GIBBSCAM® 14

Version 14 : September 2020

SolidSurfacer Tutorials

CAMBRIO

Proprietary Notice

MODELING EXERCISES

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: Phone

- Creating The Phone
- Modifying the Phone

Creating The Phone

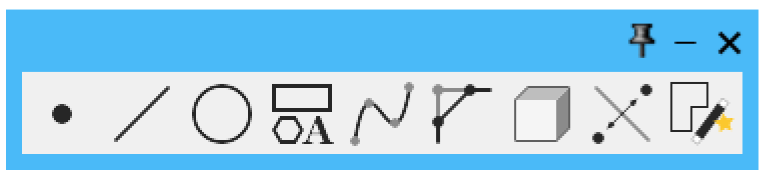
Part Set-Up

For the Part Print for this exercise, see "Part Prints" on page 163.

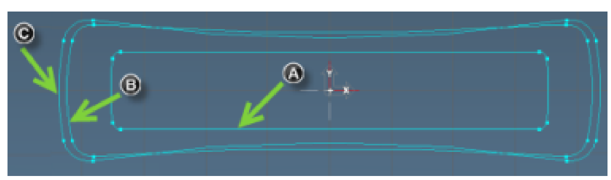
1. Create a metric 3 Axis Vertical Mill part with the following workspace stock size: +X= 120, -X= -120, +Y= 50, -Y= -50, +Z= 0, -Z= -75
2. Save this part as `Phone.vnc` for later use in the machining exercises.

Lofted Profile

1. Open the Geometry Creation  palette.



2. Create the geometry for segments A, B and C.



Segment A

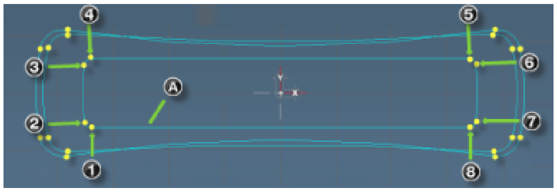
Two horizontal lines and two tangent arcs define the top shape. In order to create tangent arcs, a point and line need to be created. Create the vertical lines and a point tangent to the lines at Y0, then define the arcs tangent to the line and the point. Since the points appear at zer0 you may find it quicker to use the Mid-Point feature. Add the fillets after connecting the inner circles to the horizontal lines. Alternately you may define this shape easily in Geometry Expert, since the center points of each circle can be determined by subtracting the tangent distance from the radius. However, be careful to select the correct connection points using this method.

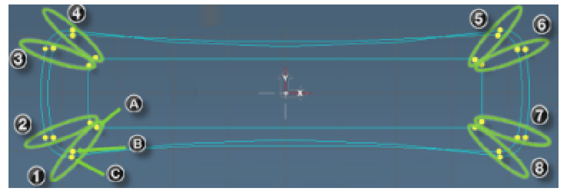
Segment B & C

Four arcs define the next two shapes. Similar to Segment A the tangent lines will need to be created first. Connect the appropriate geometry and add the fillets. These shapes are also well suited for Geometry Expert. Use Modify > Force Depth to adjust the Z depth of segment B and C.

We will now create a solid from the geometry in WG 1 using the Loft tool. For a smooth blend, we will need to select as many alignment points on each of the three shapes as possible.

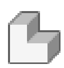
3. Ctrl-click the sync points shown. Select each point in sequence and in the direction of the loft, from Segment A to B followed by C (As above).

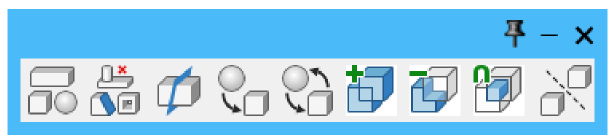


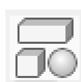


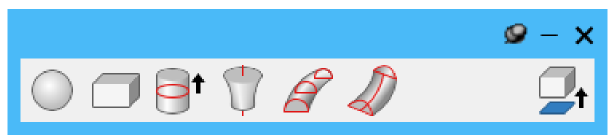
Method 1 - Shape sequence starting with segment A	Method 2 - Shape to Shape

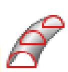
Alignment points are used to determine how the shapes will be aligned and blended. Alignment points must be selected in the right order. Alignment points can be selected in two ways, selecting alignment points from shape to shape or select a single shapes alignment points, in order, followed by the next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

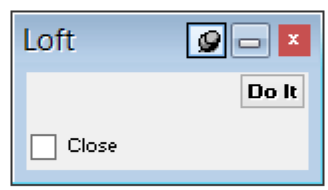
4. Open the Solid Modeling  palette.



5. Open the Create Solid  palette.

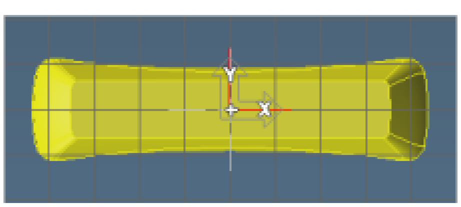


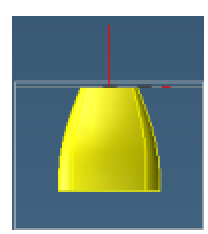
6. Open the Loft  dialog.



7. Click the Do It button.

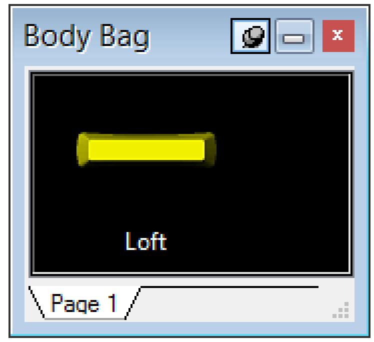
The results of the loft will look like the images shown here.





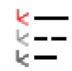
Top View	Side View

8. Double-click the loft to place it in the Body Bag.

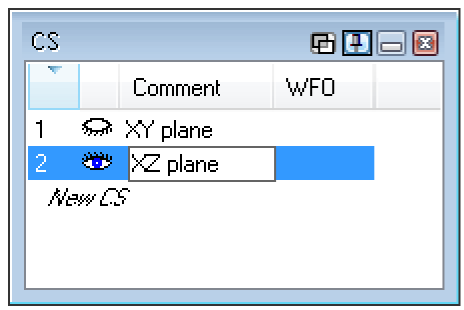


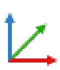
Extruded Profile

1. Open the CS list.



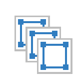
2. Create CS2 *New CS* and label it XZ plane.



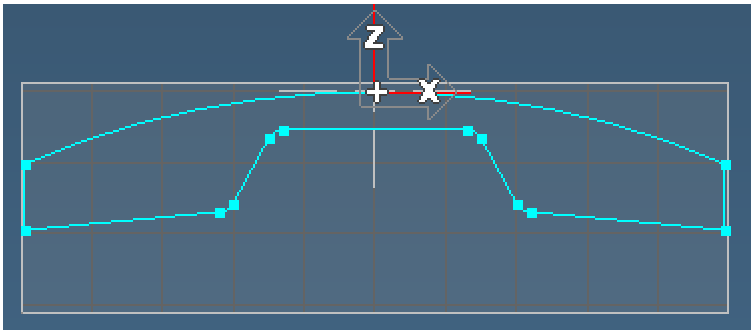
3. Open the CS  palette.

4. Click the XZ button.

5. Open the Workgroup list and create WG 2 *New WG* for the XZ geometry.



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



7. Create the geometry for segment D, in CS2 XZ plane.

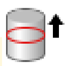
(For the Part print for this exercise, see Part Prints)

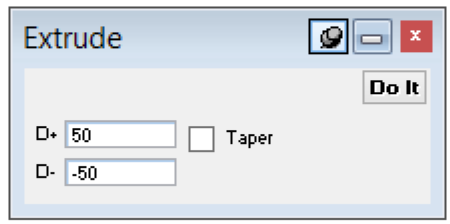
Segment D

This shape is made up of two main tangent arcs, two vertical lines and four angled lines.

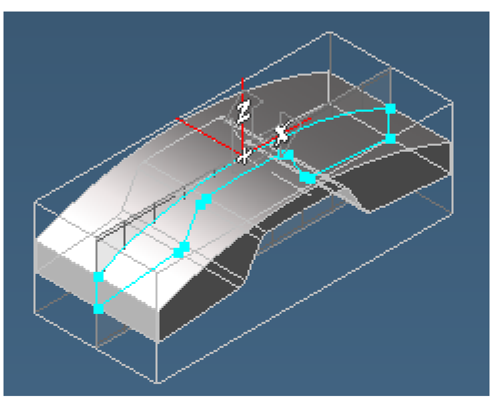
We will now create an extrusion from the geometry. Extrusions are done along the depth axis of the current coordinate system.

8. Select any part of the segment D geometry and switch to the isometric view (Ctrl+I).

9. Open the solid Extrude  dialog and enter the depth shown.



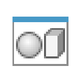
10. Click the Do It button.



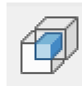
Profile Intersection

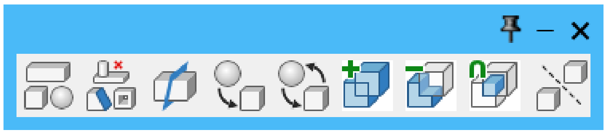
We will intersect this extrusion with the loft in the Body Bag.

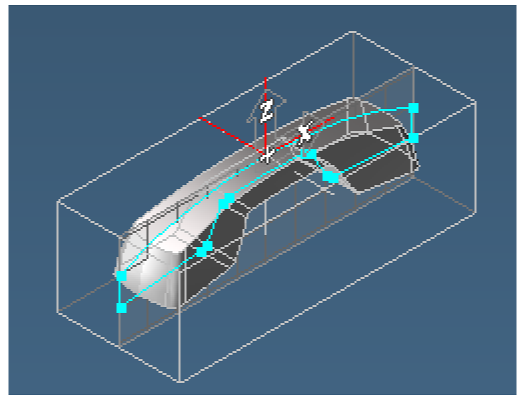
1. With the Body Bag open, press (Ctrl+A) to select all.



Since there are only two solids selected we can now perform an intersection. The selected geometry will be ignored when performing the function. Unlike other Booleans the order of selection doesn't matter with Intersection.

2. Click the Intersection button  in the Solid Modeling palette. You will be left with the shape shown below.



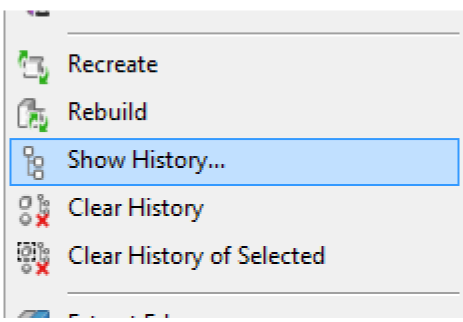


Extracting a Body

Now we will create a core for the mold using the History list.

1. Click the show geometry button in the floating taskbar to hide the geometry. 

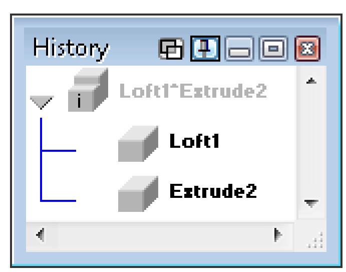
2. Double-click the model to place it in the Body Bag.



3. Right-click the model you just placed in the Body Bag and choose Show History.

   The History list will open with the Extrude and Loft bodies that were used to create the intersection.

4. Double-click the atomic body  Extrude2.



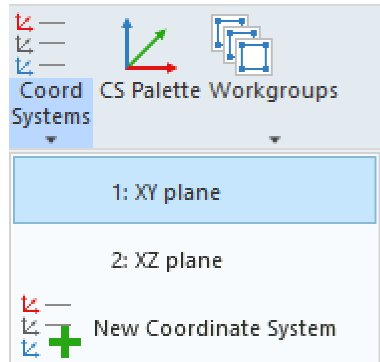
The extrusion will be brought back selected in the workspace.

5. Place Extrude2 in the Body Bag.

> ⓘ If a History list is large, an easy way to find the correct body is to open the body properties dialog and click each entry in the tree. A preview image will be shown of each item.

Making the Base

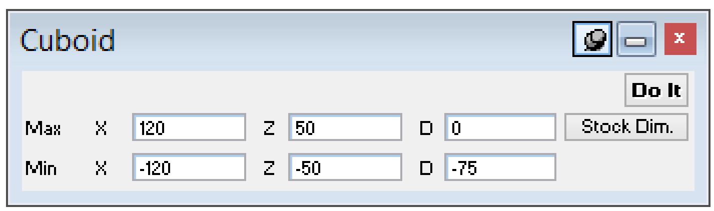
1. Switch to CS1: XY plane. (Hover over the CS button, then slide mouse down).



We will now create a cube based on the workspace stock size. it is important to be in CS1 XY coordinate system so that the cube will be in the correct position.

2. Open the Cuboid dialog from the Create Solid palette.

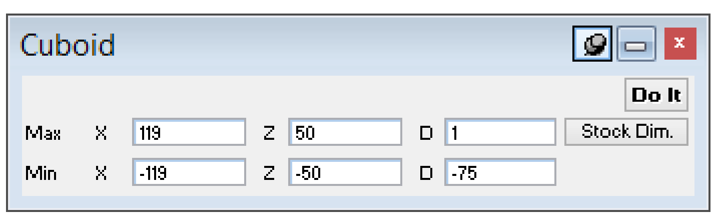
3. Click the Stock Dim. button.

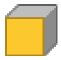


The Stock Dimension button applies the workspace stock size from the Document control dialog to the cuboid text fields.

The core size needs to be adjusted to perform a subtraction. If the solids were subtracted with these values it would result in an invalid body because of the tangency in the loft at Z0. We also need the result to be two separate bodies so we need to adjust the X values of the cube to match the extrude. 1mm will be enough to create the needed solid.

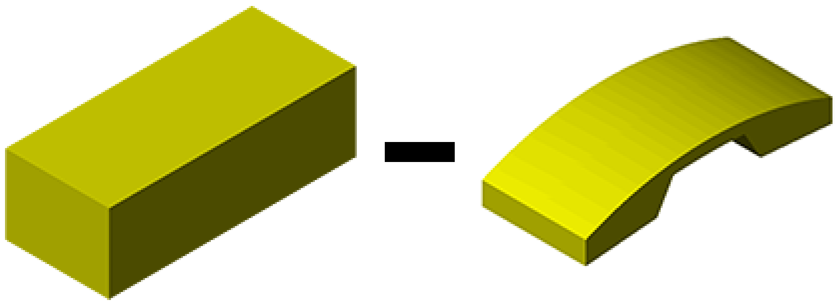
4. Adjust the X and Z values as shown and press Enter or Do It.



5. Make sure Face  and Edge  selection are turned off and `click` the cube, then (`Ctrl+click`) the extrusion in the Body Bag.

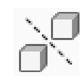
When performing subtraction operations, the order the bodies are selected in is important. The second body selected is subtracted out from the first body selected.

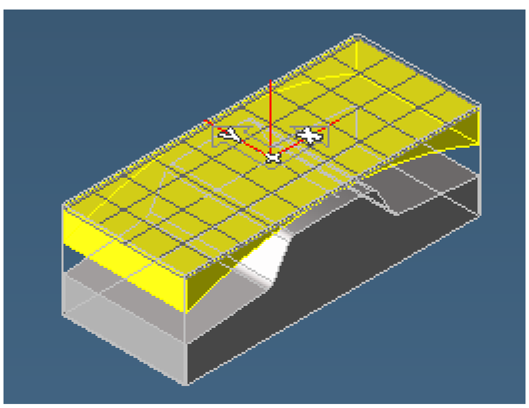
6. `Click` the Subtraction button.



Only the bottom portion of the new solid is needed for the mold core. The body created from the subtraction is an unconnected, multi-lump body that can be separated into two pieces.

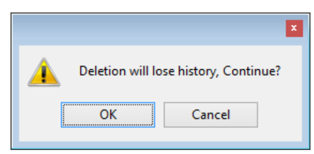
7. `Click` the subtracted body and `click` the Separate button.





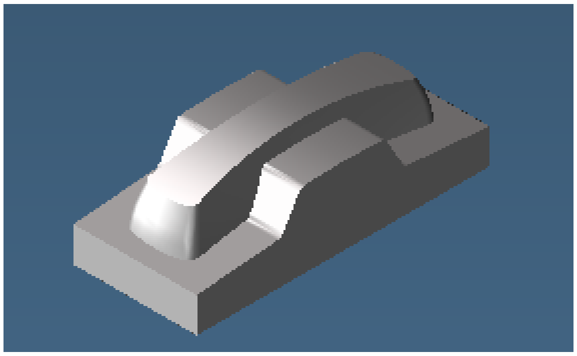
8. `Click` the top portion as shown and `click` the Trash button  or press `Delete`.

A message will appear indicating that the history of the deleted object will be lost if deleted. That portion of the body is unnecessary for the model, so deleting it is not a problem. `Click` OK.



9. Select the two remaining solids and add them together.

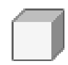
10. Place this solid in the Body Bag.

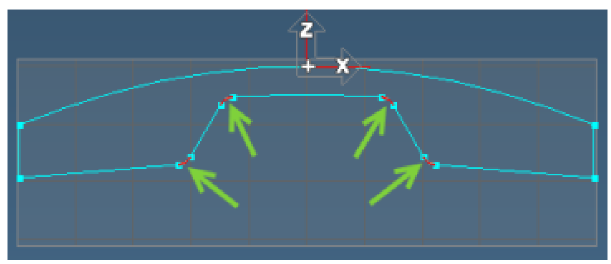


Modifying the Phone

Filleting Corners

A model review has found that the underside of the phone needs to have a 6mm fillet that wasn't in the original plan. Luckily there is an easy way to fix the part.

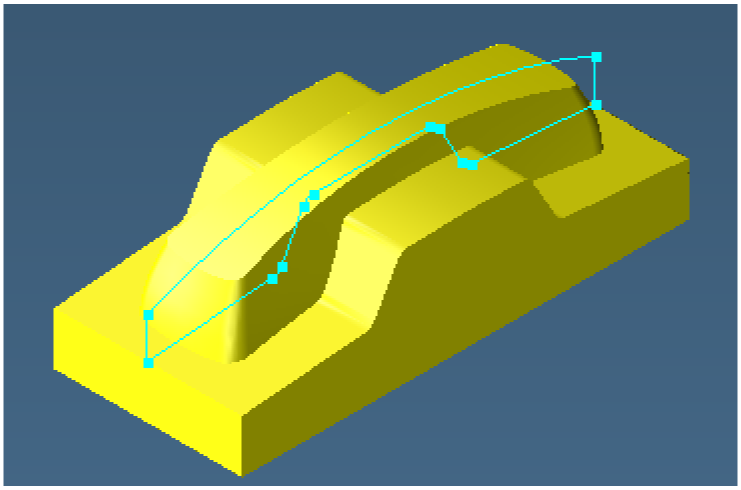
1. Click the show geometry button  and the show solids button  to hide the solids.
2. Switch to CS 2 XZ plane and WG 2.
3. Add the 6mm fillets as shown.



4. Extrude a new solid with the fillets.

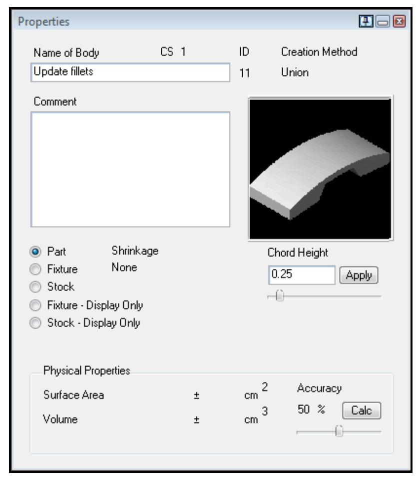
Solids will become un-hidden when creating or modifying them.

With the edges still active you can now see the fillets clearly. Because the revision is small however, it is a good idea to rename this body so it is easy to distinguish it from the other extrude.



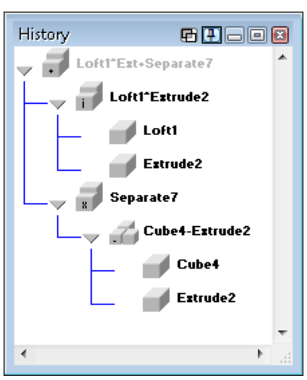
Renaming a Body with the Properties Dialog

1. Right-click the new extrusion and choose Show Properties from the context menu.



2. Rename this solid Update fillets and place it in the Body Bag.

Replacing a Body in the History

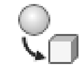


1. Open the History of the main model.

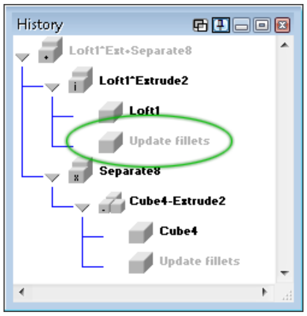
2. Double-click Extrude2 to extract it from the History of the main model.

3. Select the Update fillets solid and (Ctrl+click) the Extrude2 solid.

4. Click the Replace button and Delete Extrude2.



Rebuilding the Model



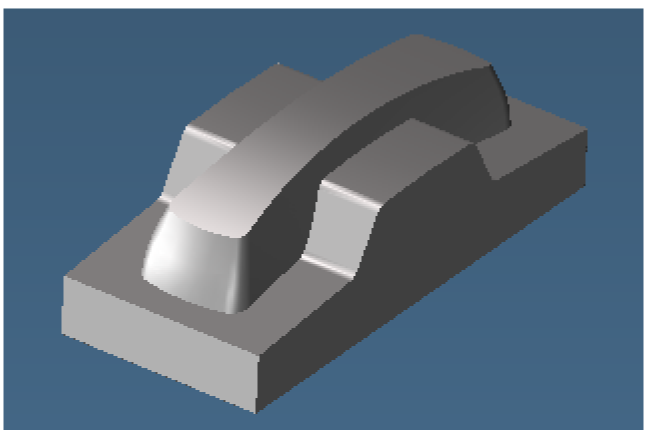
1. Click the main model to see the Update fillets solid is in the History list where the Extrude2 solid was.

   The entry is disabled because it hasn't been incorporated with the rest of the model.

2. Right-click the main model and choose Rebuild.

   The new model is rebuilt using the Update Fillets body. The part is now complete.

3. Load the model into the main workspace.



4. Save the part as we will use it in a machining exercise.

#2: Hot Punch

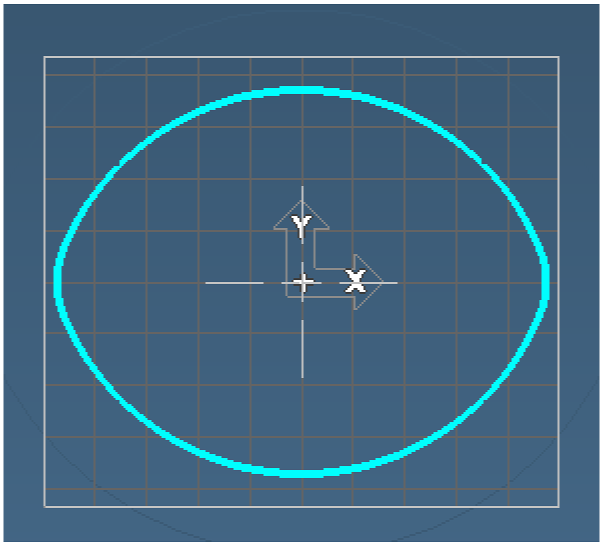
For the part print for this exercise, see "Hot Punch" on page 164.

Part Setup

This exercise is designed to teach you more about lofting, specifically the importance of selecting the appropriate alignment points to create the desired body.

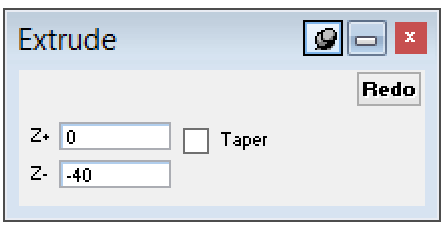
1. Create a metric 3 Axis Vertical Mill part with the following workspace stock size: +X = 38, -X = -38, +Y = 50, -Y = -50, +Z = 0, -Z = -40

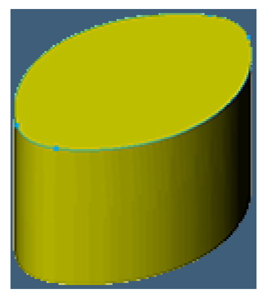
Extruded Profile



1. Create the extrude geometry in the XY plane.

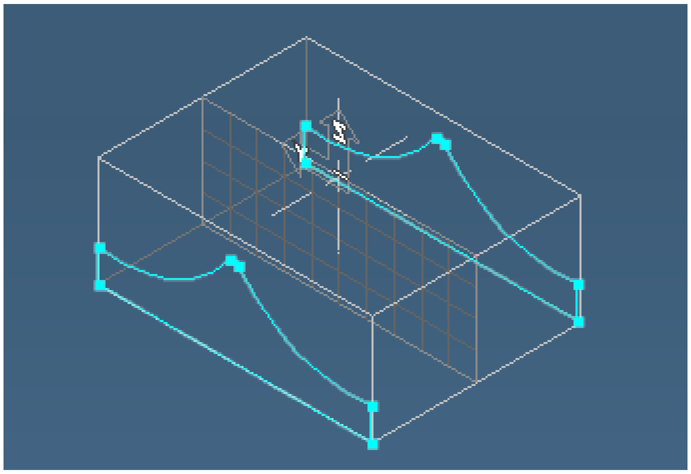
2. Extrude this geometry and place it in the Body Bag.





