason for such notification is that the technology of such operations requires that the tool tip be inside the machining surfaces.

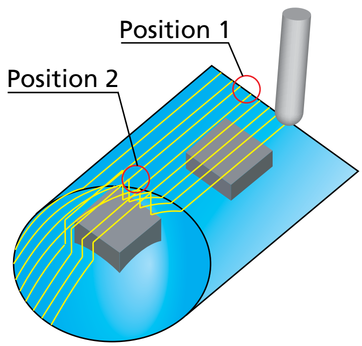
The **Report remaining collisions** option enables you to detect too small retract and approach distances or too low Clearance level. In such case, report about collisions enables you to solve the potential problems.

Check gouge between positions



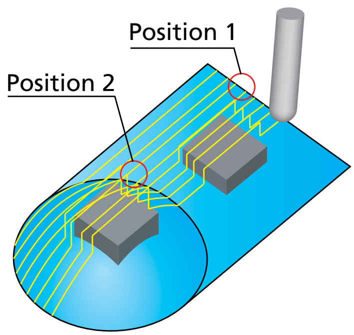
The **Check gouge between positions** option enables you to avoid the possible gouges between tool path positions. When the 5-axis movement is performed between two successive tool path positions, this option enables you to check for possible collisions of the tool and tool holder with drive and check surfaces.

This option is especially useful for flat faces machining, where the tool path positions are generated only at the drive surface edges.



When the **Check gouge between positions** option is not used, the gouge checking of the tool path on the flat face is not performed because of absence of tool path positions on the face. The gouging of a boss can occur.

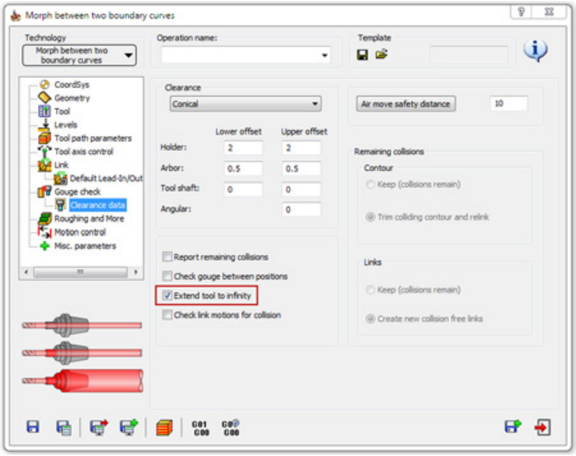
When the **Check gouge between positions** option is used, the gouge checking between tool path positions on the flat surface is performed. The gouging of a boss is avoided.



The **Check gouge between positions** option has no effect on the gouge checking of the tool path spherical surface, because of the many tool path positions that were generated on this face. The gouge checking for this face is performed for these positions avoiding possible collisions.

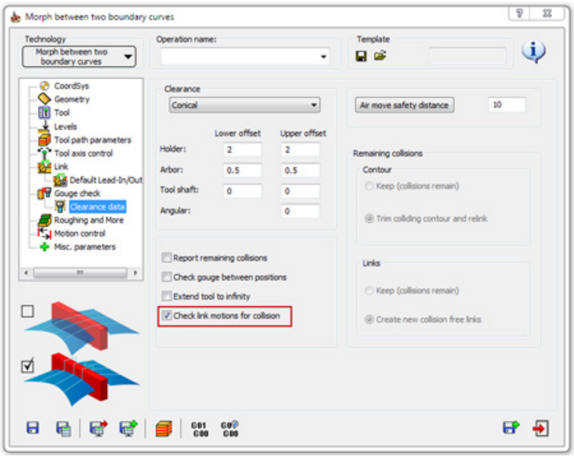
Extend tool to infinity

This option enables you to consider the tool as being extended to infinity during collision check in order to make sure that all active surfaces are checked for collision, no matter where they are located in space.



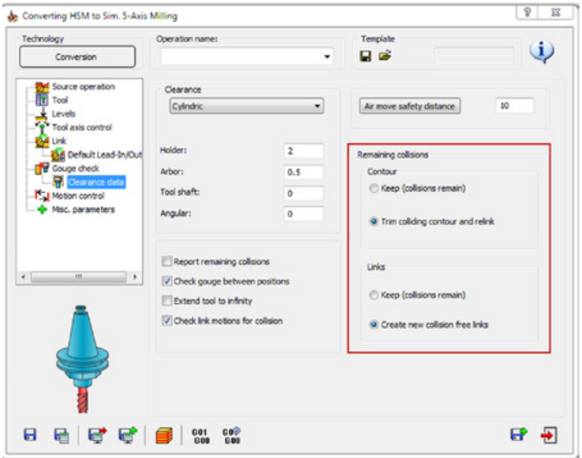
Check link motions for collision

When this option is selected, SolidCAM automatically performs the gouge checking for link movements in order to avoid possible collisions.



9.2.2 Remaining collisions

This section enables you to handle remaining collisions for links and contours independently. SolidCAM specifies whether to keep the detected colliding contours or trim and relink them.



Contour

This section enables you to select how to treat remaining collisions in cutting contours.

- **Keep (collisions remain)**

  When this option is selected, SolidCAM does not alter the tool path and it continues to have remaining collisions if any in the contour.

- **Trim colliding contour and relink**

  When this option is selected, SolidCAM trims the colliding portions of the tool path and relinks the area using the linking options.

Links

This section enables you to select how to treat remaining collisions in connecting links.

- **Keep (collisions remain)**

  When this option is selected, SolidCAM does not alter the tool path and it continues to have remaining collisions in the link.

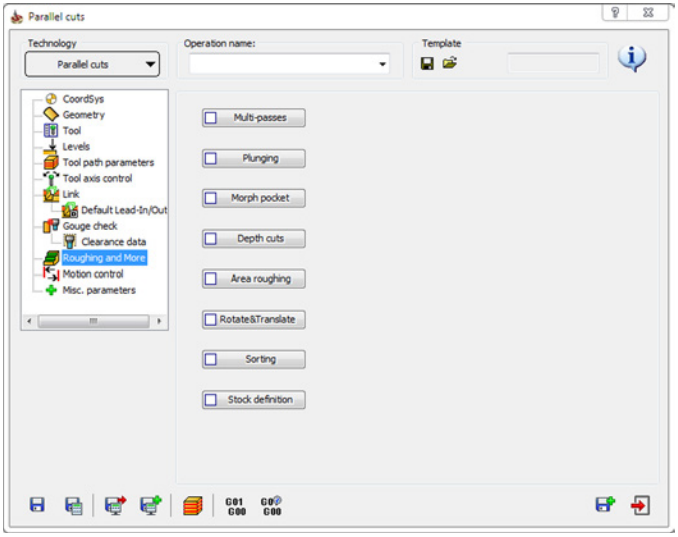
- **Create new collision free links**

  When this option is selected, SolidCAM trims the colliding portions of the links and relinks the area to remove the collisions in the links.

Roughing and More

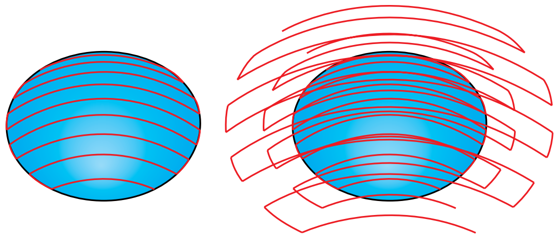
10

The **Roughing and More** page enables you to define the parameters of the rough 5-axis machining.



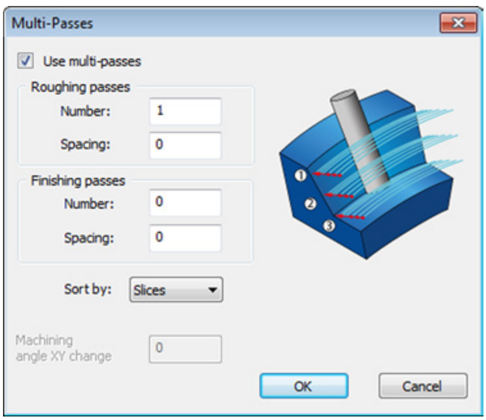
10.1 Multi-passes

The **Multi-passes** option enables you to perform the machining with a number of roughing and finishing layers. During the tool path calculation, SolidCAM generates the initial cutting pass located on the drive surface and then creates a specified number of roughing and finishing passes at different offsets specified for roughing and finishing.



Initial tool path   Multi passes tool path

When this option is used, SolidCAM generates for each tool path point a number of offsets in the direction of the surface normal. Connecting these points, SolidCAM generates a number of evenly spaced cuts with a similar tool path shape.



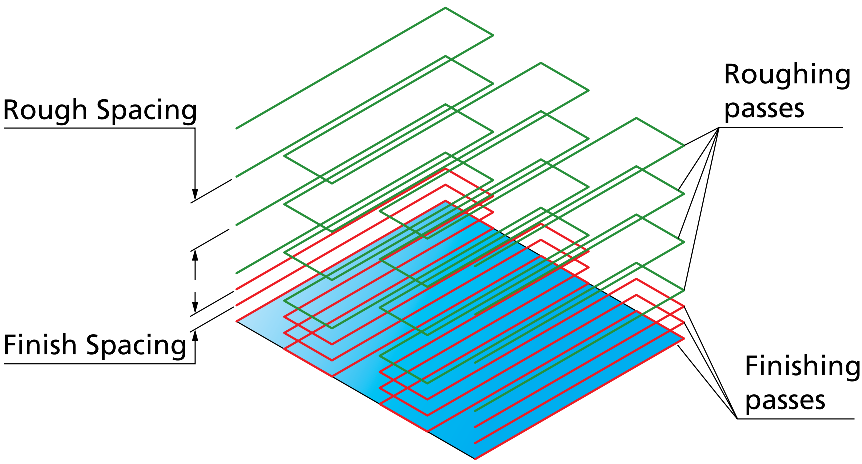
Click the **Multi-passes** button to define the parameters in the **Multi-passes** dialog box. Select the **Use Multi-passes** check box to enable the related options.

Roughing and finishing passes

The **Roughing passes** section enables you to define the **Number** of roughing passes and distance between them (**Spacing**).

The **Finishing passes** section enables you to define the **Number** of finishing passes and distance between them (**Spacing**).

During the tool path calculation, SolidCAM generates the initial cutting pass located on the drive surface and then creates a specified number of finishing passes using the specified **Spacing**. After the finishing, SolidCAM generates a specified number of roughing passes. The distance between the last finishing pass and the first roughing pass is defined by the **Spacing** parameter in the **Finishing passes** section.

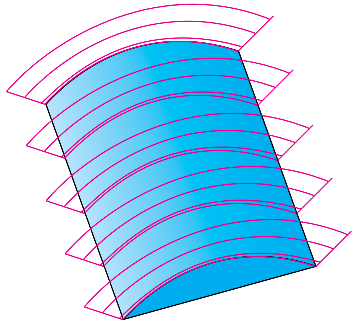


In the illustration above, SolidCAM generates three finishing passes with the specified finishing spacing and then generates four roughing passes with the specified roughing spacing. The distance between roughing and finishing parts of the tool path is equal to the finishing spacing.

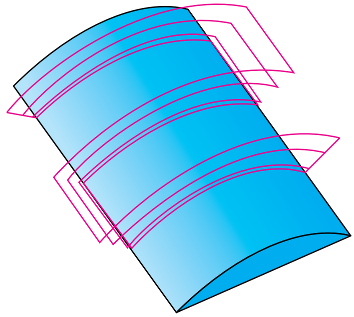
Sort by

This option enables you to define the way how the generated passes are sorted and linked.

- **Slices**. When this option is chosen, all the roughing and finishing offsets of the current cutting pass are performed before moving to the next cutting pass.

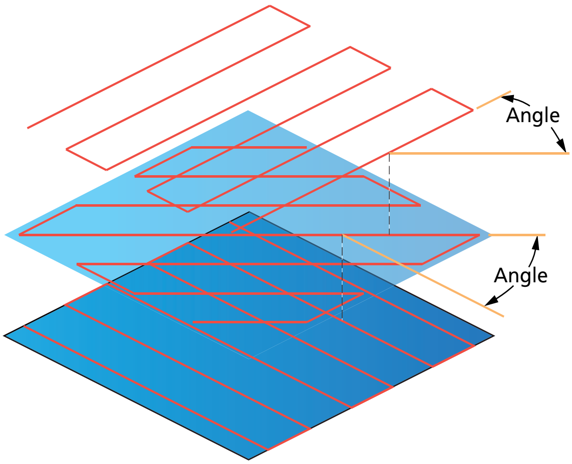


- **Passes**. When this option is chosen, all the cutting passes of the current offset level are performed before moving to the next offset level.



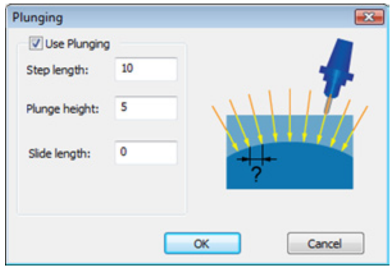
Machining angle XY change

This option enables you to define the gradual change of the machining angle for multi-cut roughing. **Machining angle XY change** is useful when the material needs to be removed at different angles in each pass. This option is enabled for **Parallel cuts** strategy, when the **Linear** work type is chosen in the **Geometry** page.



The value defines the angle to which the tool path rotates with every new pass.

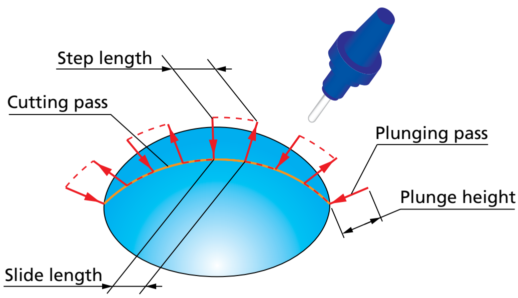
10.2 Plunging



The **Plunging** option enables you to perform 5-axis machining using the plunging technology. Plunging is a totally different concept of removing material with a special tool. Instead of milling the material, the tool moves up and down in a motion similar to drilling, at the points along the tool path.

Click the **Plunging** button to display the **Plunging** dialog box. This dialog box enables you to define the parameters of plunging.

When the **Plunging** option is used, SolidCAM generates for each cutting pass a number of tool path positions. These positions are evenly spaced along the cutting pass. The distance between two successive tool positions is defined by the **Step length** value. In each such position SolidCAM generates a plunging tool path in the direction of the drive surface normal; the height of this tool path is defined by the **Plunge height** value. The **Slide length** value defines the distance the tool travels after the plunging move.

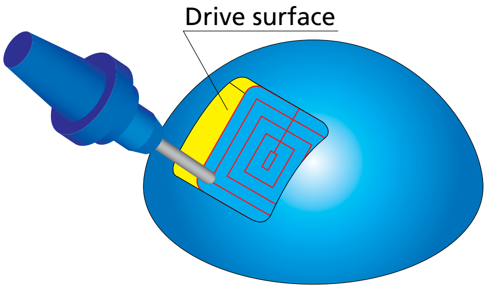


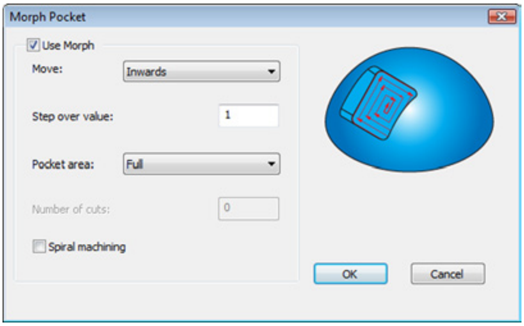
Note that setting high **Slide length** values can cause the tool breakage.

10.3 Morph pocket

The **Morph pocket** option enables you to perform 5-axis pocket machining. When this option is used, all the side faces of the pocket have to be defined as the **Drive surface**.

Click the **Morph pocket** button to display the **Morph pocket** dialog box. This dialog box enables you to define the parameters of the pocket machining.

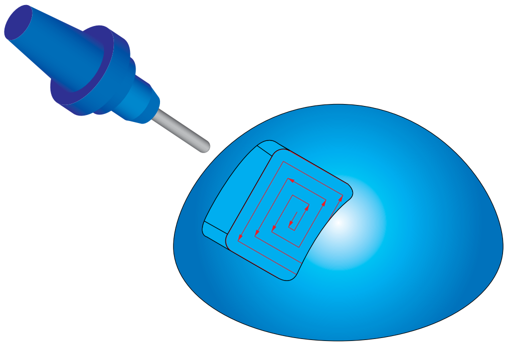




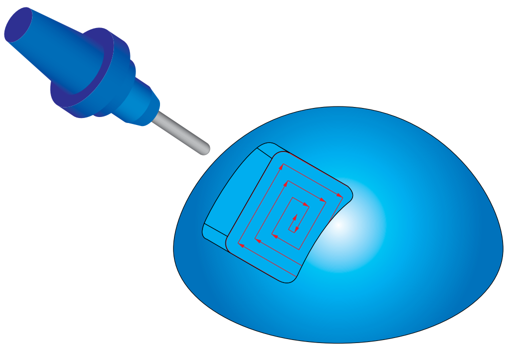
Move

This option enables you to define the direction of the pocket machining.

- **Outwards**. This option enables you to work in a pocket area starting from the middle of the pocket and cutting towards the outside border of the pocket.

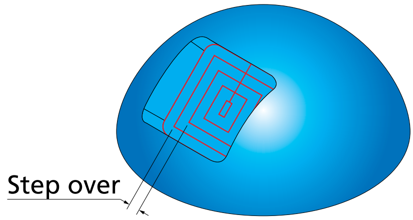


- **Inwards**. This option enables you to work in a pocket area starting from the outside border of the pocket and cutting towards the middle of the pocket.



Step over value

This parameter enables you to define the distance between two successive cuts in the pocket pass.



Pocket area

This option defines the pocket area to be machined.

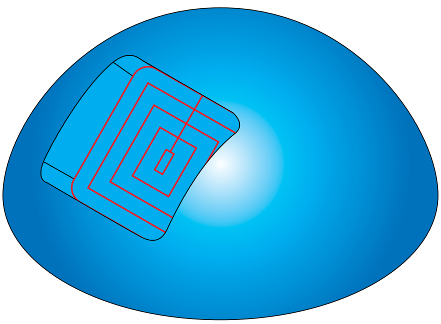
When the **Full** option is chosen, SolidCAM performs the machining of the whole pocket.

The **Determined by number of cuts** option enables you to machine a partial pocket area determined by the number of cuts (specified by the **Number of Cuts** parameter).

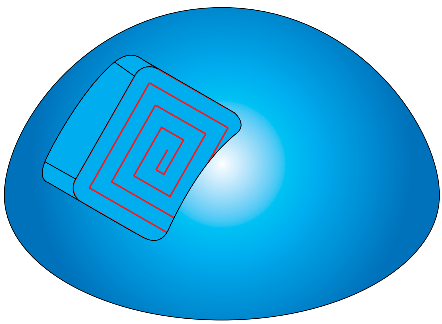
Spiral machining

When this check box is not selected, the machining is performed by a number of evenly spaced offsets connected with a straight tool movement.

When this check box is selected, SolidCAM generates a spiral tool path to machine the pocket.



Spiral machining is turned off



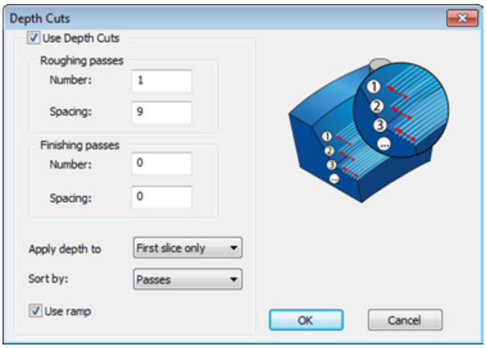
Spiral machining is turned on

10.4 Depth cuts

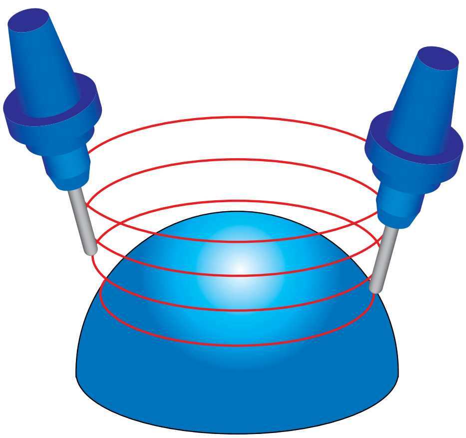
The **Depth cuts** option enables you to perform 5-axis rough and finish machining similar to the **Multi-passes** option. Using the **Multi-passes** option, SolidCAM generates roughing and finishing passes in the direction of the surface normal, independent of the tool orientation. The **Depth cuts** option enables you to perform roughing and finishing cuts in the direction of the tool axis.

SolidCAM enables you to use a combination of the **Multi-passes** and **Depth cuts** options for the machining. In this case, for each depth cut, SolidCAM generates a specified number of Multi-passes. For example, if you define 5 depth cuts and 10 Multi-passes, SolidCAM generates 5 * 10 = 50 cuts.

Click the **Depth cuts** button to display the **Depth cuts** dialog box. This dialog box enables you to define the parameters of the depth cuts.



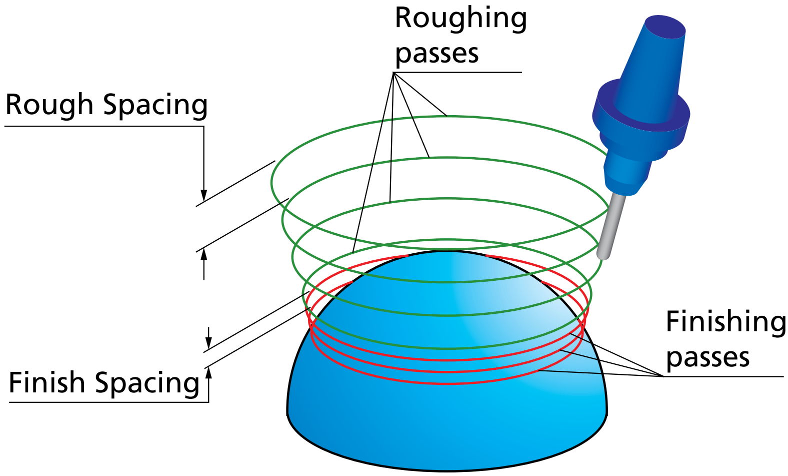
When this option is used, SolidCAM generates for each tool path point a number of offsets in the direction of the tool axis. Connecting these points, SolidCAM generates a number of evenly spaced cuts with the similar tool path shape.



Roughing and finishing passes

The **Roughing passes** section enables you to define the **Number** of roughing passes and distance between them (**Spacing**).

The **Finishing passes** section enables you to define the **Number** of roughing passes and distance between them (**Spacing**).



During the tool path calculation, SolidCAM generates the initial cutting pass located on the drive surface and then creates a specified number of finishing passes using the specified **Spacing**. After the finishing, SolidCAM generates a specified number of roughing passes. The distance between the last finishing pass and the first roughing pass is defined by the **Spacing** parameter in the **Finishing passes** section.

In the illustration above, SolidCAM generates three finishing passes with the specified finishing spacing and then generates four roughing passes with the specified roughing spacing. The distance between roughing and finishing parts of the tool path is equal to the finishing spacing.

Apply depth to

This option enables you to choose whether the defined depth cuts parameters will be applied to the entire tool path or only to the first pass/slice.

If **Multi-passes** is not enabled, choosing the **First pass only** option produces the same tool path as the **Whole tool path** option.

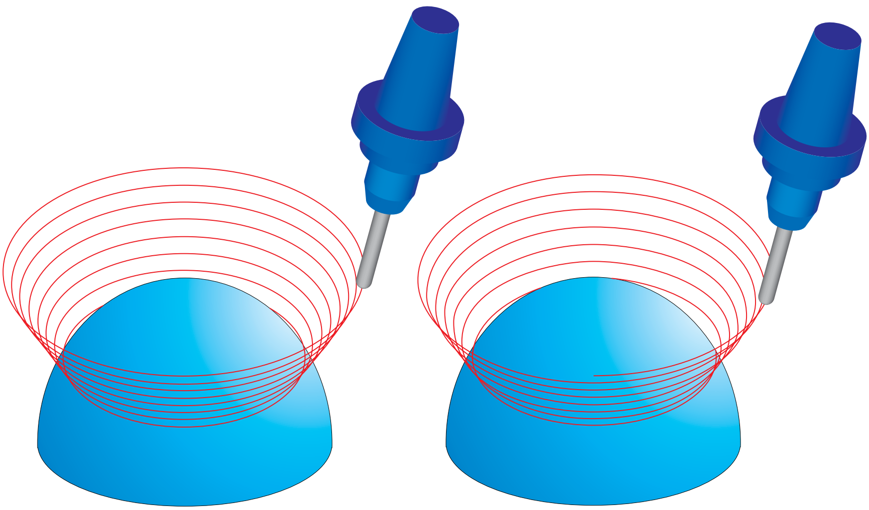
Sort by

This option enables you to define how the generated passes are sorted and linked.

- **Slices**. When this option is chosen, all the roughing and finishing offsets of the current cutting pass are performed before moving to the next cutting pass.
- **Passes**. When this option is chosen, all the cutting passes of the current offset level are performed before moving to the next offset level.

Use ramp

The **Use ramp** option enables you to perform a single spiral cutting pass instead of several separate passes generated by the **Depth cuts** option.



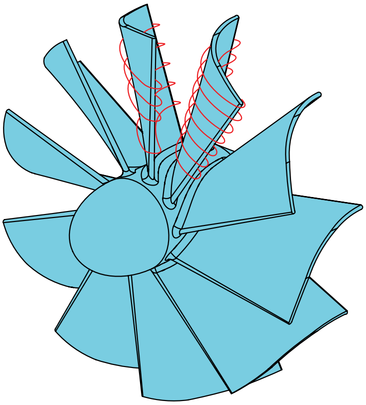
Use Ramp is turned off

Use Ramp is turned on

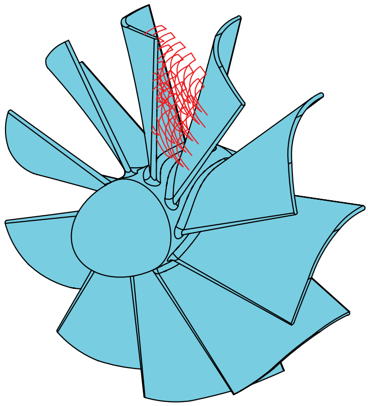
The image above illustrates the **Use ramp** option to convert a number of circular separate passes into a single taper spiral cutting pass.

10.5 Area roughing

The main purpose of the **Area roughing** strategy is impeller machining. In this strategy, the roughing tool path is created inside the initial tool path. E.g. the floor area between impeller blades can be machined using this strategy if the initial tool path describes the left and right side of the area limitations.

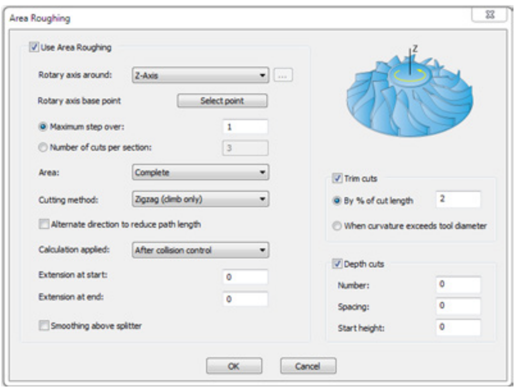


Initial tool path



Area roughing tool path

The **Area roughing** dialog box is displayed enabling you to define the parameters of the area roughing.



- **Rotary axis around**. This parameter defines the rotary axis. SolidCAM enables you to choose an axis of the Coordinate System (X, Y, Z) or define a rotary axis vector by an end point (the start point is automatically considered to be in the Coordinate System origin).

- **Rotary axis base point**. With this option, SolidCAM enables you to define the position of the rotary axis directly on the solid model.

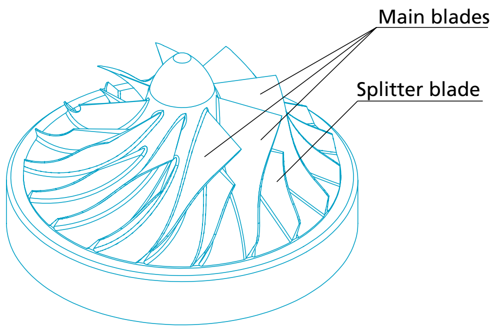
- SolidCAM enables you to define a number of cuts either by the **Maximum step over** parameter (the distance between two successive cutting passes) or by the **Number of cuts per section** parameter.

- SolidCAM enables you to machine the area enclosed between two main blades and containing a splitter blade. The **Area** option enables you to define the area where the machining will be performed.

  Complete. The machining is performed in the complete area between the two main blades.

  Left side. The machining is performed in the area between the left main blade and the splitter blade.

  Right side. The machining is performed in the area between the right main blade and the splitter blade.



- The **Cutting method** options enable you to define the passes direction and the way how the single passes will be connected into a complete tool path. The following options are available:

  One way (along rotary axis). With this option, the machining of the pass starts at the upper edge of the impeller floor face, continues along the blades and stops at the lower edge of the floor. Then the tool retracts to the start position of the next cutting pass.

  One way (along reversed rotary axis). With this option, the machining of the pass starts at the lower edge of the impeller floor face, continues along the blades and stops at the upper edge of the floor. Then the tool retracts to the start position of the next cutting pass.

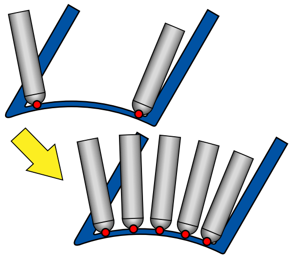
  Zigzag. With this option, the machining starts at the edge of the impeller floor face, continues along the blades to the other edge, steps over to the next cut at the same edge and continues machining to the first edge. The sequence for the cuts is from the left to the right.

  Zigzag (climb only). With this option, the machining begins in the center of the surface and progresses outwards to the sides.

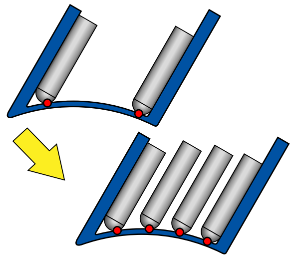
  - **Alternate direction to reduce path length**. With this option, SolidCAM changes the start position of the cut in order to minimize air cuts.

  This option is available only when the **Zigzag (climb only)** cutting method is chosen.

- **Calculation applied**. With this option, SolidCAM enables you to define when the calculation of the area roughing is performed. The area roughing calculation can be performed either before the tilting calculation (the **Before tilting** option) or after the collision control (the **After collision control** option). If the area roughing calculation is performed after the collision control, the resulting tool path is checked again for collisions.



Calculation is applied **Before tilting**



Calculation is applied **After collision control**

When the **After collision control** option is used, SolidCAM enables you to extend the tool path using **Extension at start** and **Extension at end** parameters.

- **Smoothing above splitter**. With this option SolidCAM enables you to create a morphed tool path in the area above the splitter. This smoothing is used for finishing operation.

  This option is available only if **After collision control** is selected in **Calculation applied**.

- **Trim cuts**. This parameter enables you to define the cut length of the cuts. Two options can be used for this:

  By % of cut length. This option enables you to determine the percentage of the tool path length that must be trimmed.

  When curvature exceeds tool diameter. This option enables you to trim the tool path while it is moving around the upper radius of the blade, when curvature of the blade gets bigger than the tool radius.

- **Depth cuts**. This option enables you to copy the tool path pattern into tool contact line direction. This option generates a collision free tool path pattern and upper cuts.

  Number. This parameter defines the number of total cuts.

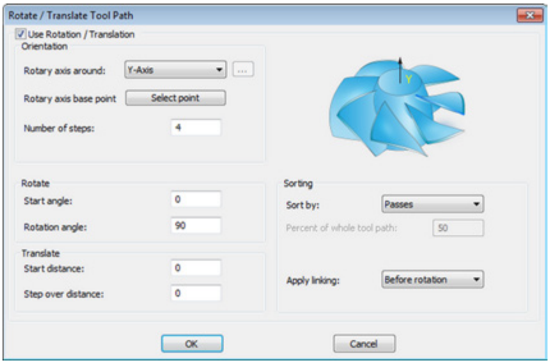
  Spacing. This parameter defines the number of depth cuts for area roughing.

  Start height. This parameter defines the start distance from tool path and depth cuts to their original position.

  These options are available only if **After collision control** and **Depth cuts** are selected.

10.6 Rotate and translate

The **Rotating** strategy is useful for parts with multiple identical elements arranged in a circular pattern. Instead of adding a separate operation and defining the same parameters for each of these patterns, you can have the same tool path repeated a given number of times by rotation around a specific axis.



The **Rotate/Translate Tool Path** dialog box enables you to define the parameters of rotation.

Orientation

- **Rotary axis around**. This option provides you the choice of the axis around which your tool path will be rotated. You may choose between the X-, Y- or Z-axis of the current Coordinate System or define a rotary axis vector by an end point (the start point is automatically considered to be in the Coordinate System origin).

- **Rotary axis base point**. This option enables you to define the position of the rotation axis. When you click the **Select point** button, the **Select point** dialog box is displayed with the coordinates of the point you pick on the model.

- **Number of steps**. This parameter enables you to define the number of instances of the circular pattern. In other words, it defines how many times the initial tool path will be repeated around the rotation axis.

Rotate

- **Start angle**. This parameter enables you to define the rotation angle for the first tool path instance of the circular pattern.

- **Rotation angle**. This parameter enables you to define the angle between two adjacent instances of the circular pattern.

Translate

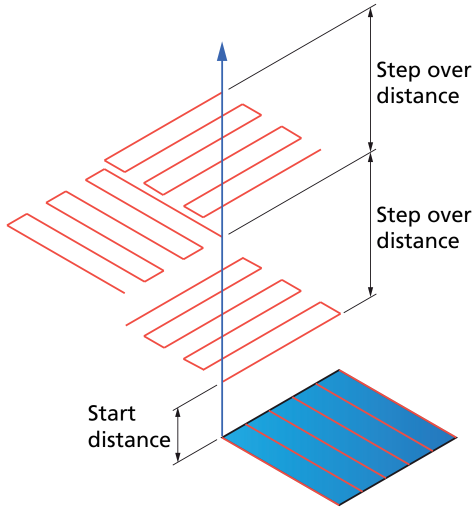
This section contains two parameters that enable you to transform a 5-axis tool path:

- **Start distance**

  This parameter defines the distance between the initial tool path instance and the next one in the transformed pattern.

- **Step over distance**

  This parameter defines the distance between two adjacent tool path instances of the transformed pattern.



These distances are measured along the transformation axis defined in the **Rotary axis around** list.

Sorting

- **Sort by**. This option enables you to choose whether the whole tool path will be rotated or only a certain part of it. The following options are available:

  Complete tool path. With this option the whole tool path will be rotated.

  Passes. With this option the whole tool path will be rotated. The resulting tool path will be sorted and linked by passes.

  Slices. With this option the whole tool path will be rotated. The resulting tool path will be sorted and linked by slices.

  Partial tool path. With this option the portion of tool path specified by a percentage is rotated. The percentage is specified by the **Percent of whole tool path** parameter.

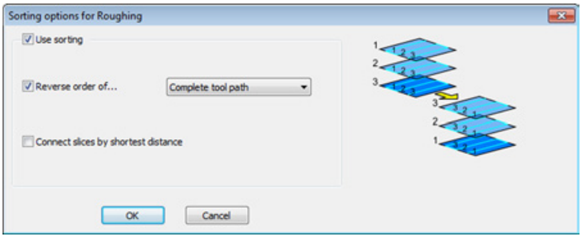
- **Apply linking**. This option enables you to link the tool path either before or after the rotation.

  Before rotation. With this option, SolidCAM generates the initial tool path, links it and then performs the rotation. In this case the link movements in all the rotated instances of the tool path are the same.

  After rotation. With this option, SolidCAM applies linking after the tool path rotation. It is recommended to use this option with the collision control activated to avoid possible collisions in the link movements.

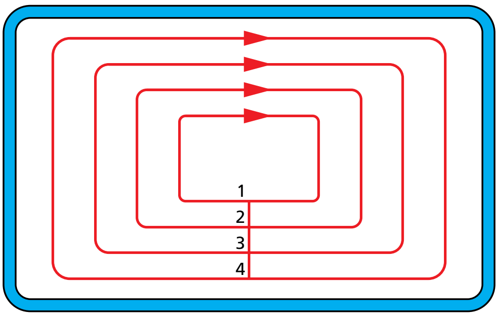
10.7 Sorting

The **Sorting** button displays the **Sorting options for Roughing** dialog box that enables you to define the sorting of the tool path passes.

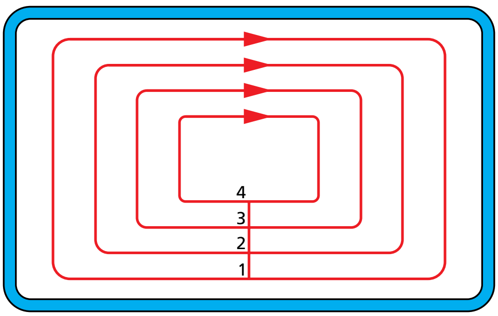


10.7.1 Reverse order of passes/tool path

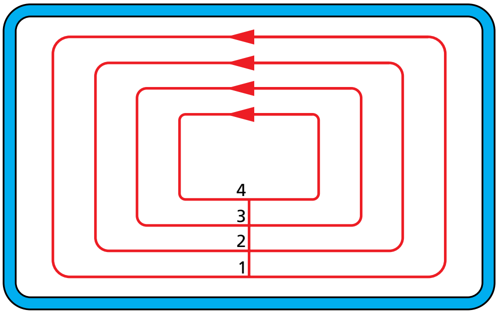
When this check box is not selected, the tool path passes are performed in the default order and in the direction of the geometry.



When the check box is selected and the **Passes** option is chosen from the list, the tool path passes are performed in the reversed order.

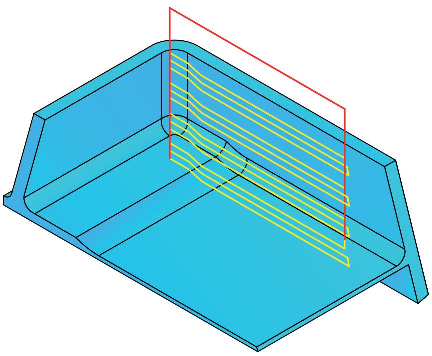


When the check box is selected and the **Complete tool path** option is chosen from the list, the tool path passes are performed in the reversed order and direction.



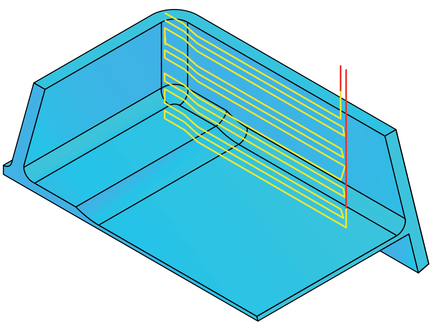
10.7.2 Connect slices by shortest distance

When this check box is not selected, the connection between tool path slices is performed through the Clearance level: after a certain slice has been machined, the tool goes up to the Clearance level



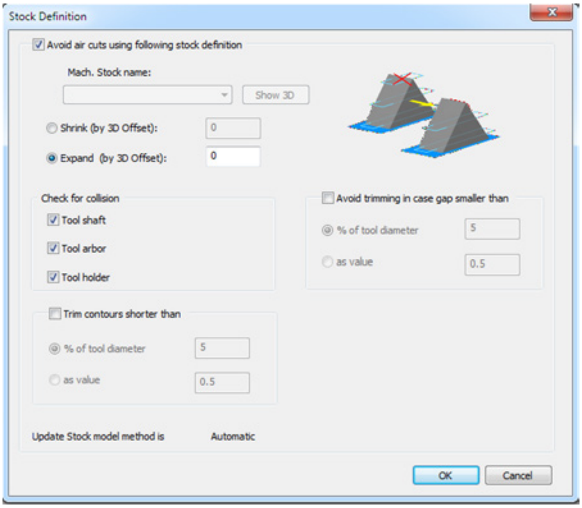
and then descends to machine the next slice.

When this check box is selected, the tool path slices are connected by lines automatically calculated by SolidCAM, so that the distance between the end of one slice and the start of the next one is minimal.



10.8 Stock definition

This button displays the **Stock definition** dialog box that enables you to define trimming of the Sim. 5-axis passes to the pre-machined or casting stock faces to avoid unnecessary air cutting.



When the **Avoid air cuts using following stock definition** option is chosen, SolidCAM calculates the **Updated Stock model** after all the previous operations. SolidCAM automatically compares the updated stock model with the operation target geometry and machines the difference between them.

SolidCAM provides you with two modes for the Updated Stock model calculation: **Automatic** and **Manual**. For CAM-Parts, the mode can be specified in the **Updated Stock calculation** page of the **Part Settings** dialog box. In the **Automatic** mode, SolidCAM automatically calculates the Updated Stock model for the previous operations. In the **Manual** mode, SolidCAM enables you to manually save the Updated Stock model after the **SolidVerify** simulation and use it for avoiding air cuts. SolidCAM notifies you about chosen Update Stock model method using the **Update Stock model method is** parameter.

For more information about Updated Stock model methods refer to the **SolidCAM Milling User Guide** book.

Machined Stock Name

This option enables you to choose the previously generated **Updated Stock model** for the tool path calculation.

This field is active only when the **Manual** method of the Updated Stock model calculation is used.

The **Show 3D** button displays the difference between the updated stock model and the target geometry used in the operation.

Shrink/Expand

SolidCAM provides you with the possibility to shrink/expand the stock model used for avoiding air cuts. The defined **Shrink/Expand** value enables you to define the 3D offset by which the stock model will be modified.

Check for collision

SolidCAM enables you to prevent the collisions between the tool/holder components and the machined stock model.

- **Tool shaft**

  Select this check box to check for collision between the tool shaft and the machined stock.

- **Tool arbor**

  Select this check box to check for collision between the tool arbor and the machined stock.

- **Tool holder**

  Select this check box to check for collision between the tool holder and the machined stock.

Trim contours shorter than

This option enables you to exclude the contours of the tool path that are shorter than a specified length.

When this check box is selected, the contours whose length is smaller than the specified contour length value are excluded from the tool path. The specified contour length can be defined as percentage of tool diameter or as a value.

Avoid trimming in case gap smaller than

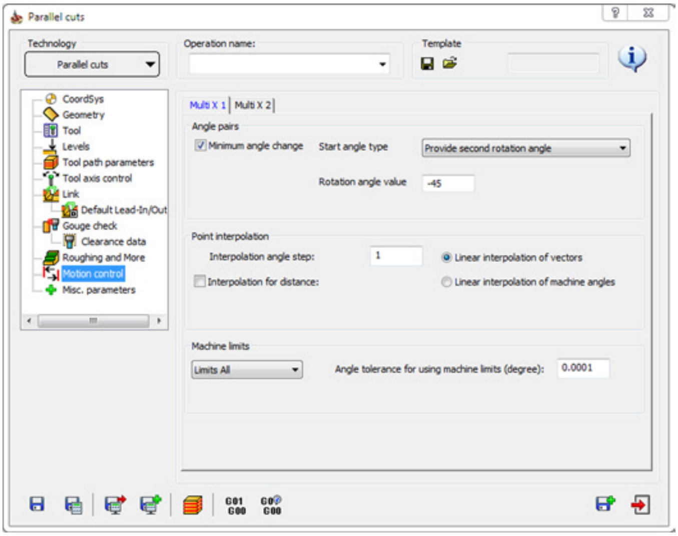
This option enables you to ignore small gaps on the tool path that are shorter than a specified length.

When this check box is selected, the gaps whose length is smaller than the specified length value are ignored, and linking is not applied to them. The specified gap length can be defined as percentage of tool diameter or as a value.

Motion control

11

Using the parameters of the **Motion control** page, you can optimize the calculated tool path according to the kinematics and special characteristics of your CNC-machine.



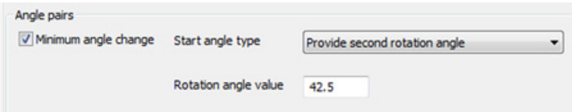
The default values of these parameters can be defined in the VMID file of your CNC-machine.

11.1 Angle pairs

For a 5-axis machine, the tool axis vector can always be mapped into two different angle pairs. During the tool path generation, SolidCAM calculates for each tool axis orientation both of these two angle pairs; only one of the two has to be chosen for the GCode generation. The following options enable you to choose the angle pair.

Minimum angle change

When this check box is selected, the necessary angle pair is determined automatically in such manner that the angle deviation from the previous tool axis orientation is minimal.



The **Start angle type** option enables you to define the control over the solution that will be chosen for the first angle pair. The following options are available:

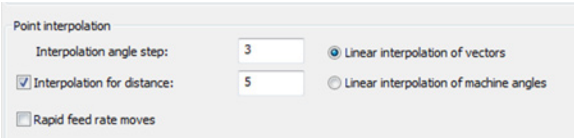
- **Choose between two solutions**. With this option you can specify the necessary solution that will be used for the first angle pair. It enables you to choose either the first solution (the **First angle pair** option) or the second solution (the **Second angle pair** option).
- **Provide first rotation angle**. In this case, SolidCAM chooses an angle pair where the first rotation angle (rotation around the first axis) is closer to the value determined by the **Rotation angle value** parameter.
- **Provide second rotation angle**. In this case, SolidCAM chooses an angle pair where the second rotation angle (rotation around the second axis) is closer to the value determined by the **Rotation angle value** parameter.

First/Second angle pair

Some machines can only use one of the angle pairs due to mechanical limitations. In this case, the **Minimum angle change** option must be deactivated and the angle pair will then be chosen as the **First angle pair** or **Second angle pair**.

These options are not available when the **Minimum angle change** check box is selected.

11.2 Point interpolation



The point interpolation provides the ability to create intermediate points by setting a certain maximum angle step distance (for 5-axis motions) or by splitting long linear motions (3-axis and 5-axis tool paths) for feed rate moves and rapid rate moves.

