GIBBSCAM® 14

Version 14 : September 2020

Radial Milling Tutorials

CAMBRIO

Proprietary Notice

RADIAL MILLING TUTORIALS

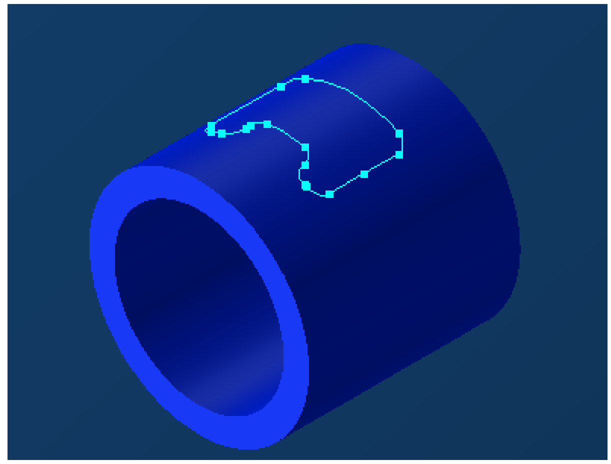
These short tutorials provide an introduction to using the Radial Milling module. As with most GibbsCAM tutorials, the parts are in metric units and are set up with an Aluminum Alloy as the Material type.

The tutorials assume you have existing knowledge of GibbsCAM Mill machining. The parts are already created and make extensive use of solids for the stock shape and are simplified to focus on learning how to use Radial Milling.

In some of these exercises we extract geometry from solids. If you do not have any solids capabilities you may open the completed part files to work from geometry we have already extracted.

BASIC USE

We will use a single closed shape in order to create three contour operations and a pocketing op. Three separate contour ops will be created to show how the various settings affect the toolpath.



1. Open the part Basic Use.vnc that is in the Sample Parts/Radial Milling - 4-Axis/Tutorial Parts - Required folder that came with the software.

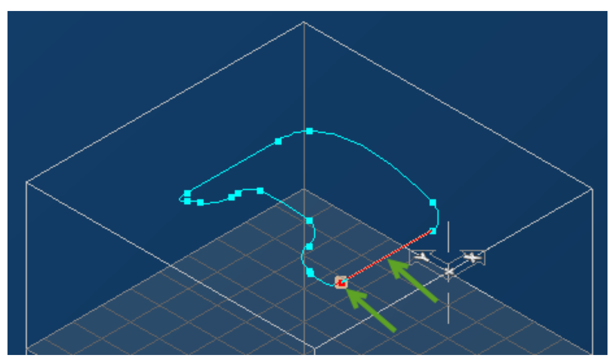
   Please note that the part geometry is not flat and not in a rotary mill workgroup. The geometry was created radially.

Op 1 - Basic Contour

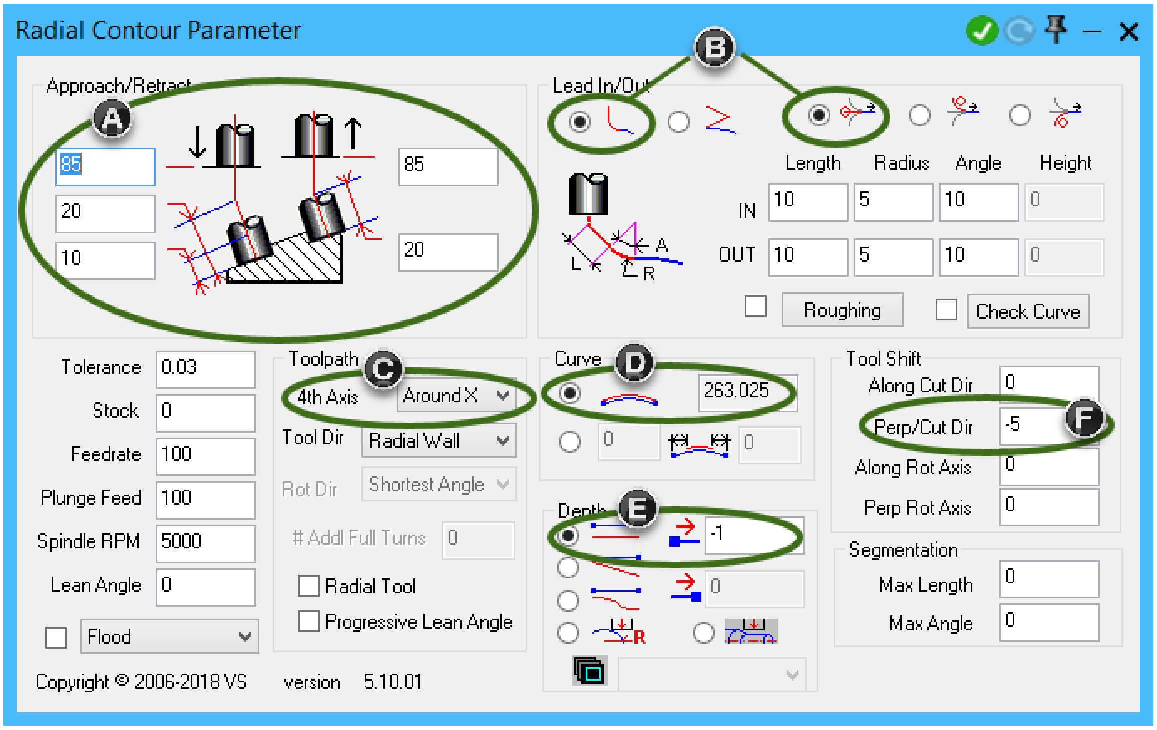
Radial Milling Contouring requires you to select a point for the start location and a feature that indicates the tool travel direction. As you select the geometry before creating the contour process, the system can calculate the total shape length. This can be very useful if you wish to enter setbacks for the toolpath. Setbacks start and finish a specified distance from the beginning and end of the shape.

1. Select point and line as shown.

   We will start by etching outside the shape, offset by 5mm.



2. Create contour process using Tool #1, the 13mm Spot Drill, using the parameters shown. Parameters that are particularly important in this situation are circled and explained below.



a. We have set the clearance plane to 85mm. The tool will rotate into position at 20mm above the part and rapid to 10mm above the part.

b. We are ramping onto the part on the center of the shape.

c. We are rotating about X.

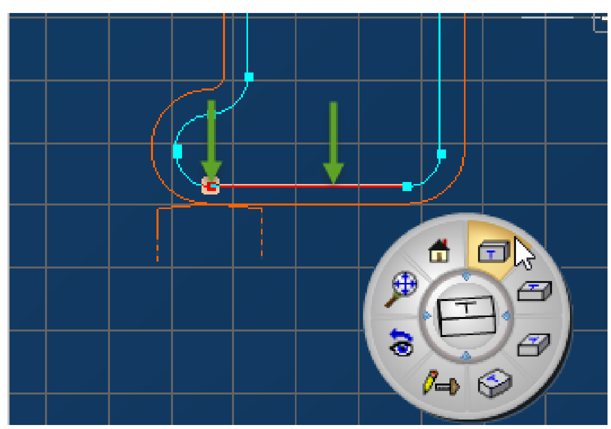
d. The full shape is being cut.

e. We are cutting 1mm deeper than the shape.

f. We are offsetting the tool by 5mm to the right of the shape.

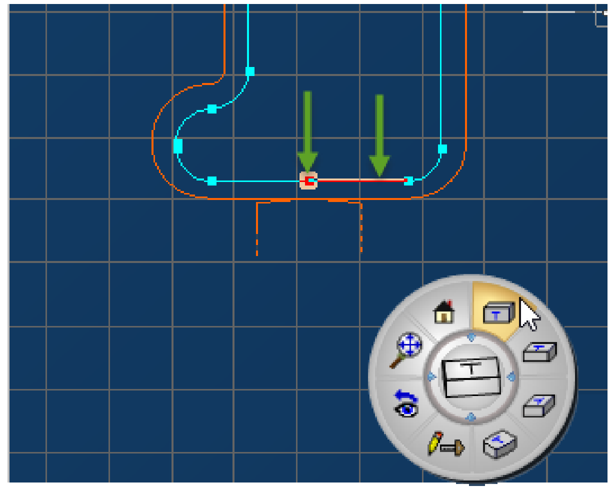
7. Click Do It ✓ to create the operation.

As it turns out, we do not want to start in the corner of the part so we need to modify the geometry to have a better starting point.



8. Select the line we started the operation on and create a center point on it. Be sure to connect the geometry so that the new point is part of the shape.

9. Select the new point and the line to the right of it. Redo the operation.
   Your results should look similar to this image.

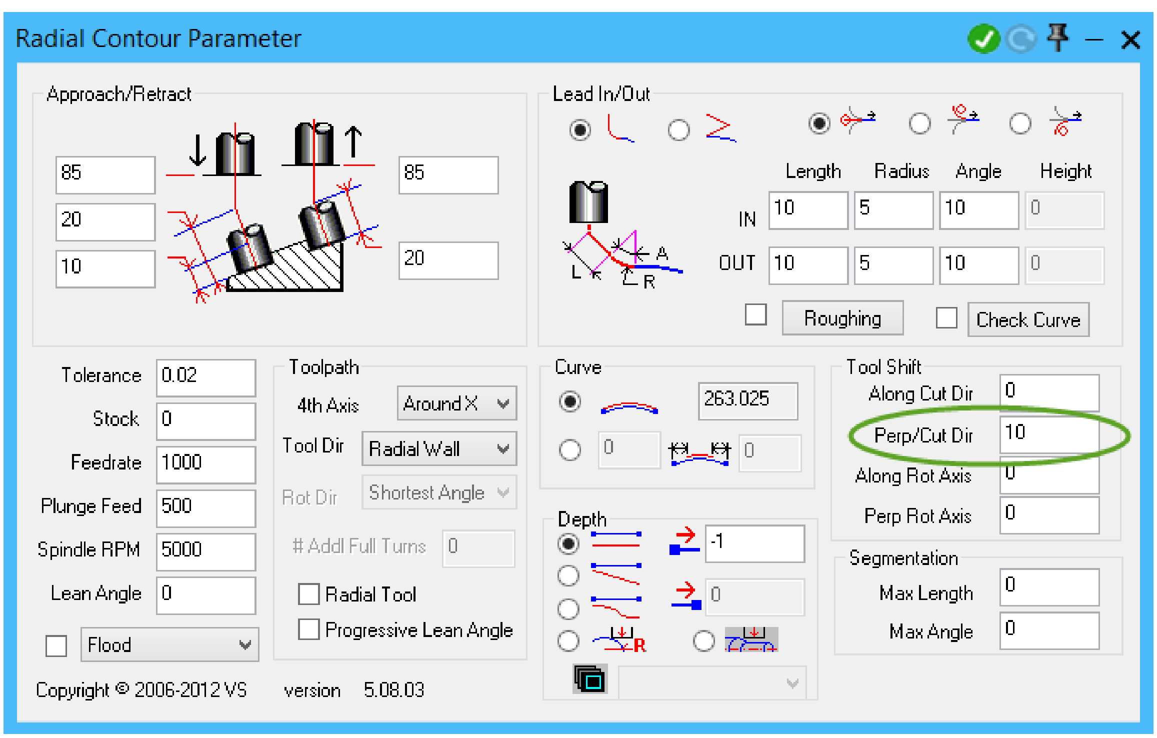


10. Deselect this operation.

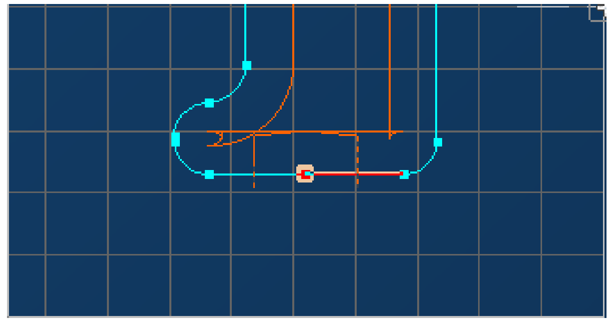
Op 2 - Inside Etching

We will now create toolpath that is offset to the inside of the shape. We will use the same values as the first operation except we will change a single value which will move the toolpath to the opposite side of the geometry by 10mm.

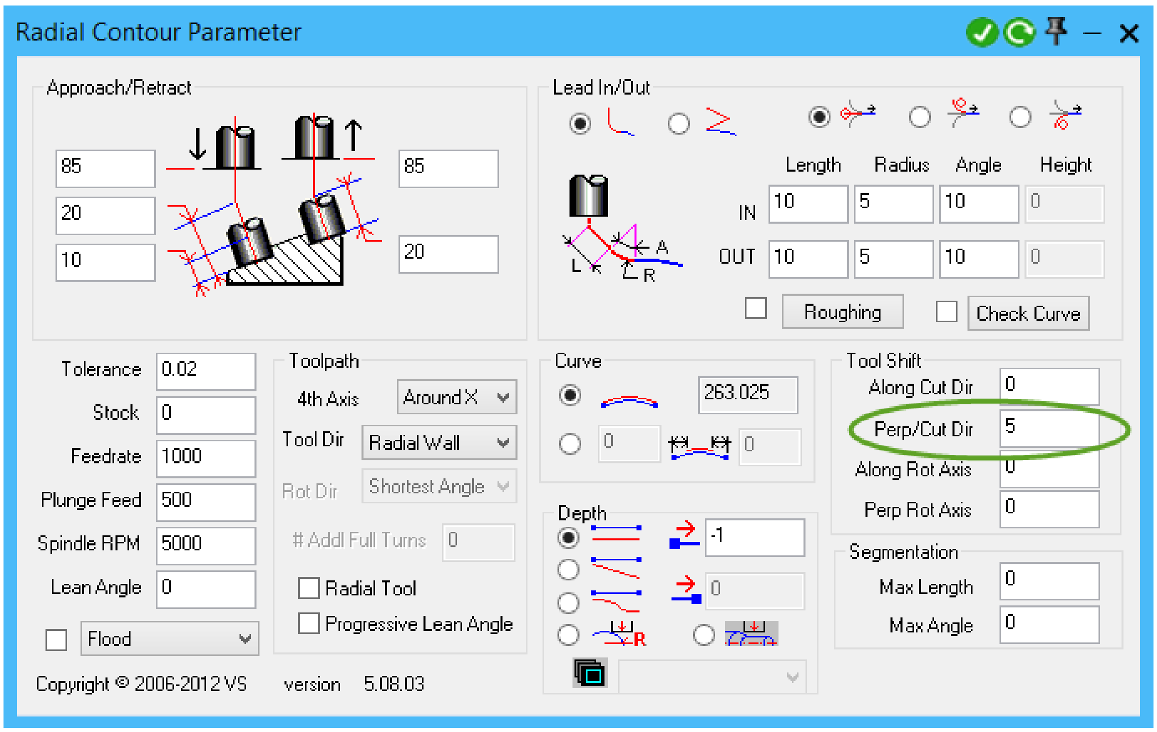
1. Create a new contour operation using the same setup as the first operation but change the `Perp Cut/Dir` value to `10`.

