Version 14 : September 2020

2.5D Solids Tutorials

Proprietary Notice

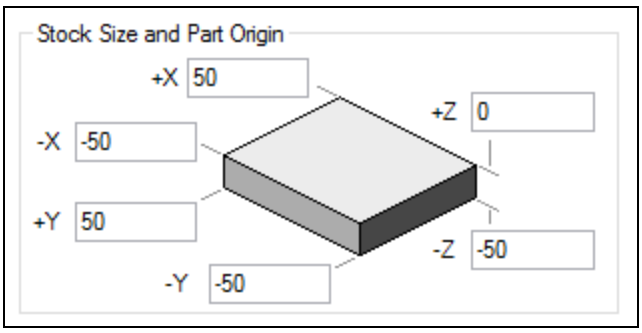
2.5D SOLIDS TUTORIALS

About the Tutorials

This chapter contains step by step instructions for creating models for a variety of parts. The models in this chapter will be machined in the Machining Exercises so be sure to save them for later use.

#1: 2.5D CAP

1. Create a new part named `Cap.vnc` with the mm dimensions shown.

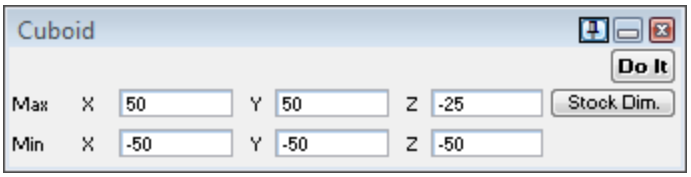


2. Open the Solid Modeling palette.

3. Open the Create Solid palette.

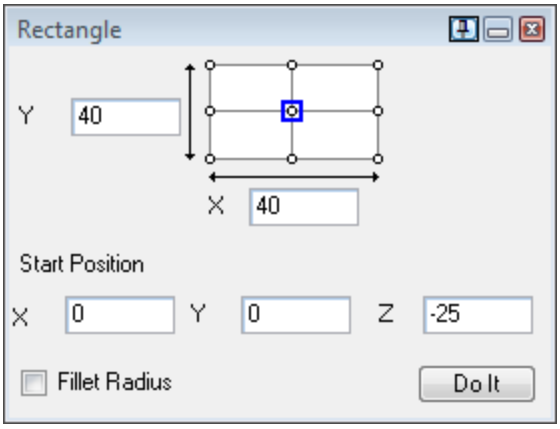
4. Open the Cuboid dialog.

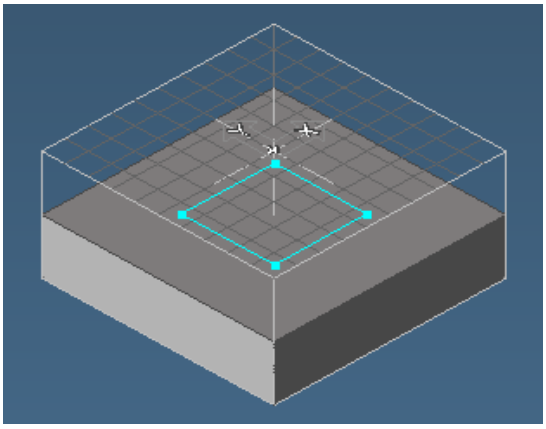
5. Enter the data in the Cuboid dialog as shown and `Click` the Do it button (or press `Ctrl+.`).



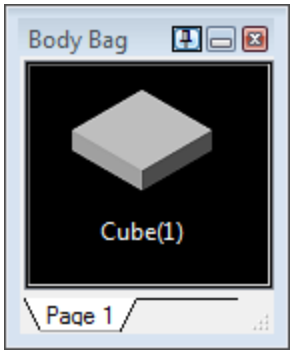
Pressing the Stock Dim. button will enter most of the data so that only the `Max Z` field needs to be changed.

6. From the Geometry Creation palette, select Shape then rectangle and create the rectangle as shown. Change view to isometric (`Ctrl+I`).



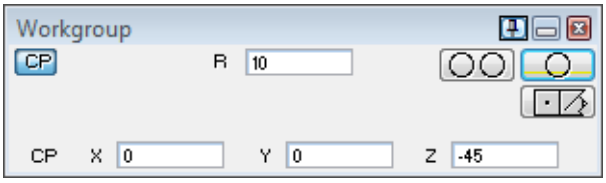


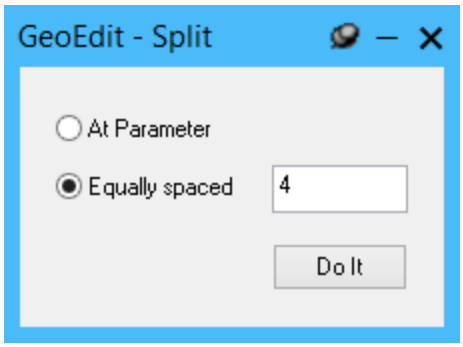
7. `Double-click` the cube to place it in the `Body Bag`.



8. Switch to the home view (`Ctrl+H`).

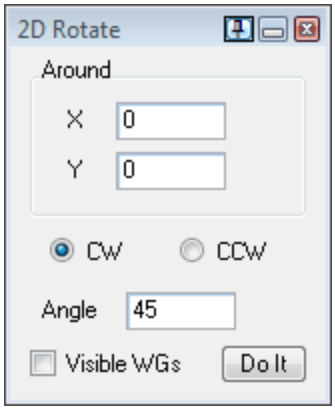
9. Create the circle  shown.

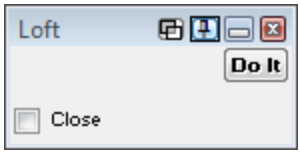




10. Select the circle and choose `Plug-Ins >`  `Geoedit Split`. Enter 4 for the number of sections then `Click` Do It (or press `Ctrl+.`).

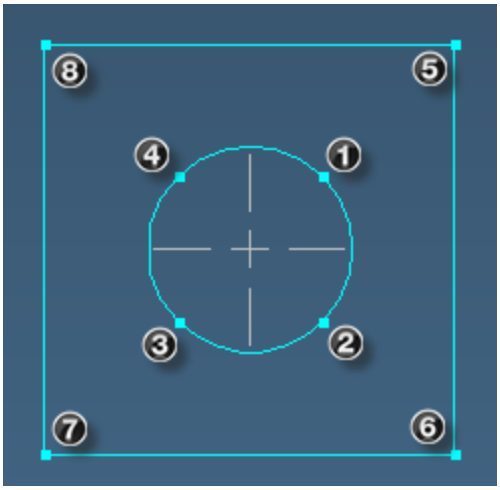
11. Choose Modify > Transform >  2D Rotate, enter the values shown. Double-click the circle to select all of it, then Click the Do it button.

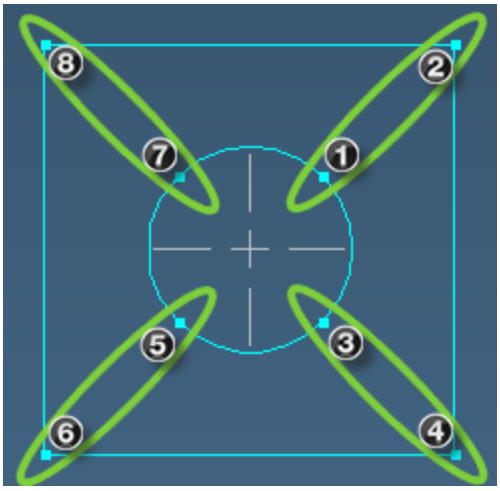




12. Close the Rotate dialog. Click away from the part to deselect and from the Create Solid  palette open the loft  dialog.

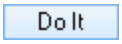
13. Ctrl+click the sync points as shown below.





Method 1 - Shape Sequence Method 2 - Shape to Shape

Select each point in sequence and in the direction of the loft, from the circle to the square, Click Do it.

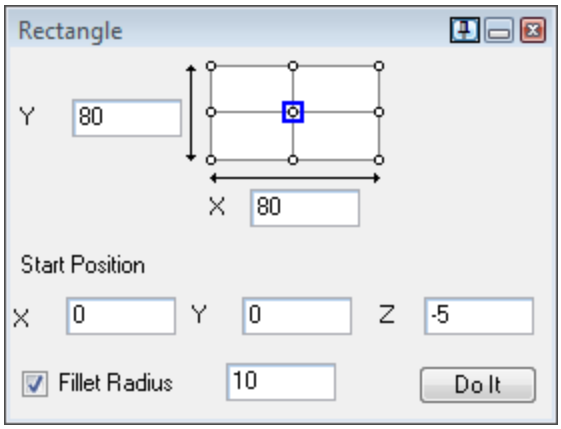


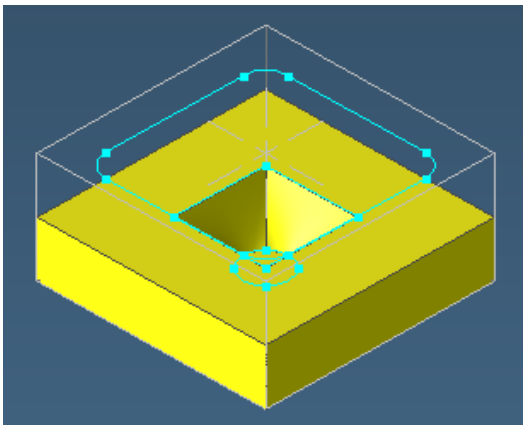
Alignment points are used to determine how the shapes will be aligned and blended. Alignment points must be selected in the right order. Alignment points can be selected in two ways, selecting alignment points from shape to shape or select a single shapes alignment points, in order, followed by the next shape in the same order.

14. Click the cube then Ctrl+click the loft shape. (Make sure Face selection  is deselected to select entire body).

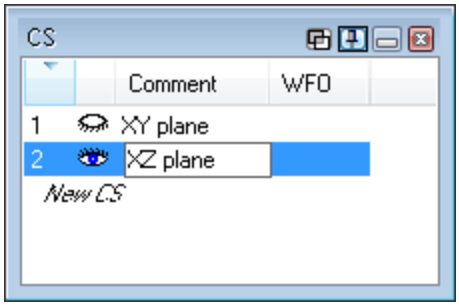
15. Subtract  the loft from cube.

16. Create a Rectangle with the Fillet Radius shown.



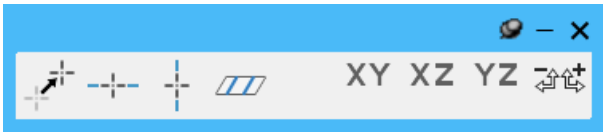


17. Click New CS from the CS list.



18. Create CS2 and label it XZ plane.

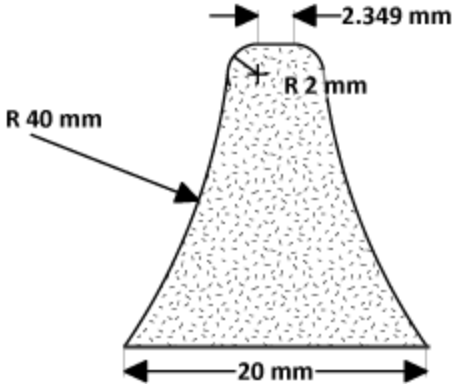
19. Open the CS  palette.