Interpolation angle step

Using this parameter, SolidCAM enables you to interpolate the angular movements. A new interpolated tool axis position is defined at each angle, defined by the **Interpolation angle step** parameter.

Interpolation for distance

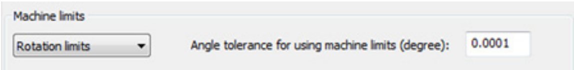
Using this option, SolidCAM enables you to perform interpolation for the linear tool movements. When this option is active, a new interpolated tool position is defined at each distance, defined by the **Interpolation for distance** parameter. For example, when the linear tool movement is performed from 0, 0, 0 to 0, 0, 100 and the **Interpolation for distance** option is used with the Distance value of 10, SolidCAM adds 9 tool positions between start and end positions (0, 0, 10, then 0, 0, 20 etc.).

- **Rapid feed rate moves.** When this option is selected, a new interpolated tool position is defined also for rapid moves.

This option is available only when **Interpolation for distance** is selected.

The interpolation between two end points of a segment can be performed either in the shortest possible way or gradually, considering the machine kinematics. In the first case, you can choose the **Linear interpolation of vectors** option, and the motion between these vectors of the end point is performed in a flat plane. In the second case, you can choose the **Linear interpolation of machine angles** option, so the motion between two vectors of the end point is no longer in a flat plane.

11.3 Machine limits



With this option, SolidCAM enables you to use the machine limits defined within the machine definition to limit the tool path movements in translation and/or rotation axis.

The following options of machine limits use are available:

- **No limits**

  All the machine limits defined in the machine definition are ignored.

- **Translation limits**

  SolidCAM uses the machine limits defined in the machine definition for translation movements.

- **Rotation limits**

  SolidCAM uses the machine limits defined in the machine definition for rotation movements.

- **All limits**

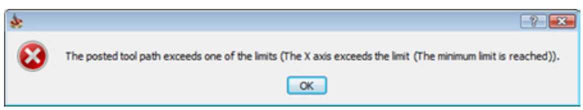
  SolidCAM uses the machine limits defined in the machine definition for both translation and rotation movements.

When machine limits are used, the calculated tool path is checked in order to avoid exceeding the machine limits. The check is performed using the angle tolerance defined by the **Angle tolerance for using machine limits** parameter.

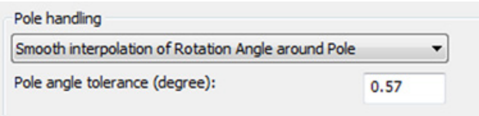
When the **Minimum angle change** option is used together with machine limits, SolidCAM uses the machine limits to choose the necessary angle pair. Consider that the B-axis movements are limited between 0 and 90°. For each tool axis orientation, SolidCAM offers two angle pairs in the calculated tool path. According to the machine limits, only pairs located in the defined range will be chosen. For example, SolidCAM offers you both B=-30° and B=+30° as possible solutions. Taking into account the machine limits, the solution B=-30° is not acceptable because it exceeds the defined range; the solution of B=+30° is in range and will be chosen by SolidCAM for further post-processing. If both of the offered angle pairs are in the range of the machine limits, the angle pair with the smallest variation of the angle (from the previous position) is used.

If the machine limits are exceeded, SolidCAM displays an error message.

The default value for the **Machine limits** option is defined in the VMID file of the CNC-machine.



11.4 Pole handling



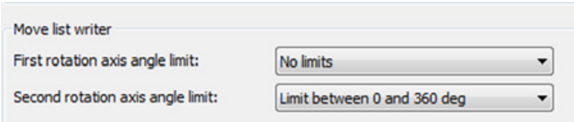
Generally, in a 5-axis machine, the tool axis vector can be mapped into two different angle pairs. There is only one exceptional case when the rotation is performed around a coordinate system axis (rotation axis), and the tool axis is parallel to the same rotation axis. In this case, any rotation angle value properly describes the tool position. Therefore, the rotation angle value can be arbitrary. Such tool axis orientation is referred to as «singularity» or «pole».

SolidCAM enables you to detect such pole areas and handle them with the following methods:

- **Freeze Angle around Pole.** In the pole areas, the arbitrary rotation angle is «frozen» when the tool axis orientation is parallel to the rotation axis.
- **Use Rotation Angle around Pole to stay within linear axis limits.** If some areas of the tool path cannot be reached by the linear axes, this option can adjust the linear axes and use the rotation axes to substitute linear motions. E.g., if you machine a cube on two opposite faces, left (-X) and right (+X), on a head-table machine, and machine limits do not allow -X movements, you can use this option to rotate the table axis to flip the cube.
- **Linear interpolation of Rotation Angle around Pole.** In 5-axis machining, the tool can be vertical and any value of the rotary axis (usually C) can be used if the X- and Y-axis values are changed accordingly, i.e. the C-axis value can be chosen arbitrarily. The linear interpolation is distributed according to the number of intermediate points and their relative distance from each other.
- **Smooth interpolation of Rotation Angle around Pole.** The smooth interpolation is similar to the linear interpolation, with the difference that the change of C-axis at first non-vertical position and second non-vertical position is performed smoothly.
- **Force table rotation available for pole handling strategies.** This option is used on 4-axis table, 5-axis table-table and 5-axis head-table machines. In some particular cases of 4-axis tool path, when the orientations become closer to a 3-axis tool path, the post processor makes only translation/linear moves. This option is also suitable for mill-turn machines. Also, instead of moving the tool in XYZ, the Motion control can rotate the part, while the tool is fixed.

The **Pole angle tolerance** value defines the maximal angular deviation of the tool axis from the rotation axis to consider the tool axis parallel to the rotation axis.

11.5 Move list writer



Some CNC-machine controllers have a limitation of acceptable angular coordinates. When an angular coordinate in the GCode exceeds such limitation, an error is returned by the controller. Almost all controllers have limitation from 0 to 360° or from -180° to 180°.

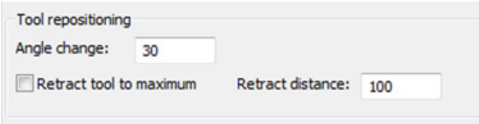
The **Move list writer** options enable you to define the angle limits for the output tool paths in order to generate a tool path compatible with the angular limits of the specific CNC-machine controller.

The **First rotation axis angle limit/Second rotation axis angle limit** options enable you to define the angle limit of the first and rotational axes respectively.

The following options are available:

- **No limit.** In this case there are no angle limitations of the output tool path. The angles can be in the range of -∞ to +∞.
- **Limit between 0 and 360 deg**. With this option, the angle coordinates in the output tool path are limited by the range from 0 to 360°.
- **Limit between -180 and 180 deg.** With this option, the angle coordinates in the output tool path are limited by the range from -180° to 180°.

11.6 Tool repositioning



These parameters enable you to control the angular tool movements in the calculated tool path.

Angle change

This parameter enables you to define the maximal angle variation between two successive tool positions. If the angle variation is greater than the specified value, a retract movement is added.

- **Retract distance**

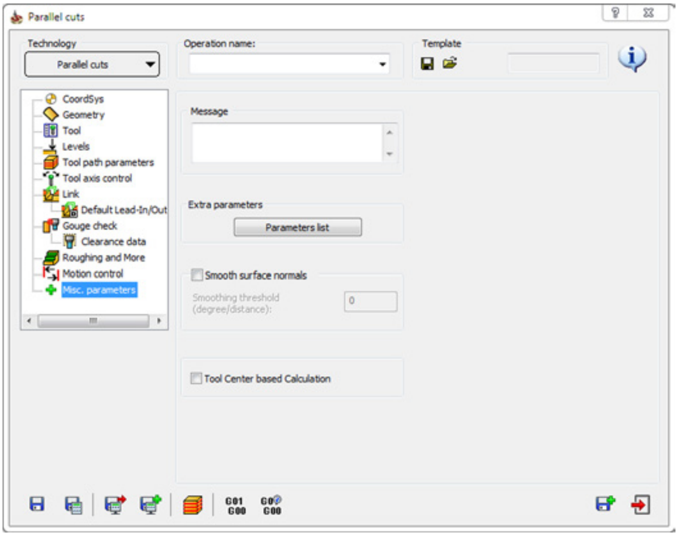
  This value determines the distance of the retract movement that is performed when the angle change between two successive tool positions exceeds the **Angle change** value. For example, when the **Angle change** parameter is set to 100°, and the C-axis orientation at the first position is 10° and at the second position is 170°, SolidCAM considers such angle variation as inadmissible and performs a retract motion.

- **Retract tool to maximum**

  When the angle change between two successive tool positions exceeds the **Angle change** value, the retract movement can be performed to the maximal distance defined by the machine limit.

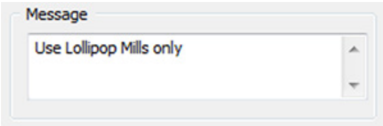
12 Misc. parameters

The **Misc. parameters** page enables you to define a number of miscellaneous parameters and options related to the 5-axis tool path calculation.



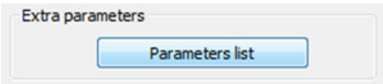
12.1 Message

In this field, you can type a message that will appear in the generated GCode file.

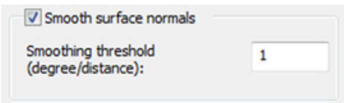


12.2 Extra parameters

The **Extra parameters** option enables you to use special operation options implemented in the post-processor for the current CAM-Part. The **Parameters List** button enables you to display the list of additional parameters defined in the post-processor.



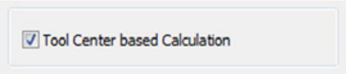
12.3 Smooth surface normals



Using this option, SolidCAM enables you to smooth the drive surface normals.

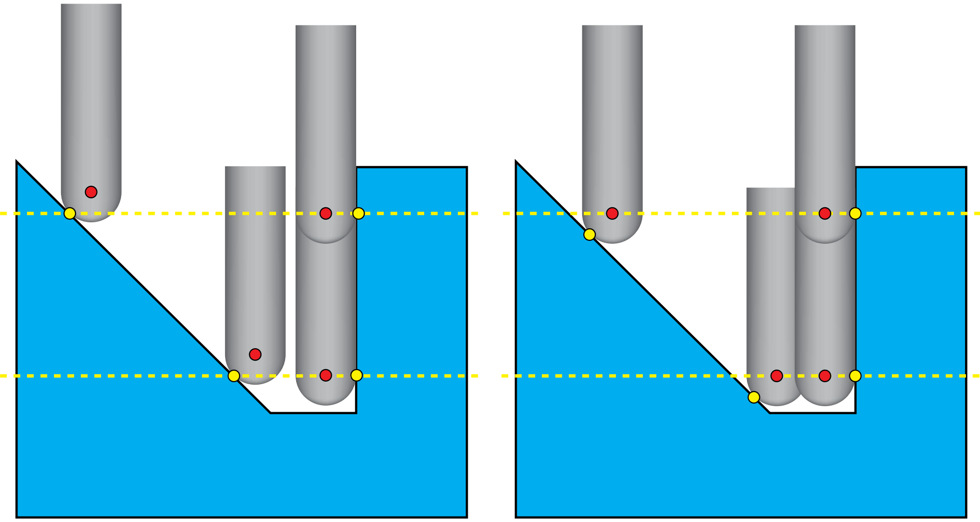
- **Smoothing threshold**. This value defines the limit of the surface normal angular variation. If along the drive surface slice the surface normal is changing more than the specified **Smoothing threshold** value per distance unit (inch or mm), SolidCAM defines for this segment a new surface normal calculated using a linear interpolation from the surface normals at the start and end segment points.

12.4 Tool center based calculation



This option enables you to perform the tool path calculation based on the tool center.

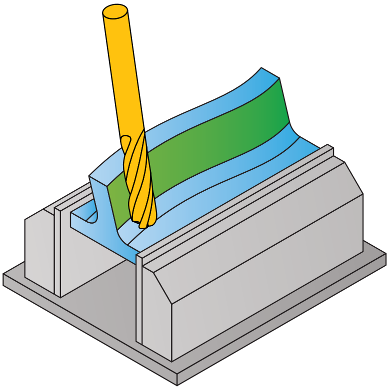
The illustration shows the use of the **Tool Center based Calculation** option for Constant-Z machining. When the option is turned off, the contact points between the tool and machined surface are located at the specified Z-levels. When the **Tool Center based Calculation** check box is selected, the tool center points are located at the specified Z-levels.



Check box is not selected   Check box is selected

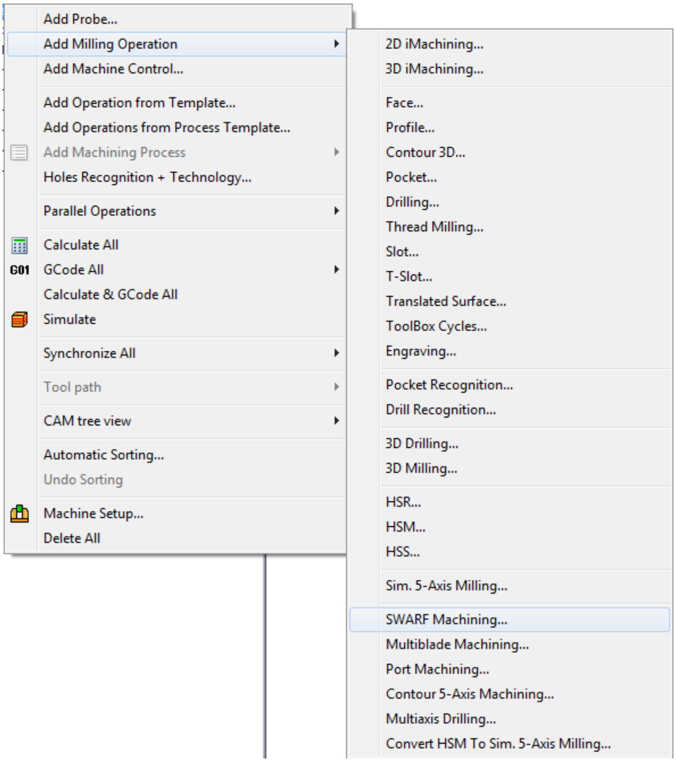
13 SWARF Machining

SolidCAM provides you with the **SWARF Machining** operation. The **SWARF Machining** strategy provides you with a number of advantages in steep areas machining. In SWARF operation, machining is performed by the tool side. The contact area between the tool and the workpiece is a line, therefore a better surface quality can be achieved with a minimum number of cuts.

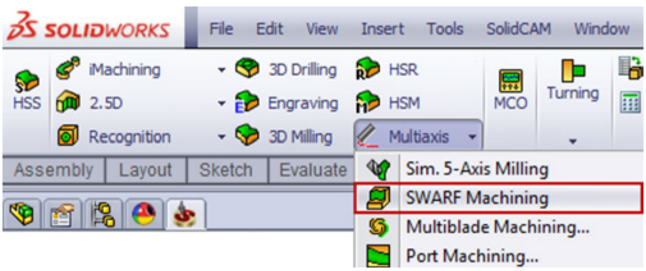


13.1 Adding a SWARF Machining Operation

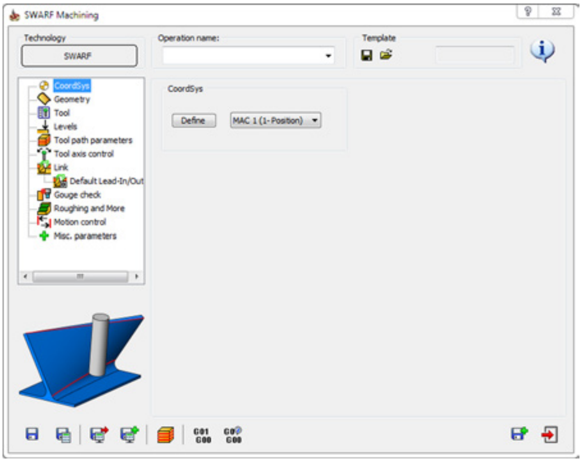
To add a **SWARF Machining Operation** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose the **SWARF Machining** command from the **Add Milling Operation** submenu.



You can also choose the **SWARF Machining** command from the **Multiaxis** menu on the **SolidCAM Operations** toolbar.



The **SWARF Machining** dialog box is displayed.

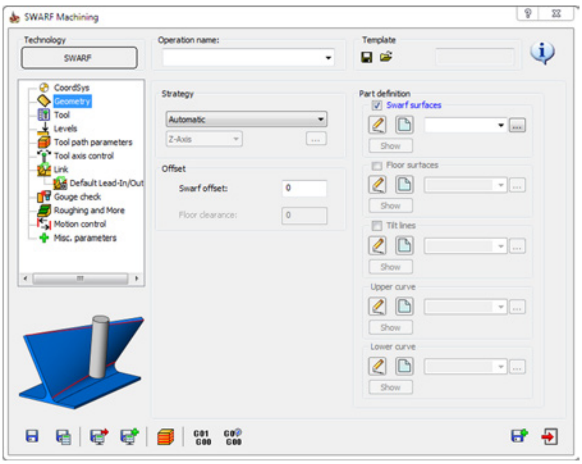


13.2 Coordsys

The **Coordsys** page of the **SWARF Machining** dialog box is similar to the **Coordsys** page of other Sim 5-axis Milling operations.

For more information, refer to chapter **2**.

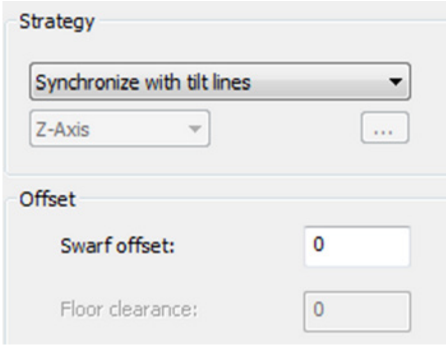
13.3 Geometry

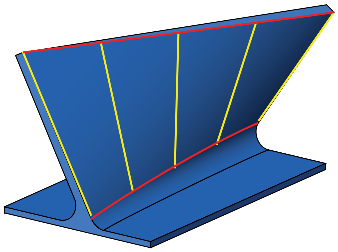


The **Strategy** section enables you to define the strategy of the **SWARF Machining**. SolidCAM offers you the following strategies:

13.3.1 Synchronize with tilt lines

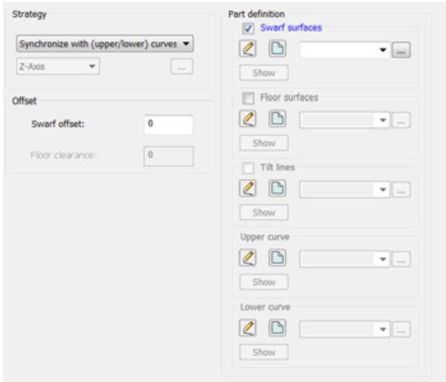
This strategy enables you to perform machining, while the tool axis is aligned to the **Tilt lines** along the **Upper curve** and **Lower curve**. SolidCAM automatically interpolates the tool axis between the tilt lines. Since this strategy provides manual control over the lead and lag angles, it can be used when all other strategies fail.

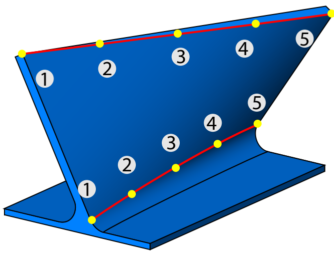




13.3.2 Synchronize with (upper/lower) curves

This strategy enables you to divide the tool path along the **Upper** and **Lower curves** into equidistant length steps. Then the tool axis is aligned to each pair of steps.

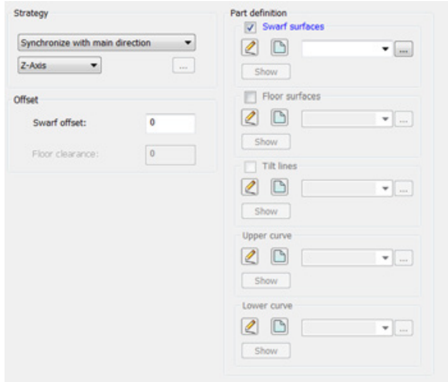


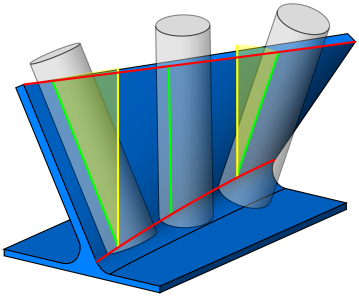


If the **Upper** and **Lower curves** have different lengths and shapes, the synchronized tool path is not always distributed correctly.

13.3.3 Synchronize with main direction

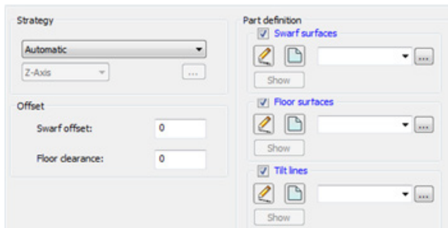
This strategy enables you to generate a tool path with a tool axis located as close as possible to the defined axis. In this case, the tool mainly tilts away to the side, while tilting around the axis is minimized.

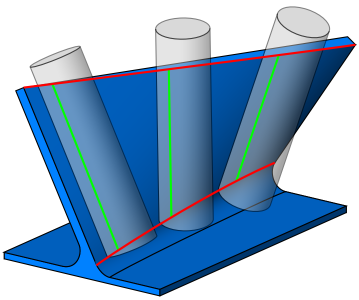




13.3.4 Automatic

This strategy enables you place the tool onto a swarf surface in such a way as to achieve a line contact between the tool and surface.





The **Swarf surfaces** must be selected as the geometry.

The tool tilts only to the side but always sticks to the main direction in one orientation.

13.3.5 Shortest distance

This strategy enables you to align the tool with upper and lower curves by using the shortest distance between these two curves.

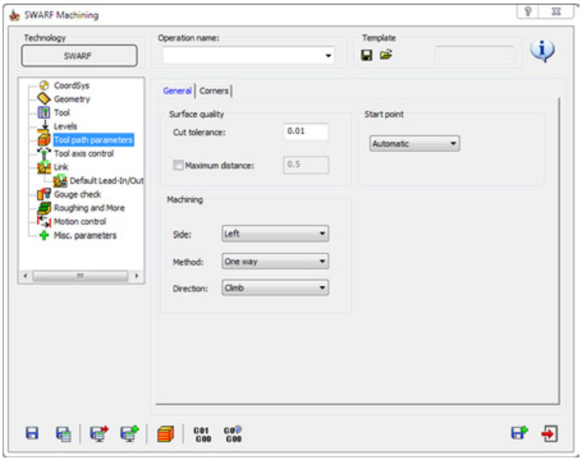
Depending on the strategy chosen, the following geometries can be defined in the **Part definition** section:

- **Swarf surfaces**. This section enables you to define the surfaces where the machining is performed in the **Automatic** strategy.
- **Floor surfaces**. This section enables you to define the **Floor surfaces** that will be avoided during the machining.
- **Tilt lines**. This section enables you to select the lines where the machining should start in the **Synchronize with tilt lines** strategys.
- **Upper curve.** This section enables you to define the upper contact point of the tool. It should be the upper edge of the swarf surface.
- **Lower curve.** This section enables you to define the lower contact point of the tool. It should be the lower edge of the swarf surface.

Offset

- The **Swarf offset** field enables you to define the offset for the **Swarf surfaces**.
- The **Floor clearance** field enables you to define the offset for the **Floor surfaces**. The machining is performed at the specified distance from the **Floor surfaces**.

13.4 Tool path parameters



For **SWARF Machining** definition, this page enables you to define a number of tool path parameters.

13.4.1 General parameters

The **General** tab displays the major parameters that affect the generation of tool path parameters.

Surface quality

This section enables you to define the parameters that affect the surface finish quality.

- **Cut tolerance.** Ths parameter defines the tool path accuracy (see topic **6.1.1**).
- **Maximum distance.** This parameter defines the maximum distance between two consecutive cuts (see topic **6.1.2**).

Machining

This section enables you to define the order and direction of the cuts.

Side

The **Side** option defines the position of the tool relative to the cutting direction or the geometries. **Right** and **Left** side are defined on open contours relative to the chaining direction of the lower curve. **Inside** and **Outside** options are defined for closed contours enabling machining inside or outside of a contour. **Autodetect** option enables the tool to detect the machining side automatically without needing inputs to specify the cutting side.

Method

The **Method** option (see topic **6.2.1**) enables you to define how the cuts are connected. SolidCAM provides you with two possibilities: **Zigzag** or **One Way**.

Direction

When the **One Way** option is chosen for the **Method**, SolidCAM enables you to choose the direction of cuts from the **Direction** list. This list offers **Climb, Conventional,** or **Follow lower curve** direction for the cutting passes.

Start point

The **Start point** section (see topic **6.2.7**) enables you to define the start point of the tool tip on the lower curve and the tool axis orientation defined by the start point on the upper curve.

The **Exact** option enables you to define the start point as the bottom and upper curves chaining start point.

The **Automatic** option enables you to define the start point automatically. If the curves are closed contours, the start point on the bottom curve is the middle point of the longest tool path segment. The start point on the upper curve is the nearest point to the start point of the bottom curve. If the curves are open contours, the start point on the bottom curve is the start point of bottom curves chaining. The start point of the upper curve is the upper curves chaining start point.

The **2 points** option enables you to pick the points on the surface using the **Limit Cuts Between 2 Points** dialog box.

The **Tilt line** option enables you to define the start point as a tilt line. Clicking the button enables you to define the tilt line coordinates by typing them in the **Start point Tilt line** dialog box or picking the points directly on the solid model.

The **One point** option enables you to define one point as a tilt line. Clicking the button enables you to define one point coordinate by typing it in the **Start point One point** dialog box or picking the point directly on the solid model.

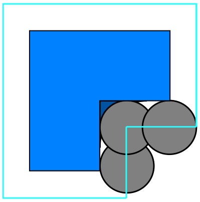
13.4.2 Corners parameters

The **Corners** tab enables you to define the tool movement in the corners.

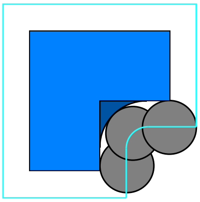
Inside corners

This section enables you to define how inner corners are machined.

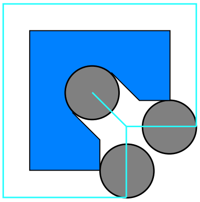
The **Sharp corner** option enables you to bring the tool as close as possible into the corner resulting in a sharp inner corner tool path.



The **Round corner** option enables you to create a fillet in the inner corner. You can also specify a fillet radius.



The **Relief groove** option enables you to apply a relief cut into the corner. You have to specify the length of the relief cut. The cut is placed at the bisector position.



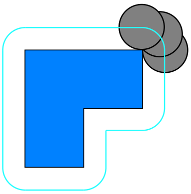
- **Radius.** This parameter enables you to add an additional fillet to the inner corner.
- **Length.** This option enables you to apply a relief value to the inner corner.
- **Detection angle.** This parameter enables you to define a threshold value starting from which the corner option is used. If the threshold is exceeded, the option is applied.

This parameter is available for the options of **Round corner** and **Relief groove**.

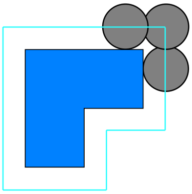
Outside corners

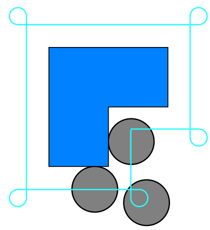
This section enables you to define how outer corners are machined.

The **Roll around** option enables you to roll the tool around the outer corner.



With the **Sharp corner** option the tool overruns straight in the corner and connects on the other side.



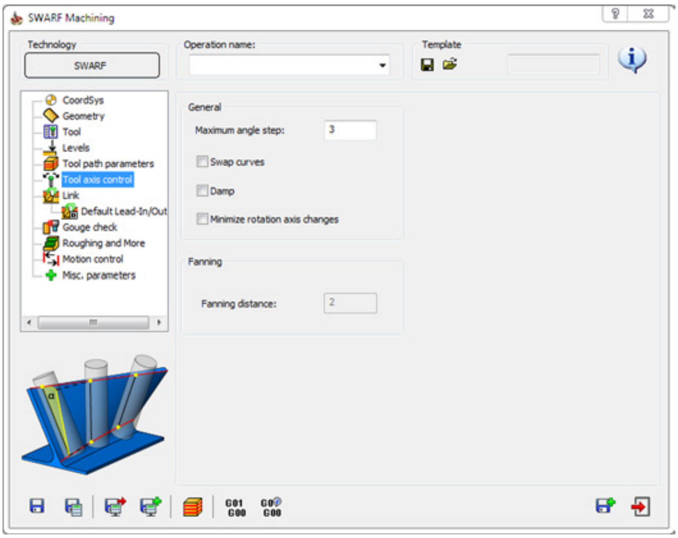


The **Loop** option enables you to create a loop at the outer corner.

- **Radius.** This parameter enables you to define the loop radius that can be applied to the outer corner.
- **Detection angle.** This parameter enables you to define a threshold value starting from which the corner option is used. If the threshold is exceeded, the option is applied.

This parameter is available for the options of **Sharp corner** and **Loop**.

13.5 Tool axis control



The following parameters enables you to control the tool axis orientation during the SWARF machining.

Maximum angle step

The **Maximum angle step** parameter enables you to define the maximum allowed angle change between two consecutive tool path points.

Swap curves

This option allows you to exchange the upper and lower curves for defined **Swarf surfaces**.

This option is available only if **Swarf surfaces** are defined in the **Part definition** section of the **Geometry** page.

Damp

This option enables you to apply axial damping to the tool. This means that the tool path is smoothed in order to avoid vertical jumps.

Minimize rotation axis changes

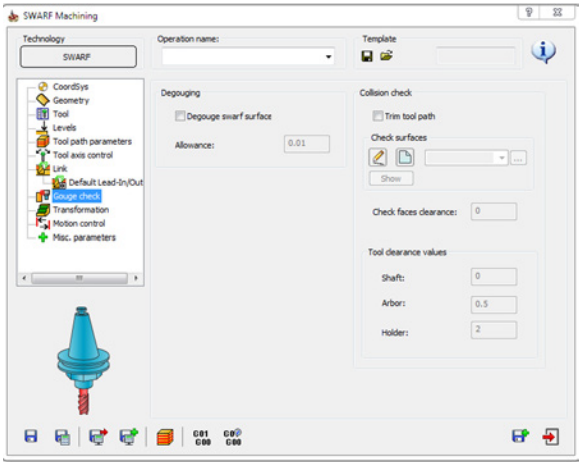
In a situation when the tool is located in the center of the machine, the machine rotary axis can rotate very fast, and singularity occurs. This option enables you to minimize the axis changes, providing smooth tilting of the tool along the surface.

Fanning distance

If the upper and lower curves have different length, the tool cannot cut with the flute full length, therefore only the tool tip moves. This movement is called fannining. To avoid the tool stoping at the shorter curve, you can set a fanning distance. When the tool arrives to the point located at the specified distance from the end of the curve, it starts to tilt and moves further with the flute full length.

This option is available only for the strategy of **Shortest distance**.

13.6 Gouge check



The **Gouge check** page enables you to automatically detect and avoid the possible collisions between the tool, the tool holder, and the workpiece.

Degouging

The **Degouge swarf surface** option enables you avoid the tool collision with the swarf surface by retracting the tool in the direction orthogonal to the contact line between the upper and buttom curves. The accuracy of degouging is defined in the **Allowance** field.

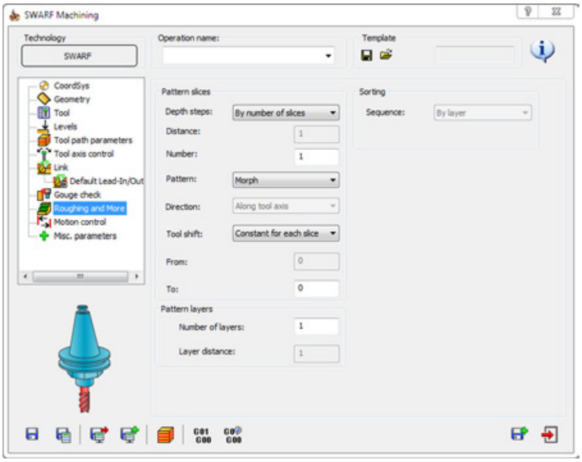
Collision check

The **Trim tool** path section enables you to trim out the tool path segments that collide with the selected **Check surfaces**. The offset from the checked faces can be defined in the **Check faces clearance** field.

- **Tool clearance values**

  The **Shaft**, **Arbor** and **Holder** parameters define the offsets applied to the corresponding parts of the tool.

13.7 Roughing and More



SolidCAM provides you with the following options to control the rough Swarf Milling:

13.7.1 Pattern slices

This section enables you to perform the machining in a single slice or in multiple slices. The machining area should be cut with multiple step depths in case the tool flute has short length. When the number of slices it set to **1**, a single slice will be generated at the bottom curve.

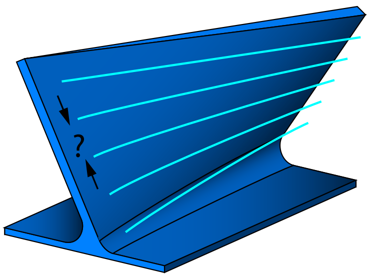
Make sure that the flute length is sufficient for this cut.

Depth steps

This option enables you to define multiple cuts along the tool axis direction.

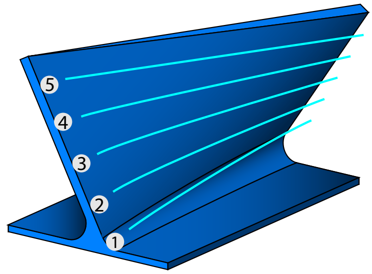
- **By slice distance**

  This option enables you to define the **Distance** between two consecutive slices.



- **By number of slices**

  This option enables you define the **Number** of slices between two curves.



Pattern

This parameter enables you to create the tool path pattern using the options of **Morph**, **Step from top**, and **Step from bottom**.

- **Morph**

  In this option the tool path is created as a morph between the upper and bottom edge.

- **Step from top**

  In this option the tool path pattern is parallel to the upper edge.

- **Step from bottom**

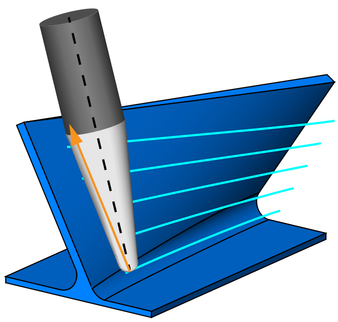
  In this option the tool path pattern is parallel to the bottom edge.

Direction

The multiple slices are copies of the initial slice. Their direction can be defined by two options. They can either be along the tool axis or along the contact line. In case that a conical tool is used, the retraction along the tool axis is not applicable. The tool would leave the surface in the upper cuts due to the conic angle.

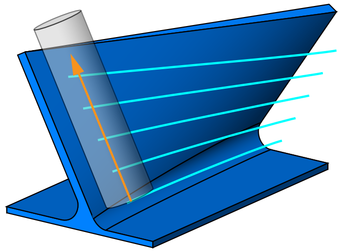
- **Along contact line**

  This option enables you to set the conical tool in contact with the actual swarf surface. The material is machined only downwards.



- **Along tool axis**

  This option enables the tool retraction into tool axis direction or tool contact line while performing the depth steps. The multiple slices are equidistant between the upper and the bottom curves.



Tool shift

- **Constant for each slice**

  In this option the axial shift is performed with a constant value for each slice.

- **Gradual for each slice**

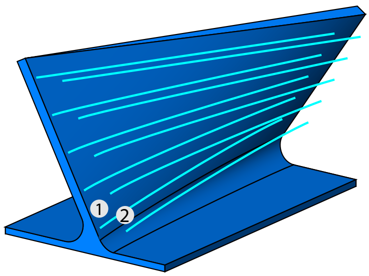
  In this option the tool tip point is shifted deeper with each consecutive slice. You have to specify a start value (**From**) and an end value (**To**). The tool shift is gradually added to each slice.

13.7.2 Pattern layers

This section enables you to define multiple layers in the direction of the material.

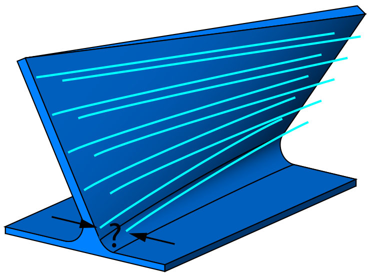
- **Number of layers**

  This field enables you to define the number of multiple offset layers from the slices.



- **Layer distance**

  This field enables you to define the layer distance along the direction of tool axis.



The **Layer distance** option is available only when the **Number of layers** is more than 1.

13.7.3 Sorting

This section enables you to define the linking method of the passes for machining.

Sequence

This option enables you to link the tool path by the layers or by slices.

- **By layer**

  When this option is chosen, all the roughing and finishing offsets of the current cutting pass are performed before moving to the next cutting pass.

- **By slice**

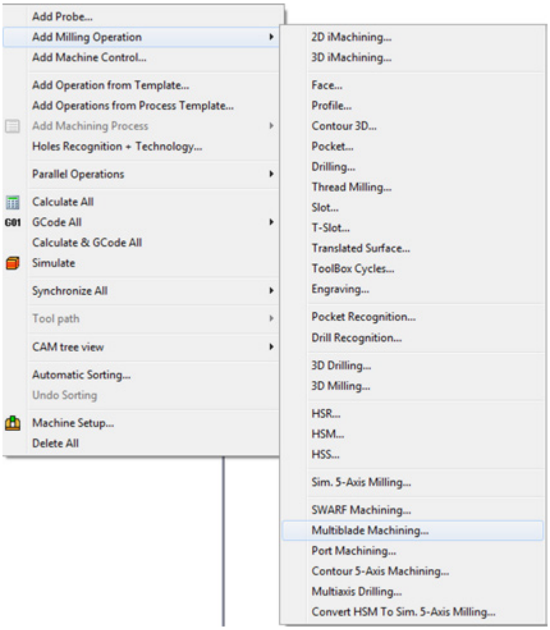
  When this option is chosen, all the cutting passes of the current offset level are performed before moving to the next offset level.

Multiblade Machining 14

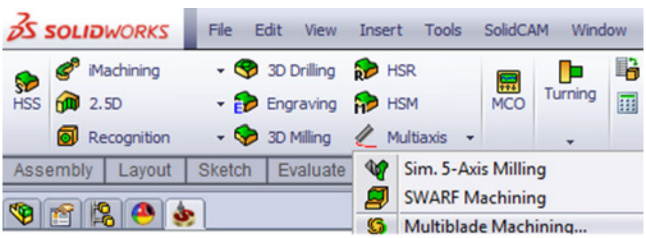
SolidCAM provides you with the **Multiblade Machining** operation. This operation generates tool paths for different configurations of impellers and bladed disks (blisks).

14.1 Adding a Multiblade Machining Operation

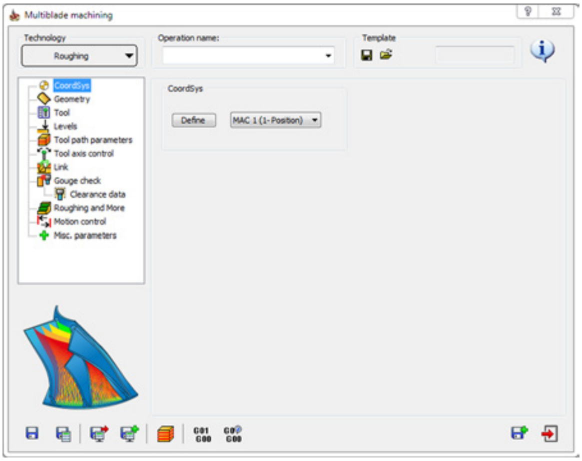
To add a **Multiblade Machining Operation** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose the **Multiblade Machining** command from the **Add Milling Operation** submenu.



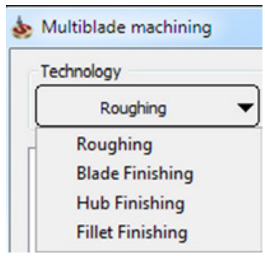
You can also choose the **Multiblade Machining** command from the **Multiaxis** menu on the **SolidCAM Operations** toolbar.



The **Multiblade Machining** dialog box is displayed.



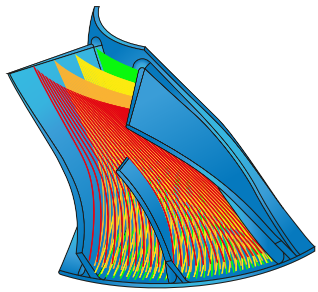
Technology



This section enables you to define the type of **Multiblade machining** operation. SolidCAM provides you with the following types of the **Multiblade machining** operations:

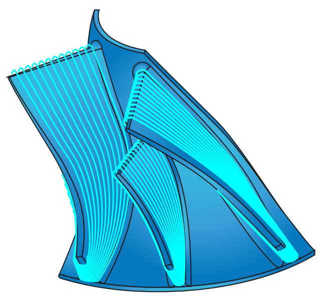
- **Roughing**

  This strategy enables you to create the basic roughing tool path between the main blade and the splitter. The pattern consists of layers and slices. Each layer consists of slices. The layers are placed on top of each other, and the slices are placed beside each other.



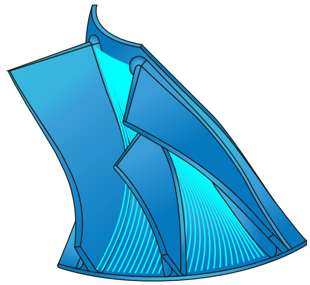
- **Blade finishing**

  This strategy enables you to create finishing slices for the blade and splitter surfaces.



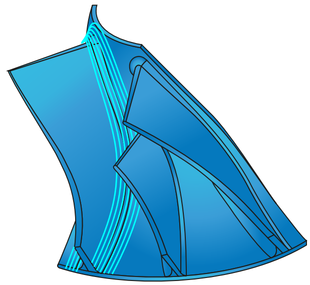
- **Hub finishing**

  This strategy enables you to create a single layer on the hub surface when the distance between the slices is very small.

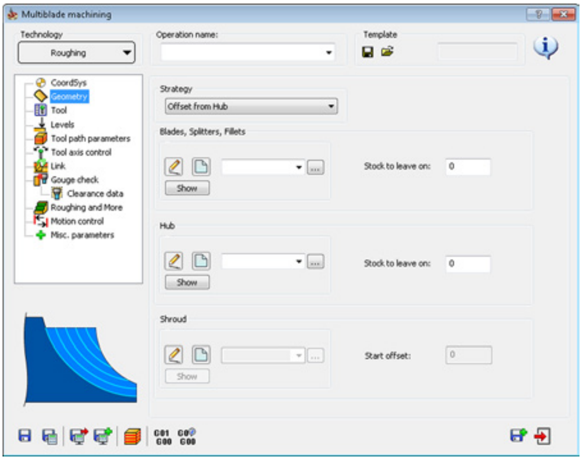


- **Fillet finishing**

  This strategy enables you to create a finishing tool path on the fillet area between the hub and blade. SolidCAM automatically detects the fillet of the part.



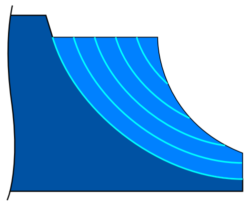
14.2 Geometry



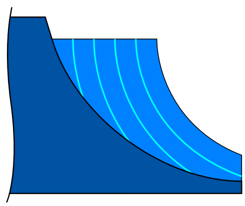
14.2.1 Strategy

This section provides you the following machining strategies:

Offset from Hub. In this strategy, all layers are offsets from the hub surface. The hub and the shroud surfaces are not parallel, therefore, the slices intersect with the shroud surface at some point. The slices are trimmed.

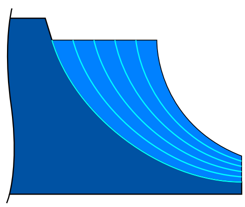


Offset from Shroud. In this strategy, all layers are offsets from the shroud surface. The hub and the shroud surfaces are not parallel therefore, the slices intersect with the shroud surface at some point. However, the slices are not trimmed away. Instead, the slices are extended till they reach the hub surface edge.



This strategy is not available for the **Hub Finishing** technology.

Offset between Hub and Shroud. In this strategy, the layers are equally distributed between the shroud and hub. The cuts are neither trimmed nor extended.



This strategy is not available for the **Hub Finishing** technology.

14.2.2 Part definition

Blades: The blade surface is a free form surface with a double curved shape. Each blade has a leading edge and a trailing edge. The leading edge is the suction side for the transported medium. The trailing edge is the exhaust side.

Splitters: A splitter is a short blade; similar to the main blade. It is located between the main blades. Usually a single splitter is used, however there are impellers that might have two splitters or even more. The leading edge can be horizontal or tilted.

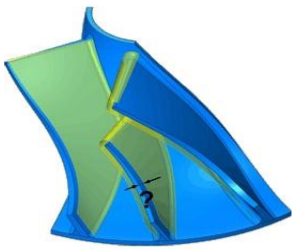
Fillets: The fillets are part of the blade. They guarantee a rounded connection between the blade and hub. The fillets can have a constant radius as well as a variable radius.

Hub: A hub is a revolved floor surface on which all the blades are placed.

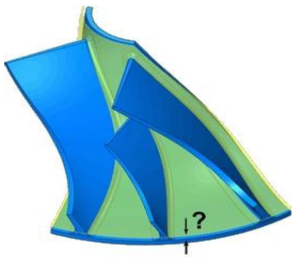
Shroud: The shroud surface is the top surface of the blade and the splitter. Usually it is an overturned surface from the stock. However, it can also be a free form surface.

14.2.3 Stock to leave on

This option sets a clearance offset between the specified parts of the tool and check surface.



