GIBBSCAM® 14

Version 14 : September 2020

5-Axis

CAMBRIO

Proprietary Notice

Contents

Introduction to 5-Axis

Congratulations on your purchase of GibbsCAM 5-Axis and welcome to the 5-Axis documentation. This document covers the description and use of the 5-Axis product.

Before using the 5-Axis documentation or product, we highly recommend that you read *Getting Started* and *Mill*, and that you become familiar with the GibbsCAM Mill product and working with solids.

The 5-Axis product requires 2.5D Solids or SolidSurfacer, as well as a 4-axis/5-axis post processor. To use 5-Axis, the current MDD must be of type Mill, Mill/Turn, or MTM.

About 5-Axis

You can use the 5-Axis module to easily create almost any type of 5-axis toolpath. Toolpath-calculation strategies include surfaces, triangle mesh, wireframe, and Swarf machining.

The 5-Axis module allows you to tilt the tool axis intelligently, with numerous tilting strategies. Support is provided for all basic endmill types, including flat, ball and bull nose, conical, and convex-tip endmills, as well as for such undercutting tools as lollipop, barrel, dovetail, and slot milling tools. Tapered and stepped shanks are supported for all applicable mill-type tools.

Gouge checking for each tool covers the cutting length, the shaft, the shank, and the holder. All gouge checking is done against the drive surfaces and any additional check surfaces. 5-Axis helps you set the right retract strategy for your situation.

Whether you machine in three axes, four axes, or all five axes, the 5-Axis module allows you set limits for the machining area and to control tool angles.

Using 5-Axis in GibbsCAM

Part Setup

To use 5-Axis functions, the current MDD must be of type Mill, Mill/Turn, or MTM. Since the GibbsCAM 5-Axis module can generate 3-axis toolpath as well as 4-axis or 5-axis toolpath it does not require the MDD to have rotary axes. However, most of the functionality in the 5-Axis module is designed for machines with one or two rotary axes.

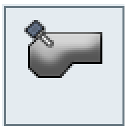
Save a Copy – Warning

For parts with 5-Axis toolpath at this release:

WARNING: We recommend against using Save a Copy to GibbsCAM 12 or earlier. This type of toolpath from the current release may not be preserved by Save a Copy, and so you may be unable to Redo, render, or post the part in the earlier release.

Creating Toolpath

5-Axis adds another machining function to the Machining (CAM) palette – a very powerful one.



A 5-Axis process combines a single tool from the Tool List with the 5-Axis machining function from the Machining palette. As with any other process, you click Do It to generate a new 5-Axis operation or Redo to regenerate an existing operation.

Converting Toolpath

You can use an Operation Modifier, 5-Axis Toolpath Conversion, to create persistent changes to a 5-Axis operation. Each time the 5-Axis operation is regenerated, the modifier is reapplied. The most common application is to convert 3-axis input toolpath to 5-axis toolpath with automatic collision-checking and tilting to accommodate a much shorter tool in a tool holder.

5-Axis Toolpath Conversion behaves as if the tool is a ball mill, regardless of the actual tool used to generate the input toolpath. This is because the toolpath conversion can only model the cutter location by using the input toolpath and the tool radius. Therefore, the contact point of the tooltip is used to calculate a virtual tool centerpoint, and it is this calculated centerpoint that the tool tilts around.

If the actual tool has a larger diameter than the tool that was used for creating the input toolpath, there will be collisions with the target surface. If the actual tool is smaller than the input toolpath's tool, it will not reach the target surface and there will be no contact point.

What is 5-Axis Machining?

The term "5-axis machining" refers to machining with 3 linear axes and 2 rotary axes. The rotary axes can rotate either the tool or the part, or in some cases, both the tool and the part.

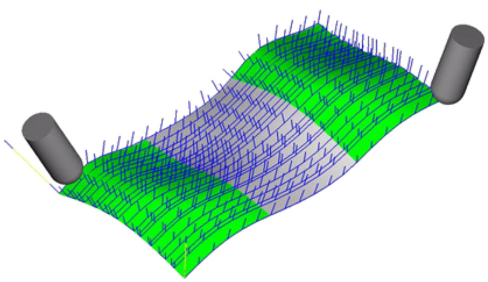
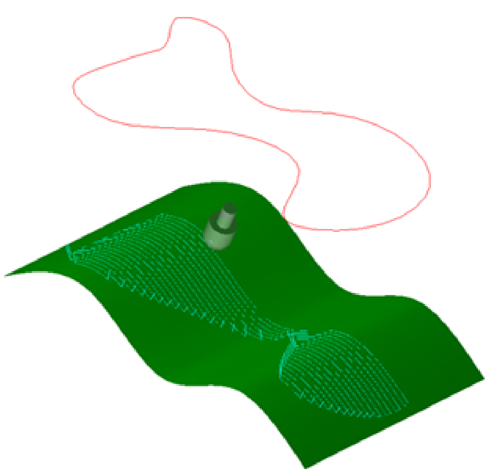
Toolpath for many machine types. Although the 5-Axis module emphasizes full 5-axis machining, it can generate toolpath for many kinds of machines, including 4-axis and 3-axis machines. For more information, see "Types of Machines" on page 15.

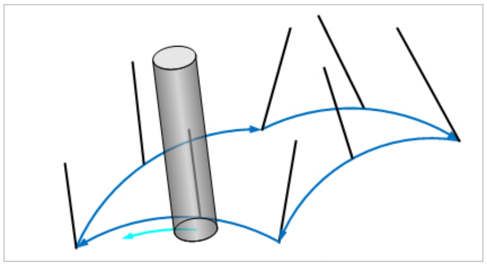
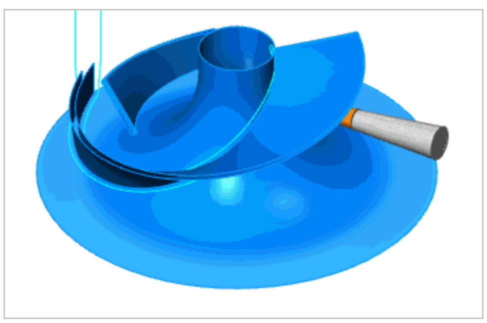
Advantages. The 5-Axis module can help you achieve shorter cycle times, smoother finishes, longer tool life, and sophisticated collision-avoidance methods. For more information, see Advantages to 5-Axis Machining.

Industries. 5-Axis machining is used for machining molds and dies, cutting tools, plastics, furniture parts, impellers, turbine blades, and many other applications. For more information, see Industrial Uses of 5-Axis Machining.

5-Axis Toolpath Calculation Strategies and Patterns

The GibbsCAM 5-Axis module offers a variety of options for calculating and generating toolpath.

Calculation	Strategies
Surfaces calculation generates toolpath points on parametric surfaces. For details, see Surface Machining.  To use Surfaces: After you choose a cut pattern – see “Pattern Settings (Calculation based on Surfaces)” on page 62 – you select drive surfaces and specify values for cutting area, sorting, surface quality, and stepover.	Cut strategy depends on Pattern: Parallel cuts (Calculation based on Surfaces) Perpendicular to curve Morph between two curves Parallel to curve Project curves Morph between two surfaces Parallel to surface
Triangle Mesh calculation generates toolpath points by dropping a collision-free contour from a set direction onto the machining surface. When the tool tilts, it rotates around a fixed contact point on the mesh. 	Cut strategy depends on Pattern: Rough Parallel cuts Project curves Constant Z Constant cusp Flatlands Pencil Projection

Calculation	Strategies
To use Triangle Mesh: After you choose a cut pattern – see “Pattern Settings (Calculation based on Triangle Mesh)” on page 76 – you select machining surfaces and then specify values for cutting area, sorting, surface quality, and stepover.	
Wireframe calculation generates a single toolpath along a drive curve, without machining surfaces, interpolating between orientations set by user-selected lines.  To use Wireframe: You select drive curves and orientation lines – see “Pattern Settings (Calculation based on Wireframe)” on page 91 – and then specify values for cutting area, sorting, and surface quality.	
Swarf Machining calculation produces the target surface with only one cut, using the whole flute length of the tool.  To use Swarf Machining:If you choose the Automatic strategy (recommended), you define the part's surfaces and upper/lower curves, choose machining approaches and start point type, specify values for surface quality and additional settings in the Multi Cuts and Corners pages. For more information, see “Calculation Strategy: Swarf Machining” on page 93	When calculation is based on Swarf Machining, the 5-Axis user interface changes in several ways. Surface Paths tab for Swarf Machining Tool Axis Control tab for Swarf Machining Gouge Check tab for Swarf Machining Multi Cuts tab Corners tab

Types of Machines

The 5-Axis module will generate effective and efficient toolpaths for almost any type of part requiring 3-axis, 4-axis, or 5-axis tool motion, with special attention given to full 5-axis machining.

Many different machine tools, with various sizes, shapes, and configurations, offer 5-Axis capability, such as horizontal and vertical milling machines, mill/turn machines, and multi-tasking machines (MTM machines). The GibbsCAM 5-Axis product is an effective tool for driving all of these types of machines.

Advantages to 5-Axis Machining

5-Axis machining has many significant advantages. With the GibbsCAM 5-Axis module, these benefits can be realized by developing efficient 5-Axis toolpaths for your 4-axis and 5-axis machine tools.

Improved cycle times and a better surface finish

In very large surfaces with large radii of curvature (such as Car Body Forming Dies), machining times may be reduced by 20 to 30 percent. This is because of the enormous advantage of having the toolpath calculations done on the smooth mathematically correct surfaces. The machine runs more smoothly and therefore the resulting surface finish is better than those systems that calculate on triangles or comparable entities.

When machining steep regions in cores of injection molds, GibbsCAM 5-Axis creates 5-axis swarf toolpaths on convex surfaces that use the outer diameter of the cutter at maximum cutting speed. This has tremendous advantages that again reduce time and improve surface finish.

Improved surface finish and longer tool life

When machining regions with small surfaces and higher radii of curvature, ball end mills have to be used. In some systems, the tool axis can only be kept normal to the surface, where the tool to surface contact point is at the bottom center of the tool, which is not the best part of the tool to cut with. With GibbsCAM 5-Axis the cutter can be tilted to the surface at an optimized angle to achieve a constant chip load and high feedrate at the contact point.

3-axis vs. 5-axis

In 3-axis machining of deep cavities with small radii at the bottom, very long cutters are required which increase delivery time and also result in vibrations or deflections when machining. This can be overcome with 5-axis machining using standard tools and tilting the cutter and holder away from steep walls when necessary to avoid collisions.

Collision avoidance

Even small movements of the cutter on the surface of the work piece can result in very large movements in all axes of the 5-axis machine. This is amplified via the tool, holder and spindle as a mathematically complex 3D Swarf body. Because of this, every surface inside of the piece is at risk of a collision.

GibbsCAM 5-Axis prevents these collisions by calculating all tool and holder collisions with the part and any fixtures. The user is provided with many options for avoiding the collisions and creating smooth and effective tool motion.

The simplest, but least effective of the collision avoidance strategies is the retraction of the cutter along the tool axis until there is no further collision; this is sometimes referred to as “comp-movement”. This strategy is only useful in some limited cases. An example of this is that of using drive surfaces to machine to comp-surfaces. The geometry of the toolpath is determined by the drive surfaces and the tool is retracted to the comp surfaces. This strategy is often used in the machining of tire moulds and door seals in the automotive industry.

In many cases, a more efficient way to avoid collision is to tilt the tool. This applies especially for holder collisions. As the holder has a considerably larger diameter than the tool combined with a long distance to the tool tip, even tilting a small angle can avoid the collision. Powerful algorithms in the 5-Axis module keep tool and axis movements smooth to avoid problems on the work piece surface due to excessive axis motions.

Another method to avoid collisions is to push the tool out of the work piece in a given direction. A good application for this collision avoidance strategy is the shaft of a turbine blade.

Axis limits

As an additional option to the automatic collision avoidance the rotational axis can be restricted in different planes to user-defined angle limits. An overtravel of the spindle is suppressed by locking the axis within the limit angles. This option saves calculation time, because fixtures may generally be excluded from automatic collision calculations.

Post processors and machine tool simulation

When used together with the GibbsCAM Machine Simulation, the 5-Axis module provides a very powerful tool for visualizing and optimizing 5-Axis toolpaths. GibbsCAM Machine Sim provides a virtual simulation of the whole machine tool, showing the motion of all rotational and linear axis. Even the work piece is shown in detail, with material removal, and can be placed on the rotational table at a user-defined position. The objective has been to provide a valuable tool to determine the optimum part-setup and toolpath strategies to avoid unnecessary try-outs on the real machine.

Reliable post processors for most 5-axis machines are available. Additional post processors are continually under development due to new customer requirements. Both custom GibbsCAM posts and ProAXYZ drivers can be developed to match customer machine control specifications.

Industrial Uses of 5-Axis Machining

The following sectors make particular use of 5-axis machining:

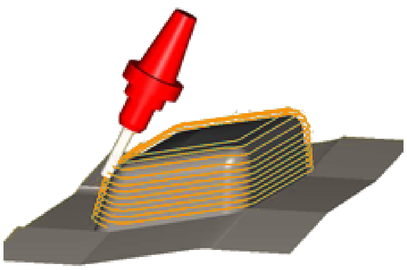
- Mold and Die Industry
- Machining Cutting Tools
- Plastics Industry
- Cylinder Head Machining

- Impeller Machining
- Turbine Blades
- Eccentric Shafts
- Extrusion and Injection Molding Screws

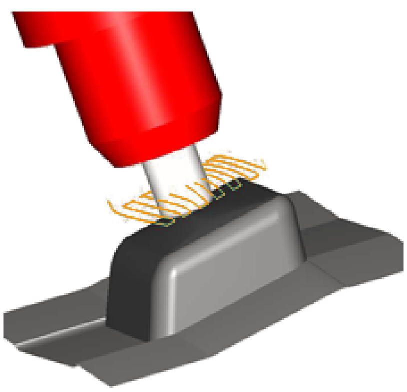
Mold and Die Industry

Machining of the core

In steep sections of the mold swarf machining on steep surface areas provides great advantages. The contact geometry between cutter and work piece is a line, therefore a smooth surface finish can be achieved with less cuts and less time. Small radii and sharp inside corners are marked as remaining stock for subsequent machining. Mold parting surfaces can be defined as check surfaces to be left unmachined by retracting the tool.



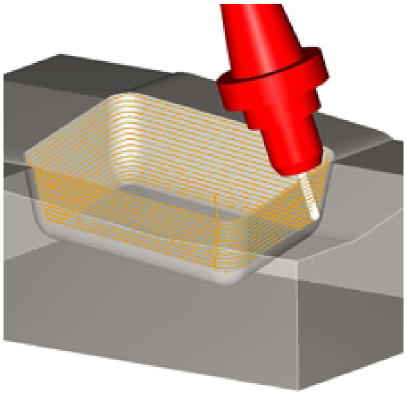
In shallow areas machining time is reduced using a large Bull Nose cutter with a small lag angle. The advantage is again the smaller number of steps to reduce machining time with an improved surface finish.



Machining of the cavity

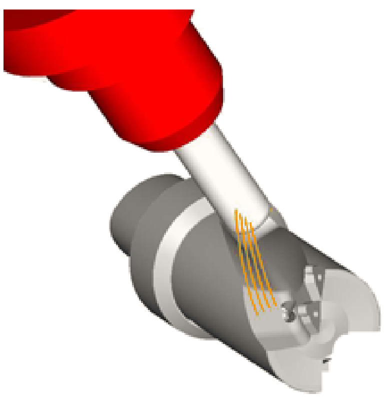
With deep cavities especially the advantages of 5-Axis are very evident. The powerful algorithms to automatically tilt the tool and holder away from the work piece in case of collision provide the ability to cut deeper molds with small radii with standard tools, without surface-finish spoiling vibrations.

Providing this strategy, GibbsCAM 5-Axis enables milling of mould sections which formerly had been done by Sinker type EDM, thus dramatically shortening the mold making time.



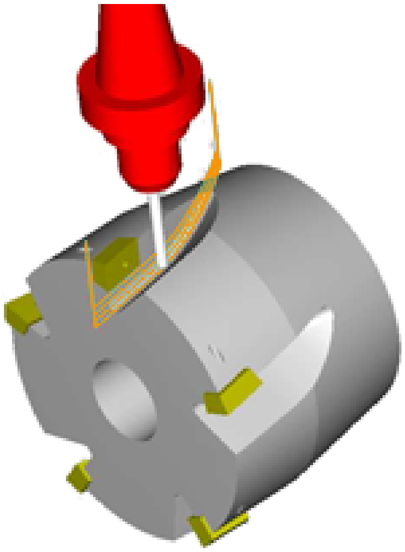
Machining Cutting Tools

When machining metal cutting tools, it is necessary to mill the flutes in 5-axis simultaneous mode. In many cases it is necessary to machine in several depth cuts as well as from the side, depending on the tool type. Shorter machining time is key because it is a very cost-sensitive business. The reduced machining time required can only be achieved with a very smooth toolpath, which is provided in the GibbsCAM 5-Axis module.



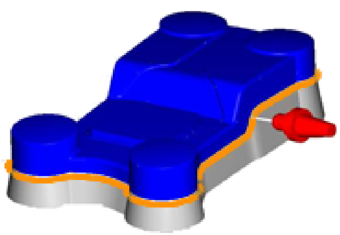
Conical tools

For finishing toolpaths on cutting tools the utilization of conical tools is a very good practice. Collision control, even on sharp corners between cuts, is an essential prerequisite.



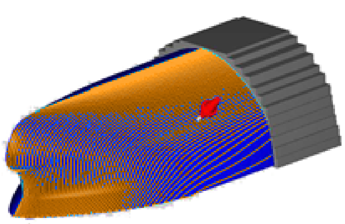
Plastics Industry

Trimming of plastic parts



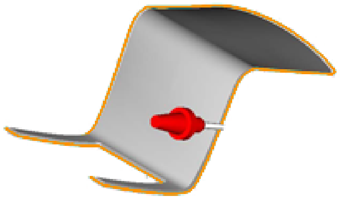
After production, vacuum-formed or fibre-reinforced plastic parts have to be trimmed, drilled, tapped, grooved etc. from all sides. The GibbsCAM 5-Axis provides maximum control of the tool direction because with these types of parts both rotary axes are on the tool and possible collisions still need to be avoided. Furthermore automatic collision detection between work piece/fixture versus tool and/or using calculated tool positions is a valuable feature.

Pattern Making and Woodcutting



Making patterns, especially patterns of very large parts using 5-axis machining provides dramatically reduced machining time compared with 3-axis machining. A large Flat End Mill is oriented perpendicular to the surface thus machining the maximum possible surface area. Another important feature of GibbsCAM 5-Axis allows the user to define a Stock Surface Model for roughing the CAD model to avoid unnecessary air moves.

Trimming of Furniture Parts

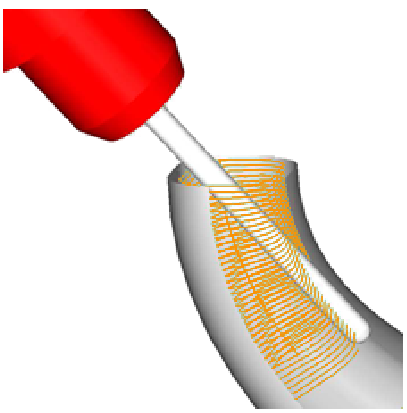


In the furniture industry router machines are often used for 5-axis machining. These machines typically have their rotational axis in the spindle. With these kinds of milling machines, collisions of aggregate and work piece can produce costly damage. GibbsCAM 5-Axis, cut part rendering, and Machine Simulation can avoid that and optimize programming time by just trying different setups of the part and different starting angles in the verification.

Cylinder Head Machining

Toolpath Strategies

One of the most complex tasks in building prototype motors for the automotive industry is the optimization of porting of the cylinder head which in turn determine the fuel consumption, power and torque. With its powerful cutting strategies, GibbsCAM 5-Axis provides solutions to decrease machining time.

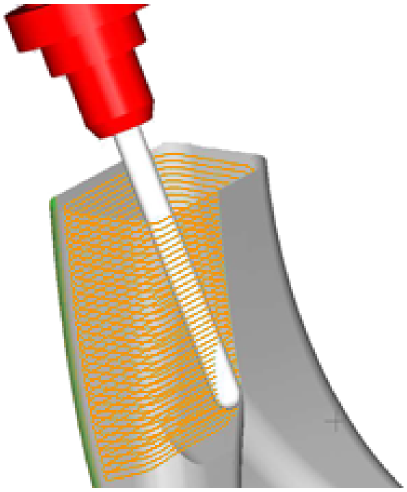


Collision Control

Collision control is available in the 5-Axis module. Multiple strategies for avoiding collisions are available, and multiple sets of gouge protected surfaces, each with their own parameter settings can be used at the same time.

Spiral Cutting

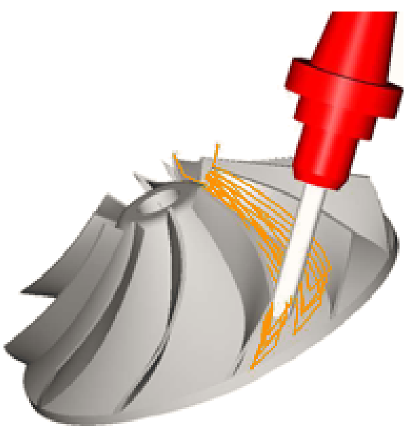
Using the spiral cut option without stepover provides improved surface quality. This option removes surface marks created by the standard stepover approaches.



Impeller Machining

Roughing of Impellers

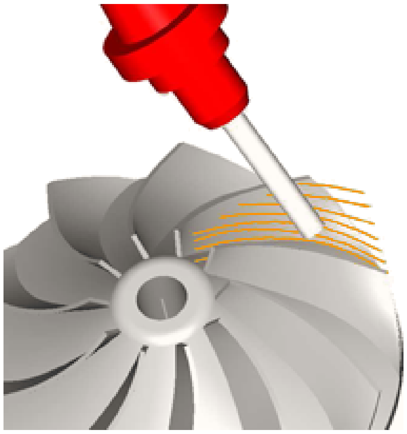
Machining of impellers is one of the most complex tasks of milling. Some of the reasons are that the very small available space for the tool to offset the necessary angular motion of the rotational axis has to be combined with highest requirements of surface quality. Further requirements due to economics, is a reduced machining time with very smooth toolpaths on a Multisurface-Model. These models, which can have very thin ribs, tend to be destroyed by vibrations.



Best results are achieved with depth cuts based on the stock definition of the upper and lower surface of the impeller ribs using conical cutters. Another important feature of GibbsCAM 5-Axis is the ability to dynamically adjust the feedrate based on surface radius of curvature.

Finishing of Impeller Blades

The critical requirements of finishing the blades of the impeller are accuracy and smooth axis movements, because anything else will result in surface marks and/or surface cut-off. To achieve this kind of toolpath GibbsCAM 5-Axis provides the ability to assign tool orientation to user-defined tool axis vectors.



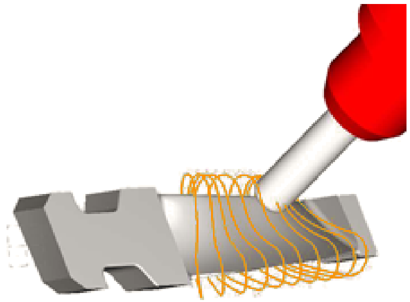
Achievements

GibbsCAM 5-Axis provides all of the necessary tools for successful 5-Axis Impeller machining. Shortest cycle time is paired with an optimum surface finish which can only be possible by calculating directly on surface representation and taking the 5-axis collision control between points into account for collision detection.

Machining of Turbine Blades

Roughing of Turbine Blades

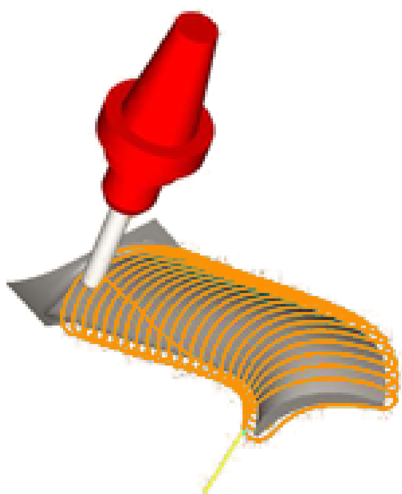
Turbine blades have always been a classical task for 5-axis machining. It combines the easy and collision free handling and programming of tilted tool plane roughing toolpaths using large end mills, with complex 5-Axis simultaneous toolpaths.



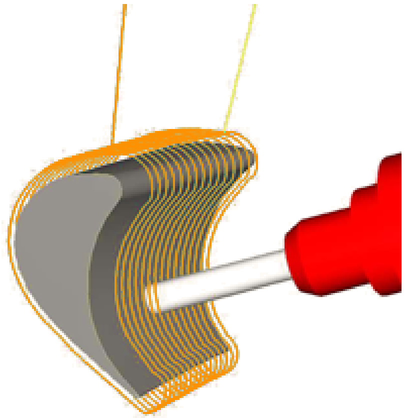
Finishing of Turbine Blades

There are two strategies to finish turbine blades.

The first strategy is to finish with a ball end mill and to tilt the tool to the rotational axis, with a defined angle, to optimise cutting conditions on the blade surface and avoid holder collisions. As a further strategy, the spiral toolpath is selected to improve surface quality by avoiding marks otherwise left by tool stepover.

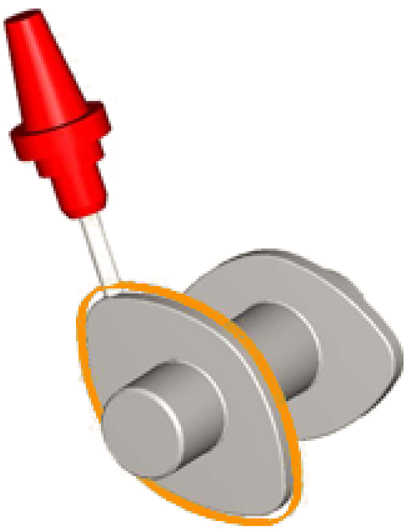


The second strategy uses a bull nose end mill. Using a large cutter diameter and utilizing the 5-Axis module's ability to detect collisions between the cutter and the work piece and to avoid it by tilting the tool, machining time is improved compared with strategy #1. A spiral toolpath can also be used in this case.



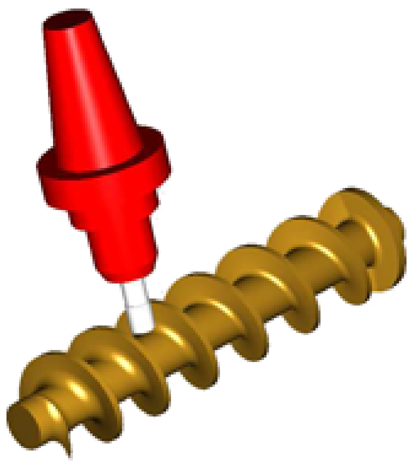
Machining of Eccentric Shafts

GibbsCAM 5-Axis offers a variety of functions for machining eccentric shafts. Camshafts and connecting rod journals can be milled with simultaneous 4-axis with collision check. The stock can be defined as well as a cutter shift into the cutting direction for rough machining.



Mill/Turn Machining for Extrusion and Injection Molding Screws

Another application for the 5-Axis module is extrusion and injection molding screws for plastics and rubber processing. It is good practice to use conical tools for this kind of application. You can define multi-passes for roughing and finishing toolpaths, keeping them closely oriented at the complicated screw shape to avoid unnecessary air moves.

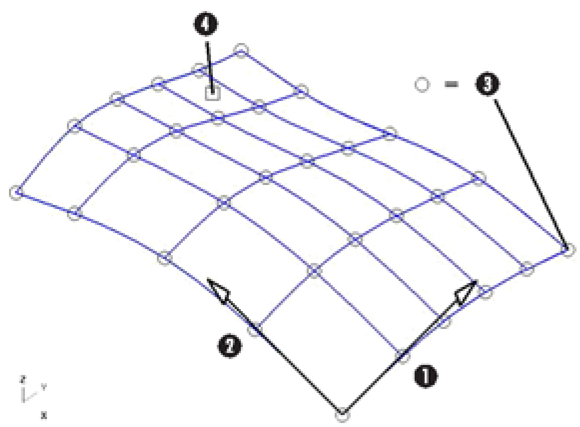


Surface Machining

To be successful with 5-Axis machining with calculation based on Surfaces, you need to have an understanding of surfaces and how toolpath can be created from the surfaces. We will start with a discussion about Single Surface 5-Axis Flow Line Machining below, expand the discussion to “Characteristics and Restrictions of Multi Surface Flow Line Machining” on page 29, and finally “Real Multi-Surface Machining” on page 30.

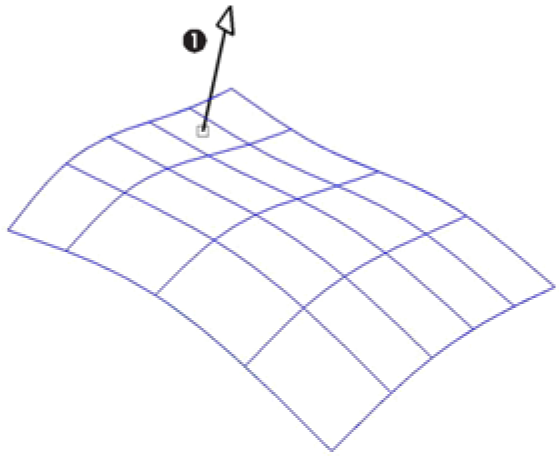
Single Surface 5-Axis Flow Line Machining

CAD surfaces are generally built on interpolation points. CAD/CAM systems usually define the surface XYZ points as a 2-parameter representation. These parameters are referred to as U and V.



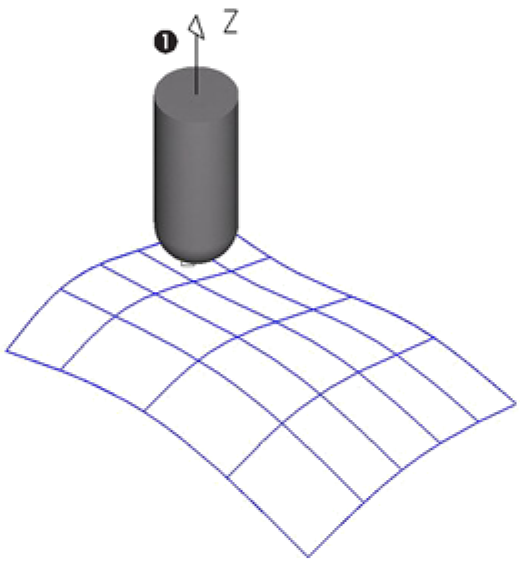
1. Parameter U
2. Parameter V
3. Interpolation Point
4. Surface Point (U, V)

Each surface point’s X, Y and Z coordinate can be calculated from a unique pair of U and V. Each surface point is associated with a surface normal that is always perpendicular to the surface at that point.



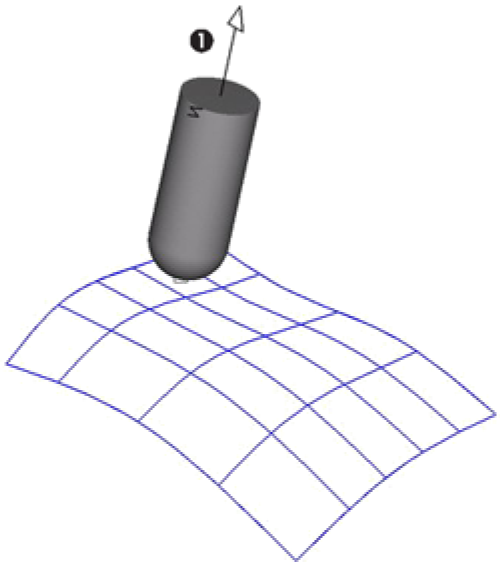
1. Surface Normal

In 3-axis machining this surface normal points to the cutter center of a ball end mill. The cutter axis always comes from one direction and it is usually aligned with Z. In some rare cases the cutter is aligned with the Y axis.



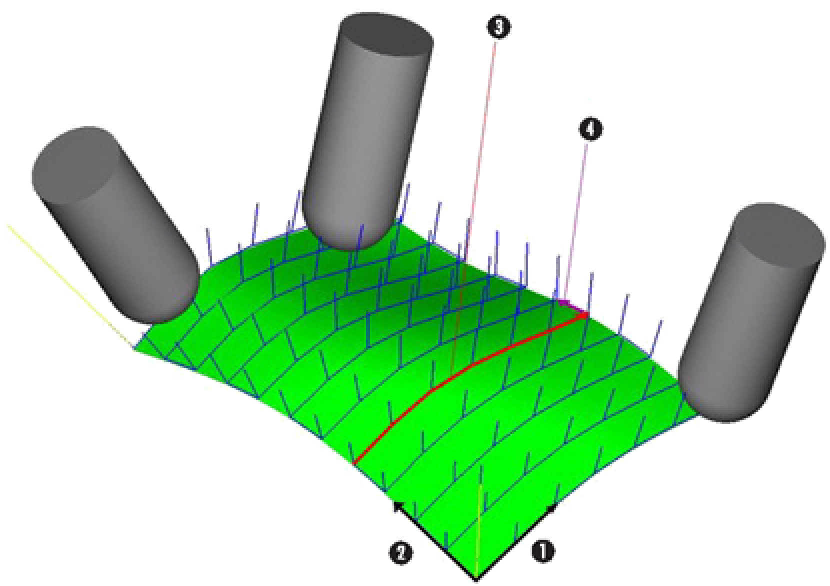
1. Surface normal dependent cutter orientation in Z for 3-axis machining using cutter center.

In 5-axis machining the surface normal may not only determine the cutter center but the cutter orientation as well. There are other ways to control the tool axis to achieve a 5-axis machining toolpath, but this will be discussed later.



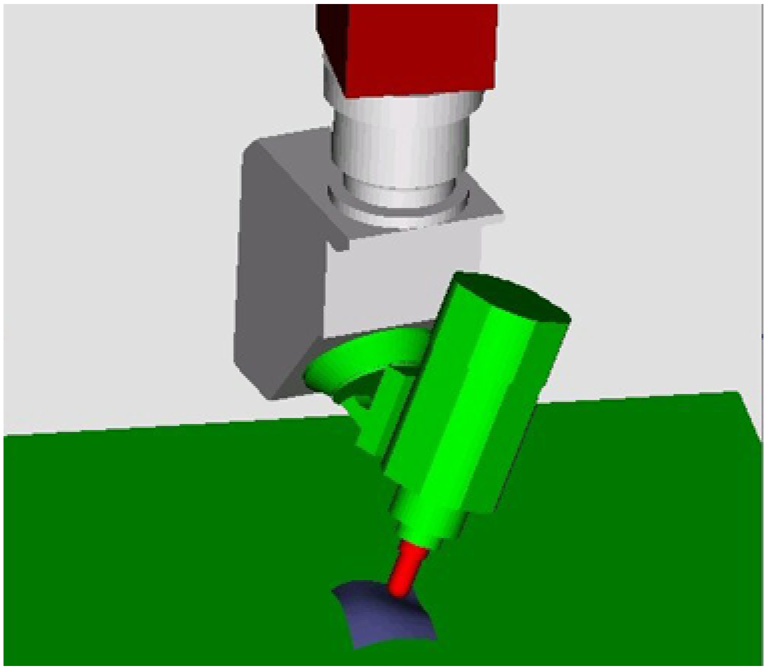
1. Surface normal dependent cutter orientation for 5-axis machining using cutter center.

A flow line 5-axis toolpath follows only the U-direction and V-direction of the surface. In the following figure a 5-axis flow line toolpath is shown which is calculated mainly in the U-direction. As soon as the surface edge is reached the tool steps in V and then continues movement in the reversed U-direction to achieve a Zig Zag (bidirectional) toolpath. The tool axis direction is changed at every single point of the toolpath according to the local surface normal. This kind of machining is called a single-surface 5-axis flow line toolpath, typically found in many CAM systems.



1. Parameter U
2. Parameter V
3. Tool movement in U direction
4. Tool movement in V direction

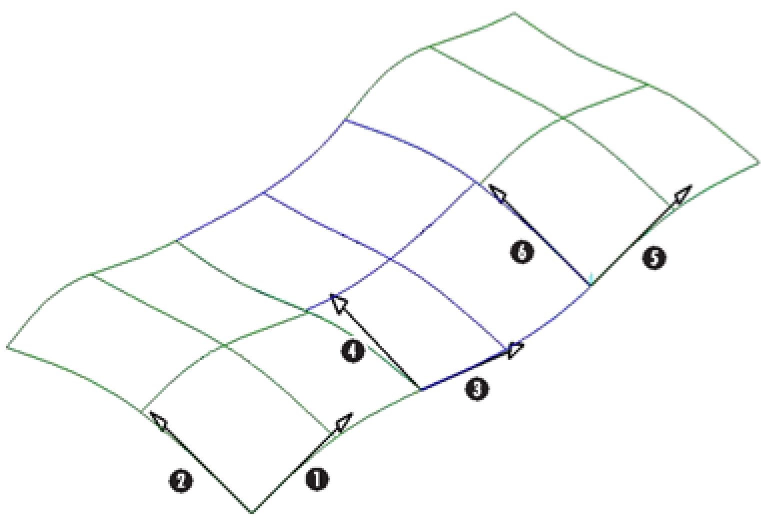
On a real machine the machine has to move its axis to rotate the tool to the required direction as shown below.



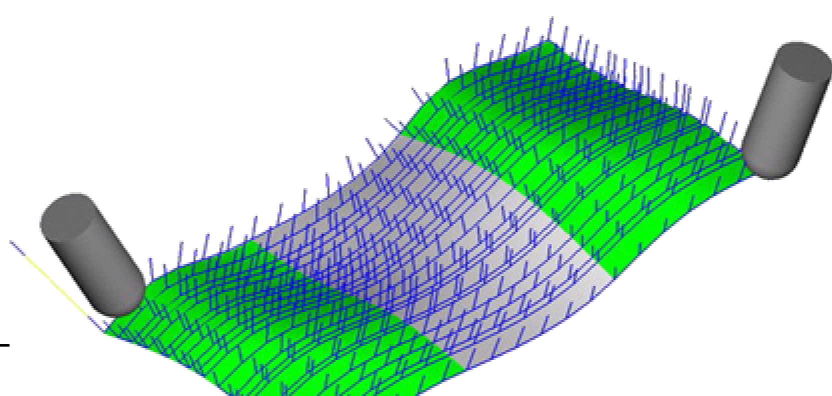
We expand this concept in Characteristics and Restrictions of Multi Surface Flow Line Machining.

Characteristics and Restrictions of Multi Surface Flow Line Machining

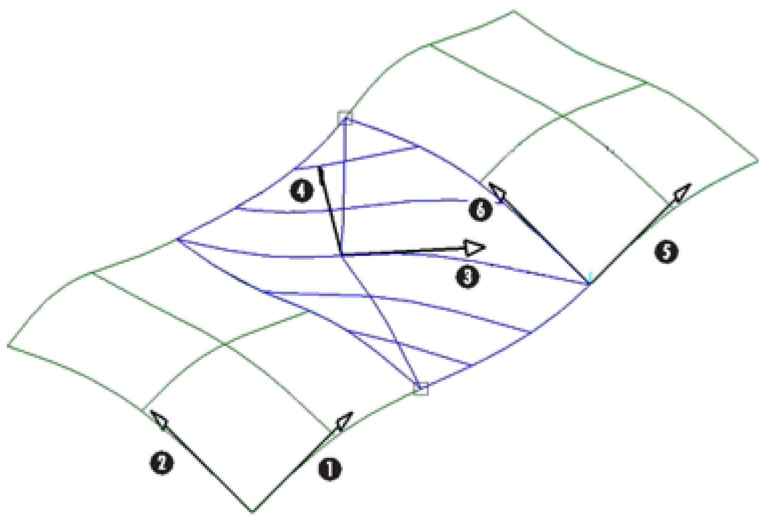
Multi surface flow line machining requires all surfaces to have the same U- and V- parameter direction. The figure below shows a sample set of 3 surfaces, which have the same u- and v-direction and the resulting Multi Surface Flow line toolpath.



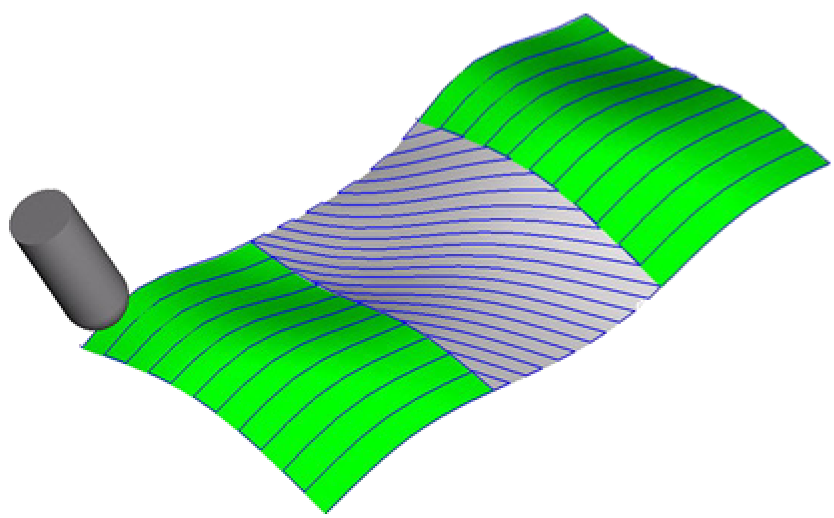
1. Parameter U1
2. Parameter V1
3. Parameter U2
4. Parameter V2
5. Parameter U3
6. Parameter V3



If surface number 2 does not have the same U- and V-direction, as in the sample shown below, a calculation of the toolpath based on the flow line of the surfaces is no longer reasonable.



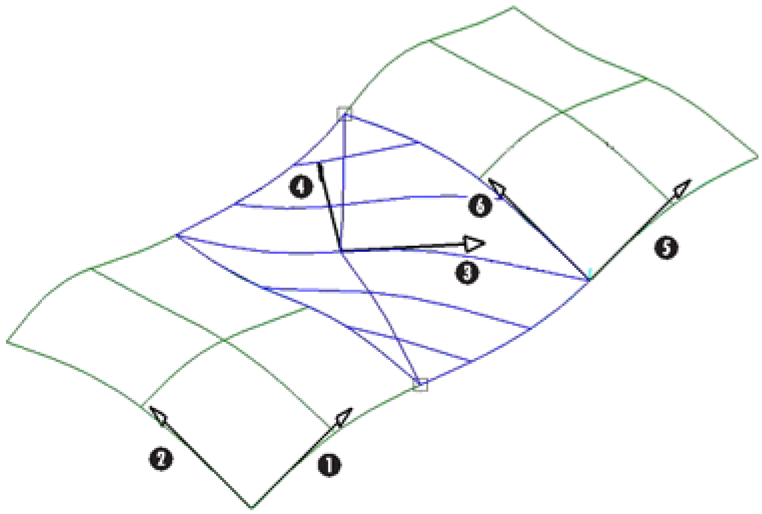
1. Parameter U1
2. Parameter V1
3. Parameter U2
4. Parameter V2
5. Parameter U3
6. Parameter V3



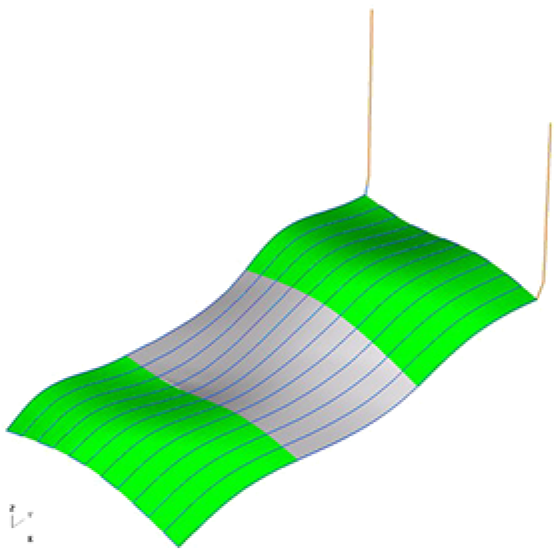
This can be handled using Real Multi-Surface Machining.

Real Multi-Surface Machining

When surfaces do not have the same U- and V-direction, a more sophisticated approach needs to be applied to solve the machining task. GibbsCAM 5-Axis has been developed to generate a smooth toolpath even on these arbitrarily orientated surfaces. The 5-Axis option provides a full suite of toolpath creation strategies to control the toolpath using drive and check surfaces, one or more curves, axis directions, line vectors, and other parameters independent of the U and V direction of the underlying surfaces.



1. Parameter U1
2. Parameter V1
3. Parameter U2
4. Parameter V2
5. Parameter U3
6. Parameter V3

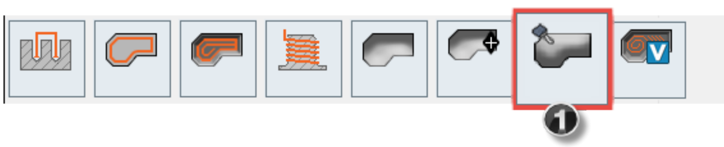


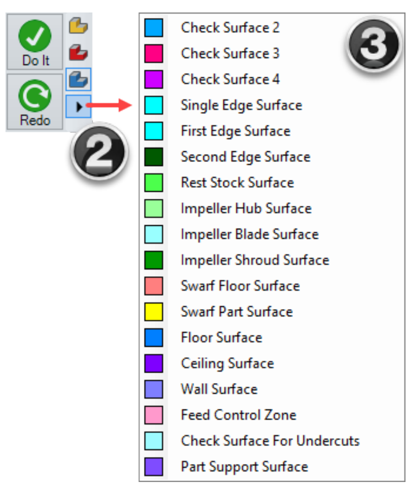
This is only one issue addressed by GibbsCAM 5-Axis that is necessary to generate an efficient 5-axis toolpath. There are a number of issues, such as surfaces or edge curves, collision avoidance, and post processor output that have been addressed by the product to enable GibbsCAM users to be productive on 5-axis machining.

5-Axis Interface

Machining Palette for 5-Axis

When the 5-Axis product is installed and enabled, the 5-Axis machining function tile is added to the Machining palette (item 1 in the following illustration), and a fly-out button (item 2) allows access to the Custom Mode menu (item 3):



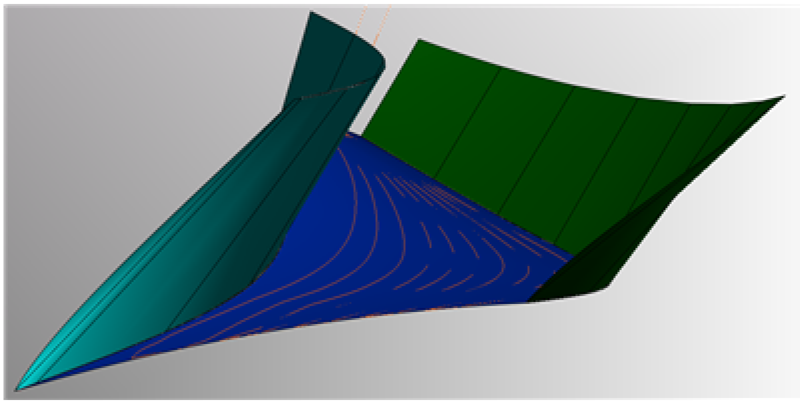


1. The 5-Axis function tile
2. Fly-out button to open the Custom Mode menu
3. Color choices available in the Custom Mode menu

Custom Mode

The Custom Mode menu lets you identify check surfaces, edge surfaces, and the like. It provides an extension to the selection mode buttons Part/Constraint/Stock. The extension lets you select additional types of surfaces that some 5-Axis dialogs prompt you to select, such as “First Edge Surface”.

In 5-Axis, when the selection mode is set to Part, you can select Drive Surfaces or Machining Surfaces. When the selection mode is set to Constraint, you can select Check Surface 1. Other surface designations, such as First and Second Edge Surface (shown below in a Morph Between 2 Surfaces pattern) are displayed in the colors of the Custom Mode menu.

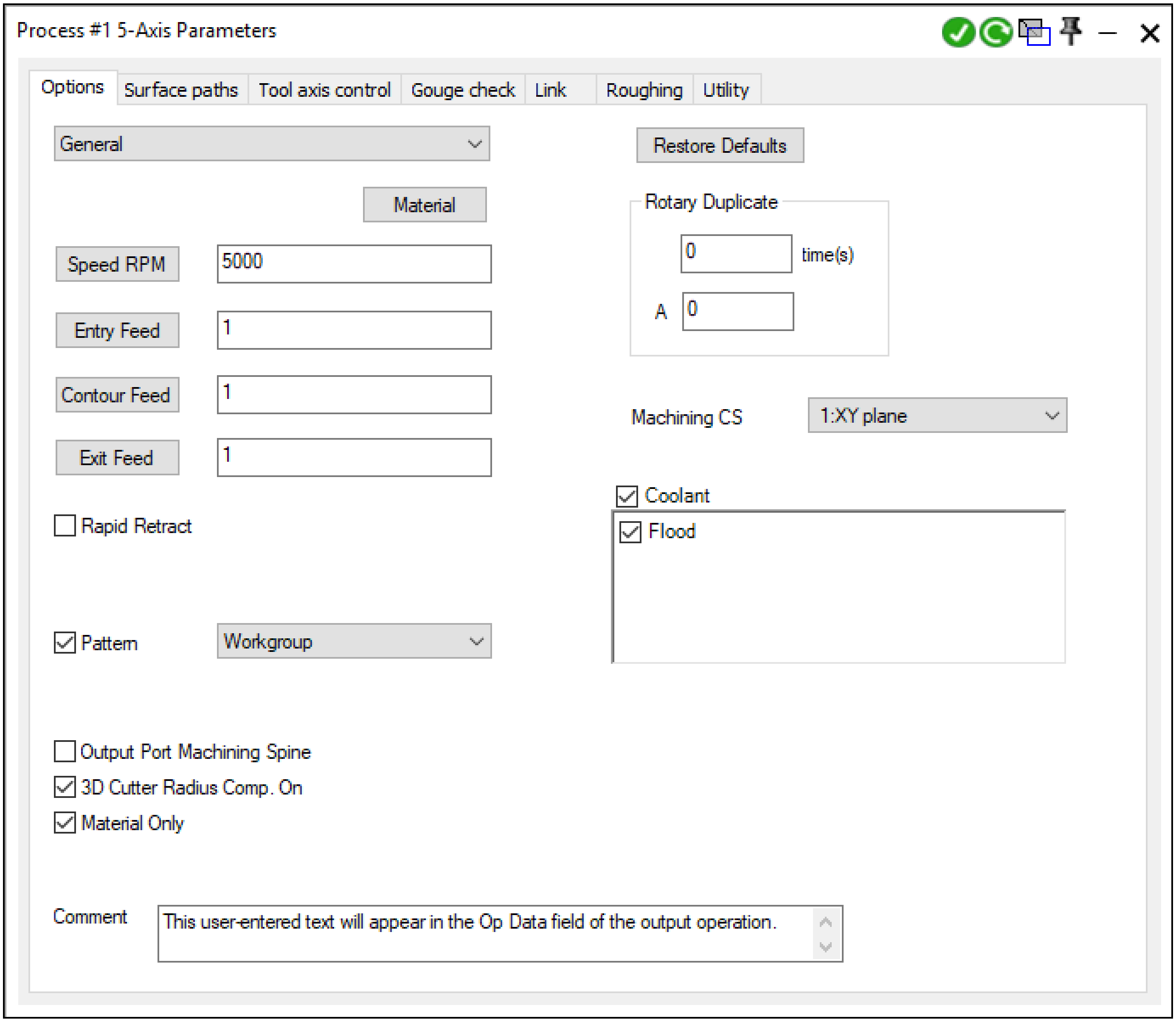


Part with Morph Between 2 Surfaces pattern, showing Drive Surface (blue, center, with toolpath) between First Edge Surface (left, cyan) and Second Edge Surface (right, deep green).

The 5-Axis Dialog Box

When a 5-Axis process tile is first created or double-clicked, the 5-Axis Parameters dialog box opens. When the top-level pull-down menu in the Options tab is set to General, the dialog box offers up to seven tabs that help you define and control 5-axis toolpath:

- “Options tab” on page 34
- “Surface Paths tab” on page 35
- “Tool Axis Control tab” on page 35
- “Gouge Check tab” on page 35
- “Link tab” on page 35
- “Roughing tab” on page 35
- “Utility tab” on page 35



If the top-level Options pull-down is set to a more specific choice than General, the number of tabs is decreased and choices are narrowed.

All tabs except Options make extensive use of graphics to help you visualize the options you are setting. The upper right area of most tabs and many sub-dialogs display one or more graphics that update according to your most recent user-interface actions.

The user interface is very dynamic: Setting a control in one tab can affect whether or not other controls appear, both on the same tab and on other tabs. For example, in the Surface paths tab, if you set the calculation to Swarf machining, then the number of tabs is reduced to four.

Options tab

The Options tab on the far left offers controls for the most basic of toolpath functions. Here you set common machining data such as feeds and speeds. Additionally, you can choose the type of interface to use, either the General interface which provides access to all of the system's options and parameters, or one of the specialized interfaces aimed at specific types of machining. For in-depth information, see "Options tab" on page 38.

Surface Paths tab

The Surface Paths tab lets you specify how 5-Axis calculates toolpath:

- For calculation based on surfaces, triangle mesh, or wireframe, additional controls let you set pattern options, cut area options, cut sorting options, and surface quality options.
- For calculation based on swarf machining, additional controls let you set sync options, pattern slice and layer options, machining options, and surface quality options.

Where applicable, this tab also allows you to select surfaces and/or edges (for driving, orientation, and/or part definition); to choose options for sorting, start point, and shifting; and to set stepover parameters. For in-depth information, see "Surface Paths tab" on page 58.

Tool Axis Control tab

The Tool Axis Control tab, when available, has controls that let you define the tool orientation. On this tab you can also set machining limit angles and define the contact point between tool and surface. For in-depth information, see "Tool Axis Control tab" on page 153.

Gouge Check tab

The Gouge Check tab has options for defining how to prevent the tool from gouging selected drive and check surfaces. For in-depth information, see "Gouge Check tab" on page 214.

Link tab

Surfaces defining the work piece may have gaps and holes. In such cases you can define the desired behavior of the toolpath. For example, small gaps can be ignored and milled without retracting or when big gaps are detected the tool can retract back to the rapid plane and skip the gap. Options such as this are set on the Link tab. For in-depth information, see "Link tab" on page 238.

Roughing tab

The Roughing tab, when available, has controls that let you define stock as well as the control the multi-passes option, the depth of cuts setting, any pocketing options and how plunging is performed. Any moves the tool makes in the air, i.e. movements that do not remove material, can be trimmed using the stock definition on this tab. For in-depth information, see "Roughing tab" on page 267.

Utility tab

The Utility tab, when available, has controls for special functions like optimizing the feedrates within the toolpath, creating toolpath with smoothed surface normals, or adding an axial shift to the resultant toolpath. For in-depth information, see "Utility tab" on page 291.

Operation Modifier: 5-Axis Toolpath Conversion

A special Operation Modifier named 5-Axis Toolpath Conversion lets you use much of the 5-Axis functionality on any Mill-type operation. Even on a 3-axis machine, this allows for sophisticated gouge checking and fine control over linking moves. On 4-axis and 5-axis machines, you can also control tool tilting.

Please Note: Parts from v10.1 and v10.3 can contain operation modifiers that used an incompatible combination of choices for "Tool axis will..." in the **Tool axis control** tab. If such a part is opened in the current release, an error message will be displayed, and the settings will be made compatible.

5-Axis Toolpath Conversion behaves as if the tool is a ball mill, regardless of the actual tool used to generate the input toolpath. This is because the toolpath conversion can only model the cutter location by using the input toolpath and the tool radius. Therefore, the contact point of the tooltip is used to calculate a virtual tool centerpoint, and it is this calculated centerpoint that the tool tilts around.

If the actual tool has a larger diameter than the tool that was used for creating the input toolpath, there will be collisions with the target surface. If the actual tool is smaller than the input toolpath's tool, it will not reach the target surface and there will be no contact point.

When you add or edit a 5-Axis Toolpath modifier, a dialog box appears that offers up to six tabs that help you define and control toolpath:

- Options tab
- "Tool Axis Control tab " on page 36
- "Gouge Check tab " on page 37
- "Link tab " on page 37
- "Roughing tab " on page 37
- "Utility tab" on page 37

All tabs except Options make extensive use of graphics to help you visualize the options you are setting. The upper right area of most tabs and many sub-dialogs display one or more graphics that update according to your most recent user-interface actions.

Options tab

The Options tab offers only two controls:

- Cut tolerance. This sets the basic tolerance for toolpath accuracy. For complete information, see "Cut tolerance" on page 148.
- Maximum Distance. This allows you to set the maximum distance between toolpath points. Smaller values will generate more points. For example, if this option is activated and the distance is set to 0.5mm, then at every 0.5mm (or less), a new toolpath position is calculated on the surface. If set, the value must be greater than 0. For complete information, see "Maximum distance" on page 148.

Tool Axis Control tab

For all output formats, the Tool Axis Control tab offers options for defining the contact point between tool and surface. When output format is 4-axis or 5-axis, you can also set tool orientation, limits on tool angles, and many other parameters. For in-depth information, see “Tool Axis Control tab” on page 153.

Gouge Check tab

The Gouge Check tab has options for defining how to prevent the tool from gouging selected drive and check surfaces. For in-depth information, see “Gouge Check tab” on page 214.

Link tab

Surfaces defining the work piece may have gaps and holes. In such cases you can define the desired behavior of the toolpath. For example, small gaps can be ignored and milled without retracting, or when big gaps are detected the tool can retract back to the rapid plane and skip the gap. Options such as this are set on the Link tab (provided that the Conversion link type is set to Relink). For in-depth information, see “Link tab” on page 238.

Roughing tab

The Roughing tab offers options that let you define stock as well as to control the multi-passes option, the depth of cuts setting, any pocketing options and how plunging is performed. Any moves that the tool makes in the air (that is, movements that do not remove material) can be trimmed using the stock definition on this tab. For in-depth information, see “Roughing tab” on page 267.

Utility tab

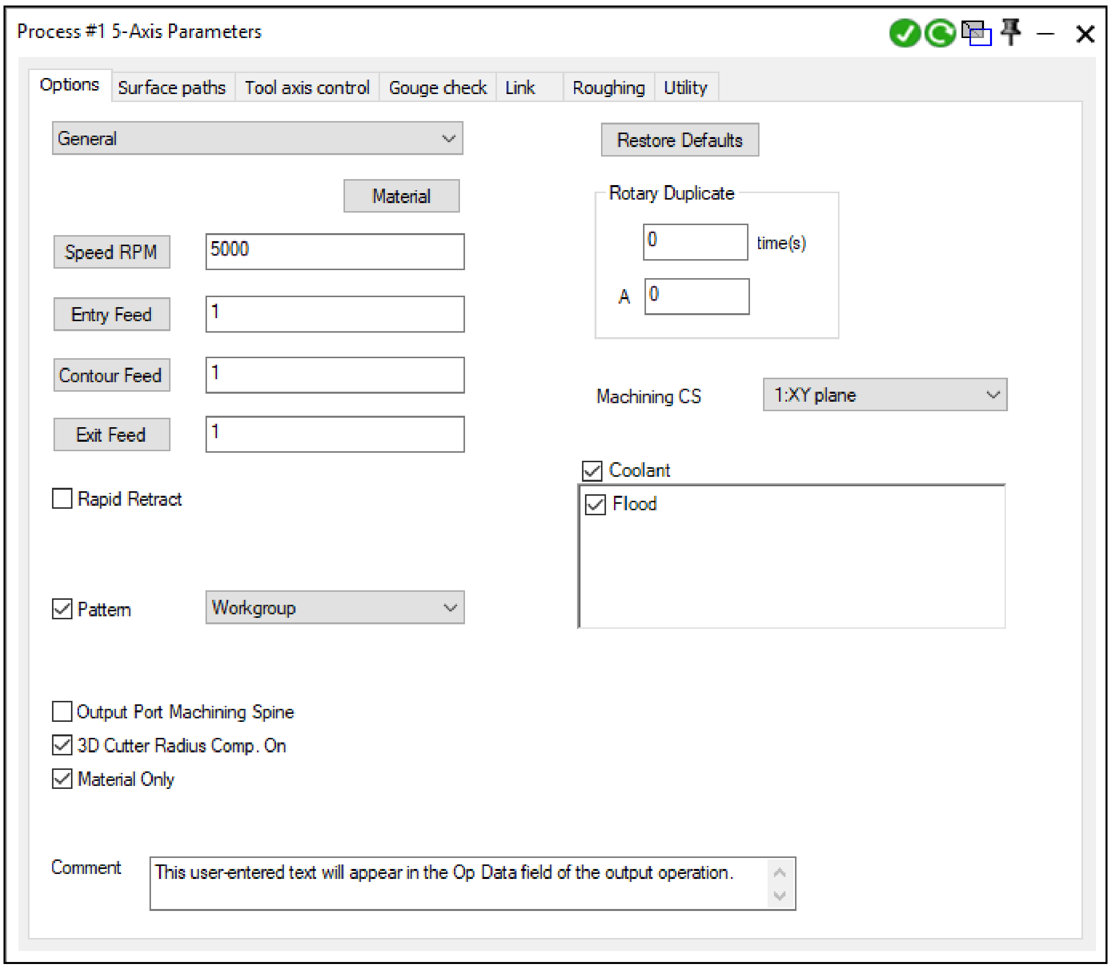
The Utility tab offers controls for special functions like optimizing the feedrates within the toolpath, creating toolpath with smoothed surface normals, or adding an axial shift to the resultant toolpath. For in-depth information, see “Utility tab” on page 291.

Options tab

About the Options tab

The **Options** page contains the very basic information that is common to all machining (see "Common Machining Controls" on page 39) such as feeds and speeds, coolant control, and patterns. This tab also lets you set rotary duplication (see "Rotary Controls" on page 40) and has a button to reset all 5-Axis controls across all tabs (see "Restore Defaults" on page 40).

Additionally, the **Options** page lets you change the 5-Axis system from a very general interface where you have many choices for creating your toolpath to a very specific interface that is specialized towards a particular type of machining. For complete information, see "Type of Machining" on page 41.



Common Machining Controls

Material
Clicking this button lets you modify the contents of the material database.

Speed RPM
Clicking this button will load a suggested speed based on the part material and the tool. You can also manually enter a value in the text box.

Entry Feed
Clicking this button will load a suggested speed for the tool when approaching the part based on the part material and the tool. You can also manually enter a value in the text box.

Contour Feed
Clicking this button will load a suggested speed for the tool when cutting the part based on the part material and the tool. You can also manually enter a value in the text box.

Exit Feed
Clicking this button will load a suggested speed for the tool when leaving the part based on the part material and the tool. You can also manually enter a value in the text box.

Rapid Retract
Activating this option will cause the tool to rapid when pulling off of the part to move between slices or passes.

Pattern
You can define a pattern from a workgroup. With 5-Axis Pattern operations, the tool just moves over in XY. All approaches and retracts are handled on the Link tab. The Retracts Dialog and other settings affect the moves between pattern instances just as they affect the moves before and after the original cutting. For more information on Patterns, see the *Mill* guide.

Spindle / Part Station
If your current MDD has more than one spindle or part station, you can choose which one to use. For more information on multi-task machining, see the *MTM* guide.

Output Port Machining Spine
When this option is selected, the curve calculated for the spine will be added to the part. This is helpful for troubleshootingunexpected results from toolpath generation or from using the Automatic spine setting in 5-Axis Porting.

3D Cutter Radius Compensation
See "3D CRC (3D Cutter Radius Compensation) " on page 53.

3D Material Only
With 3D Material Only, material can be shared across different operation types (such as Milling and Turning operations) or different CS's, and 3D milling can use rest material. It also provides Machine Sim with a more accurate picture. When a process dialog's **Solids** tab is bolded, it can use 3D Material Only.

Machining CS

You can use this drop-down list to choose which coordinate system (CS) will be the default CS to use for 5-Axis functionality that expects or requires a specific machining CS. If only one CS is defined, then the drop-down list offers it as the only choice.

Coolant

Select whether to use coolant for this operation. For most MDDs, the only option is Flood.

Comment

Enter text here that you want to appear as a comment in the Operation Data of the output operation.

Rotary Controls

The Rotary Duplicate control lets you duplicate the toolpath around the rotary axis (e.g. A, B, or C depending on your machine setup). This is an operation rotary position or rotary repeat. This functionality is fully described in the Mill manual if you need more detailed information, Simply put, you can define a 3, 4 or 5-axis toolpath complete with Entry/Exit settings as defined on the Link tab, and then duplicate that around a part at an angle. An example of this use would be defining an operation that finishes the base of a turbine. Rather than creating the same operation numerous times you can simply enter that you want to repeat this same toolpath one or more times. For example, repeat 9 times at 36 degrees increments. Note that the G-code output for repeated operations using the Rotary Duplicate option will always be in longhand (no subroutine) format for 5-Axis operations.

The moves between iterations is assumed to be “clear” and has no gouge protection. This move between iterations is dependent on MDD settings, especially for rotary head machines. For rotary table machines it is assumed that the tool is clear in Z. This is a bit more complex than a 4-axis rotary duplicate as there is a second rotary axis moving to the start point of the next iteration.

As an example, consider a BC rotary table machine, using rotary repeats around C. Unlike 4-axis rotations, the machine is also moving B to position the tool at the next start point. The Z clearance value needed to stay clear of the part may be surprising. When using this Rotary Duplicate function, please be aware of your MDD settings for clearance moves and the operation’s Last Exit Z move position for the connect move to the repeated operation.

Restore Defaults

Clicking this button resets the values of all fields to their initial system-supplied default values. This can be useful when you have modified several parameter values and you are unsure which parameter is affecting the toolpath calculations.

> *Warning:* The Restore Defaults button restores all settings of all controls across all tabs.

Type of Machining

The 5-Axis Parameters dialog box is typically in a generic interface appropriate to all types of machining. This is the case when the pull-down menu reads General. In addition to this, you can change the interface to be focused on a specific type of machining, such as: Projection, Swarf Milling, Cavity Tilt Curve, Cylinder Head, Electrode Machining 4+1 Axis, Impeller Floor Surface, Impeller Roughing, Impeller Blade Swarf Finishing, Turbo Shaft Finishing 4+1 Axis, and Drilling Options. Each of these items will change the interface to have fewer tabs. This will help you focus on the controls that are relevant to that kind of machining. Each of these items will set defaults within the system that are geared towards the specific type of machining, even for parameters that are not displayed in the dialog for the specific type of machining. It is recommended that you click the Restore Defaults button when changing between the machining types.

You may also use these specialized interfaces to help you set the defaults aimed at a type of machining and then switch over to the General interface. This can help you learn what parameter settings are important for a particular type of machining. For detailed information, refer to the following:

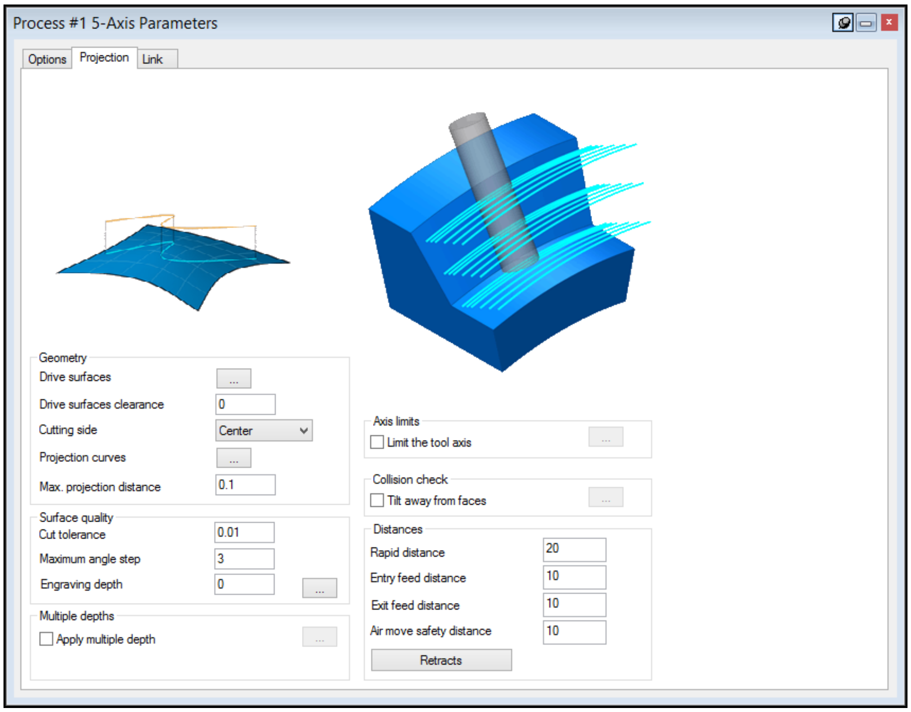
- Projection below
- “Swarf Milling” on page 43
- “Cavity Tilt Curve” on page 44
- “Cylinder Head” on page 45
- “Electrode Machining 4+1 Axis” on page 46
- “Impeller Floor Surface” on page 46
- “Impeller Roughing” on page 48
- “Impeller Blade Swarf Finishing” on page 49
- “Turbo Shaft Finishing 4+1 Axis” on page 50
- “Drilling Options” on page 50

General

When General is active, the 5-Axis Parameters dialog box has seven available tabs for defining 3, 4 and 5-axis machining. See the section for the specific tab for a description of its contents.

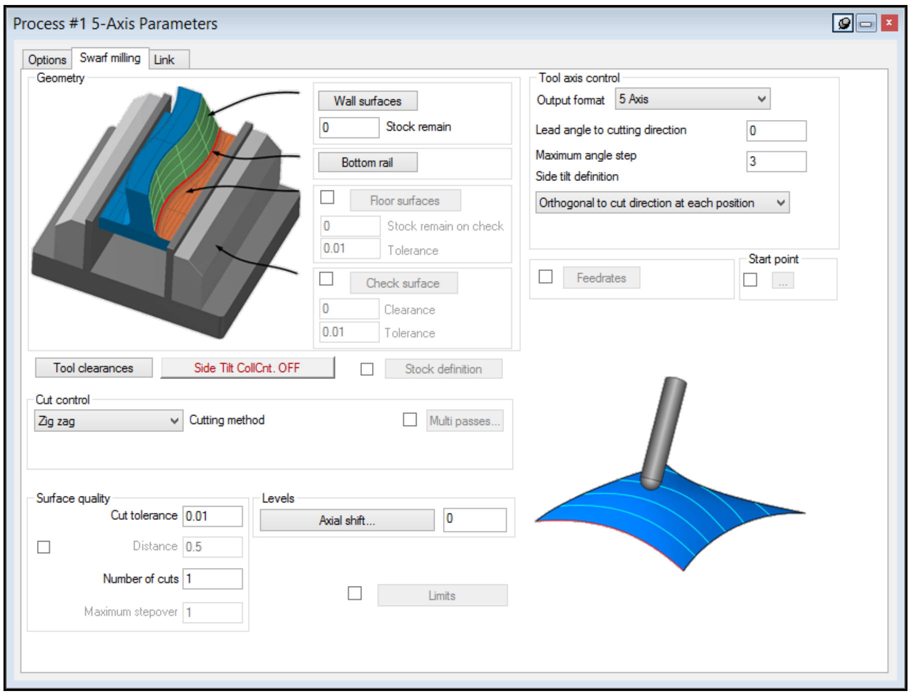
Projection

This interface is specialized towards projecting geometry onto a solid for machining. This machining option requires you to select drive surfaces and projection geometry. All of the controls for this machining are found in the general interface. The gouge check uses the tool shaft, front end of the holder and back end of the holder. The tool tip is not checked for gouges.



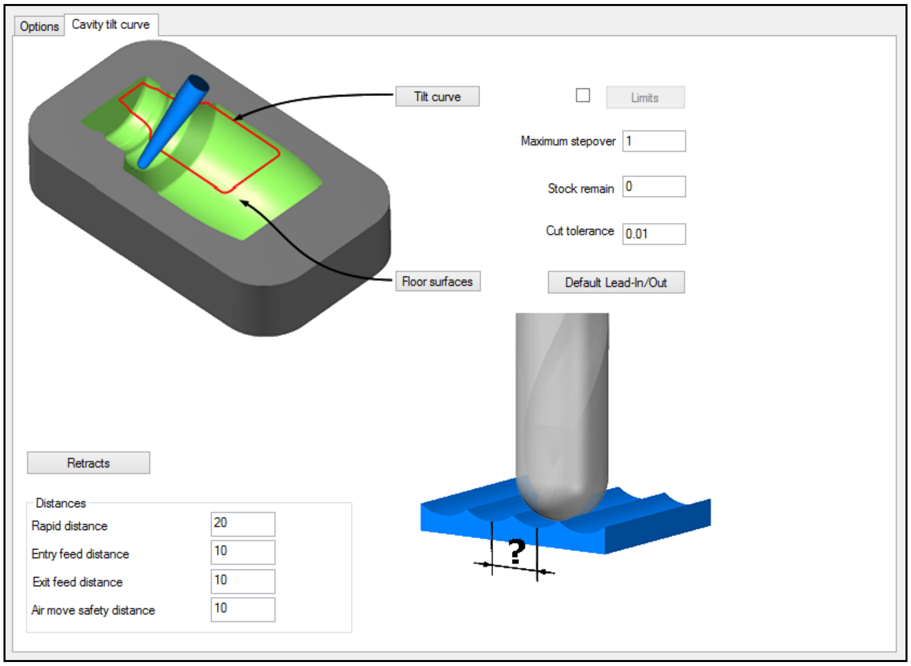
Swarf Milling

This interface is specialized towards swarf milling or cutting with the side of the tool. You choose the walls to be machined, a bottom edge of the wall, floor faces, and check surfaces. The gouge check uses the entire tool definition to check against gouges. Unlike most other focused machining types, Swarf milling gives you full control over the items in the Link tab (for more information, see "Link tab" on page 238).



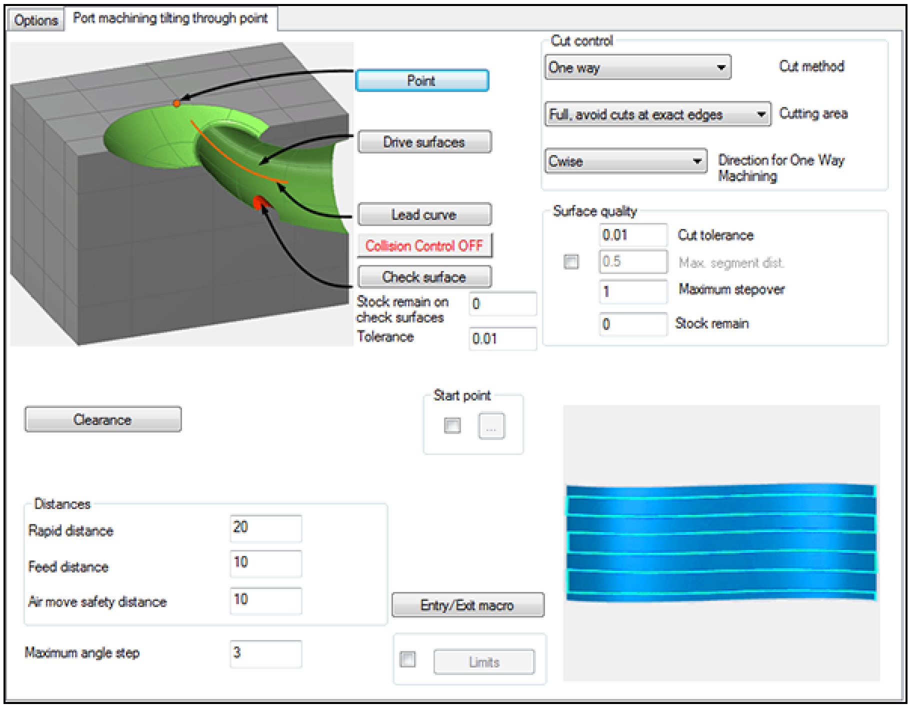
Cavity Tilt Curve

This interface is specialized towards machining cavities. You choose the surfaces to be machined and a tilt curve.



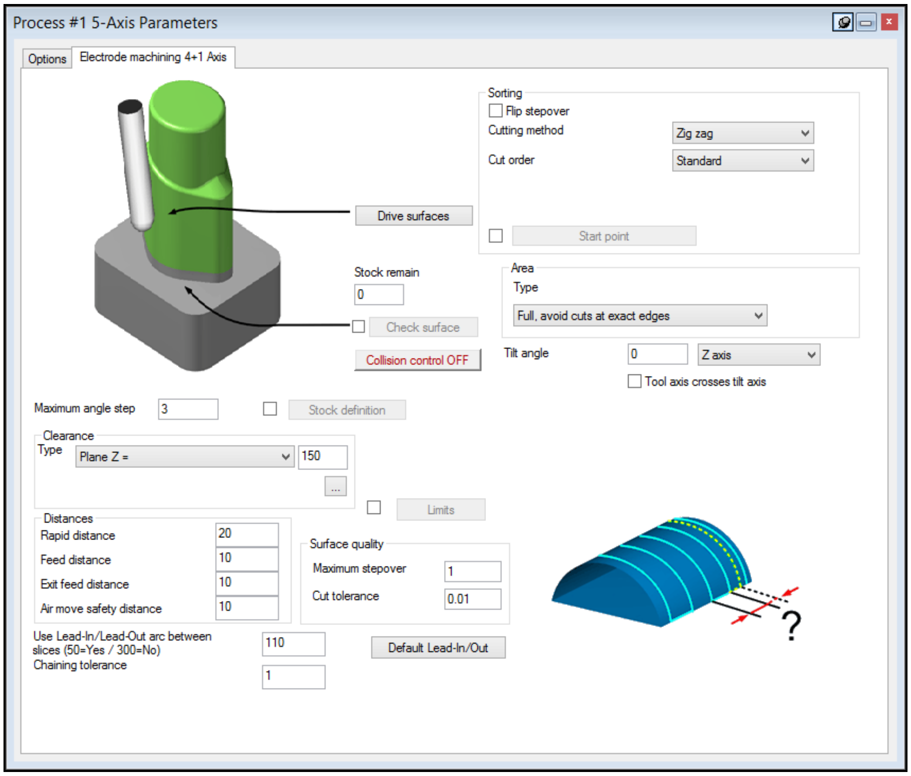
Cylinder Head

This interface is specialized towards port machining using a point through which the tool tilts. You choose the tilt point, the drive surfaces, a curve to follow, and check surfaces. The gouge check uses the entire tool definition to check against gouges.



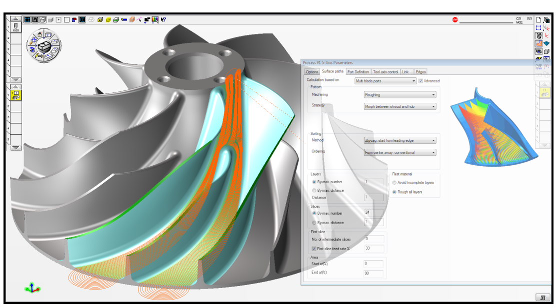
Electrode Machining 4+1 Axis

This interface is specialized towards machining electrodes. You choose the drive surfaces and check surfaces. The gouge check uses the tool shaft and the front and back ends of the holder to check for gouges against the drive surfaces. The tip and shaft are checked against check surfaces.



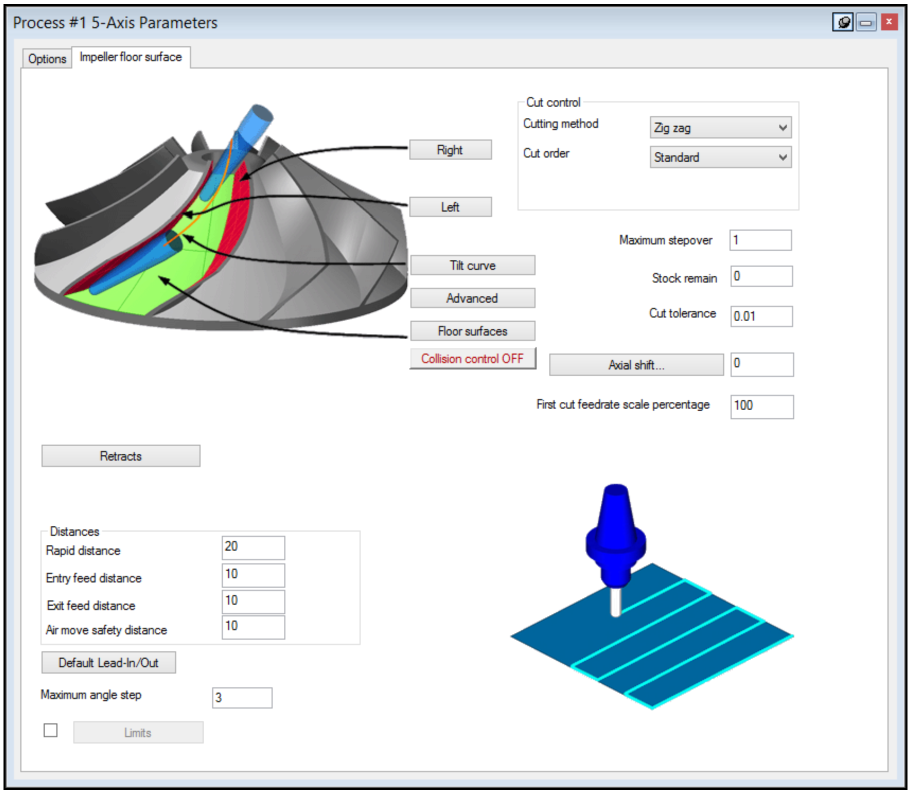
Impeller Floor Surface

5-Axis Impeller Machining Options Compared to 5-Axis MultiBlade



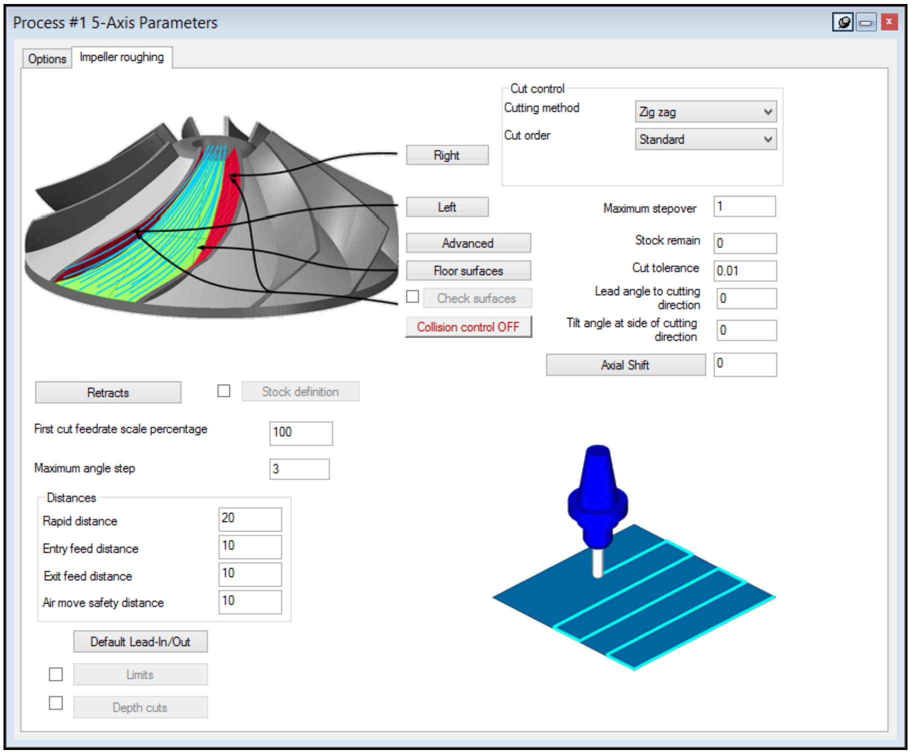
Although the base 5-Axis product includes options for impeller machining, the preferred solution is the 5-Axis MultiBlade product option (or 5-Axis MultiBlade Level 2 for finest control of all aspects of impeller machining). Because MultiBlade is only for impellers and blisks, it automatically detects and leverages radial symmetry, accommodates blades and splitters of any curvature, and provides options and controls that are specific to impellers, such as special settings for leading and trailing edges.

The Impeller Floor Surface interface is specialized towards finishing the floors of an impeller. You choose left and right blade walls, a tilt curve to follow, and the floor surfaces. Additionally, you can also choose to machine around the impeller blades, or only between the blades, using the Advanced controls. The gouge check uses the entire tool definition to check against gouges.



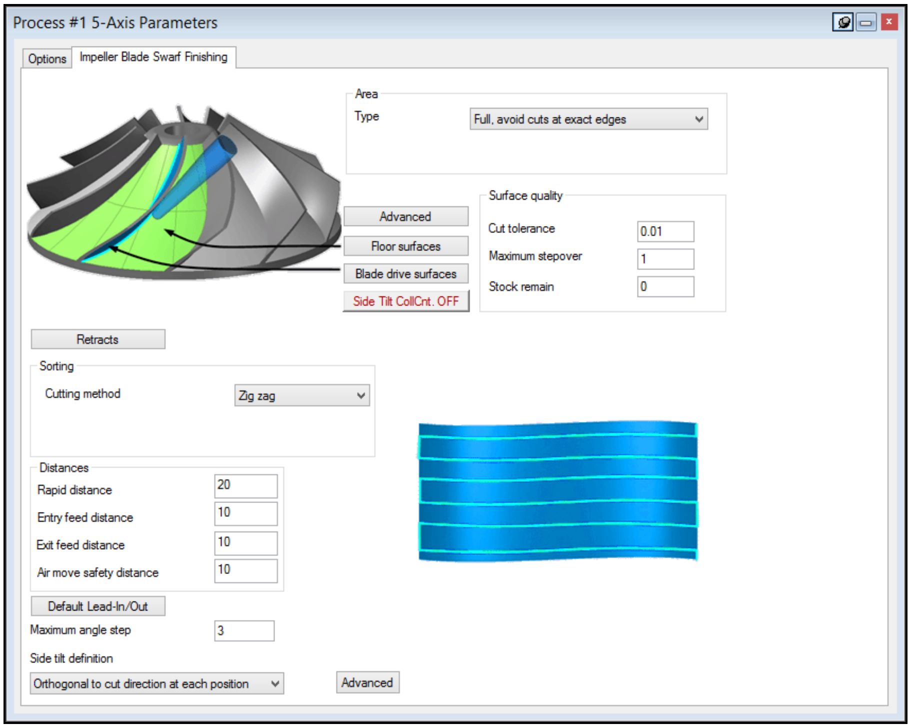
Impeller Roughing

This interface is specialized towards roughing out impellers. You choose left and right blade walls, the floor surfaces, and check surfaces; additionally, you can choose to machine around the impeller blades or only between the blades using the Advanced controls. The gouge check uses the tool tip and shaft to check for gouges against the drive surfaces. The entire tool (tip, shaft, front and back ends of the holder) is checked against check surfaces.

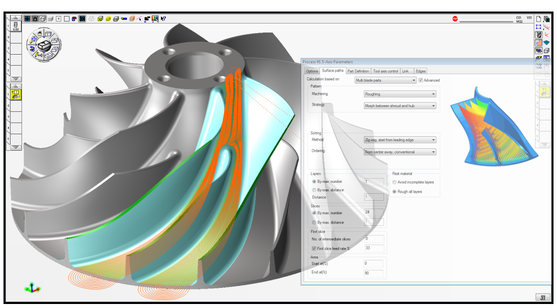


Impeller Blade Swarf Finishing

This interface is specialized towards finishing the walls of impeller blades. You choose the floor surfaces, drive surfaces, and check surfaces; additionally, you can choose to machine around the impeller blades or only one side of the blades using the Advanced controls. The gouge check uses the tool shaft and the front and back ends of the holder to check against gouges. This option does not check the tool tip for gouges.



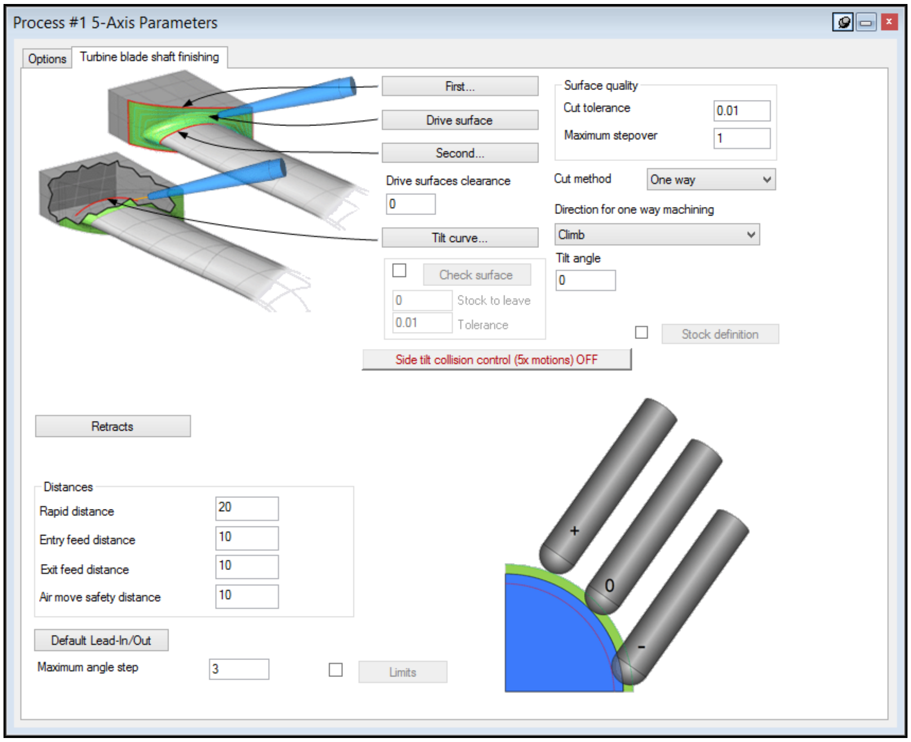
5-Axis Impeller Machining Options Compared to 5-Axis MultiBlade



Although the base 5-Axis product includes options for impeller machining, the preferred solution is the 5-Axis MultiBlade product option (or 5-Axis MultiBlade Level 2 for finest control of all aspects of impeller machining). Because MultiBlade is only for impellers and blisks, it automatically detects and leverages radial symmetry, accommodates blades and splitters of any curvature, and provides options and controls that are specific to impellers, such as special settings for leading and trailing edges.

Turbo Shaft Finishing 4+1 Axis

This interface is specialized towards finishing the shaft of a turbine blade. You choose the edges between which to machine and the drive surfaces that are to be machined. Additionally, you can select a tilt curve to follow. The gouge check uses the tool shaft and the front and back ends of the holder to check for gouges against the drive surfaces. The tip and shaft are checked against check surfaces.