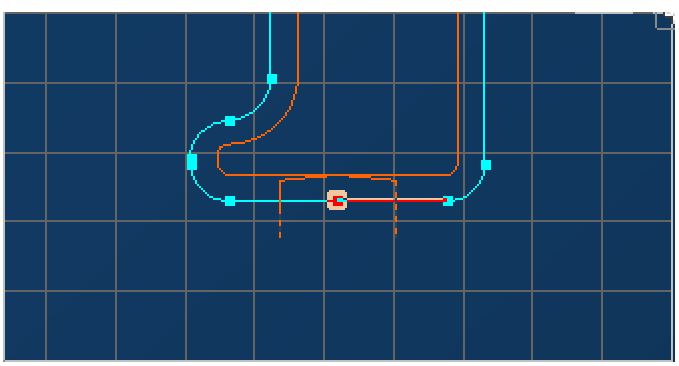


The results of this operation look rather odd. In fact, the toolpath is violating the shape to handle the rotations we require. The large offset value is bigger than the shape can handle. Let's change this as the 10mm offset was used as an example. Essentially we were forcing the tool into a place it could not fit.

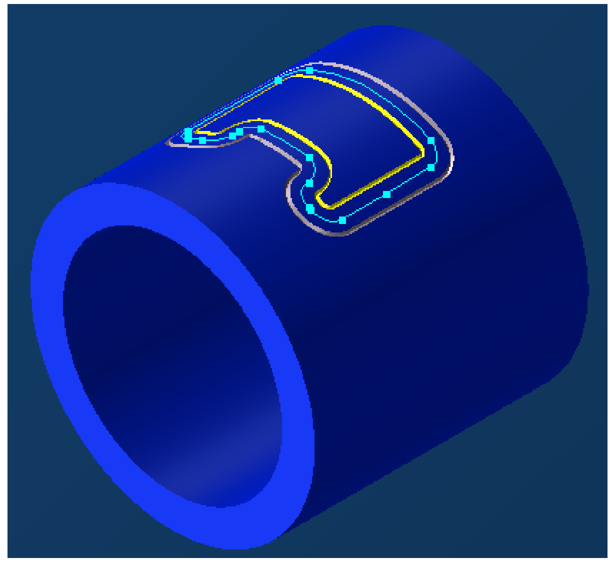


2. Change the Perp/Cut Dir value to 5 and redo the operation.





If you render the part at this point it should look like this image.

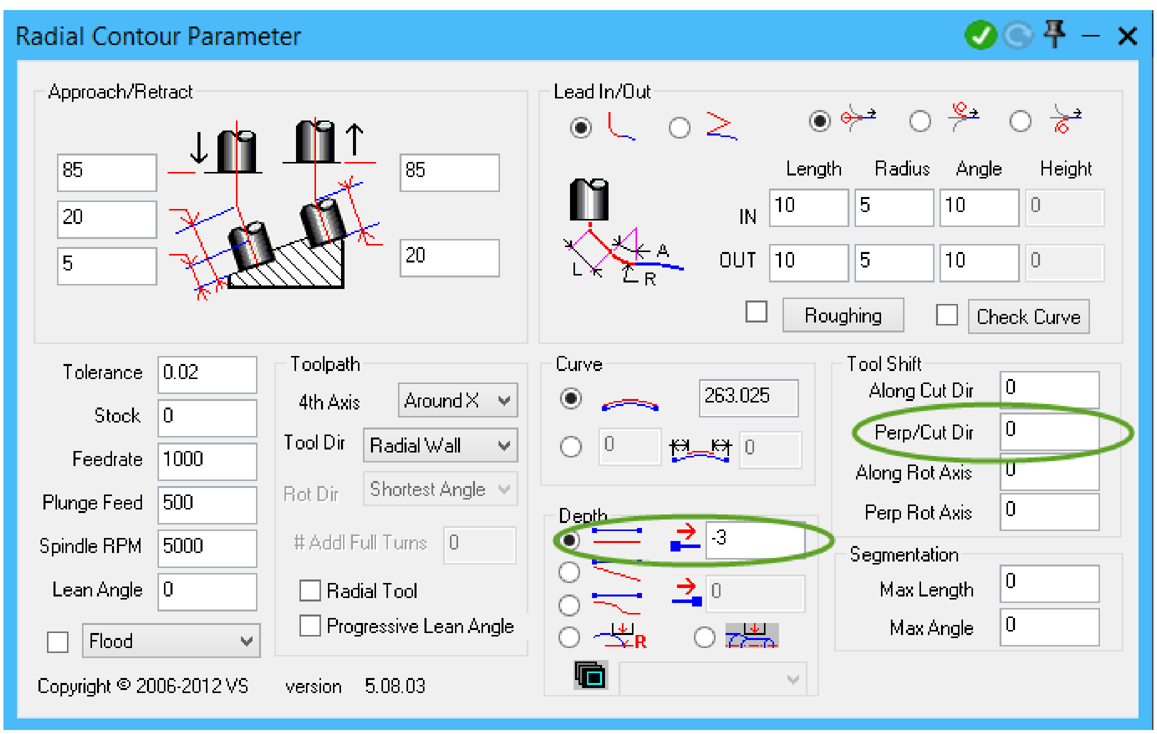


3. Deselect operation #2.

Op 3 - Contour On Center

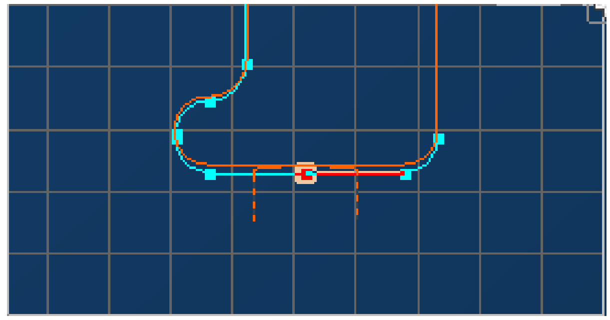
We will now machine a contour on the center of the geometry.

1. Create a contour operation using Tool #2, the 5mm Finish Endmill, using the parameters shown. Parameters that are important to getting this operation right are circled.



We have specified that we are going to machine on center (`0` `Perp/Cut Dir` value) to a depth of 3mm below the geometry.

The results of the operation should look like this image. This is the last of our contouring ops.



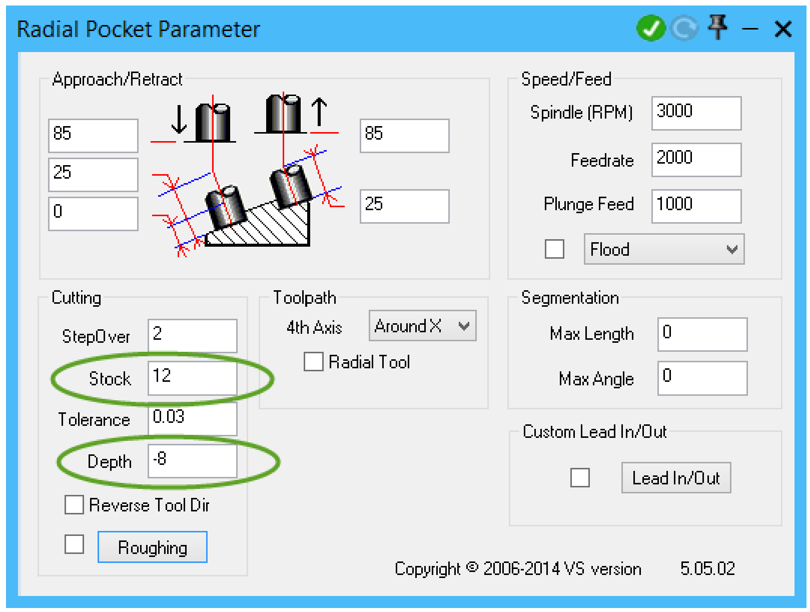
2. Deselect this operation.

Op 4 - Pocketing

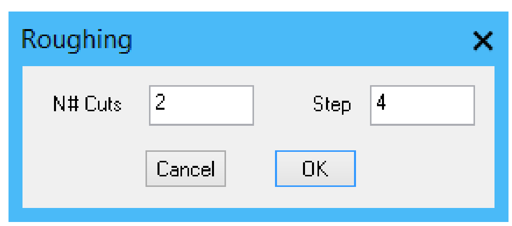
We will now pocket this same geometry.

1. Create a pocket operation using Tool #2 as shown.

   We are leaving 12mm of stock and cutting 8mm below the geometry.



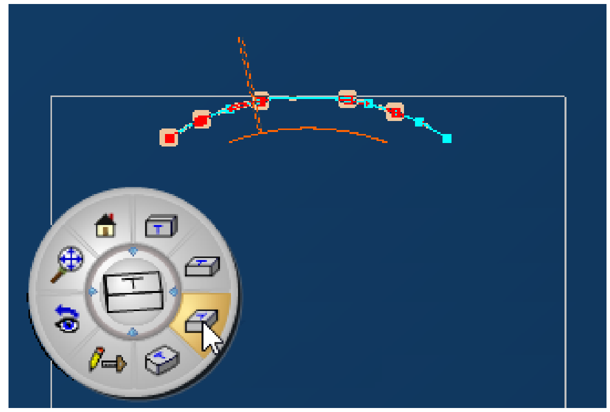
2. Click the Roughing button and set the values as shown. Then click OK.



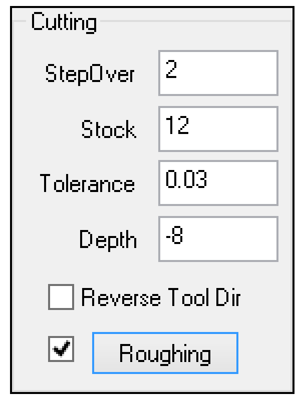
3. Click the Close button in the Pocket dialog to save the data and then click Do It to create the operation.

It seems we have made a mistake. The pocket is produced but only in one step.

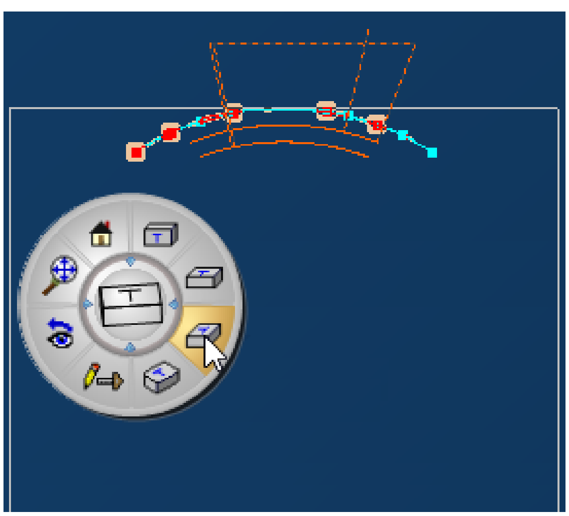
To get pocketing or contouring to actually make more than one pass you must set the values and also select the Roughing checkbox.

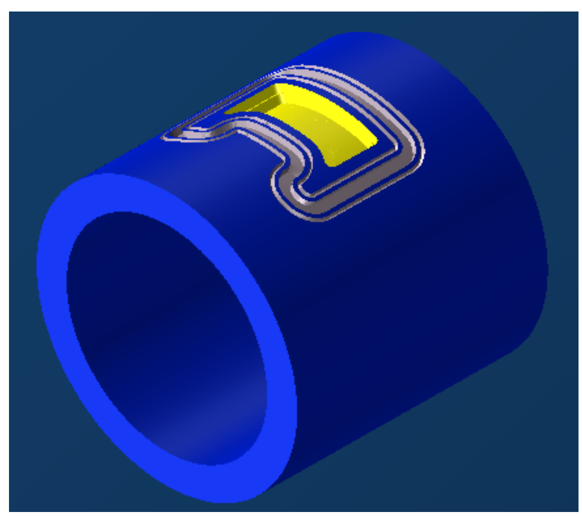


4. Open the Roughing process, select the Roughing checkbox and re-do the operation.



The results should look like this. The roughing op leaves an obvious ledge, as would be expected in a Radial Milling operation. A contour operation would be needed to clean this wall.