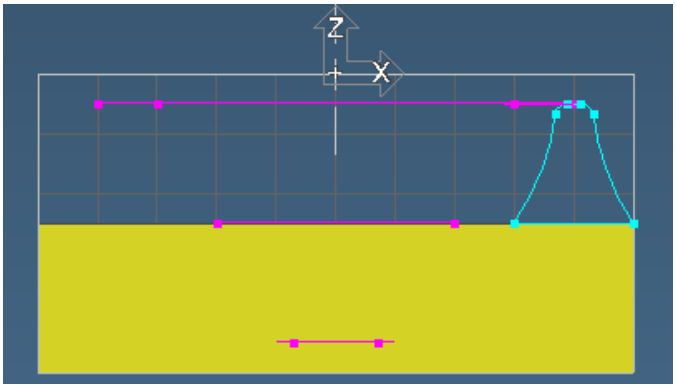


20. Click the XZ button.

21. Switch to the home view (Ctrl+H).

22. Create the profile geometry in the XZ plane.

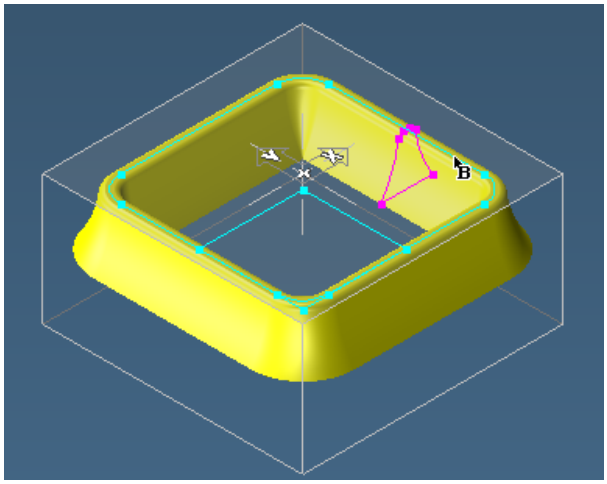




23. Switch to XY plane.

24. Switch to the isometric view (Ctrl+I) and Double-click the cube base to place it in the Body Bag.

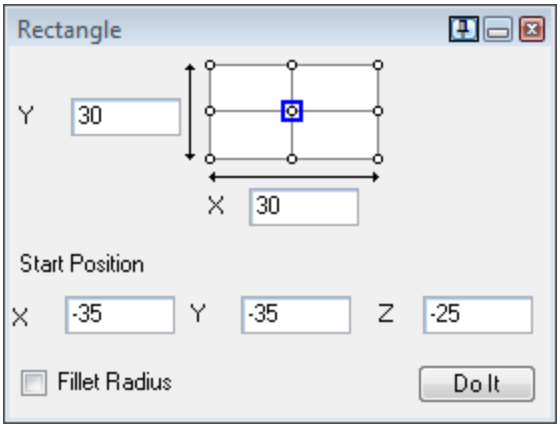
25. Select the profile and open the Sweep Solid dialog.

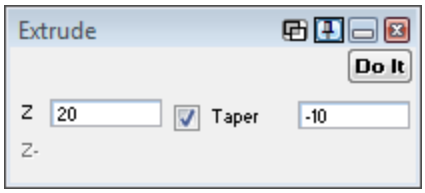


26. Place the Base Curve pointer 

on the rectangle with fillets and Click the Do it button (or press Ctrl+.).

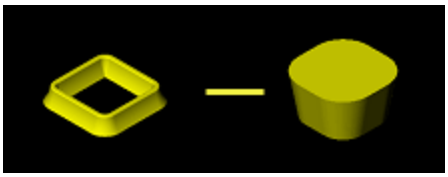
27. Open the Rectangle dialog and enter the following dimensions.





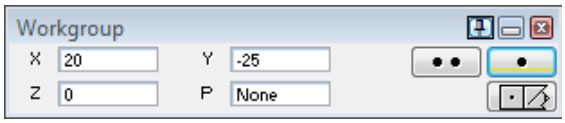
28. Select the square and create the extrusion  shown.

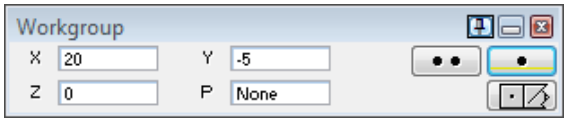
29. Subtract  the tapered extrusion from sweep.



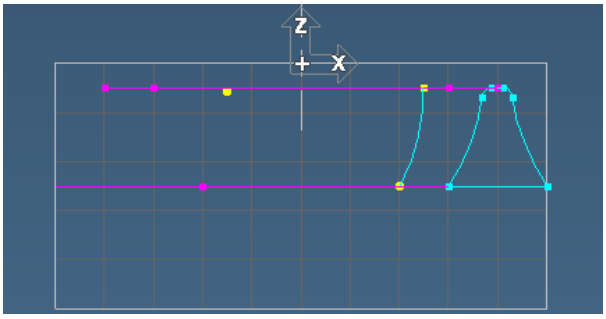
30. Switch to the XZ plane.

31. Create the points XYZ • shown below.

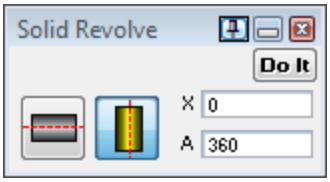




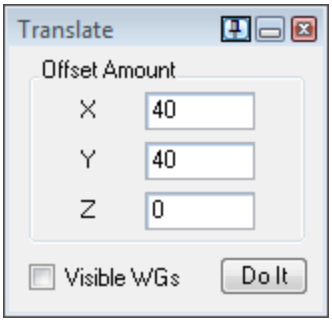
32. Create the 40mm arc terminated • between the two points as shown.



33. Select the arc and Revolve  the arc with the following information.



34. Choose Modify >Transform>  Translate and enter the following values.



35. Select the revolve and Click the Do it button.

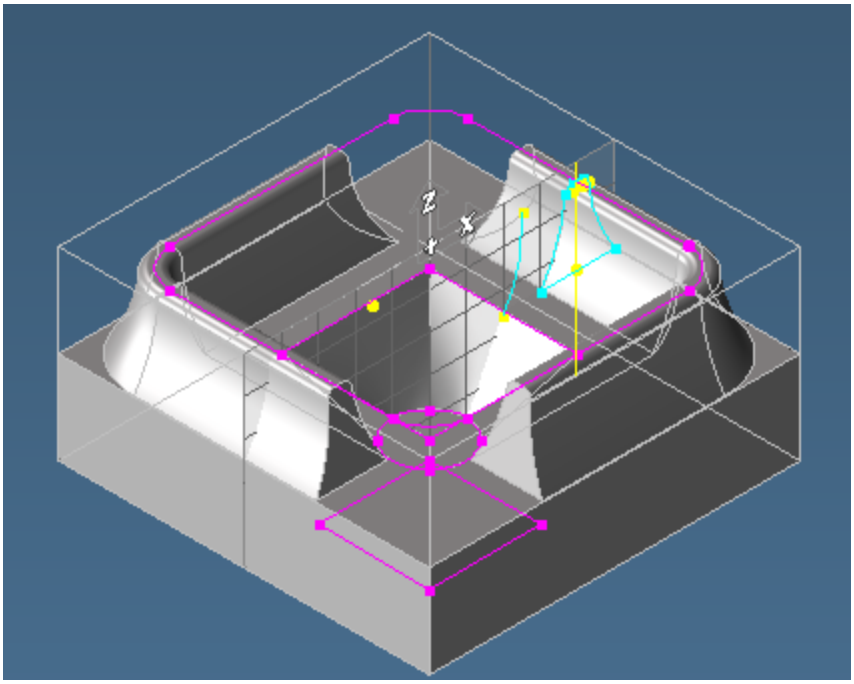
36. Subtract  revolve from sweep.

Note that the sweep is now two disjunct shapes but one multi-lump body.

37. Select the sweep and Separate  it.

38. Add  the swept shapes to the modified cube.

39. Save the part file.


Z
X

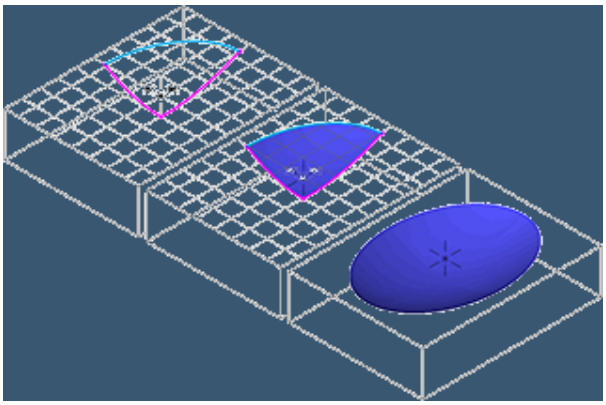
#2: BUILDING A SPHERICAL ELLIPSE

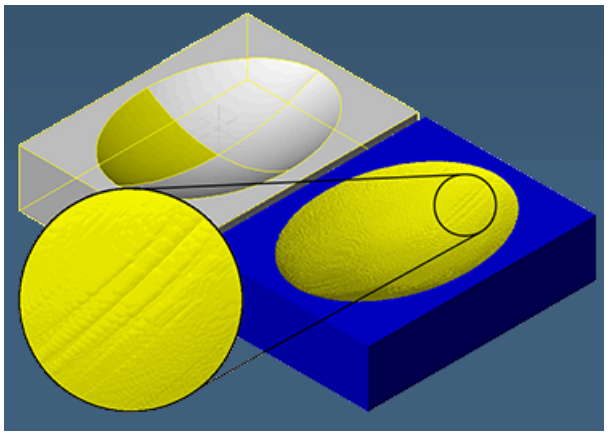
The Challenge

You need to create a concave elliptical where the lowest point is in the center.

The Problem

The print specifies values for one quadrant of the 3D shape; this is how we might normally go about building the part, one quadrant at a time. We build the first quadrant and then create a Coons Patch between the geometry. We then duplicate and mirror the Coons Patch sheet over the X and Y center lines and stitch the sheets together. This result looks like the finished shape we want.





However, there is a problem with this model that cannot be clearly seen until the part is machined. If you turn on “Show Edges”, you can see the intersecting lines running through the model. These intersecting lines run parallel to the X and Y axes at the centerline. When the part is machined, these lines will show up as if they have been magnified. The model, not the machining, is the cause of the problem.

Is this problem unique to GibbsCAM? No, this is the result of poor modeling techniques and will be reproduced in any other CAD or CAD/CAM system unless a better set of modeling techniques is used. GibbsCAM is machining the model exactly as it was built.

The Solution

To create a single continuous 3D flowing shape we will create a “3 Point Arc” on the XZ CS, another “3 Point Arc” on the YZ CS and then Sweep a sheet over these two arcs. Create a 3 Axis Vertical Mill part, X:-40/+40, Y:-30/+30, Z:-20/0mm. Follow the steps below:

Step #1

1. Create an XZ plane.
2. Create points at X-35, D0, Z0 then at X0, D0, Z-8 and lastly X+35, D0, Z0.

3. Select the points we just created.
4. Create a “3 Point Circle” (the circle will have a radius of 80.563mm).
5. Terminate the circle with the points at X-35 and X+35.

Step #2

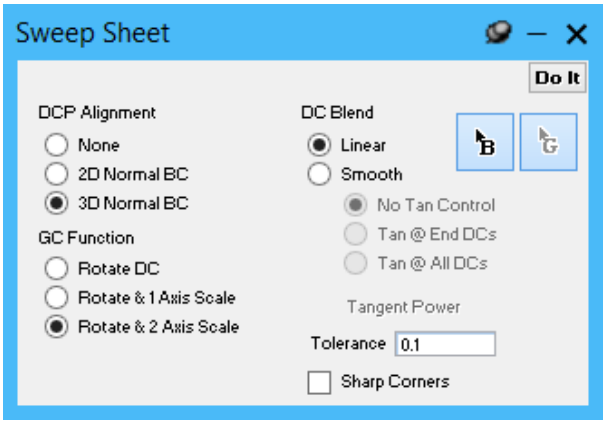
1. Create a YZ plane.
2. Create points at Y-25, Z0, X0 then Y0, Z-8, X0 and Y25, Z0, X0.
3. Select the points we just created.
4. Create a “3 Point Circle” (circle will have a radius of 43.063mm).
5. Terminate the circle with the points at Y+25 and Y-25.

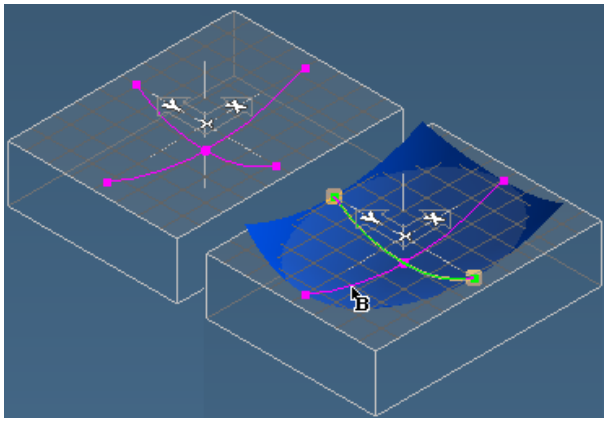
Step #3

1. Switch to the XY + XY plane.
2. Open the Surface Modeling palette.

3. Open the Sweep Sheet dialog.

4. Set the sweep options as shown.





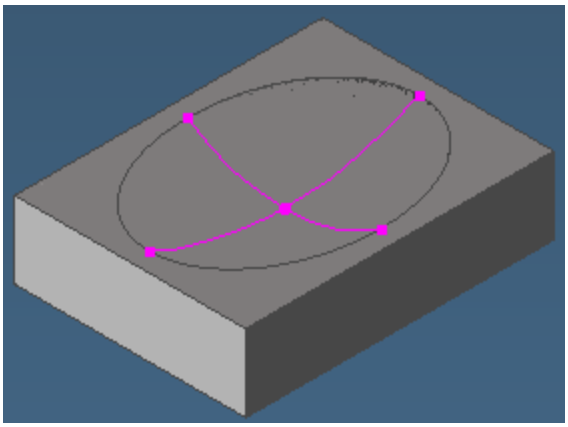
5. Place the base curve pointer on the curve shown and double-click the other curve as the drive curve.