Drilling Options

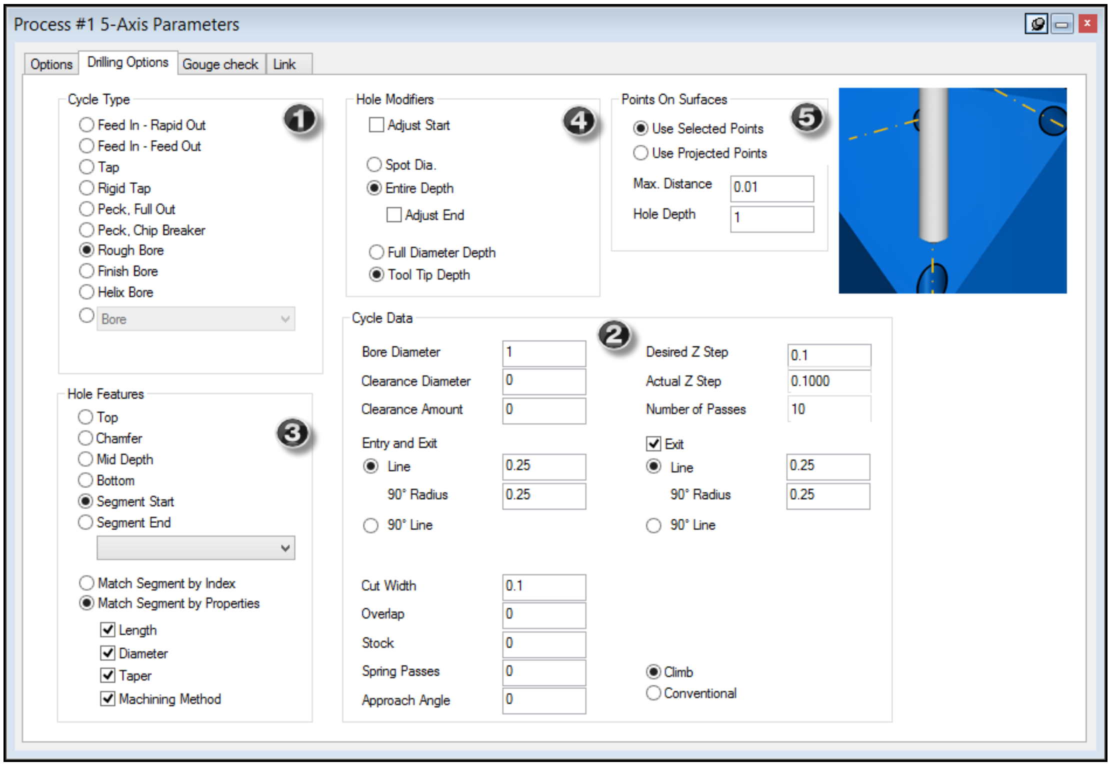
This interface allows you to perform drilling operations in a 5-axis context. Unlike most other focused machining types, Drilling gives you full control over items in three other tabs (for more information, see Tool Axis Control tab, Gouge Check tab, and “Link tab” on page 238).

The items in the Drilling Options tab are similar to the Holes process in Mill, and are grouped into five areas:

1. Cycle Type, next
2. “Cycle Data” on page 51
3. “Hole Features” on page 52
4. “Hole Modifiers” on page 52

5. “Points on Surfaces” on page 53

Except for Points on Surfaces, all controls offered on the **Drilling Options** page are discussed in much further detail in the *Mill* guide, under "Holes Process".



Cycle Type

The selections made here determine the cycle that the drill will use to make its entry and exit moves. The choices are exactly the same as for the **Holes** process dialog. For complete information, see the *Mill* guide, under "Drill tab".

Additionally, if you have a custom Post Processor that supports additional drill cycles, you may use a pop-up menu for Custom Cycle options, including variable-peck cycles.

Please Note: Tap cycles and custom drill cycles require a post-processor modification (available at no additional cost). If you use any of these cycles with a post that does not support them, you will receive an error.

Cycle Data

Depending on the option chosen for Cycle Type, one or more of the following can be set:

Clearance:
Available only for the Peck, Full Out cycle type. Enter a value to specify the incremental distance away from the material where the tool will start its next peck.

Peck:
Available only for Peck cycle types. Enter a value to specify the depth the tool will plunge on each peck.

Retract:
Available only for the Peck, Chip Breaker cycle type. Enter a value to specify how far the tool will retract after each peck.

Tap %:
Available only for the Tap cycle type. Enter a value to specify the percentage of the feedrate that will be used on the tapping cycle.

Dwell:
Available for all cycle types other than Bore. Enter a value to specify the length of time the drill will pause at the hole bottom with the spindle on.

Bore parameters:
The parameters offered in **Cycle Data** section for the Rough Bore, Finish Bore, and Helix Bore cycles are the same as the parameters offered in the Mill **Holes** process dialog for the corresponding cycles. For complete information, see the *Mill* guide, "Holes Process" > "Bore Tab".

Hole Features

A complex hole feature might have several diameters, each with different Z values. You can choose whether to apply the process values to the Top of the hole, the Chamfer depth, the Mid Depth, or the Bottom of the hole.

For information on Top / Chamfer / ... / Segment Start / Segment End, see the *Mill* guide, "Holes Process" > "Hole Feature Tab" > Settings, Options, and Parameters".

The same section in the *Mill* guide also discusses segment-matching (Match Segment by Index and Match Segment by Properties) and provides examples.

Hole Modifiers

Adjust Start:
If this checkbox is not selected, or if the value is `0`, the tool tip rapids to and from the retraction points without any incremental shift. Enter a positive value if you want to move the tool tip to an Entry Clearance Plane above the R Level.

Spot Diameter / Entire Depth:
Choose one of these options to specify whether the tool is spot-drilling or plunging to the entire depth.

For Entire Depth, you can optionally enter a value for Adjust End to specify the distance you want to tool tip to be from the bottom of the hole (an incremental adjustment to Hole Depth). You also choose whether Entire Depth refers to the depth of the tool's full diameter or just the depth of the tool tip.

Points on Surfaces

Use Selected Points / Use Projected Points:
Choose one of these options to specify whether the drilling operation is based on the selected points or, instead, on points projected from them onto the workpiece. In both cases, the orientation of the tool axis is determined by the surface normal.

Max Distance:
This value specifies the farthest distance that a Selected Point can be located, or that a Projected Point can be projected, and still be considered in the calculation. Any point farther away than the specified distance (measured along the normal to the surface) is ignored.

Hole Depth:
This value specifies the absolute depth position of the hole. This represents the maximum infeed into the surface.

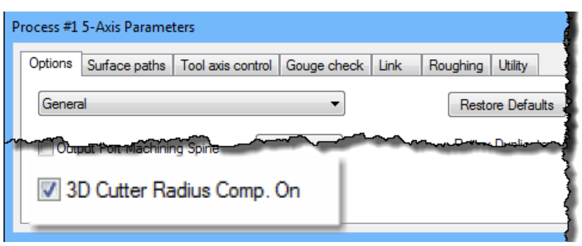
3D CRC (3D Cutter Radius Compensation)

For 5-Axis and related modules (MultiBlade and Port Machining), you can now use a 3D version of CRC (cutter radius compensation).

> **Please Note:** Because 3D CRC appends new toolpath data to the operation, a small post processor change is required.

To enable 3D CRC

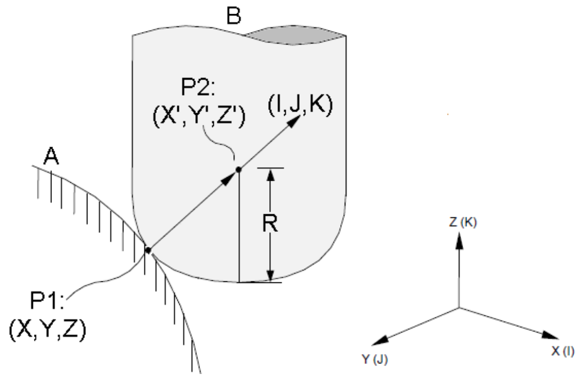
In the process dialog, **Options** tab, bottom left, select the checkbox 3D Cutter Radius Comp. On.



What Is 3D CRC?

3D CRC is three-dimensional tool compensation for straight-line blocks. Apart from the XYZ coordinates of the straight-line end point, these blocks must also contain the components of the surface-normal vector.

A *unit vector* is a mathematical quantity with a magnitude (length) of `1` and a direction. The direction of a surface normal vector is determined by the components `I,J,K` (or, in the case of Heidenhain, `Nx,Ny,Nz`) such that $I^2 + J^2 + K^2 = 1$.



A: Workpiece

B: Tool

P1: Programmed coordinates

P2: Tool center coordinates

R: Radius of tool

I,J,K: plane-normal vector: the direction from P1 to P2

As shown in the diagram above, the tool is moved through the tool radius R in the plane-normal direction of (I,J,K) from the program coordinates (X,Y,Z) to the offset tool center coordinates (X',Y',Z').

Unlike two-dimensional CRC, which generates vectors perpendicular to the direction of `I,J,K`, 3D CRC generates vectors *in the direction of* `I,J,K`. The vectors are generated at the ending point of that block.

Applications

3D CRC allows you to use tools with dimensions that do not correspond with the dimensions calculated by the CAD system, just as you might use regular 2D CRC.

For face milling: 3D CRC provides compensation of the milling machine geometry in the direction of the surface-normal vector. Cutting is usually with the end face of the tool.

For peripheral milling: 3D CRC provides compensation of the mill radius perpendicular to the direction of movement and perpendicular to the tool direction. Cutting is usually with the lateral surface (side) of the tool.

Post Changes

The changes that need to be made to a post to support 3D CRC are straightforward:

- Add the `Output3DCRCNormal` command after the `MoveAllAxes` command (or its equivalent) inside the Line/Rapid/Arc-segmentation toolpath sub-routines.
  For example:

```
TPRAPIDFEAT:

   ...

   SEQC ABSORINC CRCC RAPIDC MOVEALLAXES OUTPUT3DCRCNORMAL CRCOFFSETC SPEEDC
   TOOLOFFSETC EOL

   ...
```

```
RETURN
```

Notice that the `CRCC` and `CRCOffsetC` commands are used. These are existing 2D CRC commands that have been updated to work with 3D CRC as well. For example, the `CRCC` command will output `G141` for Haas 3D CRC-On if the current operation is using 3D CRC, or it will output `G41`/`G42` if it is regular 2D CRC.

- If you need to format the surface normal vectors in a way not supported by the input fields in the Form dialog, then you can use the `3DCRCNormalI#` commands.
  For example:

```
3DCRCSURFACENORMALS:

  IF 3DCRC? AND NOT CRCOFF?

  'I' OUTPUT('I', 3DCRCNORMALI#) 'J' OUTPUT('J', 3DCRCNORMALJ#) 'K' OUTPUT('K',
  3DCRCNORMALK#)

  END

RETURN
```

Then you can call the `3DCRCSurfaceNormals` subroutine instead of `Output3DCRCNormal` within the toolpath subroutines.

Form Changes

- To output 3D CRC, you must enable it within the post processor's Form: Program Options > Options > Operation Support tab, 3D CRC checkbox.
- To specify the 3D CRC output G-codes, edit the Form: Offsets and Workplanes > Tool Offsets > 3D CRC section.
- It is recommended to set the post's Segmentation Tolerance to `0`. This is found in Form > Movement > Rotary > Rotary Toolpath Options.
  With a setting of `0`, the post will not introduce any additional tool path segmentation, which is important while using 3D CRC with 5-Axis.
- If the post introduces more segmentation, GibbsCAM must interpolate the 3D CRC surface normal vector between the segmented points. This interpolation is a best guess at what the CNC needs to machine a smooth toolpath, and so adding more segmentation increases the uncertainty in the accuracy of the interpolated surface-normal vector.

3D CRC Support

Two types of 3D CRC output are common among machine controllers, but only one of them is supported by the 5-axis process in the current release. The two types can be described as *automatic* 3D CRC and **explicit** 3D CRC.

Automatic 3D CRC (Unsupported)

Automatic 3D CRC output type means the CNC automatically calculates the tool compensation plane that is perpendicular to the tool vector. This output type is useful in limited 5-axis machining situations, specifically in Swarf milling where the side of the tool makes contact with the material and follows a guide curve. Automatic 3D CRC output is similar to 2D CRC where a `CRC-left` or `CRC-right` G-code indicates the cutter side relative to the cutting direction.

In the current release, automatic 3D CRC output type is ***not*** supported. This is because the cutter side (`CRC-left` vs `CRC-right`) is not known and therefore cannot be output.

Explicit 3D CRC (Supported)

Explicit 3D CRC output type means the G-code block must output the three dimensional cutter compensation vector – the surface normal vector. This output type is used in 5-axis machining situations where a surface is used for machining, including Swarf milling. Explicit 3D CRC requires surface normal vectors to be output, but it also does not use `CRC-left` and `CRC-right` G-codes to dictate cutter side compensation. Instead, the CNC uses G-code to turn explicit 3D CRC on or off.

Starting in GibbsCAM 13, the explicit 3D CRC output type ***is*** supported with 5-Axis 3D CRC.

Example CNC Output Formats (Explicit 3D CRC)

Below are some sample output codes for various machine controllers.

Fanuc/Mazak

- `G41` (`G42`) `X Y Z I J K D`
- The G41 or G42 turns 3D CRC on. When G42 is used, the CNC interprets the surface normal I J K vector in the opposite direction. GibbsCAM will not indicate when a post should use G41 versus G42, and therefore the post should always output in G41 (or output G42 and the post can invert the I J K surface normal).
- The I J K parameters represent the surface normal vector.
- G41 and G42 are also used in regular 2D CRC. The difference on this G-code block is the introduction of the I J K parameters, indicating 3D CRC.

Haas

- `G141 X Y Z D I J K`
- `G141` turns 3D CRC on and `G142` is not used. `G40` cancels 2D and 3D CRC.
- The I J K parameters represent the surface normal vector.

Heidenhain TNC

- `LN X Y Z NX NY NZ TX TY TZ R0`
- `LN`, as opposed to `L`, represents a 3D CRC feed move – it turns on 3D CRC.
- The `NX NY NZ` parameters represent the surface normal vector.

Notes

All existing regular CRC commands have been updated to work in parallel with 3D CRC, if applicable. For example, the `CRCLeft?` command makes no sense for outputting 3D CRC surface normal vectors, because 3D CRC does not rely on cutter side G-codes (`G41` vs `G42`).

For this reason, if you use the `CRCLeft?` command with 3D CRC, and if 3D CRC is on (in other words, if `3DCRC?` returns `true`), then the `CRCLeft?` and `CRCRight?` commands will both return `true`.

5-Axis Operation Programming Tips

When you activate 3D Cutter Radius Comp. in the 5-axis process dialog, you need to consider the type of toolpath being generated and determine whether it is suitable for 3D CRC friendly. Here are some scenarios to consider:

- Make sure the toolpath has sufficient lead-in and lead-out moves when CRC is turned on and off; otherwise, it will gouge.
- If the machining surface has a sufficiently acute concave corner, then the tool will gouge the opposite surface while machining the current surface.

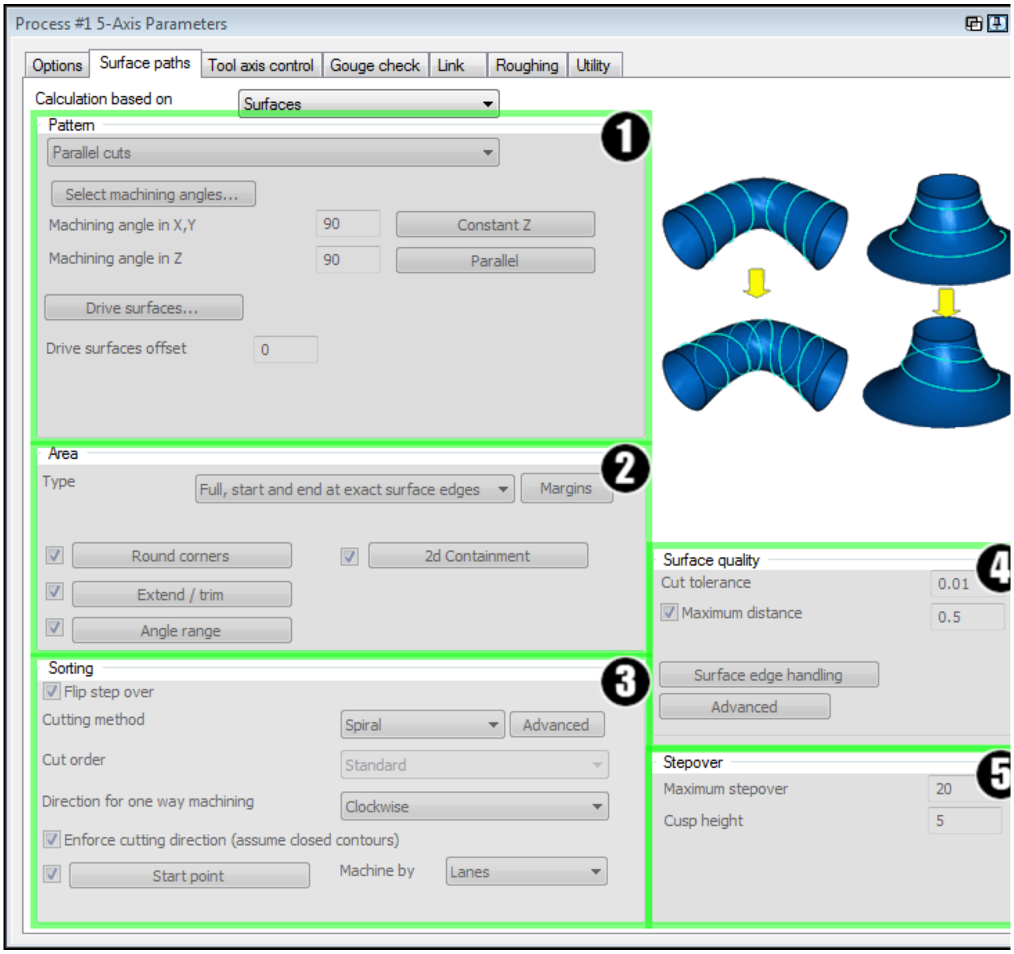
Surface Paths tab

The **Surface Paths** page (available only when the main **Options** pull-down choice is General) lets you specify how 5-Axis calculates toolpath:

- For calculation based on Surfaces, Triangle Mesh, or Wireframe, additional controls let you set pattern options, cut area options, cut sorting options, and surface quality options. For details, see below.
- For calculation based on Swarf Machining, additional controls let you select surfaces and curves, choose machining options, set surface quality values, and specify other swarf-specific settings. For details, see "Calculation Strategy: Swarf Machining" on page 93.
- For calculation based on Multi-blade Machining or Port Machining, the tab presents a different interface streamlined for that calculation. For details, see the *5-Axis-MultiBlade* guide or the *5-Axis Porting* guide.
- For calculation based on Geodesic Machining, additional controls let you set pattern options, select guide curves, specify containment type, and set parameters for area, sorting, surface quality, and stepover. For details, see "Calculation Strategy: Geodesic" on page 104.
- For calculation based on Deburring, the tab presents a different interface streamlined for that calculation: For details, see "Calculation Strategy: Deburring" on page 108.

Where applicable, the **Surface Paths** page also allows you to select surfaces and/or edges (for driving, orientation, or part definition); to choose options for sorting, start point, and shifting; and to set stepover parameters.

When toolpath calculation is based on Surfaces, Triangle Mesh, or Wireframe, the **Surface Paths** page offers five groups of controls:

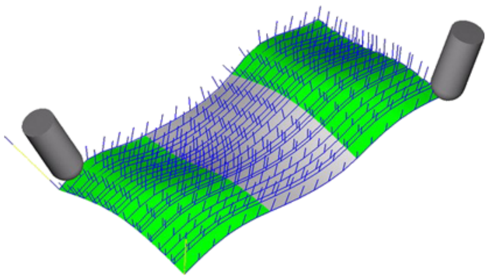
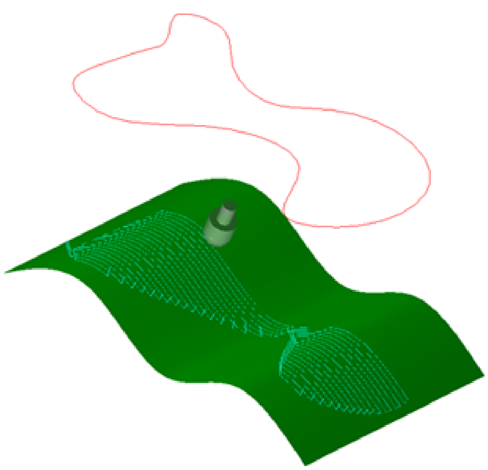


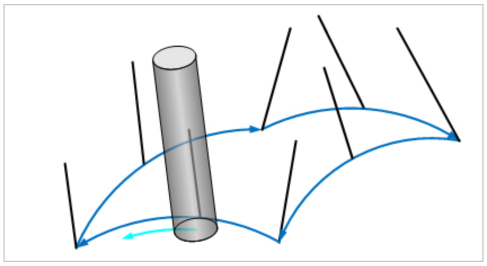
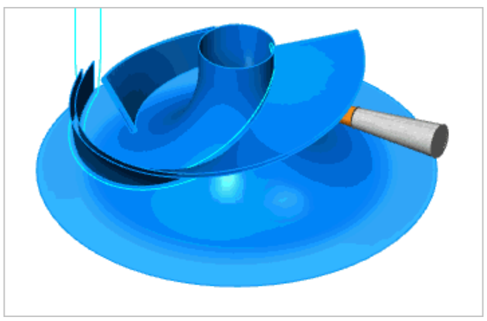
Controls for calculation based on Surfaces, Triangle Mesh, or Wireframe

1. “Pattern Settings” on page 61
2. “ Area ” on page 113
3. “Sorting Settings” on page 129
4. “Surface Quality” on page 147
5. “Stepover” on page 151

Calculation Strategies

In the Surface Paths tab, the first option, Calculation based on, affects options displayed in this tab and others.
The GibbsCAM 5-Axis module offers a variety of options for calculating and generating toolpath.

Calculation	Strategies
Surfaces calculation generates toolpath points on parametric surfaces. For details, see Surface Machining.  To use Surfaces: After you choose a cut pattern – see “Pattern Settings (Calculation based on Surfaces)” on page 62 – you select drive surfaces and specify values for cutting area, sorting, surface quality, and stepover.	Cut strategy depends on Pattern: Parallel cuts (Calculation based on Surfaces) Perpendicular to curve Morph between two curves Parallel to curve Project curves Morph between two surfaces Parallel to surface
Triangle Mesh calculation generates toolpath points by dropping a collision-free contour from a set direction onto the machining surface. When the tool tilts, it rotates around a fixed contact point on the mesh.  To use Triangle Mesh: After you choose a cut pattern – see “Pattern Settings (Calculation based on Triangle Mesh)” on page 76 – you select machining surfaces and then specify values	Cut strategy depends on Pattern: Rough Parallel cuts Project curves Constant Z Constant cusp Flatlands Pencil Projection

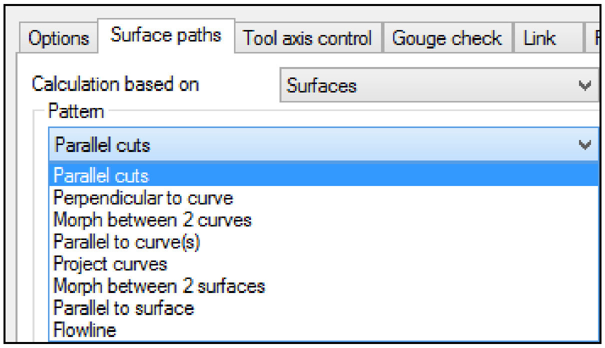
Calculation	Strategies
for cutting area, sorting, surface quality, and stepover.	
Wireframe calculation generates a single toolpath along a drive curve, without machining surfaces, interpolating between orientations set by user-selected lines.  To use Wireframe: You select drive curves and orientation lines – see “Pattern Settings (Calculation based on Wireframe)” on page 91 – and then specify values for cutting area, sorting, and surface quality.	
Swarf Machining calculation produces the target surface with only one cut, using the whole flute length of the tool.  To use Swarf Machining:If you choose the Automatic strategy (recommended), you define the part's surfaces and upper/lower curves, choose machining approaches and start point type, specify values for surface quality and additional settings in the Multi Cuts and Corners pages. For more information, see “Calculation Strategy: Swarf Machining” on page 93	When calculation is based on Swarf Machining, the 5-Axis user interface changes in several ways. Surface Paths tab for Swarf Machining Tool Axis Control tab for Swarf Machining Gouge Check tab for Swarf Machining Multi Cuts tab Corners tab

Pattern Settings

The first thing to do is determine what type of toolpath calculation strategy to use and then which type of machining strategy to use.

For calculation based on Surfaces or Triangle Mesh, the Pattern list determines the type of cut:

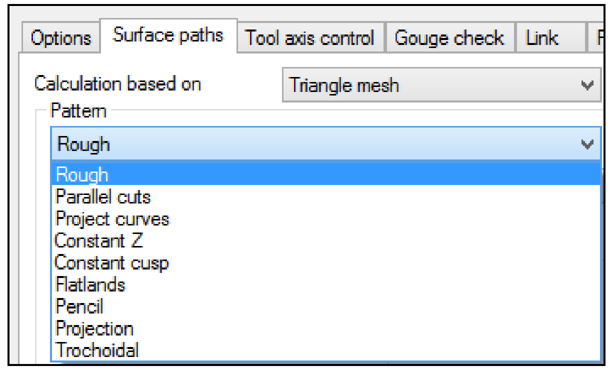
Calculation based on Surfaces



How it works. For machining strategies with calculation based on Surfaces: You choose a Pattern, select the faces to be cut (Drive surfaces), and specify values for Drive surface clearance and other settings.

For complete information, see "Pattern Settings (Calculation based on Surfaces)" on page 62.

Calculation based on Triangle Mesh



How it works. For machining strategies with calculation based on Triangle Mesh: You choose a Pattern, select the faces to be cut (Machining surfaces) and specify values for Z height (Heights), tool offset (Offset), and other settings.

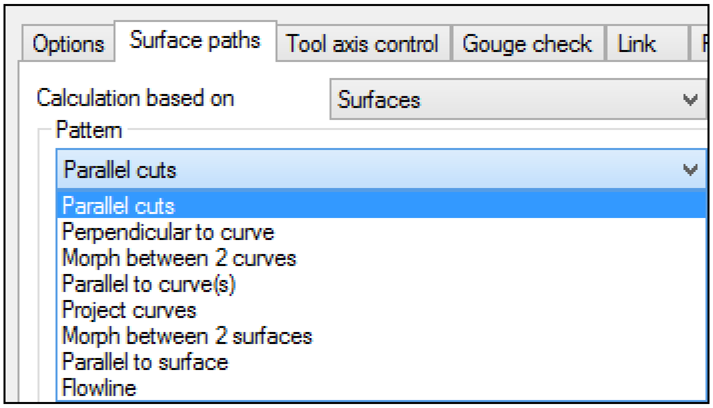
For complete information, see "Pattern Settings (Calculation based on Triangle Mesh)" on page 76.

For machining strategies with calculation based on Wireframe: You select the curves to be cut (Drive curves), select Orientation lines, and then specify other values and settings. For complete information, see "Pattern Settings (Calculation based on Wireframe)" on page 91.

For machining strategies with calculation based on Swarf Machining, Pattern is replaced by Swarfing options. See "Calculation Strategy: Swarf Machining" on page 93.

Pattern Settings (Calculation based on Surfaces)

In the Surface paths tab, when the calculation is based on Surfaces, several different machining strategies are available. The Pattern list determines the type of cut:



- Parallel cuts (Calculation based on Surfaces)
- “Perpendicular to curve” on page 66
- “Morph between two curves” on page 66
- “Parallel to curve” on page 67
- “Project curves” on page 68
- “Morph between two surfaces” on page 69
- “Parallel to surface” on page 70
- “Flowline” on page 71

How it works. After you choose the type of cut (Pattern), you then select the faces to be cut, known as the *drive surfaces* (see “Drive Surfaces” on page 74) and optionally specify an offset value (see “Drive surface clearance” on page 75) and other settings. All pattern types except Parallel cuts require you to select one or more surfaces or curves in addition to the drive surfaces.

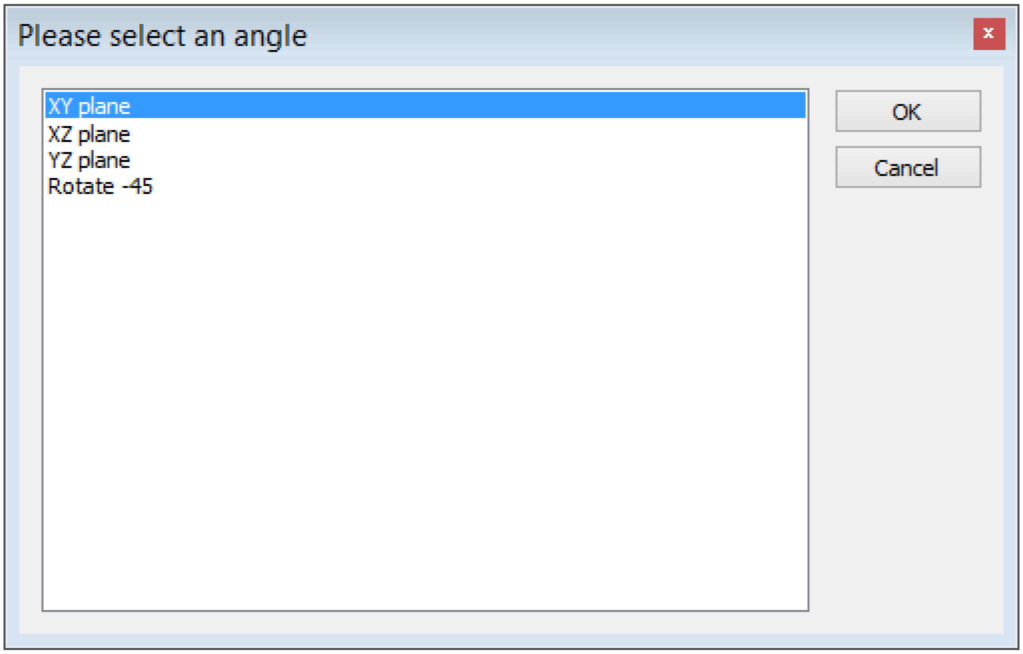
Parallel cuts (Calculation based on Surfaces)

The Parallel cuts pattern will create toolpaths that are parallel to each other. The direction of the cuts is defined by the two angles: Machining angle in X,Y and Machining angle in Z. The distance between two neighboring cuts is the *stepover* (see “Stepover” on page 151). Once your parameters are set, define the area to be machined using the Drive surfaces options (see “Drive Surfaces” on page 74). See “Parallel Cut Examples” on page 64 for samples of how the Machining angle settings are used.

> ⓘ Imagine slicing an apple: You can slice it with a knife parallel from the top down or from the left side to the right side. The pictures in the dialog symbolize how to set the desired cutting direction using the angles.

Select machining angles

This button opens a dialog that allows you to choose one of the coordinate systems in the part file to set the machining angles. Selecting a CS and clicking OK will automatically fill in the Machining angle in X,Y and Machining angle in Z fields.



Machining angle in X, Y

This is the angle of the toolpath, referenced to the XY plane. An angle of `0` degrees will produce toolpath that is parallel to the Y axis where an angle of `90` degrees will produce toolpath that is parallel to the X axis. Any value between `-360` and `360` is valid.

Machining angle in Z

This item controls the toolpath pattern relative to the Z axis. An angle of `90` degrees is the default for parallel machining, which results in parallel passes that are orthogonal to a virtual line rotated `90` degrees from the Z axis. An angle of `0` degrees is the default for Constant Z, which results in Z slices that are orthogonal to the Z axis (a virtual line rotated `0` degrees from the Z axis).

Constant Z

Clicking this button will disable the Machining angle in X, Y parameter and create cuts parallel to Z.

Parallel

Clicking this button will enable both the Machining angle in X,Y and Machining angle in Zoptions.

Drive surfaces

See “Drive Surfaces” on page 74.

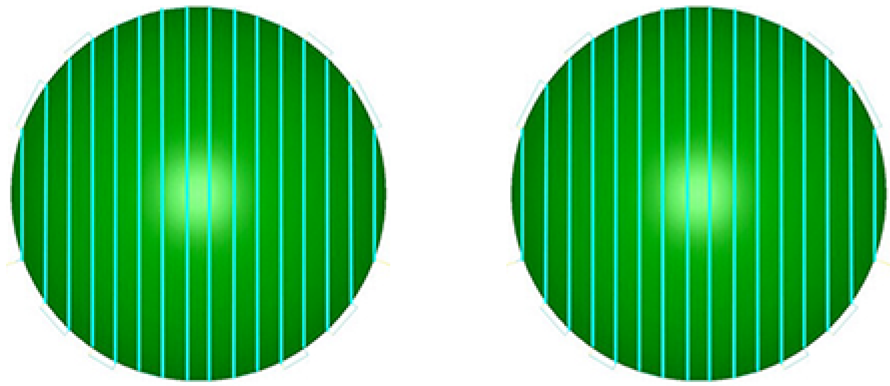
Drive surfaces clearance

See “Drive surface clearance” on page 75.

Parallel Cut Examples

Cuts parallel to the Y axis

Setting the machining angle in Z to `90°` and the angle in XY to `0` results in toolpath parallel to Y axis with a constant X stepover.

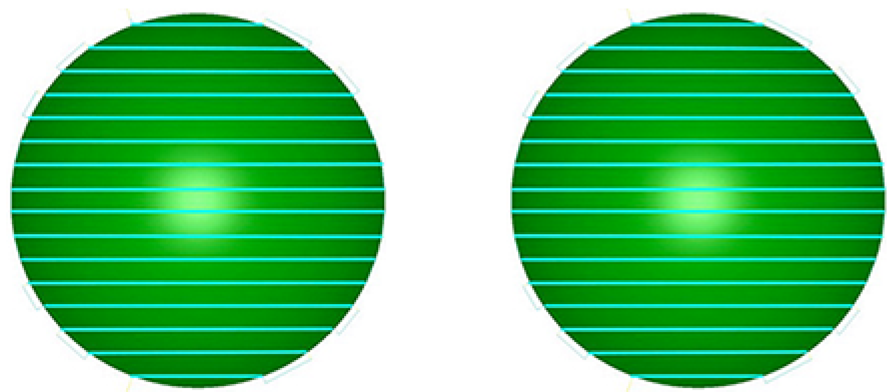


Cuts Parallel to the Y Axis

The file `Pattern - Parallel Cuts.vnc` and shows an example of this toolpath.

Cuts parallel to the X axis

Setting the Machining angle in Z as well as the Machining angle in X, Y to `90` degrees creates toolpath parallel to X axis with a constant Y stepover.

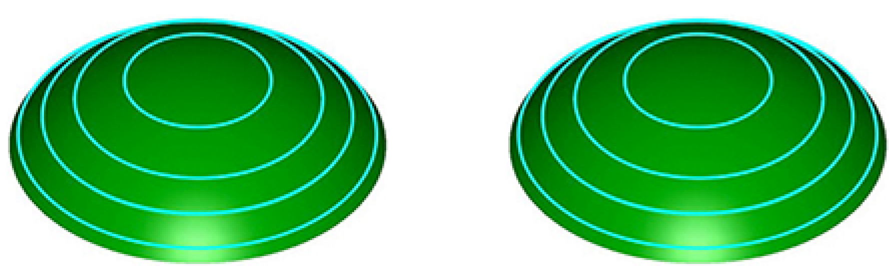


Cuts Parallel to the X Axis

The file `Pattern - Parallel Cuts.vnc` and shows an example of this toolpath.

Cuts parallel to the Z axis

To get Z constant cuts enter a Machining angle in Z of `0` degrees or click the Constant Z button. Now the Machining angle in X, Y blanks out because you are no longer working in that plane. In this example Machining angle in Z and Machining angle in X, Y are set to `0` degrees to create circular toolpath with a constant Z distance.



Cuts Parallel to the Z Axis

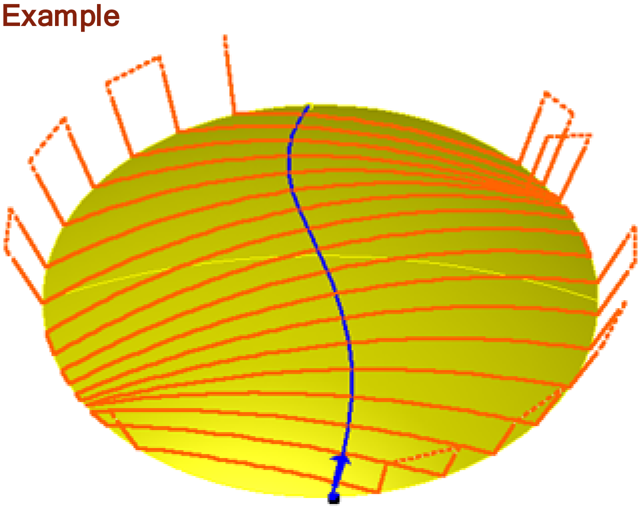
Perpendicular to curve

The Perpendicular to curve pattern will generate toolpath orthogonal to a leading curve. This means that when your selected curve is not a straight line the cuts are not parallel to each other.

Click the Lead button to select the drive curves (geometry or edge of a solid) to be used as the guide. For more information on curve selection, see “Drive Curves” on page 76. You will also need to select one or more faces to define the machining area by clicking on the Drive Surfaces button (see “Drive Surfaces” on page 74). The distance between two neighboring cuts (at the crossing point of curve and toolpath) is the “Stepover” on page 151.

- The curve does not need to be located exactly on or above the surface. It can be placed anywhere in your part.
- If the selected curve bends too much then the toolpath can intersect itself. The quality of this Pattern type is only as good as the curve/surface selections.

Example



In this example you can see the leading curve and the generated toolpath. It is important that the toolpath at the edge of the drive surface do not cross each other. In this example the cuts come very close to each other but do not touch.

See the file `Pattern - Cuts Along Curve.vnc` for this example.

Morph between two curves

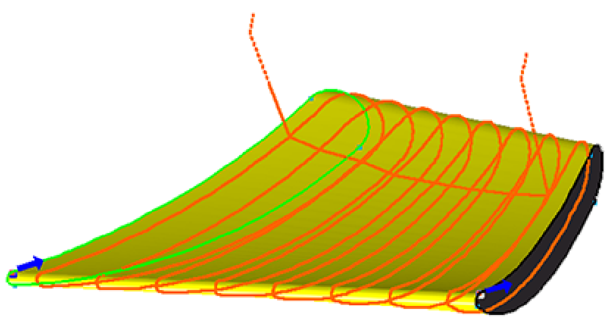
The Morph between 2 curves pattern will create a morphed toolpath between two leading curves. Morphed means that the generated toolpath is approximated between the tilt curves and evenly spread over the surface. This option is very suitable to machine steep areas for mold making. To use this option you need to select one or more faces to be machined and the two curves to morph between. Click the First and Second curve buttons to pick the curves from your geometry (see “Drive

Curves" on page 76). You may manually select the faces to be machined or you may use the "Drive Surfaces" on page 74 button to select and save the faces.

The more accurate the guide curves are to the real surface edges the better this function works. So the best result would be an exact curve on the edge of the drive surface. The number of the cuts is not clearly defined since you have a morphed toolpath and the distances between the cuts at the end of the faces are very different. If you want a certain amount of cuts, set the Area type to be " Determined by number of cuts " on page 116.

> When you set the cutting area to Full, start and end at exact surface edge you can set margins to the curves.

Example



In this example you see a cut out of a wing. The black and green lines on the edge of the sides are the first and second selected curves, respectively. As you can see the angle between these curves as well as the shape are totally different. The generated toolpath is approximated between the tilt curves and evenly spread at the thin and thick radius side. From this you can see the difference and advantage of this function over a parallel toolpath.

The example can be seen in the file `Pattern - Morph Between Two Curves.vnc`.

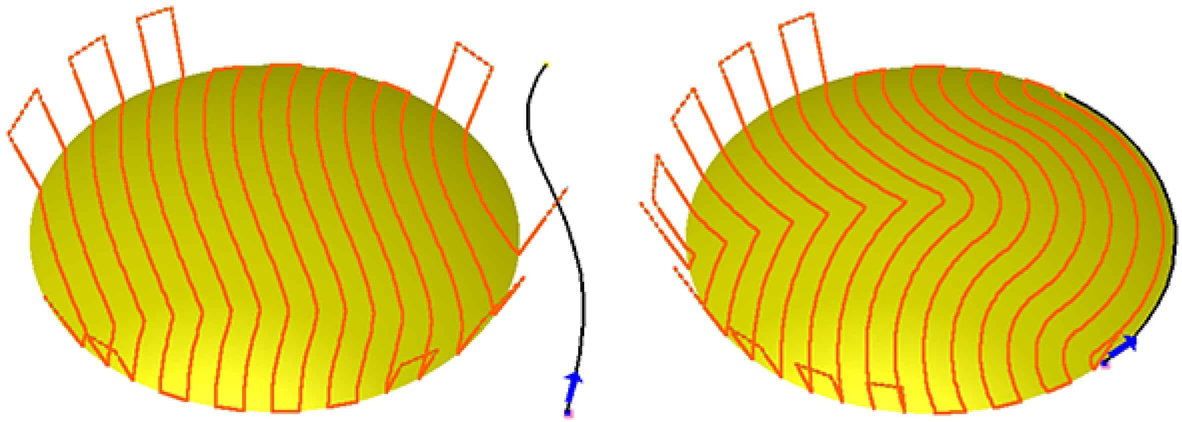
Parallel to curve

The Parallel to curve pattern will align the cut direction along a leading curve. The leading curve does not need to be located exactly on or above the surface; it can be placed anywhere in your part. The neighboring cuts are parallel to each other.

To use this option you need to select one or more faces (see "Drive Surfaces" on page 74) and a drive curve, selected using the Single Edge button (see "Drive Curves" on page 76).

Example

The following images are examples of operations using the Parallel to curve machining strategy. The example can be seen in the file `Pattern - Parallel To Curve.vnc`.



Project curves

The Project curves pattern will generate a single toolpath along a curve. It thus is a good choice for engraving. You will need to choose one or more faces (see “Drive Surfaces” on page 74) and projected geometry to act as a guide curve, which is selected with the Projection button (see “Drive Curves” on page 76). Ideally, the curve to be machined is located directly on the drive surface.

Projection direction

The default, Surface Normal, will project the curve in a direction normal to the surface. Other choices allow you to project instead in the direction parallel to one of the X, Y, or Z axes, or parallel to a line that you choose.

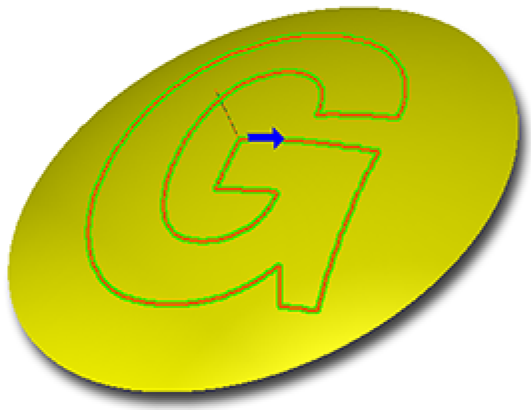
Max Projection Distance

This is the maximum amount the precess will attempt to project from the selected curves. If a letter “T” is centered one inch above a sphere and the max distance equals one, then only the vertical part of the “T” will be projected as the horizontal cross of the “T” needs to be projected more than one inch to reach the surface.

Type

Just as it does in other patterns, the Type menu allows you to choose from various cutting approaches. For details, see “Radial is generally used as a finishing operation. It is particularly effective on circular shaped components and shallow areas. ” on page 83 (for Triangle Mesh).

Example / Sample Part



Here you can see generated toolpath along a curve. To see this example, open the file `Pattern - Project Curve.vnc`.

Morph between two surfaces

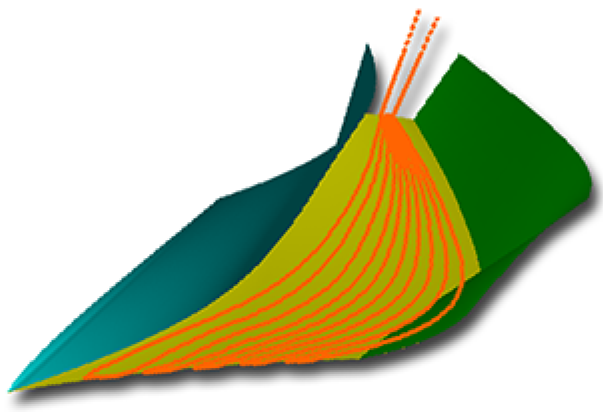
The Morph between 2 surfaces pattern will create a morphed toolpath on a drive surface. The drive surface is enclosed by two check surfaces. Morphed means that the generated toolpath is approximated between the check surfaces and evenly spread over the drive surface. Machining the floor of an impeller between two blades is a common use for this pattern type. To set up this process choose Morph between 2 surfaces from the drop down menu, then pick the first and the second check surface (the two surfaces surrounding the drive surface) by clicking the First and Second buttons (see “Check Surfaces” on page 75). The Advanced button lets you control the toolpath’s behavior between the check surfaces; see “Advanced button for Pattern” on page 72.

- The drive surface edge and the check surface edge must be coincident.
- To ensure the check surface is not violated due to tool tilting it is important to activate gouge checking.

A big advantage to Morph between 2 surfaces is the possibility to compensate the tool to the drive surface and the check surface in the left and right corners of the work piece. This is the concept of “margins”. When you work with margins, the tool must be a ball endmill and “Calculation Based on Tool Center” on page 294 must be enabled; see “Utility tab” on page 291. Also when you work with margins the value should be your tool radius or bigger. A smaller value would destroy the faces. For an example, see “Stepover” on page 151.

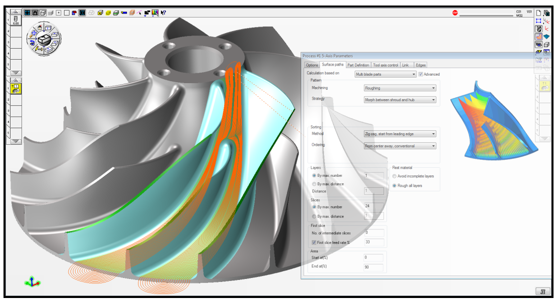
Example of Morph Between Two Surfaces

This example shows an impeller section. The tool diameter is 10 mm, so the start margin is 5 mm. As you can see the distances between the drive surface and the check surface to the sphere center of the tool is 5 mm. This is also essential for the end surface.



To see this example, open the file `Pattern - Morph Between Two Surfaces.vnc`.

5-Axis Impeller Machining Options Compared to 5-Axis MultiBlade



Although the base 5-Axis product includes options for impeller machining, the preferred solution is the 5-Axis MultiBlade product option (or 5-Axis MultiBlade Level 2 for finest control of all aspects of impeller machining). Because MultiBlade is only for impellers and blisks, it automatically detects and leverages radial symmetry, accommodates blades and splitters of any curvature, and provides options and controls that are specific to impellers, such as special settings for leading and trailing edges.

Parallel to surface

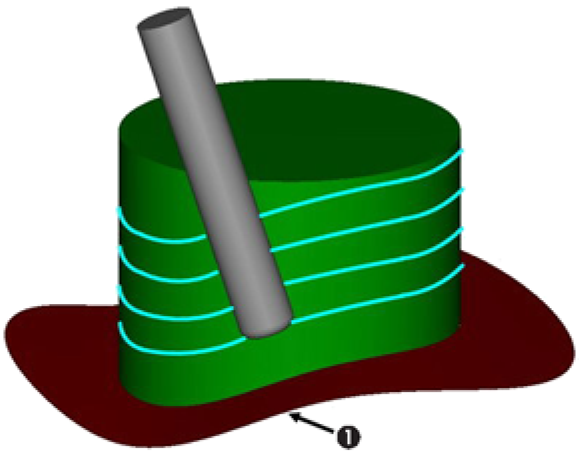
Using the Parallel to surface pattern means that the cuts on your drive surface will be generated on a drive surface (see “Drive Surfaces” on page 74) parallel to a check surface (see “Check Surfaces” on page 75). This option is particularly useful when your drive surface encounters an uneven check surface. The distance between two neighboring cuts is the Stepover. With this strategy you can define a margin to get the tool to be as close to both the check and drive surfaces as possible without gouging. Additionally, there are items that give you additional control over the surfaces to be cut which are accessed by clicking the Advanced button. For more information, see “Advanced button for Pattern” on page 72.

- The drive surface edge and the check surface edge must be coincident.
- To ensure the check surface is not violated due to tool tilting it is important to activate gouge checking.

If you are working with margins (see “Stepover” on page 151 for an example) the Area Type has to be set to Full, start and end at exact surface edge because the distance between the margin and first cut depends on the exact position of the surface edge. Then select the Advanced button to open the Margins window. The start margin belongs to the first surface and the end margin to the second surface. The values should be the radius of your tool. If you are using a ball endmill and would like to have double tangency (pencil tracing), you have to switch on "Calculation Based on Tool Center" on page 294 in the Utility tab. If the calculation is not based on the tool center, the wrong toolpath will be generated.

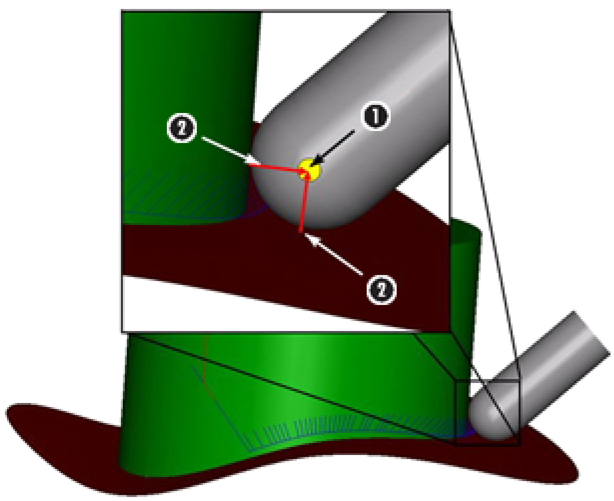
Example

Here you can see the tool running on the drive surface parallel to the wavy check surface. All cuts are parallel; it does not matter how far they are away from the check surface.



1. Check Surface

This shows how the margins option works. With a margin like the tool radius the tool exactly is located in the edge.

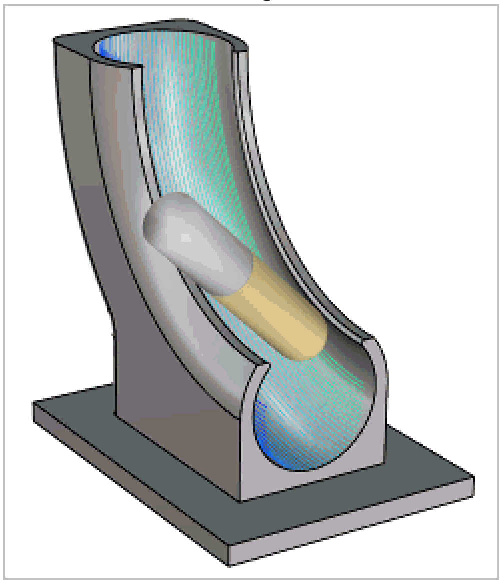


1. Tool center point
2. Margin

To see this example, open the file `Pattern - Parallel To Surface.vnc`.

Flowline

Toolpath generated by Flowline will be aligned or mapped to the U or V direction of a single machining surface according to your choices, as illustrated below. This lets you avoid having to select additional bounding geometries like wall surfaces or edge curves, and the maximum stepover can be maintained with a constant distance even if the surface topology is very complex. Calculation time is very fast. This pattern is for only one surface at a time, and the surface must have two or more edges.



You will need to choose a Style and a Drive Surface (see “Drive Surfaces” on page 74) and to specify a value for Drive surface clearance (see “Drive surface clearance” on page 75).

Style

Choose one or the other of two possible directions.

Advanced button for Pattern

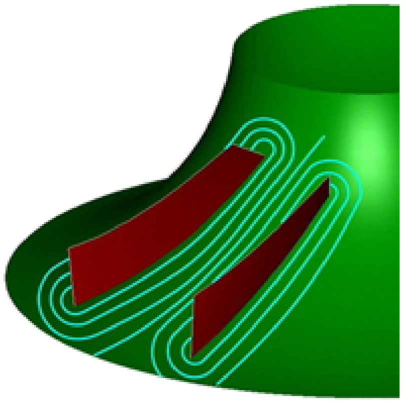
In the Pattern region of the Surface paths tab (for certain choices of Pattern), you can click the Advanced button to open the Advanced options of surface paths pattern dialog box.

Generate toolpath only at front side

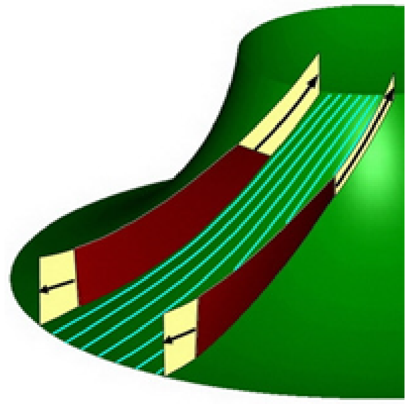
The effect of this item depends upon the Pattern being used.

Morph between two surfaces

When Pattern is set to Morph between two surfaces, then selecting this checkbox will restrict toolpath to between the two surfaces on the side that the surface normal points away from. The default toolpath is all around the blades, even on the backsides. When this option is active, the toolpath is only created between the first and second surface. What happens is that the two enclosing surfaces will be virtually extended until they reach the end of the drive surface. The red surfaces are the real surfaces. The yellow ones are the virtual extended surfaces.

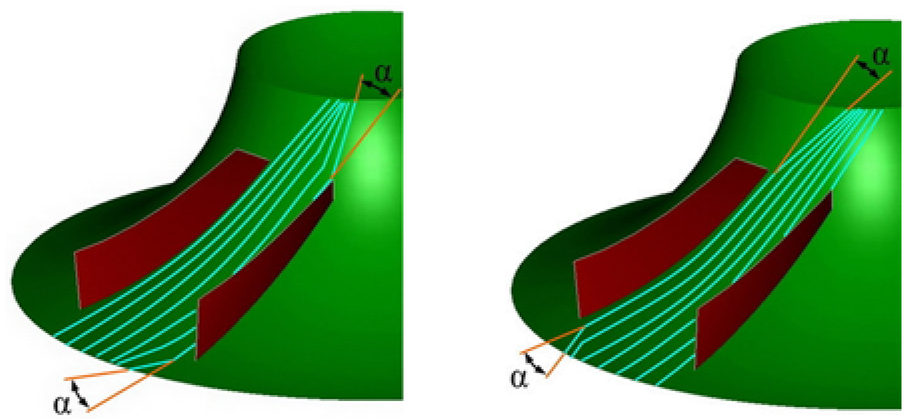


Generate toolpath only at front side disabled



Generate toolpath only at front side enabled

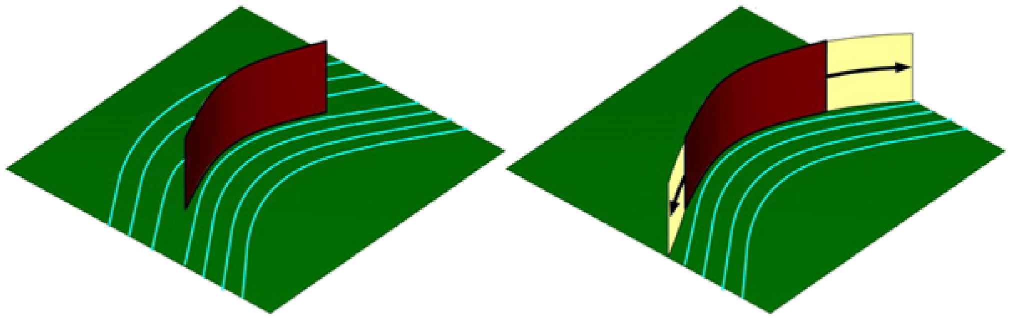
The First surface toolpath tangent angle and Second surface toolpath tangent angle items limit the toolpath generation. Imagine that you tilt the virtual extended surfaces with an angle. You can set it for the first and for the second surface. A positive angle value lets the path tilt inside, a negative angle tilts the path outward.



Examples of Surface toolpath tangent angle limits

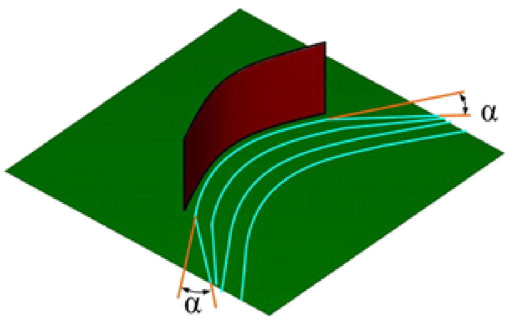
Parallel to surface

When Pattern is set to Parallel to surface,then selecting this checkbox will restrict toolpath so that it is parallel to the selected surface on the side that the surface normal points away from. The following images show the toolpath when the checkbox is not selected (on the left) and when it is selected (on the right). What happens is that the surfaces are virtually extended to the end of the drive surface. In this image the red surface is the real surface, the yellow ones are virtually extended.



An example of default toolpath compared to toolpath when Generate toolpath only at front side is selected

The Single edge toolpath tangent angle option limits the toolpath generation. This tilts the virtual extended surfaces with an angle.



Edit Curves / Edit Surfaces

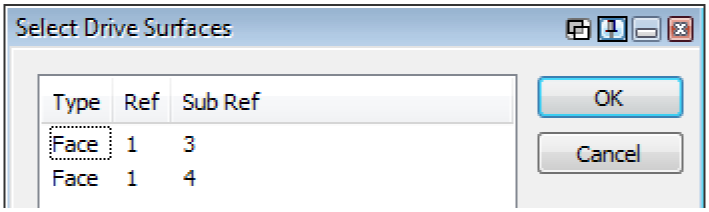
The following patterns all require the selection of one or more curves or surfaces to fully define the process: Perpendicular to curve, Morph between two curves, Parallel to curve, Project curves, Morph between two surfaces and Parallel to surface. The buttons in the Edit Curves or Edit Surfaces area open a dialog box that lets you choose the required element from the workspace. For more information, see “Drive Surfaces” on page 74 or “Drive Curves” on page 76.

Drive Surfaces

A *drive surface* is the body, face, or group of faces you want to machine. You can machine either solids or sheets; if you select sheets, be sure that the correct side of the sheet is pointing out.

Every pattern type requires you to choose the faces to be machined. Clicking this button lets you select the face or faces to work on; the 5-Axis Parameters dialog box disappears and is temporarily replaced by the Select Drive Surfaces dialog box opens. You can also select drive surfaces directly by selecting the body or faces when the selection mode in the Machining palette is set to Part (see “Machining Palette for 5-Axis” on page 32).

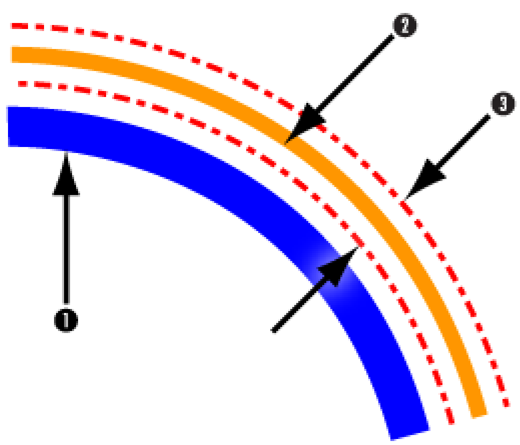
Select Drive Surfaces



This dialog shows which faces will be used as drive surfaces for the current process. Surfaces are added by selection in the workspace.

Drive surface clearance

Drive surface clearance is a virtual offset to the drive surface. The parameter enables you to specify the amount of material or stock allowance to remain on drive surface after the completion of the toolpath. All values are relative to the surface. The tool will not come closer to the surfaces by less than this value for positive offset values, and will not enter into the surface by more than the absolute value of the offset for negative offset values. For example, with a drive surface offset of `0.3` the tool will not come closer than 0.3mm + tolerance. This can also be seen as a remaining stock value on the surfaces.



1. Drive Surface
2. Drive Surface Clearance
3. Tolerance

The offset is 3-dimensional and expands the faces in every direction. This function affects the tool *tip* only; therefore, the tool shaft, holder front, and holder back do not necessarily stay away from the drive surfaces by the offset value when swarf cutting. To get an offset for the tool parts, you must use the options found in “Clearances for Tool Parts” on page 234.

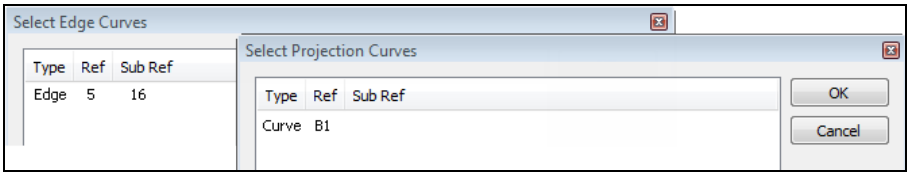
Check Surfaces

A check surface is a face used to contain toolpath or set an area the tool may not enter or cut. Check surfaces are also used to control the shape of the toolpath in that the tool can follow the check surface’s topology. The tool behavior when it encounters a check surface depends on the Pattern Settings being used, the selected Gouge Check strategy and the Link settings.

Drive Curves

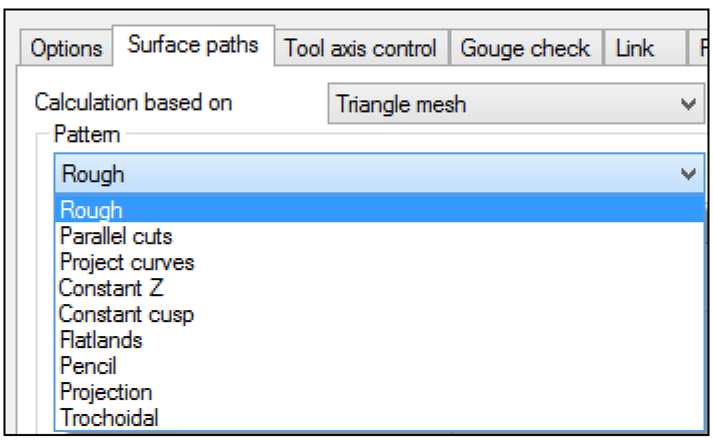
A drive curve is geometry or the edge of a solid that is being used to control the toolpath. Depending upon the Pattern Settings type selected, the exact name of the guide curve may be different. Examples include: Lead, First Curve / Second Curve, and Edge or Single Edge.

When you select an edge or curve, you will be presented with a dialog similar to one of the following. These dialogs show which element or elements will be used in the operation.



Pattern Settings (Calculation based on Triangle Mesh)

The first thing to do is set up the machining strategy for your machining surface. The Pattern list determines the type of cut:



- Common Controls, next
- “Rough” on page 77
- “Parallel cuts” on page 80
- “Project curves” on page 81
- “Constant Z” on page 85
- “Constant cusp” on page 86
- “Flatlands” on page 87
- “Pencil” on page 89
- “Projection” on page 90

How it works. After you choose a type of cut (Pattern), you select the faces to be cut (Machining surfaces) and specify values for Z height (Heights), tool offset (Offset), and other settings. Optionally, you can select the ellipsis button (...) to specify a Machining direction – either Top (the default) or an Other direction based upon either an XYZ vector that you enter or a tool plane that you select.

Common Controls

The following controls are found in multiple patterns.

Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights

Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Offset type

This drop-down menu lets you specify a virtual offset to machining surfaces (such as for material or stock allowance) either globally or individually.

- If you select Global, the value you specify for Offset is applied equally in radial and axial directions.
- If you select Radial and Axial, you can specify individual values for Radial offset and Axial offset.

Offset

This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

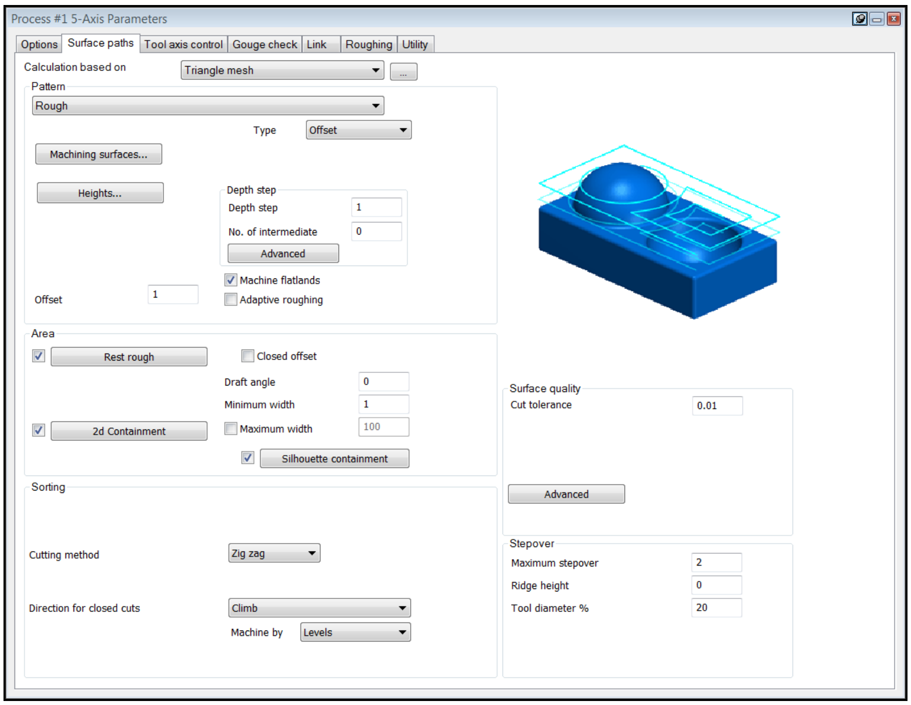
A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

Rough

With calculation based on Triangle Mesh, the Rough pattern is used to clear large volume of excess material very quickly and leave small amount of stock material for semi-finishing and finishing strategies. You can use this strategy to create a rough component from a rectangular or core shaped block. Machining is done in planar layers perpendicular to the tool axis.

How it works. Toolpath cuts material in successive Z-levels working from top down. The distance between two Z levels is defined by the value of Depth step. The default setting for Type is Offset: toolpath is created from model slices and offset outwards. Alternatively, you can use the Parallel setting and specify a single cut direction. For both cut types, the distance between successive passes is defined by Stepover. Toolpath segments are trimmed to block limits. The result is a rough component with a staircase effect over the whole component. It differs from a finished component by a thickness whose value you specify in Offset.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see “ Area ” on page 113, “Sorting Settings” on page 129, Surface Quality, and “Stepover” on page 151.



Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights

Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Type

The default setting, Offset, is automatic when some options are in effect (such as Adaptive roughing). Toolpath is created from model slices and offset outwards.

If Parallel is available, you can specify the X,Y slope of the parallel toolpath passes by entering a value for Machining angle in X,Y. Parallel is best suited for shallow areas.

Depth step

The Depth step value defines the distance between the Z levels.

Number of intermediate slices

You can add intermediate slices to the Rough toolpath to reduce the staircase effect. Reasons for adding intermediate slices include:

- Less material left for smaller tools while rest-roughing
- Uniform thickness for semi-finishing toolpaths
- Uniform tool load on semi-finishing tools
- More stock removal with larger tools with fewer steps
- Reduction in Rough machining time

Intermediate slices are rest roughed; additional offset passes are added if the stock left is too extensive, so as to avoid load on the cutter.

How it works. The ordering of the intermediate toolpath segments is from top to bottom to reduce the machining time. The number of intermediate slices defines the number of additional slices that will be added between each Depth step. *Example:* If Depth step is set to 4 mm and Number of intermediate slices is set to 2, then the effective stepdown is 1 mm.

Machine flatlands

This checkbox lets you machine true flat areas of 3D workpiece with toolpath passes that are offset segments of the flat area boundary. It is best suited to machine large flat areas at multiple Z levels.

Adaptive roughing

Selecting the Adaptive roughing checkbox ensures that the cutting conditions remain almost constant. This offers great improvements compared to conventional constant-offset roughing strategies. The strategy avoids full-width cuts by constantly measuring the engagement volume of the tool with material and gradually removing material off the remaining stock. Because it guarantees a stable load on the tool, it allows for a higher rate of material removal at higher feedrates and it reduces the overall machining time.

Advantages:

- Reduction in machining cycle time
- Extended tool life

Features:

- Supports 2-axis and 3-axis models
- No full-width cuts
- Consistent engagement with material
- Trochoidal like passes to progressively cut corners

Offset

This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

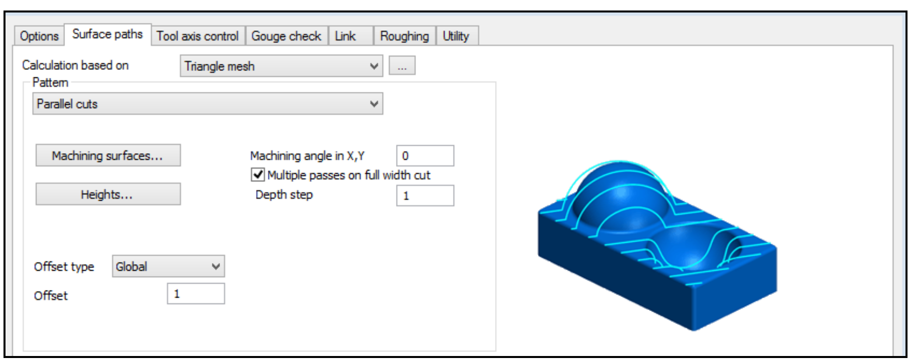
Parallel cuts

With calculation based on Triangle Mesh, the Parallel cuts pattern will create toolpath passes that are parallel to each other, all of them at the same angle in the XY plane (Machining angle in X,Y).

How it works. You select one or more Machining surfaces, choose either automatic or user-defined Z Heights, specify the Machining angle in X,Y, and specify a positive, zero, or negative value for tool Offset.

> ⓘ Imagine slicing a block of cheese each slice parallel to all the others. The pictures in the dialog symbolize how to set the desired cutting direction using the angles.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see " Area " on page 113, "Sorting Settings" on page 129, Surface Quality, and "Stepover" on page 151.



Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights

Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Machining angle in X, Y

This is the angle of the toolpath, referenced to the XY plane. An angle of 0 degrees will produce toolpath parallel to the X axis; an angle of 90 degrees will produce toolpath parallel to the Y axis. Any value between -360 and 360 is valid.

Multiple passes on full width cut

To reduce tool load, you can specify a positive value for Depth step. The Z depth (depth-of-cut) for each pass will not exceed this value.

Offset

This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

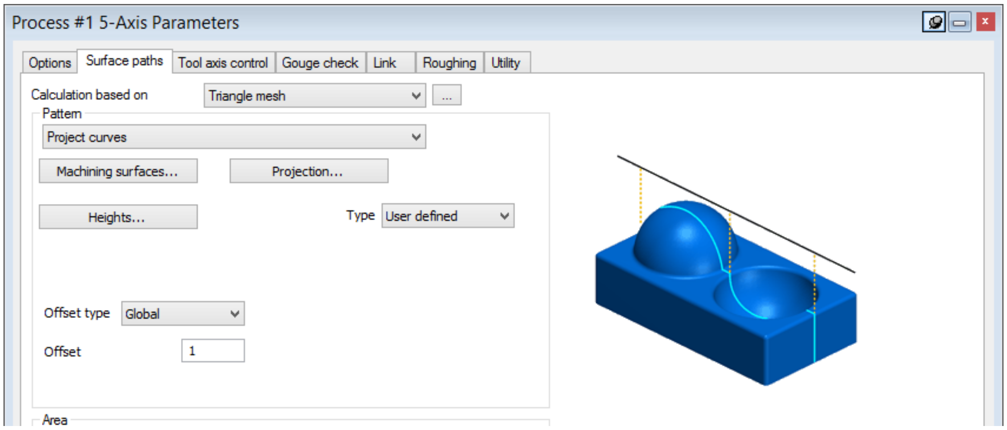
A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

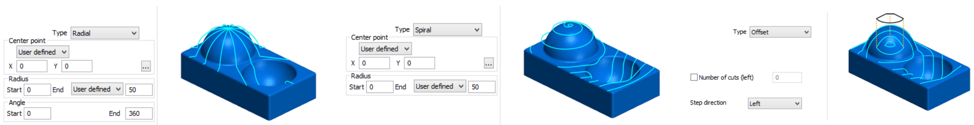
Project curves

With calculation based on Triangle Mesh, the Project curves pattern lets you create a 2D curve pattern and drop it onto the mesh in the Z direction to create toolpath.

How it works. You choose a Type such as Radial, Spiral, Offset, or User-defined, select one or more Machining surfaces (and Projection curves if necessary), choose either automatic or user-defined Z Heights, specify other values and options as needed depending on type, and specify a positive, zero, or negative value for tool Offset.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see “ Area ” on page 113, “Sorting Settings” on page 129, Surface Quality, and “Stepover” on page 151.


Process #1 5-Axis Parameters
Options
Surface paths
Tool axis control
Gouge check
Link
Roughing
Utility
Calculation based on
Triangle mesh
...
Pattern
Project curves
Machining surfaces...
Projection...
Heights...
Type
User defined
Offset type
Global
Offset
1
Area

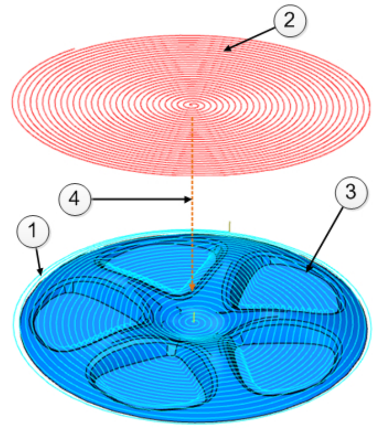


Type

- **Radial** is generally used as a finishing operation. It is particularly effective on circular shaped components and shallow areas.

  How it works. A 2D radial pattern(2) with a constant pitch is projected onto the 3D workpiece(3), based on a Center point and start and end Radius. Center point and RadiusEnd can be either specified manually by choosing User defined, or detected automatically by choosing Autodetect.

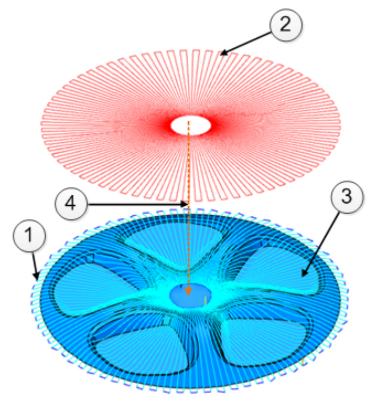
  The area can be limited by start and end Angle between 0 and 360 degrees. The resulting 2D pattern (2) is dropped onto the workpiece(3) in the Z direction (4) and limited by the machining zone, to create the resulting toolpath(1).



Radial

- **Spiral** is generally used as a finishing operation. It is particularly effective on circular shaped components and shallow areas. This toolpath can be used on high-speed machining centers.

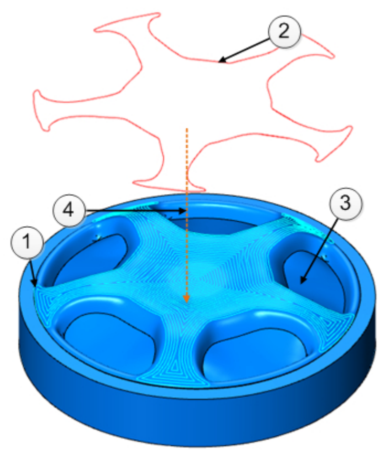
  How it works. A 2D spiral pattern(2) with a constant pitch is projected onto the 3D workpiece(3), based on a user-defined or automatically detected Center point and Radius: Start and End. Machining starts from inner circle defined by start radius and cuts spirally outward with stepover constant in the 2D plane.



Spiral

- **Offset** is used to create a 2D offset pattern based on user-defined curves – which can be open or closed – projected on the 3D workpiece to finish-mill complex shapes. It is particularly effective on shallow areas.

  How it works. An offset pattern(2D) is created on a 2D plane from the specified curve segments. A constant stepover creates ***N*** parallel offsets of the curve in the Left or Right directions (or ***2N*** for Both), where ***N*** is the value you specify for Number of cuts. The created pattern is then dropped onto the workpiece(3) in the Z direction(4) to create the toolpath(1).



Offset

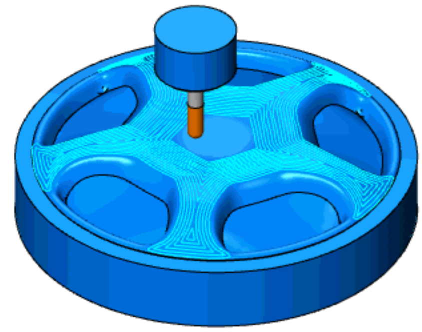
Number of cuts

Specifies the number of offset cuts Left and/or Right *in addition to the center cut*. For example, if you specify 3 and Both, then a total of *seven* cuts will be made: three each to the left and to the right, plus the center cut.

Note that the system automatically detects closed or open segments in the drive curve:

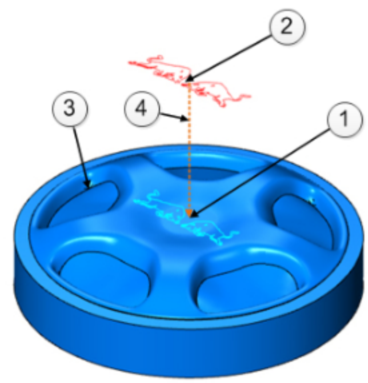
- Closed drive curve: The offsets are calculated inside a closed drive curve. In this case, Number of cuts is automatically calculated based on stepover value.
- Open drive curve: The offsets are calculated on the right or left or on both sides of the open drive curve segment. In this case, you need to specify the Number of cuts.

The stepover parameter value corresponds to the maximum 2D distance between any two consecutive passes.



- **User-defined** is generally used as a finishing operation. It is particularly effective on shallow areas.

  How it works. The selected free-form curves(2) are dropped onto the workpiece(3) in the Z direction(4) and limited by the machining zone to create toolpath (1).



User-defined

Machining surfaces
Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Projection
For type Offset or User-defined, clicking this button opens a dialog that lets you select the curves to be projected.

Heights
Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Offset
This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a

3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

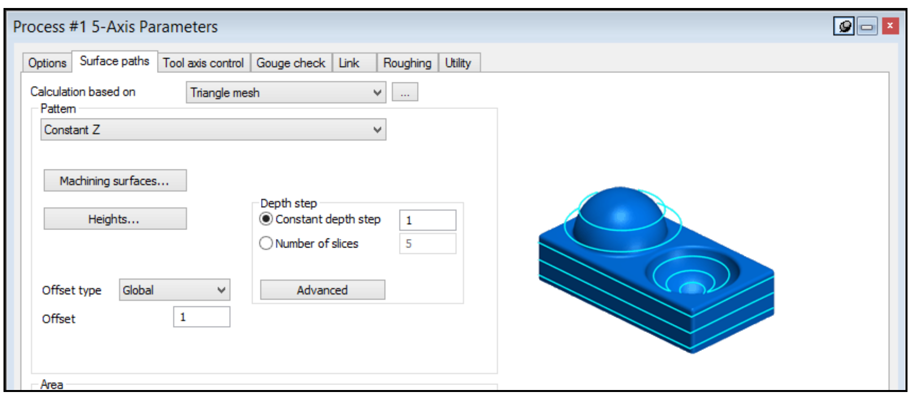
A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

Constant Z

With calculation based on Triangle Mesh, the Constant Z pattern lets you machine a 3D workpiece with toolpath passes that are parallel to a plane that depends on the machining direction. Imagine the component being sliced from top to bottom. This is generally used to semi-finish or finish a workpiece, and is best suited to machine steep areas: The stepover is defined in the machining direction and is used to machine vertical or near-vertical walls of a 3D workpiece.

How it works. You select one or more Machining surfaces, choose either automatic or user-defined Z Heights, specify whether or not to Machine flatlands, and specify a positive, zero, or negative value for tool Offset.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see “ Area ” on page 113, “Sorting Settings” on page 129, Surface Quality, and “Stepover” on page 151.



Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights

Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Machine flatlands

This checkbox lets you machine true flat areas of 3D workpiece with toolpath passes that are offset segments of the flat area boundary. It is best suited to machine large flat areas at multiple Z levels.

Offset

This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

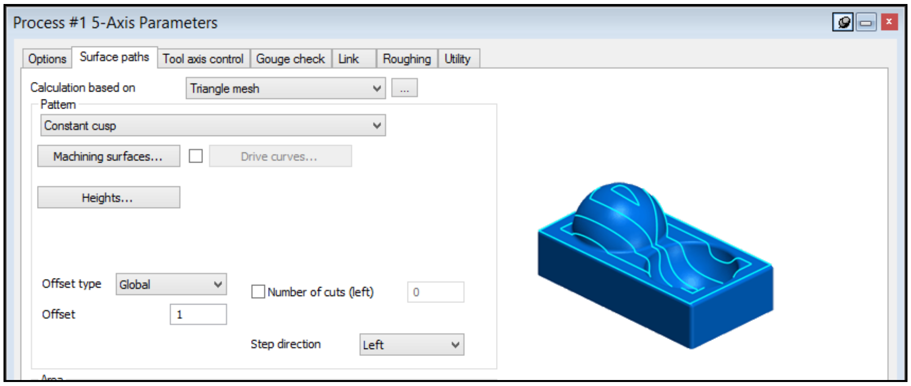
A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

Constant cusp

With calculation based on Triangle Mesh, the Constant cusp pattern lets you create an equidistant cut pattern on the machining surfaces. The aim is to have a constant distance between each contour so that the cusps created will have the same height. This is generally used to semi-finish or finish a workpiece, and is best suited to machine steep areas as well as shallow areas.

How it works. You select one or more Machining surfaces, choose either automatic or user-defined Z Heights (note that you must define both start and end heights), and specify a positive, zero, or negative value for tool Offset. In the Stepover section, the distance between two consecutive toolpath segments can be defined as constant stepover or using ridge height.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see “ Area ” on page 113, “Sorting Settings” on page 129, Surface Quality, and “Stepover” on page 151.



Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights

Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Offset

This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

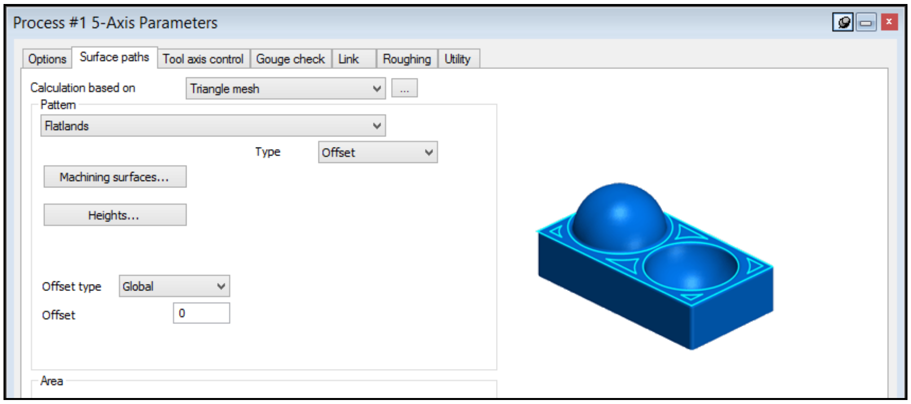
Flatlands

With calculation based on Triangle Mesh, the Flatlands pattern lets you machine true flat areas of a 3D workpiece with toolpath passes that are offset segments of the flat area boundary. It is best suited to machine large flat areas at multiple Z levels. Flat areas such as parting surfaces can be machined by end mill or bullnose mill cutters using the Flatlands pattern.

How it works. You select one or more Machining surfaces, choose either automatic or user-defined Z Heights, and specify a positive, zero, or negative value for tool Offset.

Note that only true flat areas are detected; you can define the minimum width of the flat area to be detected by the algorithm using the Minimum width parameter under Area.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see " Area " on page 113, "Sorting Settings" on page 129, Surface Quality, and "Stepover" on page 151.



Machining surfaces
Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights
Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Offset
This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

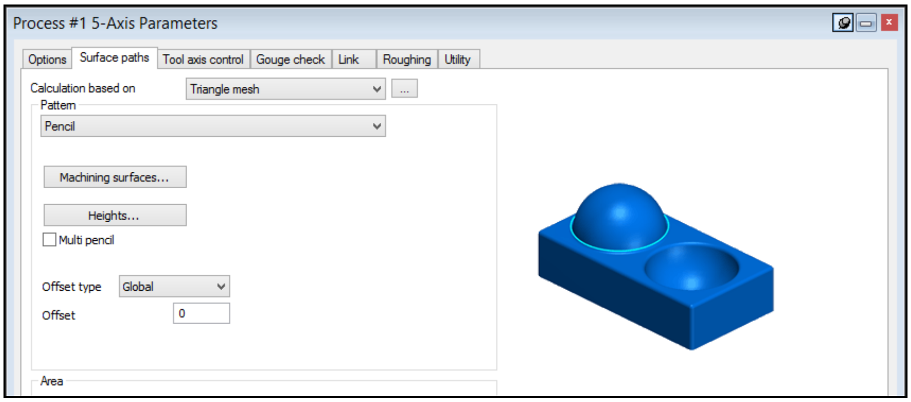
A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

Pencil

With calculation based on Triangle Mesh, the Pencil pattern lets you automatically detect inlying edges and create a single toolpath along them.

How it works. You select one or more Machining surfaces, choose either automatic or user-defined Z Heights, and specify a positive, zero, or negative value for tool Offset.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see " Area " on page 113, "Sorting Settings" on page 129, Surface Quality, and "Stepover" on page 151.



Machining surfaces
Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Heights
Clicking this button opens a dialog that lets you define the heights of the area to be machined. You can choose from several Automatic options (which create a bounding height around the selected drive surfaces), or you can specify the heights manually (User defined) either by entering values directly or by clicking the ellipsis button (...) and querying points.

Note that you do not set width or length here. They are set in the Area portion of the tab.

Offset
This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

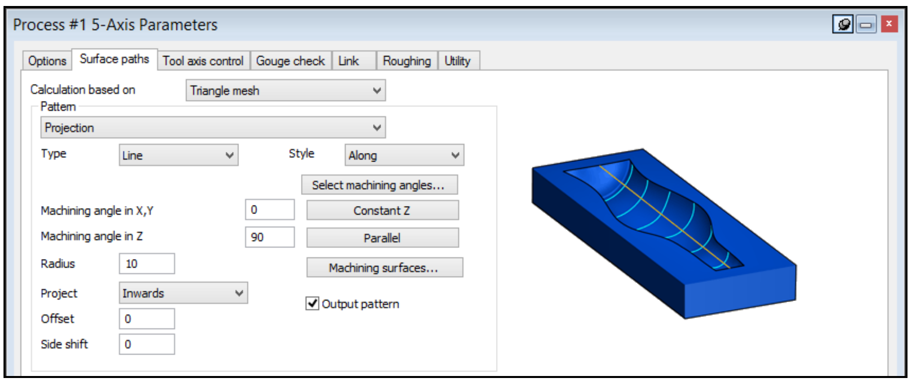
A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

Projection

With calculation based on Triangle Mesh, the Projection pattern lets you create a line projection on machining surfaces. It is especially useful for molds, but can be used for rotary machining of any rotary part.

How it works. You choose a Style, choose either Constant Z (with restricted angle values) or Parallel, specify a value for Radius and choose whether to project inwards or outwards, select one or more Machining surfaces, and specify a positive, zero, or negative value for tool Offset.

For the Pattern portion of this tab, options and parameters are described below. For other portions of this tab, see "Area Options for Projection Pattern" on page 128, "Sorting Settings" on page 129, Surface Quality, and "Stepover" on page 151.



Style

Along: Creates the toolpath projected along the line.

Around: Creates the toolpath perpendicular projected from the line

Radius

This value represents the maximum projection distance measured from the line.

Project

This defines the tool orientation. For concave shapes, select Inwards. For convex cavities, select Outwards.

Machining surfaces

Clicking this button opens a dialog that lets you select the surfaces to be machined. Or, instead of a surface, you can select a facet body.

Offset

This value represents a virtual offset to the surfaces. The parameter enables you to specify the amount of material or stock allowance to remain on the surfaces. The offset can be understood as a 3-dimensional offset that expands the faces in all directions. For example, with an offset of 0.3 mm, the tool comes no closer than 0.3 mm to the selected faces.

Note: The offset from the surface is only as accurate as the machining accuracy (cut tolerance). That means that the offset can deviate with the selected tolerance. For example, with an offset of 0.1 mm and a cut tolerance of 0.1 mm, the real offset can go from 0.0 mm to 0.2 mm.

The offset will always be considered within gouge checking tool part clearances and the stock to leave value. All values will be added together, so in the end there is an overall clearance consisting of *offset* + *stock to leave* + *tool clearance*.

A positive value offset lets stock remain. A negative value lets the tool undercut the surface, and necessarily causes collisions.

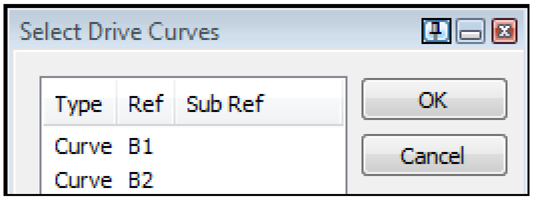
Pattern Settings (Calculation based on Wireframe)

In the Surface paths tab, when the calculation is based on Wireframe, no machining surfaces are used, and the output is a single toolpath along one or more specified drive curves.

- For pattern 5-Axis Profiling, you select drive curves and orientation lines, designate a snap distance and cutting side, and optionally specify an offset value.
- For pattern 2-Axis Rough, you select drive curves, specify start and end heights, choose whether to specify a constant step or to specify the number of slices, and optionally specify an offset value.
- For pattern 2-Axis Profile, you select drive curves, specify start and end heights, choose whether to specify a constant step or to specify the number of slices, designate a cutting side and cutting method, and optionally specify an offset value.
- For pattern Engrave, you select drive curves, specify start and end heights, choose whether to specify a constant step, and designate cutting method and direction.
- For pattern Face, for regular 2D face milling, you select drive curves and specify a machining angle and offset. Various extension values can be specified under Area.
- For pattern Trochoidal, you select drive curves and specify start and end heights, choose whether to specify a constant step, and set the width, radius, and direction of the trochoidal loops.
- Pattern Extrude is for situations where a cutter with very large diameter cuts along and across an extruded profile. Machine limits and collision checking are fully supported. You specify a profile curve and sweep rail, and choose amongst several options: Across or Along; Inside or Outside; Zigzag or One way; Bottom to top or Top to bottom; and Climb or Conventional.