5. Save this file as it is complete.

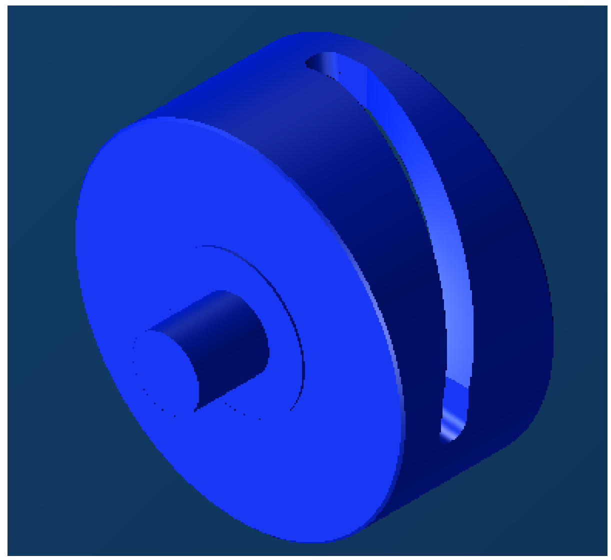
RADIAL TOOL ON A CAM

In this exercise we will force a tool to remain radial on a part.

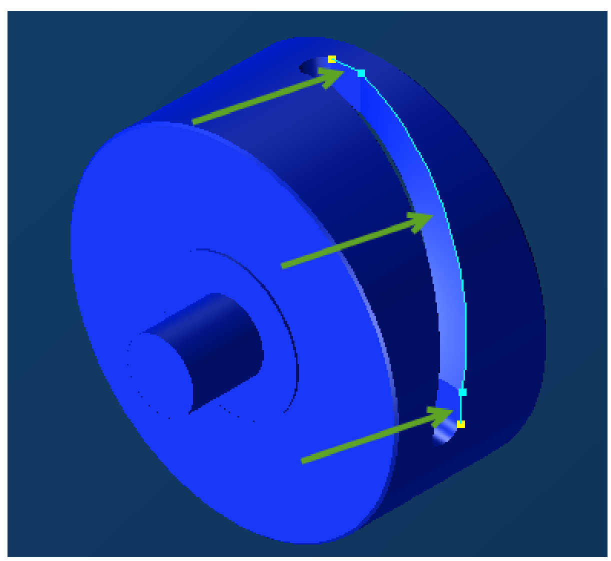
Getting the Geometry

1. Open the file Barrel Cam.vnc that is in the Sample Parts/Radial Milling - 4-Axis/Tutorial Parts - Required folder.

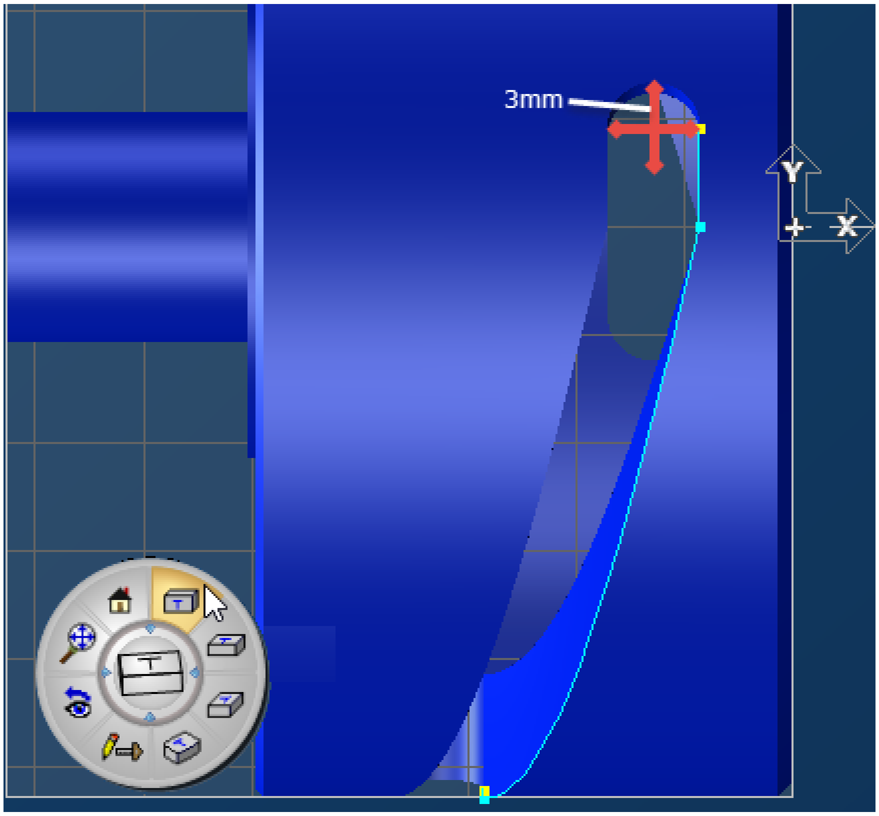
   This is a barrel cam with a 6mm slot. We are going to generate toolpath to cut this slot but the tool will remain radial as that is how the cam was designed.



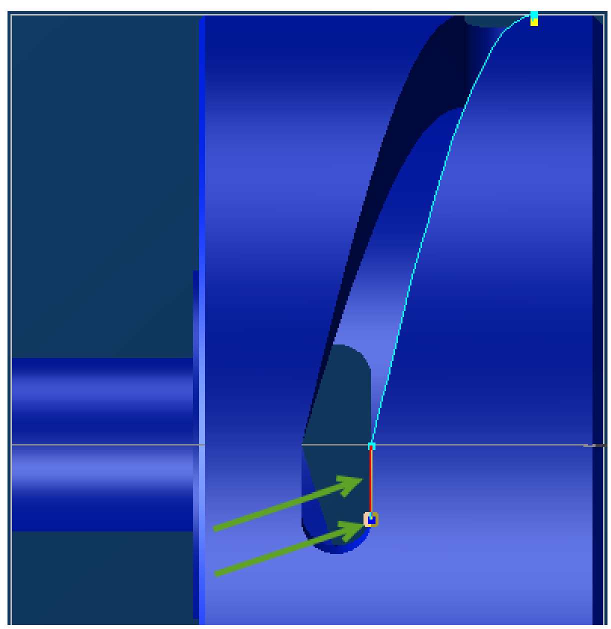
2. Extract edges to create the geometry shown.



Note that this does not include the rounded ends of the slot. You should select only three edges. We want to select the geometry that the tool will have to travel to create the shape. The end of the slot is a 3mm radius, the center of which goes through the center of the part. A 6mm tool can cut this slot and remain radial. This is illustrated in the following image which has all of the geometry extracted from the shape.

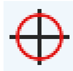


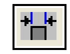
Machining the Slot



1. Select the lowest point and arc on the slot geometry.

   We do this so that the contour process knows the length of the geometry that will be cut. While we will not use this data in this operation, always selecting your geometry first is a good habit to form when using 4-Axis. We also need to know how deep to cut.

2. Select the  Show Position item from the Plug-Ins menu.

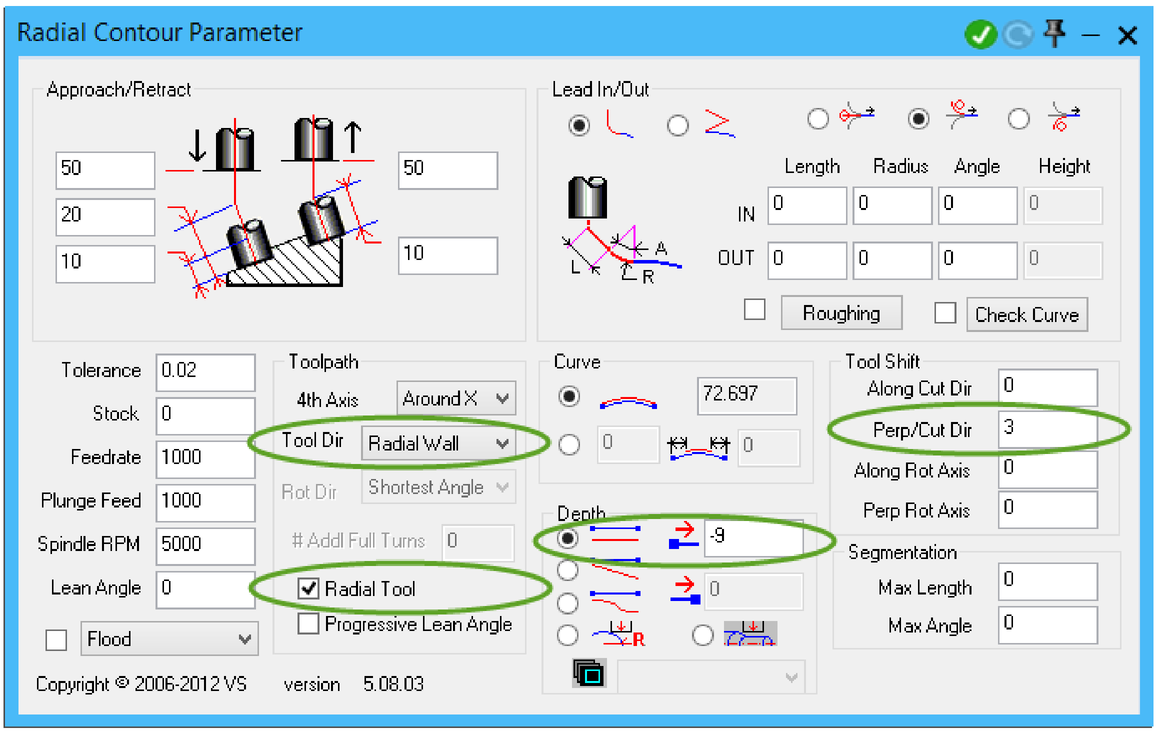
3. Switch the plug-in to determine thickness. 

4. Click anywhere near the slot on the curved surface of the barrel cam.

   This will display the thickness of the solid.

   We now know we need to cut at least 8mm deep.

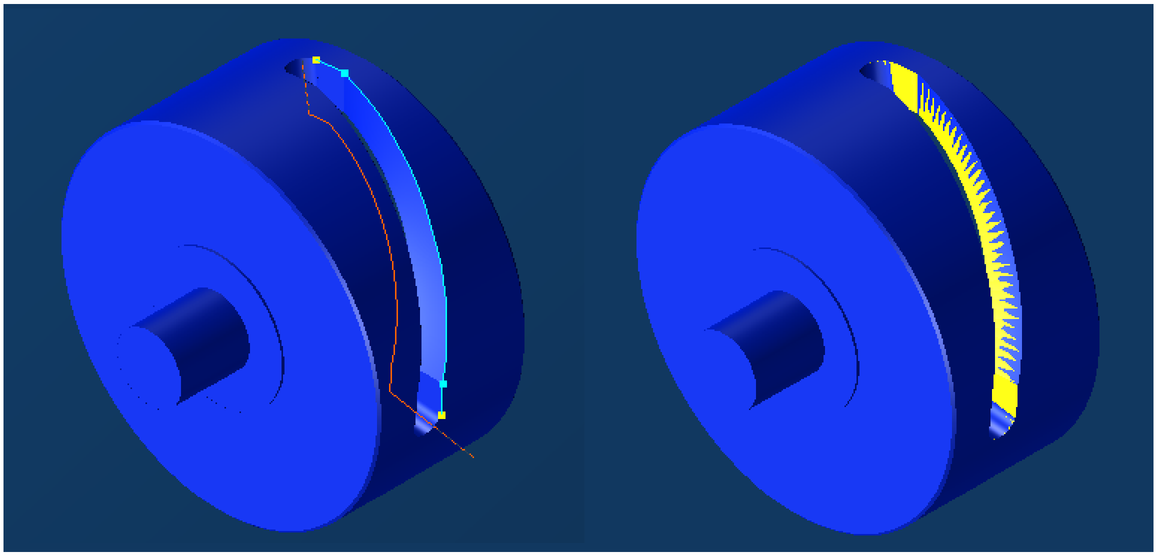
5. Create a Radial Milling Contour process as shown using the 6mm Finish Endmill.



We have set the process to force the tool to remain radial (the Radial Tool checkbox option). If we did not select this, the tool would simply follow the geometry and may generate moves in Y. We are offsetting the depth by -9mm. This will ensure the tool clears the bottom of the slot. We have also specified a Perp/Cut Dir value of 3mm. We are cutting a wall, so Perp/Cut Dir is a more appropriate choice than Along Cut Dir. The 3mm value moves the 6mm tool off the geometry by a radius.

6. Click OK to close the dialog and click Do It to create the Contour operation.

   The results should look like the following image.



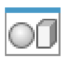
7. Save the part as it is complete.

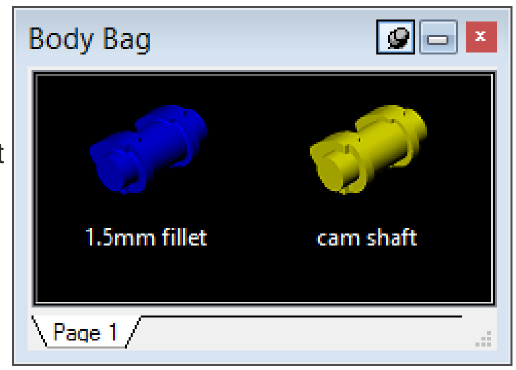
USING DIFFERENT GEOMETRY

This tutorial focuses on using the Constant Radius option and how the walls of solids can help control the toolpath.

