GIBBSCAM® 14

Version 14 : September 2020

5-Axis Tutorials

CAMBRIO

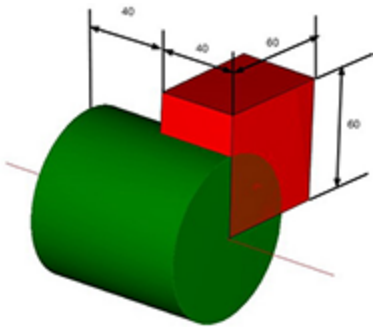
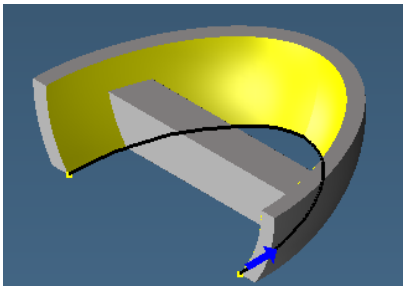
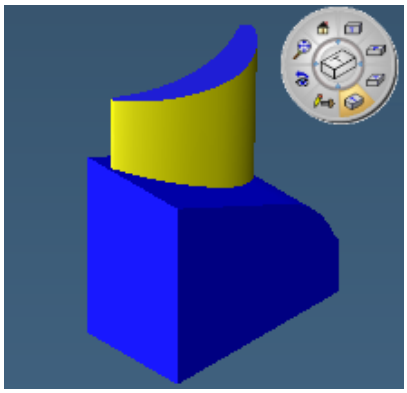
Proprietary Notice

5-AXIS TUTORIALS

These tutorials provide an overview of the 5-Axis basic features, methods and strategies. We will use pre-defined parts and cover the whole range of details of 5-Axis machining in some tutorials while others will simply cover a small part of the 5-Axis features.

ABOUT THE TUTORIALS

We will work on the three following files to introduce the 5-Axis features.

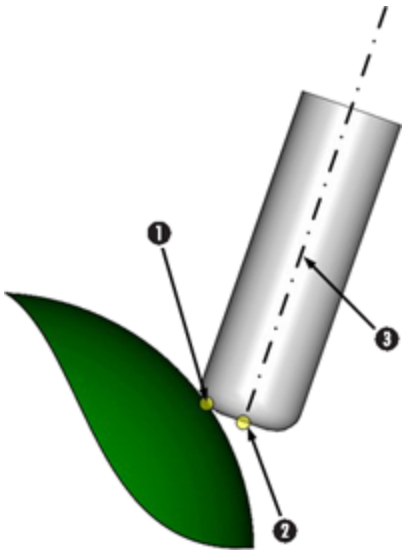
Tutorial 1	Tutorial 2	Tutorial 3
		
Parallel cuts on a cylinder with a fixture.	Toolpath on a model without creating additional geometry.	Z Constant Toolpath on a blade.

Before starting the tutorials there are several concepts you should understand.

Concepts

The following diagram describes some general 5-Axis terms that will be used in the following tutorials:

1. Contact point.
2. Tool center point.
3. I,J,K, tool orientation.



Contact point
the point at which the tool touches the material.

Cutter location point
the point that is defined by the generated toolpath.

I, J, K tool orientation
values that represent the orientation of the tool axis.

Tips

When defining a tool tilt, it will rotate around the contact point. The point on the material is fixed and Cutter location point on the tool will change according to the tool orientation (unless they are the same position as in sharp tip tools).

When using surfaces you should always check sheet sides before creating a new operation. This will ensure the toolpath is applied to the correct side of the surface.

TUTORIAL 1

Parallel Cuts on a Cylinder with a Fixture

We will be creating a single process to cut the cylinder and redoing operations many times to discuss different features in 5-Axis processes on the following tabs.

1. Surface paths.
2. Tool axis control.
3. Link.
4. Collision control.

> ⓘ It is recommended that when creating the process that you work through the tabs in this order to create toolpath.

Surface paths
On this tab you can set the options for the Pattern, Area, Sorting and Surface quality. The Pattern includes the Drive Surfaces which are the surfaces to be machined.

Tool axis control
This tab allows you to define the tool orientation and machining Limits.

Link
Here you can setup the motion of the tool between slices, cuts and passes.

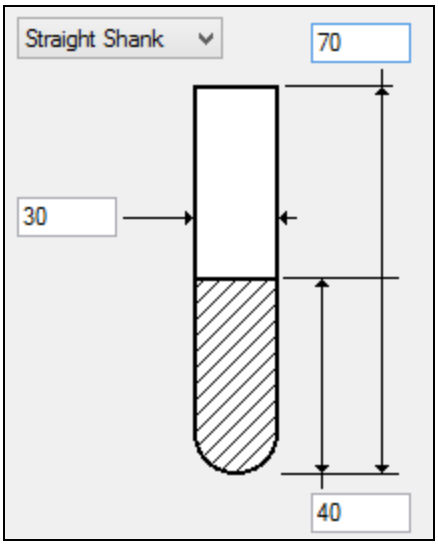
Collision Control
This tab contains all options to avoid the tool gouging selected drive and check surfaces.

Part Setup

1. Create a new 5 Axis Vertical Mill part with the following mm. workspace dimensions:
   +X: 5 -X: -85 +Y: 45 -Y: -45 +Z: 45 -Z: -45.

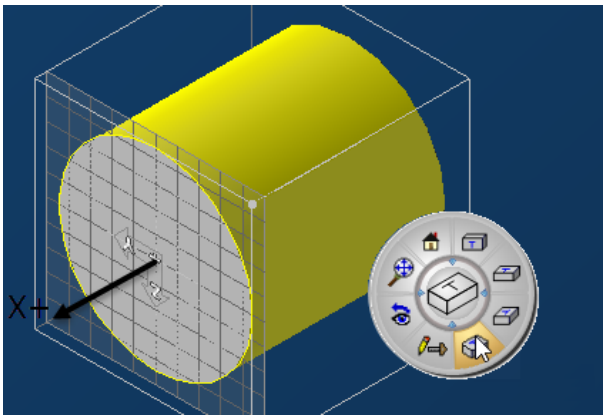
2. Create a 30mm Ball Endmill with an overall length of 70mm without the holder.

   We will first create the parallel cuts without the fixture.



3. Create a cylinder with 40mm radius and a length of 80mm or open the example file cylinder.vnc.

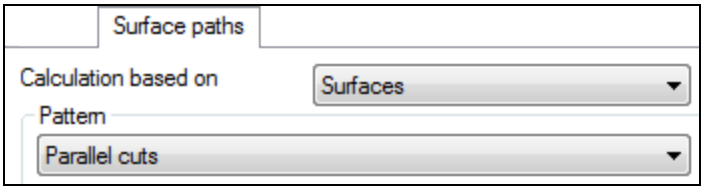
   Note that the view shown is the right-click isometric view.



Ensure the cylinder is parallel to the X axis and starts in X0, Y0 and Z0, and lies in X- direction. This is important for the Surface Paths pattern orientation.

Surface Paths

1. Create a 5-Axis process  with the Ball Endmill.
2. Switch to the Surface paths tab.
3. Accept the default Parallel cuts as we wish to make parallel cuts in the X axis.

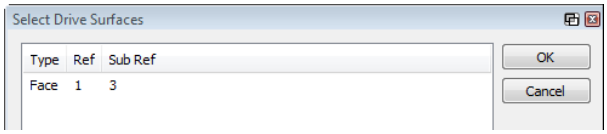


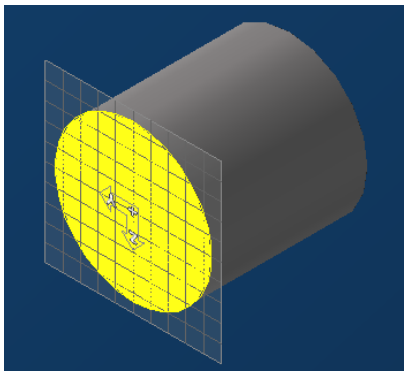
4. Make sure the Machining angle in Z is set to 90° or click the Parallel button.
5. Change the Cutting method to One way.
6. Also change the Maximum stepover to 8mm.
7. Click the Drive surfaces button. 

   A dialog opens that displays the surfaces selected to machine.

8. Turn on  Face selection.

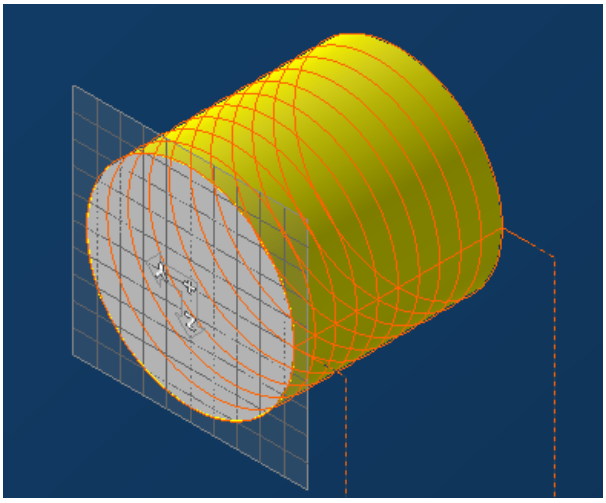
9. Select the round cylinder face as shown and click OK.





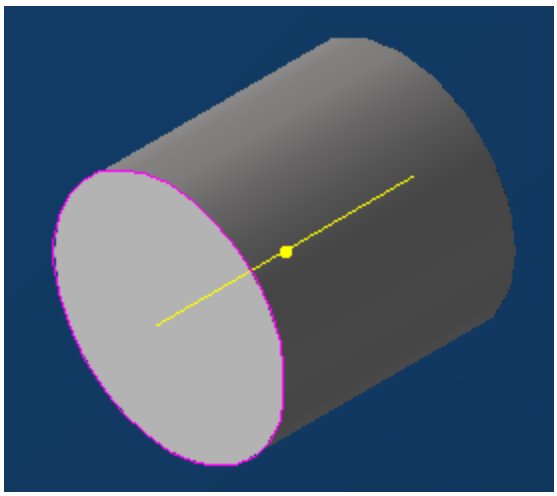
10. Click the cylinder and create the operation.

Do It



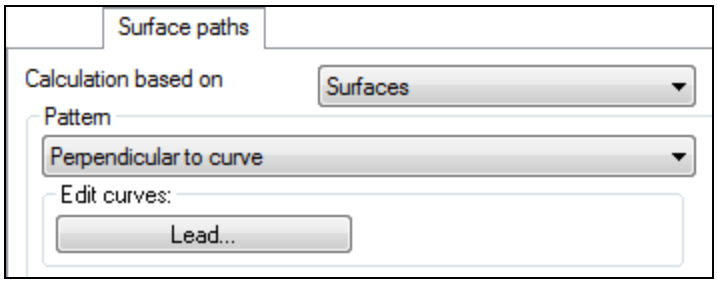
Now we will change the pattern to Perpendicular to curve. This is another way to get the desired motions without having to calculate the angles needed as in the previous definition.

11. In the XY plane create a 180° line at Y0.



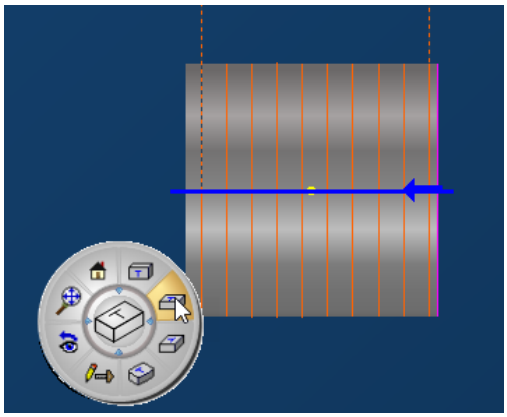
To ensure the desired direction of the toolpath it is important to set the direction of the lines movement in the -X direction.

12. Change the Pattern to Perpendicular to curve.



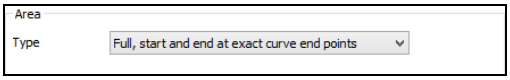
13. Now click the Lead button and select the line. Click OK.

14. Click Redo. 



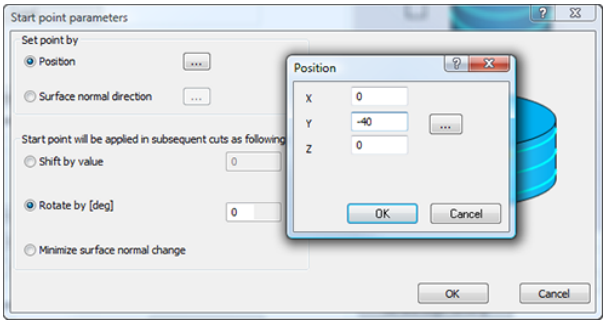
The start point was randomly selected. We will now define the start point. We want the machining to start on the edge of the cylinder.

15. Change the Area Type to Full, start and end at exact curve end points.



16. Click the checkbox to enable Start Point and click the Start point button.

17. Enter a Y value of -40mm and clickOK.



18. Redo the operation.

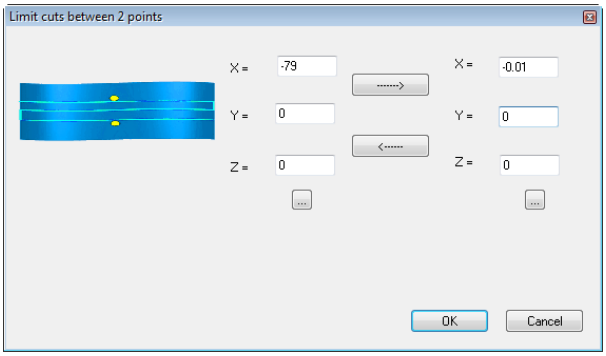
The toolpath is generated exactly up to the surface edge or to the nearest possible position.

We will now create toolpath that will limit the cuts between points.

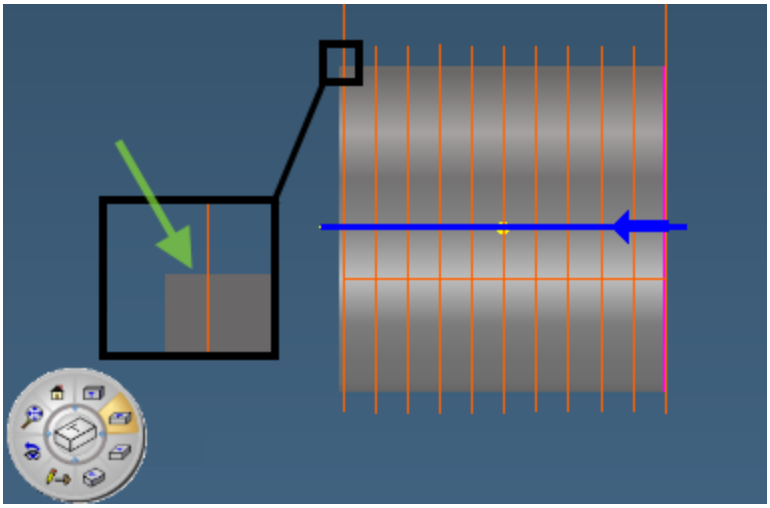
19. Change the Area Type to Limit cuts by one or two points.

20. Click the Set Points button.

21. Enter -79mm for the first X value and -0.01mm for the second X value and clickOK.



22. Redo the operation.



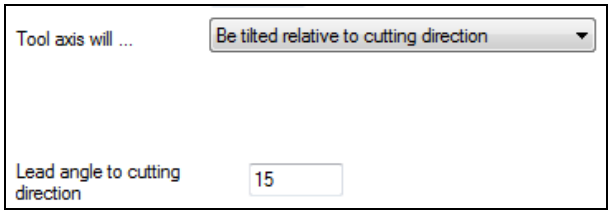
The cuts are made between the limit points.

Tool Axis Control

1. Switch to the Tool axis control tab.

   We will now define a Lead angle.

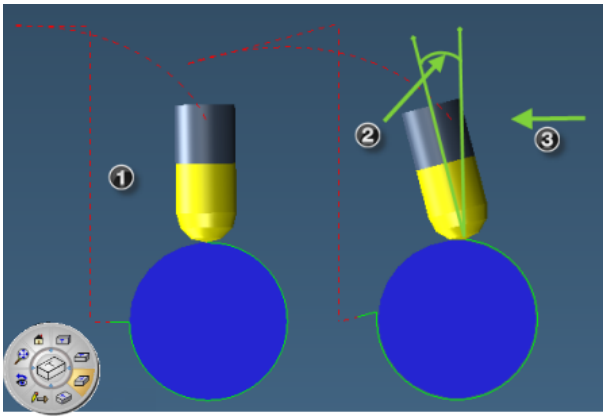
2. Ensure the Tool axis control strategy is set to Be tilted relative to cutting direction and set a Lead angle to cutting direction of 15°.



A positive value will lean the tool in the direction of movement and a negative value will lean the tool away (lag).

3. Redo the operation and render the results.

   From the right side view you can see the difference between them.

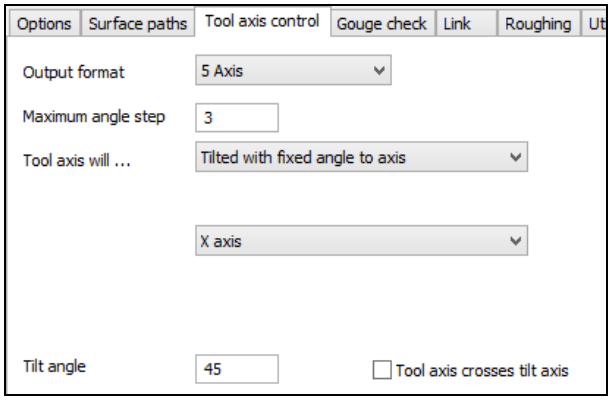


1. No lead angle.
2. Lead angle.
3. Motion direction.

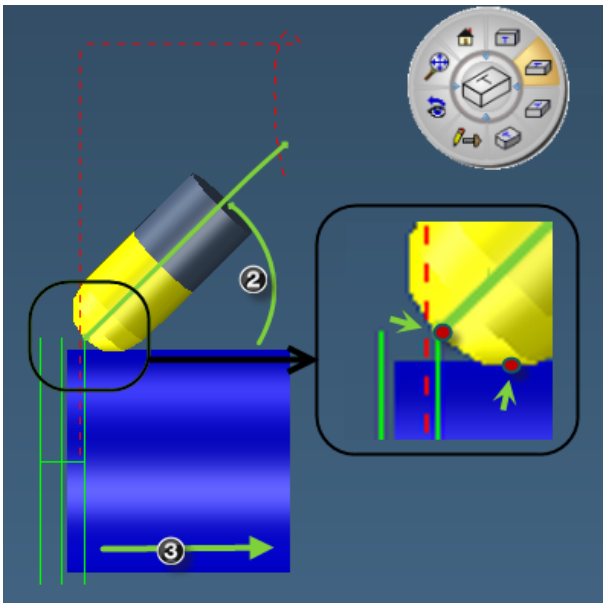
4. Now change the tool axis to Tilted with fixed angle to axis.