3 Profile Loft

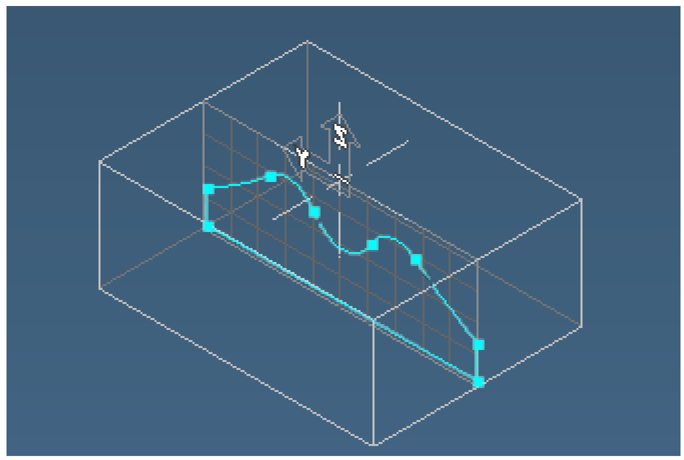
1. Create a YZ axis CS2 and Workgroup #2.



2. Create segments A and C.

   Create one segment at a depth of zer0 and then Modify > Duplicate Dup + Force Depth/... to make a copy at X= -38 and use Modify > Transform> Force Depth... to move the first segment to X= 38.

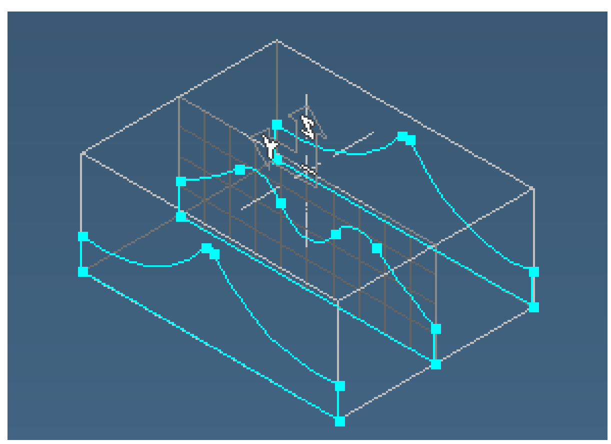
3. Create Workgroup #3 for segment B and create the geometry.



Moving Geometry to a Workgroup

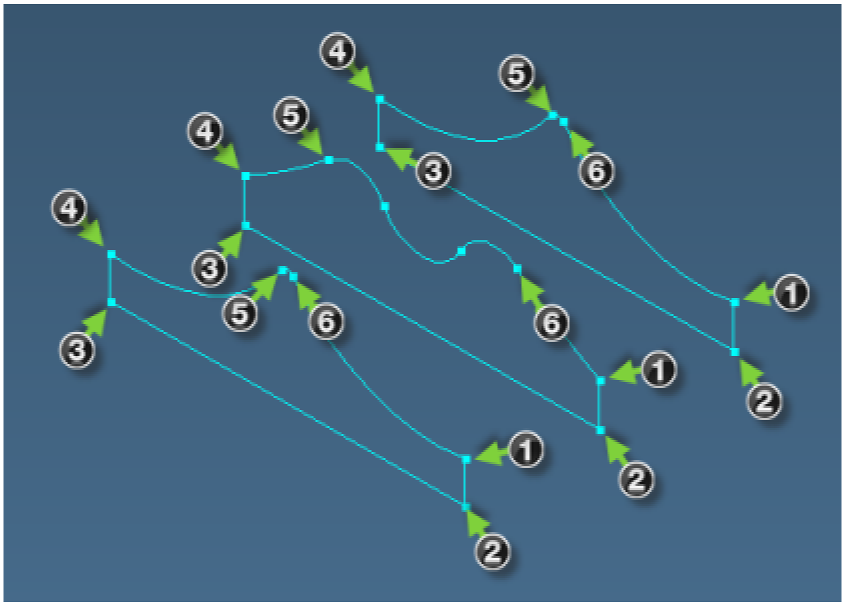
1. Double-click anywhere on the geometry and press (Ctrl+X) (Edit > Cut).

2. Switch to WG2 and click (Ctrl+V) (Edit > Paste).



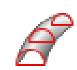
Now we will loft these three shapes into a solid. In order to create the desired solid, we will need to select the alignment points we want the system to use to blend the body. In order to achieve the desired body, we want to align the two outside connectors of segment B with the top connectors on segments A and C.

Selecting Alignment Points

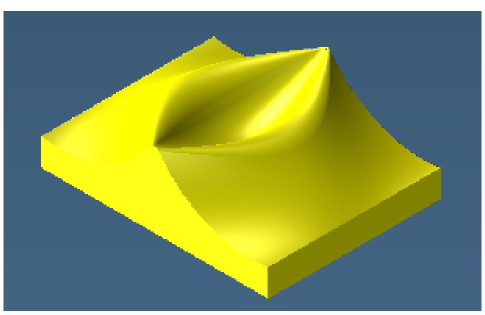


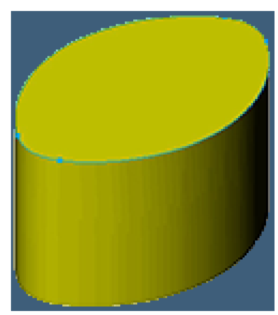
1. `Ctrl-click` the alignment points shown.

2. Loft the geometry.

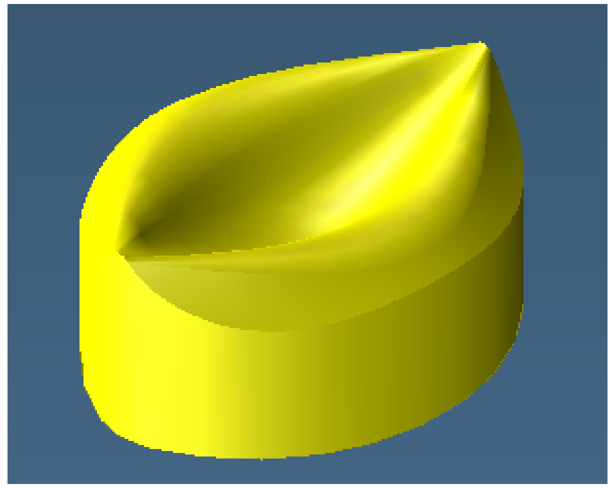


3. Select the loft and the extrusion.





4. Create an Intersection  from the two solids.



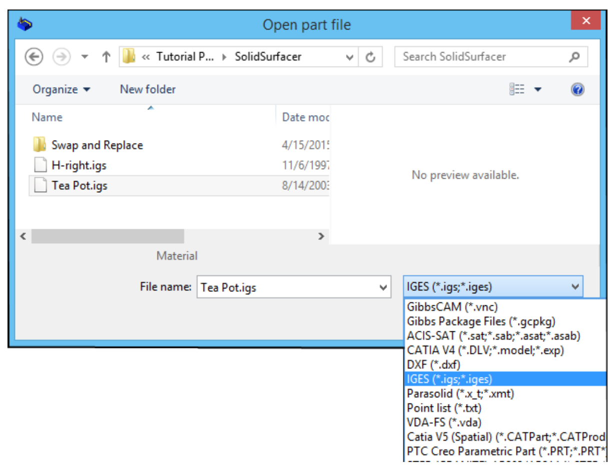
5. Save the part for the machining exercises.

#3: Teapot

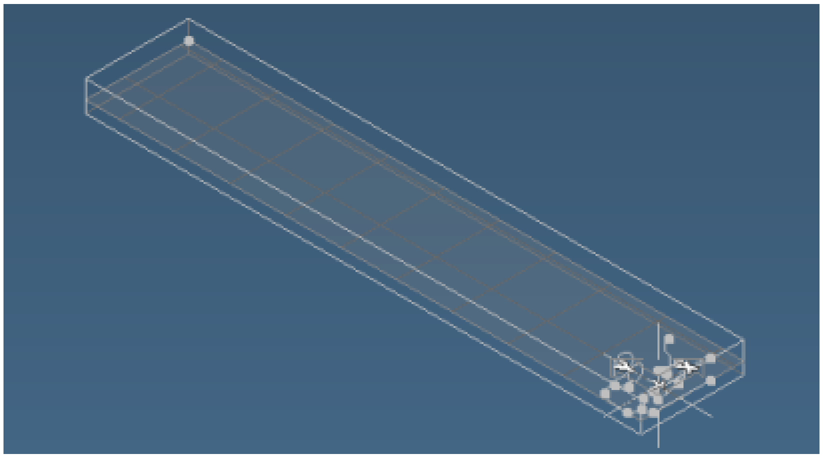
The 2D geometry for this part will be imported from an IGES file. The file can be found in either the Sample parts folder or the installation CD. This file may be found on the Sample Parts/Solids/Tutorial Parts - Required/Solid Surfacer folder in the application directory. (default is Program Files)

Opening an IGES File Directly

1. Click (Ctrl+O) (Open) and change the Files of type list to IGES (*.igs) files.



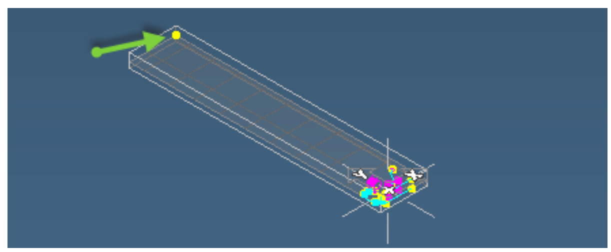
2. Open the Tea Pot.igs file.

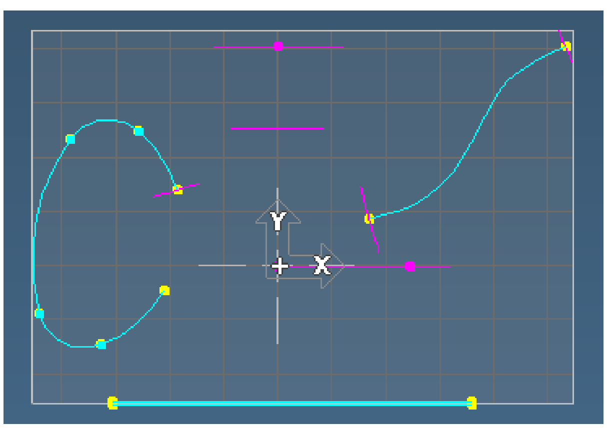


Auto-Size the Workspace

There is a stray point in the top corner.

1. Switch to WG3 and `Delete` the point.





2. Select Modify > Shrink Wrap (`Ctrl+'`) and then (`Ctrl+H`) to show the Home view.

This will adjust the workspace stock to fit the existing geometry.

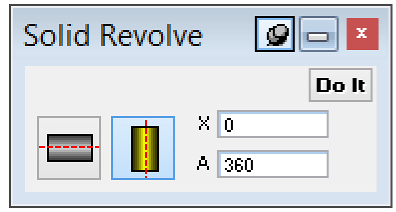
Creating the Teapot

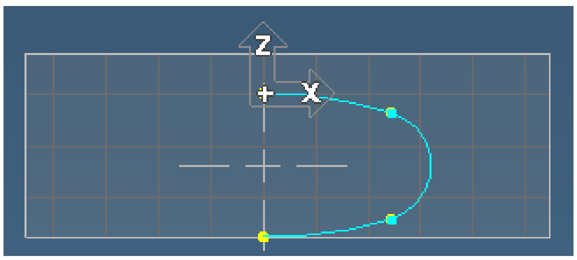
Revolving the Main Body

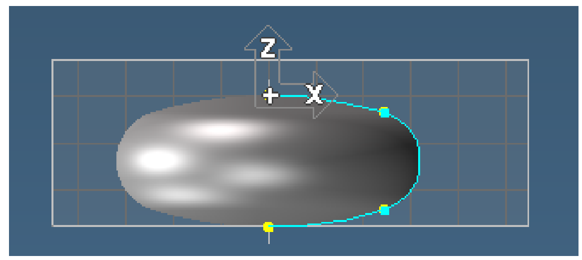
1. `Click` the CS marker at the origin (CS2) and switch to the home view (`Ctrl+H`).

2. Switch to WG2 and hide the other workgroups. 

3. Click anywhere on the geometry and revolve  it as shown below.

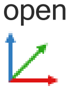


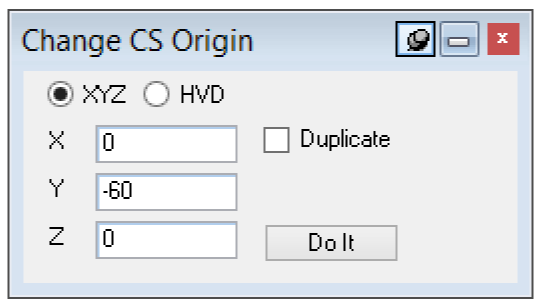




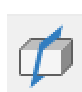
4. Right-click on the body to open the Show Properties dialog and rename it Body.

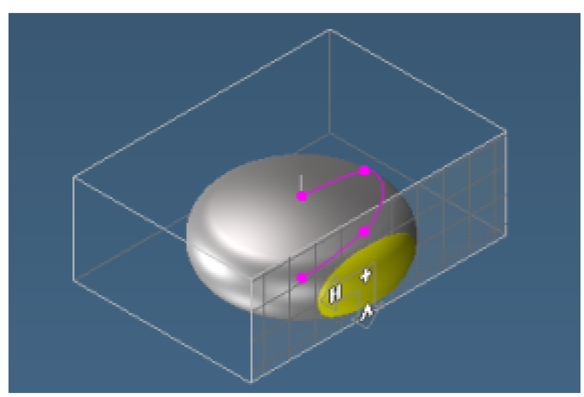
We will now create a slicing plane to make a base for the model.

5. Create a new CS based on the current XZ and open the Change CS Origin  dialog in the CS  palette.



This will move the CS to the base of the stock.

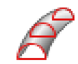
6. Click the Body then slice  and Delete the small slice. (Click OK to lose history and Continue).

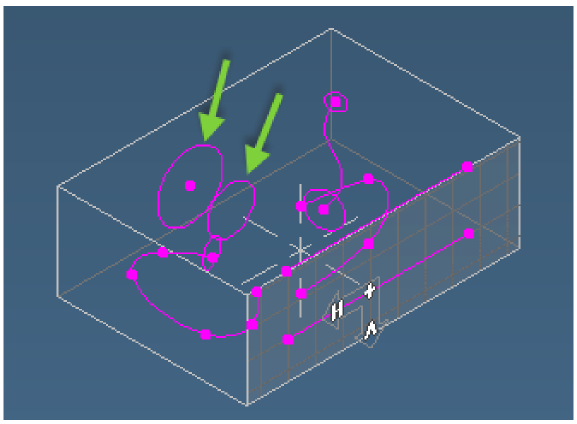


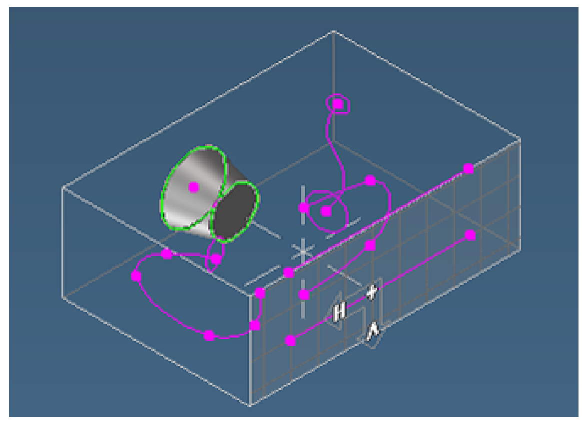
7. Place the remaining solid in the Body Bag and rename it Main Body by clicking the name as shown.

Each operation will change the name of the current state of a solid.

8. Switch to WG3 and loft the two circles shown.



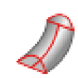


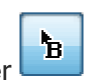


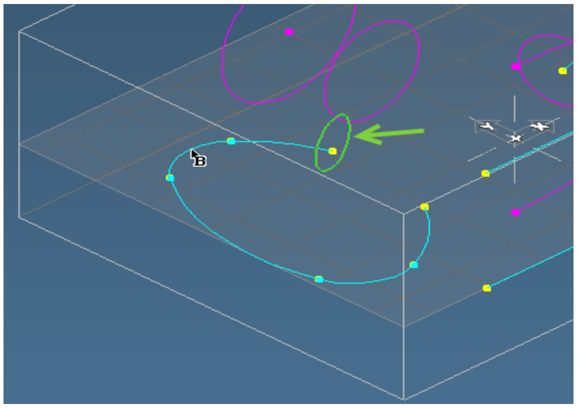
9. Name this body Top.

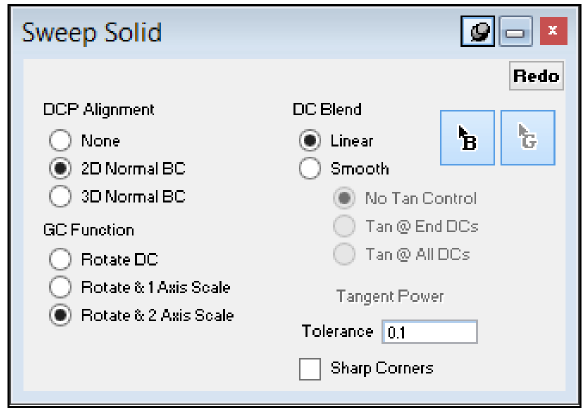
Sweeping the Handle

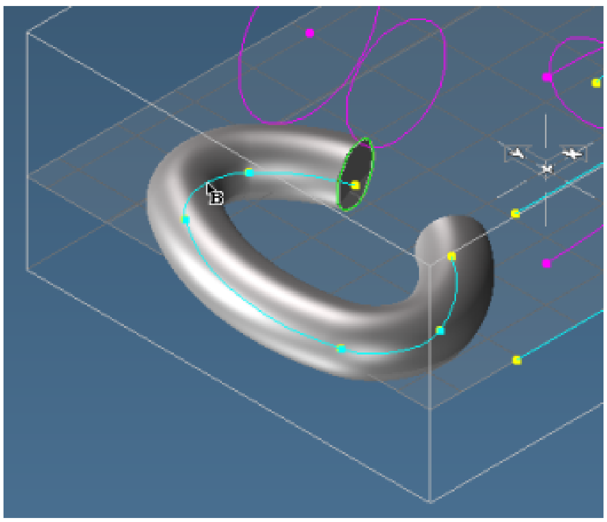
Switch to CS1 and open the Sweep Solid dialog.



1. Select the circle shown and place the base curve pointer  on the geometry. Click the Do It button.





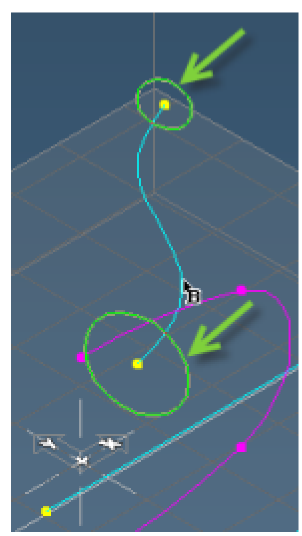


2. Name this body `Handle 1`.

Two Drive Curve Sweep

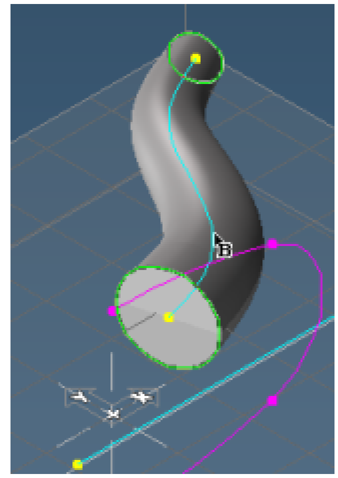
We will use the same settings for the next sweep.

1. Select the two circles and place the pointer  on the spline.

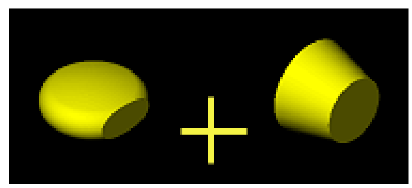


2. Apply the sweep. 

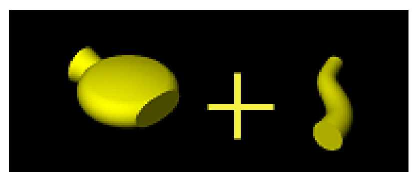
3. Name this body Spout.



Combining the Solids



1. In the body bag, select the Main Body and then the Top. Add  them together.

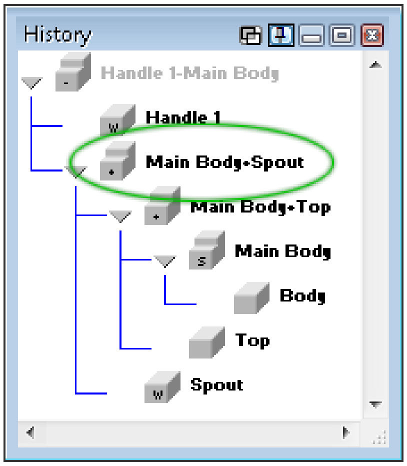


2. Click the Main Body+Top and add  the Spout.

3. Select the Handle 1 and subtract  Main Body+Spout.

4. Place the modified handle in the Body Bag and rename it Final Handle.

In order to get the Main body+Spout back we need to extract it from History.

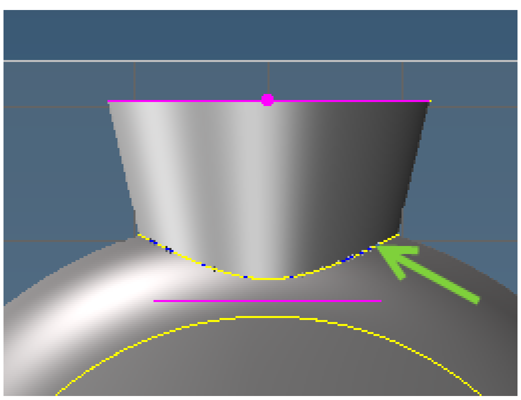


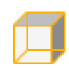
5. Right-click Final Handle and choose Show History.

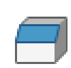
6. Double-click the Main Body+Top+Spout Union lump. 
   This will extract the body selected and place it in the workspace.
7. Change the Main Body+Top+Spout name to Main Body.

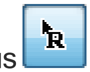
Creating a Variable Radius Blend

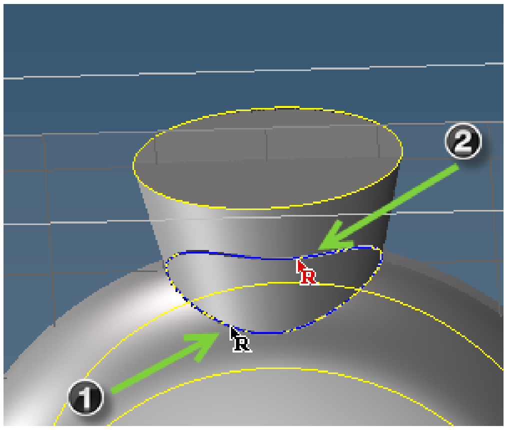
Our next step will be to round the edge where the top meets the main body. We will use the Variable Radius Rounding that allows different radii to be applied at different locations along an edge.



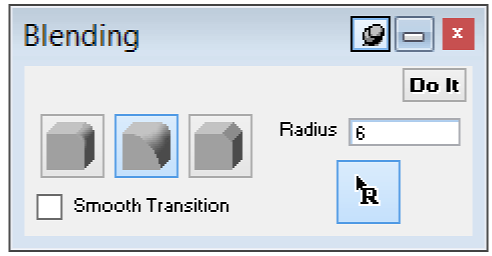
1. Turn on Edge Selection  and select the edge shown.

2. Open the Advanced Solid Modeling palette and open the Blending  dialog. 
3. Select the Variable Radius blend. 

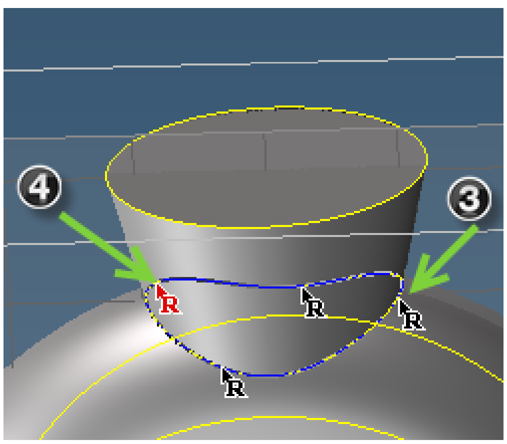
4. Adjust the view using the trackball as shown below. Place a radius  pointer as shown (1) and enter a 6mm Radius then place the second pointer.

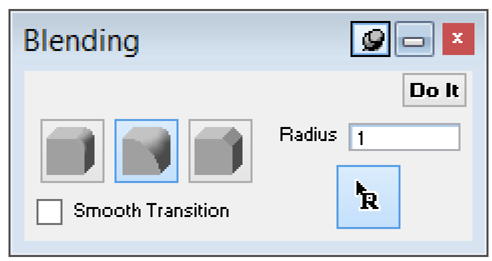


Radius pointers get their radius information when the pointer is red. To verify the radius setting for each position, select the pointer.