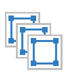
About the Part

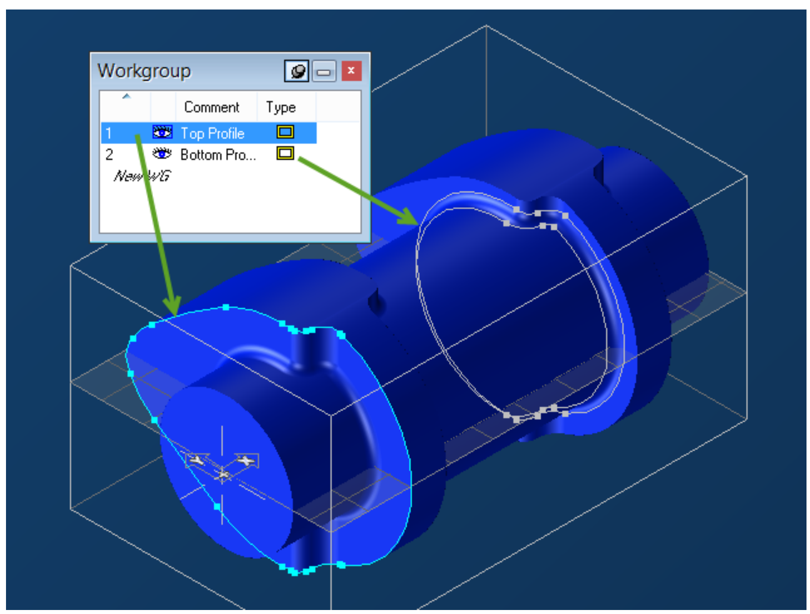
1. Open the file Cam Shaft.vnc that is in the Sample Parts/Radial Milling - 4-Axis/Tutorial Parts - Required folder.





We have two parts in this file. Both are the same cam shaft but one has fillets (representing the final cut shape) and the other does not have fillets. The body with fillets is set as the stock shape. We have created two separate models to demonstrate different techniques you may use in 4-Axis.



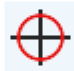


Double-click to open the Workgroup list dialog. The part has two workgroups with geometry extracted from the models. The workgroup Top Profile contains geometry extracted from the outer edge of the “cam shaft” model. The workgroup Bottom Profile contains geometry extracted from the edges of the fillet on the 1.5mm fillet model.

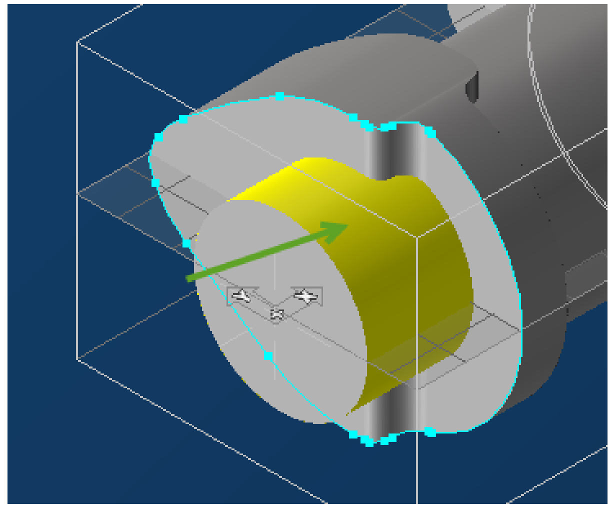
Cam Shaft Operation 1

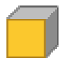
Determining the size of the Cam Shaft

In the first operation we will machine the side of the cam that the workgroup 1 geometry is on. The closed shape of the profile is fairly complex as its radial depth changes but we want to keep the tool tip at a constant depth - the depth of the shaft. We need to know what that depth is.

1. Select the  Show Position item from the Plug-Ins menu.

2. Click the picture icon until it displays  to determine curvature.



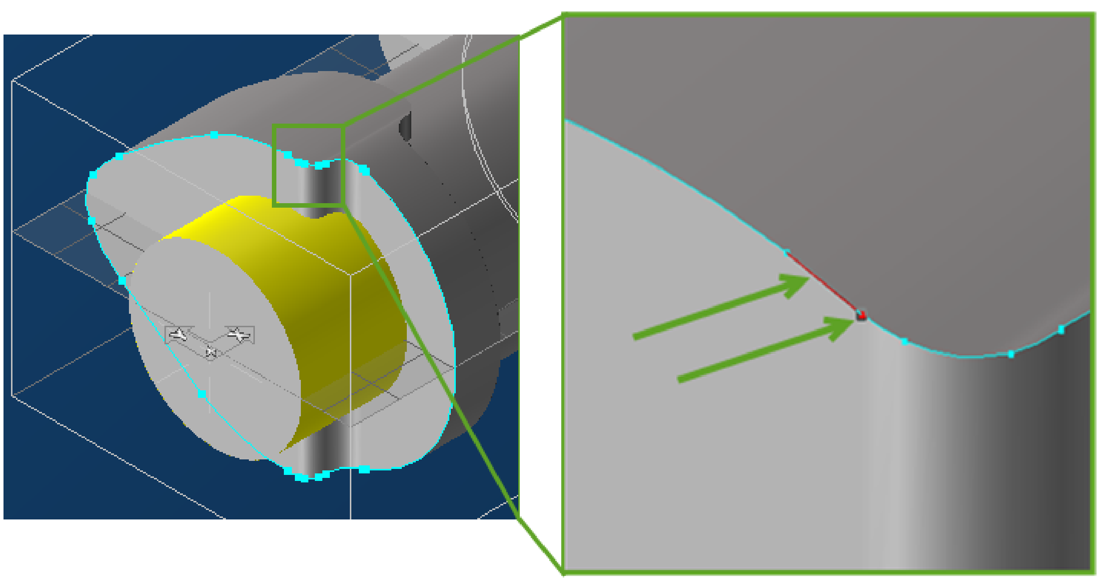
3. Activate face selection  and click anywhere on the cam shaft near the origin.

   This will display the radius of the solid.

We now know we need to cut to 12mm deep.

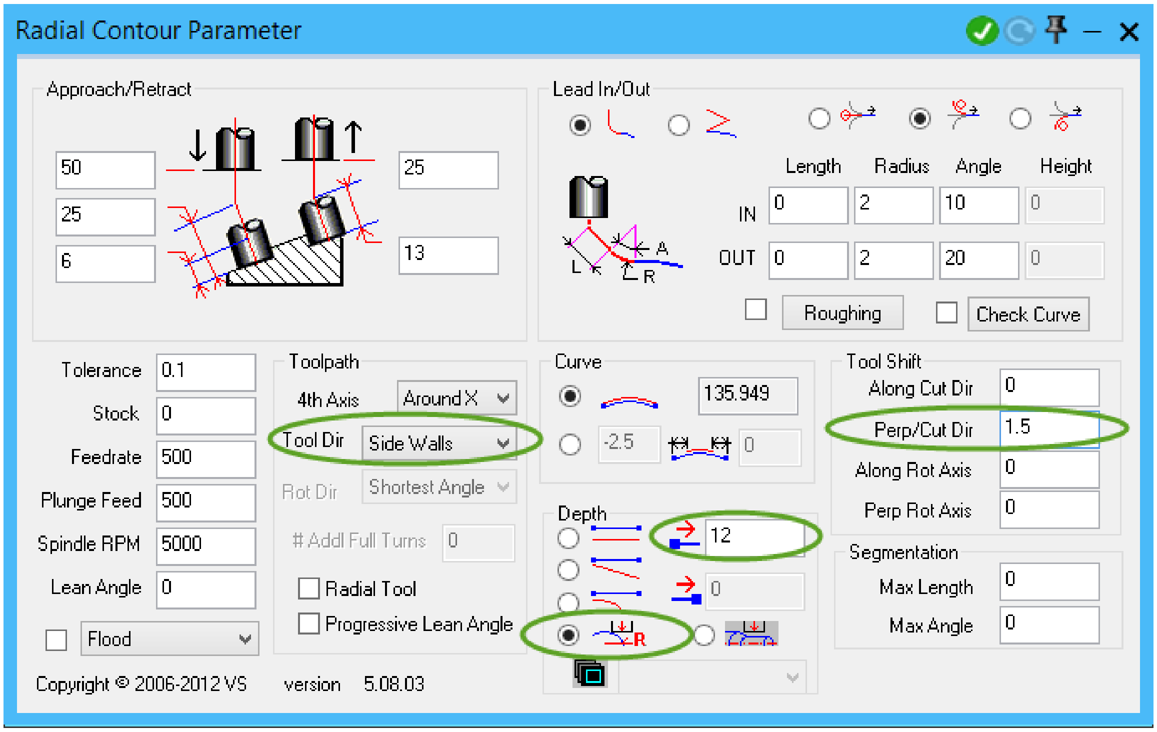
Creating the Operation

1. Select the point and arc shown.



While we do not have any lines to enter the part on, this arc is fairly flat and straight. It should work reasonably well.

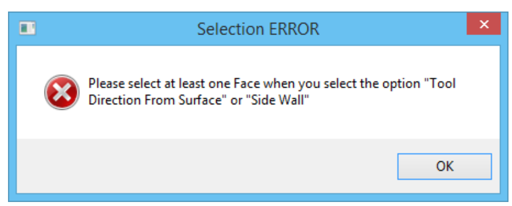
2. Create a Radial Milling Contour operation using Tool #1 as shown.



We have specified Side Walls because we are concerned with cutting the walls of the cam. The Perp/Cut Dir value is equal to the tool radius. The Depth is set to 12mm using the Constant Radius setting. That means that the tool will follow the path of the geometry but the tool will not adjust its depth.

3. Close the Contour4 dialog and `click` Do it.

An error message appears. We want the tool to remain normal to the geometry, cut the wall and remain radial. To do this GibbsCAM needs more information. We can use the solid model to help control the 4th axis rotations.

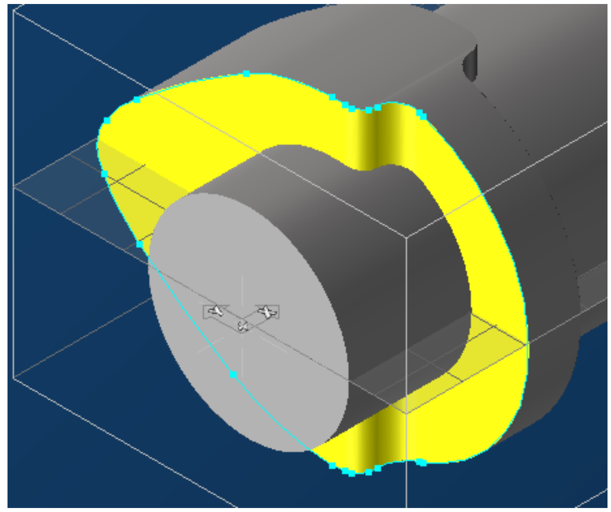


Using Wall Selection

Radial Milling does not work from solids, which would give us all the normal benefits that machining solids provides, such as gouge protection. Having said that, Radial Milling can temporarily use a solid to control the tool's angle. Basically the system makes virtual geometry from a selected face and aligns the tool to the virtual geometry. We are going to use this function to avoid the gouge.

1. `Click` OK to dismiss the error message and turn on face selection.

2. Select any face of the cam that is enclosed by the geometry, i.e. any of the faces we are attempting to cut.



3. Right click on the face and choose Select > Select Wall Faces from the menu.

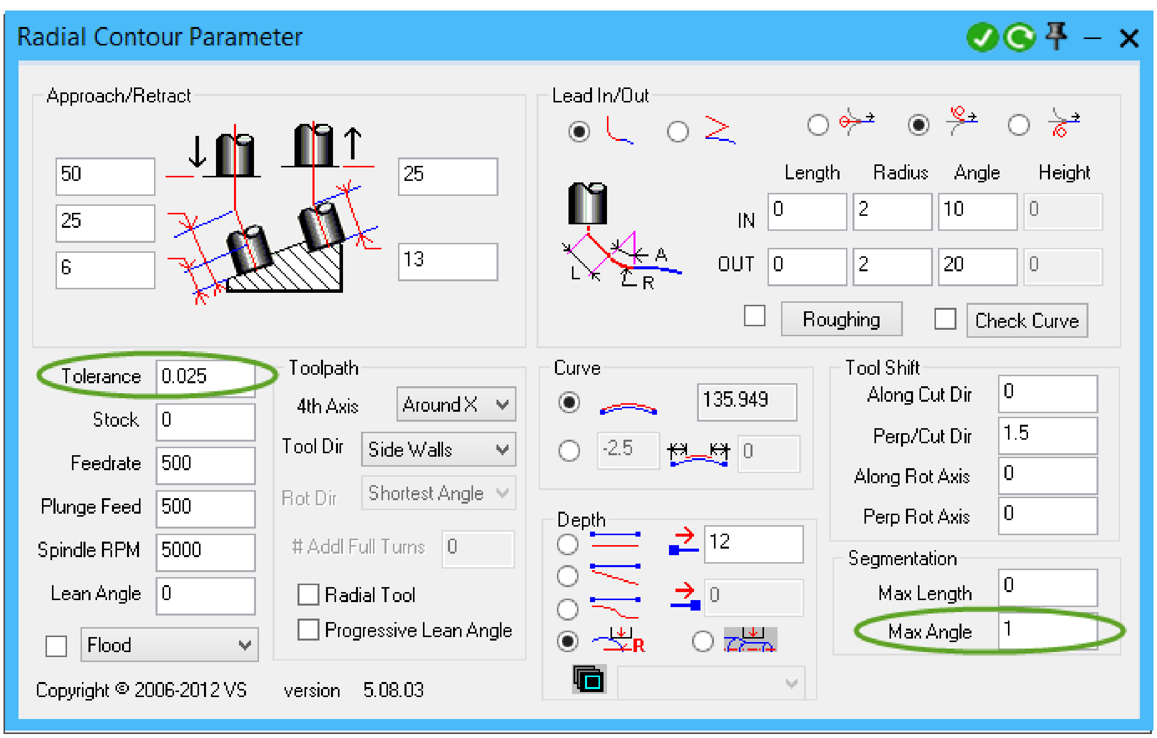
   This will select all of the faces that will be machined.

4. With the faces and geometry selected, redo the operation.

Cleaning the Toolpath

If we look at the concave corner near the top of the part we can see that there is a small section where the tool goes backwards then forwards. This is due to the change in the tool's angle based on the selected faces. We can make this a little cleaner by changing a few values.