5. Enter 45° as the Fixed tilt angle in the X-Axis.

   This will tilt the tool at 45° relative to the X axis. The tilt axis can be set to the X, Y, and Z axis or any line.



6. Redo the operation.
7. Switch to front view.

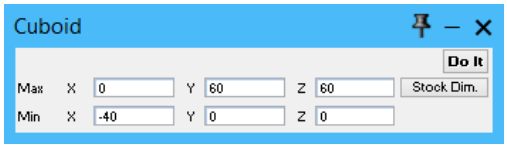


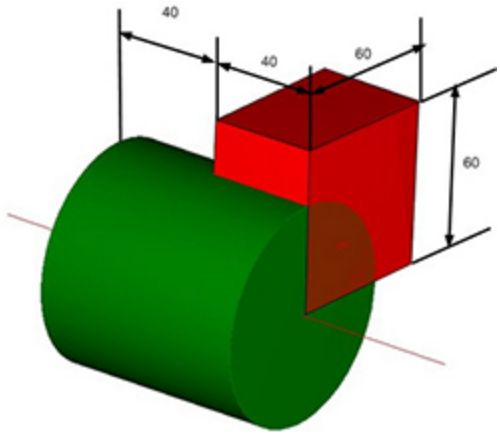
The tool is now tilted towards X+ in a constant angle of 45 degrees all along the toolpath. Notice that the contact point of the surface is maintained and the tool center is moved due to the tilt.

Gouge Check

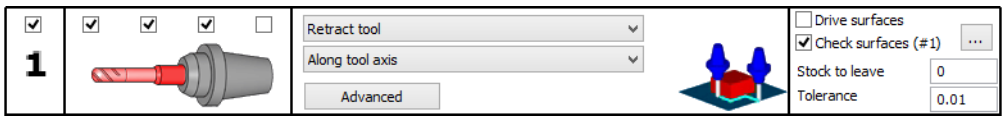
Now we will work with gouge check options. For this we will create a fixture.

1. Create a cuboid based on the dimensions shown:





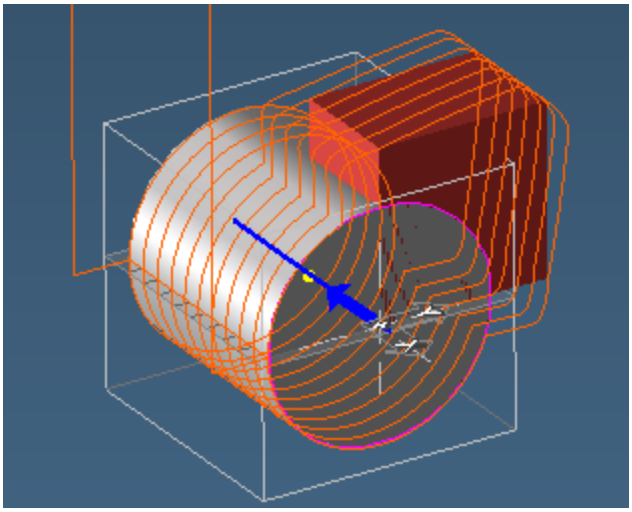
2. Change the properties of the cube to be a fixture.
3. In the Tool axis control tab change the tool axis to Not be tilted and stays normal to surface.
4. Switch to the Gouge check tab and activate Status 1.
5. Activate the options shown.



The gouge check will be checked only against the tool tip, shaft and the front of the holder. The checkboxes indicate which part of the tool is gouge checked against. The menu indicates the strategy that will be used to prevent a gouge. In this case the tool will retract along its current orientation axis until it will not gouge. Selecting the check surfaces allows a stock clearance tolerance to be set. The Clearance for tool parts button allows specific clearance values for each part of the tool to be set.

Because we defined the cuboid as a fixture the body will be used as the check surfaces. If you click the ellipsis button  you will be able to add any other surfaces or faces to the dialog. If this is necessary, turning off Face selection will allow you to select the whole body to add all the faces as Check Surfaces at once.

6. Create the toolpath.

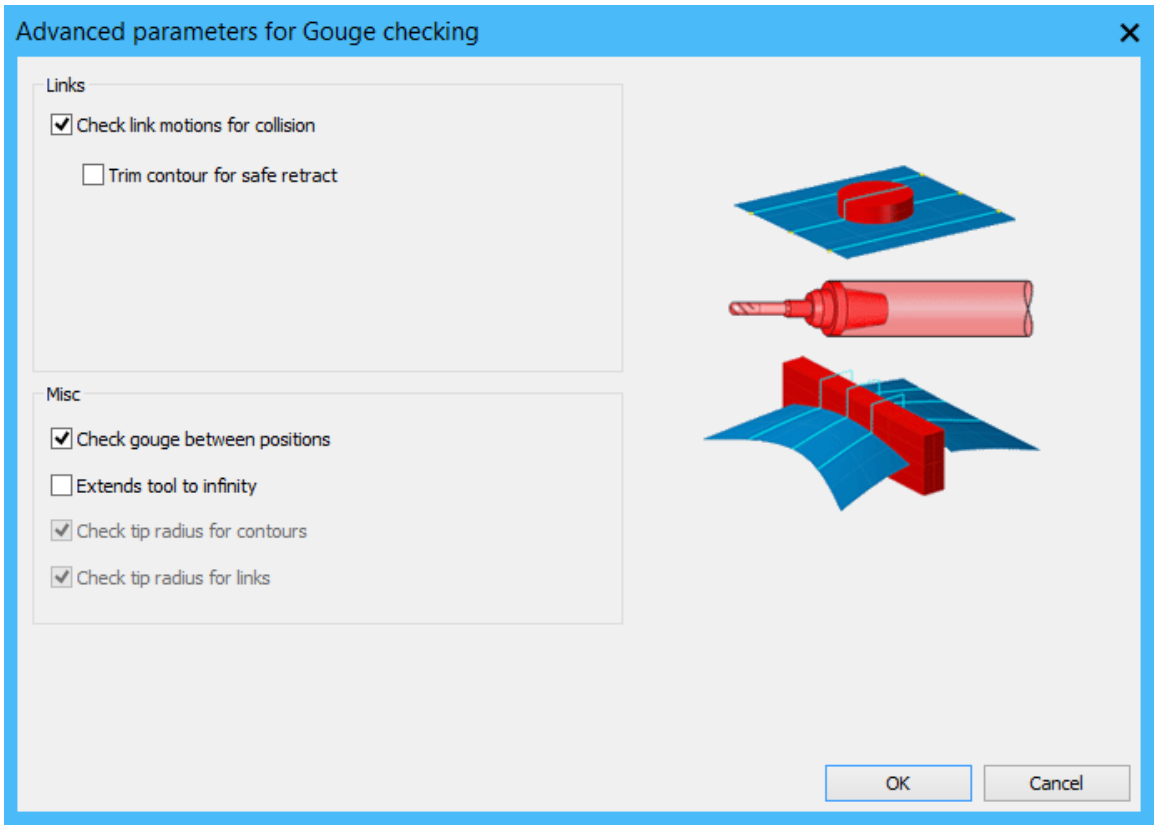


7. Render the operation.

   Notice that the check surfaces are machined as well. All the time the tool orientation is kept normal to the drive surface (cylinder).

8. In the Gouge check tab `click` the Advanced button.

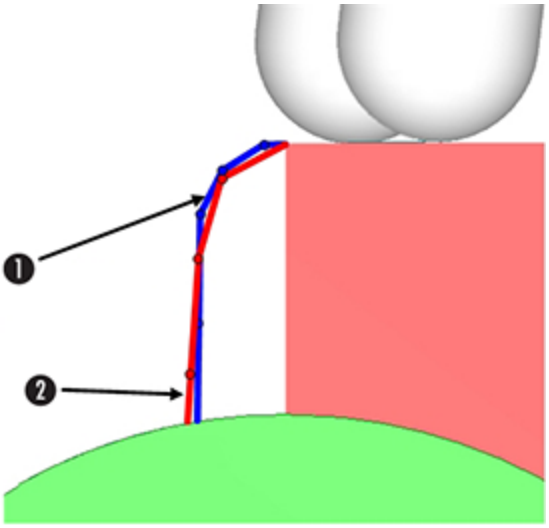
9. Deactivate Check gouge between positions and Check link motions for collisions.



   Create a new operation to compare the difference.

10. Switch to the Right view and compare the two toolpaths.

1. Using Check gouge between positions.
2. Op #2.

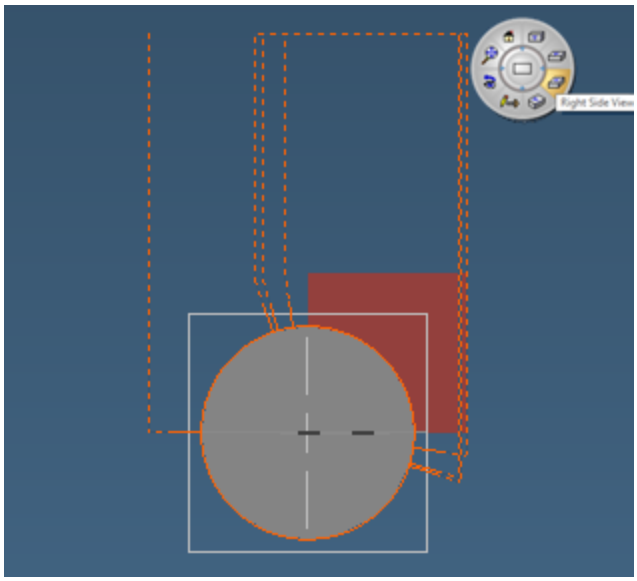


The second toolpath used less tolerance and could possibly cause a small gouge. The original toolpath is more smooth.

2. Delete the second operation.
3. In the Gouge check tab change the strategy to Leaving out gouging points.
4. Enter a 2mm Stock to leave.

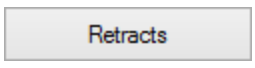
   This will limit tool movement around the check faces to be greater than 2mm.

5. Redo the operation.

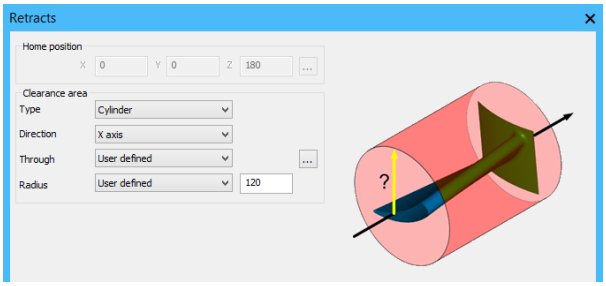


The tool now rapids around the check surfaces. The moves may still gouge the fixture depending on other settings, and we will change the tool motion in the next step.

Link

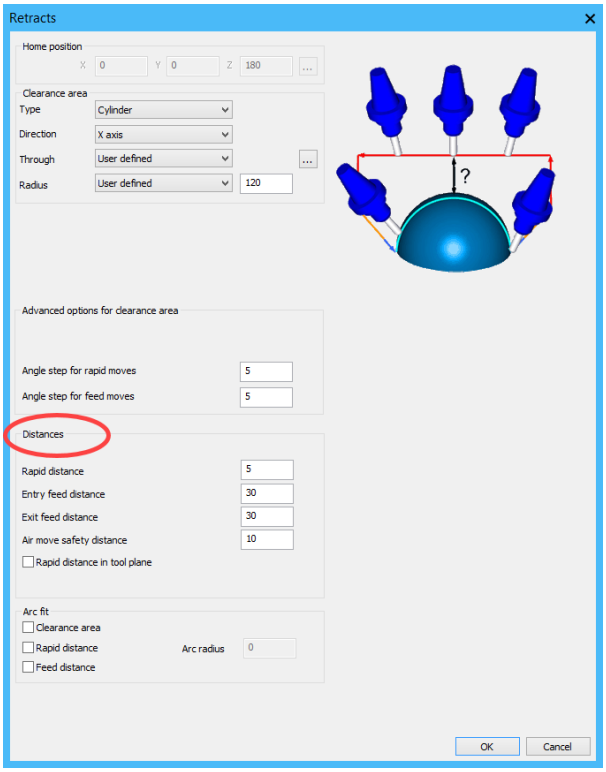
1. Switch to the Link tab.
2. Open the Retracts dialog. 

3. Set the clearance Type as Cylindrical parallel to X with a radius of 120mm.



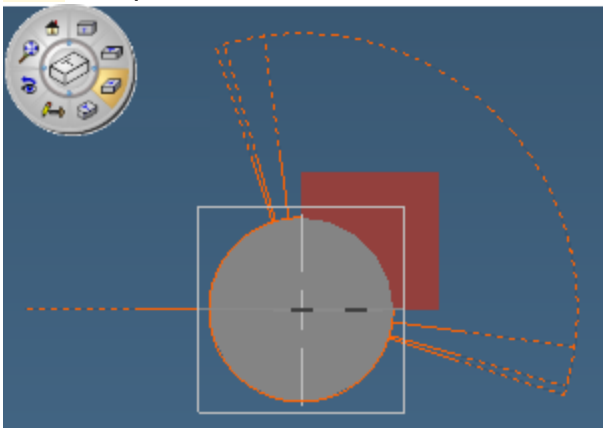
In the Distances section of the dialog.

4. Change the rapid distance to 5mm and the feed distance to 30mm. Leave the air move safety distance with default 10mm.



5. Click OK.

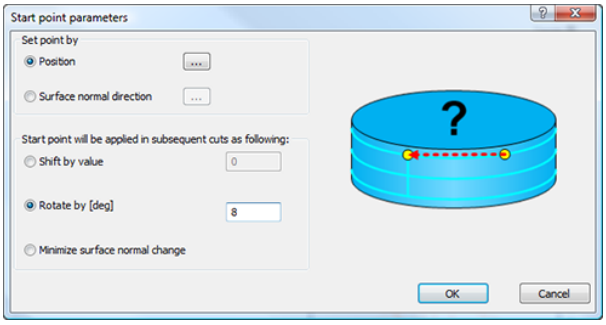
6. Redo the operation and render.



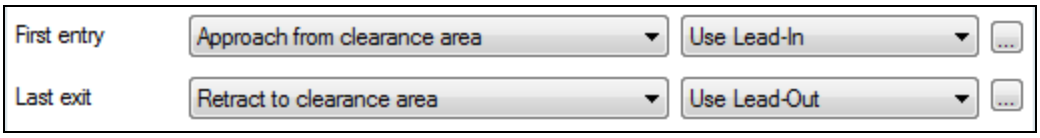
7. Switch to the Surface paths tab.

8. Open the Start point parameters dialog. 

9. Change the Rotate by [Deg] to 8.

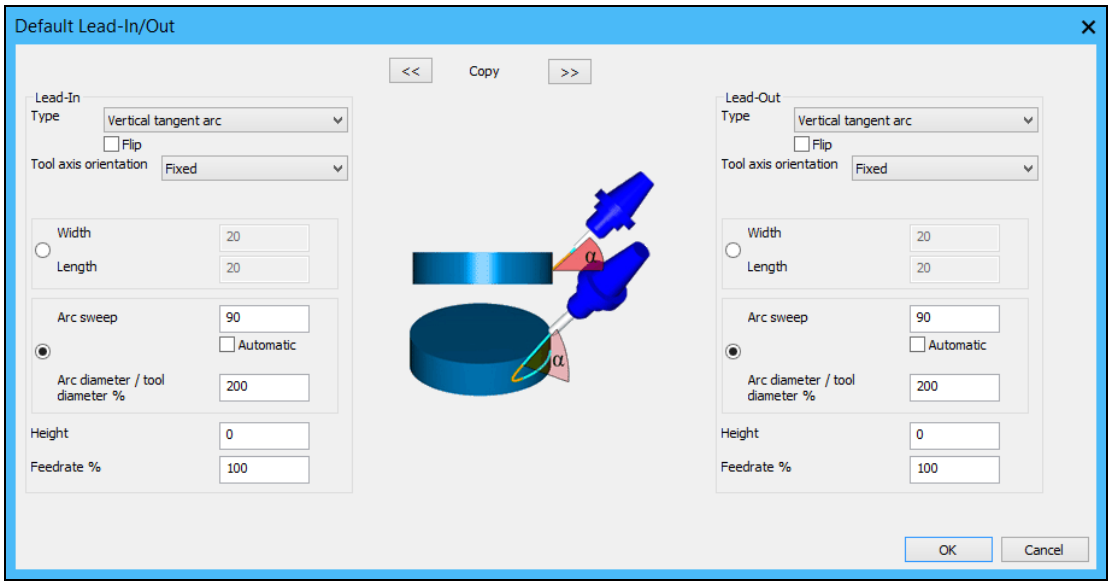


10. In the Link tab change the First entry and Last exit to Use Lead-In and Use Lead Out.



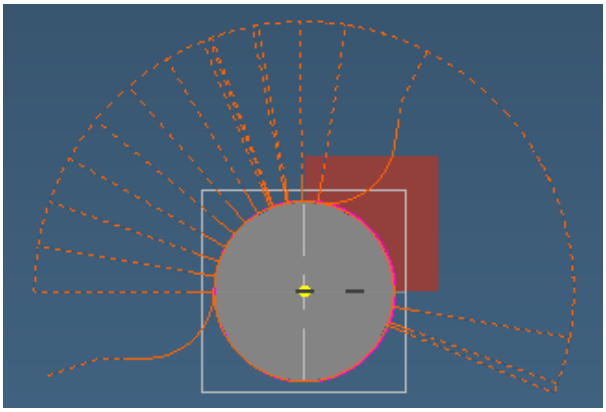
11. Open the Default Lead-In/Out dialog. 