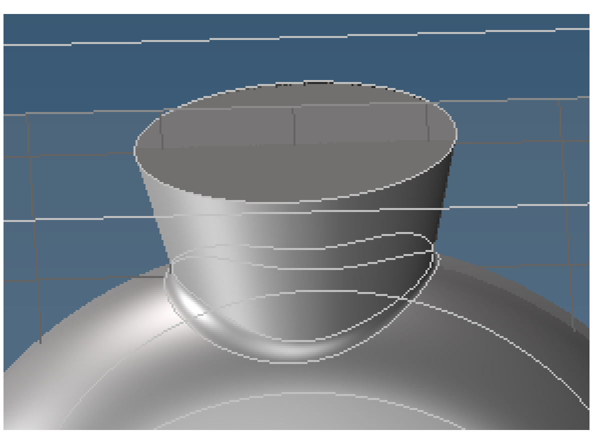


5. Place the third pointer and change the Radius to 1mm, then place the last pointer shown.



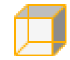


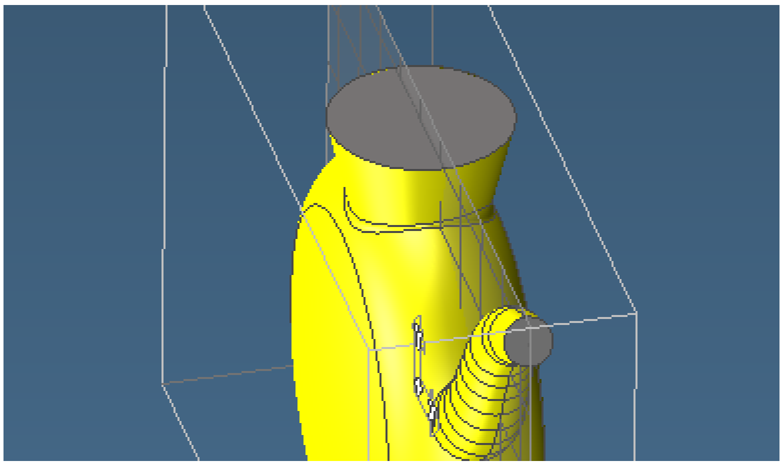
6. Click the Do It button to apply the variable radius blend.

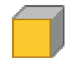


We will now hollow out the solid.

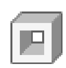
Creating the Shell

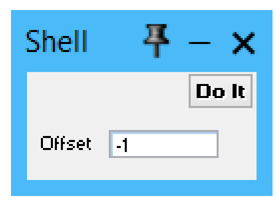
1. Turn off Edge Selection and select the solid. 



2. Turn on Face Selection  and (`Ctrl+Click`) the two openings as shown.

When using shelling, any deselected faces will create entry holes in the shelled solid. A shelling operation can create an offset outward or inward by using positive or negative values.

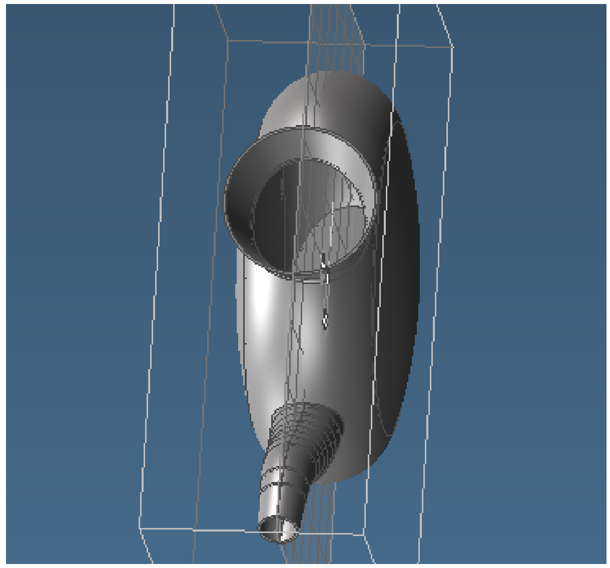
3. Open the Advanced Solid Modeling  dialog, select Shell  and enter the value shown.



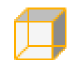
For this shelling operation we want to create the offset inward to get the cavity.

4. `Click` the Do It button.

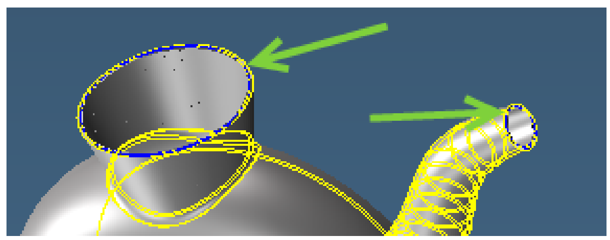
5. Place the shell into the Body bag and rename it Shell. Place Shell back into the workspace.



Unstitching the Cavity

1. Click the Edge Selection button. 

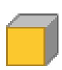
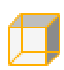
2. Double-click the inside edge of the top opening and (Ctrl+Double-click) the inside edge of the spout as shown.



3. Click the Unstitch Solid button. 

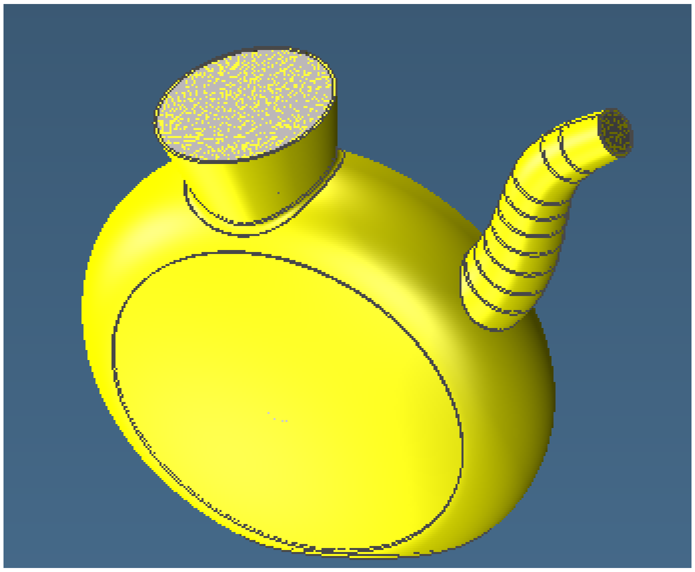
This will create two new solids, the cavity and a filled body. The shell is now only in the history. We don't need the outer solid that was created.

Measuring the Teapot Volume

1. Turn off Face Selection  and Edge Selection. 

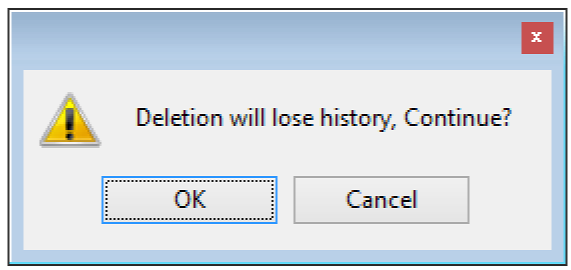
Deleting a Body

1. Delete the outer solid.



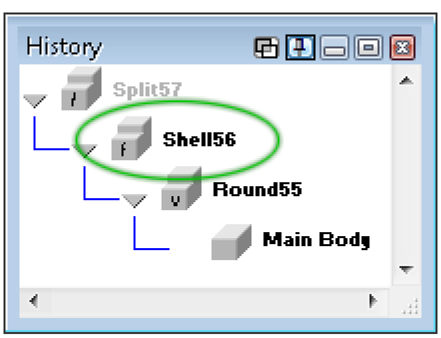
You have selected the correct body if the outer solid is yellow while the top and tip of the spout are dappled gray and yellow. This dappling effect signifies that two surfaces are shared and one is selected.

You will be asked if you wish to continue, that history will be lost.



2. Click OK or press Enter.
3. Double-click the core to place it in the Body Bag. Rename it Cavity.

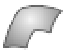
Extracting the Shell from the History

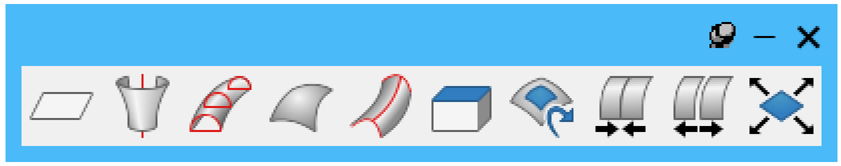


In order to get the shell back, it needs to be extracted from History.

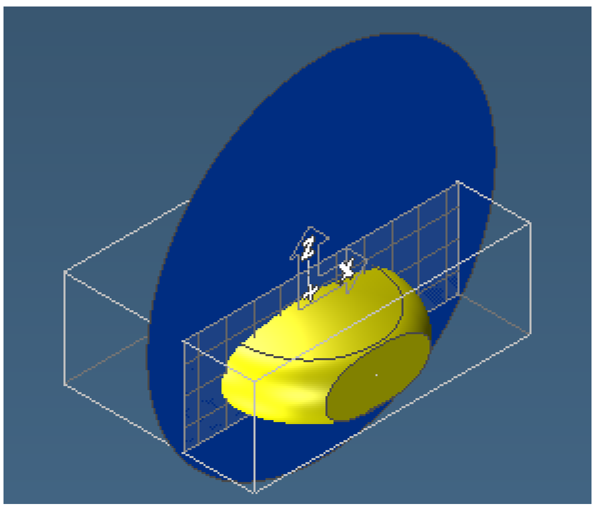
1. Show History for the cavity and extract the shell.
2. Double-click the Shell to place it in the Body Bag.
3. Double-click Cavity to move it from the Body Bag to the workspace.

Now we will slice the core using a surface created from the XZ plane. Once we have done that we will calculate the volume of the teapot.

4. Switch to CS2 and open the Surface Modeling  palette.

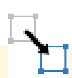


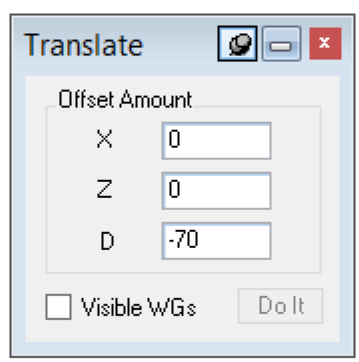
5. Click the Plane button. 



This will create a planar sheet based on the CS.

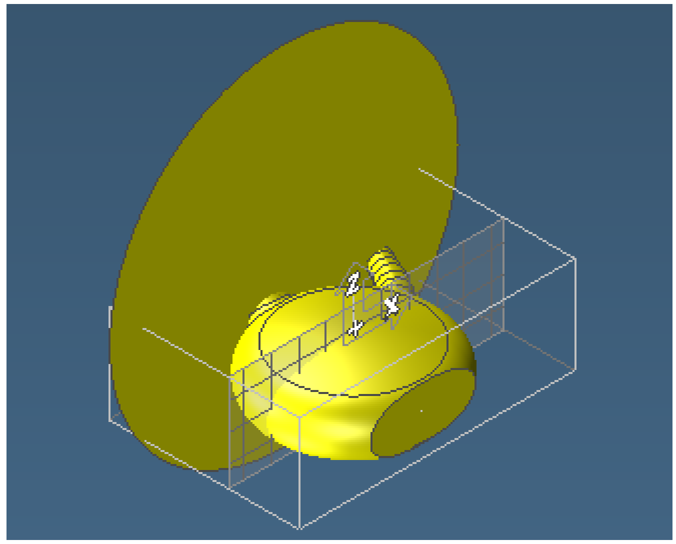
Slicing with a Plane

1. Select the plane and open the Modify>Transform>  Translate dialog.
2. Enter the values shown and click the Do It button.



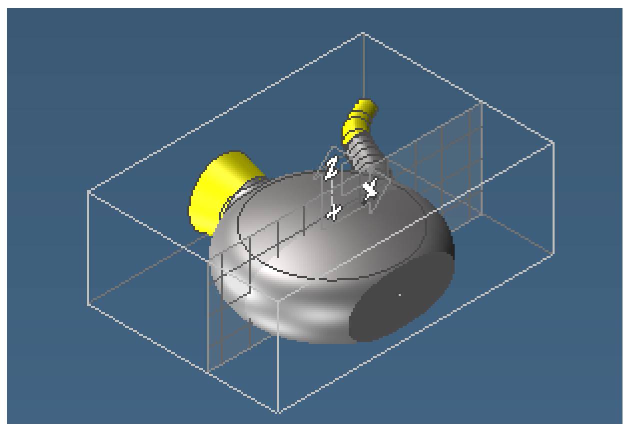
This will move the plane to approximately where the top meets the main body.

3. While holding down the `Ctrl` key, select the cavity and the plane.



A body can be sliced with either a sheet or the current coordinate system. If no sheet is selected, it will slice with the CS.

4. `Click` the Slice button,  press `Delete` and `Enter`.



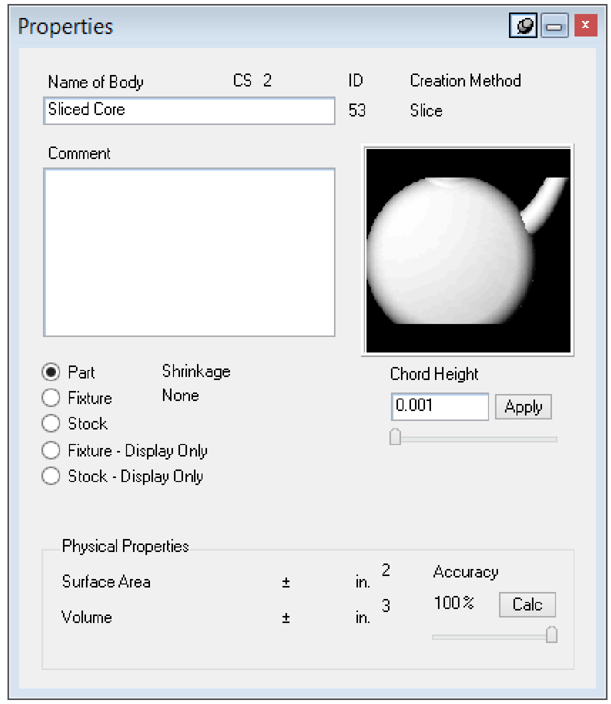
This will slice the solid into three pieces, `Delete` the small slices and accept the lost history warning.

5. Switch to the top view (`Ctrl+E`).

6. Open the Properties for the remaining slice.

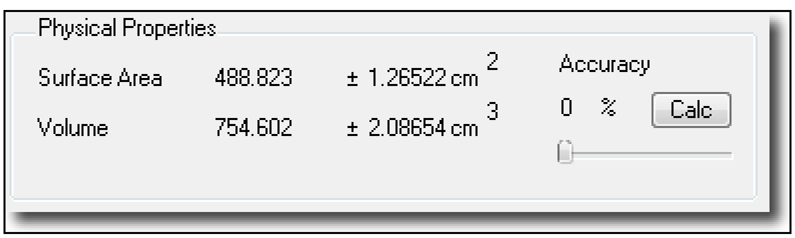
The Properties dialog can be used to name a part, identify it as a fixture, stock or part, modify the chord height of the graphic or calculate Physical Properties.

7. Rename and move the Accuracy scale to 100%.



Calculating the Volume

1. Click the Calculate button on the properties dialog.



Even low accuracy calculations can be quite close.

2. Save this file as Teapot.vnc

#4: Plumbing

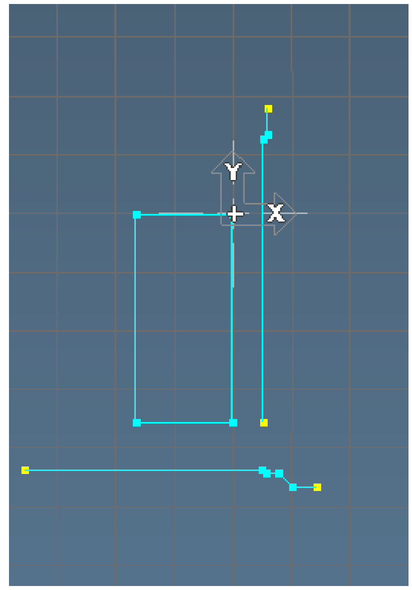
For the part print for this exercise, see “Plumbing” on page 165.

Creating the Plumbing

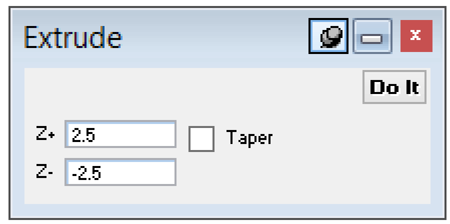
Part Setup

1. Create a Metric 3 Axis Vertical Mill part with the following workspace stock size:
   +X = 60, -X = -80, +Y = 50, -Y = -90, +Z = 0, -Z = -50

2. Create the geometry shown.

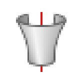


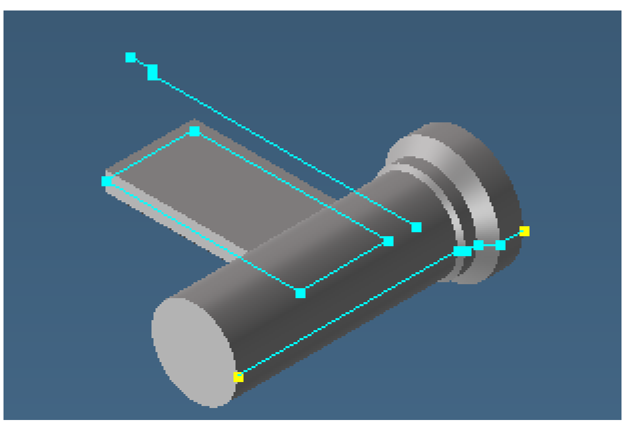
3. Extrude  the rectangle.

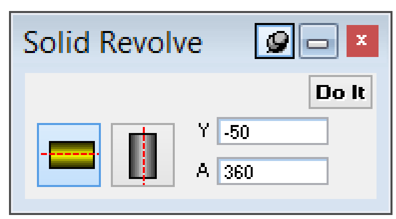


Next, we will revolve the horizontal geometry about Y -50.

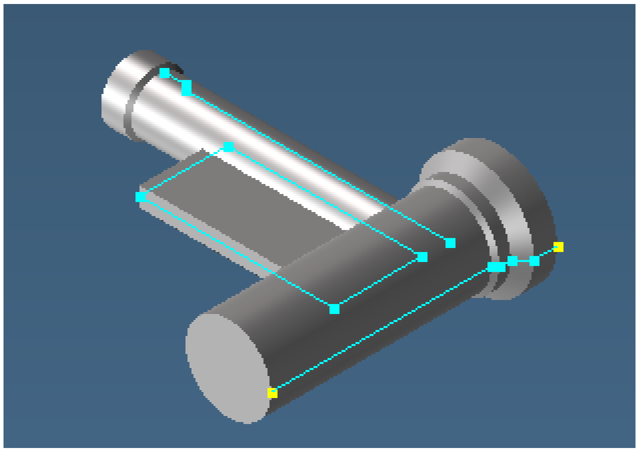
4. Open the Revolve dialog and revolve the horizontal geometry.

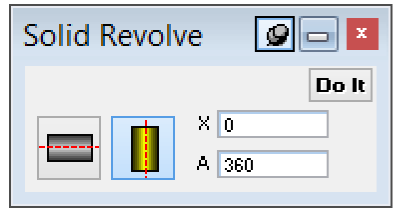




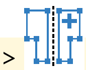


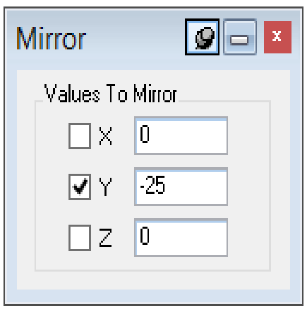
5. Revolve the vertical geometry.

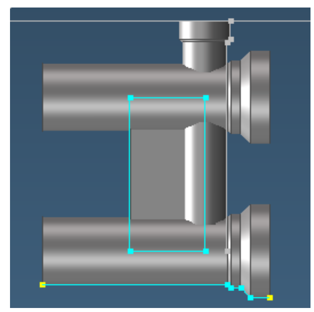




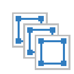
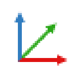
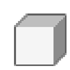
Mirroring

1. Select the first revolve and the extrusion and choose Modify >Duplicate>  Duplicate + Mirror and enter the Values To Mirror.

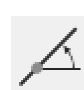
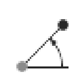


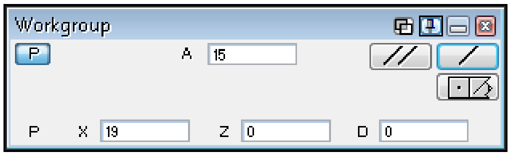


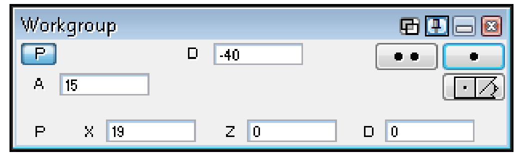
We will now create an angled shape.

2. Create a WG2 and CS2 .  
3. Hide the solids. 
4. Switch to the front (`Ctrl+F`).
5. Create a new XZ CS. 

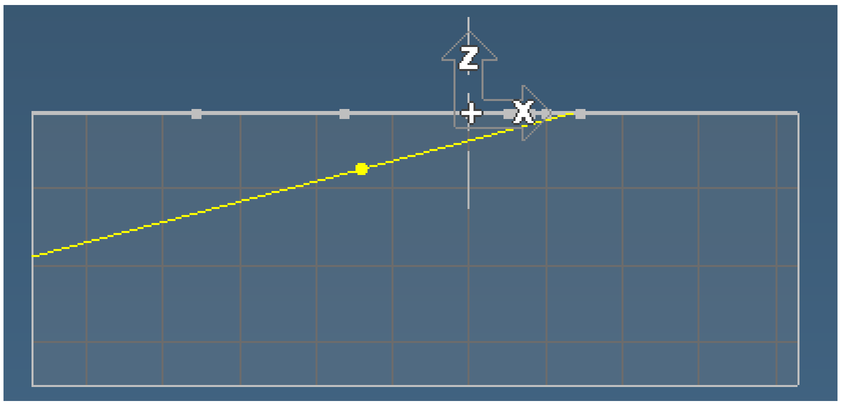
Aligning a CS

1. Create an angled line and point as defined in the print.  

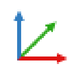


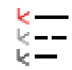


The example shown is based on creating an angled line first, then creating a point on that angle the specified distance. It could also be done the opposite way by creating the point and then creating an angled line through that point. In the latter case we wouldn't have needed to define the point for each feature as was needed for the example.

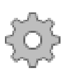


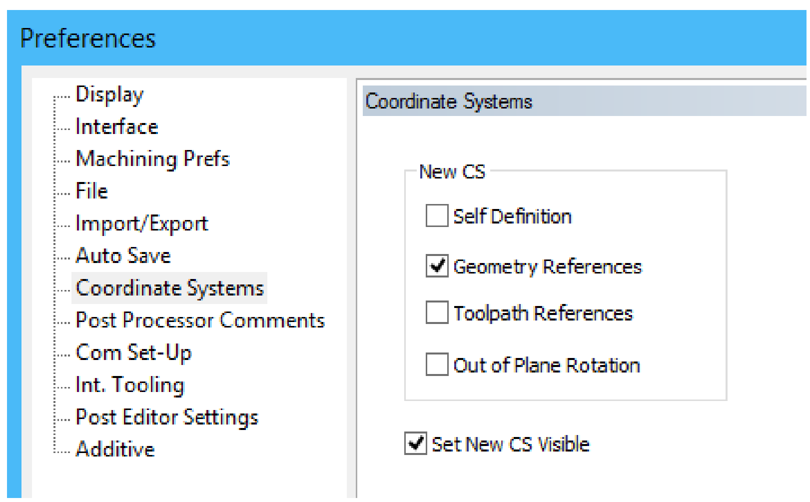
2. Open the CS palette and CS list.





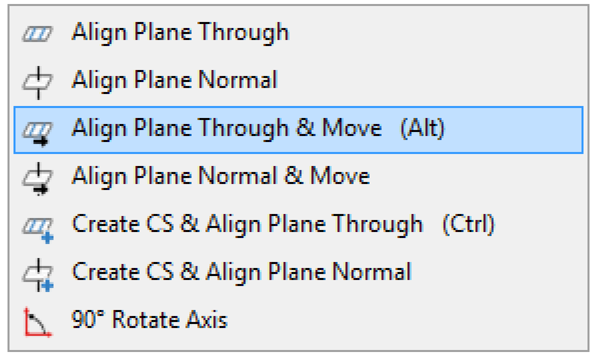
3. Select the point and line.

4.  Set the CS preferences.

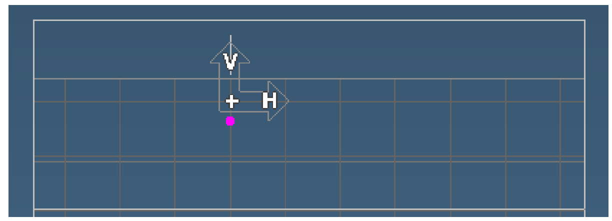


5. `Right-click` the align CS button.

6. Choose Align Plane Through & Move (Alt).



7. Deselect the geometry and switch to the home view (`Ctrl+H`).

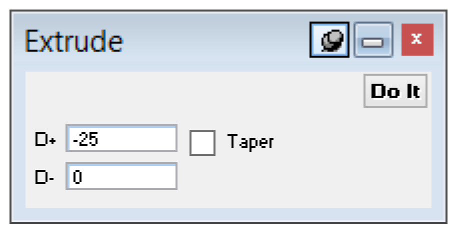


If the home view does not look like the image press the Toggle Depth button and switch to the home view (`Ctrl+H`) again.