1. Open the contour process and change the values as shown.



By providing a Max Angle value we will restrain the tool from going more than 1° past parallel with a face, and the tighter Tolerance will make cleaner toolpath.

2. Redo the operation.

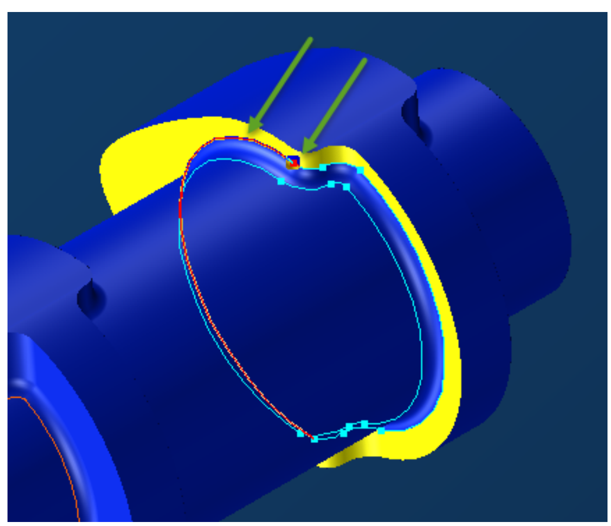
Now the part should be correct. The toolpath still has a location where the tool cuts back, and this is not entirely avoidable. This toolpath represents the tool tip, not the contact point so any location where the tool rapidly changes its orientation can lead to a situation similar to this. We have minimized the effect of this and were we to use the Analyze Cut Part option in Op Sim Rendering, we would see that we are neither gouging nor leaving any material behind. If we really did not want to have the tool cut back we could extract the geometry from the bottom of the shape and have the tool follow that curve rather than the top shape.

Cam Shaft Operation 2

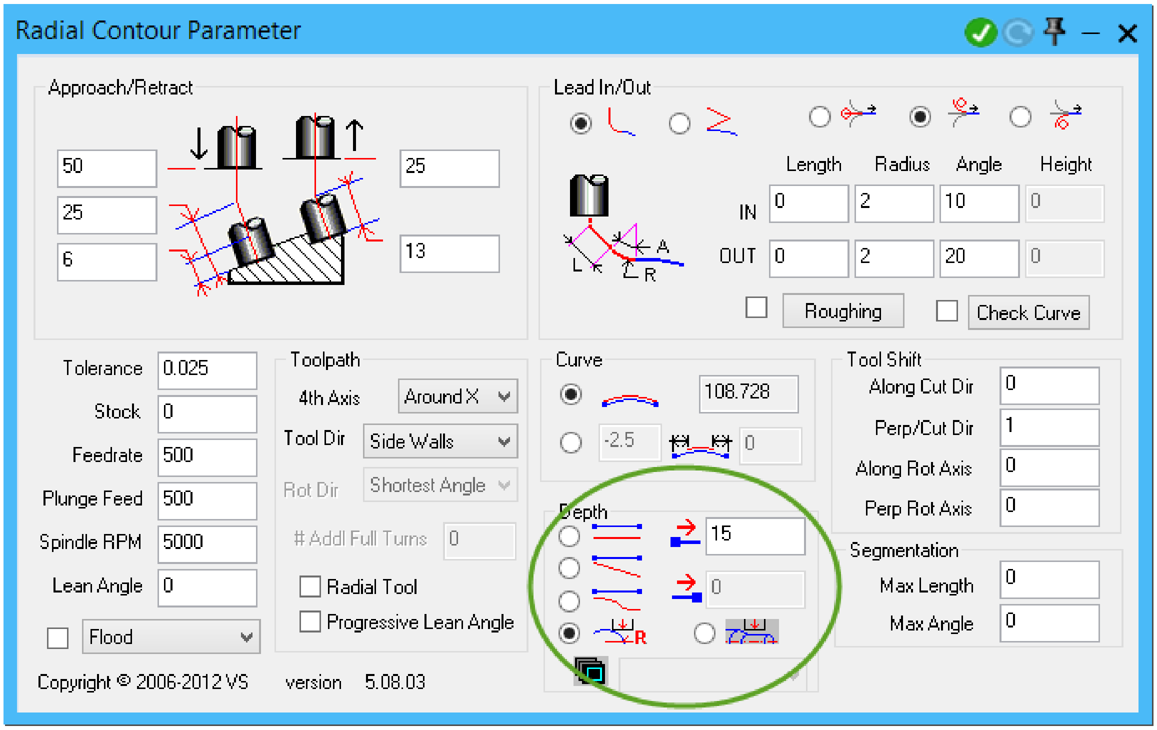
Setting up the Operation

The next operation will use the geometry (found in WG2) from the fillets on the second cam. The depth of the floor here is 15mm. If you wish to confirm this for yourself, use the Show Position plug-in as we did earlier in the tutorial.

1. Put the cam shaft part body in the body bag. We will now use the stock body with 1.5mm fillet.
2. Using the same selection technique as before, select the walls that need to be machined.
3. In addition to the faces, select the point and arc on the outer geometry as shown.



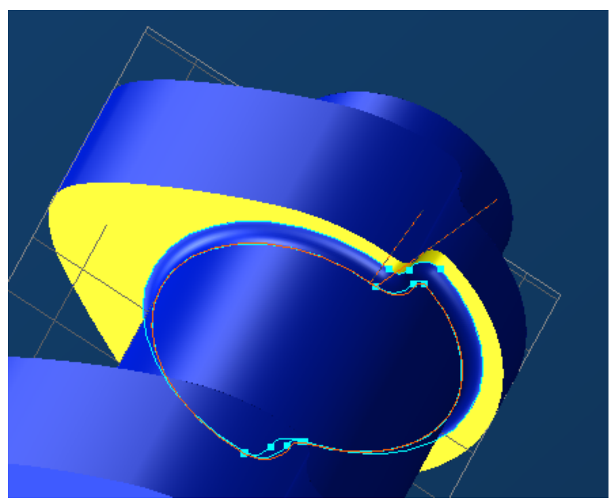
4. Using Tool #1, create a contouring operation as shown.



The information is the same as before except we have changed the constant depth to 15mm.

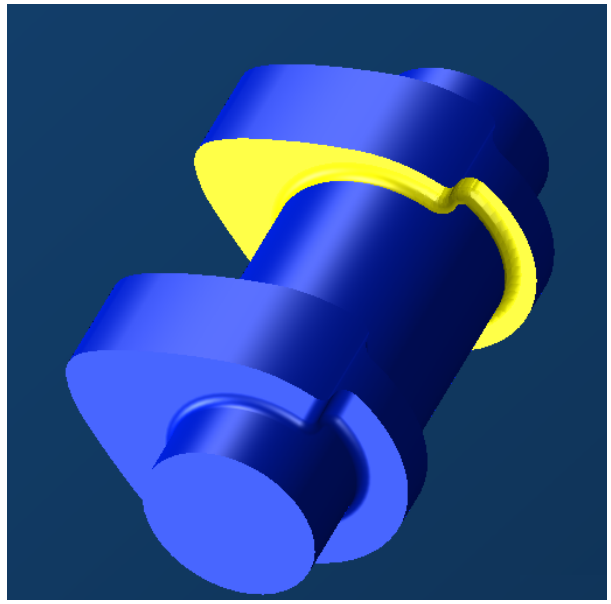
5. Close the dialog and then create the operation.

This creates good toolpath that follows the selected shape, does not gouge the walls and remains at a constant depth. The nice thing is that it does not matter which set of geometry we choose.



6. Deselect the operation, then double-click the Process tile.

7. While holding down the Ctrl key, deselect the point and arc then select the matching point and arc on the geometry that represents the bottom of the fillet.



8. Click Do it.

   The toolpath is identical. The tooltip is going to be in the same location for either set of geometry because we are using selected walls and the Perp/Cut Dir value have not changed. If the geometry were different, that is to say, not matching the fillet, then the toolpath would be different, but that would also be an entirely different part.

9. Save the part.

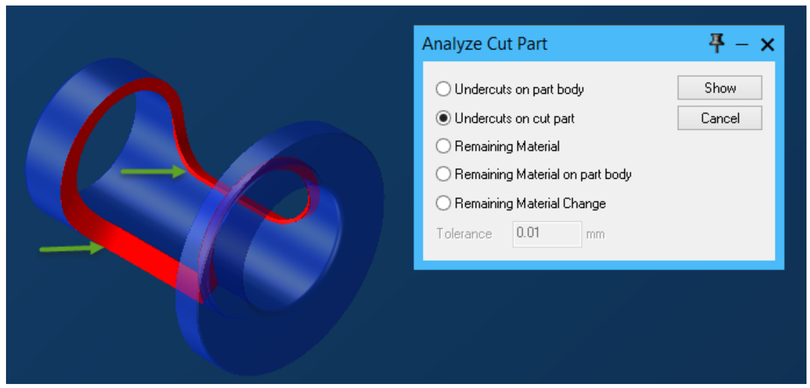
SIDE CUT AND SELECTING FACES

About the Pipe Cut Part

This part has an existing Polar & Cylindrical Milling operation. We will see that this part cannot be properly cut using the Polar & Cylindrical Milling option even though it looks like it can. We will create a Radial Milling operation that properly cuts the part. As stated, this part cannot be cut with Polar & Cylindrical Milling– the geometry goes through center. The tool must go through center for Polar & Cylindrical Milling.

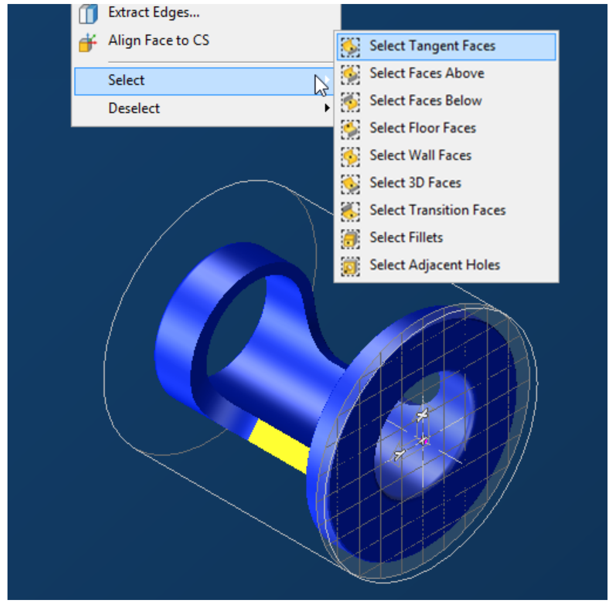
1. Open the file "Pipe Cut.vnc".
2. Select the body and render.
3. When the operation is finished activate Analyze Cut Part and select the Undercuts on cut part option.

   We can clearly see that the tool is cutting too deep at the bottom of the shape. Let's look at why this is happening.



The Shape

4. Click the CS command icon and switch to CS2 (the XY plane) and then Workgroup 1.

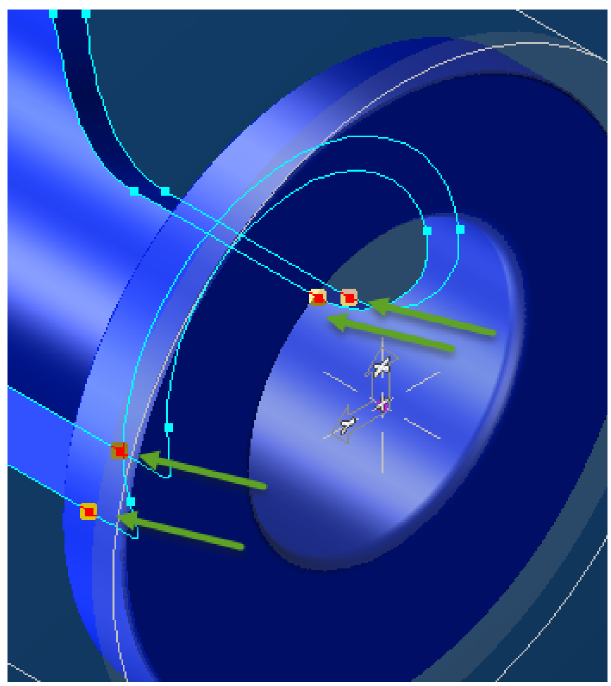


5. Turn on face selection and right click anywhere in the cutout section of the model. Then choose Select Tangent Faces.

   Now that the loop of faces that define the cutout are selected we will extract geometry and see what is going on.

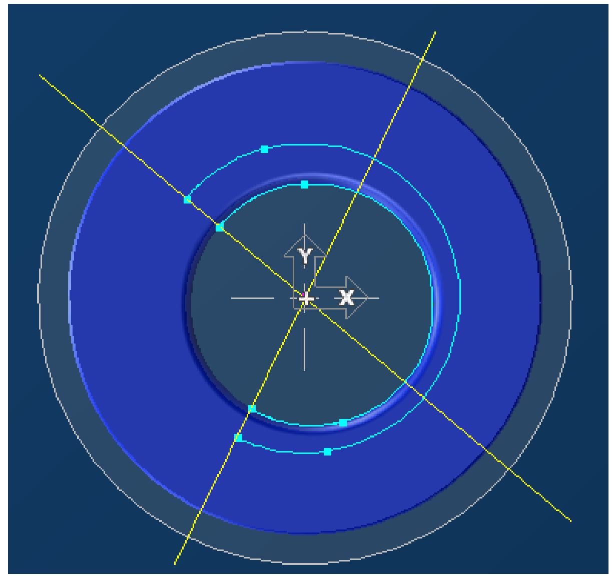
6. Right-click anywhere on the model and from the menu choose Extract edges .
7. Extract the geometry with a tolerance of 0.

   We will now create some lines through points.