12. Select Vertical tangent Arc in Lead-In as well as Lead-Out.



13. Redo the operation and render.

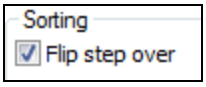
    Notice the tangent approach and retract as a result of using Lead-In/Out.



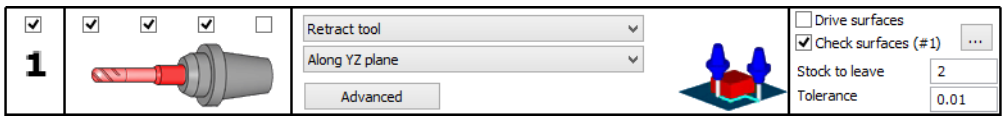
At each approach the start point is shifted 8°.

Advanced Options

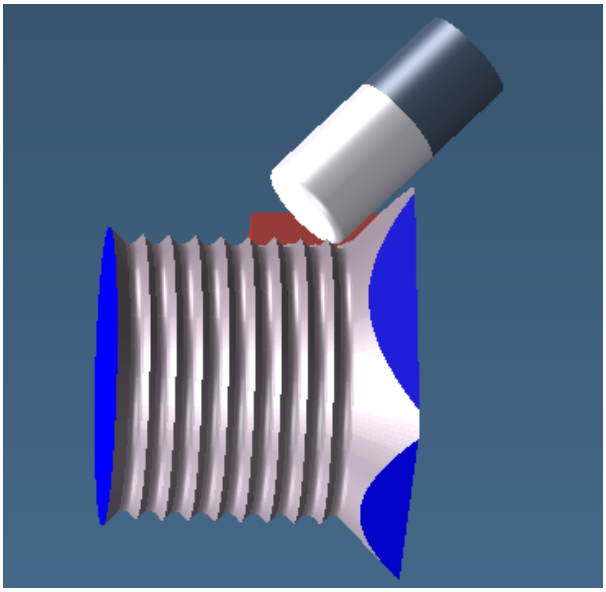
1. In the Surface paths tab, Sorting section, check the Flip step over.



2. In the Gouge check tab change strategy to Retract tool and let the tool retract in YZ plane.

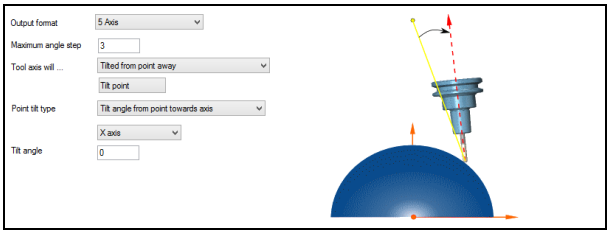


3. Redo the operation and render.

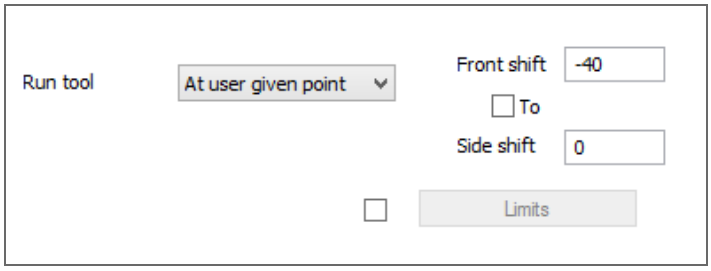


Notice that due to Flip step over the machining starts from the other side.

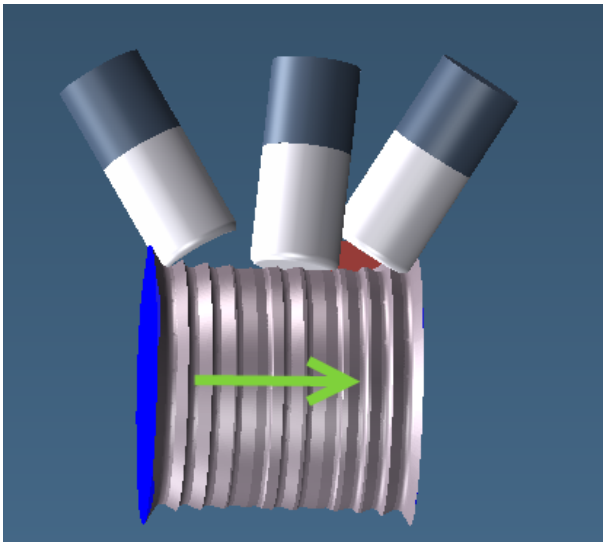
4. Go to the Tool axis control tab and change the tilting strategy to Tilted from point away.



5. Set the tilt angle to 0 and the tilt axis to the X Axis.
6. Click the Run tool dropdown.



7. Enter the values as shown.

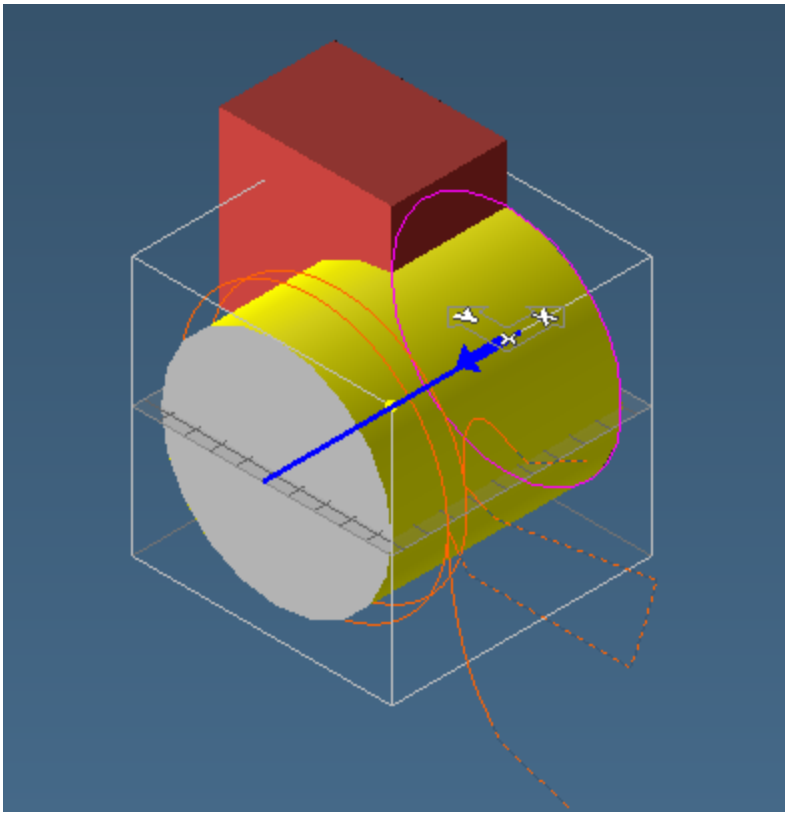


8. Redo the operation and render.

Notice that at all positions the tool axis is pointing to the point.

9. In the Gouge check tab change the strategy to Stop toolpath calculations.

10. Redo the operation and render.



Only the non-gouging motions remain.

This is the end of the first look at how a 5-Axis process works with some of the basic features. If you would like to keep modifying this part you may wish to save it. We will now continue with the next tutorial.

TUTORIAL 2

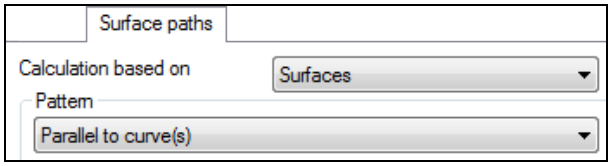
Model Surface Machining

In this tutorial, you will learn about the following 5-Axis options:

- Parallel to curve.
- Swarf toolpath.
- Gap handling methods.
- Principles of the workflow.
- Tool tilt control.
- Multiple gouge protection features.
- Gouge protection against walls and sharp corners.
- Apply different stock settings to different sets of surfaces.
- Gouge protection of tool holders.

Multiple Surface Toolpath

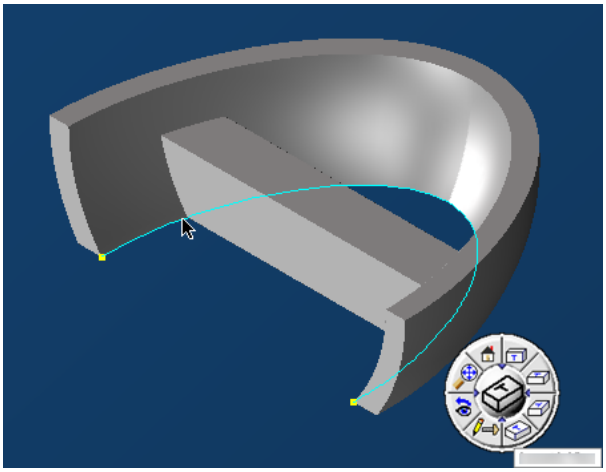
1. Open the file `simple.vnc`.
2. Create a 5-Axis process  with the tool.
3. In the Surface paths tab select Parallel to curve(s) as the pattern.



The cut direction will be along a leading curve.

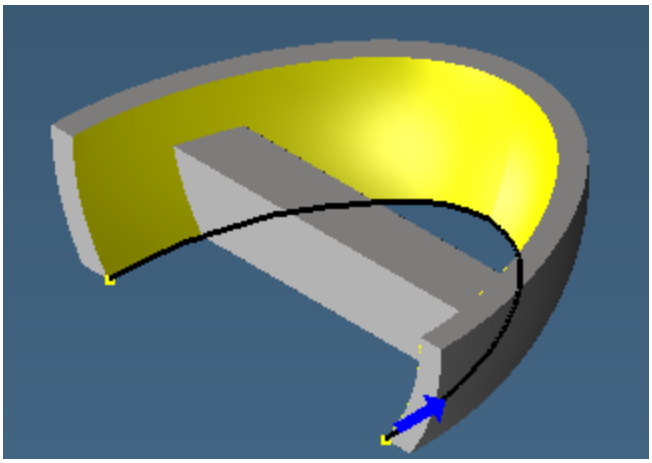
4. Click the Edge curve(s)... button. 

5. Select the geometry as shown.



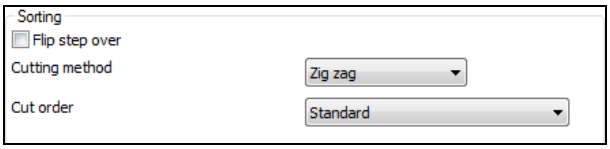
6. Click the Drive surfaces button 

and select the inner convex walls as shown.

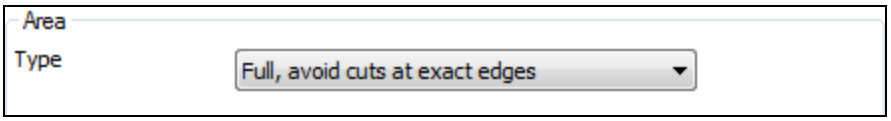


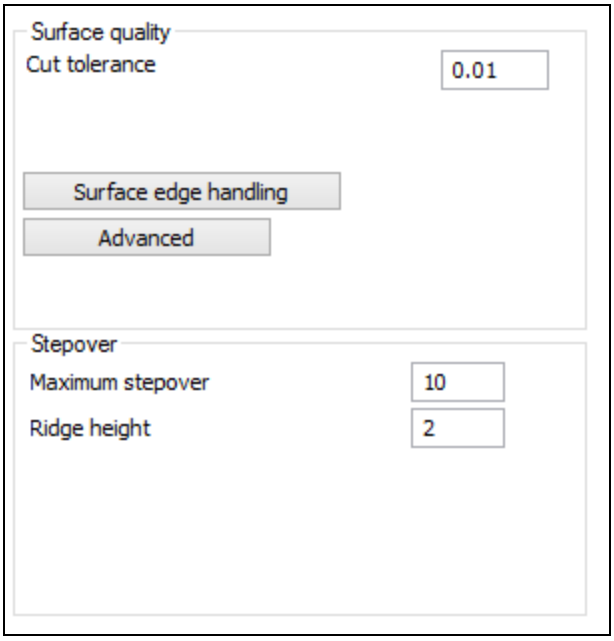
7. Click OK.

8. Ensure the Cutting method is set to Zigzag.



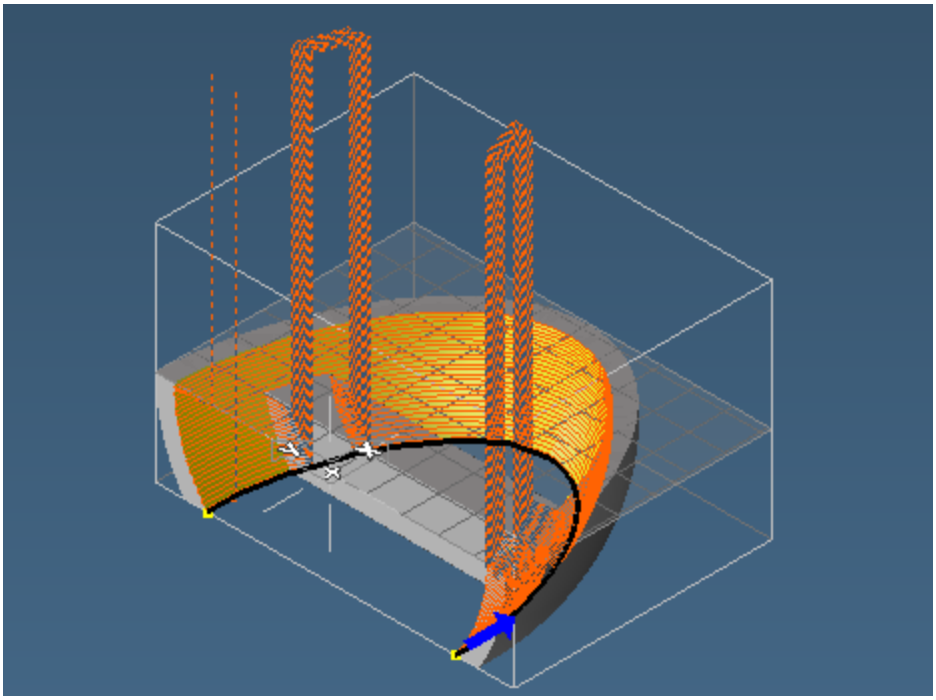
9. Set the Area Type to Full, avoid cuts at exact edges.





10. In the Surface quality settings set the Cut tolerance to `0.01`.
11. Set the Maximum stepover to 2mm.

    Leave all other settings as default.
12. Create the toolpath.



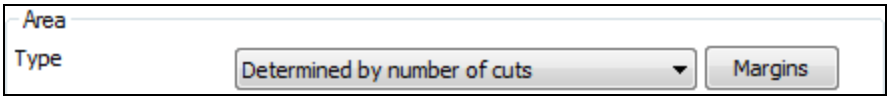
    Although this would be a clean way to cut the drive surfaces the toolpath is not usable because it gouges the crossbeam. We will need a different tool angle approach so that we do not destroy the part.
13. Render the part. `Click` the Collision checking icon and ensure that Collision checking is disabled  (not blue).

    This will ensure we do not get any collision checking errors in the next exercise.

Gap Handling

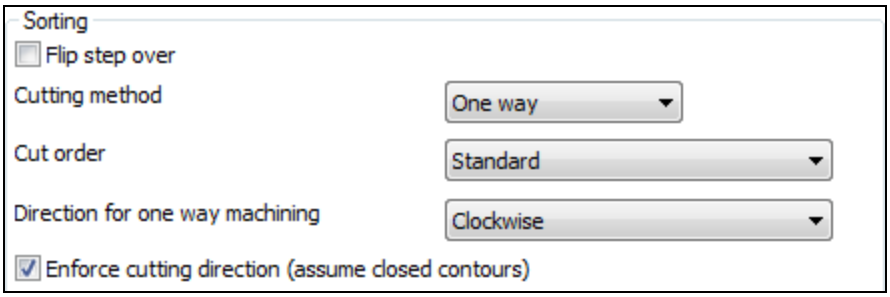
The crossbeam is a “gap” in the tool movement. We will now change to a swarf style toolpath to deal with the gap. This type of toolpath is used for machining with the side of the tool for pocket sides and walls rather that the bottom of the tool.

1. In the Surface paths tab change the Area Type to Determined by number of cuts with 1 cut.



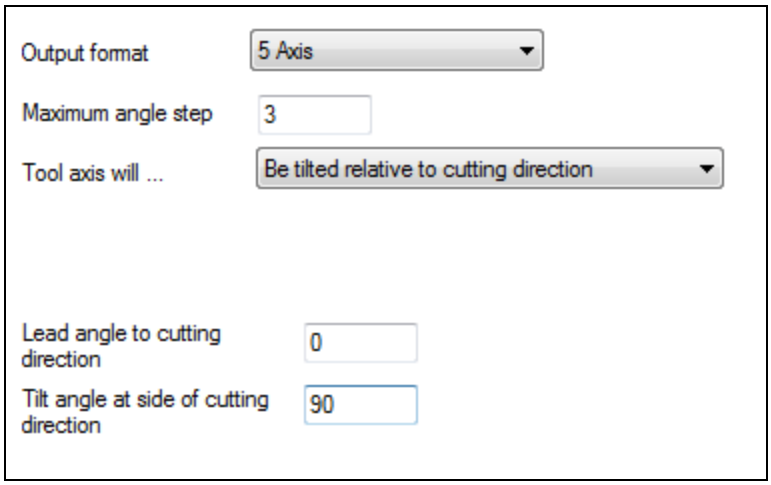
We will use a single cut to examine our choices in the gap settings. Later we will increase the steps to cut the whole drive surface area.