Drive curves

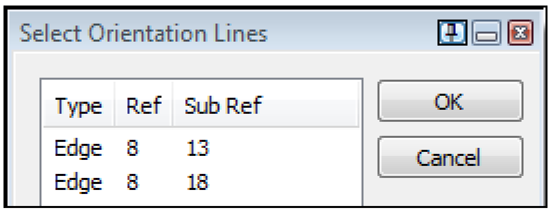
A drive curve is an edge, curve, or connected set of curves that define the path followed by the tool. Clicking the Drive curves button temporarily hides the 5-Axis Parameters dialog and opens the Select Drive Curves dialog. This dialog shows which elements will be used as drive curves for the current process. Curves are added by selecting them in the workspace.



Note that the tool will be automatically offset with the tool radius.

Orientation lines

An orientation line provides a tilt line for aligning the tool axis. Clicking the Orientation lines button opens a dialog that shows which elements will be used. Lines and edges are added by selecting them in the workspace.



Maximum snap distance

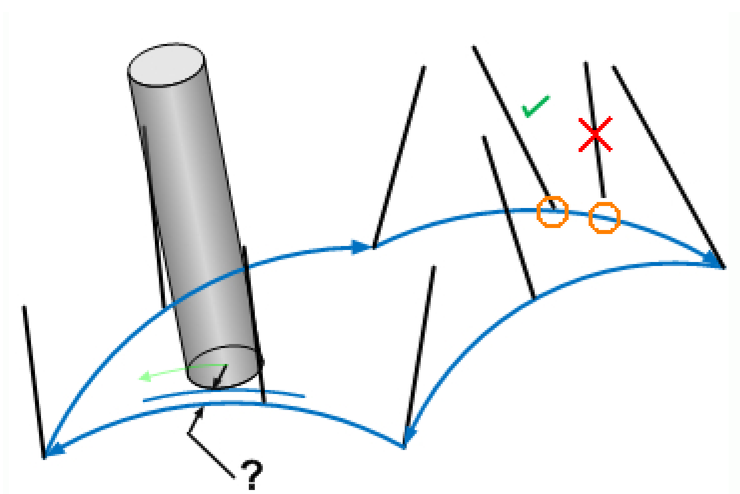
This value specifies the farthest distance that a tilt line can be located and still be considered in the toolpath calculation. If the line touches the drive curve or lies within this distance, it will be considered for tilting. Any element farther away than the specified distance is ignored.

Cutting side

Choose Left or Right to position the tool to the left or right of the cutting direction. Cutting direction is determined by the chaining of the drive curves from the start point of the chain.

Offset

This value specifies the minimum distance between tool and drive curve.



Blue arrows: Drive curves (chained clockwise)

Black lines: Orientation lines

Orange circles: Maximum snap distance

Cutting side: Right

Offset: **?** > 0

Calculation Strategy: Swarf Machining

In the Surface Paths page, the first option, Calculation based on, affects options displayed in this tab and others. If you choose Swarf machining, the 5-Axis process dialog changes its tabs dynamically, suppressing Roughing and adding Multi Cuts and Corners.

- "Overview of Swarf Machining", next
- Surface Paths tab for Swarf Machining
- "Tool Axis Control tab for Swarf Machining" on page 98
- "Gouge Check tab for Swarf Machining" on page 98
- "Link tab" on page 238
- "Multi Cuts tab" on page 100
- "Corners tab" on page 102

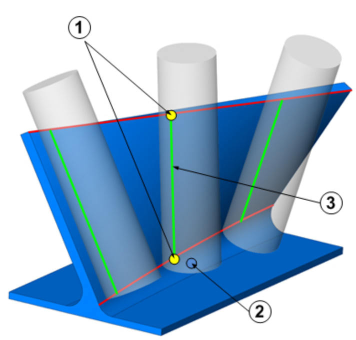
Overview of Swarf Machining

Swarf machining, or also called "flank milling", is a 5-axis simultaneous milling process. It is most often used for machining fluid parts for turbo engines or aeronautical parts like integral elements.

The goal is to produce the target surface with only one cut, using the whole flute length of the tool. Achieving this goal can mean:

- Better surface finish quality (without hand finish)
- Shorter finish cycle time
- Full access of machining areas through simultaneous 5-axis vector orientation
- Constant cutting conditions: increased material removal at constant (low) cutting force

Input geometry always includes one upper curve and one lower curve (shown in red in the illustration).



The tool is always aligned between contact points (item 1) on each curve. Tilt cycles are used to control tool axis orientation (item 3). The cutter location point (item 2) is the point that is represented by the coordinates of the machine program.

The tool flank is straight geometry; the actual surface is free-form, and can be curved in any direction. Therefore, swarf cutting cannot create a target surface perfectly if the surface between the two curves has double curvature: if the surface is convex, the tool will gouge, and if the surface is concave, it will leave rest material. The deviation is usually not large, however, and an automatic tilting cycle is available to minimize the error.

Surface Paths tab for Swarf Machining

When toolpath calculation is based on Swarf Machining, the Surface Paths page offers the following types of settings.

- Geometry selection: See Geometry Selection, next
- Machining: See “Machining” on page 95.
- Start point: See “Start Point” on page 95.
- Surface quality: Same as for strategies other than Swarfing. See “Surface Quality” on page 147.
- Advanced control: See “Strategy” on page 96.

Geometry Selection

Swarf surfaces

When this checkbox is selected, the selector button (...) opens a dialog that allows you to select surfaces. These are the surfaces to be machined.

Swarf offset

The swarf offset is an offset onto the upper and lower curve. It creates rest material.

Floor surfaces

When this checkbox is selected, the selector button (...) opens a dialog that allows you to select surfaces. These are the surfaces that represent the axial limit of the tool.

Floor Clearance

Specifies a minimum distance between tool and floor. Set an appropriate value to avoid scratching the floor with the tip of the tool.

Tilt lines

When this checkbox is selected, the selector button (...) opens a dialog that allows you to specify vectors (for example, by selecting lines). These vectors represent the preferred tilt of the tool.

Guide curves

When this checkbox is selected, you can specify upper and lower guide curves

Upper curve

The upper curve defines the upper contact point of the tool. It should be the upper edge of the swarf surface.

Lower curve

The lower curve defines the lower contact point of the tool. It should be the lower edge of the swarf surface.

Machining

Side:
Left or Right applies to open contours, and is relative to the chaining direction of the lower curve.

Inside or Outside applies to closed contours.

Direction:
Choose which direction to use, from options Climb, Conventional, or Follow lower curve chaining.

Start Point

The choice for Start point implicitly defines the starting tool axis orientation from a point on the lower curve and a point on the upper curve.

Exact:
Uses the chaining start point on both lower and upper curves.

Automatic:
Sets the start point automatically, depending on contour type:

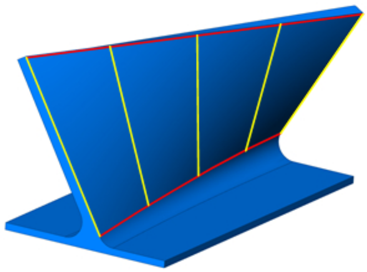
- When curves are closed contours, the start point on the lower curve is the mid point of the longest toolpath segment, and the start point on the upper curve is the nearest point to the start point of the lower curve.
- When curves are open contours, the start point on the lower curve is the actual chaining start point of the lower curve, and the start point of the upper curve is the chaining start point of the upper curve.

2 points:
Uses points that you select: one on the upper curve and one on the lower curve. This is meaningful only for closed curves.

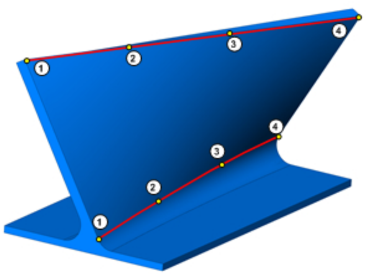
Tilt line:
Offers a selection button (...) to open a dialog that allows you to select a tilt line where the machining should start. This is meaningful only for closed curves.

1 point:
Offers a selection button (...) to open a dialog that allows you to select a point located upon either the upper or the lower curve. Initial tilting is not affected, because it is determined by the choice of Strategy. This is meaningful only for closed curves.

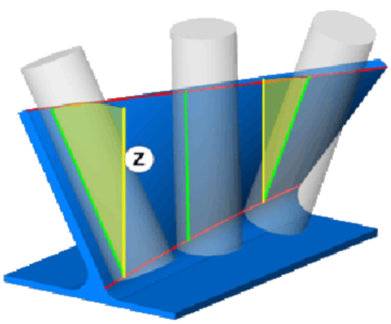
Strategy



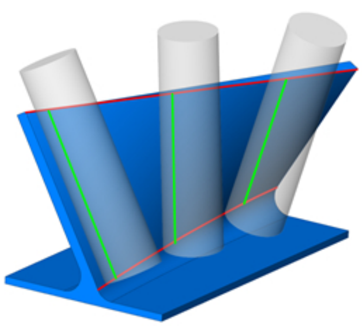
Sync with tilt lines:
The tool axis is aligned to tilt lines along the upper and lower curves. Tilt lines (yellow) must be selected as geometry.



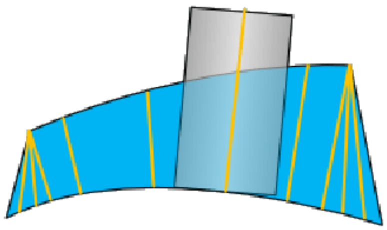
Sync with (upper/lower) curves:
The two curves are divided into steps of equidistant length. The tool axis is aligned to the pair of points in each step.



Sync with main direction:
The tool proceeds along either a principal axis (X, Y, or Z) or else a line that you select.

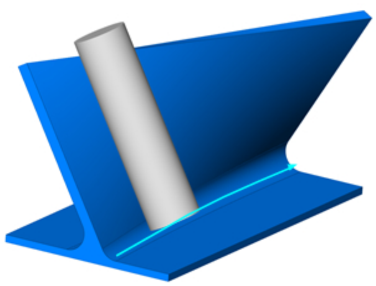
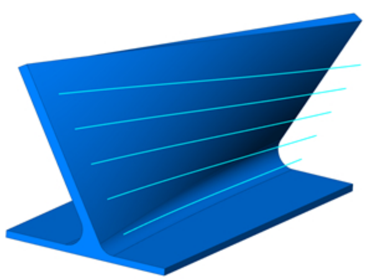


Automatic:
The system calculates tool placement onto the swarf surface so as to minimize gouging and rest material. A swarf surface must be selected as geometry.

	**Shortest distance:** The tool will be aligned with the shortest distance between the two curves. This strategy is useful for machining flat surfaces with sharp corners. When upper and lower curves are of different lengths, it is advisable to set a fanning distance: see “ Fanning” on page 98.

Pattern Slices

You can choose Single slice or Multiple slices. The purpose of multiple slices is to swarf-cut the machining area with multiple step depths in case the tool flute length is short.

	**Single slice:** This generates a single slice at the lower curve. Ensure that the flute length is sufficient.
	**Multiple slices:** Slices can be defined by providing either a maximum slice distance or a maximum number of slices. Slices are equidistant between the upper and the lower curve; the picture shows a total of 5 slices. Multiple slices are copies of the initial slice. Copy direction can be defined as either Along tool axis or Along contact line. Note that if a conical tool is being used, retraction along tool axis is not applicable, as the tool would leave the surface in the upper cuts; therefore, for conical tools, always use Along contact line.

Shift

Constant for each slice

This option applies the specified value for axial shift (To) to all slices.

Gradual for each slice

This option allows you to shift the tool tip point slightly deeper with every new slice. Specify an ending value (To) and a starting value (From).

Extensions

You can choose how the tool approaches and leaves the cutting operation. If you choose Automatic or Align to edges, you can specify the can choose the amount to extend before the start and after the end. Alternatively, you can choose Start with angle and specify the angle of the tool when it starts to engage.

Tool Axis Control tab for Swarf Machining

When toolpath calculation is based on Swarf Machining, the Tool Axis Control page offers the following types of settings.

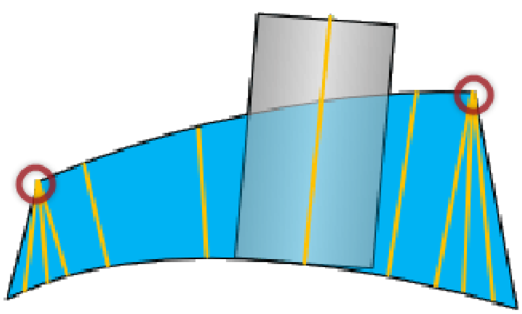
General

Provides settings for the following:

- Max. angle step: Specify the maximum angle through which the tool axis can vary. A smaller angle will require the calculation of more points.
- Swap curves: Select this checkbox to exchange the upper and lower curves.
- Damp: Select this checkbox to apply axial damping to the tool while swarfing along the wall. This is beneficial if the upper or lower curve is not tangent, and has sharp corners that would cause the tool to immediately retract or plunge.
- Minimize rotation axis changes: Select this checkbox to reduce problems with singularities that can cause extreme machine movements.

Fanning

Available only when the Strategy (on the Surface paths page, under Advanced control) is set to Shortest distance. For Fanning distance, specify a positive value to separate tool axis vectors that would otherwise originate in a single point because the upper and lower curves are of different lengths, as illustrated below.



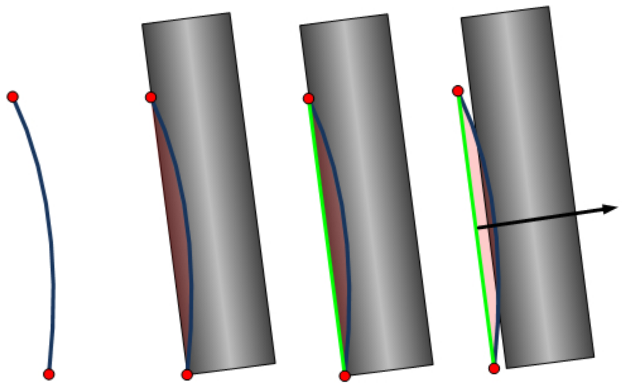
Gouge Check tab for Swarf Machining

When toolpath calculation is based on Swarf Machining, the Gouge Check page offers the following types of settings.

- “Degouging”, next
- “Collision check” on page 99

Degouging

Degouging avoids collisions that are greater than the specified allowance, by pushing the tool away in a direction orthogonal to the contact line between the upper and lower curves, as illustrated below.



To enable degouging, select the Gouge allowance checkbox and enter a value greater than `0`.

Gouge allowance:
Specifies the maximum distance between cutter flank and the surface to be protected.

Check against swarf surfaces:
When this is selected, the swarf surface is protected against gouging.

Additional check surfaces:
When this checkbox is selected, the selector button (...) opens a dialog that allows you to select surfaces.

Collision check

Trim toolpath:
Select this checkbox to enable the trimming of toolpath segments that collide with selected collision check surfaces.

Check against swarf surfaces:
When this is selected, toolpath is trimmed away from the swarf surface.

Additional check surfaces:
When this checkbox is selected, the selector button (...) opens a dialog that allows you to select surfaces. Toolpath will be trimmed away from the selected surfaces.

Tool clearance values:
You can specify separate clearance values for separate portions of the tool.

Multi Cuts tab

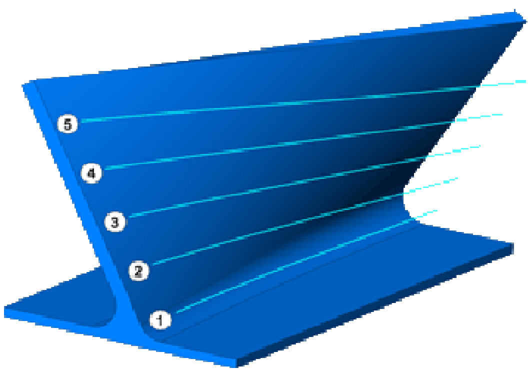
When toolpath calculation is based on Swarf Machining, the Multi Cuts page offers the following types of settings.

- “Pattern Slices”, next.
- “Pattern Layers” on page 102
- “Sorting” on page 102

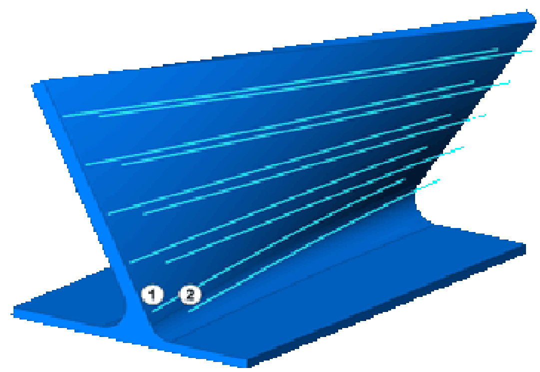
Terminology: Slice and Layer

A *slice* is along the upper-to-lower order of cuts: The first slice is closest to one curve, and the last slice is closest to the other curve. A small slice is narrow; a large slice is broad.

A *layer* is along the outer-to-inner order of cuts. A small layer is thin; a large layer is thick.



Slices (5)



Layers(2)

Pattern Slices

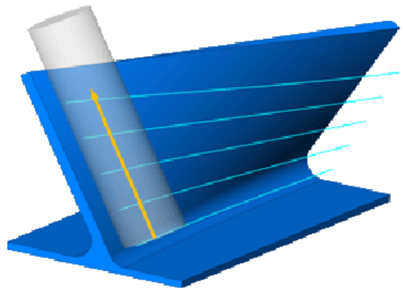
Depth steps:
Choose a method and enter a value, as follows. Also specify a Direction (see farther below).

- For the choice By slice distance, specify a real number greater than `0`. Each slice will be separated from the next by this amount.
- For the choice By number of slices, specify an integer greater than `0`. This quantity of cuts will be evenly divided between lower curve and upper curve.

Pattern:
Choose a method:

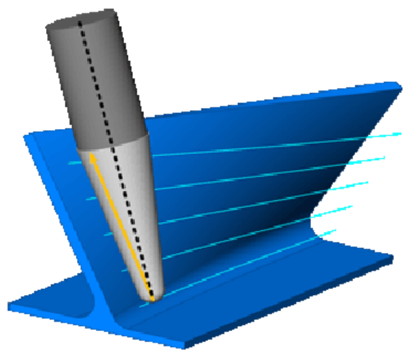
- Morph will create a series of slices that interpolates between upper and lower curves.
- Step from top will create a series of slices parallel to the upper curve.
- Step from bottom will create a series of slices parallel to the lower curve.

Direction:
Available only when Depth steps is set to By slice distance. The following choices are presented.



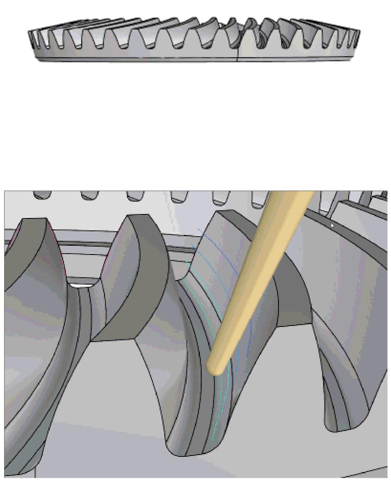
Along tool axis

Along tool axis: Distance from one slice to the next is measured along the axis of the tool.



Along contact line

Along contact line: Distance from one slice to the next is measured along the contact line where the tool meets the surface.



Follow surface topology

Follow surface toplogy: With this method, the direction for the number of slices is defined in an optimal direction, along surface tangency.

Sorting algorithm lets you set the Cut from bottom checkbox. This provides better material removal performance by starting the first cut at the bottom.

Tool shift:
Choose a method and enter one or two values. Default: `0` (no tool shift).

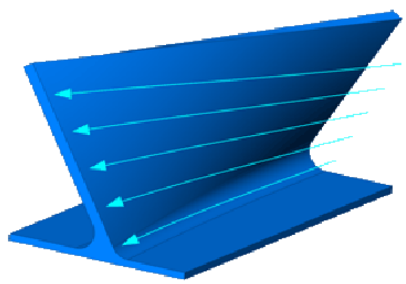
- For the choice Constant for each slice, you can specify a single tool shift amount to be applied to each slice.
- For the choice Gradual for each slice, you can specify a range of tool shift amounts (To and From) to be applied gradually from the first slice to the last.

Pattern Layers

For Number of layers, enter an integer greater than `0`.

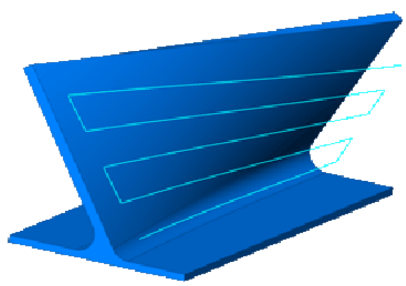
Sorting

For Method, the following choices are presented.



One way

One way: Toolpath direction is approximately the same for all slices.



Zig zag

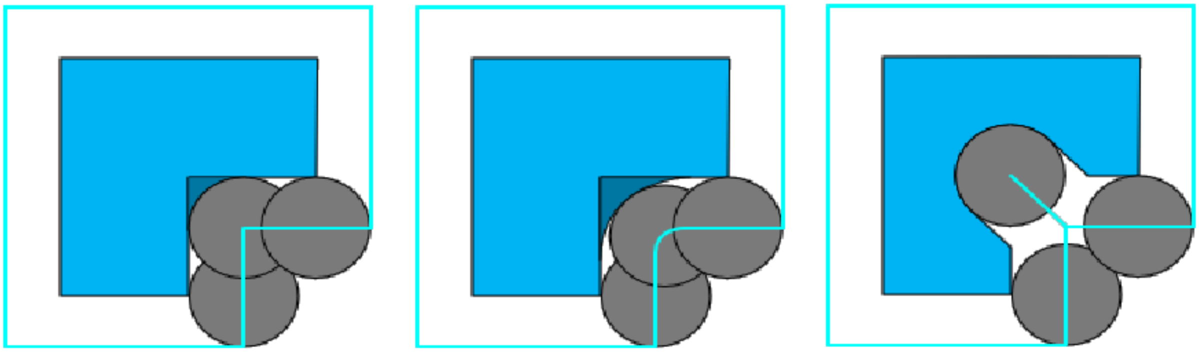
Zig zag (available only when the choice for Depth steps is By slice distance): Toolpath direction reverses on alternating slices. Moves are added to connect the end of one slice to the start of the next one.

Corners tab

When toolpath calculation is based on Swarf Machining, the Corners page offers the following types of settings.

- "Inside Corners", next.
- Outside Corners

Inside Corners



Sharp corner | Round corner | Relief groove

Sharp corner:
Toolpath will make an abrupt change at the inside corner point.

Round corner:
Toolpath will make an arc transition (fillet) at the inside corner point, leaving rest material.

Radius:
Radius of the transition arc.

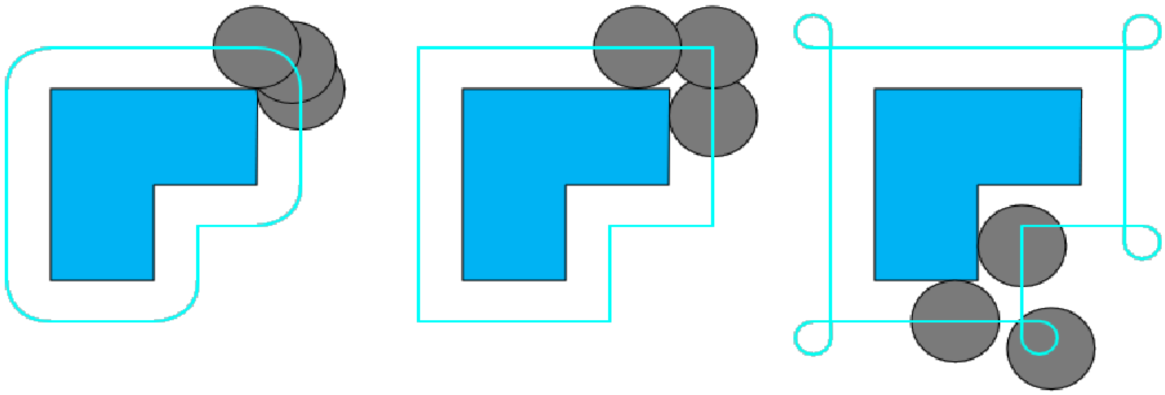
Detection angle:
Threshold angle for applying this inside corner treatment.

Relief groove:
Toolpath will create a groove inside the inside corner.

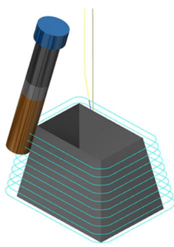
Length:
Depth of groove.

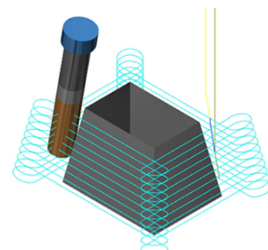
Detection angle:
Threshold angle for applying this inside corner treatment.

Outside Corners



Roll around	Sharp corner	Loop





Roll around:
Toolpath will make an arc transition around the outside corner point.

Sharp corner:
Toolpath will make an abrupt change at the outside corner point.

Detection angle:
Threshold angle for applying this outside corner treatment.

Loop:
Toolpath will make a smooth outside loop around the outside corner point.

Radius:
Radius for the outside loop.

Detection angle:
Threshold angle for applying this outside corner treatment.

Calculation Strategy: Geodesic

In the Surface Paths page, the first option, Calculation based on, affects options displayed in this tab and others. If you choose Geodesic machining, the 5-Axis process dialog offers seven tabs, from

Options through **Utility**. Controls in tabs not mentioned below are covered in the material for each of the 5-Axis tabs generally.

- "Overview of Geodesic Machining", next.
- Surface Paths tab for Geodesic Machining
- "Roughing tab for Geodesic Machining" on page 107
- "Utility tab for Geodesic Machining" on page 108

Overview of Geodesic Machining

Background

For non-prismatic features of a work piece, a common process of surface finishing is cutting with several parallel cuts to produce the final work piece geometry. Originally, it was quite common to project parallel lines onto the free-form surface topology or slice with planes in a fixed direction to create "waterline" patterns. These patterns have the strengths of robustness and a relatively simple algorithm. The weakness is that the distance between the cuts is not constant, but depends instead on the surface topology/curvature and the local changes between two slices.

Geodesic

The Geodesic Machining calculation for toolpath goes farther, generalizing the concept of a "straight line" mapped onto "curved spaces". Geodesic patterns take into account distances on the surface topology.

Using a global distance field provides full flexibility to calculate various pattern types while maintaining consistent distances between cuts.

Features of Geodesic

- For offset or morph pattern types, single or multiple guide curves can be used as input. Even in an undercut situation (relative to a fixed direction) the algorithm outputs the same consistency of slices when generating the pattern.
- Tool center mode allows collision-free pattern generation even in sharp inner corners or strongly curved areas with multiple surface patches.
- The morph pattern provides more flexibility in terms of supported geometry. As the distance is calculated on the surface topology, a much higher precision of morph distance can be achieved even if the guide curves are located close to each other.
- Boundary detection is provided for guide and containment curves.
- Extensions and hole-filling are available to avoid edge rolling and easy pattern extrapolation. Guide curve extension allows short curves to be extended to the surface boundary.

Surface Paths tab for Geodesic Machining

When toolpath calculation is based on Geodesic Machining, the Surface Paths tab offers the following types of settings.

- Pattern: See Surface Paths tab for Geodesic Machining, next.
- Guide curves: Similar in concept to guide curves in Surface paths > Morph between 2 curves or Project curves, but with additional options. See “Guide Curve Parameters” on page 106.
- Containment: Similar in concept to 2D Containment options in Triangle Mesh > Area, but with additional options. See “Containment Parameters” on page 107.
- Area: Same as for strategies other than Geodesic. See “ Area ” on page 113.
- Sorting: Same as for strategies other than Geodesic. See “Sorting Settings” on page 129.
- Surface quality: Same as for strategies other than Geodesic. See “Surface Quality” on page 147.
- Stepover: Same as for strategies other than Geodesic. See “Stepover” on page 151.

Pattern Parameters

Calculation type

Two calculation types are offered:

Contact point

Contact point mode supports all tools. Its output is similar to surface based pattern, and it does not guarantee a collision-free pattern with the surrounding geometry – for example, in inner corners.

Tool center

Tool center mode supports ball tools only. The calculation is generated in offset space to avoid collisions with the surrounding geometry .

Machining surfaces

Opens a dialog that lets you select one more surfaces from the workspace.

Advanced

Opens a dialog that lets you set special parameters for unusual situations.

Machining surfaces offset

Enter a nonnegative value to offset from all selected surfaces.

Guide Curve Parameters

Pattern type

Two pattern types are offered:

Morph between two curves

Lets you select any two curves to build a pattern that changes gradually from one to the other.

Parallel to multiple curves

Lets you select many curves to build a pattern that is parallel to each selected curve.

Input type

Four input types are offered:

Automatic (machining area)

This will create toolpath automatically based on the selected surfaces and the parameters you specify in the **Area** section.

Automatic (surface boundary)

This will create toolpath automatically based on the edges of the selected surfaces.

Automatic (center)

This will create toolpath automatically in a centered pattern.

User-defined curves

Provides a button (Guide curves) that opens a dialog where you manually select the curves instead of having the system choose them automatically.

Advanced

Opens a dialog that lets you refine parameters for guide curves, such as projection direction, offset from the curves, and tolerances.

Containment Parameters

Type

Four types are offered:

Automatic

Delegates the containment strategy to the system.

User-defined

Provides a button (Containment curves) that opens a dialog where you manually select the curves instead of having the system choose them automatically.

Silhouette

Specifies a silhouette containment strategy but delegates details to the system.

User-defined and silhouette

Specifies a silhouette containment strategy and also provides a button (Guide curves) that opens a dialog where you manually select additional curves – for example, designating an area you want to exclude.

Roughing tab for Geodesic Machining

For Geodesic Machining, the Roughing tab offers the following subset of controls provided in other calculation strategies:

- Stock definition: See “Stock Definition Parameters” on page 267.
- Depth cuts: See “Depth Cuts” on page 274.
- Transform / Rotate: See “Transform/Rotate” on page 283.
- Sorting options: See “Sorting options” on page 285.

Utility tab for Geodesic Machining

The **Utility** tab presents several advanced controls to refine your toolpath.

- Feedrate control: See “Feedrate Control” on page 291.
- Axial shift: See “Axial Shift” on page 293.

Calculation Strategy: Deburring

In the Surface Paths page, the first option, Calculation based on, affects options displayed in this tab and others. If you choose Deburring, the 5-Axis process dialog changes its tabs dynamically, suppressing tabs for Gouge Check, Roughing, and Utility. Controls in tabs not mentioned below are covered in the material for each of the 5-Axis tabs generally.

- “Overview of Deburring”, next
- “Surface Paths tab for Deburring” on page 108

Overview of Deburring

After machining, burrs can be found on all parts that have straight edges or non tangent outer surface topologies, appearing whenever the tool chips metal off that edge. This can not only destroy the functionality of the part, it can endanger the worker. In almost all machining it is necessary to deburr the part after it is machined.

Manual deburring can take as much time as the entire automated processing of a part. The Deburring calculation strategy can greatly speed throughput by automating this last portion of the cycle, creating a deburring toolpath on the outer edges of part geometry. The position of the tool relative to the edge is always the bi vector between the two surfaces of that edge.

The system provides automatic feature detection, automatic linking, automatic lead in, and automatic collision avoidance. The goal is to create toolpath in a completely automated fashion just by selecting the part geometry.

Please Note: Only ball mill cutters are supported, and the geometry input (a mesh) must be in a good quality for feature detection to work properly.

Surface Paths tab for Deburring

When toolpath calculation is based on Deburring, the Surface Paths page offers the following types of settings.

- Geometry input: See Geometry Input, next.
- Path parameters: See “Path parameters” on page 109.
- Extensions/Overlap: See “Extensions / Overlap” on page 110.

- Surface quality: The same as for strategies other than Deburring. See "Cut tolerance" on page 148.

Geometry Input

Be meticulous in specifying geometry input to Deburring. Poorly defined geometry will prevent the automatic features from working properly.

Part surfaces

Click the ellipsis [...] button to open a dialog where you can select the surfaces for the deburring operation. Part edges suitable for deburring can be automatically detected, and part surfaces will be protected against collisions with the holder geometry.

Part surface selection is mandatory. You can optionally specify additional surfaces for collision checking – see Check surfaces, below.

Edge definition

Two edge-definition modes are offered.

User-defined

Lets you choose edges manually, using the ellipsis button for User-defined edges. Multiple curves can be selected, and only the selected curves will be deburred.

Autodetect

Instructs the system to extract all edges. All extracted edges will be deburred except the ones manually choose to exclude, using the ellipsis button for Exclude edges. Note: If automatic detection fails for a particular edge, then this edge canot be deburred even if you select it manually.

The Advanced button opens a dialog where you can refine parameters like Minimal sharp edge angle and Minimal detected edge length , and you can limit the detection area.

Check surfaces

When the checkbox is selected, you can click the ellipsis [...] button to open a dialog where you can specify additional surfaces that will be used in collision checking and specify a value for Check surfaces clearance. Check surfaces typically include fixtures and clamps that would interfere with the deburring.

Path parameters

In this section, you specify how the deburring cycle is to proceed.

Edge shape

Two ways of specifying edge shape are offered.

Constant width

The distance between the edges and the edges of the chamfer will be held constant.

Constant depth

The distance from the center of the ball mill cutter will be held constant. In this case, the width depends on the edge angle: the width for a shallow angle will be much bigger compared to an edge with a small angle. This becomes important when specifying a large value so as to create a chamfer.

Inner corners

Inner corners can be trimmed so that the toolpath will have a sharp corner or relief grooves can be added. (Relief grooves provide a smoother transition between areas that can be machined according to specification and areas that cannot be reached by the tool in inner corners.)

Relief grooves will only be applied to inner corners with equal edge angle on both sides of the inner corner.

Direction

Extensions / Overlap

Length

Normally set to `0`, but you can specify a positive value to extend a deburring pass, or a negative value to overlap it.

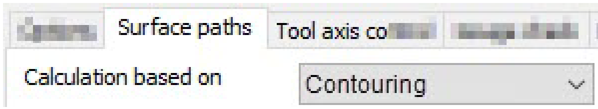
Calculation Strategy: Contouring

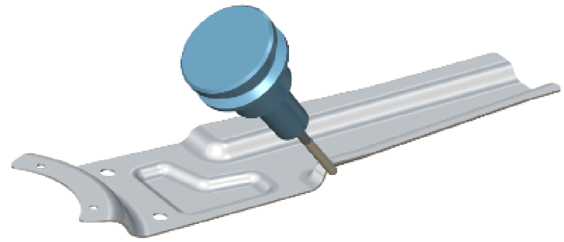
In the Surface Paths page, the first option, Calculation based on, affects options displayed in this tab and others. If you choose Contouring, the 5-Axis process dialog changes its tabs dynamically, suppressing tabs for Roughing and Utility, and adding one for Corners. Controls in tabs not mentioned below are covered in the material for each of the 5-Axis tabs generally.

- "Overview of Contouring", next
- "Surface Paths tab for Contouring" on page 111
- "Link tab" on page 238
- "Corners tab" on page 112

Overview of Contouring

5-Axis Contouring is a specialized strategy for edge trimming of thin-walled hulls and other thin materials.

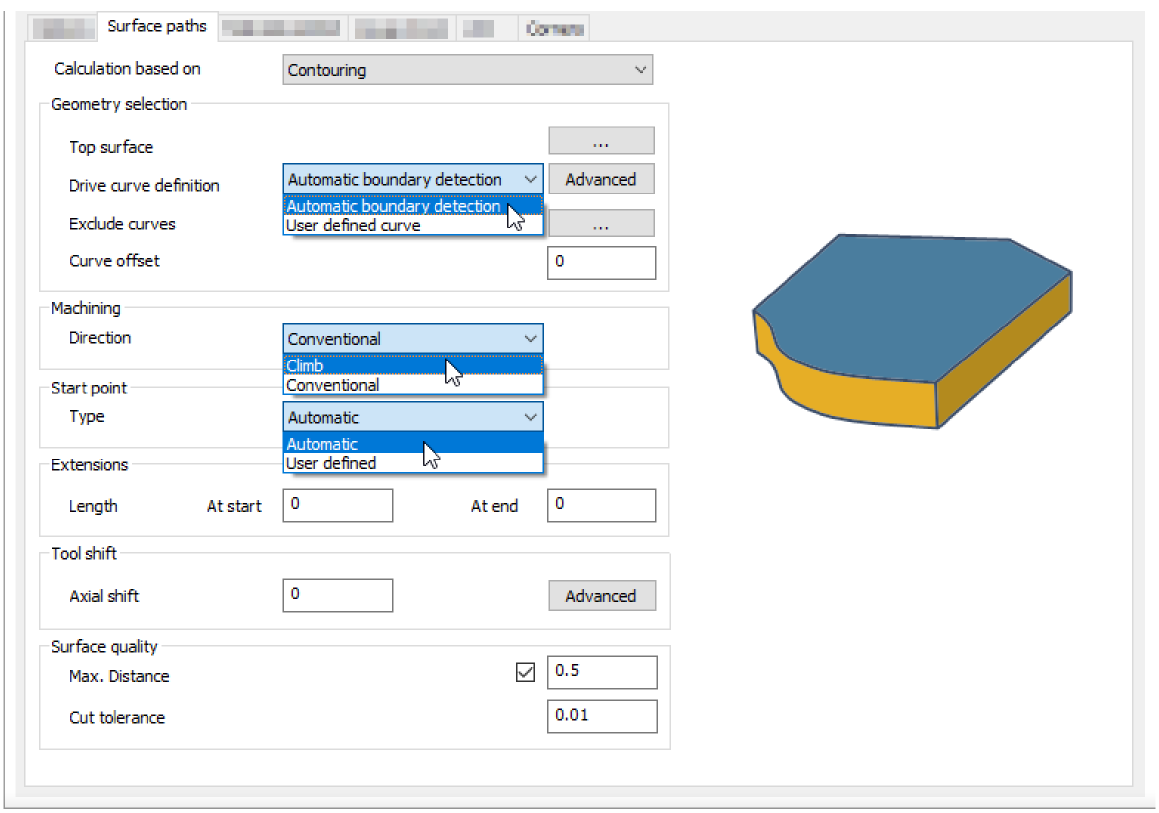




Surface Paths tab for Contouring

When toolpath calculation is based on Contouring, the Surface Paths page offers the following types of settings.

- Geometry selection: See Geometry Selection, next.
- Machining: Choose Climb or Conventional. See "Machining direction " on page 112.
- Start point: Either choose Automatic or else choose User defined and then specify X, Y, and Z values for the start point.
- Extensions: Normally set to `0`, but you can specify a positive value to extend At start and/or At end.
- Tool Shift: You can enter an axial shift offset to be added to the tool along its axis, or you can click Advanced to specify more detailed parameters in the **Tool shift advanced** dialog. For more information, see "Axial Shift" on page 293.
- Surface quality: The same as for strategies other than Contouring. See "Cut tolerance" on page 148.



Geometry Selection

Specify a Top surface by clicking the ellipsis [...] button and selecting a surface from the workspace.

For Drive curve definition, do one of the following:

- Choose Automatic boundary detection and optionally use the Advanced button to identify the automatic boundaries as outer curves and/or holes, as well as the ellipsis button for Exclude curves to select curves in the workspace that are **not** to be considered drive curves; or
- Choose User defined curve and then click the ellipsis button for User defined curves to select one or more curves in the workpace that **are** to be considered drive curves.

Relief grooves will only be applied to inner corners with equal edge angle on both sides of the inner corner.

Machining direction

For more details, see “Surface Paths tab for Contouring” on page 111.

Link tab for Contouring

The **Link** page offers controls for how the tool moves when it is not cutting, such as how the tool will move when entering or exiting the part (see “Entry/Exit” on page 238).

* - Some special-purpose calculation strategies offer a simplified interface with fewer controls: See the guides for *5-Axis MultiBlade* or *5-Axis Porting*.

Additionally, items on this page provide control over how the tool will move when it encounters air or check surfaces while cutting (see “Gaps Along Cut” on page 241), how the tool will move between stepovers (see “Links Between Slice” on page 245), and how the tool will move between passes (see “Links Between Passes” on page 250). The page also offers controls for defining clearance areas and clearances for feed and rapid moves (see “Retracts Dialog” on page 252), and powerful custom controls defining how the tool will move onto and off of the part (see “Default Lead-In/Out” on page 259).

Corners tab

When toolpath calculation is based on Swarf Machining, the Corners page offers the following types of settings.

Inside Corners

Options

Choose from Sharp corner, Round corner, or Relief groove.

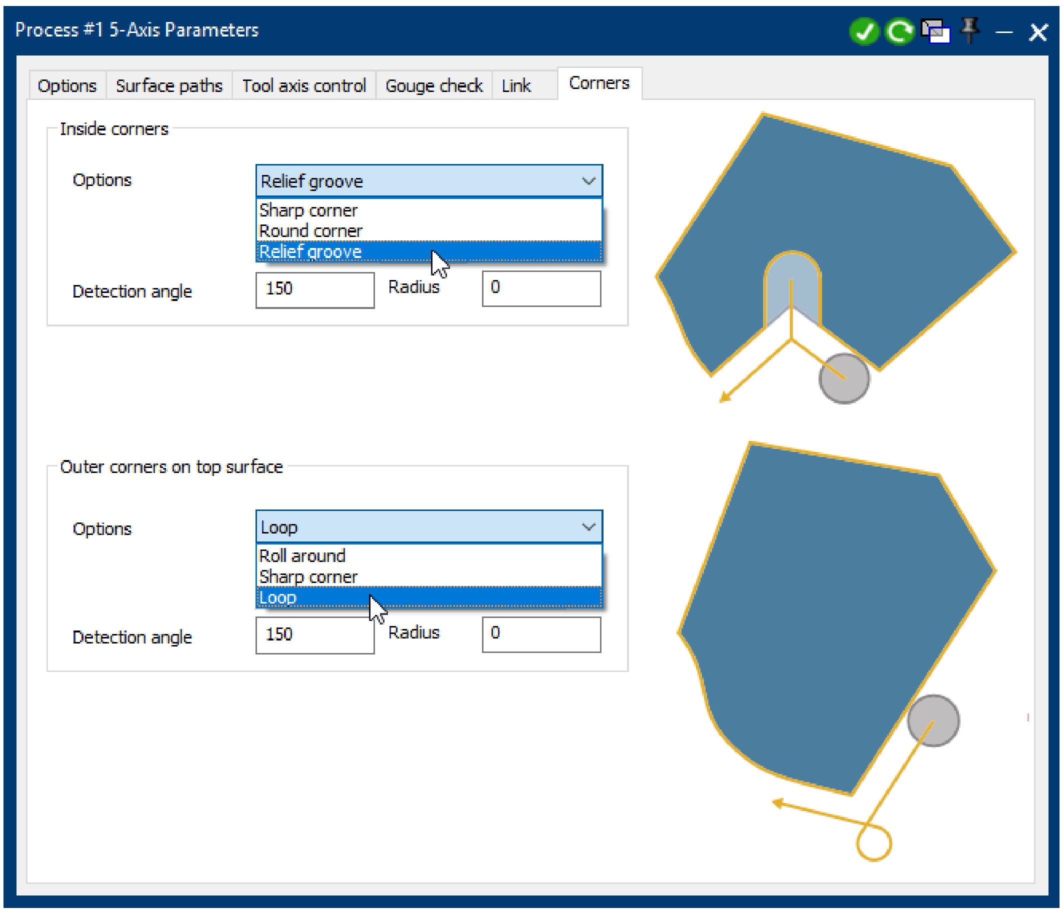
For Round corner or Relief groove, specify a Detection angle at which to consider it an inside corner. For Relief groove (pictured below), also specify a Radius for the groove.

Outside Corners

Options

Choose from Roll around, Sharp corner, or Loop.

For Sharp corner or for Loop (pictured below), specify a Detection angle at which to consider it an outside corner, as well as a Radius for the sharp corner or loop.



Area

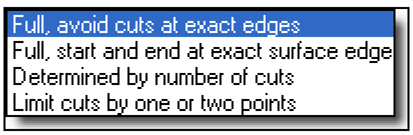
In the Surface paths tab with calculation based on Surfaces or Triangle Mesh, you choose a Pattern and set its options and values, and then you make choices for controlling the Area to be machined.

For calculation based on Surfaces:
First you choose how the area will be machined, using the entries in the Type menu; see Type , next. Then, you can select up to four Area options for limiting the area that will be machined; see “Area Options” on page 120.

For calculation based on Wireframe:
No Pattern choices are presented, and Area options are a subset of those offered by Surfaces; see “Area Options” on page 120.

For calculation based on Triangle Mesh:
Most patterns let you specify two to four Area options for limiting the area that will be machined; see “Area Options” on page 120.

Type



Generally the cutting area Type lets you define the area on the drive surface to be milled. Different options have advantages for different machining strategies. Choose your strategy from the pull-down menu:

- Full, avoid cuts at exact edges , next
- “Full, start and end at exact surface edges ” on page 115
- “ Determined by number of cuts ” on page 116
- “Limit cuts by one or two points” on page 117

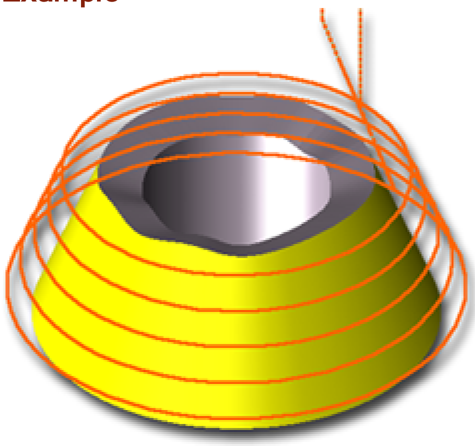
Full, avoid cuts at exact edges

Here the toolpath will be generated on the entire drive surface, avoiding the surface edges. Because the idea here is to avoid cutting at the exact surface edges, the distance that the tool cuts from the edges is always less than one half the step distance. This distance from the edge cannot be modified directly, but it will change as the step distance is modified.

This option is useful in cases where the boundary of the drive surfaces is not smooth, meaning that the edge of the surface is wavy or there are small gaps. If the surface edge is too wavy and the gaps are too big and the half of the stepover size is not enough to compensate, then the system recognizes a Gaps Along Cut instead (see “Gaps Along Cut” on page 241). They are handled on the **Link** tab; see “Link tab” on page 238.

When you are swarf milling, it might happen that the tool does not reach the edge at the end of the surface. This occurs because the remaining cuts after the first have exactly the maximum stepover distance and your surface usually ends somewhere between one cut. To mill these parts, you can set a value for axial shift (see “Axial Shift” on page 293). This value is an infeed to the tool position in axial direction. Please note that this axial shift will be set to every cut then. The value for Axial Shift is entered on the Utility tab; see “Utility tab” on page 291.

Example



In this picture, you can see that the tool does not start at the exact edge of the surface. Therefore, the wavy upper edge has no influence on the toolpath.

To see this example, open the file `Cutting Area - Type.vnc`.

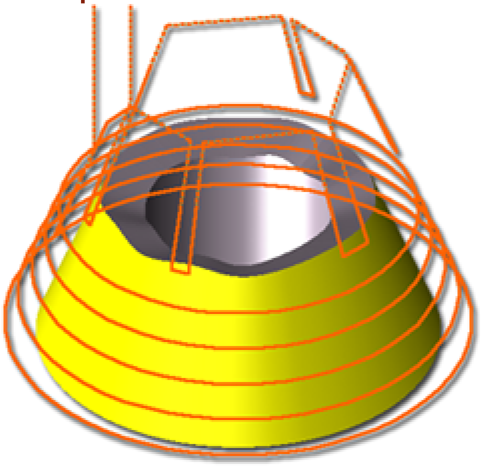
Full, start and end at exact surface edges

With this option, the toolpath will be generated on the whole surface and exactly to the surface edge or to the nearest possible position. Additionally, you can set a start and/or an end margin by clicking on the Advanced button. The margins must be positive. The start margin belongs to the first curve/surface and the end margin belongs to the second curve. For more information, see the example in "Stepover" on page 151.

The number of cuts depends on the stepover size. Because the first and last cuts are exactly on the edge and the distances between the cuts is equal, the number of cuts will be calculated with length of surface/maximum stepover. Therefore, the actual maximum stepover value is smaller than the value you set before.

> ⓘ Please be aware that this option is very sensitive to situations where either the edge of the surface is wavy or there are small gaps in the surface. These situations can cause undesirable retracts by the tool. This can be alleviated by using the "Gaps Along Cut" on page 241 option or by setting the cutting area to Full, avoid cuts at exact edges .

Example



In this picture, you can see that the tool starts at the exact edge of the surface. Because the first cut starts at the upper end of the wavy surface, you can see that this is not the best strategy. A better use here might be Full, avoid cuts at exact edges or setting a margin. Thus, at the end of the surface, the last cut is at the exact edge.

To see this example, open the file `Cutting Area - Type.vnc`. Changing the Gaps Along Cut option lets you see how the different clearance options will affect your toolpath.

Determined by number of cuts

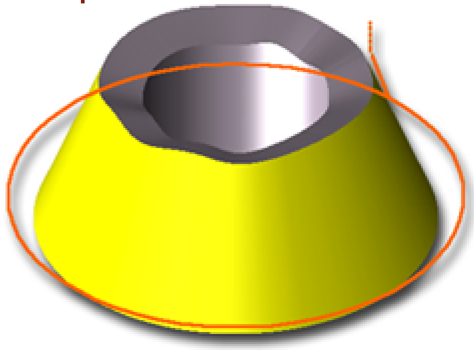
This option, which is not available for all toolpath patterns, allows you to set a defined number of cuts. It also provides a way to test the toolpath without generating a lot of cuts, thus saving toolpath computation time.

The first cut is at the exact edge, but it can be shifted with a margin that can be added by clicking on the Advanced button. The margins must be positive. The start margin belongs to the first curve/surface, and the end margin belongs to the second curve. For more information, see the example in “Stepover” on page 151.

In patterns such as Parallel to curve and Parallel to surface, you often need only one single cut to finish the whole contour, and so more than one cut is not necessary. In patterns such as Morph between two surfaces and Morph between two curves, the number of cuts is not clearly defined: with a morphed toolpath, the distances between the cuts at the end of the faces are very different.

When selecting this option, the Maximum step over parameter is hidden, because it is a result of the machined area and the number of cuts.

Example



In this picture, you see a typical application for Determined by number of cuts. You only have one single cut, but you can mill the final contour.

To see this example, open the file `Cutting Area - Type.vnc`.

Type Options

In the Area portion of the Surface paths tab, some choices for Type offer additional controls:

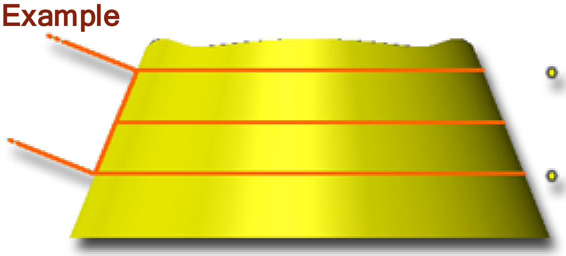
- Clicking the Set Points button opens the Limit cuts by one or two points dialog; see below.
- Clicking the Margins button opens the Margins dialog box; see "Margins" on page 118.

Limit cuts by one or two points

Setting Type to Limit cuts by one or two points allows you to limit the machining between two points, enabling you to work on only a certain area of the part's surfaces. When you click the Set points button, a dialog box appears that allows you either to define a point explicitly by entering XYZ coordinates, or to choose a point by clicking the [...] button. If the coordinates for the two points are identical, the toolpath will make one pass through that point only. Pressing one of the arrow buttons ("--->" and "<---") copies the coordinate from one side to the other.

> ⓘ You can only limit the toolpath if the points are beside the cuts along the toolpath.

Example

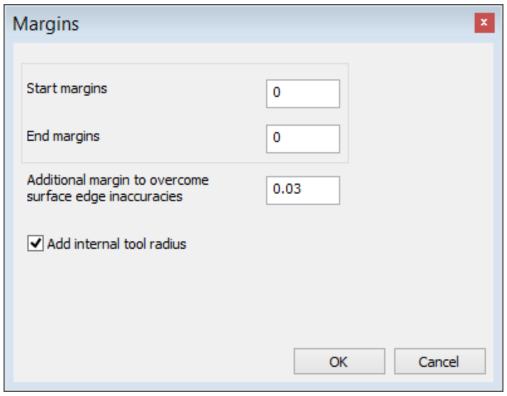


In this picture you see the machining only is in the center area of the surface, between the two points. The sample part also shows this option using through a point technique.

To see this example, open the file `Cutting Area - Type.vnc`.

Margins

The first cut starts at the edges of the drive faces in cases where you have set the Area Type (see “Type ” on page 114) to Full, start and end at exact surface edges or to Determined by number of cuts. The toolpath on the surface edge now has a defined position. With this position it is possible to define a certain margin from the surface edge for the first cut and the last cut. The Margins dialog lets you set an additional Start and End margin to overcome surface edge inaccuracies.



Additional margin to overcome surface edge inaccuracies

Toolpath strategies that use edge curves and surfaces sometimes encounter difficulties, because CAD systems deliver the drive surfaces and the edge geometry (curves or surfaces) only within some accuracy. If you wanted to start the toolpath *exactly* at `0` distance from the edge geometry, this would be problematic, because the geometry can never be exactly aligned. For this reason, an edge tolerance is used. The toolpath generated will be at the distance of surface edge curve tolerance plus the margin value entered by the user.

For example, to get a toolpath at 5mm distance, you can keep the surface edge tolerance at 0.03 and enter a margin of 4.97mm.

Add internal tool radius

For a pencil trace machining, it is necessary to have at least the tool radius margin to the leading curve or surface. When this checkbox is selected, the tool radius will be added on top to the margin and the additional margin.

Availability and Examples

Following are the combinations of patterns and area types that let you set margin values as well as examples of how they can be used.

Morph between two curves

This pattern lets you supply values for Start margins and End margins when the Area Type is set to Full, start and end at exact surface edges or Determined by number of cuts .

Example: Turbine with two floor faces.
An example could be a turbine blade with two floor faces. Although you are using Morph between two curves to limit the toolpath to the blade, you still have to be concerned about the

floor faces, which will be gouged if you just follow the bottom edges of the turbine blade. By setting a margin equal to the tool radius the tool will maintain that distance from floor faces and will not gouge. So always use at least the tool radius as margin to get the proper calculation of the tool center from the wall to floor. Please note that the start margin belongs to the first curve and the end margin to the second curve.

Parallel to curve

This pattern lets you supply a value for Start margins when the Area Type is set to Full, start and end at exact surface edges or Determined by number of cuts .

Examples: Electrode machining; Swarf machining.
An example could be electrode machining. The electrode has floor faces you don't want to gouge. When you set a margin of the tool radius, the tool always maintains a certain distance from the floor faces, so you would set the tool radius or greater as the margin value.

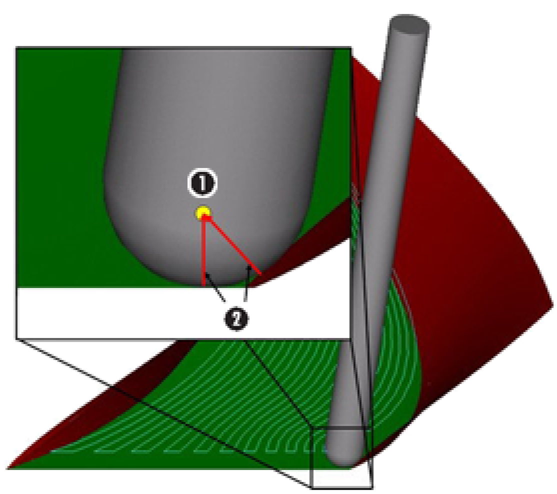
Another example is when you are swarf cutting and the wall surfaces don't come to the bottom edge curve along the entire perimeter, or if there are holes modeled in the wall. To avoid having gaps, you can set a start margin, then set an axial shift value that is the inverse of the shift to put the path back in the proper location.

Morph between two surfaces

This pattern lets you supply values for Start margins and End margins when the Area Type is set to Full, start and end at exact surface edges or Determined by number of cuts .

The distance between the margin and the first cut depends on the exact position of the surface edge. You can see in the picture below why this is so important.

Example: Impeller.
In this impeller example you have an inlaying edge. The toolpath has to fit in that edge to avoid a gouge. When you set a margin of the tool radius, the tool always maintains a certain distance from the floor face and blade face, so always use at least the tool radius as margin. Please note that the start margin belongs to the first surface and the end margin to the second surface.



1. Sphere Center
2. Start Margin

Parallel to surface

This pattern lets you supply a value for Start margins when the Area Type is set to Full, start and end at exact surface edges or Determined by number of cuts .

Example: Drive face plunges through a crossing check face.
An example could be two crossing faces where the drive face plunges through a check face. Because you do not want the tool to gouge, the machining must stop before the drive face plunges into the check face. When you set a margin of the tool radius, the tool has a certain distance from floor faces and it won't gouge. Therefore, always use at least the tool radius as margin.

Parallel cuts

This pattern does not allow you to specify values for start margins or end margins, but it is possible to set a value for Additional margin to overcome surface edge inaccuracies.

Area Options

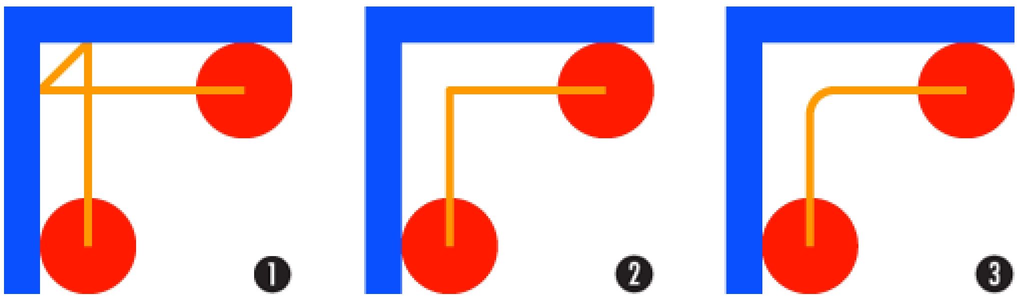
The Area portion of the Surface paths tab offers several optional checkboxes:

- Corner Cleanup (with calculation based on Surfaces or Wireframe) lets you deal with sharp corners. See Corner Cleanup, below.
- Extend/Trim (with calculation based on Surfaces or Wireframe) lets you control the ends of the toolpath. See “Extend/Trim” on page 121.
- Trim to flute length (with calculation based on Triangle Mesh for some patterns) lets you remove cuts that would extend deeper than the flute length. See “Trim to flute length” on page 122.
- Angle Range (with calculation based on Surfaces and most Triangle Mesh patterns) lets you control the machining area by surface normals. See “Angle Range” on page 123.
- 2D Containment (with calculation based on Surfaces, Wireframe and most Triangle Mesh patterns) lets you control the machining area by 2D shapes. See “2D Containment” on page 125.
- Rest rough (with calculation based on Triangle Mesh Rough pattern) lets you quickly clear unmachined areas left by a previous roughing. See “Rest rough” on page 127.
- Rest finishing (with calculation based on Triangle Mesh for several patterns) lets you control areas to be finished after a previous pass. See “Rest finishing” on page 127.
- Silhouette containment (with calculation based on Triangle Mesh for most patterns) lets you limit the machining area by the silhouette of the machining surfaces. See “Silhouette containment” on page 127.
- In the Triangle Mesh Projection pattern, you specify start/end points for a line and then values for start/end heights and angles. See “Area Options for Projection Pattern” on page 128.

Corner Cleanup

This choice lets you find small-radius areas and sharp inner edges in the surface model. Inside corners cause “fish tails” or “dove tails” in toolpaths; using this option helps you eliminate such unwanted motion. This option can also be considered to be a fillet generator. The surface model is rounded (filleted) in the direction of the toolpath slices with a radius to avoid small radii and inner sharp corners. The applied radius is the main tool radius plus the current stock value. The fillet generation is independent of tool type and shape. In most cases this option is used with a ball cutter, lollipop cutter or a conical cutter with ball tip. If swarf machining (side cutting) is applied then cylindrical or torus cutters can be used with this option.

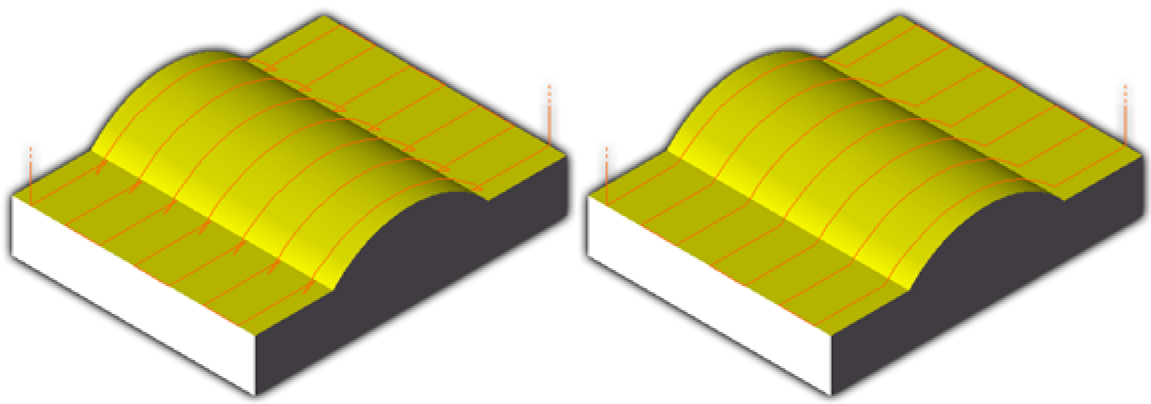
Clicking the Corner Cleanup button lets you apply an Additional Adjustment value, which must be positive, to the radius move in the toolpath. This makes the corner move value equal to the tool radius, plus the stock to leave, plus the additional adjustment to the radius.



Examples of: (1) Toolpath in a sharp corner; (2) The same toolpath with Chamfer Corners option active; and (3) With an additional adjustment to the radius added to the corner

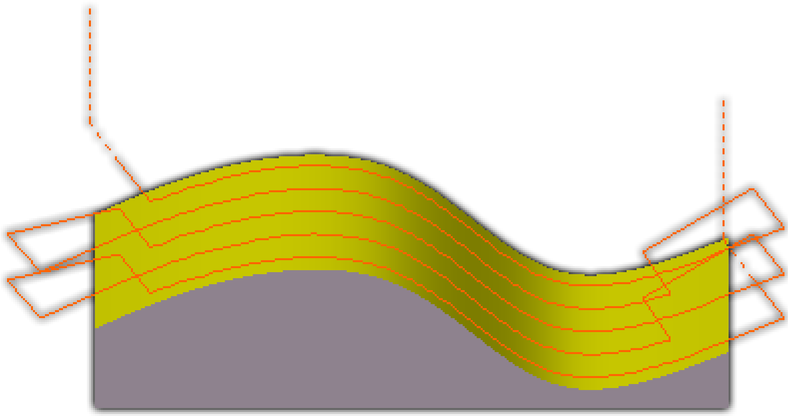
Example

Here you can see an example of a part with unwanted “fish tails” and the same toolpath with “Chamfer Corners” applied. To see this example, open the file `Cutting Area - Round Corners.vnc`.



Extend/Trim

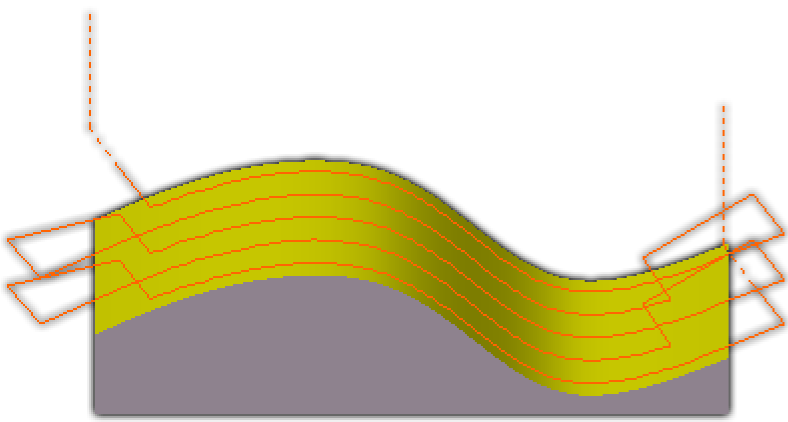
This choice lets you extend or trim the toolpath. The toolpath will be trimmed and/or extended tangentially to its orientation. When “extending”, the tool moves past the end or ends of the surface. When “trimming”, the tool does not reach the end of the surface. In the case of round or curved surfaces, the tool also leaves the surface tangentially but continues on straight. This function is useful if you do not want the tool to step over to the next cut while contacting the drive surface. When you enter a percentage of tool diameter distance greater than 50 percent, the tool tip extends past the surface and does not contact it at all during the step over to the next slice. The values you enter may be positive or negative numbers. The Extend/Trim gaps option allows the extend/trim settings to be applied to any gaps in the drive surfaces encountered along the toolpath in addition to the surface edges.



This function is similar to the Entry and Exit macros. The advantage of this function over the macros is this function provides better control with enclosed contours.

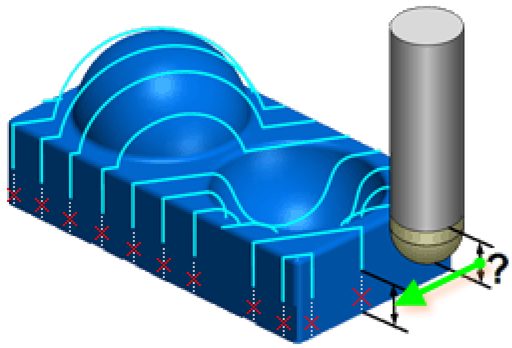
Example

In this picture you see toolpath extended at the start and trimmed at the end. To see this example, open the file `Cutting Area - Extend Trim.vnc`.



Trim to flute length

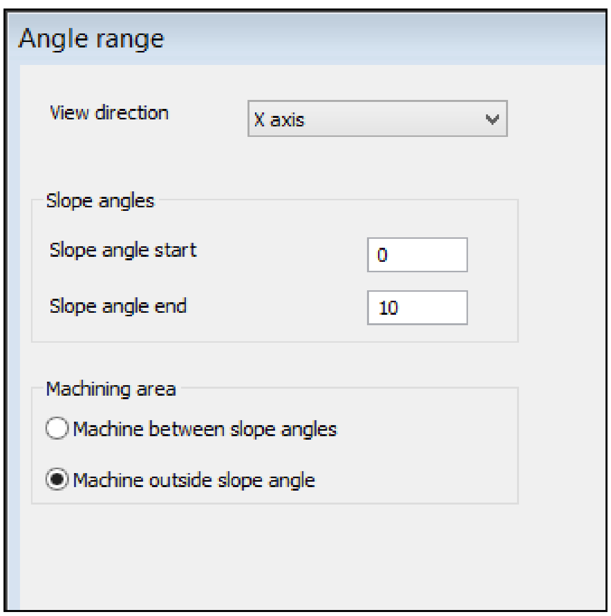
Available only for Triangle Mesh patterns Parallel cuts and Project curves. If this checkbox is selected, the toolpath is trimmed so that any cuts deeper than the flute length are removed.



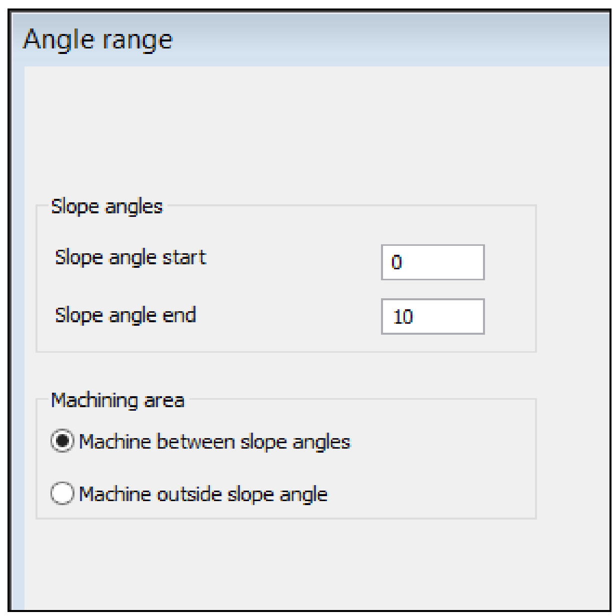
Angle Range

The definition of shallow and steep areas for a mold is obvious. For 5-axis machining on parts with undercuts and complex topology, the definition of shallow and steep areas is more abstract than the definition used for mold-making and 3-axis machining. This function lets you define the areas to be machined above the surface normal angles.

Angle Range (calculation based on Surfaces)



Angle Range (calculation based on Triangle Mesh)

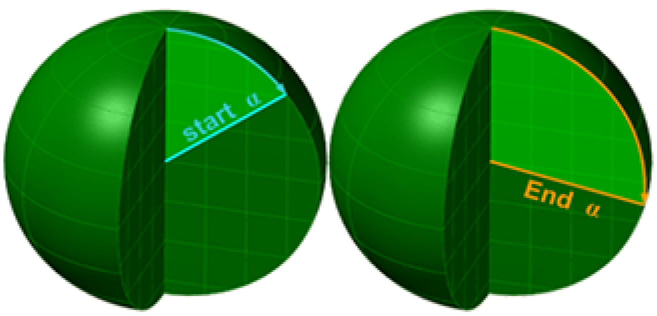


(Not all options are available for all patterns.) A distinction is drawn by the steep areas and shallow areas. The shallow and steep areas are defined by a view direction and two angles describing an angle interval. Then you have the ability to machine everything inside this angle interval or outside of this interval.

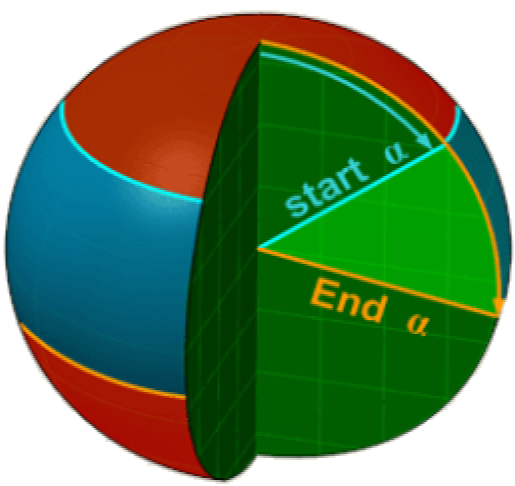
- The start angle must be smaller than the end angle. For example, the start angle might be 10° and the end angle 20°.
- If the stepover is bigger than the area you have defined between the start and end angle, then no toolpath will be generated.
- “Shallow” and “steep” calculation is based purely on surface contact points. In other words, some portions of the surface geometry are virtually trimmed in order to split the part into shallow or steep regions.

To set up this process for Calculation based on Surfaces, you must first choose the view direction from which steep and shallow will be defined. You can choose among X, Y, Z, or a user-defined direction. For example, choosing the Z axis will align the angles about the Z axis. If you choose the User Defined Axis option, the Select Axis button becomes available. Clicking this button opens the View direction window. This lets you define a vector.

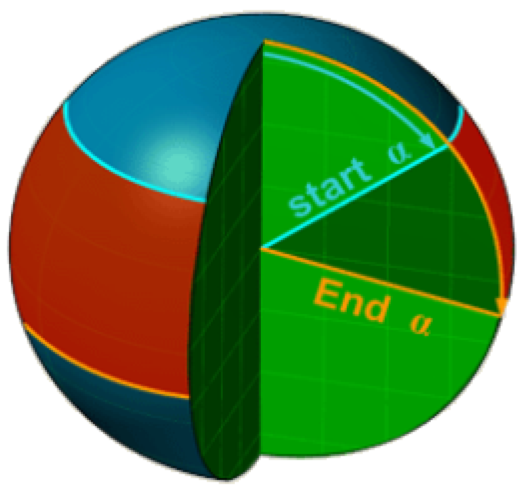
For all calculation types, you set the start and end slope angles. The start angle has to be smaller than the end angle. A good way to set up the angles correctly is analyzing what the surface normals are.



The last thing to do is to select which area will be machined. The area between the angles is the “steep” area; everything else is “shallow”.



Machining the “steep” angles



Machining the “shallow” angles

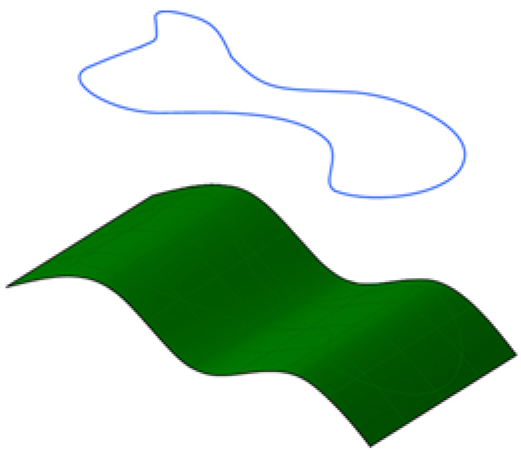
2D Containment

This choice lets you use a 2D shape to act as a machining boundary. You must select the containment curves (see “Drive Curves” on page 76) and an axis from which to project the curve. In typical 3-axis machining, containment boundaries are often used to define or limit the area where the tool should be cutting the material. There is a slightly different definition of containment boundaries for 3 to 5-axis machining. You can define 2D containment boundaries (multiple closed curves and nested shapes are allowed), and the drive surfaces are “virtually” trimmed by the given containment boundaries. Because the calculation is based on surface contact points, it is not guaranteed that the tool is actually “contained” within the given boundary. The projection axis direction is used to project the given 2D or 3D containment curves to the part and the part is “virtually” trimmed by the given curves.

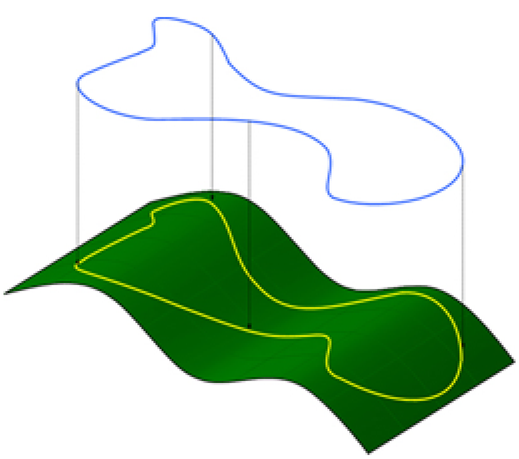
How it works

To use a 2D Containment, you need one or more closed shapes. The shapes may be nested.

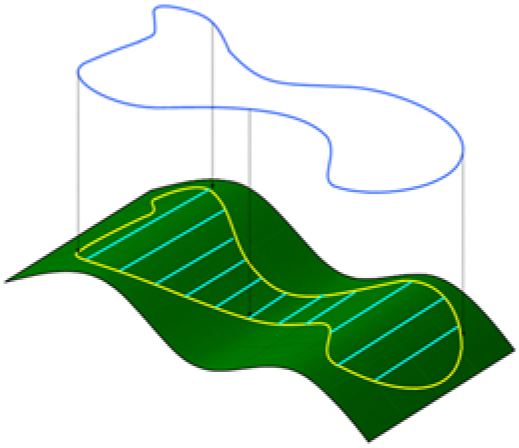
The contour may lie above the drive surface or directly on the drive surface.



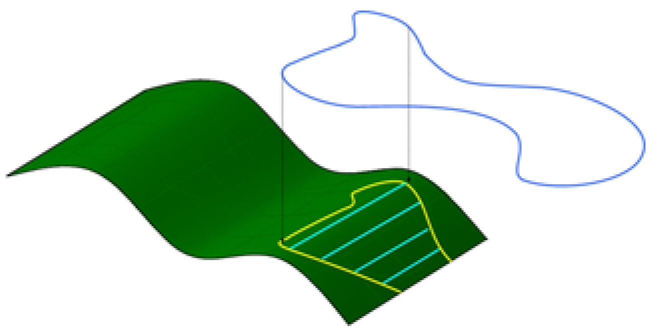
When the contour is not on the surface it will be projected to the drive surface. The projection axis is very important and the surfaces to be cut must be in that plane in some way. If they are not the contour will project improperly or not at all.



When you generate the operation, the toolpath is trimmed at the contour but the pattern is the same.

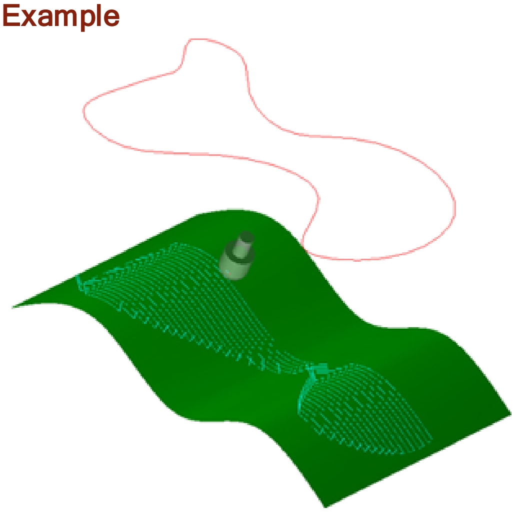


What happens if the contour is not entirely surrounded by the surface but only a part of the two overlap?



In this case, only that part of the contour that is above the surface will be projected. The toolpath only reaches the edge of the surface.

Example



In this picture you see that the toolpath is trimmed by the contour in Z projection direction. To see this example, open the file `Cutting Area - 2D Containment.vnc`.

Rest rough

This option is available only for Triangle Mesh, Rough pattern. It calculates a toolpath that will remove all unmachined areas left by a previous large roughing tool. The previous tool is used to identify accurately the areas on a 3D component by sweeping the diameter across the whole part being machined. Unmachined areas are thus identified and passed to the system, and the toolpath is calculated.

Rest rough toolpath does not require the whole part to be machined again. It will machine only those areas that are left out by previous tool. Intricate parts may require multiple rest rough toolpaths to remove as much material as possible before running semi finishing or finishing toolpath. In rest rough toolpath, you normally use a smaller step down as the cutter size reduces than the cutter used for the previous roughing tool path.

You can optionally set a value for Draft angle.

When you click Rest rough, the Rest roughing dialog offers these options:

Roughing tool diameter:
Defines the diameter of the tool that was used for the previous roughing operation. The previous tool is used to identify the areas on a 3D component by sweeping the diameter across the whole part being machined.

Roughing tool corner radius:
Defines the corner radius of the tool that was used for the previous roughing operation.

Roughing offset:
Allows you to compensate the offset being used in the main roughing cycle. The toolpath area will be extended by the value entered here.

Rest finishing

This option is available only for Triangle Mesh patterns Parallel cuts, Project curves, Constant Z, and Constant cusp.

Silhouette containment

This option is available only for some Triangle Mesh patterns, such as Rough, Parallel cuts, Constant Z, and Constant cusp. It lets you limit the machining area by the silhouette of the machining surfaces, created in the direction of the machining direction.

When you click Silhouette containment, a dialog offers these options:

Offset:
Expands the containment outward by the value supplied here, to extend the machining area.

Choices under Defined by:

Part silhouette:
Default setting. The area to be machined is limited to the tool centerline, which exactly follows the actual part shadow. The tool does not reach beneath that containment. Note that for shallow areas the tool does not reach the outer edge.

Part end height:
With this silhouette, unlike Part silhouette, the tool *does* reach the lower end of steep walls.

Tool contact:
Silhouette is determined by the tool contact point. For steep walls, the silhouette is the exact tool tip. For shallow areas, the tool reaches a bit over the tool tip in order to machine the complete surface.

Top of vertical walls:
Available only for pattern Constant cusp. In this silhouette, the machining stops at the top of the vertical walls. Only the shallow areas will be machined.

Bottom of vertical walls:
Available only for pattern Constant cusp. In this silhouette, the machining stops at the end of the vertical walls; thus, all shallow areas will be machined as well as the steep wall. Note that for the steep wall, the tool does not reach to the floor.

How it works:
A radial pattern is created on a 2D plane with user-defined parameters (Center point, Start radius, End radius, Angle, Stepover). The 2D radial pattern is then projected on the workpiece to create a toolpath.

Area Options for Projection Pattern

For the Triangle Mesh pattern Projection, you limit the machining area by selecting a line (or otherwise specifying start point and start/end height) and supplying values for start angle, end angle, and step angle.

Line start point:
This defines the line's position and orientation. You can specify the X,Y,Z values of the line's start point or you can click Select line, which opens a dialog that allows you to define the vector of the line or (by clicking the ellipsis button, ...) to select a line from the geometry.

Start height along line:
This defines the start position of the toolpath along the line.

End height along line:
This defines the end position of the toolpath along the line.

Start angle:
This defines the start angle position of the toolpath around the line.

End angle:
This defines the end angle position of the toolpath around the line.

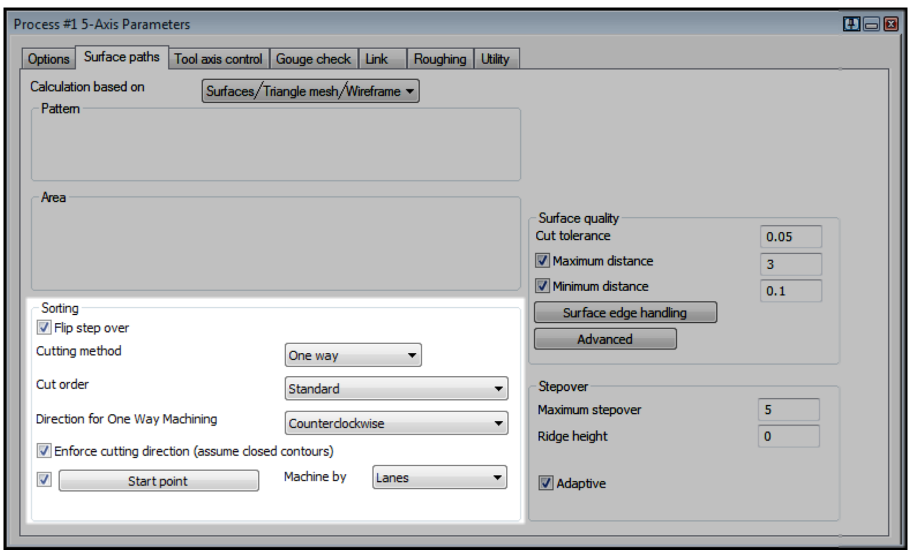
Step angle:

This defines the angle step between the projected line. Note that this value applies only to the Along choice of projection style.

Sorting Settings

The controls for setting Sorting options are located in the lower left portion of the Surface paths tab when calculation is based on Surfaces, Triangle mesh, or Wireframe. Not all controls are available for all calculation and pattern types.

- "Flip Stepover" on page 129 *(calculation based on Surfaces)*
- "Reverse Radial Sorting" on page 130 *(calculation based on Triangle Mesh)*
- "Cutting Method" on page 130
- "Cut order" on page 132
- "Direction for One Way machining" on page 132
- "Machine by Lanes or Regions" on page 138 *(calculation based on Surfaces)*
- "Machine by Levels or Regions" on page 139 *(calculation based on Triangle Mesh)*
- "Start corner" on page 140 *(calculation based on Triangle Mesh)*
- "Start point" on page 141

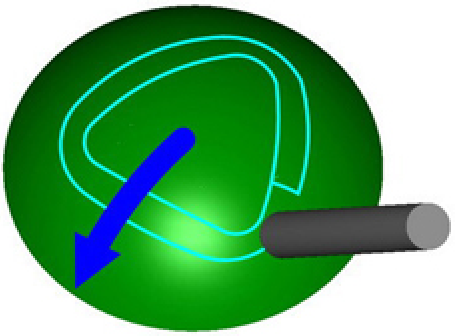


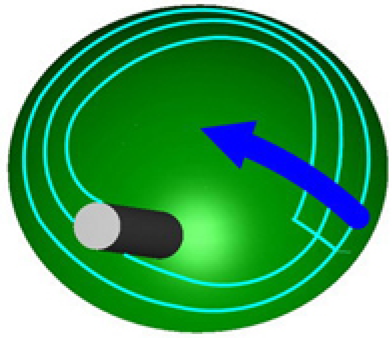
Flip Stepover

The Flip Stepover option changes the direction of the toolpath's cut sequence. This can change machining direction from the outside to the inside or from the left to the right.

Example

To see this example, open the file `Sorting - Flip Stepover.vnc`.





Here the machining begins at the top of the work piece.	By activating the “Flip Step over” option the machining begins at the edge

Reverse Radial Sorting

The Reverse option is offered for Radial-type cuts when calculation is based on Triangle mesh using the Project curves pattern. When the checkbox is selected, cuts will proceed inward, towards the center point.

Cutting Method

The choices for Cutting method allow you define how to connect from one cut to the next. The machining can be One way, Zig zag, or Spiral. Some choices might not be offered for some calculation types or patterns.

One way

With closed geometries, the tool moves always around the part in the same direction.

With geometry that is not completely closed, it is recommended to set the option Enforce Cutting Direction. This causes the surface to be machined like a closed contour.

With open geometry, the tool moves to the end of the drive surface, retracts with the Links Between Slice settings, and begins at the start of the drive surface again.

Zig zag

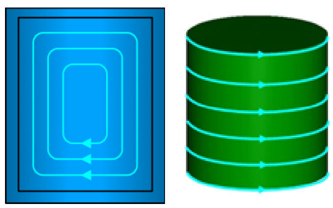
With closed geometries with every cut, the tool moves around the surface until the start point is reached. Then it steps over according to the Links Between Slice settings and continues machining in opposite direction.

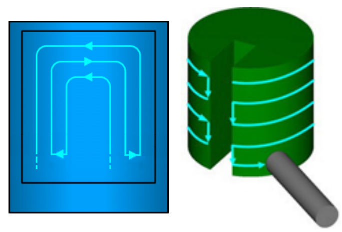
With open geometry, the tool starts at one end of the surface, steps over according to the Links Between Slice settings at the end of the surface and continues machining in opposite direction.

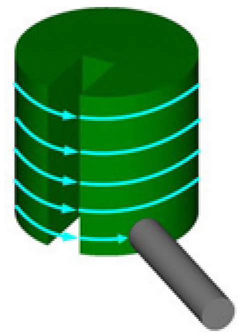
When used in combination with the tool axis orientation Be tilted relative to cutting direction and a side tilt angle (see Side tilt definition), then the tool has a constant orientation along its way. That means that the tool always keeps its absolute orientation.

Sometimes you need the tool to flip its orientation with every new cut. That means that the orientation of the tool is relative to the cutting direction. For this, activate Allow flipping side direction.

Example
To see this example, open the file `Sorting - One Way - Zig_Zag.vnc`.



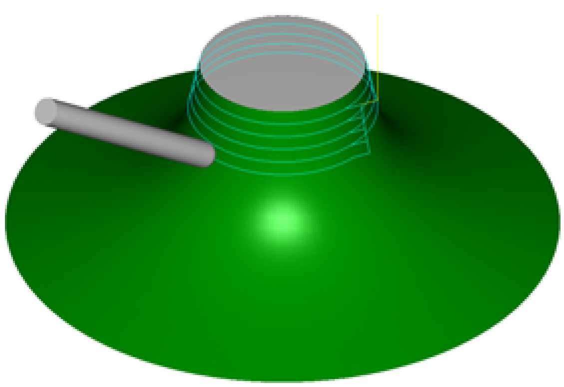


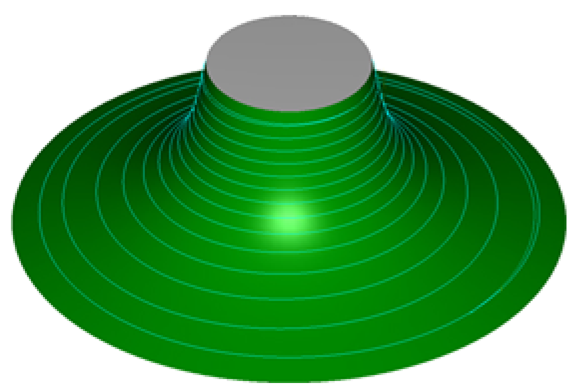


When Cutting Method is set to One way, the tool continues its cutting direction as it moves around the part.	When Cutting Method is set to Zig zag, the tool changes direction with every new pass.	When Cutting Method is set to One way with Enforce cutting direction, the tool continues in one direction, and gaps are ignored as toolpath attempts to follow a closed contour.

Spiral

This option will generate spiral cuts on your surface. This option can be used with all patterns and the spiral shape is projected back to the original surfaces. This helps to assure the requested surface tolerances. The first and last cut is parallel to the surface edge shape.





A parallel cut	A parallel cut with the Spiral method.

Advanced options for spiral machining

In this window you can set the toolpath behavior at the start and end of the spiral. You can close the spiral with a complete contour at the top (First contour) and/or bottom (Last contour) of the spiral. Additionally you may set the spiral mode to either Full Spiral (which is the default mode) or to Blend along distance, which requires a Blend Distance. The Blend along distance option will create a spiral only along the distance specified.

Cut order

The cut order defines the sequence of the cuts. There are several possible options, depending on the calculation type and pattern.

Standard
Standard sets a default cut order, usually from one side to the other.

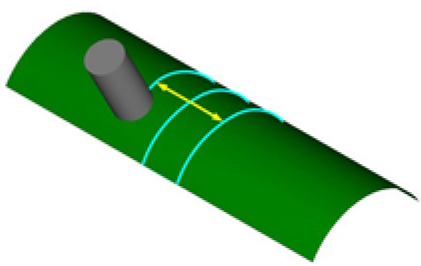
From center away
The machining begins in the center of the surface and progresses outward.

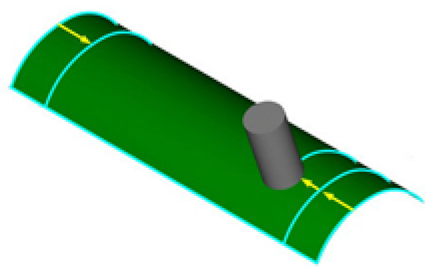
From outside to center
The machining begins from the outside of the surface and progresses inward.

From top to bottom
The machining begins from the top of the surface and progresses downward.

From bottom to top
The machining begins from the bottom of the surface and progresses upward.

Example
To see this example, open the file `Sorting - Cut Order.vnc`.





Cut order choice: From center away. The first cut is in the middle. The subsequent cuts proceed in alternate sides in an outward direction.	Cut order choice: From outside to center. The first cut is at the edge. The subsequent cuts proceed progressively inward.