8. Create lines through the two sets of points shown.

Radial Lines



9. Switch to the Home view.

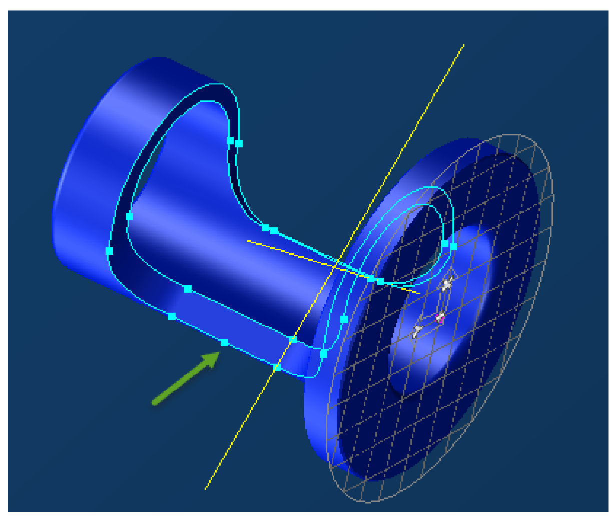
   We can see that the lines run through X0Y0, i.e. the lines are radial. With Polar & Cylindrical Milling, the tool must be radial. That is why we are removing too much material.

Fortunately, Radial Milling handles Y offsets. We need to recreate this operation using Radial Milling.

Pipe Cut Operation 1

1. Delete the existing operation 1.

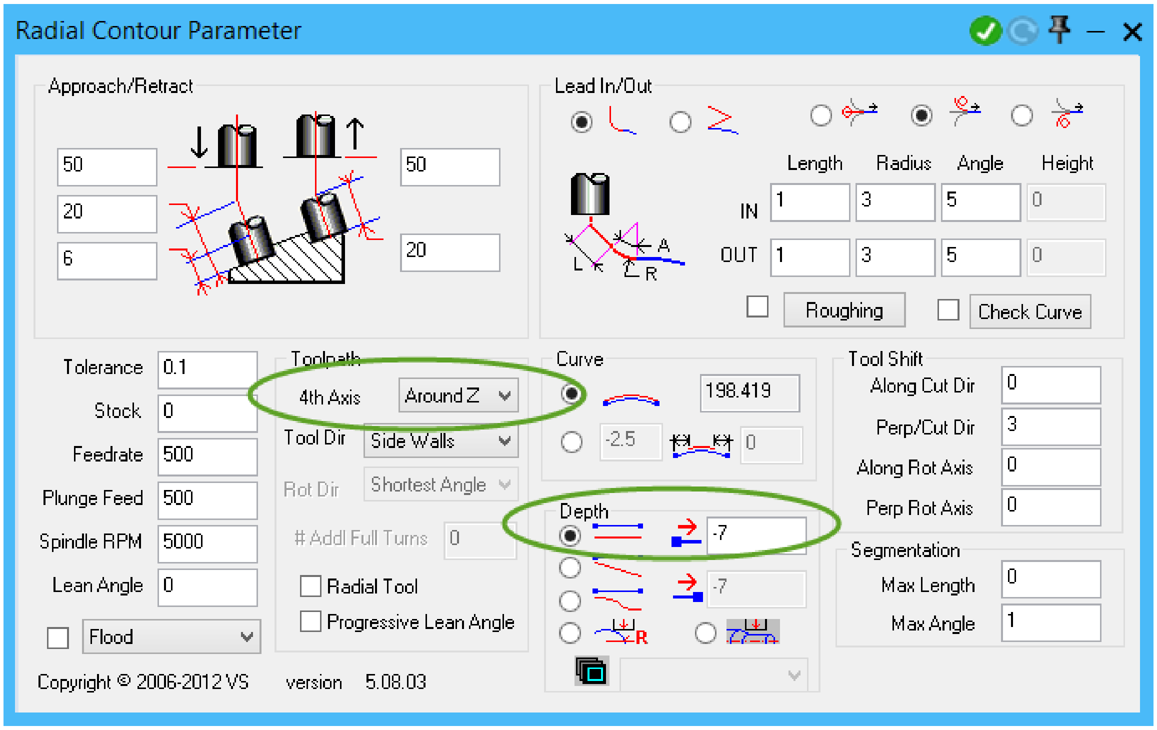
   We will use the geometry we just created as our contour shape with a mid-point added for the start location.



2. Create a mid-point on the line on the Y+ side of the outer profile geometry.

   You will need to disconnect, create a duplicate line and reconnect geometry to complete this.

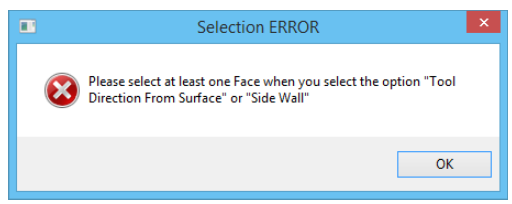
3. Create a Radial Milling Contour process with Tool #1 as shown.

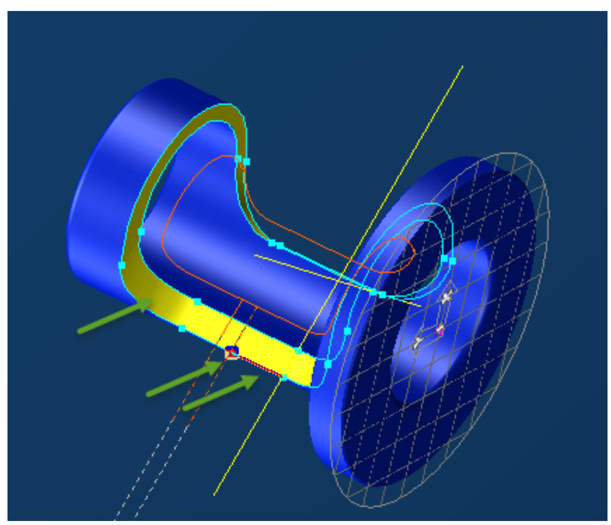


Be sure to set the 4th Axis to Around Z as this is on a Mill/Turn machine.

4. Select the midpoint and line then `click` Do it.

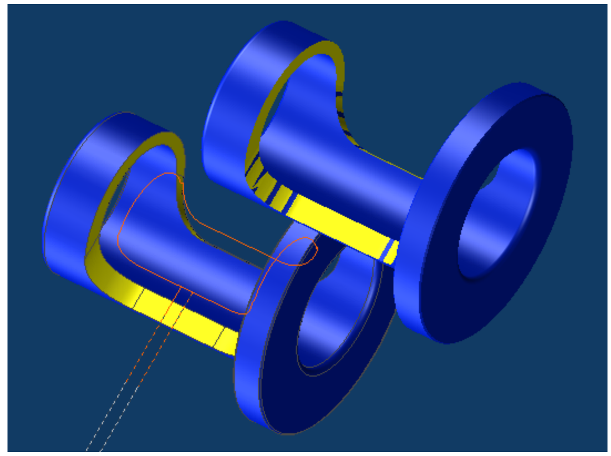
We need to select the walls of the solid to control the tool's alignment.





5. Using `right-click` Select >Tangent Faces, add the walls to the selected geometry.

6. Redo the operation.



The rendered operation is good. You might be wondering about the little “hitches” near the corner of the toolpath. These are normal and to be expected as tools swing around the transition from one wall to another. We have told the tool to maintain its tip at -7mm and that is what it is doing during the transition.

7. Save the part as it is complete.

MOVING THE TOOL RELATIVE TO THE GEOMETRY

About the Ellipse part

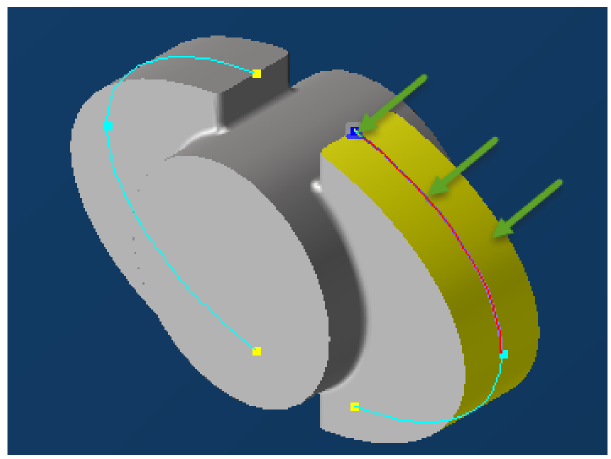
In this exercise we are going to experiment with the Tool Direction From Geometry option and look at another way to use Tool Shift.

1. Open the file Ellipse.vnc.

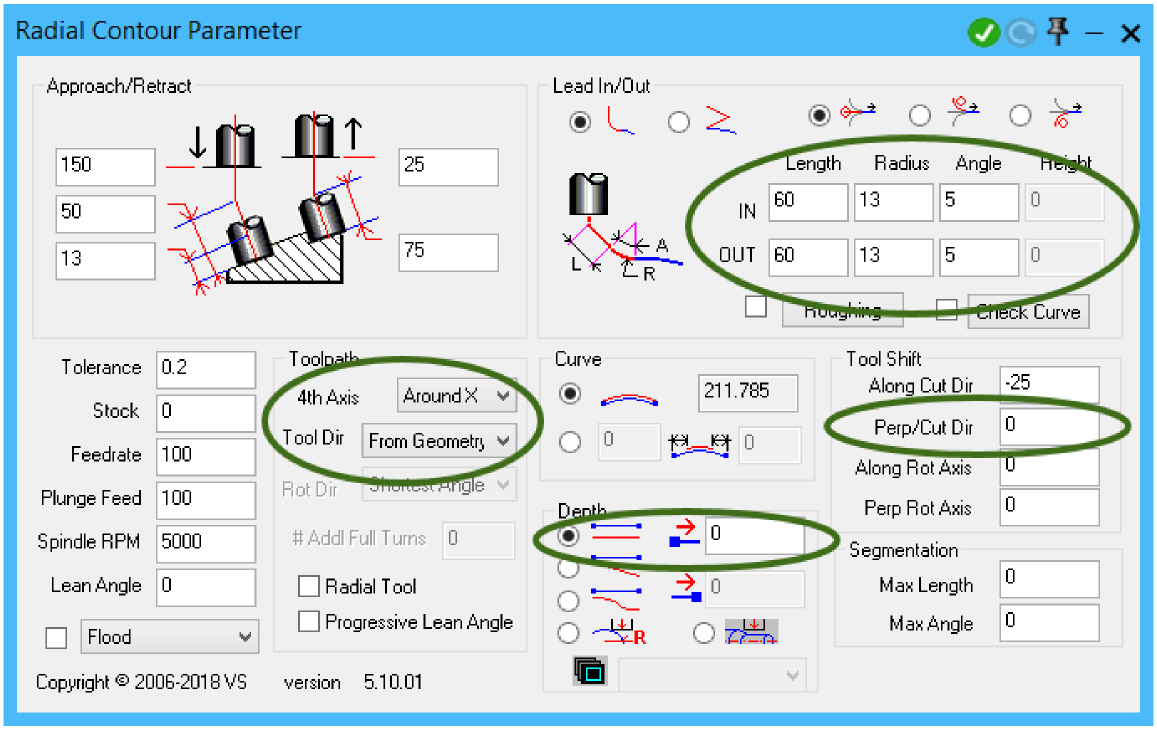
   We are only concerned with the elliptical geometry shapes around the profile of the part. While other sections of this part can be cut with Radial Milling, we are focusing on these particular profiles to learn more about controlling Radial Milling toolpath.