8. Create a 14mm diameter circle around the point.

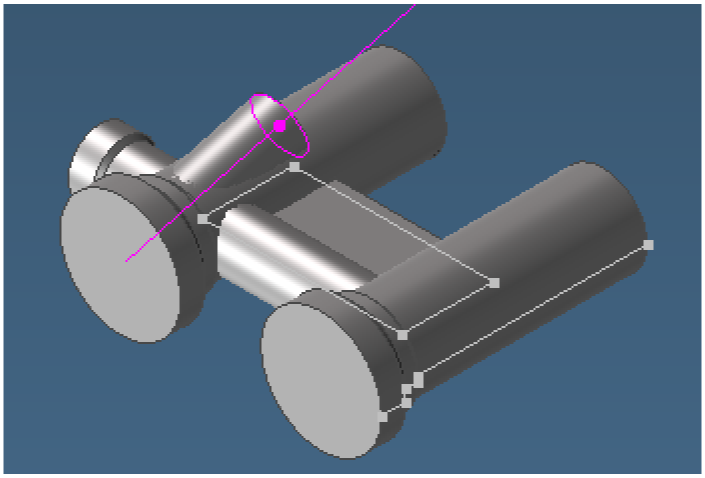
Extruding the Slants

1. Extrude the circle.



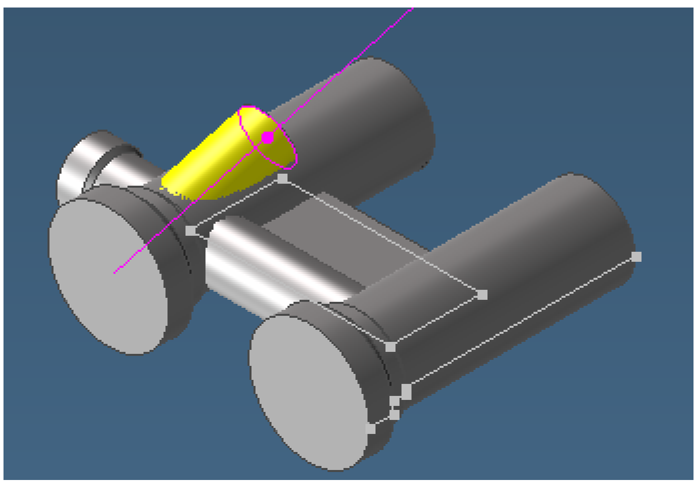
Notice the solids become visible.

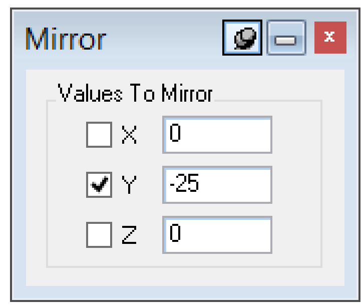
2. Switch to CS 1.



3. Switch to the opposite isometric view (`Ctrl+Alt+I`).

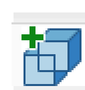
4. `Click` the extrusion and Duplicate + Mirror the extrusion using the same values from earlier.

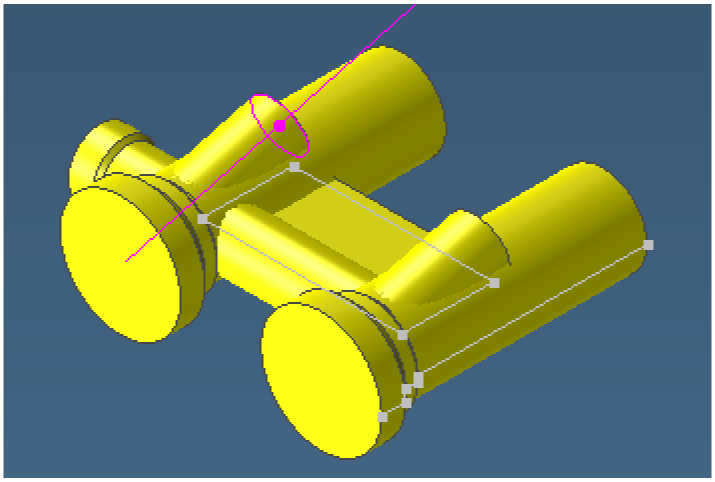




We will now add the bodies to create the final part.

5. Add all the solids together.





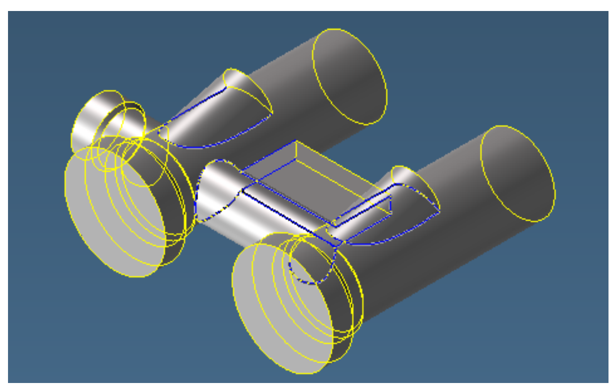
Our next step will be to add a radius to selected edges. We will be selecting edges that when machined, would not otherwise be rounded by a ball end mill, effectively giving that edge a radius.

6. Hide geometry, stock and origin, and the CS grid.

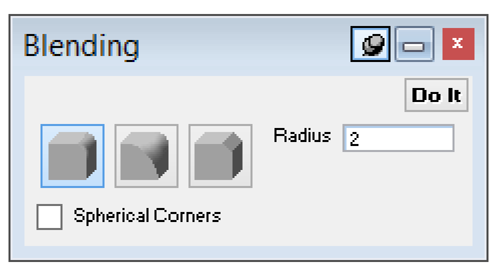
7. Turn on edge selection.

Adding Intersection Blends

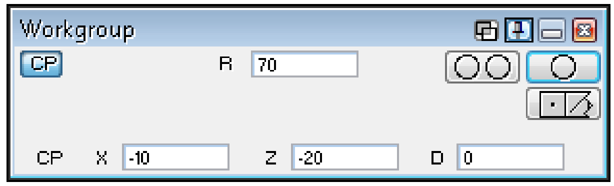
1. While holding down the `Ctrl` key select the intersecting edges shown.



2. Blend the selected edges.

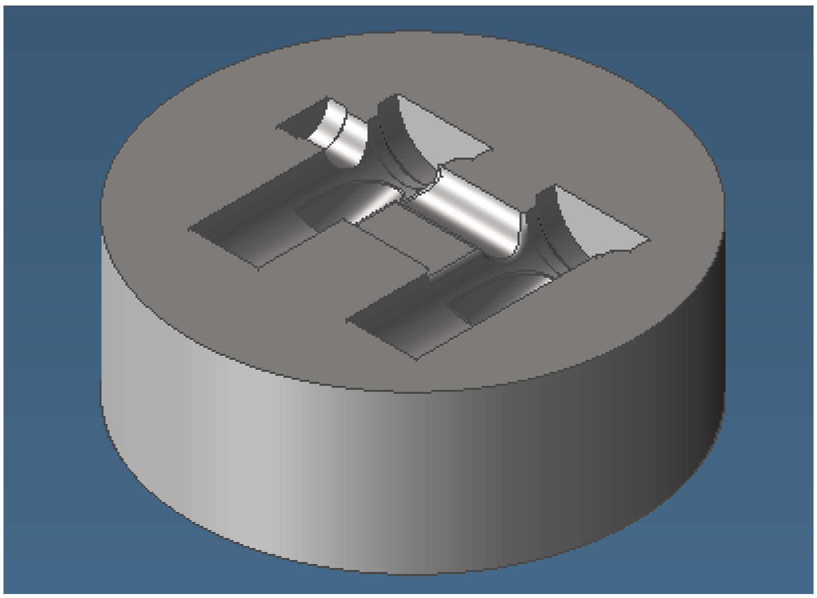


3. Create a circle and extrude it to the base of the stock (-Z = `-50`).



4. Switch to the isometric view (`Ctrl+I`) and turn off edge selection.

5. Subtract the model from the extrusion.

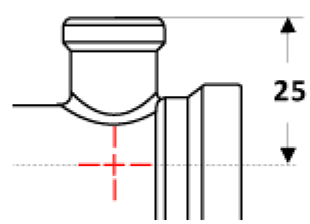


Correcting A Design Flaw

The revolved cross pipe is too long. Instead of manually rebuilding the entire model we will use Recreate and Rebuild to change entire Mold Cavity.

1. Hide solids and show the geometry, and the CS grid.

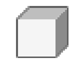
2. Create the revision geometry or edit the original.



An easy way to edit the original geometry would be to select the top part of the geometry, Modify > Translate and reconnect it.

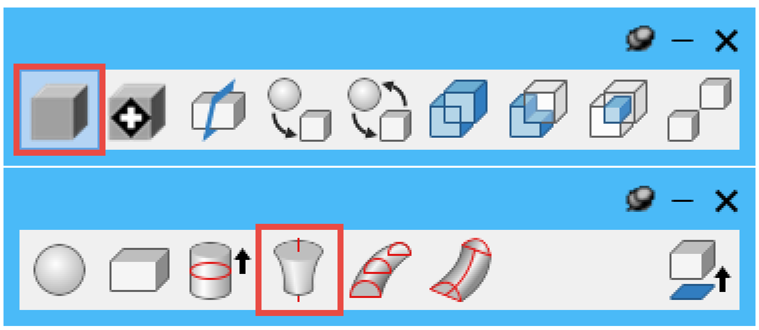
Recreating the Cross Pipe

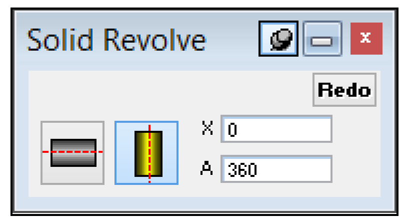
1. Show solids and open the History list.



2. Right-click the revolve and choose  Recreate.

   The solid will be sent to the Body Bag and will be colored red. The action to create the solid is accessed and opened. The path to a dialog is highlighted in red on each palette. The Do It button changes to a Redo button. At this point the revolve data could be edited but we don't require changes.





3. Select the revision geometry and press the Redo button.

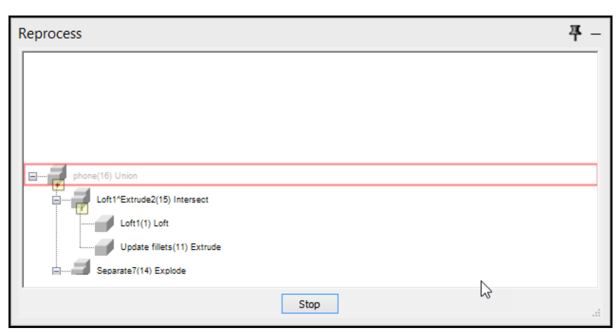
   If you created new geometry in a new workgroup you must switch to that Workgroup.

Rebuilding the Model

> ⓘ If you have selected the wrong solid or do not wish to perform a recreate you may get out of Recreate mode by clicking on the red solid or by right-clicking a body and choosing Exit Recreate.

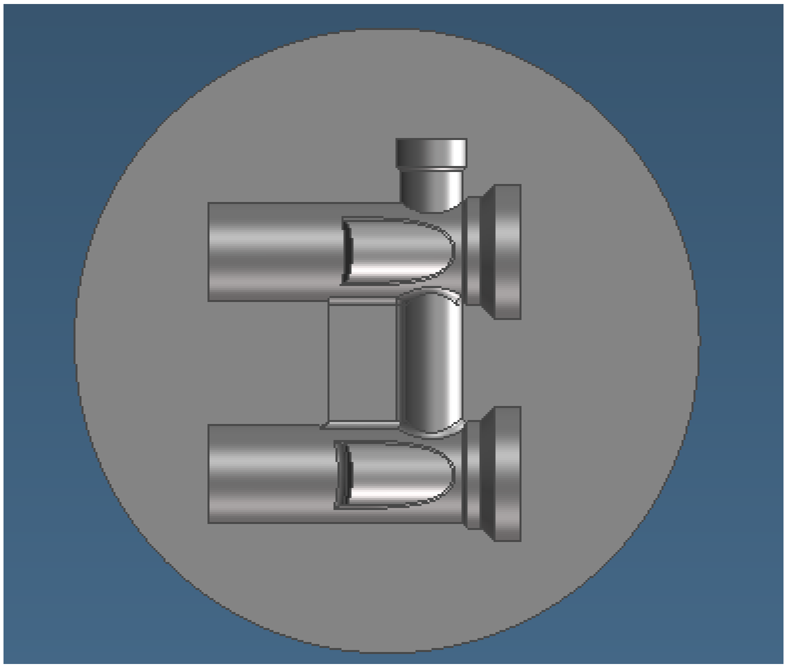
1. Right-click the main model and

   choose  Rebuild.



The rebuild is complete. All solids created that contain the recreated body are affected.

We can now see that the cross pipe is the correct size.



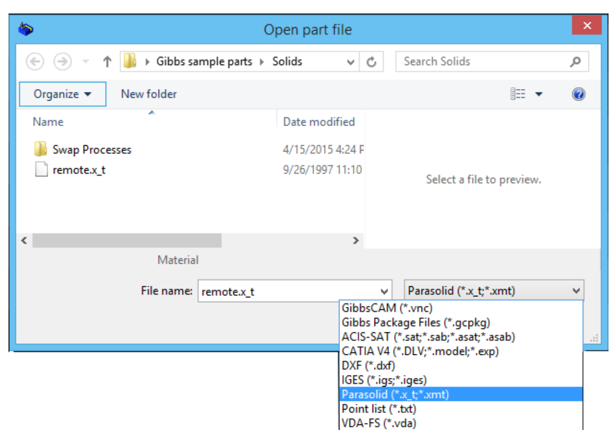
2. Save this part to be used later in the machining exercises.

#5: Remote Control

In this exercise we will import a model. We will be importing a Parasolids file included on theGibbsCAM CD. The file extension (.ext) of a Parasolid file can be different depending on the design software. This file's name is remote.x_t. To ensure we can find and open the file we will need to check the system preferences.

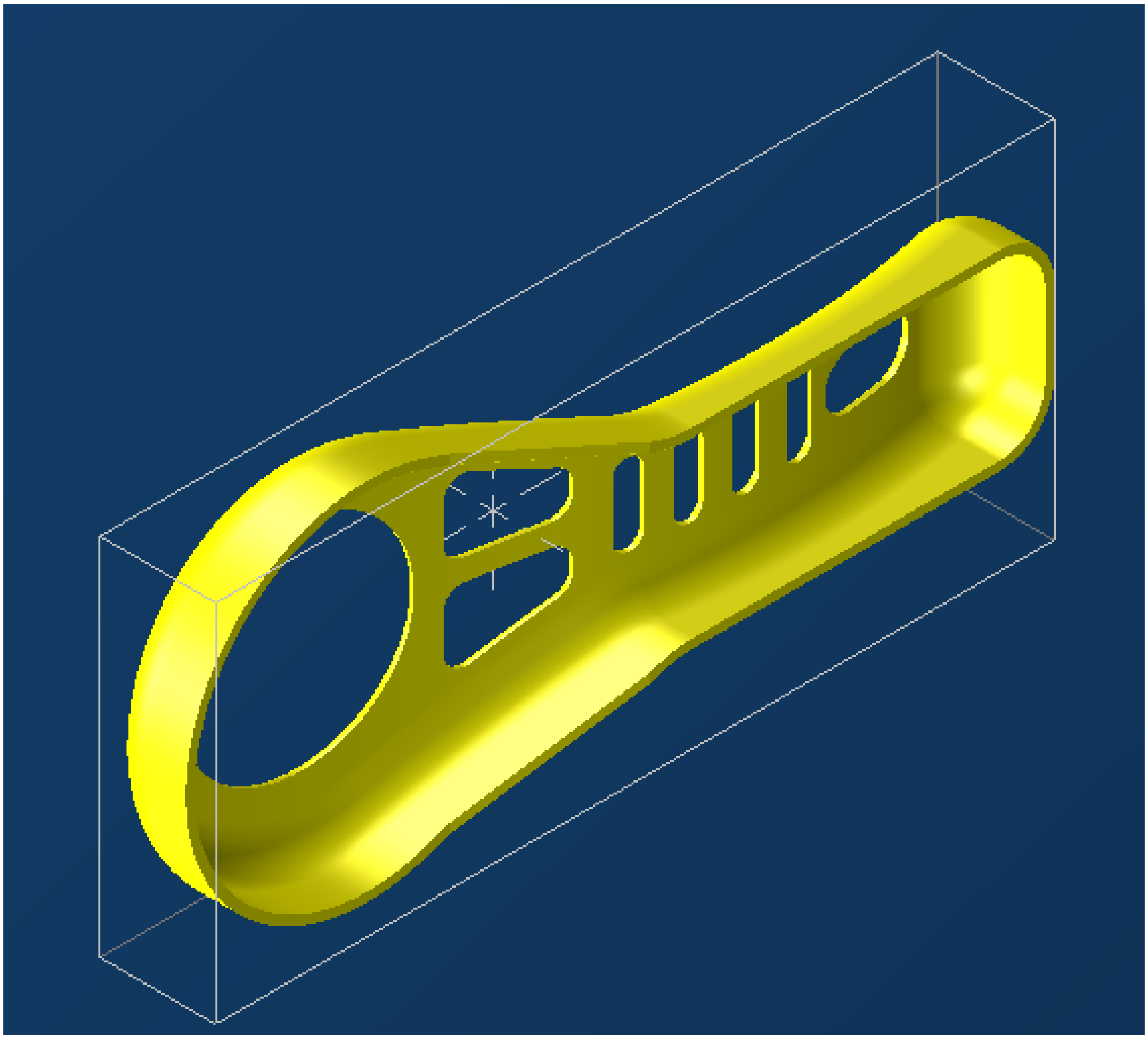
Creating the Remote Mold

1. Press (`Ctrl+O`) (Open) and change the Files of type list to Parasolid (*.x_t; *.xmt) files.



2. Open 

the remote.x_t file.

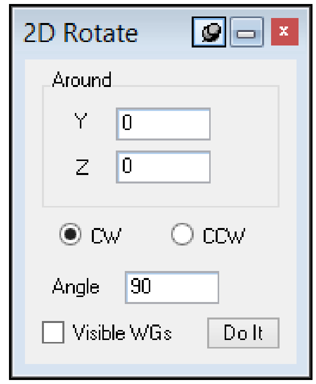


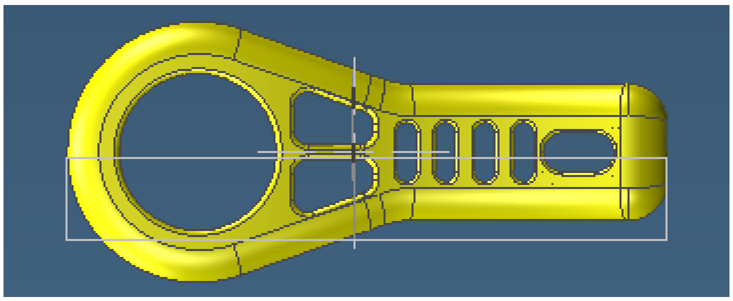
Re-positioning the Model

We need to rotate the model to the XY CS.

1. Create a YZ CS.

2. Select and Modify > 2D Rotate the model as shown.





3. Select Modify>Stock> Shrink Wrap (`Ctrl+‘`).

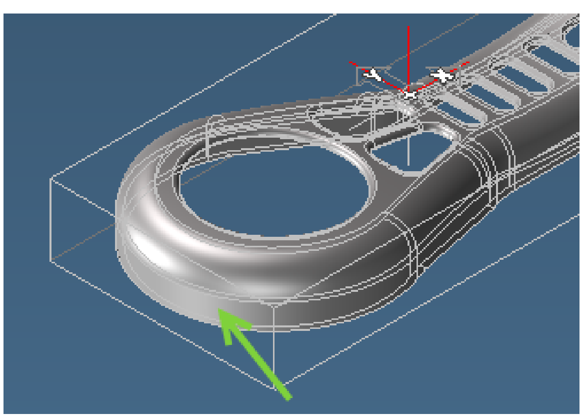
Our next step will be to apply a draft angle to the walls. We will need to select the faces this will be applied to, as well as the reference edge to act as a pivot point.

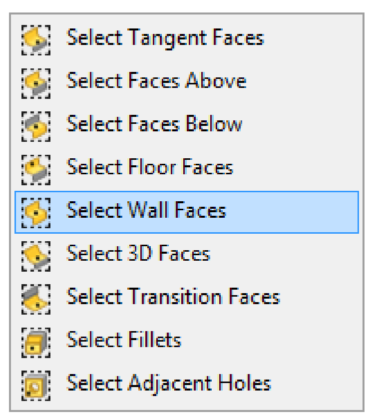
Applying the Draft Angle

1. Turn on face selection and edge selection and switch to CS 1 XY, isometric view (`Ctrl+I`).

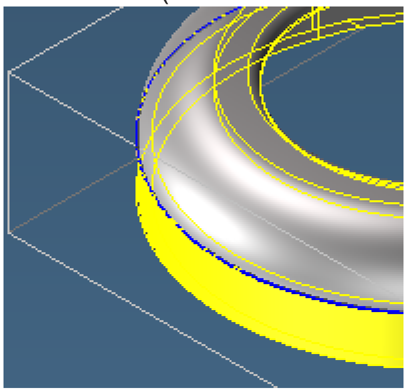
2. Open the Advanced Solid Modeling palette.

3. `Right-click` the face shown and choose Select > Select Wall Faces.

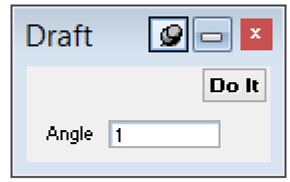




4. Zoom in and (Ctrl+Double-click) the outer edge above the selected face.



5. Draft the wall with the value shown.

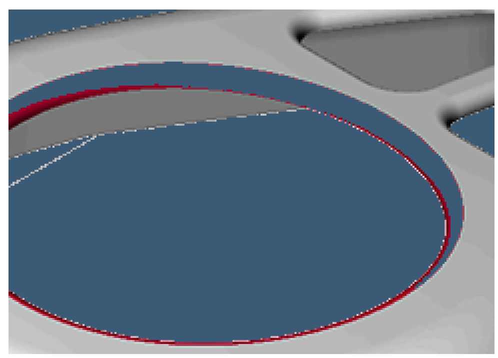


A close inspection of the body will show that the sidewalls are no longer normal to the XY plane.

Creating Shutoffs

1. Turn off edge selection and pan (Ctrl+←) left until the large opening is in view.

2. Right-click the blend of the largest circle and choose Select> Select Wall Faces.



3. Click the Unstitch Solid button.

This will create two new solids, the model with the large hole filled in and the hole.

4. Pan (Ctrl+drag) to the next set of holes.

We will now make a compound selection to unstitch the next two holes.

5. Right-click the blend of one of the next two holes and choose Select> Select Fillets.

6. (`Ctrl+Right-click`) the selected fillets and choose Select> Select Wall Faces.

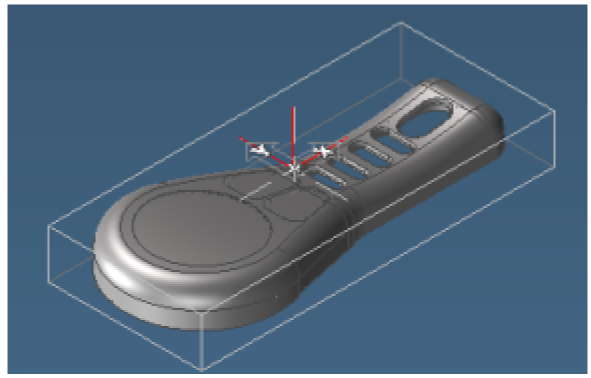
   This will add the adjacent walls to the current selection.

7. (`Ctrl+Right-click`) the blend of the other hole and choose Select> Select Fillets.

8. (`Ctrl+Right-click`) the selection and choose Select> Select Wall Faces.

   We now have all the fillets and walls of the two larger holes selected.

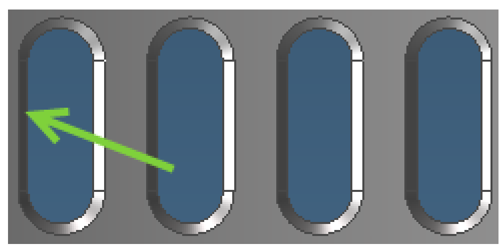
9. `Click` the Unstitch Solid button .



For the next four holes we will use a similar technique but use select tangent faces rather than fillets.

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

11. `Right-click` the chamfer face of the first hole and choose Select> Select Tangent Faces.

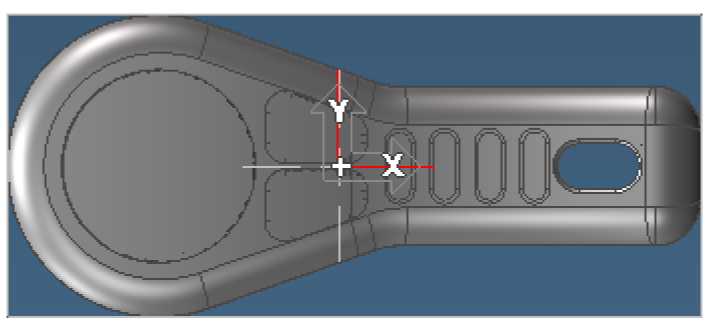


12. (`Ctrl+Right-click`) the chamfer and choose Select> Select Wall Faces.

13. Continue to add (`Ctrl+click`) the tangent faces and wall faces for the other three holes.

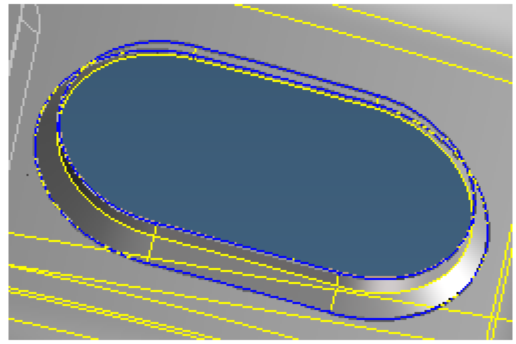
    Once all the hole walls and chamfers are selected we can now unstitch them.