Direction for One Way machining

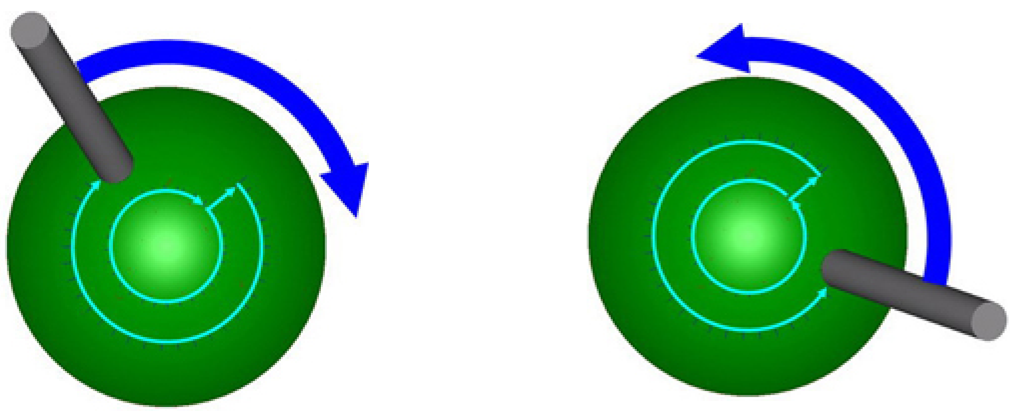
This pull-down menu lets you define the moving direction of the tool on the part. Two choices – Clockwise and Counterclockwise – are independent of the rotation direction of your spindle (see “How Clockwise/Counterclockwise Works” on page 135, below). Other choices depend on the spindle rotation direction (see “How Climb/Conventional Works” on page 133). For certain settings for Pattern and Cutting method, behavior is also affected by the chaining direction.

Direction for One Way Machining is intentionally unavailable if Cutting method is set to Zigzag. (However, a different set of choices is provided for calculation based on Triangle Mesh: Direction for closed cuts offers Climb and Conventional, and Machine by offers Levels and Regions.)

- When Conventional is selected, the tool movement is opposite to the spindle rotation. Conventional milling is preferred for the milling of castings or forgings with very rough surfaces.
- When Climb is selected, the tool movement and the spindle rotation have the same direction. Climb milling is preferred when milling heat treated alloys. It causes chipping in milling hot rolled materials due to hardened layer on the surface.
- When Clockwise is selected, the tool movement has a clockwise direction.
- When Counterclockwise is selected, the tool movement has counterclockwise direction.

ⓘ The choices Clockwise and Counterclockwise do not refer to spindle rotation. These choices determine whether the tool should move around a closed surface in clockwise or counterclockwise direction.

Example
To see this example, open the file `Sorting - Direction For One Way Machining.vnc`.



Clockwise machining direction around the part | Counterclockwise machining around the part

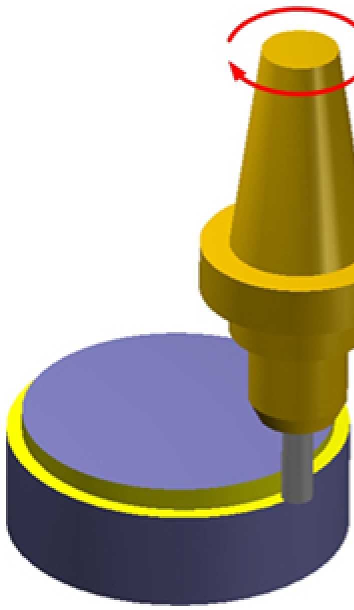
How Climb/Conventional Works

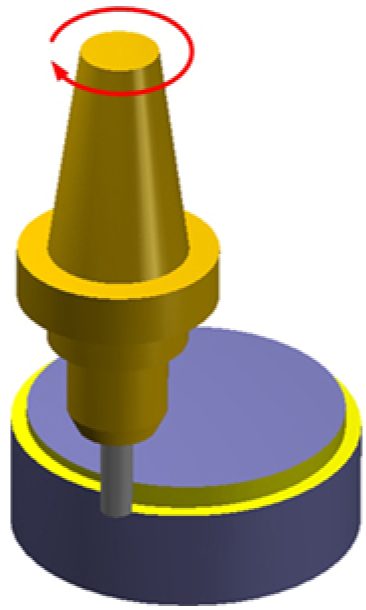
Conventional | Climb

When using Climb or Conventional it is best to choose the Be tilted relative to cutting direction axis control option. Other axis control options let the tool change between climb and conventional cutting as needed where Be tilted relative to cutting direction does not. By setting this option, the operation can only use one of two calculation routines based on only one factor – namely, the value of the side tilt angle of the cutting direction.

In cases where the side tilt angle is bigger than 45°

In this case, the machining will be recognized as swarf machining and the definition of climb or conventional is very easy. The spindle typically (except in a very few cases) turns clockwise. The tool movement is opposite to the spindle rotation. So you can say that if the tool is milling on the right side (relative to the moving direction of the tool), it is always conventional. When Climb milling is set, the tool movement and the spindle rotation have the same direction. The tool always machines on the left side.

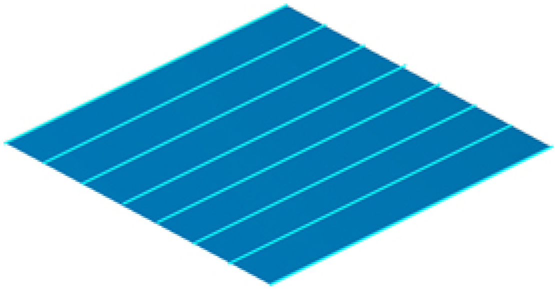




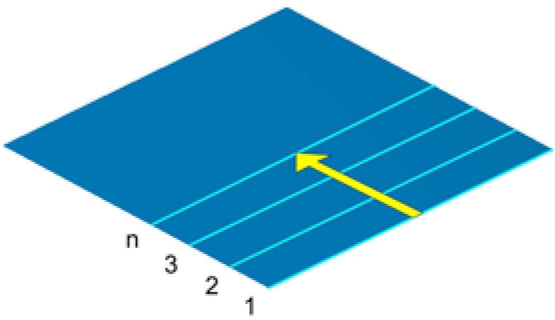
The tool moves counterclockwise while the spindle is turning clockwise. To the model, the tool is on the right side.	The tool and the spindle are turning clockwise. To the model, the tool is on the left side.

In cases where the side tilt angle is smaller than 45°...

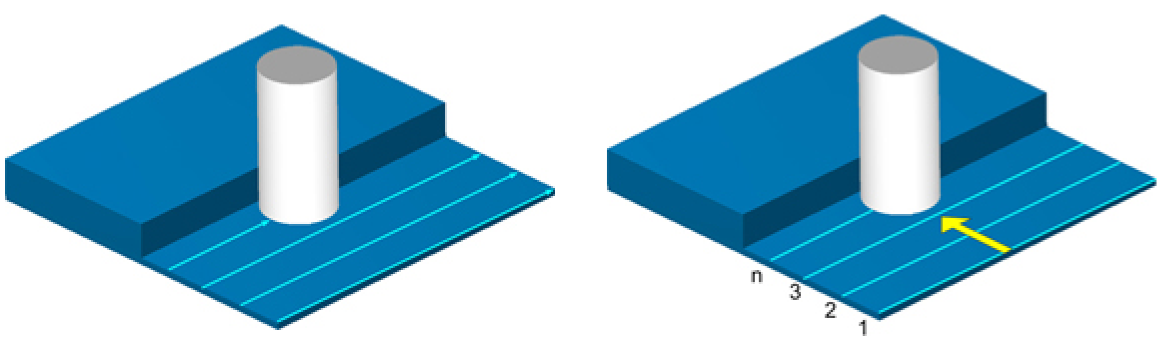
This situation is more complicated than with swarf milling. In this case, you do not have a swarf face and do not work with the side of the tool. You cannot define whether you are positioned right or left of the contour. Imagine that you are working on a flat face, machining simple parallel cuts. You do not know where to align the tool, because there are no side faces stating where the material is. Despite this, the function works. How is that? Let's look at the flat face again, where the paths are parallel to each other.



At first the only information we have about any direction is the direction of the cut sequence because the machining must start from one side.



With this information you know where your material is. With this you also know in which direction you have to move the tool to get climb or conventional cutting. The images below show how real machining would look. You can see that the side the material is on depends on the cut sequence. Since the spindle always rotates clockwise (for conventional cutting as shown in the picture), the tool has to move from the left to the right.

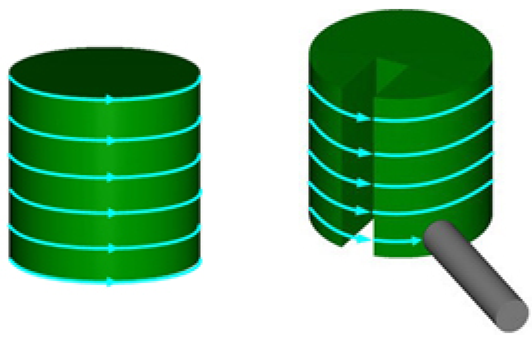


How Clockwise/Counterclockwise Works

The Clockwise option gives the tool movement a clockwise direction; Counterclockwise gives the tool movement a counterclockwise direction. Despite this seemingly straightforward definition, there are restrictions you have to consider because the direction cannot be clearly defined for every pattern strategy. Most important here is that for all strategies you have to have a closed toolpath. That means that a cut has to end where it started.

For open toolpath (non-closed faces that have a gap) you can force the toolpath to be closed. That means that you force the tool to move above the gap in the face. For information on link motion between two slices, see “Links Between Slice” on page 245.

Below, the image on the left shows a normal closed path on a contour, and the image on the right shows an open contour toolpath with an enforced cutting direction.

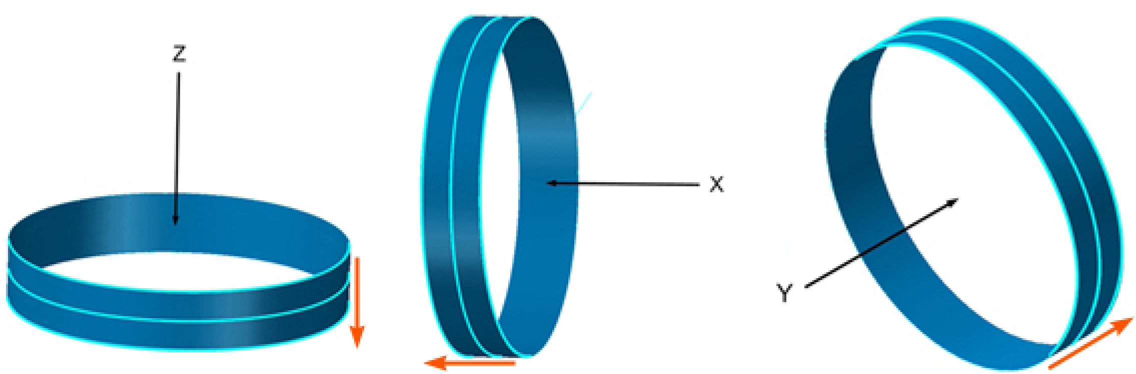


Determining “Clockwise” and “Counterclockwise” based on the Machining Pattern

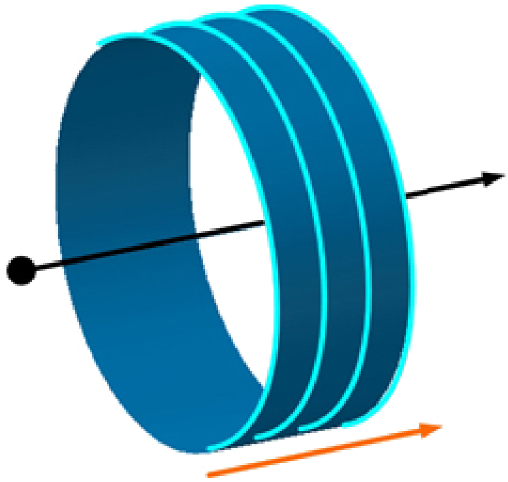
Parallel cuts (calculation based on Surfaces)

Here the direction is defined about the view direction onto the first cut. With this function you define two angles which span a plane. The cuts are parallel to this plane. Perpendicular to the plane is the direction from which the cuts start. This defines the cutting direction. With the Parallel cuts pattern, Clockwise and Counterclockwise always work as defined.

The following examples illustrate this. In the images below, the black arrows show the view direction (defined by the cut plane) and the orange arrows show the toolpath start direction.



Parallel cuts



Here the direction is defined with the curve and the cut planes. The chaining of the curve is important here. The chaining controls which side the toolpath starts on and where it goes to. This defines the view direction. If you start the chain from the other side, the machining would begin from the other side. With the Parallel cuts pattern, Clockwise and Counterclockwise always work as defined. In the picture the black point is the start of the chaining of the curve.

Morph between two curves

With this pattern type the direction cannot be clearly defined. Setting the direction to Clockwise or Counterclockwise does not always work as expected. There are two reasons for this.

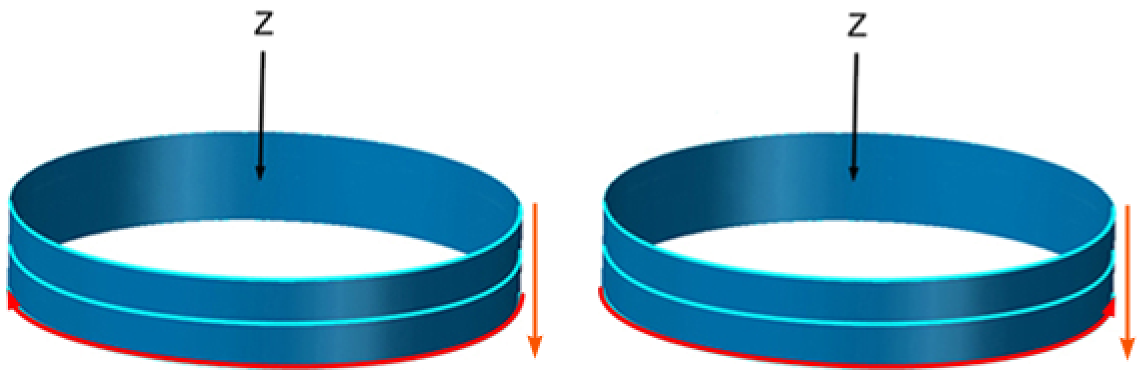
- The direction depends of the chaining of the curves. If the area to be machined has two curves, the system does not decide which curve defines the direction because both curves are equally important in the calculation toolpath calculation.
- Which curve is the first and which is the second is not clearly defined. You can define the cut start by selecting which curve is the first and which the last, but this has no influence on the cut direction.

Parallel to curve

With this pattern the direction is defined by the curve and the cut sequence direction. The chaining of the curve is important here. The chaining determines on which side the toolpath starts and where it goes to.

If in your closed contour the curve points clockwise, the Clockwise setting lets the tool run clockwise. If the chaining is counterclockwise and you set the direction to Clockwise, the machining will be counterclockwise; if you choose the Counterclockwise setting, the machining will still be clockwise.

In the following pictures, the cut sequence is top down (the orange arrow) and the view direction of the first cut is from the top. In the first picture, the curve (the red arrow) shows a clockwise chaining. In the second picture, the curve shows a counterclockwise chaining direction. When setting the direction parameter to Clockwise, the machining in the first picture will be clockwise; in the second picture, it will be counterclockwise.



The chaining direction determines clockwise or counterclockwise movement for the “Parallel to curve” machining pattern.

Project curve
For this pattern the direction is defined only by the curve chaining direction. So if in your closed contour the curve points clockwise, the Clockwise setting lets the tool run clockwise. If the chaining is counterclockwise and you choose the Clockwise setting, the machining would be counterclockwise. If you set here the direction to Counterclockwise, the machining would be counterclockwise.

Morph between two surfaces
For this pattern the direction cannot be clearly defined. Setting the direction to Clockwise or Counterclockwise does not always work in this case.

Parallel to surface
For this pattern the direction cannot be clearly defined. Setting the direction to Clockwise or Counterclockwise does not always work in this case.

Enforce Cutting Direction

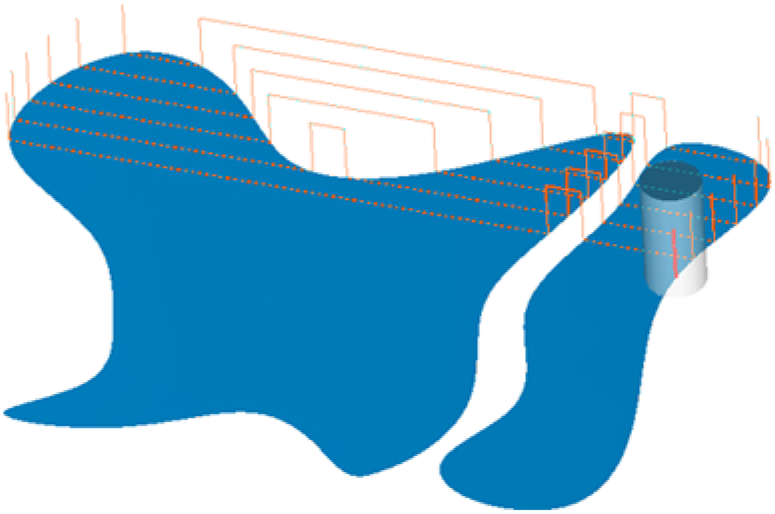
This checkbox is available for calculation based on Surfaces (except when Cutting method is set to Zigzag) or Wireframe. Selecting the Enforce Cutting Direction (assume closed contours) checkbox will force the toolpath to consider open contours (a shape with a gap) as closed. The tool will continue across the gap, and change of direction will not occur.

Machine by Lanes or Regions

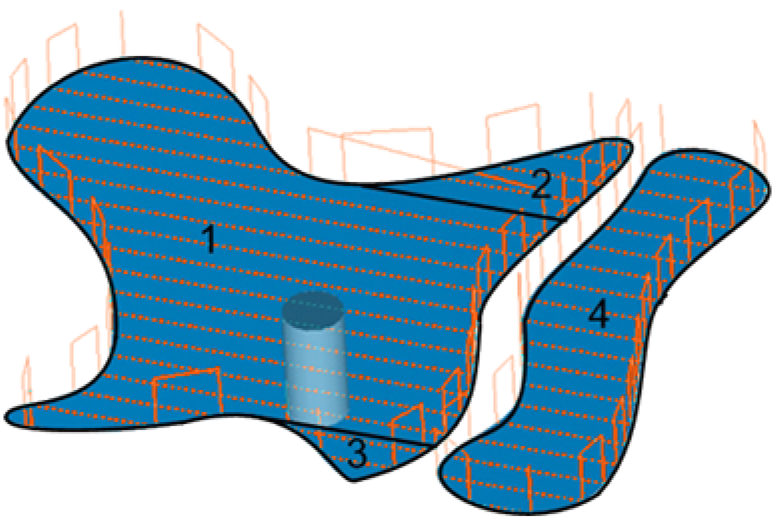
(This pulldown menu is available only for calculation based on Surfaces. For the similarly named pulldown for Triangle Mesh, see “Machine by Levels or Regions” on page 139.)

The usual toolpath that is generated has a topology of multiple contours (lanes) on the drive surfaces. This machining area mode tells the system to follow the machining by Lanes or by Regions. Machining by lanes is the default behavior. When toolpath is generated on many areas it may be desirable to break the toolpath into regions.

Take an example of a wavy shape that is parallel to the machining plane. The operation is set to use parallel cuts using the Lanes default behavior. During machining the tool retracts because of the gaps and because you are working on two separated drive faces. Generally the machining will be continued over all faces assumed as one big face.



When you sort the cuts by regions, the system divides the drive faces into individual regions and machines them successively. This is not limited to only between the separate faces, but even on the faces where you find gaps the machining will be separated into regions. So in this example, the cuts are organized into regions exactly where there were link jumps. In the end you get many regions to machine.



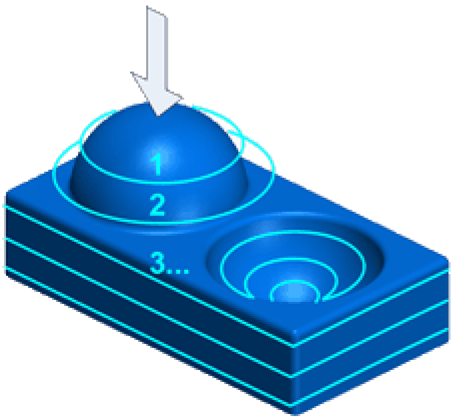
Machine by Levels or Regions

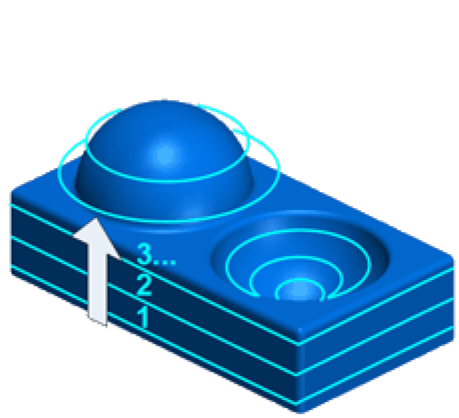
(This pulldown menu is available only for calculation based on Triangle Mesh. For the similarly named pulldown for Surfaces, see “Machine by Lanes or Regions” on page 138.)

When rough toolpath is generated, it usually has a topology of multiple contours. For models that have multiple contours or pockets, it might be preferable to machine regions independently to reduce air moves. This machining area mode tells the system to follow the machining by Levels or by Regions.

Start at

The Start at pulldown menu, available only when Pattern is set to Constant Z, lets you specify whether the machining levels or regions start at the Top and proceed downward, or start at the Bottom and proceed upward.

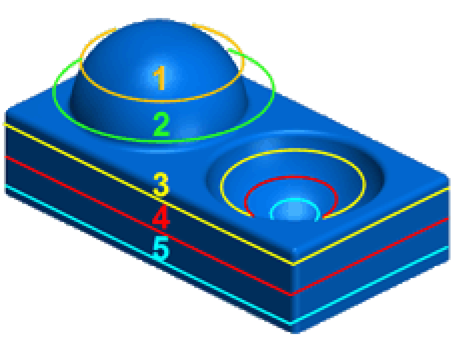


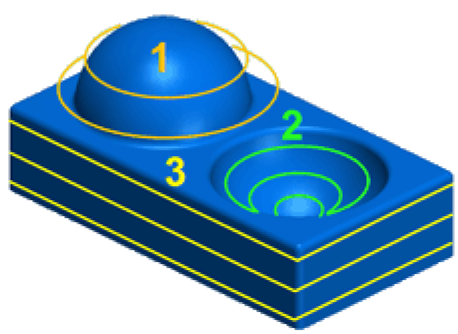


Start at Top	Start at Bottom

Machine by

The Machine by pulldown menu, available when Pattern is set to Rough or Constant Z, lets you specify whether to sort the machining passes according to Levels (and thus possibly making many moves from one pocket to pocket to another) or according to Regions (thus machining each pocket independently of the others).





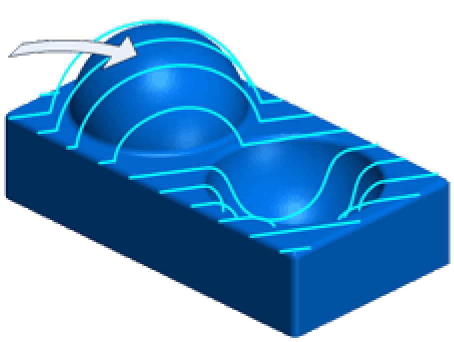
Machine by Levels	Machine by Regions

Start corner

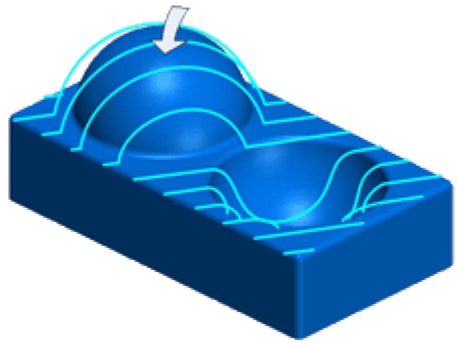
This pulldown menu is available only for calculation based on Triangle Mesh.

Start corner

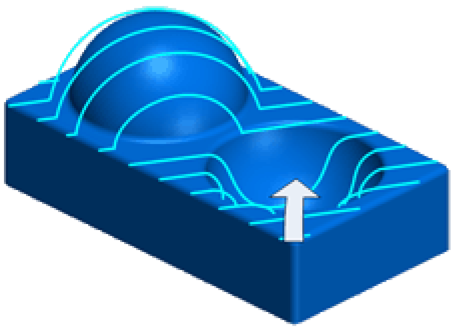
The Start corner pulldown menu, available only when Pattern is set to Parallel cuts, lets you specify which of four possible corners to start parallel cuts, as illustrated below.



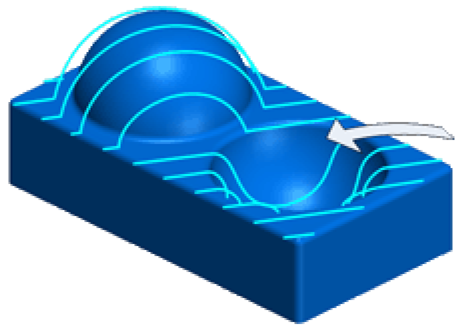
Start corner: Upper left



Start corner: Upper right



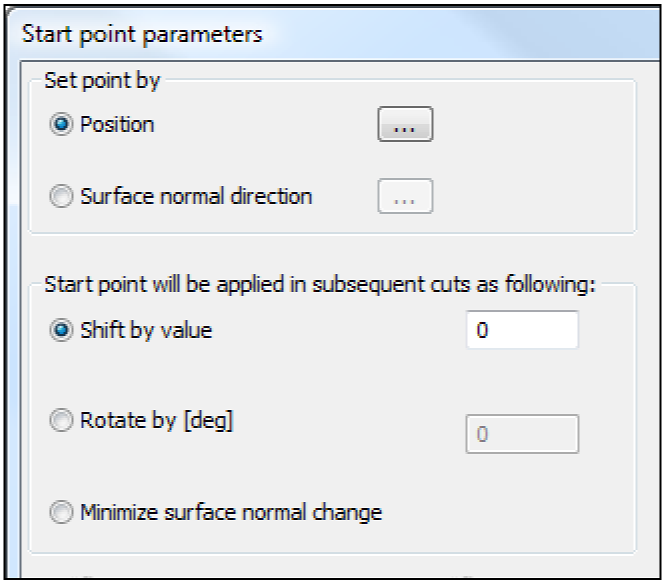
Start corner: Lower left



Start corner: Lower right

Start point

Using the Start Point option you can define a start position for the first toolpath slice. Selecting a start point does not change the machining order even if the selected point is closer to the last lane than to the first one. It is only set for the first contact point on the first calculated toolpath slice. If the order of the toolpath slices need to be changed, the Cut order option should be used.



ⓘ If you have selected One way in your cutting direction and Enforce Cutting Direction is also active, the new start point may not work.

Set point by

The start point can be set by specifying its Position. For calculation based on Surfaces, it can alternatively be set by specifying its Surface normal direction.

Position

This can be a point picked from your geometry or a point set with fixed values. The position values are absolute X, Y and Z coordinates. If the start point is not on the drive surfaces, then the closest surface point to your selected start point is used as the start position.

Surface normal direction

Available only for calculation based on Surfaces. The start point will be defined by a vector. That point of the toolpath which has its surface normal direction closest to the vector defines the new start point.

Start point will be applied in subsequent cuts as following

There are up to three methods by which the start point can be applied.

Shift by value

*Not available for calculation based on Wireframe.*This option defines the start position for subsequent toolpath slices. The start position will be incremented along the path by this value for each slice. This will help eliminate witness marks. Shifting does not work on open contours.

Rotate by [deg]

Available only for calculation based on Surfaces. This option also defines the start position for subsequent toolpath slices, but the shift distance is defined as an incremental angular amount. The angle values are relative to the previous cut. This will help eliminate witness marks. For example, when machining a cylinder the start point can be rotated 3 degrees for each toolpath slice in order to shift the mark on the part generated due to tool step over from one slice to the next one. Rotate does not work on open contours or flat surfaces.

Minimize surface normal change

Available only for calculation based on Surfaces. This option is used for blades or turbine blades machining. In these cases it is desirable to have the start point at the small radius of the wing. Unfortunately the toolpath's start points move and leave the edge position. To avoid this the start points can be forced to always be at a position with the same surface normal direction, thus the start points will always stay at the edge.

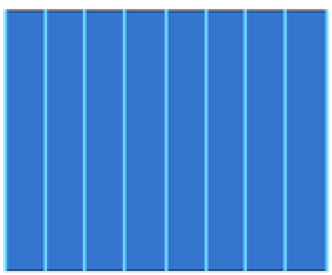
With this option the system will attempt to minimize any changes in the tool's angle of approach by finding a surface normal that is as close to the start position as can be found on the part. For example, multiple slices on a cylinder would all use the same surface normal. On a part with a flowing surface the surface normals can be very different at each slice. Minimizing the angle of approach can be very useful if there are fixtures or clamps on the part.

How it works

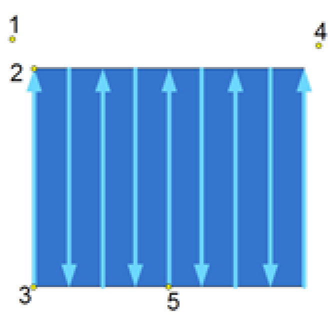
Following are different scenarios with combinations of cutting methods, and surfaces with open or closed contours to help you understand how this works.

Scenario #1

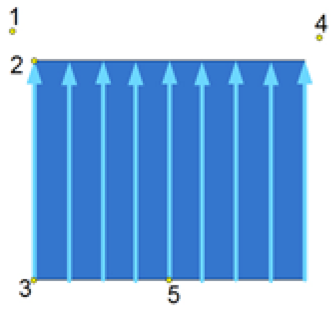
This scenario shows an open surface contour with parallel cuts.



The default start point is defined by the toolpath pattern. The picture shows that the Cutting Method is set to Zig zag and the machining starts at position #3 by default.



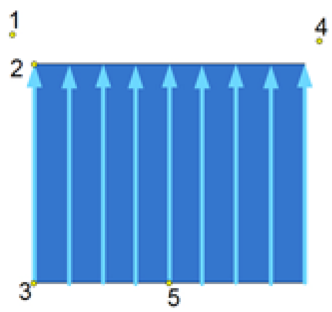
If a new start point is set at position #5 then the machining will start from the initial start point - position #3. The new start point cannot change the order of the cuts. So the machining won't start from the middle.



If a new start point is set at position #1, 2 or 4 the starting point for the machining will be at position # 2. The machining direction swaps and starts from the opposite side. Again, a new start point doesn't change the cut order, just the start position on the initial surface edge.

Scenario #2

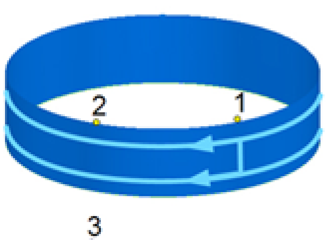
This scenario shows the same surface but this time the Cutting Method is set to One way. In this case it does not matter which start point is picked, the start point is always at position #3.



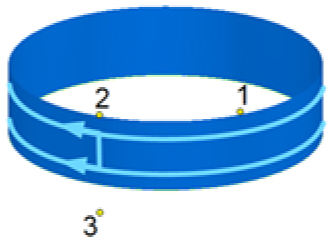
Scenario #3

This scenario shows a closed surface contour with parallel constant Z cuts. The cuts are closed contours, meaning that the cuts end where they started.

The Cutting Method is set to One way. The standard start point is at position #1



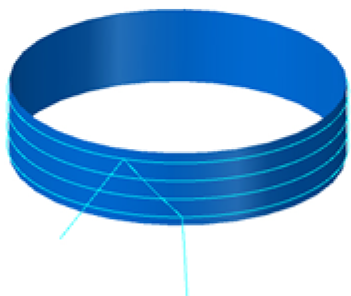
The new start point is set to position 2. Accordingly the start point moves to position #2



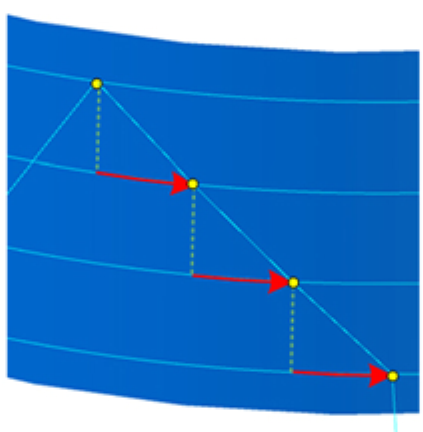
If the new start point is set to position #3 then machining would start at position #2. The reason is the same as before: when the start point is changed only the start position on the initial start edge will change, not the cut order.

Scenario #4

This scenario shows a closed contour with constant Z cuts in one way. The Shift by value is set to 1 mm. In this case the start point is shifted by 1 mm at every complete rotation.

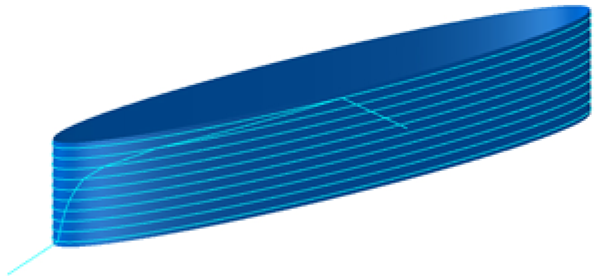


The shifting is done along the toolpath. In the picture below the red arrows are the shift and the yellow dots are the start points.

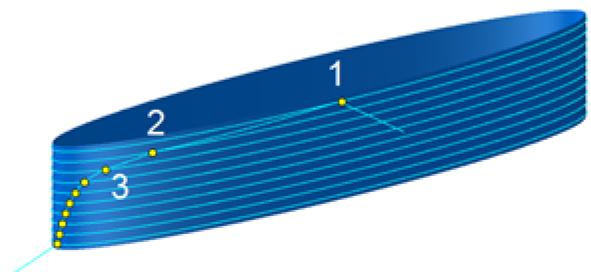


Scenario #5

This scenario shows a closed contour with constant Z cuts in one way. The Rotate by [deg] is set to 5 degrees. In this case, the start points for subsequent slices are shifted by 5 degrees.



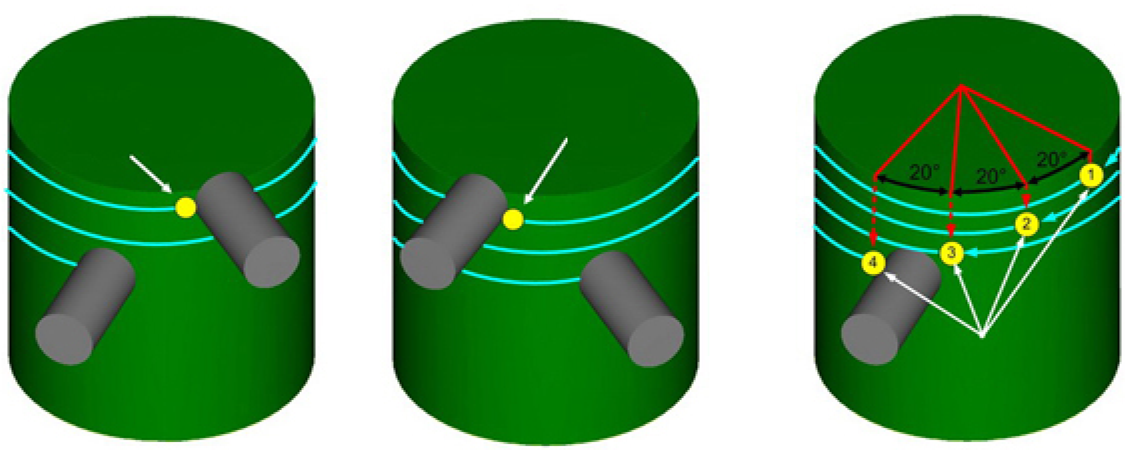
Each new start point position will be calculated based on the surface normal direction. This means that in the area where the surface radius is very large, the start points are more spread out (positions #1, 2 and 3). In the area where the surface radius becomes smaller, the rate of surface normal change is larger which results in start positions that are much closer to each other.



If the radius of the surface is infinite, that means the surface is flat. A rotational start point will not work in this case.

Example

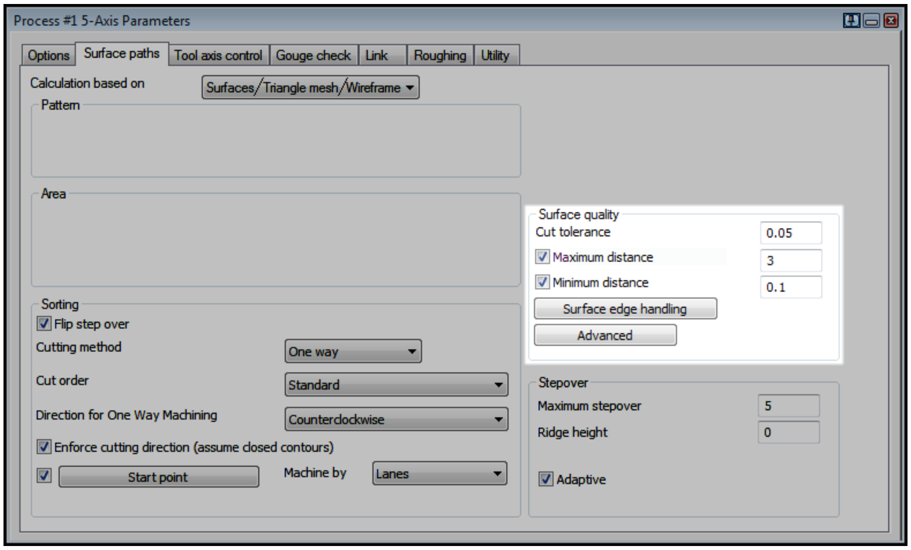
To see this example, open the file `Sorting - Start Point.vnc`.



Default toolpath start position	Toolpath start position with new start point	This is a toolpath start position with a new start point and a 20 degrees rotation angle.

Surface Quality

The controls for setting Surface Quality options are located near the lower right portion of the Surface paths tab when calculation is based on surfaces, triangle mesh, wireframe, or swarf machining. Not all controls are available for all calculation and pattern types.



The Surface Quality options control the toolpath's approximation of the cut surface:

- The value for Cut tolerance (see "Cut tolerance" on page 148) is the basic tolerance for toolpath accuracy.
- The value for Maximum distance (see "Maximum distance" on page 148) can create an extremely close approximation on flat surfaces by ensuring that no section of the toolpath will be more than the value entered.
- The value for Minimum distance specifies the minimum separation between successive passes. Available only for some triangle mesh patterns.
- The Surface edge handling button (see "Surface edge handling" on page 149 lets you make an open path into a virtual closed path. This button is available only when calculation is based on surfaces.
- The Advanced button (see "Advanced Button for Surface Quality" on page 150) gives you several controls for handling how toolpath is generated based on the surface quality. This button is available only when calculation is based on surfaces.
- When calculation is based on swarf machining, you set the value for Maximum angle step here instead of in the Tool axis control tab; see "Maximum angle step" on page 156.

Cut tolerance

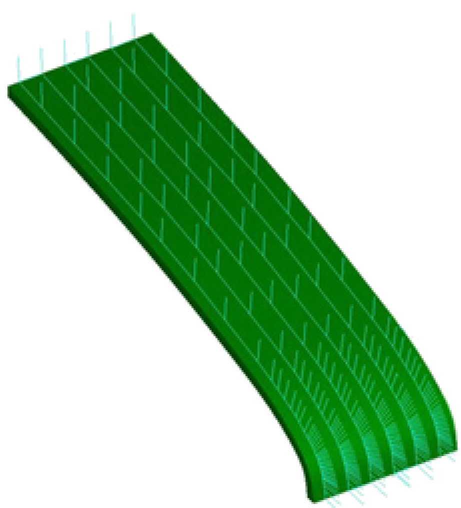
The Cut tolerance is the tolerance for the accuracy of the toolpath. This value is the chordal deviation of the toolpath against the surfaces to be machined. In other words, the toolpath can have a maximum error to the surfaces in the range of plus or minus cut tolerance.

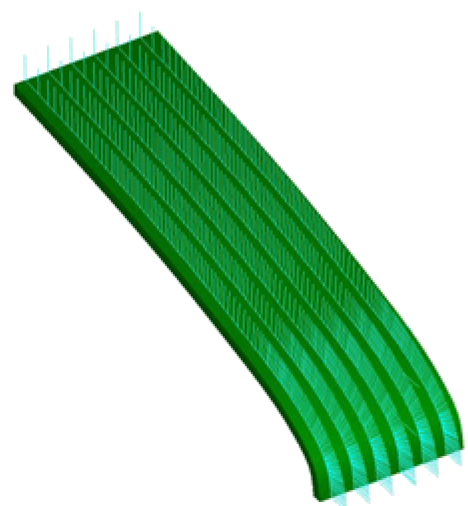
A small cut tolerance gives you more path points on the drive surface. Thus, the generated toolpath is more precise to the surface contour. The result of the machining is a very good surface quality that requires more time to calculate.

For many calculation strategies and patterns, you can generate even more path points by setting a value for Maximum distance (see “Maximum distance” on page 148), for Stepover (see “Stepover” on page 151) or for Maximum angle step in the Tool Axis Control tab (see “Maximum angle step” on page 156).

Example

To see this example, open the file `Surface Quality - Cut Tolerance.vnc`.





A large cut tolerance generates fewer points on the toolpath. After machining, the surface is rougher. Therefore the calculation time much faster.	Small cut tolerance

Maximum distance

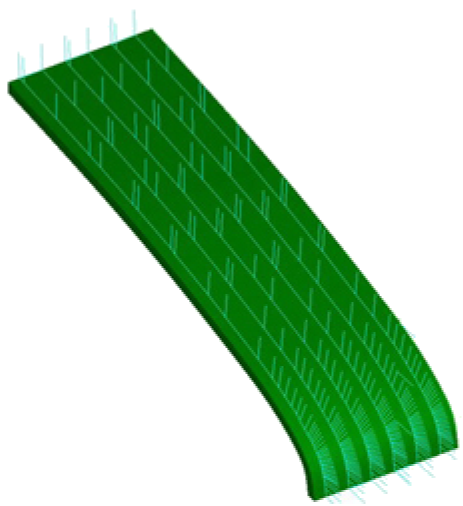
Depending on the value for Cut tolerance, you will have many or relatively few points on the surface. This is especially true for round surfaces where you have more points because the toolpath always changes direction. For more points on flat surfaces, select the Maximum distance checkbox. Smaller values will generate more points; although the Cut tolerance is the same, you get more points on straight or flat surfaces because the distance is a maximum user given distance to each other. For example, if this option is activated and the distance is set to 0.5mm, then at every 0.5mm (or less), a new toolpath position is calculated on the surface. If set, the value must be greater than 0.

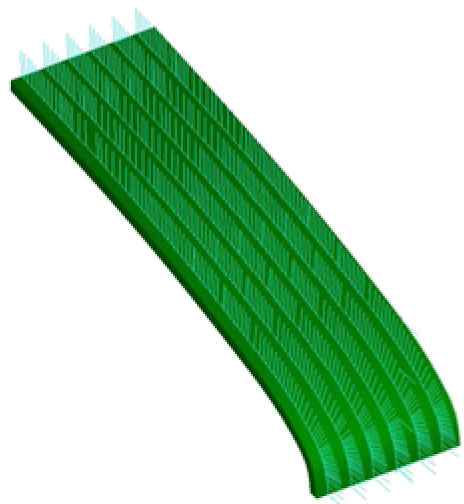
Depending on the Cut tolerance and "Maximum angle step" on page 156 values, then some toolpath segments may be closer than the value set for Maximum distance.

If Maximum distance is unchecked, then the toolpath positions will be influenced only by the values for Cut tolerance and Maximum angle step.

Example

To see this example, open the file `Surface Quality - Cut Tolerance.vnc`.





Result without distance	Result with distance.

Surface edge handling

This lets you create single longer surfaces where there may otherwise be gaps. Surface paths are created on individual surfaces and are then merged together to longer surface paths. The decision about how to merge the paths is based on a merge distance. If all surface paths on a toolpath slice are merged the system checks to determine if a closed surface path can be built by connecting the start to the end. The same merge distance value is used for deciding this. So all surface paths that are within this value will be merged together.

Maintain outside sharp edges

With this option enabled the system can maintain a sharp edge at the outside of two intersecting surfaces. This is achieved by extending the toolpath with a loop that leaves and enters the faces tangentially.

Sharp edges detection angle

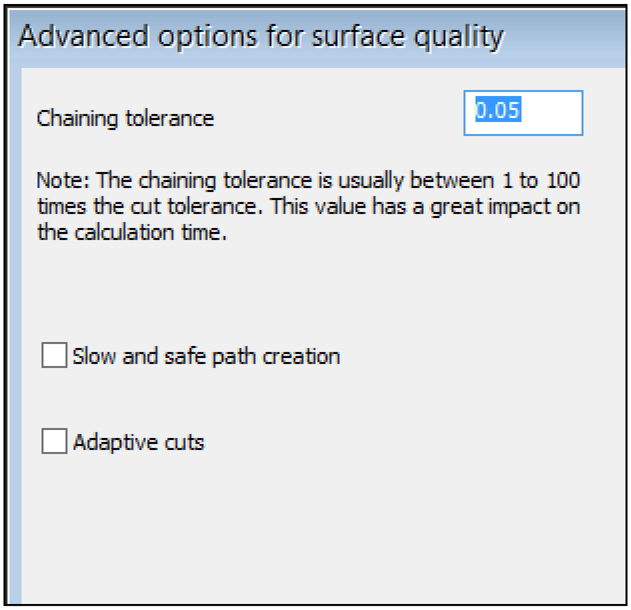
Whether a loop is created or not depends upon this angle threshold. A value between 0 and 360 degrees is valid.

Radius for loops

This value specifies the size of the loop.

Advanced Button for Surface Quality

In the Surface quality area of the Surface paths tab, the Advanced button opens the Advanced options for Surface Quality dialog box.



Chaining Tolerance

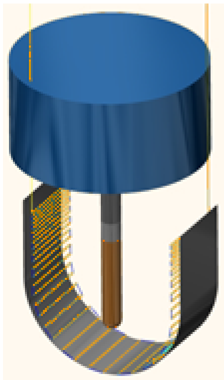
The value for Chaining tolerance is an internal value for toolpath generation. It should be set to 1 to 10 times the cut tolerance. If you have untrimmed simple surfaces, then this value can be set to 100 times of the cut tolerance, drastically increasing the calculation speed.

Slow and safe path creation

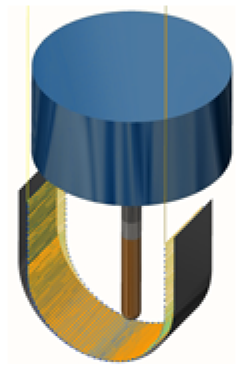
When toolpath is being generated surfaces are analyzed using a grid. When the toolpath topology becomes very complex (being parallel to the curve or very large surfaces) it can become inaccurate. When the Slow and safe path creation checkbox is selected, a finer grid is applied to the surface, based on the Stepover tolerance. This results in slower but more accurate results for surface contact points.

Adaptive cuts

Adaptive cuts can be used to provide a constant stepover when using a cut type of Morph between two curves, Morph between two surfaces, Parallel to curve, or Parallel to surface. Due to the way the morphing algorithm works, the stepover is not always constant. This is especially true with steep surfaces like U-shaped parts or molds. When the Adaptive cuts checkbox is selected, the calculation time is longer but the stepover is exact now.



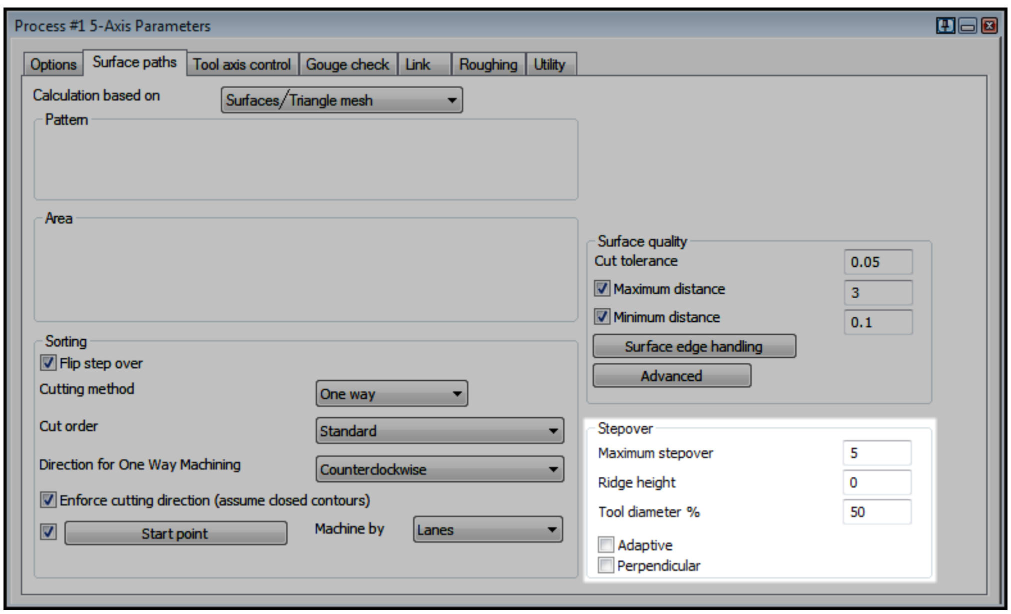
Adaptive cuts off



Adaptive cuts on

Stepover

The controls for setting Stepover options are located in the lower right portion of the Surface paths tab when calculation is based on surfaces or triangle mesh. Not all controls are available for all calculation and pattern types.



The Stepover is the distance between two neighboring parallel cuts. The distance for the stepover can be defined as a side step value (Maximum stepover) or as a cusp height (Ridge height), or as a percentage of the tool diameter (Tool diameter %; available only when pattern is Rough and Adaptive roughing is not selected). The stepover distance can actually be smaller or bigger than your set value depending on the pattern option used. This is especially true with Morph between two curves and Morph between two surfaces, where the stepover varies. The Ridge height option only works with ball endmills.

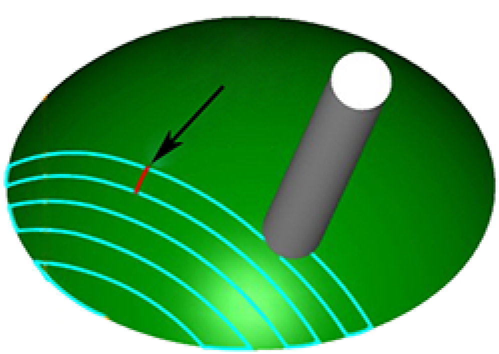
When pattern is Parallel cuts, selecting checkbox Perpendicular makes available two other parameters: Detect threshold distance and Pass extension. For areas areas steep enough to meet the criteria you specify, the system creates regions with passes that areperpendicular to the regular ones.

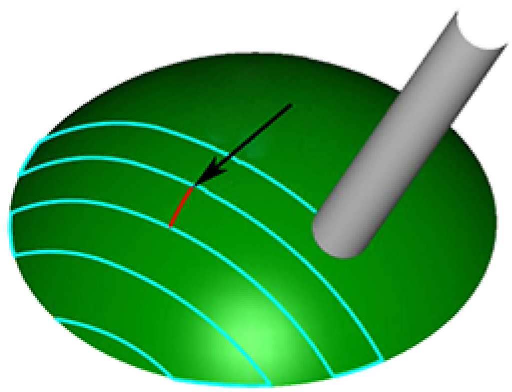
The Stepover distance is measured differently with each pattern used.

- When pattern is Parallel cuts, stepover is the distance between the parallel planes.
- When pattern is Perpendicular to curve (see “Perpendicular to curve” on page 66), stepover is the distance along the curve perpendicular to which cutting planes are used.
- When pattern is Morph between two curves or Morph between two surfaces (see “Morph between two curves” on page 66 and “Morph between two surfaces” on page 69), the lanes are distributed so that the maximal distance (along the drive surface) between the curves or surfaces is the user-defined value.
- When pattern is Parallel to curve or Parallel to surface (see “Parallel to curve” on page 67 and “Parallel to surface” on page 70), stepover is the 3D distance between two consecutive lanes.
- When pattern is Project curves, only one cut is performed and therefore no stepover occurs.

Example

To see this example, open the file `Surface Quality - Maximum Stepover.vnc`.





Small stepover	Big stepover

Tool Axis Control tab

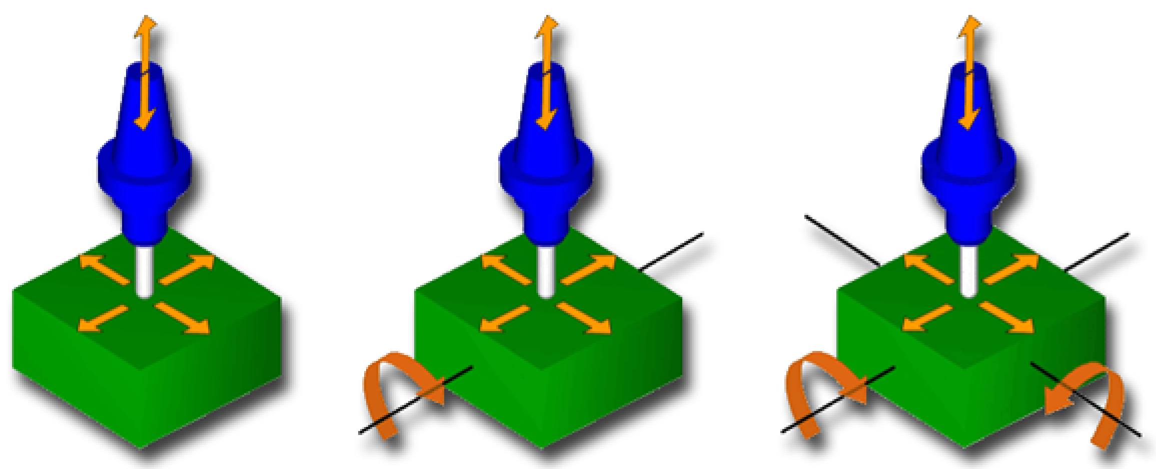
The **Tool Axis Control** page offers the following types of settings (except as noted below*):

- Output Format
- "Maximum angle step" on page 156
- "Tool Axis will..." on page 157 (includes information on lead and tilt angles, side tilt definitions, and related topics)
- "Run tool" on page 202
- "Limits" on page 208

* - Some special-purpose calculation strategies offer a simplified interface with fewer controls: See "Tool Axis Control tab for Swarf Machining" on page 98 or the guides for *5-Axis MultiBlade* or *5-Axis Porting*.

Output Format

The Output format pull-down menu lets you control how many axes the tool will be able to move in.

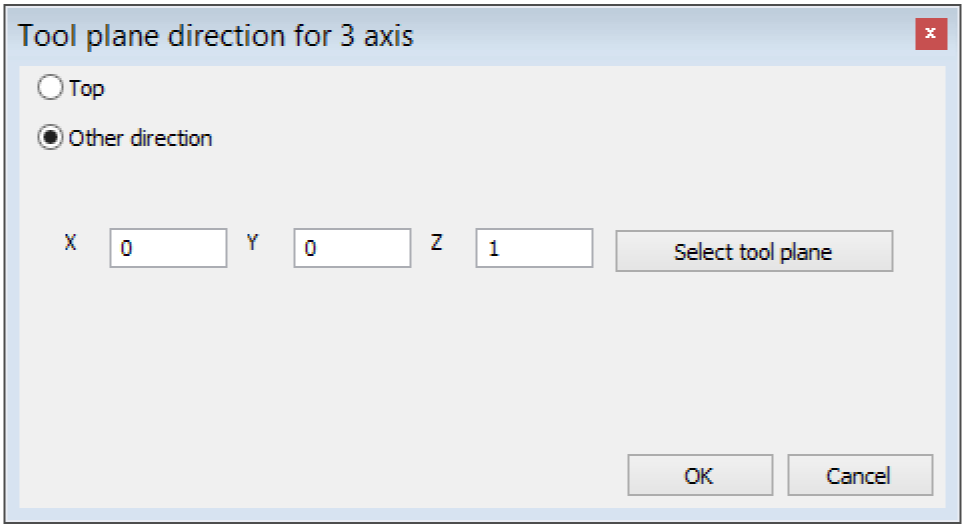


3-Axis Output	4-Axis Output	5-Axis Output

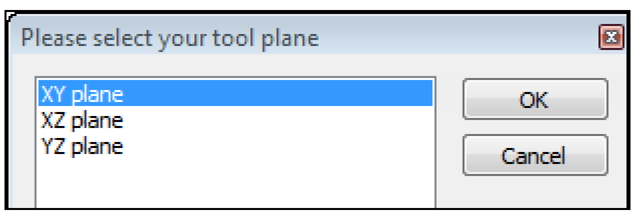
3-Axis

The tool axis direction must be defined by the user as a 3D vector, commonly referred to as I, J, K. For example, to machine with the tool axis normal to CS1 (the XY plane) the I, J, K is `0`, `0`, `1`. This parameter defines the 3-axis cutting direction (tool plane direction). By using this parameter, toolpaths can be generated where the tool comes from: the top view, the side view, the front view or any other user-defined direction. The toolpath generated is formatted as 5-axis moves, but its tool axis orientation is always parallel to this vector. The I, J, K values define a vector in the part coordinate system or CS1. This vector defines the direction of the spindle. For example, the value 0,0,1 implies that the spindle is parallel to the top view. Basically the vector can be understood as a vector starting from the tool tip and pointing to the spindle, parallel to the spindle axis of rotation.

To use this option choose the output format 3-Axis from the pull-down menu and then click the ellipsis button (...) to access the Tool plane direction parameter window. When this window is open, you can select the Top view or a custom vector (Other direction).



Clicking the Select tool plane button lets you choose a tool plane direction from any coordinate system defined in your part.



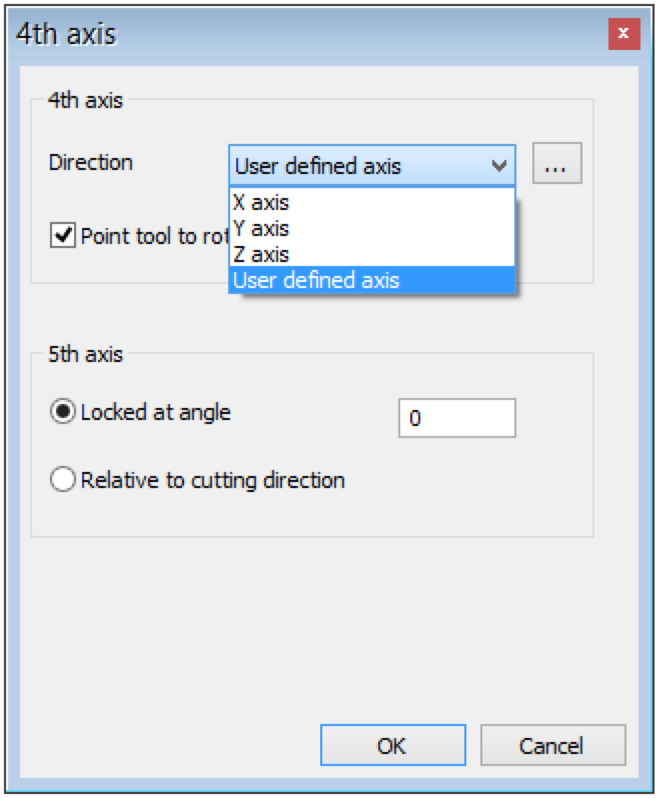
4-Axis for General Calculation Strategies

In some cases it is desirable to have the fifth axis locked to a particular fixed angle. Valid values are from `-90` to `+90` degrees. In the case of regular 4-axis machines, the machine tool is built such that the tool axis direction (the spindle direction) is perpendicular to the rotary axis. In such cases, the default value of `0` must be used.

Sometimes, the 4-axis machines have a head mounted at a fixed tilt angle, like a 45-degree head. In this case the spindle direction is 45 degrees tilted to the rotary axis vector. Then the locked axis value must be set to 45 degrees. A value of `+45` means that the vector from tool tip towards the spindle and the rotary axis vector (e.g. X axis vector which is (`1`, `0`, `0`) have a +45 degree angle to each other.

Another usage of this parameter is in combination with a 5-axis machine. In order to reduce rotary axis motion it might be desirable to use a 5-axis machine but to limit toolpath output to 4-axis with a fixed fifth angle. This means that one of the rotary axes will be fixed for the whole toolpath.

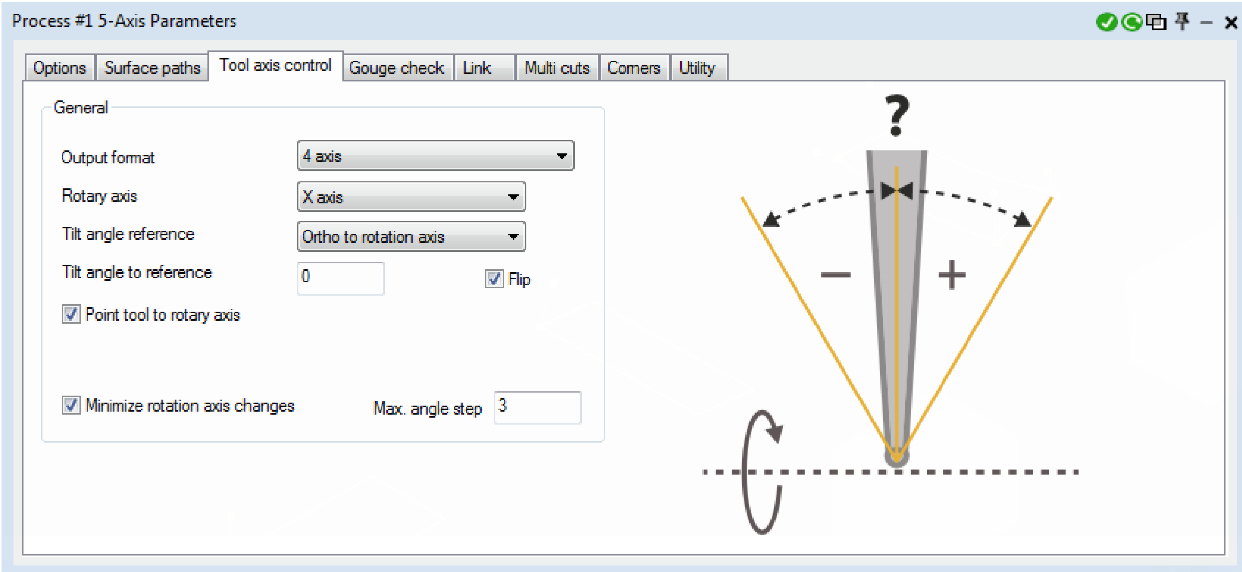
To use this option, choose the output format 4 Axis from the pull-down menu and then click the Rotary Axis button to open the **4th Axis** dialog box:



You can specify any direction as a User-defined axis by clicking the selector button ( ) and defining a vector. In addition, if you select the checkbox Point tool to rotary axis for the 4th axis, you can specify how the 5th axis is treated: either Locked at angle whose value you specify, or Relative to cutting direction.

4-Axis for Swarf Machining

When calculation strategy is set to Swarf Machining, if the geometry is capable of being machined using 4-axis, you should consider setting Output format to 4-axis. The **Tool axis control** tab presents a different set of controls:



For Rotary axis, you can choose from any of the three principal axes or define your own.

For Tilt angle reference, you can choose either Ortho to rotation axis (pictured above) or Parallel to rotation axis. Then enter an angle value (Tilt angle to reference) that the tool can tilt relative to this direction. Also check or clear the Flip checkbox to specify the positioning of tool.

Check the Minimize rotation axis changes checkbox and specify a value for Max. angle step to reduce large swings in tool axis orientation.

5-Axis

The default setting for the Output format is 5 Axis, and all of the tool axis tilting and limiting parameters can be used in a 5-axis toolpath.

For 4-axis or 5-axis output, you can set the following parameters: Maximum angle step (see Maximum angle step); Tool Axis will... (see "Tool Axis will..." on page 157); Run tool (see "Run tool" on page 202); and Limits (see "Limits" on page 208).

Maximum angle step

The Maximum angle step value sets the maximum allowed angle change between two adjacent toolpath positions. The calculation engine outputs 5-axis toolpath that contains the tool tip position and the tool axis vector. The tool axis vectors are not allowed to have an angle change more than the value specified here. Any number of degrees greater than 0 is a valid entry.

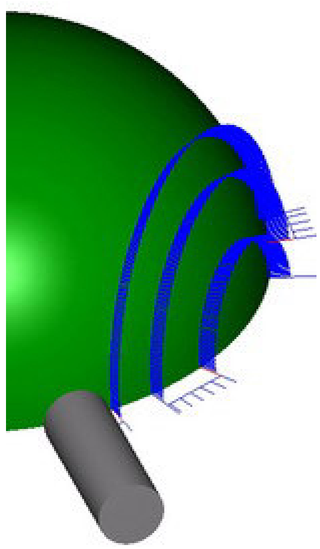
ⓘ

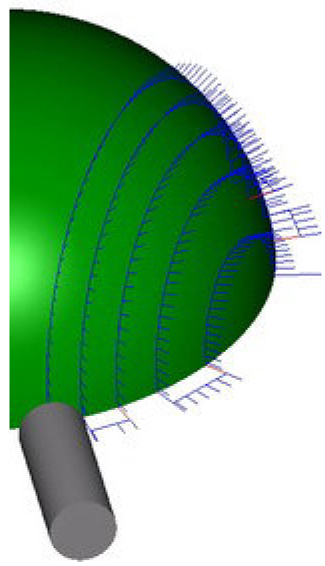
- Be careful when setting the gouge check strategy to Tilt Tool as you may still get collisions with the tool despite checking against all of the tool parts as the tool will tilt away with the maximum angle step. This is because the tool is checked for collision only at each tool position, not between positions. Let's say you have set the default to 3°. If there is another collision within the 3° the gouge check might not recognize this. To fix this, use a smaller value here.

- Depending on the values for Cut tolerance and Maximum distance, there can be some toolpath positions where the angle step is less than this value.
- Decreasing the Maximum angle step generates more points; increasing the value generates fewer points.

Example

To see this example, open the file `Maximum Angle Step.vnc`.





With a small angle step you get more points on the surface.	With a bigger angle step you get less points.

Tool Axis will...

This pull-down menu specifies a tilting strategy for your tool axis and how it relates to the surface normal. Tool axis will ...offers the following types of settings:

- “Not be tilted and stays normal to surface” on page 158
- “Be tilted relative to cutting direction” on page 158
- “Tilted with the angle” on page 171
- “Tilted with fixed angle to axis” on page 172
- “Rotated around axis” on page 173
- “Tilted through point” on page 174
- “Tilted through curve” on page 175
- “Tilted through lines” on page 188

- “Tilted from point away” on page 189
- “Tilted from curve away” on page 190
- “Tilting Relative to Impeller Machining Layer” on page 200
- “Be tilted relative to contact point” on page 201

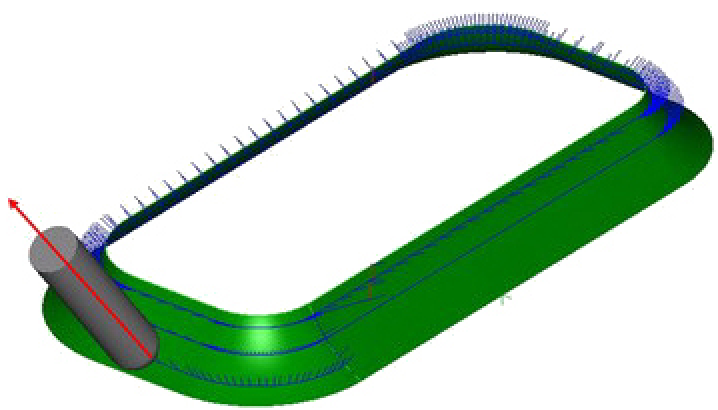
Each menu choice is described in detail, along with additional controls specific to that particular tilting strategy.

Not be tilted and stays normal to surface

Using this option, the surface normal and the tool axis vector are the same. If you use this option but need to keep angle limits, you can set parameters to do so. For more information, see “Limits” on page 208.

Example

Here you can see that the tool is normal to the surface at every position. To see this example, open the file `Tilting Strategies.vnc`.



Be tilted relative to cutting direction

With this Tool axis control strategy, you can give a lead angle to the cutting direction as well as tilt angle to the side of the cutting direction. All angles are in degrees.

Lead angle to cutting direction

This angle defines the lead/lag angle of the tool axis from the surface normal in the direction of the toolpath slice. Positive angles tilt the tool forward in motion direction (lead angle). Negative angles tilt the tool back against motion direction (lag angle).

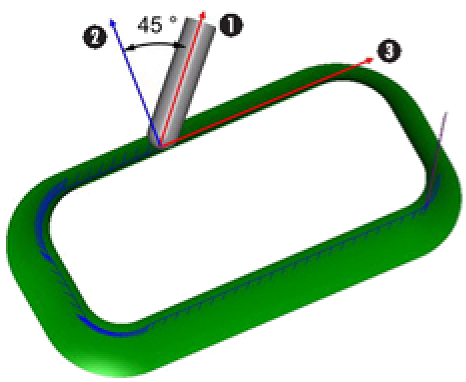
- The lead angle is relative to the cutting direction.
- In case of Zig zag machining, the tool orientation flips with every new cut.
- In case of One way machining, the tool orientation does not change.

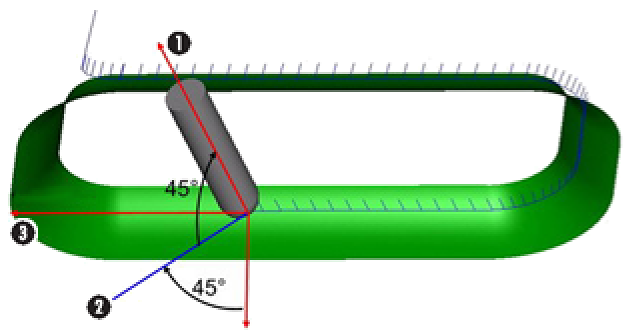
Tilt angle at side of cutting direction:
This angle defines the side tilt value of the tool axis from the surface normal direction based on the toolpath direction. Positive angles tilt to the left side (in motion direction), negative angles tilt the tool axis to the right side (in motion direction).

- The side tilt angle is absolute to the cutting direction.
- In case of Zig zag machining, the tool orientation does not flip with every new cut. The tool keeps its orientation as specified for the first cut.
- To set the side tilt angle relative to the cutting direction of each pass of a zigzag cut (tool axis flips absolute orientation with every new cut), activate Allow flipping side direction by clicking the Advanced button under Side tilt definition.

Side tilt definition:
Side tilt definitions can be set up when you use a tilt angle at the side of the cutting direction. They are additional settings in case of side machining. For more information, see “Side tilt definition” on page 159.

Example
This example shows two toolpaths and how the tool axis (#1) relates to the surface normal (#2) based on the cutting direction (#3). To see this example, open the file `Tilting Strategies.vnc`.





The first toolpath only has set lead angle tilted to cutting direction with 45°.	In the second toolpath the tool is tilted with a lead angle to cutting direction of 45° and a side tilt angle of 45°.

Side tilt definition

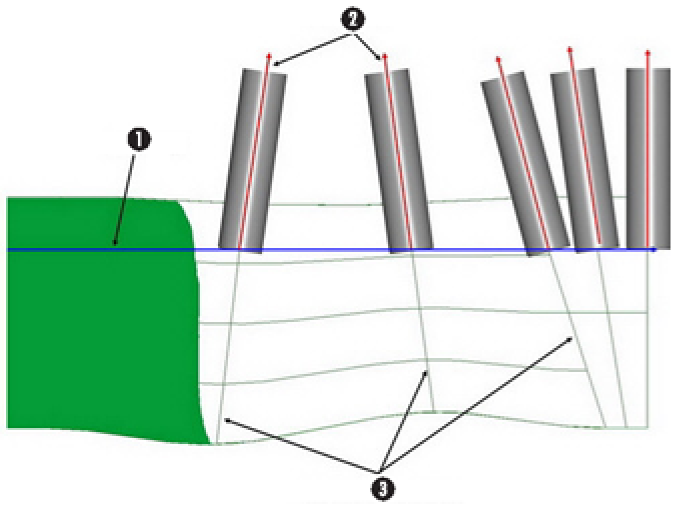
Follow surface iso direction
This option is a good choice if linear surfaces are present. Multiple surfaces can be used here. If any surface does not have a compatible U and V direction with the neighboring surfaces, then this function tries to correct these areas automatically.

Please note that fanning can be applied by the user to avoid quick changes of tool orientation due to irregularities of the surface geometry. For more information, see “Side Tilt Fanning Distance” on page 166.

Clicking the Advanced button lets you set options for greater control over the tool axis. This side tilt option supports the Gradual lead angle change, Gradual side tilt angle change, Side Tilt Fanning Distance and Ruled surface radius limit advanced options. For more information, see “Advanced Options for Tilting Relative to Cutting Direction” on page 164.

Example

When you look at the surface grids you can see that the iso directions are different. The tool axis orientation follows that direction.



1. Toolpath
2. Tool Axis
3. Surface Iso Direction

To see this example, open the file `Side Tilt Definition.vnc`.

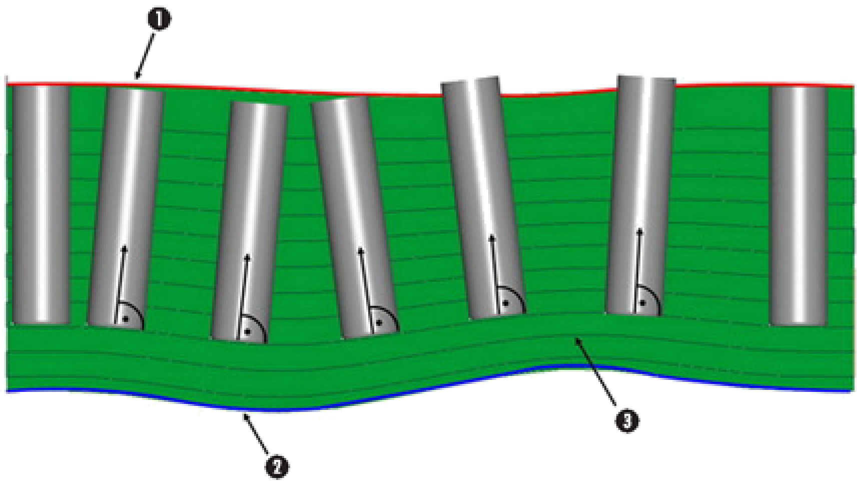
Ortho to cut direction at each position

The side tilt direction is determined by a normal line from the current surface contact point to the lower edge curve (whatever curve is driving the generation of the toolpath). That means that the tool axis is always normal to the toolpath. This option is most useful when the lead angle direction should be defined by the direction in which the toolpath is moving. This option can be used for all toolpath patterns. See “Pattern Settings” on page 61.

> If you set a lead angle in the Be tilted relative to cutting direction option, then this angle will be added to the orthogonal orientation.

Example

The surface is curved at the upper and lower edge. The toolpath is parallel to the lower edge. You can see the tool is always orthogonal to the toolpath.



1. Upper Curve
2. Lower Curve
3. Toolpath

Clicking the Advanced button lets you set options for greater control over the tool axis. This side tilt option supports the Allow flipping side direction, Gradual lead angle change and Gradual side tilt angle change advanced options. For more information, see "Advanced Options for Tilting Relative to Cutting Direction" on page 164.

Orthogonal to cut direction at each contour

The tool axis orientation is determined by a single orthogonal line calculated from a complete contour segment. The system analyzes this segment and approximates from all orthogonal vectors a single vector.

Clicking the Advanced button lets you set options for greater control over the tool axis. This side tilt option supports the Allow flipping side direction, Gradual lead angle change and Gradual side tilt angle change advanced options. For more information, see "Advanced Options for Tilting Relative to Cutting Direction" on page 164.

The Approximate option selects the calculation method used to determine this vector. There are three types of approximation:

Approximate by one vector

There is one orthogonal vector that replaces each orthogonal vector on the contour. The replacement vector is given by the "least-squares mean" of the orthogonal vectors of the contour, and then this mean vector is normalized.

Approximate by two vectors

The orthogonal vectors are computed from a polynomial of degree one that fits the original orthogonal vectors and is computed by the least-squares method. Therefore, there are two vectors representing the coefficients of the polynomial.

Approximate smooth

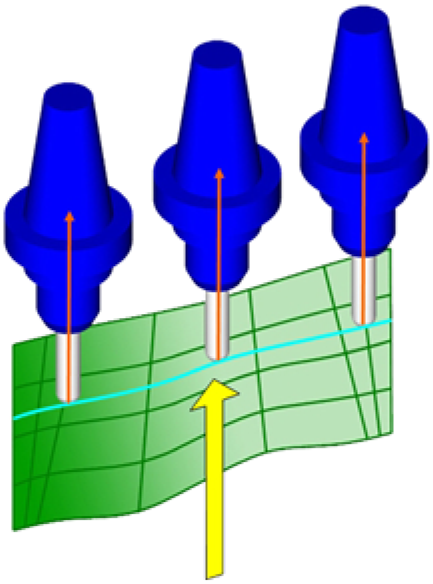
The orthogonal vectors are computed from a polynomial of degree two that fits the original orthogonal vectors and is computed by the least-squares method.

Smooth (local)

The ortho vectors are computed by a distribution of the local neighboring ortho vectors.

Use spindle main direction

This option uses the machine definition's spindle main orientation (default tool axis vector) definition as the reference for finding the side tilt direction. The side tilting is always calculated from the surface normal towards the spindle main orientation.

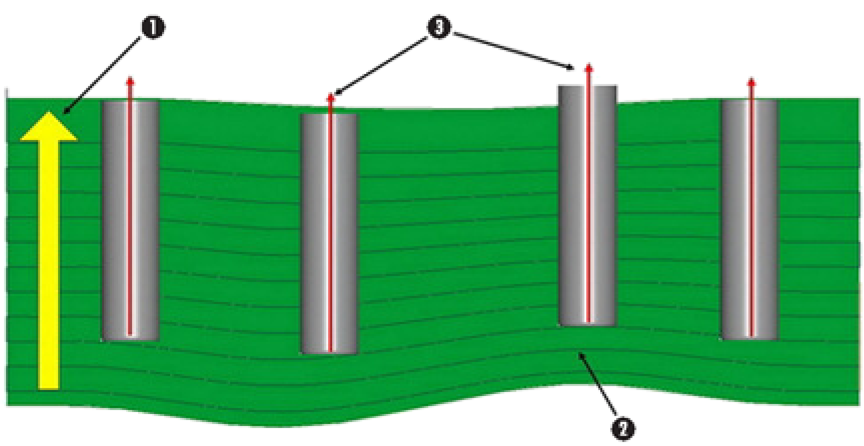


For example, if the spindle main direction vector is the Z axis and side tilting of 90 degrees from surface normal takes place, then the tool axis orientation is the surface normal rotated 90 degrees towards spindle main direction. In practical terms, such a rotation can be handled by a machine tool without utilizing the C axis.

- You may add a lead angle and a side tilt angle to this toolpath. When applied, the tool tilts from the surface normal in direction of the main axis.
- This option does not work with Allow flipping side direction.

Example

In this example the spindle main direction is the Z axis. With a side tilt angle of `90°`, the tool axis is orientated to this direction. To see this example, open the file `Side Tilt Definition.vnc`.



1. Spindle Direction
2. Toolpath
3. Tool Axis

Clicking the Advanced button lets you set options for greater control over the tool axis. This side tilt option supports the Gradual lead angle change and Gradual side tilt angle change advanced options. For more information, see "Advanced Options for Tilting Relative to Cutting Direction" on page 164.

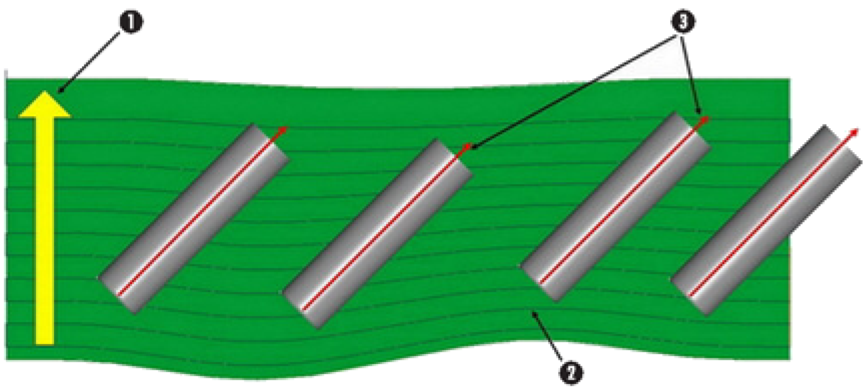
Use user-defined direction

This option allows a user-defined fixed direction vector as the reference for finding the side tilt direction. Clicking the selection button (the button with the ellipsis, ...) opens a dialog box where you can either enter coordinates for the vector or select a point in the geometry.

- You can add a lead angle and a side tilt angle to this toolpath. When angles are applied, the tool tilts from the surface normal in direction of the main axis.
- This option does not work with Allow flipping side direction.

Example

In the example, the main spindle direction is set to `45°` between z and x. With a `90°` side tilt angle the toolpath looks like the picture. To see this example, open the file `Side Tilt Definition.vnc`.



1. Spindle Direction
2. Toolpath
3. Tool Axis

Clicking the Advanced button lets you set options for greater control over the tool axis. This side tilt option supports the Gradual lead angle change and Gradual side tilt angle change advanced options. For more information, see "Advanced Options for Tilting Relative to Cutting Direction" on page 164.

Use tilt line definition

This option utilizes user given tilt line elements as the side tilt direction. This option gives the user the freedom of defining the side tilting direction manually by just passing lines.

Clicking the Advanced button lets you set options for greater control over the tool axis. This side tilt option supports the Gradual lead angle change and Gradual side tilt angle change advanced options. For more information, see "Advanced Options for Tilting Relative to Cutting Direction" on page 164.

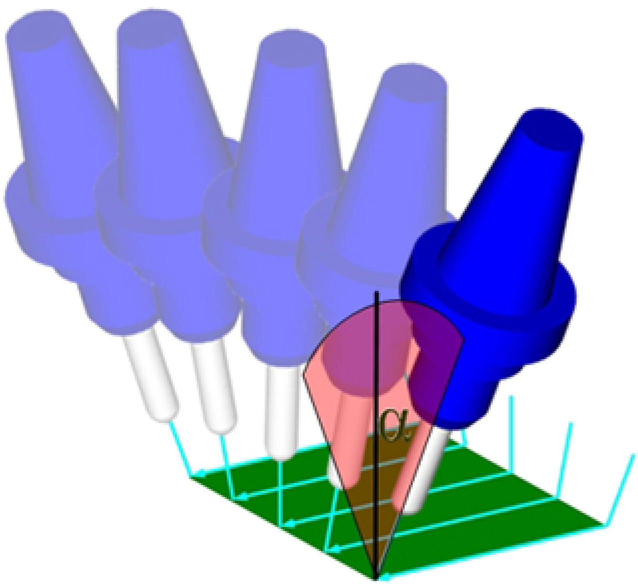
Tilting lines maximum snap distance

The maximum snap distance parameter defines the maximum distance between tilt line end points and the machining contour. When tilting is applied to a contour, then only lines within this distance will be used, other lines that are far from the contour will be 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

Gradual lead angle change

The value for Gradual lead angle change is an additional angle offset added to the lead angle setting and applied equally to subsequent slices. During machining, this gradual lead angle change value will be divided by the number of cuts to yield gradual lead angle increments. This new angle increment will be added to the lead angle for each slice. By the last slice, at the end of the toolpath, the final tool axis orientation is the lead angle value plus the gradual lead angle change value. The first cut of the toolpath is tilted with only the lead angle value. The parameter accepts values from –180° to 180°. Positive angles let the tool tilt to the front (in the moving direction); negative angles let the tool tilt backwards (in the moving direction).

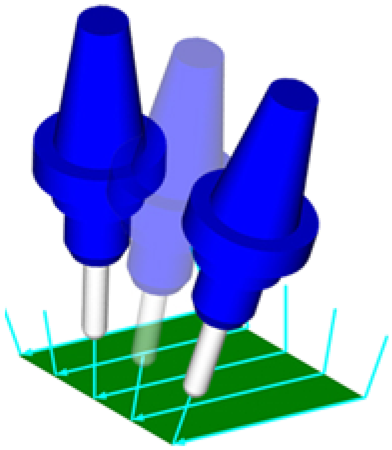


Example: Lead angle is set to 5 degrees and gradual lead angle is set to 10 degrees.
Results: At the beginning of the toolpath the lead angle will be 5 degrees. At the end of the toolpath, the lead angle will be 5 + 10 = 15 degrees. At the halfway point of the toolpath, the lead angle will be 5 + (10 * 0.50) = 10 degrees.

The concept behind this option is machining a blisk (a single-engine component composed of a rotor disc and blades also known as an integrally bladed rotor). When cutting a blisk you may want a side tilt angle of 40 degrees when near the top and as the machining progresses downward you might want to reduce the tilt to 10 degrees. The change is applied gradually for each contour.

This option is only available when using the tool axis orientation Be tilted relative to cutting direction. This parameter also works with Allow flipping side direction and Gradual side tilt angle change. When both of these advanced options are used, the side tilt angle as well as the gradual side tilt angle change will alternate with every cut.

Example



When you simulate the operation you see the tool tilting with the new lead angle increments. The lead angle is set to `10°`, so the first cut has a `10°` lead angle. The gradual lead angle change is set to `35°`. With ten cuts, the angle increment is `3.5°`. Therefore, the second cut has a total lead angle of `13.5°`, the third cut `17°`, and so on. In the end you get a total lead angle of `45°`. To see this example, open the file `Gradual Tilt Angle Change.vnc`.

Gradual side tilt angle change

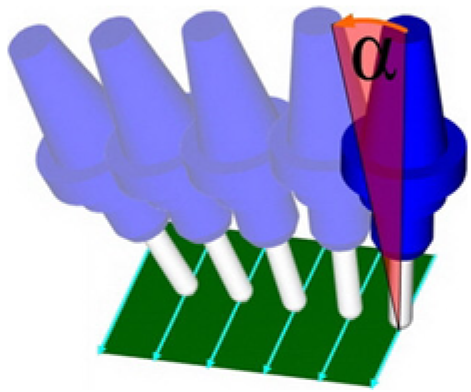
The Gradual side tilt angle change is an additional angle offset added to the Side tilt definition setting and applied equally to subsequent slices. During machining this gradual side tilt angle change value is divided by the number of cuts to give us gradual side tilt angle increments. This new angle increment will be added onto the side tilt angle value for each slice. By the last slice at the end of the toolpath the final tool axis orientation is the side tilt angle value plus the gradual side tilt angle value. The first cut of the toolpath is tilted with only the tilt angle value. The parameter accepts values from `-180°` to `180°`. Positive angles let the tool tilt to the left (in the moving direction) while negative angles let the tool tilt to the right (in the moving direction).

For example, a side tilt angle is set to 5 degrees and the gradual side tilt angle is set to 10 degrees. This means that at the beginning of the toolpath the side tilt angle will be 5 degrees and at the end of the toolpath the side tilt angle will be 5 + 10 = 15 degrees. At the halfway point of the toolpath, the side tilt angle will be 5 + (10 * 0.50) = 10 degrees.

The concept behind this option is machining a blisk (a single-engine component composed of a rotor disc and blades also known as an integrally bladed rotor). When cutting a blisk you may want a side tilt angle of 40 degrees when near the top and as the machining progresses down you might want to reduce the tilt to 10 degrees. The change is applied gradually for each contour.

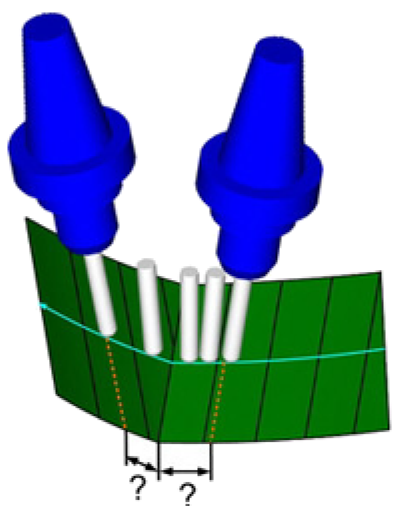
This option only is available when using the tool axis orientation Be tilted relative to cutting direction. This parameter also works with Allow flipping side direction and Gradual lead angle change. When using both of these advanced options the lead angle as well as the gradual lead tilt angle change will alternate with every cut.

Example



You can see the tool tilting with the new side tilt angle increments. To see this example, open the file `Gradual Tilt Angle Change.vnc`.

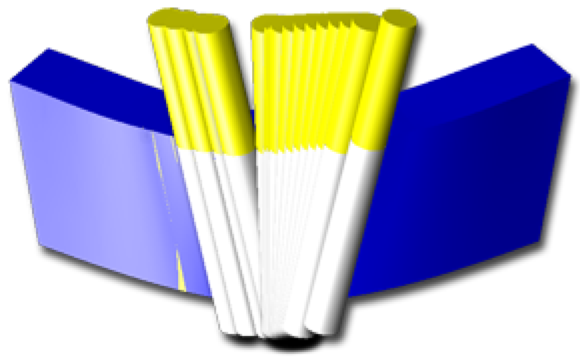
Side Tilt Fanning Distance



This option is for two crossing curved surfaces with different iso directions. To make a proper toolpath from the one surface iso direction to the other surface iso direction a smooth toolpath with constant cross fading will be generated. The fanning distance is the distance from the crossing point of the surfaces and the point where the cross fading of the tool axis begins. The distance you set will be applied to all surfaces, beginning from the crossing point. Please note that this option is only available when the tool axis option Be tilted relative to cutting direction is combined with the side tilt definition Follow surface iso direction. Cutting with the side of a tool requires defining the best lead/lag direction at each position of the toolpath. The best lead/lag direction jumps between surfaces when surfaces that are almost ruled and have a slight curvature meet. In such cases, the fanning distance is used to dampen out abrupt orientation changes.

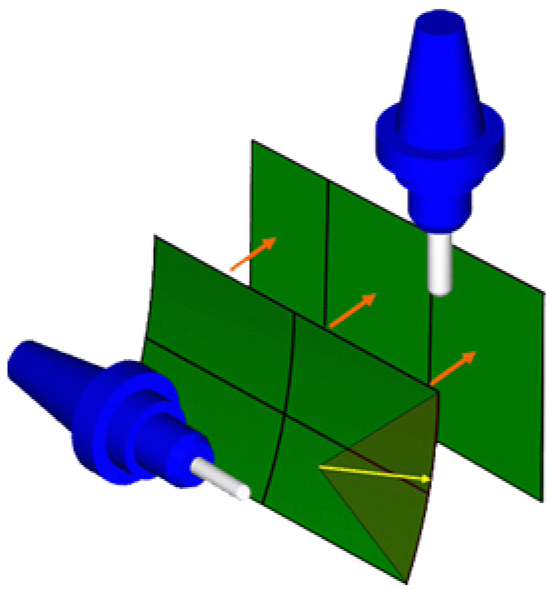
Example

The two examples below show the difference using a fanning distance. The first image is without a fanning distance and the second is with a 15 mm fanning distance. The toolpath is only a single cut at the lower edge. The point where the two faces cross. is where the fanning distance takes effect. To see this example, open the file `Side Tilt Fanning Distance.vnc`.