Ellipse operations

1. Select the point, line and surface shown.

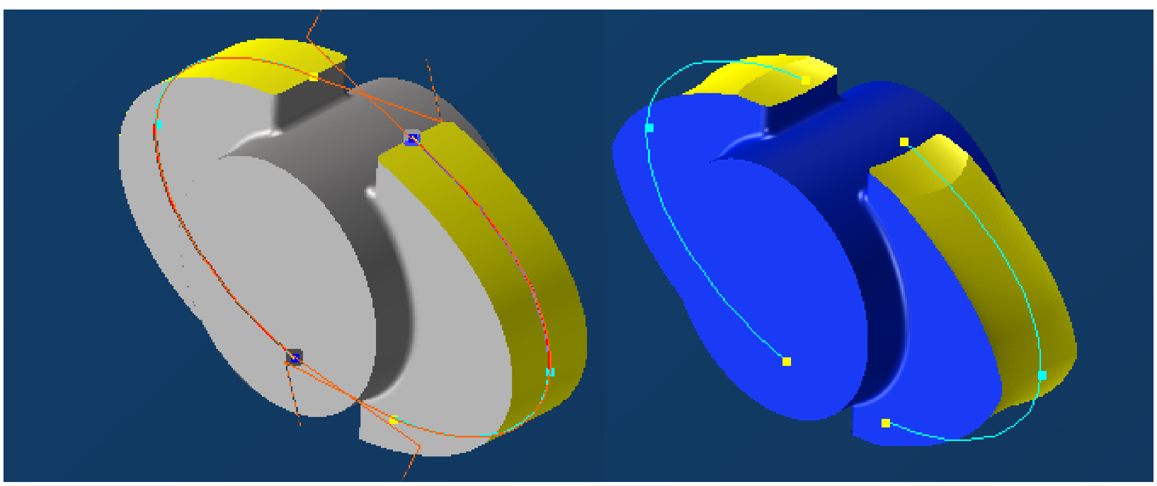


2. Create a Radial Milling Contour operation using Tool #1 and the process parameters shown below.



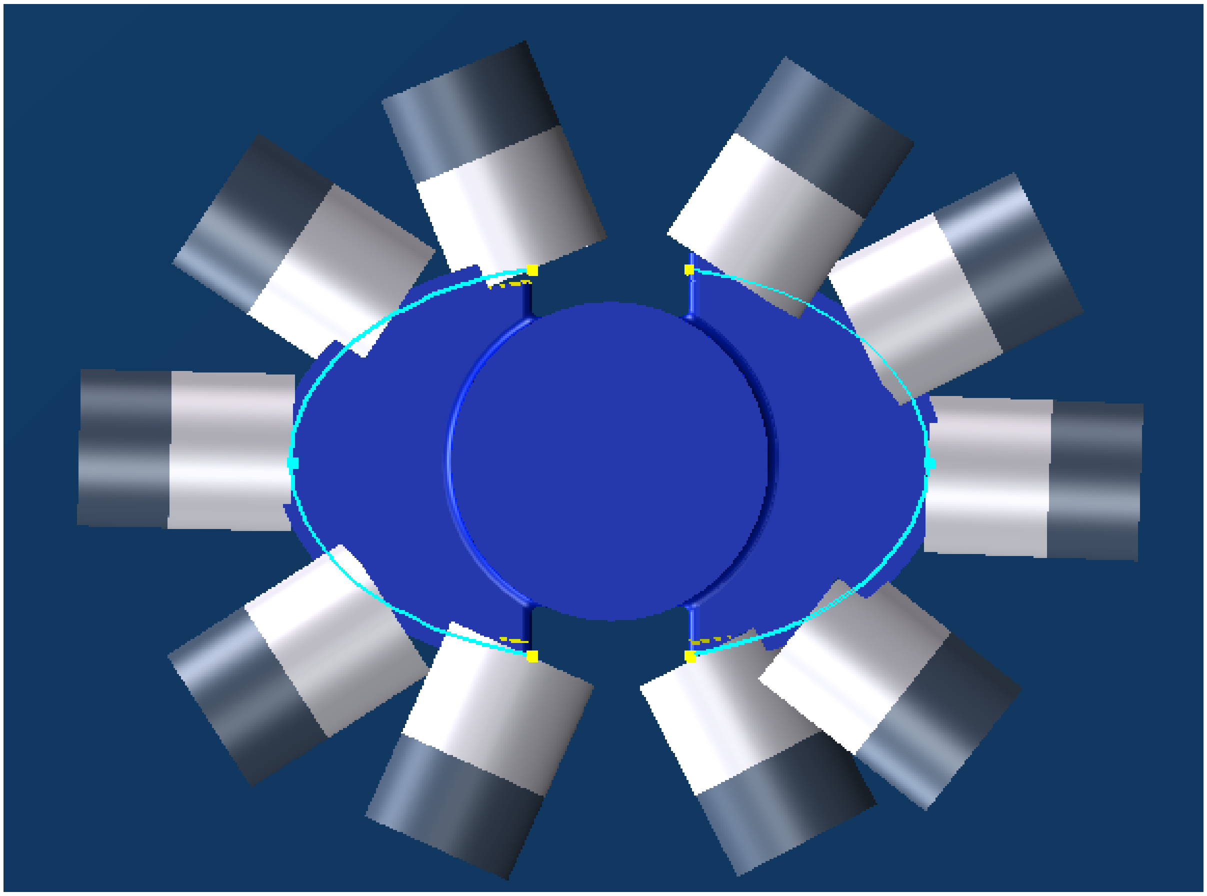
3. Create a second Radial Milling Contour operation, again using Tool #1 but instead of starting at the top of the part, select the point line and surface at the bottom of the part so that the tool will retract, reposition and then feed back onto the part.

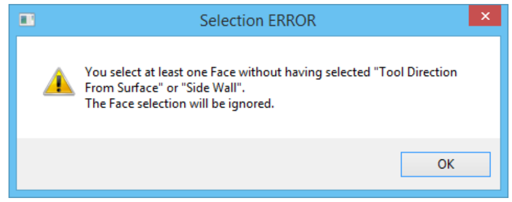
The resulting toolpath and rendered image should look as below.



Correcting the Operations, Part 1

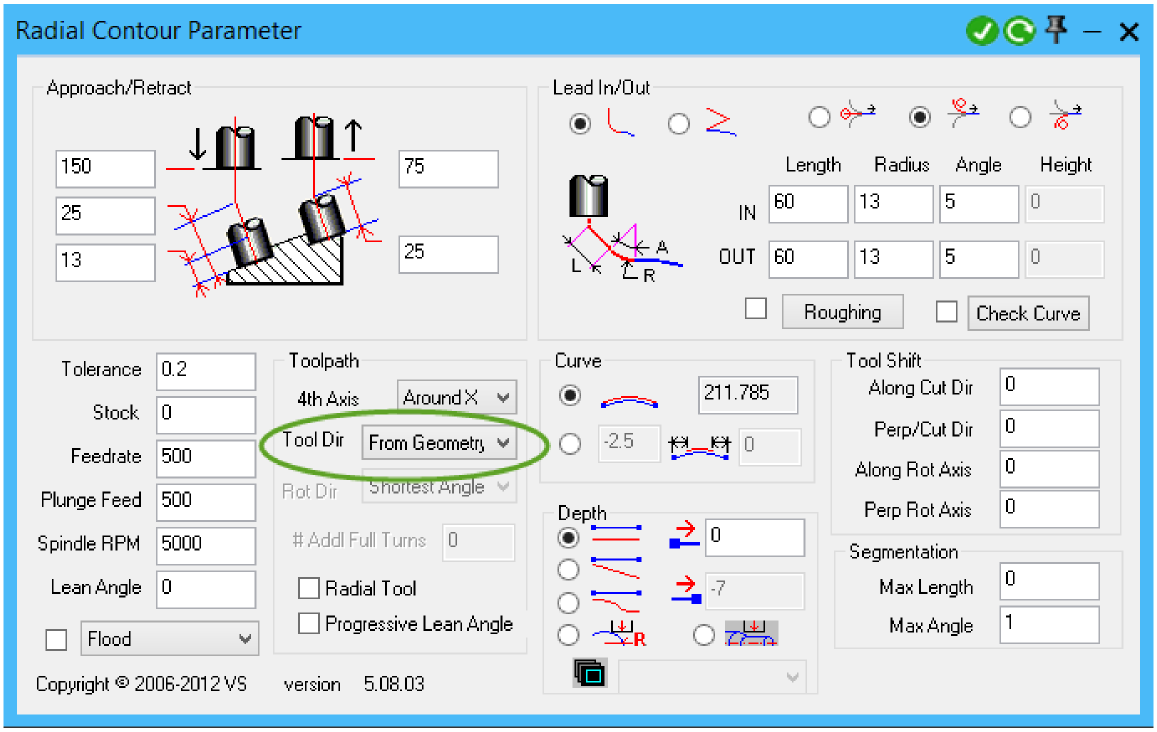
Unfortunately, when we examine the rendered part, we see this is a very bad result. The tool starts off tangent to the point selected and is radial. As the tool progresses around the part, it remains radial and removes a lot of material it should not remove.



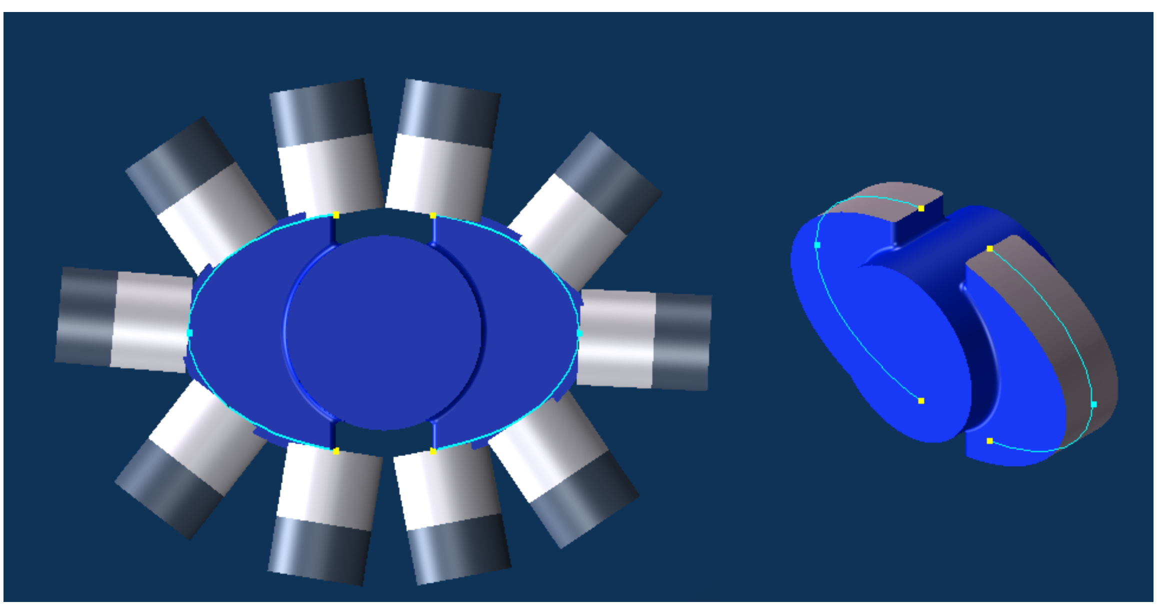


4. Open Operation 1, set the Toolpath option to Tool Dir From Geometry then redo the operation and repeat this process for Operation 2.

   Ignore the warning message - we do not use the Face selection.



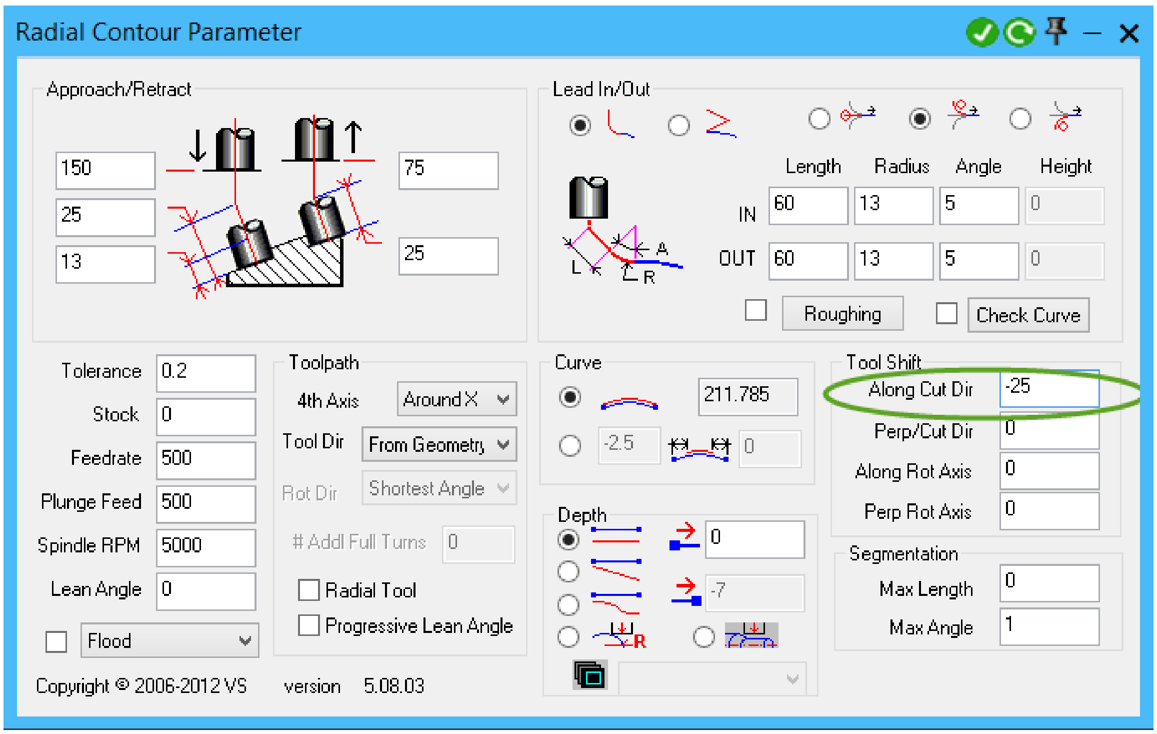
Now when rendered we can see that the tool does not gouge the part. The tool orientation is defined by the geometry. The tool is kept normal to the geometry and since the Depth setting is 0 the tool tip follows the geometry exactly and all angle changes occur above the tip.



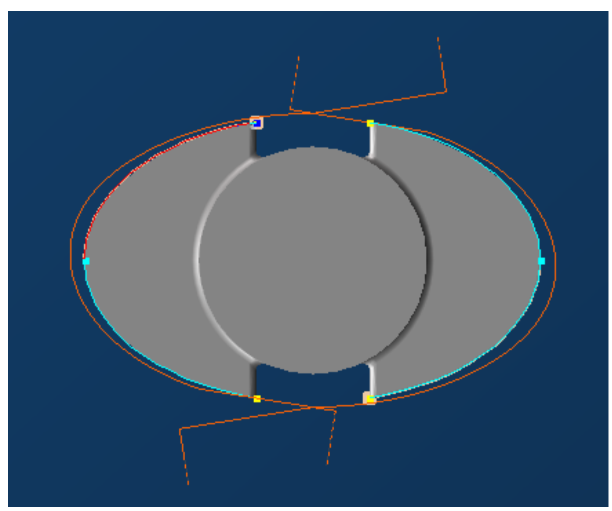
Correcting the Operations, Part 2

We are not quite done with this part yet. You may have noticed that the part is being cut with the bottom of the tool. This is most noticeable as the tool goes around the apex of the open ellipses. Cutting with the bottom of the tool is not optimal, so we are going to pull the tool back a little so that it cuts with its leading edge.