Blade side



Hub side

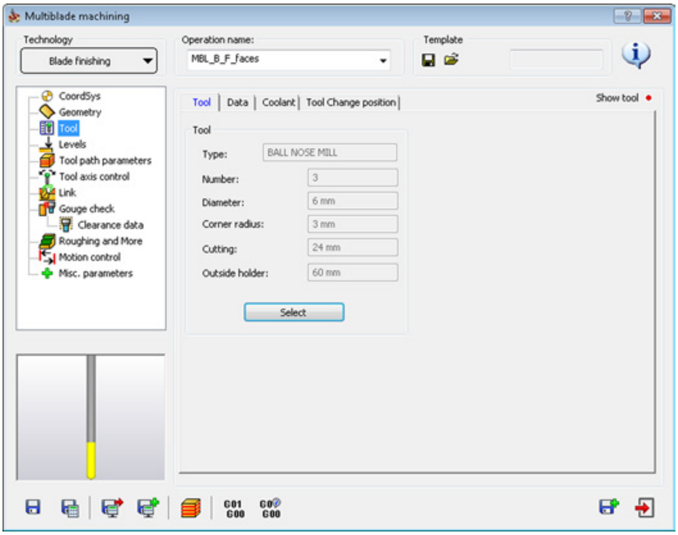
14.2.4 Start offset

This option allows you to define an offset value below the shroud for cutting.

This option is available only in the **Offset from Shroud** and **Offset between Hub and Shroud** strategies.

14.3 Tool

This page enables you to define the tool for the **Multiblade machining** operation and to set the cutting parameters (feed and spin).

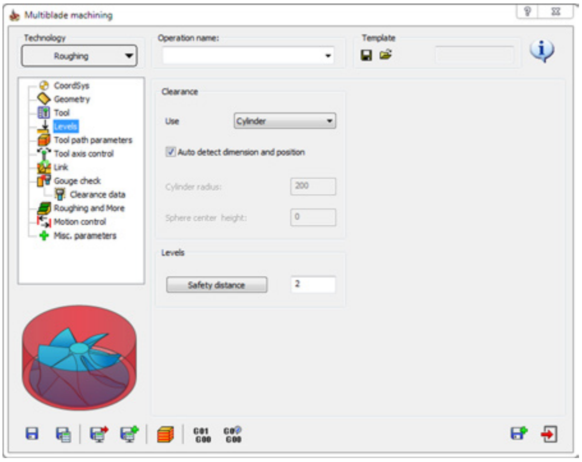


The following tool types are compatible with the **Multiblade machining** operation:

- **Ball Nose Mills**
- **Taper Mill with full radius tip**

14.4 Levels

For **Multiblade machining**, the **Levels** page enables you to define the machining levels.



14.4.1 Clearance

With this option, you can set clearance values.

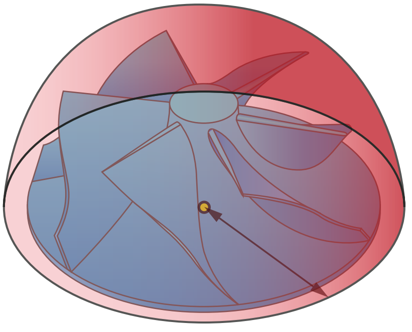
Use

In this section two geometries are used to define the clearance area:

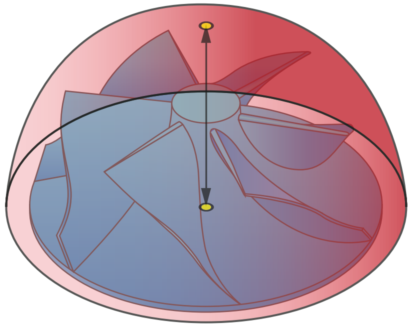
- **Sphere**

  When this option is chosen, the **Clearance** area has a spherical shape; it should enclose the multiblade part geometry completely. The tool performs a retract movement to the **Clearance** sphere and then a rapid movement along the sphere surface. The axis of the sphere is always parallel to the rotary axis of the multiblade part.

  Sphere radius: this parameters enables you to define the radius of the clearance sphere.



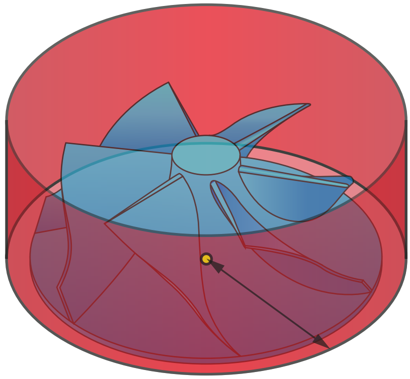
Sphere center height: this parameters enables you to define the distance from the sphere center to the sphere surface.



- **Cylinder**

This option enables you to define the Clearance as a cylindrical surface enclosing the multiblade part. The tool performs a retract movement to the Clearance cylinder, and then a rapid movement along the cylinder surface. The axis of the cylinder is always parallel to the rotary axis of the multiblade part.

Cylinder radius: this parameter enables you to define the radius of the clearance cylinder.



Auto detect dimension and position

In this option SolidCAM automatically defines the best fit sphere radius and the best position of the sphere center of the multiblade part.

14.4.2 Levels

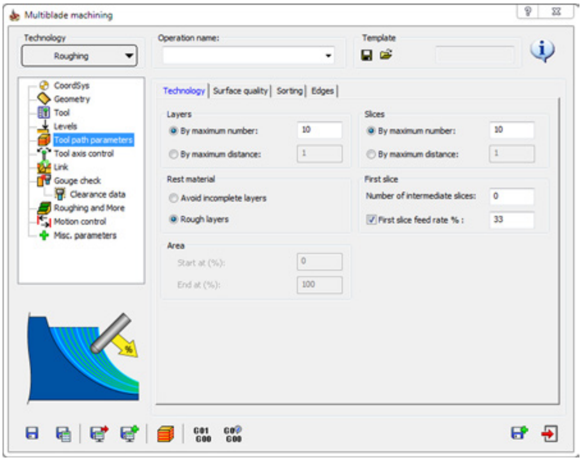
This option enables you to define the safety distance to approach and retract from the part.

Safety distance

After the descent movement to the **Retract distance** level, the tool starts the approach movement to the material. The approach movement consists of two segments. The first segment is performed with a rapid feed up to the **Safety distance**. From the **Safety distance** level, the approach movement is performed with the cutting feed.

14.5 Tool path parameters

The **Tool path parameters** page enables you to define the parameters of finish machining.



14.5.1 Technology

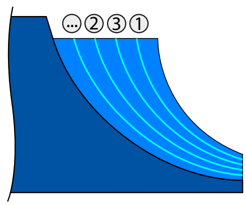
This page enables you to define the technological parameters of the **Multiblade machining** operation.

Layers

The layers are different depth levels of the roughing pattern. They are positioned on top of each other. The layers consist of the slices and can only be set if the technology is defined as **Roughing** or **Blade finishing**. The number of layers can be determined using one of two options:

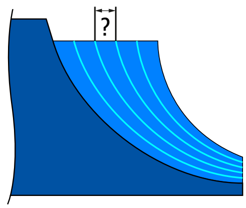
- **By maximum number**

  This option sets the maximum number of layers for roughing.



- **By maximum distance**

  This option sets the maximum distance between two layers for roughing.

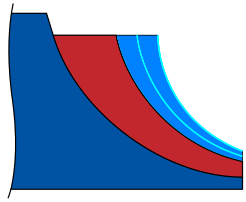


Rest material

This option enables you to machine the areas in which the stock is left out by previous operations.

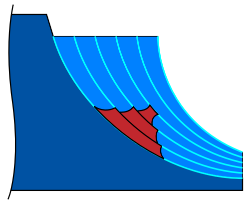
- **Avoid incomplete layers**

  If some material is left for machining, this option enables you to machine the entire layer and not only the portion where the material remains.



- **Rough layers**

  This option creates a tool path for the entire area of the blade considering only the height of the remaining stock. The width of the stock is not considered.



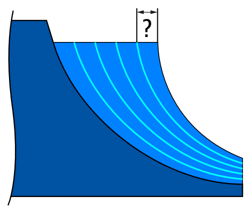
This option is available only in the **Roughing** technology.

Area

This option enables you to limit the area using a start distance from either the hub or the shroud. The area value must be entered as a percentage of the height of the blade.

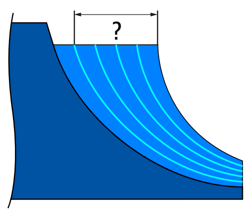
- **Start at (%)**

  This option enables you to determine a margin at the starting edge of the surface to avoid the inaccuracies of the surface edge and get a smooth cut.



- **End at (%)**

  This option enables you to determine a margin at the end edge of the surface to avoid the inaccuracies of the surface edge and get a smooth cut.



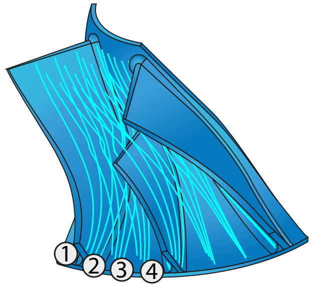
This option is available in **Roughing** and **Blade finishing** technologies, with the **Offset between Hub and Shroud** strategy.

Slices

This option enables you to form the layer for **Roughing** and the pattern for the **Hub finishing** technology.

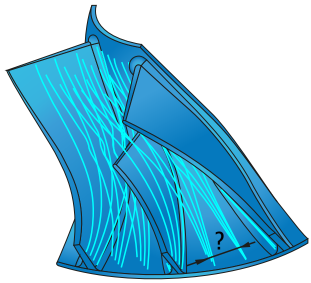
- **By maximum number**

  This option sets the maximum number of slices for roughing.



- **By maximum distance**

  This option sets the maximum distance between two slices for roughing.



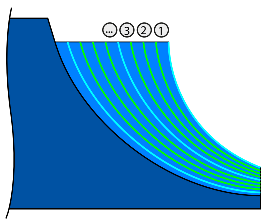
This option is available only in the **Roughing** and **Hub finishing** technologies.

First slice

This option controls the first slot cuts where the tool is fully engaged.

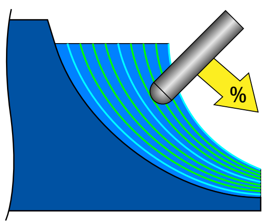
- **Number of intermediate slices**

  This option creates multiple depth cuts on the first slice.



- **First slice feed rate %**

  This option enables you to reduce the feed rate of the first slices to a percentage of the machining feed rate.



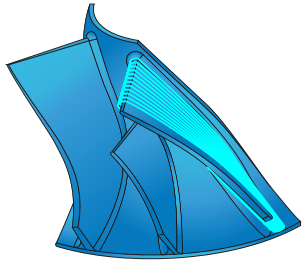
This option is available only in the **Roughing** technology.

Contour

This option sets the area to machine the blade and the fillet.

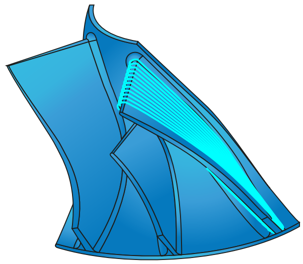
- **Full**

  This option enables you to activate a tool path created as a full contour around the blade.



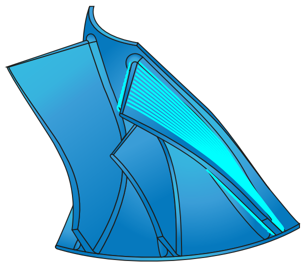
- **Full (trim trailing edge)**

  In this option, the tool does not roll around the trailing edge.



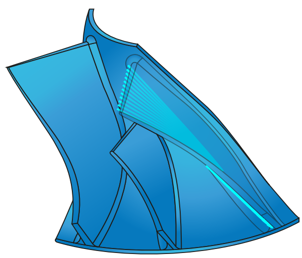
- **Full (trim trailing/leading edge)**

  In this option, the tool does not roll around the leading and trailing edge.



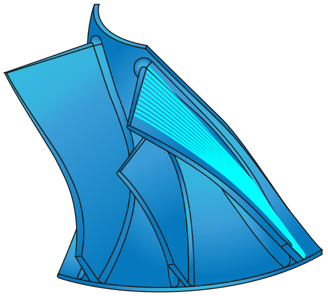
- **Left side**

  In this option, the tool path is created only on the left side of the blade.



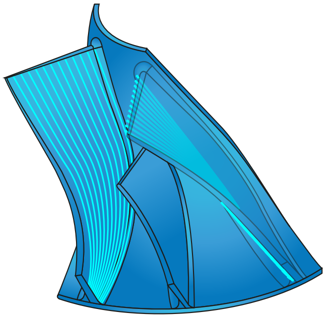
- **Right side**

  In this option, the tool path is created only on the right side of the blade.



- **Pocket**

  In this option, the tool path is created on the right side of the left blade and on the left side of the right blade.



The **Contour** option is available only in the **Blade finishing** and **Fillet finishing** technologies. The **Pocket** parameter is not available in the **Fillet finishing** technology.

Blade side

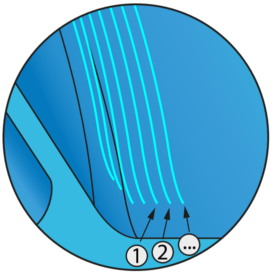
This section enables you to define how the blade side is machined.

Area

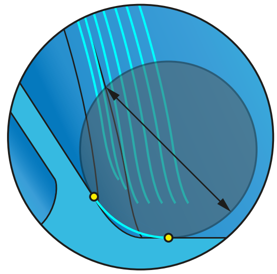
This parameter defines the method of cutting.

- **By number of cuts**

  This option sets the **Number of cuts** to machine the area.



- **By big tool diameter**



In this option, a sphere of a specific diameter is rolled along the fillet to determine the two contact points to define the area to be machined. The **Blade overlap** parameter enables you to set the values for blade overlap.

- **Same as Hub side**

This parameter uses the values defined in **Hub side**.

Hub side

This section enables you to define how the hub side is machined.

Area

This parameter defines the method of cutting.

- **By number of cuts**

This option sets the number of cuts to machine the area.

- **By big tool diameter**

In this option, a sphere of a specific diameter is rolled along the fillet to determine the two contact points to define the area to be machined. The **Hub overlap** parameter enables you to set the values for hub overlap.

- **Same as Blade side**

This parameter uses the values defined in **Blade side**.

By maximum number

This option defines the maximum number of cuts for finishing operation.

By maximum distance

This option defines the maximum distance between two passes of the tool path for finishing operation.

Both sides

Big tool diameter

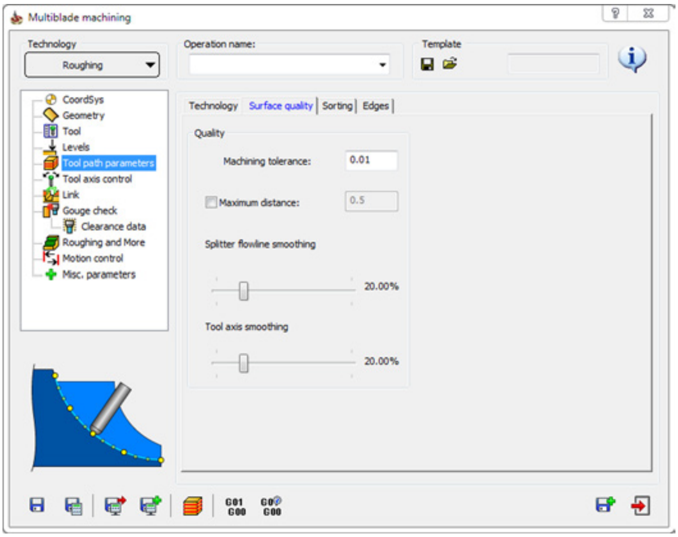
In this option, SolidCAM rolls a sphere of a specified diameter along the fillet to determine the two contact points to define the area to be machined.

Side step

This option allows you to define the distance between two subsequent passes of the tool path.

The options of **Blade side**/**Hub side**/**Both sides** are available only in the **Fillet finishing** technology.

14.5.2 Surface quality



The **Surface quality** tab enables you to define the parameters that affect the surface finish quality.

Machining tolerance

The machining tolerance is the tolerance for the accuracy of the tool path. This value is the chordal deviation of the tool path against the surfaces to be machined. In other words, the tool path can have a maximum error to the surfaces in the range of plus or minus machining tolerance.

Maximum distance

This parameter enables you to get more points on surface to be machined even if the machining tolerance is the same.

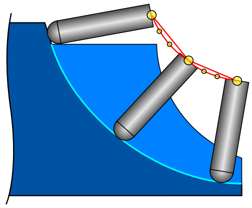
Splitter flowline smoothing

In this option, the slider sets the degree of smoothing around the leading edge of the splitter.

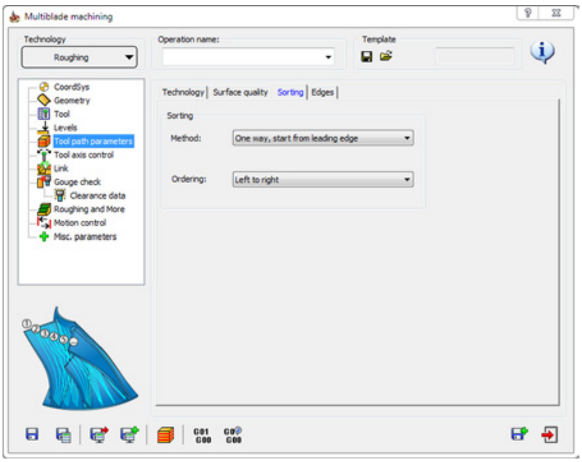
This option is available only in the **Roughing** and **Hub finishing** technologies.

Tool axis smoothing

This value sets the degree of post smoothing for the tool axis. At 0% the tool is oriented exactly in its initially calculated point. The higher the smoothing, the more deviation is allowed from the initial orientation so that the different orientations bend together.



14.5.3 Sorting



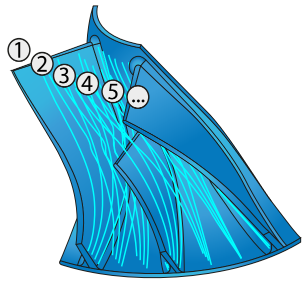
The **Sorting** tab enables you to define the order and direction of the cuts.

Method

The **Method** list defines working methods. The following option are available in this:

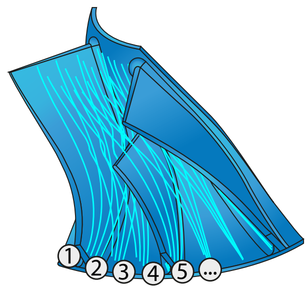
- **One way, start from leading edge**

  In this option, the slices always start at the leading edge.



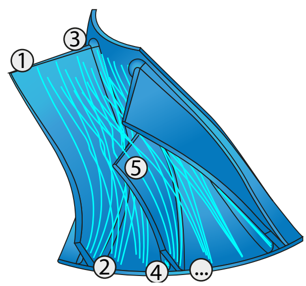
- **One way, start from trailing edge**

  In this option, the slices always start at the trailing edge.



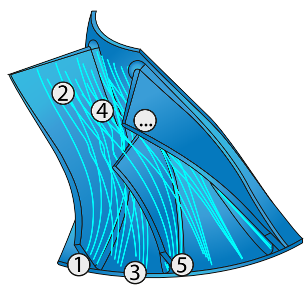
- **Zigzag, start from leading edge**

  In this option, the first slice is from the leading edge.



- **Zigzag, start from trailing edge**

  In this option, the first slice is from the trailing edge.



The options of **Zigzag, start from leading edge** and **Zigzag, start from trailing edge** are available only in the **Roughing** and **Hub finishing** technologies.

- **Spiral, start from leading edge**

  In this option, the first slice is from the leading edge.

- **Spiral, start from trailing edge**

  In this option, the first slice is from the trailing edge.

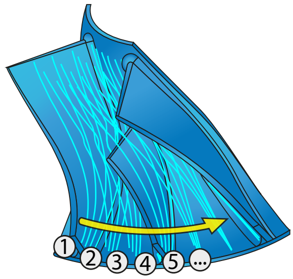
> These two options are available only in the **Blade finishing** technology when the option of **Full** is selected in **Contour**.

Ordering

The **Ordering** list defines the sequence of the slices. The following options are available in this:

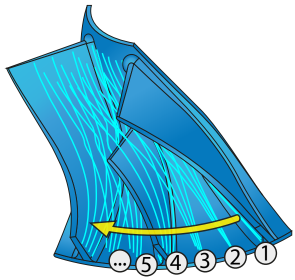
- **Left to right**

  In this option, the slices are applied from left to the right side.



- **Right to left**

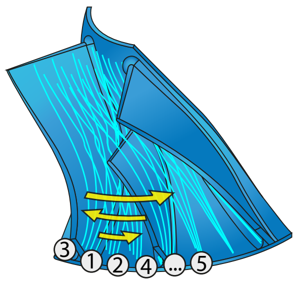
  In this option, the slices are applied from right to the left side.



- **From center away**

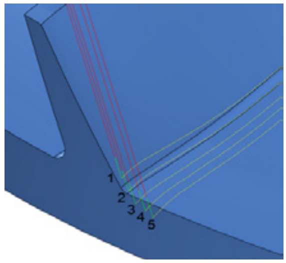
  In this option, the first slice is applied at the center, then it proceeds outwards while alternating the sides.

> These three options are available only in the **Roughing** and **Hub finishing** technologies.



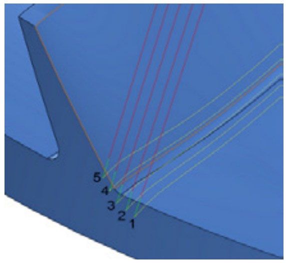
- **Top down**

  In this option the machining starts at the blade and ends on the hub.



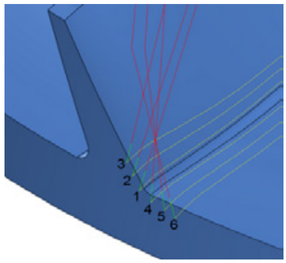
- **Bottom up**

  In this option the machining starts at the hub and ends on the blade.



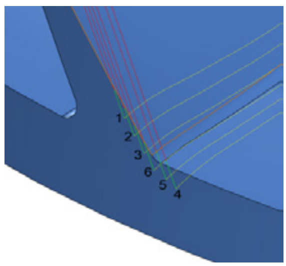
- **Inside to outside**

  In this option the machining starts at the center and then uses alternate passes on the hub and blade to complete the machining.



- **Outside to inside**

  In this option the machining starts from the hub or shroud and moves towards the center while machining.



These options are available only in the **Fillet finishing** technology.

Cut direction

This option enables you to define either **Climb** or **Conventional** direction for the cutting passes.

The **Cut direction** is available only in the **Blade finishing** and **Fillet finishing** technologies.

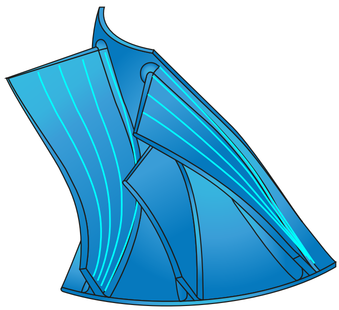
14.5.4 Edges

Edge rolling

This section defines how far the tool must roll around the leading and trailing edges of the blade and the trailing edge of the splitter.

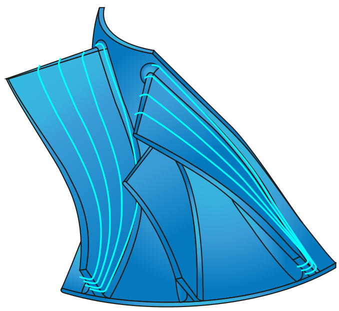
- **Automatic**

  In this option, the tool path is automatically trimmed or extended in regards to tangency while approaching to and retracting from the material.



- **Full (Without trimming)**

  In this option, the tool rolls around the entire leading and trailing edge up to the back side.

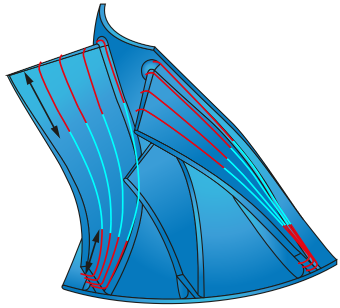


- **Trimmed by tool radius**

  In this option, the tool path is trimmed when the radius of the leading and trailing edge exceeds the tool radius.

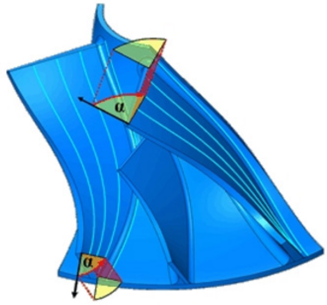
- **Trimmed by length**

  In this option, the tool path is trimmed by a certain value defined in the **Leading edge length**/**Trailing edge length** fields.



- **Trimmed by angle**

  In this option, the tool path trimming is defined by the angle that is spanned between the virtual extension of the blade edge and the cutting side. The values of the **Leading edge angle** and **Trailing edge angle** are set in the separate fields.



Extention

This option enables you to extend the tool path in the direction of the leading and trailing edge.

- **Leading edge**

  This section enables you to extend the leading edge in the tangential (along the cutting direction) or in radial (towards the rotation center) direction. The extension values are defined in the **Tangential** and **Radial** fields.

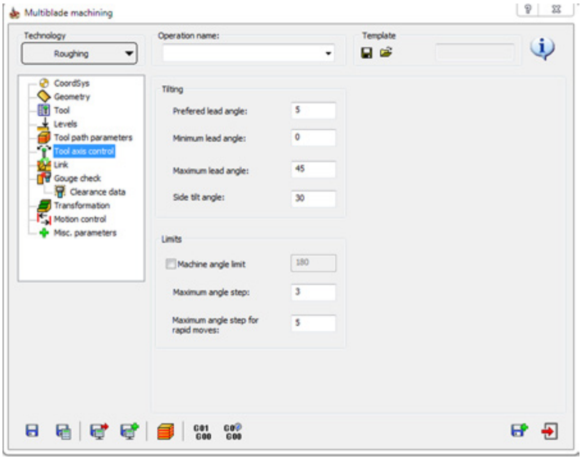
- **Trailing edge**

  This section enables you to extend the trailing edge in the tangential (along the cutting direction) or in radial (outwards from the rotation center) direction. The extension values are defined in the **Tangential** and **Radial** fields.

Tilting

This option controls the tilting limits at leading and trailing edges so that the tool does not tilt beyond the specified value. This helps in keeping the tool within the limits of the machine angles.

14.6 Tool axis control



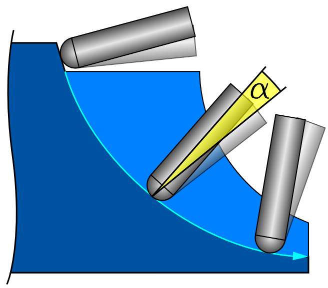
The **Tool axis control** page enables you to define the orientation of the tool axis during the machining.

Tilting

This option enables you to define a tilting range for the lead angle. It allows you to machine most of the area by providing the optimal tilting.

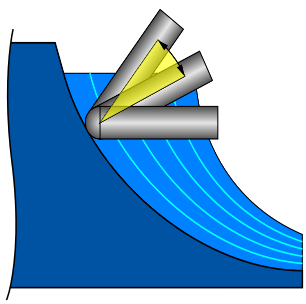
- **Preferred lead angle**

  SolidCAM uses this option as default. If not, then the lead angle ranges from the minimum to the maximum angles.



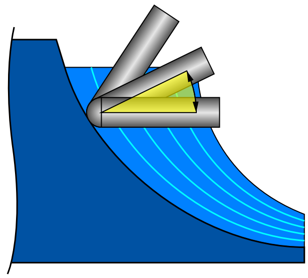
- **Minimum lead angle**

  This parameter sets the minimum lead angle for the preferred lead angle.



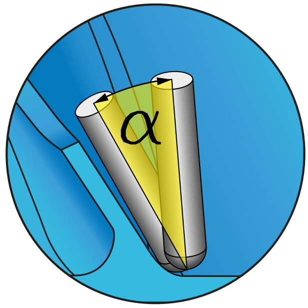
- **Maximum lead angle**

  This parameter sets the maximum lead angle for the preferred lead angle.



- **Side tilt angle**

  This parameter enables the tool tilting to the side of the cutting direction, towards the blades. At zero degrees the tool is oriented perpendicular to the hub surface.

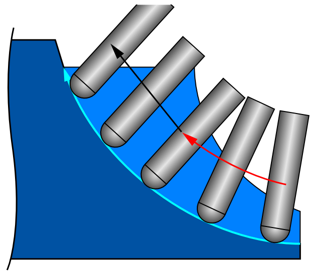


Limits

With this option, SolidCAM enables you to use the machine limits.

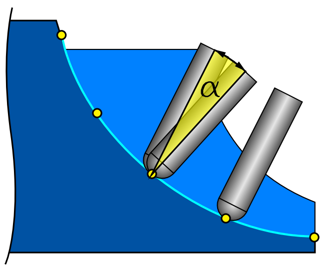
- **Machine angle limit**

  This parameter enables you to control the maximum tilting of the tool in respect to machine angle limitations. If the tool cannot reach areas in the tool paths in order to fit the limits, the portion of the tool path is trimmed.



- **Maximum angle step**

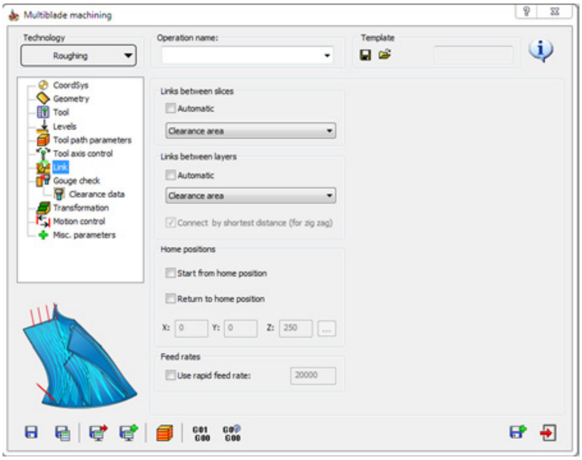
  This parameter controls the maximum angle change between two segments. When the angle step is smaller, it enables calculation of more segments.



- **Maximum angle step for rapid moves**

  This parameter controls the maximum angle change between two segments on the clearance area. When the angle step is smaller, it enables calculation of more segments.

14.7 Link



The **Link** page defines the shape of the links between layers or slices.

You can set the links either automatically or manually. To edit the linking parameters, clear the **Automatic** check boxes in the **Links between slices** and **Links between layers** sections.

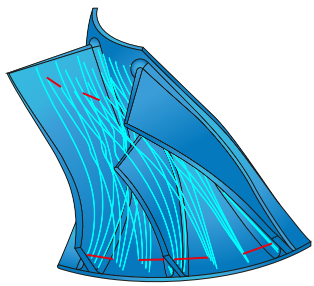
Links between slices/layers

- **Direct blend**

  This link type is a combination of direct and blend spline links. Using this option allows the links to stay close to the part.

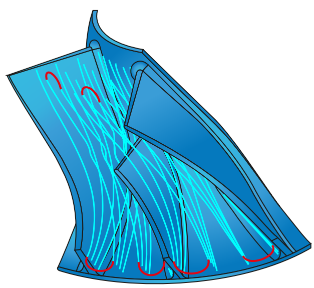
- **Direct**

  This option enables a straight connection between two slices/layers.



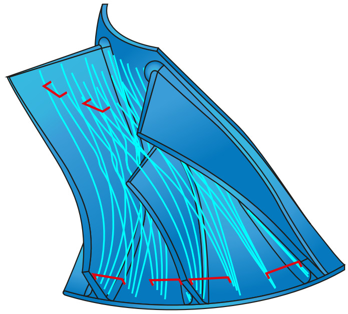
- **Blend spline**

  SolidCAM connects two layers/slices with a spline tangential to both segments.



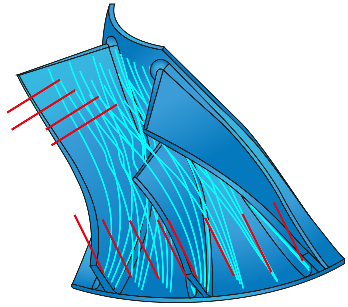
- **Safety distance**

  This option allows a small retraction along the tool axis. Then the tool steps over to the next slice/layer.



- **Clearance area**

  The clearance is either a cylinder or a sphere. The tool moves rapidly when retracting to the clearance area. You can define the appropriate diameter in manual linking.



In the **Blade finishing** technology, **Links between slices** option is not available. In the **Hub finishing** technology, **Links between layers** option is not available.

Connect by shortest distance (for zig zag)

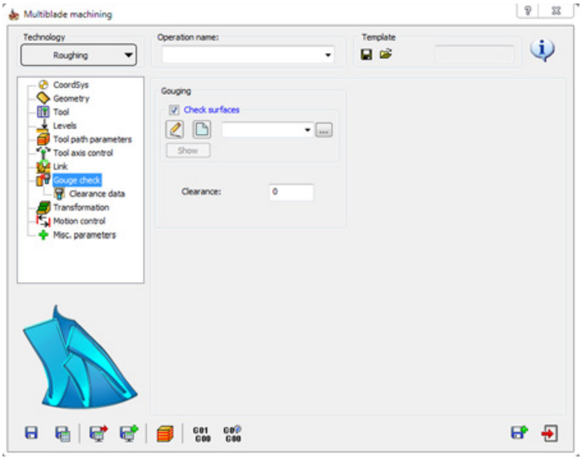
This option connects layers using the shortest distance. This parameter can be used with all the linking options.

This option is available in **Roughing** technology when the **Sorting** method is set to **Zigzag, start from trailing edge** or **Zigzag, start from leading edge**.

Feed rates

The **Use rapid feed rate** option enables you to define the feed rate used in links.

14.8 Gouge check



The **Gouge check** page enables you to define the gouge checking parameters.

14.8.1 Gouging

Check surfaces

With this option, you can define additional check faces that are not floor, blades, fillets and splitter.

Clearance

With this option, you can set a clearance value for gouge check.

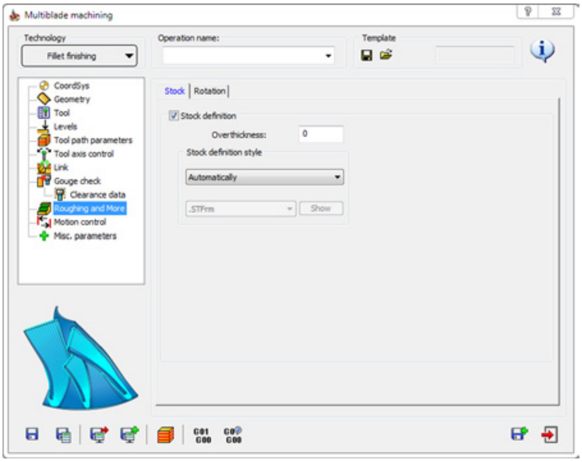
14.8.2 Clearance data

The **Clearance data** page enables you to define the clearance offsets for arbor and tool holder in order to get a guaranteed clearance gap between the arbor, tool holder, and workpiece.

Clearance

The **Tool Shaft** parameter defines the offset applied to the tool shaft cylinder from all sides. The **Arbor** defines the offset applied to the arbor cylinder from all sides. The **Holder** parameter defines the offset applied to the tool holder cylinder from all sides. The **Clearance angle** parameter sets a minimum clearance angle between the blade and tool.

14.9 Roughing and More



This page enables you to define parameters for setting the stock material.

14.9.1 Stock

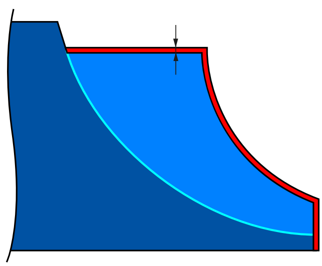
This page enables you to modify the stock definition.

Stock definition

This option enables you to load a part as rest material. The tool path is not created if the stock is not defined.

- **Overthickness**

  This option enables you to expand the defined offset value and defines an extra thickness that can be temporarily applied to the tool.



Stock definition style

This option enables you to specify the method of machining area definition.

- **Automatically**

  In this option SolidCAM calculates the updated stock model after all the previous operations. The **Overthickness** value is added as offset to the stock, which is used as stock for the current operation.

- **By selected operations**

  In this option you can select the operations to calculate the updated stock.

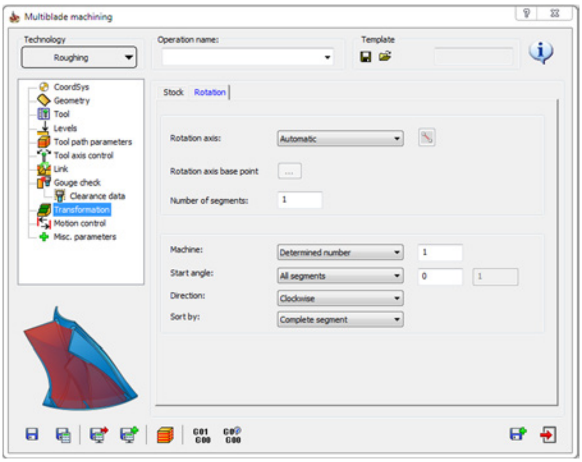
- **Stock by *.FCT file**

  In this option machining is performed in the area defined by an offset from the updated stock, defined in FCT file located in the CAM-Part folder. The offset is defined by the **Overthickness** parameter.

- **Stock by *.STL file**

  In this option machining is performed in the area defined by an offset from the updated stock, defined in STL file located in the CAM-Part folder. The offset is defined by the **Overthickness** parameter.

14.9.2 Rotation



This option rotates and copies the tool path around the axis of the multiblade.

Rotation axis

This option enables you to define the axis of rotation of the tool path.

- **Automatic**

  In this option SolidCAM automatically sets the axis for rotation.

- **User defined**

  In this option you can manually set the axis for rotation.

Rotation axis base point

This option enables you to define the base point of the rotation axis that is needed in addition to the direction.

Number of segments

This option defines the total number of tool path segments in the part that need to be rotated.

Machine

This option allows you to set the segments for machining.

- **All**

  In this option you can decide to machine all the segments that were defined in the part definition or only a determined number.

- **Determined number**

  In this option you can decide to machine a determined number of segments that were defined in the part definition.

Start angle

The start angle enables you to define the start angle position of the tool path according to its initial position.

Direction

This parameter enables you to define if the rotation sequence should be clockwise or counterclockwise.

Sort by

This parameter enables you to define the sorting method.

- **Complete segment**

  This option enables you to machine each segment completely before moving to the other segment.

- **Layer**

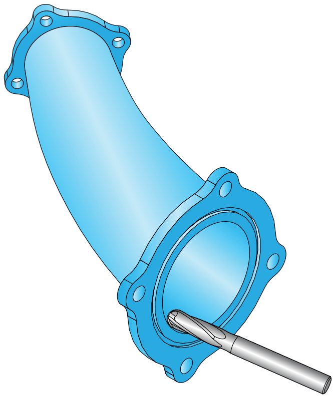
  This option enables you to machine all the layers of all the segments sequentially.

- **Slice**

  This option enables you to machine all the slices of all the segments sequentially.

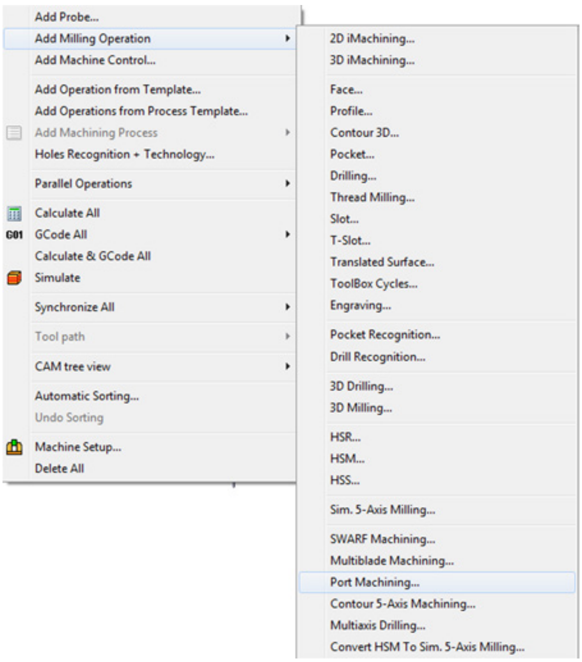
15 Port Machining

SolidCAM provides you with the **Port Machining** operation. The **Port machining** is used to create either a roughing or finishing tool path for port type geometries. This operation enables you to reach the full area with a single tool path, machining from the top and the bottom. The tool path is calculated on the triangle mesh elements.

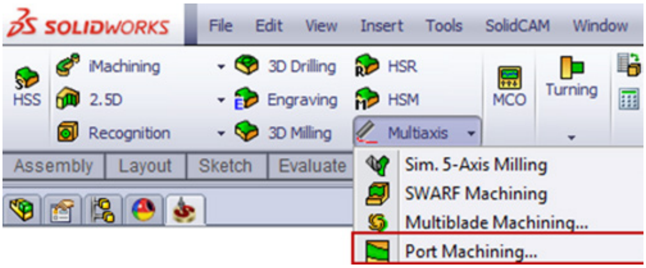


15.1 Adding a Port Machining Operation

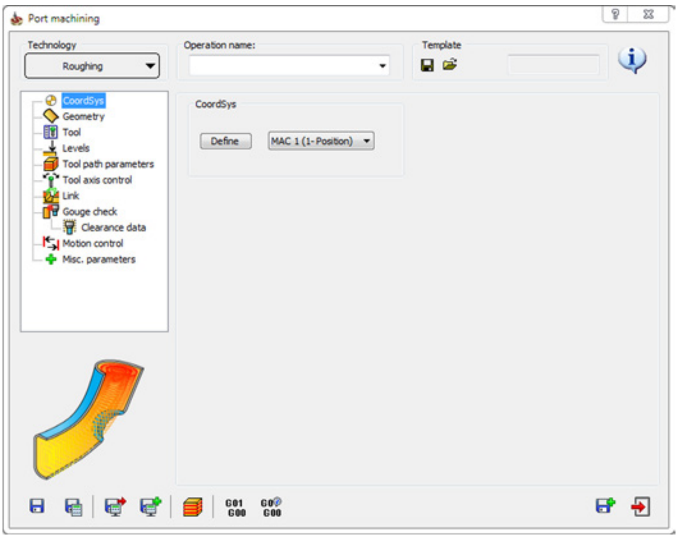
To add a **Port Machining Operation** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose the **Port Machining** command from the **Add Milling Operation** submenu.



You can also choose the **Port Machining** command from the **Multiaxis** menu on the **SolidCAM Operations** toolbar.



The **Port Machining** dialog box is displayed.

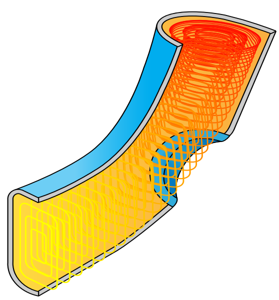


Technology

This section enables you to define the type of **Port machining** operation. SolidCAM provides you with the following types of the **Port machining** operation:

- **Roughing**

  This strategy enables you to create the basic roughing tool path to remove a lot of material at a time within the port. The pattern consists of the layers and slices. This helps in keeping minimum material on the walls for finishing operation.

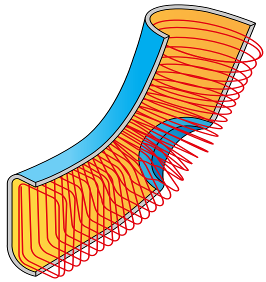


- **Rest Roughing**

  This strategy enables you to create a roughing tool path based on the updated stock model. With this strategy you can target only the specific areas where material is remaining and eliminate the possibility of unnecessary air cuts.

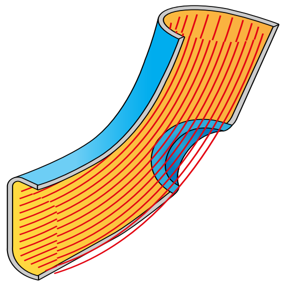
- **Spiral Finishing**

  This strategy creates a tool path to machine the entire surface in a descending helical manner, avoiding unnecessary retracts and ensuring constant contact between the cutter and machining surface.

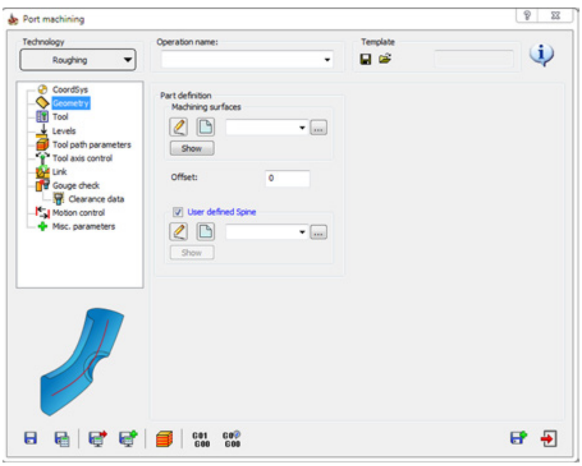


- **Plunge finishing**

  This strategy enables you to create multiple cuts along the flow line direction of the port. The start point is always outside, as it creates scallops along the flow of the gases.



15.2 Geometry

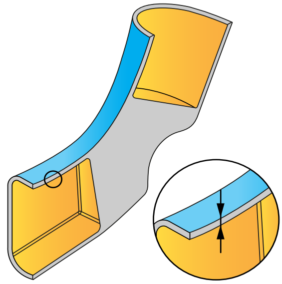


Part definition

- **Machining surfaces**

  These surfaces are the actual port surfaces that must be selected as an input geometry.

- **Offset**



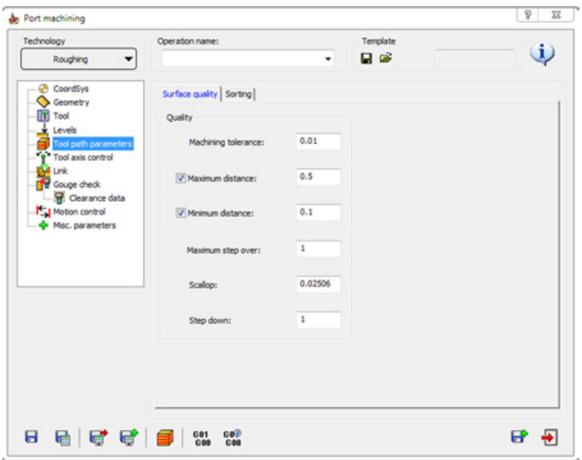
This parameter sets the rest material on the port surfaces that can be used for a roughing tool path where the rest material is left for finishing.