14. `Click` the Unstitch Solid button .



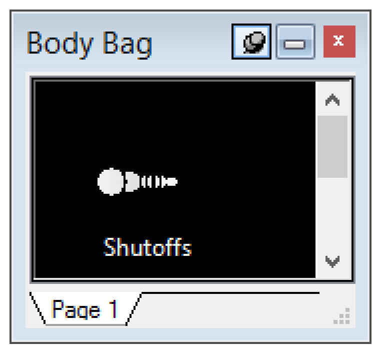
For the last hole we will select the edge loops to unstitch.

15. Turn off face selection and turn on edge selection.
16. Zoom into the last hole and rotate (`Shift+` +) the view slightly.
17. Select the outer and inner edges of the remaining hole. (You may need to rotate the part to see the other side.)



18. `Click` the Unstitch Solid button.
19. Place the main body in the Body Bag.

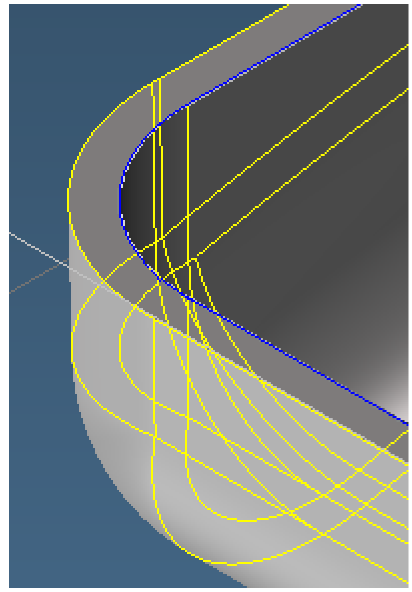
Combining the Shutoffs



1. Add the holes together and place them in the Body Bag and rename this body `Shutoffs`.
2. Un-Bag the main model.
3. Switch to the opposite isometric view (`Ctrl+Alt+I`).

Creating the Core

1. Zoom in and `Double-click` the inside edge.



2. `Click` the Unstitch Solid button.

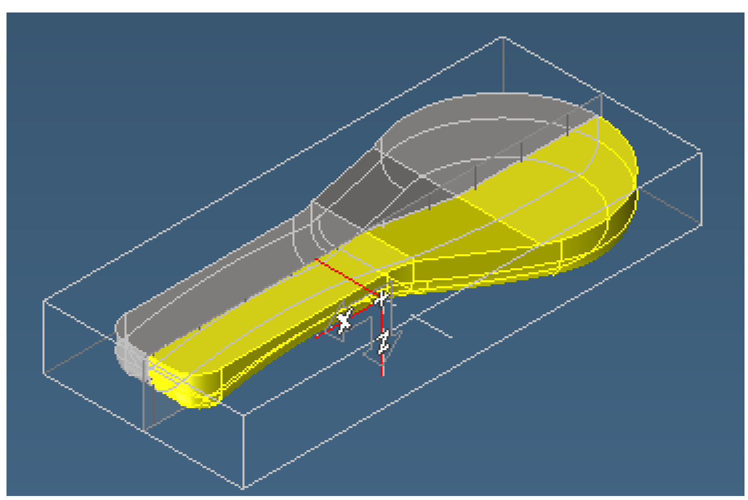
3. Place the outer solid in the Body Bag and name it `Cavity`.
4. Show properties of the inner solid and change the name to `Core`.

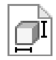
   In order to create the base for our mold with a proper parting line, we will extract geometry from a slice of the Core.

5. Create an XZ CS.

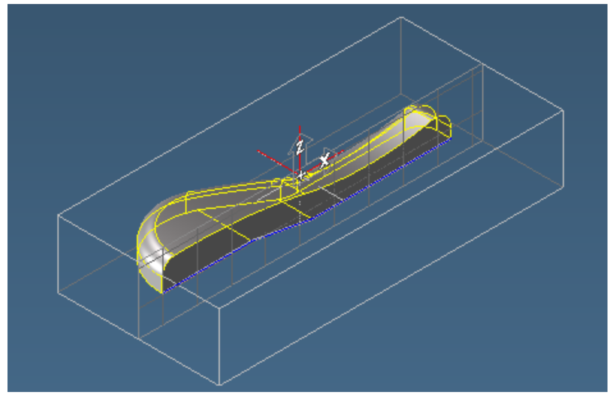
Extracting a Parting Line

1. Select the Core, then Slice  with the XZ CS and `Delete` the selected front half. (`Click`OK to accept losing history and continue.)



2. Open the Document control  dialog and increase the workspace stock size to: +X = `90`, -X = `-80`, +Y = `40`, -Y = `-40`, +Z = `3`, -Z = `-30`.

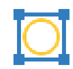
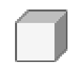
3. Switch to the unzoomed (`Ctrl+U`) isometric view (`Ctrl+I`).



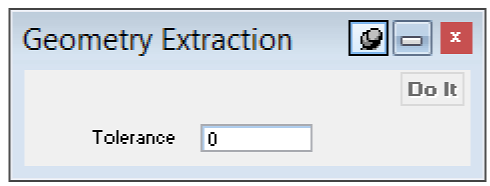
4. `Double-click` the bottom edge of the slice.

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

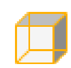
6. Open the Geometry From Solids palette within the Geometry palette.

7. Extract the geometry  from the edge.



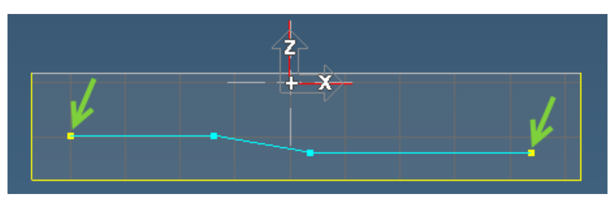
8. Turn off edge selection.



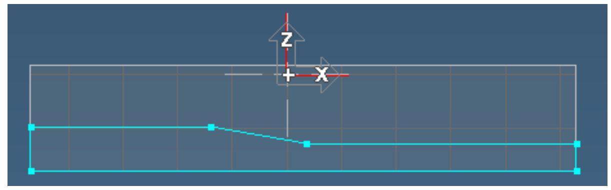
9. Place the sliced Core in the Body Bag.

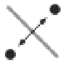
Creating the Base with the Parting Line

1. In the XZ plane create lines parallel to the axis at Z= -30, X= -80 and X= 90.

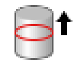


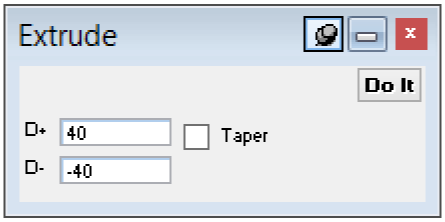
2. `Delete` the two terminator points.

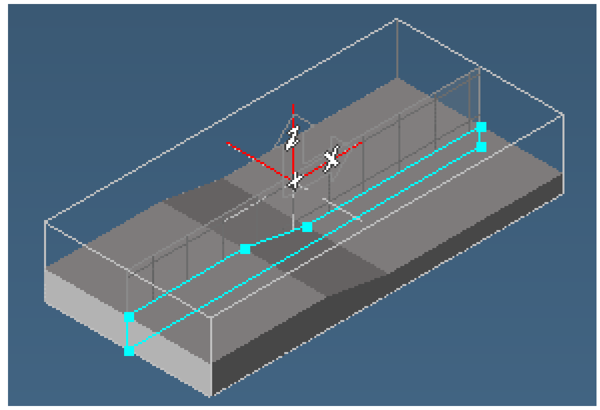


3. Connect  the lines.

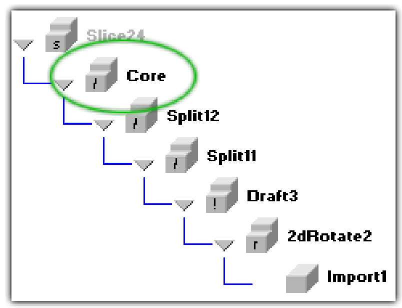
4. Extrude the geometry and name it `Base`.



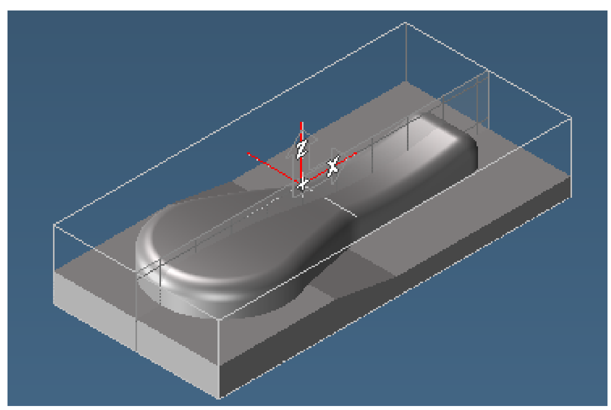


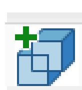


5. Switch to the isometric view (`Ctrl+I`).



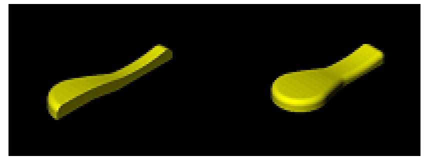
6. Open the history of the sliced Core in the Body Bag and extract the Core body.



7. Non-destructively add (`Alt+` ) the two solids and rename the union `Final Core`.

We will use the extrusion again to make the top part of the mold.

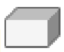
8. `Delete` the Core and the sliced Core in the Body Bag.

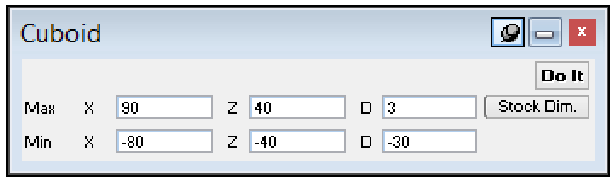


We will no longer need these solids.

9. Switch to the XY CS.

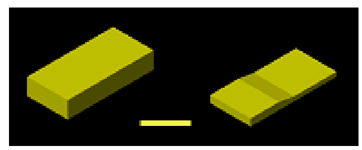
Creating the Cavity

1. Create a Cuboid  using the stock dimensions. Stock Dim.

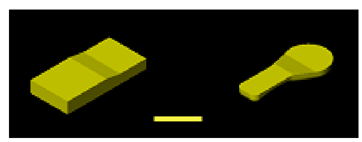


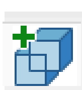
Boolean Operations

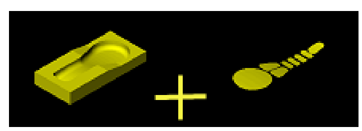
1. Subtract  the Base from the Cube and name the result `Cap`.



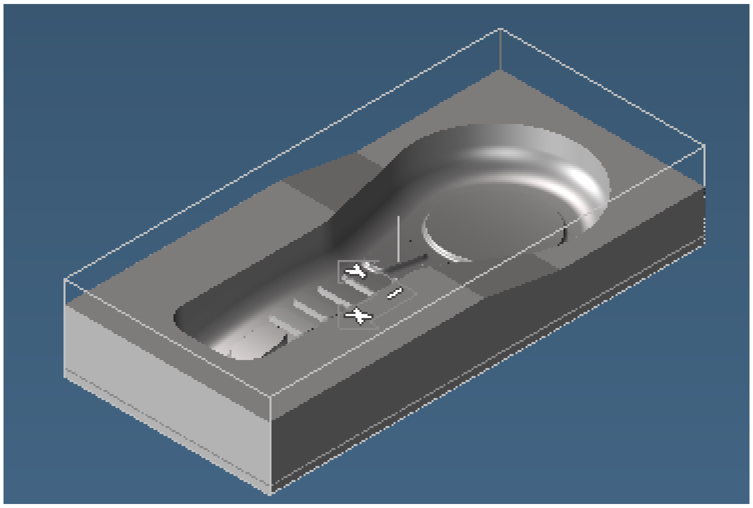
2. Switch to the opposite isometric view (`Ctrl+Alt+I`) and subtract  the Cavity from the Cap and name it `Prelim Cavity`.



3. Add  the Shutoffs to the Prelim Cavity and rename it `Final Cavity`.



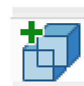
4. Save the part here, to be used in the machining section of this manual.



This is the end product of the exercise, but lets verify the model. We will combine the two halves of the mold and subtract a cube to see if our mold will create a model.

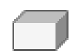
5. Switch to the isometric view (`Ctrl+I`).

6. Perform a non-destructive addition (`Alt+`) of the Final Core and Final Cavity.

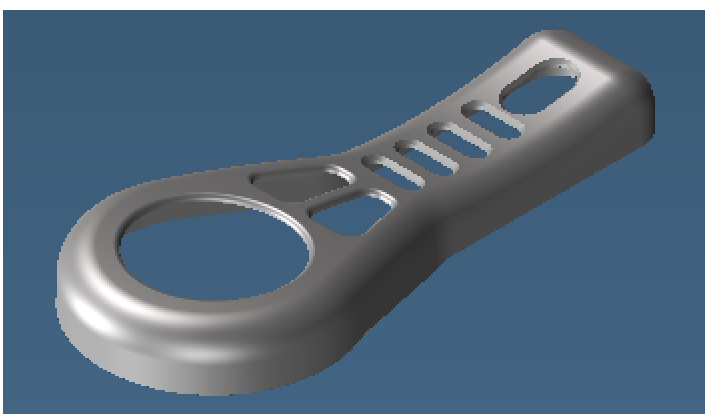


Remember, non-destructive Booleans are accomplished by holding down the `Alt` key when the Boolean operation is performed.

7. Create another Cuboid using the workspace stock size.



8. Subtract the union from the new cube.



MACHINING EXERCISES

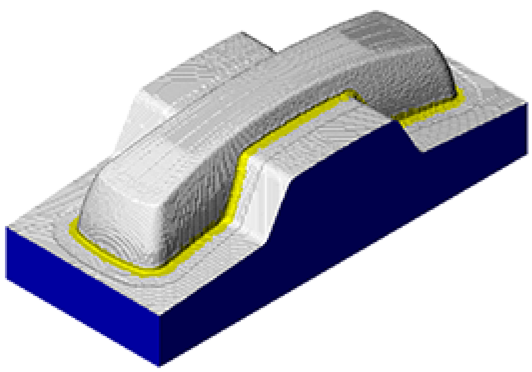
The part models for all of the exercises were created in the modeling exercises of this manual. If you have not gone through those exercises and created the solid models for these parts, you should do that now. We assume you are familiar with the interface and the principles used in Mill and Advanced CS.

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.

#1: The Phone

- The Set Up
- Creating Operations

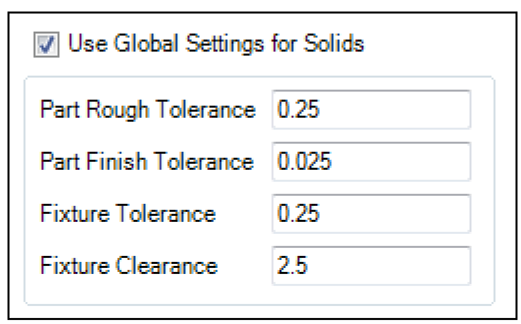
The Set Up



In this exercise, we will use a roughing process to remove the bulk of the stock. Lace cut surfacing toolpaths will then be created to semi-finish the phone core. Then we will create finishing passes using the 2 Curve Flow and Surface Flow options. We will complete the machining with an Intersection toolpath.

1. Open the Phone part created in modeling exercises.

2. Make the following changes in the Document Control Dialog Machining Preferences tab.



3. Increase the original stock +Z to 3mm and add a Clearance value of 15mm.

4. Create the following tool list.

#	Type	Total Length	Diameter	Bottom Radius	# Flutes	Flute Length	Material
1	Rough EM	121mm	25mm	0	3	45mm	HSS
2	Ball EM	74mm	12mm	n/a	4	25mm	TiN Coated
3	Ball EM	73mm	9mm	n/a	4	22mm	TiN Coated
4	Ball EM	50mm	6mm	n/a	4	19mm	TiN Coated
5	Finish EM	50mm	6mm	1mm	4	19mm	TiN Coated

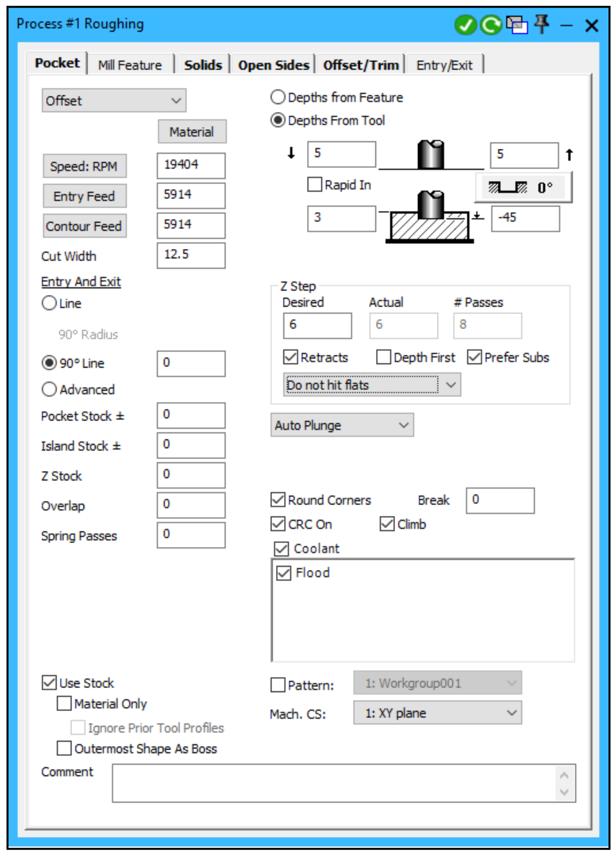
Creating Operations

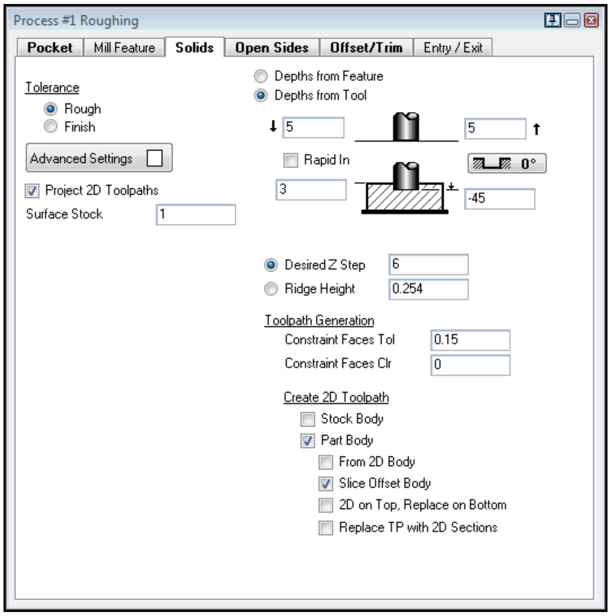
#1: Roughing

First, we will create a roughing process to remove the bulk of material from our starting stock.

1. Create this Roughing process with the 25mm Rough EM (Tool #1).

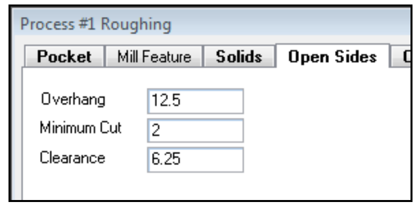
The Depth First option is not used in this process. While the part may appear to create multiple pockets, the model is really an open sided pocket with a boss in the middle. The roughing process is designed to generate routines which will remove material from within a closed shape. You may select only a body for the cut shape; the toolpath generated will use the stock as a constraint shape for the roughing. The roughing toolpath will cut within the stock shape, around the selected body.



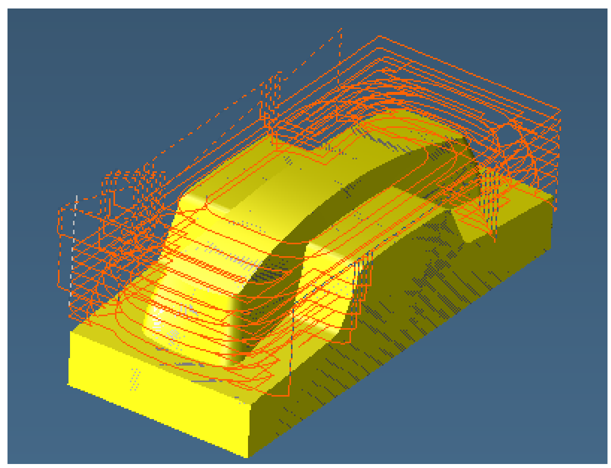


2. Enter this information in the Solids tab.

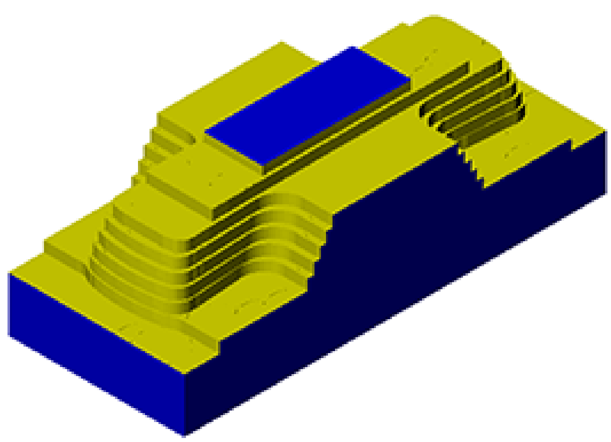
   Select the Rough tolerance to generate faster toolpath. Using the Slice Offset Body option will generate 2D optimized toolpath. We will use the default settings in the Open Sides tab.

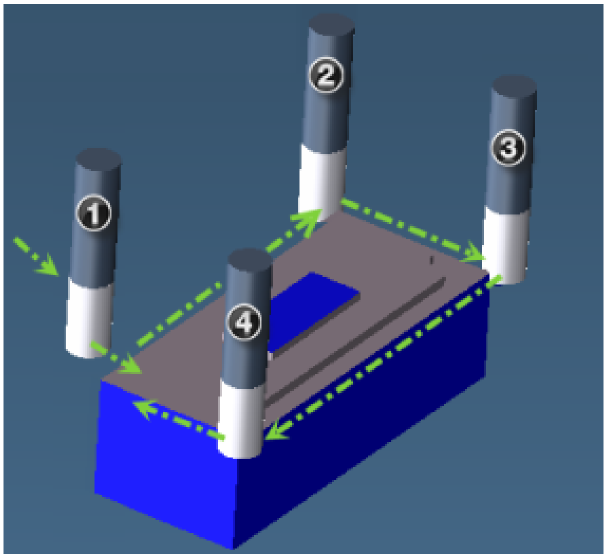


3. Select the model and `click` the Do it button.



4. Render (`F6`) the cuts  of the operation to verify the results.





Rendering after each operation is highly recommended when machining solids.

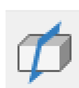
Since this is an open pocket, the tool enters from the outside of the part and cuts towards the center.

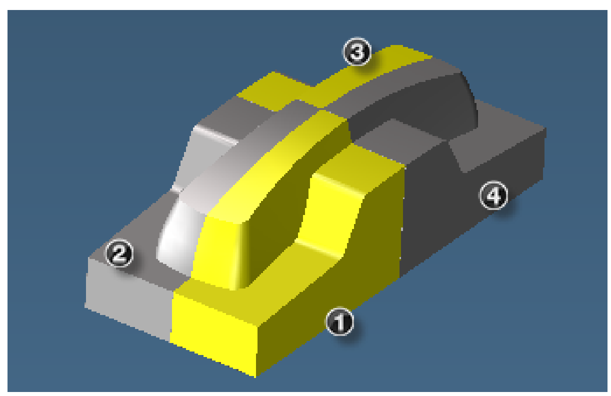
5. Deselect the operation in the Op list.
6. `Click` in an empty space in the Process list and choose Select All (`Ctrl+A`) and press `Delete`.

   This deletes all processes in the list.

#2-5: Lace Cuts

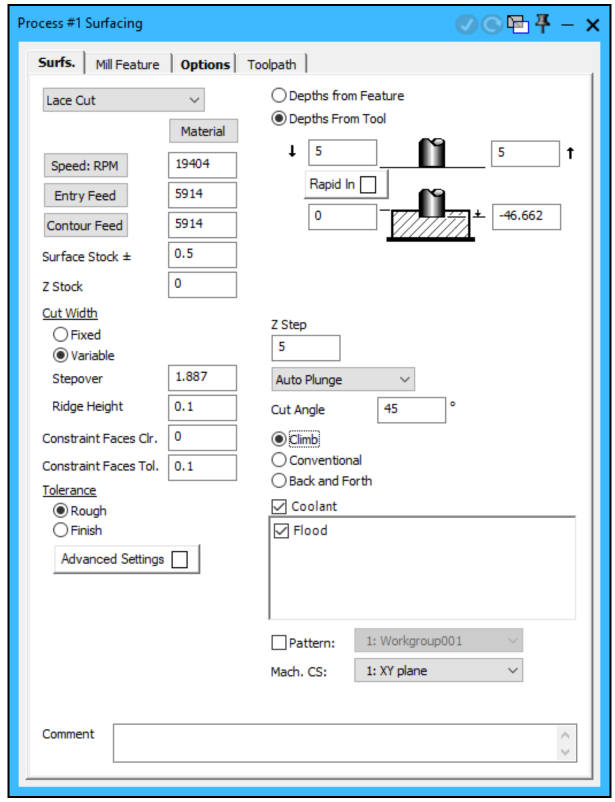
In order to create the semi-finishing lace cut toolpaths, we will slice the body into four quadrants. This will allow us to machine the part in sections; each section using a different cut angle and cutting towards the top of the part in a clockwise direction.