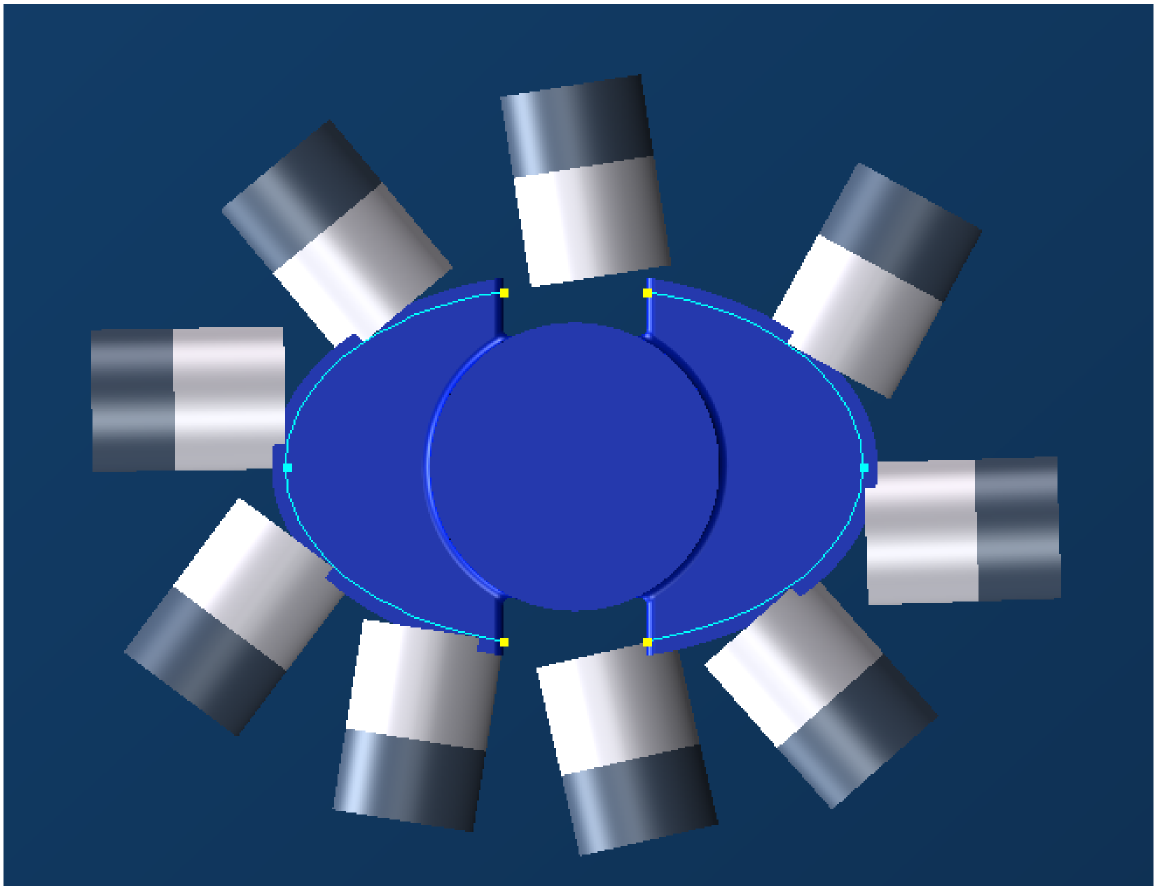
5. Open Operation 1, set the Along Cut Dir option to `-25`mm, and then redo the operation. Repeat this process for Operation 2.



We can see that the toolpath is no longer exactly on the geometry. This is because the toolpath represents the tooltip.



Looking at the rendered image below we can see that the center of the tool is exactly on the toolpath. By offsetting the tool by a radius the leading edge of the tool is now cutting the material first. The leading edge of the tool is effectively following the geometry.



Modifying the Tool Shift along the direction of cut is also a very useful technique for cutting threads.

6. Save this part as it is complete.

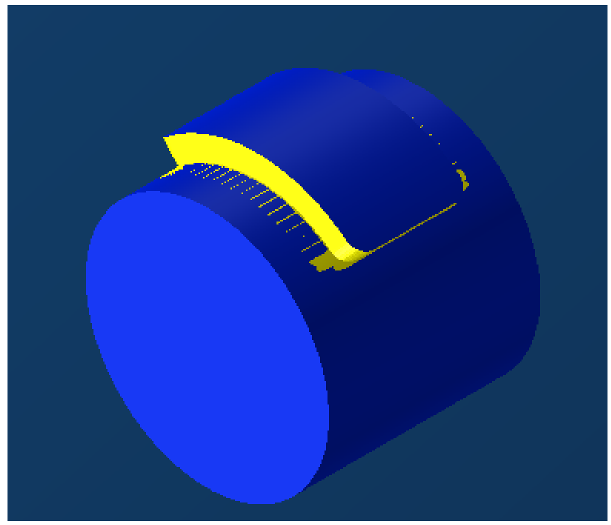
ADDING A CHAMFER

About the Part

In this tutorial we are going to add two types of chamfers to an existing part. The first chamfer will simply use a spot drill to create a 1mm chamfer around the top of the boss. The second chamfer will create a 30° chamfer on two sides of the boss using the Progressive Lean Angle option.

1. Open the file Progressive Lean.vnc.

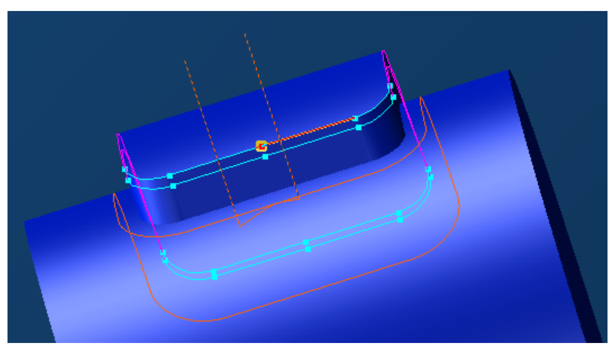
This is a rather simple part with an existing 4-Axis Contour operation that trims a little material off of the boss.



Adding The Chamfers

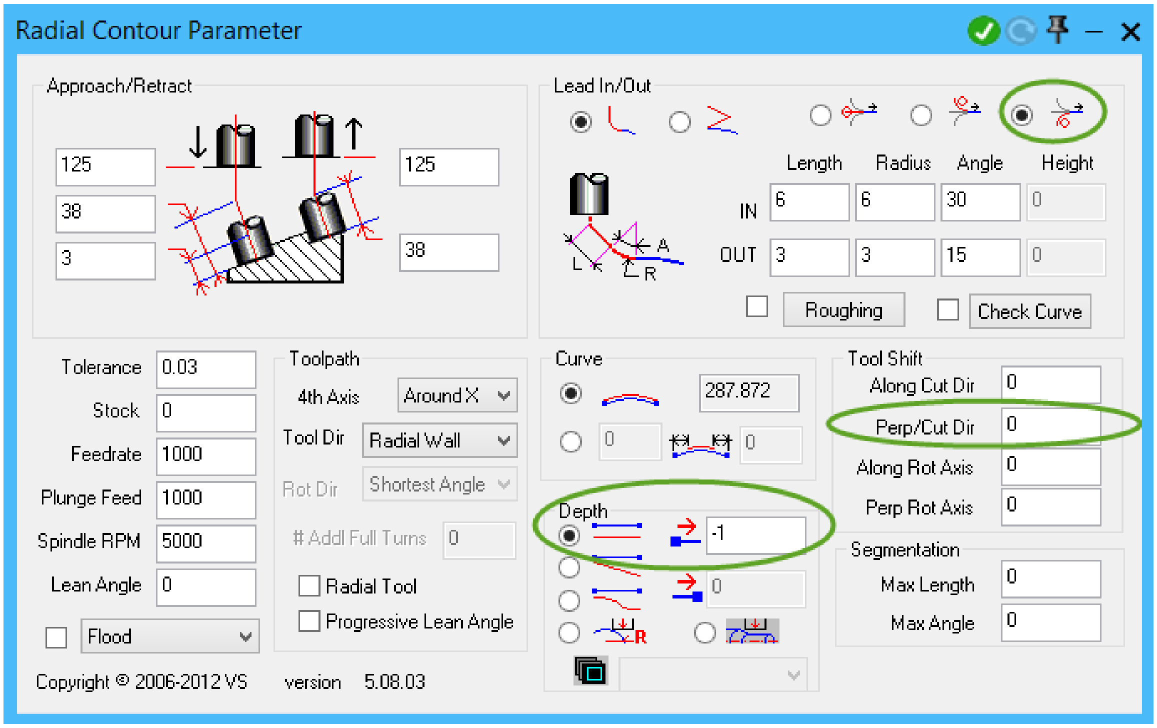
Progressive Lean - Operation 2

1. Select the same geometry used in Operation 1.



Be sure to deselect operation 1, as it is still required.

2. Create a Radial Milling contour operation using Tool #2 as shown.



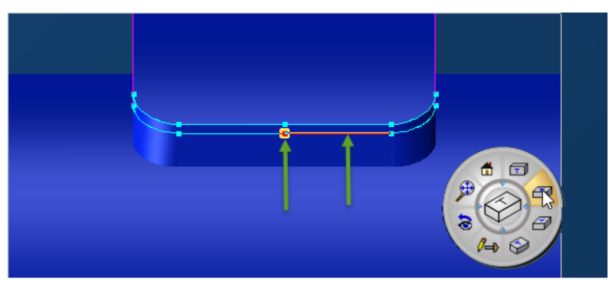
This is a very basic operation where we are running the tool on the outside of the geometry, cutting 1mm deep. When rendered you should see the edge of the boss has been given a chamfer. This is an operation that could easily be done with Polar & Cylindrical Milling.

Progressive Lean - Operation 3

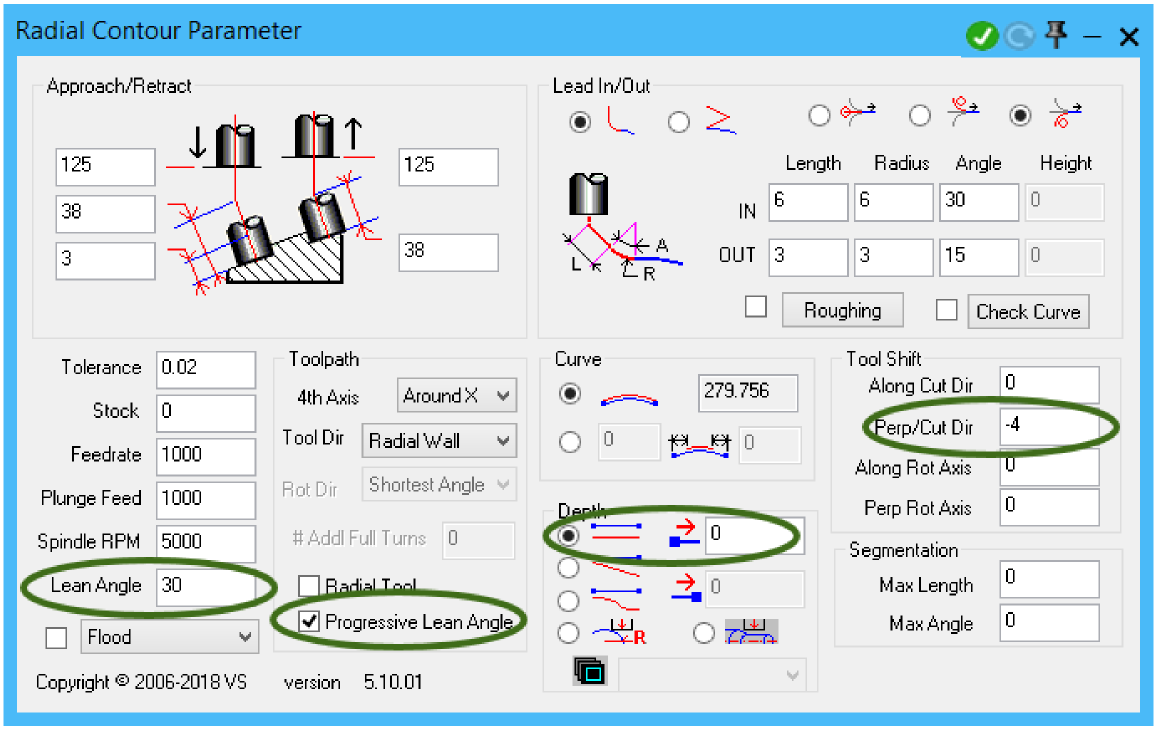
We will now make a chamfer that Polar & Cylindrical Milling cannot do. Let's imagine that this part was designed with a different chamfer. Rather than a standard 45° edge the sides of the boss have a 30° edge. We need to be able to lay the tool over by 30°. Using the Lean Angle we can lay the tool over but that fixes the tool to the angle specified. Fortunately the Toolpath section has a Progressive Lean Angle option. This option will progressively move the tool to the lean angle as it approaches parallel to the rotary axis. As the tool moves towards perpendicular to the rotary axis the tool is moved back to 0°.

3. Select the point and line shown on the lower closed shape.

   These are the same points as in the other operations, but using the lower closed shape. We are using a different shape to set the bottom of the chamfer.

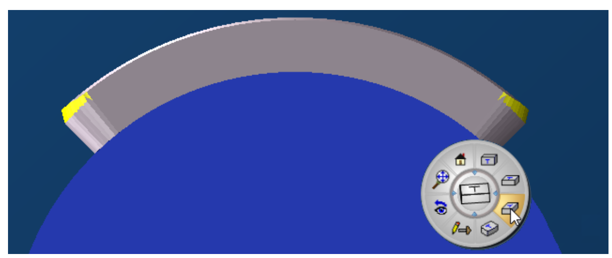


4. Create a Radial contour operation using Tool #1 as shown.



The Depth could actually be set to a different value and we would get a virtually identical result. We used this value so we can clearly see what is happening. The Perp/Cut Dir value is the same as is used in Operation 1.

When rendered the part should look like this image. Note how the tool changes its angle. This is a very useful technique.



5. Save the file as it is complete.