- **User defined spine**

This section enables you to define the curve that guides the tool path. When this check box is not selected, the system automatically creates a spine according to the defined surfaces. When this check box is selected, the spine can be defined manually. You have to ensure that the spine is always positioned inside the port and the tool must always fit between the port and the spine.

If the spine is too short, not all the machining surfaces can be reached.

15.3 Tool path parameters



The **Tool path parameters** page enables you to define the parameters of **Port machining**.

15.3.1 Surface quality

- **Machining tolerance**

  This parameter sets the machining accuracy, the maximum deviation from the machining surface.

- **Maximum distance**

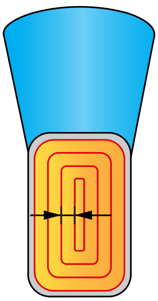
  When this check box is selected, you can define the maximum distance between two points of the tool path.

- **Minimum distance**

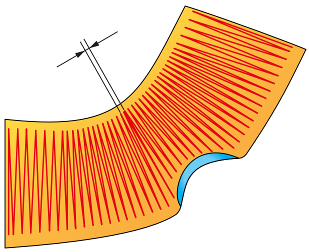
  When this check box is selected, you can define the minimum distance between two points of the tool path.

- **Maximum step over**

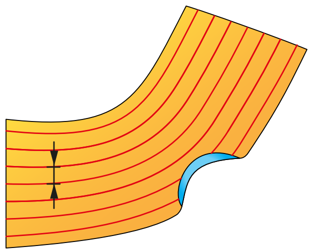
  When this parameter is used in case of a roughing operation, the step over is the distance between two slices. In case of a finishing operation, the step over is the distance between two layers.



Roughing Max. step over



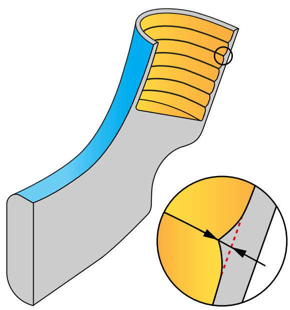
Spiral Finishing Max. step over



Plunge Finishing Max. step over

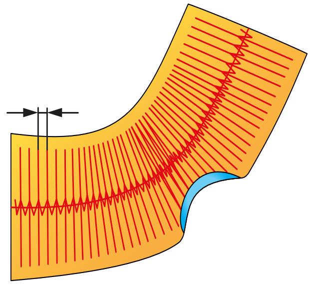
- **Scallop**

  This parameter enables you to define the cusp height of the machined surface.



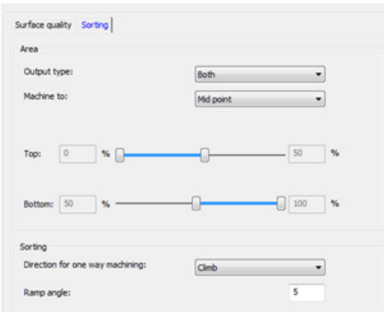
- **Step down**

  This parameter enables you to set the distance between two layers for roughing pattern.



  This option is available only in the **Roughing** and **Rest Roughing** technologies.

15.3.2 Sorting



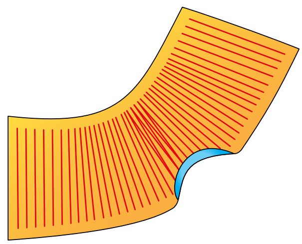
Area

This section enables you to define the machining area and direction.

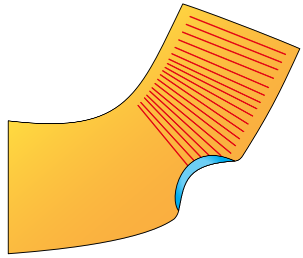
Output type

This parameter sets the main machining area.

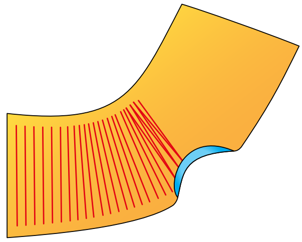
In the option of **Both**, the tool path cuts are automatically linked from top to bottom.



In the option of **Top**, the tool path is defined by the spine start point.



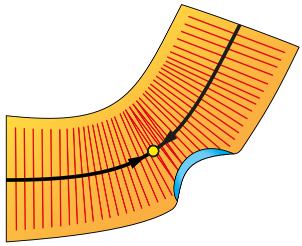
In the option of **Bottom** the tool path is defined through the spine end point.



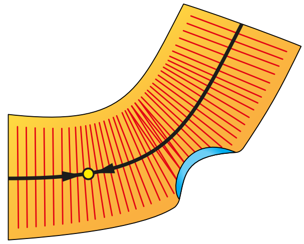
Machine to

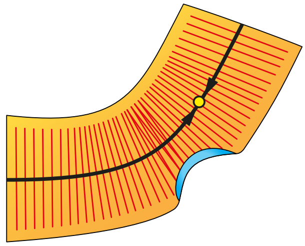
This parameter sets the depth of cutting for each type of output.

In the option of **Mid point**, SolidCAM makes connection between the top and bottom segments of the tool path in the center of the port.



In the option of **Maximum from top**, SolidCAM machines as far as possible from the top.





In the option of **Maximum from bottom**, SolidCAM machines as far as possible from the bottom.

In the **User defined** option, the area to be machined can be entered in percents of the spine. The **Top** slider sets the machining area limits starting from the top surfaces. The **Bottom** slider sets the machining area limits starting from the bottom surfaces.

Sorting

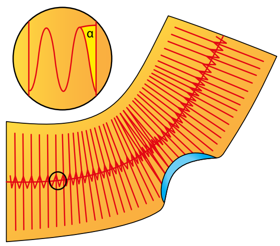
This section enables you to define the direction of machining.

Direction for one way machining

This parameter enables you to choose **Conventional** or **Climb** milling.

This option is not available in the **Plunge Finishing** technology.

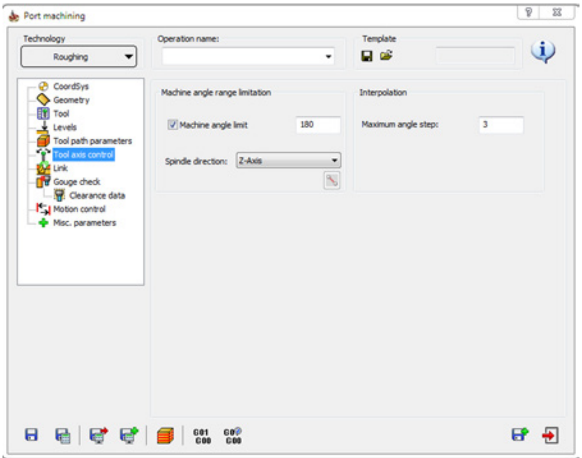
Ramp angle



This parameter defines the pitch of the helix when the tool plunges to the next deeper layer.

This option is not available in **Spiral Finishing** and **Plunge Finishing** technologies.

15.4 Tool axis control



The **Tool axis control** page enables you to define the orientation of the tool axis during the machining.

- **Machine angle limit**

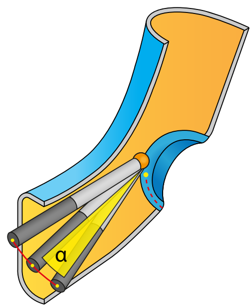
  This parameter enables you to take into account the rotational axis limitation of the machine. In case the tilting cannot be applied due to the limits, SolidCAM trims the tool path.

- **Spindle direction**

  This parameter enables you to set the machine angle limit around **X-Axis**, **Y-Axis**, or **Z-Axis**. The **User-defined** direction enables you to manually pick the direction of the spindle in case the port has two exhaust points.

- **Maximum angle step**

  This parameter sets the maximum angular deviation between two tool path points.



15.5 Gouge check

The **Gouge check** page enables you to automatically detect and avoid the possible collisions between the tool, the tool holder, and the workpiece.

- **Check surfaces**

  This option enables you to check the surfaces that need to be protected against collisions as there are additional attachments to the port ends which need to be considered in gouge checking.

- **Check against machine surfaces**

  This option gives you the possibility of not checking against the machining surfaces and/or against user defined check faces. It provides you with tilting which is not influenced by the machining surfaces. This option is helpful when a virtual tool path is created.

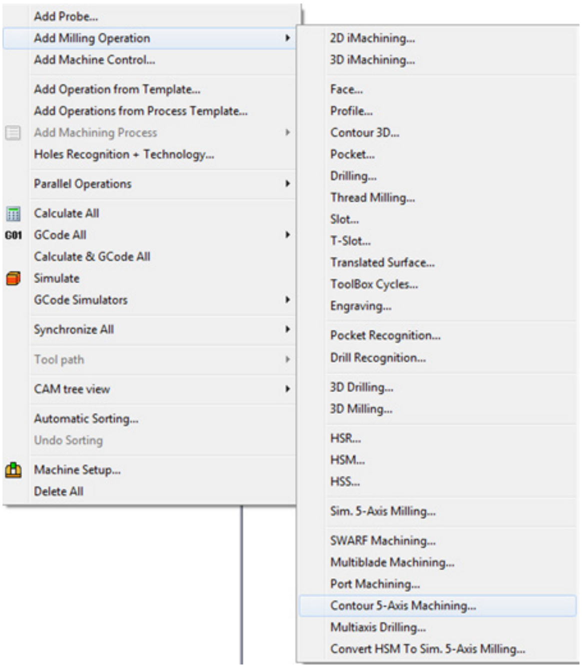
Contour 5-Axis Machining

16

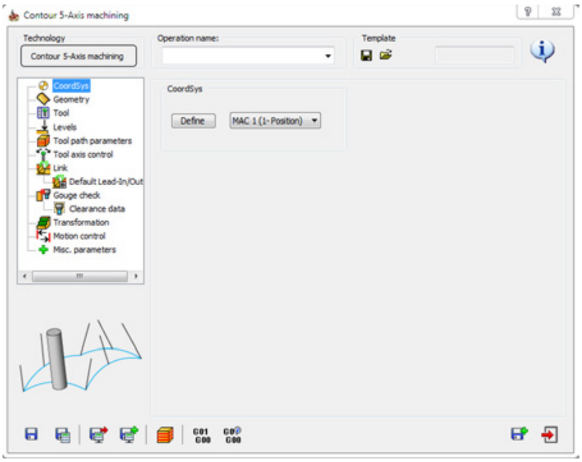
SolidCAM provides you with the **Contour 5-Axis machining** operation. The calculation based on this operation creates tool path with a wireframe as input drive curve. This strategy works without any machining surfaces.

16.1 Adding a Contour 5-Axis Machining Operation

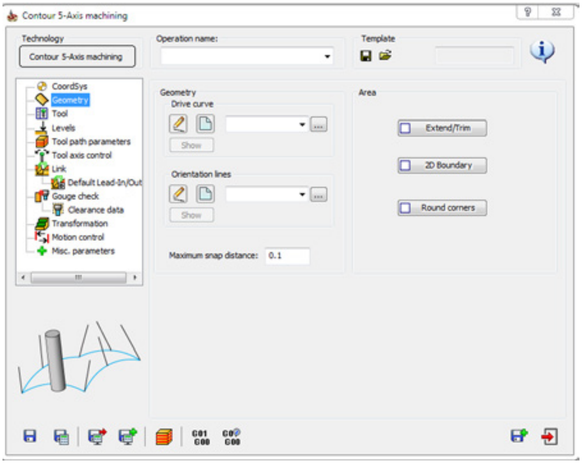
To add a **Contour 5-Axis machining** to the CAM-Part, right-click the **Operations** header in **SolidCAM Manager** and choose the **Contour 5-Axis machining** command from the **Add Milling Operation** submenu.



The **Contour 5-Axis machining** dialog box is displayed.



16.2 Geometry



The **Geometry** section enables you to define the geometry for the Contour 5-Axis operation.

- **Drive curve**

  This section enables you to define the curve on which machining is performed. The tool is automatically offset with the tool radius.

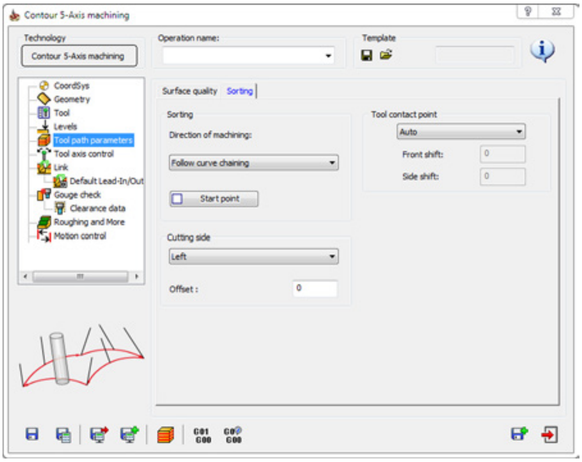
- **Orientation lines**

  This section enables you to define the lines that control the tool axis orientation along the drive curve.

- **Maximum snap distance**

  This parameter defines the maximum distance between orientation line end points and the drive curve. When tilting is applied to a contour, only lines within this distance are used, while other lines that are far from the contour are ignored.

16.3 Tool path parameters

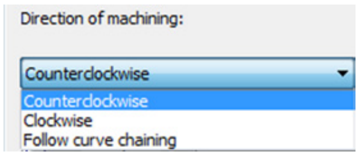


The **Surface quality** tab of the **Tool path parameters** page in **Contour 5-Axis machining** is similar to other Sim 5-axis Milling operations.

Sorting

- **Direction of machining**

  This option allows you to set **Counterclockwise**, **Clockwise**, or **Follow curve chaining** direction for machining.

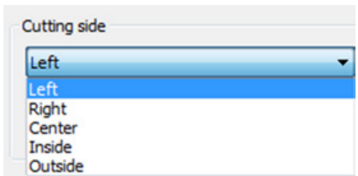


- **Start point**

  This option defines the direction of the start point.

- **Cutting side**

  This parameter positions the tool in the **Left**, **Right**, **Center**, **Inside**, or **Outside** of the selected contour.

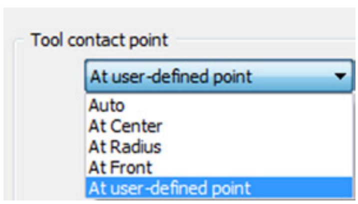


- **Offset**

  The offset is a minimum distance between tool and drive curve.

- **Tool contact point**

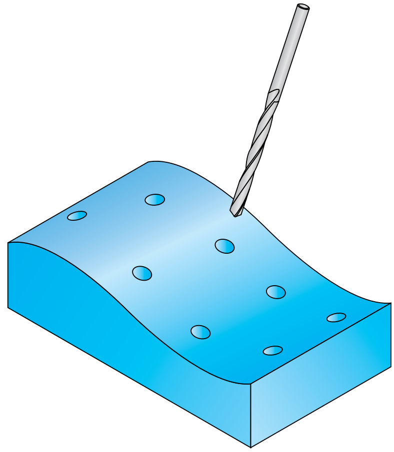
  The **Tool contact point** section enables you to define the point on the tool surface that contacts with the drive surfaces during the machining.



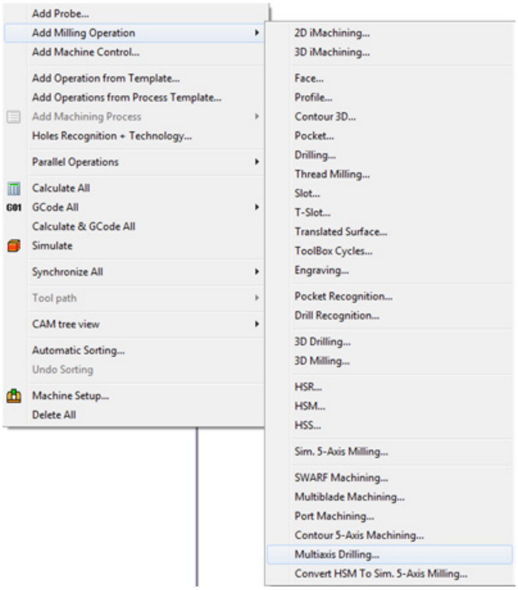
Multiaxis Drilling Operation

17

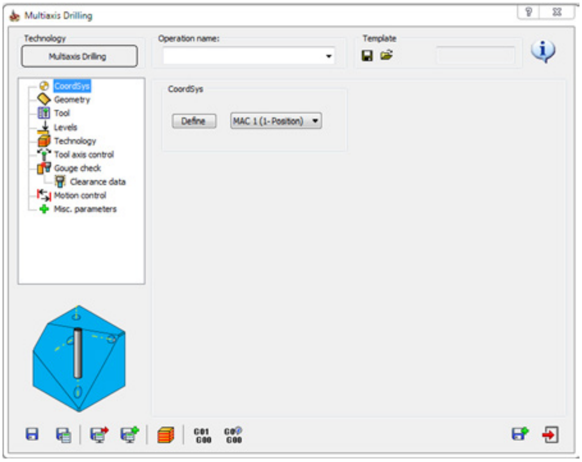
Multiaxis Drilling operation enables you to machine a series of drills that have different orientations.



To start the operation, choose the **Multiaxis Drilling** command from the **Add Milling Operation** menu in the **SolidCAM Manager** tree.

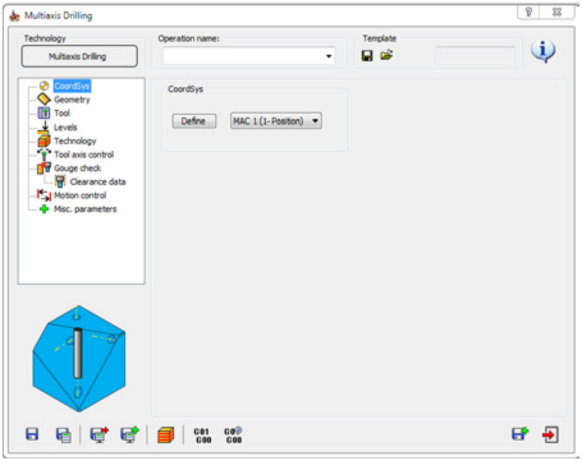


The **Multiaxis Drilling** dialog box is displayed. This dialog box enables you to define the parameters of the **Multiaxis Drilling** operation.



17.1 CoordSys page

This page enables you to define the Machine Coordinate System for the **Multiaxis Drilling** operation.



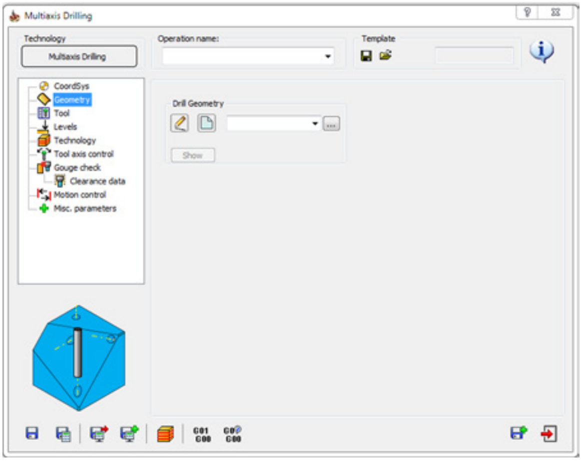
In the **Multiaxis Drilling** operation, you have to choose only the Machine Coordinate Systems. The Multiaxis drilling tool path positions and tool axis orientation at each tool path position are generated relative to the Machine Coordinate System. The tool path is generated in the 4/5-axis space, relative to a Machine Coordinate System. The Machine Coordinate System is defined relative to the center of rotation of the machine (CNC-machine origin).

You can choose an existing Coordinate System from the list or click the **Define** button to define a new one using the **CoordSys Manager** dialog box. This dialog box enables you to define a new Coordinate System directly on the solid model.

When the Coordinate System is chosen for the operation, the model is rotated to the selected CoordSys orientation.

For more information on the Coordinate System definition, refer to the **SolidCAM Milling User Guide** book.

17.2 Geometry page

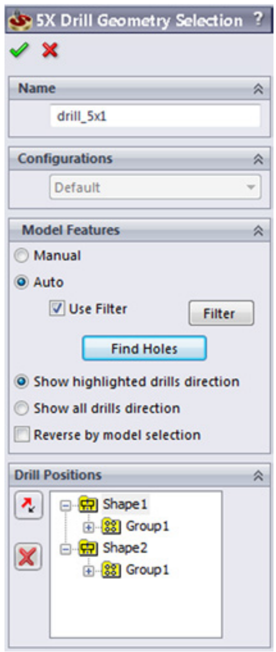


This page enables you to define the geometry data for the **Multiaxis Drilling** operation.

You can choose an existing geometry from the list or click the **New** button to define a new one. When the geometry is chosen, the **Show** button enables you to display it on the model.

Geometry definition

When you click the button, the **5X Drill Geometry Selection** dialog box is displayed.



Geometry definition is not available if the Target model is not defined.

- **Name**

  Define the name of the geometry.

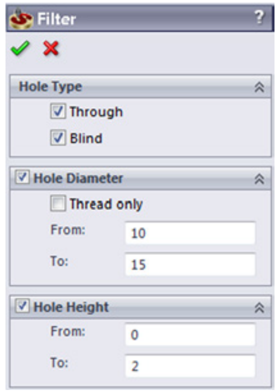
- **Configurations**

  Switch between SolidWorks configurations and choose the relevant configuration for the geometry definition.

- **Model Features**

  Define the mode of geometry selection (**Auto**/**Manual**), the filtering of drills and the direction of drilling.

- **Auto**



When the **Auto** mode is chosen, clicking the **Find Holes** button enables SolidCAM to find automatically all the drills present in the solid model.

You can filter the drills by selecting the **Use filter** check box and clicking the **Filter** button. The **Filter** dialog box is displayed. It enables you to filter the drills by **Hole type** (**Blind**/**Through**) and setting the **Hole diameter** and **Hole height** range.

- **Manual**

When the **Manual** mode is chosen, you can select the drills manually by picking the hole faces on the solid model. The **Use filter** option and the **Find Holes** button are unavailable.

The **Show highlighted drills direction** option enables you to display the direction of drilling by surface normals for the specific highlighted drills in the list.

The **Show all drills direction** option enables you to display the direction of drilling for all drills in the list.

The **Reverse by model selection** option enables you to reverse the direction of specific drills by selecting their faces on the model.

- **Drill Positions**

This section lists all the drills found in the model.

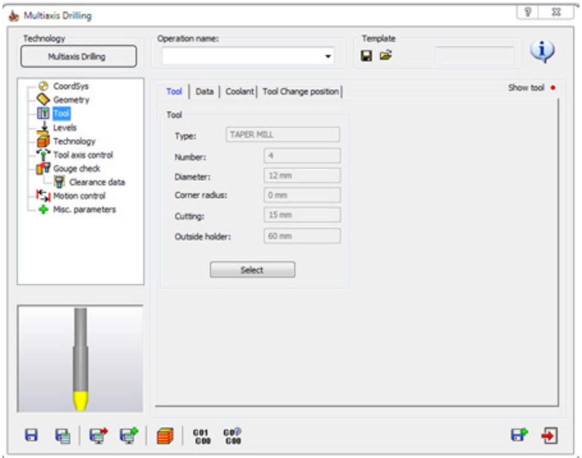
The **Reverse**  button enables you to change the direction of the drill or a group of drills selected in the list.

The **Reject**  button enables you to remove the selected drill or a group of drills from the list.

For more information on drilling geometry definition, refer to the **SolidCAM Milling User Guide** book.

17.3 Tool page

This page enables you to define the tool for the **Multiaxis Drilling** operation and to set the cutting parameters (feed and spin).

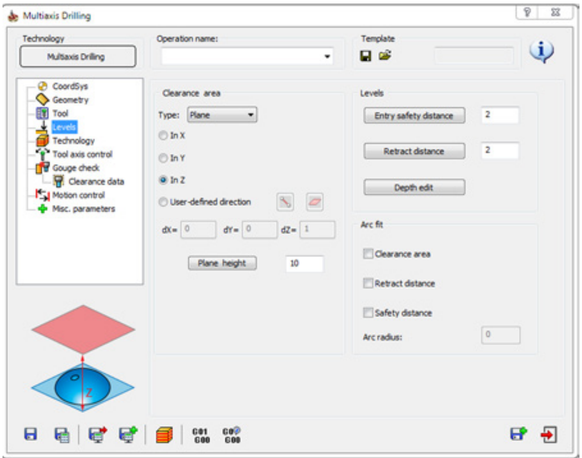


The following tool types are compatible with the **Multiaxis Drilling** operation:

- **End, Bull Nose & Ball Nose Mills**
- **Drill, Centre, Chamfer & Spot Drill**
- **Reamer**
- **Bore**
- **Tap**
- **Taper, Slot, Lollipop, Dovetail & Face Mill**
- **Thread Mill and Taper**

17.4 Levels page

This page enables you to define the machining levels for the **Multiaxis Drilling** operation.



17.4.1 Clearance area

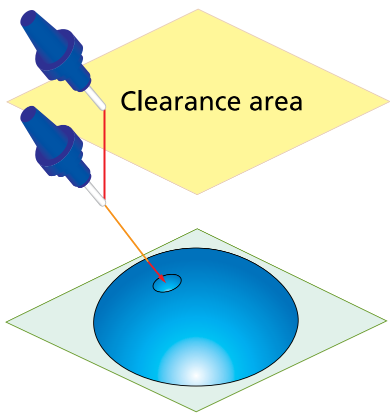
The **Clearance area** (see topic **5.1**) is the area where the tool movements can be performed safely without contact with the material. The tool movements in the **Clearance area** are performed with rapid feed.

Depending on the part shape, you can choose different clearance area types:

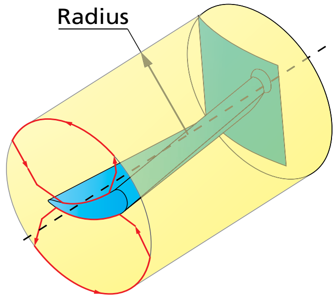
- **Plane**

  This option enables you to define the **Clearance area** by plane. The tool performs a retract movement to the **Clearance plane** and then a rapid movement in this plane.

  SolidCAM enables you to define the location and the orientation of the **Clearance plane**.



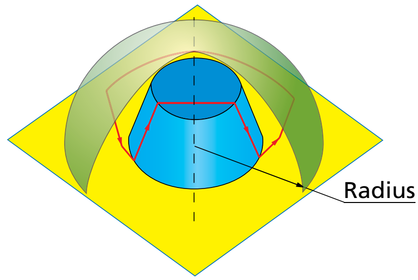
- **Cylinder**



This option enables you to define the **Clearance area** as a cylindrical surface. The tool performs a retract movement to the **Clearance cylinder**, and then performs a rapid movement along the cylinder surface.

SolidCAM enables you to define the location, orientation and radius of the **Clearance cylinder**.

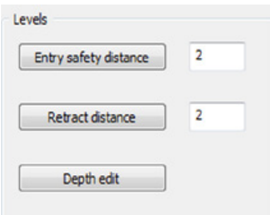
- **Sphere**



When this option is chosen, the **Clearance area** has a spherical shape. The tool performs a retract movement to the **Clearance sphere** and then a rapid movement along the sphere surface.

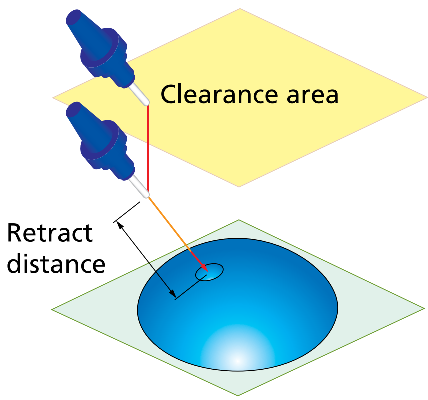
SolidCAM enables you to define the location and radius of the **Clearance sphere**.

17.4.2 Levels



This section enables you to define the **Retract** and **Safety distance** (see topic **5.2**) for the tool to approach and retract from the part.

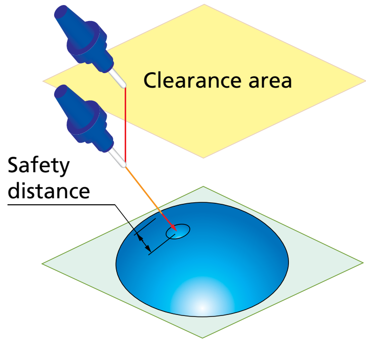
Retract distance



In the **Clearance area**, the tool rotates to the final orientation for the first cut. After the rotation, the tool performs a rapid descent movement to the level specified by the **Retract distance** parameter. The **Retract distance** is measured from the start position of the drill.

Safety distance

After the descent movement to the **Retract distance** level, the tool starts the approach movement to the material. The approach movement consists of two segments. The first segment is performed with rapid feed up to the **Entry safety distance**. From the **Entry safety distance** level, the approach movement is performed with the cutting feed.



Depth edit

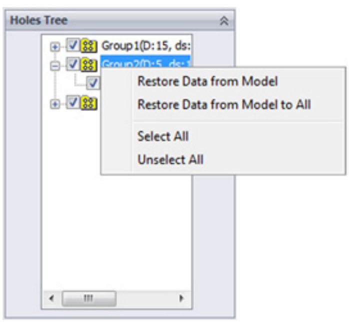
This button displays the **Depth Edit** dialog box that enables you to choose the drills to be included in the geometry and modify the geometrical parameters of the chosen drills.

- **Holes Tree**

  This section displays the list of all drills chosen for the geometry. All the drills in the list are structured in Groups. Each Group has the same **Delta start**, **Drill depth**, **Delta depth** and **Depth type** data displayed in parentheses.

  When one of the items of the list is selected, the relevant parameters are displayed in the **Delta start**, **Drill depth**, **Delta depth** and **Depth type** sections. The corresponding drills are highlighted on the solid model with the arrow indicating the machining direction.

  The right-click menu is available on each item in the list:



Restore Data from Model

This command enables you to restore the default parameters recognized on the Target model for the selected item (a group or a single drill). When this command is applied, SolidCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters.

Restore Data from Model to All

This command enables you to restore the default parameters for all the drills in the list. When this command is applied, SolidCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters.

Select All/Unselect All

These commands enable you to toggle the selection of all recognized drills.

- **Hole Diameter (D)**

  This section enables you to set the diameter value and apply it to selected drills.

- **Delta start (ds)**

  This section enables you to change the Z-value of the default drilling start point recognized on the Target model. When a positive value is entered, the start point is moved upwards from the default position. When a negative value is entered, the start point is moved downwards from the default position.

- **Drill Depth (d)**

  This section enables you to define the value of the drilling depth and apply it to selected drills.

- **Delta Depth (dd)**

  This section enables you to set the offset for the cutting depth and apply it to selected drills.

- **Depth Type**

  This section enables you to define the **Depth Type** for selected drills. You can define the diameter on the conical part of the drilling tool that will reach the specified drilling depth during the machining. You can also deepen a drilled hole in order to obtain a given diameter at the specified drill depth.

The following options are available:

Cutter Tip

The tool tip reaches the defined drilling depth.

Full Diameter

The tool reaches the defined drilling depth with the full diameter.

Diameter Value

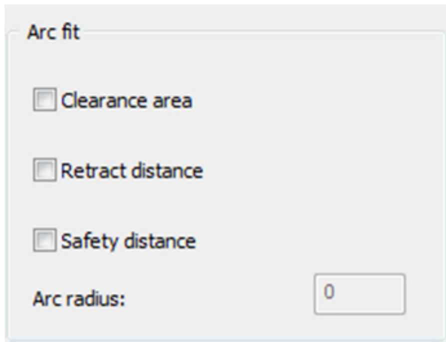
The tool reaches the defined depth with the drill cone diameter specified by the **Diameter Value** parameter.

The **Apply** button, in each of the sections described above, enables you to apply the defined parameter to the selected list item (a group or a drill).

The **All** check box enables you to apply the updated values to all the items in the list.

Arc fit

This option provides tangential arcs for the approaching and retracting link segments. You can specify the radius of the arc. This option can be applied to **Clearance area**, **Retract distance**, and **Safety distance**.

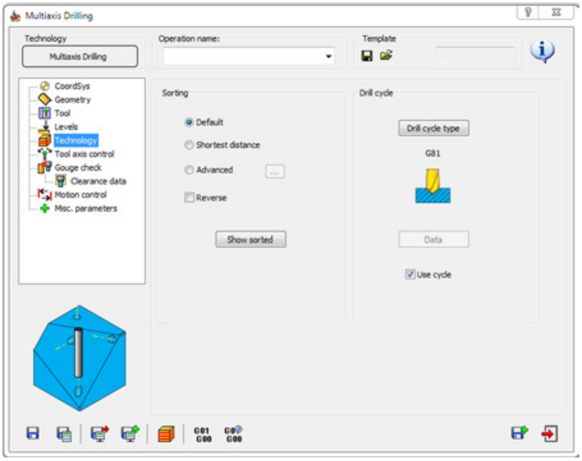


Arc radius

This option fits an arc to sharp angles in the checked areas and distances.

17.5 Technology page

This page enables you to define the technological parameters of the **Multiaxis Drilling** operation.

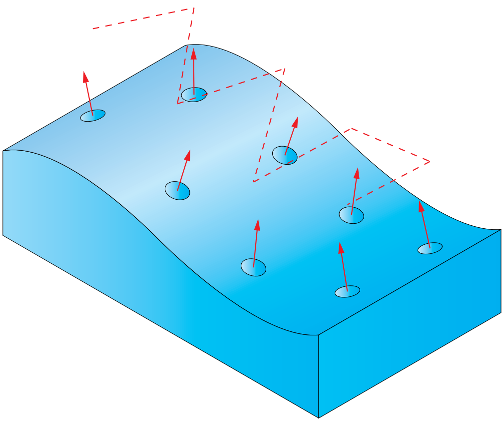


17.5.1 Sorting

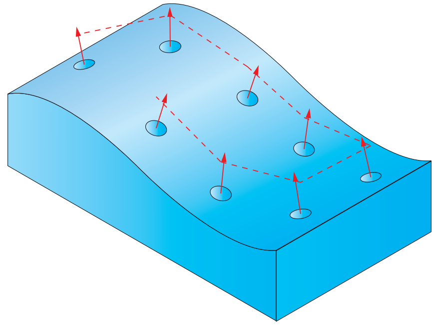
This section enables you to define the order of the drilling sequence. This option provides you with the following modes of drilling positions sorting:

Default

In this option sorting is not performed. The drills are machined in the initial order provided by the **Multiaxis Drilling** geometry.



Shortest distance



In this option drills are sorted by the shortest distance. When this option is used, SolidCAM minimizes the length of the necessary tool movement. Machining of the first drill is performed at the first point defined in the drilling geometry. Then the nearest drilling instance is chosen, i.e. the one that is located at the shortest distance from the previous drilling position.

Advanced

This option enables you to sort the drilling positions for machining of linear, circular and cylindrical drilling patterns. The ... button displays the **Advanced Sorting** dialog box.

The **Linear** tab contains the sorting methods appropriate for machining of linear drilling patterns.

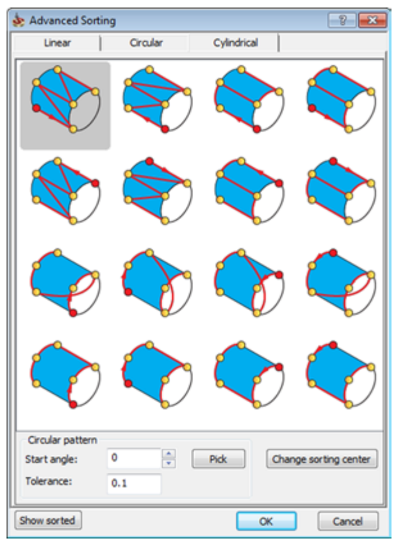
The **Circular** tab contains the sorting methods appropriate for machining of circular drilling patterns.

The **Cylindrical** tab contains the sorting methods appropriate for machining of cylindrical drilling patterns (see topic **15.5.2**).

Reverse

This check box enables you to choose the opposite direction of the drilling.

17.5.2 Sorting of cylindrical drilling patterns

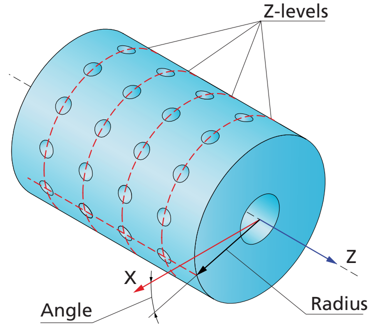


The **Cylindrical** tab of the **Advanced Sorting** dialog box contains the sorting methods applicable to cylindrical drilling patterns.

For the ordering of drilling positions by all of the cylindrical sorting methods, SolidCAM uses the center point around which the cylindrical pattern is defined. For each position of the chosen geometry, a radial vector passing through the cylinder axis and the drilling position is determined. SolidCAM then determines the angle between this vector and the X-axis at the center point. The Z-coordinates of the drilling positions also serve as a criterion for sorting.

Circular pattern

This section contains the following parameters:



- **Start angle**

  This parameter enables you to define the angle at which the start position will be chosen. This angle is defined according to the positive direction of the X-axis at the pattern center.

  The **Pick** button enables you to define the **Start angle** by picking a point on the model. The **Pick angle point** dialog box enables you to pick the position on the model and displays the coordinates of the picked position.

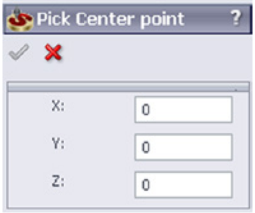
  When the position is picked and the dialog box is confirmed, SolidCAM determines the direction vector from the pattern center towards the picked position. SolidCAM automatically calculates the angle between the direction vector and the X-axis at the pattern center and displays the angle value in the **Start angle** edit box.

- **Tolerance**

  During the cylindrical sorting, SolidCAM classifies the drilling positions into groups of those located at the same angle.

  The **Tolerance** value determines if drilling positions belong to the same angle group. For each group of drilling positions located at the same angle, SolidCAM determines the start position; each additional drilling position to be included into this group must be located at the angle that is within the angular tolerance (calculated automatically according to the specified tolerance) from the angle of the start position.

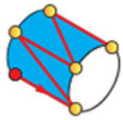
- **Change sorting center**

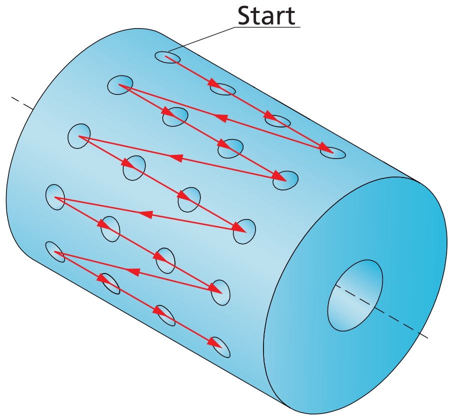
  

  By default, the pattern center is automatically defined at the origin of the Coordinate System used in the current operation. The **Change sorting center** button enables you to change the pattern center location by picking on the model. The **Pick Center point** dialog box enables you to pick the center position and displays the coordinates of the picked point.

Cylindrical sorting methods

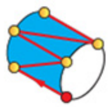
The following methods are available for advanced **Cylindrical** sorting:

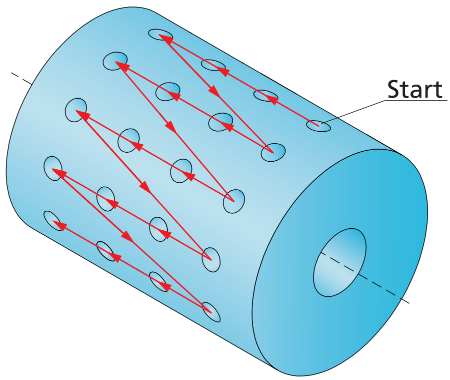




The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in an increasing order of Z-coordinates.

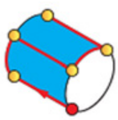
When all positions at the same angle are reached, the above order is repeated according to the angle increments in the CW direction.

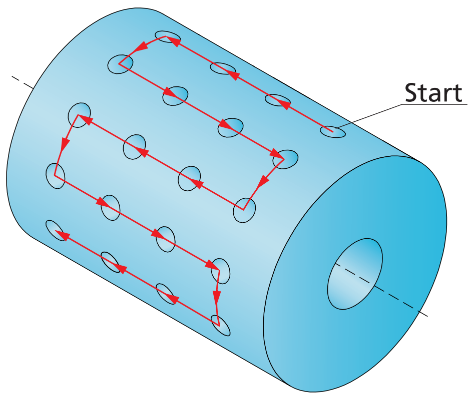




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in an increasing order of Z-coordinates.

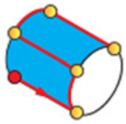
When all positions at the same angle are reached, the above order is repeated according to the angle increments in the CW direction.

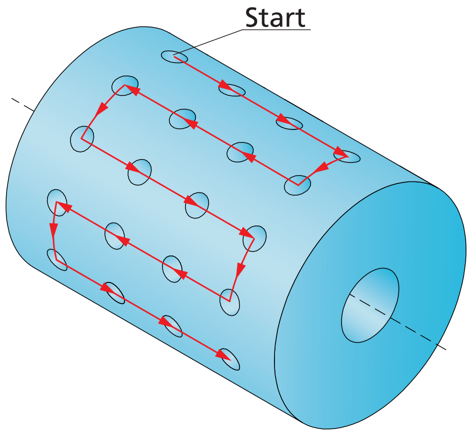




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in a decreasing order of Z-coordinates.

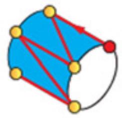
When all positions at the same angle are reached, the above order starts at the minimal Z-coordinate after an angle increment in the CW Direction. Drilling now is performed in an increasing order of Z-coordinates.

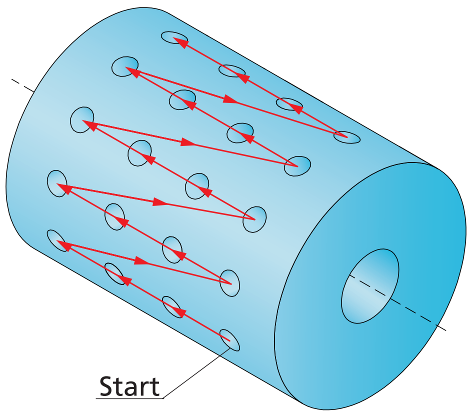




The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in an increasing order of Z-coordinates.

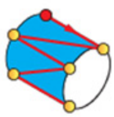
When all positions at the same angle are reached, the above order starts at the minimal Z-coordinate after an angle increment in the CW direction. Drilling now is performed in an decreasing order of Z coordinates.

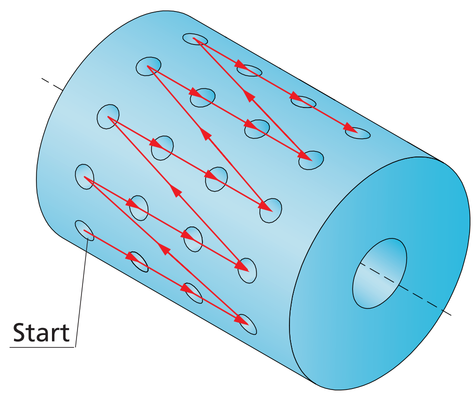




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in a decreasing order of Z-coordinates.

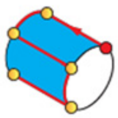
When all positions at the same angle are reached, the above order starts at the maximal Z-coordinate after an angle increment in the CCW direction. Drilling now is performed in a decreasing order of Z coordinates.

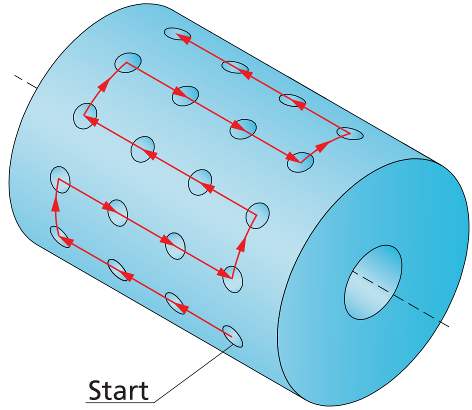




The start drilling position is placed at the minimal Z coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in an increasing order of Z-coordinates.

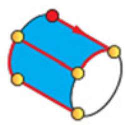
When all positions at the same angle are reached, the above order starts at the minimal Z-coordinate after an angle increment in the CCW direction. Drilling now is performed in an increasing order of Z-coordinates.

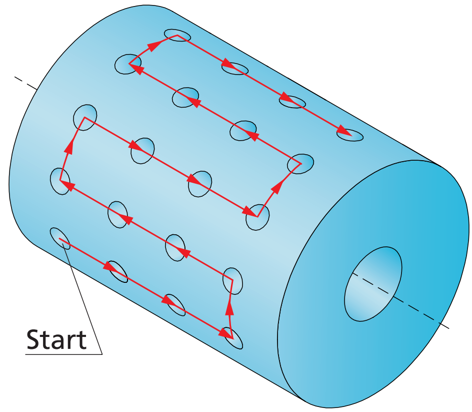




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in a decreasing order of Z-coordinates.

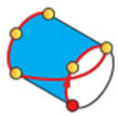
When all positions at the same angle are reached, the above order starts at the minimal Z-coordinate after an angle increment in the CCW direction. Drilling now is performed in an increasing order of Z-coordinates.

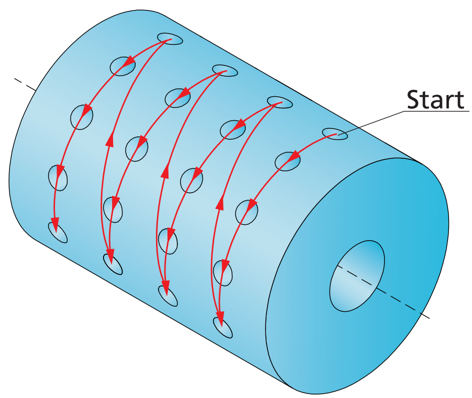




The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same angle is performed in an increasing order of Z-coordinates.