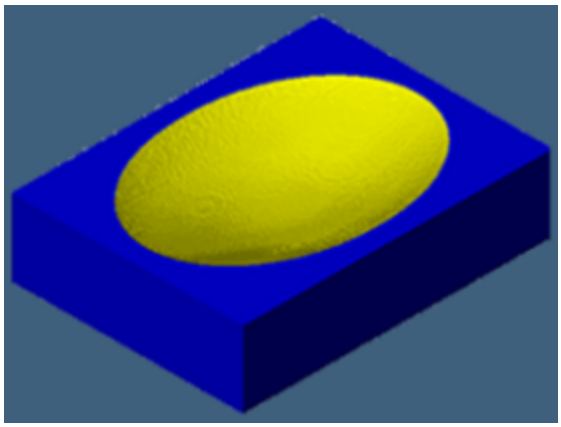
6. Click the Do It button to create the sheet.

7. Create a cube based on the workspace stock dimensions.

8. Slice the cube with the swept sheet function.

Now you have a continuous shape which does not have quadrant intersections. Turn on Edge Selection and you will only see the edges of the spherical ellipse at the top surface of the body. There are no edges in the middle of the spherical ellipse like we saw with the quadrant building technique. You may machine as desired (the sample part uses a lace cut) and you will not see any lines in this area as you did before.





9. Save the file.

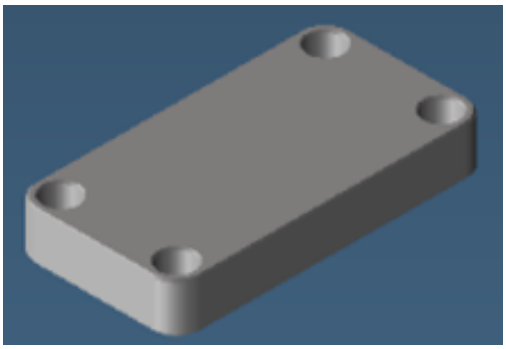
REPLACE HISTORY

In this tutorial we are going to modify a model to create a pocket. We will machine the pocket with saved processes. After we machine the part we will import a new body that represents a redesigned pocket. Using Replace, we will recreate the model and redo the machining.

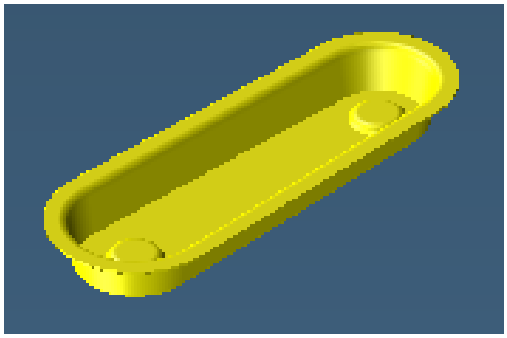
Part Creation

1. Open part file Swap Example.vnc.

   This part file has 2 bodies – Block and Pocket. Pocket is currently not visible as it is in the Body Bag.



Block



Pocket

Saved Processes

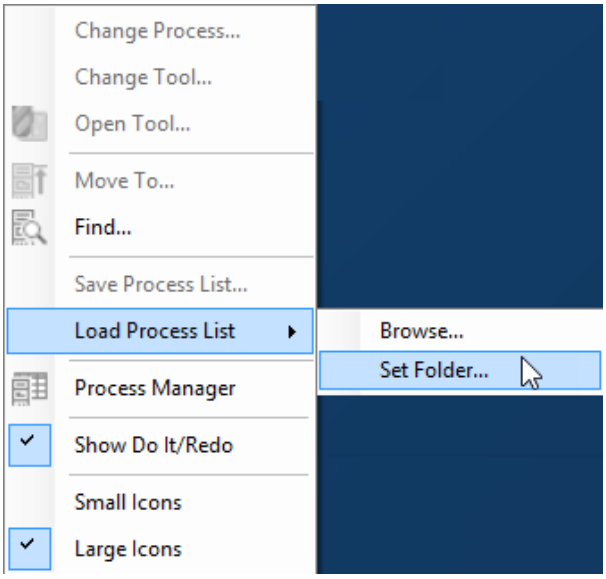
The first thing we will do is make the saved processes available for use.

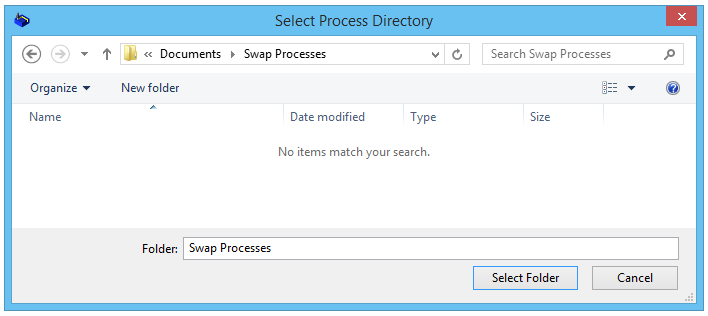
1. Open the Sample Parts\Solids\Required\SolidSurfacer\Swap & Replace folder and copy the Swap Processes folder to My Documents.

   Placing the saved processes here will make it easy to find them when we need them.

1. Right-click a Process tile and from the menu choose Load Process List, then Set Folder.

This will now be the default location that processes can be loaded.

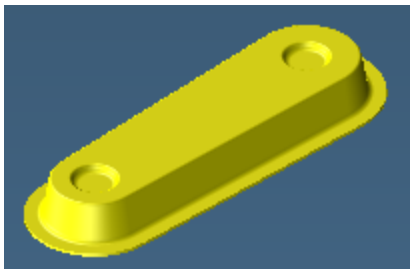




2. Navigate to the My Documents/ Swap Process directory and click Select Folder.

Placing the saved processes here will make it easy to find them when we need them.

Unstitching and Subtracting "Pocket"



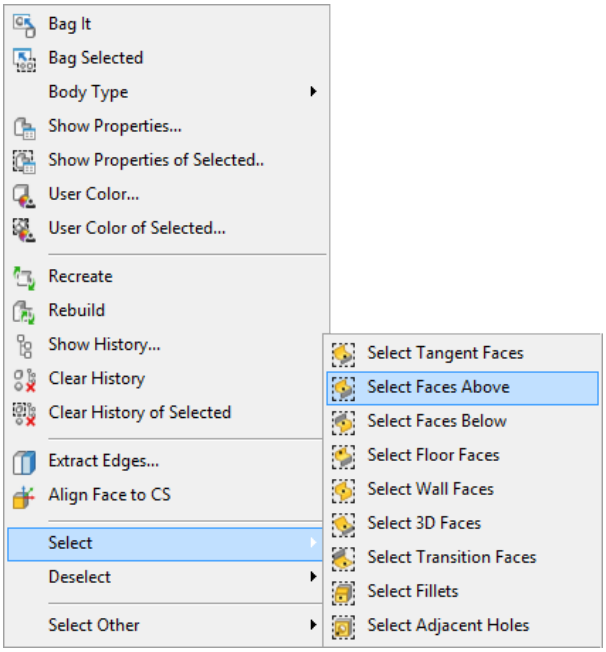
We are going to unstitch the body named Pocket to close off the shape, thereby creating a core. The interior topology is not important; it is the exterior shape we will be subtracting from the Block.

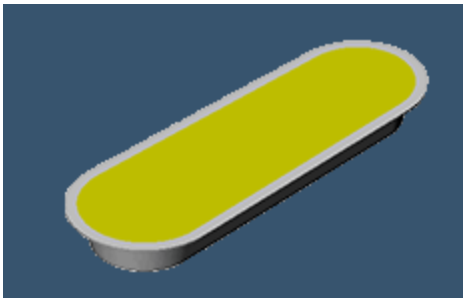
Opposite isometric view (Ctrl+Alt+I)

1. Turn on face selection and Right-click the bottom flat face of the pocket body.

2. Choose the Select>  Select Faces Above option.

All faces inside the pocket except the flats on top of the bosses should be selected.





3. Select the two flat tops of the bosses, as shown above.

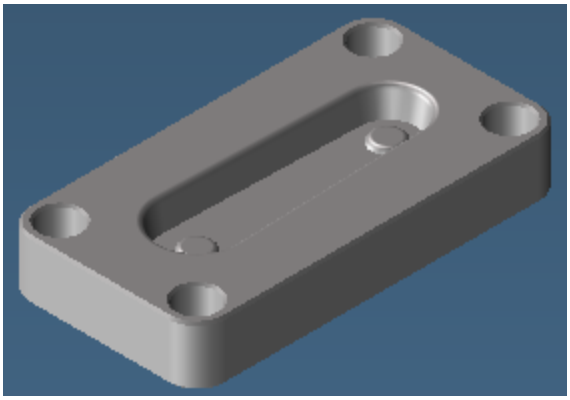
4. Click the Unstitch button. 

   This results in two solids.

5. Select and Delete the smaller of the two resulting solids.

   We do not need the core that was created, but we need the filled pocket body.

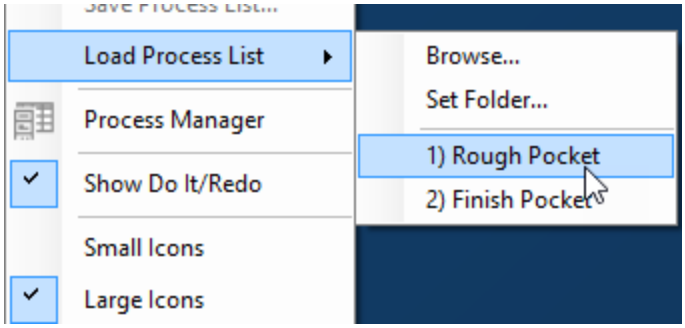
6. Subtract the filled pocket from the Block.



Machining the Part

Loading Processes.

1. Right-click the Process list and select Load Process List. This time the menu will display files in the folder we set.



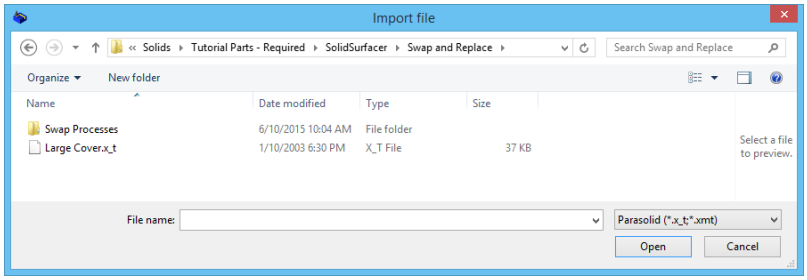
2. Choose 1) Rough Pocket.prc. There will be a slight delay as the Tool and Process that comprise this pocketing process are loaded.
3. Select the faces in the pocket the same way we selected the faces for the unstitch. Create the toolpath.
4. Deselect the Operation tile and Clear the process tile.
5. From the Processes menu select Load and choose 2) Finish Pocket.prc and create the toolpath.
6. Render the operation.
7. Deselect the operations and place the model in the Body Bag.

Modifying the Part

Importing the Changed Pocket

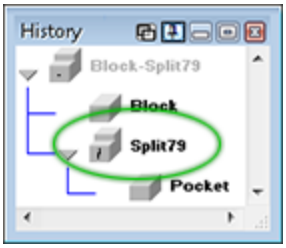
We are now going to import a model that is a different pocket shape. We will use the Replace function to insert it where the filled Pocket body was used.

1. File> Import the file Large Cover.x_t from the Swap & Replace folder.



Replacing the Model History

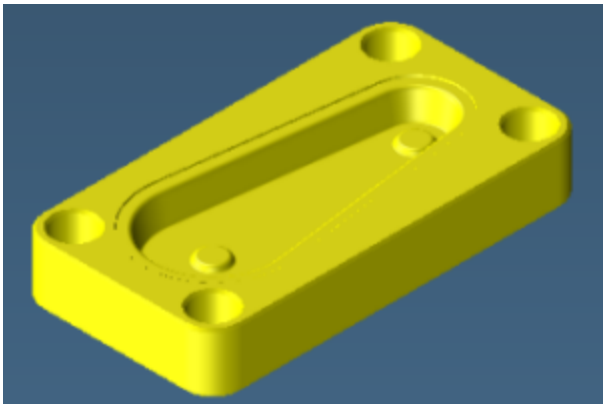
1. As with the smaller pocket shape, `Right-click` the bottom face and choose Select>  Select Faces Above and then select the tops of the two bosses.
2. Unstitch the solid to fill it and `Delete` the smaller solid.