2. Now set the Cutting method to One way.



3. Go to the Tool axis control tab.

4. Change the tool axis to Be tilted relative to cutting direction from the strategy drop-down menu.

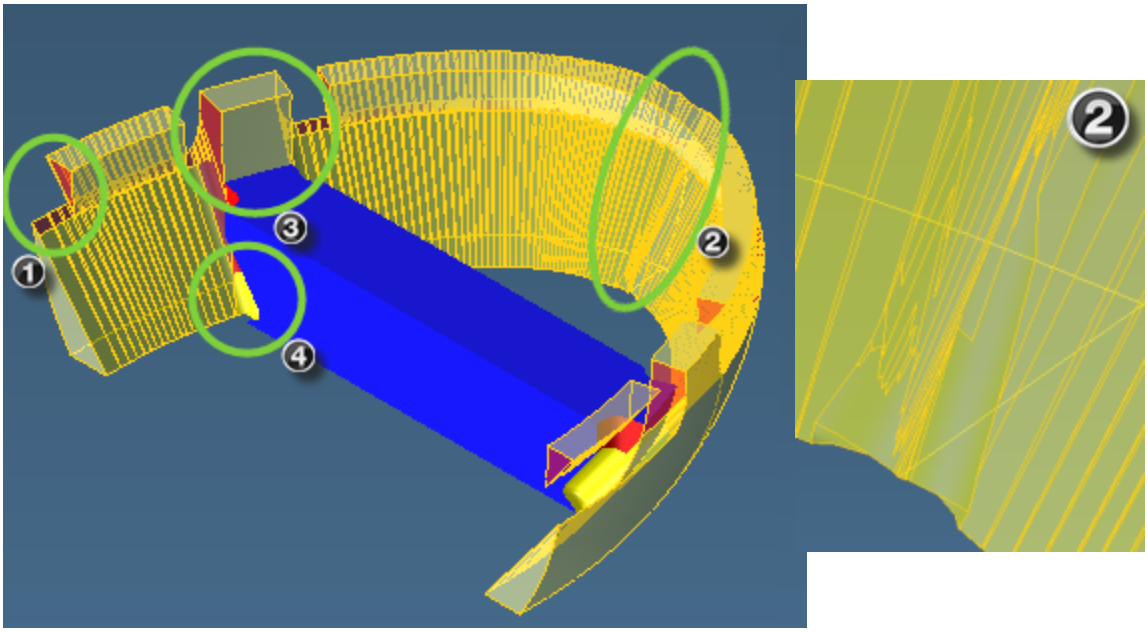


5. Change Tilt angle at side of cutting direction to 90°.

This will tilt the tool so that it is almost normal to the XY plane.

6. Redo the operation. You may need to reselect Edge curves or Drive surface.

7. Render the results.



We still have a few problems with this toolpath, as shown above.

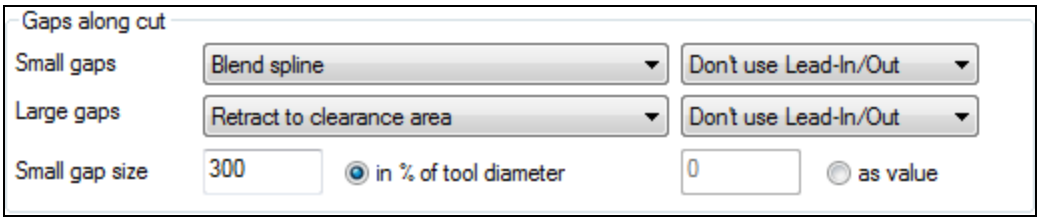
A. The tool is gouging areas of the part because of the tool tilt settings.
B. When you look at the inner radius where the surfaces meet, you will find small unnecessary movements. This area creates a “fish tail”, which as you can see from the rendering does more damage to the interior wall of the part. Note the toolpath movement as shown at a high zoom level.
C. We are cutting the crossbar and skipping the section above the bar.
D. Lastly the tool and holder gouges the fixture.

But first we will handle the crossbar problem.

8. Go to the Link tab.

The best solution is to create a toolpath that ignores the crossbar and then we can retract the tool with gouge protection strategies to better cut around the crossbar by changing the Gaps along cut settings.

9. Set the Small gaps to use a Blend spline.



We do not have any large gaps, so we can leave it as Retract to clearance area.

10. Set the Small gap size in % of tool diameter to 300.

Generally you want different settings for small toolpath gaps and large ones. A gap is any area that contains sections that are either to be avoided or ignored when generating the toolpath. In

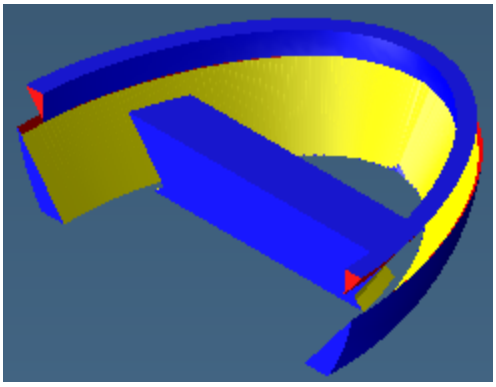
our case we want to ignore the large crossbar that creates a gap in the tool movement. Any area smaller than 300% of our tool (30mm) will be ignored when creating toolpath.

Now we will work with the “fish tail” problem.

11. Go to the Surface paths tab.
12. Enable Corner Cleanup. 

13. Enter an Additional Adjustment radius of 2 in the Corner Cleanup dialog.

    This option will attempt to round corners in the toolpath like the fish tail.
14. Redo the operation and render.

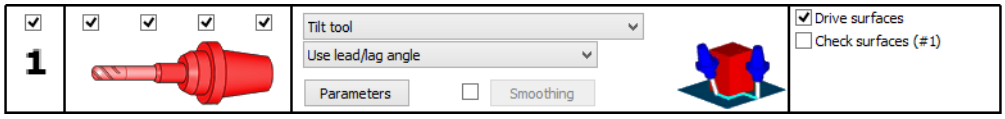


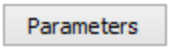
You can see that the tool does not retract to the clearance area over the crossbar area. Also in the inner edge the fish tail disappeared.

Use Different Collision Avoidance Strategies

In this section you will learn how to use and combine different gouge protection. The first we will set up will be to avoid collision with the machined surfaces by changing the tool tilt.

1. Switch to the Gouge check tab.
2. Activate Status 1 and enter the settings shown.

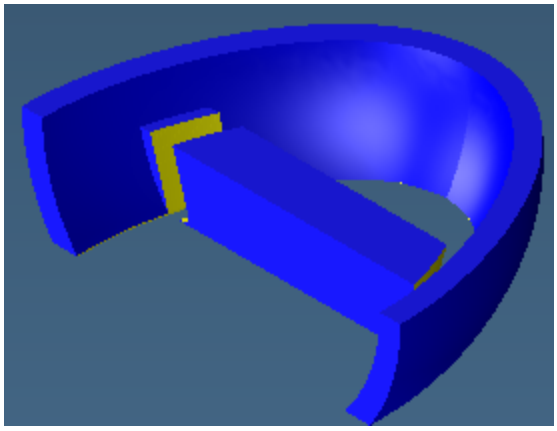


3. In the Parameters dialog 

enter 0.5mm clearance for the Clearance angle.

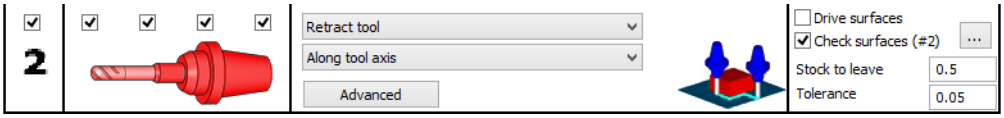
4. Redo the operation.

   The tool does not gouge the drive surface at all. But it still cuts the crossbar.



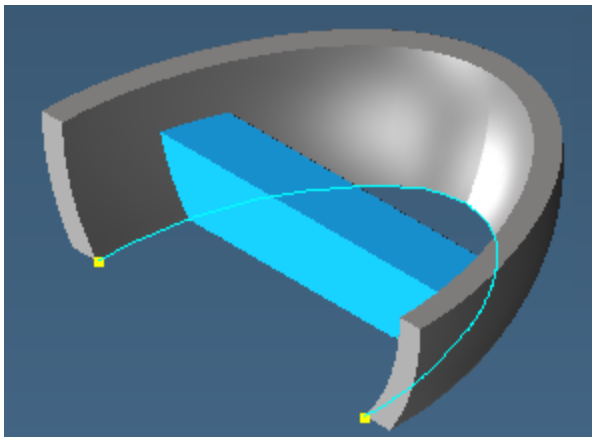
We will now combine multiple gouge check strategies to avoid collision with crossbar by retracting the tool along the tool axis over the crossbar. This way we will not have a full retract of the tool but will only retract to avoid check surfaces.

5. In the Gouge check tab Activate Status 2 and enter the settings shown.



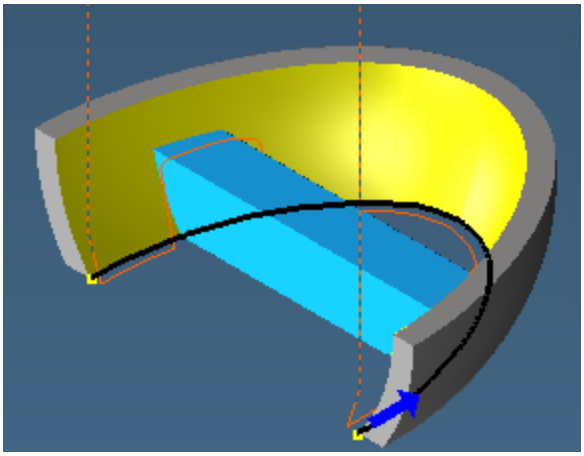
6. Open the Check surfaces ellipsis dialog. 

7. Select all the crossbar faces.



8. Click OK.

9. Redo the operation and render.

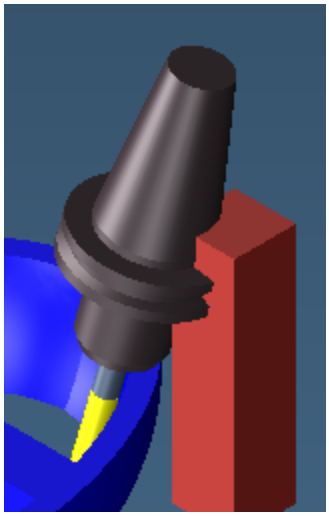


The tool retracts along its axis over the crossbar.

Holder Collision Check

In many cases it is not only necessary to check tool collisions but necessary to check the tool holder too. In some cases it can be useful to extend the holder so it represents the machines spindle or complete head.

1. Enable Show tool holder  if it is not yet enabled.

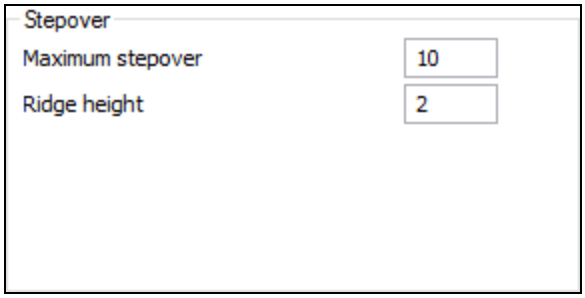


2. Render the operation with the holder.

Notice that the holder is crashing into the fixture we have.

3. Go to the Surface paths tab.
4. Change the Area Type to Full, avoid cuts at exact edges.

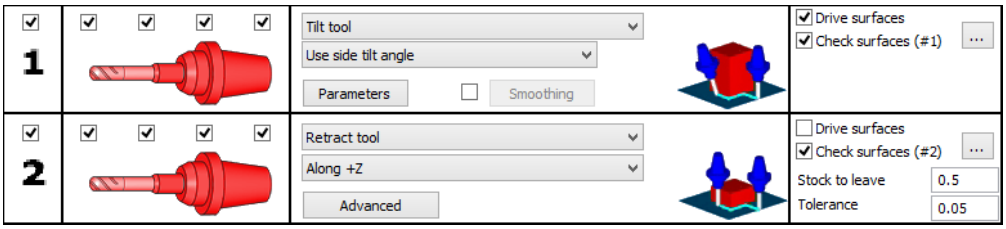
5. Change the Maximum stepover to 10mm.



6. Go to the Gouge check tab.
7. Check Check Holder back in Status 1.
8. Also in Status 1, activate Check surfaces.
9. Add the vertical fixture to the check surfaces list.

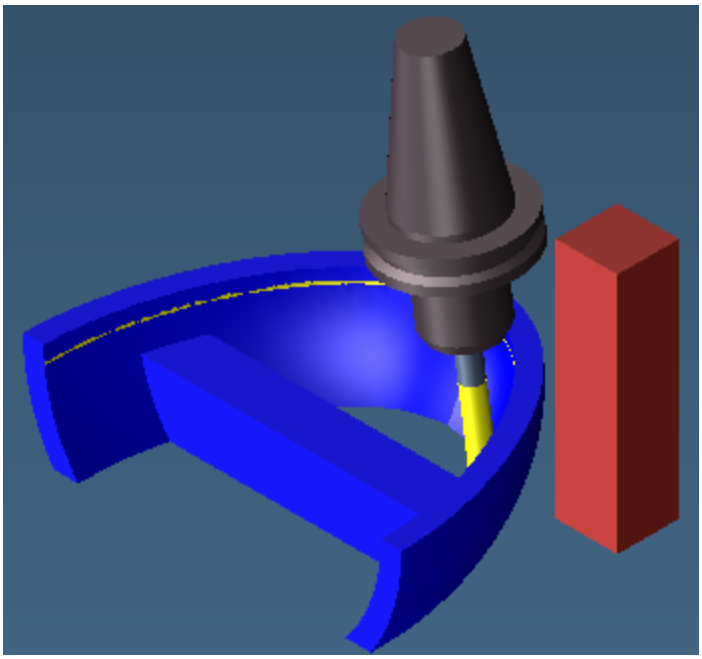
   You may need to get the fixture from the body bag.

10. Go to Status 2 and choose Moving tool away as the strategy.
11. Set the retracting parameter to Retract tool in +Z.



As it says, the tool now will retract only in positive Z direction when it hits the check surface. The tool orientation does not change.

Redo the operation and render.



Consider turning Collision Checking ( icon in the CPR dialog) back on, and make sure you have Auto Tool size enabled in the Display section of the File>Preferences>Display>CPR preference settings.

TUTORIAL 3

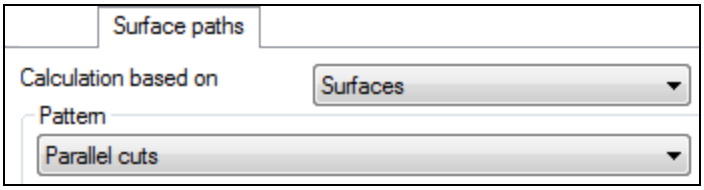
Z Constant Toolpath on a Turbine Blade

In this tutorial you will create a Z constant toolpath and a morph toolpath on the turbine blade using collision control by tilting the tool axis.

Finishing Toolpath

1. Open the file `Simple Turblade.vnc`.
2. Create a 5-Axis process with Tool #1.
3. Go to the Surface paths tab and select Parallel cuts as surface paths pattern.

   This should be the default selection.



   We want to work top down so that the motion has Z constant cuts.

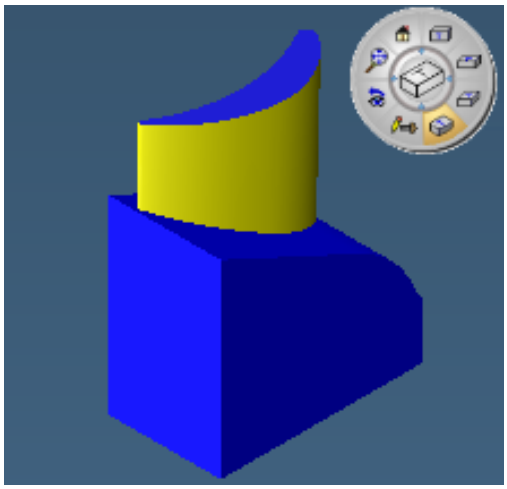
4. `Click` the Constant Z button. 

   The Machining angle in X,Y blanks out, as we do not work on that plane any more when the Machining angle in Z is 0°.

5. `Click` the Drive surfaces button 

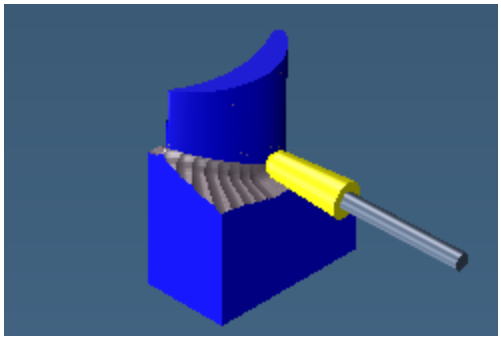
and select the surface shown below as drive surface.

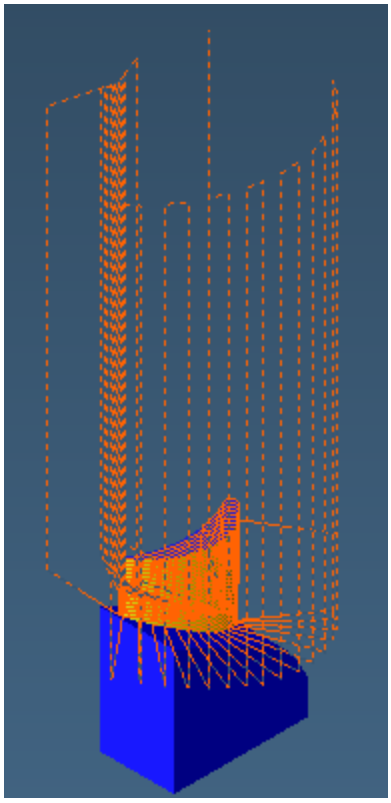
   Leave all other settings as default.



6. Create the operation and render the result.

The result is a toolpath with z constant cuts. The default retract (clearance) distance is set to 150 mm which is too far. Also when you look closer, you see that the tool is gouging the floor face.