No Fanning Distance	15mm Fanning Distance

Ruled surface radius limit

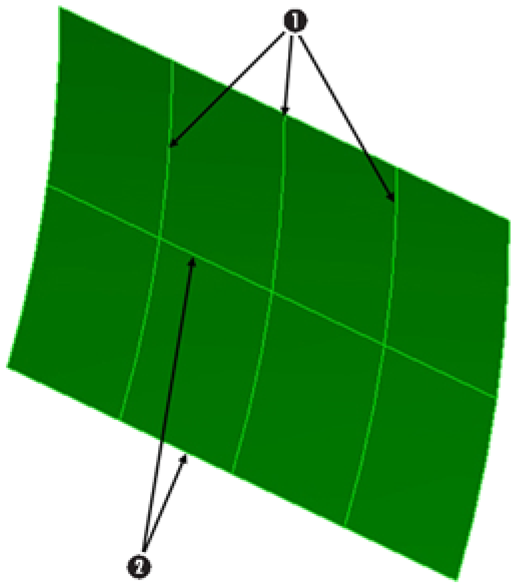


Swarf machining is defined by having a line contact between the cutter (either cylindrical or conical) and the surface. This line contact can only be achieved if the surfaces are ruled surfaces. Surfaces are spanned in U and V and a ruled surface must have an infinite radius (a flat) in one of these directions. Practically speaking, many surfaces appear to be ruled, but an analysis of their surface mathematics show that although they do not have an infinite radius in one direction, the radius is very large radius. This large radius can be considered almost flat. This parameter lets you set how large or small the radius must be to be considered "flat" so that the surfaces can be used for swarf machining.

The value of this setting (a small or large radius) does not affect the resulting toolpath in terms of gouging. To ensure the surface is not gouged you will need to enable gouge protection. This option is only available when using the tool axis orientation Be tilted relative to cutting direction and the side tilt definition Follow surface iso direction.

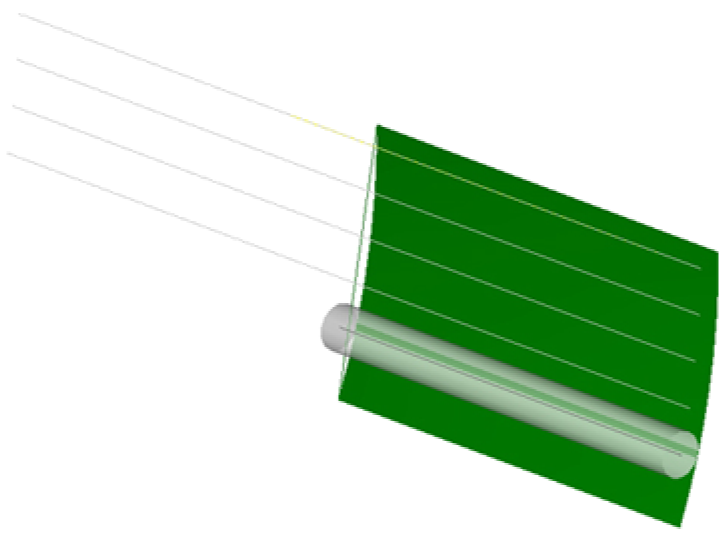
Example

In this example you see a part with a curved surface with a radius of 147.727 mm. (Please note that typically much smaller radii will be involved, but the large radius helps illustrate this function.) The toolpath is a single path parallel to the lower edge but tilted `90°` to the side. In the vertical direction the iso lines of the surface are not ruled, instead the horizontal iso direction is ruled. To see this example, open the file `Ruled Surface Radius Limit.vnc`.

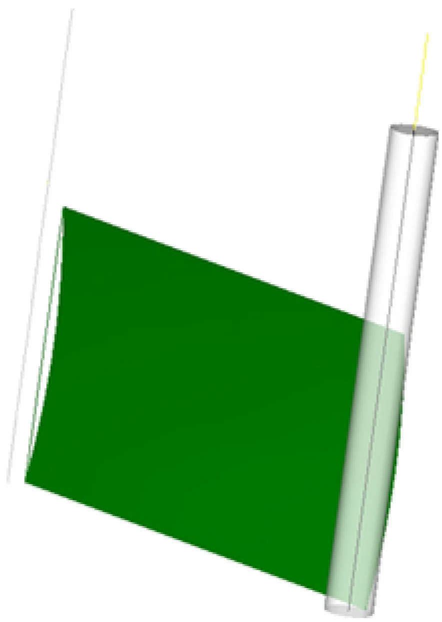


1. Curved vertical iso direction
2. Ruled horizontal iso direction

The system assumes the horizontal iso direction will be used to orient the tool axis, even though you have set a side tilt angle of 90°. Therefore the toolpath is wrong. The image below is what happens when you simulate the part. Here the radius limit is set to 148 mm. That means that all surfaces with the radius larger than 148 mm are assumed as ruled. Our surface with the radius 147.727 mm is smaller than this value, so it is not seen as ruled.



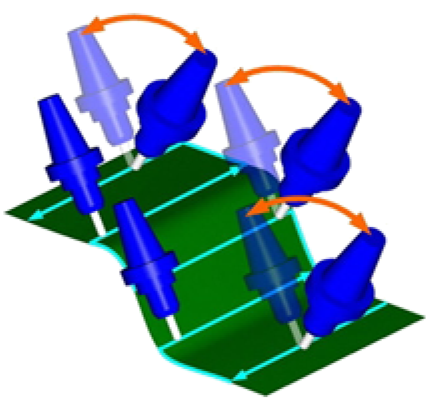
When you simulate the second operation, the limit is set to 147 mm. Now the radius limit takes effect and the toolpath will be correct.



Allow flipping side direction

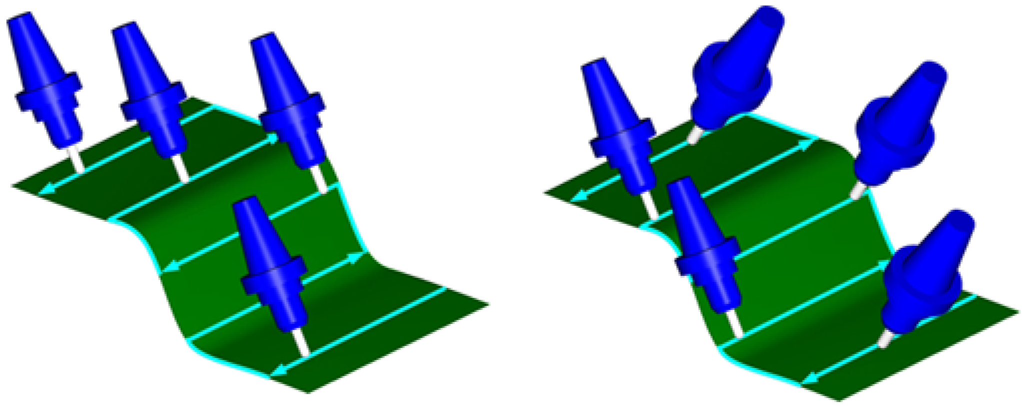
This option works in conjunction with Gradual lead angle change and Gradual side tilt angle change, and is available only when the Cutting method is set to Zig zag (or, with calculation based on Triangle Mesh, One way), combined with tool axis orientation set to Be tilted relative to cutting direction and certain choices for Side tilt definition.

If this option is not selected, then the tilt angle for all cuts is the same as the angle of the first cut. If this option is selected, then the side direction is changed based on the current cutting direction – in other words, the tool will always tilt to the right or left depending on the cutting direction.



Example

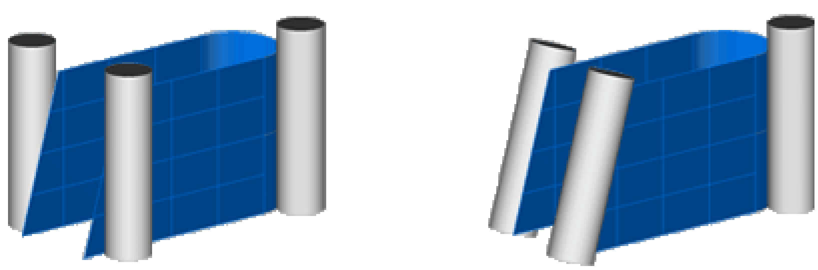
Open the file `Allow Flipping Side Direction.vnc` and simulate the first operation. You see that the tool orientation will be kept all along its path. Now simulate the second operation and see how the tool flips its orientation with every new cut. To see this example, open the file `Allow Flipping Side Direction.vnc`.



Side Flipping is off | Side Flipping is on

Align tool axis to planar surface edges

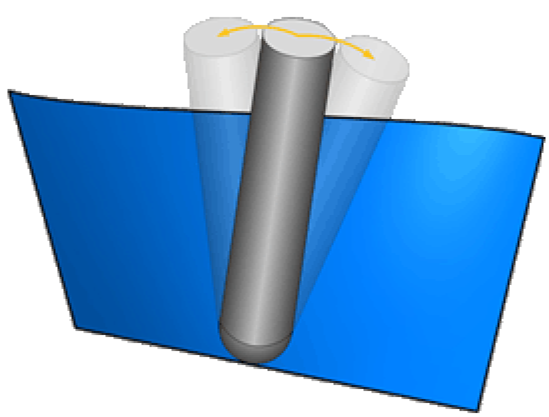
This option will force the tool axis to be parallel to the edges of a surface. This is available when the Side tilt definition is set to Follow surface iso direction.



Off: Not aligned to surface edges | On: Aligned to surface edges

Improve side tilt definition for twisted surfaces

This option can be used for swarf-cutting impeller blades and similar ruled twisted surfaces. The aim is to provide an optimized tilting with a line contact between tool and surface.



Side tilt improved for twisted surfaces

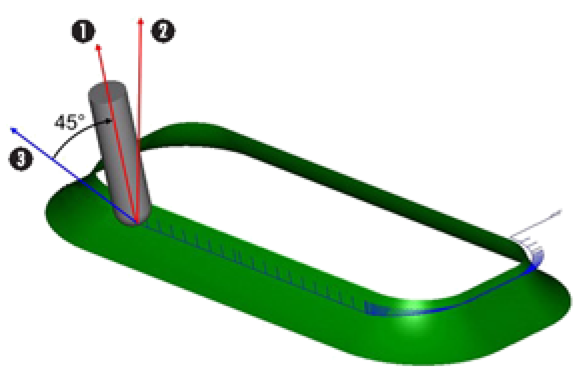
Tilted with the angle

The tool axis will be tilted from the surface normal direction towards the tilt axis. The tilt axis can be the X, Y and Z axis or any line created in the geometry. Imagine that the tilt axis and surface normal span a plane, the tool can tilt only on this plane. When the surface normal is parallel to your desired tilt axis there is no plane to be spanned, resulting in the tool axis not tilting despite your setting a tilt angle. The option Tool axis crosses tilt axis (see Tool axis crosses tilt axis) is available to force the tool axis to intersect with the tilt axis.

> When machining in the XY plane, the tool can be tilted using X or Y, but not Z. `0°` will set the tool normal to the plane. When X is selected, a positive angle will tilt the tool toward X-positive and when Y is selected, a positive angle will tilt the tool toward Y-positive. Negative angles will tilt the tool in the opposite direction. Typically, the tilt is performed on the axes which appear in the CS axis label. In other words, XY allows tilting in XY, XZ allows tilting in XZ, and YZ allows tilting in YZ.

Example

In this example the tool is tilted with 45° against the Z axis (tilt axis). You can see how the surface normal and the tilt axis span a plane in which the tool tilts. See the file `Tilting_Strategies.vnc` for a working example.



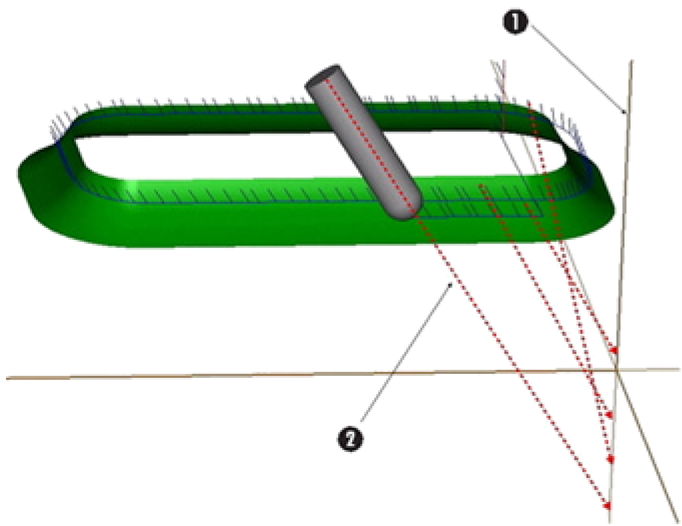
1. Tool Axis
2. Tilt Axis
3. Surface Normal

Pole limit

When this option is active, the tilting of the tool axis is limited to the selected tilt axis. When this item is disabled the tool may tilt beyond the selected axis's pole. For example, if the Z axis is selected then the maximum angle for the tool axis is the Z axis. When this option is disabled the tool is allowed to tilt beyond the Z axis.

Tool axis crosses tilt axis

If this option is enabled, the extension of the tool axis will always intersect with the defined axis.



1. Z Axis
2. Tool Axis

Reverse tool

If this switch is set to true, then the tool direction is reversed. E.g. if Reverse tool is off then the drive surfaces are machined from the positive side of the Tool Axis (the positive side of the drive surfaces). But if Reverse tool is on then the drive surfaces are machined from the negative side of the Tool Axis (the negative side of the drive surfaces).

Tilted with fixed angle to axis

The tool axis will be tilted from the tilt axis towards the surface normal.

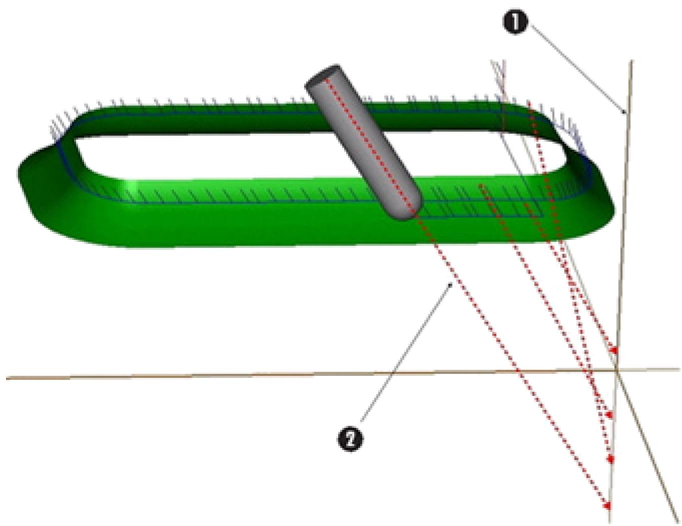
You choose a reference vector from the pull-down menu – X-axis, Y-axis, Z-axis, or a Line that you define or select from the workspace – and a Tilt angle relative to that orientation

When the surface normal is parallel to your desired tilt axis there is no plane to be spanned, resulting in the tool axis not tilting despite your setting a tilt angle. The option Tool axis crosses tilt axis is available to force the tool axis to intersect with the tilt axis. This function works almost identically to Tilted with the angle. However, where Tilted with the angle considers 0° to be normal to the surface, Tilted with fixed angle to axis considers 0° to be parallel to the chosen axis and 90° to be normal to the axis.

When the Line option is selected, click the ellipsis button (...) to either manually define a line or select a line in your part file. Defining a line manually requires a coordinate and a vector. To choose a line, click on the ellipsis button.

Tool axis crosses tilt axis

If this option is enabled, the extension of the tool axis will always intersect with the defined axis.



1. Z Axis
2. Tool Axis

Reverse tool

If this option is enabled then the tool direction is reversed. E.g. if this option is off, then the drive surfaces are machined from the positive side of the Tool Axis (the positive side of the drive surfaces). But, if this option is on then the drive surfaces are machined from the negative side of the Tool Axis (the negative side of the drive surfaces).

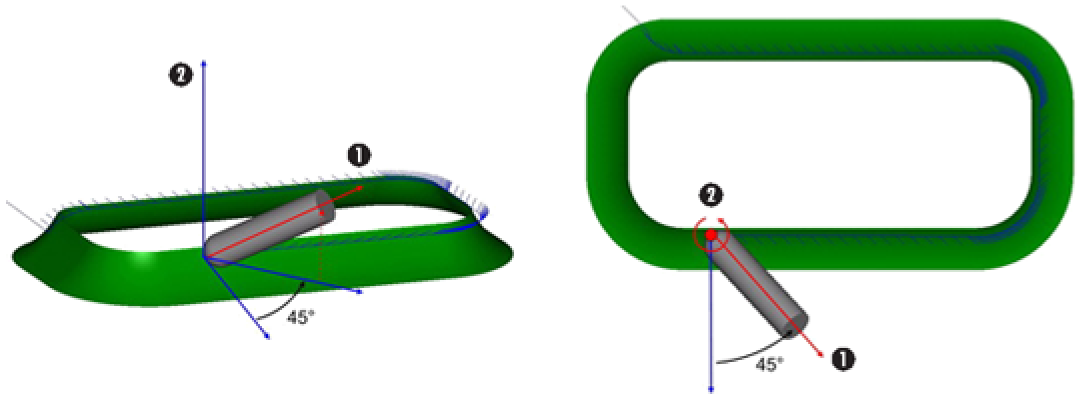
Rotated around axis

With this option the tool axis has the same direction as the surface normal but is tilted around a specified axis. This function works almost identically to Tilted with the angle. However, where Tilted with the angle causes the tool to tilt parallel to the selected axis or line, Rotated around axis cause the tool to tilt radially around (perpendicular to) the selected axis or line. If you consider the axis labels of the CS plane, such as XY, the tilt can only be performed around axes which are flat to that plane and cannot tilt around Z, for instance. The reference axis can be the X, Y, Z or any line. When the surface normal is parallel to your desired tilt axis there is no plane to be spanned, resulting in the tool axis not tilting despite your setting a tilt angle.

When the Line option is selected you may click on the ellipsis button (...) to either manually define a line or select a line in your part file. Defining a line manually requires a coordinate and a vector. To choose a line click on the ellipsis button.

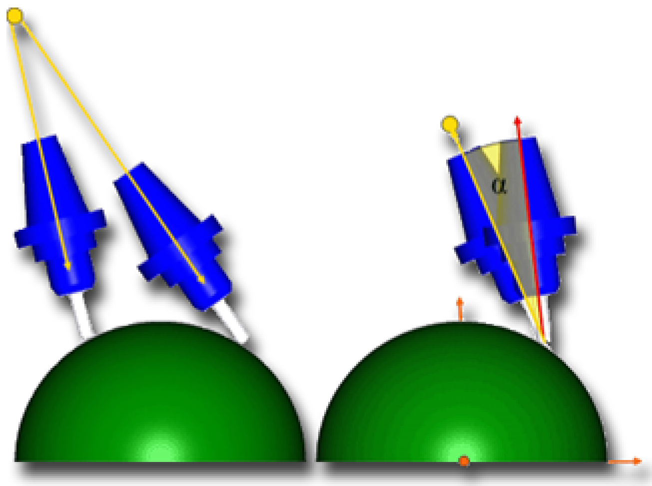
Example

In this example the tool axis direction (#1) is the same as the surface normal but tilted with a 45 degrees angle around the main Z axis (#2). From the top view you can see the tilted angle of 45° better. See the file `Tilting_Strategies.vnc` for a working example.



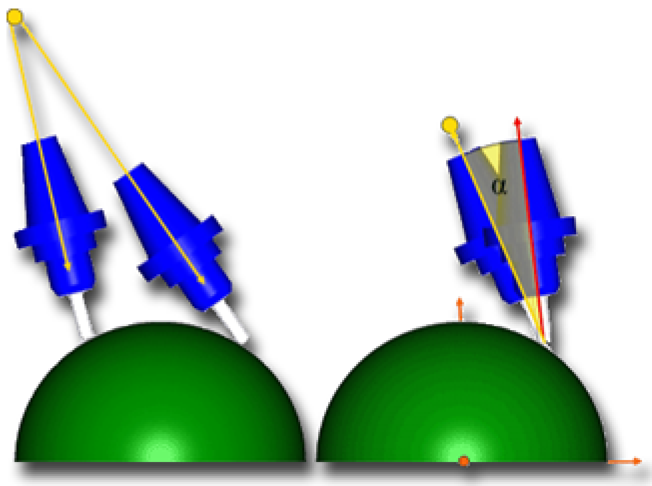
Tilted through point

With this option the tool axis is always pointing from a designated point in your geometry to the surface. Click the Tilt point button to specify the absolute X,Y,Z location of the point or select it from the workspace.



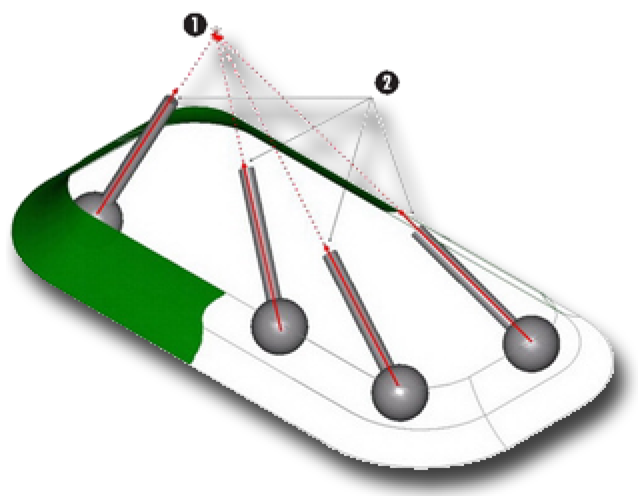
According to the option you choose for Point tilt type, the tool axis will align either from point towards axis or from axis towards point.

Additionally, you can set a separate Tilt angle. This means the tool axis will be aligned through the specified point, but the tool will then be tilted at the specified angle, either relative either to a primary axis (X-axis, Y-axis, Z-axis) or relative to a Line that you define or choose from the workspace by clicking the ellipsis button (...).



Example

In this example you can see how the tool axis is always aligned to the point above the drive surface. See the file `Tilting Strategies.vnc` for a working example.



1. Designated Point
2. Tool Axis Direction

Tilted through curve

With this option the tool axis is aligned to a tilt curve while machining.

Select the curve by clicking on the Tilt curve button. For Curve tilt type, choose one of these menu options: Closest point, Angle from curve, Angle from spindle, main direction, From start to end, Automatic curve and From start to end for each contour.

Additionally, you can set a separate Tilt angle. This means the tool axis will be aligned through the specified point, but the tool will then be tilted at the specified angle, either relative either to a primary axis (X-axis, Y-axis, Z-axis) or relative to a Line that you define or choose from the workspace by clicking the ellipsis button (...).

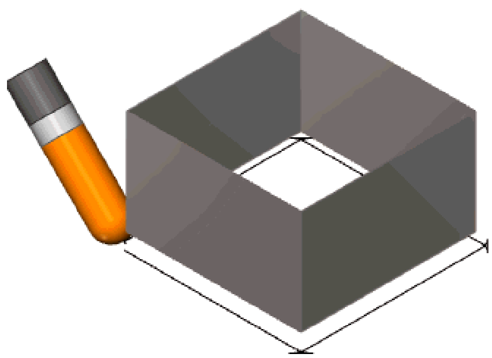
Advanced options for Curve tilt type

For some choices of Curve tilt type, an Advanced button becomes available. Depending on the type, the **Advanced** dialog allows you to exert fine control over several aspects of toolpath, noted as follows.

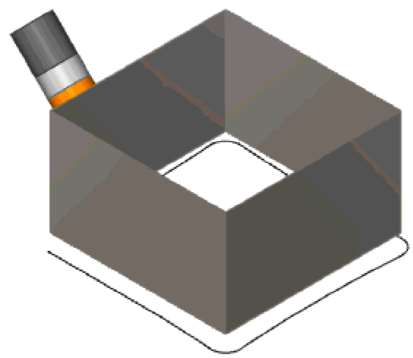
Side tilt fanning distance

This option is when the drive surface geometry has two crossing curved surfaces. The aim is to avoid a sudden jump in tool axis when changing the surfaces.

The fanning distance is the distance from the crossing point of the surfaces and the point where the cross fading of the tool axis begins. The bigger the fanning distance, the earlier the tool begins to tilt.



Toolpath without fanning



Toolpath with fanning

Note: Cutting with the side of the tool requires a proper definition of the best side direction at each position of the toolpath. In case of surfaces that are almost ruled and have a slight curvature in the secondary direction, the best side direction between two surfaces jumps at the boundary between such two surfaces. In such case, the fanning distance is applied to dampen out such jerky changes of orientation.

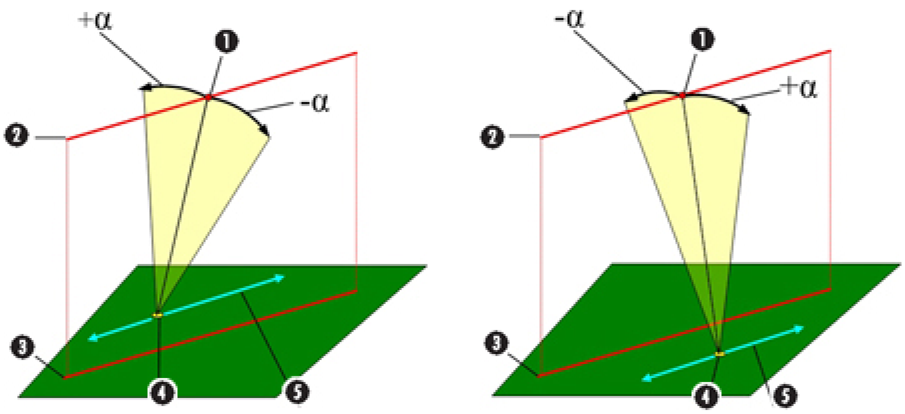
Closest point

During machining the tool axis is aligned to a tilt curve with an optional Fixed tilt angle setting. The orientation of the tool axis is aligned with the point represented by the shortest distance between the present toolpath point and the tilt curve. The tilt curve must be located above the drive surface. The tool's maximum tilting is vertical (90°) or horizontal (0°), so if your tool axis is already tilted 45° from the present toolpath point to your curve and you set a fixed tilt angle of 60° the tool would tilt just 90° (vertical).

Fixed tilt angle

The Fixed tilt angle tilts the tool axis away from the tilt curve. The direction is defined by the surface spanned by the curve point, surface point and the direction from the curve to the surface point.

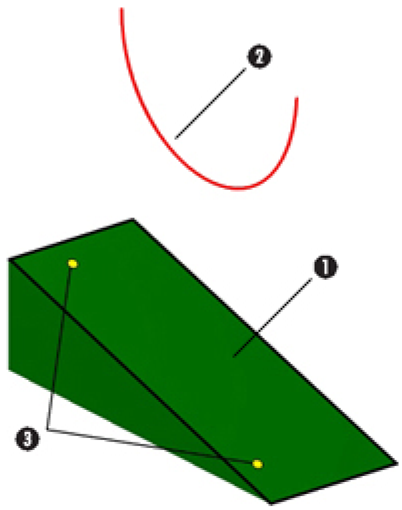
- When using positive tilt angles the tool tilts outwards (in moving direction).
- When using negative tilt angles the tool tilts inwards (in moving direction).



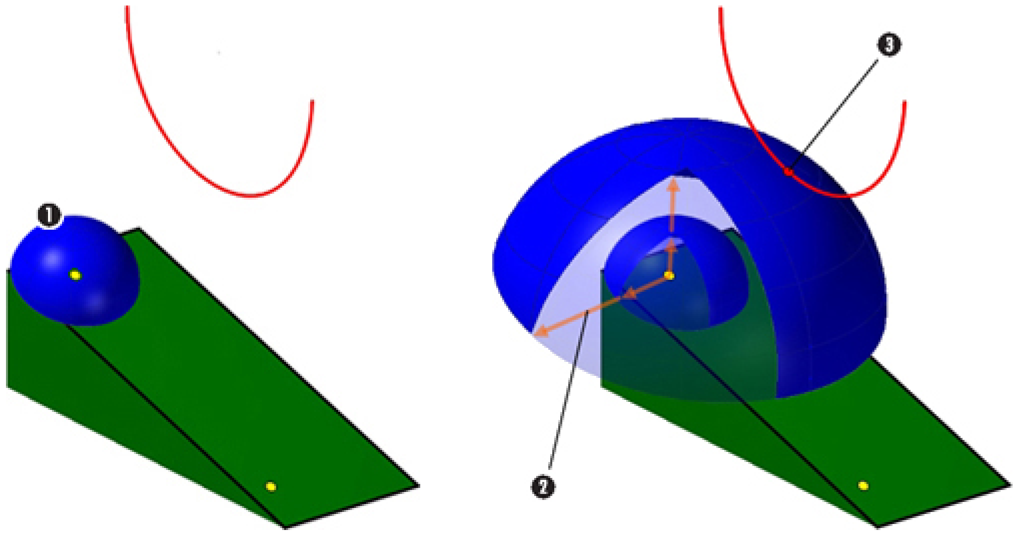
1. Curve Point
2. Tilt Curve
3. Projected Tilt Curve on the surface
4. Toolpath Point
5. Direction of movement

How this works - a simple example

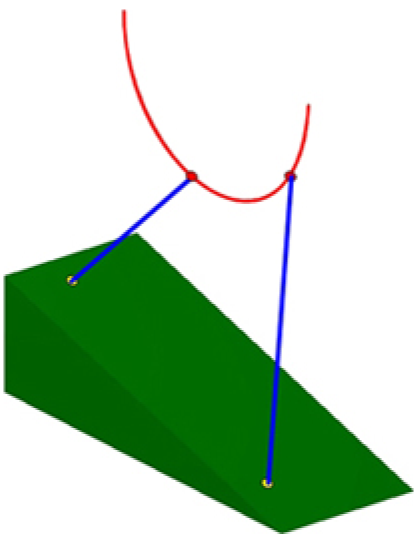
Here we have a drive surface (#1) with a tilt curve (#2) above it. Two random points along the toolpath (#3) are shown.



Now imagine that for each position (point) along the toolpath, a small sphere is created about that point (#1). Next, the sphere is expanded (#2) until it contacts the tilt curve. The point of contact between the sphere and the tilt curve (#3) becomes the reference point along the tilt curve.

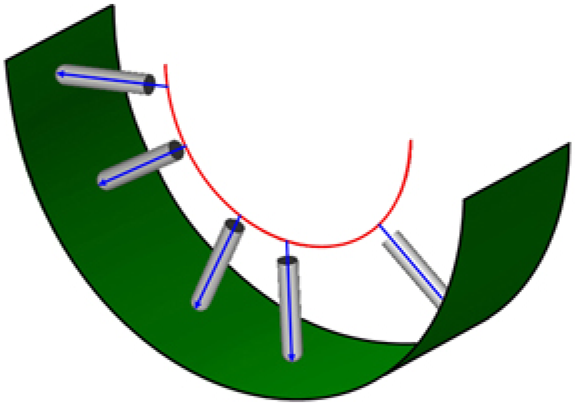


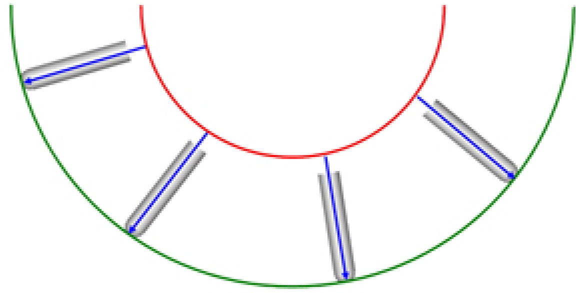
A line is created between the point on the curve and the present toolpath point. This line is the tool axis orientation. This process is repeated for each point along the toolpath.



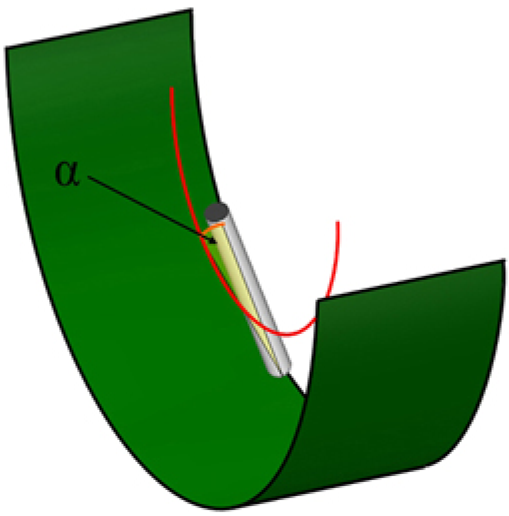
Example - Closest Point

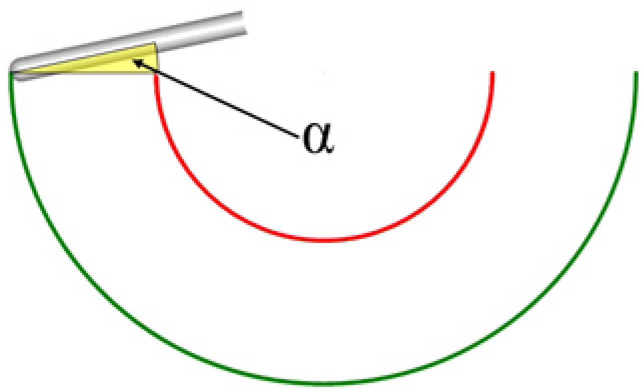
The example shows a curved surface with a certain radius. The tilt curve is above the surface. The tilt curve has the same shape as the surface but a smaller radius. Any point on the surface can be "connected" to the curve by the shortest distance. The result is that the tool axis is always normal between the curve and surface.





Example - Closest point with tilt angle:
In this example the tilt angle is set to 10°. Looking from the curve to the surface (below left) you'll see that the tool is tilting to the right on the right side from the curve and to the left from the left side of the curve (when following the moving direction). See the file `Closest Point-TiltAngle.vnc` for a working example.

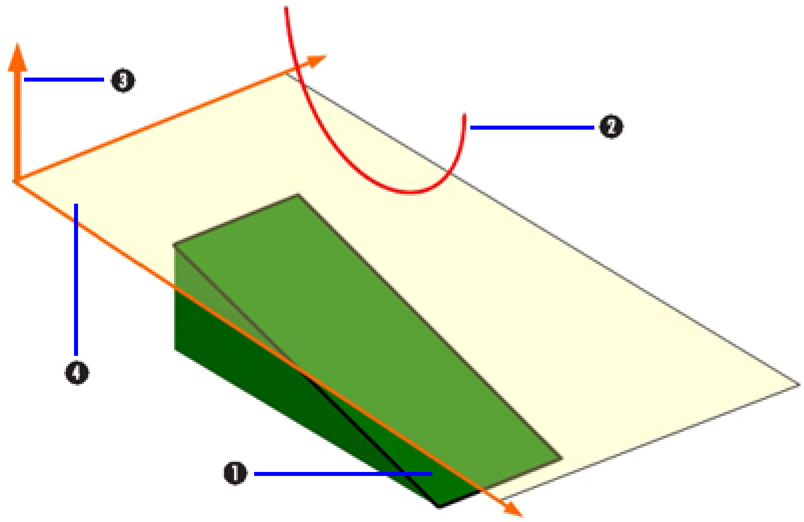




Angle from curve

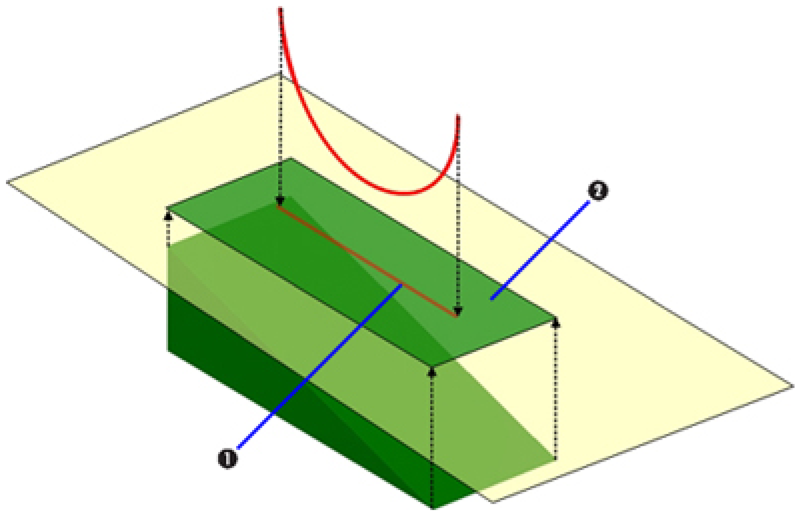
With this option the tool axis is aligned to a tilt curve during machining. The orientation of the tool axis is the projected length between your present toolpath point and the tilt curve. The tool's maximum tilting is vertical (90°) or horizontal (0°). For example if your tool axis is already tilted `35°` from the present toolpath point to your curve and you set a fixed tilt angle of `-60°` the tool would tilt just `0°` (horizontal).

How this works - a simple example
Here we have a surface with a tilt curve above the surface. To calculate the toolpath the system uses the vector of the main spindle, typically the Z axis, and the plane that is normal to this axis, usually XY plane.



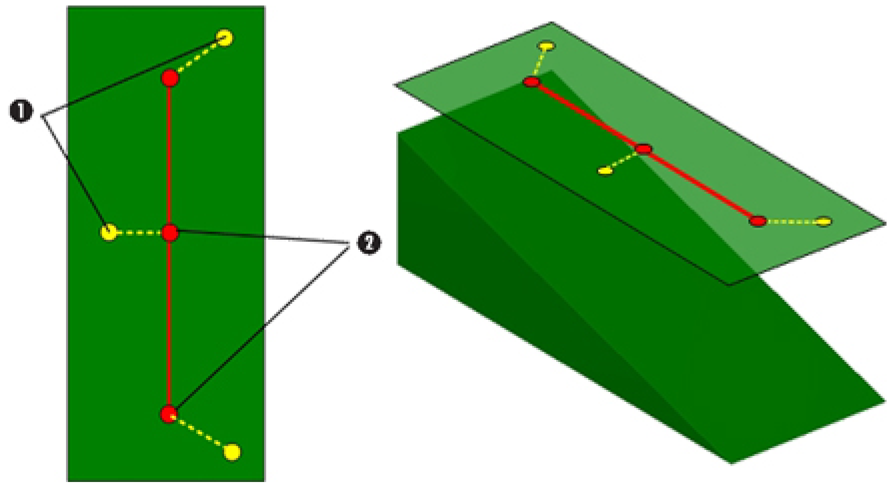
1. Drive Surface
2. Tilt Curve
3. Main Spindle Direction
4. Plane

Using this information the drive surface and tilt curve are projected onto the plane.



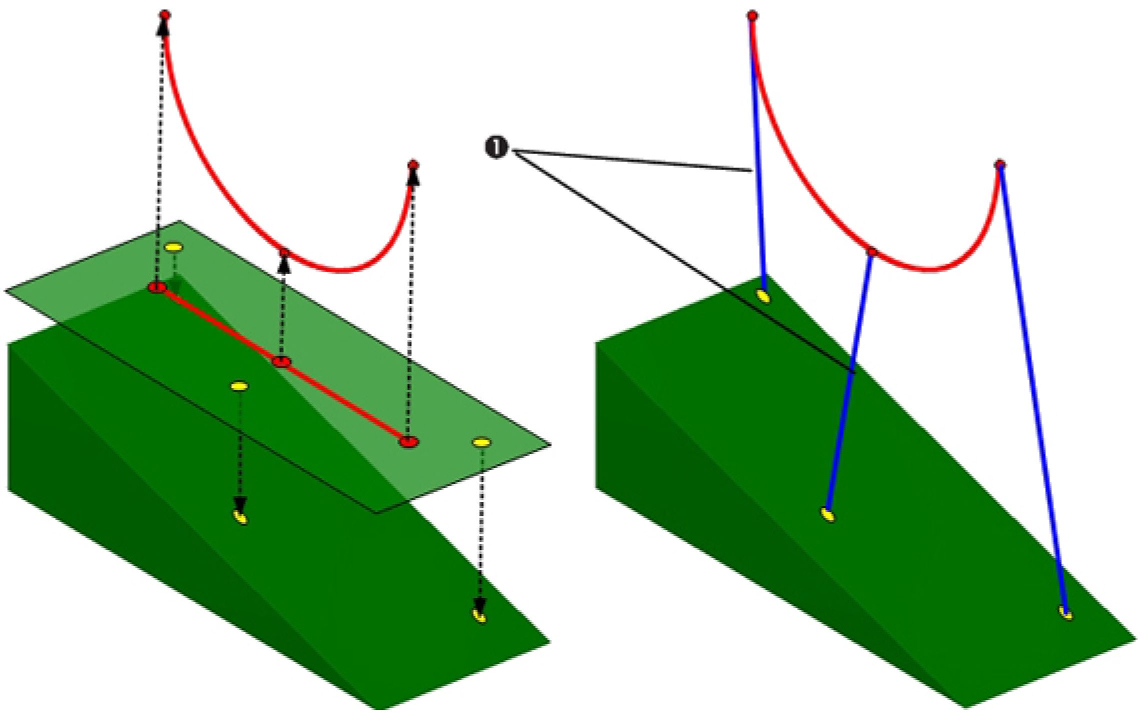
1. Projected Surface
2. Projected Curve

The system looks at all positions within the toolpath and finds the closest point on the projected curve to the toolpath position. Here we can see three random toolpath points on the surface (yellow dots) and the matching closest position on the projected curve along with a connecting line between them.



1. Toolpath Point
2. Curve Point

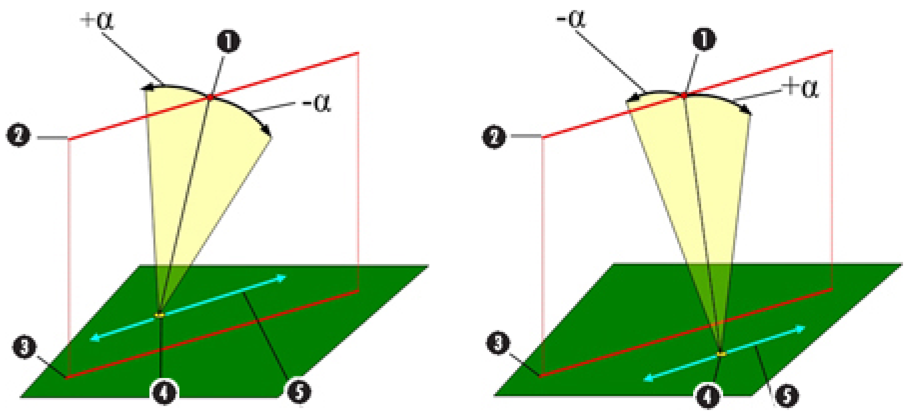
The points are projected back to the surface and up to the curve. By connecting these points we get the tool axis orientation (#1).



Fixed tilt angle

The Fixed tilt angle tilts the tool axis centerline away from the tilt curve. The direction is defined by the surface spanned by the curve point, surface point and the direction from the curve to the surface point.

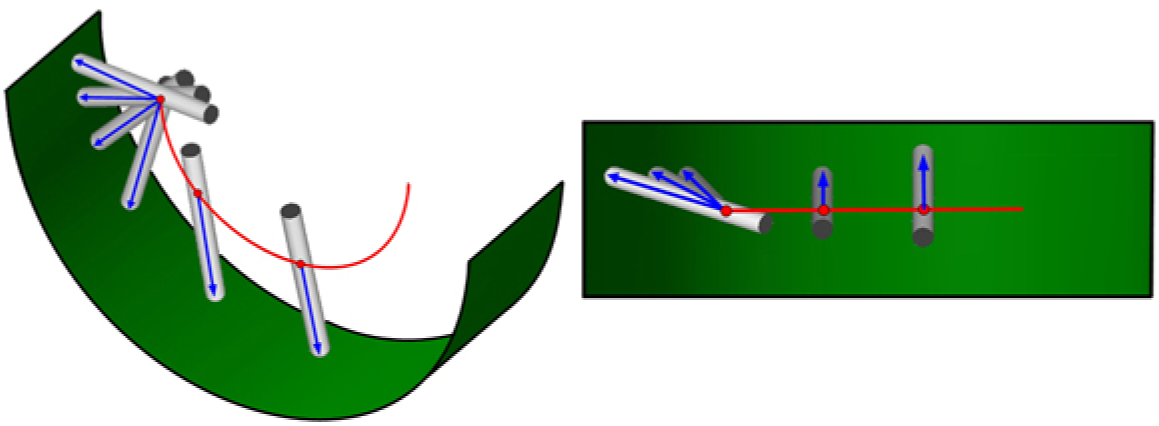
- When using positive tilt angles the tool tilts outwards (in moving direction).
- When using negative tilt angles the tool tilts inwards (in moving direction).



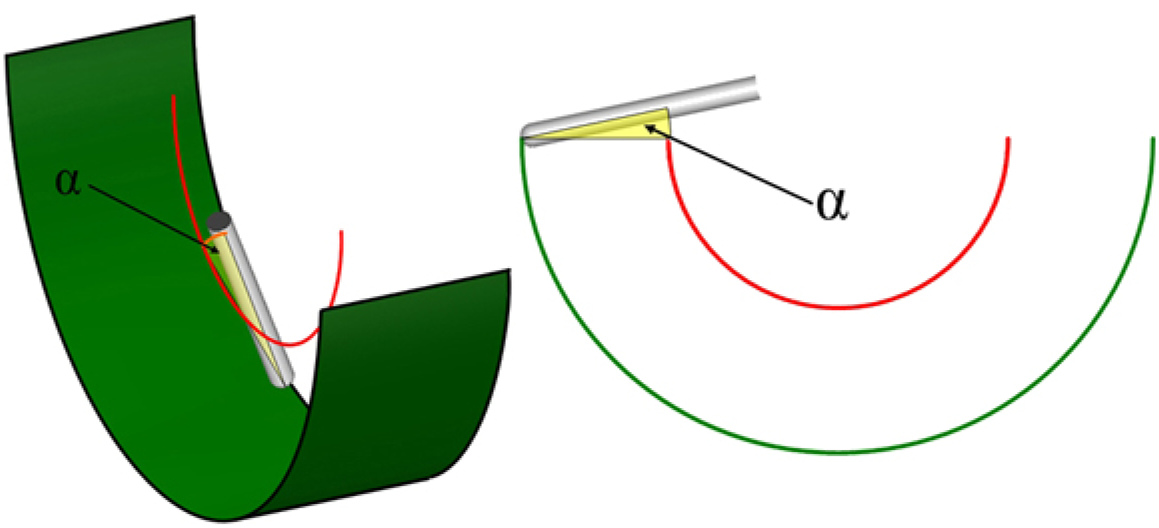
1. Curve Point
2. Tilt Curve
3. Projected Tilt Curve on the surface
4. Toolpath Point
5. Direction of movement

Example - Angle from curve:

In this example we have a curved surface with a certain radius and a tilt curve above the surface that has the same shape as the surface but a smaller radius. If we focus on the tool axis orientation at the end of the curve we see that the tool goes through that point because it is the closest point when viewed from the top (second image below). See the file `Closest Point-TiltAngle.vnc` for a working example.



When the tool is tilted by 10° we get a different result, as seen below.

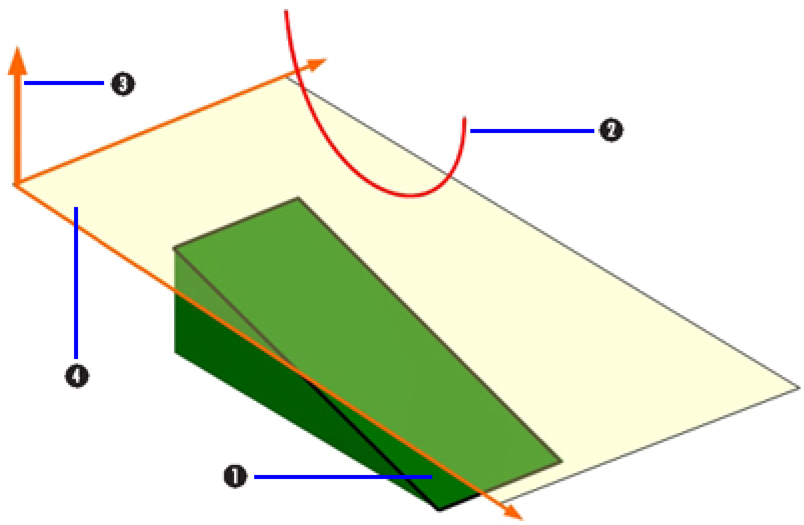


Angle from spindle, main direction

With this option the tool axis is aligned to a tilt curve during machining. The orientation of the tool axis is the projected length between your present toolpath point and the tilt curve. This option is similar to the Angle from curve option but the difference is that the tilting starts from the spindle main direction towards the tilt curve. The angle from the spindle main direction to the tilting curve is defined by the Fixed tilt angle. Therefore the default value of 0 degrees would make the tool axis orientation parallel to the spindle main direction. The tool's maximum tilting is vertical (90°) or horizontal (0°). The tilt curve must be above the drive surface.

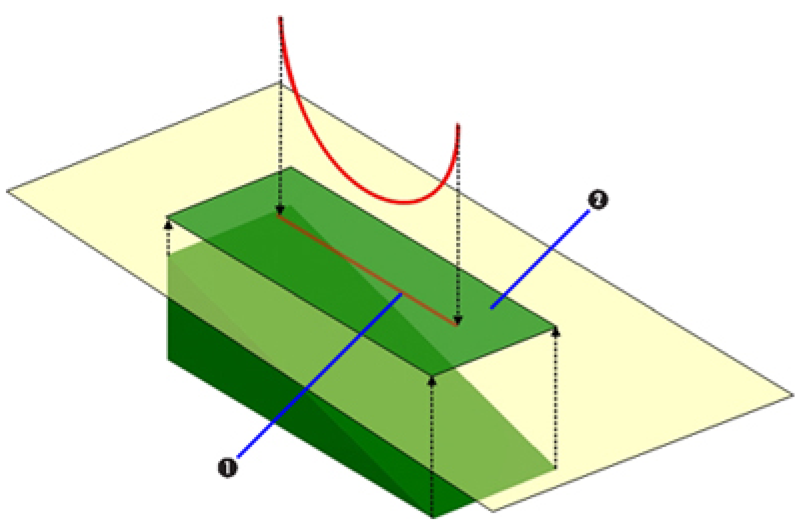
How this works - a simple example

Here we have a surface with a tilt curve above the surface. To calculate the toolpath the system uses the vector of the main spindle, typically the Z axis, and the plane that is normal to this axis, usually XY plane.



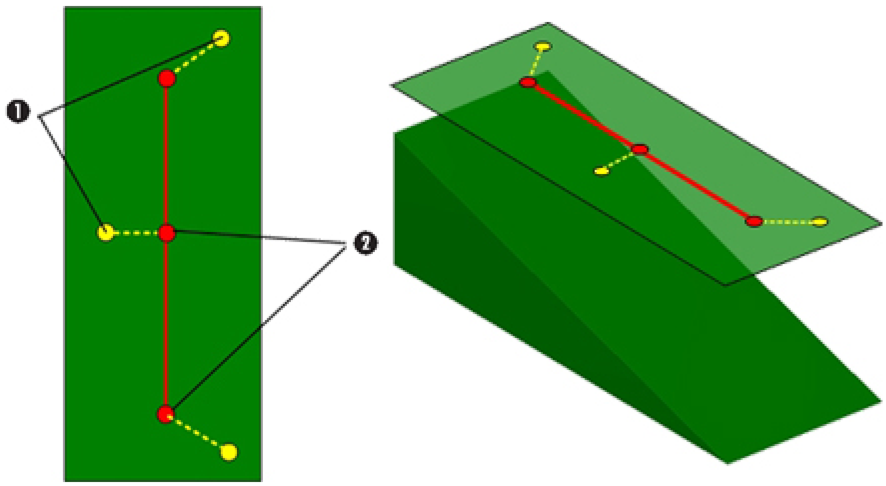
1. Drive Surface
2. Tilt Curve
3. Main Spindle Direction
4. Plane

Using this information the drive surface and tilt curve are projected onto the plane.



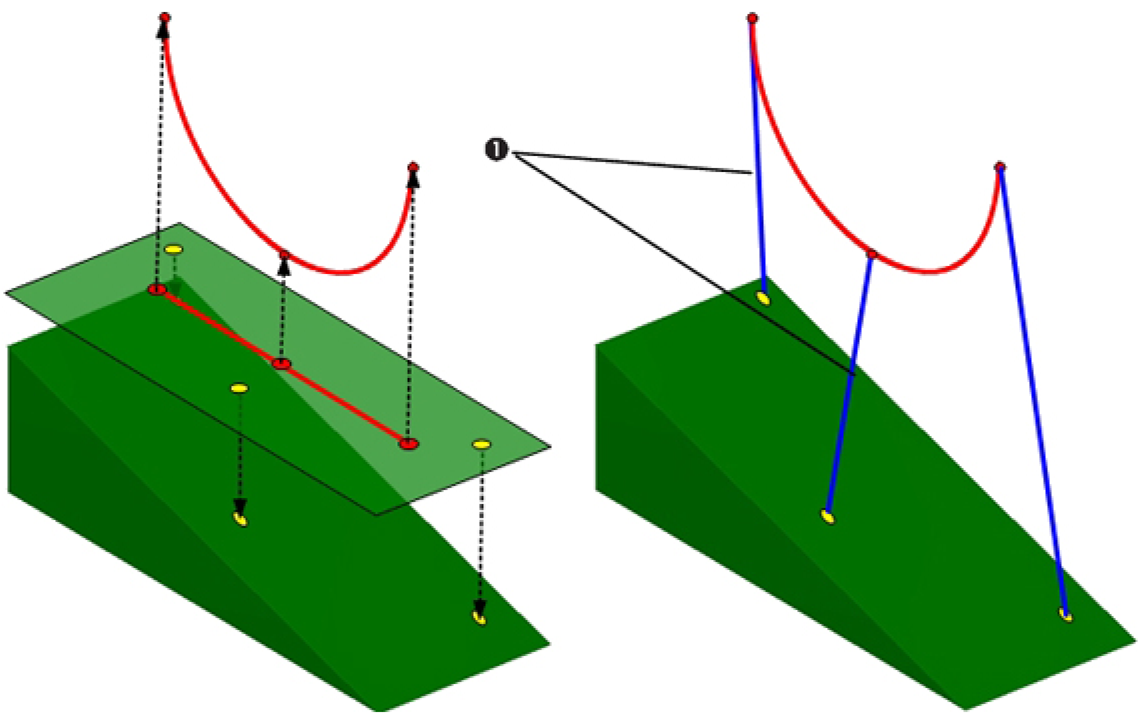
1. Projected Surface
2. Projected Curve

The system looks at all positions within the toolpath and finds the closest point on the curve to the toolpath position. Here we can see three random toolpath points on the surface (yellow dots) and the matching closest position on the curve along with a connecting line between them.



1. Toolpath Point
2. Curve Point

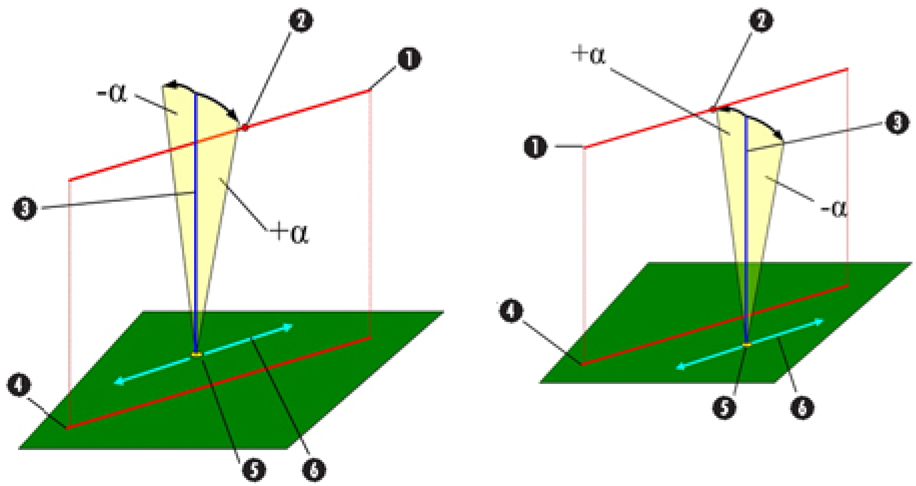
The points are projected back to the surface and up to the curve. By connecting these points we get the tool axis orientation (#1).



Fixed tilt angle

The Fixed tilt angle tilts the tool from the machine definition's spindle main direction vector to the tilt curve. The direction is defined by surface spanned by the curve point, surface point, and direction from curve to surface point.

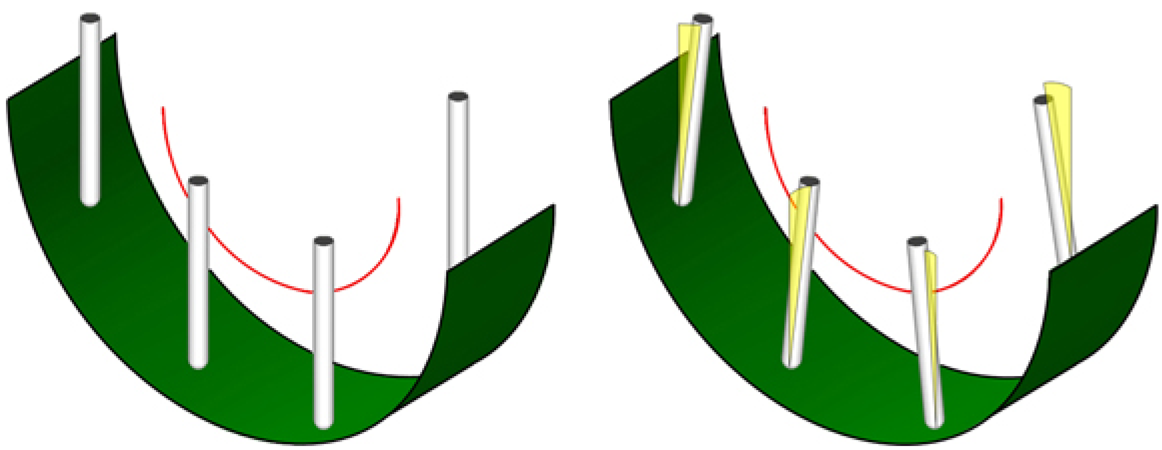
- When using positive tilt angles the tool tilts against the tilt curve (in the moving direction).
- When using negative tilt angles the tool tilts away from the curve (in the moving direction).



1. Tilt Curve
2. Tilt Curve Point
3. Main Direction Vector
4. Projected Tilt Curve on the surface
5. Toolpath Point
6. Direction of movement

Example - Angle from spindle, main direction:

The image below left shows the default toolpath. The tool is not tilted at all because the system uses the spindle main direction vector which is the Z axis in this case. In the image below right the tool is tilted 10 degrees from the spindle main direction towards the tilt curve. See the file `Closest Point-TiltAngle.vnc` for a working example.



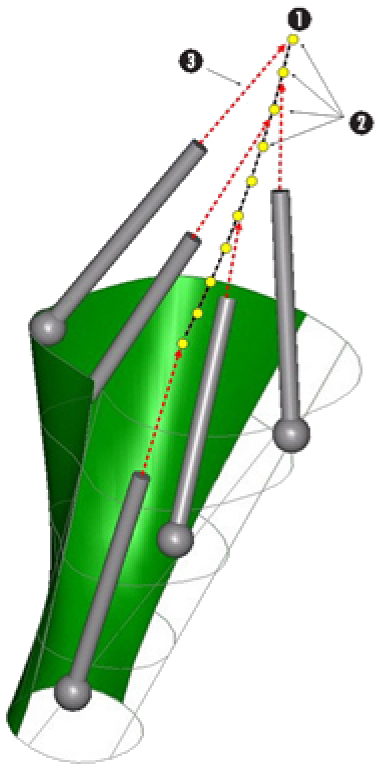
From start to end

This tilt type is useful for generating toolpath for tube milling and port machining (engine inlets). The tube milling usually is machined in constant Z cuts, resulting in cut slices. The amount of the constant Z cuts depends on the Stepover. The tilt curve is divided by the number of slices of the toolpath. Every slice is now aligned to its corresponding point on the curve.

When tube milling be sure that your tilt curve is located inside and/or above your drive surface and that the beginning of the curve starts on the correct end.

In this example the maximum step over is 10 mm. The toolpath for the tube has 10 slices, so the curve has 10 corresponding points for orienting the tool axis.

Note that if you use a collision avoidance strategy “Stop toolpath calculation” or “Leaving out gouge points” and this causes the last slices not to be machined, it will look as if in the last cut the spindle is pointing to a point on the curve other than the last one.



1. Tilt Curve
2. Curve Points
3. Tool Axis Direction

Automatic curve

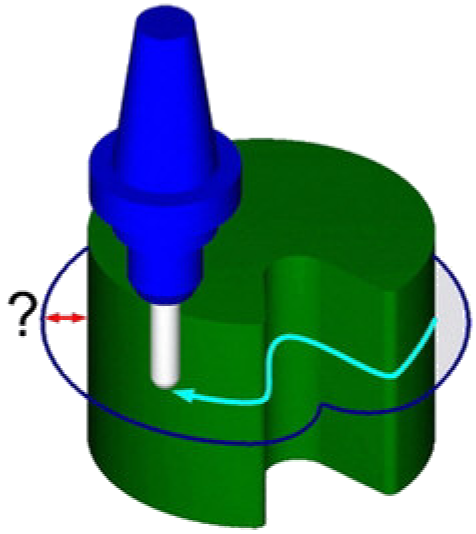
The Automatic curve option is the only strategy where the curve is calculated automatically by the system for each contour and the user does not need to provide any tilting curve geometry. The toolpath curve is determined by slicing the surface with a plane parallel to the XY plane at each cut depth. The automatically generated curve tries to dampen the tool motion by a user defined Damping Distance. This type of tilting is especially helpful for cutting deep cavities. Any Fixed tilt angle is defined from the Z axis towards the internally calculated automatic curve at each point of the tool motion.

Damping Distance

The Damping distance is the distance between the drive surface and the generated curve. This can be set as an actual value or as a percentage of the tool diameter.

Example

In this picture you can see the drive surface compared to the automatic curve. The curve is similar to the shape of the drive surface but the curve is smoother than the actual drive surface when inside the indent. See the file `Automatic Curve.vnc` for a working example.

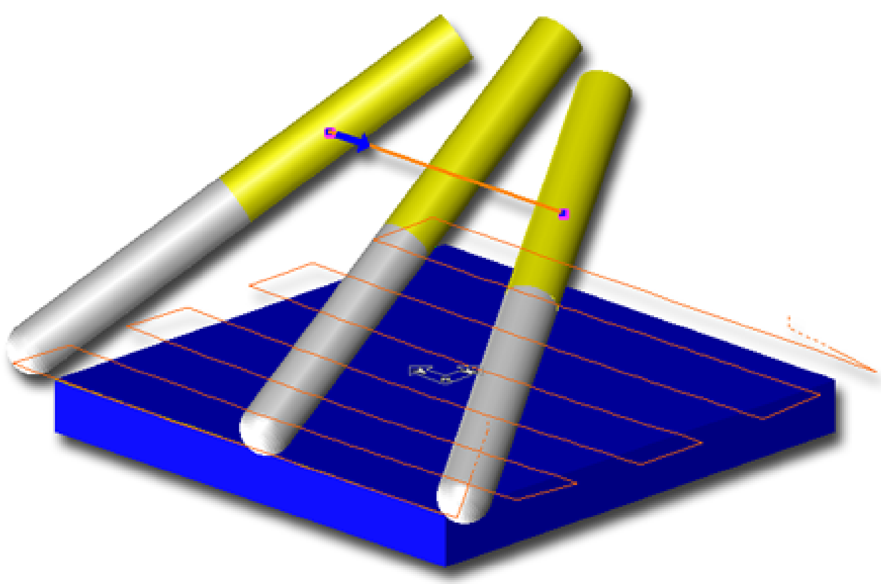


From start to end for each contour

This option gradually tilts the tool through the defined drive curve as the toolpath is followed. At the start and end of the toolpath the tool tilts through the start and end of the curve. Halfway through the toolpath the tool will be at the midpoint of the curve. A typical toolpath contains many contours so the gradual tilting will happen for EACH of the toolpath contours.

Example

In this picture you can see the tool tilting along the drive curve. The tool goes through the start and transitions to the end position. This will be repeated for each back and forth pass. See the file `Tilted Through Curve - From Start To End For Each Contour.vnc` for a working example.



Tilted through lines

With this option the tool axis will be approximated along the toolpath to lines defined in the geometry. This parameter only works with 4-axis and 5-axis output. There are two options for controlling the tilt: All lines weighted by distance and Always closest two lines. The line is selected by clicking on the Tilt Line button.

Use Tilt Through

All lines weighted by distance

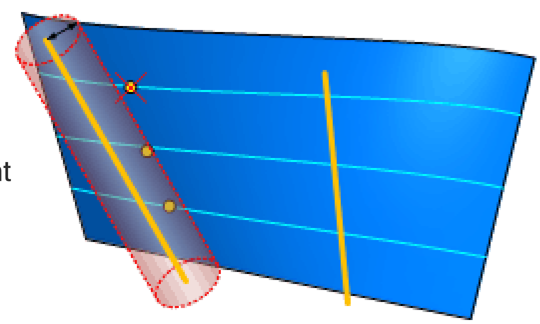
Here the direction of the tool axis will be approximated through all lines which are close to your toolpath. So if you have many lines tilted in different directions, the tool axis is tilted with the average of these lines. That also means that with this option you will almost never have the same orientation as a specific line.

Always closest two lines

Here the direction of the tool axis will be approximated through two neighboring lines along the toolpath. These lines are also the closest to each other. The tool axis will follow the two lines when it is at the closest point to those two lines (first move between the pair with the first line and last move between the pair with the 2nd line). All moves between the pair will be approximated using a linear interpolation between the two lines.

Tilting lines maximum snap distance

This value specifies the farthest distance separating a tilt line from the toolpath. If the line lies within this distance, it will be considered for tilting. Any element farther away than the specified distance is ignored.

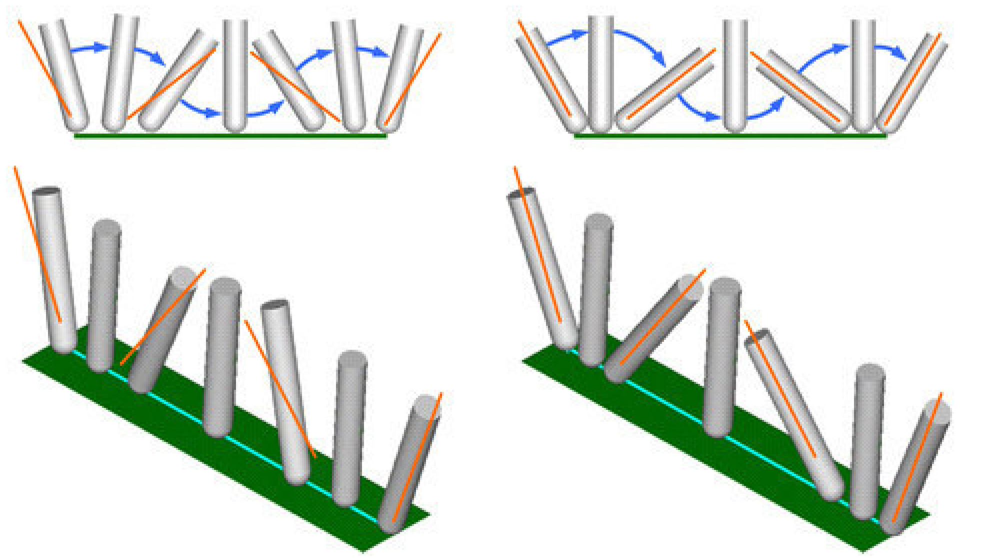


Imagine a tube drawn around the toolpath line so that its radius is equal to the maximum snap distance. Every tilt line that you select is checked to see if it intersects this imaginary tube (see illustration). If this tube is intersected, then the tilt line is used in the calculation, with the result that the tool will tilt between the two most appropriate of chosen tilt lines.

Please note that when you choose the lines from your geometry, make sure that the chaining of all the lines point in the same relative direction. That is to say that all lines point away from your drive surface or all lines should point to the drive surface. If you mix the chaining directions the tool will also make direction changes.

Example

In these images you can see the green drive surface and the four orange tilt lines. In the first image (All Lines Weighted By Distance) you can see the toolpath is approximated through all lines, meaning the orientation of the tool axis never has the same orientation as any single line. In the second image (Always Closest Two Lines) you can see the toolpath is approximated through two neighboring lines, meaning the orientation of the tool axis is the same as the tilt lines. See the file `Tilted Through Lines.vnc` for a working example.



All Lines Weighted By Distance | Always Closest Two Lines

Tilted from point away

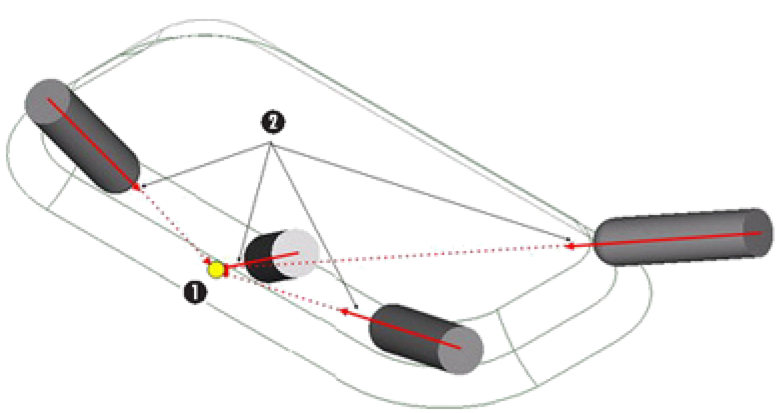
With this option the tool axis is always pointing away from a designated point in the part. This function is the opposite of Tilted through point; see “Tilted through point” on page 174. The selected point must be located under your drive surface, not above or on the surface.

According to the option you choose for Point tilt type, the tool axis will align either from point towards axis or from axis towards point.

Additionally, you can set a separate Tilt angle. This means the tool axis will be aligned through the specified point, but the tool will then be tilted at the specified angle, either relative either to a primary axis (X-axis, Y-axis, Z-axis) or relative to a Line that you define or choose from the workspace by clicking the ellipsis button (...).

Example

In this example the point is located somewhere under the surfaces. During machining, the tool axis is always aligned away from this point.



1. Point
2. Tool Axis

Tilted from curve away

During machining on your drive surface the tool points from the tilt curve away. Depending on your curve tilt type the tool orientation and alignment to the curve changes. The selected curve must be located under your drive surface, not above or on the surface.

Select the curve by clicking on the Tilt curve button. For Curve tilt type, choose one of these menu options: Closest point, Angle from curve, Angle from spindle, main direction, From start to end and From start to end for each contour.

Additionally, you can set a separate Tilt angle. This means the tool axis will be aligned through the specified point, but the tool will then be tilted at the specified angle, either relative either to a primary axis (X-axis, Y-axis, Z-axis) or relative to a Line that you define or choose from the workspace by clicking the ellipsis button (...).

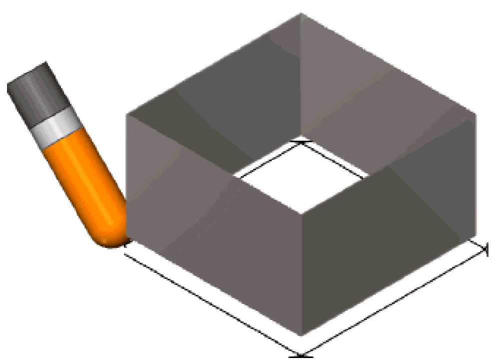
Advanced options for Curve tilt type

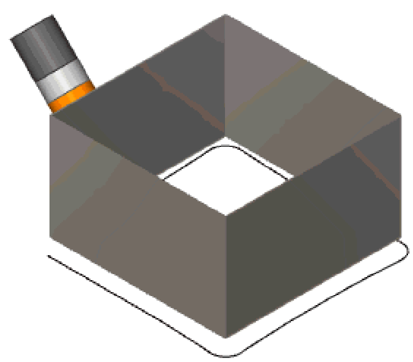
For some choices of Curve tilt type, an Advanced button becomes available. Depending on the type, the **Advanced** dialog allows you to exert fine control over several aspects of toolpath, noted as follows.

Side tilt fanning distance

This option is when the drive surface geometry has two crossing curved surfaces. The aim is to avoid a sudden jump in tool axis when changing the surfaces.

The fanning distance is the distance from the crossing point of the surfaces and the point where the cross fading of the tool axis begins. The bigger the fanning distance, the earlier the tool begins to tilt.





Toolpath without fanning	Toolpath with fanning

Note: Cutting with the side of the tool requires a proper definition of the best side direction at each position of the toolpath. In case of surfaces that are almost ruled and have a slight curvature in the secondary direction, the best side direction between two surfaces jumps at the boundary between such two surfaces. In such case, the fanning distance is applied to dampen out such jerky changes of orientation.

Curve Tilt Type

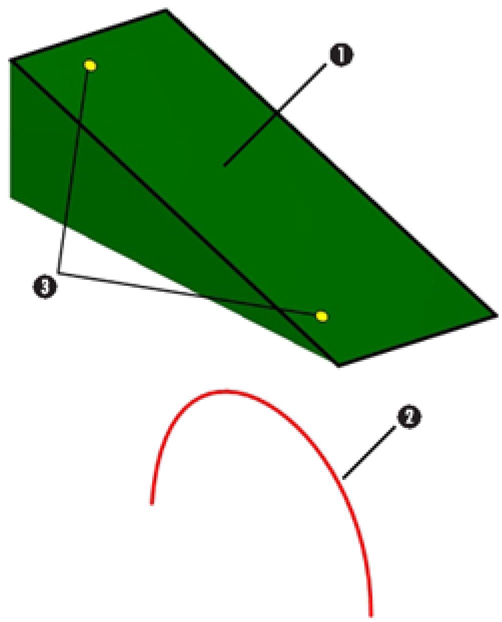
Closest point

Here the direction of your tool axis is the same as the shortest distance between your present toolpath point and the tilt curve. This item uses the Fixed tilt angle parameter.

The following example shows a surface with a tilt curve beneath. You can see that the tool axis has the same direction as the shortest 3D distance between the surface toolpath point and the curve.

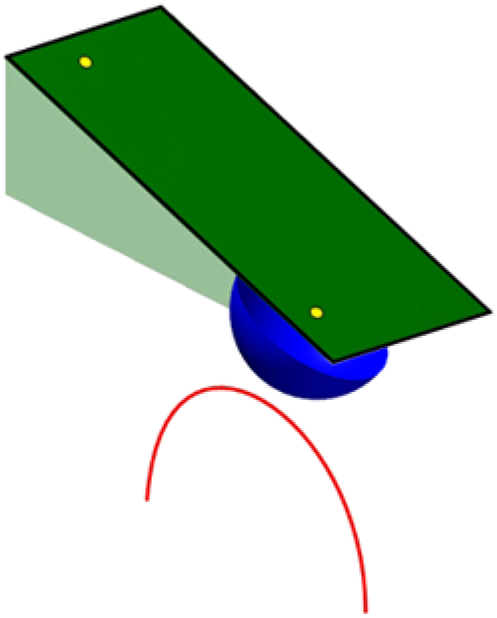
How it works explained with a simple example

Let's start with a surface and a tilt curve beneath the surface. Additionally, we will look at two random points on the toolpath.

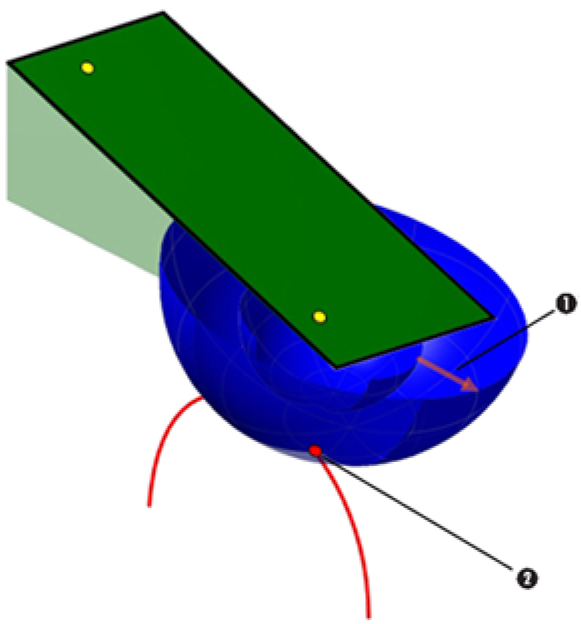


1. Drive Surface
2. Tilt Curve
3. Toolpath points

Now, let's imagine a small sphere built around one of the toolpath points.

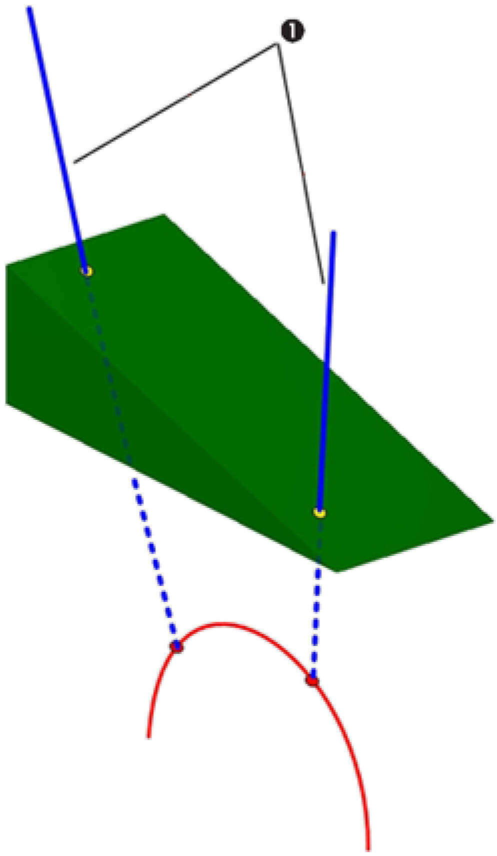


Then the sphere is expanded until the sphere touches the curve. The location of the sphere/curve contact gives us a point.



1. Expanding curve
2. Point of contact

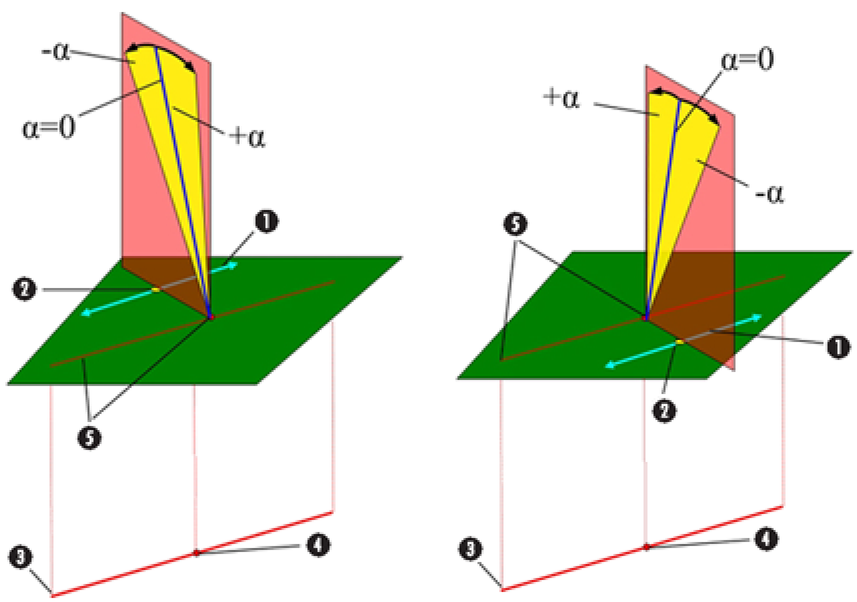
Now imagine a line between this point on the curve and the toolpath point. This line is the tool axis orientation.



1. Tool Axis Orientation

Fixed tilt angle

This parameter allows you to set an optional fixed tilt angle. This angle tilts the tool axis centerline away from the tilt curve. The direction is defined by the surface spanned by the curve point, a surface point and the direction from the curve to the surface point. Positive tilt angles cause the tool to tilt inwards (in the moving direction). Negative tilt angles cause the tool to tilt outwards (in the moving direction). The maximum tilt is vertical (90°) and horizontal (0°).



1. Moving Direction
2. Toolpath Point
3. Tilt Curve
4. Contact Point on the Tilt Curve
5. Projected Tilt Curve and Point on the Surface

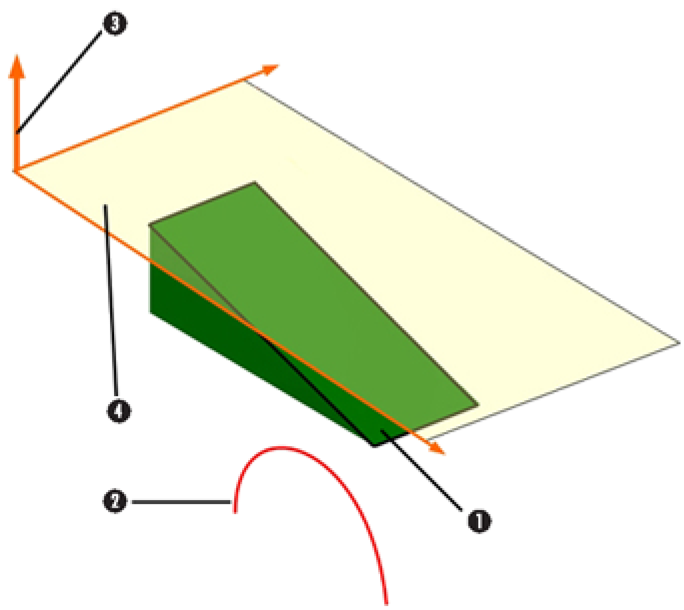
Left Side	Right Side

Angle from curve

This option works similar to the Closest point option. The difference is that an additional Fixed Tilt angle can be given to tilt the tool axis centerline from the tilt curve to the direction of spindle direction. For example, a value of 0 degrees here would make this option behave like the Closest Point option. The direction of your tool axis is the projected length between your present toolpath point and the tilt curve. This is the 2D distance. This item uses the Fixed tilt angle parameter.

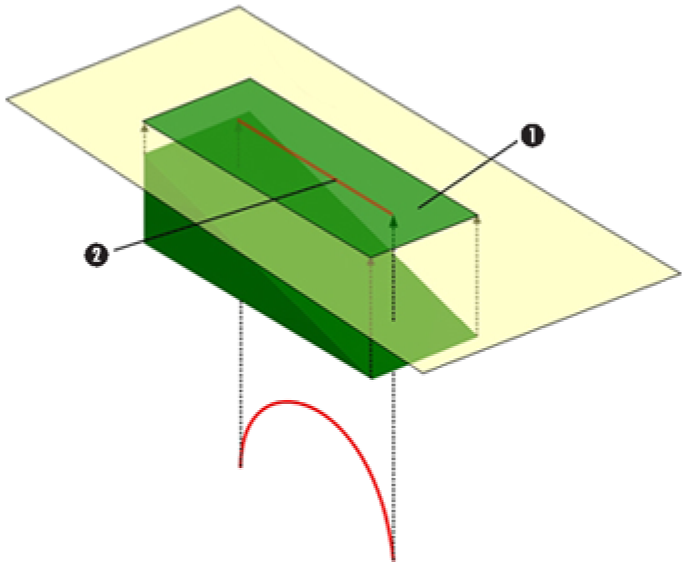
How it works explained with a simple example

We start with a drive surface and a tilt curve beneath the surface. The system looks at the machine definition's spindle main direction vector (usually the Z axis) and the plane to which this axis is normal (usually XY plane).



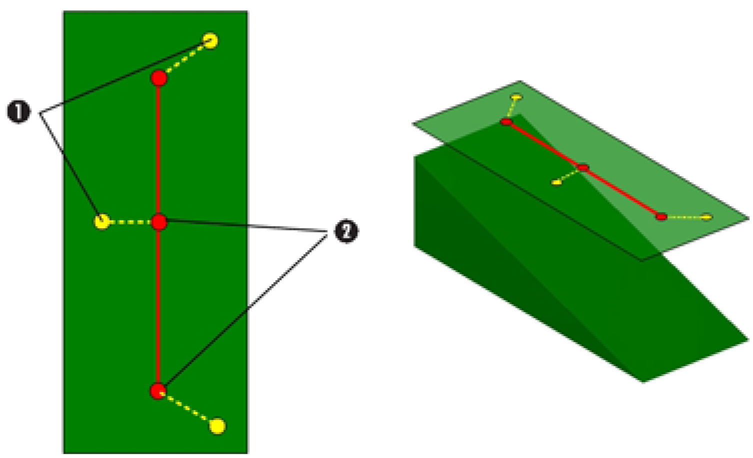
1. Drive Surface
2. Tilt Curve
3. Main Spindle Direction
4. Plane

The system then projects the drive surface and the tilt curve onto the plane.



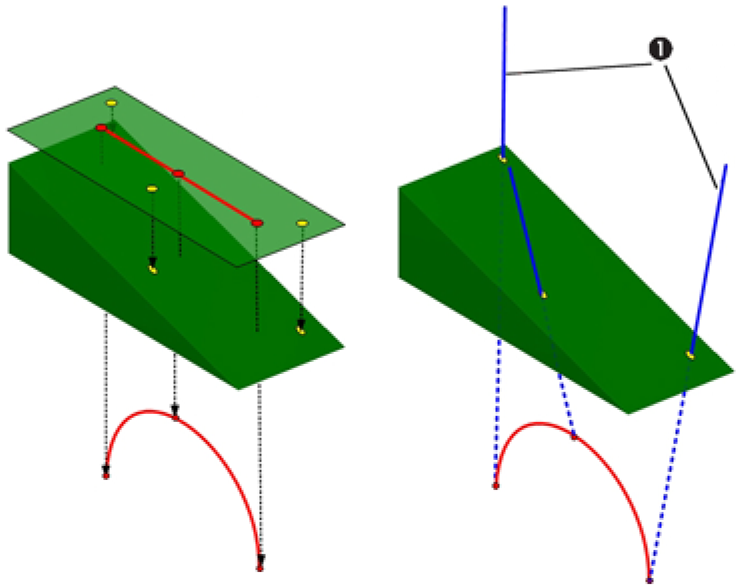
1. Projected Surface
2. Projected Curve

Here we see three random toolpath points on the projected surface. The system finds the closest point on the projected curve to the toolpath point. This is represented by the yellow one.



1. Toolpath Point
2. Curve Point

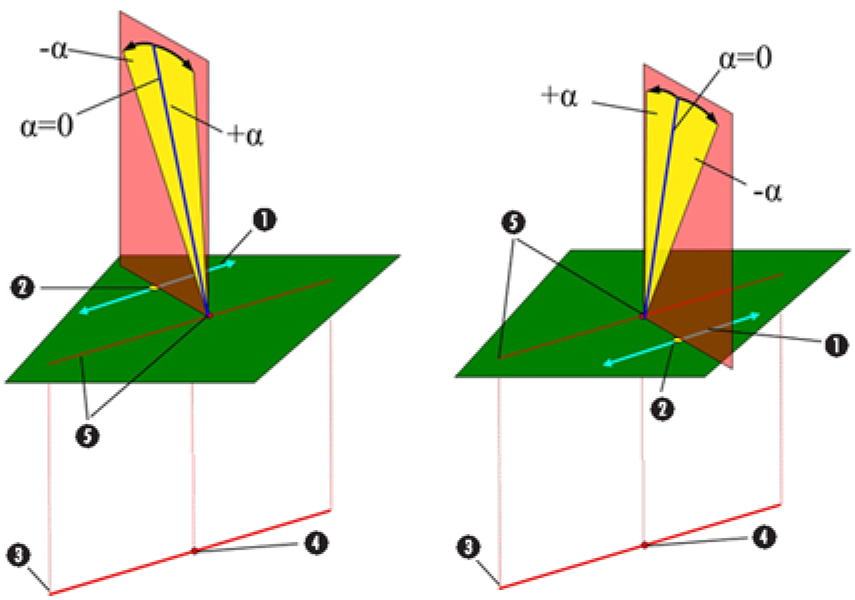
The curve points are projected back to the curve. By projecting a line between the toolpath points and the curve points the tool axis orientation is determined.



1. Tool Axis Orientation

Fixed tilt angle

This parameter allows you to set an optional fixed tilt angle. This angle tilts the tool axis centerline away from the tilt curve. The direction is defined by the surface spanned by the curve point, a surface point and the direction from the curve to the surface point. Positive tilt angles cause the tool to tilt inwards (in the moving direction). Negative tilt angles cause the tool to tilt outwards (in the moving direction). The maximum tilt is vertical (90°) and horizontal (0°).



1. Moving Direction
2. Toolpath Point
3. Tilt Curve
4. Contact Point on the Tilt Curve
5. Projected Tilt Curve and Point on the Surface

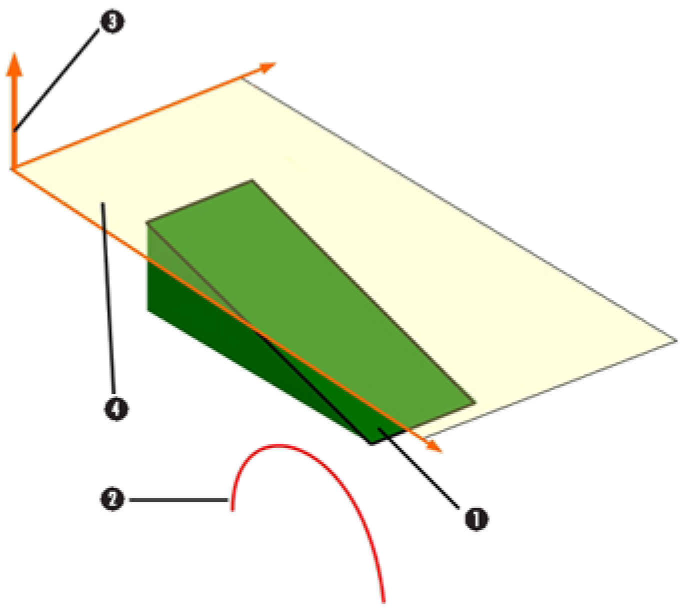
Left Side	Right Side

Angle from spindle, main direction

This option works similar to the Angle from curve option. The difference is that the tilting starts from the spindle main direction towards the tilt curve. The angle from the main direction to the tilt curve is defined by the Fixed Tilt angle. A value of `0` degrees makes the tool axis orientation parallel to the spindle main direction.