3. `Double-click` the graphic to extract the smaller filled pocket from the History of the Block.
4. Select the filled Large Cover solid then the smaller filled pocket. `Click` the Replace button. 

   You will not see any changes but you may be wondering why we had to unstitch the larger pocket. Shouldn't we be able to just extract the original smaller body and replace that? The answer is, in this case, no. The modeler is not able to match up the faces of the two bodies. If we were using a function that was not face-dependent (translate, booleans, and so forth) we could just select the two original bodies and perform a swap or replace.

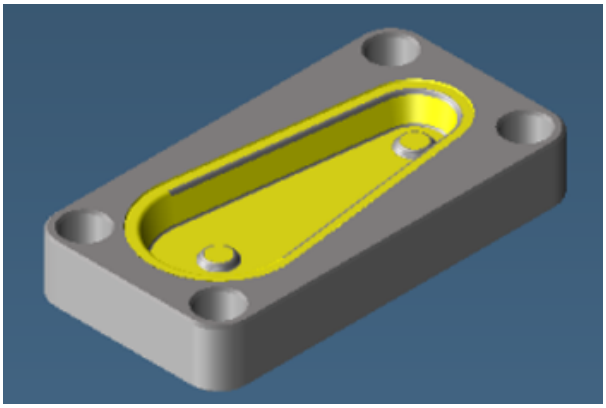
5. `Right-click` the Block model and choose  Rebuild.



Redoing the Operations

Now we will update the operations based on the new model.

1. Select Edit >  Redo All Ops and render the part.



2. Save the file; it is complete.

MACHINING SOLIDS

About these Tutorials

The part models for all of the following exercises were created in the Modeling Exercises chapter of this manual. If you have not gone through those exercises and created the solid models for these parts, you should do that now. We assume you are familiar with the interface and the principles presented in the Mill and Advanced CS Modules. If you are unfamiliar with that information please read through those manuals before attempting the following tutorials.

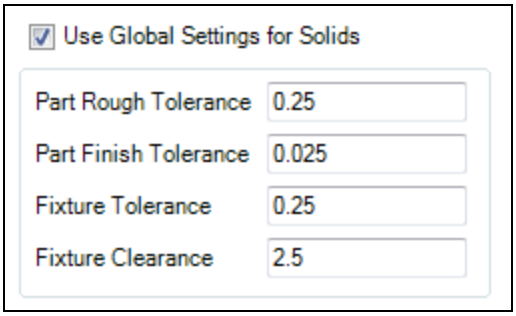
All parts in these exercises are assumed to be made of cast aluminum alloys. The feeds and speeds are all defaults, based on the CutDATA™ values. The exercises do not provide a step for setting the material but if you have CutDATA, please set the part material when you open the part file. If you do not have CutDATA simply use the default material – stainless steel. The feeds and speeds may be set by clicking on the calculation buttons.

Pockets and Contours

In this exercise, we will use a pocketing process to rough the part. The part will be finished using Contour operations on the whole body and selected faces.

Part Setup

1. Open the 2.5D Solids part Cap.vnccreated in modeling exercises. Set the Global Settings in the Machining Preferences tab of the DCD as shown.



2. Ensure that the part has a Clearance value of 15mm.
3. Create the following tool list.

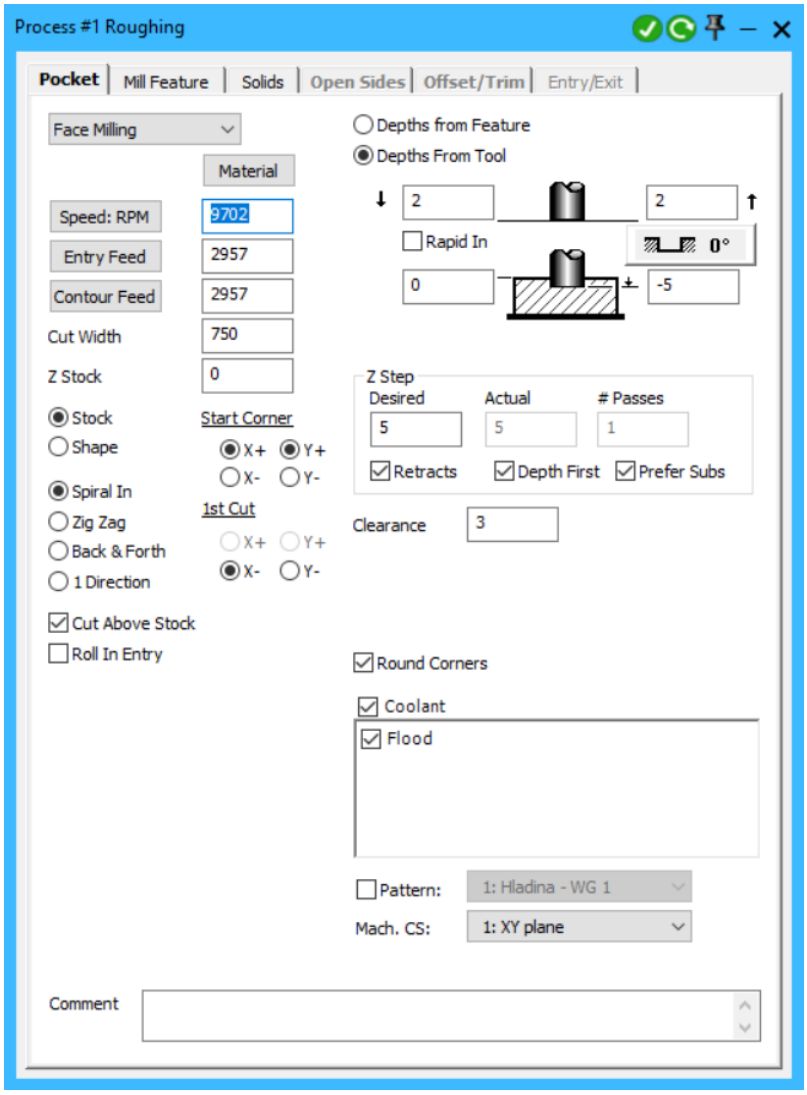
#	Type	Total Length	Diameter	Bottom Radius	# Flutes	Flute Length	Material
1	Face Mill	50mm	50mm	0mm	5	11.5mm	HSS
2	Rough EM	92mm	16mm	0mm	3	32mm	TiN Coated
3	Finish EM	66mm	10mm	2mm	3	16mm	TiN Coated

Machining the Part

#1, Face Milling

First we will Face Mill the part.

1. Create this roughing process with tool #1.

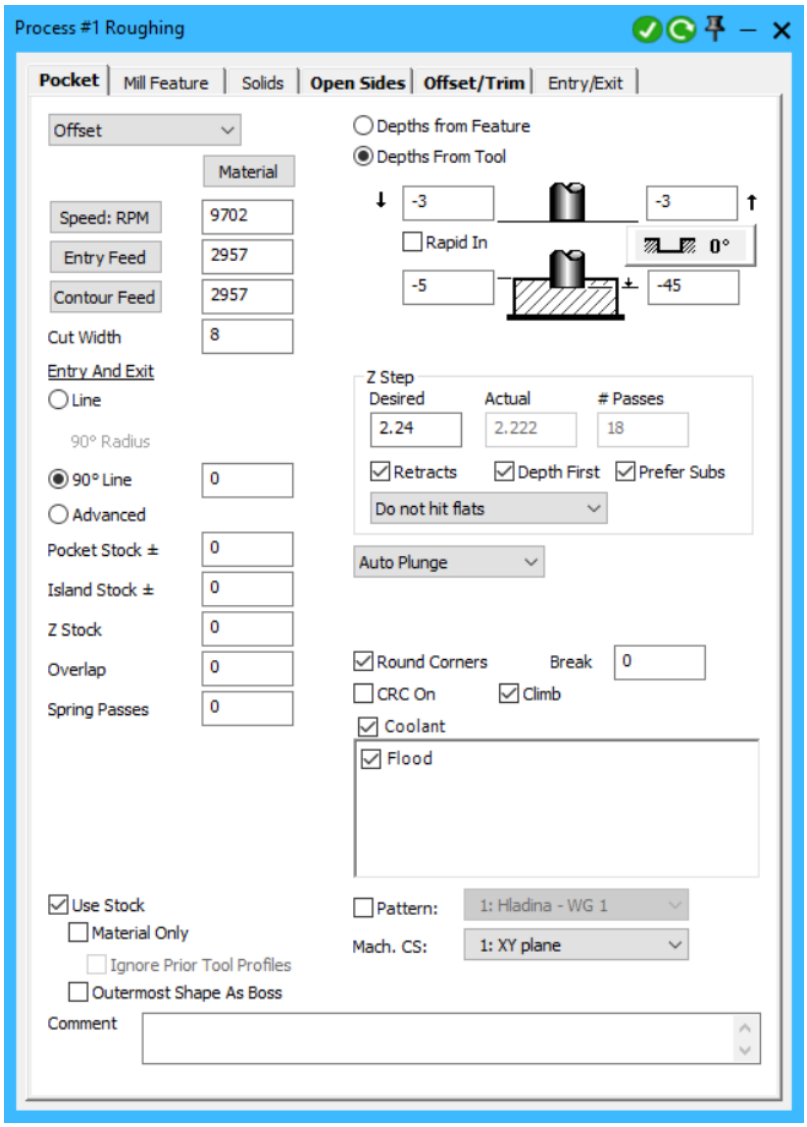


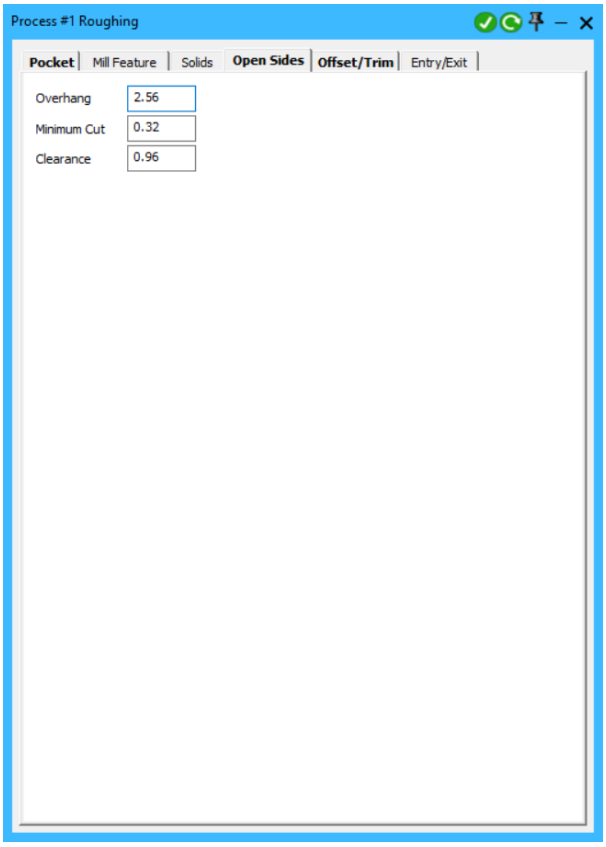
This will clear off the top of the part. Face Milling is not dependent on having a solid to machine. Face milling machines either selected geometry or the workspace stock.

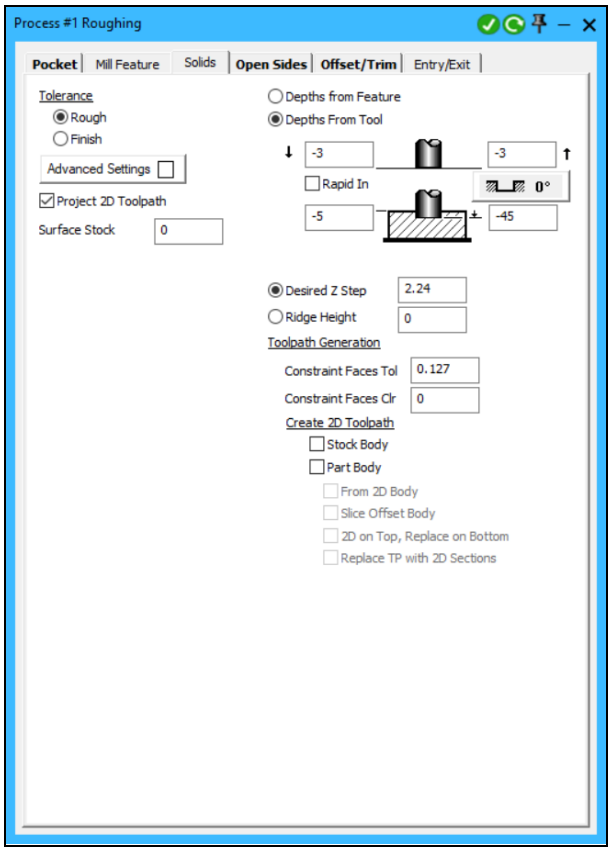
2. Create the toolpath.