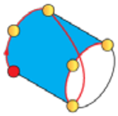
When all positions at the same angle are reached, the above order starts at the minimal Z-coordinate after an angle increment in the CCW direction. Drilling now is performed in a decreasing order of Z-coordinates.

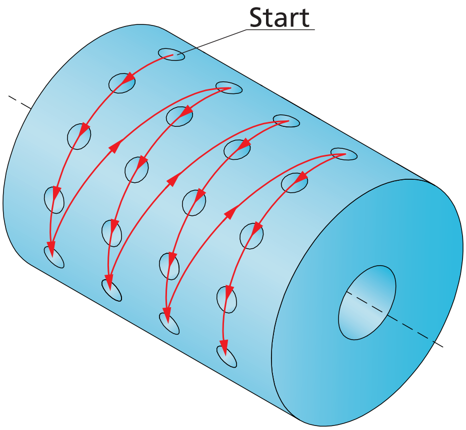




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CW direction.

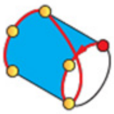
When all positions at the current Z-level are reached, the position moves to the next decreasing Z-level, at a CCW direction at an angle maximally close to the **Start angle** value. Holes at the new Z-coordinate are performed in a CW direction.

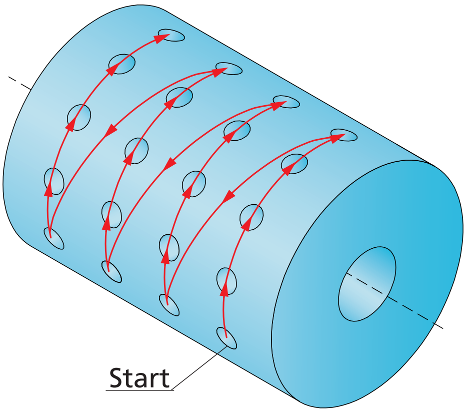




The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CW direction.

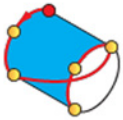
When all positions at the current Z-level are reached, the position moves to the next decreasing Z-level, at the CCW direction at an angle maximally close to the **Start angle** value. Holes at the new Z-coordinate are performed in the CW direction.

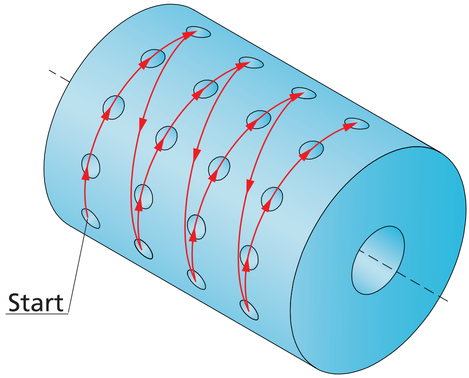




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CCW direction.

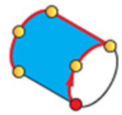
When all positions at the current Z-level are reached, the position moves to the next decreasing Z-level, at the CW direction at an angle maximally close to the **Start angle** value. Holes at the new Z-coordinate are performed in the CCW direction.

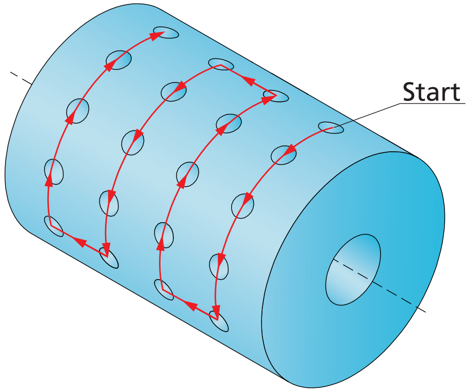




The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CCW direction.

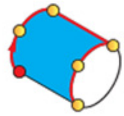
When all positions at the current Z-level are reached, the position moves to the next increasing Z-level, at the CW direction at an angle maximally close to the **Start angle** value. Holes at the new Z-coordinate are performed in a CCW direction.

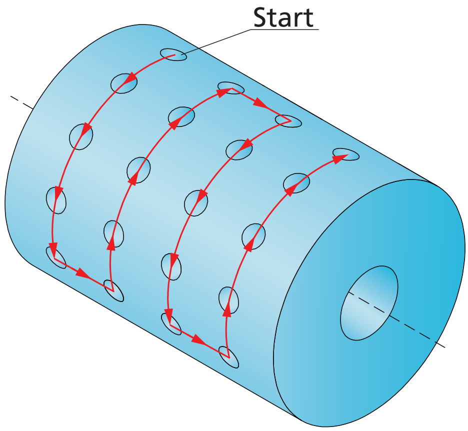




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CW direction.

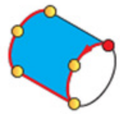
When all positions at the current Z-level are reached, the position moves to the next decreasing Z-level. Holes at the new Z-coordinate are performed in the CCW (opposite) direction.

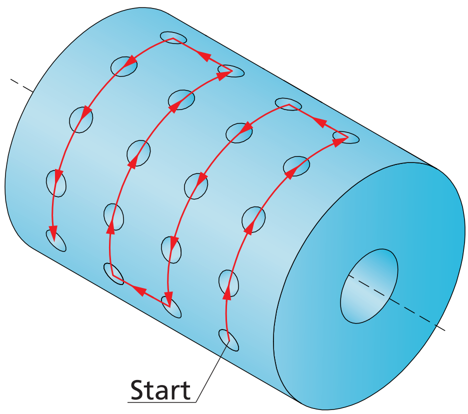




The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CW direction.

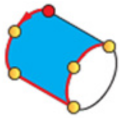
When all positions at the current Z-level are reached, the position moves to the next increasing Z-level. Holes at the new Z-coordinate are performed in the CCW (opposite) direction.

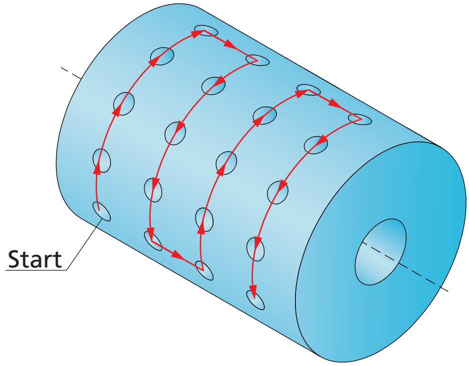




The start drilling position is placed at the maximal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CCW direction.

When all positions at the current Z-level are reached, the position moves to the next decreasing Z-level. Holes at the new Z coordinate are performed in the CW (opposite) direction.





The start drilling position is placed at the minimal Z-coordinate and at the angle maximally close to the **Start angle** value. Drilling the holes at the same Z-coordinate is performed in the CCW direction.

When all positions at the current Z-level are reached, the position moves to the next increasing Z-level. Holes at the new Z-coordinate are performed in the CW (opposite) direction.

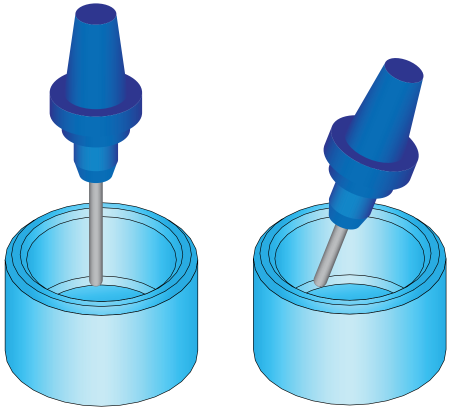
17.5.3 Drill cycle type

This section enables you to define the drilling cycle that will be used in the current operation. For more information about the drilling cycles, refer to **SolidCAM Milling User Guide** book.

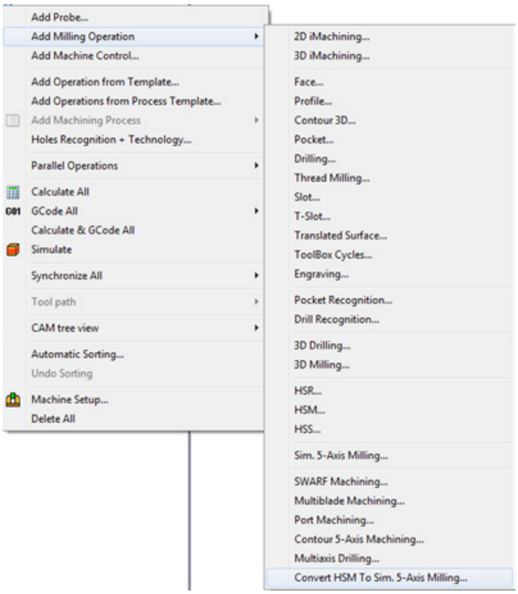
Converting HSM to Sim. 5-Axis Milling

18

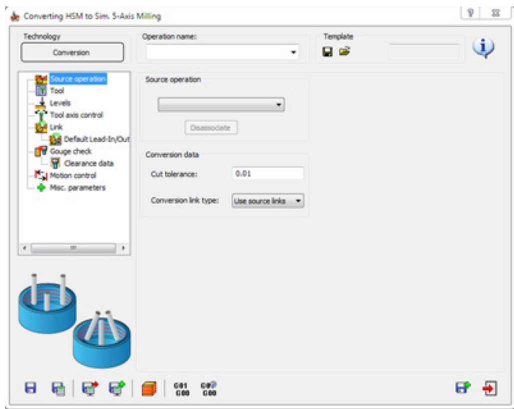
In some machining cases, there is a need to perform machining of 3D parts using the 5-axis capabilities. For example, 3D machining of deep cavities requires the use of tools of great length, which can cause tool breakage; the same cavities can be machined using a tool of smaller length while tilting this tool to follow the same tool path.



To convert a HSM operation into a 5-Axis one, choose **Convert HSM to Sim. 5-Axis Milling** from the **Add Milling Operation** submenu in the **SolidCAM Manager** tree.



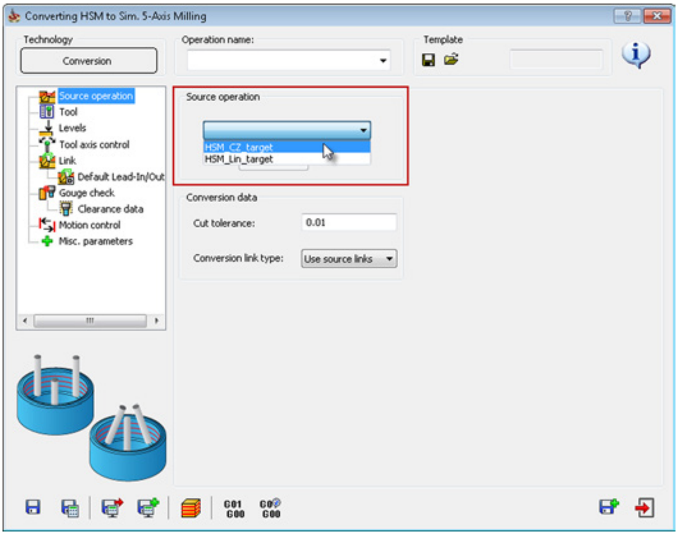
The **Converting HSM to Sim. 5-Axis Milling** dialog box is displayed.



When the converted operation is saved and calculated, the source operation is suppressed in the **SolidCAM Manager** tree.

18.1 Source operation

This page enables you to choose original operation that will be converted and define a number of the conversion parameters.



Source operation

In this section, you need to choose the operation to be converted from the corresponding combo-box. The **Disassociate** button cancels the connection between the source operation and the converted one.

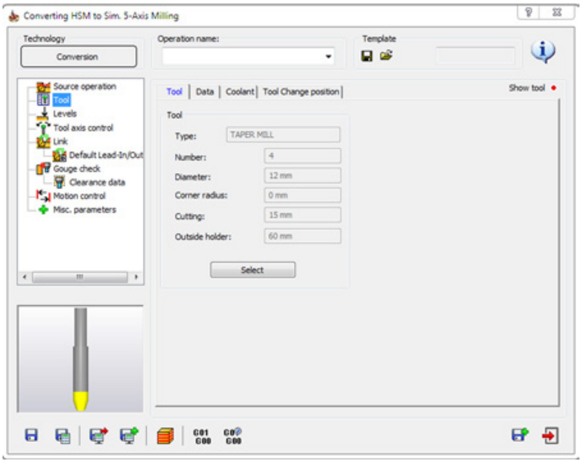
Conversion data

The **Cut tolerance** value defines the tool path accuracy.

The **Conversion link type** option lets you to choose between using the original source links from the HSM operation or relinking the tool path. When the **Relink** option is chosen, the options in the **Levels** and **Link** pages become available for editing.

18.2 Tool

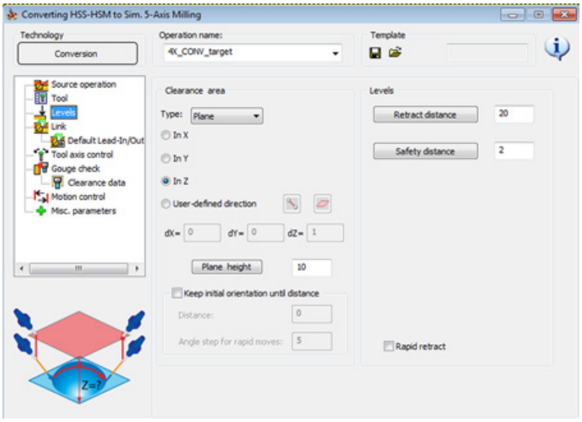
This page enables you to define the tool and the related parameters such as feed and spin for the operation.



To convert a HSM operation into a 5-axis one, a tool of the **Ball-nosed mill** type must be used in the source operation.

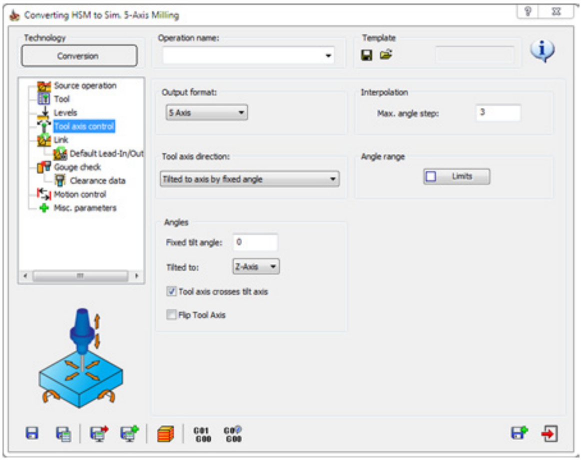
18.3 Levels

This page enables you to define the machining levels for the operation (see chapter **5**).



18.4 Tool axis control

This page enables you to define the orientation of the tool axis during the Sim. 5-axis machining.



Output format

This parameter (see topic **7.1**) enables you to define the output format of the current Sim. 5-Axis operation. For this operation, either **3-Axis** or **4-Axis** format can be used.

Tool axis direction

This section (see topic **7.3**) enables you to choose the tool tilting strategy. The tool tilting strategies enable you to define the orientation of the tool axis during the machining relative to the surface normal.

Angles

This parameter (see topic **7.3**) enables you to define the tilting angles and related parameters.

Interpolation

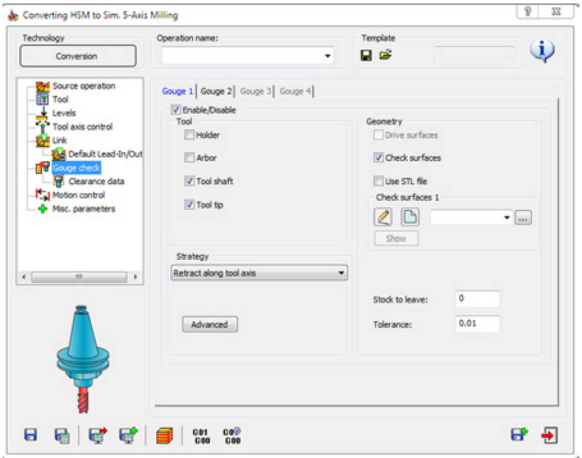
The **Max. angle step** parameter (see topic **7.2**) enables you to define the maximal allowed angle change between the tool axes, at two consecutive tool positions.

Angle range

The **Limits** option (see topic **7.4**) displays the **Limits** dialog box that enables you to limit the tool tilting along the tool path.

18.5 Gouge check

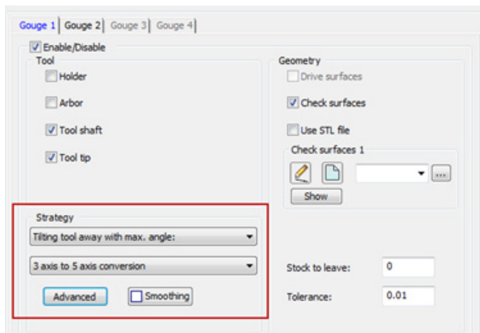
This page enables you to avoid the tool gouging of the selected drive and check surfaces (see chapter **9**).

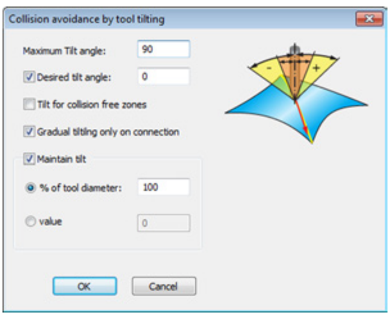


In this page, SolidCAM enables you to define the parameters of automatic collision avoidance for 5-axis tool paths by tool inclination. The basic principle of automatic collision avoidance is to free the user from manual choice of tilting angles: you only need to specify the desired tilting angle, the maximal angle allowed, and whether tilting will occur in areas where no collisions are detected.

3 axis to 5 axis conversion

This option enables you to define the advanced parameters for tool tilting. The **Advanced** button displays the **Collision avoidance by tool tilting** dialog box that enables you to define the following tilting parameters.





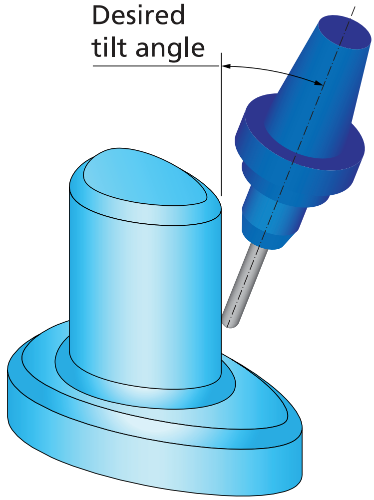
- **Maximum tilt angle**

  This parameter defines the maximal angle of tool tilting allowed to avoid collision. When this value is found not sufficient to avoid collisions, the tool returns to the Clearance level, and the system searches the remaining calculated tool path for gouge-free passes that can be performed according to the angle limitation. If no possible passes are found, machining is not continued.

- **Desired tilt angle**

  When this check box is selected, the tool is tilted to the preferred angle specified in the related edit box, when a collision is detected. When this value is found not sufficient to avoid collision, it is increased up to the specified **Maximum tilt angle** value.

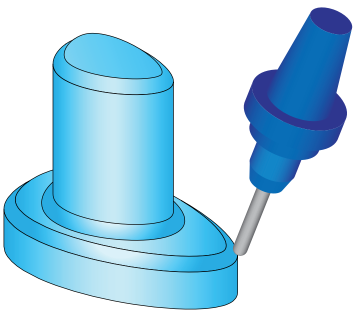
  When this check box is not selected, the system automatically calculates the tool tilting angle in the range from **0°** up to the specified **Maximum tilt angle** value.



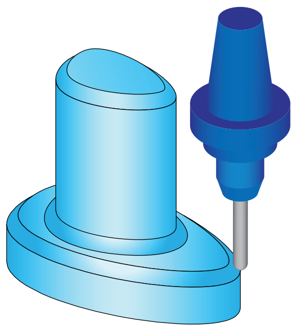
- **Tilt for collision free zones**

  When this check box is selected, the tool is also tilted to the specified **Desired tilt angle** in areas where no collisions are detected.

  When this check box is not selected, the tool sticks to its initial axis defined for 3-axis tool path, when it machines the collision-free areas.



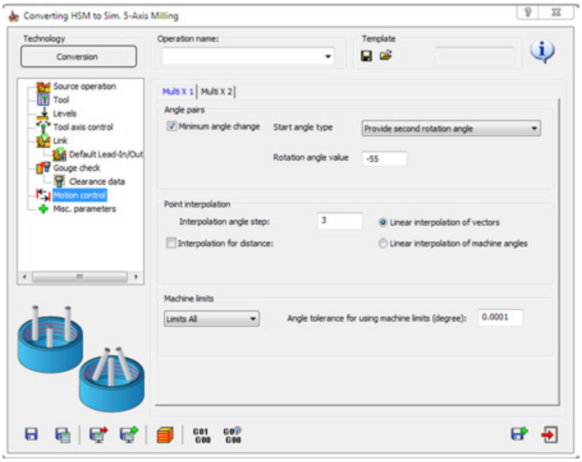
Tilt for collision free zones check box is selected



Tilt for collision free zones check box is not selected

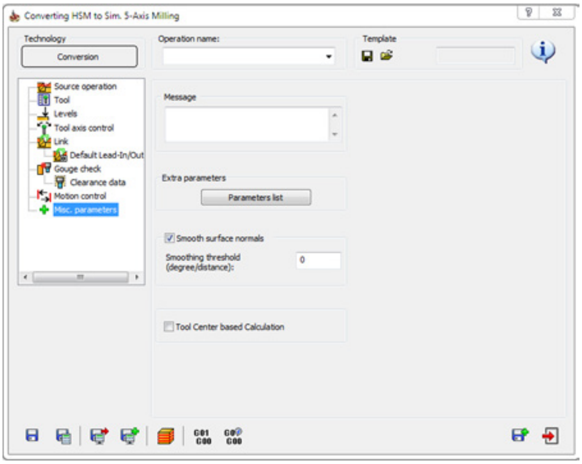
18.6 Motion control

This page enables you to define the parameters related to the kinematics and special characteristics of the CNC-machine.



18.7 Miscellaneous parameters

This page enables you to define a number of miscellaneous parameters and options related to the 5-axis tool path calculation.

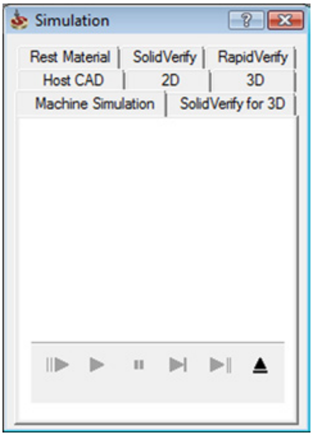


Machine simulation

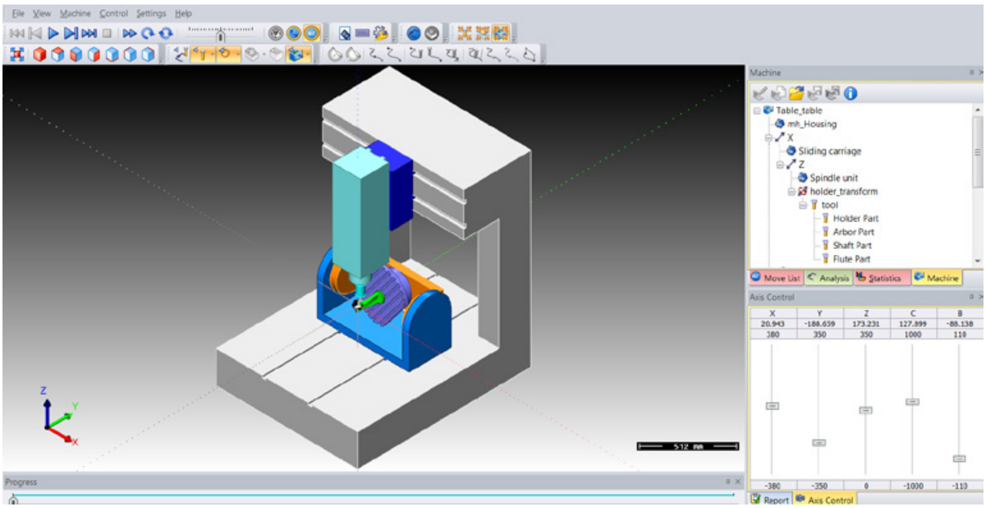
19

SolidCAM provides you with the **Machine simulation** mode that enables you to perform the machining simulation and tool path verification using the kinematics of the CNC-machine.

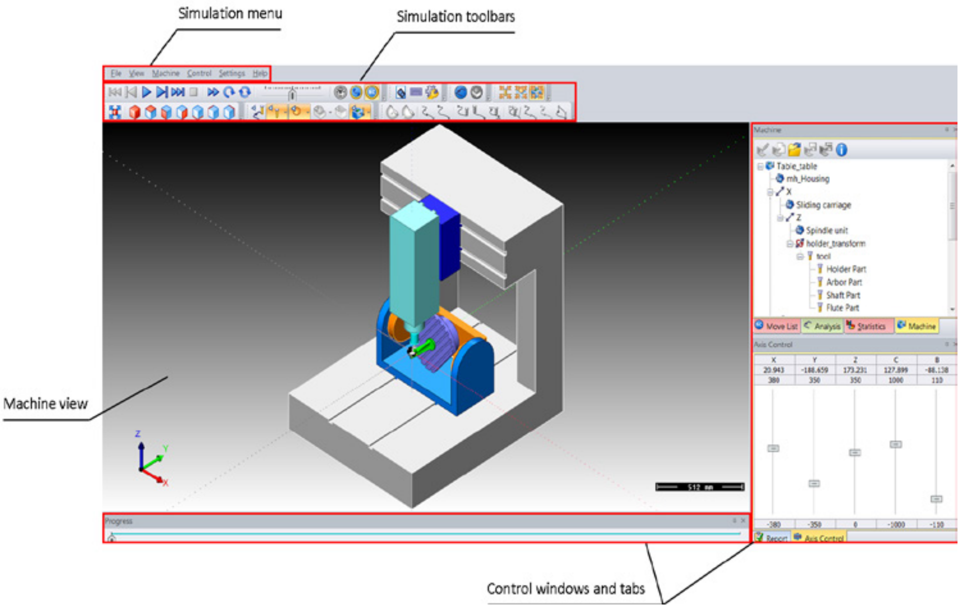
In the **Simulation** control panel choose the **Machine simulation** mode.



The Machine Simulation window is displayed.



19.1 Machine simulation user interface



The user interface of the **Machine simulation** window is divided into the following areas:

- **Machine view**

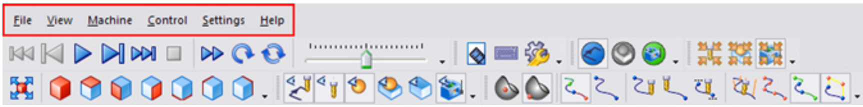
This area is the graphic area where you can see your CNC-machine and the machined part. All the tool movements along the tool path and motions of the CNC-machine components are displayed in this area.

- **Simulation control**

This area is divided into the following elements:

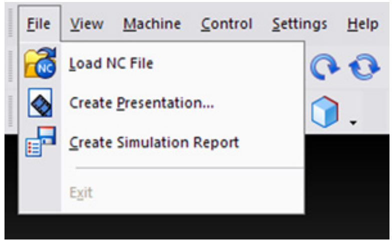
- **Simulation menu**
- **Simulation windows and tabs**
- **Simulation toolbars**

19.1.1 Simulation menu



The Windows-style menu located above the graphic area contains the following sets of simulation control options:

File



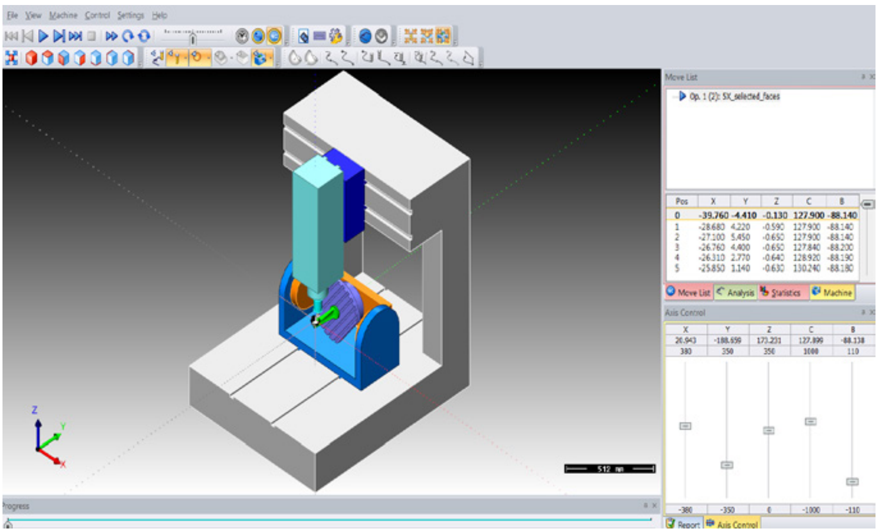
This menu item contains the file managing commands:

- **Load NC File**

  When you perform simulation in the **Machine simulation** mode, SolidCAM saves all the simulation data in the **Output.sim** file located in the CAM-Part folder. The **Load NC file** option enables you to load an existing NC file containing the simulation data.

- **Create Presentation**

  This option enables you to create a self-extracting executable file containing the current CNC-machine and NC file. When you run the generated executable file, it unpacks in the system temporary folder and displays the standalone window.

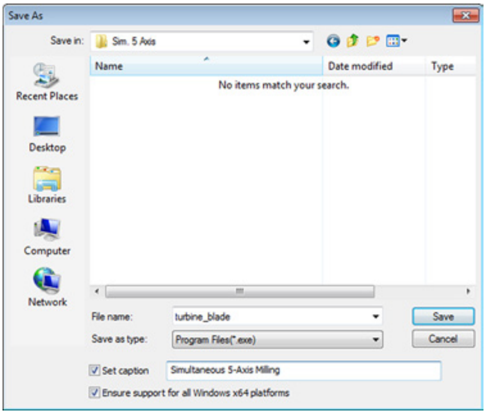


This window provides you with the CNC-machine data, tool path and all the tools to run and control the simulation and verification of the tool path. Using this file, the simulation can be displayed on any computer even without SolidCAM being installed.

When you choose this command, the **Save As** dialog box is displayed. In this dialog box, you have to enter the name for the presentation file and specify its location.

To add a title to your presentation, select the **Set caption** check box and enter the title in the relevant text box.

To enable the possibility to run the presentation file on all Windows platforms, select the **Ensure support for all Windows x64 platforms** check box.

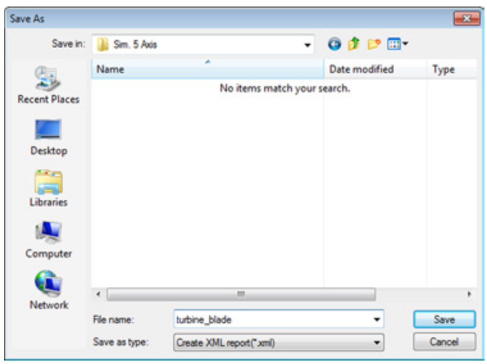


The **Create Presentation** option is also available upon clicking the  button in the **Setting** toolbar.

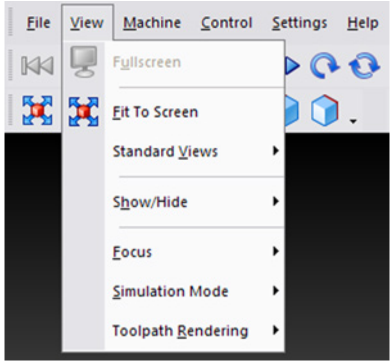
- **Create Simulation Report**

This option enables you to save a simulation report file in the XML format. When you choose this command, the **Save As** dialog box is displayed. In this dialog box, you have to enter the name for the report file and specify its location.

The generated report file contains the information about the simulation running time and the memory it requires, the measurement units, collision checking, the tools used, etc.



View



This menu item contains the commands responsible for the display of the simulation model in the **Machine view** area.

Fullscreen

This command enables you to extend the simulation window to the entire screen size.

Fit to screen

This command enables you to adjust the simulation model size to the graphic area.

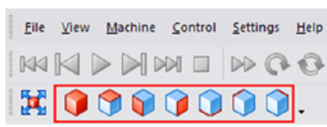
This option is also available as the button in the **View** toolbar.

Standard Views

This submenu contains commands that enable you to rotate the simulation model to the appropriate standard view.

- **Isometric**: this button enables you to rotate the simulation model into the isometric view.
- **Top**: this button enables you to rotate the simulation model into the top side view.
- **Front**: this button enables you to rotate the simulation model into the front side view.
- **Right**: this button enables you to rotate the simulation model into the right side view.
- **Bottom**: this button enables you to rotate the simulation model into the bottom side view.
- **Left**: this button enables you to rotate the simulation model into the left side view.
- **Back**: this button enables you to rotate the simulation model into the back side view.

These commands are also available as buttons in the **View** toolbar.



Show/Hide

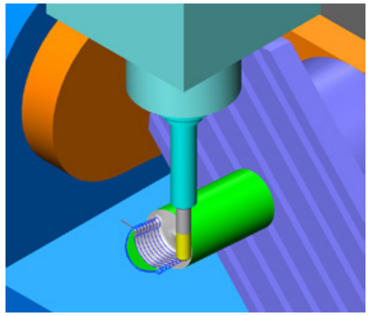
This submenu contains commands that enable you to control the display of various machine and model components in the **Machine view** area.

-  **Tool path**: this button enables you to toggle the display of the tool path in the graphic area of the simulation.

-  **Tool**: this button enables you to toggle the display of the tool path in the graphic area of the simulation.

-  **Workpiece**: this button enables you to toggle the display of the workpiece in the graphic area of the simulation.

-  **Stock**: this button enables you to toggle the display of the stock model in the process of machining.



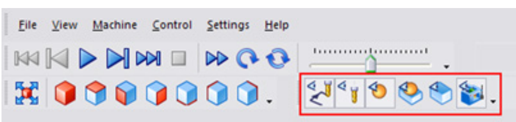
When the stock is displayed, you can perform solid verification of the material cutting process in the **SolidVerify** mode integrated into the Machine simulation. The simulation is performed by dynamic subtraction of the tool solid model (using solid Boolean operations) from the stock solid model.

To perform solid verification on the stock model, select the **Enable verification** check box under **Solid verification** section in the **Machine simulation** page of the **SolidCAM Settings** dialog box.

-  **Initial Stock**: this button enables you to toggle the display of the stock initial state before the machining.

-  **Machine Housing**: this button enables you to toggle the display of the machine housing in the graphic area of the simulation.

These commands are also available as buttons in the **Visibility** toolbar.



- **Compass**: this button enables you to toggle the display of the coordinate system in the bottom left corner of the graphic area of the simulation.

- **Ruler**: this button enables you to toggle the display of the ruler in the bottom right corner of the graphic area of the simulation.

- **Center point**: this button enables you to toggle the display of the center point in the graphic area of the simulation.

Focus

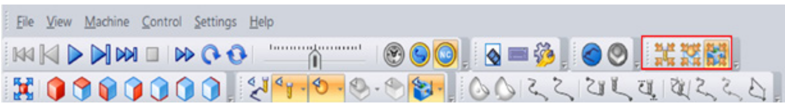
This submenu contains commands that enable you to directly zoom and observe simulation model elements in stationary position.

- **Tool Focus**: this button enables you to display the tool in a close view.

- **Workpiece Focus**: this button enables you to display the machined workpiece in a close view.

- **Machine Focus**: this button enables you to display the entire machine model in a close view.

These commands are also available as buttons in the **Focus** toolbar.



Simulation mode

This submenu contains the following modes:

- **Toolpath mode:** the tool path is displayed during the simulation.

- **Material mode:** the material removal is displayed during the simulation.

- **Kinematic mode:** the machine kinematics is displayed without the tool path and material removal.

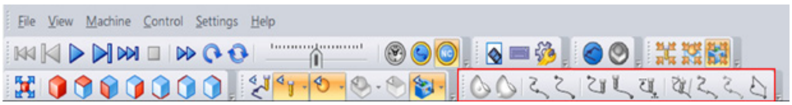
Toolpath mode and **Material mode** are also available as buttons in the **Simulation modes** toolbar.

Toolpath rendering

This submenu contains commands that enable you to choose the mode of tool path display.

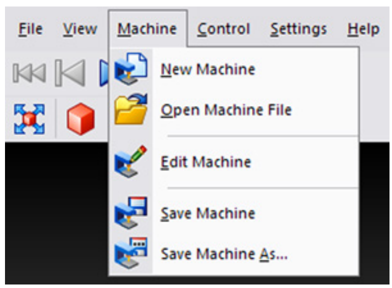
- **Tool center/Tool tip:** these commands enable you to display the tool path relative to the center or to the tip of the tool.
- **Display All Operations/Display Current Operation:** these commands enable you to display the tool path for all of the part operations all at once or only for the current one.
- **Follow/Trace/Segment:** these commands enable you to display the already machined tool path (**Follow**), the one to be machined (**Trace**) or the segments of the tool path which are currently being machined (**Segment**).
- **Tool Vector/Toolpath points:** these commands enable you to display the vectors of the tool tilting relative to the machined surface (**Tool Vector**) and the tool path by sequence of points (**Toolpath points**).
- **Leads/Links**: these commands enable you to toggle the display of the tool approach and linking movements.

These modes are also available as buttons in the **Toolpath rendering** toolbar.



Machine

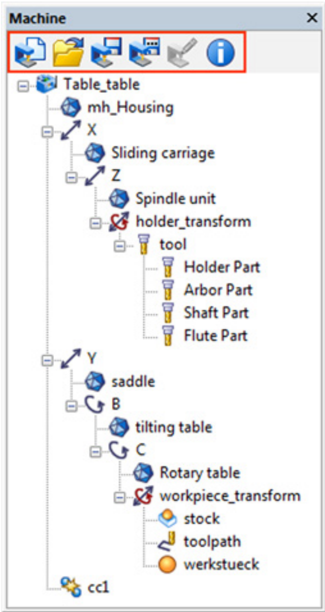
This menu item contains the following commands:



- **New Machine**: this command enables you to create a new machine definition.
- **Open Machine File**: this command enables you to load a different existing machine.
- **Edit Machine**: this command allows editing the current machine definition. This option is enabled only in special cases.

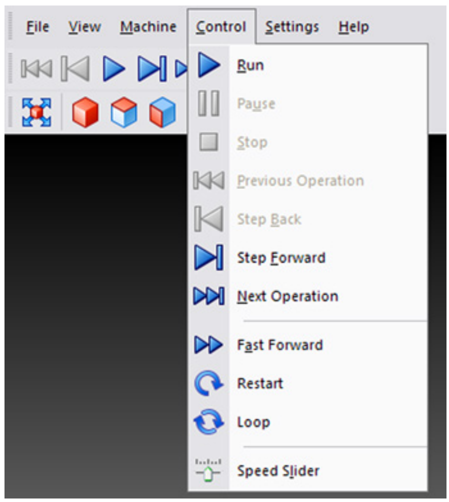
- **Save Machine**: this command enables you to save the edited machine definition.
- **Save Machine As**: this command enables you to save the edited machine definition under a different name and/or in a different folder.

These commands are also available as buttons in the **Machine** tab.



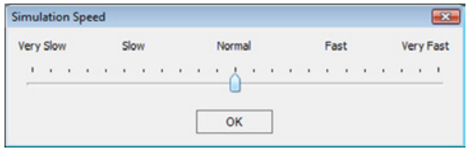
Control

This menu item contains commands that enable you to control the progress of the simulation.

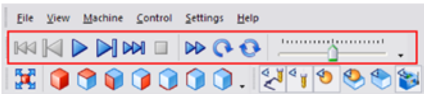


- **Run**: this command enables you to start the simulation.
- **Pause**: this command enables you to pause the simulation.
- **Stop**: this command enables you to stop the simulation.
- **Step Back/Step Forward**: these commands enable you to perform the simulation in the step-by-step mode.

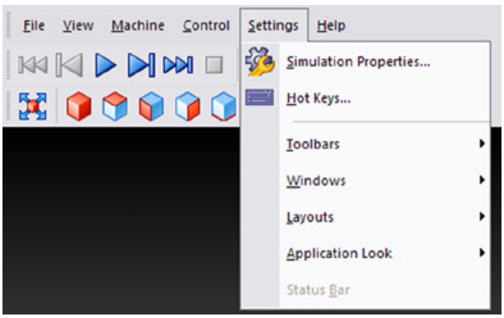
-  **Previous Operation/Next Operation**: these commands enable you to switch to the previous/next operations.
-  **Fast Forward**: this command enables you to perform the simulation in the system memory and display the final result.
-  **Restart**: this command enables you to start the simulation from the very beginning.
-  **Loop**: this command enables you simulate the operations in a closed cycle.
-  **Speed Slider**: this command enables you to display the **Simulation Speed** dialog box. This dialog box contains the slider to control the simulation speed.



 These commands are also available as buttons in the **Control** toolbar.



Settings



This menu item contains the commands that enable you to manage the simulation settings and to adjust the simulation interface in accordance with your requirements.

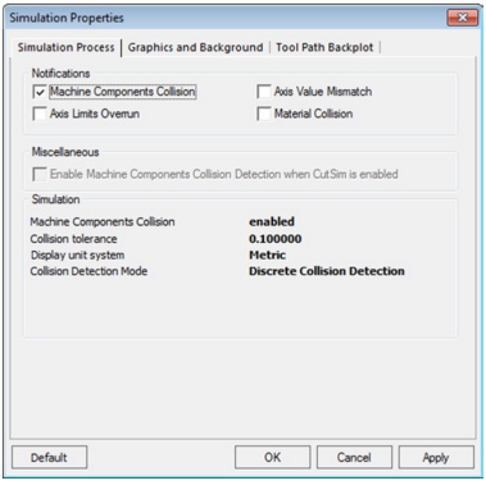
Simulation Properties

This command displays the **Simulation Properties** dialog box that enables you to edit the simulation settings.

This dialog box contains the following sets of parameters for managing the Machine simulation settings:

Simulation Process

This tab contains options that affect the simulation when it is running.



The **Notifications** section enables you to turn on the display of notifications in case of the following events reported:

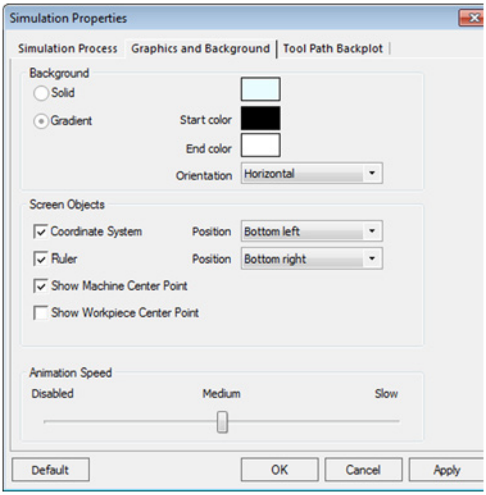
- **Machine Components Collision:** collision between tool and machine components
- **Axis Limits Overrun:** when one of the moving parts of the CNC-machine exceeds a limit defined in the CNC-machine definition
- **Axis Value Mismatch:** when wrong data is detected, e.g. wrong tool dimensions
- **Material Collision:** collision between tool and workpiece

The **Miscellaneous** section contains the **Enable geometry collision checking when CutSim is used** check box that activates the collision checking on the geometry in the material removal mode.

The **Simulation** section contains the summary of the simulation data.

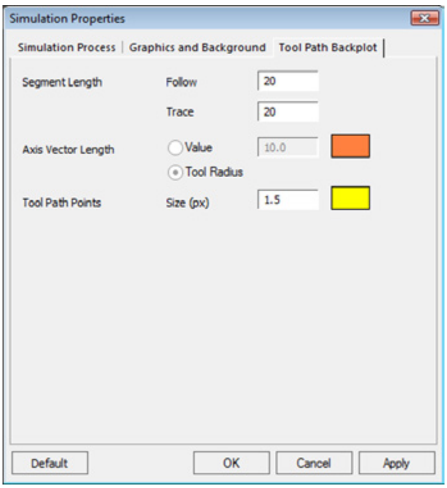
Graphics and Background

This tab contains options for setting the background color of the simulation window, for display of Coordinate System and ruler, and the slider to control the visualization speed.



Tool Path Backplot

This tab contains parameters of tool path visualization.



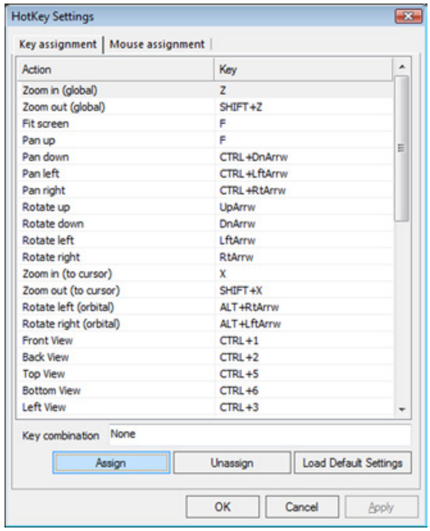
The **Segment Length** option enables you to set the length of tool path segments displayed when running the simulation in the **Follow/Trace** modes.

The **Axis Vector Length** option enables you to set the length of the tool tilting vector when running the simulation in the **Tool Vector** mode. This length can be set as a value or as the radius of the tool used for the operation. The colored rectangle on the right enables you to set the color of the vectors visualization.

The **Tool path points** option enables you to set the size (in pixels) of the tool path points when running the simulation in the **Tool path points** mode. The colored rectangle on the right enables you to set the color of the points visualization.