1. Non-destructively (`Alt`) slice  the model with the XZ and a new YZ plane.



When sliced, you should have four pieces in addition to the original phone body. We will now lace cut each section with the same process with a different Cut Angle, starting with the piece in the -X, -Y quadrant and continue clockwise around the model.

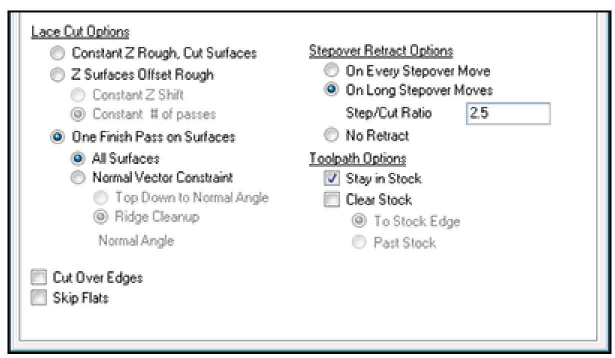
2. Create this Surfacing process with the 12mm Ball EM (Tool #2).



We are going to need to select only the faces we intend to machine. If we do not deselect certain faces the toolpath will automatically cut over the edge of the body. While this does not violate the solid quarter, it does cut into the final part.

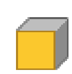
The information in the Toolpath tab is of no concern to us right now. The format you would select depends on the capabilities of your control.

3. Enter this information in the Options tab.

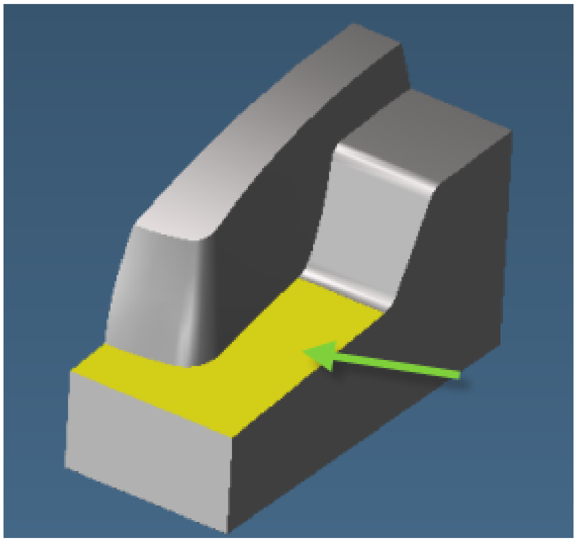


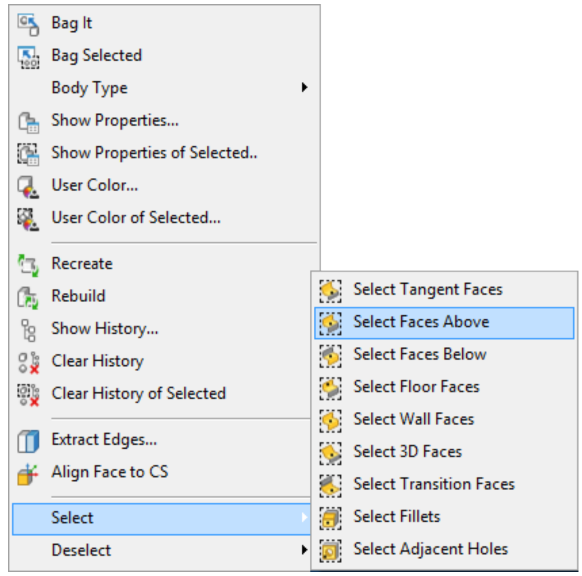
4. Switch to the XY plane.

5. Turn on Face Selection.



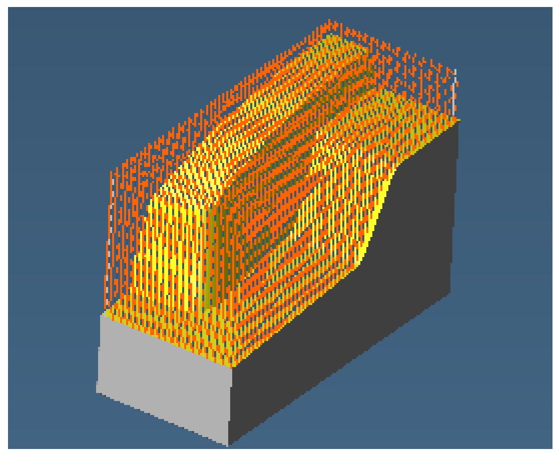
6. Right-click the face shown and choose Select>Select Faces Above.





This will select only the faces we need to machine. The walls and bottom face will not be selected because the faces are selected based on the current CS.

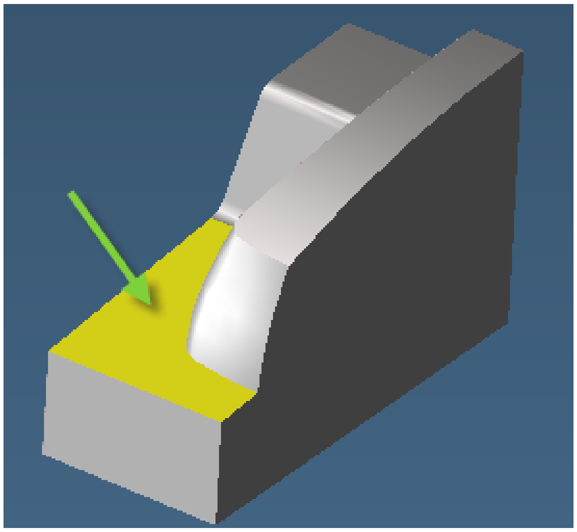
7. Create the toolpath.

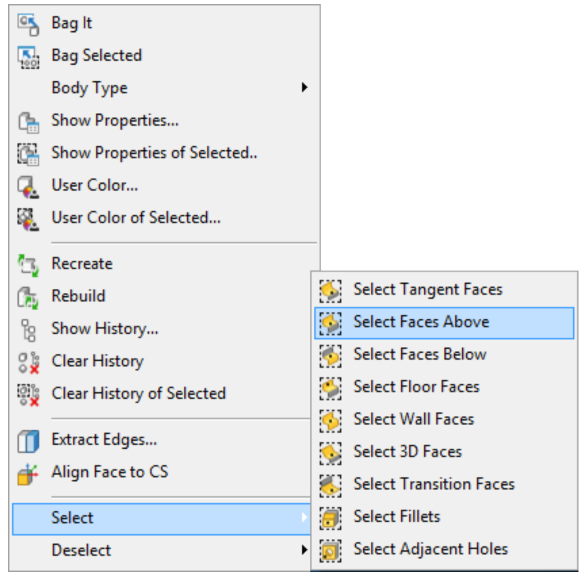


The next quarter will only use a different Cut Angle.

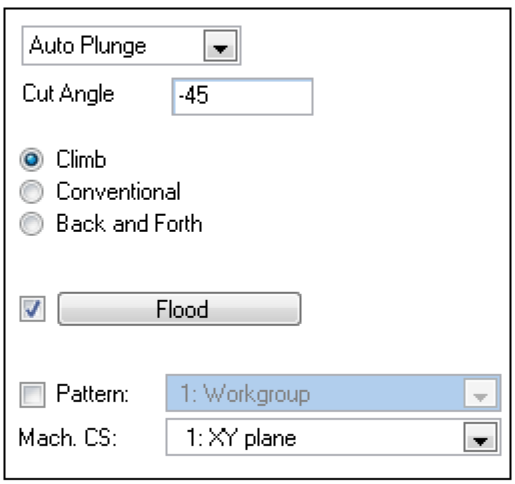
8. Deselect the Operation in the op list.

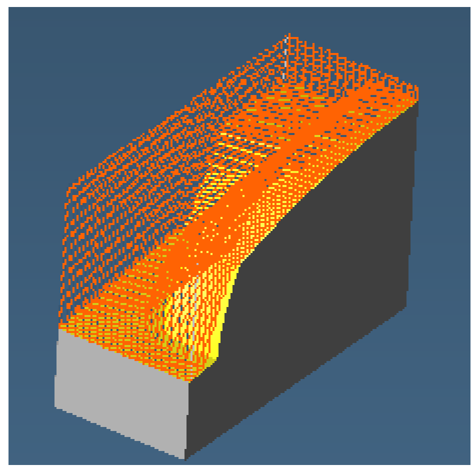
9. Select the faces on the next quarter as we did for the last operation.





10. Using the same process change the Cut Angle and create the toolpath.

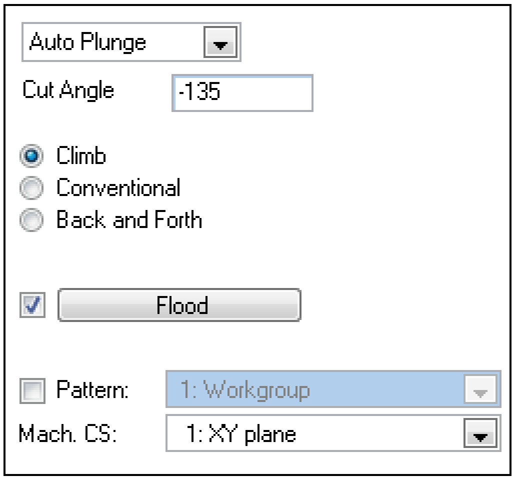


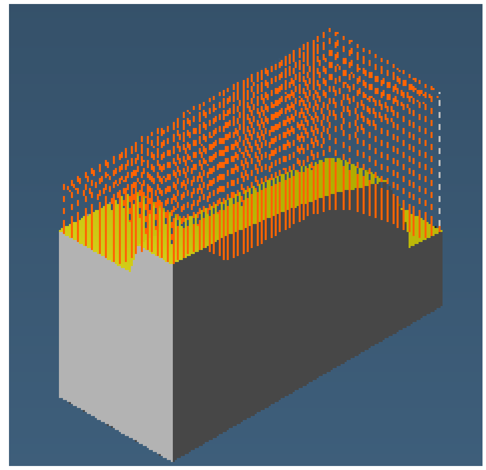


The information contained in the Options window remains the same on the following two lace cut operations.

11. Deselect the operation.

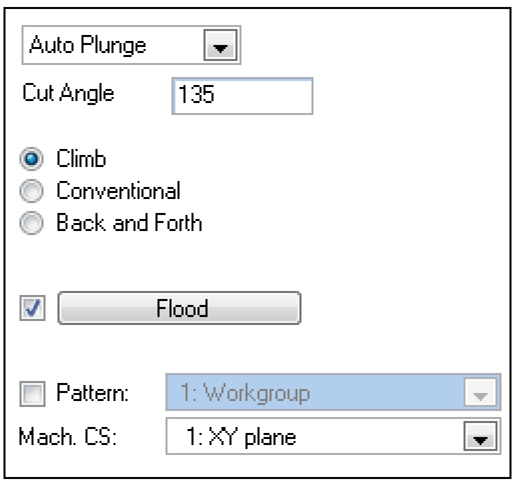
12. Select the faces for the next quarter, change the Cut Angle and create the toolpath.

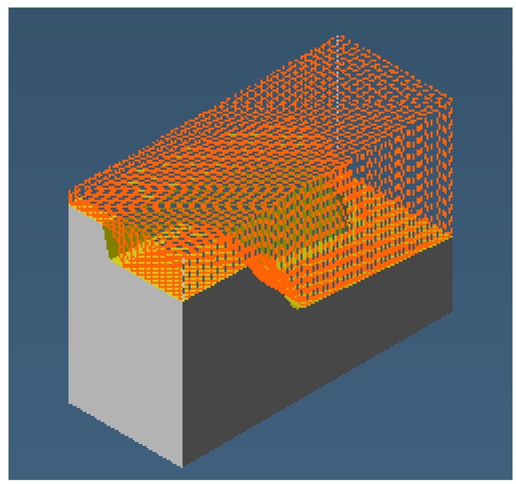




13. Deselect the operation.

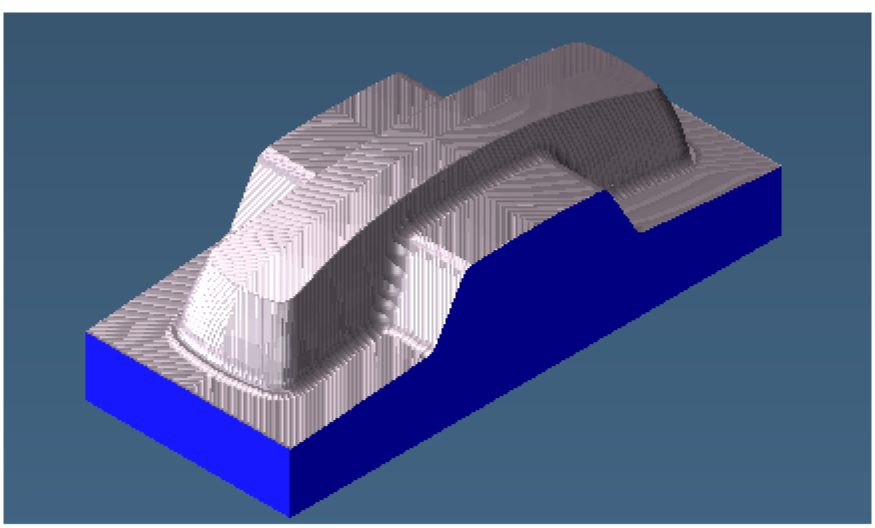
14. Select the faces for the last quarter, change the Cut Angle and create the toolpath.





We have specified that all tool cuts go toward the center of the part.

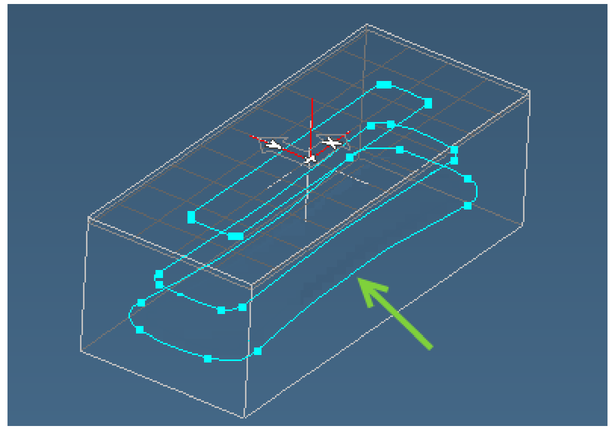
15. Render the operations.

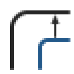


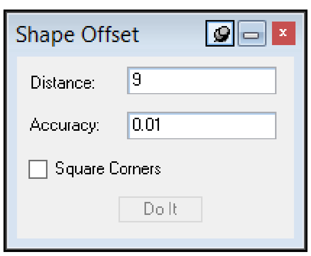
#6: 2 Curve Flow

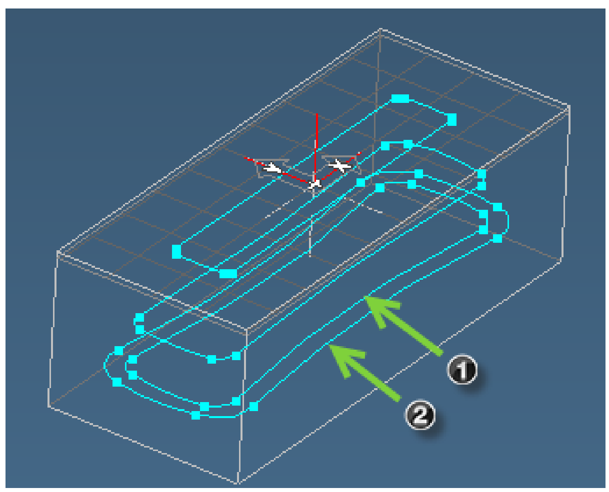
The next operation to be performed will be a 2 Curve Flow around the handset. For the tool to be able to perform the operation we will need to offset the handset geometry. Geometry acts as a constraint area in 2 Curve Flow operations. If we did not offset the geometry the tool would have very little room to move, resulting in cutting only a few millimeters down from the top of the handset.

1. In the XY plane, select the base geometry.



2. Open the Shape Offset ( >  > ) dialog and create the offset shown.



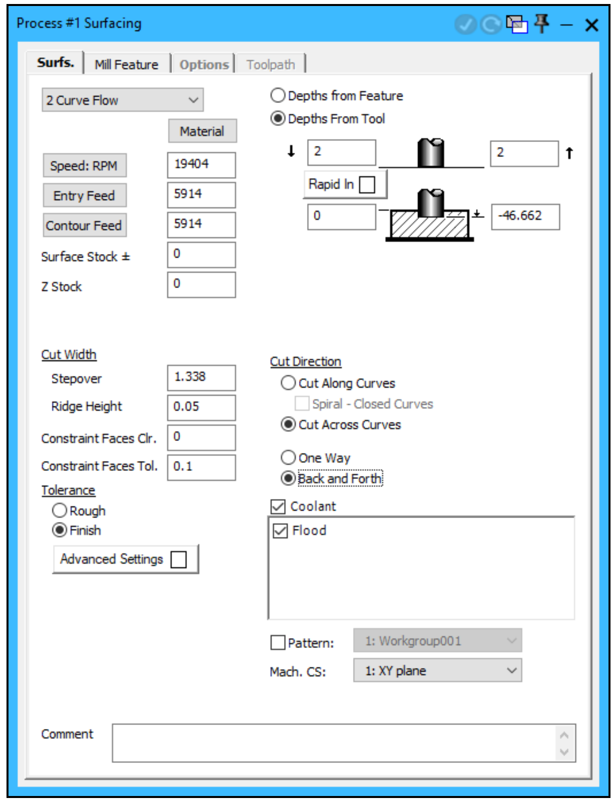


1. Original Shape
2. **Outer** Offset Shape

Shape Offset creates offsets in two directions from the original shape.

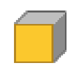
3. Delete the smaller interior offset that was created.

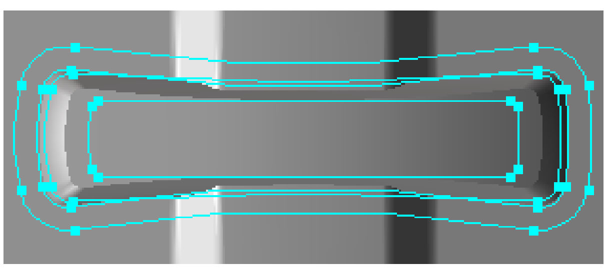
4. Create this Surfacing process with the 9mm Ball EM (Tool #3).



Be sure to set the Cut Direction to Cut Across Curves. Cut Along Curves will follow the contour of the shapes. We want to cut up and down around the handset.

5. Turn off Face Selection.

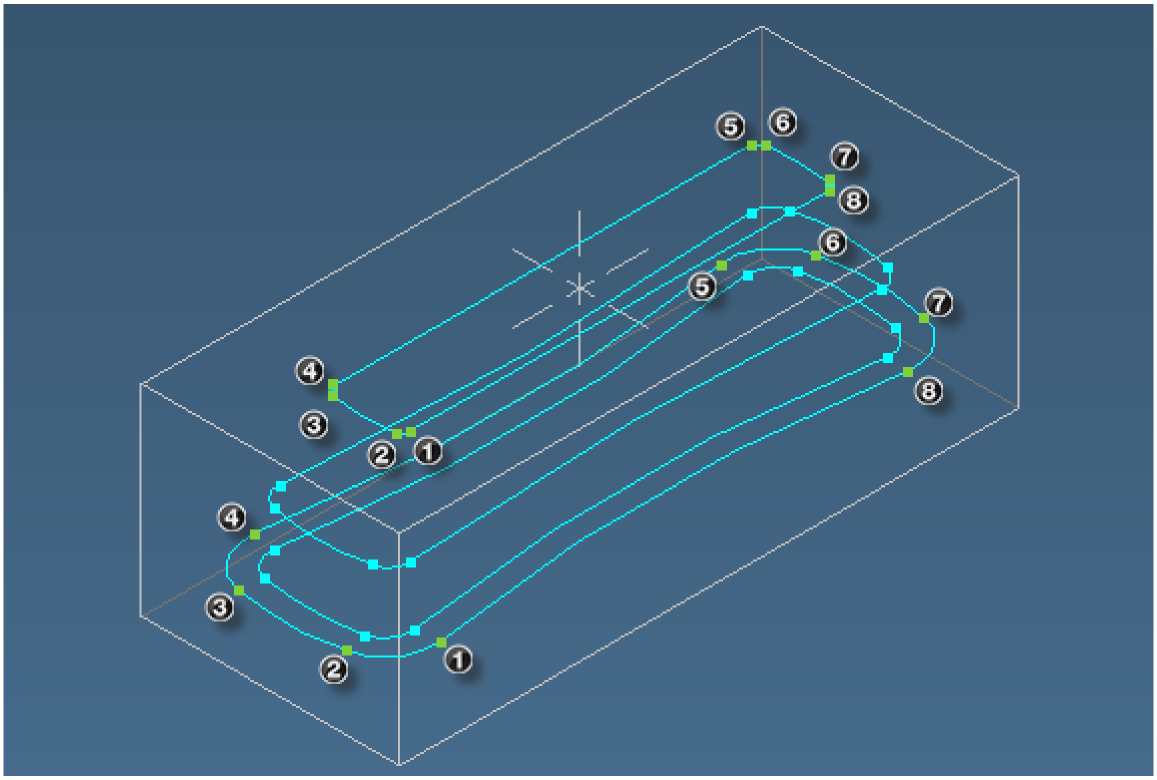




It is clear from the top view why we created the offset. The offset of 9mm gives the tool room to perform the 2 Curve Flow operation to the base of the phone.

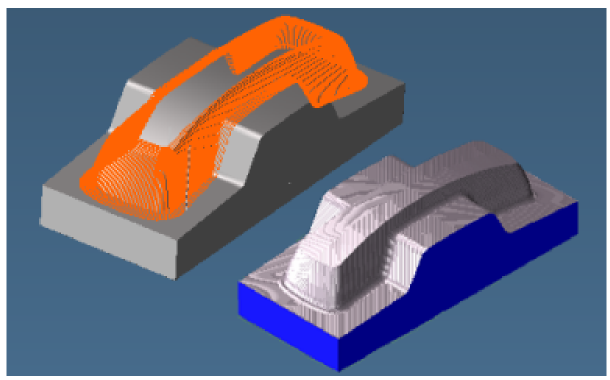
To create the 2 Curve Flow operation we will need to select alignment points in the geometry similar to lofting a solid. Then, we will need to select the body.

6. Select alignment points in the order shown.



The points may be selected using the shape sequence method or the shape to shape method described in the modeling exercise for this part.

7. (`Ctrl+click`) the body and create the toolpath.

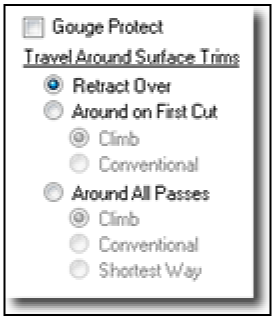


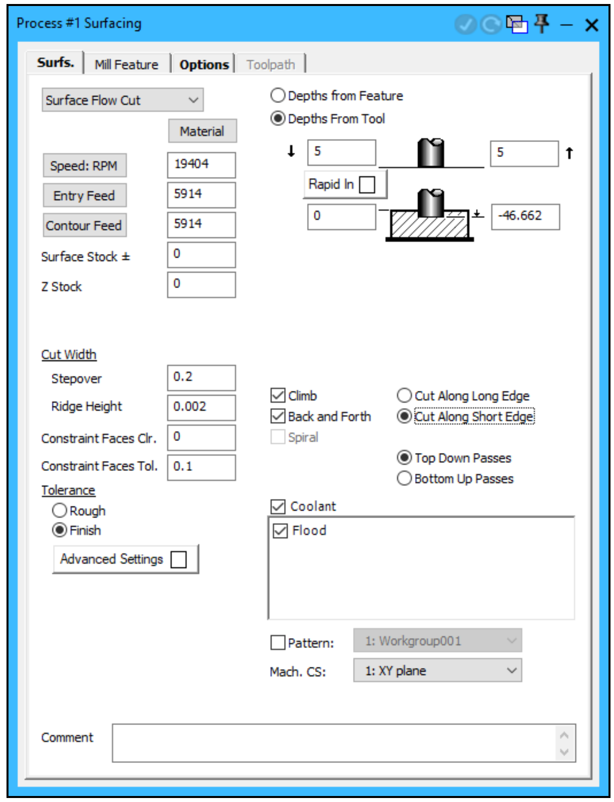
8. Render the part.

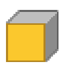
#7: Surface Flow

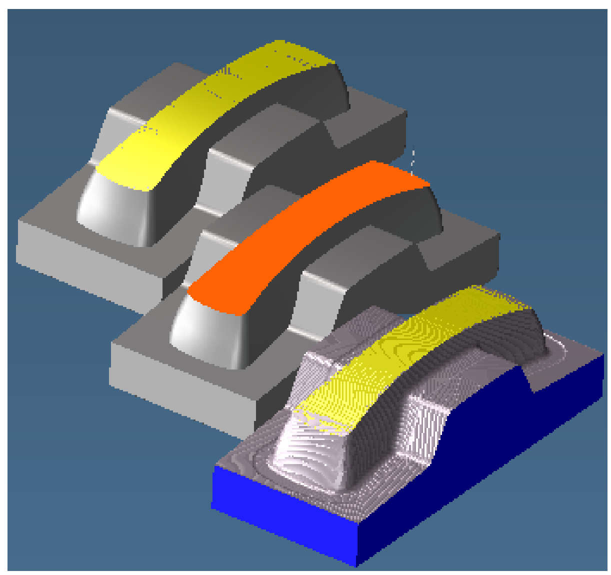
The next operation will machine the top of the handset.