#2-3, Pocketing

1. Create this Roughing process with tool #2.





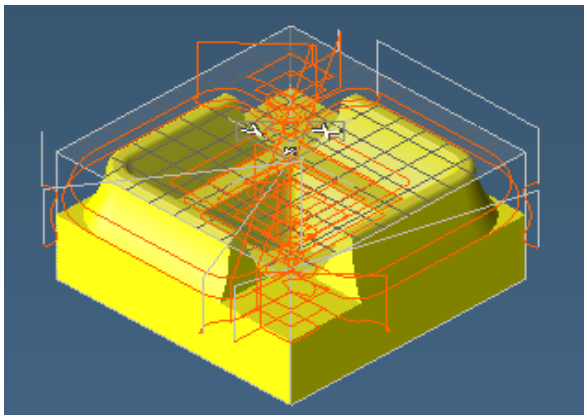


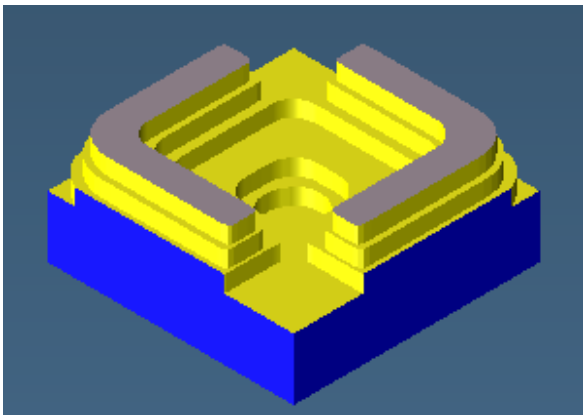
When all of the options are set, create the operation.

The Use Stock option will trim the toolpath to the stock condition. Since this is an open pocket we can get some drastically different results. If you wish to see the difference, try re-doing the operation with Use Stock off.

The Rough tolerance setting allows us to generate toolpath faster with looser tolerances. This will be fine since we are going to finish the surface in the next group of operations.

2. Create the toolpath and render.





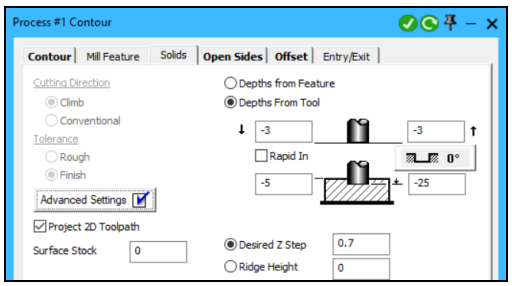
This process creates two operations.

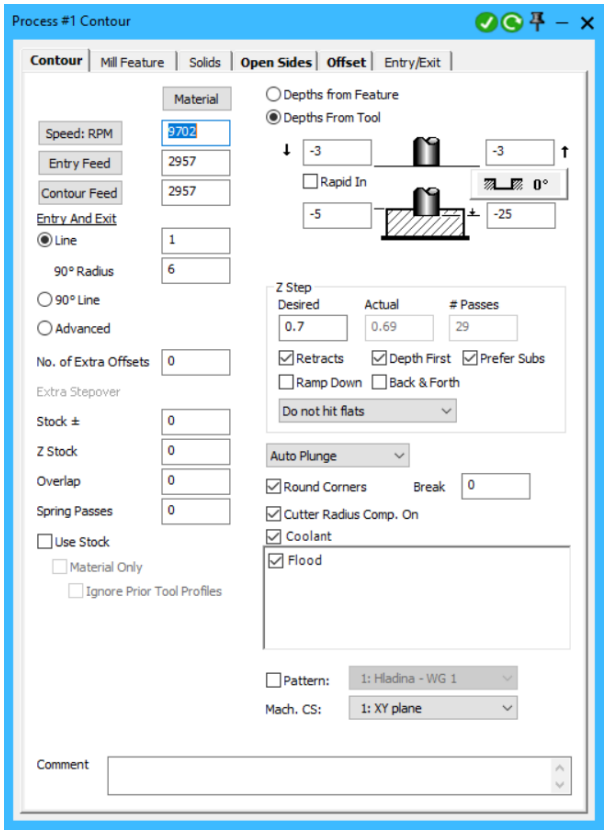
#4-5, Contouring

We will now do the first of two sets of contour operations to finish the profile.

1. Create this Contour process with tool #3.

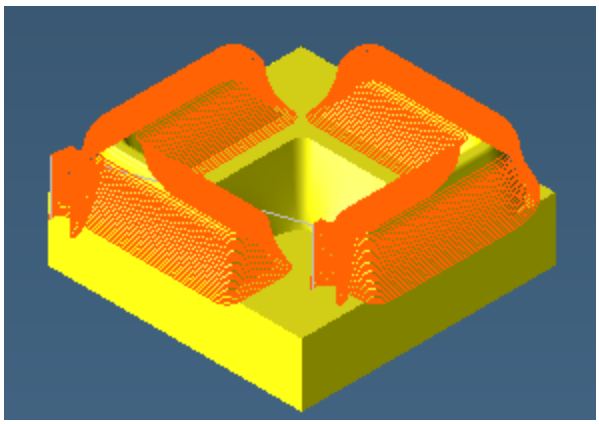
   The Desired Z Step determines how smooth the wall will be. Alternatively, the Ridge Height option can be used to specify the part's smoothness. Often the rendered part is far more accurate in showing any scallops that are on the part than our eyes can discern.

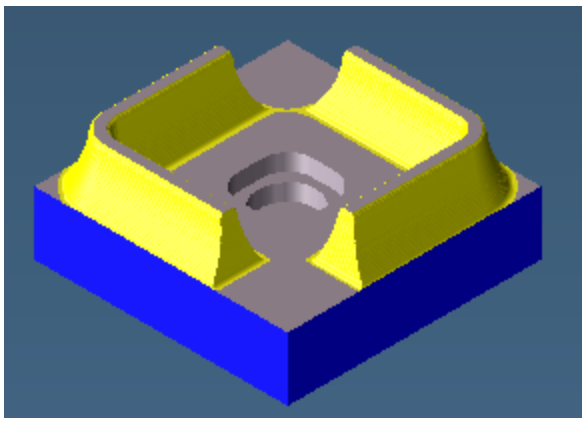




2. Create the toolpath and render.

   This operation produces smooth toolpath that wraps around the walls of the part. When rendered your part will look similar to the image to the right. Because we specified Depth First, the toolpath is completed around one wall before rapiding over and machining the other.

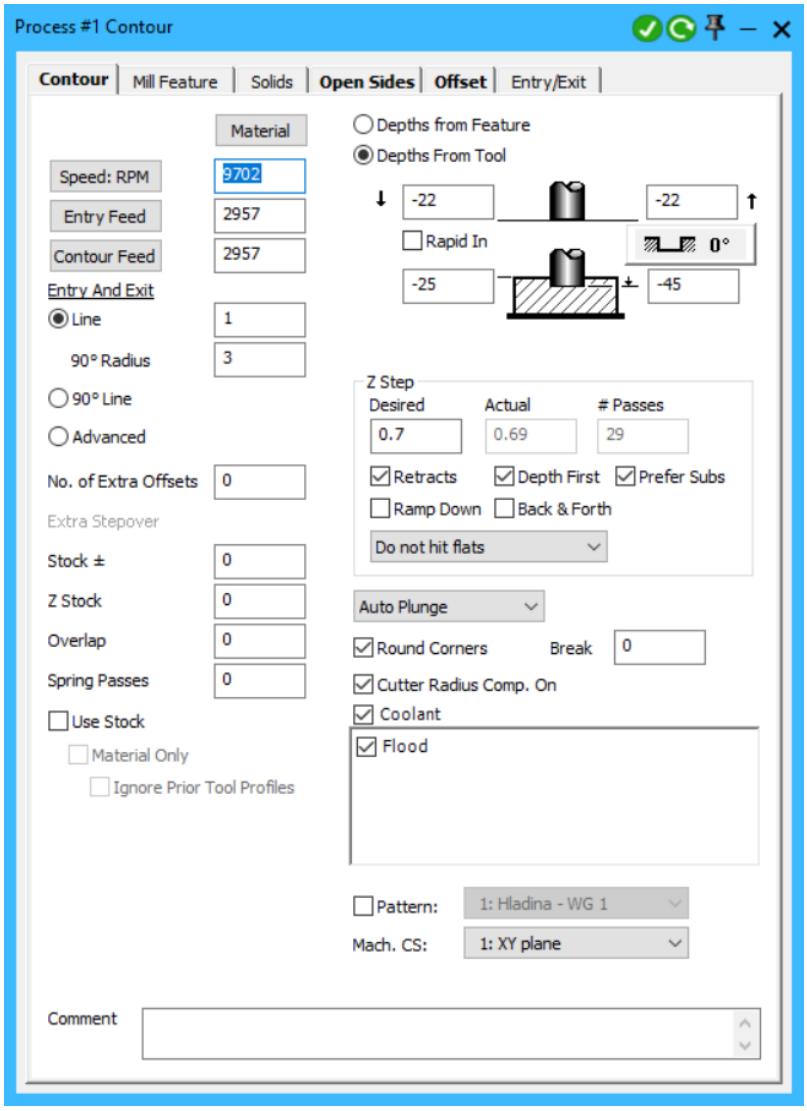




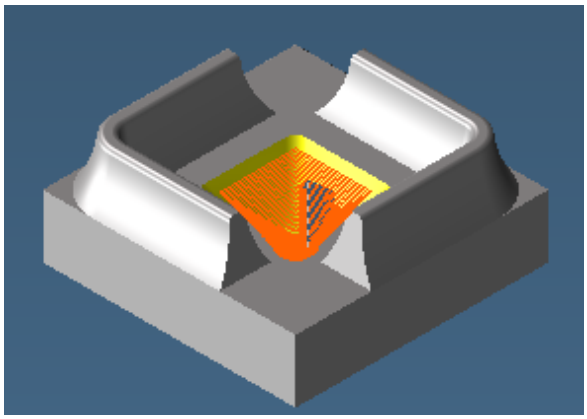
#6, Contour

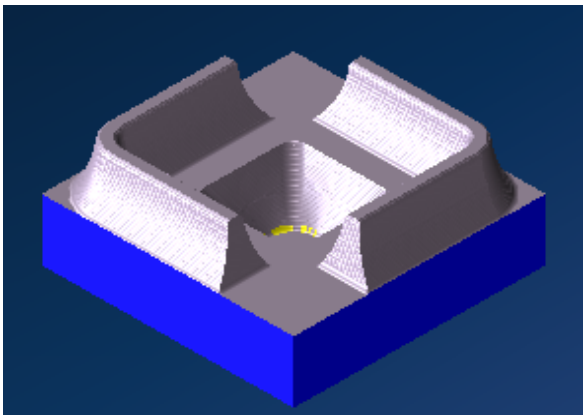
The last operation will contour the inner pocket.

1. Modify the existing Contour process as shown.



2. Turn on face selection mode.
3. Select the faces of the angled pocket.
4. Create the toolpath and render.





5. Save the part.

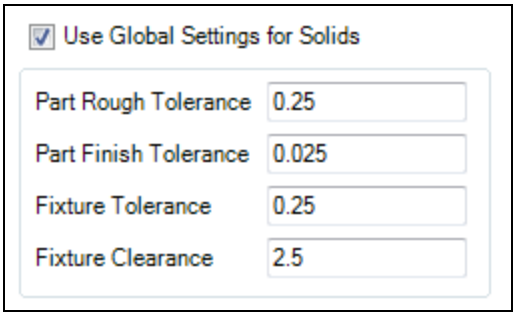
#2: The Hot Punch

The Hot Punch will only have one operation – a projected engraving operation. The part consists of a stock body. We will create a text shape and project the contour toolpath onto the model.