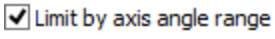


Limit Finishing Toolpath to Work with Non-Undercutting Machines

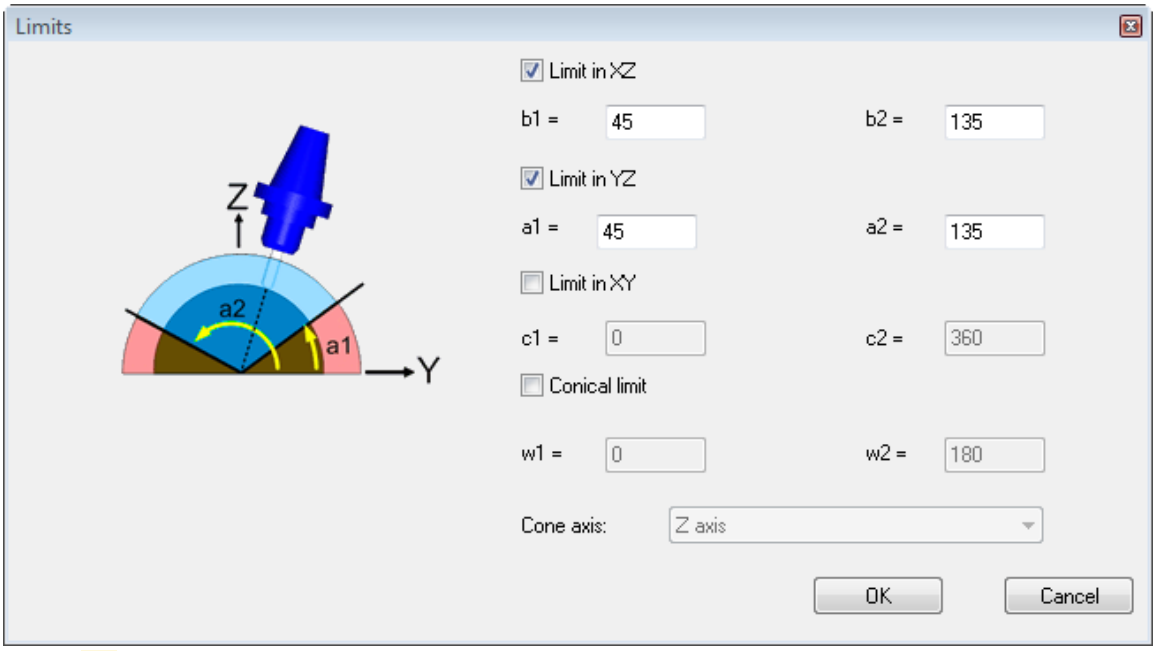
Some milling machines are limited by their axis setup to a certain range of angles, which makes it impossible to machine undercuts. For this purpose we will check the angle output using the angle limits.

> Some machines with a typical 45 degrees tilted axis are not able to reach angles more than 90 degrees (e.g. DMU70V or DMG80P).

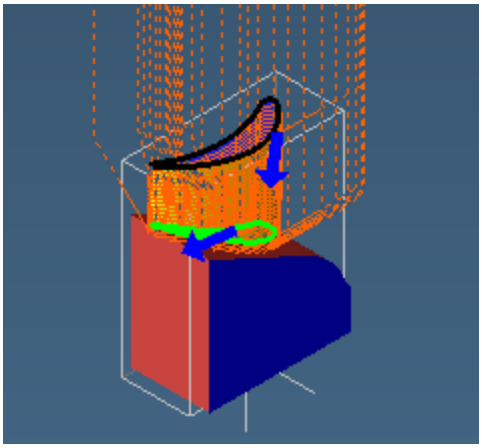
In this example the machine angle limit will be set to 45° on the Z axis.

1. Go to the Tool axis control tab.
2. Check and open the Limits dialog at the bottom of the dialog. 

3. Check Limit in XZ and Limit in YZ.
4. Enter the values shown.

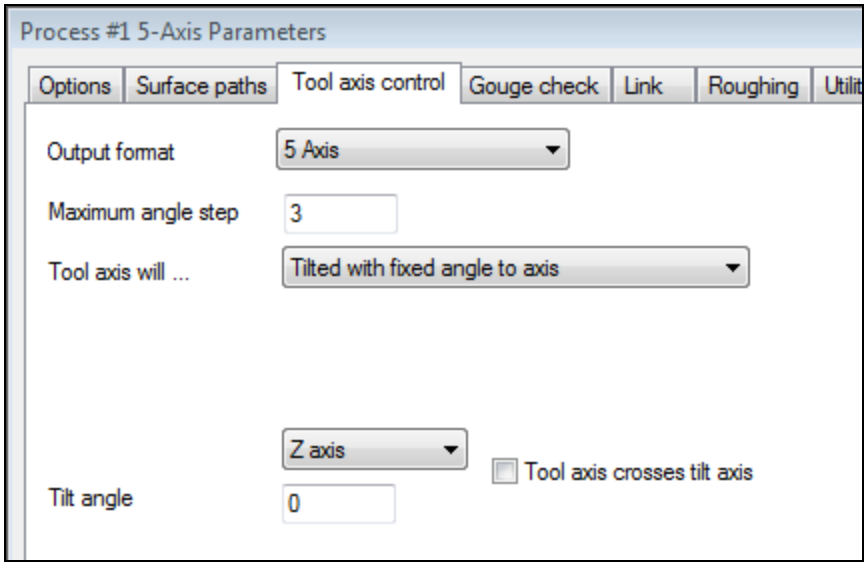


5. Click OK to close the Limits dialog.



6. Redo the operation.

   Even with the Axis Limit the tool still gouges the floor. So we will work with tool tilt to tilt the axis towards the surface normal. A tilt axis can be the X, Y and Z axis or any line.



7. Go to the Tool axis control tab and set the tool axis to Tilted with fixed angle to axis.

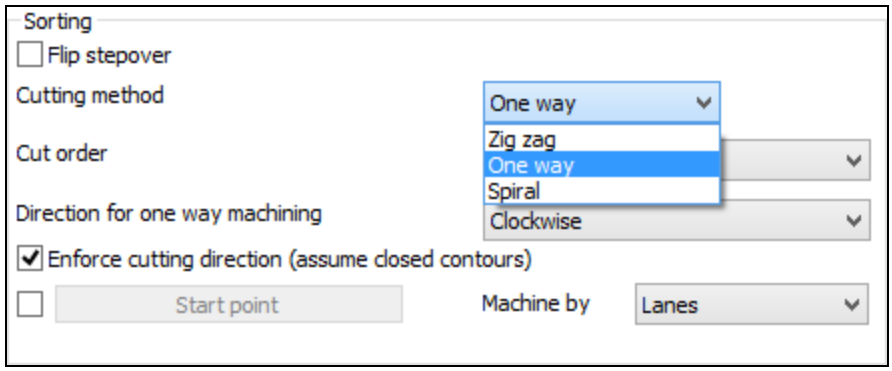
8. Enter a tilt angle of 45° in the Z Axis.
9. Redo the operation.

   In this case the result is the same as the Limit operation. But remember: When you set the limits, the tool can tilt in all directions within your limit range. With a fixed tilt angle, the orientation of the tool axis will never change.

One Way Cut

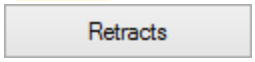
The current toolpath will change direction with each new cut. For a smoother path we will use One way cutting. We will also change the retracts.

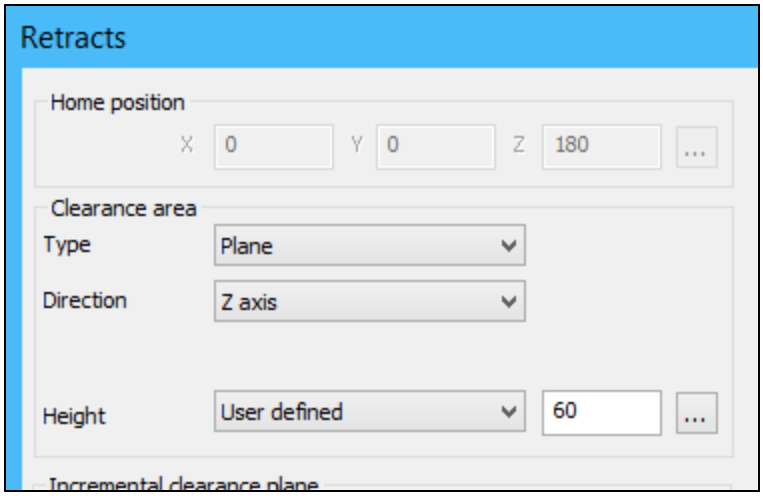
1. Go to the Surface paths tab.
2. Change the Cutting method to One way.



3. Now go to the Link tab.

   The Clearance area is set much higher than we need so we will decrease the distance at which the tool will retract.

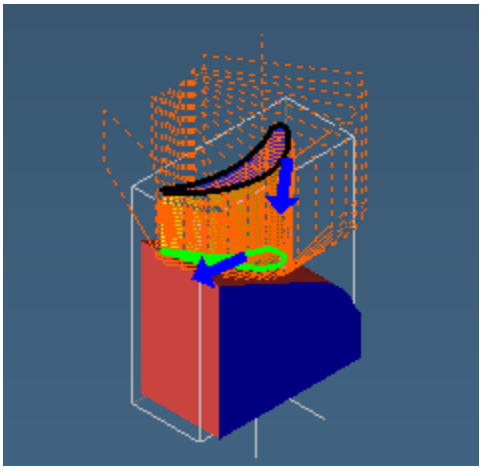
4. Click the Retracts button at the bottom. 



5. Enter a Height of 60mm.
6. Click OK.

7. Redo the operation.

   The retraction distance now is closer to the part. The tool keeps its motion direction.



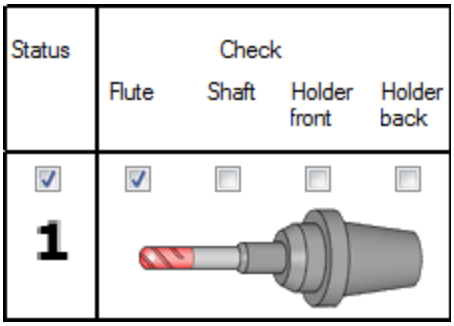
Gouge Checking

The floor surface of the blade will still be gouged by the tool. There are several gouge strategies available. Here in our case a good strategy would be to retract the tool along the tool axis.

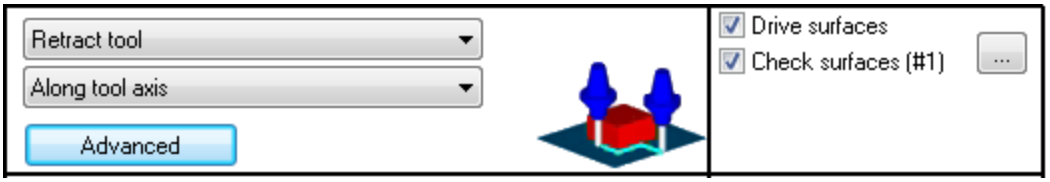
1. Go to the Gouge check tab.

2. CheckStatus 1.

   By default only the tool is selected. In our case it is not obvious that the other parts of the tool will gouge since the tool axis is tilted 45°.



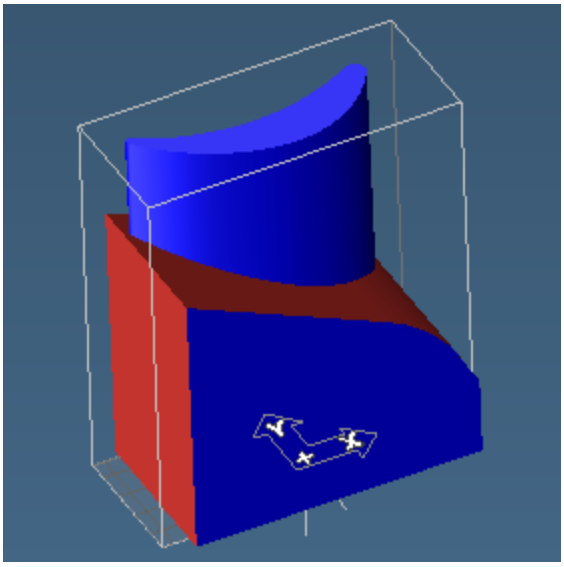
3. Change the strategy to Retract tool.



4. Set the retract to Along tool Axis.

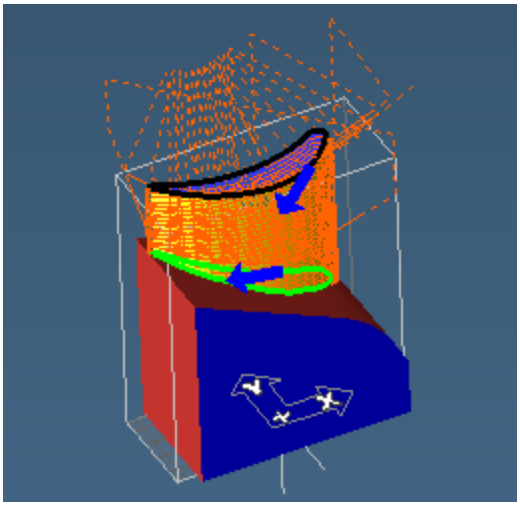
5. Activate Check surfaces and click the ellipsis button. 

6. Select the floor faces around the boss and the face that is closest to the blade.



7. Click OK button to check against those surfaces.

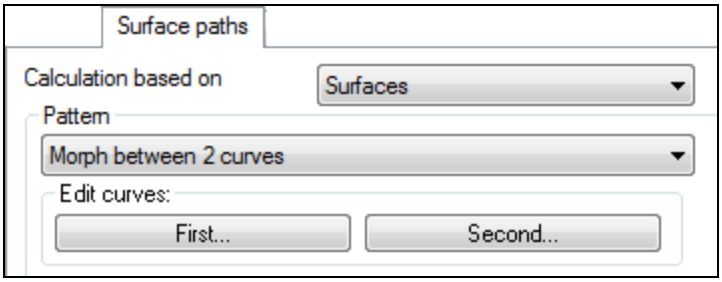
8. Redo the operation.



Morph Toolpath

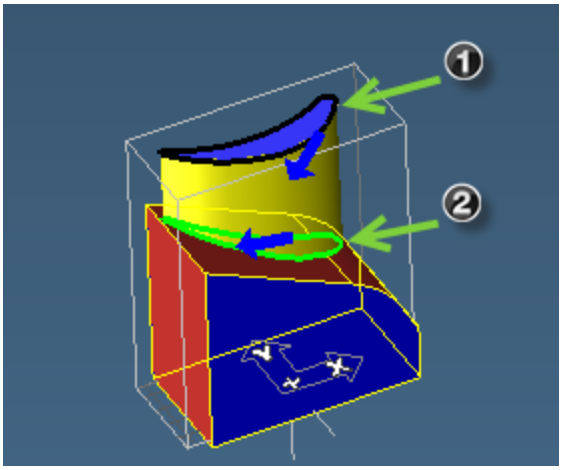
The Parallel cuts strategy does not represent the shape of our turbine blade. The upper and the lower edge of the curve are not parallel. To lower the number of retract moves we will use Morph between two curves.

1. Open the Surface paths tab and change the pattern to Morph between 2 curves.



2. Switch on Edge selection, then `click` the First button 

and select the upper curve.



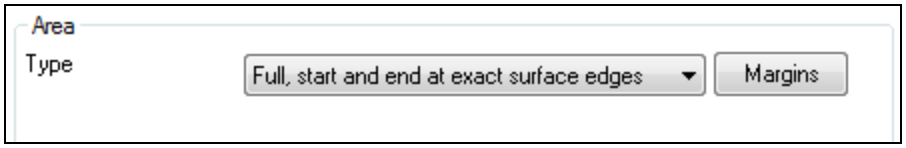
3. `Click` the Second button 

and select the lower curve.

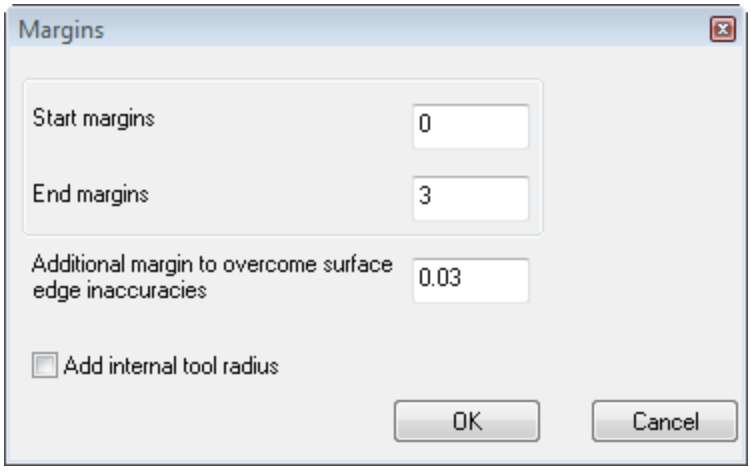
- The more accurate the guiding curves are to the real surface edges the better this function works. So the best result would be an exact curve from the edge of a body.
- Edges can be selected directly without extracting geometry.

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 Determined by number of cuts.

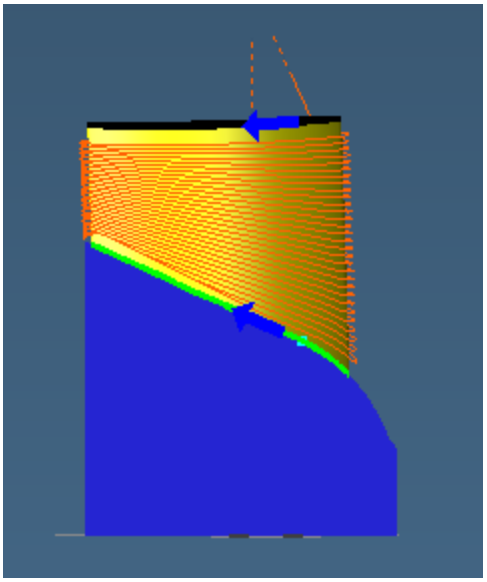
When you set the cutting area to Full, start and end at exact surface edge, you can set margins to the curves.