Hot Keys

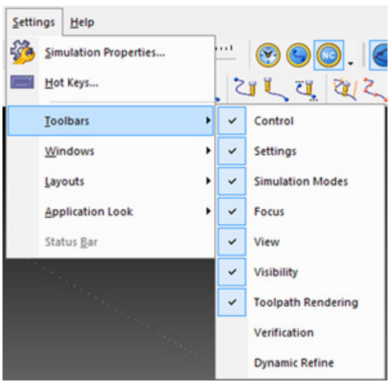
This command displays the **HotKey Settings** dialog box that enables you to define hot keys and mouse settings for the simulation control.

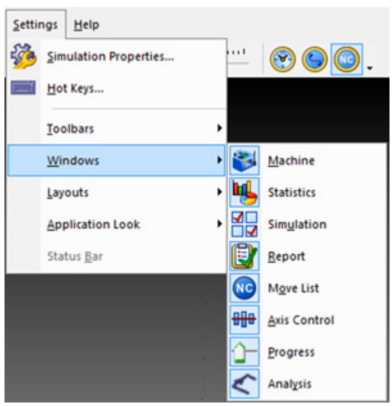


Choose an appropriate operation in the **Key assignment** or **Mouse assignment** section and assign the required combination of the keyboard keys and mouse buttons.

Toolbars

This submenu lists the simulation control toolbars that can be displayed or hidden by clicking on each item in this list. The **Customize** option enables you to customize the toolbars and separate commands that can be used during the simulation.

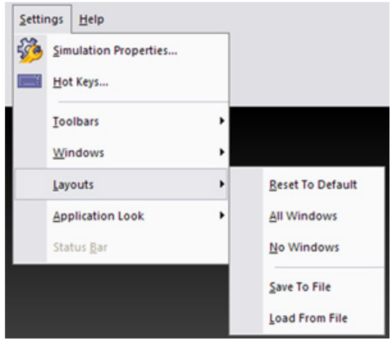




Windows

This submenu lists the simulation windows and tabs that enable you to define and control the CNC-machine and other data used for the simulation. These windows are displayed in tab view to the right from the graphic area (the **Report**, **Machine** and **Analysis** tabs and the **Move List** window) or at the bottom of the main SolidWorks window (the **Progress** bar and the **Axis Control** window).

Layouts



This submenu lists the simulation window layout options.

The **Reset to default** option resets the window layout to default.

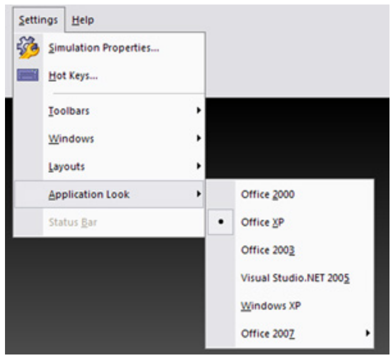
The **All Windows** option displays all windows and tabs listed in the **Windows** submenu.

The **No Windows** option hides all windows and tabs listed in the **Windows** submenu. Only the graphic area is displayed.

The **Save to file** option enables you to save the current layout scheme as a ***.layout** file in the machine definition folder.

The **Load from file** option enables you to load a previously saved layout file.

Application Look



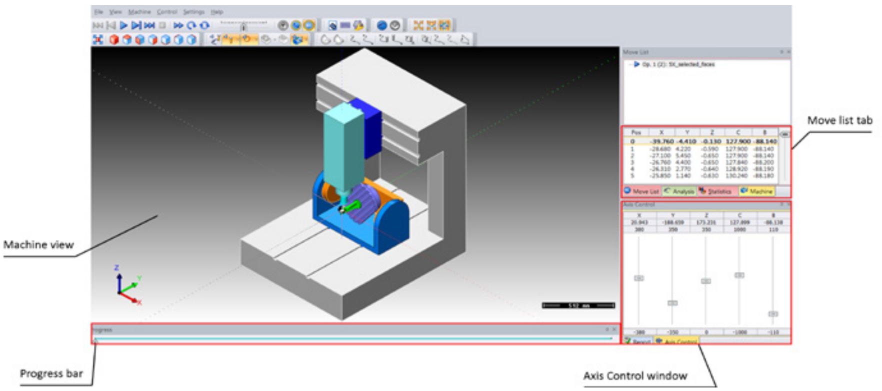
This submenu enables you to adapt the machine simulation interface to the color schemes of commonly used applications such as **MS Office**, **Windows XP**, etc.

Help

This menu item displays the information and help issues regarding the Machine simulation feature.

19.1.2 Simulation windows

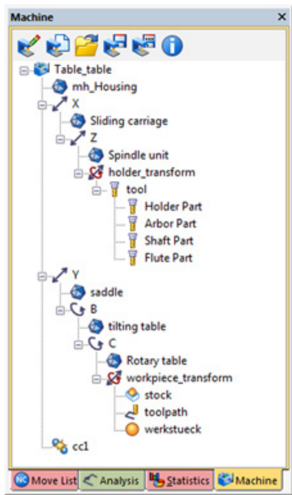
The simulation windows and tabs enable you to define and control the CNC-machine data and other parameters used for the simulation. Each window or tab can be undocked from its default location by clicking on its caption and dragging, and docked at your convenience around the **Machine view** area.



Machine

This tab displays the CNC-machine definition tree and enables you to define the CNC-machine and manage the CNC-machine components displayed in the graphic area.

The buttons in the tab toolbar enable you to manage existing machine definitions and add new ones.

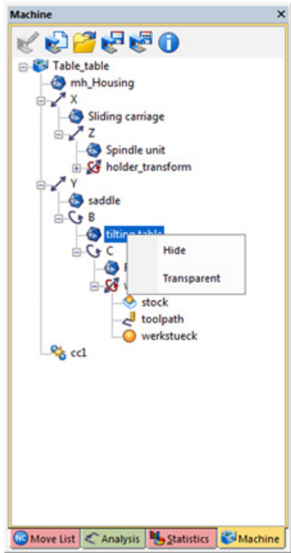


- **Edit Machine**: this button allows editing the current machine definition. This option is enabled only in special cases.
- **New Machine**: this button enables you to create a new machine definition.
- **Open Machine File**: this button enables you to load a different existing machine.
- **Save Machine**: this button enables you to save the edited machine definition.
- **Save Machine As**: this button enables you to save the edited machine definition under a different name and/or in a different folder.

- **Info**: this button toggles the display of the information about the listed machine components.

The CNC-machine definition tree displays all components of the CNC-machine used for the machining of the current CAM-part. The tree displays all the structure of the CNC-machine and the relation between all the CNC-machine components.

SolidCAM enables you to manage the CNC-machine components using the right-click menu available on each component.



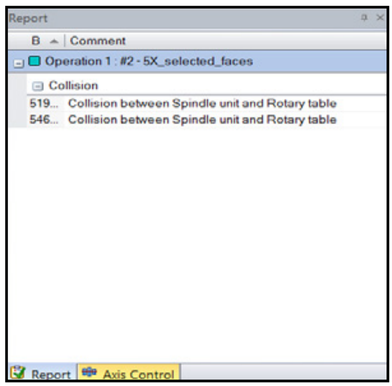
- **Show/Hide**

  This option enables you to show/hide the chosen component of the CNC-machine.

- **Transparent/Opaque**

  This option enables you to control the transparency of the chosen component of the CNC-machine.

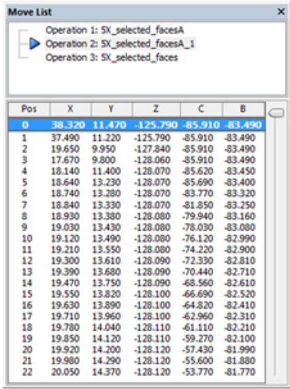
Report



This tab lists the operations with tools used and all events that happen during simulation. The items in the report are listed in a tree format structure as operations followed by the tool number and the tool definition. The following event types can be reported:

- **Value underflows and overflows**
- **Collision events**

Move list



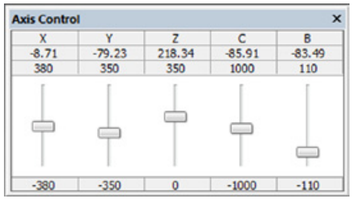
This window displays the lines of the GCode as the operation is running on; the active GCode line is highlighted.

SolidCAM enables you to display coordinates relative to the CAM-Part coordinate system or to the CNC-machine origin, depending on the **Machine simulation** settings.

The slider to the right enables you to navigate through the GCode.

The **Collision** and **Out of limits** icons appear to the left of the GCode string in case of an appropriate event.

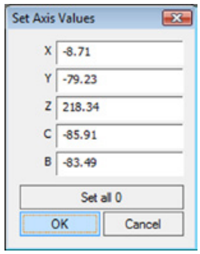
Axis Control



This window enables you to control the tool location manually using the axis sliders. It displays the current coordinates of the CNC-machine. Each axis has a control slider that enables you to perform manual movements within the specified limits.

The manual axis control cannot be used when the simulation is in progress. Stop the simulation to enable it.

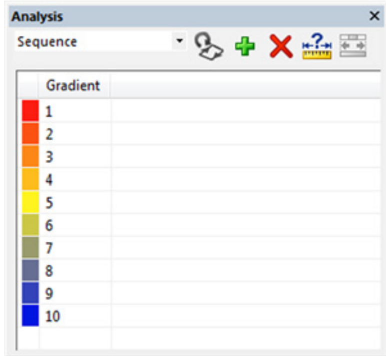
Right-clicking on the coordinate sliders displays the **Set Axis Values** dialog box that enables you to enter the values for the coordinates.



Progress

The **Progress** bar shows the advance of the simulation process. It consists of a slider that moves as the simulation is running on and a colored stripe that represents different tools by different colors. The colors of the tools are also displayed in the **Report** tab.

Analysis



This tab contains color representation of various elements of the simulation to facilitate the visualization. Choose an element from the list to display its analysis in colors.

You can change the color for each item by double-clicking on the corresponding rectangle and choose the desired color from the displayed Windows-style **Color** dialog box.

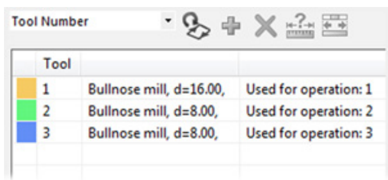
Toolbar buttons

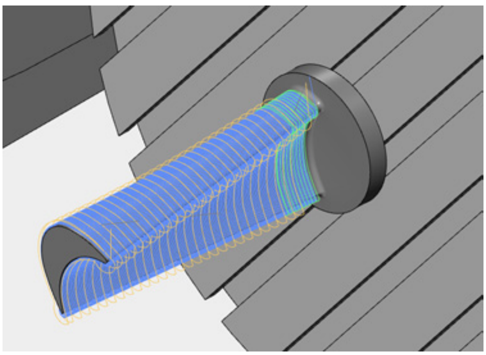
-  **Refresh**: this button enables you to update the simulation when changes have been made in the analysis settings.
-  **Add**: this button enables you to add values into the table.
-  **Remove**: this button enables you to remove selected values from the table.
-  **Adjust**: this button enables you to set limitations for specific parameters to display the tool path in different colors according to the defined settings.
-  **Auto adjust**: when you click this button, the system automatically sets the parameter ranges for the defined settings.

Parameters

- **Tool number**

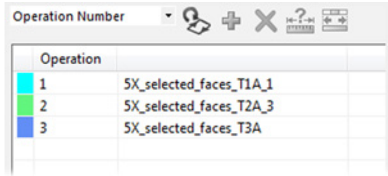
  When you choose this element from the list, the table below displays the tool path color scheme according to the tools used in part operations. The tools are numbered in the corresponding column and represented by rectangles of different colors in the left most column.

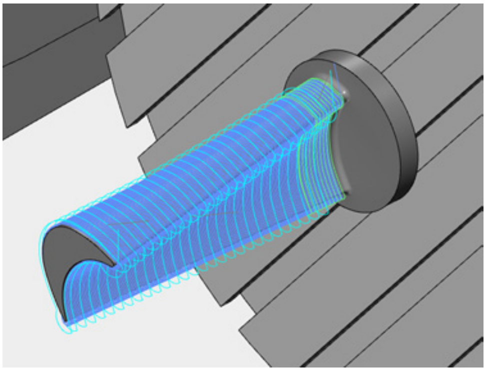




- **Operation number**

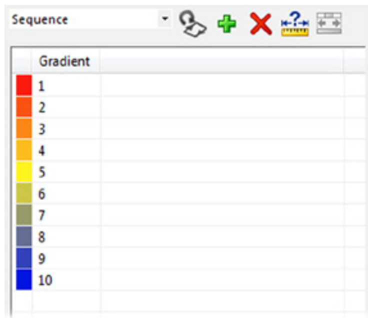
When you choose this element from the list, the table below displays the tool path color scheme according to the part operations. The operations are numbered in the corresponding column and represented by rectangles of different colors in the left most column.

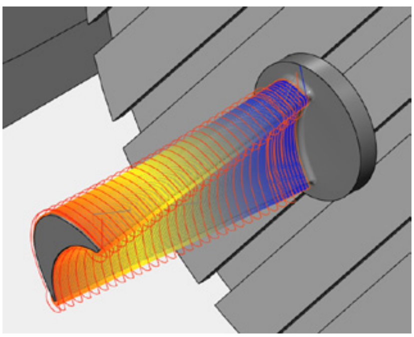




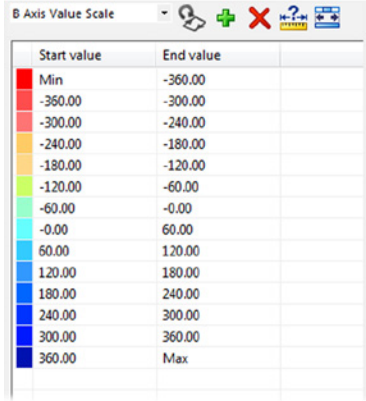
- **Tool path sequence**

When you choose this element from the list, the tool path is represented in a gradient color scale according to the progress of machining. This scale enables you to easily identify the start point and the end point of the machining, the cutting method (e.g. **Zigzag** or **One way**), the cut order (e.g. from outside to inside), and other machining parameters.





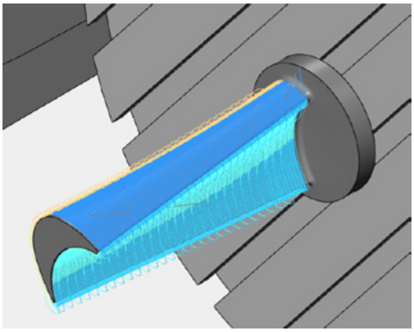
- **B Axis Value Scale/C Axis Value Scale**



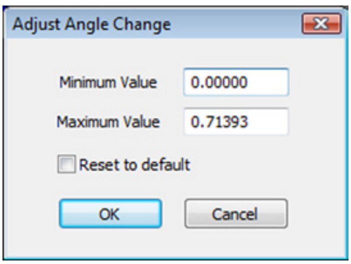
	Start value	End value
	Min	-360.00
	-360.00	-300.00
	-300.00	-240.00
	-240.00	-180.00
	-180.00	-120.00
	-120.00	-60.00
	-60.00	-0.00
	-0.00	60.00
	60.00	120.00
	120.00	180.00
	180.00	240.00
	240.00	300.00
	300.00	360.00
	360.00	Max

When you choose this element from the list, the tool path is represented in a gradient color scale according to tilting angles of the machine rotation axis. This scale enables you to identify the rotation axis angle range used in the operation, the rotation angle used for machining of specific areas, and limit overruns that occur during the simulation.

The angle range values are displayed in the corresponding columns of the table.



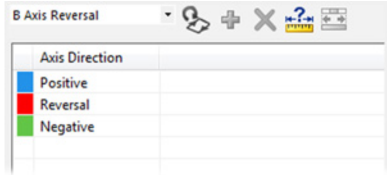
You can define a specific angle range to view the tool path in the corresponding colors. Click the **Adjust**  button in the toolbar to the right from the options list. The **Adjust Angle Scale** dialog box is displayed. This dialog box enables you to enter the minimal and maximal values for the angle range and return to default values, if necessary.

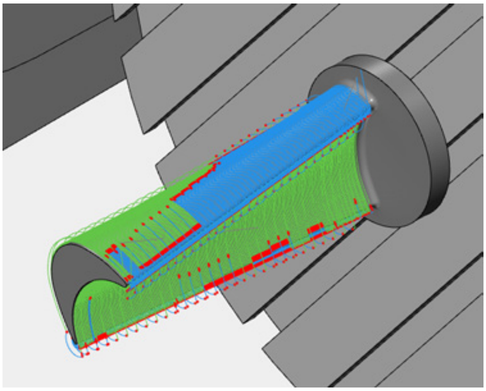


Click the **Refresh**  button in order for the change to take effect.

- **B Axis Reversal/C Axis Reversal**

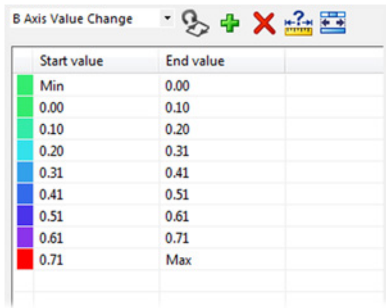
  When you choose this element from the list, the tool path is represented in colors according to change of direction of the machine rotation axes. These colors enable you to identify the areas where possible contouring errors have negative influence on the machining result (surface quality). Every time when a rotation axis changes its direction, the tool path segment changes its color.

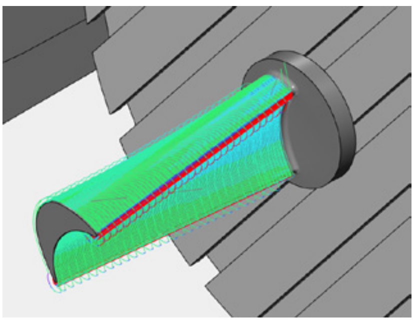




- **B Axis Value Change/C Axis Value Change**

  When you choose this element from the list, the tool path is represented in a gradient color scale according to change of tilting angles of the machine rotation axis. This scale enables you to identify the rotation speed range used in the operation, the rotation speed used for machining of specific areas and determine the areas where machine speed limits are reached.



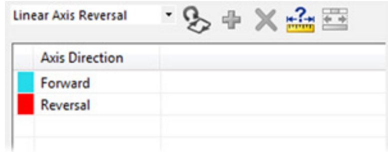


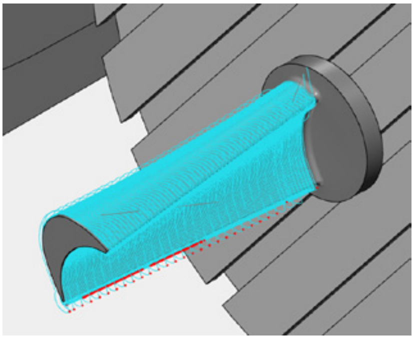
You can define a specific angle range to view the tool path in the corresponding colors by clicking the **Adjust** button and entering the values into the **Adjust angle scale** dialog box.

Click the **Refresh** button in order for the change to take effect.

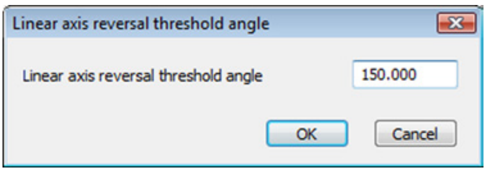
- **Linear axis reversal**

When you choose this element from the list, the tool path is represented in colors according to change of direction of the machine linear axes. Every time when a linear axis changes its direction, the tool path segment changes its color.





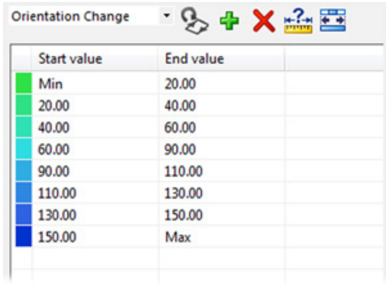
You can define a threshold angle value for the axis reversal to view the tool path in the corresponding colors by clicking on the **Adjust**  button and entering the values into the **Linear axis reversal threshold angle** dialog box.



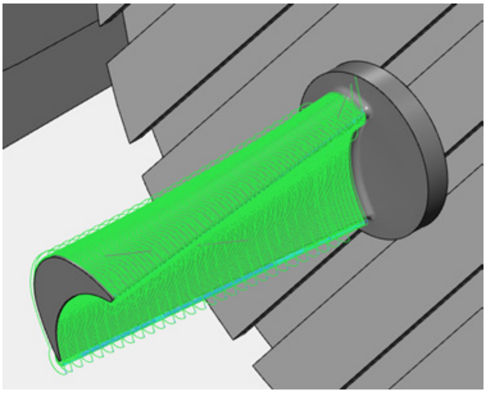
Click the **Refresh**  button in order for the change to take effect.

- **Orientation change**

When you choose this element from the list, the tool path is represented in a gradient color scale according to change of orientation of the machine rotation axes. This scale enables you to identify the rotation speed range used in the operation, the rotation speed used for machining of specific areas and determine the areas where machine speed limits are reached.

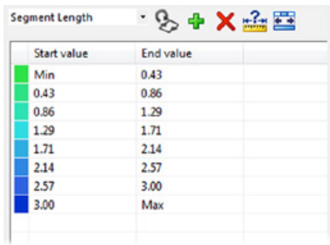


Start value	End value
Min	20.00
20.00	40.00
40.00	60.00
60.00	90.00
90.00	110.00
110.00	130.00
130.00	150.00
150.00	Max

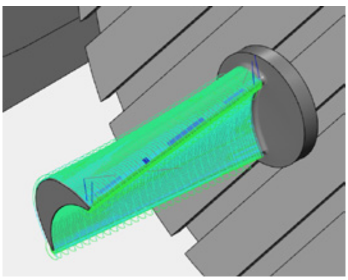


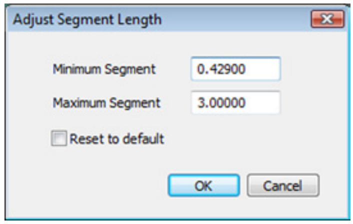
- **Segment Length**

When you choose this element from the list, the tool path is represented in colors according to the length of its segments. These colors enable you to identify the areas where you have long linear motions, usually in roughing tool path or where the segments become very short, e.g. for finishing.



Start value	End value
Min	0.43
0.43	0.86
0.86	1.29
1.29	1.71
1.71	2.14
2.14	2.57
2.57	3.00
3.00	Max



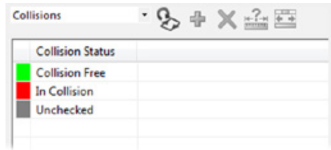


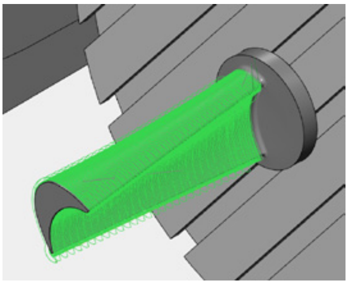
You can define a specific segment length range to view the tool path in the corresponding colors. Click the **Adjust**  button in the toolbar to the right from the options list. The **Adjust Segment Length** dialog box is displayed. This dialog box enables you to enter the minimal and maximal values for the length range and return to default values, if necessary.

Click the **Refresh**  button in order for the change to take effect.

- **Collisions**

When you choose this element from the list, the tool path is represented in colors according to the collision status.

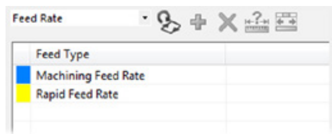


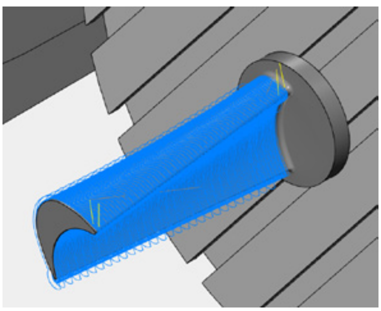


- Segments with collisions are marked red.
- Collision-free segments are marked green.
- Segments that were not checked yet are marked gray.

- **Feed rate**

When you choose this element from the list, the tool path is represented in colors according to the feed rate.

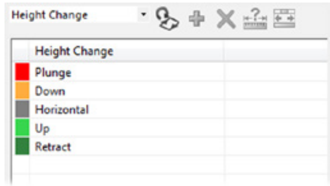


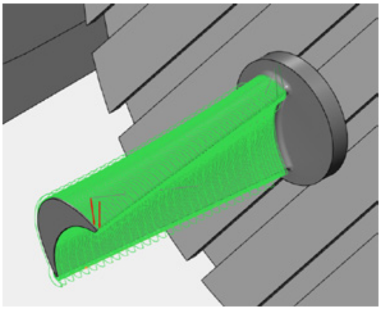


- Segments with machining feed rate are marked blue.
- Segments with rapid feed rate are marked yellow.

- **Height Change**

When you choose this element from the list, the tool path is represented in colors according to the tool orientation relative to the tool path.

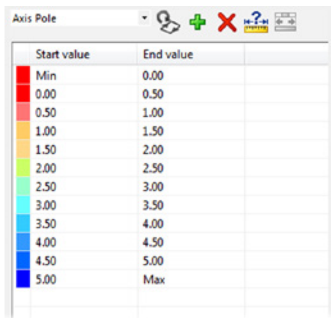


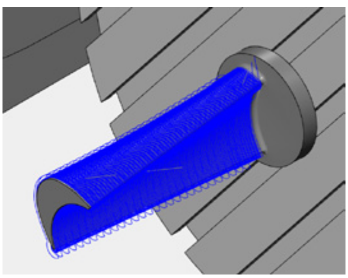


- Segments where the plunging is performed in the tool axis direction are marked red.
- Segments with the lag angle tool orientation are marked orange.
- Segments with the normal tool orientation are marked gray.
- Segments with the lead angle tool orientation are marked light green.
- Segments where the tool retracts along the tool axis are marked green.

- **Axis Pole**

When you choose this element from the list, the tool path is represented in a gradient color scale which enables you to determine whether the two rotational axes are collinear. The more the axes get collinear, the tool path turns more red. This option is useful for simulation of impeller parts machining.



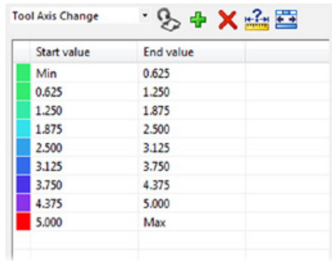


You can define a specific tool axis angle range to view the tool path in the corresponding colors. Click the **Adjust**  button in the toolbar to the right from the options list. The **Adjust Axis Pole** dialog box is displayed. This dialog box enables you to enter the minimal and maximal values for the angle range and return to default values, if necessary.

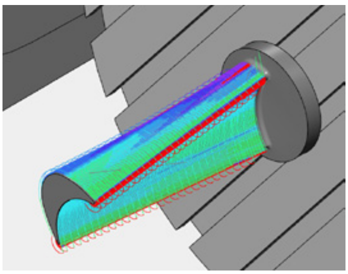
Click the **Refresh**  button in order for the change to take effect.

- **Tool Axis Change**

When you choose this element from the list, the tool path is represented in a gradient color scale according to the change of tilting angle of rotational axes. This scale enables you to identify the rotation speed range used in the operation, the rotation speed used for machining of specific areas and determine the areas where machine speed limits are reached.



Start value	End value
Min	0.625
0.625	1.250
1.250	1.875
1.875	2.500
2.500	3.125
3.125	3.750
3.750	4.375
4.375	5.000
5.000	Max

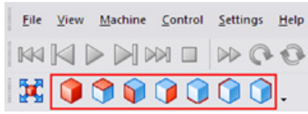


19.1.3 Simulation toolbars

The simulation control provides you with a number of toolbars enabling you to control the simulation process and the model visualization in the graphic area. The arrow near each toolbar enables you to customize this toolbar by displaying or hiding certain buttons.

View

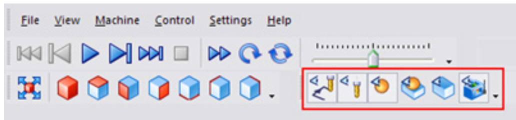
The buttons of this toolbar enable you to rotate the simulation model to the appropriate view (**Isometric**, **Right**, **Front**, etc.).



These buttons are also available as commands in the **Standard Views** submenu of the **View** menu (see topic **19.1.1**).

Visibility

The buttons of this toolbar enable you to control the display of various machine and model components in the graphic area.



These buttons are also available as commands in the **Show/Hide** submenu of the **View** menu (see topic **19.1.1**).

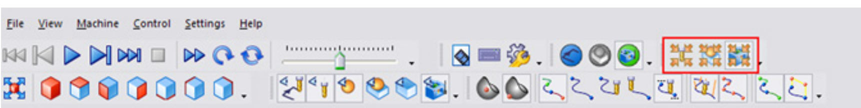
Simulation modes

This toolbar contains the modes of simulation model display.

These buttons are also available as commands in the **Simulation mode** submenu of the **View** menu (see topic **19.1.1**).

Focus

The buttons of this toolbar enable you to directly zoom and observe elements such as tool, workpiece and machine in stationary position.



These buttons are also available as commands in the **Focus** submenu of the **View** menu (see topic **19.1.1**).

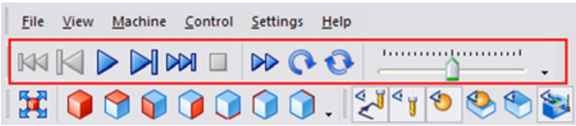
Tool path rendering

The buttons of this toolbar enable you to choose the mode of tool path display.

These buttons are also available as commands in the **Focus** submenu of the **View** menu (see topic **19.1.1**).

Control

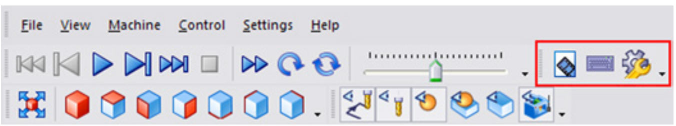
The buttons of this toolbar enable you to control the progress of the simulation.



These buttons are also available as commands in the **Control** menu (see topic **19.1.1**).

Settings

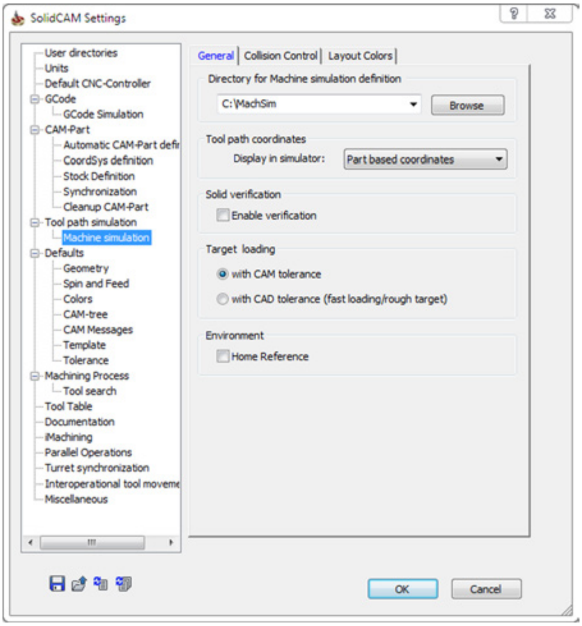
The buttons of this toolbar enable you to manage the simulation settings and to adjust the simulation interface in accordance with your requirements.



- The **Create Presentation** button enables you to create a self-extracting executable file containing the current CNC-machine and NC file (see topic **19.1.1**).
- The **Hot Keys** button displays the **HotKey Settings** dialog box that enables you to define hot keys and mouse settings for the simulation control (see topic **19.1.1**).
- The **Simulation Properties** button displays the **Simulation Properties** dialog box that enables you to edit the simulation settings (see topic **19.1.1**).

19.2 Machine simulation settings

The **Machine simulation** page of the **SolidCAM Settings** dialog box enables you to define a number of settings of the **Machine simulation**. The page includes three tabs: **General**, **Collision Control**, and **Layout Colors**.



19.2.1 Directory for Machine simulation definition

This parameter defines the location of the CNC-machines definition files used for the **Machine simulation**. A number of CNC-machine subfolders are located under this folder. According to the settings of the MAC file, the appropriate machine is chosen for the **Machine simulation**.

19.2.2 Tool path coordinates

This section enables you to define the type of coordinates that will be displayed in the **Move list** tab (see topic **19.1.2**). The **Part based coordinates** option enables you to display the coordinates related to the CAM-Part coordinate system (part coordinates). The **Absolute machine axis values** option enables you to display coordinates related to the CNC-Machine origin (machine coordinates).

19.2.3 Solid verification

SolidCAM enables you to perform the solid verification of the material cutting process in the **SolidVerify** mode integrated into the **Machine simulation**. Using this functionality, you can display the stock model and perform the simulation by dynamic subtraction of the tool solid model (using solid Boolean operations) from the stock solid model.

When the **Enable verification** check box is selected, the solid verification is performed in the **Machine simulation** mode.

19.2.4 Target loading

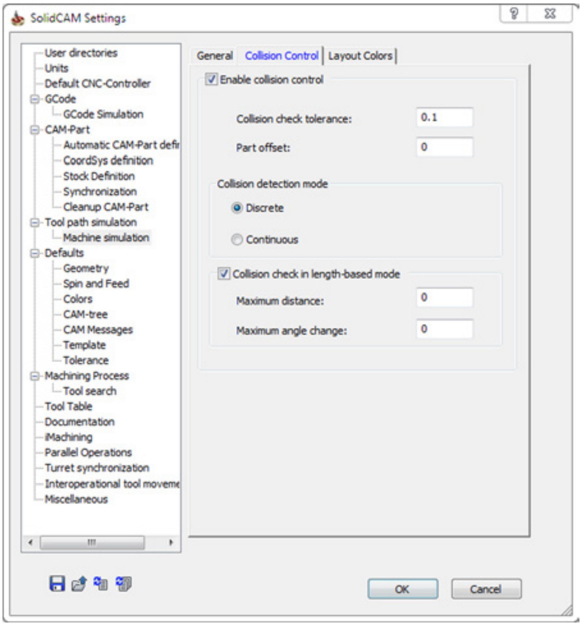
This option enables you to set the mode of target model loading during the **Machine simulation**.

- **with CAM tolerance.** When you choose to load the target with CAM tolerance, the target model with the tolerance value defined for the CAM model will be loaded.
- **with CAD tolerance (fast loading/rough target).** When you choose to load the target with CAD tolerance (fast loading/rough target), the target model with the tolerance value defined for the CAD model will be loaded.

19.2.5 Environment

The **Home reference** check box enables you to start the **Machine simulation** with all the machine devices returned to their **Home reference points** defined in the Machine ID file.

The **Collision control** tab enables you to define parameters for collision control.



19.2.6 Enable collision control

This option enables you to detect and avoid possible collisions between all components of the CNC-machine in the process of machining. When the **Enable collision control** check box is selected, SolidCAM performs the collision checking according to the collision control parameters in the CNC-machine definition. When the **Enable collision control** check box is not selected, SolidCAM ignores the collision control parameters in the CNC-machine definition and does not perform the collision control.

Collision check tolerance

This parameter enables you to specify the tolerance of the collision check. SolidCAM ignores all the collisions with tolerance smaller than the specified value and alerts when the collision tolerance is greater than the tolerance specified value.

Part offset

This parameter enables you to specify a value to offset the entire part for collision check.

19.2.7 Collision detection mode

Discrete. This option enables you to perform the collision control along the tool path connecting two positions with a straight line.

Continuous. This option enables you to perform the collision control along the tool path connecting two positions with an arc.

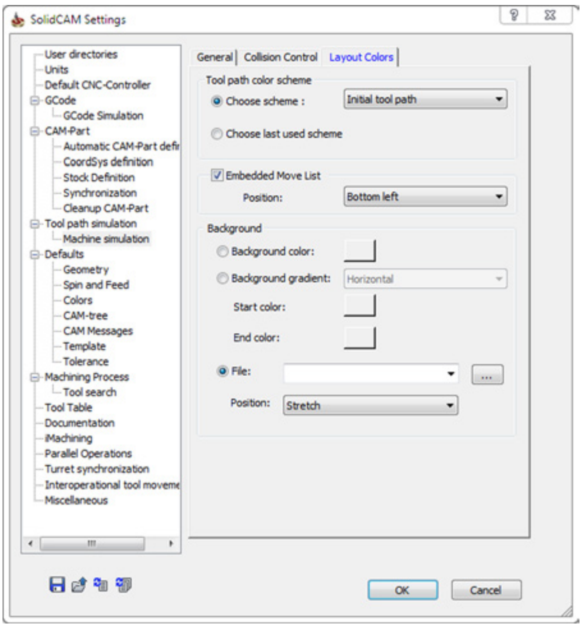
19.2.8 Collision check in length-based mode

This section enables you to perform the collision check when the **length-based mode** is chosen.

Maximum distance. This parameter enables you to specify the step between two consecutive checking positions.

Maximum angle change. This parameter enables you to specify the maximal angle change allowed for the tool per move.

The **Layout Colors** tab enables you to define the tool path color scheme for **Machine simulation**.



19.2.9 Tool path color scheme

This option enables you to choose the appearance of the tool path during the simulation.

Choose scheme

This option enables you to choose one of the color schemes defined in the **Analysis** section of the **Simulation window**.

Choose last used scheme

This option stores the last color scheme used in the **Machine simulation** and retrieves it when a new simulation is run.

19.2.10 Embedded move list

When this option is selected, it enables you to display the **Move List** directly in the **Simulation window** and to choose the list position in the window.

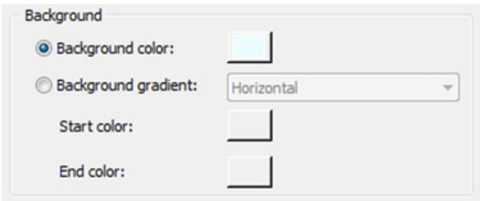
The following positions are available:

- **Bottom left**
- **Bottom right**
- **Bottom center**
- **Top left**
- **Top right**

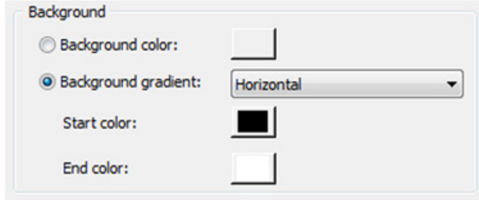
19.2.11 Background

Background color

This option enables you to choose the color for the Machine simulation background. The **Color** dialog box enables you to choose the appropriate color.

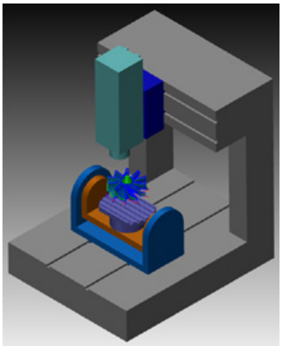


Background gradient

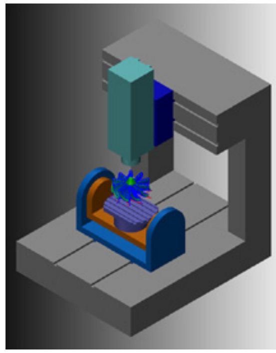


This option enables you to choose the colors for the **Machine simulation** gradient background. The **Color** dialog box enables you to choose the appropriate colors.

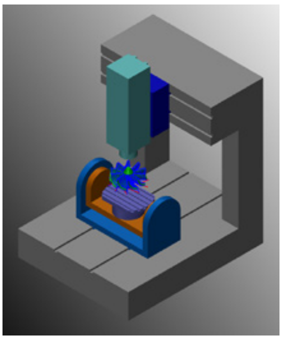
The following options are gradient backgrounds:



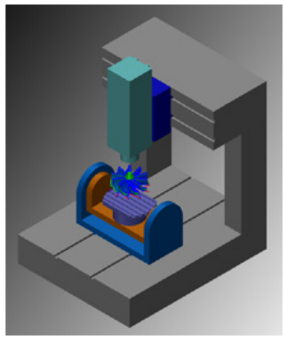
Horizontal



Vertical



Diagonal descending



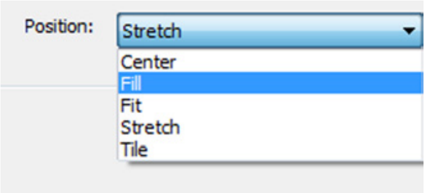
Diagonal ascending

19.2.12 File

This option allows you to use any image as the background of the **Machine simulation**.

19.2.13 Position

This option enables you to choose the position of the selected image.



The following positions are available:

- **Center**
- **Fill**
- **Fit**
- **Stretch**
- **Tile**

CNC-machine definition

20

20.1 CNC-machine definition

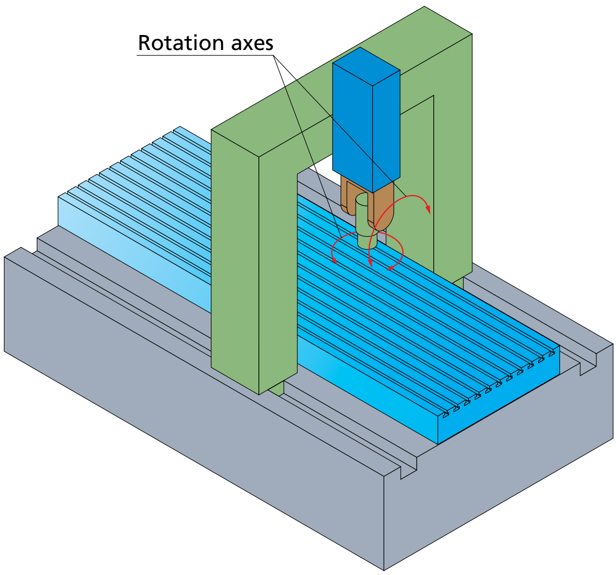
SolidCAM enables you to define a number of the CNC-machine parameters in the VMID file. These parameters enable you to take into account custom properties of the CNC-machine. These parameters are used on the different stages of the tool path calculations.

20.1.1 CNC-machine kinematic type

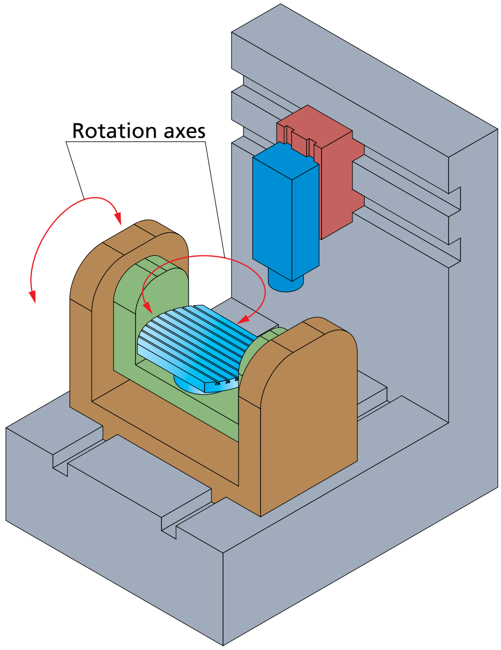
Choosing a set of certain axes for various machine devices enables you to define the type of the kinematics of the CNC-machine.

The following types are supported by SolidCAM according to the location of the rotation axes on the devices of CNC-machine:

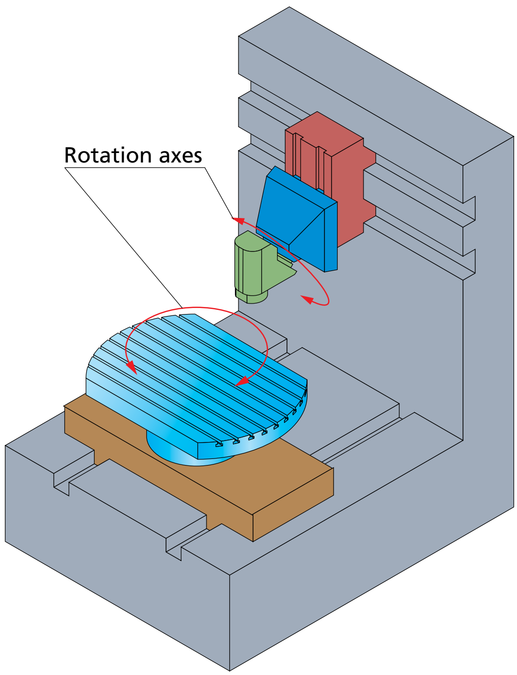
- **Head-Head**. In this type of CNC-machines, both rotation axes are mounted on the turret (spindle) of the CNC-machine.



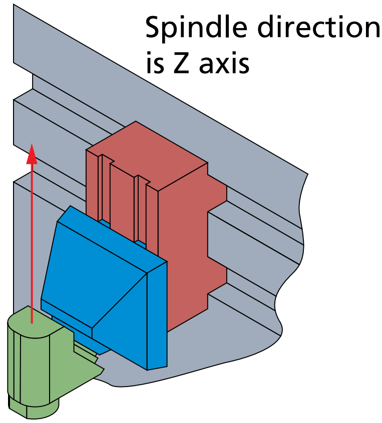
- **Table-Table**. In this type of CNC-machines, both rotation axes are mounted on the CNC-machine table.



- **Head-Table**. In this type of CNC-machines, one rotation axis is mounted on the turret (spindle) and the other is located on the table of the CNC-machine.

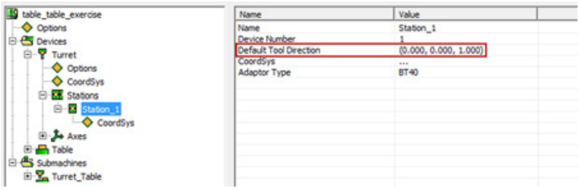


20.1.2 Spindle direction

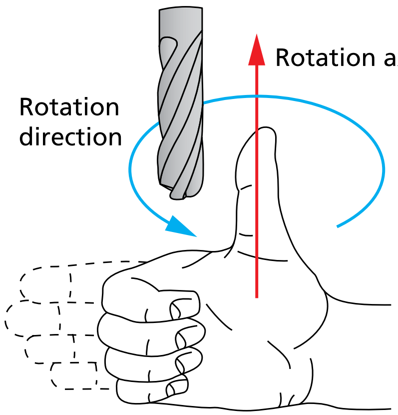


The **Default Tool Direction** parameter enables you to define the direction of the tool mounted on a spindle axis of the CNC-machine. The direction is defined by a vector, e.g. when the spindle is parallel to the Z axis, the vector is (0,0,1).