Part Setup

Document Control Dialog

1. Open the Hot Punch.vnc part. Set the global settings in the Machining Preferences tab of the DCD as shown.



2. Ensure that the part has a Clearance value of 15mm.

Tool List

1. Create the following tool.

#	Type	Total Length	Diameter	Corner/Tip	# Flutes	Flute Length	Material
1	3.15mm Center Drill	31.5mm	3.15mm	118°	2	1.9mm	Carbide Solid

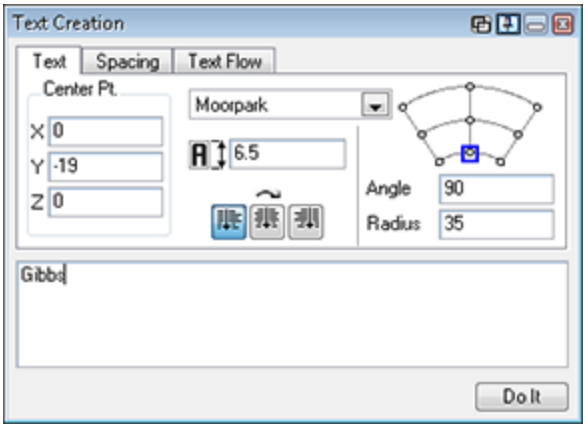
Creating Operations

#1: Engraving

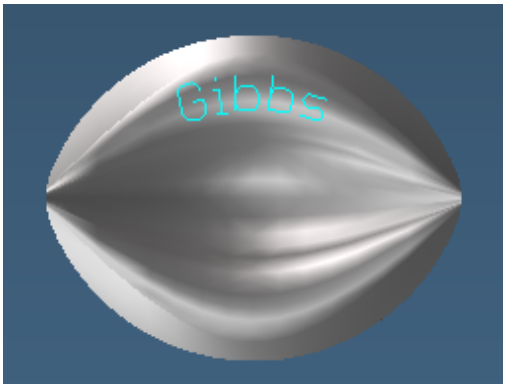
We will create text and engrave the top face of the part. The contouring process provides for centerline machining of all selected shapes which include text and artwork. Contouring operation toolpaths can be projected on to sheets and bodies.

We will need to create text geometry that can be machined. The system can create spline geometry from any TrueType font. You may need to set the directory that contains the fonts for your system. There is a Font Directory item in the Preferences > File submenu which allows you to designate a directory that contains your system fonts. The Moorpark TrueType font used in this exercise is shipped with each order.

1. Open the Text Creation dialog from the Shape palette in the Geometry Creation palette.

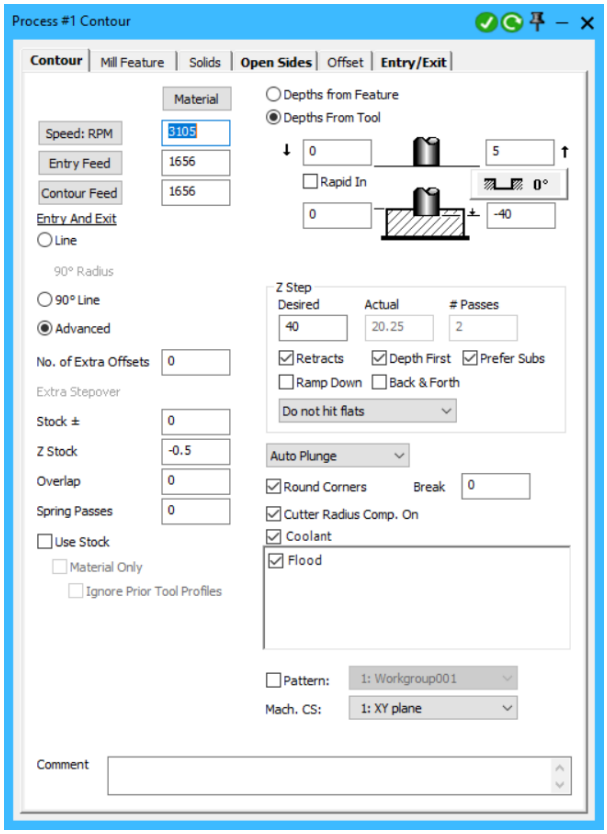


2. In the Text Flow tab select the clockwise arc.
3. In the Text Flow tab enter the information shown.
4. Click the Do It button to create the text.



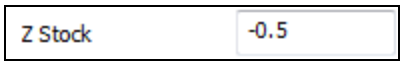
If you have any problems with text creation, refer to the Text Creation Exercise in the Geometry Creation Manual.

5. Select the text geometry.

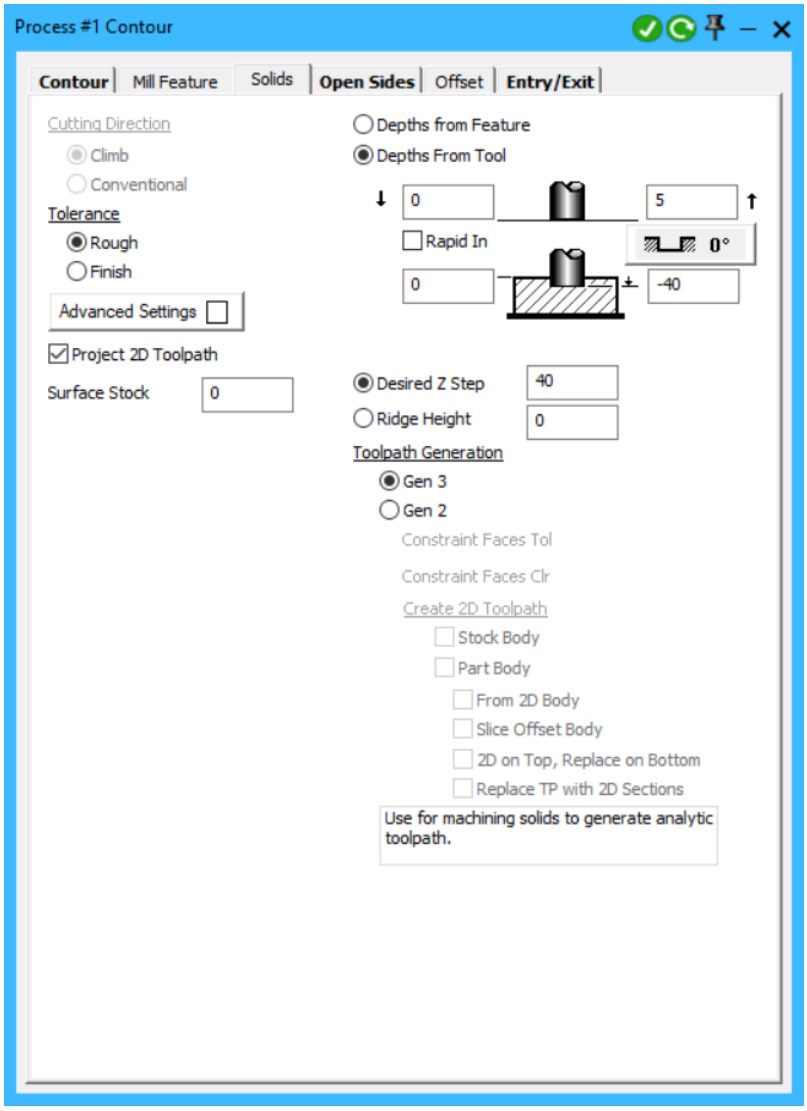


6. Create this Contouring process with tool #1.

   When more than one shape is selected prior to creating a contouring process, the system automatically assumes that you are doing engraving. When this is the case, several of the items in the Contouring Process dialog will be grayed out. The Z Stock option is used in this case so that the engraving toolpath will cut into the selected body. The toolpath generated by the operation will be shifted down along the Z axis only by the amount specified. The toolpath itself will be projected onto the surface of the body and then will be shifted down in Z to cut into the body.

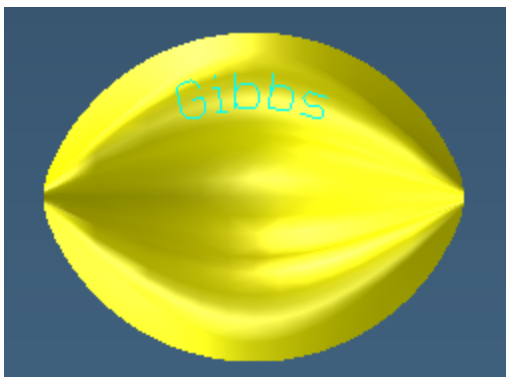


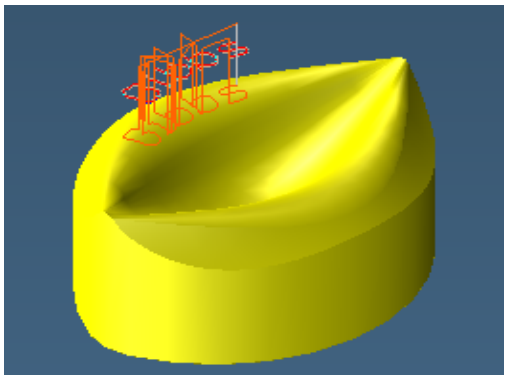
7. In the Solids tab enter the information shown.



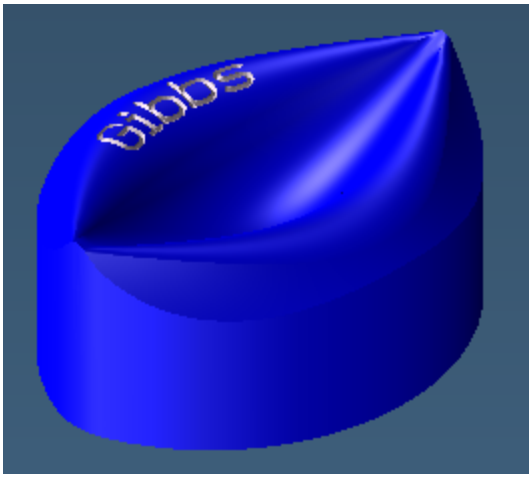
8. `Ctrl+click` the model.

   The text and the model should both be selected. When 2D geometry and a solid or sheet are selected for the cut shape of a process, the toolpath will be a projection of the 2D toolpath onto the body or sheet.





9. Create the toolpath
10. Render the operation.



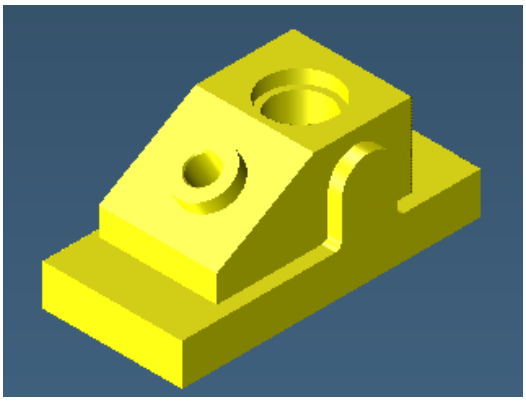
#3: The Base

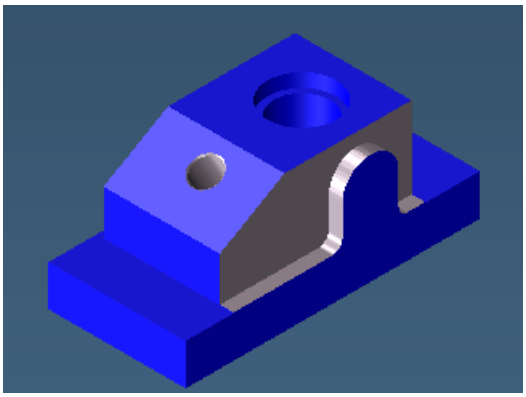
About the Part

This part is intended to help you become familiar with two important tools - the Profiler and using constraint shapes and faces.

1. Open the part file Base.vnc.

   This part has an existing stock condition and two groups of operations including two drilling operations and a roughing operation. Drilling in 2.5D Solids, as with the standard Mill package, is accomplished with geometry or the Hole Wizard so we do not need to go over the Holes process.



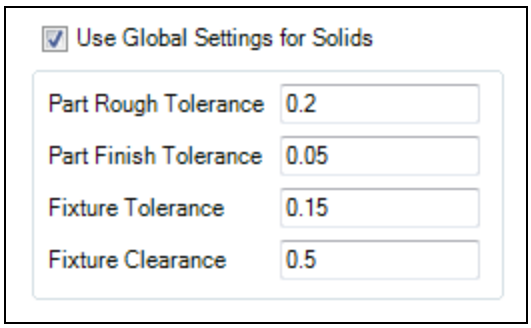


   The roughing operation uses a combination of containment faces to achieve the desired results. Let's look at how this works.

Part Setup

The part should be ready for you to apply operations to it but to be safe we will go over certain aspects.

1. Ensure that the part is using a 5 Axis Vertical Mill MDD, that the Clearance Plane is set to 350mm and that the Global Settings in the Machining Preferences tab are as shown.



The part has four tools as detailed below.