4. Click the Margins button and enter the following values:



5. Redo the operation.



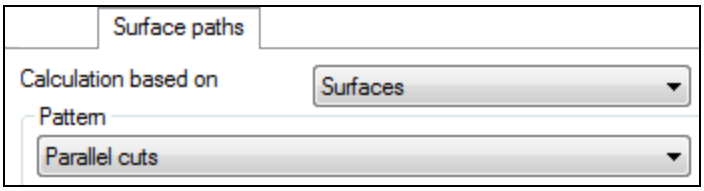
The toolpath is now evenly spread over the drive surface.

QUICK TUTORIALS

4-Axis Machining Compressor

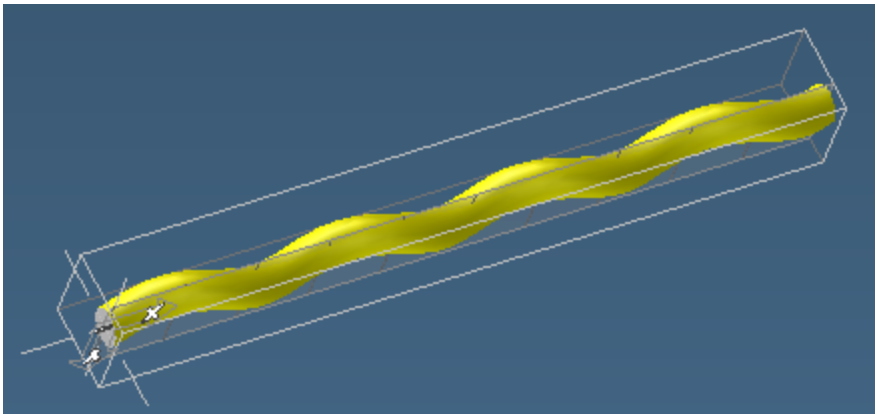
This example shows how to do a 2 axis + rotary toolpath that will follow the surface of a part rather than the just the rotary axis. Typically, a 4-Axis toolpath will only follow the rotary and the length axes during a feed. We will now demonstrate a feed move that uses depth moves during rotary cutting.

1. Open the file `4 Axis Compressor Armature.vnc`.
2. Create a 5-Axis process  with the tool.
3. Go to the Surface paths tab and select Parallel cuts as surface path pattern.

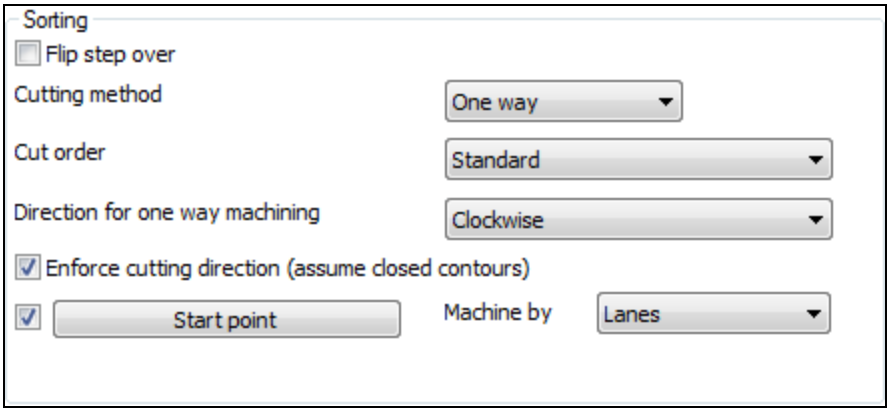


4. Set Machining angle X,Y to `0` and Machining angle in Z to `+90`.
5. `Click` the Drive surfaces button. 

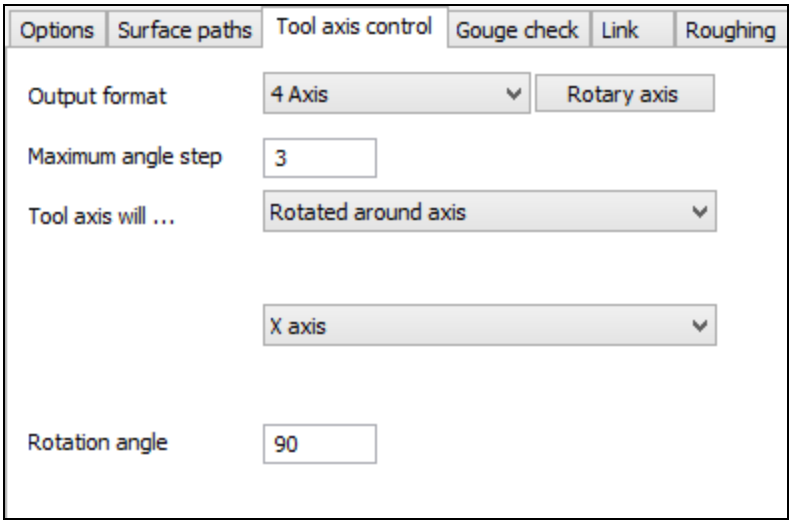
6. Select the two arm surfaces.



7. Set the cutting method to One Way.

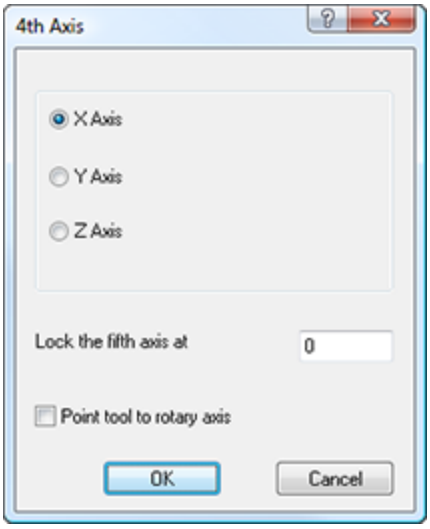


8. Set the Maximum stepover to 10mm.
9. Now switch to the Tool axis control tab.
10. Change the output format to 4 Axis.



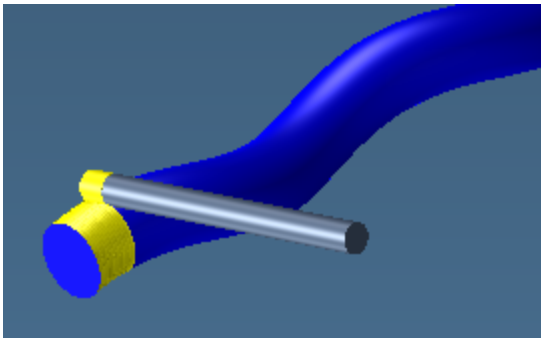
11. Click the Rotary Axis button 

and declare the X axis as rotary axis.



This should be the default.

12. Set the tool axis to Rotated around axis.
13. Define the Rotation angle as 90° around X Axis.
14. Create the toolpath and render.



The tilted tool rotates around the X axis and moves in 2 axes during the rotation.

5-Axis Engraving

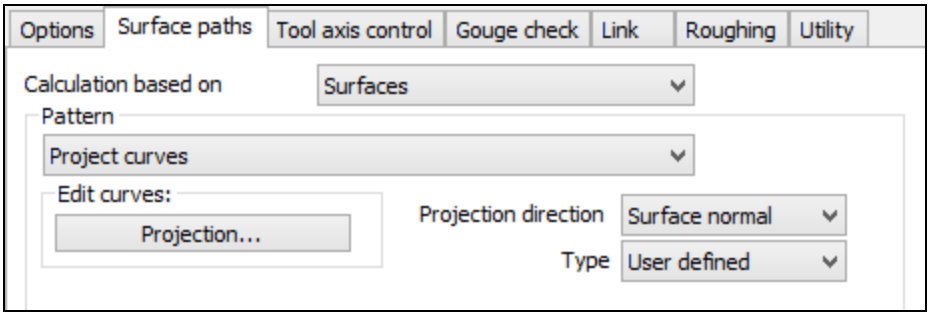
This exercise will demonstrate how to do 5-Axis engraving using the surface normal to determine the tool orientation.

1. Open the file 5 Axis Engraving.vnc.

   We will engrave the geometry on the scissor handle.

2. Create a 5-Axis process  with the tool.

3. Go to the Surface paths tab and select Project curves as the pattern.



We will use the lettering for the projection curves.

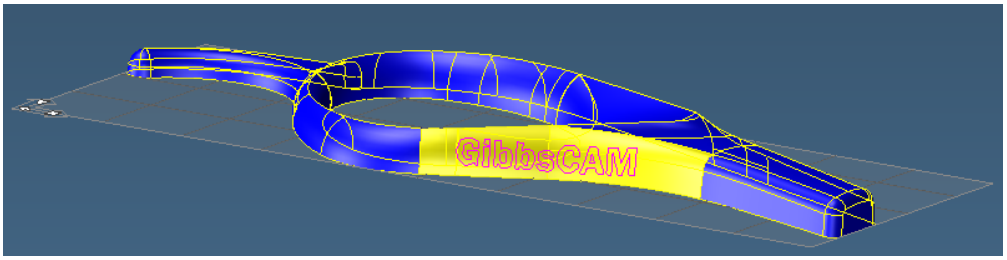
4. Click the Projection button. 

5. Select all the geometry for the letters and click OK.

Turning off Show Solids  and using a (Shift+drag) marquee selection is the easiest way to select the text.

6. Click the Drive surfaces button. 

7. Select the 4 faces under the geometry.



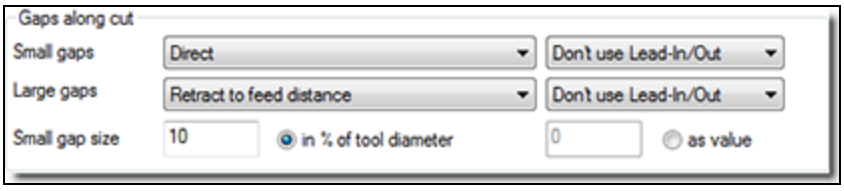
8. Go to the Tool axis control tab.

9. Set the tool axis to Not be tilted and stays normal to surface.

Using the surface normal the tool will always be certain to have a uniform cut for all letters.

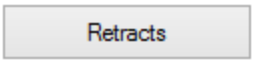
10. Go to the Link tab.

11. Set the Large gaps to Retract to feed distance.

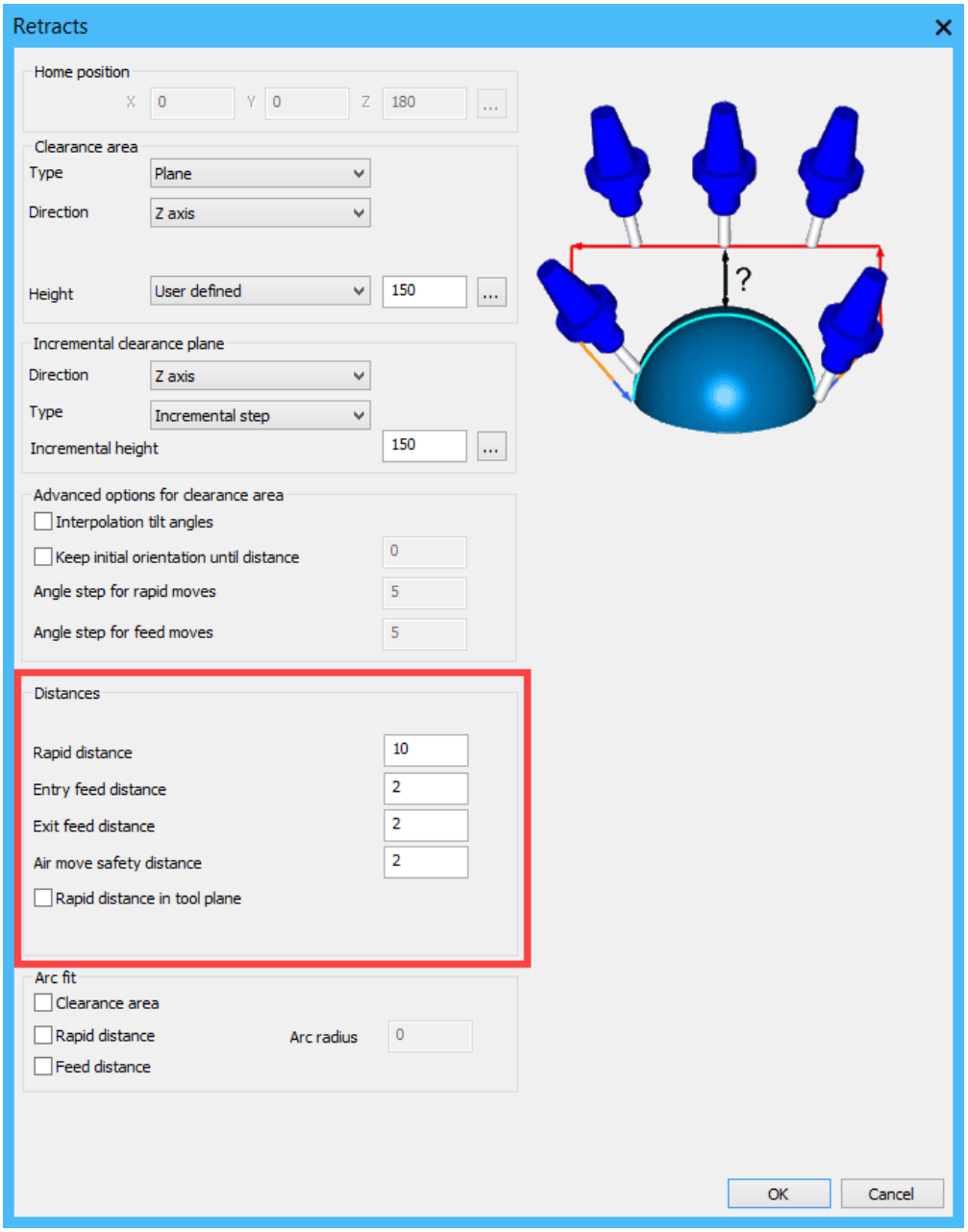


12. Set the Small gap size in % of tool diameter to 10%.

13. Set the Small move size in % of tool diameter to 50%.

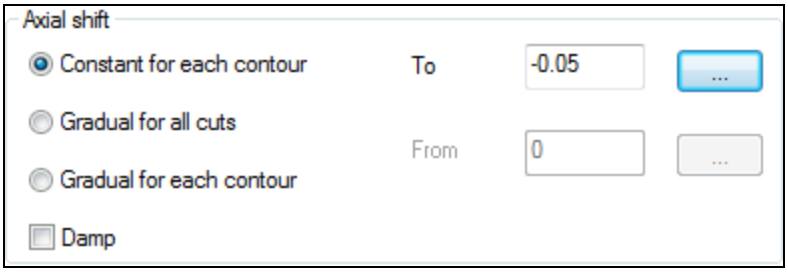
14. Open the Distances dialog at the bottom of the link tab. 

15. Enter the values shown.



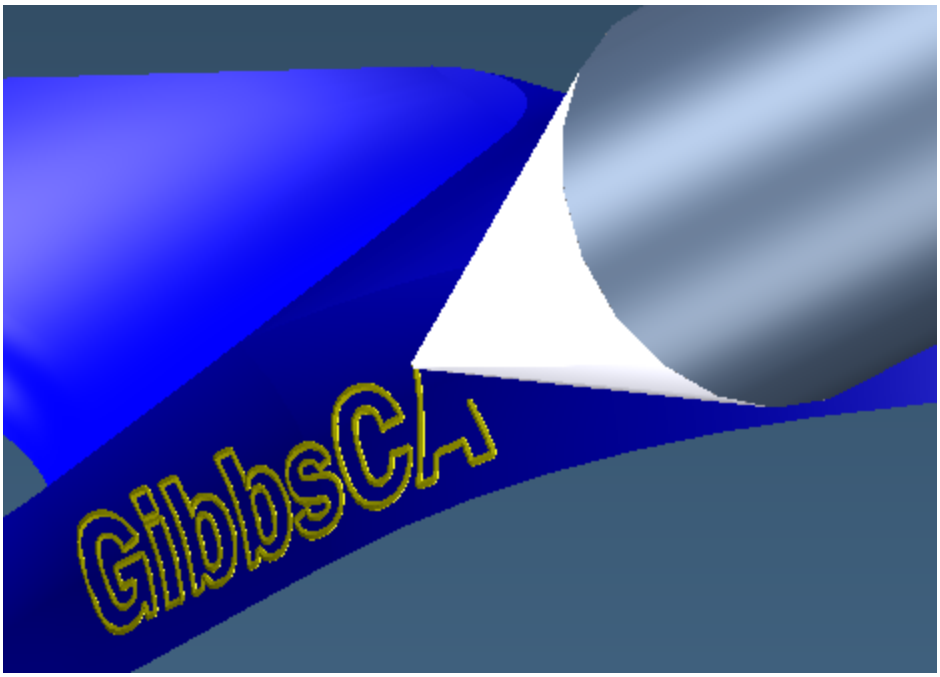
16. Go to the Utility tab.

17. Enter `-0.05`mm as the Axial Shift value.



This is the depth of the engraving.

18. Create and render the operation.

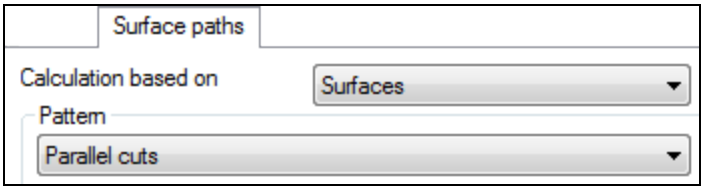


Camshaft

This example will demonstrate how to machine a Camshaft on a 5-Axis lathe.