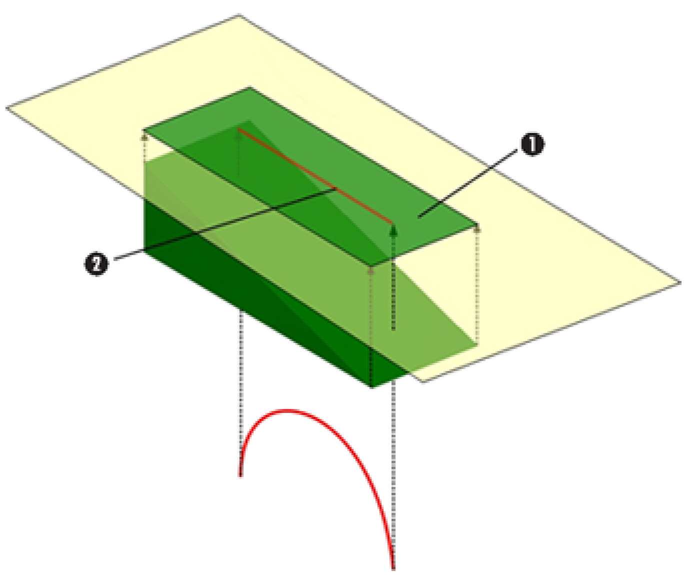
How it works explained with a simple example

We start with a drive surface and a tilt curve beneath the surface. The system looks at the machine definition’s spindle main direction vector (usually the Z axis) and the plane to which this axis is normal (usually XY plane).



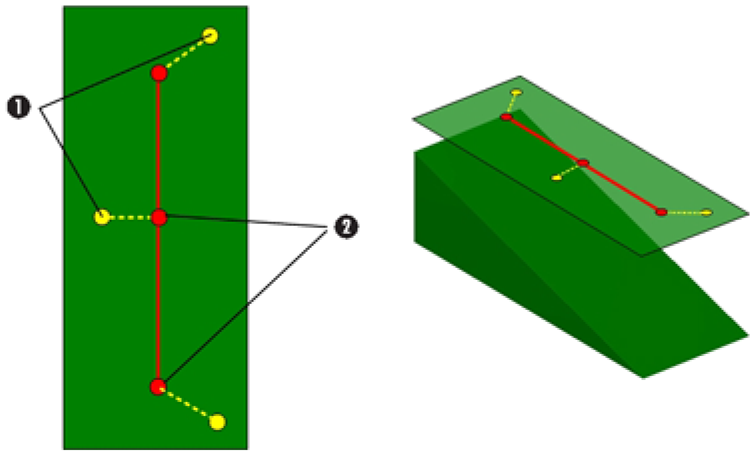
1. Drive Surface
2. Tilt Curve
3. Main Spindle Direction
4. Plane

The system then projects the drive surface and the tilt curve onto the plane.



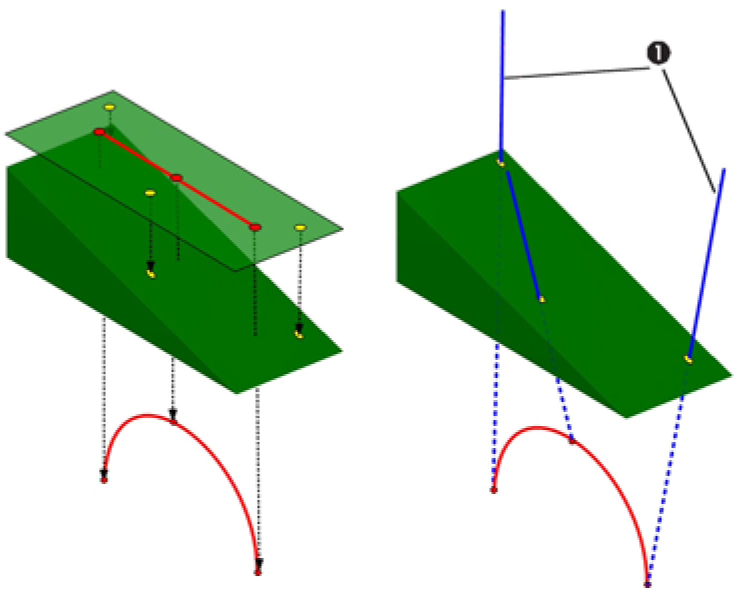
1. Projected Surface
2. Projected Curve

Here we see three random toolpath points on the projected surface. The system finds the closest point on the projected curve to the toolpath point. This is represented by the yellow one.



1. Toolpath Point
2. Curve Point

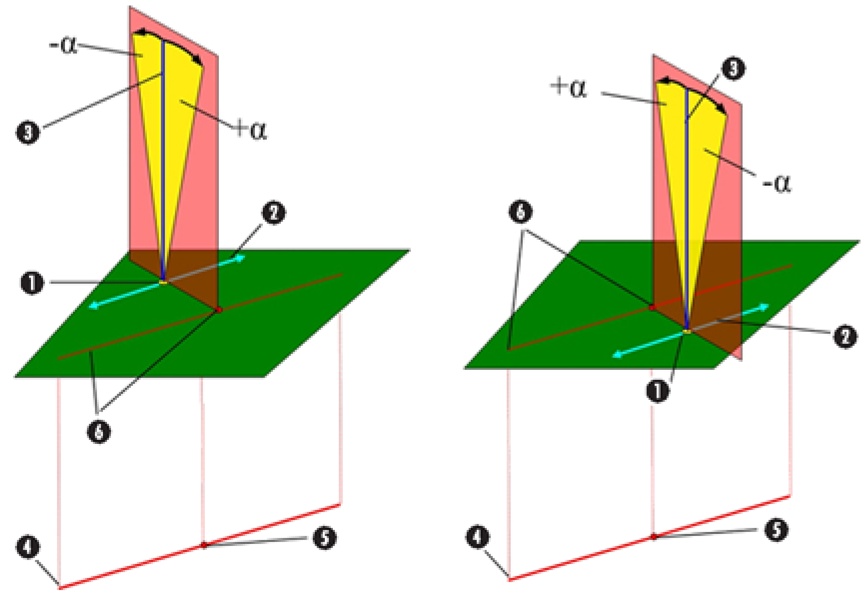
The curve points are projected back to the curve. By projecting a line between the toolpath points and the curve points the tool axis orientation is determined.



1. Tool Axis Orientation

Fixed tilt angle

The Fixed tilt angle tilts the tool from the machine definitions spindle main direction vector to the tilt curve.The direction is defined by the surface spanned by the curve point, the surface point and the direction from the curve to surface point. Positive tilt angles cause the tool to tilt inwards (in the moving direction). Negative tilt angles cause the tool to tilt outwards (in the moving direction). The maximum tilt is vertical (90°) and horizontal (0°).

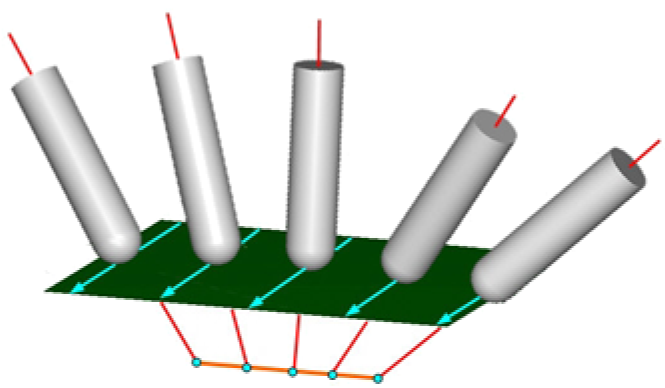


1. Toolpath Point
2. Moving Direction
3. Main Direction Vector
4. Tilt Curve
5. Tilt Curve Point
6. Projected Tilt Curve and Point on the Surface

Left Side	Right Side

From start to end

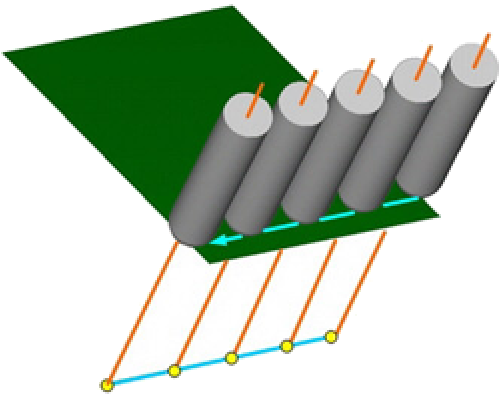
This tilt type is used for generating toolpaths for tube milling and port machining (engine inlets). The tube milling usually is machined in constant Z cuts resulting in cut slices. The amount of the constant Z cuts depends on the Stepover. The tilt curve is divided by the number of slices in the toolpath. Every slice is now aligned to its corresponding point on the curve. The tilt curve must be below the drive surface.



- It is recommended that the drive curve end you select is in the same area of the part as where you want the toolpath to start.
- In some cases using the collision avoidance strategies Stop toolpath calculation or Leaving out gouge points can cause the last slices to not be machined. If this occurs the last cut will look as if the spindle is pointing to a point on the curve other than the last one.

From start to end for each contour

The tilt curve is divided by the number of toolpath points in the present cut. The orientation of the tool axis at the present toolpath point is aligned to its corresponding point on the curve. This process repeats with every new cut. The tool will gradually transition from point to point.



Tilting Relative to Impeller Machining Layer

This tilting option is meant to be used for impeller-type parts. The tool will stay normal to the floor face of the impeller. The tilting to the lead and lag can be adjusted by a global lead/side angle and additionally with a local lead angle at the leading edge, splitter edge, and trailing edge. The edges of the geometry are defined with a line.

Tilt lines

The lines you select will be used to apply a local tilting at the leading edge, splitter edge and trailing edge. So the line must be located and orientated along this edge. Note that a positive angle tilts against the rotation axis and a negative angle tilts against the rotation axis.

Impeller rotation axis

This setting should represent the impeller rotation axis (usually the Z axis).

Global lead angle

The global lead angle defines the lead angle applied to the tool which is initially normal to the floor. The aim is to provide a leading angle to avoid the tool cutting with its backside. Note that a positive angle tilts against the rotation axis, a negative against the rotation axis.

Additional lead angle

This lets you set an local lead angle at the leading edge, splitter edge, and trailing edge.

Global side angle

The side lead angle defines the side angle applied to the tool which is initially normal to the floor. The aim is to influence side tilting into a preferred general direction.

Approximate

The Approximate option selects the calculation method used to determine a side tilt vector. There are two types of approximation.

None

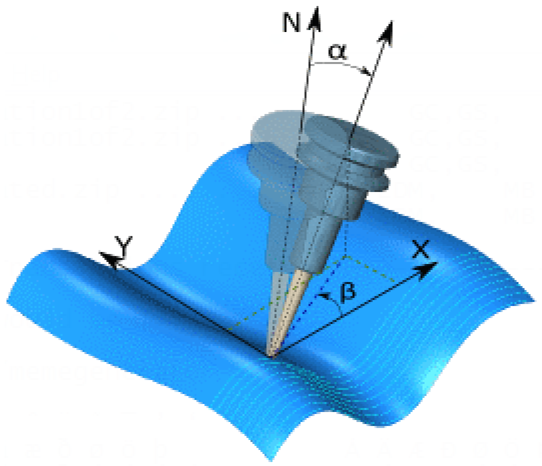
There is no approximation.

Smooth (local)

The orthogonal vectors are computed by a distribution of the local neighboring orthogonal vectors.

Be tilted relative to contact point

With this Tool axis control strategy, you can give a lead angle to the cutting direction as well as tilt angle to the side of the cutting direction. All angles are in degrees.



Be tilted relative to contact point:
N: normal vector;
α =lead angle (in this case towards **X**)
β =lead angle (in this case towards **Y**)

Lead angle to cutting direction

This angle (labeled α in the illustration) defines the lead/lag angle of the tool axis from the surface normal in the direction of the toolpath slice. Positive angles tilt the tool forward in motion direction (lead angle). Negative angles tilt the tool back against motion direction (lag angle).

- The lead angle is relative to the cutting direction.
- In case of Zig zag machining, the tool orientation flips with every new cut.
- In case of One way machining, the tool orientation does not change.

Tilt angle at side of cutting direction:

This angle (labeled β in the illustration above) defines the side tilt value of the tool axis from the surface normal direction based on the toolpath direction. Positive angles tilt to the left side (in motion direction), negative angles tilt the tool axis to the right side (in motion direction).

- The side tilt angle is absolute to the cutting direction.
- In case of Zig zag machining, the tool orientation does not flip with every new cut. The tool keeps its orientation as specified for the first cut.

Run tool

- About Run Tool
- Tool Area Definitions

About Run Tool

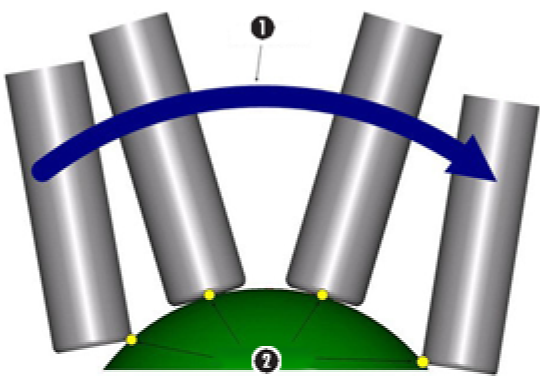
This parameter defines the contact point between the tool and drive surfaces. The parameters include Auto, Center, Front, Radius and User given point. The actual point of contact depends upon the tool used. See Tool Area Definitions for illustrations of the contact points.

Auto:
The Auto option is the automatic mode. In this mode, the system determines where the tool contacts the surface.

If you change the orientation of the tool, then the surface contact point remains and the contact point on the tool moves from the tip of the tool to the radius of the tool still maintaining the tangency between tool and surface.

Example

In this example, the tool axis limit angle is 70 degrees. Now, if the tool machines under this limit, the system chooses run tool option “center”. When the tool comes to the limited areas, the touch point changes to the tool radius. For a working example, see sample part `Run Tool-Auto_Front-Radius-Center.vnc`.

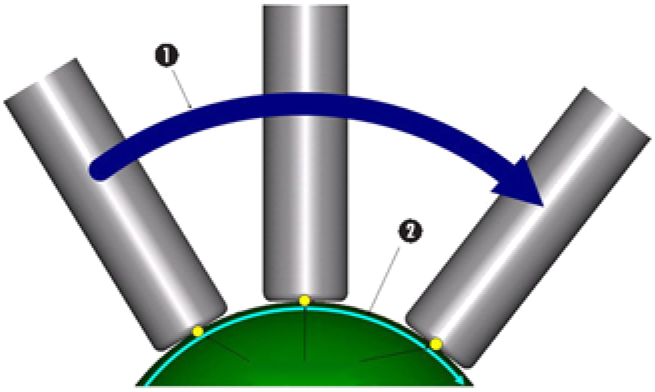


1. Move Direction
2. Touch Point

Center:
If this parameter is set to Center, then the tip of tool is touching the surface contact point. If the tool axis orientation is changed due to tilting options, then the tool is tilted around this tip point. In such cases, the tool and surface are not tangential anymore and the tool will gouge the surface. To avoid this, turn on gouge checking and set the first gouge check strategy to Retract Tool.

Example

Here you can see the tool always contacting the surface at the center. For a working example, see sample part `Run Tool-Auto_Front-Radius-Center.vnc`.



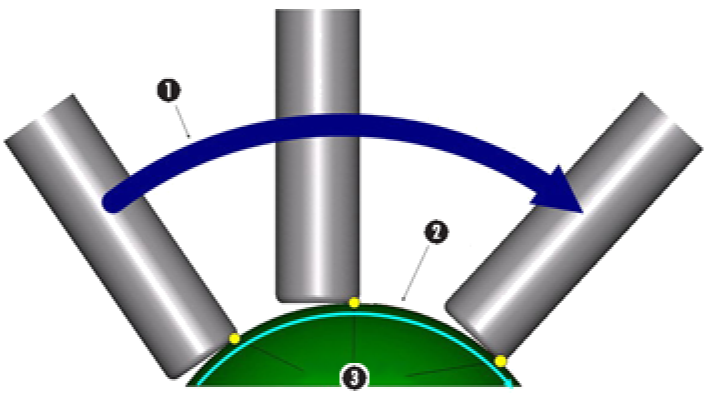
1. Move Direction
2. Toolpath

Front:

The option Front is similar to Tool Area Definitions and forces the tool contact point to be a fixed point on the tool. All changes to tool orientation are done around this pivot point and this will also cause gouging of the drive surfaces. Setting a gouge control strategy is critical to work with this option.

Example

Here you can see the tool always contacting the surface at the front. For a working example, see sample part `Run Tool-Auto_Front-Radius-Center.vnc`.



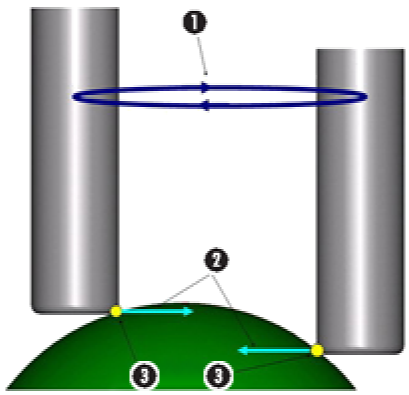
1. Move Direction
2. toolpath
3. Tool Front

Radius:

If this parameter is set to Radius, then the tangency is maintained as in the case of Tool Area Definitions. The difference is that for a bull nose tool, the tip of the tool is never used as the contact point on the drive surfaces.

Example

In this example, a bull mill machines around the sphere. Regardless of toolpath position, the tool contact point is always at the radius. For a working example, see sample part `Run Tool-Auto_Front-Radius-Center.vnc`.



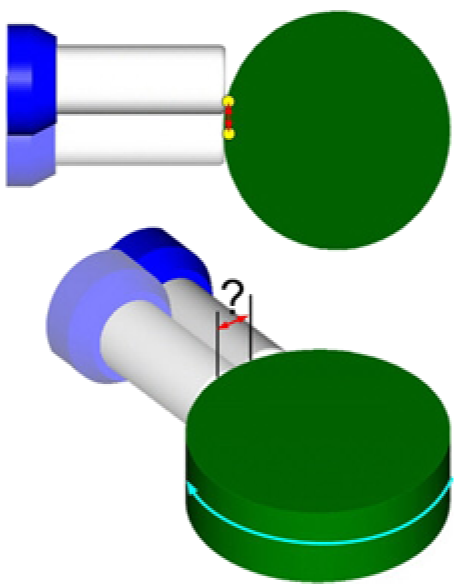
1. Move Direction
2. toolpath
3. Tool Radius

User Given Point:

You can specify the contact point between the tool tip and surface in terms of a front and/or side shift distance. The tool contact point is offset by this distance. These values are relative to tool at center contact point and the toolpath direction. Positive values for side shift move the tool to the left (in machining direction). Positive values for front shift move the tool forward (in machining direction).

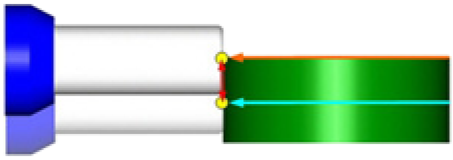
Front Shift:

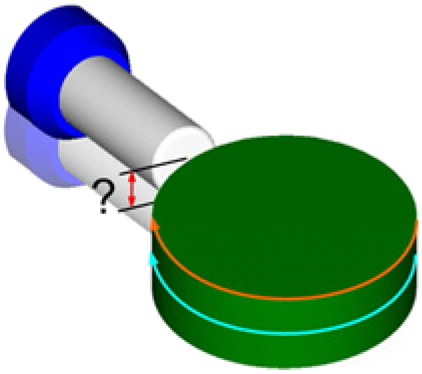
This amount shifts the tool contact point along the toolpath direction. A positive value causes the contact point to be forward of the center point with respect to the toolpath direction. A negative value results in a contact point that is behind the center point with respect to the toolpath direction. A value larger than the tool radius results in a “virtual” contact point where the tool no longer contacts the part.



Side Shift:

Side means perpendicular to or across the motion direction. A positive value ensures that from the tip center of the tool, the side part of the tool contacts the surface.

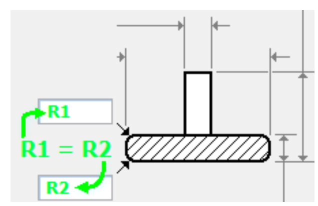


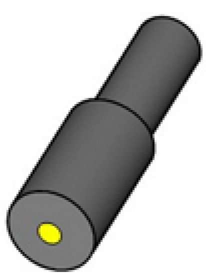
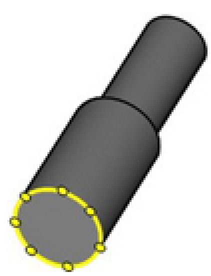
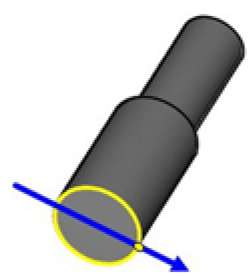


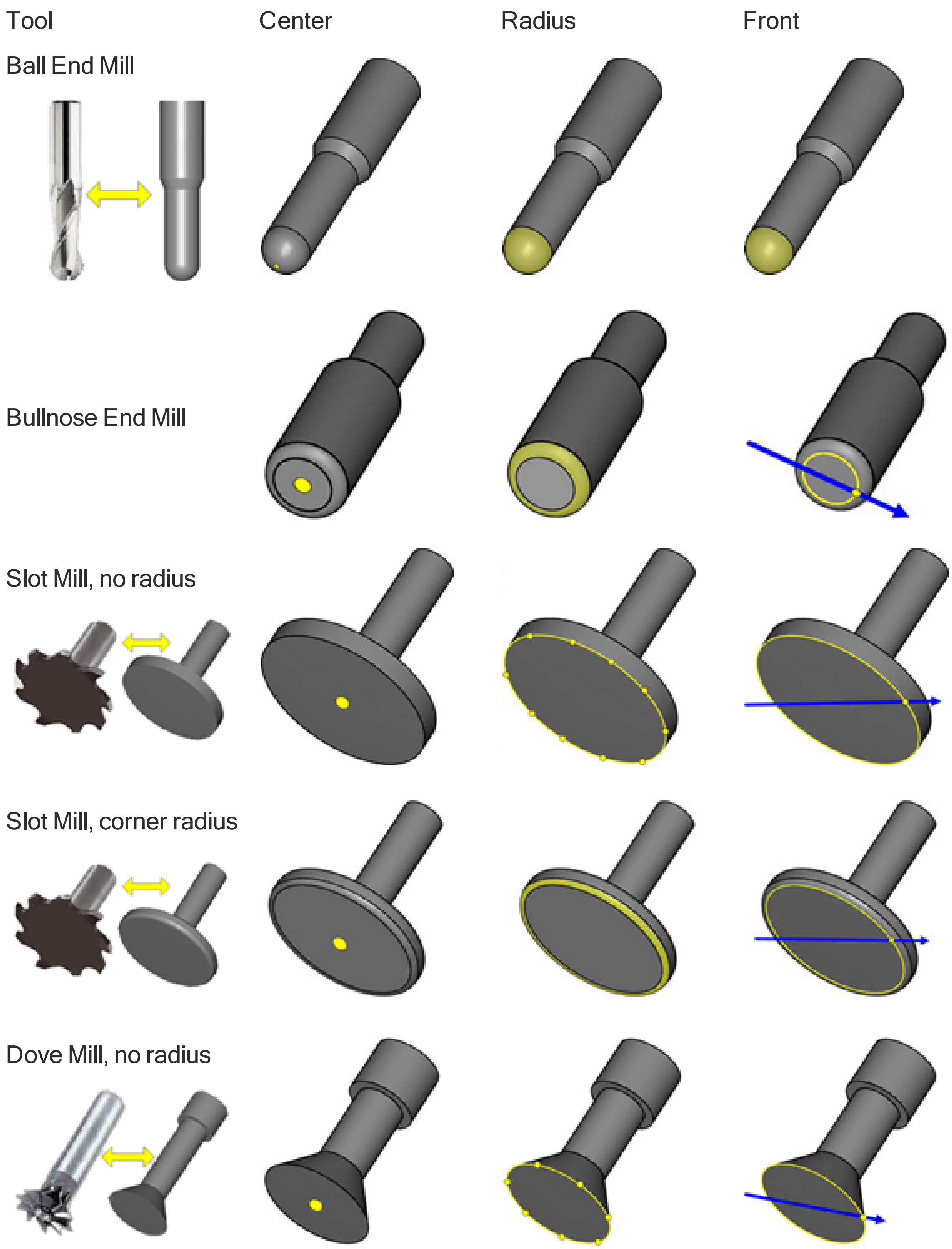
Tool Area Definitions

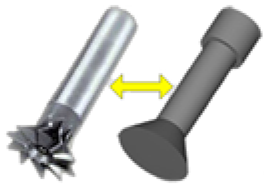
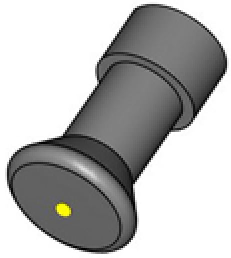
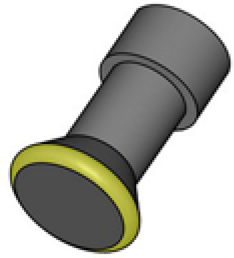
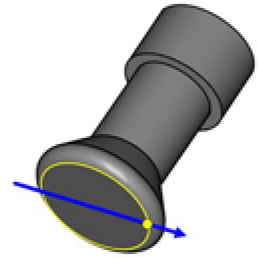
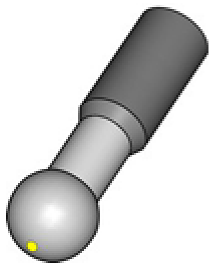
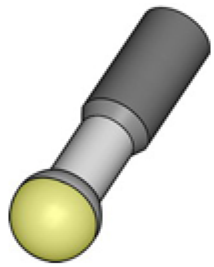
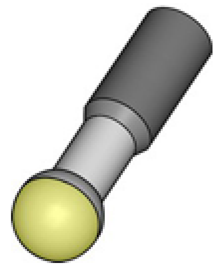
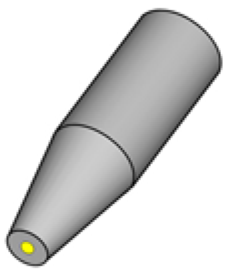
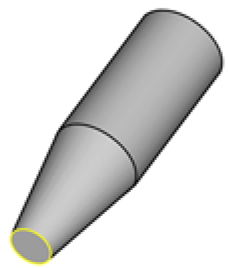
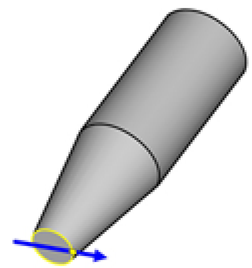
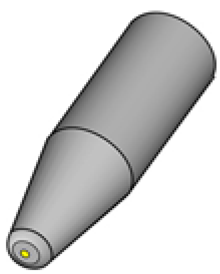
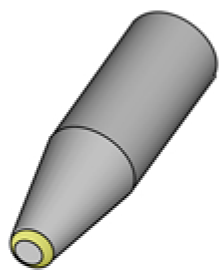
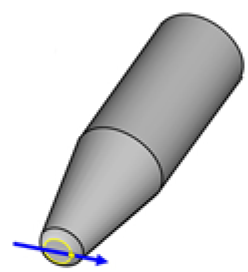
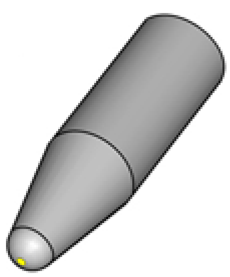
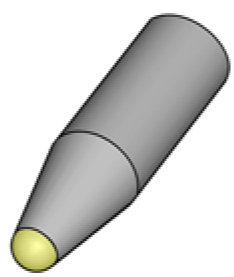
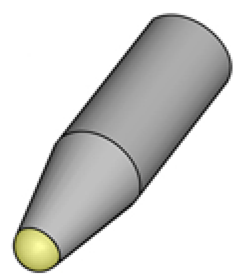
For each supported tool type, the run tool option “auto, center, front and radius” affect different points and areas on the tool. An explanation of how each of the run tool options affects the location of the contact point is shown below.

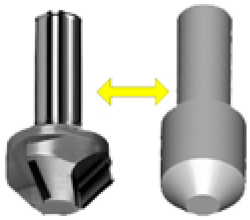
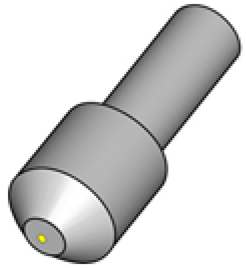
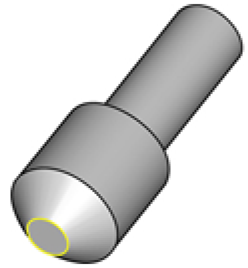
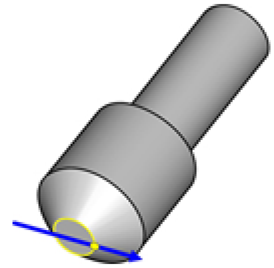
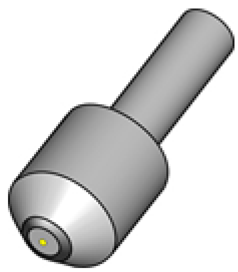
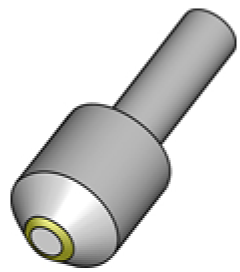
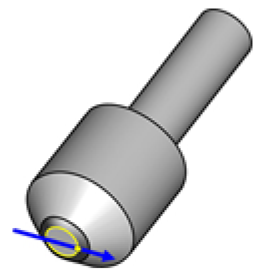
> If you use a key cutter tool in 5-Axis, it must have top radius equal to bottom radius. Key cutters are treated as bullnose mills if top and bottom radii are unequal.



Tool	Center	Radius	Front
Flat End Mill			


Tool
Center
Radius
Front
Ball End Mill
Bullnose End Mill
Slot Mill, no radius
Slot Mill, corner radius
Dove Mill, no radius

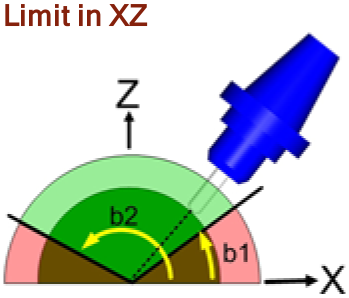
Tool	Center	Radius	Front
Dove Mill, corner radius 			
Lollipop Mill			
Taper Mill, no radius			
Taper Mill, corner radius			
Taper Mill, full radius			

Tool	Center	Radius	Front
Chamfer Mill, no radius 			
Chamfer Mill, corner			

Limits

Activating this option allows you to set direction limits on the tool axis. With these limit parameters you can control the tool axis orientation along the toolpath. The available limit types are XZ, YZ, XY, and Conical.

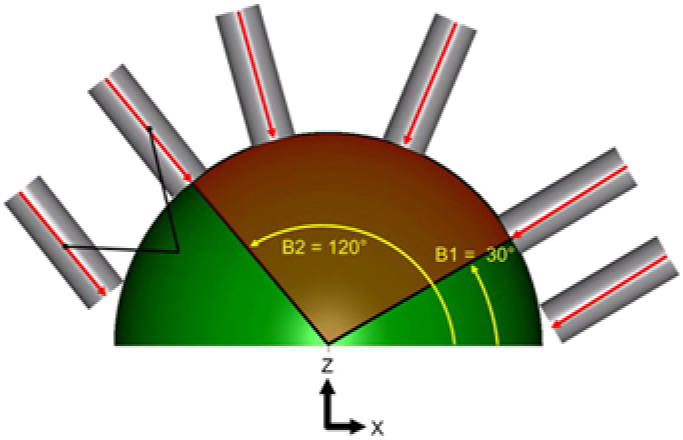
Limit in XZ



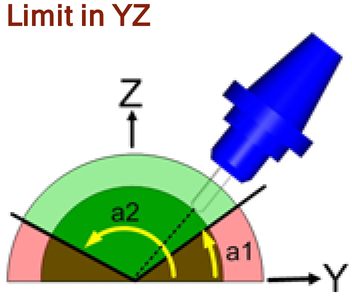
With this option you can limit the tool on the XZ plane between two angles. Angle `b1` sets the start limit parameter, and angle `b2` sets the end limit parameter. These angle values can range from `0°` through `360°` and are absolute values.

Example

In this example you can see that the minimum tool limit angle `b1` = `30` degrees and the maximum angle `b2` = `120` degrees.



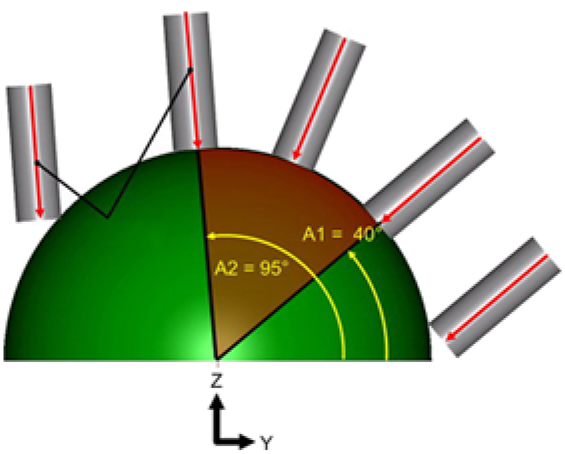
Limit in YZ



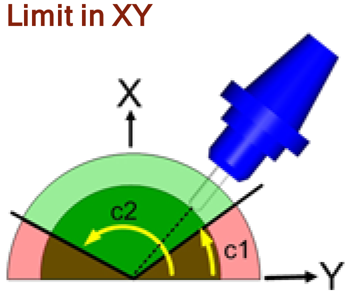
With this option you can limit the tool on the YZ plane between two angles. Angle a1 sets the start limit parameter, and angle a2 sets the end limit parameter. These angle values can range from 0° through 360° and are absolute values.

Example

In this example you can see that the minimum tool limit angle a1 is 40 degrees and the maximum angle a2 is 95 degrees. You can use any angle from 0 through 360 degrees.



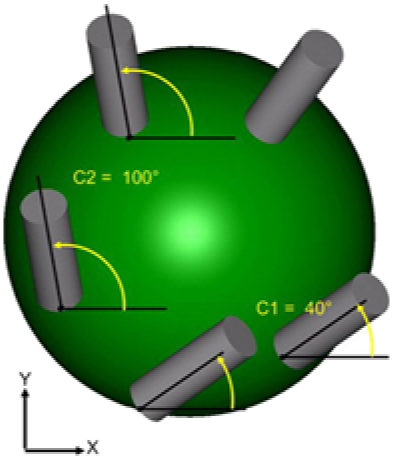
Limit in XY



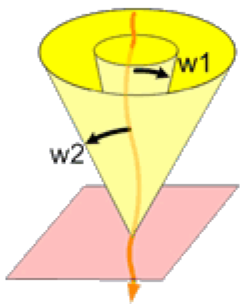
With this option you can limit the tool on the XY plane between two angles. Angle c1 sets the start limit parameter, and angle c2 sets the end limit parameter. These angle values can range from 0° through 360° and are absolute values.

Example

In this example you can see that the minimum tool limit angle c1 is 40 degrees and the maximum angle c2 is 95 degrees. You can use any angle from 0 through 360 degrees.



Conical limit

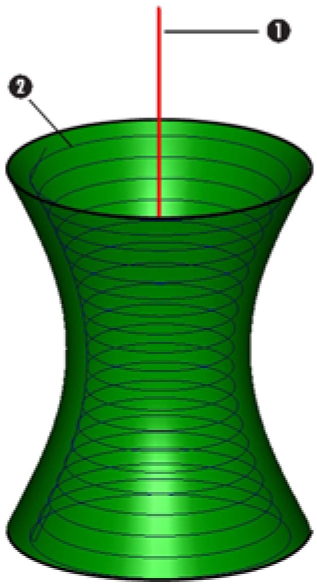


Use this option to limit the tool between two angles starting from the toolpath slice normal vector. In other words, imagine two cones with different opening angles, w1 and w2. The tool axis direction is enforced to be between these two cones. The orientation of the cones depends on the cone axis settings. You can set the orientation to X, Y, or Z directions, or to a user-defined direction. If your cone axis is a line, then you can use the X axis for lines parallel to X, or Y axis for lines parallel to Y, or Z axis for lines parallel to Z, or User-defined direction for lines parallel to an element that you select from the workspace using the selector ([...]) button. These angle values can range from 0° through 360° and are relative values.

If your toolpath is related to a leading curve, such as when you use the strategy Perpendicular to curve, you can set the cone axis to Dynamically using leading curve. Then you can limit the tool axis along this curve and its toolpath.

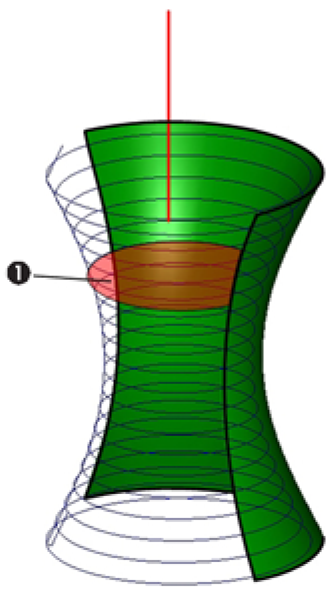
How it works explained with an example

Below is a curved cylinder using the toolpath pattern strategy Parallel cuts. The generated toolpath moves are constant parallel cut slices. The limit setting is Dynamically using leading curve.



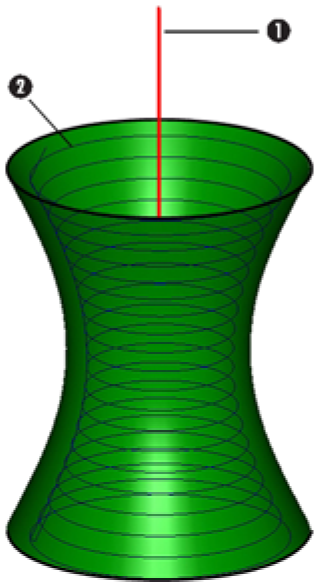
1. Leading Curve
2. Parallel Cuts

To get a better view, one half will be cut out and we will take just one random slice. Imagine the slice as a plane.



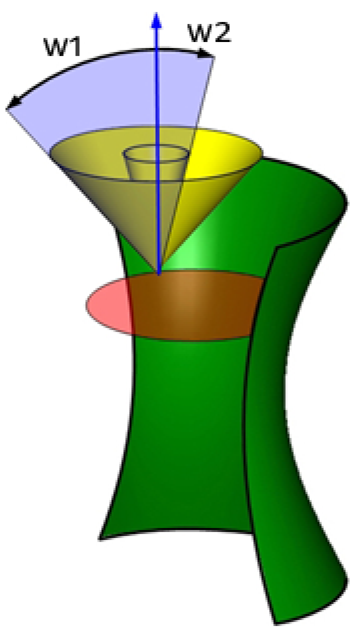
1. Cut Plane

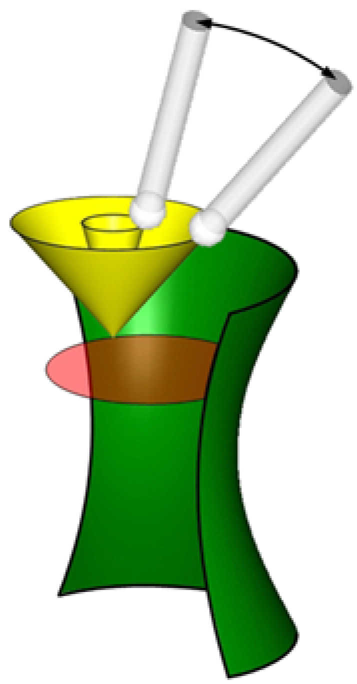
Now we will pick a random toolpath point on the surface and plane. From this point create a line normal to the cut plane.



1. Line Normal to the Cut Plane
2. Toolpath Point

From this normal the limit angles will be spanned. w1 is the inner limit angle and angle is the outer limit angle. To which direction it tilts depends on the tool axis strategy. Finally, from this example you can see the range in which the tool can tilt.



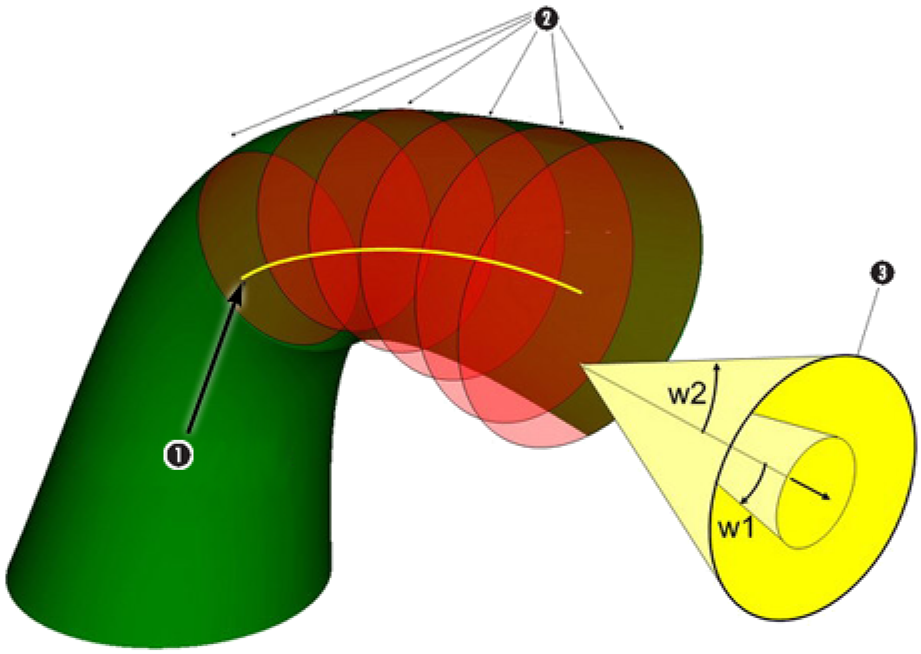


Span of Limit Angle	Limit Range

Example

This is a typical example for tube milling. You see the leading curve running through the tube. The toolpath pattern is set to “perpendicular to curve” which produces parallel cuts through the curve. With the dynamic limits setting you can now force the tool between the angles and avoid collisions, for example, without having gouge check activated. Open the file `Tool Axis`

`Direction Limit Parameters - Contain Tool Within Conical Limits.vnc` for a working example.



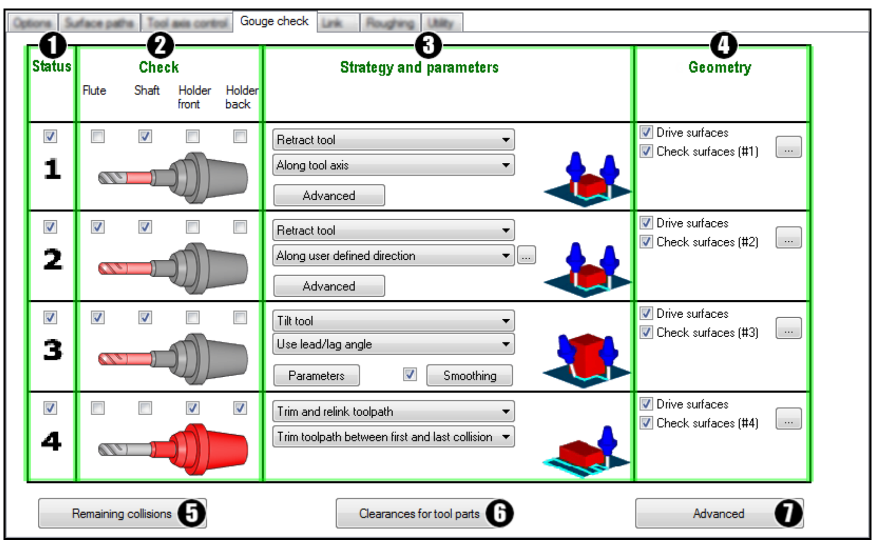
1. Leading Curve
2. Slices
3. Surface Normal

Gouge Check tab

The 5-Axis module offers powerful gouge-checking capabilities, giving you significant control over what is checked and how. For most strategies (exceptions are noted below*), you can set up to four completely different gouge checking methods (Status column) with different parameters, including which parts of the tool and holder are checked (Check column), how the tool should move to avoid gouges (Strategy and parameters column), and which faces to use for collision control (Geometry column).

* - Some special-purpose calculation strategies offer a simplified interface with fewer controls: See "Gouge Check tab for Swarf Machining" on page 98, or the guides for *5-Axis MultiBlade* or *5-Axis Porting*.

The buttons in the lower portion of the **Gouge Check** page — Remaining Collisions, Clearances for tool parts, and Advanced — open dialogs for specifying collision-handling actions, for setting tool and holder clearances, and for setting special parameters.



1. "Status" on page 215
2. "Check" on page 215
3. " Strategies and Parameters " on page 215
4. "Geometry" on page 232
5. "Remaining Collisions" on page 232
6. "Clearances for Tool Parts" on page 234
7. "Advanced Button for Gouge Check" on page 235

Controls on the Gouge Check tab

Status

You can create up to four different combinations of settings, where each combination consists of settings for tool components (under Check), gouge avoidance strategies (under Strategy and parameters), and surfaces (for Geometry). In the Status column, a checkbox indicates whether the corresponding gouge-checking combination is activated or deactivated.

The numerals are simpy labels, without any no numerical or sequential hierarchy. For example, you could deactivate combinations 2 and 4 and leave 1 and 3 activated, or you could leave all combinations deactivated except 3.

Check

This option defines which parts of the tool and holder will be used for the gouge check calculation. You can choose one to all four of the following, in any combination:

- Flute: the tool length down to the tool tip. For tools used in 5-Axis, flute length must be less than or equal to the value for Length out of Holder (also called the "stick-out length") – if flute length is larger than stick-out length, gouge checking will not detect tool holder collisions properly.
- Shaft, also known as the *shank*, or non-cutting parts of the tool
- Holder Front, sometimes called the *arbor*
- Holder Back (sometimes called simply *tool holder*).

In addition, you can add clearance distances to the basic dimensions for shaft, holder front, and holder back. For more information, see "Clearances for Tool Parts" on page 234.

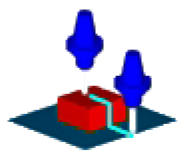
Strategies and Parameters

Gouge check strategies control the retracting process and describe the movement and tilting of the tool to avoid gouging. Five main strategies are offered, and the first three of these five are modified by sub-strategies:

- **Main strategy 1:** Retract Tool
  See Retract Tool, next.
- **Main Strategy 2:** Tilt tool
  Sub-strategies: Use lead/lag angle, Use side tile angle, Automatic
  See "Tilt Tool" on page 222.
- **Main Strategy 3:** Trim and relink toolpath
  Sub-strategies: Trim collision only, Trim toolpath after first collision, ..., Trim toolpath after last collision
  See "Trim and Relink Toolpath" on page 230.
- **Main Strategy 4:** Stop toolpath calculation
  See "Stop Toolpath Calculation" on page 231.

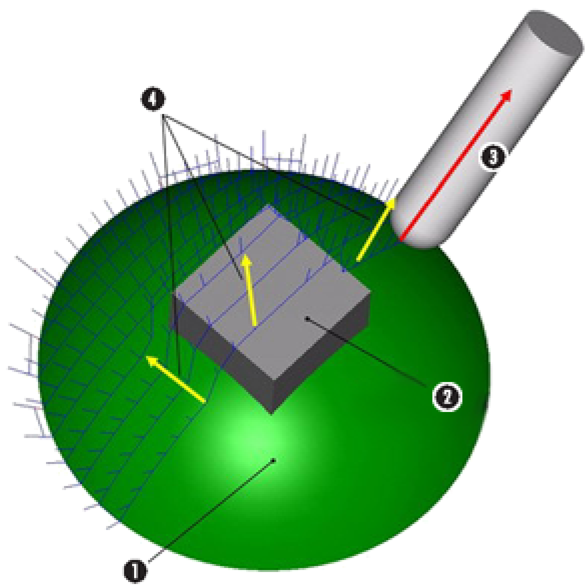
- **Main Strategy 5:** Report collisions
  See "Report Collisions" on page 232.

Retract Tool

	If the gouge check strategy Retract tool is selected, then the system will avoid gouges by retracting the tool. The resulting toolpath will be free of gouges. When the toolpath encounters a gouging point, it will retract the tool along the tool axis and clear the gouge point by the user specified distance, then make a connection move (line) to where the toolpath can resume after avoiding the gouging point.

Example

Here you can see the tool retracting along the tool axis. Open the file `Gouge Check Strategy-Retract Along Tool Axis.vnc` for a working example.



1. Drive Surface
2. Check Surface
3. Tool Axis
4. Retracting Direction

Advanced Options for Retract Tool + Along Tool Axis

To access the following options, set the sub-strategy to Along tool axisand click the Advanced button located immediately below the sub-strategy pull-down menu.

Drop tool down wherever needed

This option can be used to project the toolpath from a drive surface onto a check surface.

Remove areas where tool drop fails

This option will remove areas from the toolpath where the tool will not contact the check surface or where there will be a collision.

Smooth Retracts / Smooth distance

This option allows you to specify a value for Smooth distance. Abrupt tool retracts will be smoothed by this amount.

Advanced Options for Retract Tool + Other Sub-Strategies

To access the following options, click the Advanced button located immediately below the sub-strategy pull-down menu.

Project tool on direction wherever needed

This option can be used to project the toolpath along the direction selected as the sub-strategy. Selecting this checkbox makes the following checkboxes available.

Move tool outwards wherever needed

This option allows you to specify the maximum outward distance that the tool can be nudged when needed.

Project tool inwards wherever needed

This option allows you to specify the maximum inward distance that the tool can be nudged when needed.

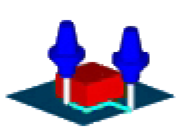
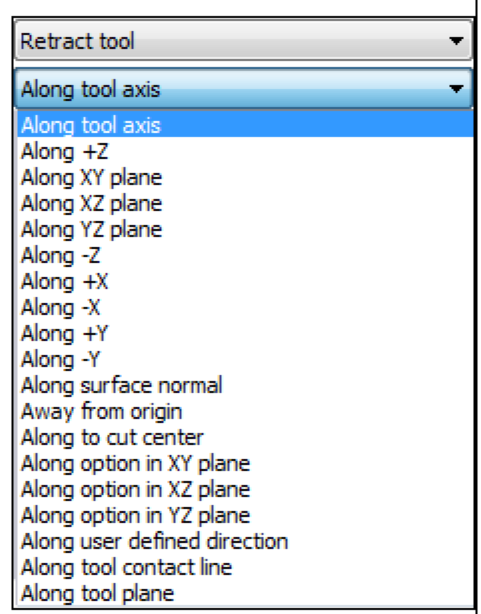
Remove areas where project tool fails

This option allows you to specify that, for areas where the tool cannot be projected as directed, the areas should be removed.

Reverse

This option reverses the direction of projection.

Retract Tool Sub-Strategies

	The Retract Tool strategy has numerous sub-strategies specifying how the tool should retract. These options can be grouped into several categories; see Retract Tool Along X, Y, or Z below, “Retract Tool Along Surface Normal” on page 218, “Retract Tool Away From Origin” on page 219, “Retract Tool to Cut Center” on page 220, “Retract Tool in User-Defined Direction” on page 221, “Retract Along Tool Contact Line” on page 221, and “Retract Along Tool Plane” on page 221. For each option, the Advanced button provides additional settings; see “Advanced Options for Retract Tool” on page 221.	

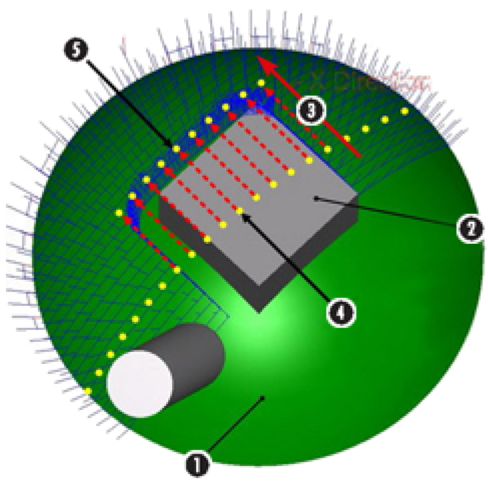
Retract Tool Along X, Y, or Z

These sub-strategies control the direction in which the tool will move away from the check and drive surface. While retracting, the tool always uses the shortest distance to go around the check surface. With this option the tool moves away from a detected gouge point in only the selected retracting direction.

Available retracting directions include in the +X, -X, +Y, -Y, +Z, and -Z axis direction, in the XY, XZ, and YZ plane, and also optimized in XY, XZ, and YZ. For the optimized options instead of using the surface normals for each point, the complete contour is looked at and an offset is generated based on the tool's radius. Then a move is made in the closest direction to the offset from the original contour. The idea here is to eliminate the effect of inner corners on the surface normal when using retract in XY plane.

Example

Here a gouge is detected. If you select Move tool in –X, the affected toolpath points will be moved away only in -X direction until the check surface ends and the tool is able to pass.



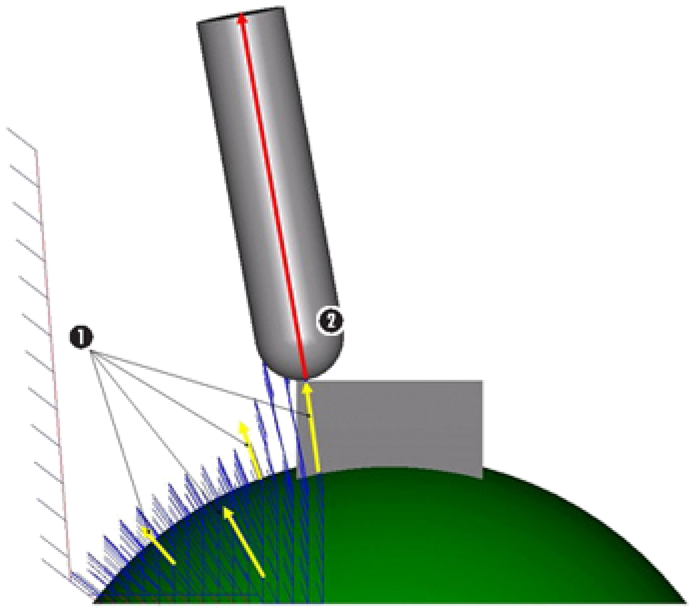
1. Drive Surface
2. Check Surface
3. X Direction
4. Old toolpath points
5. New toolpath points

Retract Tool Along Surface Normal

If you choose this option, the tool always retracts along the drive surface normal.

Example

As you can see in this example, every toolpath vector has the same direction as the surface normal. Open the file `Gouge Check Strategy-Retract Along Tool Axis.vnc` for a working example.



1. Surface Normal
2. Tool Axis

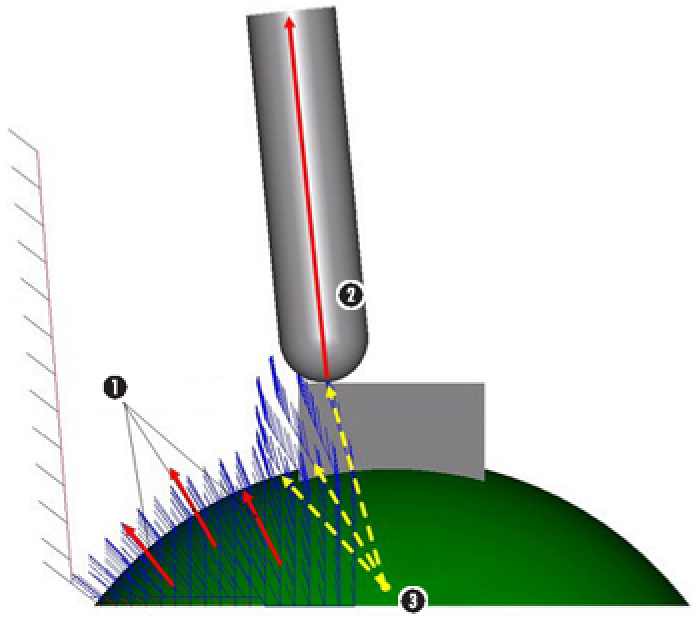
Retract Tool Away From Origin

With this option the tool always retracts away from the origin. This means that a vector is created starting at the origin, and going through the tool position. Then the retract occurs along this vector.

> This option works well when the origin is concentric within a sphere. However, when the origin is at a point on a planar part and a check surface is not near the origin then the vector from the origin to the tool position can cause a gouge.

Example

Here you can see that the tool retracts not from the surface normals but from the origin. Open the file `Gouge Check Strategy-Retract Along Tool Axis.vnc` for a working example.

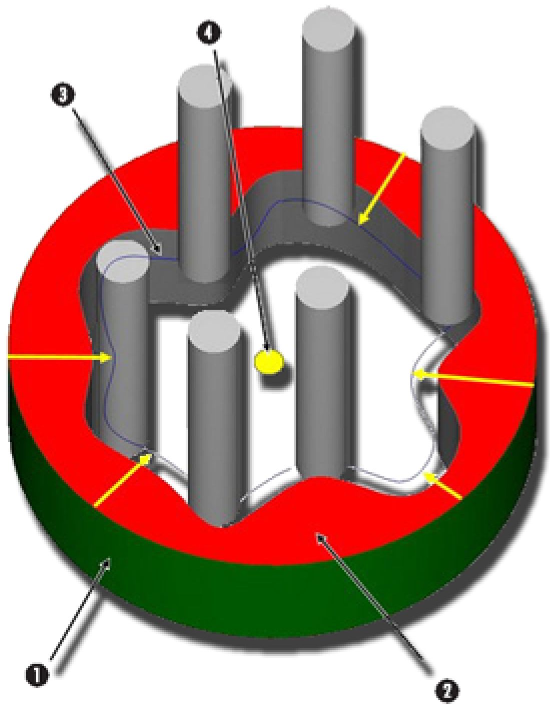


1. Surface Normal
2. Tool Axis
3. Origin

Retract Tool to Cut Center

This gouge check option is ideal for tube milling. To avoid a gouge the cutter will be retracted to the cut center. The cut center is the center point of your cut slice.

Example
In this example we see a cut-out through a tube. The green drive surface is machined in parallel z cuts. The red surface is the check surface. With the gouge check on the cutter retracts along the check surface to the cut center. Open the file `Gouge Check Strategy - Moving Tool Away - Retract Tool To Cut Center.vnc` for a working example.



1. Drive Surface
2. Check Surface
3. Toolpath
4. Cut Center

Retract Tool in User-Defined Direction

This will retract the tool in a direction you specify. You can specify a vector or select geometry that defines the vector.

Retract Along Tool Contact Line

This gouge check option is useful for tapered tools. It will retract the tool along its contact line with the drive faces. For a tapered tool, the retraction direction is along the tapered angle.

Retract Along Tool Plane

This gouge check option will avoid collisions while retaining tool orientation and height.

Advanced Options for Retract Tool

The Advanced button gives you access to additional options on the retract.

Project tool on direction wherever needed

Selecting this option will project the tool from its position in the direction selected in Retract Tool Sub-Strategies. When this checkbox is selected, other options are available.

Move tool outwards wherever needed / Maximum outward distance

The tool will be retracted up to this amount. If this option is not used, then the maximum distance is considered to be infinite.

Project tool inwards wherever needed / Maximum inward distance

The tool will be projected inward up to this amount.

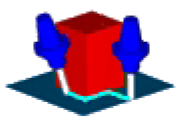
Remove areas where project tool fails

This option will remove areas where a tool projection is infinite or where projection fails.

Reverse

This option will reverse the direction of projection.

Tilt Tool

	If the gouge check strategy Tilt tool is selected, then the system will avoid gouges by tilting the tool away from the surface. Methods for tilting require you to specify an array of settings: See Use lead/lag angle below, Use side tilt angle”, or “Automatic” on page 224. All of the values are relative angles in degrees. The first two methods have a limited set of parameters. The third method has an extensive set of parameters; see “Automatic” on page 224. When the Smoothing checkbox is selected, additional settings are available; see “Side Tilt Angle Smoothing” on page 223. For the special case of the 5-Axis Toolpath Conversion operation modifier, a fourth method is available, with its own set of parameters; see “Advanced Options for 3-Axis to 5-Axis Conversion” on page 229.

- Gouge checking requires significant computing time. The best approach is to use limit angles and tilt angles to create gouge-free toolpath and then use one or more gouge checking strategies and the Report remaining collisions from all strategies option to prove that there are no gouges.

- It is still possible to get collisions with the tool even though gouge checking is activated for all of the tool. This can occur when there is a gouge that occurs between points on the toolpath, such as collisions that lie between the Maximum angle step. For example, suppose you have an operation where the tool tilts away with the maximum angle step set to 3°. If there is a collision within these 3° the gouge check may not recognize this. This is remedied by using a smaller value for the maximum angle step.

Use lead/lag angle

Using this option the tool tilts to the front or rear, relative to the cutting direction. Positive angles tilt the tool to the front, negative to the rear.

Use side tilt angle
Using this option the tool tilts to the sides, relative to the cutting direction. Positive angles tilt the tool to the left, negative to the right.

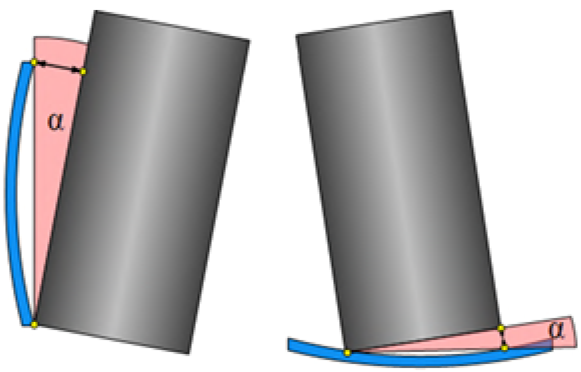
Parameters for Tilt Tool + Use Lead/Lag Angle or Use Side Tilt Angle

Maximum tilt angle
For Maximum tilt angle, you can specify angles in the range `0`° through `180`°.

Minimum tilt angle
For Minimum tilt angle, you can specify angles in the range `-180`° through `0`°.

Clearance Angle

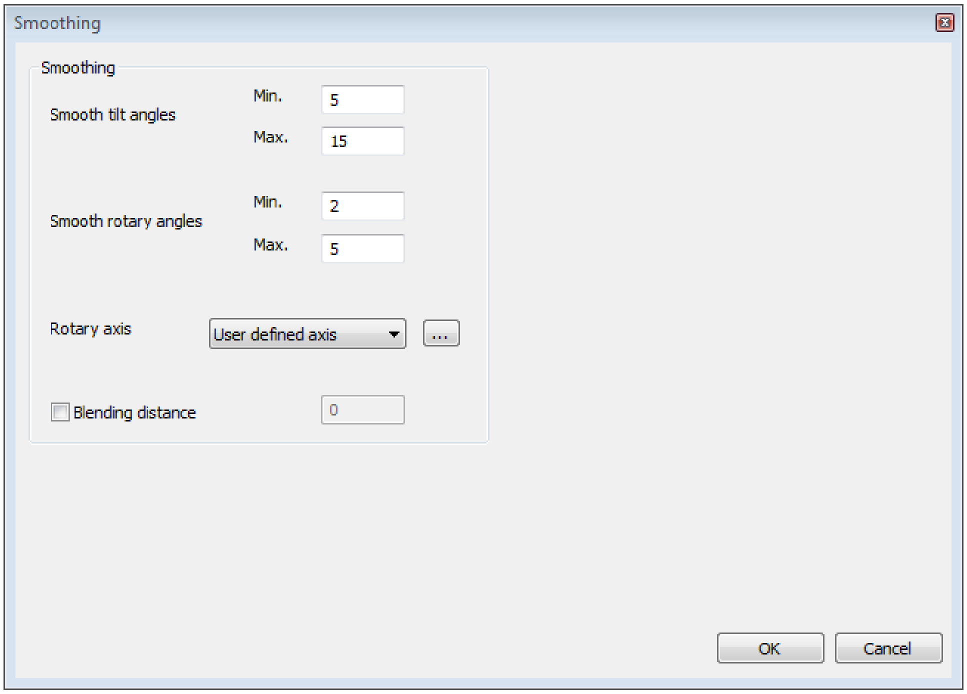
For Clearance angle, you can specify a minimum distance between the tool and the check surface. The distance is expressed by the opening angle between the surface contact point, the point on the tool that contacts the check face and the contact point on the check face that contacts the tool. The clearance angle can be applied to the side of the tool and the front face of the tool.



Side Tilt Angle Smoothing

Smoothing

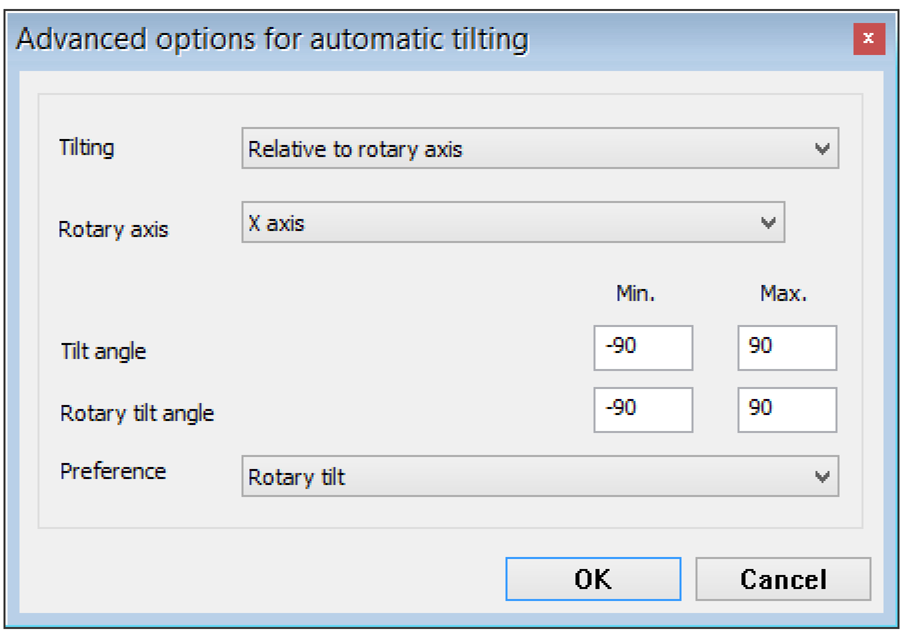
For sub-strategies other than Automatic, you can select the Smoothing checkbox and specify smoothing parameters.



If you do not use Blending distance, you can specify minimum and maximum values for tilt angles and rotary angles. If you select Blending distance, you can specify how far the blending will be performed. In either case, for Rotary axis, you can specify the X, Y, or Z axis, or an axis that you define.

Automatic

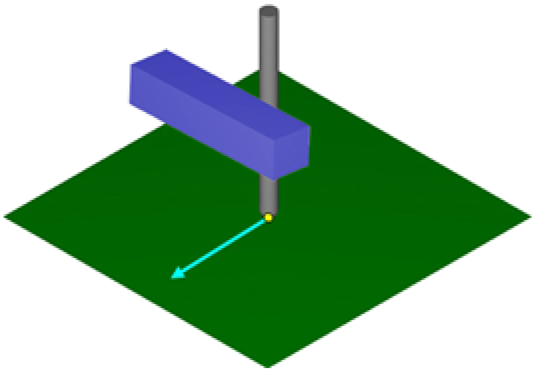
When you choose the Automatic sub-strategy, the system decides whether to avoid collisions by tilting from a rotary axis, by tilting around a rotary axis, by tilting the lead angle, by tilting the sideangle tilting, or using a combination. You specify whether the tilting, rotary, or neither should be preferred, and you specify the rotary axis.



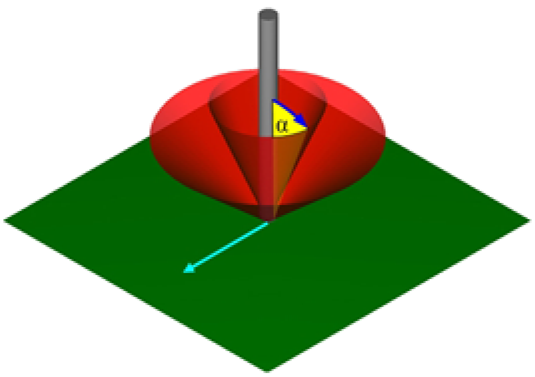
You can set an angle range in which the system can tilt the axes. The option accepts values for lead/lag from a minimum of -90° to a maximum of +90°. For side tilt, the minimum is 0° and maximum is 180°.

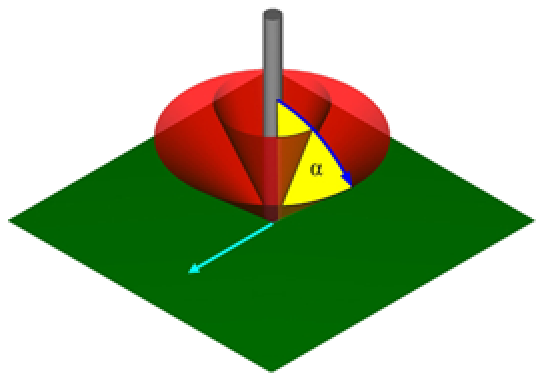
How it works:

A tool going along its toolpath is going to hit a check surface. The gouge checking system now starts calculating a solution.

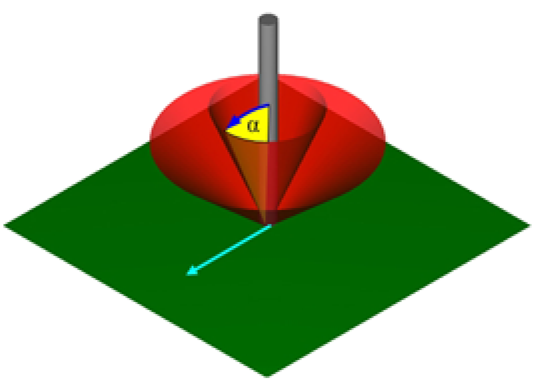


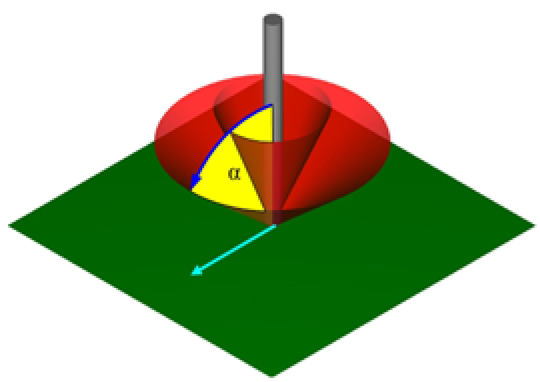
The system generates two cones starting at the toolpath point. The angles of these cones is defined by the angles you set. First the minimum angle for the tilting to the side of the cutting direction is calculated and then the maximum angle is calculated.





Then the minimum angle allowance in cutting direction and then the maximum allowance in cutting direction are calculated.





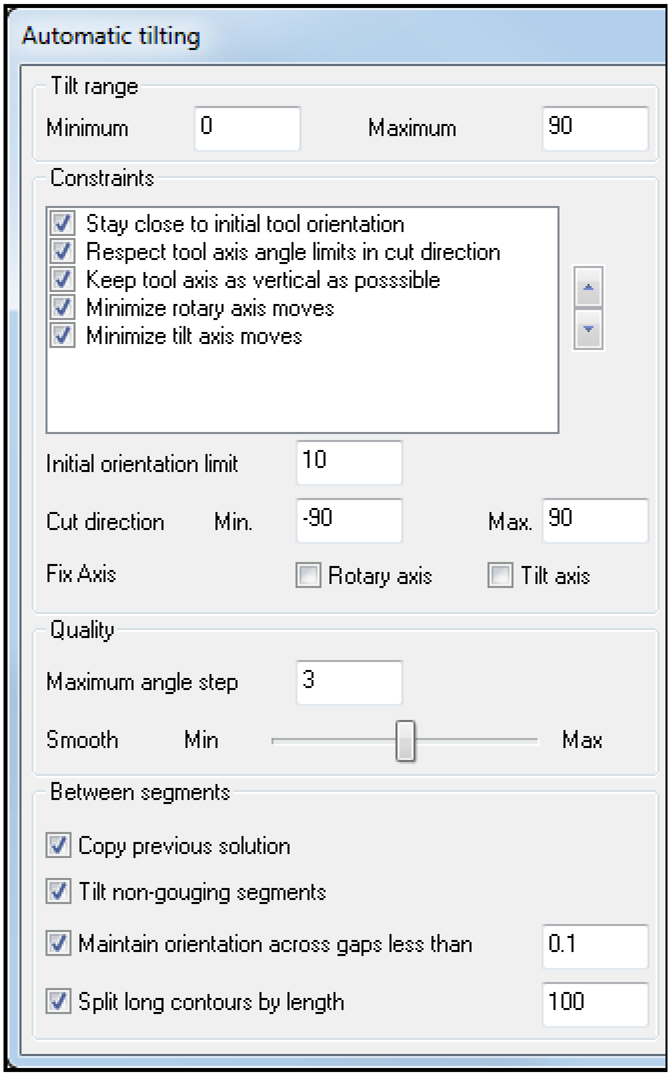
The result can be a tilting just in one direction or a combination of both tilting directions.

Automatic Tilting

Advanced parameters for tilting tool away

This enhancement gives the user far greater control over the algorithm that the 5-axis toolpath generation system uses to calculate an appropriate toolpath. In cases where gouging occurs, the tool can be tilted away from the gouging position.

Potential inputs can be grouped into five subcategories: See Tilt range below, “Constraints: Preferences” on page 227, “Constraints: Parameters” on page 227, “Quality” on page 228, and “Between segments” on page 228. Taken together, these subcategories provide a structured set of priorities for communicating your wishes to the algorithm.



1. Tilt range
2. Constraints: Preferences
3. Constraints: Parameters
4. Quality
5. Between segments

Tilt range

Enter the desired minimum and maximum tilt (side-to-side). This is the *domain* of the solution search: no solution checking will be performed outside of the specified domain. When you enter a value for Minimum or Maximum, the graphic to the right of the dialog indicates the domain as a yellow shaded area.

Constraints: Preferences

The top portion of the Constraints area offers an *ordered series* of preferences: you can select up to five checkboxes to activate the corresponding preference. Activating a preference asks the algorithm to use your input if possible, but in case of failure the selection is ignored and a solution provided.

You can customize the order in which preferences are applied. To move an item up or down in the list, click to highlight it and then click the up or down arrow to the right of the list.

The five constraint preferences are explained below in their default order.

Stay close to initial tool orientation

This preference will ask the algorithm to restrict the domain of search to the specified angle around the current tool orientation. By default, this is toggled on and placed at the top of the list.

Respect tool axis angle limits in cut direction

This preference will ask the algorithm to respect the angle limits specified in the cut direction. By default, this preference is toggled off.

Keep tool axis as vertical as possible

This preference ask the algorithm to keep the tool as close as possible to the Z axis. By default, this is toggled off.

Minimize rotary axis moves

This preference will ask the algorithm to minimize the rotary moves as much as possible. By default, this is toggled on.

Minimize tilt axis moves

This preference will ask the algorithm to minimize the tilt moves as much as possible. By default, this is toggled on and placed at the end of the list.

Constraints: Parameters

Initial orientation limit

The tilting angle solution is searched in a cone around the initial orientation without surpassing the tilting minimal and maximal values around Z axis. Note that the initial orientation limit becomes active when the preference ”Stay close to initial tool orientation” is checked.

If this option is turned on and an angle value is supplied, then the system will try to use the tool axis control defined in the Tool axis control tab. (An example might be Tilted through a point.) For example, it might be desirable to gouge-check the tool motion and allow the system to use the original tool axis orientation for each point (in this example, calculated through the point) and allow the deviation angle to resolve collisions. The algorithm will try to make the toolpath collision-free within the range of deviation angle that you supply here.

Cut Direction
Enter the desired minimum and maximum allowed angle (forward and back, or lead/lag.)

Fix Axis
You can opt to fix the specified axis (or axes) if a solution is available. You can choose to fix the Tilt axis, the Rotary axis, or both. Note that fixing any axis disables the ability to change the order of constraint preferences.

Quality

Maximum angle step
The maximum angle step represents the precision used to find a solution. Enter a value less than the minimum free gouge domain angle.

Smooth
This slider directs the algorithm to compensate towards minimizing corrections in the toolpath (Min) or towards making smoother toolpath (Max).

Between segments

Copy previous solution
For optimization purposes, the algorithm will try to copy the previous solution to the current solution. In case of failure the algorithm will try to find the proper solution and the previous solution will not be used. This parameter defines whether the last orientation from the previous contour should be used for the first orientation for the current contour. If this option is enabled the system will try to make the tool orientation continuous on linking the two contours. The orientation obtained from the previous contour will be applied to the current contour and only in the case of a collision will the previous orientation be ignored and a new one computed.

Tilt non-gouging segments
The algorithm will be applied to contours that are not gouging at all.

Maintain orientation across gaps less than
The algorithm will try to make the tool axis orientation continuously between two consecutive contours. This parameter defines the minimum distance allowed from the last point of the previous contour to the first point of the current contour to keep the tool orientation continuous. That is to say if the distance to the next cut is smaller than the value the tool will not change its orientation. If the distance is bigger the tool orientation will be reevaluated.

Split long contours by length
The contour will be divided accordingly and the algorithm will be applied to all sub-contours. This option becomes available only when the Maintain orientation across gaps less than option is selected. The system may have difficulty finding a single solution for tilting the tool when the there is a long, single shape or contour that defines the toolpath. This option will break the shape into sections or sub-contours, each of which can have its own tilt angle.

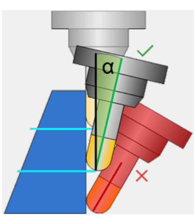
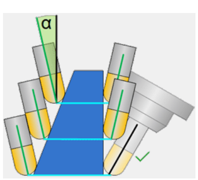
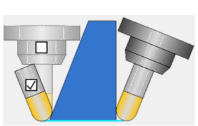
Because each shape is different, it is difficult to recommend a specific value. You may want to try different values for a result that works.

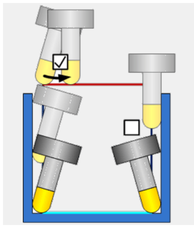
Advanced Options for 3-Axis to 5-Axis Conversion

When the 5-Axis Toolpath Conversion operation modifier is used, option 3 axis to 5 axis conversion is also available in some cases. This lets you convert a 3-axis input toolpath to a fully collision-checked 5-axis toolpath. Because the automatic tilting compensates for the tool holder, this option allows you to use a 3-axis toolpath with a much shorter tool.
Parts from v10.1 and v10.3 can contain operation modifiers that used an incompatible combination of choices for “Tool axis will...” in the **Tool axis control** tab. If such a part is opened in the current release, an error message will be displayed, and the settings will be made compatible.

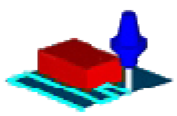
> 5-Axis Toolpath Conversion behaves as if the tool is a ball mill, regardless of the actual tool used to generate the input toolpath. This is because the toolpath conversion can only model the cutter location by using the input toolpath and the tool radius. Therefore, the contact point of the tooltip is used to calculate a virtual tool centerpoint, and it is this calculated centerpoint that the tool tilts around.
>
> If the actual tool has a larger diameter than the tool that was used for creating the input toolpath, there will be collisions with the target surface. If the actual tool is smaller than the input toolpath’s tool, it will not reach the target surface and there will be no contact point.

The Parameters button opens a dialog that offers the following.

	Maximum tilt angle The maximum deviation allowed to find a solution. The tool tilt angle will never exceed the value set here. If the only way to avoid a collision is by exceeding this angle, then the toolpath will be trimmed. In the picture, the green angle indicates a permitted tilt angle; red indicates a tool that exceeds this angle.
	Desired tilt angle Allows you set a preferred tilt angle for avoiding collisions. This angle will be applied even if when a smaller deviation would suffice, but it is not a maximum allowance. If the angle set here is not enough to avoid the collision, then the tool is permitted to tilt beyond this angle. In the picture, the green orientation lines show the desired tilt angle applied even when a smaller tilt angle would avoid a collision. The black line shows a tool being permitted to exceed the desired tilt angle.
	Tilt for collision-free zones If this checkbox is selected, the system will not try to keep to a `0` tilt angle. Instead, it will tilt the tool smoothly almost all the time – from toolpath start to end, and from one collision area to another.

	Gradual tilting only on connections If this checkbox is selected, then the tool axis orientation will remain unchanged during approach and retract moves. Reorientation will occur only during connection moves, as shown in the left portion of the picture. If this checkbox is unselected, then tool axis reorientation can occur during approach moves, retract moves, or connection moves.

Trim and Relink Toolpath

	If the gouge check strategy Trim and Relink Toolpath is selected, the system will trim the toolpath when a collision is detected. Instead of moving or reorienting the tool when a collision with a check surface is detected, the toolpath positions producing a gouge are removed (trimmed). The machining will be continued as much as possible. The Advanced Button for Gouge Check option should be used in conjunction with this option.

The Advanced button opens a dialog that offers the following.

Don’t trim toolpath
When using this option the toolpath will be created in its entirety.

Trim toolpath after first collision
This option allows the first collision to occur and then stop.

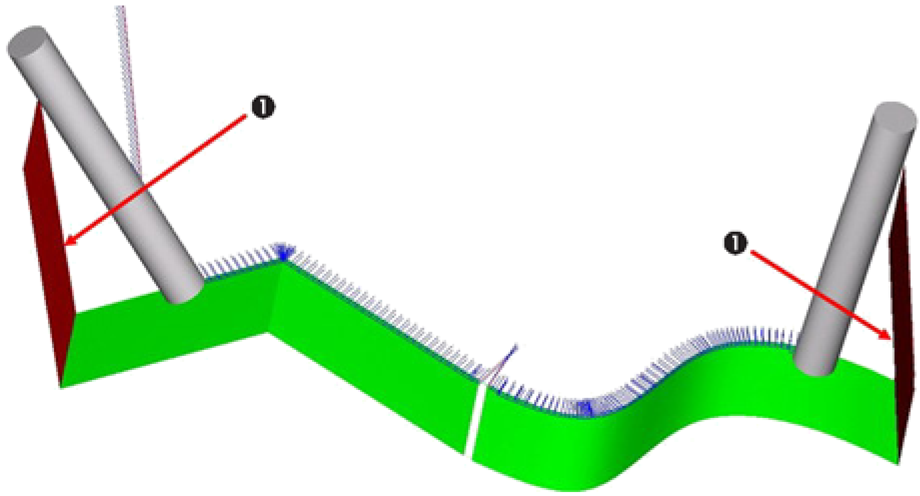
Trim toolpath before last collision
This option creates all of the toolpath up to just before the last collision.

Trim toolpath between first and last collision
This will stop the toolpath at some point in the middle.

Trim toolpath before first collision
This will create the toolpath up until the first collision.

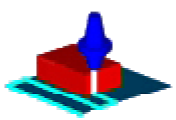
Trim toolpath after last collision
This will create the toolpath until just after the last collision.

Example
Here we see a tool following the toolpath. The surface normals show the tool orientation up to the gouge protection trimming point, where the toolpath is stopped and typically retracted to a safe clearance distance, then moves past the gouge point and resumes toolpath, if possible. Open the file `Gouge Check Strategy - Leaving Out Gouging Points.vnc` for a working example.



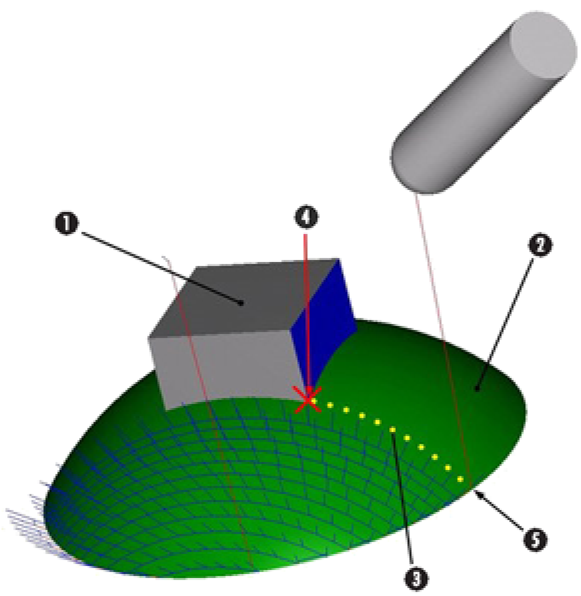
1. Check Surface

Stop Toolpath Calculation

	This gouge check strategy will create toolpath only up to the point that the first gouge is detected. The toolpath will stop at this point, ending the operation.

Example

Here you can see that the next cut would cause a contact with the check surface. The toolpath is created only up to the point that the first gouge is detected. Open the file `Gouge Check Strategy - Retract Along Tool Axis.vnc` for a working example.



1. Check Surface
2. Drive Surface
3. Toolpath
4. First Gouge
5. Stop Position

Report Collisions

This gouge check strategy simply checks for collision between the tool and the check faces without trying to avoid the collision. The result is a information message reporting between which tool part and surface a collision happened.

This option will report collisions found in the specified tolerance only. Therefore, it might more collisions (or fewer) than would actually occur in another strategy if different tolerances were set for the other strategy.

Note that this option will report a different set of collisions than the Report remaining collisions from all strategies option in the Remaining collisions dialog. For more information on this, see "Remaining Collisions" on page 232.

Geometry

In this section you define which type of faces will be used for collision control.

Drive Surfaces

When you activate this option, then all selected drive surfaces (see Drive Surfaces) will be checked for collisions. This ensures that your drive surface will not be damaged.

Check Surfaces

When you activate this option you can set additional surfaces to be checked for collisions. For example this selection might include other surfaces from your part which are not drive surfaces. When you only have the Check surfaces option activated you can set two parameters:

Stock to leave

This is an additional offset distance to your check surfaces. This is useful to create a "safety zone" around the check surfaces.

Tolerance

This is the variance allowed in checking the surface. Giving the system a larger tolerance to use for the check surfaces will speed up the calculation.

Remaining Collisions

When at least one gouge checking strategy is active, the Gouge check page offers a Remaining colliisions button. Clicking it opens the **Remaining collisions** dialog.

Remaining colliding contours

Usually, collisions are undesirable behavior. In some cases, however, collisions might remain in the toolpath. For example:

- If Check gouge between positions has been disabled (in **Gouge Check** > Advanced > Miscellaneous), then collisions might remain. Disabling this option speeds up the calculation but leaves a possibility of remaining collisions if sharp corners and complex geometries are present. In this case, it is helpful to use Remaining collisions.

- In case of engraving or trimming, the tip of the tool is inside the surfaces to be machined. Such a machining strategy will trigger Remaining collisions.
- User-entered values might be too small for retract distance or approach distance, or the rapid plane might be too low. In this case, Remaining collisions can provide details.
- The order in which gouge-check strategies are selected might result in a scenario where valid toolpath generated by the first gouge-check option conflicts with a second or subsequent gouge-check option. In this case, Remaining collisions can provide details.

The tolerance used for determining the collisions is the maximum tolerance of all collision strategies (*except:* the tolerance from fifth gouge check strategy, Report collisions, is ignored).

For reporting remaining collisions, you can choose from the following three option buttons:

Keep (collisions remain)
Keep the toolpath despite any collisions that might be found.

Trim colliding contours and relink
Trim back the toolpath in an effort to avoid collisions, and then re-run the linking steps.

Stop toolpath calculation
Halt the calculation and report an error.

Miscellaneous

Report remaining collisions from all strategies
This option is useful when using gouge checking strategies, particularly when Check Gouge Between Positions is disabled. Report remaining collisions is intended for use when Check Gouge Between Positions is disabled to increase system performance. This option runs much faster than checking between positions. Report remaining collisions will not perform collision avoidance for gouge checking but it will tell you if there is a problem so that you can review the process and fix the collision.

The calculated toolpath is applied with a tolerance that is double the user-supplied tolerance to detect collisions. Report remaining collisions only checks the *tool* component (tip, shank, etc.) and the drive/check surface combinations as specified for each of the active gouge-checking strategies.

There are several conditions in which the system is, technically speaking, generating collisions purposefully. The Report remaining collisions option helps you work with these situations. For example:

- When the collision control between positions is disabled. This will accelerate system performance, but leaves the possibility of remaining gouges if sharp corners and complex geometries are present. In this case, Report remaining collisions is helpful with identifying the collisions.
- When engraving or trimming, the tool tip is actually inside the surfaces to be machined, which is technically a gouge. This will result in remaining collisions being reported.

- You may have entered retract and approach distances that are too small or rapid planes that are too low. These cases will result in remaining collisions.

This gouge check option simply checks for collision between the tool and the check faces without trying to avoid the collision. The result is a information message reporting between which tool part and surface a collision happened.

This option will report collisions found in the specified tolerance only. Therefore, it might more collisions (or fewer) than would actually occur in another strategy if different tolerances were set for the other strategy.

Note that this option will report a different set of collisions than the Report collisions strategy on the main Gouge Check page. For more information on this, see “Report Collisions” on page 232.

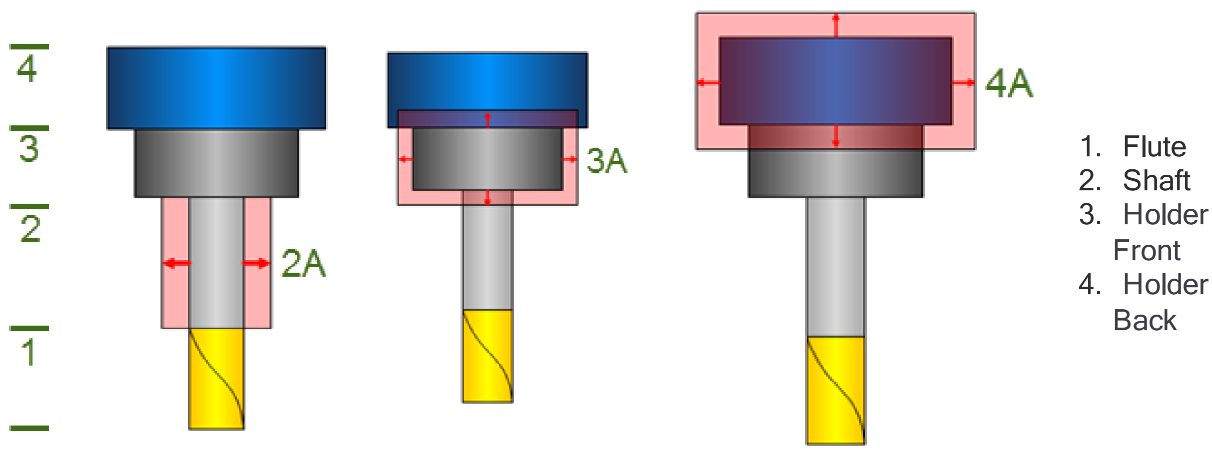
Clearances for Tool Parts

The system looks at the holder back, holder front, and shaft as simple cylinders or cones, no matter what the shape actually looks like. These clearances are a virtual stock added to the diameters of your holder back, holder front, and shaft.

If the surfaces to be considered have a value set for Stock to leave, then the clearance and Stock to leave values are added together to keep the holder front away from the part by that distance. For example, if the holder front clearance is 0.2 and you set Stock to leave to 0.5 on the surfaces, then the holder front is not allowed to come closer than 0.2 + 0.5 = 0.7 to the part.

Example: Cylindrical clearances.
For cylindrical clearances, you specify three linear values: One for the shaft diameter, one for the holder front diameter and length, and one for the holder back diameter and length.