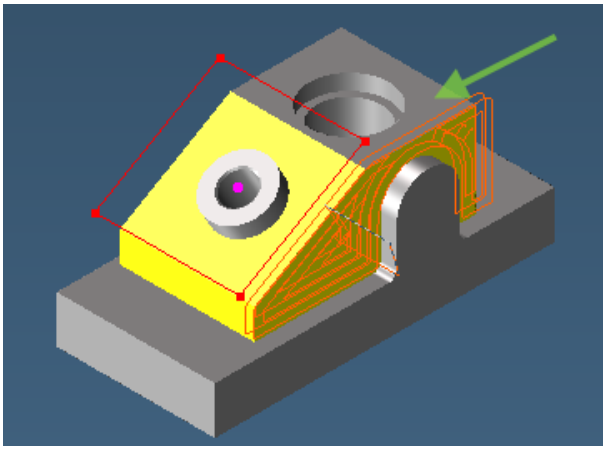
#	Type	Total Length	Diameter	Bottom Radius/Tip	# Flutes	Flute Length	Material
1	Drill	260mm	80mm	90°	2	n/a	TiN Coated
2	Drill	260mm	40mm	118°	2	n/a	TiN Coated
3	Rough EM	121mm	25mm	0mm	3	45mm	TiN Coated
4	Finish EM	72mm	10mm	0mm	3	22mm	TiN Coated

Machining the Part

Op 3, Roughing

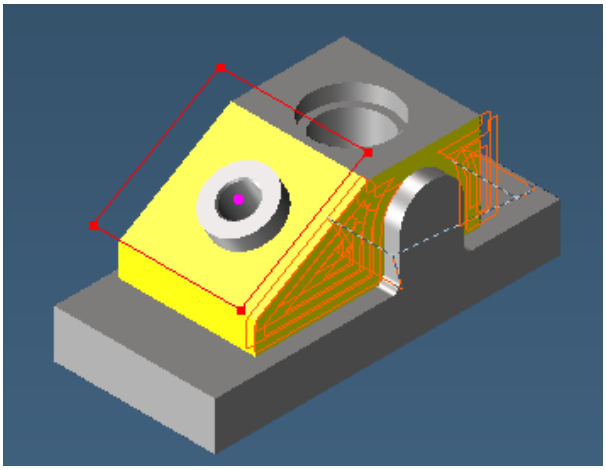
1. Double-click operation #3.

2. Turn on Face Selection and deselect the top flat face as shown.



3. Redo (Ctrl+,) the operation.

   The results are very different. By deselecting the face we have told the system that face is a constraint and we cannot machine by it.



4. Reselect the top face and Re-do the operation to return it to its original state.

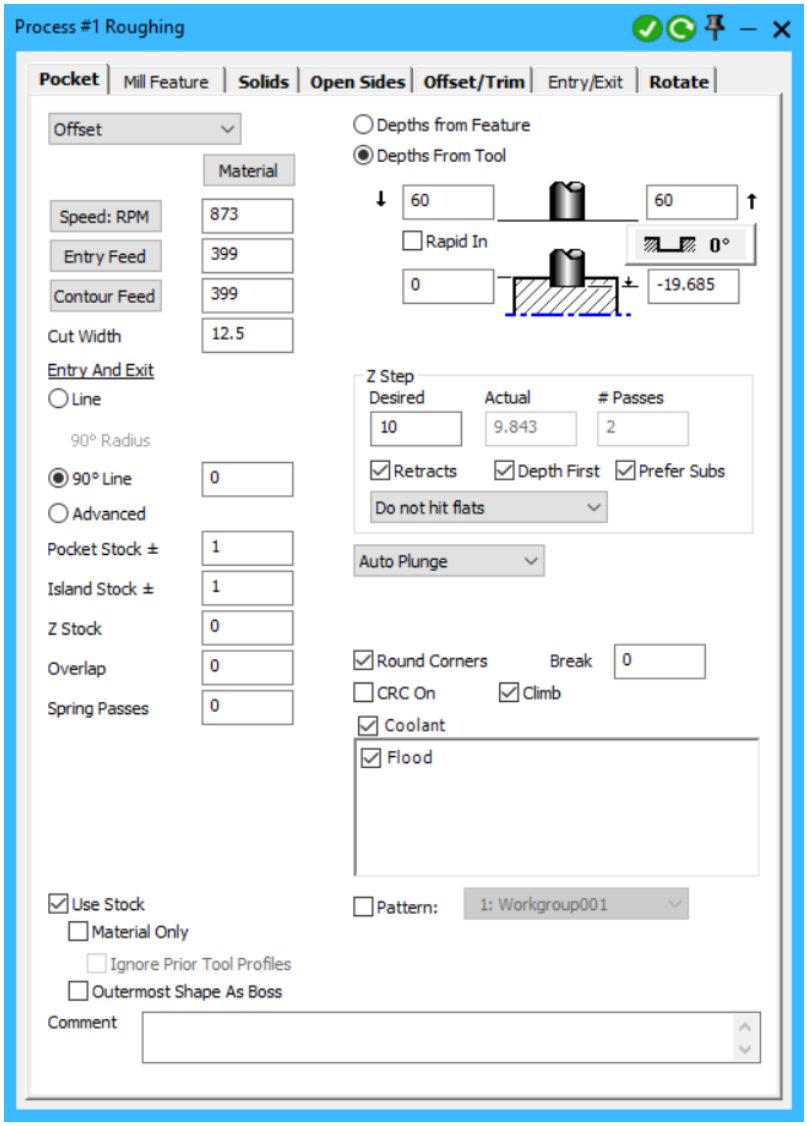
Ops 4-6, Rough and Contour

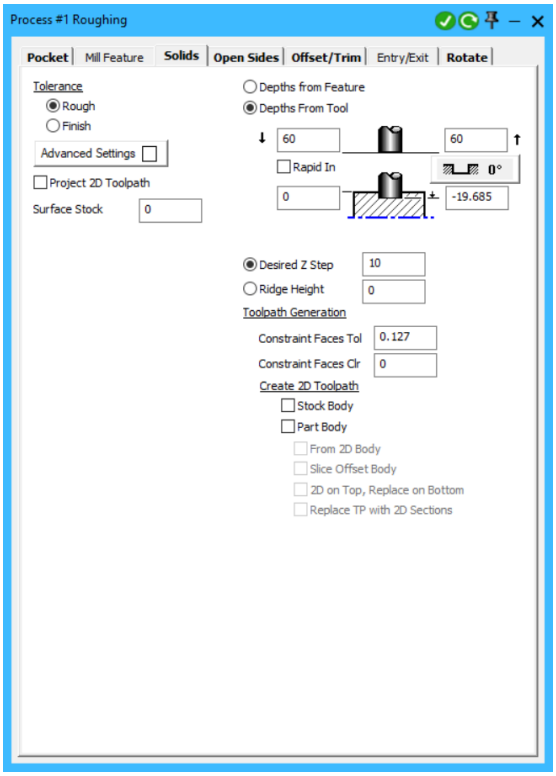
We will now create a roughing and contouring process group to machine the angled face.

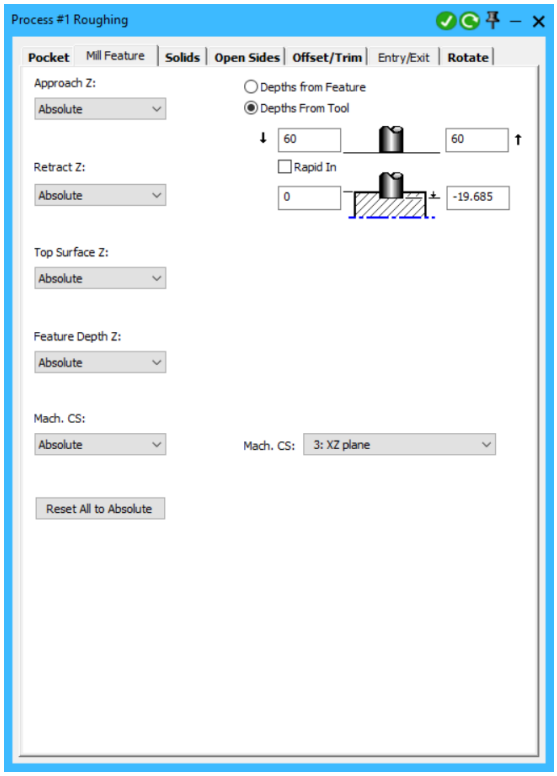
1. Switch to CS2.

2. Create this roughing process with tool #3.

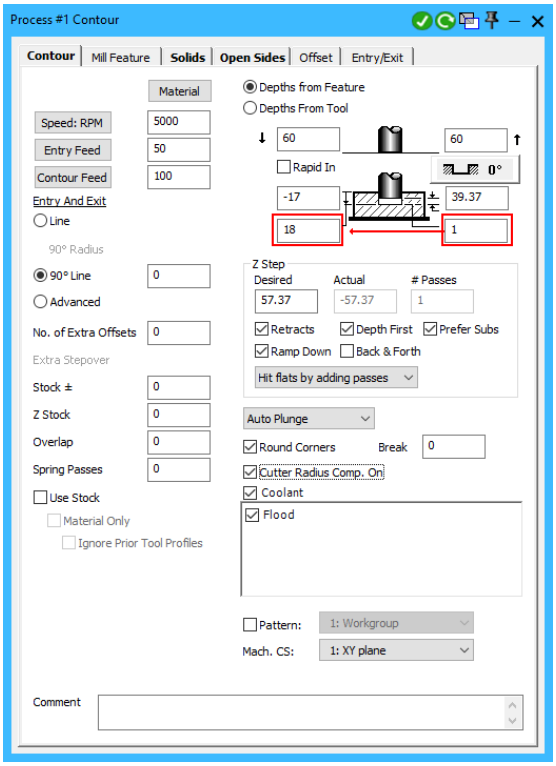
   The Surface Z and Depth Z values are acquired by interrogating the top of the circular boss and the large flat angled face. If your machining CS is set to something other than CS 2 you may get different results.

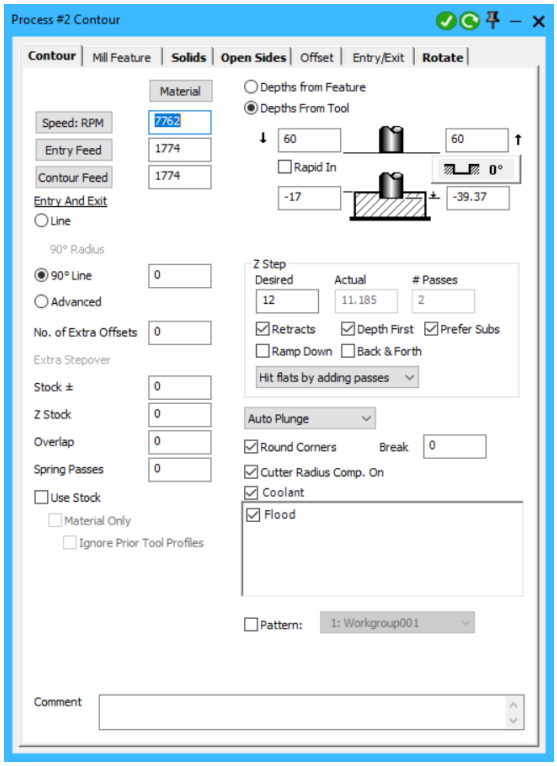






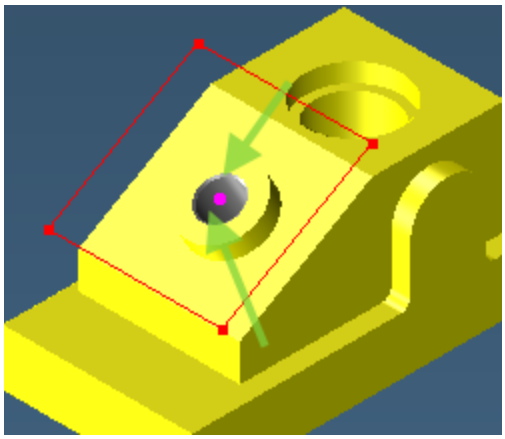
3. Create this contour process with tool #4.





As usual, the rotations information is automatically acquired.

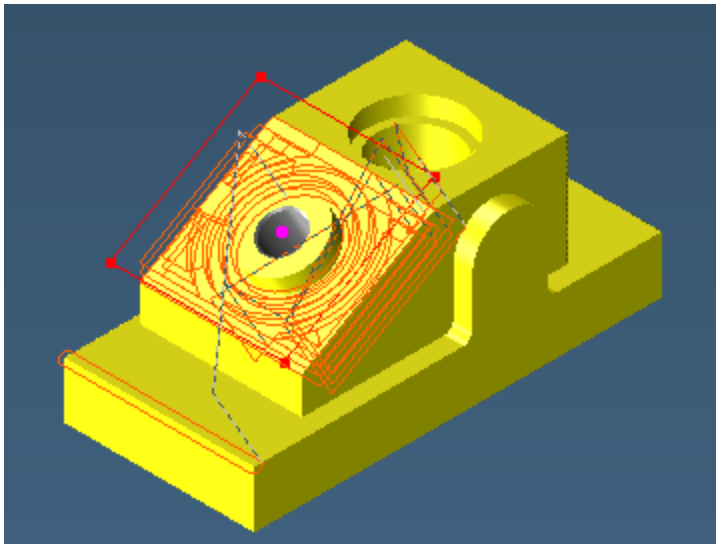
4. Select All (`Ctrl+A`) the entire model, then deselect the two faces that define the hole in the boss.