1. Create this Surfacing process with 6mm Ball EM (Tool #4). Set the Options tab items as shown.





2. Turn on Face Selection  and select the top face.



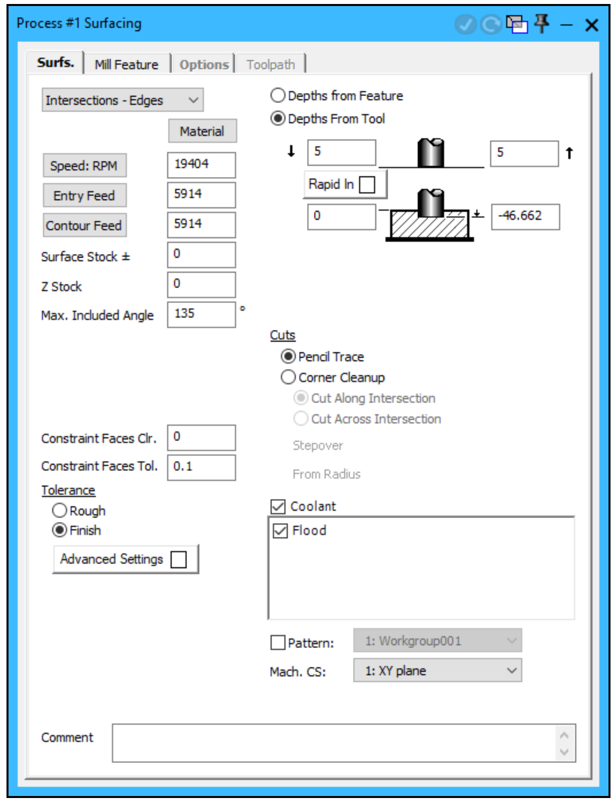
This operation will produce toolpath that cuts back and forth along the Y axis, producing a smooth finish on the handset.

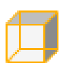
3. Create the toolpath.
4. Render the part.

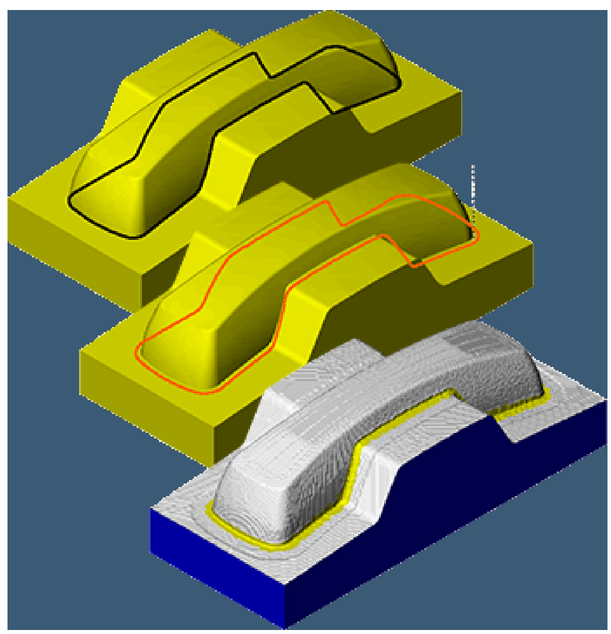
#8: Edge Intersection

Finally, we will create an Intersection cut using a selected edge. This will serve to clean up the body where larger tools could not reach.

1. Create this Surfacing process with the 6mm Finish EM (Tool #5).



2. Turn on Edge Selection  and `Double-click` anywhere on the edge shown.



Make sure you are using the 3D chain mode in the edge context menu. For more information, see "Edge Context Menu" chapter in Solid Surfacer User guide.

3. (`Ctrl+click`) anywhere on the body and create the toolpath.

4. Render the part.

#2: The Hot Punch

The Hot Punch will be machined with three operations – a contouring operation to clear the perimeter of the part, followed by a lace cut surfacing operation to finish the part, followed by a projected engraving operation.

Part Set-up

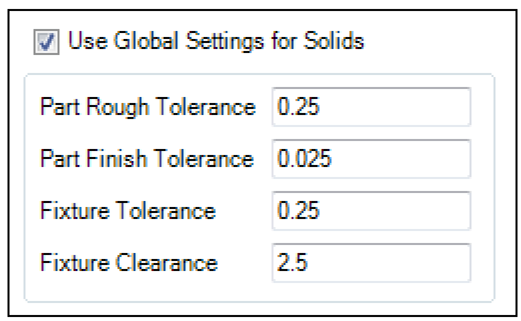
Document Control Dialog

1. Open the Hot Punch part created in Modeling Exercise #2.

2. Change the workspace stock size to the following dimensions:
   +X = `40`, -X = `-40`, +Y = `35`, -Y = `-35`, +Z = `0`, -Z = `-45`

3. Add a Clearance value of `15`mm.

4. Set the Global Settings in the Machining Preferences tab as shown.



Tool List

1. Create the following tool list.

#	Type	Total Length	Diameter	Corner/Tip	# Flutes	Flute Length	Material
1	Rough EM	121mm	25mm	0mm	3	45mm	HSS
2	Ball EM	73mm	10mm	n/a	4	22mm	TiN Coated
3	3.15 Center Drill	31.5mm	3.15mm	118	2	1.9mm	Carbide Solid

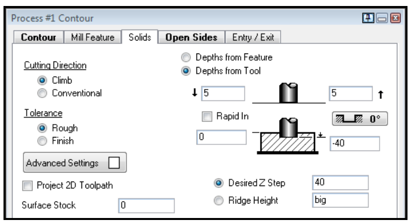
*For the Center Drill you will only need to specify the Size from the menu.

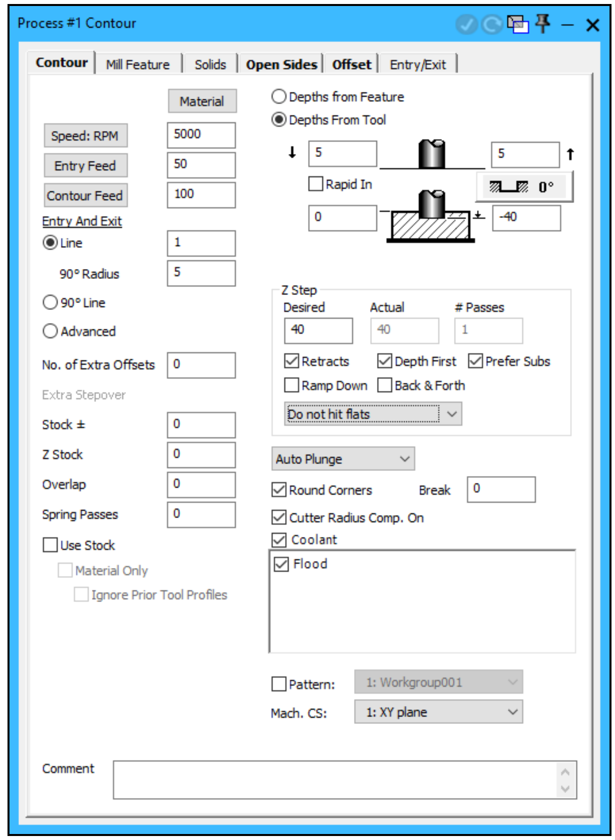
Creating Operations

#1: Contouring

First, we will create an operation to contour around the part, creating a single pass along the outside wall.

1. Create this Contour process using the 25mm Rough EM (Tool #1).
2. Enter this information in the Solids tab.

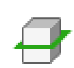




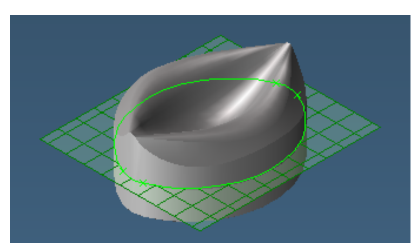
The settings in the Open Sides tab do not influence this operation so we can ignore it.

To define the cut shape for this process, we will use only the Profiler, no solids or sheets. The system will only use the profile shape and machining markers to create the toolpath.

3. Click the Toggle Profiler button.

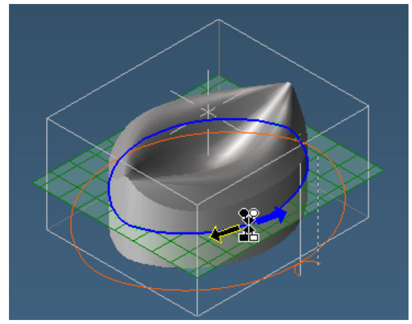


4. Drag the grid to where the entire outer perimeter is selected. (if necessary check in Side view)



5. Select the profile.

6. Set the machining markers as shown.



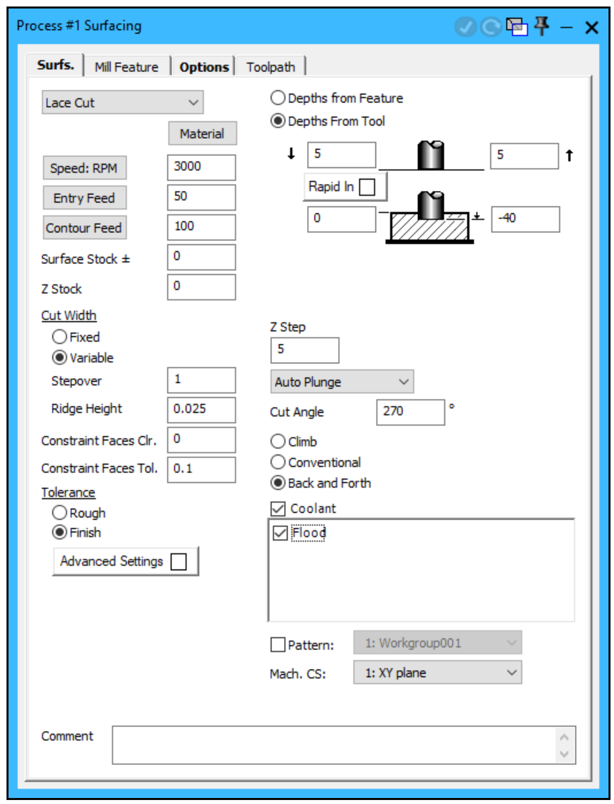
The markers should be set for conventional cutting.

Make sure the solid is not selected. If the solid is selected, the system will attempt to project the contouring toolpath onto the selected body. The result would be the same toolpath, however it takes much longer to process.

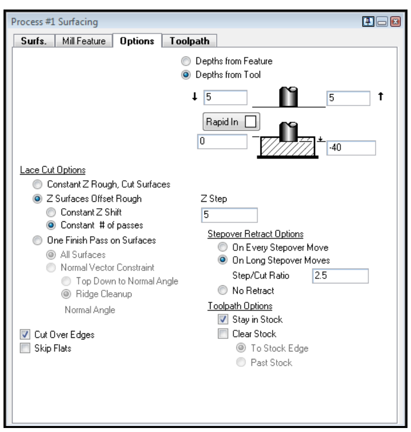
7. Create the toolpath.

#2: Lace Cut

The next operation will lace cut across the top face of the part.



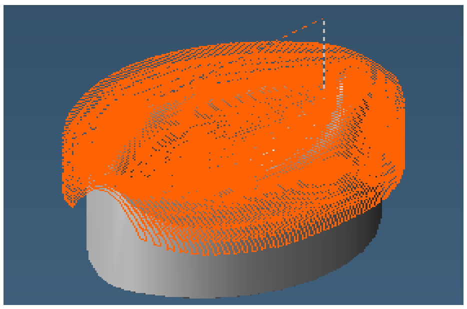
1. Create this Surfacing process with the 10mm Ball EM (Tool #2).
2. Enter this information in the Options tab.



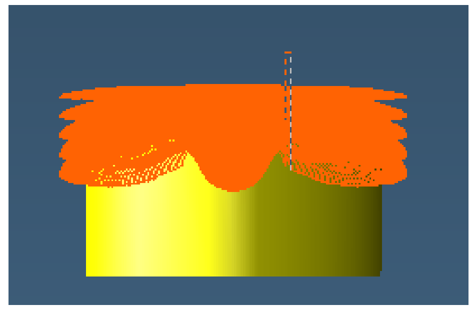
The Z Surfaces Offset Rough Lace Cut option is very useful with custom stock bodies, particularly those that have undulating shapes. The Constant # of passes option uses the desired Z Step value to break up the range of the cut into sections. Each of these sections will be completely cut by the pass, there are no retract moves. Each pass will be a different shape which cuts over the entire selected body.

The Clear Stock option is not selected because we do not want the surfacing toolpath to cut the area of the part that has been removed by the previous contouring operation. When Clear Stock is off, the toolpath generated by the process will only look at the selected part model rather than the stock, whether it be a stock body or the stock definition entered in the Document Control dialog. By turning Stay In Stock on we will confine the generated toolpath to the stock boundaries. Were this option off, the toolpath would have two additional passes outside of the stock bounds to complete the body's shape. This is not necessary in this part.

3. Select the solid and create the toolpath.



Isometric View



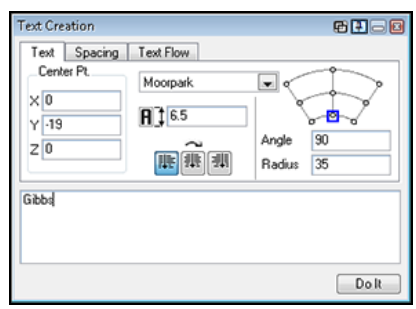
Right Side View

#3: Engraving

Finally, 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 tab 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 A dialog.

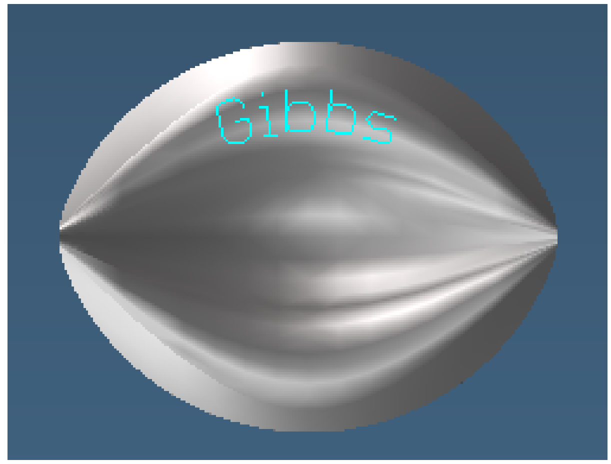


2. In the Text Flow tab select the clockwise arc button for the Shape.

3. Enter the above information in the Text tab.

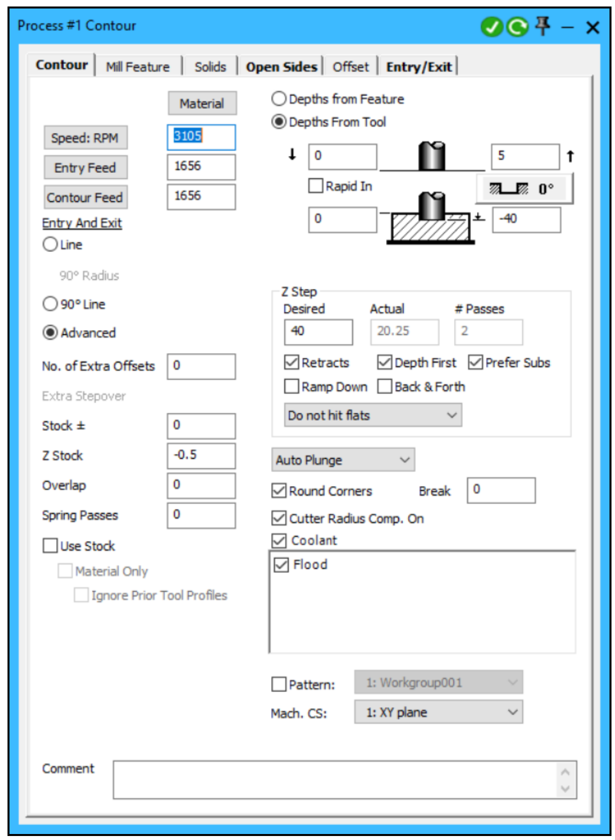
4. Create the text. If your text looks too unclear, make sure you have View>Draw points (Ctrl+J) unchecked. This turns off display of points connectors and terminators.

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



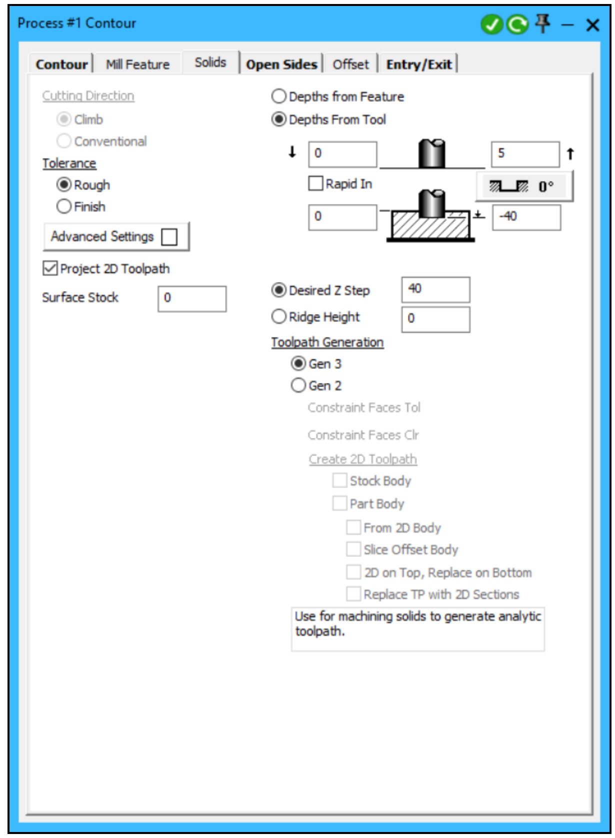
5. Select the text geometry. (Shift+drag) to draw a window and select an area, then (Ctrl+click) to deselect the solid.

6. Create this Contouring process with the Center Drill (Tool #3).



When more than one shape is selected prior to creating a contouring process, the system automatically assumes that you are doing engraving. Several items will become disabled. 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. Enter this information in the Solids tab.



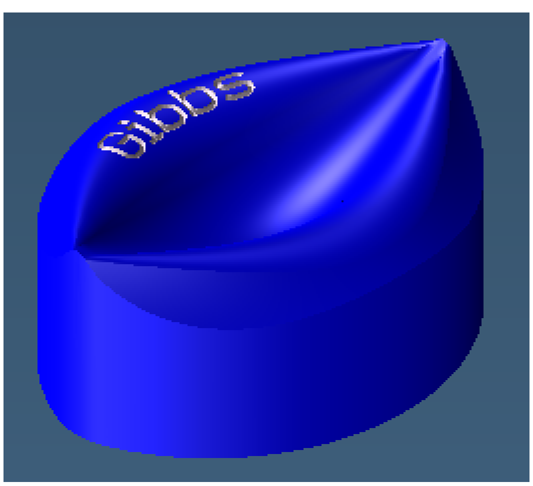
8. (`Ctrl+click`) the solid.

   Make sure the text geometry is still selected.

   The text and the body 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 operations.



11. Save this part.

#3: Plumbing

- Part Set-Up
- Machining The Part

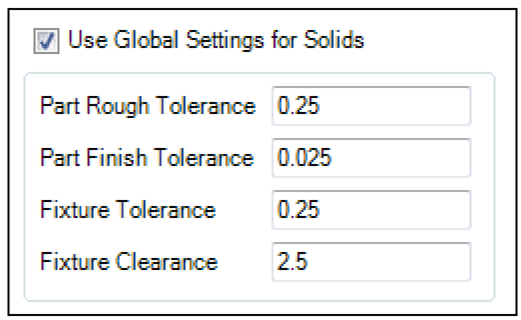
Part Set-Up

The Basics

To machine this mold we will first use a roughing process to clear out the greatest amount of material. We will then use a series of lace cutting operations to machine specific areas of the part. Next, we will use a contour operation to cut a particular face. Lastly we will select certain faces for an intersection.

1. Open the Plumbing part created in Exercise #1 in the Solid Modeling Exercises.
2. Ensure the workspace stock size is:

   +X = 80, -X = -80, +Y = 80, -Y = -80, +Z = 0, -Z = -50.
3. Add a Clearance value of 15mm.
4. Set the Global Settings in the Machining Preferences tab as shown.



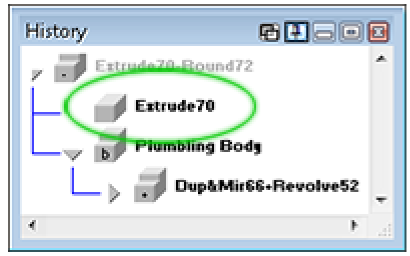
5. Create the following tool list.

#	Type	Total Length	Diameter	Corner/Tip	# Flutes	Flute Length	Material
1	Rough EM	73mm	10mm	2mm	2	22mm	HSS
2	Ball EM	50mm	6mm	n/a	4	19mm	TiN Coated
3	Finish EM	100mm	25mm	0mm	4	38mm	TiN Coated
4	Ball EM	38mm	3mm	n/a	4	20mm	TiN Coated

Stock Body

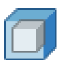
The first thing we will do is designate a stock body. We will use the extruded cylinder as a basis for this.

1. From Show History list extract the extruded cylinder.



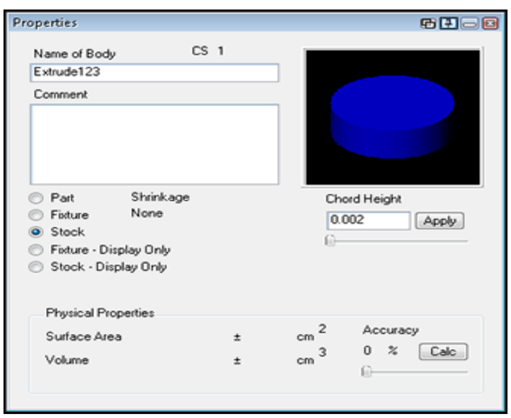
We will need to modify this solid so it will work as a stock body. A stock body must be larger than the part to be machined on all sides. The general formula for how much larger the stock must be is: surface stock + surface tolerance. This is a general rule. In this case we will just offset the stock body larger by 1mm.

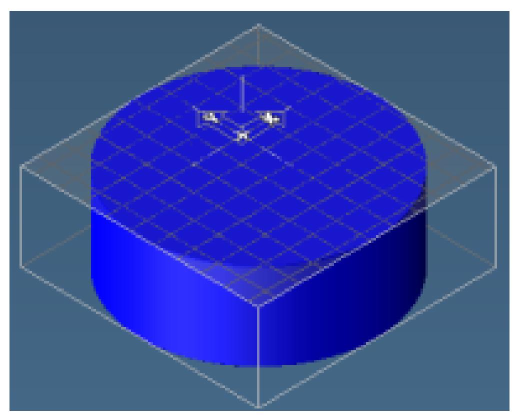
2. Select the extrude.

3. Offset  the cylinder by the amount shown.

4. Right-click the offset cylinder and choose Properties.

5. Designate the body as Stock.



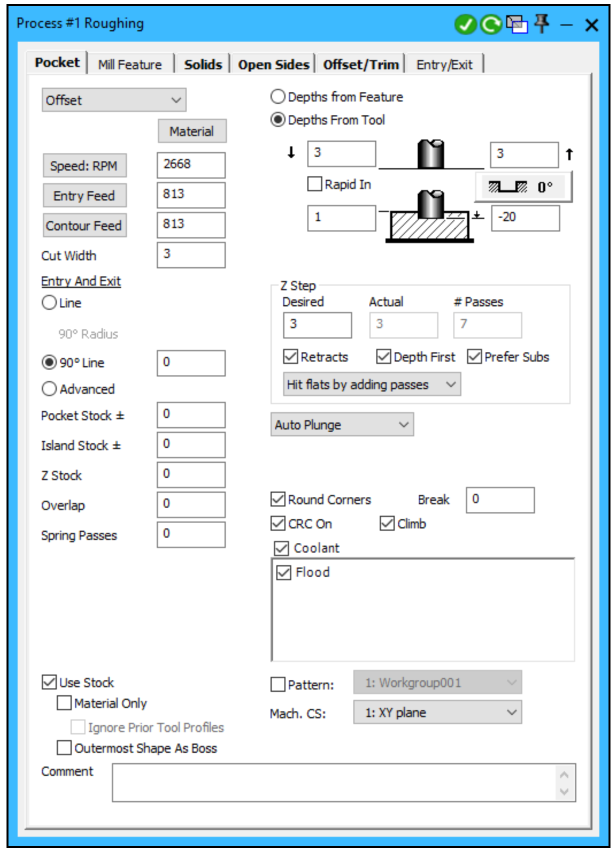


The stock body now extends 1mm outside the workspace stock. This will not affect the machining.