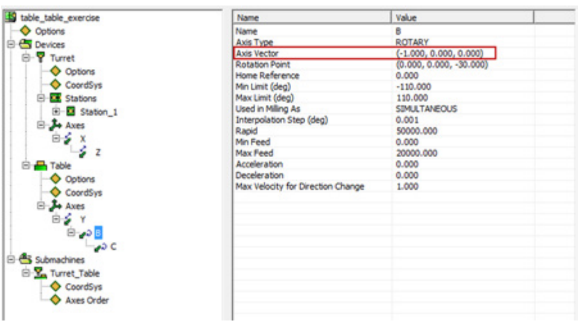
This parameter is located in the station definition.



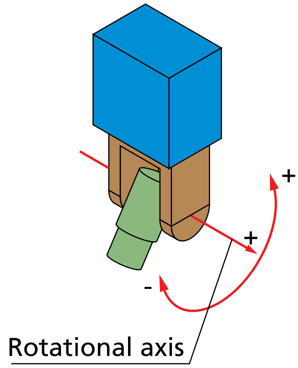
20.1.3 Rotation axes direction



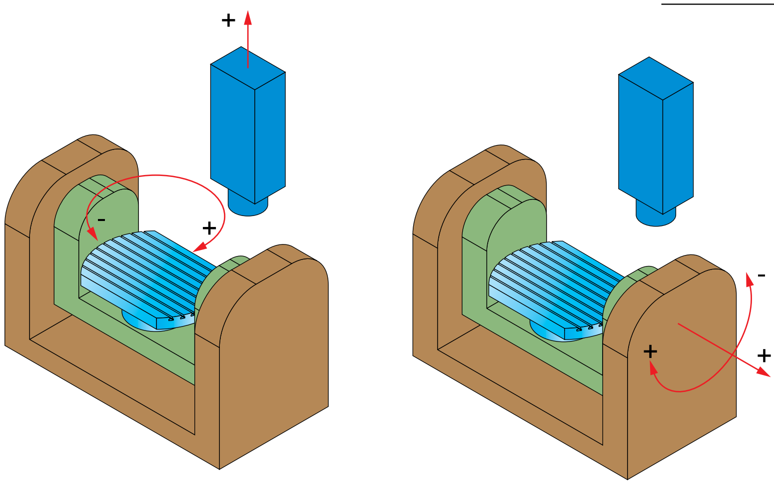
The direction of the rotation axes is defined according to the tool movements around the axis. The right hand rule is used for the direction definition (the fingers of the right hand are curled in the positive tool rotation direction, and the thumb indicates the positive direction of the rotation axis). Correspondingly, when the rotation axis is pointing away from the observer, the positive tool rotation direction is clockwise; when the rotation axis is pointing towards the observer, the positive tool rotation direction is counterclockwise.



If the rotation around the axis is performed by the spindle, the direction of the rotational axis is defined as shown.



In some cases, the rotation around the axis is not performed by the spindle. For example, in CNC-machines of the **Table-Table** type, the rotation is performed by the rotary table and tilting is performed by the tilting table. In this case, the direction of the rotational axes is defined according to the virtual spindle rotation around the axis as shown below. The negative rotation of the tilting table causes the positive tool tilting relative to the rotation axis.



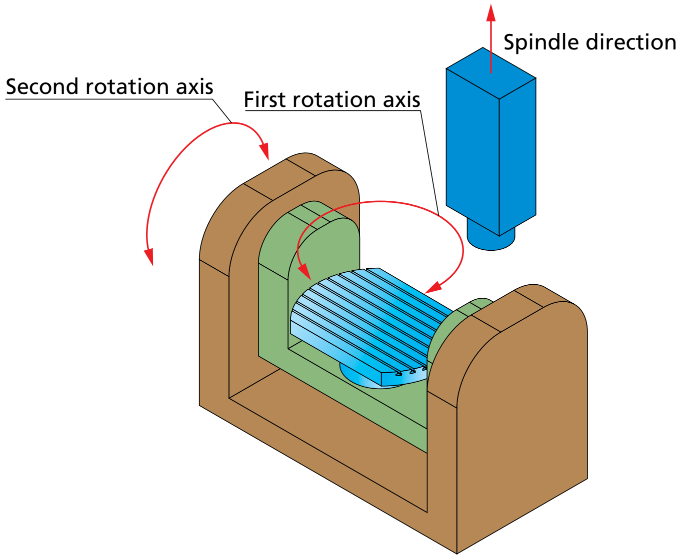
Generally, the first rotation axis is the axis of rotation around the spindle direction (see topic **20.1.2**). E.g. when the spindle direction of the **Table-Table** machine is parallel to the Z-axis, the first rotation axis has to be axis of the rotation around the Z-axis.

The **Axis Vector** parameter defines the positive direction of the rotation axis. The direction is determined with the right hand rule according to the positive direction of the rotation of the CNC-machine part performing the rotation. In this case, the positive direction of the rotary table rotation is clockwise. Therefore, using the right hand rule, the axis direction is the negative Z-direction (0, 0, -1).

The first and the second rotational axis depend on the CNC-machine kinematic type:

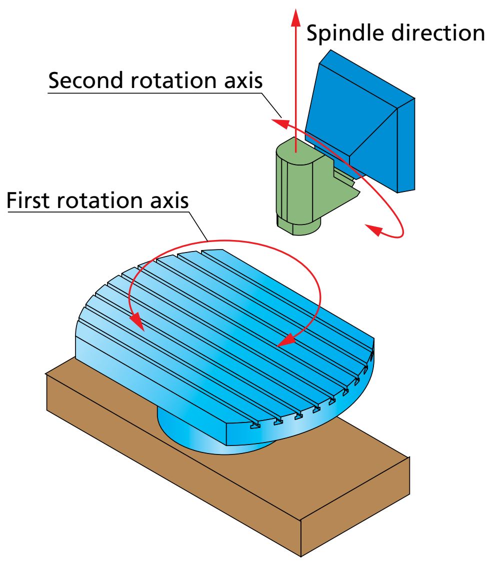
- **Table-Table**

  In this case, the first rotation axis performing the rotation around the spindle axis is mounted on the second rotation axis. The second rotation axis is mounted on the CNC-machine table.



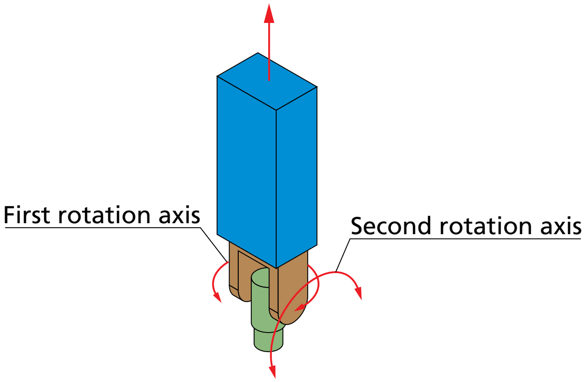
- **Head-Table**

  In this case, the first rotation axis performing the rotation around the spindle axis, is mounted on the table. The second rotation axis is mounted on the head of the CNC-machine.



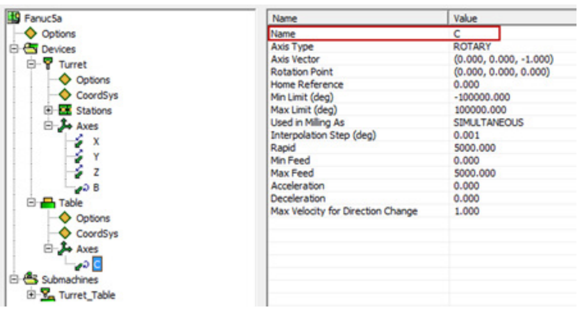
- **Head-Head**

  In this case, the first rotation axis performing the rotation around the spindle axis is mounted on the CNC-machine head. The second rotation axis is mounted on the first rotation axis.



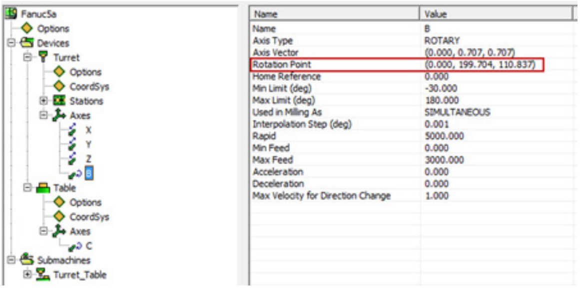
20.1.4 Rotation axes names

The axes names are defined in the VMID file.



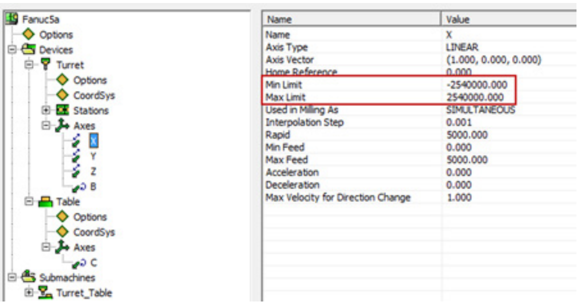
20.1.5 Rotation point

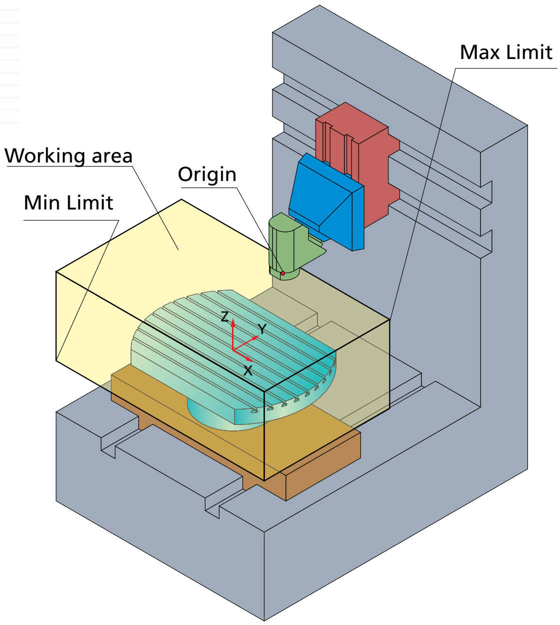
The **Rotation Point** parameter defines the location of the rotation axis relative to the CNC-machine origin.



20.1.6 Translation axis limits

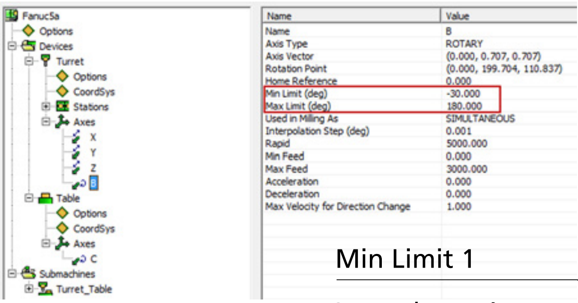
Each linear axis has a set of limits defined relative to the origin point. Using these limits you can define virtual 3D box of the working area of the CNC-machine.

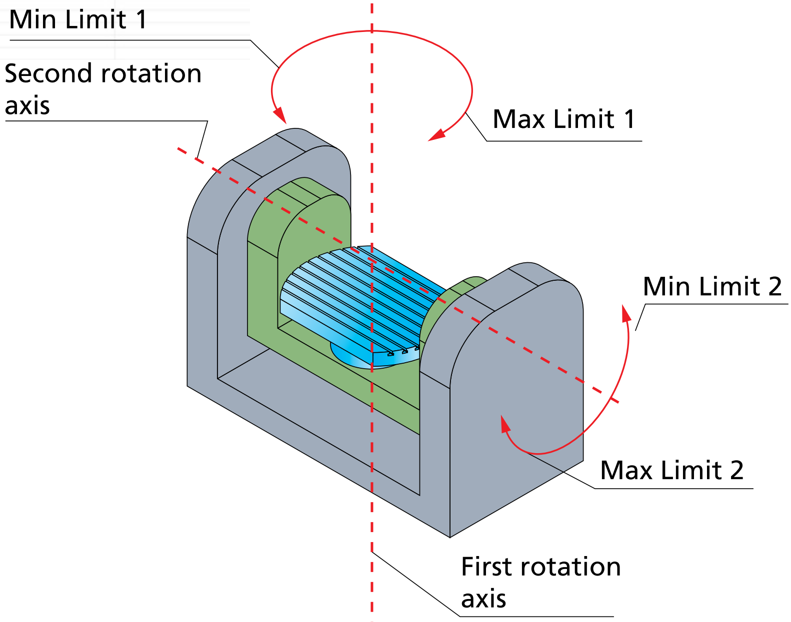




20.1.7 Rotation axis limits

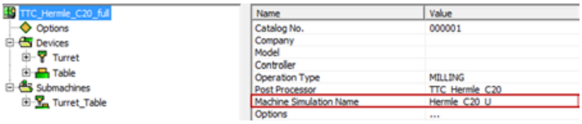
Similar to linear axes, each rotation axis also has a set of limits for rotation. The **Min Limit** and **Max Limit** values define the limit rotation angle in degrees.





20.1.8 Machine simulation name

This parameter defines the name of the CNC-machine model used for the Machine simulation.



The location of the appropriate model of the CNC-machine is defined in the **Directory for Machine simulation definition** defined by the SolidCAM **Machine simulation settings**.

Consider that the **Directory for Machine simulation definition** is:

C:\Users\Public\Documents\SolidCAM\SolidCAM2013\Tables\MachSim

In this case, the data of the CNC-machine **Hermle_C20_U** mentioned in the example above is located in the following folder:

C:\Users\Public\Documents\SolidCAM\SolidCAM2013\Tables\MachSim\xml

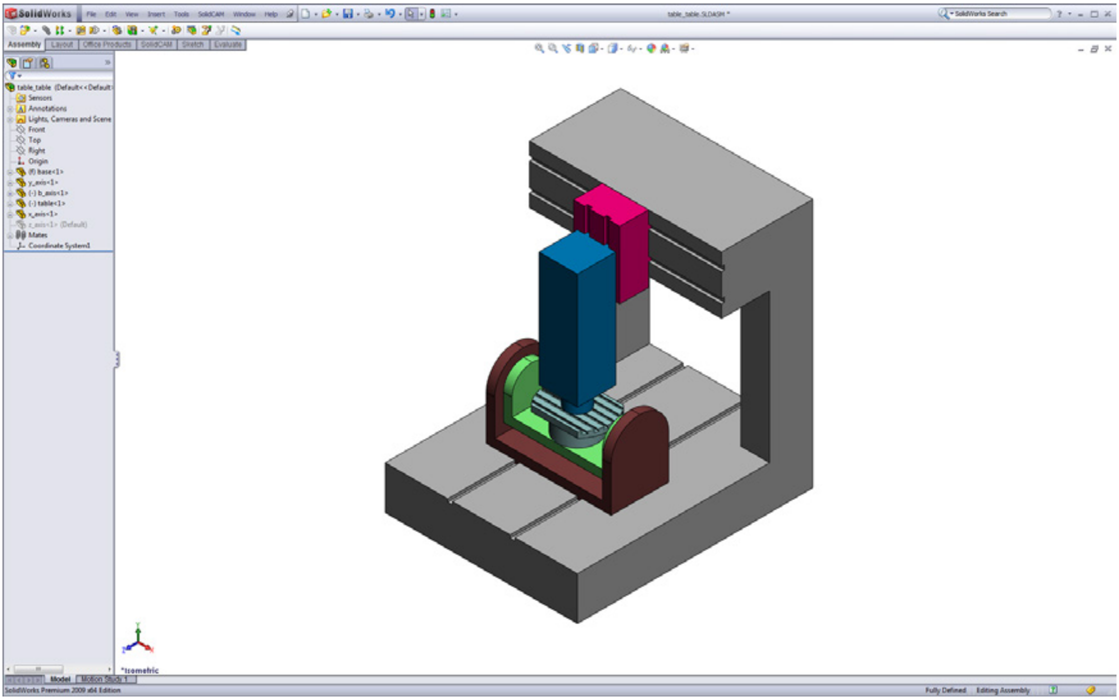
20.2 CNC-machine model definition

The **Machine simulation** (see chapter **19**) is performed on the model of a CNC-machine. This topic describes and explains all stages of the CNC-machine model definition.

20.2.1 Preparing a CNC-machine model

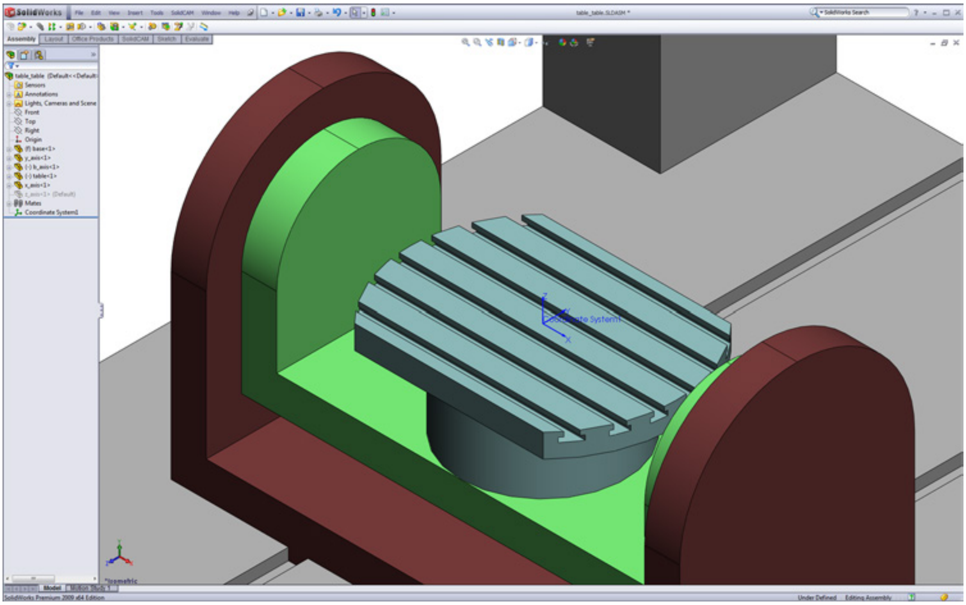
Machine simulation requires a model of the CNC-machine in the STL format. This model is usually supplied by a CNC-machine vendor. When the CAD model of the CNC-machine is prepared, it can be exported into a number of STL files, each one representing a different component of the CNC-machine.

The image below shows a schematic model of a **Table-Table** CNC-machine built in SolidWorks.

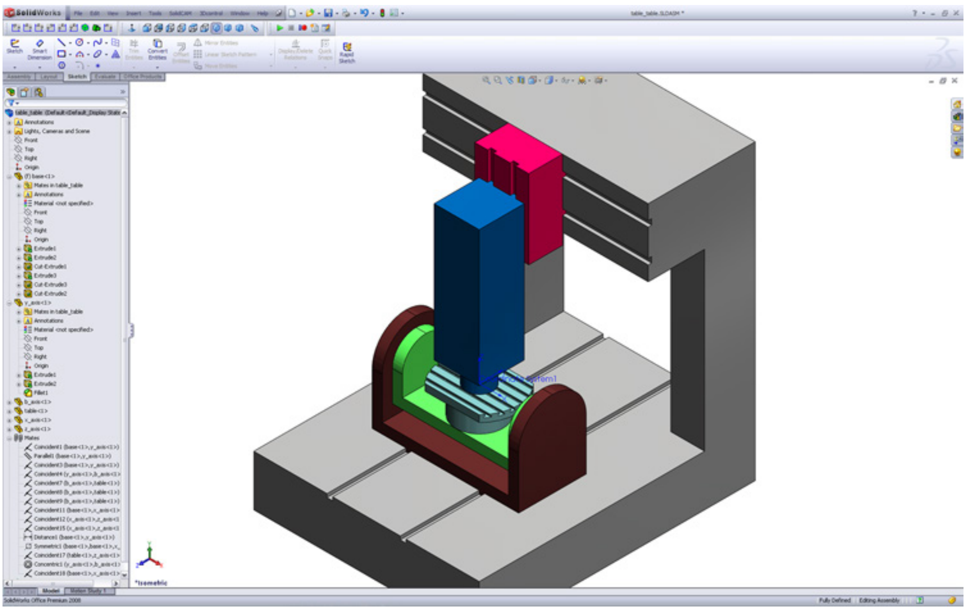


Each STL file was created using an output coordinate system located at the CNC-machine origin. Since the coordinate system of your CNC-machine assembly is different, an additional coordinate system was defined with the proper location and axes orientation.

In the case of the **Table-Table** machine shown above, a new coordinate system located in the intersection of the top face of the table and the rotational axis (CNC-machine origin) was defined.



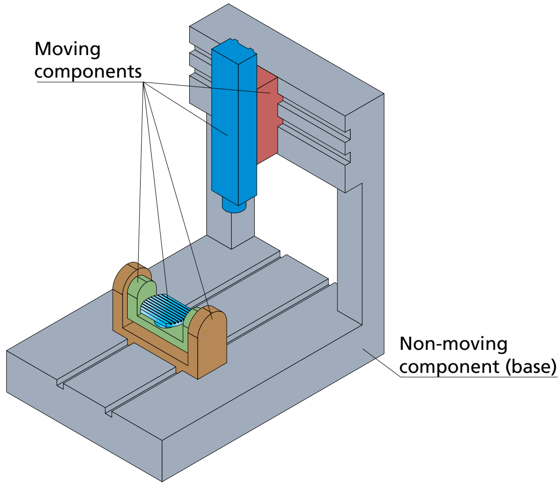
Then all components were moved into their initial state (the components performing rotational axes movements have to be placed into their initial state at C=0, B=0; the components performing translational axes movements have to be placed at X=0, Y=0, Z=0).



After the coordinate system is defined and all the CNC-machine components are placed into their initial state, the CNC-machine model was exported into the STL format.

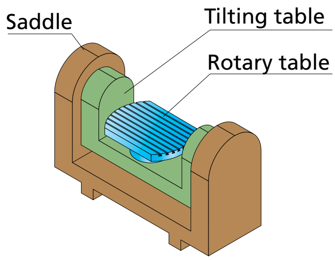
A CNC-machine model consists of a number of components. It is recommended to try to define the CNC-machine with the minimum number of STL files. To reduce the number of STL files, several components can be put together in one sub-assembly that is exported into a single STL file; the criterion for putting several components into one STL file is the common movement of these components. When assembly components always move together, they can be combined into a sub-assembly. For example, the model of the spindle unit of a **Table-Table** CNC-machine consists of a number of components that have a common movement; according to the criterion above, all these spindle unit components can be combined into a sub-assembly.

20.2.2 Understanding the structure of the CNC-machine



Before studying your CNC-machine components, you have to analyze the machine kinematics. Generally, all the components of the CNC-machine can be classified into two groups: non-moving components and moving components. The first group includes the CNC-machine base, controller, doors etc. The moving parts are the components of the transitional and rotational axes and the spindle unit.

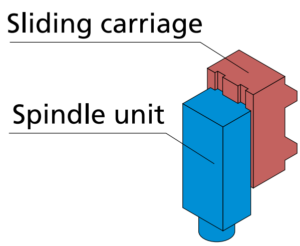
In the case of the **Table-Table** machine mentioned in topic **20.2.1**, the base is a non-moving component.



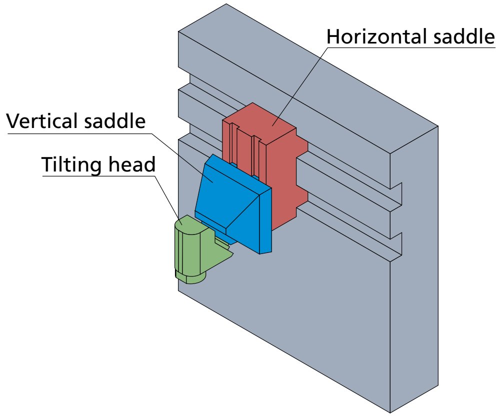
All the moving components can be classified according to the dependency of the movements. For example, in the **Table-Table** CNC-machine the rotary table that provides the rotation around a vertical axis (C-axis) is mounted on the tilting table. The tilting table that provides the rotation around the X-axis (B-axis) is mounted on the saddle. The saddle performs the X-axis movements. Movement of the saddle affects the location of the B-axis and C-axis (the tilting and rotary tables are moved). Movement of the tilting table (rotation around B-axis) affects the orientation of the C-axis (the rotary table is moved together with the tilting table), but does not affect the X-axis. The rotary table movement (rotation around C-axis) does not affect the B-axis and X-axis orientation.

Hierarchically, we can describe the structure of the rotary table, tilting table and saddle using "parents-children" relations. The saddle is the "parent' of the tilting table because the tilting table ("child") is mounted on the saddle. Similarly, the tilting table is a "parent" of the rotary table because the rotary table is mounted on the tilting table.

Another separate set of the CNC-machine components provides the movements in the YZ-plane. The sliding carriage providing movements along the Y-axis, and the spindle unit performs the Z-axis movements. In this hierarchy the sliding carriage is a "parent" and the spindle unit is a "child".

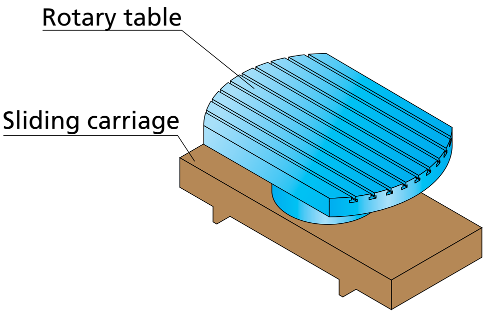


The structure of a **Head-Table** CNC-machine can be described by two separate sets of components. The components of these sets are combined together according to the "parents-children" criterion.

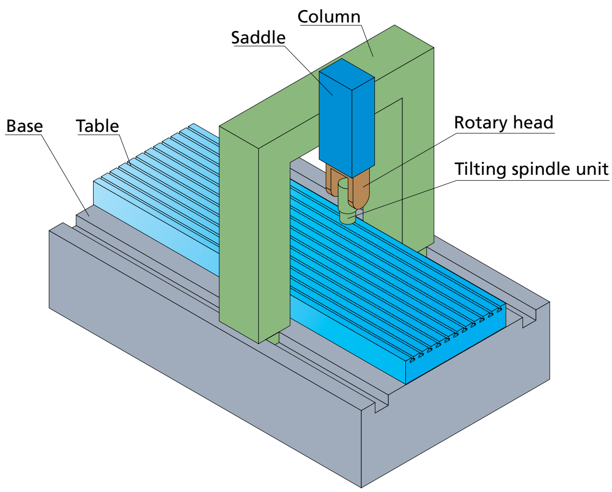


The first set consist of the tilting head with the spindle unit (B-axis) mounted on the vertical saddle (Z-axis). The vertical saddle is mounted on the horizontal saddle (X-axis).

The second set consists from the rotary table (C-axis) mounted on the sliding carriage (Y-axis).



The table of a **Head-Head** CNC-machine is a non-moving component mounted on the base. The moving parts are described by the following "parents-children" hierarchy. The tilting spindle unit (B-axis) is mounted on the rotary head (C-axis). The rotary head is mounted on the saddle performing movements in the YZ plane. The saddle is mounted on the column (X-axis).

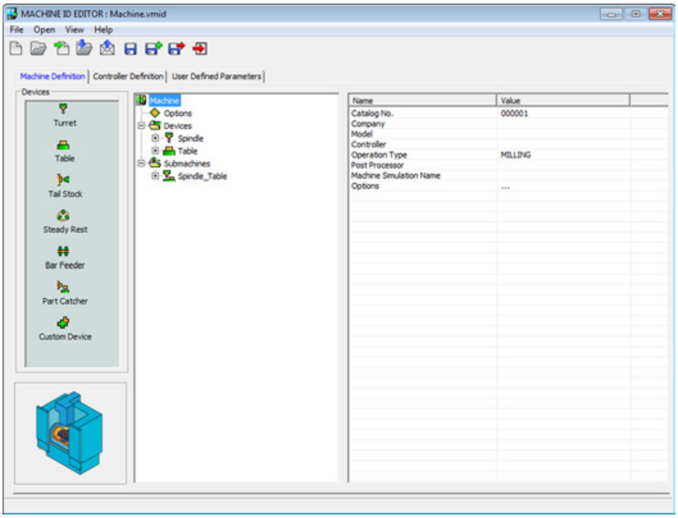


The CNC-machine has to be defined according to the "parents-children" relations between the CNC-machine components; these relations determine the order of the components definition and dependencies between them.

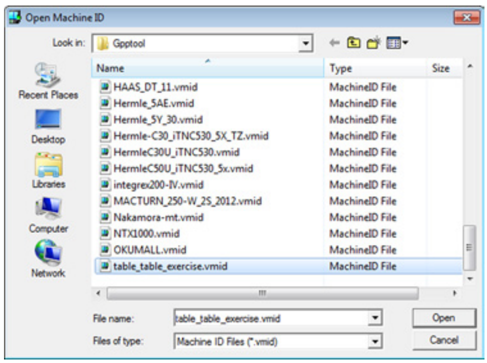
20.2.3 Reviewing the CNC-machine properties

The **Exercises** folder supplied with this book contains three files that fully define the CNC-machine: **table_table_exercise.vmid**, **table_table_exercise.prp**, and **FANUC.gpp**. For correct work of Machine Simulation, these files should be copied into **GPPtool** folder on your hard drive (the default location is **C:\Users\Public\Documents\SolidCAM\SolidCAM2013\Gpptool**). Also, you have to copy the Machine Simulation folder **Table_table_exercise** into the corresponding folder on your hard drive (the default location is **C:\Program Files\SolidCAM2013\Tables\Metric\MachSim\xml**).

In the **Solidcam** folder (**C:\Program Files\SolidCAM2013\Solidcam**) locate the **MachineIdEditor.exe** file and double-click it. The **Machine ID Editor** window is displayed.

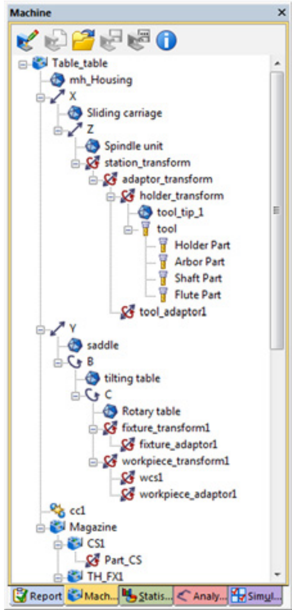


In the menu, choose **File**, **Open.** In the **Look in** section, browse the **GPPTool** folder and select the **table_table_exercise.vmid** file. Click **Open**. The Machine ID file is loaded.



In the menu, choose **Open**, **Machine Simulation**. The **MachSimIntegration** window containing the machine simulation is displayed.

The **Machine** window located in the right part of the screen lists all machine components in the hierarchical order. These components are divided into the following categories:



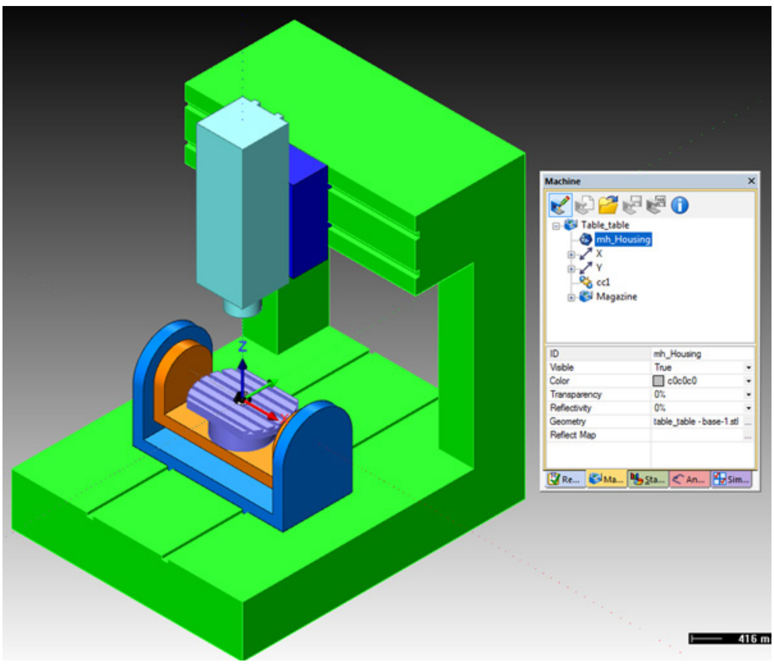
- **Axes**
- **Coordinate System Transforms**
- **Geometries**
- **Heads**
- **Dynamic elements**

Click the **Edit** icon to turn on the editing mode.

20.2.4 Defining the CNC-machine housing

At the first stage of the CNC-machine definition, non-moving components of the machine such as housing are defined.

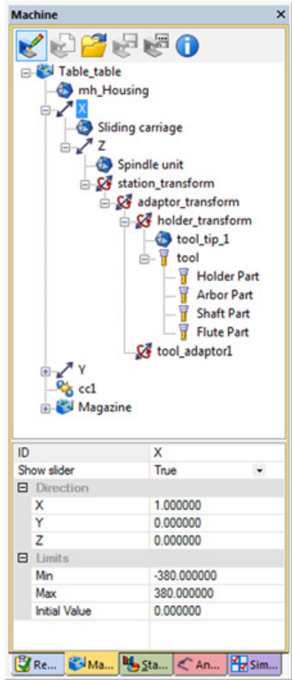
Click the **mh_Housing** item in the Machine tree. The machine housing is highlighted and its properties are displayed in the lower part of the window.



When the **mh_** prefix appears in the **ID** of a CNC-machine component, this component is considered as a housing part. The components defined with this prefix can be hidden during the simulation using the **Show/Hide machine housing** button. In this case, the **ID** value is set to **mh_Housing**.

The element properties table also enables control over the visibility and the visual properties (color, transparency, reflectivity) of the CNC-machine components.

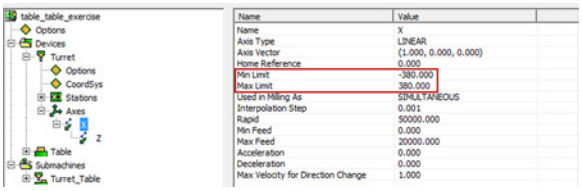
20.2.5 Defining the translational axis



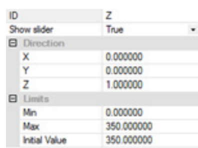
After the non-moving component of the CNC-machine, the moving components could be defined. Moving components have to be defined according to their "parents-children" dependencies. The order of the definition is the following: "parents" have to be defined before the "children". In case of **Table-Table** machines, the first component to be defined is the sliding carriage that performs the X-axis movements and then the spindle unit that moves along the Z-axis. These two components are joined into a separate set; the movements of this set are independent from the movements of Y-, B- and C-axes.

Click the **X** item in the CNC-machine definition tree. In the element properties table, the orientation of the axis is defined by a vector with three coordinates. The following limits are set for the axis: the **Min Limit** value is **-380** and the **Max Limit** value is **380**.

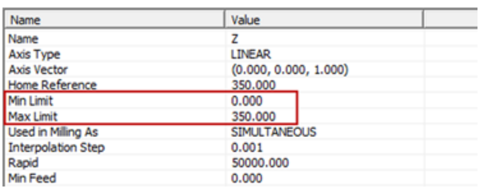
Take a note that the same values must be set in the corresponding VMID file.



Click the **Z** item. Notice that the Z-axis is a "child" of the X-axis. The following limits are set for the Z-axis: the **Min Limit** value is **0** and the **Max Limit** value is **300**.

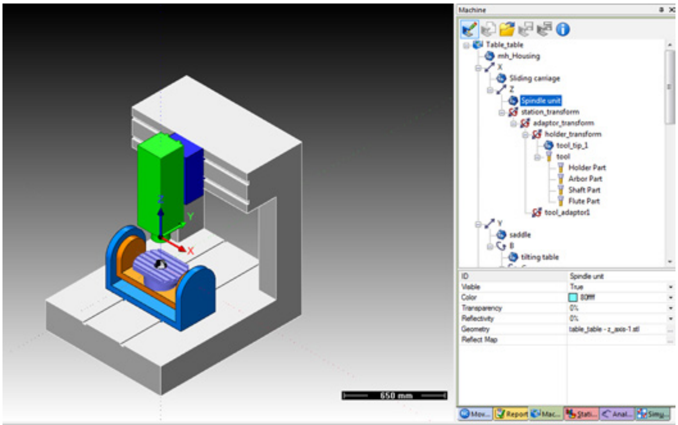


Take a note that the same values must be set in the corresponding VMID file.



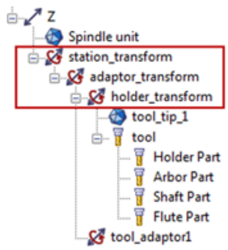
Use the sliders of the **Axis Control** tab to check the translational movements of the geometry along the axis.

Clicking the **Spindle unit** item highlights the spindle and displays the component properties.



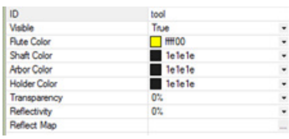
20.2.6 Defining the tool

At this stage, you can review the default tool and its kinematic relationship with the CNC-machine components.



The tool with holder is mounted on the CNC-machine spindle unit (Z-axis). The tool item is preceded by a series of coordinate system transformations that define the kinematic relationship between the tool and the CNC-machine components. In the default state, the **station_transform**, **adaptor_transform**, and **holder_transform** items have the same values, but they can change when Machine Simulation of a certain part is loaded.

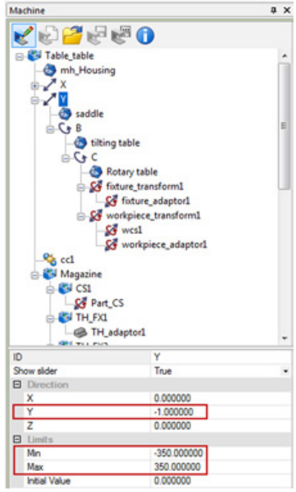
The **tool_tip_1** item defines the default model of the tool.



ID	tool
Visible	True
Flute Color	ffff00
Shaft Color	1e1e1e
Arbor Color	1e1e1e
Holder Color	1e1e1e
Transparency	0%
Reflectivity	0%
Reflect Map	

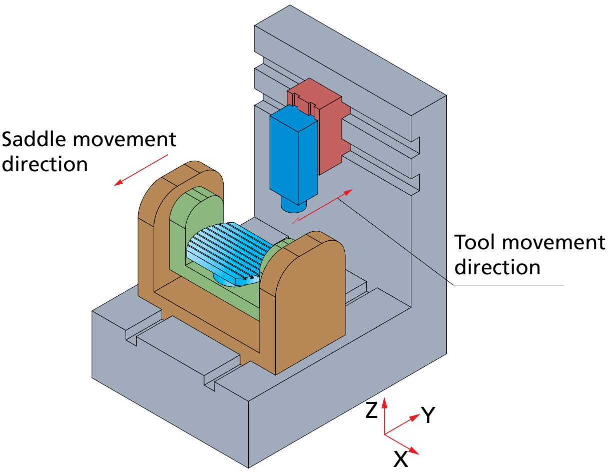
The tool geometry with the built-in name **Tool** is related to the **tool.stl** file. In the process of the CNC-machine definition this file does not exist. It will be automatically created during the simulation. Every time during the simulation this file will be overwritten with the actual tool data.

20.2.7 Defining the translational axis



Click the **Y** item in the CNC-machine definition tree. In the element properties table, the orientation of the axis is defined by a vector with three coordinates.

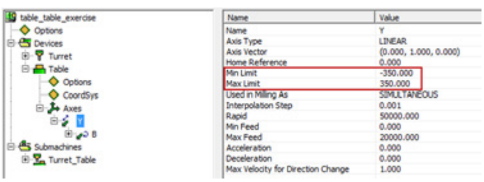
In this case, the defined translational axis is the Y-axis. Notice that the direction is defined according to the tool movements along the axis. The tool movement in the positive Y-axis direction causes the saddle movement in the negative Y-direction.



Therefore, the direction is defined with the following values: **0.00000 -1.00000 0.00000**. This vector defines the direction of the CNC component movement when the tool moves in the positive axis direction.

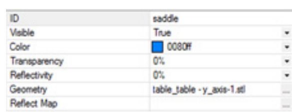
The saddle performs the Y-axis movements within the range of the minimal and maximal limit values. The **Min Limit** value is set to **-350** and the **Max Limit** value is set to **350**.

Take a note that the same values must be set in the VMID file.



Name	Value
Name	Y
Axis Type	LINEAR
Axis Vector	(0.000, 1.000, 0.000)
Home Reference	0.000
Min Limit	-350.000
Max Limit	350.000
Used in Milling As	SIMULTANEOUS
Interpolation Step	0.001
Rapid	50000.000
Min Feed	0.000
Max Feed	20000.000
Acceleration	0.000
Deceleration	0.000
Max Velocity for Direction Change	1.000

After the translational axis parameters, the geometry of the part performing the translational movement along the axis is defined. In this case, this part is the saddle.



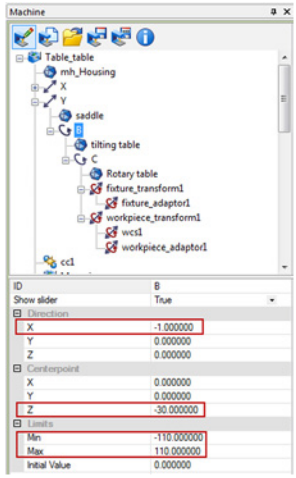
Use the sliders of the **Axis Control** tab to check the translational movements of the defined geometry along the axis.

20.2.8 Defining the rotational axis

Defining the tilting table

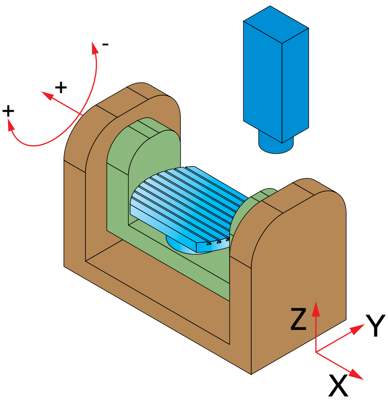
At this stage, you can review the definition of the tilting table providing the B-axis rotation.

Click the **B** item in the CNC-machine definition tree.



In the element properties table, the **Direction** and the **Center point** parameters determine the location and orientation of the rotational axis.

The tilting axis performs the rotation around the X-axis. Notice that the positive tool movement around the rotational axis (positive direction of the B-axis) is performed by the negative rotation of the tilting table.

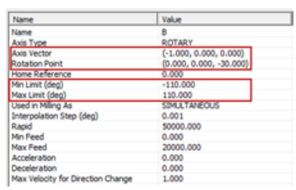


The positive direction of the rotational axis of the tilting table is defined by the right hand rule. Therefore, the axis **Direction** parameters values must be the following: **-1.00000 0.00000 0.00000**.

The **Center point** parameters define the location of the axis relative to the CNC-machine origin. In this case, the tilting axis is located 30 mm below the table. Therefore, the **Center point** values must be defined as follows: **0.00000 0.00000 -30.00000**.

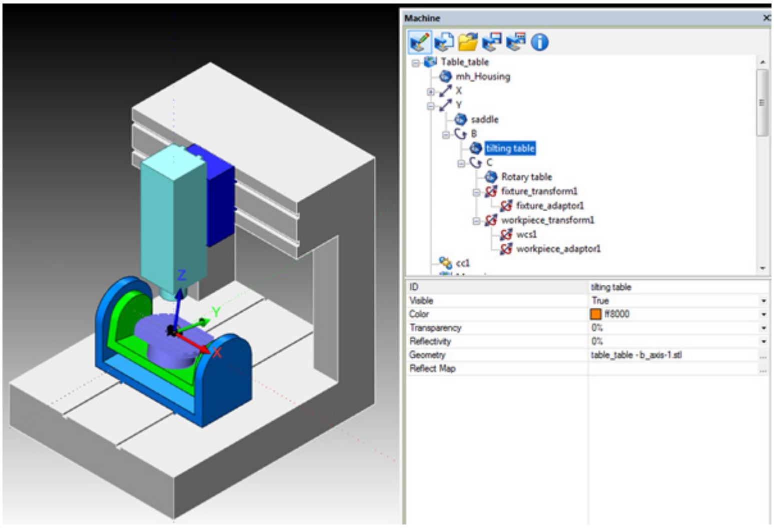
The minimum and maximum rotation angle limits are set in the **Limits** section: the **Min Limit** value is **-100°** and the **Max Limit** value is **100°**.

Take a note that the same values must be set in the VMID file.



Name	Value
Name	B
Axis Type	ROTARY
Axis Vector	(-1.000, 0.000, 0.000)
Rotation Point	(0.000, 0.000, -30.000)
Home Reference	0.000
Min Limit (deg)	-110.000
Max Limit (deg)	110.000
Used in Milling As	SIMULTANEOUS
Interpolation Step (deg)	0.001
Rapid	50000.000
Min Feed	0.000
Max Feed	20000.000
Acceleration	0.000
Deceleration	0.000
Max Velocity for Direction Change	1.000

Now consider the geometry of the tilting table. In the CNC-machine definition tree, click the **tilting_table** item. The tilting table is highlighted in the graphic area, and its parameters are displayed.



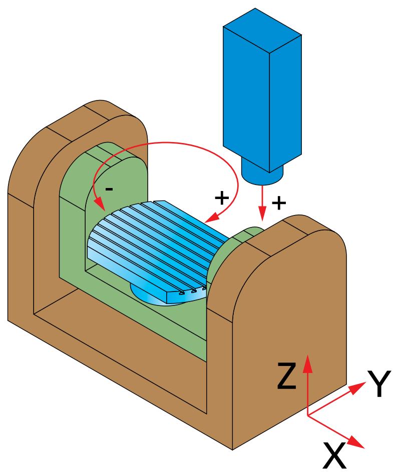
Use the sliders of the **Axis Control** tab to check the translational movements of the defined geometry along the axis.

Defining the rotary table

At this stage, you can review the definition of the rotary table that provides the C-axis rotation.