We have already drilled out this hole and we do not need to waste toolpath on it.

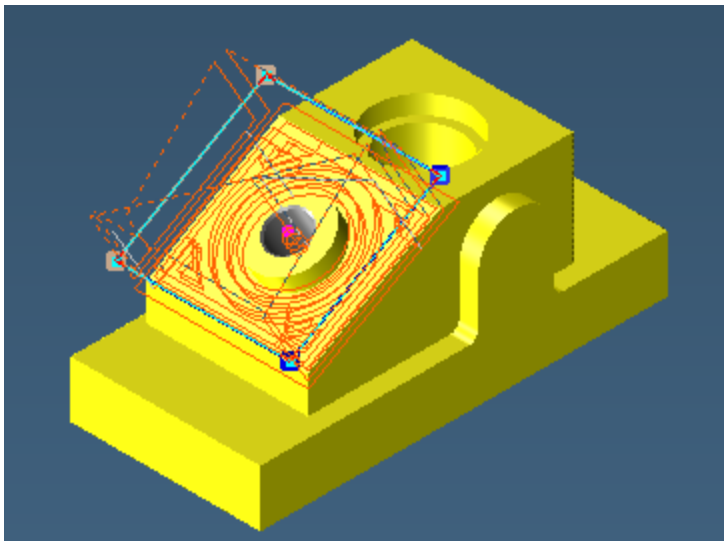
5. Create the operations.

   You can see that we have some toolpath that is not necessary. We forgot to select the constraint geometry.



6. `Ctrl+double-click` on the “air” geometry and redo (`Ctrl+,`) the operation.

   These results are more desirable.



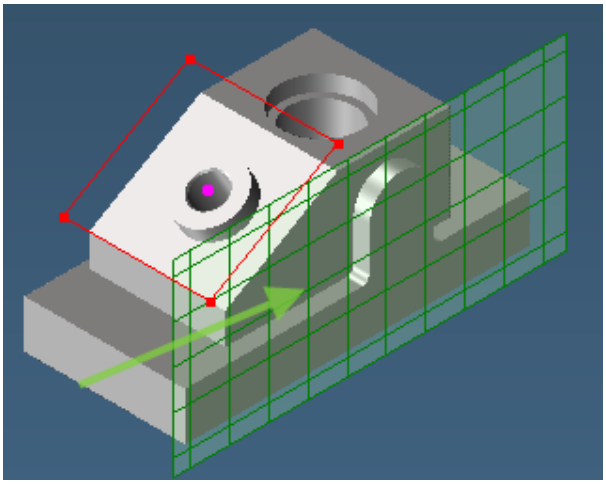
Op 7, Contouring with the Profiler

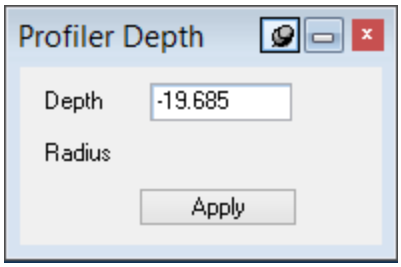
The last thing we will do is use the Profiler to machine a selected area of the part. Operation #3 leaves stock on the rounded wall.

1. Switch to the XZ plane.
2. Activate the Profiler. 
3. Turn off the Coordinate system grid.

   You will find it easier to see and work with the Profiler if the CS grid is off.

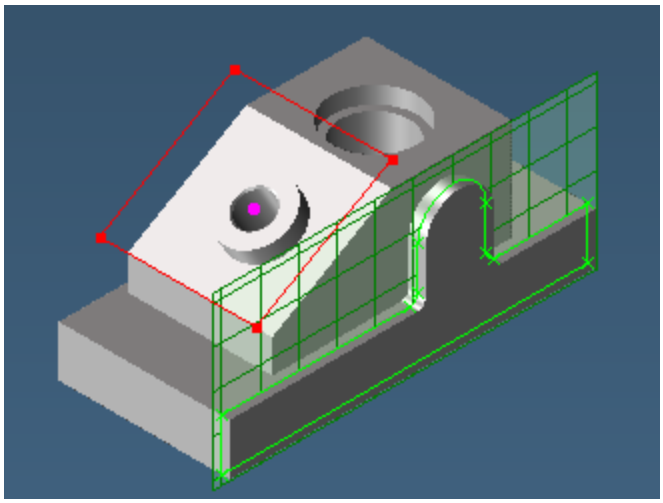
4. `Right-click` the profile grid and choose  Profiler Depth.
5. Interrogate (`Alt+click`) the face shown and `Click` the Apply button.

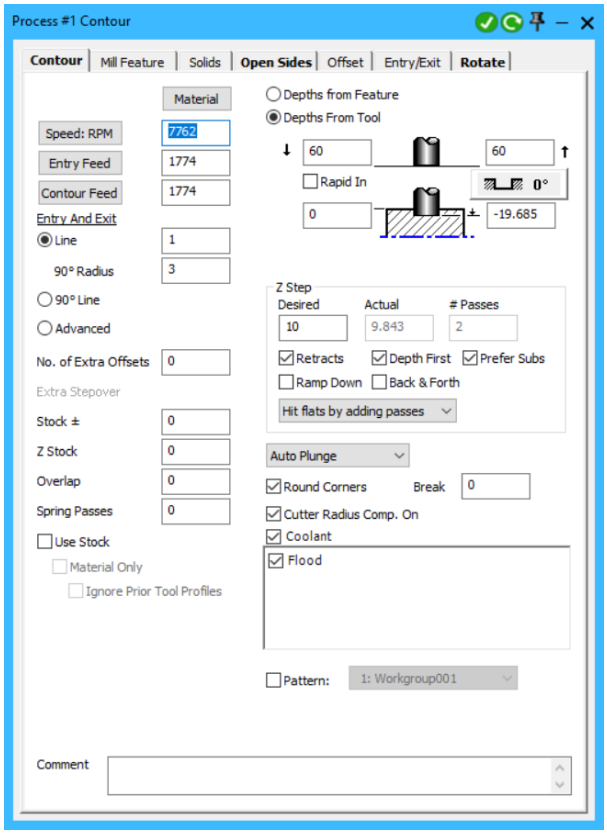




Unfortunately at this depth the Profiler does not see the rounded wall. We will have to manually move the Profiler.

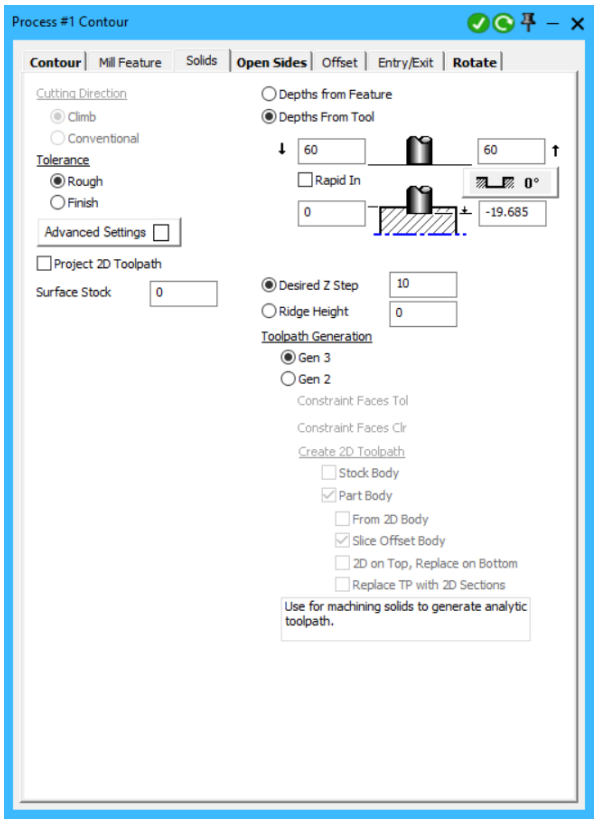
6. `Click` anywhere on the Profiler's grid and drag the Profiler to a location in the center of the rounded wall, as shown above.

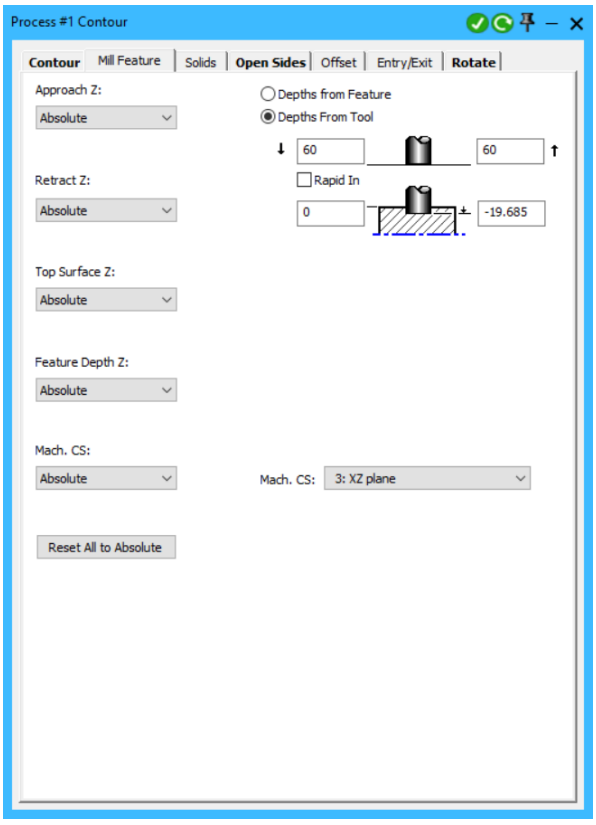




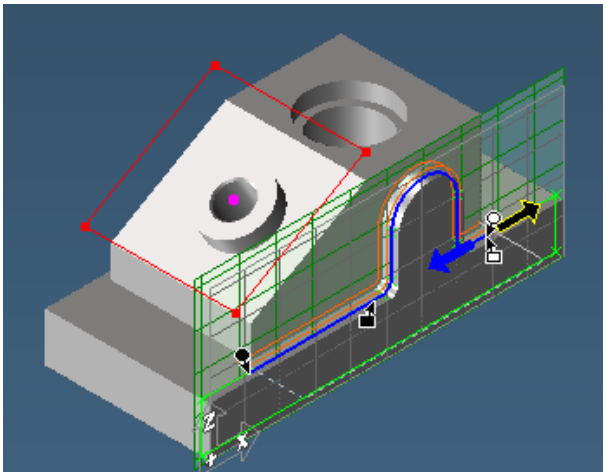
7. Create a Contour process with tool #4 as shown.

   The Final Z depth can be acquired by interrogating the same face we tried to use for the Profiler depth.



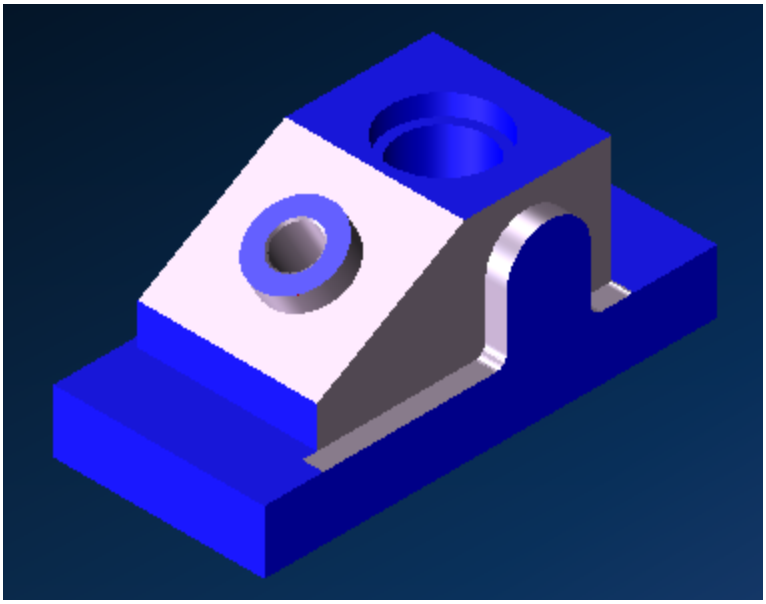


8. Click the Profiler and set the machining markers as shown, then create the operation.



You may find it necessary to go to the Home view to get the markers in the correct location.

9. Render the operations.



10. Save this part.

PART PRINTS

Spherical Ellipse