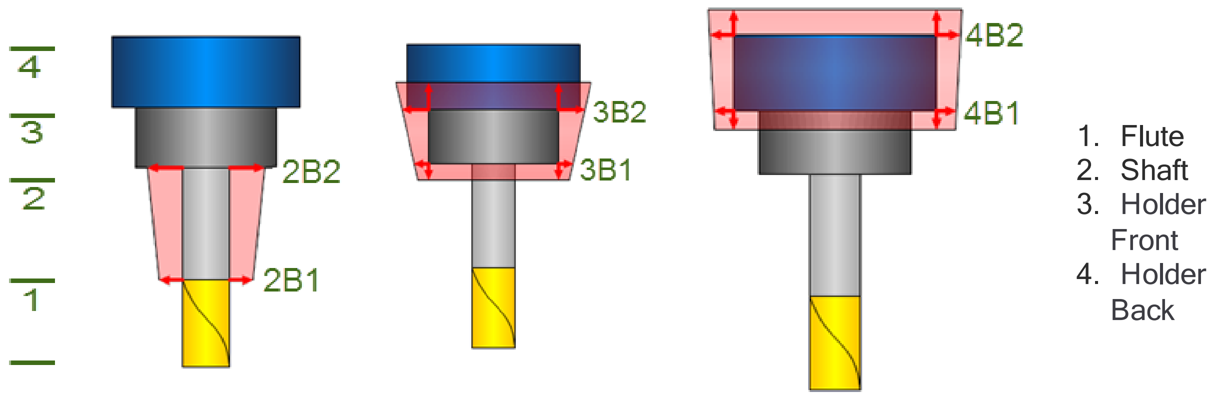
Clearances - Cylindrical



2A = Shaft diameter clearance; 3A = Holder Front clearance; 4A = Holder Back clearance

Example: Conical clearances.
For conical clearances, you specify six linear values: shaft diameters (upper and lower); holder front diameter and length (upper and lower), and holder backdiameter and length (upper and lower).

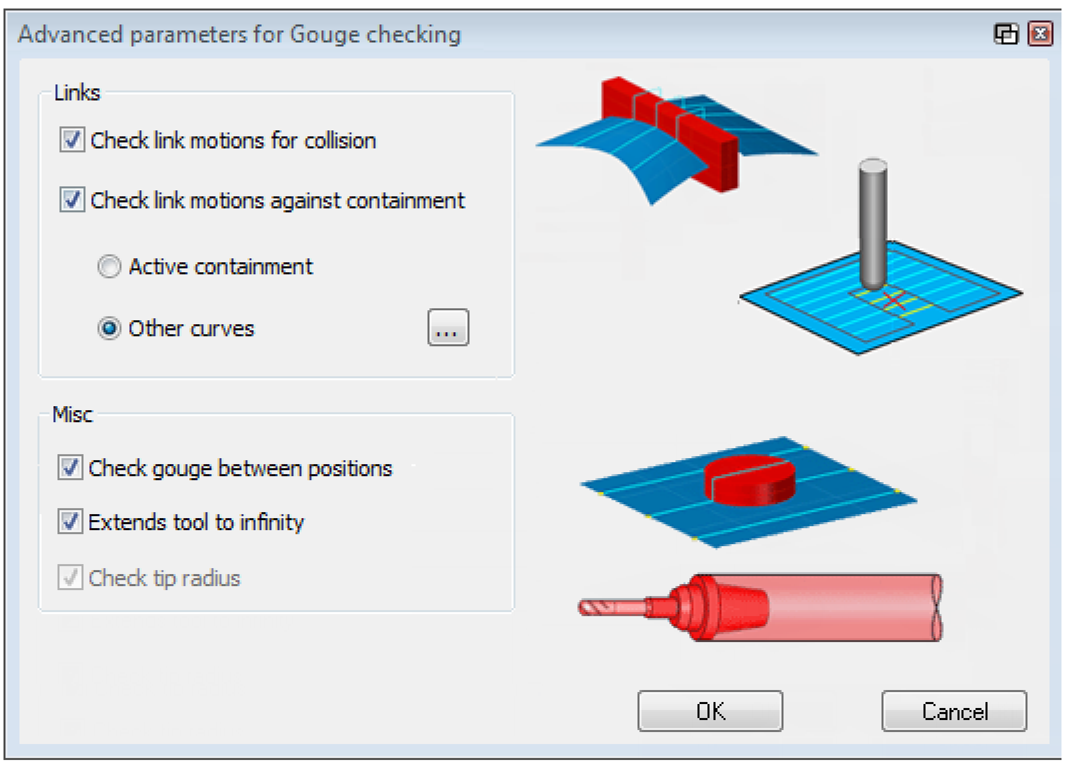
Clearances - Conical



2B1,2= Shaft diameter clearances; 3B1,2 = Holder Front clearances; 4B1,2= Holder Back clearances

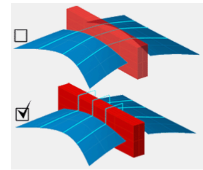
Advanced Button for Gouge Check

When at least one gouge checking strategy is active, the Gouge check page offers an Advanced button. Clicking it opens the Advanced Parameters for Gouge Checking dialog, which lets you activate one or more additional options. See Check link motions for collisions below, “Check gouge between positions” on page 236, “Extends tool to infinity” on page 237, and “Check tip radius” on page 237.



Links

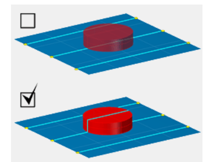
Check link motions for collisions



When this option is active, the system will check link moves for collisions. For more information on links, see “Link tab” on page 238.

Miscellaneous

Check gouge between positions



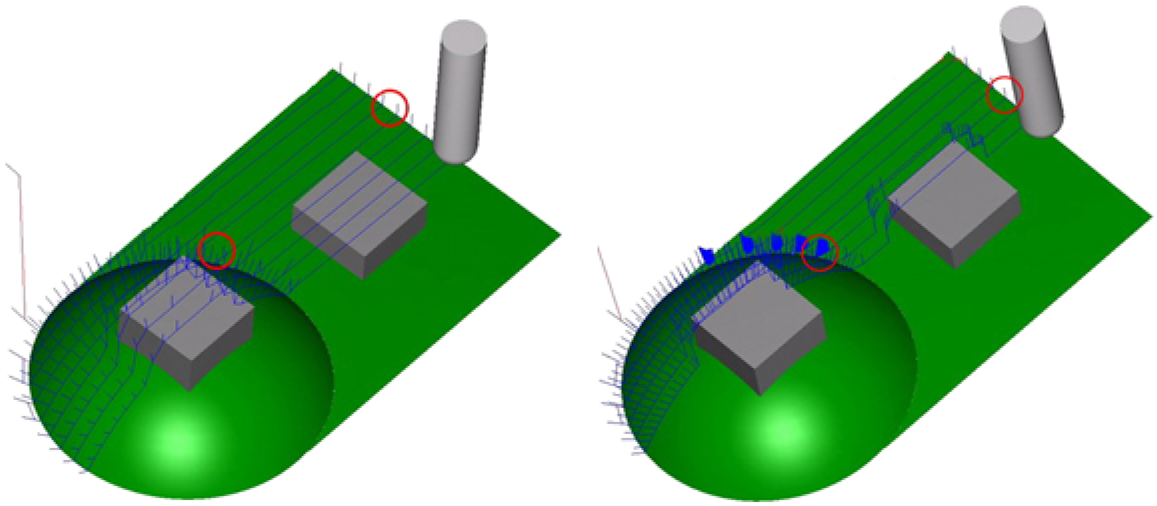
Select this option in order to activate the collision checking between toolpath positions. When there is a flat area, toolpath is typically generated at the edges of the flat; in other words, there are no points between the start and end of the flat. This may result in a gouge not being detected between the start and end points. When this option is active, the system will look at the tool’s movement from one position to the next and check for collisions with the drive and check surfaces. This option should always be used to get good toolpath that avoids gouges.

- This function may not be necessary if you have set a value for Maximum distance. See “Maximum distance” on page 148.
- Please be aware that this may slow down the calculation.

Example

The following images show a part with a rounded area and a flat. The two gray cubes in each image represent check surfaces.

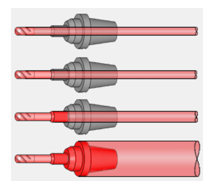
- In the illustration on the left, there are no toolpath points between the edges of the flat. The tool will gouge the check surface. On the round part there are many toolpath points. Gouge checking works here even with the function disabled.
- In the illustration on the right, there are also no toolpath points between the edges of the flat. With the option activated the system finds the check surface and avoids the gouge. Again, on the rounded surface there are many toolpath points, so the system does not necessarily need the function, but it still helps.



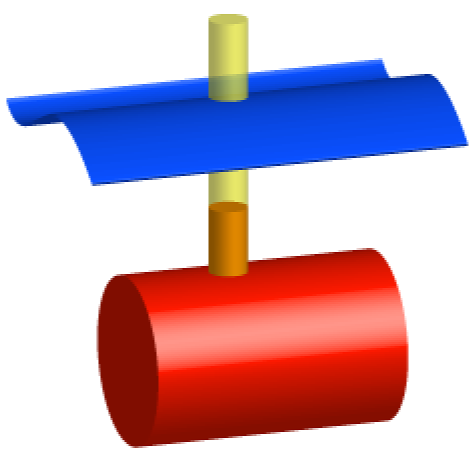
Gouge Check Between Positions is disabled.	Gouge Check Between Positions is activated.

Open the file `Gouge Check Strategy - Leaving Out Gouging Points.vnc` to see the actual sample operations.

Extends tool to infinity



When this option is active, the tool geometry in other words, the tool, the holder front, and the holder back – is stretched along its axis to infinity. This helps the collision-checking system to detect all possible collisions.



A good example for using this option is when you are using guide surfaces to control the tool's axis (for example a cylinder) and the tool is retracted to the actual part surface (which is defined as a check surface) to do the cutting. If the total tool length is not long enough – including both the holder back and the holder front– then the collision-checking system might find a collision-free location for the tool between the cylinder and the real part. This would not generate the desired toolpath. Although making the tool longer could resolve the problem and force the tool up to the check surface, there is no need for such lengthening when the option for Extends tool to infinity is active.

Check tip radius

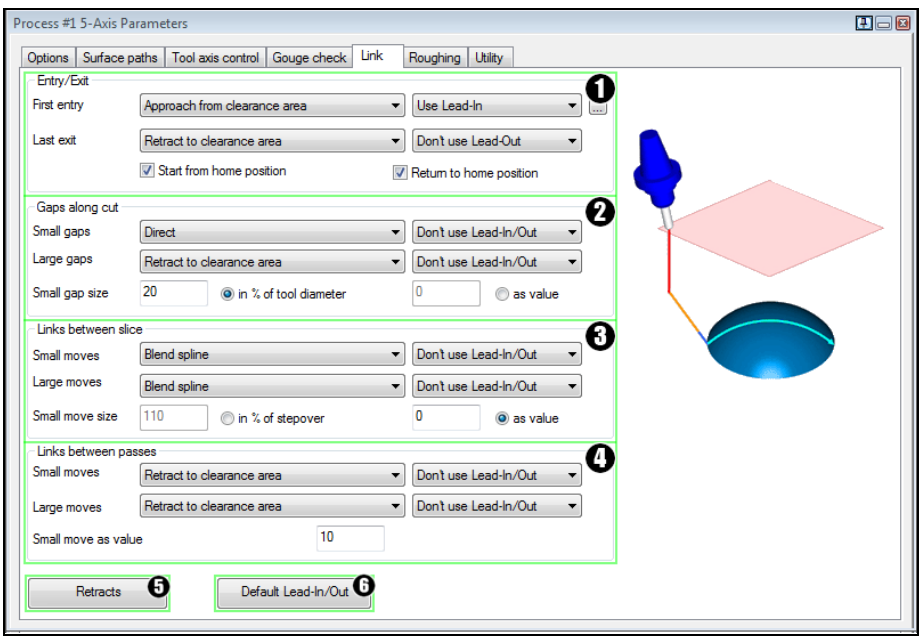
This is available when using the strategy Tilting tool away with max angle (see "Tilt Tool" on page 222) and a non-flat tool. When enabled, this option includes the radius of the tool tip in the collision checking. When the option is disabled, the radius of the tool tip is ignored.

Link tab

The **Link** page (with some exceptions, noted below*) offers controls for how the tool moves when it is not cutting, such as how the tool will move when entering or exiting the part (see "Entry/Exit" on page 238).

* - Some special-purpose calculation strategies offer a simplified interface with fewer controls: See the guides for *5-Axis MultiBlade* or *5-Axis Porting*.

Additionally, items on this page provide control over how the tool will move when it encounters air or check surfaces while cutting (see "Gaps Along Cut" on page 241), how the tool will move between stepovers (see "Links Between Slice" on page 245) and how the tool will move between passes (see "Links Between Passes" on page 250). The page also offers controls for defining clearance areas and clearances for feed and rapid moves (see "Retracts Dialog" on page 252), and powerful custom controls defining how the tool will move onto and off of the part (see "Default Lead-In/Out" on page 259).



1. "Entry/Exit" on page 238
2. "Gaps Along Cut" on page 241
3. "Links Between Slice" on page 245
4. "Links Between Passes" on page 250
5. "Retracts Dialog" on page 252
6. "Default Lead-In/Out" on page 259

Controls on the Link tab

Entry/Exit

In the Link tab, you can choose items under Entry/Exit to define link types and distances for tool approach (see First Entry below), for tool retraction (see "Last Exit " on page 239) and for home position start/return (see "Use Home Positions" on page 240).

First Entry

First entry is the first approach of the tool towards the part in a given operation. You can specify an entry clearance distance from where the tool will enter the part and whether there will be a macro move or not when entering the part.

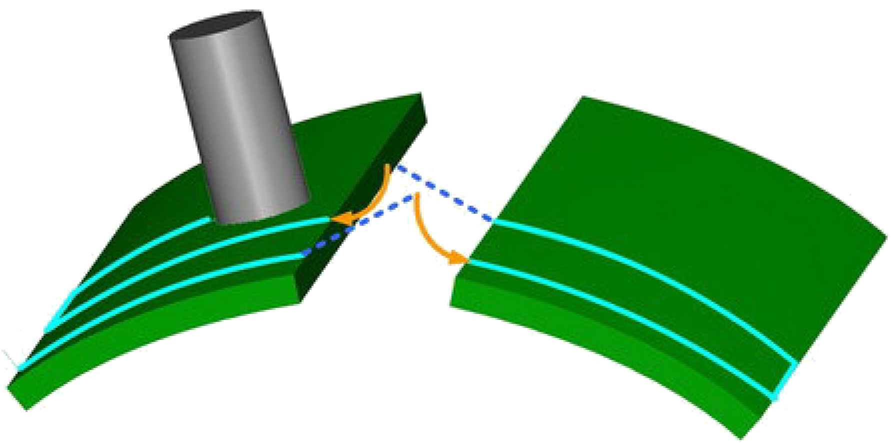
The entry options are: Approach from clearance area, Approach from rapid distance, Approach from feed distance and Direct.

In the default setting, Approach from clearance area, the tool starts at the Retracts Dialog, moves to the rapid distance, rapids to the feed distance, and then feeds to the surface. If you choose Approach from rapid distance or Approach from feed distance the first entry will be from a closer start point. If you choose Direct, no link is calculated and added to the toolpath.

After you have specified an entry clearance distance for the tool, you can define a custom entry type. The choices are Use Lead-In or Don't use Lead-In.

Use Lead-In

This option controls movement of the tool when moving onto the part. This includes gaps in the toolpath and is in addition to the gap options. The lead-in is defined in the Default Lead-In/Out button, or you can set a custom lead-in by clicking the ellipsis button.



Don't use Lead-In

If lead-in is not used, then the tool will move from the clearance position to the first point in the toolpath along the tool axis in a straight line motion.

Last Exit

Last exit defines how the tool moves when exiting the part in a given operation. You can specify an exit clearance distance to where the tool will go after finishing the toolpath and whether there will be a macro move or not when exiting the part.

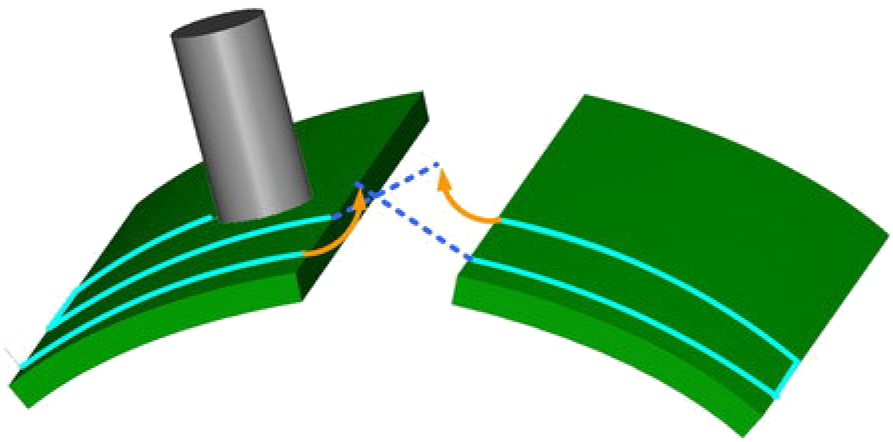
The exit options are: Retract to clearance area, Retract to rapid distance, Retract to feed distance, Retract to clearance area through tube center and Direct.

In the default setting, Retract to clearance area, the tool feeds from the drive surface to the feed distance, then rapids to the rapid distance, and then moves to the clearance area. If you choose Retract to rapid distance or Retract to feed distance, the machining ends closer to the part. If you choose Retract to clearance area through tube center, the tool retracts through the center of an enclosed contour, such as a tube or cylindrical geometry. If you choose Direct, no link is calculated and added to the toolpath.

After you have specified an exit clearance distance for the tool, you can define a custom exit type. The choices are Use Lead-Out or Don’t use Lead-Out.

Use Lead-Out

This controls movement of the tool when leaving the part. This includes gaps in the toolpath and is in addition to the gap options. The lead-out is defined in Default Lead-In/Out, or you can set a custom lead-out by clicking the ellipsis button.



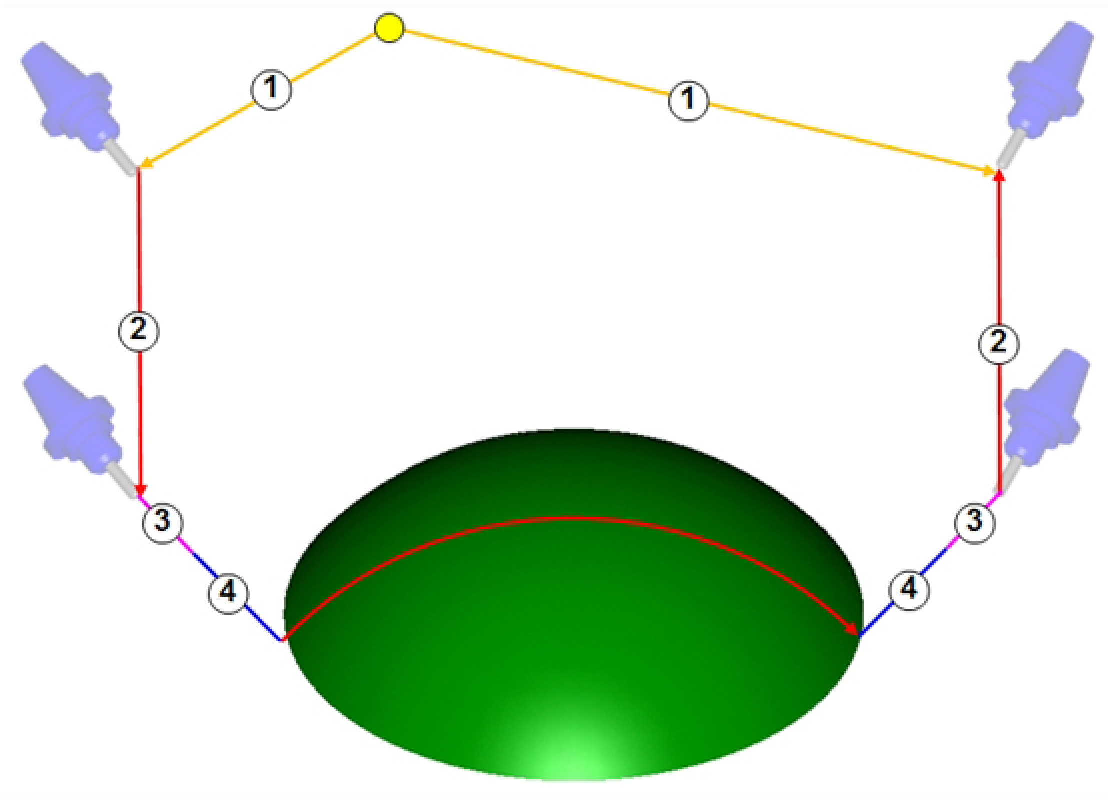
Don’t use Lead-Out

If lead-out is not used, then the tool will move away from the part to the clearance position along the tool axis from the last point in the toolpath in a straight-line motion.

Use Home Positions

The home position is a certain point located relative to the part's zero point. When both checkboxes are selected, the tool will start from this position and retract to it right after the link type is performed.

In the following picture, the yellow point symbolizes the home position, and clearance area was chosen for both First entry and Last exit. Therefore, the tool first retracts to the feed distance (4), then to the rapid distance (3), and then to the clearance area (2), before it finally retracts to the home position (1).



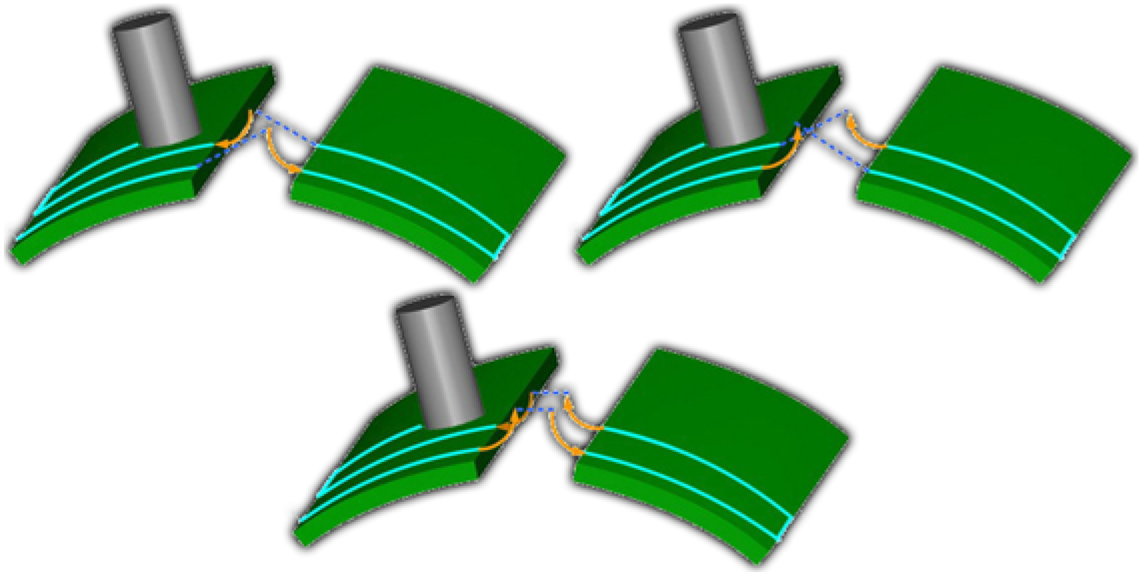
Note that the tool orientation is 0,0,1. In case a collision is detected, the tool will retract along its axis until the path to the home position is collision-free.

Gaps Along Cut

In the Link tab, you can choose items under Gaps Along Cut to control the tool's movement if gaps are found in the toolpath, such as a space between surfaces. You have different choices for how the tool should move across the gap and resume machining. These options include Direct, Follow surfaces, Blend spline, Retract to feed distance, Retract to rapid distance and Retract to clearance area.

The system recognizes a difference in the size of gaps. Depending on the size of the gap, it is possible to have different strategies based on whether the gap is big or small. In addition to the method for handling the gap you may set how the tool moves to and from the surface using the Default Lead-In/Out values. You may also set a custom definition for how the gaps are handled when Use Lead-In, Use Lead-Out, or Use Lead-In/Out are selected.

Using a lead-in for entry moves or a lead-out for exit moves will extend the toolpath.



Example of an Entry lead-in use, Exit lead-out use and using both Entry and Exit

Small Gap Size

Here you set the threshold for determining whether a gap in a toolpath segment is small or large.

in % of tool diameter

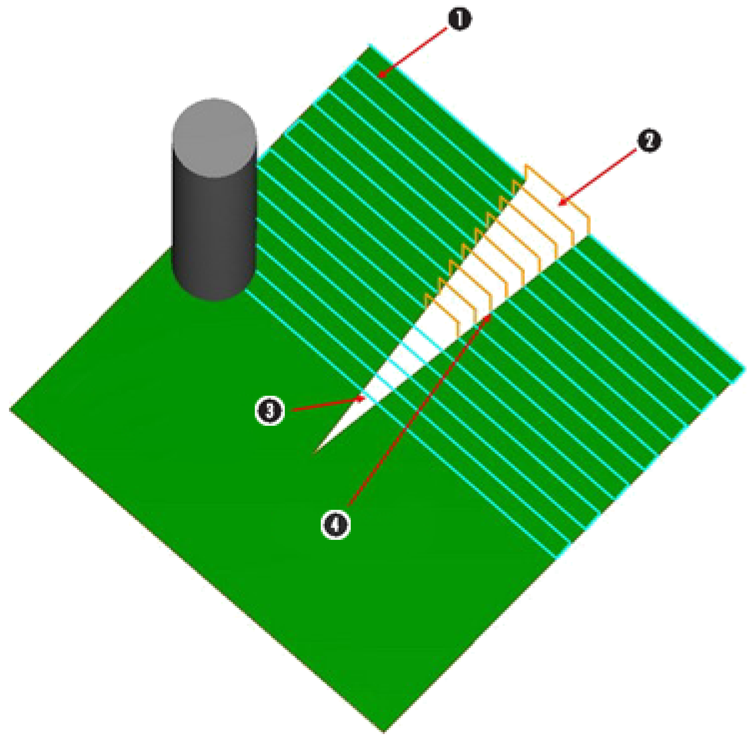
The value is defined as a percentage of tool diameter. Any gaps along the toolpath segment that are smaller than this value are considered to be small gaps and the system will use the selected strategy for traversing this space. Any gaps along the toolpath segment that are larger than this value are considered to be large gaps and the system will use the selected strategy for traversing this space. For example, if the tool diameter is 20mm and the gap size is set to 10% then the threshold is 2mm. All gaps which are smaller than 2mm are considered to be small gaps, all gaps greater than 2mm are considered to be large gaps.

As value

If you do not want the gap threshold to be based on the tool diameter you may use this option and set a value for the gap size. Any gaps which are smaller than this value are considered to be small gaps. All gaps greater than this value are considered to be large gaps.

Example

This example shows a surface with a gap which becomes smaller along the toolpath. For this operation the strategy for large gaps is Retract to feed distance and for small gaps it is Direct. The gap size is set to 50% of the tool's diameter. The tool diameter is 20mm so the small gaps are 10mm or less and the big gaps are larger than 10 mm. You can see that for the large gaps the tool pulls off the surface, feeds across the gap and then feeds back onto the part. The tool ignores the small gaps and just continues cutting across the space with a straight line connecting move.



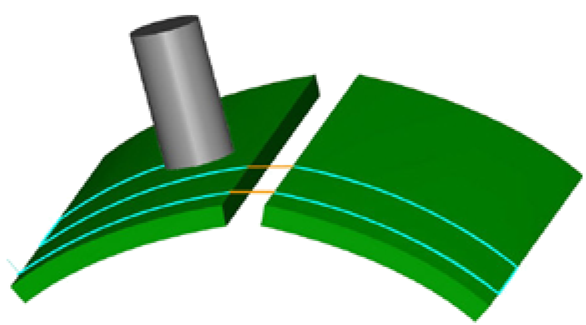
1. Toolpath
2. Gap
3. Small Gaps
4. Large Gaps

The Gap Options

These options describe the tool behavior when encountering a large or small gap. The threshold for whether a gap is large or small is set using either the Small Gap Size parameter or the As value parameter.

Direct

With this option the tool follows the shortest path to the other side of the gap without any retract movements. The toolpath in the gap is straight line and traverses the gap at the contour feedrate.

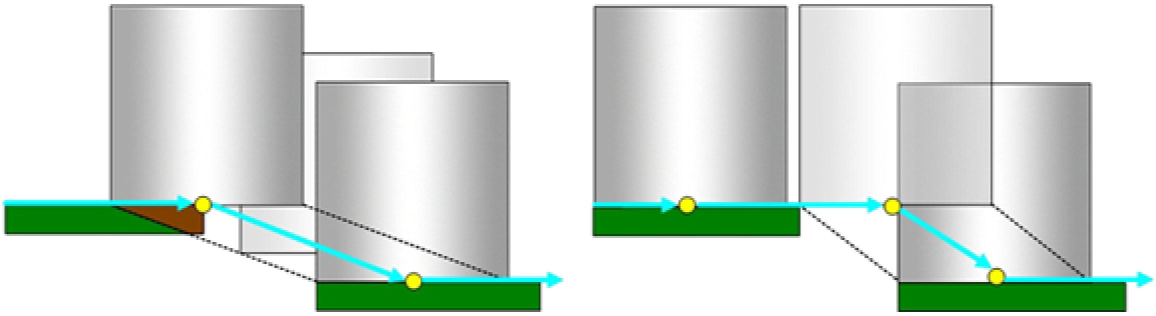


Follow surfaces

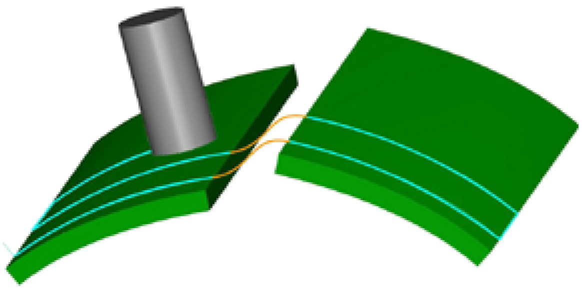
With this option the tool tries to follow the surface geometry. Toolpath will be generated similarly to the closed surface geometry, even across gaps. Please note that this strategy is gouge-protected only when gouge checking is active.

In the image below left you can see a Direct connection. The cutter location has reached the end of the surface and immediately connects to the next surface. In doing so it gouges the upper surface.

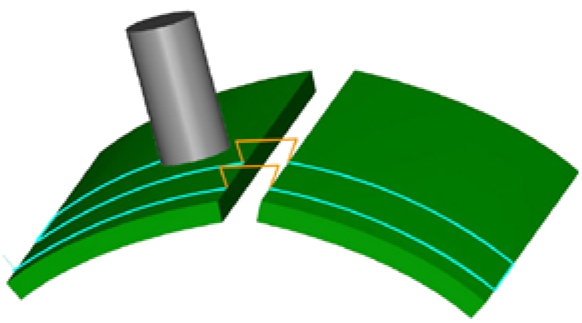
With Follow surface the tool moves to the end of the upper surface and connects only when the whole tool has left the surface.



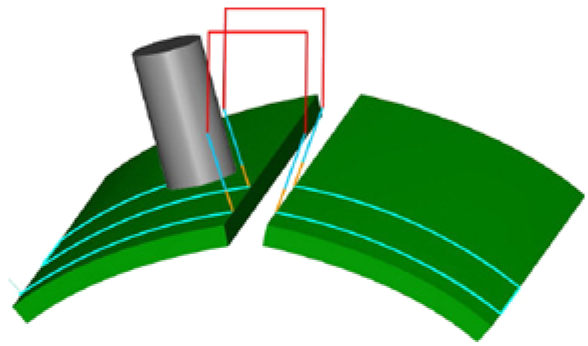
Blend spline
With this option gaps are traversed with a spline. The toolpath leaves and enters the drive surfaces tangentially. This results in very smooth toolpath.



Retract to feed distance
When a gap is detected the tool retracts to the feed distance along the tool axis. The tool then feeds over to the next point in the toolpath and feeds onto the surface.

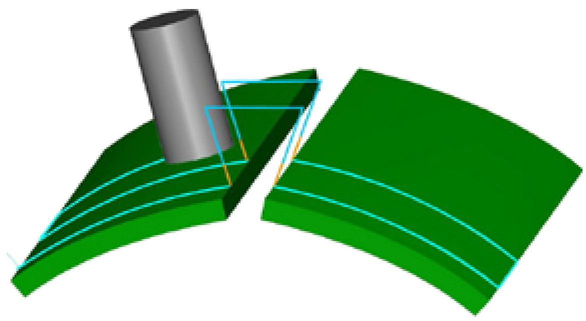


Retract to clearance area
With this option the tool retracts to the clearance area when a gap is detected. The tool feeds to the feed distance and rapids to the rapid distance along the tool axis. The tool then moves to the clearance distance before moving to the next toolpath point.



Retract to rapid distance

When a gap is detected the tool retracts along the tool axis to the feed distance and then the rapid distance. The tool then rapids to the next point in the toolpath rapids down to the feed distance and then feeds onto the surface.

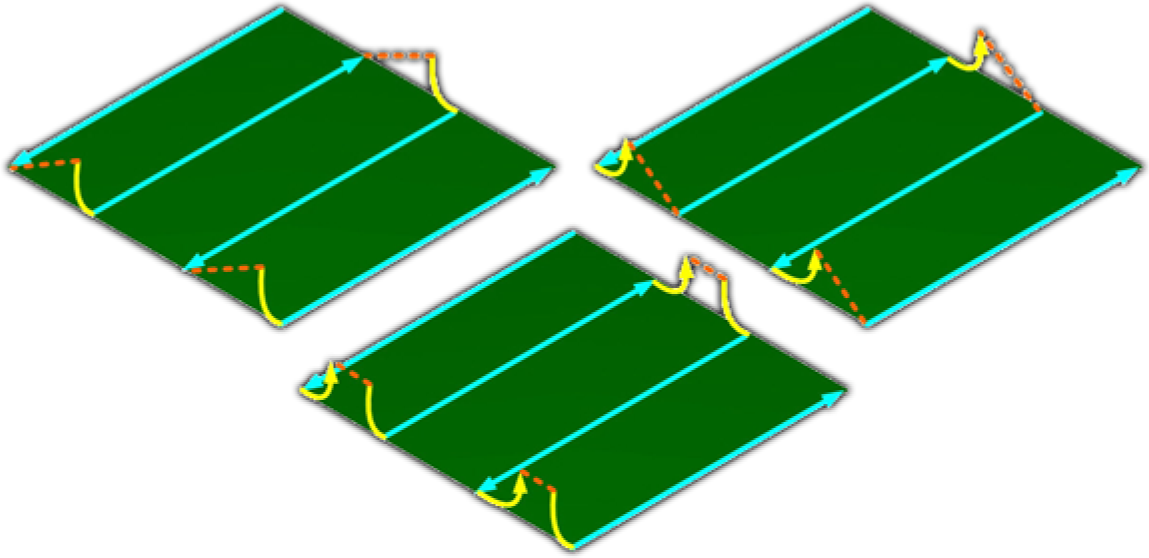


Links Between Slice

In the Link tab, you can choose items under Links between slice to control the tool's movement when stepping over to the next cut and to provide different choices for how the tool should move during the stepover. These options include Direct, Retract to feed distance, Retract to clearance area, Follow surfaces, Blend spline and Retract to rapid distance.

The system recognizes a difference in the size of links or stepovers, being either small or large moves. In addition to the method for handling the stepover, you can set how the tool moves to and from the surface using the Default Lead-In/Out option.

Using a lead-in for entry moves or a lead-out for exit moves will extend the toolpath.



Example of an Entry lead-in use, Exit lead-out use and using both Entry and Exit

Small move size

The value in this field sets the threshold for determining whether a stepover is small or large.

in % of stepover

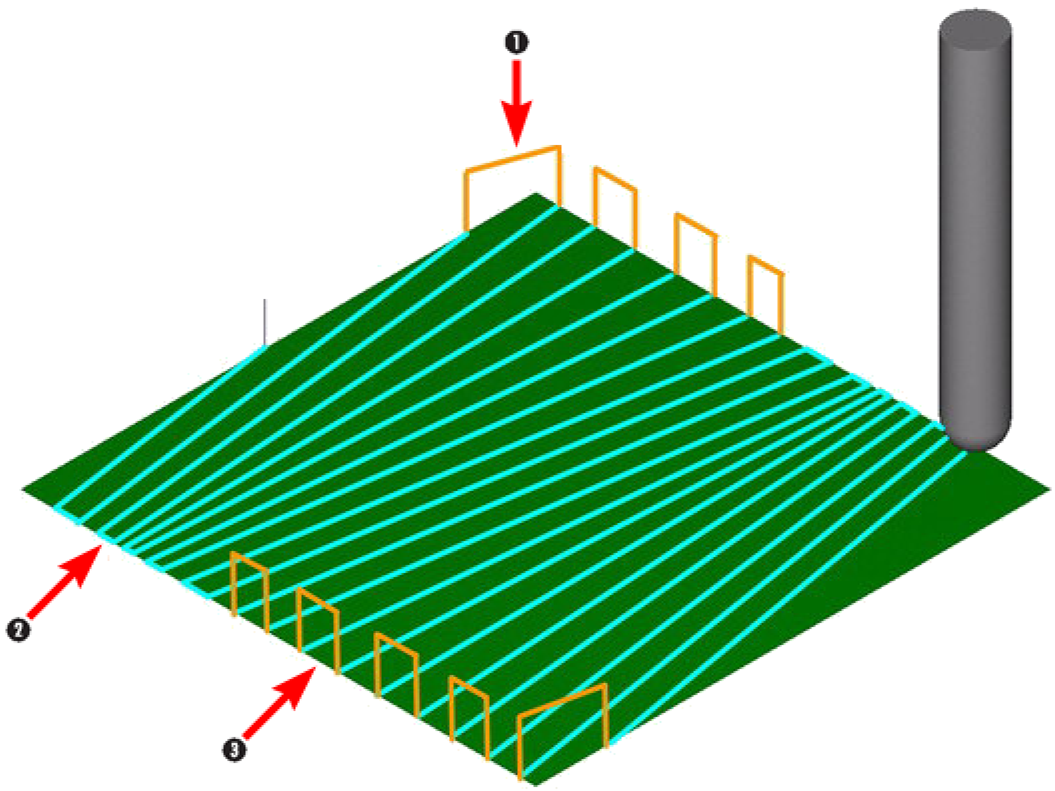
The value is defined as a percentage of the user given maximum step over value. For example, if this value is set to 150% and the maximum step over value is 0.1mm the gap threshold is 0.15mm. This means that all stepover moves from one toolpath slice to the next are checked against this 0.15mm and determined whether the gap is smaller or larger than this value.

As Value

If you do not want the gap size to be dependent on the maximum stepover you may set a value to use for the threshold. Any gaps which are smaller than this value are considered to be small moves; all gaps greater than this value are considered to be large moves.

Example

Here you see a surface with cuts that are not parallel. The distance between cuts is different at the edge of the surface where the tool steps over to next the cut. Where there are large links the tool retracts using the Retract to feed distance method and where the links are small the tool steps over using the Direct method.

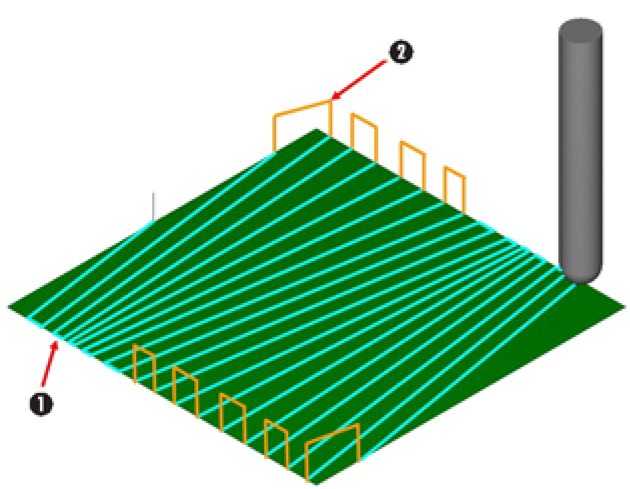


The Links Between Slice Options

These options describe the tool behavior when performing a stepover in the toolpath.

Direct

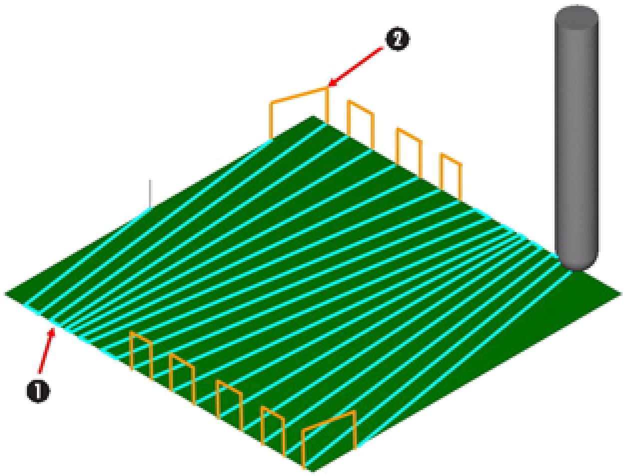
With this option the tool follows the shortest path to the next slice without any retract movements. The toolpath in the link is a straight line and the tool moves at the contour feedrate.



1. Direct
2. Broken Feed

Retract to feed distance

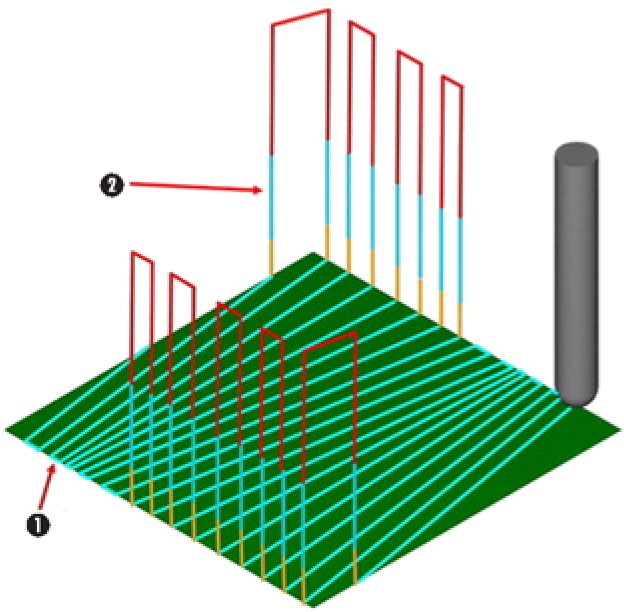
With this option the tool retracts to the feed distance along the tool axis at the machining speed. The tool then feeds to the next point in the toolpath and feeds onto the surface.



1. Direct
2. Broken Feed

Retract to clearance area

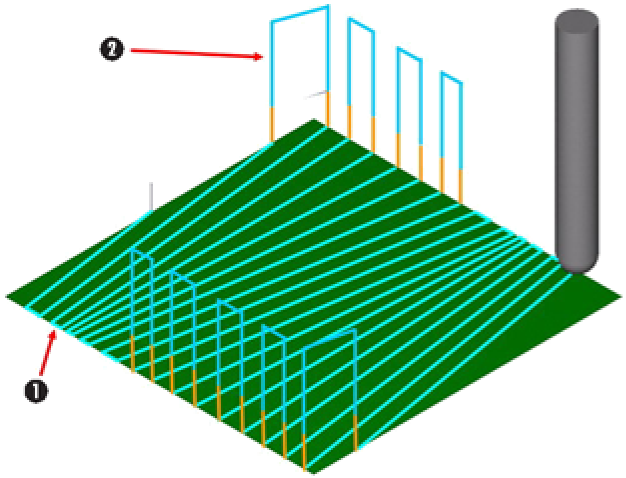
With this option the tool moves in rapid back to the clearance area. Only the return to the drive surface is done at the contour feedrate.



1. Direct
2. Clearance Area

Retract to rapid distance

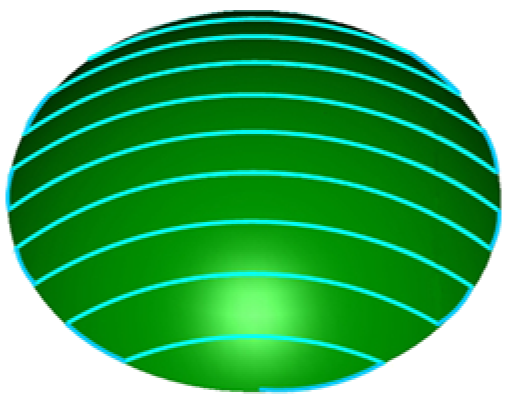
With this option the tool retracts by the feed distance plus the rapid distance along the tool axis. The tool rapids from the drive surface to the rapid distance. From there over to the next point and back down to the rapid distance (cyan moves in the image below) the tool moves at the rapid rate. The moves to and from part to feed distance are feed moves.



1. Direct
2. Broken Feed and Rapid

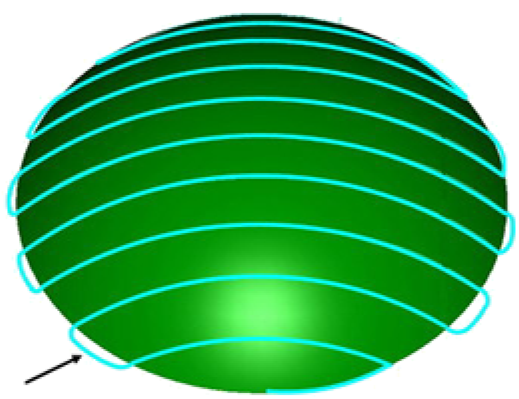
Follow surfaces

With this option the tool tries to follow the geometry and check surfaces between the slices. Please note that this strategy is gouge-protected only when gouge-checking is active.



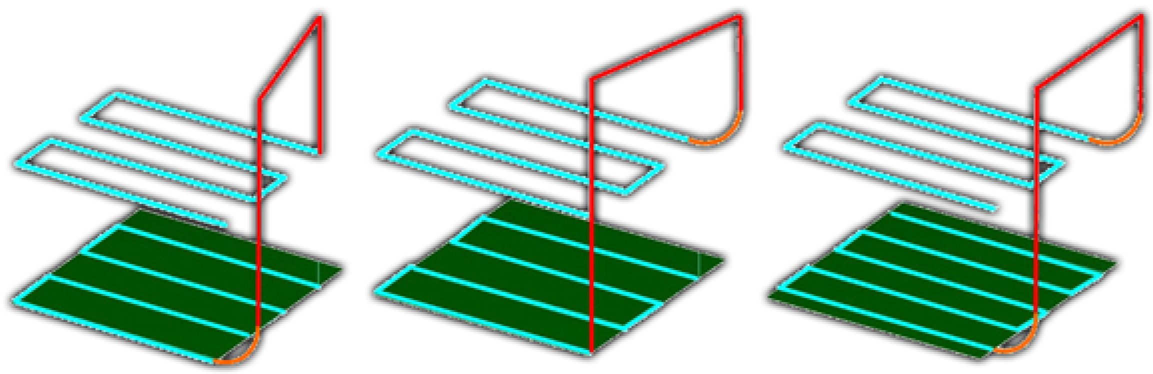
Blend spline

With this option the tool movement follows an arc that leaves and enters the drive surface tangentially.



Links Between Passes

When the multiple passes option is used, the system will generate horizontal or vertical slices (for details, see “Multi Passes” on page 271). In the Link tab, you can choose items under Links between passes to control the tool’s movement when moving from one pass to the next. The choices for how the tool should move during the stepover include: Direct, Retract to feed distance, Retract to clearance area, Follow surfaces, Blend spline and Retract to rapid distance. In addition to the method for handling the move between passes you can set how the tool moves to and from the surface using the Default Lead-In/Out option.



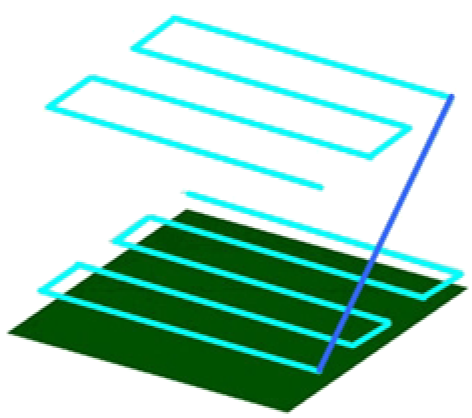
Example of an Entry macro use, Exit macro use and using both Entry and Exit

The Links Between Passes Options

These options describe the toolpath behavior when moving from one pass to the next.

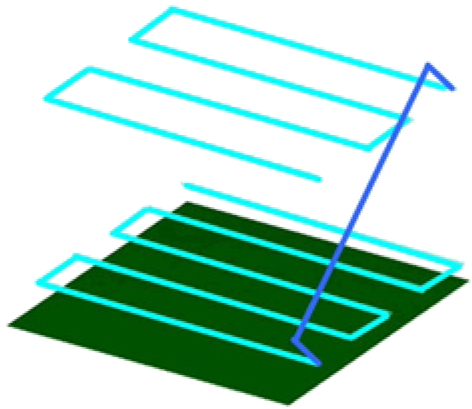
Direct

With this option the tool follows the shortest path to the next slice without any retract movements.



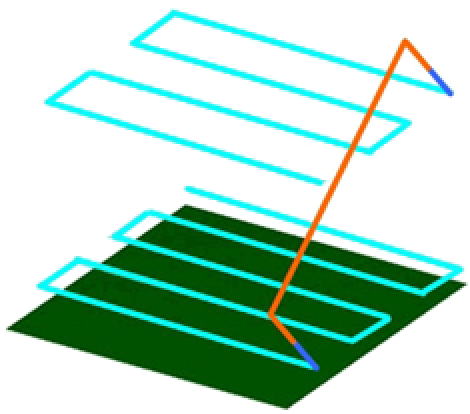
Broken feed

With this option the tool retracts by the feed distance at the contour feedrate. The tool then feeds in a straight line to above the start point in the next pass and feeds onto the surface.



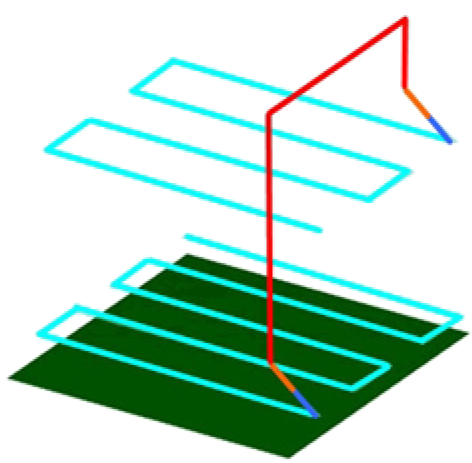
Retract to rapid distance

With this option the tool retracts by the feed distance plus the rapid distance. The tool then rapids from the drive surface and moves over to the next pass start point.



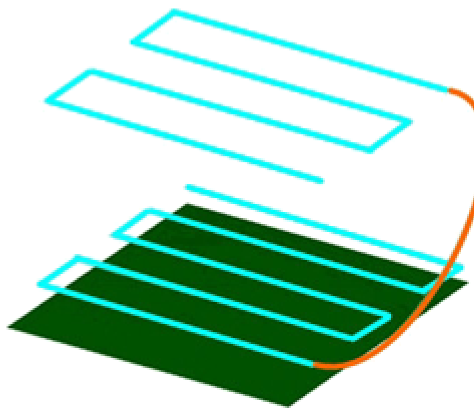
Retract to clearance area

With this option the tool moves back to the clearance area and restarts the entry process for the next pass.



Blend spline

With this option the tool moves from one pass to the next on an arc that is tangential to the end and next start point.



Follow surface

With this option the tool tries to follow the geometry and check surfaces between the slices. Please note that this function only works when gouge checking is active.

Retracts Dialog

In the **Link** page, depending on the drive surface and machining strategy, clicking the Retracts button opens the **Retracts** dialog, which offers the following types of settings:

Home position

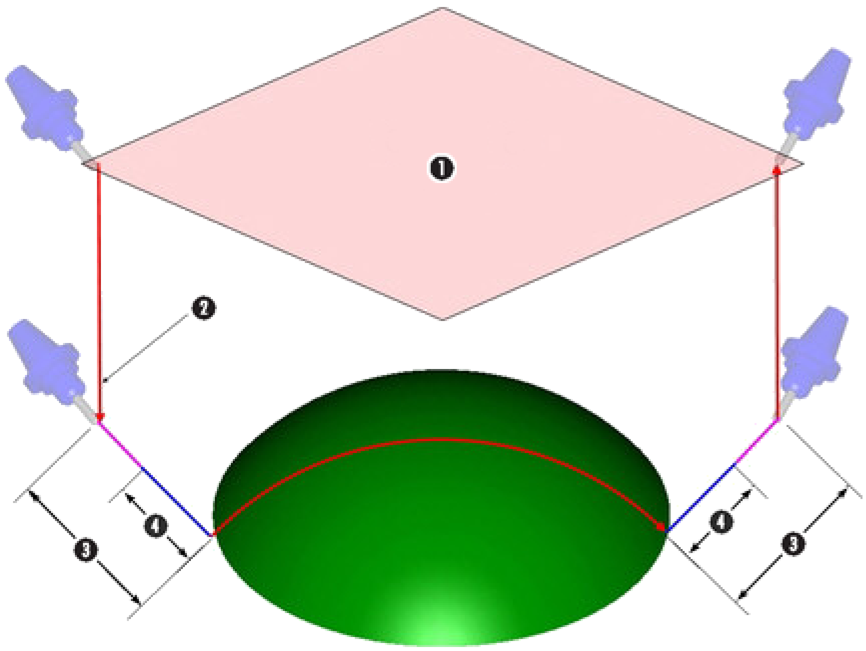
You can specify a point either by typing values for X,Y,Z or by clicking the selector button(...) to select geometry in the workspace.

Clearance area

The controls under Clearance area let you specify the area in which the tool can travel without hitting the work piece. You can choose from three different clearance area types: Plane, Cylinder, or Sphere. For detailed information, see “ Clearance Area Types ” on page 254.

Distances

The controls offered by Distances let you define the feed and rapid distance to approach the part and retract away from the part.



1. Clearance Area
2. Toolpath
3. Rapid Distance
4. Entry Feed Distance
5. Exit Feed Distance

The illustration shows toolpath (2) as the tool approaches from a clearance plane (1), then travels through a Rapid distance (3) and Entry feed distance (4), then cuts, then travels through an Exit feed distance (5) and Rapid distance (3), and then retracts back to the clearance plane.

Rapid distance

The tool moves at the rapid feedrate when it is at the clearance area, and then it moves its final orientation while moving from the clearance area to the rapid distance. After the tool is at the rapid distance, it has the correct orientation for the first cut. In the illustration, see item 3.

Entry feed distance

The feed distance upon entering the part. In the illustration above, see item 4.

Exit feed distance

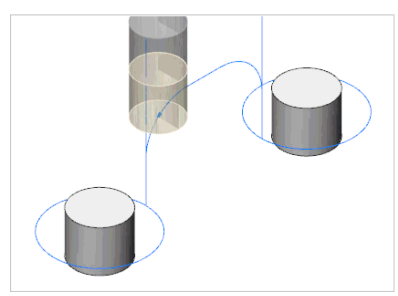
The feed distance upon exiting the part. In the illustration above, see item 5.

Air Move Safety Distance

Minimum distance between the clearance area and the drive surface or check surface.

Arc fit

Provides tangential arcs for the segments where the tool approaches and retracts. If you select one or more checkboxes (from: Clearance area, Rapid distance, or Feed distance), you can specify a value for Arc radius. The illustration shows the effect of specifying Arc fit for Feed distance.

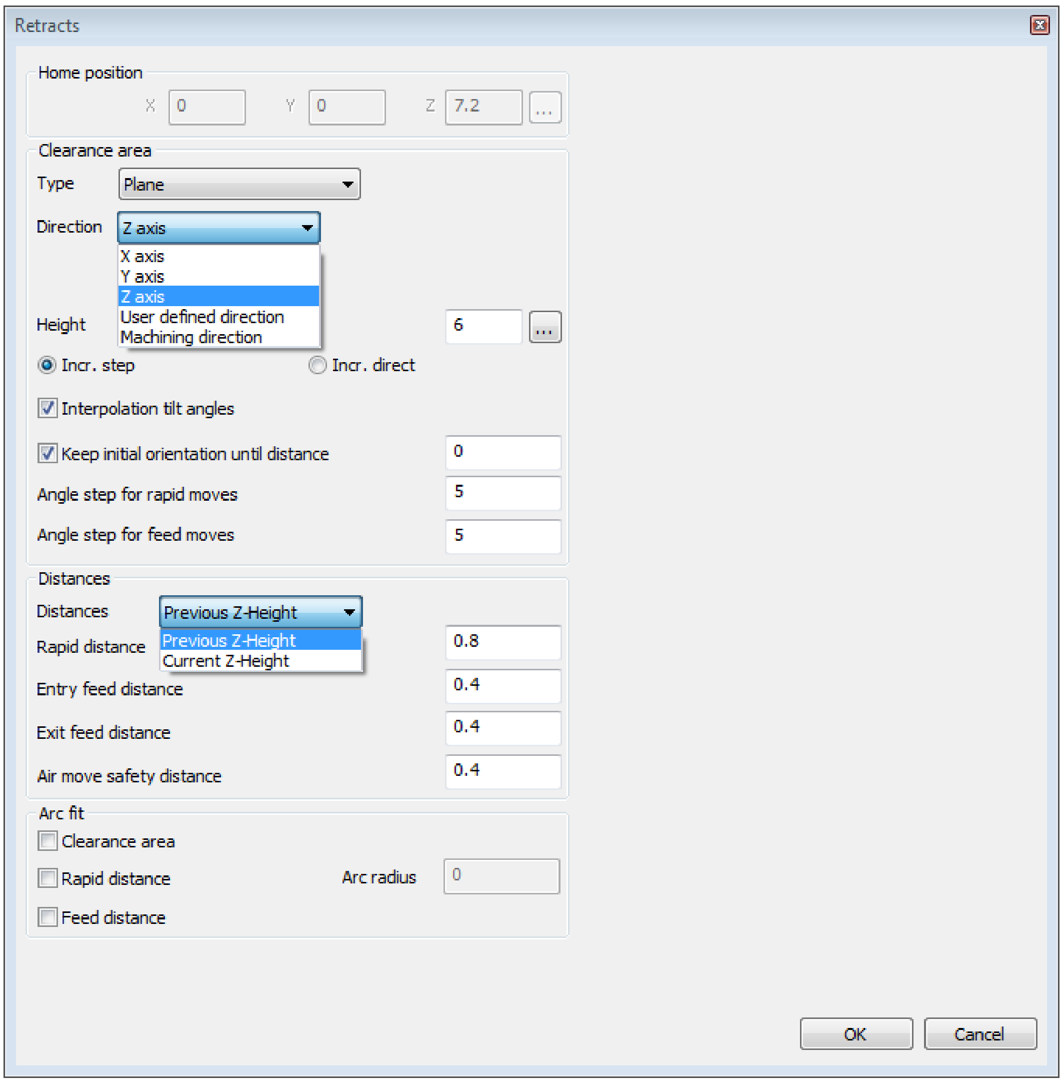


Clearance Area Types

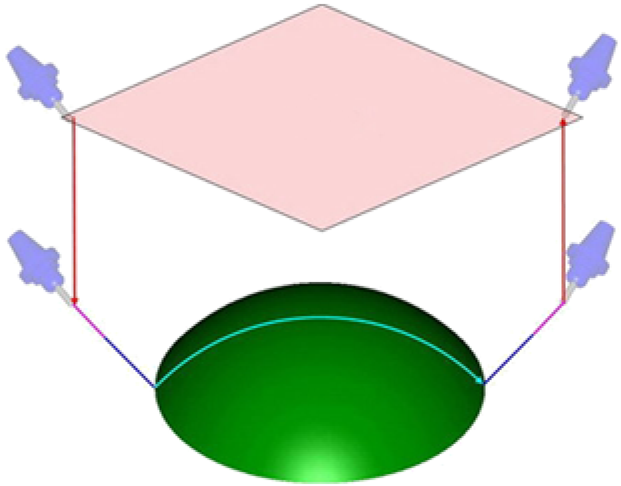
In the **Retracts** dialog, Clearance area offers the following types of settings: Plane below, “Cylinder” on page 256, and “Sphere” on page 258. For other controls in the **Retracts** dialog, see “Retracts Dialog” on page 252.

Plane

The plane is the default clearance area setting. This clearance area is a plane normal to a Direction that you choose – of X axis, Y axis, Z axis, User-defined direction, or Machining direction – , with the distance to the plane set by the Height. You can manually enter a value, or you can select a point in the workspace. If you select a point, the system will load that point’s depth from CS1 for the Height value.

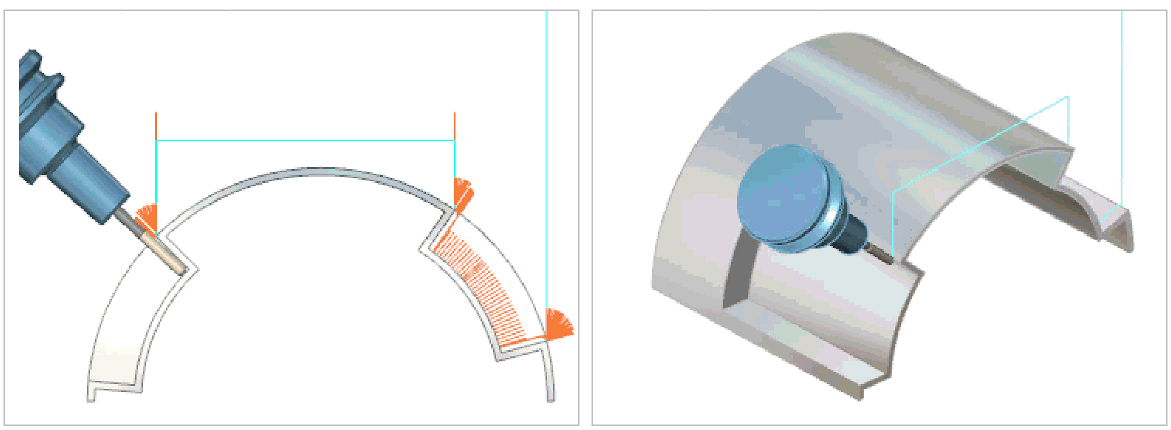


As with all of the clearance settings, the tool will rapid from the position to the rapid distance. Along the way, the tool will align to its correct orientation for the operation.



Incremental height and Incremental step or direct

You can enter a value for incremental height from the surface itself, as illustrated below, and specify either a step or a direct move from one height to the next.



Interpolation tilt angles

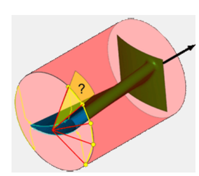
Select this checkbox if you want the system to calculate tilt angles as interpolated moves.

Keep initial orientation until distance

When moving from the last point of the rapid move to the clearance area, the tool will maintain its initial orientation until it reaches this distance above the part before orienting itself to the clearance orientation (1,0,0).

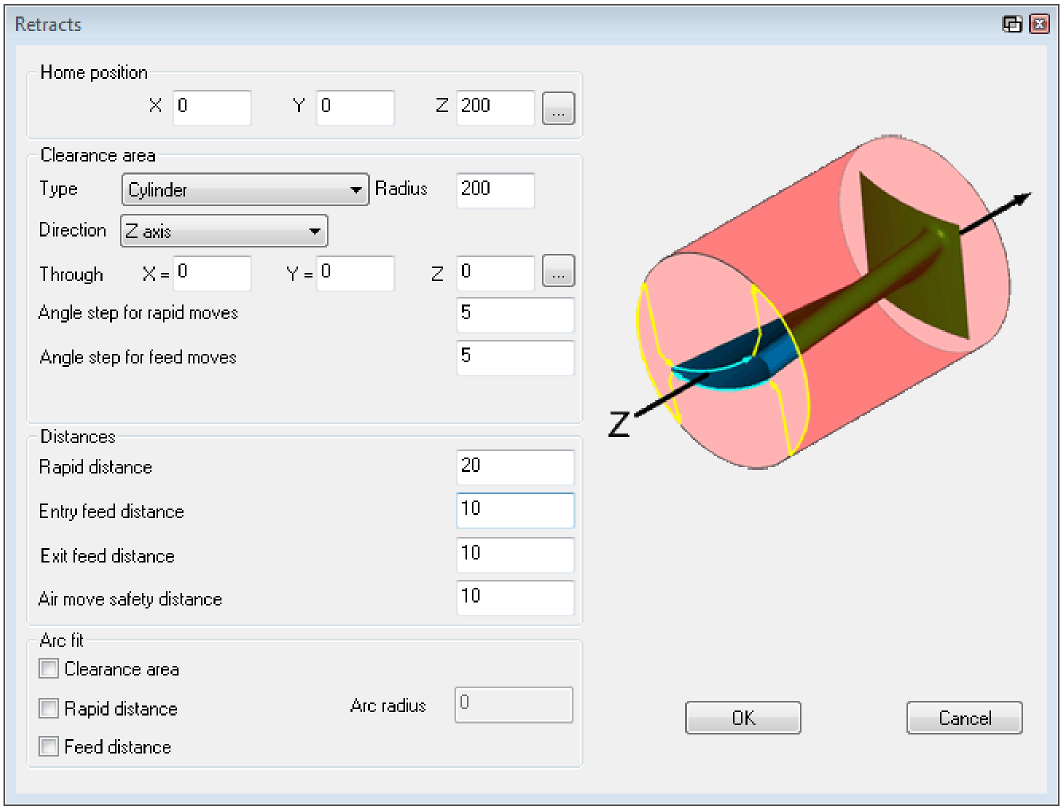
Angle step for feed moves / for rapid moves

These two parameters specify the maximum step for orientation changes during feed moves / during rapid moves.



Cylinder

This clearance area type has a cylindrical shape whose axis is aligned to a Direction that you choose, of: X axis, Y axis, Z axis, orUser-defined direction. When defining this clearance area, be sure that it completely encloses your drive surface geometry. The cylinder extends in infinity along the specified Direction.



Radius

This is the main control for this clearance option. This value is the radius of the cylinder, which is centered on the axis.

Through

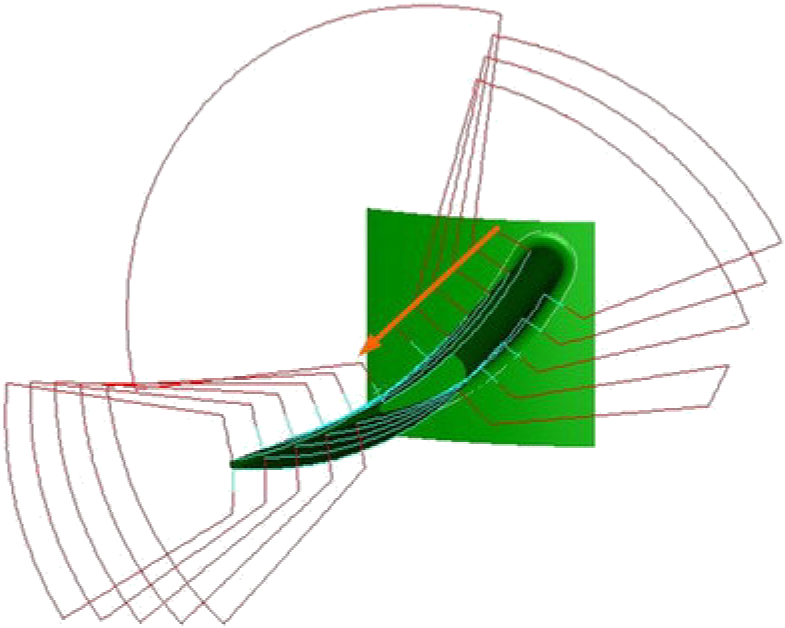
You can offset the position of the cylinder, parallel to the axis, by changing the Through position. Please note that since the cylinder is infinite in length, any value entered for a point along the parallel axis will be ignored. For example, if you choose the 3D point `X+10`, `Y-5`, `Z+15`, and the cylinder is parallel to Z, then the Z value will be displayed but ignored.

Angle step for feed moves / for rapid moves

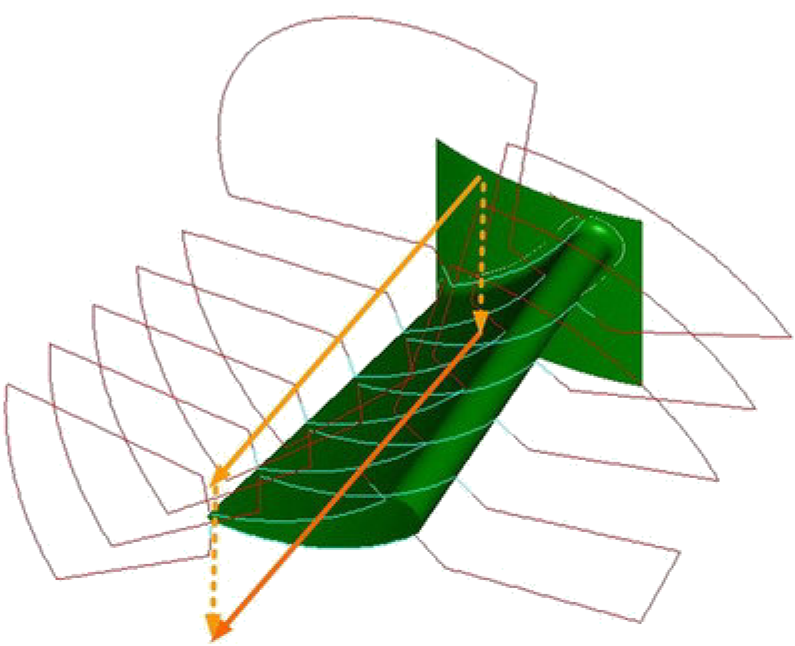
These two parameters control the length of the curved feed moves (or rapid moves) that occur when moving along a non-planar clearance area, such as a cylinder or sphere. The curved feed moves (rapid moves) are segmented into shorter line moves that do not exceed the angle step. This is similar to Angular Segmentation.

Example

Here you can see the machining of a turbine blade. Only the sides of the blade will be machined, and so the small radius edges are left out. Note how the tool retracts to the clearance area cylinder, which is parallel to the X-axis. As you can see, the turbine blade center is not exactly through the X-axis.

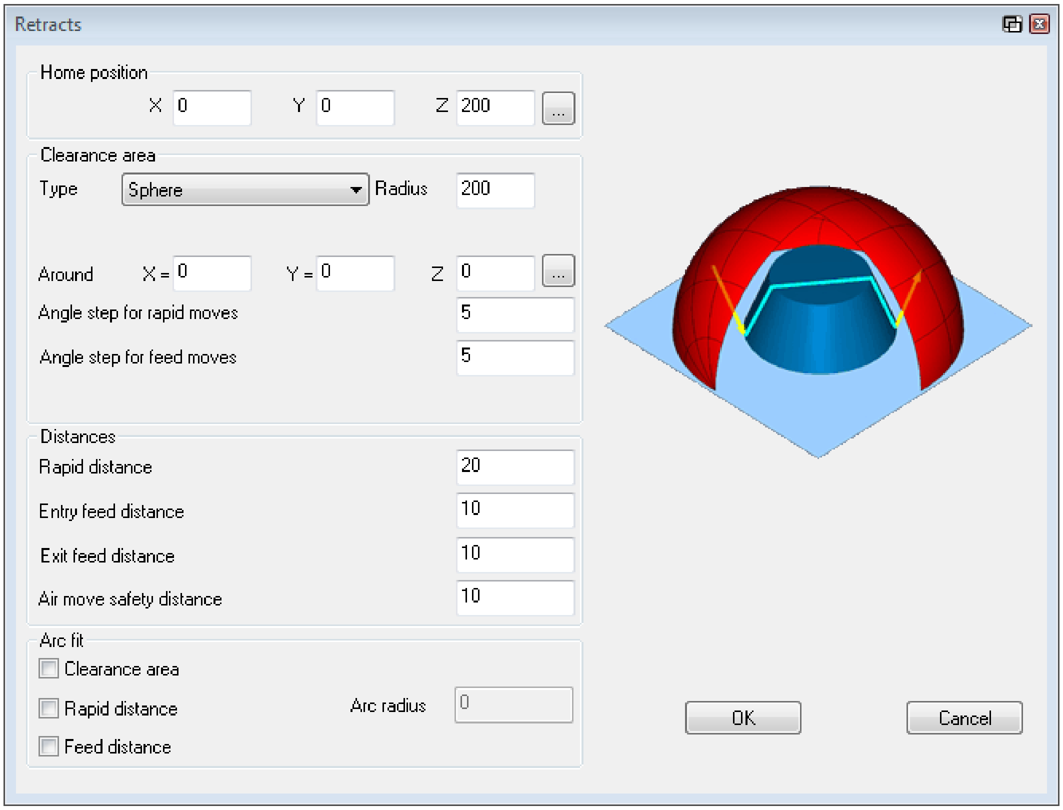


To move the cylinder, you can set an offset to the X-axis so that the axis of the cylinder moves to the center of the turbine blade. After you specify the offset, the cylinder is moved down.



Sphere

This clearance area type uses a spherical shape to surround the drive surfaces. The sphere should completely enclose your drive surface geometry. On most machines, the tool typically cannot orient to every possible angle, so you do not need the full sphere. Usually, you will be concerned with defining half of a sphere located above your drive surface.



Radius

This value sets the size of the sphere.

Around

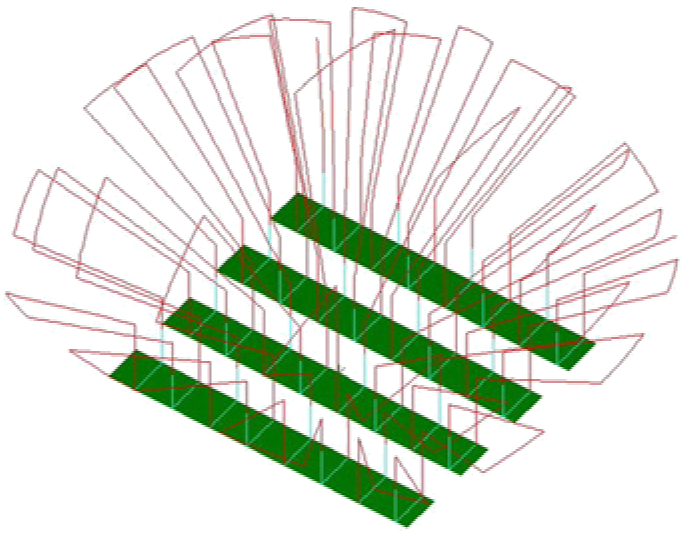
With these values you set the position of the sphere. You must be sure the sphere completely encloses your drive surfaces. These are absolute values in the coordinate system.

Angle step for feed moves / for rapid moves

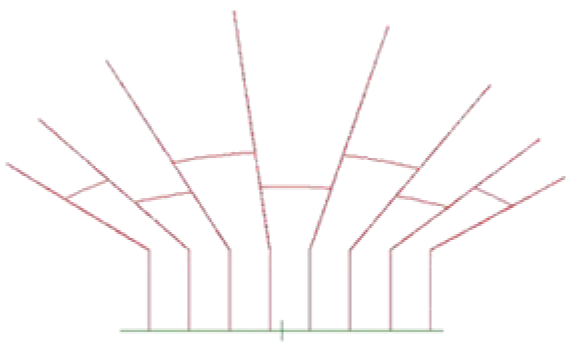
These two parameters control the length of the curved feed moves (or rapid moves) that occur when moving along a non-planar clearance area, such as a cylinder or sphere. The curved feed moves (rapid moves) are segmented into shorter line moves that do not exceed the angle step. This is similar to Angular Segmentation.

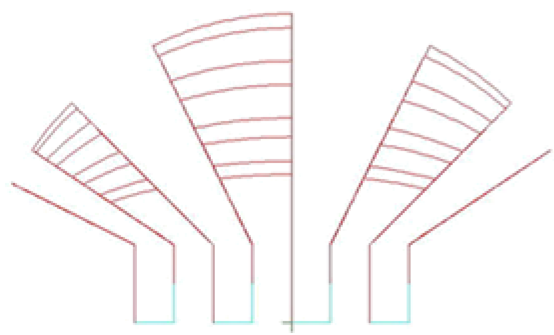
Example

In the following image you can see four drive parallel surfaces machined. At the gaps in the surface, the tool retracts to the clearance area. With all of these retracts, you can begin to see the spherical shape of the clearance area.



Here are some different views to help visualize this option in use.





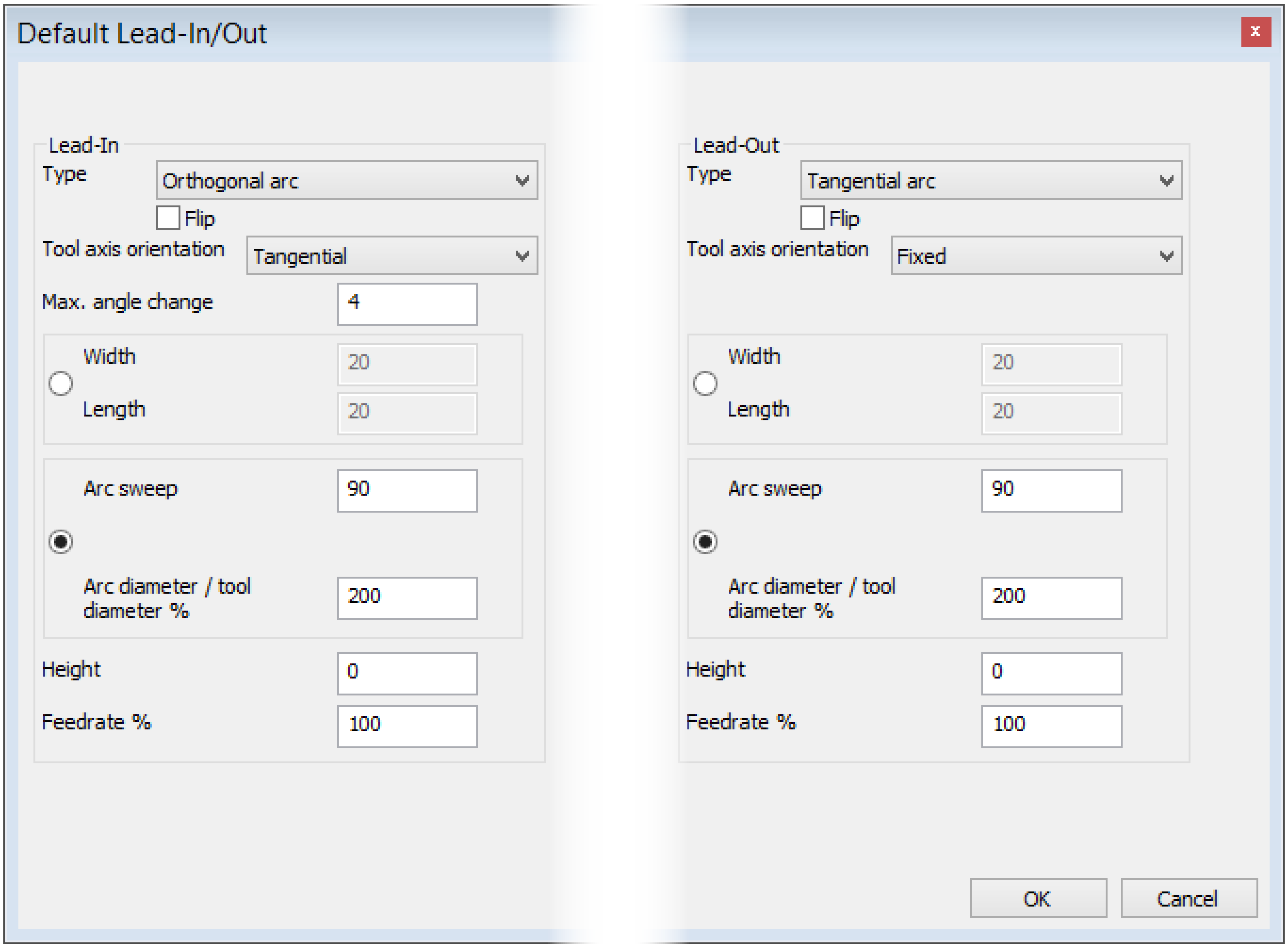
Side View	Front View

Default Lead-In/Out

In the **Link** page, the Default Lead-In/Out opens a dialog that lets you define default settings for Lead-In and Lead-Out that can be used when Lead-In and/or Lead-Out moves are desired for Entry/Exit, Gaps Along Cut, Links Between Slice, or Links Between Passes. Using a Default Lead-In/Out saves you from having to redefine the Lead-In/Out parameters for each link type. When a macro is activated, no matter which Link option you have selected, the macro moves are placed in addition to the link moves.

The Lead-In defines the toolpath before the tool enters the drive surface and the Lead-Out defines the toolpath after the tool leaves the drive surface. You can choose a different Type for Lead-In and for Lead-Out. Types of Lead-In / Lead-Out include the following: Tangential arc, Reverse tangential arc, Vertical tangential arc, Reverse Vertical Tangential Arc, Horizontal tangential arc, Orthogonal arc, Tangential line, Reverse Tangential Line, Orthogonal line, Reverse vertical profile ramp, and Position line.

Additional options can be set to achieve exactly the motion you want. For details, see “Lead-In/Out Settings” on page 264.



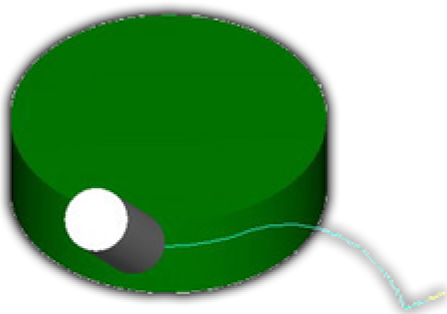
Lead-In/Out Types

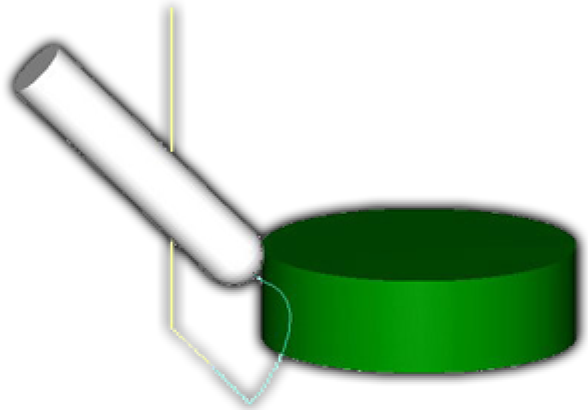
You can set up your Lead-In/Out moves with the following options.

Tangential arc

This option connects tangentially to the first toolpath point of the drive surface. The orientation of the arc is 90° or normal to the tool axis so the arc orientation depends on the side tilt orientation of the tool.

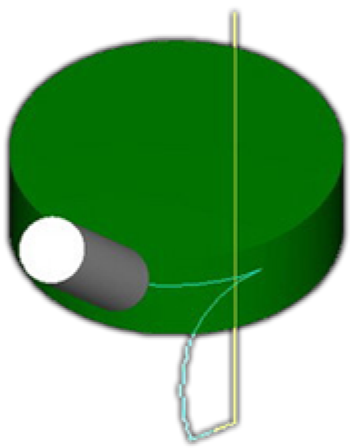
In the example below the tool is tilted 45° to the side, so the arc orientation is also 45° to the drive surface. Setting a side tilt angle of 0° would cause the arc to be vertical. With a 90° side tilt angle the arc is horizontal.





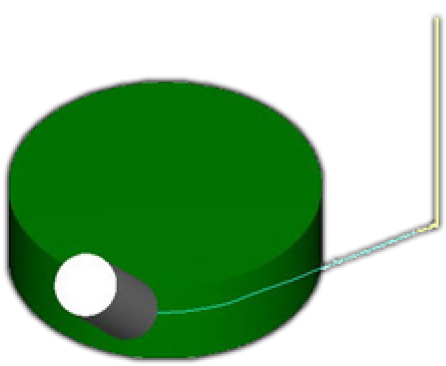
Reverse tangential arc

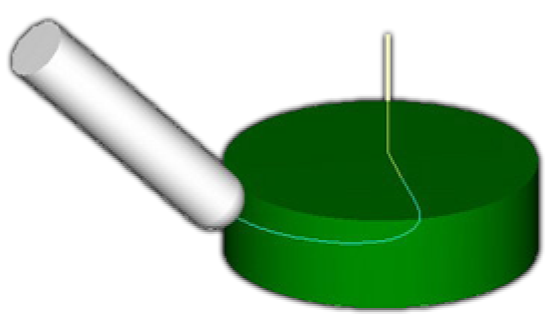
This option is basically the same as the Tangential arc in that it connects tangentially to the cut on the drive surface. The orientation of the arc depends on the side tilt orientation of the tool. With this option the direction of the arc is reverse to Tangential arc.



Vertical tangential arc

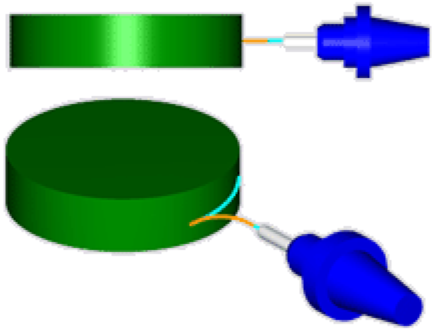
With this option the tool leads-in with a vertical arc to the drive surface. In this case “vertical” refers to the direction of the tool orientation and not with the angle of 90°, as with Tangential arc. If the tool tilts to the side, the vertical tangential arc has the same orientation.





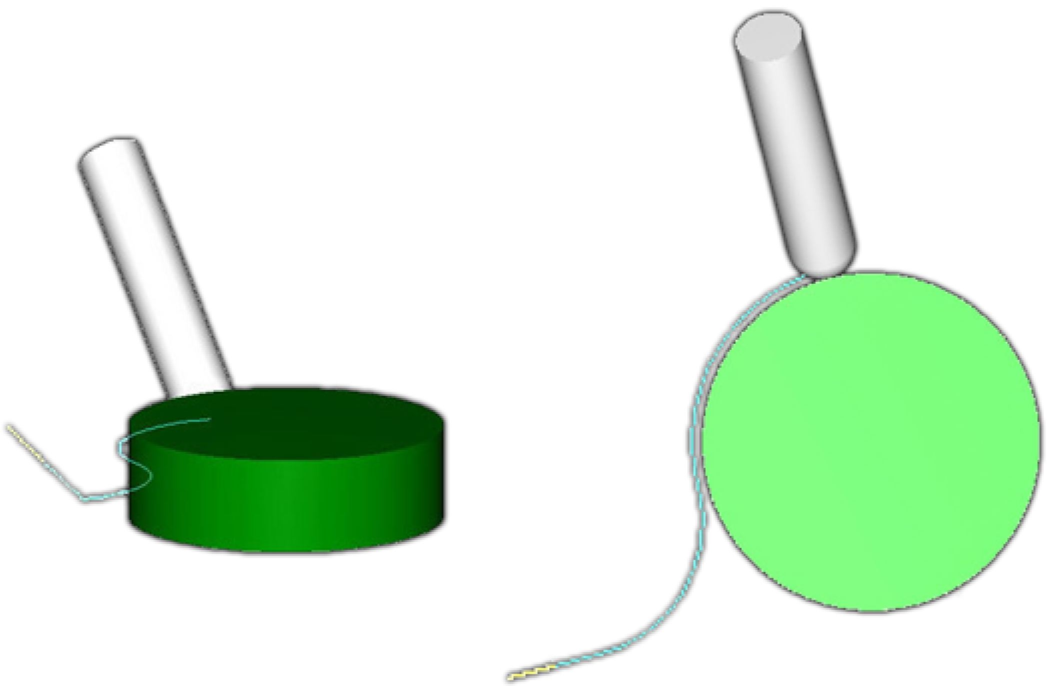
Reverse Vertical Tangential Arc

This option is functionally the same as Vertical tangential arc except with this option the direction of the arc is reversed.



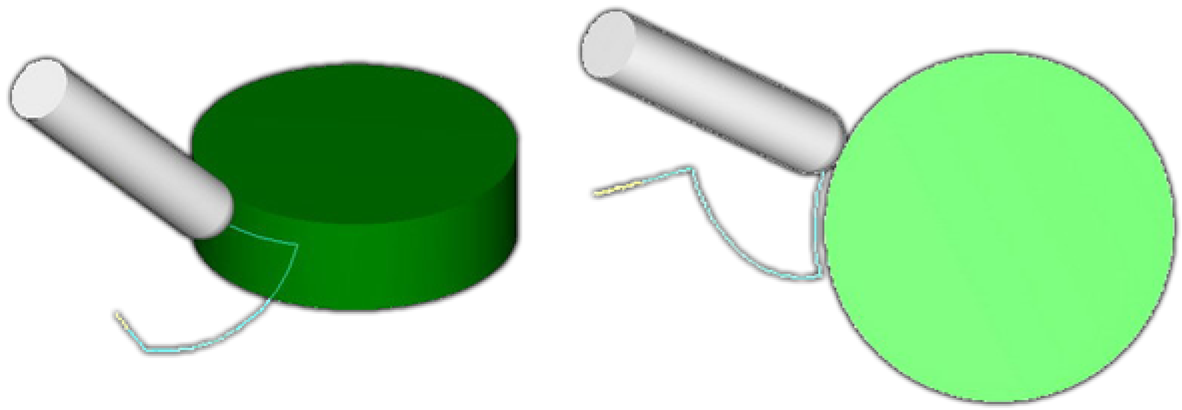
Horizontal tangential arc

With this option the tool enters the drive surface on the same horizontal level as the slice of the cut. The arc orientation is independent from the tool orientation.



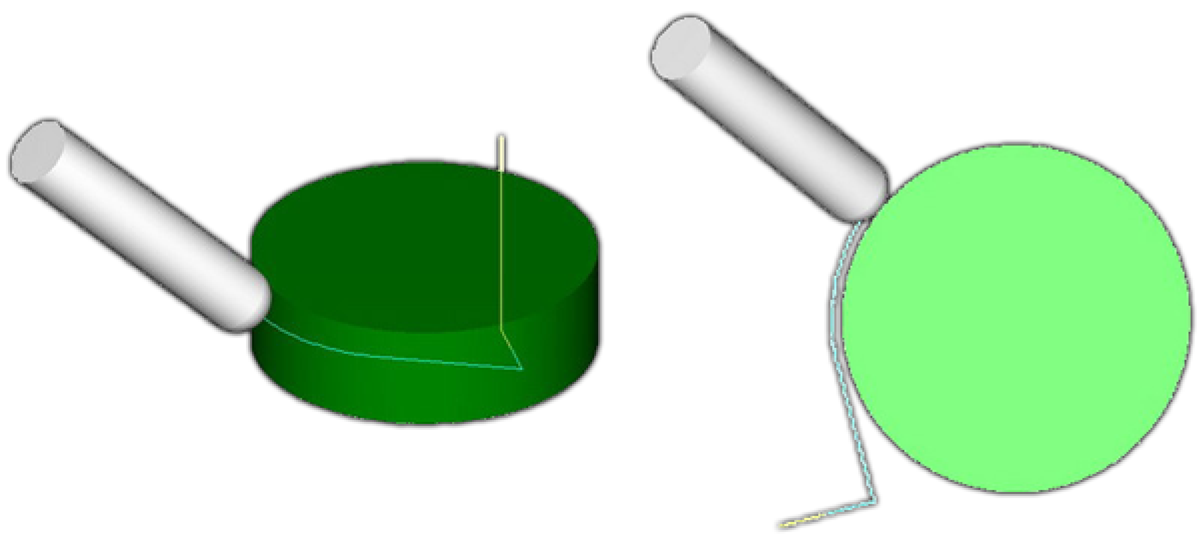
Orthogonal arc

This option connects orthogonal to the first toolpath point of the drive surface. The orientation of the arc is 90° to the tool axis, so the arc orientation depends on the side tilt orientation of the tool.