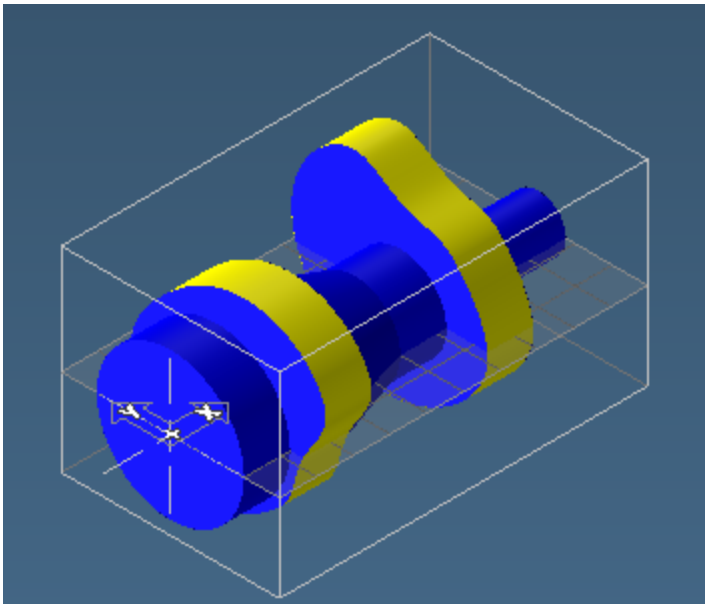
1. Open the file `camshaft.vnc`.
2. Create a 5-Axis process  with the tool.

3. Go to the Surface paths tab and select Parallel cuts as the surface path pattern.



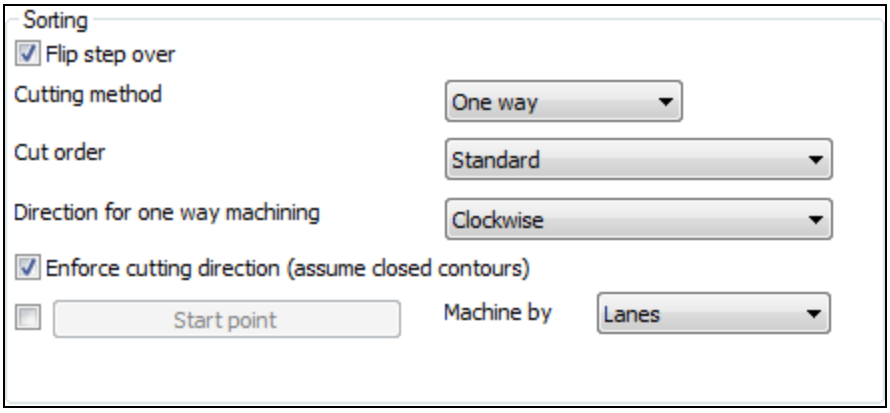
4. Enter `0` as the Machining angle in X,Y.
5. Enter `-90` as the Machining angle in Z.
6. `Click` the Drive surfaces button. 

7. Select all the tangent faces of the surfaces shown.



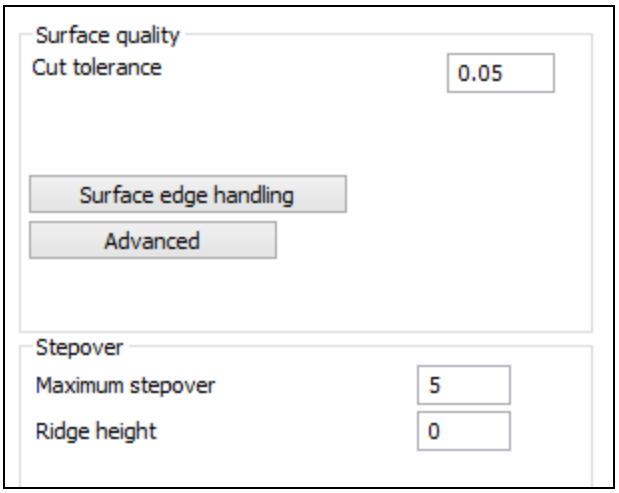
Using the Select Tangent Faces command from the context menu allows you to select these faces without having to change the view.

8. Now set the cutting method to One Way.
9. Enable the Flip step over option.

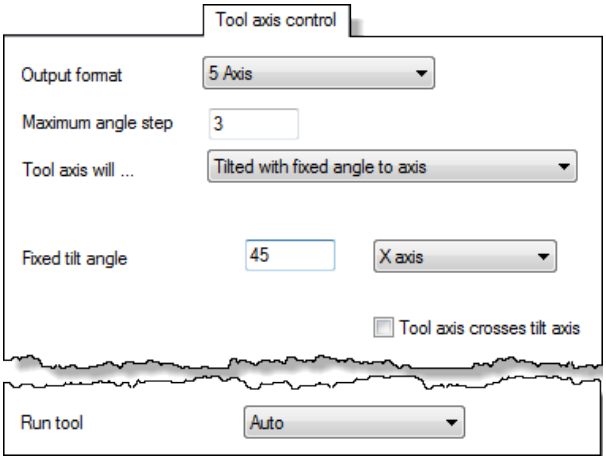


This lets the machining start from the negative side.

10. In Surface quality set the Cut tolerance to 0.05mm.



11. Set the maximum step over to 5mm.
12. Activate the Maximum distance checkbox and change the value to 3mm.
13. In the Tool axis control tab set the tool axis to Tilted with fixed angle to axis.

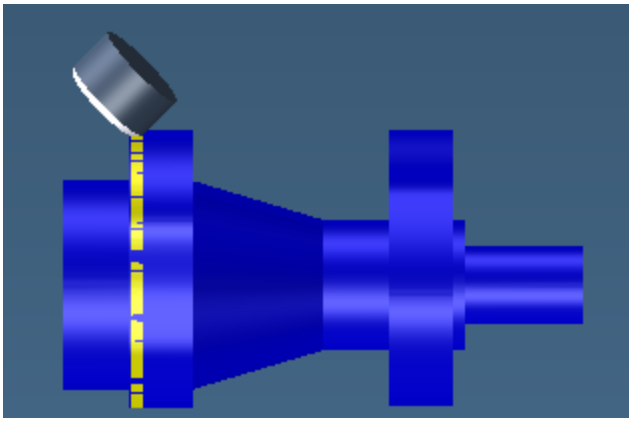


The fixed tilt angle value describes how much the tool axis direction should be tilted from the defined tilt axis direction. In other words, imagine a plane spanned by the surface normal and the tilt axis. The tool direction vector will rotate on this plane from the tilt axis direction to the surface normal direction about the fixed tilt angle.

14. In this case we set 45° to the X Axis.

    We will let the system decide where the contact point between tool and surface is by using the default Run tool to Automatic.

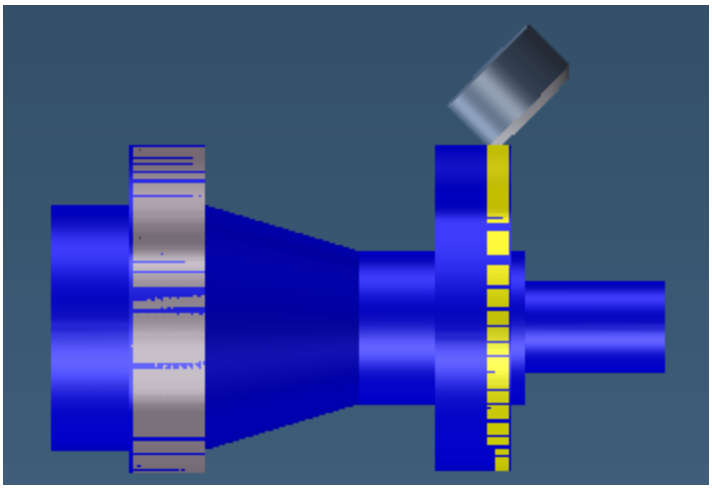
15. Create the toolpath.
16. Render the operation.



In this case both cams are machined with the same tilt angle. In some cases you may need to angle the other side because of machine constraints. We will control the tilt of the tool on the other cam in the next step.

17. Deselect the tangent faces of the second cam surface and redo the operation.
18. Deselect the current operation in the operation list.
19. Go to the Tool axis control tab and change the Fixed tilt angle to 135.
20. Reset the drive surfaces to the tangent faces of the second cam.
21. Deactivate Flip step over in the Surface paths tab.

    This defines that the machining will move from positive to negative.
22. Create the new operation and render.



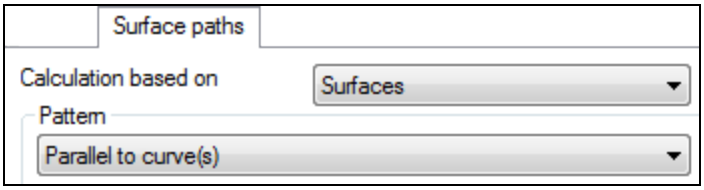
Notice the tool tilt angle is the opposite of the first operation.

Vacuum Formed Part Finishing

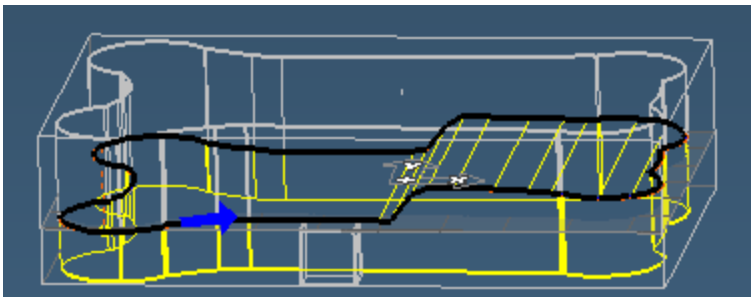
This is an example of finishing a vacuum formed part flange. Normally this would be trimmed by hand, but in the case of this part the complexity of the bottom edges make using a 5-Axis machine perfect for this situation. The part is slipped over a fixture on the machine so cutting multiple parts is as simple as running the program and slipping the part off and adding a new one. This makes the option of using a precise process with a CNC much more accurate for multiple parts instead of cutting them by hand.

The real part is complex with contours on the top. Since we do not need to use anything in the model except for the bottom edge we have simplified the part for machining purposes.

1. Open the file vacuum_form.vnc.
2. Create a 5-Axis process  with the tool.
3. Go to the Surface paths tab and select Parallel to curve(s) as the pattern.



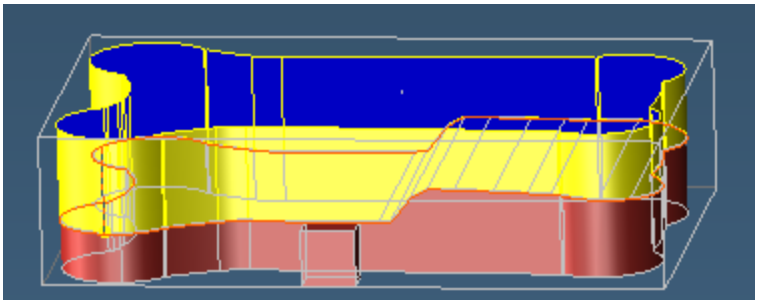
4. Click the Edge curve(s) button. 



5. Select the Exterior edge on the part using 3D Chain as the Curve geometry and click OK.

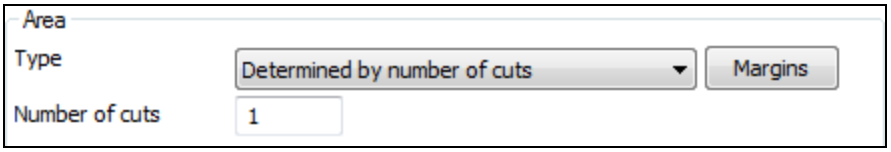
   The Wireframe view shows the edge clearly.

6. Select the Wall faces as the Drive surfaces as shown. 



   A cutting depth may be applied by entering a negative Drive surfaces clearance value. In our example we will keep this value at 0.

7. Change the area type to Determined by number of cuts with 1 cut.

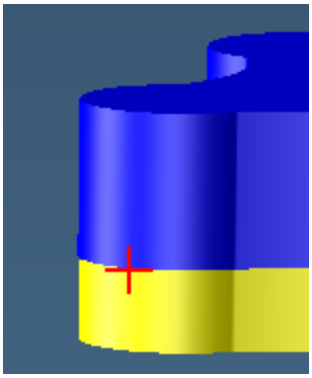


8. Check the Start Point option and click it. 

   We will select a point on the drive surface by clicking to interrogate the point.

9. Click the ellipsis button  and on Set point by Position, click the  button again.

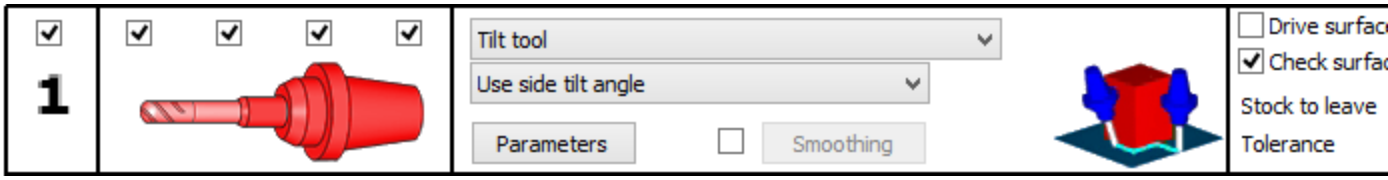
10. Click the ellipsis  button and select the point close to the position shown. Click OK.



11. Go to Tool axis control tab.
12. Change strategy to Be tilted relative to cutting direction.
13. Enter 10° as the Tilt angle at side of cutting direction.

    We use this 10° tilt to keep the tool holder and spindle away from the table. We will now set up gouge checking.

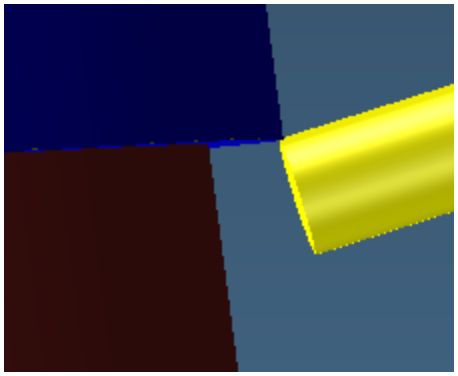
14. Go to the Gouge check tab and activate Status 1.
15. Check against all parts of the tool and holder.
16. Select Tilt tool from the strategy menu.
17. Set the Use side tilt angle and to 30°.
18. Enable gouge checks against Check surfaces and set a 2mm Stock to leave.

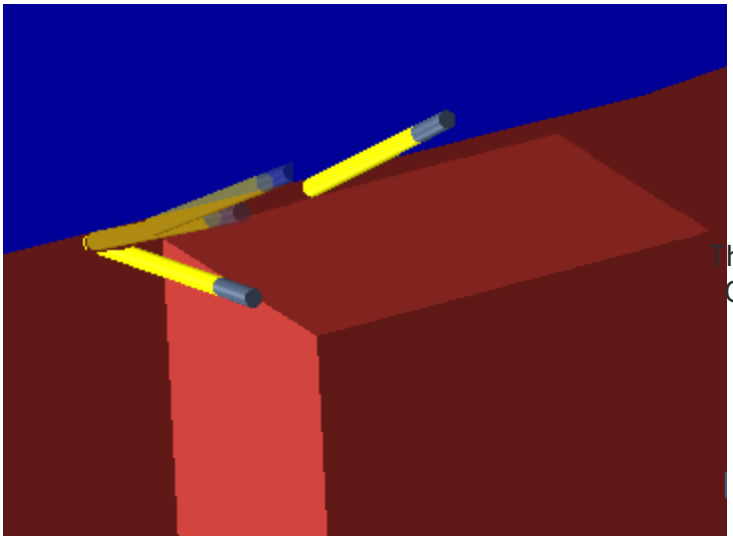


    The part has a simulated clamp as a fixture to protect against gouges.

19. Click the ellipsis button  and select the Clamp body fixture as the Check surface.
20. Click the Clearances for tool parts button at the bottom of the dialog. Change the Shaft value to 2mm.
21. Create the toolpath.
22. Render the operation.

You can see when looking closely at the rendering that we will be cutting any flange from the part that may exist and also remove the sharp edge from the bottom.





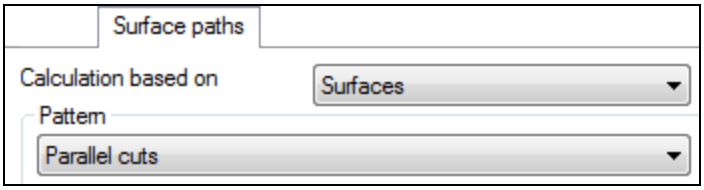
The tool tilts even more around the Clamp body to prevent a gouge.

Electrode

This example demonstrates how to machine an electrode using 4-axis plus one axis a with fixed tilting angle. The machine should rotate only its C-axis and move in Y.

> ⓘ You often use this type of operation to run a 4-axis machine with a manual adjustable head.

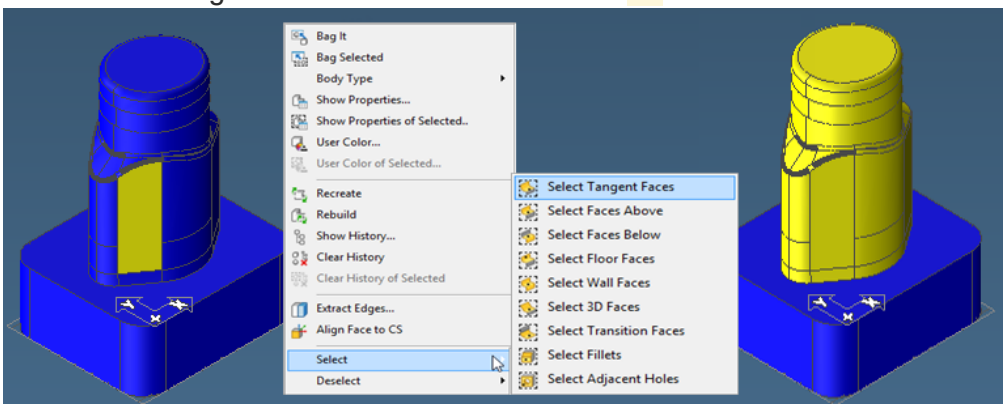
1. Open the file `Electrode.vnc`.
2. Create a 5-Axis process  with the tool.
3. Go to the Surface paths tab and select Parallel cuts as the surface path pattern.