In the CNC-machine definition tree, click the **C** item.

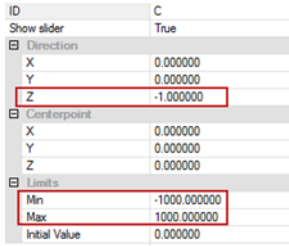
The rotary axis performs the rotation around the Z-axis. Note that the positive tool rotation around the rotational axis (positive direction of the C-axis) is performed by the negative rotation of the rotary table; the positive direction of the rotational axis of the rotary table is defined by the right hand rule. Therefore, the axis **Direction** parameters values must be the following: **0.00000 0.00000 -1.00000**.



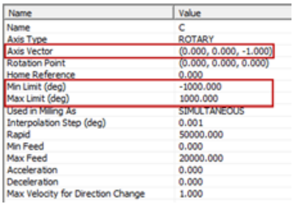
The **Center point** parameters define the location of the axis relative to the CNC-machine origin. The rotation B-axis passes through the CNC-machine origin. Therefore, the **Center point** has to be defined with the following values: **0.00000 0.00000 0.00000**.

The minimum and maximum rotation angle limits are set in the **Limits** section: the **Min Limit** value is **-1000°** and the **Max Limit** value is **1000°**.

Take a note that the same values must be set in the VMID file.



ID	C
Show slider	True
Direction	
X	0.000000
Y	0.000000
Z	-1.000000
Centerpoint	
X	0.000000
Y	0.000000
Z	0.000000
Limits	
Min	-1000.000000
Max	1000.000000
Initial Value	0.000000



Name	Value
Name	C
Axis Type	ROTARY
Axis Vector	(0.000, 0.000, -1.000)
Rotation Point	(0.000, 0.000, 0.000)
Home Reference	0.000
Min Limit (deg)	-1000.000
Max Limit (deg)	1000.000
Used in Milling As	SIMULTANEOUS
Interpolation Step (deg)	0.001
Rapid	50000.000
Min Feed	0.000
Max Feed	20000.000
Acceleration	0.000
Deceleration	0.000
Max Velocity for Direction Change	1.000

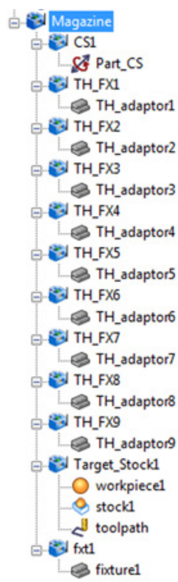
The geometry of the rotary table is defined in the **Rotary table** item.

Use the sliders of the **Axis Control** tab to check the translational movements of the defined geometry along the axis.

The **Rotary table** item is followed by a series of transforms: **fixture_transform1**, **fixture_adaptor1**, **workpiece_transform1**, **wcs1**, and **workpiece_adaptor1**. These items are reserved for coordinate system transformations performed as you load an actual part and add more data on fixture, workpiece and so on.

20.2.9 Defining the magazine

At this stage, you can review the **Magazine** part of the CNC-machine definition tree containing features loaded together with a CAM-Part, such as STL holders for Spindle, Fixtures, Target, and Stock.



The **CS1** item defines the current coordinate system. If the coordinate system axes are added to the item in an STL file, the MAC is shown during the Machine Simulation.

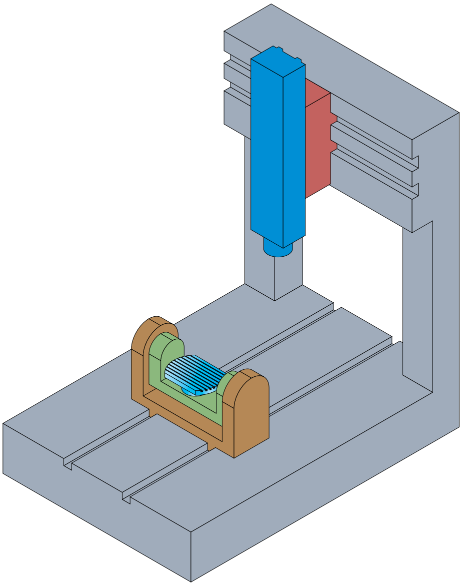
The **TH_FXx** items defines the STL holders for Spindle turret.

The **Target_Stock1** item contains the properties of the loaded CAM-Part stock and target.

The **fxt1** item contains the fixtures defined in the Setup.

20.2.10 Collision control

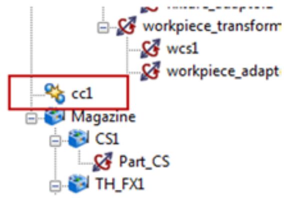
This functionality enables you to detect and avoid possible collisions between all components of the CNC-machine in the process of machining.



The understanding of the CNC-machine construction and kinematics is necessary for the collision definition. In case of **Table-Table** CNC-machines definition, the machine construction precludes the collisions between the saddle part and the tilting and rotary tables. The collisions between the sliding carriage and saddle are also precluded by the CNC-machine construction and kinematics. There is no necessity to check the CNC-machine components during their movements for such collisions.

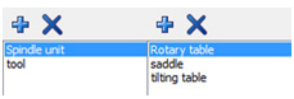
The collisions between spindle unit (with tool holder and tool) and saddle (with mounted tilting table and rotary table) are possible. Such collisions have to be detected and avoided.

The **cc1** item displayed in the machine definition tree and a table divided into two sections in the bottom part of the **Machine** tab contain the collision control definition.

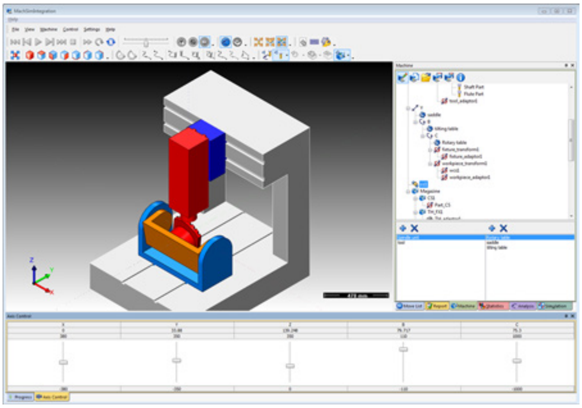


This table lists two groups of CNC-machine components for which the collision checking will be performed.

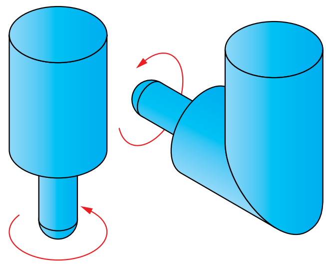
The tool and spindle unit components are included in the first group. The saddle, tilting table, and rotary table components are included into the second group. The collision checking will be performed between the components of the two groups.



Use the sliders of the **Axis Control** tab to check the collision checking definition. When a collision is detected, the contacting components are highlighted.

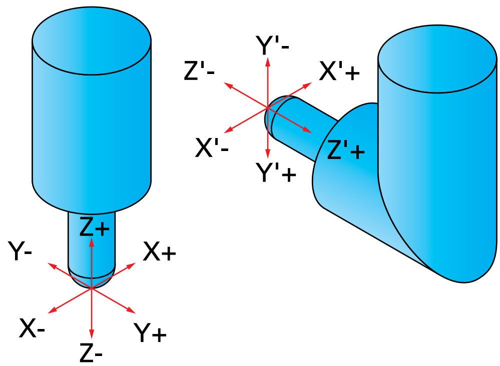


20.2.11 Defining the coordinate transformation



In some cases, the CNC-machine spindle is not aligned in the positive Z-axis direction. For example, an angular attachment enables you to transform the vertical spindle rotation (around the Z-axis) into the horizontal rotation (around the Y-axis). In this case, the Z-axis of the spindle unit is not parallel to the default main spindle direction.

In the illustration below, the default **XYZ** coordinates describe the main spindle direction (coordinate system of the CNC-machine). The **X'Y'Z'** coordinate system describes the angular attachment. These coordinate systems have different axes orientations.

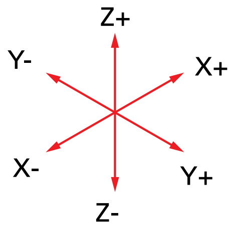


The coordinate transformation between the default coordinate system and the spindle coordinate system has to be performed in order to enable correct execution of the tool path by the tool mounted in the spindle unit.

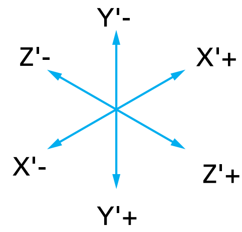
To perform the necessary coordinate transformation, a new coordinate transformation item was added into the CNC-machine definition tree. Hierarchically, this item has to be "parent" to the **holder_transform** item.

The element properties table contains a transformation matrix. Using this matrix, the transformation of the coordinate system can be defined. In other words, the orientation and location of a transformed coordinate system are defined relative (**X'Y'Z'**) to the coordinate system of the CNC-machine (**XYZ**).

The columns of this matrix describe the axis orientation of the CNC-machine coordinate system relative to the transformed coordinate system. The **X** column defines the direction of the X-axis; the **Y** column defines that of the Y-axis, and the **Z** column defines that of the Z-axis. The **Shift** column defines the offset of the transformed coordinate system origin relative to the CNC-machine coordinate system origin.



CNC machine coordinate system



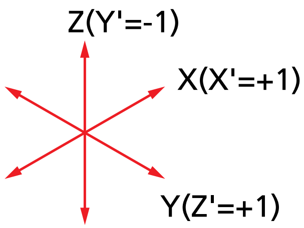
Transformed coordinate system

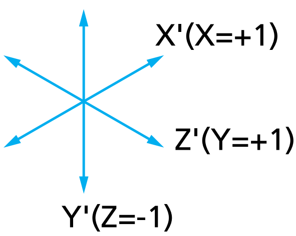
The rows of the matrix define the orientation of the transformed coordinate system relative to the CNC-machine coordinate system. The **X** row defines the direction of the X-axis, the **Y** row defines that of the Y-axis, and the **Z** row defines that of the Z-axis.

By default, the diagonal of the transformation matrix is filled with **1** values. This means that the initial coordinate system of the CNC-machine and the transformed coordinate system are the same and coordinate transformation is not performed. Such transformation matrixes were used for the *station_transform* and *adaptor_transform* definition.

In case a transformation is required, the transformation matrix has to be filled in.

The transformed coordinate system for the angular attachment discussed before is obtained by the 90° rotation of the CNC-machine coordinate system around the X-axis.





CNC machine coordinate system

Transformed coordinate system

20.2.12 XML file structure

This topic describes the XML file structure using the **Table-Table** CNC-machine example built through the topics **20.2.1** – **20.2.11**.

XML tags

The XML-based definition of the CNC-machine consists of a number of commands (tags). Each describes the specific item of the CNC-machine definition tree.

Each tag is enclosed by the "<" and ">" signs.

Example: *<machine_definition>*

Some XML constructions consist of open and close tags; the close tag starts from the "/" symbol.

Example: *<machine_definition*

CNC-machine definition tags

</machine_definition>

For some tags the opening and closing can be performed in the single tag.

Example: *<view_transform initialvalue -="1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000" />*

The XML tag can contain a number of variables.

Example: *name="Table_table"*; here the *name* is the variable name and *"Table_table"* is the value. The value must be enclosed in quotation marks.

The automatically generated XML file is the following:

The order of the commands (tags) of the XML file is the same as the order of the CNC-machine components in the CNC-machine definition tree.

```
<?xml version=”1.0” encoding=”UTF-8” ?>
```

This string is similar for each XML file; it describes the used version of XML format and encoding.

```
<machine_definition>
```

This tag starts the CNC-machine definition.

```
<machine_data name=”Table_table” version=”1.7” units=”metric”>
```

This tag defines the name of the CNC-machine and the used units.

```
<view_transform initialvalue =”1.00000000,0.00000000,0.00000000,0.000
00000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.000000
00,1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000
” />
```

```
</machine_data>
```

This tag starts the transformation definition. The *initialvalue* variable defines the used transformation matrix.

```
<axis id=”X” type=”translation” x=”1.00000000” y=”0.00000000”
z=”0.00000000” minvalue=”-380.000000” maxvalue=”380.000000”
valuetype=”cont” initial_value=”0.000000”>
```

This tag starts the X-axis definition. The *<axis>* tag enables you to define the type of the axis (translational/rotational), axis orientation and limits.

```
<geometry name=”Sliding carriage” geo=”table_table - x_axis-1.stl”
clrr=”0.00000000” clrg=”0.00000000” clrb=”1.00000000” alpha=”0.0”
reflectivity=”0.0” reflectivityBitmapFileName=”” objtype=”geometry” />
```

This tag defines the geometry of the part performing X-axis movements. The *name* variable defines the geometry ID. The *geo* variable defines the STL file used for the geometry definition. The *clrr*, *clrg* and *clrb* variable define the RGB components of the geometry color. The value of these parameters has to be in the range from 0 to 1. The *alpha* variable defines the transparency of the geometry. The *reflectivity* variable defines the reflectivity of the geometry.

```
<axis id=”Z” type=”translation” x=”0.00000000” y=”0.00000000”
z=”1.00000000” minvalue=”0.000000” maxvalue=”350.000000”
valuetype=”cont” initial_value=”350.000000”>
```

```
<geometry name="Spindle unit" geo="table_table - z_axis-1.stl"
clrr="0.50196081" clrg="1.00000000" clrb="1.00000000" alpha="0.0"
reflectivity="0.0" reflectivityBitmapFileName="" objtype="geometry" />
```

The Z-axis definition.

```
<transform id="station_transform" initialvalue ="1.00000000,0.0000000
0,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0
.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00
000000,1.00000000">
```

This tag starts the tool transformation definition. The `initialvalue` variable defines the used transformation matrix.

```
<transform id="adaptor_transform" initialvalue ="1.00000000,0.0000000
0,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0
.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00
000000,1.00000000">
```

This tag defines the transformations of the tool-adaptor unit.

```
<transform id="holder_transform" initialvalue ="1.00000000,0.00000000
,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.
00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.000
00000,1.00000000">
```

This tag defines the holder transformation.

```
<geometry name="tool" geo="tool.stl" alpha="0.0" reflectivity="0.0"
reflectivityBitmapFileName="" objtype="tool" cuttr="1.00000000"
cuttg="1.00000000" cuttb="0.00000000" noncuttr="0.11764706"
noncuttg="0.11764706" noncuttb="0.11764706" arborr="0.11764706"
arborg="0.11764706" arborb="0.11764706" holderr="0.11764706"
holderg="0.11764706" holderb="0.11764706" />
```

This tag defines the geometry of the tool (with the tool holder). The `clrr`, `clrg` and `clrb` variables define the RGB components of the tool color. The `holderr`, `holderg` and `holderb` variables define the RGB components of the tool holder color.

```
<geometry name="tool_tip_1" geo="tool_tip.stl" clrr="1.00000000"
clrg="0.00000000" clrb="0.00000000" alpha="0.0" reflectivity="0.0"
reflectivityBitmapFileName="" objtype="geometry" />
```

This tag defines the geometry of the tool tip.

```
</transform>
```

End tag for the holder transformation definition.

```
<transform id="tool_adaptor1" initialvalue ="1.00000000,0.00000000,0.
00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.000
00000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.000000
00,1.00000000" />
```

This tag defines the tool adaptor transformation. Since the station may hold more than one tool, each adaoptor is assigned with a sequential number.

```
</transform>
```

End tag for transformation definition of the tool-adaptor unit.

```
</transform>
```

End tag for the tool transformation definition.

```
</axis>
```

End tag for the Z-axis definition.

```
</axis>
```

End tag for the X-axis definition.

```
<axis id="Y" type="translation" x="0.00000000" y="-1.00000000"
z="0.00000000" minvalue="-350.000000" maxvalue="350.000000"
valuetype="cont" initial_value="0.000000">
```

Start of Y-axis definition.

```
<axisid="B"type="rotation"x="-1.00000000"y="0.00000000"z="0.00000000"
minvalue="-110.000000" maxvalue="110.000000" valuetype="cont" initial_
value="0.000000" rzx="0.00000000" rzy="0.00000000" rzz="-30.00000000">
```

Start of B-axis definition. For rotational axes the `rzx`, `rzy` and `rzz` variables define the location of the rotation base point.

```
<axis id="C" type="rotation" x="0.00000000" y="0.00000000" z="-
1.00000000" minvalue="-1000.000000" maxvalue="1000.000000"
valuetype="cont" initial_value="0.000000" rzx="0.00000000"
rzy="0.00000000" rzz="0.00000000">
```

Start of C-axis definition.

```
<geometry name="Rotary table" geo="table_table - table-1.stl"
clrr="0.50196081" clrg="0.50196081" clrb="1.00000000" alpha="0.0"
reflectivity="0.0" reflectivityBitmapFileName="" objtype="geometry" />
```

Definition of the CNC-machine part performing C-axis movements.

```
<transform id=”fixture_transform1” initialvalue =”1.00000000,0.0000000
0,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0
.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00
000000,1.00000000”>
```

This tag starts the workpiece transformation definition. The `initialvalue` variable defines the used transformation matrix.

```
<transform id=”fixture_adaptor1” initialvalue =”1.00000000,0.00000000,
0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.0
0000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.0000
0000,1.00000000” />
```

This tag defines the fixture adaptor. Since the machine may have more than one table, each item is assigned with a sequential number.

```
</transform>
```

End tag for the fixture transformation definition.

```
<transform id=”workpiece_transform1” initialvalue =”1.00000000,0.0000
0000,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.0000000
0,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0
.00000000,1.00000000”>
```

This tag defines the stock and target transformation. Since the machine may have more than one table, each item is assigned with a sequential number.

```
<transform id=”wcs1” initialvalue =”1.00000000,0.00000000,0.00000000,
0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.0
0000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.0000
0000” />
```

This tag defines the workpiece transformation.

```
<transform id=”workpiece_adaptor1” initialvalue =”1.00000000,0.000000
00,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,
0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.0
0000000,1.00000000” />
```

This tag defines the workpiece adaptor transformation.

```
</transform>
```

End tag for the workpiece transformation definition.

```
</axis>
```

End tag for the C-axis definition.

```
<geometry name=”tilting table” geo=”table_table - b_axis-1.stl” clrr=”1.00000000” clrg=”0.50196081” clrb=”0.00000000” alpha=”0.0” reflectivity=”0.0” reflectivityBitmapFileName=”” objtype=”geometry” />
```

Definition of the CNC-machine part performing B-axis movements.

```
</axis>
```

End tag for the B-axis definition.

```
<geometry name=”saddle” geo=”table_table - y_axis-1.stl” clrr=”0.00000000” clrg=”0.50196081” clrb=”1.00000000” alpha=”0.0” reflectivity=”0.0” reflectivityBitmapFileName=”” objtype=”geometry” />
```

Definition of the CNC-machine part performing Y-axis movements.

```
</axis>
```

End tag for the Y-axis definition.

```
<geometry name=”mh_Housing” geo=”table_table - base-1.stl” clrr=”0.75294125” clrg=”0.75294125” clrb=”0.75294125” alpha=”0.0” reflectivity=”0.0” reflectivityBitmapFileName=”” objtype=”geometry” />
```

Housing model definition.

```
<collcheck id=”cc1” name=”cc1” group1=”Spindle unit,tool” dynamic1=”true” dynamic2=”true” active=”true” group2=”Rotary table,saddle,tilting table” />
```

Collision checking definition. The `group1` and `group2` variables enable you to define groups of the CNC-machine components.

```
<magazine>
```

Start of the magazine components definition.

```
<head name=”CS1”>

<transform id=”Part_CS” initialvalue =”1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000,0.00000000,0.00000000,0.00000000,0.00000000,1.00000000” />

</head>
```

These tags define the current coordinate system.

```
<head name=”TH_FX1”>

<geometry name=”TH_adaptor1” geo=”fixture.stl” clrr=”0.11764706” clrg=”0.11764706” clrb=”0.11764706” alpha=”0.0” reflectivity=”0.0”
```

```
reflectivityBitmapFileName=”” objtype=”fixture” />
```

```
</head>
```

These tags define the STL holders for the Spindle Turret. The items are numbered according to the number of tools used for a certain workpiece machining.

```
<head name=”Target_Stock1”>
```

Start of Target and Stock definition.

```
<geometry name=”workpiece1” geo=”workpiece1.stl” clrr=”0.65098041”
clrg=”0.79215688” clrb=”0.94117647” alpha=”0.0” reflectivity=”0.0” refle
ctivityBitmapFileName=”tablereflection.bmp” objtype=”workpiece” />
```

Workpiece geometry definition.

```
<geometry name=”stock1” geo=”stock1.stl” clrr=”0.50196081”
clrg=”0.32156864” clrb=”0.00000000” alpha=”0.0” reflectivity=”0.0”
reflectivityBitmapFileName=”” objtype=”stock” />
```

Stock geometry definition.

```
<geometry name=”toolpath” geo=”toolpath1.asc” clrr=”0.80000001”
clrg=”0.80000001” clrb=”0.80000001” alpha=”0.0” reflectivity=”0.0”
reflectivityBitmapFileName=”” objtype=”toolpath” />
```

Tool path definition.

```
</head>
```

End of Target and Stock definition.

```
<head name=”fxt1”>
```

Start tag for fixture defined in the setup.

```
<geometry name=”fixture1” geo=”fixture.stl” clrr=”0.11764706”
clrg=”0.11764706” clrb=”0.11764706” alpha=”0.0” reflectivity=”0.1” refle
ctivityBitmapFileName=”tablereflection.bmp” objtype=”fixture” />
```

```
</head>
```

End tag for fixture defined in the setup.

```
</magazine>
```

End tag for the magazine definition.

```
</machine_definition>
```

End tag for the CNC-machine definition.

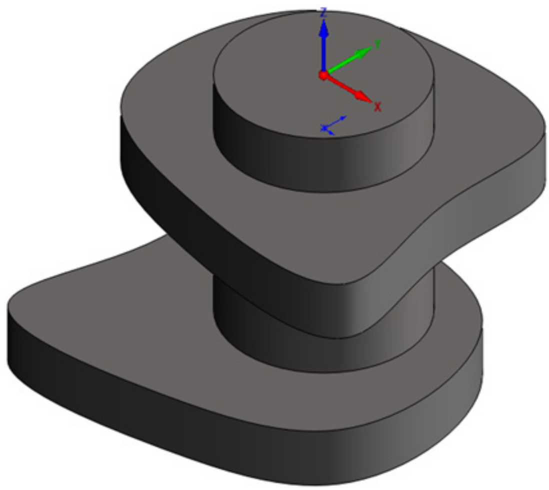
Exercises

21

Exercise #1: Eccentric shaft and cam machining

This exercise illustrates the use of the Sim. 5-Axis operation for eccentric shaft and cam machining.

A Sim. 5-Axis operation is used to perform the machining of the external shape of the cam.

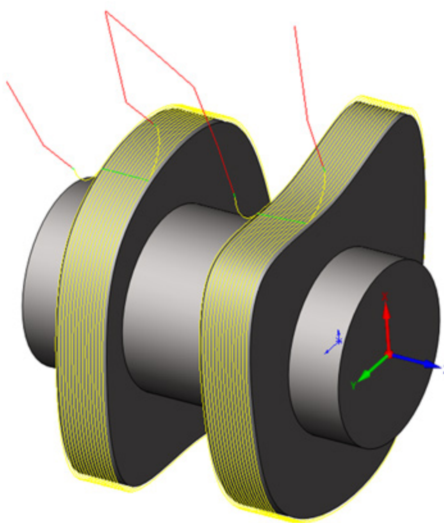


The **5X_PC_selected_faces** operation provides the cam machining using a bull-nosed tool of **Ø10** with the corner radius of 2 mm. The **Parallel cuts** strategy is used to generate a number of equidistant cuts parallel to the XY plane of the coordinate system.

For **Gaps along cut**, the **Large gaps** are linked using the **Follow surfaces** method. Also, the **Use Lead-On/Out** option is chosen for linking **Large moves** at the **Clearance area**.

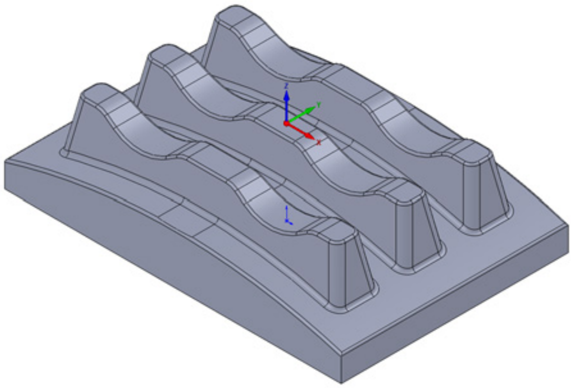
The tool tilting is defined using the **Tilted relative to cutting direction** option with the **Lag angle to cutting direction** of **20°**. The tool contact point is defined at the front tool face. This combination of parameters enables you to perform the machining by the toroidal surface of the tool.

During the **Machine simulation** of the operation, a message about exceeding the circular limits can be displayed; this is caused by the continuous rotation of the part during the machining. This problem can be avoided at the actual CNC-machine by setting in the postprocessor the rotation angle to zero, after a specific number of full turns around the rotation axis.



Exercise #2: Parallel to surface and Parallel to curve strategies

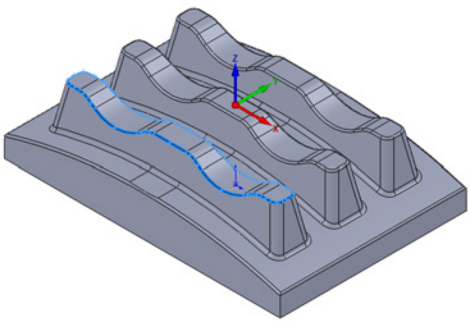
This exercise illustrates the use of the Parallel to surface and Parallel to curve operations for machining of the part shown on the illustration.



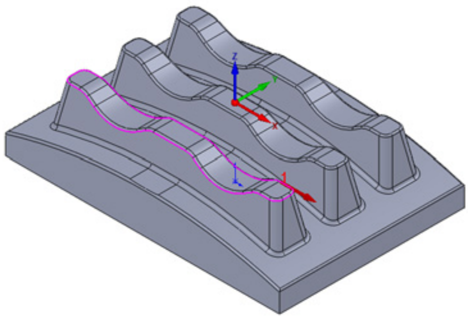
5X_ParC_faces1

This operation performs the finish machining of the part upper edges. A bull-nosed tool of **Ø16** with the corner radius of 8 mm is used for this operation.

The drive surface and edge curve are shown below:



Drive surface

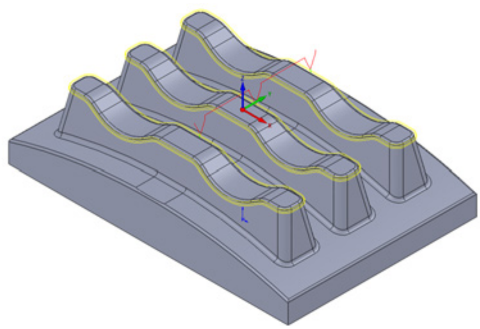


Edge curve

In the **Tool path parameters** page, for **Sorting**, the **One way** option is chosen as the cutting method, and machining is performed by **Regions**.

In the **Link** page, the **Use Lead-In** option is selected in the **First entry** section of **Approach/Retract** tab. For **Last exit**, the **Use Lead-Out** option is selected. In the **Links** tab, the **Follow surfaces** movement is selected for **Large gaps** in the **Gaps along cut** section. For **Links between slices**, the **Large moves** are performed with the **Follow surfaces** strategy.

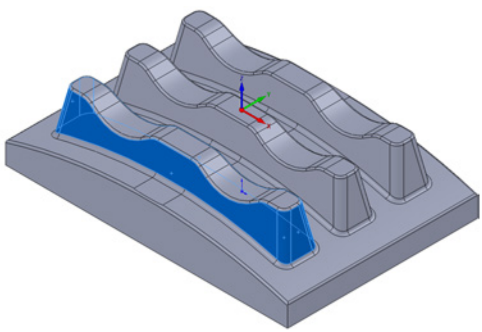
The tool tilting is defined using the **Tilted relative to cutting direction** option with the **Tilt angle at side of cutting** of **45°** to avoid the tool gouging with another rib.



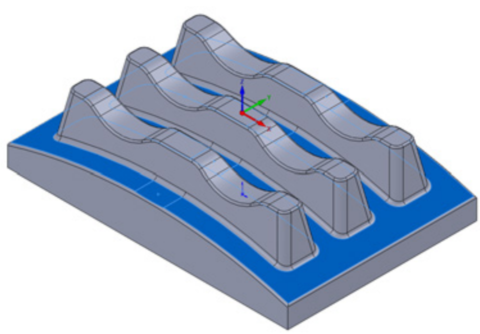
To repeat the same tool path for the other two ribs, **Use Rotation/Translation** option is selected in the **Roughing and More** page. For **Rotary axis around**, the **Y-Axis** option is chosen with **Number of steps** equal to **3**. The **Step over distance** is set to **80**.

5X_ParS_faces2

This operation performs the machining of the inclined walls of the part rib.
The drive surface and edge curve are shown below:



Drive surface

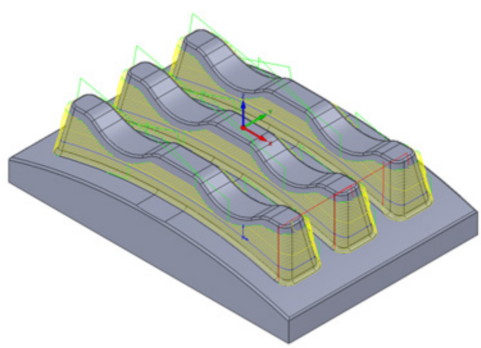


Edge curve

The **Parallel to surface** strategy is used in combination with the default **Zigzag** cutting method and **Maximum step over** of **2** to generate a 3D tool path along the rib surface. The **Machine by** option is set to **Regions**.

In the **Link** page, the **Use Lead-In** option is selected in the **First entry** section of **Approach/Retract** tab. For **Last exit**, the **Use Lead-Out** option is selected. In the **Links** tab, the **Follow surfaces** movement is selected for **Large gaps** in the **Gaps along cut** section to avoid unnecessary tool retracts.

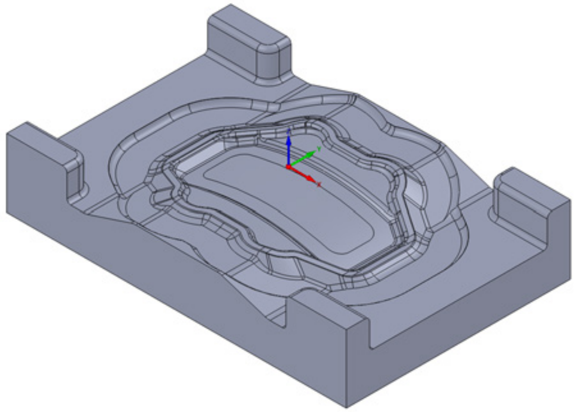
The **Tilted relative to cutting direction** option is used to define the proper tool tilting with the **Tilt angle at side of cutting** of **90°**. The side tilt is defined to **Follow surface ISO line direction**.



Gouge checking options are applied to determine the maximum possible tool entry position between the ribs using the **Tilting tool away with max. angle** strategy, while the **Use lead/lag and side tilt angles** option is chosen.

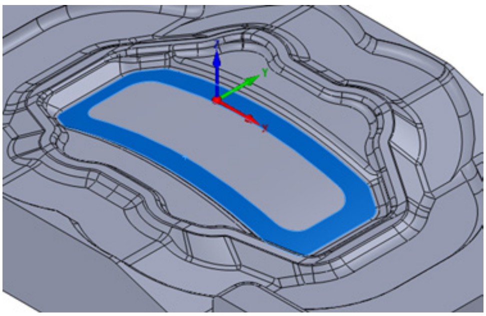
Exercise #3: Perpendicular to curve strategy

This exercise illustrates the use of the Sim. 5-Axis operation for machining of the mold core.

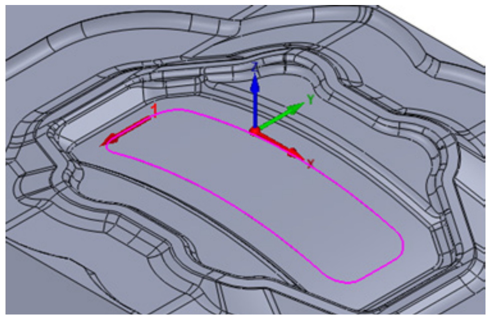


The **5X_PerpC_faces** operation is used for machining the part central face.

The drive surface and lead curve are defined as shown below.



Drive surface

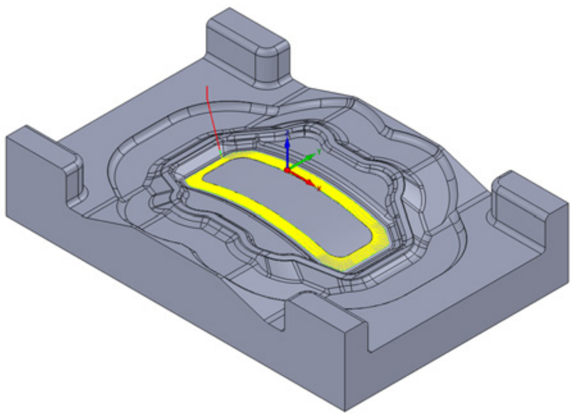


Lead curve

The **Perpendicular to curve** strategy is used to generate the cuts orthogonal to the lead curve with **Maximum step over** set to **0.2**.

A bull-nosed tool of **Ø4** with the corner radius of 1 mm is used for this operation.

The **Follow surfaces** movement is selected for **Large moves** in the **Links between slices** section to avoid the tool jumps in the corners.

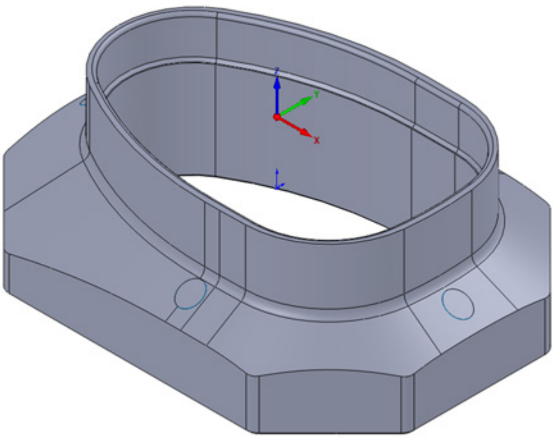


The **Reverse vertical tang. arc** is used for **Lead-In**.

Gouge checking options are used with a check surface defined on the cavity walls in order to avoid the possible tool gouging with the walls.

Exercise #4: Morph strategies

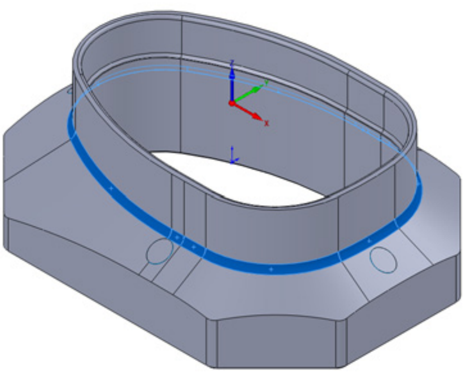
This exercise illustrates the use of the Sim. 5-Axis operations for machining of the part shown on the illustration.



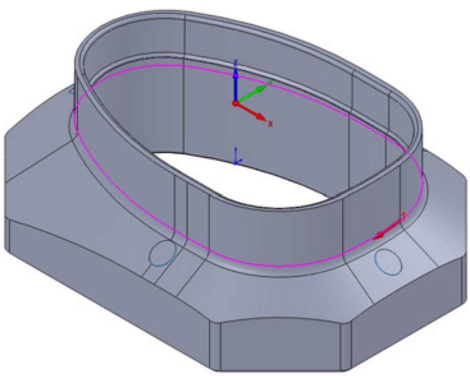
5X_MC_faces

The **Morph between two boundary curves** strategy is used with **Maximum step over** value set to **0.2**. A bull-nosed tool of **Ø6** with the corner radius of 3 mm is used for this operation.

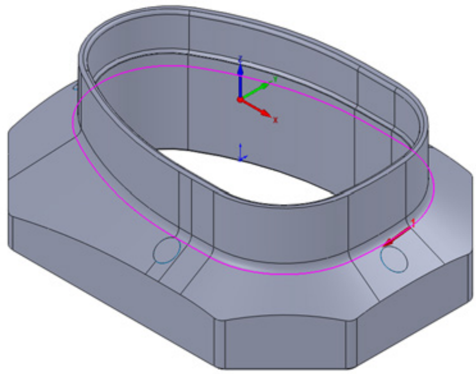
The drive surface and boundary curves are shown below:



Drive surface



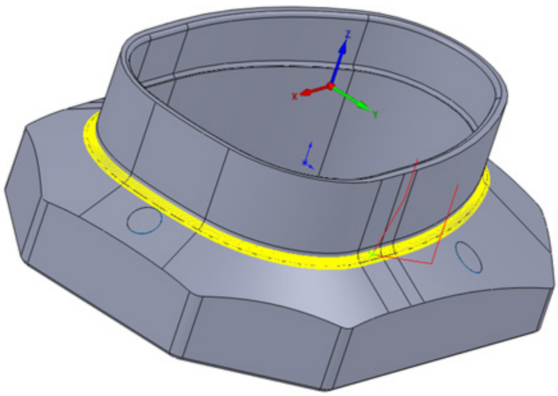
Start edge curve



End edge curve

The **Use Lead-In** and **Use Lead-Out** options are applied with **Vertical tang. arc**.

The **Tilted relative to cutting direction** option is used to define the proper tool tilting with the **Tilt angle at side of cutting** of **20°**.

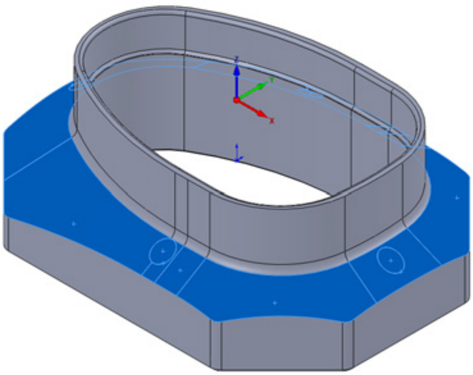


5X_MS_faces1

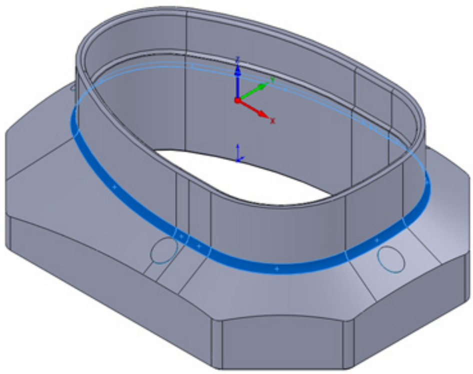
The **Morph between two adjacent surfaces** strategy is used with **Maximum step over** value set to **6**. A bull-nosed tool of **Ø12** with the corner radius of 1 mm is used for this operation.

In **Tool path parameters**, the **Flip step over** option is chosen to change the machining direction and perform it inwards.

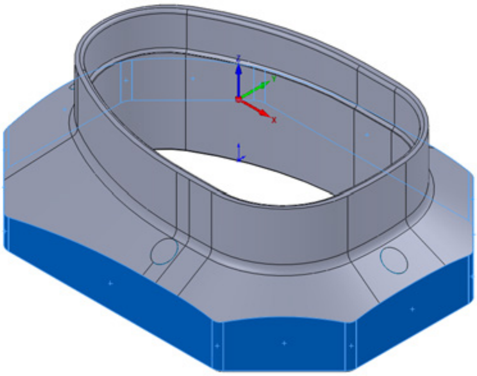
The drive surface and adjacent surfaces are shown below:



Drive surface

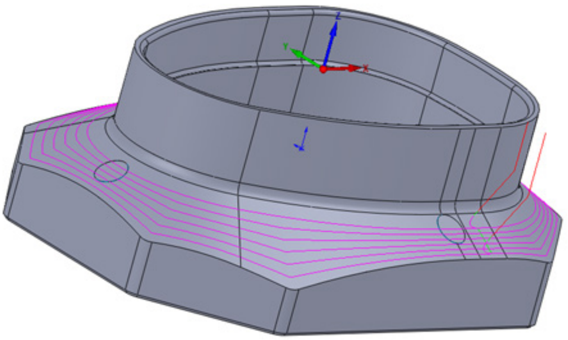


Start edge surfaces



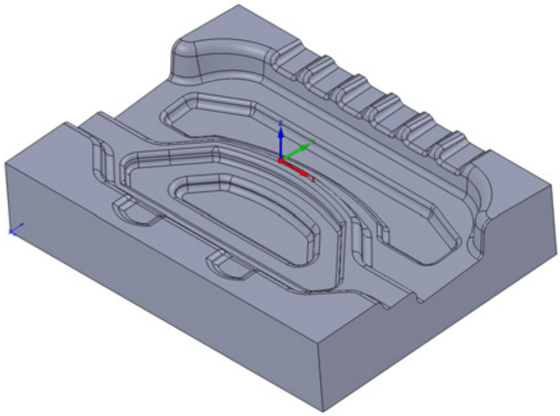
End edge surfaces

The **Use Lead-In** and **Use Lead-Out** options are applied with **Vertical tang. arc**.



Exercise #5: Projection strategy

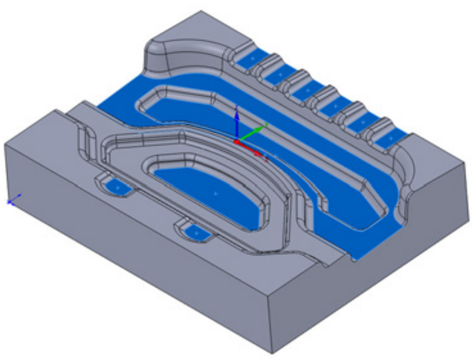
This exercise illustrates the use of the Sim. 5-Axis operation for finishing machining of the mold cavity.



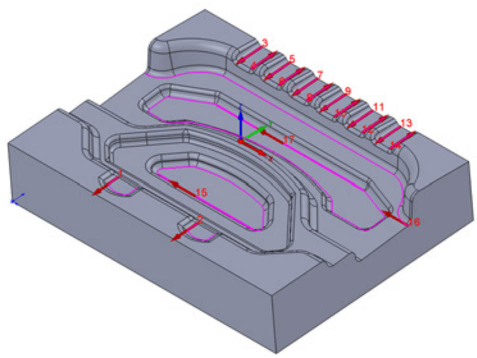
5x_Proj_faces1

This operation uses a ball-nosed tool of **Ø6** with the corner radius of 3 mm to machine in a single operation all fillets with the given radius on the part surfaces.

The drive surface and projection curves are shown below:



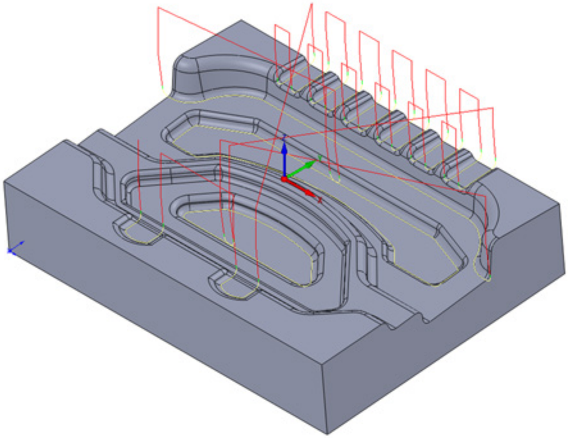
Drive surface



Projection curves

The tool contact point is set **At Center** position, which enables you to keep the tool tilted around the tool center point.

The **Use Lead-In** and **Use Lead-Out** options are applied with **Vertical tang. arc**.



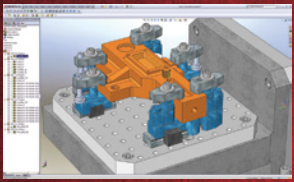
SolidWORKS
Certified Gold Product

SolidCAM + SolidWorks

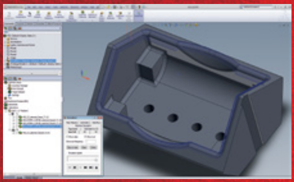
The complete integrated Manufacturing Solution

SolidCAM

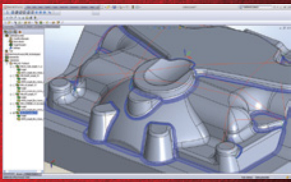
iMachining – The Revolution in CAM!



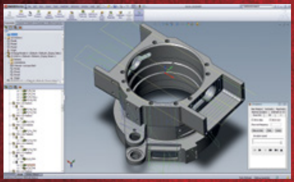
2.5D Milling



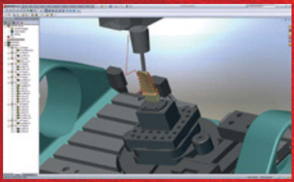
HSS (High-Speed Surface Machining)



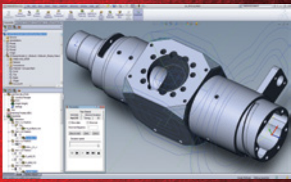
HSM (High-Speed Machining)



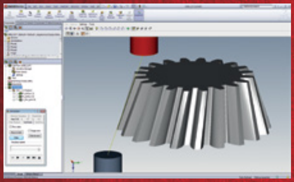
Indexed Multi-Sided Machining



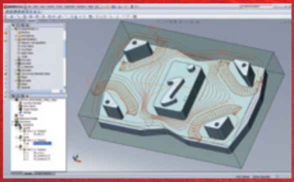
Simultaneous -5Axis Machining



Turning and Mill-Turn up to -5Axis



Wire EDM



iMachining

Service and Support

YouTube Broadcast Yourself™

www.youtube.com/SolidCAMProfessor
www.youtube.com/iMachining

www.facebook.com/SolidCAM

www.solidcam.com