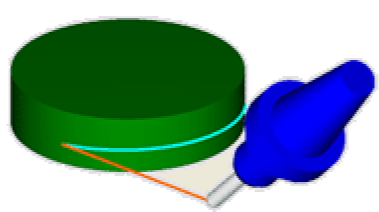
Tangential line

This option connects tangentially to the first toolpath point. The lead angle is normal to the slice level of the first cut and the side tilt angle is the same as the programmed side tilt angle.



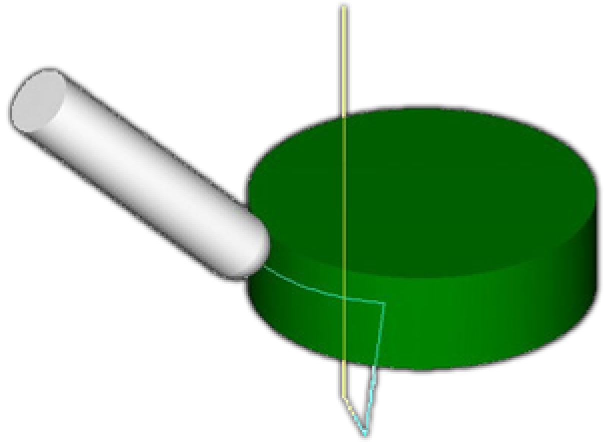
Reverse Tangential Line

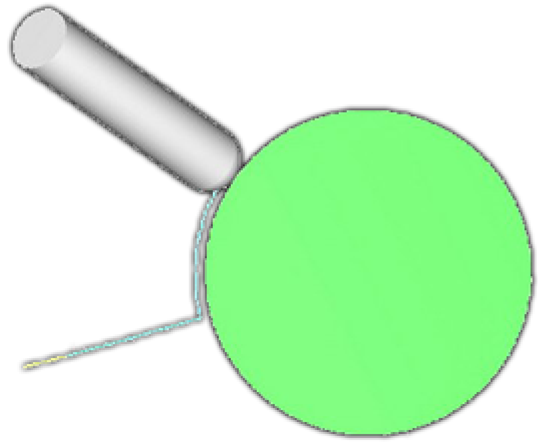
This option is functionally the same as Tangential line except that with this option the direction of the line is reversed.



Orthogonal line

This option connects orthogonally or perpendicular to the first toolpath point on the drive surface. The orientation of the line is 90° to the tool axis, so the line orientation depends on the side tilt orientation of the tool.





Position line

This option offers a selection button (...) to open a dialog that allows you to select a line, providing exact control over the positioning of the lead-in and lead-out.

Lead-In/Out Settings

These three options provide greater control over the tool.

Flip

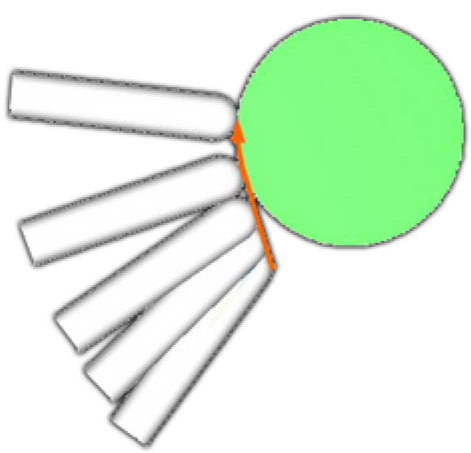
This option is available for Lead-In/Out types that are arcs. Selecting this option will flip the arc as if it were being viewed in a mirror.

Tool axis orientation

This setting defines how the tool will be orientated during the approach from the macro moves. You may choose between `Tangential` and `Fixed`.

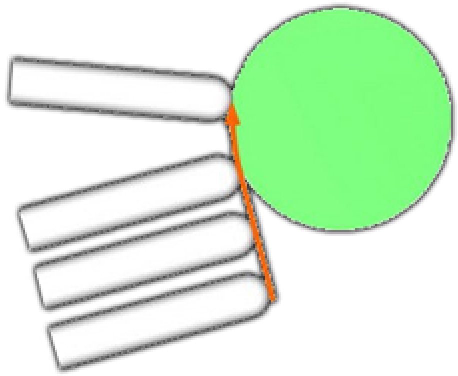
Tangential

During the entry or exit macro the tool will approximate the macro type, toolpath distance to the drive surface. This can be thought of as a virtual surface which has similar geometry to the drive surface. The result of this option is an even smoother crossover of the tool to the drive surface.



Fixed

With the fixed tool axis orientation the tool has the same orientation as in the endpoint of the macro or the first toolpath point on the drive surface.



Tilted

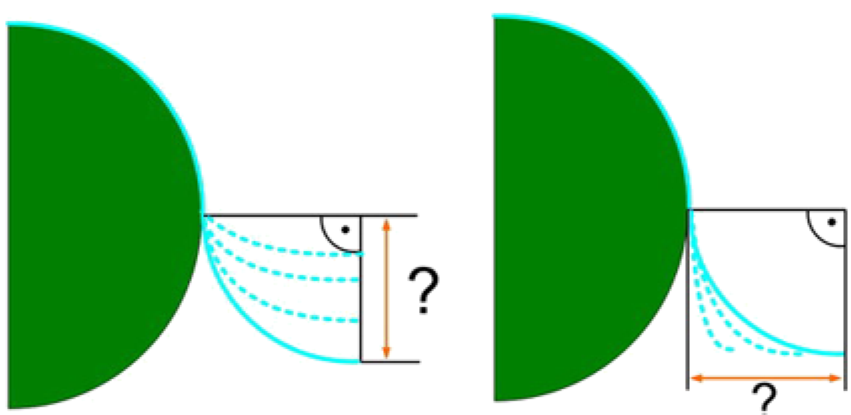
During the lead-in/out the tool axis will orient itself as defined in the Tool Axis will... setting (see Tool Axis will...) found on the Tool Axis Control tab.

Arc Parameters

When defining the arc to be used in a macro entry or exit you may use one of two possible options to define the size of the arc. The first option is to define the arc by an Arc Diameter and Arc Sweep angle. The other option is to use Width and Length values. Only one option at a time is possible, so if you have chosen one, the other option blanks out.

Width and Length

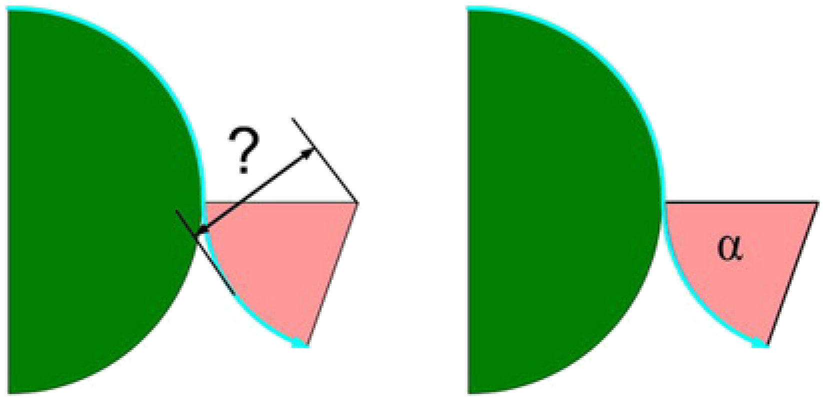
The width and length define a bounding rectangle to enclose a 90° arc. The arc will always have a 90° sweep and the arc is stretched or squeezed based on the width and length value.



Width	Length

Arc Sweep and Arc Diameter

This option lets you create an arc on or off of the part by defining its size. Arc Diameter describes the radius of the tangential move as a percentage based on the tool diameter. For example, a tool diameter of 10mm and a setting of 200% in the Arc Diameter results in the arc diameter being 20 mm. Arc Sweep describes the angle of the arc segment defining the arc move.



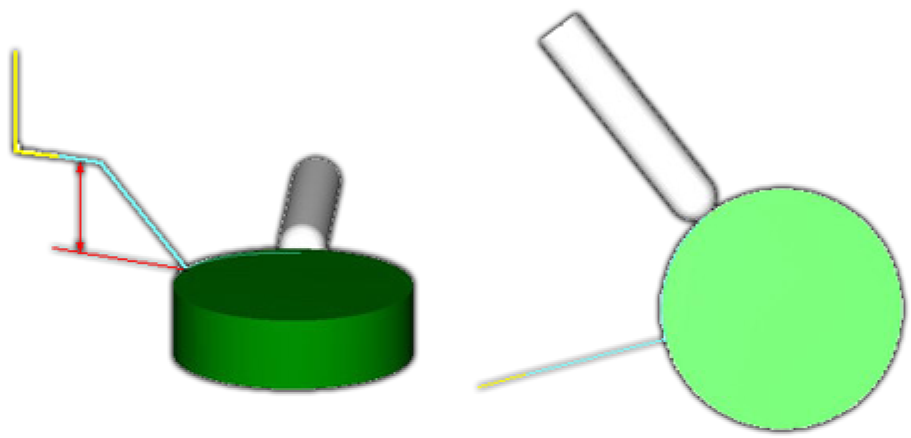
Arc Diameter Arc Sweep

Length

When a line is being used for the macro, the only parameter to be concerned with is the Length. This value describes the length of the line going to or coming from the toolpath.

Height

This parameter defines the incremental height of the macro move. In the case of using a line, the height value helps to build a ramp. In the case of an arc move, the height value helps to build a spiral.



Feedrate %

You can adjust this parameter to a value smaller than or larger than 100% if you want to use a slower or faster feedrate for this lead-in or lead-out.

Roughing tab

The options presented by the **Roughing** page let you define how to rough your part.

- For calculation based on Surfaces, Wireframe, or Geodesic, or for 5-Axis Toolpath Conversion, this includes what is to be cut (see Stock Definition), how many cuts the tool should take (see "Multi Passes" on page 271), whether the tool should plunge rough the part (see "Plunge" on page 272), how pockets are to be handled (see "Morph Pocket" on page 273), how deep each roughing pass should be (see "Depth Cuts" on page 274), how to rough an impeller floor (see "Area Roughing" on page 276), how to duplicate and rotate the toolpath (see "Transform/Rotate" on page 283), how to reflect the toolpath (see "Mirror" on page 284), and options for sorting the passes (see "Sorting options" on page 285).
- For calculation based on Triangle Mesh patterns Rough, Constant Z, or Projection, this can include what is to be cut, how to duplicate/rotate/reflect the toolpath, what type of approach to use, and various advanced options. See "Roughing Parameters for Triangle Mesh" on page 286.

Stock Definition

When you select the Stock definition option in the Roughing tab, the system will consider the defined stock as the material condition at the start of each operation. If multiple cuts are calculated for the operation the stock definition will allow the system to eliminate air cuts. All of the toolpath segments that are outside of the stock definition will be filtered out.

Various parameters are used to maximize this option's potential (see "Stock Definition Parameters" on page 267 or, for Triangle Mesh, "Stock Definition Dialog" on page 286). If Stock Definition is off, the system ignores all stock conditions and only generates toolpath based on face selection.

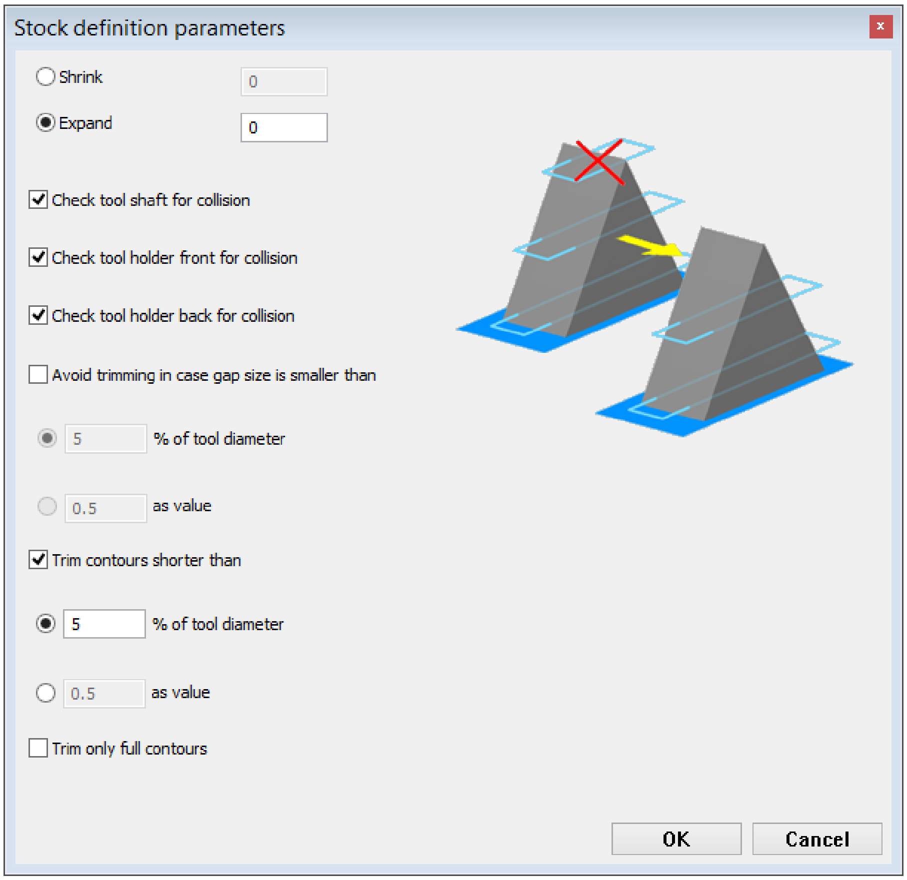
Note that the stock might be defined in several ways (such as from a set of surfaces, workgroup geometry, a bounding box, a 2D containment, or a default stock box). In all cases, the stock definition is considered to be a "shell", meaning that the surfaces define the outer layer of the stock definition, as compared to a solid stock definition.

Stock Definition Parameters

In the Roughing tab, when the Stock definition checkbox is selected, the Stock definition parameters button appears (if calculation is based on Surfaces or Wireframe). Clicking this button opens the **Stock Definition Parameters** dialog.

Items in this dialog let you set the stock tolerance and what parts of the tool will be used for trimming toolpath. Any motion where the tool, holder front, or holder back touches the stock will be kept as valid toolpath. When the tool, holder front, or holder back(as selected here) are not touching the stock, the system assumes that it is "cutting air" and the toolpath is trimmed. This can happen whenever the entire tool tip is outside of the stock material, or even if the entire tool tip is inside the stock material.

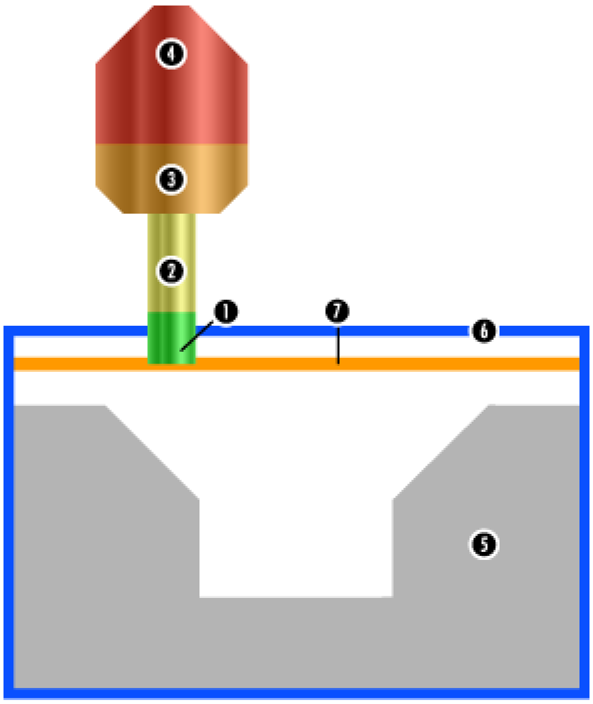
By default, the system only looks at the tool tip (the area up to the full radius) but you can also check the tool, holder front, or holder backby selecting one or more checkboxes for Check tool shaft for collision, Check holder front for collision, and Check holder back for collision.



When performing this calculation, the system looks at the stock condition for the current operation. That might be the stock set in the Document Control dialog, stock defined in a workgroup, a stock body, or stock that is locally selected for the operation from the Machining palette. Whatever selection method, that boundary or shell is used as the stock.

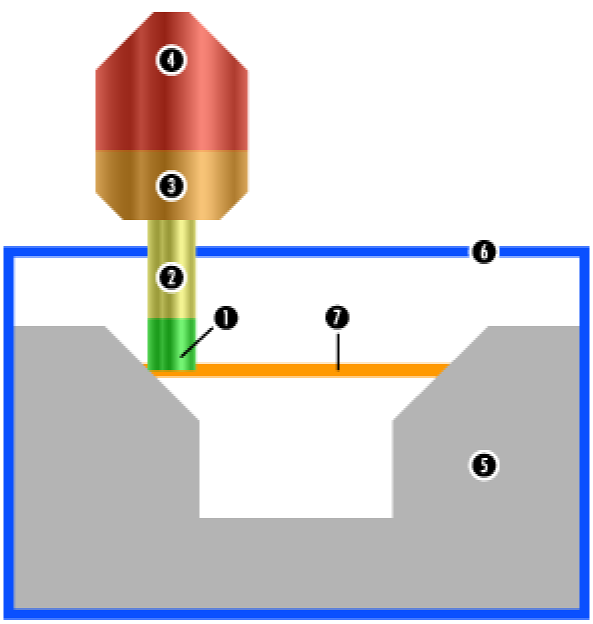
There are common cases where the tool tip is actually cutting material but the system fails to realize it, because the tool is deep inside a pocket and the only thing that intersects with the stock is a part of the holder front or holder back. Therefore, you may wish to use this option and control what parts of the shaft, holder front, and holder back are used when trimming the toolpath to avoid cutting air.

Note that one method to avoid having valid toolpath removed as air cuts would be to activate all three checking options (shaft, holder front, and holder back). But this will cause the system to take more time to calculate the toolpath. So it is recommended that you only activate the checks that are necessary for the toolpath you are creating.



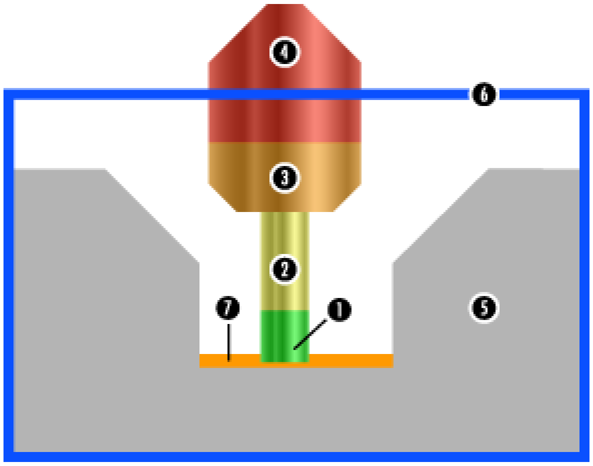
1. Tool tip
2. Tool shaft
3. Holder front
4. Holder back
5. Pocket
6. Stock “shell”
7. Toolpath

Example of valid roughing toolpath where the system would trim the toolpath due to cutting air if the Stock Definition option was not enabled



1. Tool tip
2. Tool shaft
3. Holder front
4. Holder back
5. Pocket
6. Stock “shell”
7. Toolpath

Example of valid roughing toolpath where the system would trim the toolpath due to cutting air if the Check tool shaft for collision option was not enabled



Example of where the system would trim the toolpath due to cutting air if the Check holder back for collision option was not enabled

Shrink / Expand

The stock definition tolerance is a value that defines the tolerance between the tool and the stock. In the past, this was a single value that could be positive, zero, or negative. (A negative value could be thought of as decreasing the stock size or increasing the tool size; a positive value value could be thought of as increasing the stock size or decreasing the tool size.) The idea behind this value is to compensate for the chord height / tolerance used in the tessellation of the stock. This value does not affect the final part, only the trimming of the roughing passes, and therefore overcutting or undercutting the part is of little concern. This tolerance is intended to help you minimize the amount of air cutting but give you control over how liberal or conservative the toolpath trimming will be.

At this release, the same idea is conceptualized as a value by which to shrink or expand the size of the stock. Regardless of which is chosen, a positive value (or `0`) is required:

- When a positive Shrink value is used, the effect is to shrink the stock (in all three dimensions). Another way to think about a Shrink value of `0.1` mm is to imagine that the tool is offset by 0.1 mm. The effect is to trim the toolpath so that more toolpath will stay in the material.

  Important: The value for Shrink must never be greater than half the tool radius.

- When a positive Expand value is used, the effect is to expand the stock (in all three dimensions). The effect is to trim the toolpath so that it will extend more outside the stock.

Check tool shaft for collision

If this checkbox is selected, then all motions where the tool shaft is touching (or intersecting) the stock definition are kept in the toolpath.

Check holder front for collision

If this checkbox is selected, then all motions where the holder front is touching the stock definition are kept in the toolpath.

Check holder back for collision

If this checkbox is selected, then all motions where the holder back is touching the stock definition are kept in the toolpath.

Avoid trimming in case gap size is smaller than

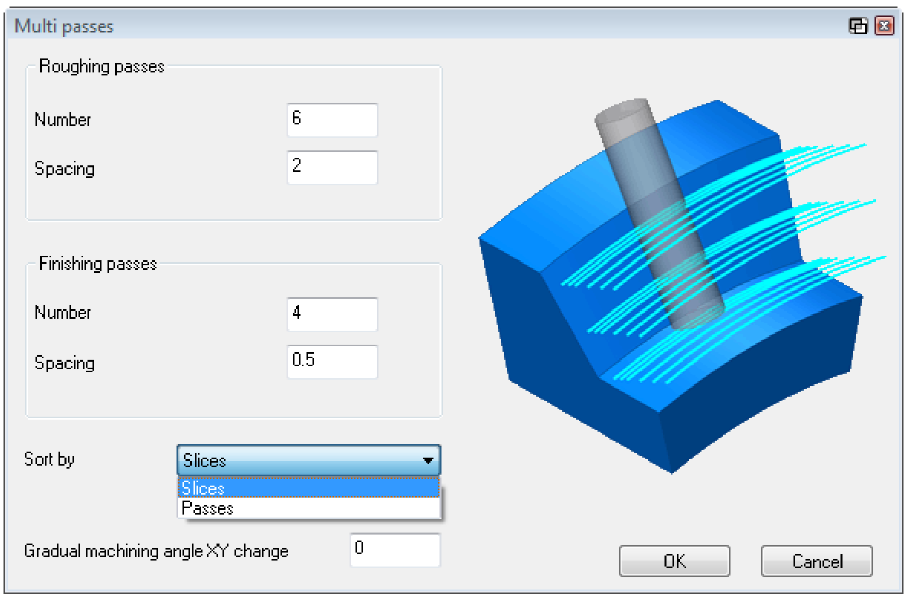
If this checkbox is selected, then you can specify a relative or absolute amount for the largest gap size where trimming will not occur. Toolpath trimming occurs only when the stock gap encountered is above a certain size.

Trim contours shorter than

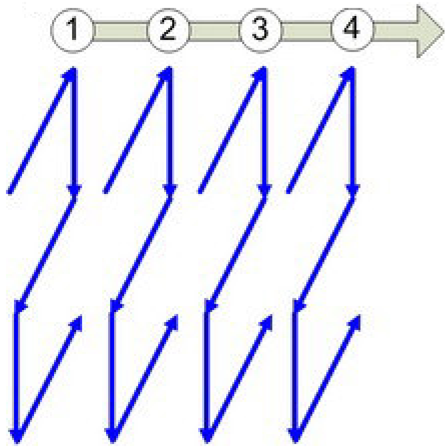
If this checkbox is selected, then all motions where the holder is touching the stock geometry are kept in the toolpath, within a relative or absolute value that you specify here.

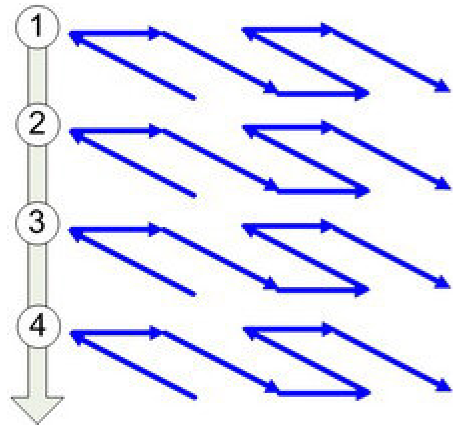
Multi Passes

When you select the Multi passes option in the Roughing tab, the system will let you create multiple toolpath passes that are offset in the direction of the surface normal, regardless of tool axis orientation. The shape of the toolpath does not change. Roughing passes are located above the finishing passes. Activate Roughing passes to remove larger amounts of material. When closer to the final surface, activate the Finishing passes option to make smaller cuts. Number defines how many cuts will be made and Spacing is the distance between the cuts.



The Sort by menu lets you choose to machine in slices using the Slices selection, or by layers using the Passes selection.

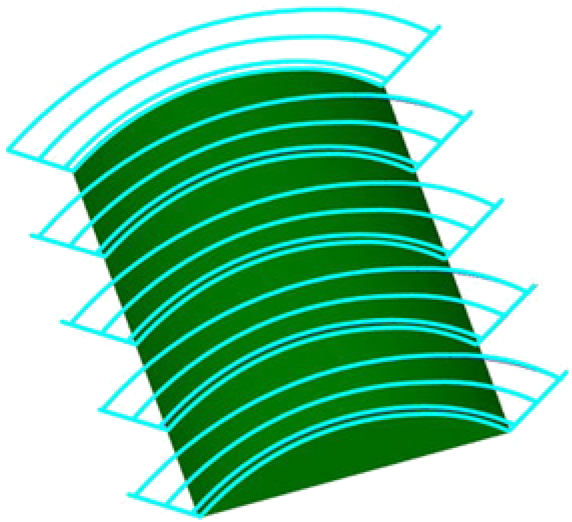


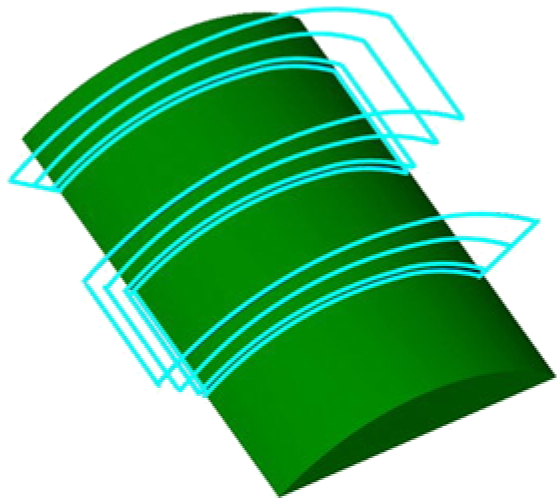


Sort By Slices

Sort By Passes

In the following images we see a comparison of machining sorted by slices versus passes. There are three roughing slices or passes with 5 mm spacing and one finishing pass with 1 mm spacing.





Sort By Slices

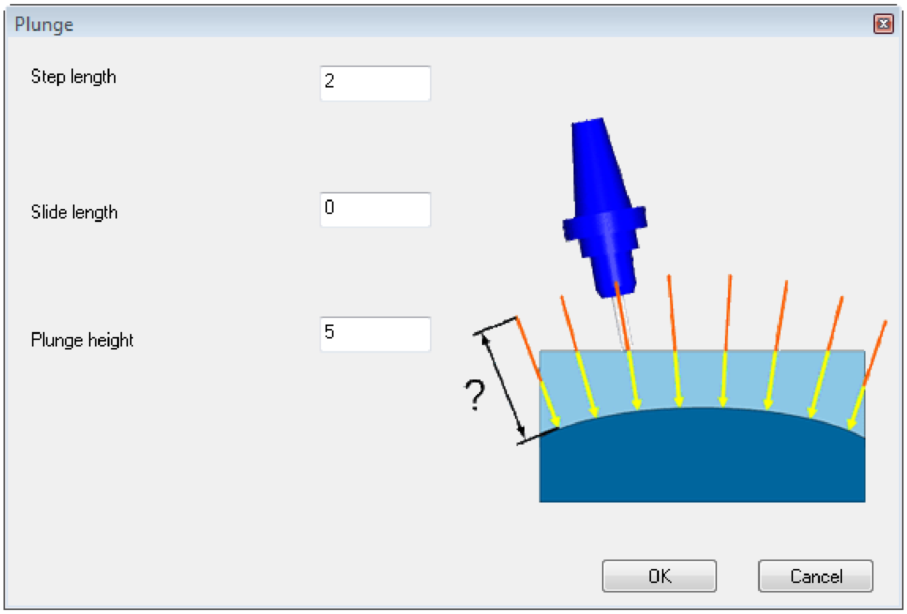
Sort By Passes

Gradual machining angle XY change

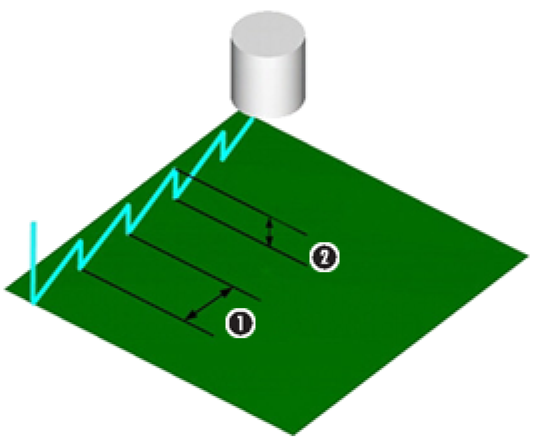
This option lets the toolpath rotate with every new pass with the set angle value. This is available with calculation based on `Surfaces`, pattern `Parallel cuts`, when the `Machining angle in Z` is not equal to zero.

Plunge

The Plunge option lets the tool plunge to the drive surface along its tool axis to rough the part.



The Step Length value describes the step-over distance between plunge moves. The Plunge height is the distance above the surface where the plunge should begin.

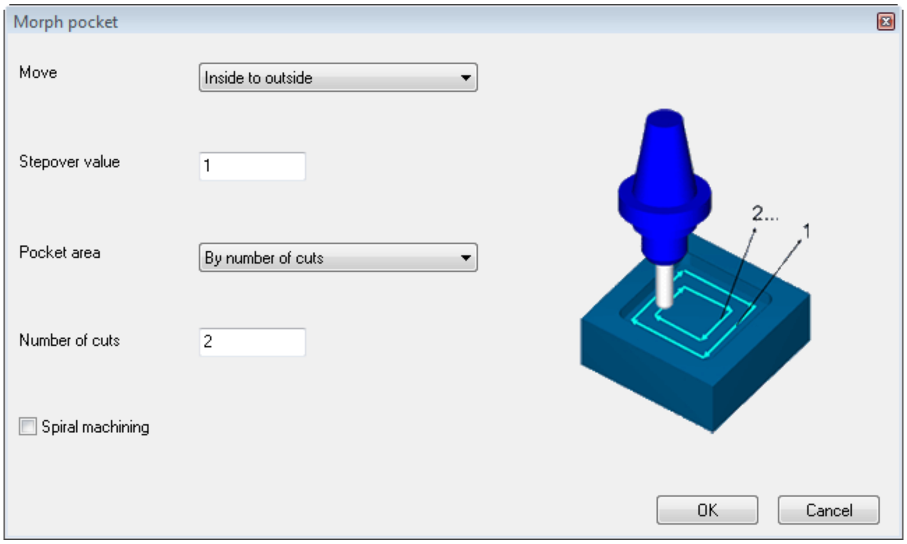


1. Length
2. Height

Slide Length and Retract Angle are not used currently and are planned to be used in a future release.

Morph Pocket

The Morph pocket option lets you generate toolpath for simple pockets. For this option it is important that you work on surfaces that define a closed pocket, so you must select the faces that define the edge of the pocket, not only the face that is the floor of the pocket.



Move
This option sets the machining direction. It can be from Outside to inside or from Inside to outside.

Stepover value
This option sets the maximum distance between two cuts.

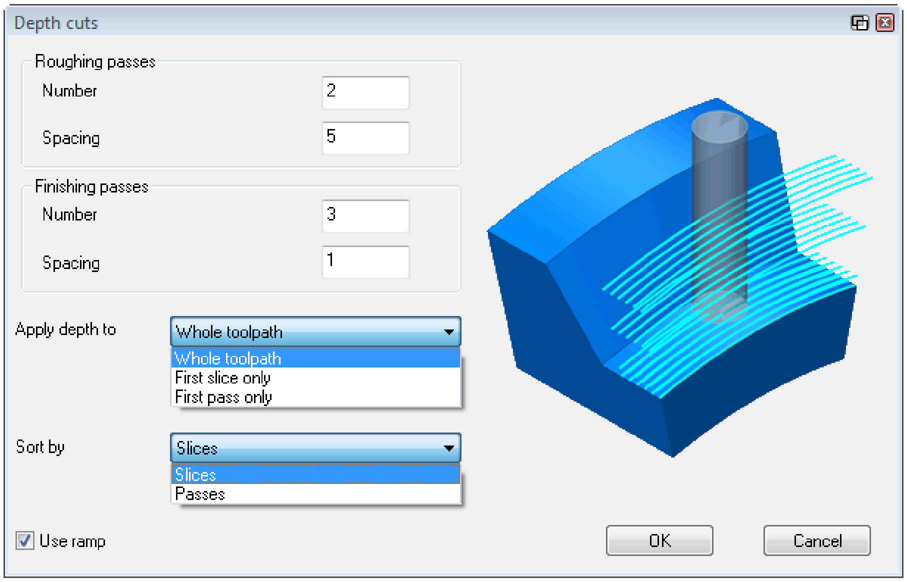
Pocket area
This option defines whether you want to machine the whole pocket using the Full option, or if you want to stop machining after a certain number of cuts using the By number of cuts option.

Number of cuts
This parameter sets the number of roughing cuts for a morph pocket. If this parameter is used, then it is probable that the whole pocket will not be machined.

Spiral Machining
When you activate spiral machining, the tool movement changes from parallel cuts to a spiral machining toolpath.

Depth Cuts

Depth cuts are similar to Multi Passes (see Multi Passes). Multi Passes are always offset in the direction of the surface normal, regardless of tool axis orientation. In contrast, the multiple passes generated by the Depth Cuts function will always be offset relative to the tool axis orientation. The roughing passes are located above the finishing passes. Activate the Roughing passes option to remove larger amounts of material. When closer to the final surface, activate the Finishing passes option to make smaller cuts. Number defines how many cuts will be made and Spacing is the distance between the cuts.



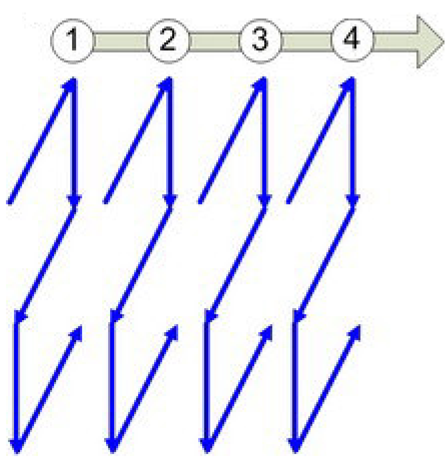
A combination of Multi passes and Depth cuts will create a mixture of both types of cuts. Hierarchically, the depth cuts come first. Each single depth cut is associated with a complete set of defined multi passes. For example, if you specify `10` depth cuts and `10` multi passes, the result will be `10 * 10 = 100` cuts. The feed from one pass to another pass is done in the direction of the tool side tilt angle.

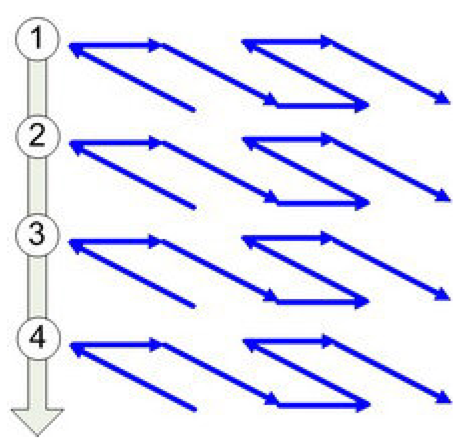
Apply Depth to

The Apply Depth to menu lets you specify how the depth cuts are applied: to the Whole toolpath, to the First slice only, or to the First pass only.

Sort by

The Sort by menu lets you choose to machine in slices using the Slices selection, or by layers using the Passes selection.



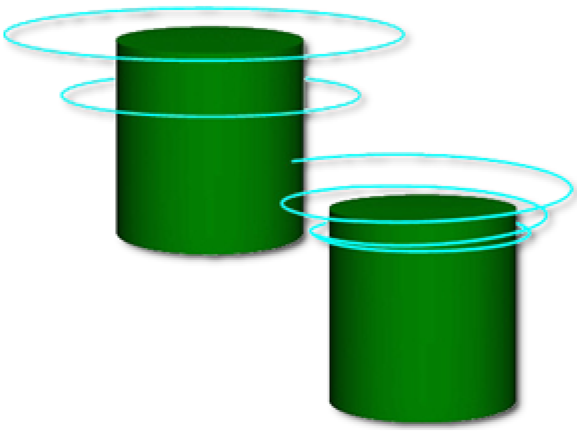


Sort By Slices

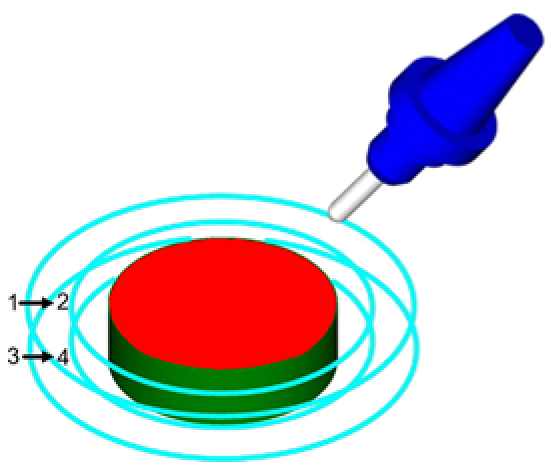
Sort By Passes

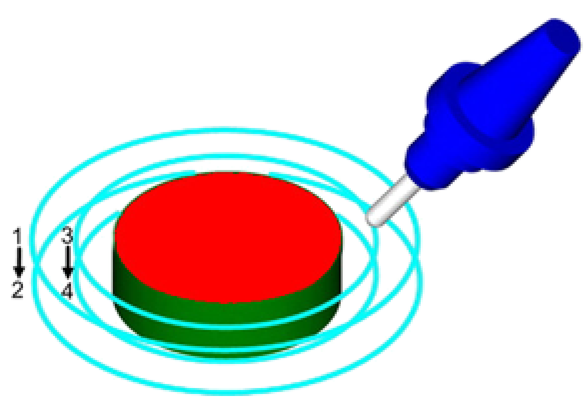
Use ramp

This option changes the different roughing and finishing slices into one spiral slice. The order is starting from the roughing passes to the finishing passes. The tool starts and stops on the same position as without the ramp option.



Example
The following pictures show machining sorted by passes and by slices. With the Sort by passes version you can see that the tool moves on the same level against the drive surface. With the Sort by slices version you can see that the tool slices against the drive surface step by step. See the file `Depth Cuts.vnc` for this example.

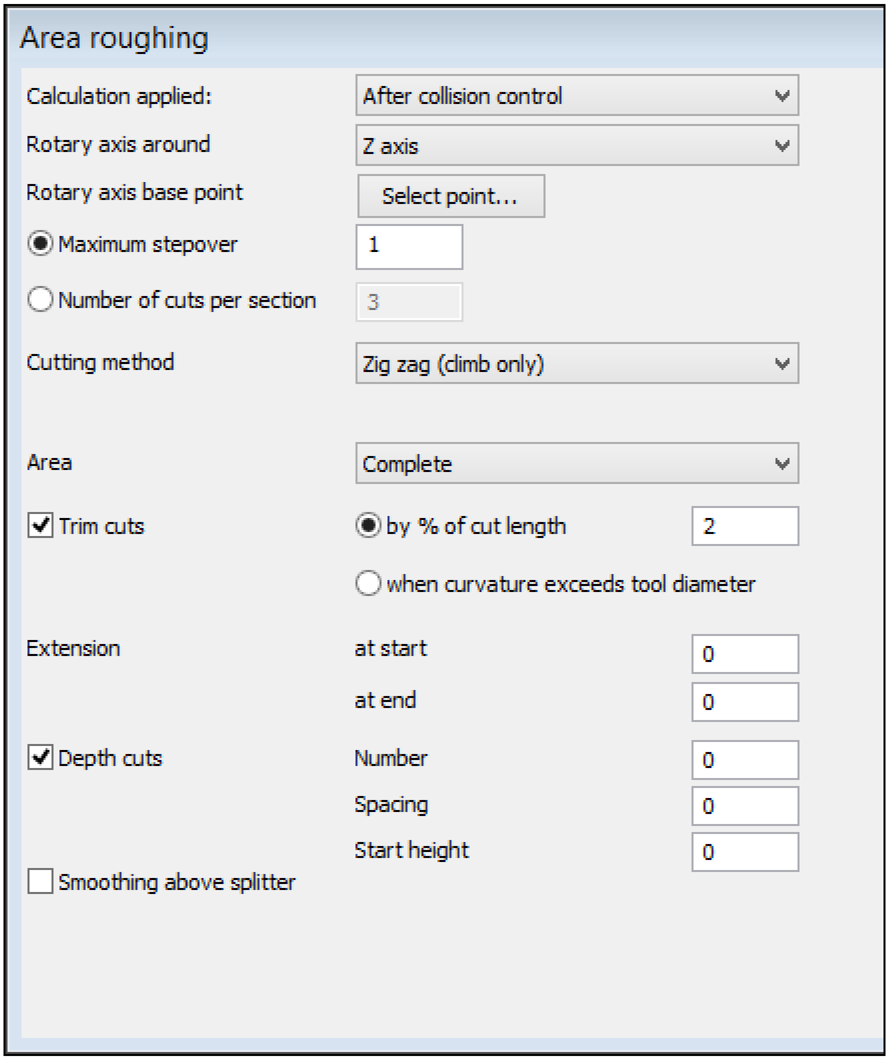




Sort By Passes	Sort By Slices

Area Roughing

This function is for creating morphed toolpath for an impeller hub. It can be used to create roughing procedures as well as floor finishing procedures and it can be used to machine the impeller with our without a splitter. The results of this function are similar to Morph between two surfaces, but with this function you have the ability to define a splitter blade that the tool can work around.



- When you work with margins, the tool must be a ball endmill and, in the Utility tab, the Calc based on tool center checkbox must be selected.
- When you work with margins, the value should be equal to or bigger than the tool radius. A smaller value will destroy the faces, except when Gouge Protection is active.

Calculation Applied

This option controls whether the roughing passes that are calculated use the tool axis control from the base toolpath (After collision control) or whether each rough pass calculates its own tool axis and collision control (Before tilting).

Rotary axis around

Select the axis the impeller is rotating around, either X, Y, Z, or a User defined axis.

Rotary axis base point
This item defines the position of the axis. For example when using the Z axis, you must set an X and Y value to position the rotation axis.

Maximum step over
This item defines the maximum distance between two cuts. The number of cuts is modified to achieve the desired stepover. This distance can be smaller than the set value but will never be larger. Instead of this you may choose to specify a number of cuts using the Number of cuts per section option.

Number of cuts per section
This option specifies how many cuts are desired per section. The stepover is modified to accommodate the number specified. Instead of this you may choose to specify a stepover amount using the Maximum step over option.

Cutting method
The items in this menu help you define the connection to the next cut between the blades.

One way (along rotary axis)
With this option the machining starts at the lower edge of the impeller floor face, continues along the blades and stops at the upper edge of the floor. Then it retracts to the beginning and starts over with the next cut.

One way (along reverse rotary axis)
With this option the machining starts at the upper edge of the impeller floor face, continues along the blades and stops at the lower edge of the floor. Then it retracts to the beginning and starts over with the next cut.

Zig zag
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, as defined when looking from outside the impeller towards the impeller axis of rotation.

Zig zag (climb only)
With this option, the machining begins in the center of the surface and progresses outward for each side. If you are using the Number of cuts per section Cutting method, then you can also set the Alternate direction to reduce path length option.

Alternate direction to reduce path length
This option is available only when Number of cuts per section is in effect and the Cutting method is set to Zig zag (climb only). When this checkbox is selected, the toolpath will not climb-cut every pass, but instead will alternate between climb and conventional.

Area
The items in this menu help you define the area around the splitter blade that is to be machined.

Complete
This option will machine the whole area between the two main blades.

Left

This option will machine only between the left main blade and the splitter blade, as defined when looking from outside the impeller towards the impeller axis of rotation.

Right

This option will machine only between the right main blade and the splitter blade, as defined when looking from outside the impeller towards the impeller axis of rotation.

Trim cuts

This defines whether the trimming is set by a percentage of the contours or whether the trimming starts when the curvature of the contour exceeds the tool diameter.

Extension

This is used to specify an extension of the toolpath at the start and/or end of the default path. This is activated by setting Calculation Applied to After collision control.

Depth Cuts

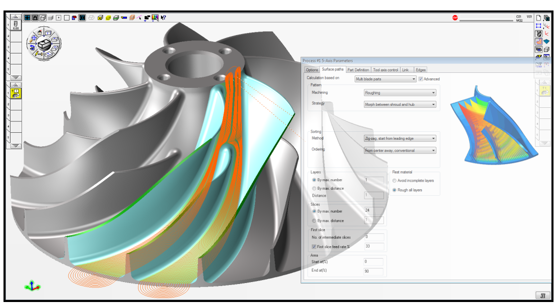
With this option applied, the toolpath pattern will be copied in the tool contact direction. The aim is to get all material out of the impeller hub. Once a collision free pattern is generated it can be used for depth cuts providing all following upper cuts to be collision free. Set the total number of cuts, the distance between cuts and a start distance above the original toolpath. This is activated by setting Calculation Applied to After collision control.

Smoothing above splitter

This option will attempt to smooth the toolpath above the splitter. This is activated by setting Calculation Applied to After collision control.

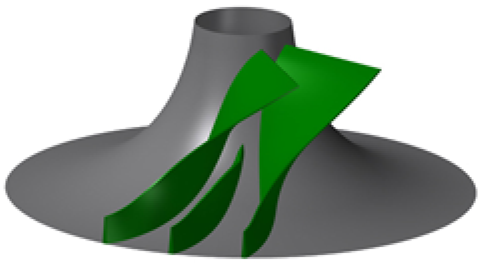
Sample: Using Area Roughing

5-Axis Impeller Machining Options Compared to 5-Axis MultiBlade



Although the base 5-Axis product includes options for impeller machining, the preferred solution is the 5-Axis MultiBlade product option (or 5-Axis MultiBlade Level 2 for finest control of all aspects of impeller machining). Because MultiBlade is only for impellers and blisks, it automatically detects and leverages radial symmetry, accommodates blades and splitters of any curvature, and provides options and controls that are specific to impellers, such as special settings for leading and trailing edges.

The area roughing is built up on an existing toolpath strategy. So first thing to do is to create a swarf toolpath around the faces. We will focus on only one set of surfaces.



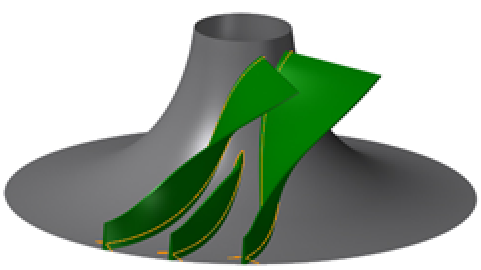
1. The toolpath pattern morph Parallel to surface is being used. The lead surface (Single edge) in this case is the (grey) floor face.

2. The Area roughing function needs an impeller segment of at least two blades defined as drive surfaces. If the impeller has no splitter, then the two given blade faces must be selected. If the impeller comes with a splitter face, then the two blades faces and also the splitter faces (in this example all green faces) must be selected.

    An impeller must be composed from two faces that make a closed contour.

3. The number of cuts depends on the cutting area. For this strategy it is necessary to have only one cut. Set the cutting area type to Determined by number of cuts and, in field Number of cuts, enter 1.

4. In the Tool axis control tab, specify 90 degrees for the value of Tilt angle at side of the cutting direction. This will make the tool parallel to the drive surface. In this case the tool will swarf. The resulting toolpath will look like this.

    Since the tool tilting is normal to the drive faces, the tool is gouging the impeller and sometimes the floor surface.

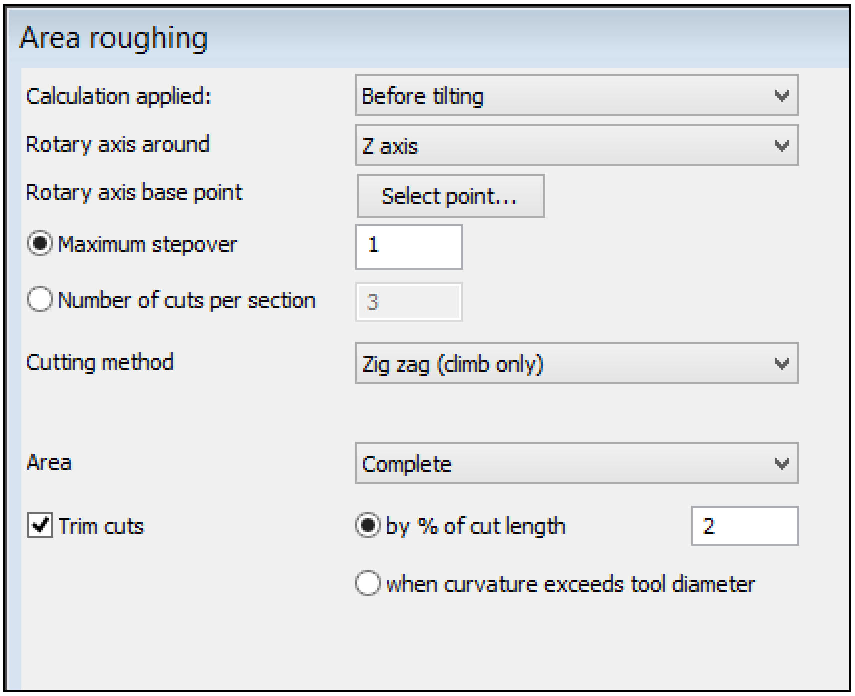
5. In the Gouge check tab, a collision control set must be activated. There are two strategies that must be activated:

    - Tilting tool away with max angle and Use side tilt angle, with the Drive surfaces checkbox selected. This will avoid collisions with the impellers.

    - Retracting tool along tool axis with the Check surfaces #*n* checkbox selected. Select the (grey) floor face (the same selected as the Single edge). This will assure that the tool will not collide with the floor.

The resulting toolpath by now should be gouge free and swarfing. With this toolpath information the system can define the floor contour between the blades, where it starts and where it ends.

Now click the Roughing tab, select the Area Roughing checkbox, and click Area Roughing to set the following parameters:



1. For Rotary axis around, choose either X, Y, Z, or User defined axis. In the case of a user-defined axis, you can either pick the axis from the geometry or you can specify X, Y, Z vectors.
2. If the rotation axis is shifted in one direction, you must select a point for Rotary axis base point.
3. If you define a value for Maximum stepover, it sets the maximum distance between two cuts. This distance in the actual toolpath could be smaller than the set value but never larger. Or, instead of specifying a maximum stepover, you could specify a value for Number of cuts per section. These cuts will be spread over the floor face with same stepover.
4. For Area to be machined, choose one of the following:

Complete	the whole area between the two main blades
Left side	only between the left main blade and the splitter blade
Right side	only between the right main blade and the splitter blade

5. The Cutting method defines the connection between the cuts. The options are:

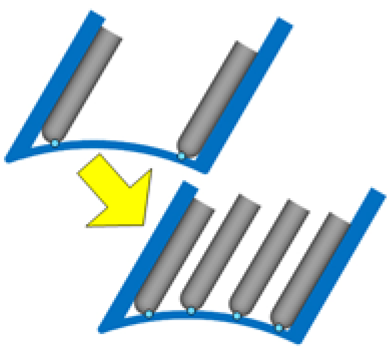
One way (along rotary axis)	The machining starts at the upper edge of the impeller floor face, continues along the blades and stops at the lower edge of the floor. Then it retracts to the beginning and starts over with the next cut.

One way (along reverse rotary axis)	The machining starts at the lower edge of the impeller floor face, continues along the blades and stops at the upper edge of the floor. Then it retracts to the beginning and starts over with the next cut.
Zig zag	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.
Zig zag (climb only)	The machining begins in the center of the surface and progresses outwards for each side.

6. For Calculation applied, choose one of the following:

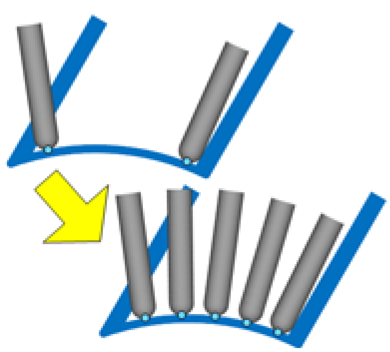
After collision control

In this case the collision check will be applied before the area roughing morphs the toolpath. If the calculation is applied after the collision control it is possible to extend the toolpath at the start and the end.

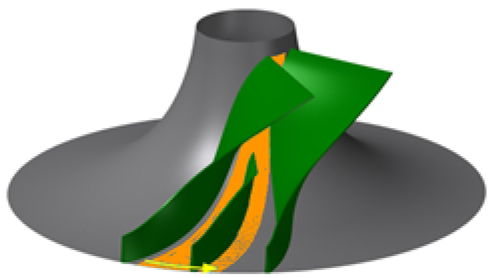


Before tilting

In this case the toolpath will be morphed over the floor face before the collision control is applied. The collision checking happens afterwards.



The resulting toolpath should look like this:



Transform/Rotate

This option allows the automatic generation of duplicate rotated copies of the toolpath for the operation. A rotation axis and position is specified along with an initial rotation, rotation angle and number of steps. Additionally, several options for sorting the toolpath are available. All toolpath rotations will always generate long hand code when post processed.

Sort by

This defines how the toolpath will be rotated.

- Complete toolpath - The complete toolpath section will be rotated in one piece.
- Passes - The rotation will be applied in layers. The order for two rotations would be: 1st layer of 1st toolpath, 1st layer of 2nd toolpath. Then 2nd layer of 1st toolpath, 2nd layer of 2nd toolpath and so on.
- Slices - The rotation will be applied in slices. The order for two rotations would be: 1st slice of 1st toolpath, 1st slice of 2nd toolpath. Then 2nd slice of 1st toolpath, 2nd slice of 2nd toolpath and so on.
- Partial toolpath - Only a percentage of the toolpath will be rotated. In this case a value must be specified in the field Perc. of whole toolpath.

Apply linking

Linking can be set to Before rotation or After rotation. If the linking is applied before the rotation then all rotated toolpaths have same linking segments as the initial toolpath. If the linking is applied after the rotation then the linking of the rotated toolpaths will be collision checked and eventually taken out. So, this option works only with activated collision control.

Apply stock

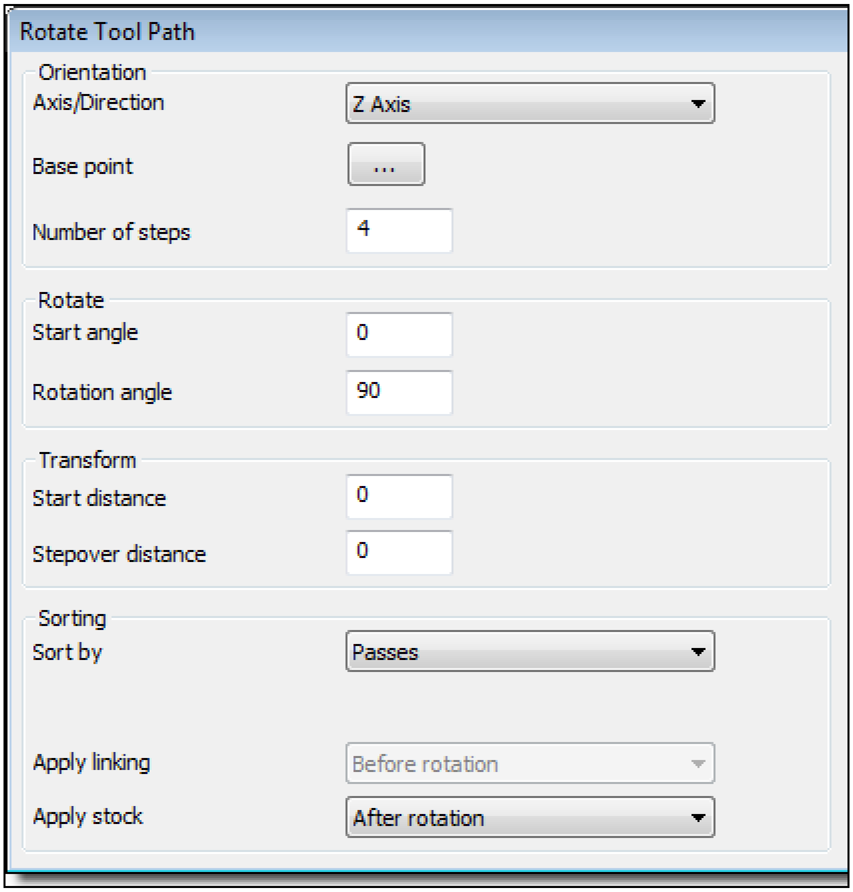
Stock may be applied to the calculation Before rotation or After rotation. If the stock is applied before the rotation then all rotated toolpaths will have the same stock definition as the initial toolpath. If the stock is applied after the rotation then the stock definition will be checked for every rotated toolpath.

How it works

Each item in this dialog should be set. Following is an example.

1. A rotation axis must be selected. Available are X, Y, Z and User defined axis directions. When selecting the User defined axis, the axis can be picked from the geometry or can be set by vectors in X, Y and Z.
2. Set a rotation axis base point if the rotation axis is shifted in one direction.

3. Set the number of steps (meaning how often should the toolpath be copied). If “1” is input then the existing toolpath will be moved.
4. Set the start angle. This angle is the position of the first rotated toolpath.
5. Set the rotation angle. The rotation angle is the increment between two rotated toolpaths.
6. Chose the sort by method. This defines whether the toolpath will be rotated.
7. Set Apply linking. The linking can be applied before or after rotation.
8. Apply stock. The stock can be applied before or after rotation.



Mirror

This option lets you reflect toolpath, generating a mirror-reversed copy of it. Click the Mirror button to open the **Toolpath Mirroring** dialog, which offers the following types of settings.

Axis/Direction

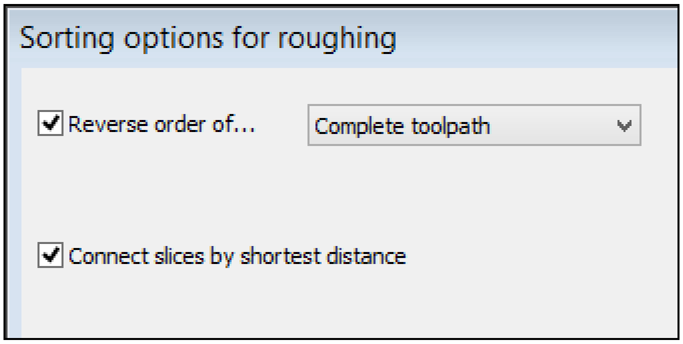
Choose from X-axis, Y-axis, Z-axis, or User defined axis. In the case of a user-defined axis, you can either pick the axis from the geometry or you can specify X, Y, Z vectors.

Base Point

You can specify a rotation axis base point if the rotation axis is shifted in one direction.

Sorting options

These options let you change the order and direction of the default roughing.



Reverse order of

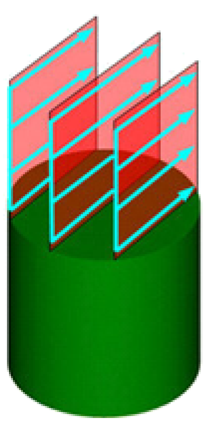
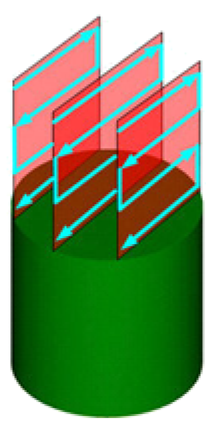
This option reverses the order of the cuts for depth cuts or multi-passes.

- Passes - by default the machining starts with the slice or pass which is considered as the last or the cut with the biggest distance to the machining surface. Setting the option to Passes simply swaps the cut order. Now the cut which is next to the drive face is the first cut.

- Complete toolpath - this options swaps the cut order of the set of passes and slices as well as the order of the single cuts with in one pass or slice. For example, if the default machining would start from the fifth pass moving from left to right, the new order with this option activated now starts from the first layer moving from the right to the left.

Connect Slices By Shortest Distance

This option works with the Multi Passes and Depth Cuts functions. With both of these functions you have the choice between Sort by slices and Sort by passes. If Sort by slices is active you will see that, even when you are using a Zig Zag motion (see Cutting Method), the machining within one slice is one way machining. The machining direction doesn't change until the next slice. That means that the tool moves long distances without cutting. If you activate Connect slices by shortest distance the machining will use the shortest distance to the next cut. The result is Zig Zag machining within one slice.

The default behavior of a Zig	A Zig Zag slice with

Zag slice toolpath	“Connect slices by shortest distance” activated

Roughing Parameters for Triangle Mesh

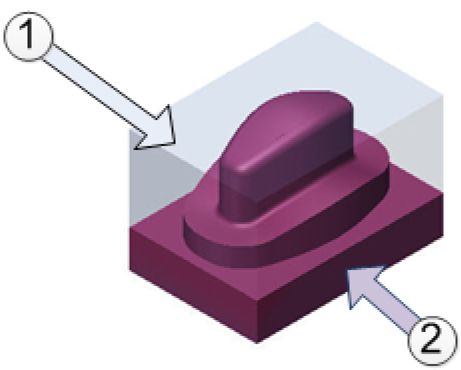
With calculation based on Triangle Mesh, for various patterns, the Roughing tab offers various settings:

- (pattern=Rough) Stock definition option: see Stock Definition Dialog below
- (pattern=Rough or Constant Z) Advanced parameters: see “Advanced Dialog (Options for Roughing)” on page 287
- (pattern=Rough) Approach move parameters: see “Approach moves” on page 289
- (pattern=Projection) Multi passes option: see “Multi Passes” on page 271

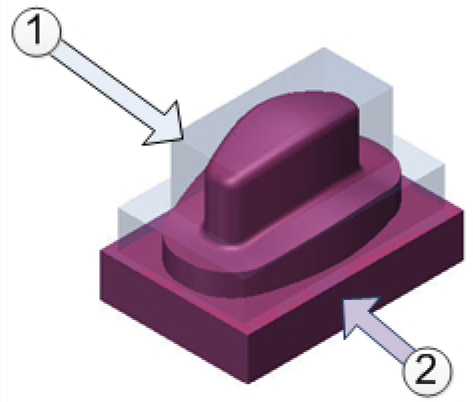
Stock Definition Dialog

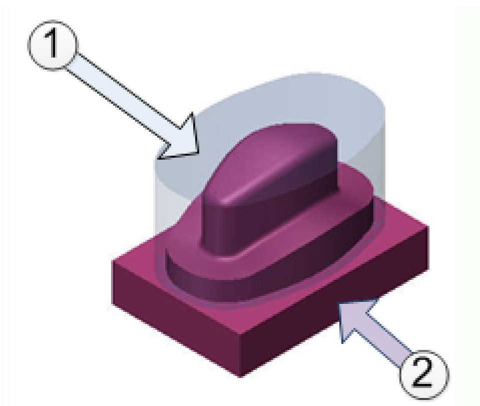
When you click the Stock definition button, the Stock definition dialog appears. Items in this dialog allow you to define the stock and to specify a value to offset the stock model either inward (Shrink) or outward (Expand). Offsetting is done in all three dimensions.

The Stock type pull-down menu offers three choices for defining the shape, size, and orientation of the raw material:



Bounding box:
To use a rectangular bounding box (1) around the machining surfaces (2). Use this option if the raw material to be machined is a rectangular block.

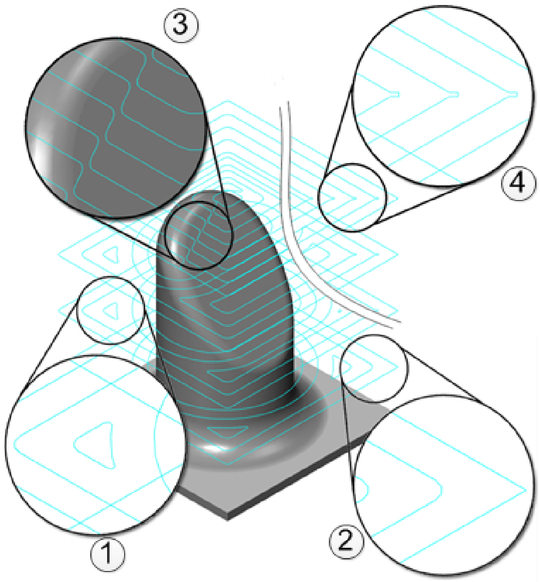
<table>
<tr><td>
</td><td>Surfaces:
To use an STL model or user-selected surface (1) to define the raw material. Use this option if machining a pre-cast or pre-machined stock.

With Surfaces, you can also optionally specify whether or not the Stock has undercuts. When this checkbox is cleared (default setting), toolpath is calculated for the entirety of the stock, possibly resulting in many air moves. When this checkbox is selected, stock slices identify pre-machined and undercut regions and toolpath is computed accordingly, often resulting in time savings.</td></tr>
<tr><td>
</td><td>2D containment:
Lets you specify one or more 2D curves to define the stock (1). This option can be used to machine a limited area or an insert.</td></tr>
</table>

Advanced Dialog (Options for Roughing)

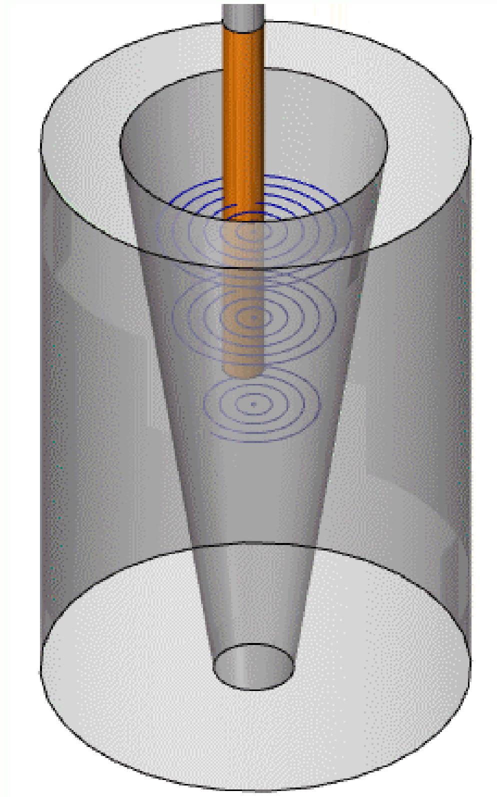
Depending on the Triangle Mesh pattern, the Advanced dialog offers various controls, such as Smoothing, Remove corner pegs, Filtering, and Approach Moves.

Smoothing and Remove corner pegs:
These controls are available only for Triangle Mesh pattern Rough. The following illustration shows the differences amongst the three Smoothing choices and the effect of the Remove corner pegs option.

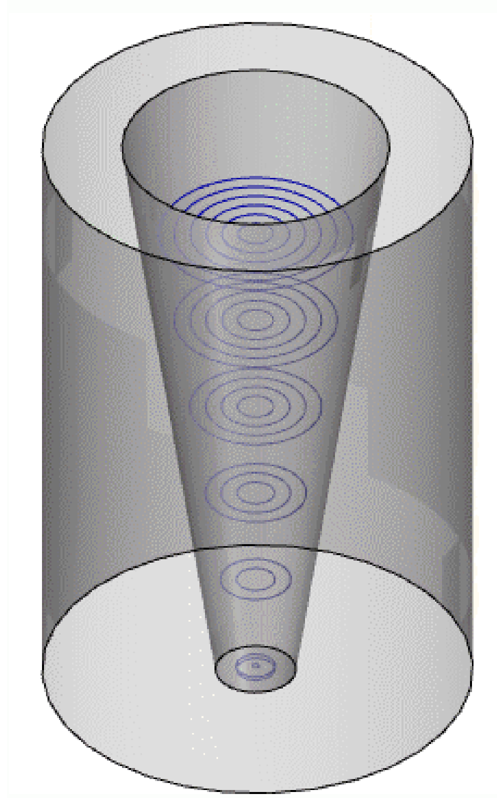


1. Smooth corners: Creates fillets in the sharp corners of the toolpath. Specify the fillet radius as a percentage of the stepover distance. Note that the fillet is not applied to the outer contour (compare Smooth final contour).
2. Smooth final counter: Creates fillets in the sharp corners of the outer contour. Specify the fillet radius as a percentage of the stepover distance.
3. Smooth links: Trims the last segments of the previous contour and the first segments of the next contour so as to smooth the links within a group. The links will connect diagonally or, in case of a blend spline linking, the connection would be an S-type link.
4. Remove corner pegs: With a stepover of greater than 50%, small pegs of material can sometimes be left in the corners. When this option is selected, an extra movement is added to corners, removing the rest material.

Filtering:
Filtering can be used to remove small pockets and segments that do not need machining. The size of these segments must be defined as percentage of the tool diameter.



Without filtering: Toolpath includes small segments



With filtering: Small segments are filtered out

The illustration shows toolpath where radial cuts are created within a drafted tube. On the left, you see that the first slice of the each layer is a very small toolpath segment, almost just a point and not necessary to machine. On the right, with filtering set to 50% of tool diameter, the useless small segments are filtered out.

Approach moves

These controls are available only for Triangle Mesh pattern Rough.

Center Cutting Tool:
Select this checkbox if the tool is able to plunge into the material. Clear this checkbox if the tool should start from the side of the material (for example, tools with inserts are unable to cut in the center, so plunging and drilling would be impossible).

Ramp type:
You can choose from the following options for lead-in ramp move. In all cases, if the selected type fails (for example, if the ramp move causes a gouge), then another ramp option is used instead. If no ramp move is possible, then a plunge move is applied.

Automatic:
This option tries the following ramp options in sequence. In other words: Line is tried first; if it fails, then Helical is tried; if it fails then Zigzag is tried; if all fail, then Profile is used.

Line:
Tries to perform a lead-in move is along an angular line. Ramp angle defines the angle of the ramp move, compared to horizontal.

Helical:
Tries to use a helical entry into stock material; the tool engages the stock with helical interpolation. Ramp angle and Ramp length are required to define the helix.

Zigzag:
When the length of the ramp is too short for Line, a zigzag angular moves is tried. The Zag move is in opposite direction to Zig move and at same angle to horizontal. Ramp angle define the angle of the ramp move compared to horizontal. Ramp length defines the length of each Zig and Zag.

Profile:
The tool engages the stock following the contour of the part or the tool path profile. Ramp angle is required to define the profile move and the angle at which it engages the stock.

Ramp angle:
Defines the angle with which the tool enters the next slice or pass. If this is set to 90°, then all ramp types degenerate to straight vertical moves.

Ramp length:
Defines one of the following, depending on Ramp type:

- Line: Length of line
- Helical: Diameter of helix
- Zigzag: Length of segment. (Each zig and zag is a segment.)
- Profile: Length of profile

Multi passes

This option is available only for Triangle Mesh pattern Projection. For complete information, see “Multi Passes” on page 271.

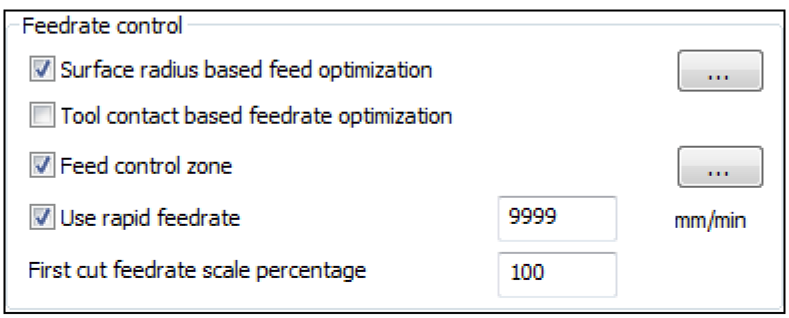
Utility tab

The **Utility** page presents several advanced controls to refine your toolpath. Not all controls are available for all calculation types and patterns; for example, when calculation is based on Triangle Mesh, the **Utility** page offers only one control under feedrate. For calculation based on Surface or Wireframe, the **Utility** controls are divided into three general areas, as follows.

- Feedrate Control
- "Axial Shift" on page 293
- Miscellaneous:
  - "Set Y Axis Machine Limits" on page 294
  - "Smooth Surface Normals" on page 294
  - "Calculation Based on Tool Center" on page 294

Feedrate Control

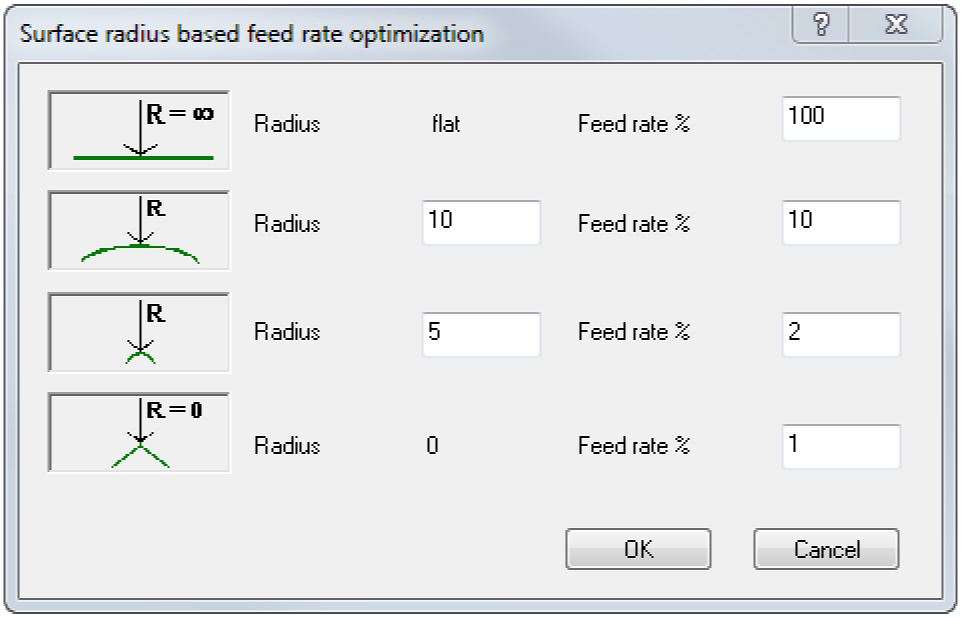
The controls presented in the Feedrate Control area of the Utility tab allow you to refine the operation's feedrates from the nominal values specified in the Options tab (see "Options tab" on page 38).



Not all controls are available for all calculation types and patterns; for example, when calculation is based on Triangle Mesh, the only available feedrate control is Use rapid feedrate.

Surface radius based feed optimization

Selecting the checkbox for Surface radius based feed optimization allows you to click [...] (the ellipsis button), opening a dialog box where you can specify feedrate parameters for amounts of surface curvature. The feedrate percent you specify for the first radius ("flat") is the feedrate to be used for all surface curvatures greater than the radius specified second. The feedrate percent you specify for last radius ("0") is the feedrate for all sharp corners of curvature smaller than the radius specified next to last.



Feed control zone

Selecting the checkbox for Feed control zone allows you to click [...] (the ellipsis button), opening a dialog box where you can select Geometry, specify an Offset value, and specify how much to decrease or increase Inside feedrate and Outside feedrate as a percentageof the base feedrate.

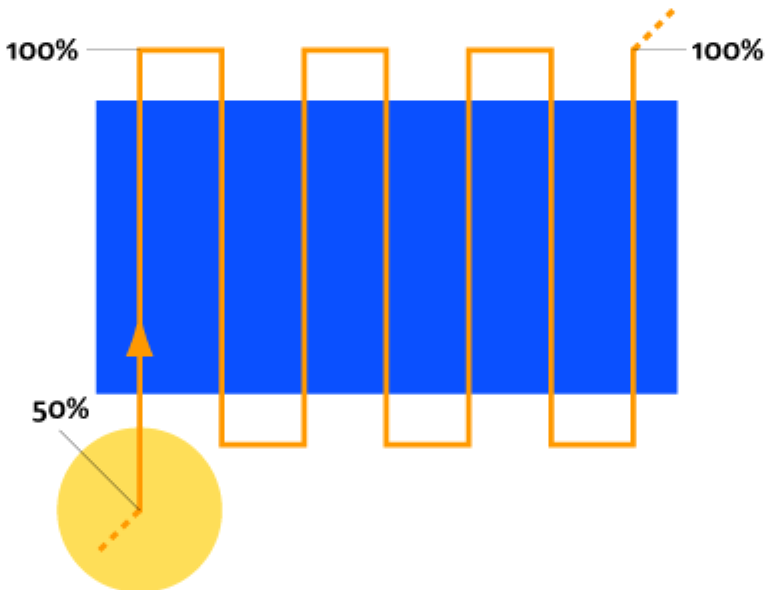
Use rapid feedrate

When the checkbox for Use rapid feedrate is selected, rapid moves will be output as a `G1` instead of a `G0`. This may help avoid potential collisions, since a high feed `G1` will be an interpolated move in all axes, unlike the `G0` move on most machines.

In the textbox that displays when this checkbox is selected, you can specify a feedrate for `G1` rapid moves.

First Cut Feedrate Percentage

This option lets you change the feedrate for the first cut in an operation. You set a percentage to adjust the feedrate. A value between 0 and 100% will decrease the feedrate. A value greater than 100% will increase the feedrate. This will help you account for the tool bearing a heavier load on the first cut. In the following image we see a tool that will be taking a heavy cut at first, followed by lighter cuts that are equal to the tool radius. To account for the first cut's heavier load on the tool, the feedrate is scaled to 50% of the rest of the operation.



Axial Shift

The Axial shift area of the Utilities tab lets you specify an *axial shift*: an offset to be added to the tool along its axis.

You can control how the axial shift affects the contact point between tool and workpiece:

- If you choose Constant for each contour, the axial shift distance is applied equally and is constant on each contour. In this case, you can enter a value (or select a point) only for To.
- If you choose Gradual for all cuts, the contact point shifts with every new cut, with the result that all of the tool's flutes will be used. In this case, you can enter values (or points) for both To and From.
- If you choose Gradual for each contour, the contact point slides gradually. In this case, you can enter values (or points) for both To and From.

To and From. You can supply values for To and From either by entering text in the text box or by clicking the [...] button and selecting an existing point in the part to load its absolute depth value. Positive values let the tool retract; negative values let the tool infeed.

Damp

The Damp checkbox lets you specify whether or not to damp out abrupt shift movements of the tool axis. When damping is in effect, virtual fillets are generated with fillet radius equal to twice the tool diameter. This helps to smooth the toolpath by preventing the tool from retracting along its axis at or near corners.

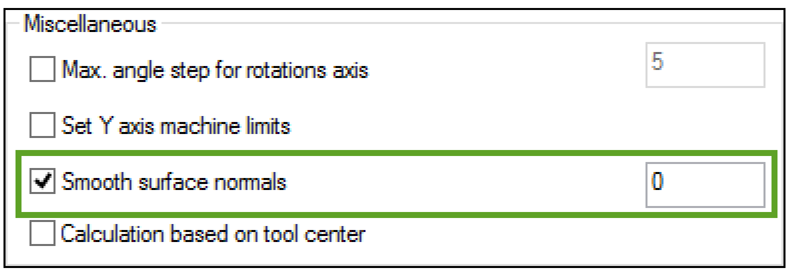
Set Y Axis Machine Limits

This is a special option to provide limits for the resulting Y values. If these limits are provided, then the tool is tilted such that the tool remains in these Y limits. This is a very special option to cut turbine blades that are wide and cannot normally be machined with a machine that has limited Y travel.

Smooth Surface Normals

When this checkbox is a selected, a function will apply a smoothing filter to the surface normals used to calculate the tool axis at each toolpath position. This will result in a smoother change in tool axis orientations along the toolpath. The smoothing threshold value is the angle change in degrees per unit (mm or inch).

The smoothing filter works this way: If the change in surface normal exceeds the smoothing threshold at any point along the toolpath then additional toolpath positions are added. The tool axis at these new toolpath positions is calculated as a linear interpolation of the tool axis between the points of the originally calculated toolpath.



Calculation Based on Tool Center

This function controls the fundamental way that the toolpath is calculated. It has an effect on how the tool is positioned so that it is touching the drive surface. When this function is on, the calculation is based on the center of the tool. When it is off, the calculation is based on the contact point of the tool. With this function the center of the tool is defined to be the center of the tool's corner radius, which for a ball end mill is on the tools axis.

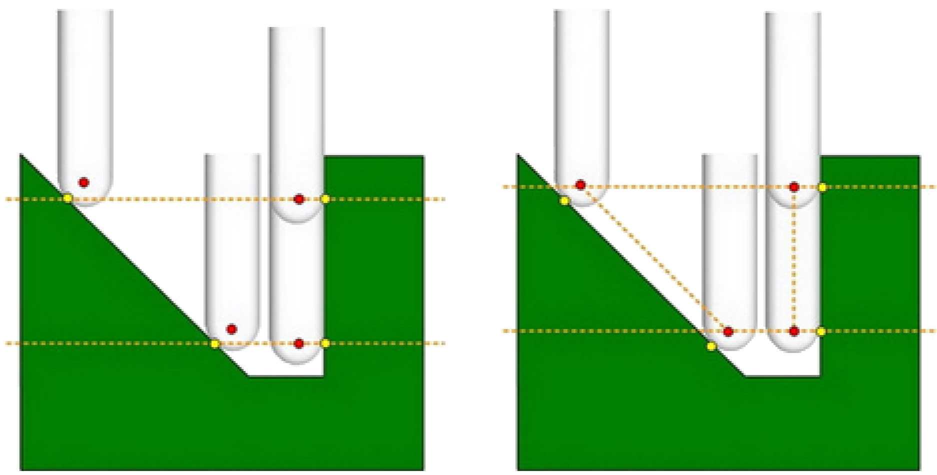
In calculating the toolpath, the tool is initially positioned so that it is touching the drive surface with an initial tool axis orientation and so that it satisfies the criteria specified by the settings on the Surface Paths tab (see "Surface Paths tab" on page 58). Then the tool is tilted so that the tool axis satisfies the criteria specified by the settings in the Tool Axis Control tab (see "Tool Axis Control tab" on page 153).

When the Calculation based on tool center option is selected, the tool axis tilting is done without changing the location of the tool nose radius center point, so that the tool axis then tilts around that center point, and the tool/drive surface contact point changes. When the Calculation based on tool center option is unselected, the tool axis tilting is done without changing the location of the tool/drive surface contact point, so that the tool axis then tilts around that contact point, and the tool nose radius center point changes.

The result of using this function is that the user has more control over the actual toolpath positioning.

Example: Side view of toolpath with constant Z cuts.
The images below compare side views of a toolpath with constant Z cuts, with the checkbox selected vs cleared.

- The illustration on the left shows the calculation with the checkbox for Calculation based on tool center unselected, so the calculation is based on the contact point. In this case, all of the contact points on the face have the same Z value, but the tool center varies as the surface normal changes– in other words, as the surface becomes steeper, the tool center point drops in Z.
- The illustration on the right shows the calculation with the checkbox for Calculation based on tool center selected, so the calculation is based on the tool center point. In this case, all of the tool center points have the same Z value, but the drive surface/tool contact point varies as the surface normal changes – in other words, as the surface becomes steeper, the tool contact point goes up in Z.



Red dots represent the tool center and yellow dots represent the surface contact points.

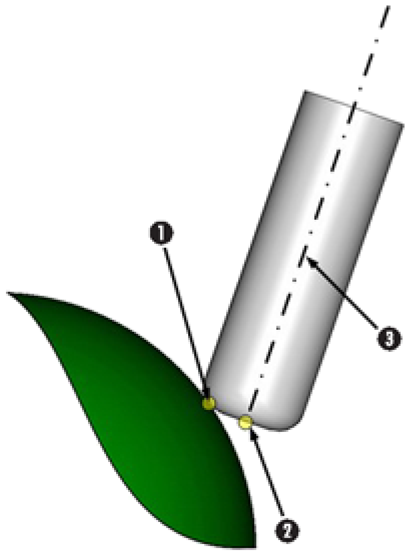
Using the Calculation based on tool center option typically requires a Margin offset value (the tool nose radius) to be entered into the Margins dialog box, summoned by the Margins button on the Surface paths tab (under Area, just to the right of certain Type drop-down choices). For more information and an example, see “Stepover” on page 151.

Glossary

The following is a list of terms and concepts used throughout the 5-Axis documentation.

Term	Definition
Drive Surface	A drive surface is the face or group of faces you want to machine. The faces may be on solids or sheets. You can select drive surfaces when the selection mode in the Machining palette is set to Part. *Tip:* When using Feature Manager as a shortcut for selecting surfaces, double-check the list shown in the Select [...] Surfaces dialog box to be sure it includes items of type Face only and does not accidentally include curves or points.
Drive Curve	A drive curve is geometry or edge of a solid that is used to control the toolpath. With calculation based on Surfaces, Triangle Mesh, or Swarf Machining, different cutting strategies refer to drive curves by different names, but they are all the same idea.
Swarf Machining	Also called “flank milling”; a toolpath calculation strategy used to produce a target surface in only one cut.
Swarf Milling	A limited 5-Axis option for cutting with the side of a tool.
Check Surface	A check surface is a face used to contain toolpath or set an area the tool may not enter or cut. Check surfaces are also used to control the shape of the toolpath in that the tool can follow the check surface’s topology. You can select Check Surface 1 when the selection mode in the Machining palette is set to Constraint. Other check surfaces are selectable when the selection mode is set appropriately in the Custom Mode menu.
Selector button / Ellipsis button	[...] A button that has three periods (an “ellipsis”) on it. Allows you to make a selection in the workspace.
Contact point	The point where the tool touches the material. When the tool tilts it rotates around the contact point. Note that the point on the material is fixed and the point on the tool itself changes according to the orientation (except for sharp corner tools).
Cutter Location Point	The point that is represented by the coordinates of the machine program.
I, J, K Tool Orientation	Values that represent the orientation of the tool axis.
Margin	The distance between the tool center and surfaces.
Relative Value, Relative Angle	A relative value is relative from the present toolpath point. A relative angle is relative to the moving direction of the tool.

Below is an image representing the concepts of the Contact point, Cutter Location Point and I, J, K Tool Orientation .



1. Contact Point
2. Cutter Location Point
3. IJK Tool Orientation

Conventions

GibbsCAM documentation uses two special fonts to represent screen text and `keystrokes or mouse actions`. Other conventions in text and graphics are used to allow quick skimming, to suppress irrelevancy, or to indicate links.

Text

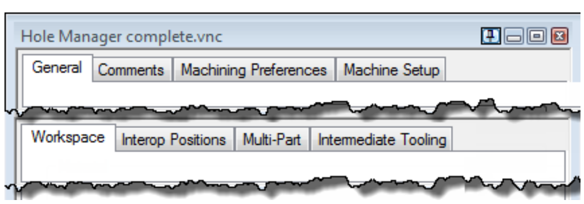
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

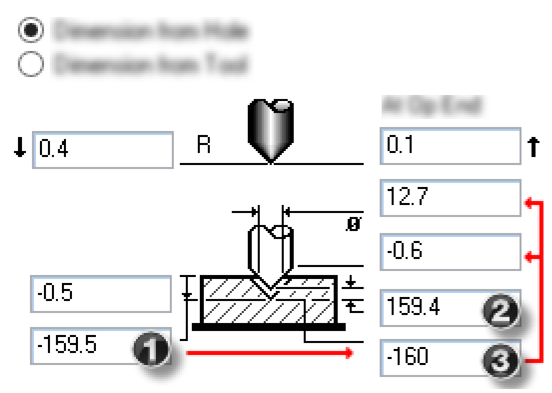
`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

Some graphics are altered so as to de-emphasize irrelevant information. A “torn” edge signifies an intentional omission. Portions of a graphic might be blurred or dimmed to highlight the item being discussed. For example:





Annotations on a graphic are usually numbered callouts (as seen above), and sometimes include green circles, arrows, or tie-lines to focus attention on a particular portion of the graphic.

Faint green borders that outline areas within a graphic usually signify an image map. In online help or a PDF viewer, you can click a green-bordered area to follow the link.

Links to Online Resources

Link	URL	Action / Description
Go	http://www.GibbsCAM.com	Opens the main website for GibbsCAM.
Go	https://online.gibbscam.com	Opens a restricted website containing materials available for download. Requires a GibbsCAM Online Services account; to set up an account, contact GibbsCAM Support.
Go	https://store.GibbsCAM.com	Opens the website for the GibbsCAM Student Store.
Go	https://macros.gibbscam.com	Opens a wiki containing documentation and examples of GibbsCAM macros. Requires a GibbsCAM account.
Go	http://kb01.GibbsCAM.com	Opens a Knowledge Base article, **Contour Operations Using Thread Mill Tools**, that explains in detail the correct way to program Contour processes using Thread Mill tools.
Go	mailto:Support@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Technical Support department for GibbsCAM.
Go	mailto:Registration@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Registration department for GibbsCAM.
Go	mailto:Sales@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Sales department for GibbsCAM.
Go	http://www.autodesk.com/inventor	Opens an external website that provides more information on Autodesk Inventor products.
Go	http://www.celeritive.com	Opens an external website that provides more information on VoluMill Ultra High-Performance Toolpath (UHPT) from Celeritive Technologies.
Go	http://www.predator-software.com	Opens an external website that provides more information on a CNC editor and a virtual CNC viewer from Predator Software, Inc.

Index

B

C

D

E

F

M

N

O

P

R

S

T

U