4. Click the Constant Z button. 

5. Click the Drive surfaces button. 

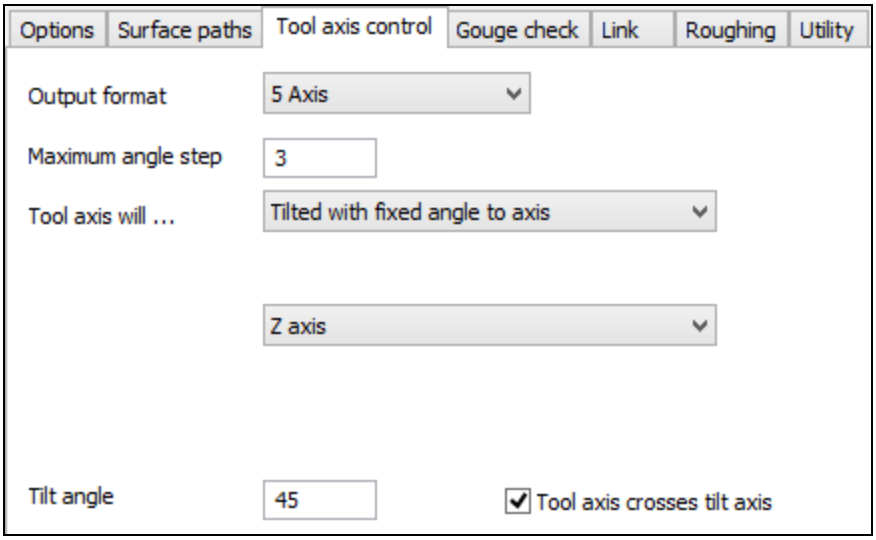
6. Select all the tangent faces of the boss and click OK.



7. Set the cutting method to One Way.

8. In the Stepover settings set Maximum stepover at 2mm.

9. Open the Tool axis control tab and set the tool axis to Tilted with fixed angle to axis.



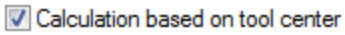
10. Change the Fixed tilt angle to 45° in the Z Axis.

11. Check the Tool axis crosses tilt axis option.

    This will force all Y moves to be C rotation moves. The result is an output that contains X and C moves.

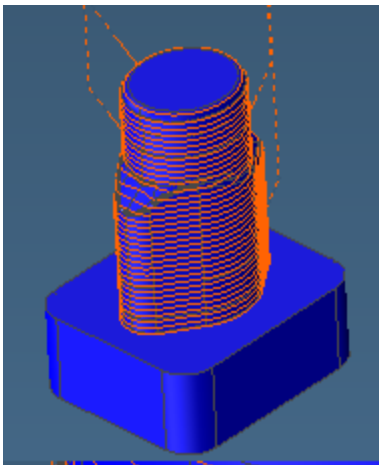
12. Go to Link tab and set the Links between slice > Small move size to 300 in % of stepover.

13. Go to the Utility tab and check the Calculation based on tool center option.

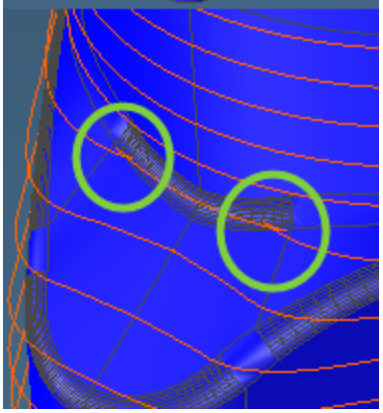


In the case of Z constant cuts, this function affects the position of the tool center height. The system keeps the tool center point on the same level during machining, independent from the surface shape, so the contact point of tool and surface vary in their height. The tool height position remains constant.

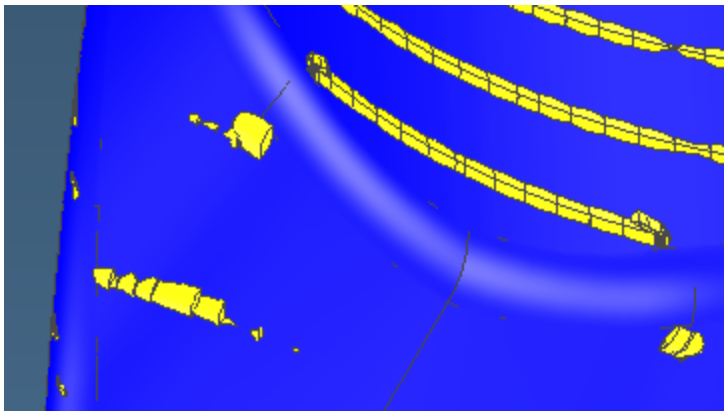
14. Create the toolpath.



After a quick look at the results, the toolpath seems to be all right. But looking closer, you will find small mistakes in the sharp inner corners. These corners cause a problem known as “fish tail”. This occurs at fillets that are smaller than the tool radius.



When rendered this tool move will gouge the part. As you can see in the picture, for example, the gouging happens when the tool steps over to the next slice. The “fish tail” moves will need to be removed.



15. Go to the Surface paths tab.

16. Check Corner Cleanup.

Normally Corner Cleanup will change small radius and inner sharp edge moves in the toolpath. Such areas will be left out from toolpath generation.

The gouge problem can be solved in two ways. We will apply both options for best results. Changing retracts between cuts in the Link tab and using options from the Gouge Check tab.

17. Go to the Link tab again and look at Links between slice setting.

    The tool is gouging because we are using a Direct move to the next slice.

18. For this surface, set the Links between slice Small and large moves to Follow surfaces.

19. Now switch to the Gouge check tab.

20. Activate Status 1.

    Check only against Flute (tool shaft, holder front and holder back are not necessary here).

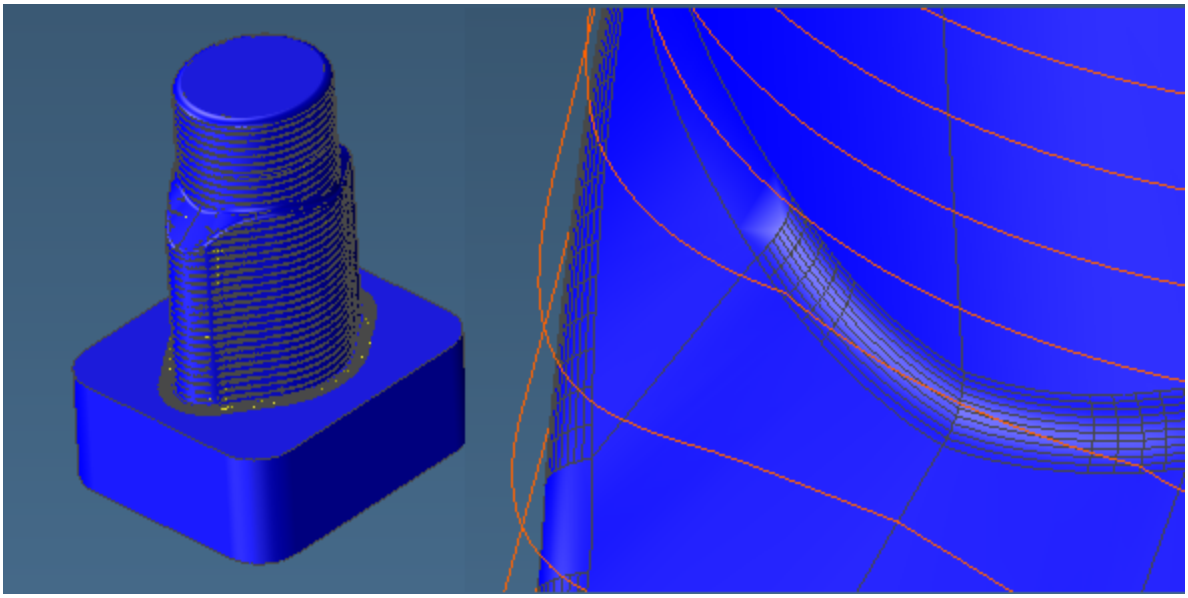
21. Set strategy to Retracting tool along tool axis.

    We do not want to change the tool axis orientation or move the tool away. Instead, this option will only retract the tool along its axis when a gouge is encountered.

22. Check Drive surfaces.

We do not have any check surfaces.

23. Redo the operation and render.



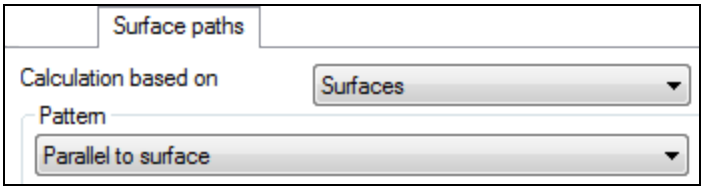
You can see that the fish tails have disappeared and the tool does not gouge the surface and follows the surface while retracting along its axis.

Pencil Trace with Tilt Curve

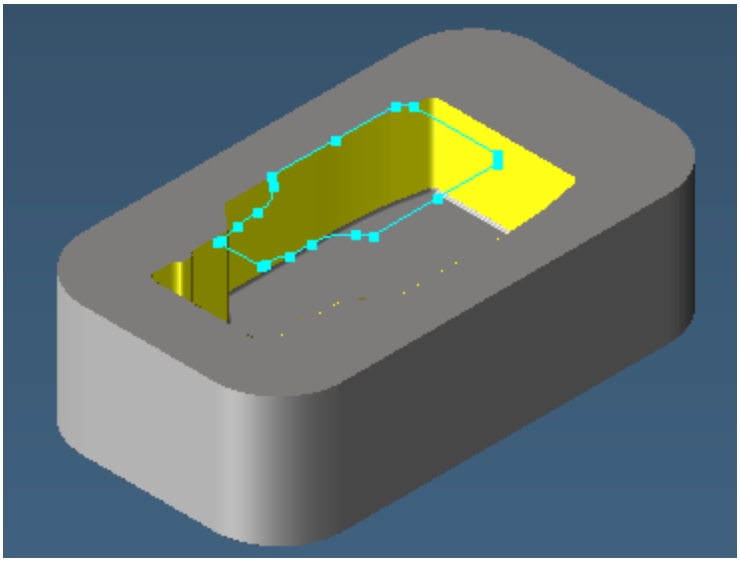
This is an example of doing a Pencil trace in a deep pocket using a Tilt curve to guide the tool axis. This is specifically useful when machining floor fillets with small tools.

We have completed the initial cuts of this part and only require the final cleanup of the corners.

1. Open the file Mold_Bottle.vnc.
2. Create a 5-Axis process  with the 3mm Ball EM (tool #4).
3. Switch to the Surface paths tab.
4. Select Parallel to surface as the Pattern.

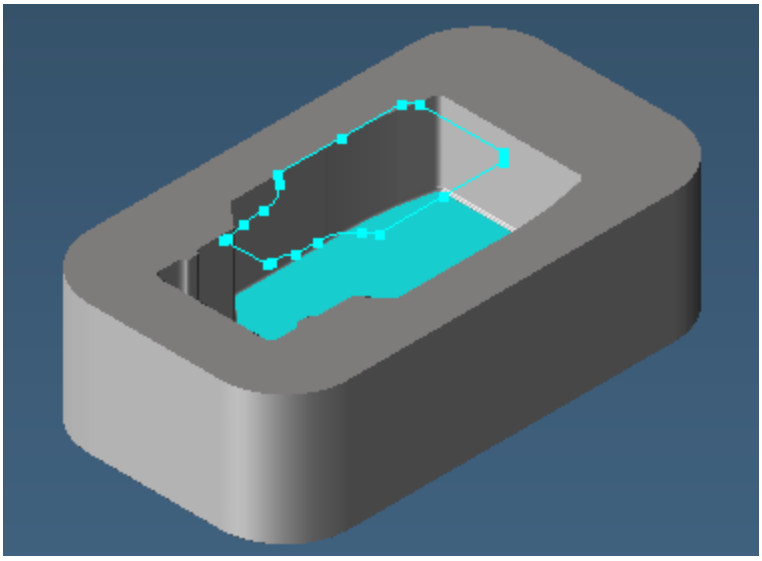


5. Select the inner wall faces as shown.

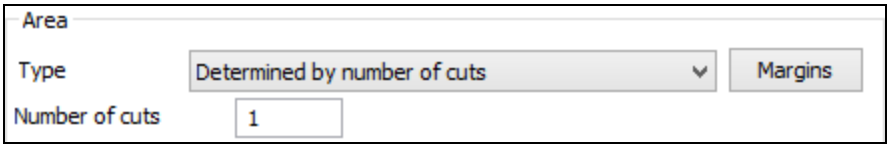


6. Click the Single edge button 

select the bottom face and click OK.



7. Now set the Area >Type to Determined by number of cuts with 1 cut.

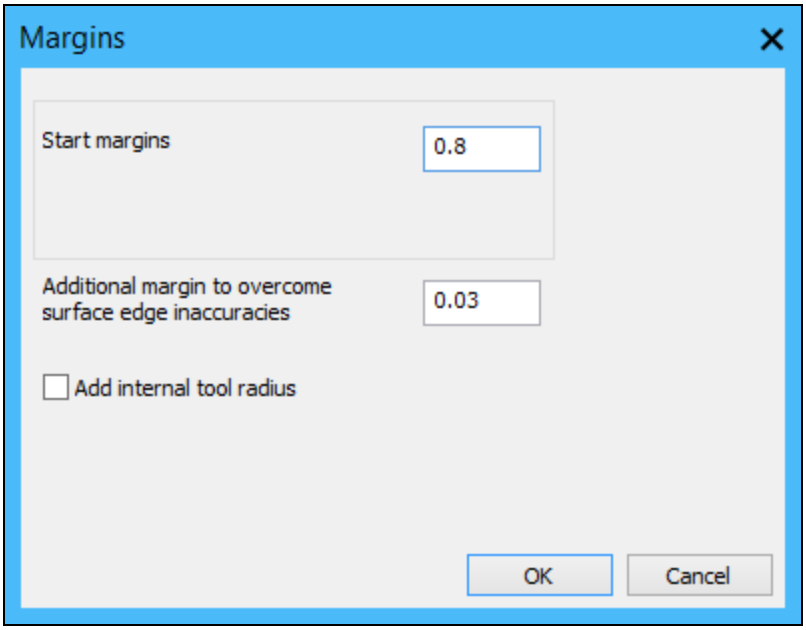


The first cut is at the exact edge, but can be shifted with a margin.

8. Click the Margins button 

to set up the margin.

9. Enter the tip radius of the tool as the margin which is `0.8` mm and `click` OK.

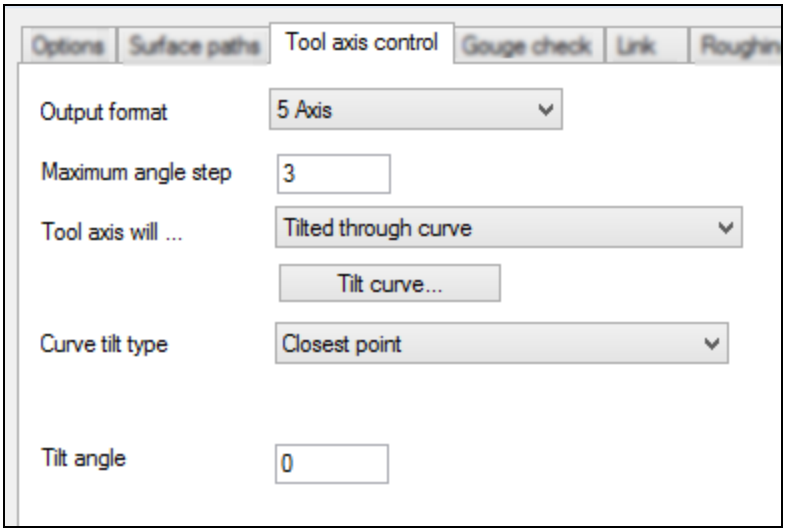


When using margins it is more common to calculate the toolpath based on the tool center.

10. `Check` Corner Cleanup.

11. Go to the Utility tab and `check` Calc based on tool center.

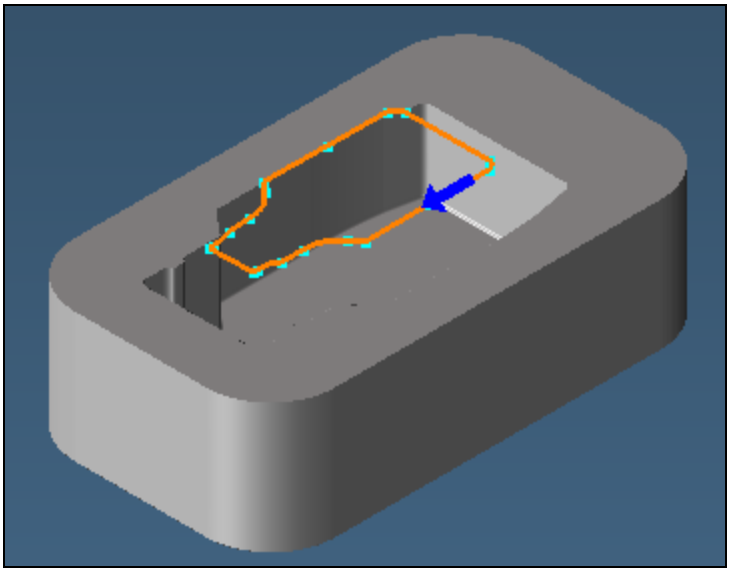
12. In the Tool axis control tab set the tool axis to be Tilted through curve.



13. Set the Curve tilt type to use Closest point.

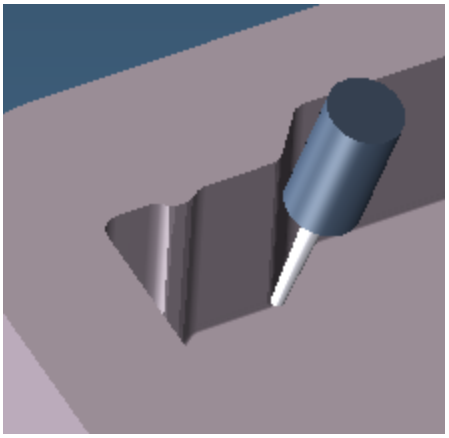
14. Click the Tilt Curve button, 

select the geometry shown and click OK.



The geometry for the curve is an offset of the mold and above the cavity. This curve will allow the tool axis to tilt away from the faces so that it will not gouge the walls.

15. Create the toolpath and render.



You see the single cut. During machining the tool is always tilted through the curve, away from the drive surface.