Machining The Part

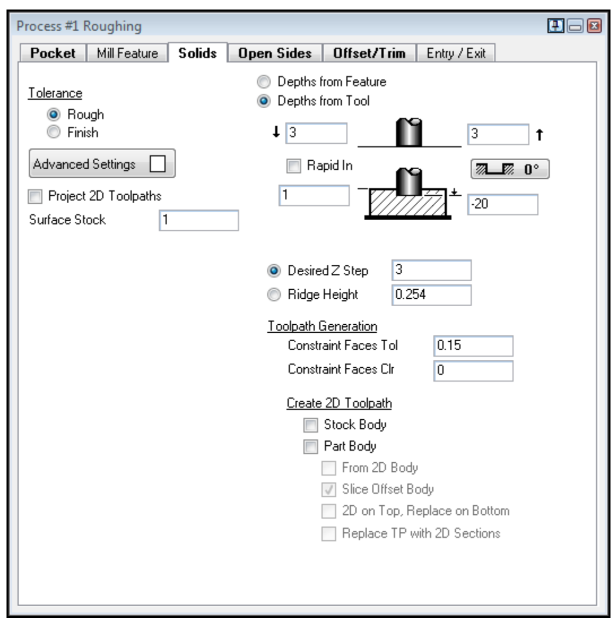
#1-3: Roughing

1. Create this Roughing process with the 10mm Rough Endmill (Tool #1).

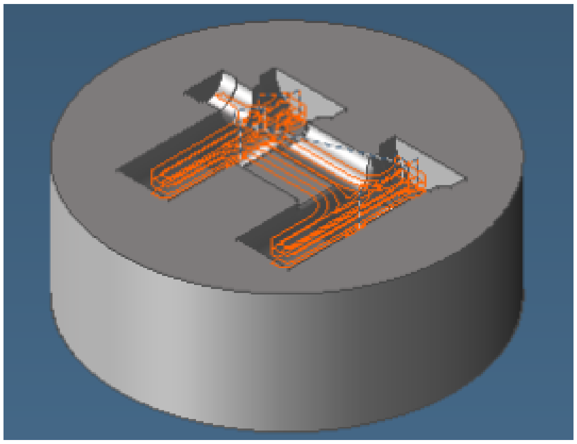


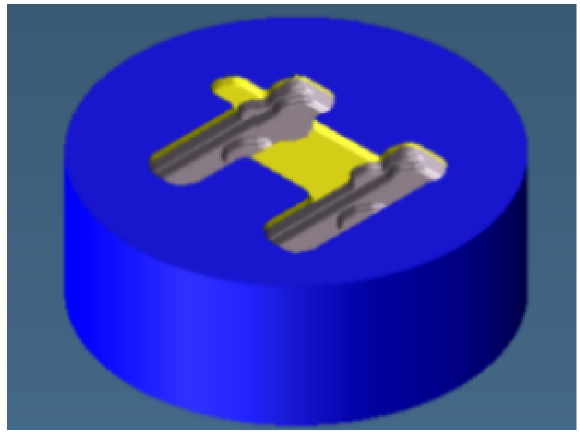
2. Set the Minimum cut to 5mm in the Open Sides tab.

3. Enter this information in the Solids tab.



4. Select the solid and create the toolpath.
5. Render the operations.

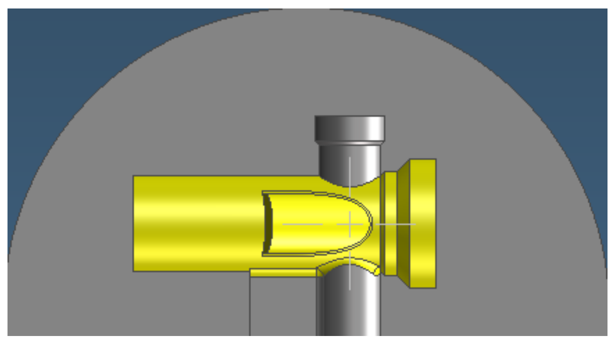




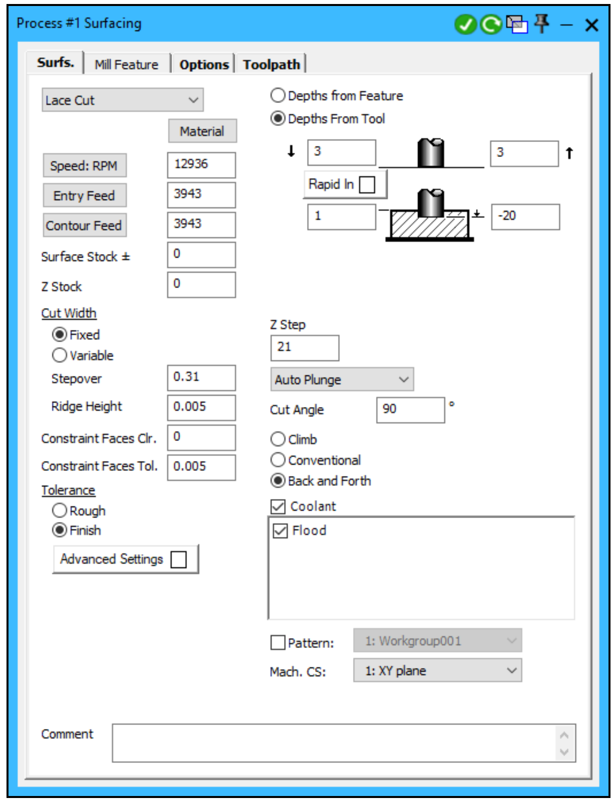
#4: Lace Cut

For this operation we will select all of the faces that make up one of the two main cavities.

1. Turn on Face Selection and (`Ctrl+click`) the faces shown.

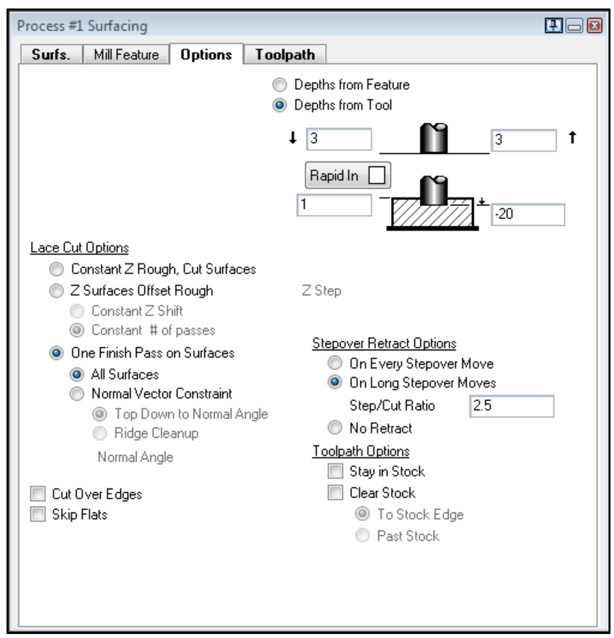


2. Create this surfacing process with the 6mm Ball EM (Tool #2).



Note that the Fixed option has been selected for the Cut Width. The Ridge Height value of `0.004`mm was entered and the XY Stepover was calculated at `0.31`mm.

3. Enter this information in the Options tab.



This process is going to be used as a semi-finishing operation to clean up or even out areas of the part that still have a lot of material after the roughing operation.

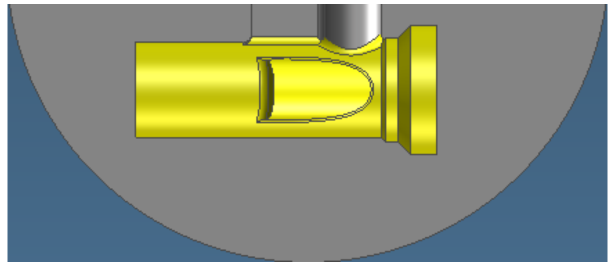
4. Create the toolpath.

#5: Lace Cut

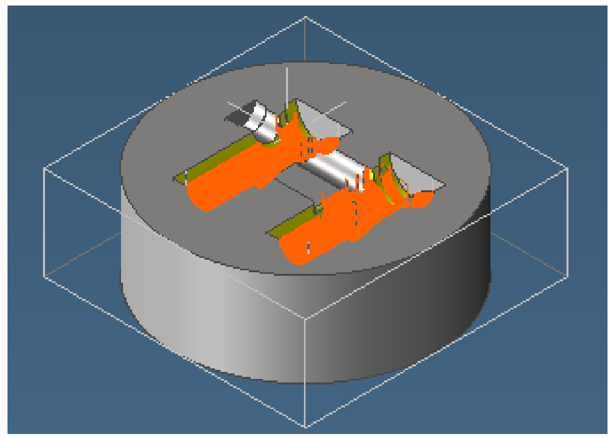
1. Deselect the operations and faces.

   For the next operation we will use the same process.

2. (`Ctrl+click`) the faces shown.

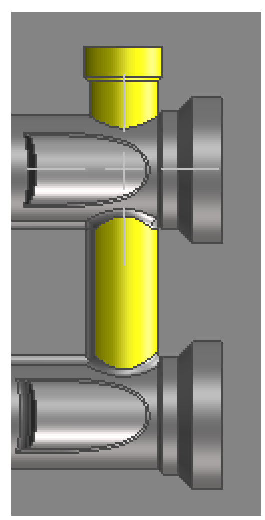


3. Create the toolpath.



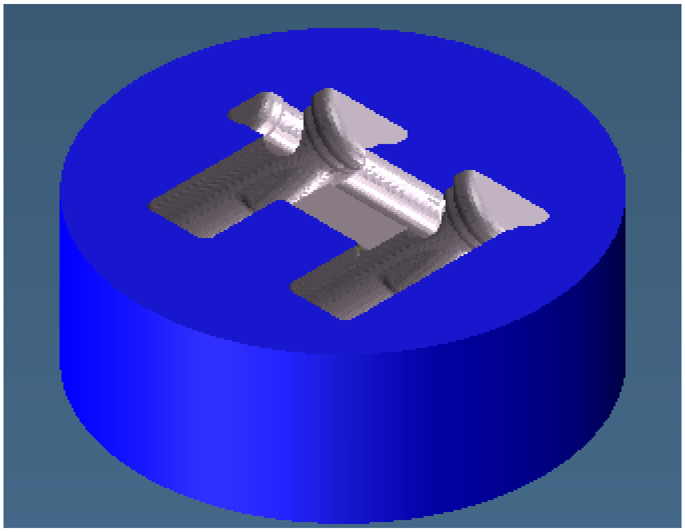
#6: Lace Cut

1. Deselect the operations and faces.
2. Change the Cut Angle to zer0 degrees.
3. (`Ctrl+click`) the faces shown.



4. Create the toolpath.

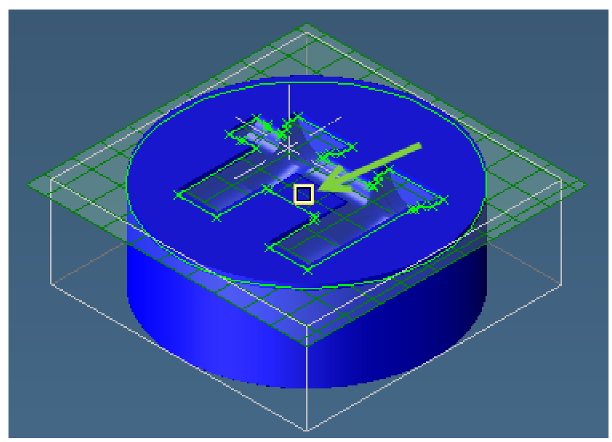
5. Render the operations.



#7: Contour with the Profiler

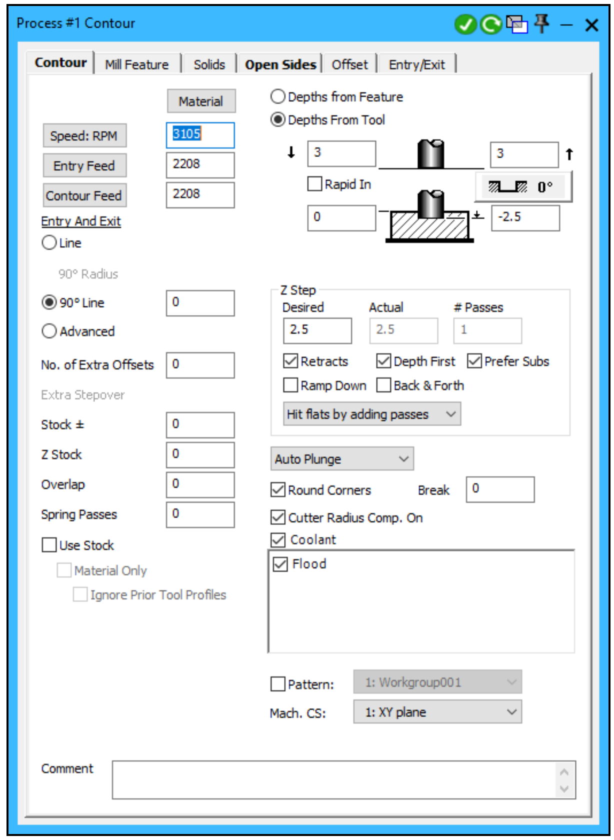
The next operation will be a contour operation using the Profiler to finish the flat face in the cavity.

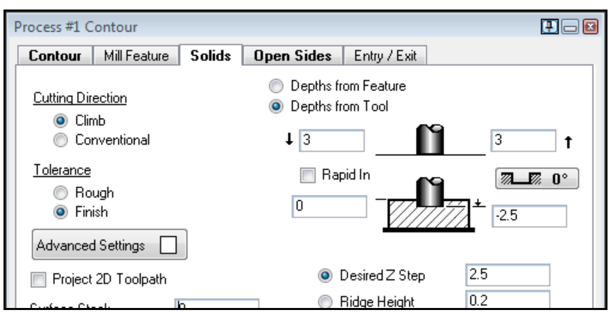
1. Activate the Profiler.
2. `Right-click` the Profile and choose Profiler Depth....
3. Interrogate the flat face in the cavity to load its depth into the Profiler Depth dialog.



The depth of the face should be –2.5mm.

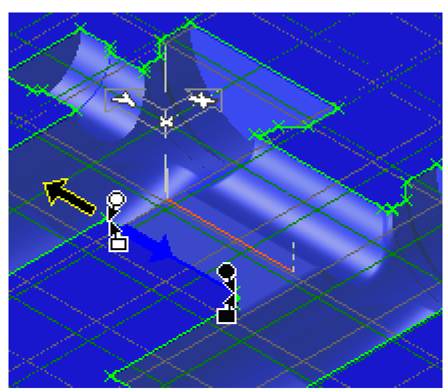
4. Create this Contour process with the 25mm Finish EM (Tool #3).





The Cutting Direction will become disabled after selecting the profile.

5. Set the machining markers as shown along the green profiler line.

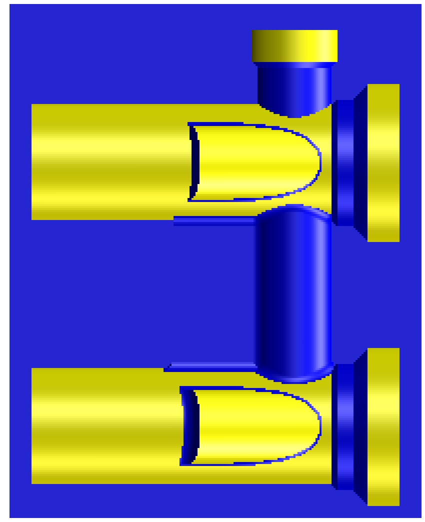


6. Create the toolpath.

#8: Corner Cleanup

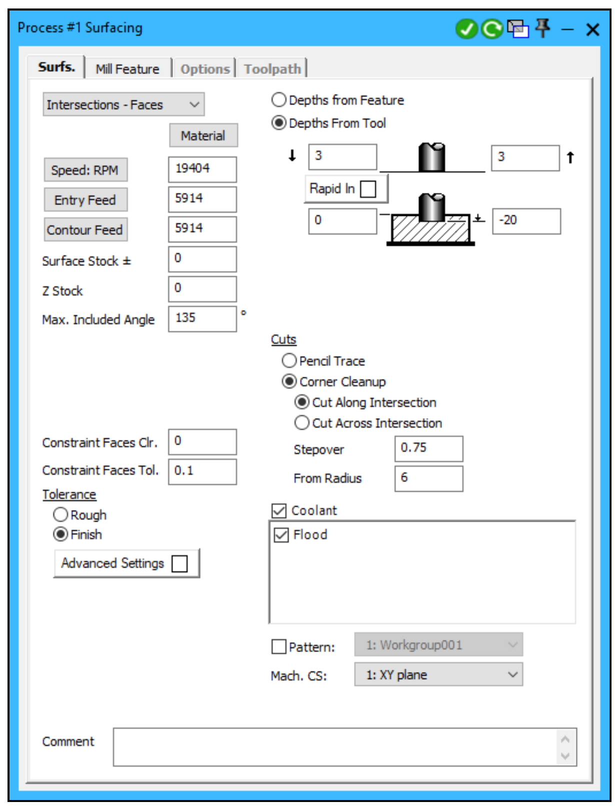
The Last thing we will do is use a small tool to finish the corners of the part. We will use an Intersections-Faces Surfacing operation to accomplish this.

1. Select the faces shown.



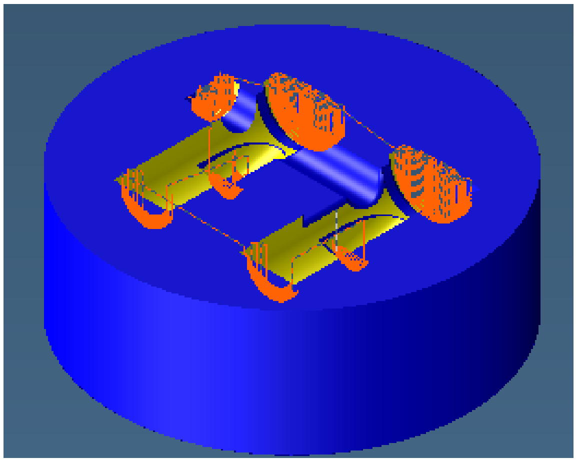
The corners of each of these faces have material the 6mm ball endmill left behind.

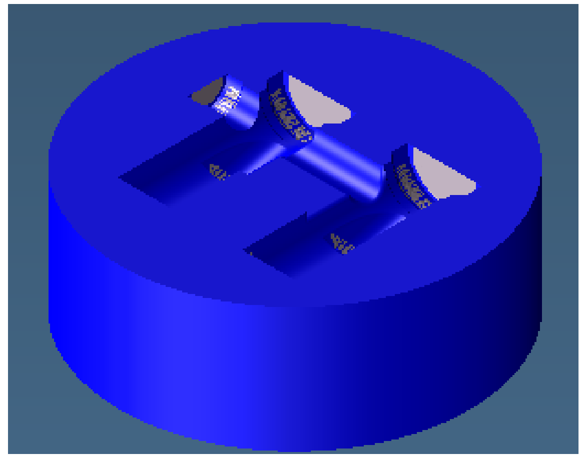
2. Create this Surfacing process with the 3mm Ball Endmill (Tool #4).



3. Create the toolpath.

4. Render the operation.





5. Save the file.

#4: The Remote Control

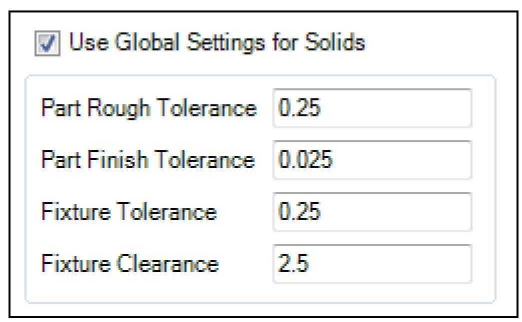
In this exercise, we will machine the cavity that was created in the corresponding modeling exercise.

Part Set-up

1. Open the Remote.vnc file that was created in Solid Modeling Exercise #5.

   Make sure that you open the .vnc file containing the core and cavity part models and not the x_t file, which was the Parasolids file that we originally imported.

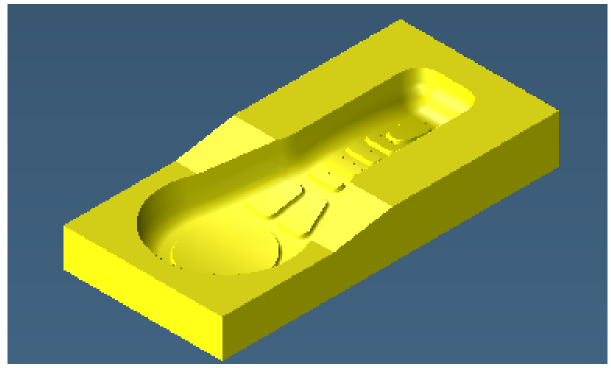
2. In the Document Control Dialog Machining Preferences tab, set the Global Settings as shown and add a Clearance value of 15mm.



Positioning the Part

Before we can machine the cavity we need to position the "Final Cavity" body so that it can be machined from the standard XY plane.

1. Select the cavity body and Modify>2D Rotate the body 180° clockwise in the YZ plane. Then, Modify>Translate the body down in Z by –27mm.



Tool List

1. Create the following tool list.

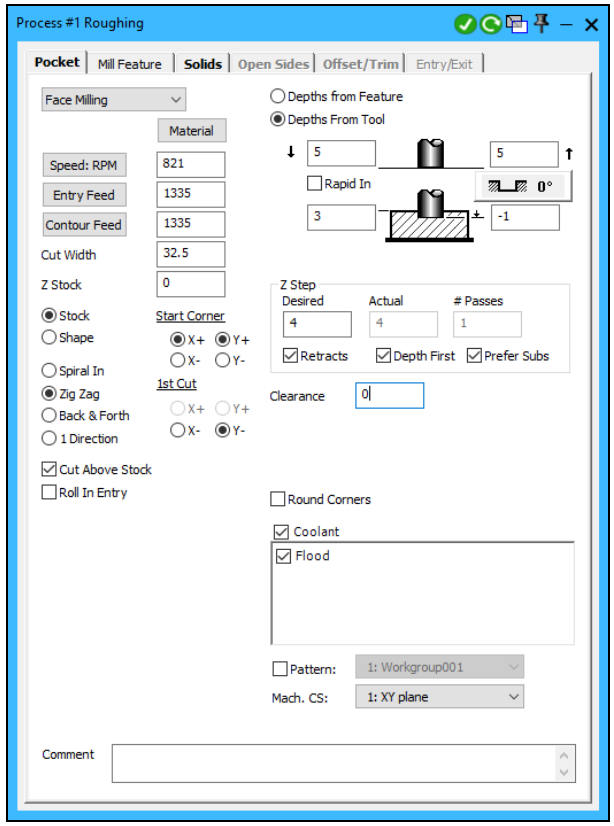
#	Type	Total Length	Diameter	Bottom Radius	# Flutes	Flute Length	Material
1	Face Mill	38mm	65mm	0mm	4	15mm	TiN Coated
2	Rough EM	38mm	15mm	0mm	4	6.3mm	HSS
3	Ball EM	38mm	9mm	n/a	4	12mm	TiN Coated
4	Ball EM	38mm	4mm	n/a	4	20mm	TiN Coated
5	Ball EM	38mm	2mm	n/a	4	6.3mm	TiN Coated

Machining the Part

#1: Face Mill

First, we will create an operation that will face mill the parting line surface.

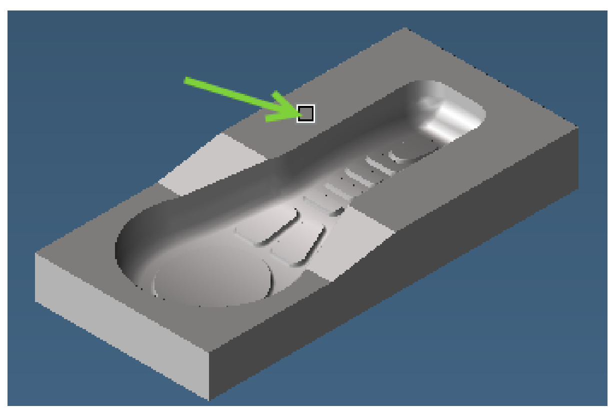
1. Create this Roughing process with the 65mm Face mill (Tool #1).

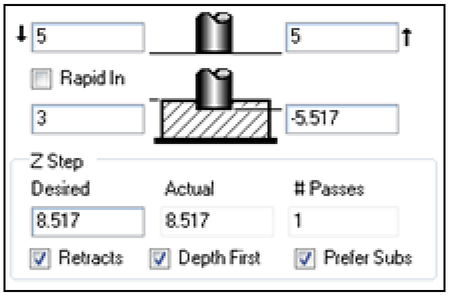


In order to specify the correct final Z depth for the operation in the Entry/Exit Clearance Diagram, we will load the Z depth of a face using the interrogation tool. In this operation, we want to cut to the depth of the first flat of the cavity.

2. `Click` in the final Z depth text box.

3. (`Alt+click`) to select the face indicated.





The Z depth of the selected face should be shown in the Final Z text box as shown in the following image.

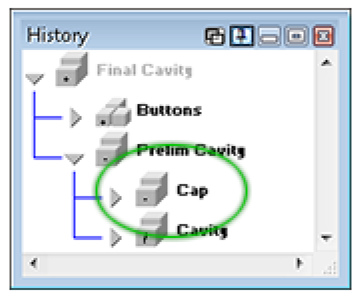
In this face mill process we have selected the Stock option for the cut shape of the operation, so we do not need to select any geometry or bodies to create the toolpath.

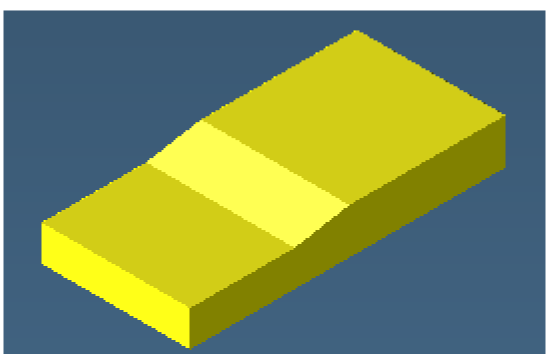
4. Create the toolpath.

Extracting the Cap

To continue the machining of the parting line body, we will lace cut the angled surface that connects the two flat faces. In order to machine the appropriate area, we need to bring the top base body (the Cap, which does not have the remote subtracted out) from the History list.

1. Open the History list of the Final Cavity body and Double-click the icon in front of the Cap item.





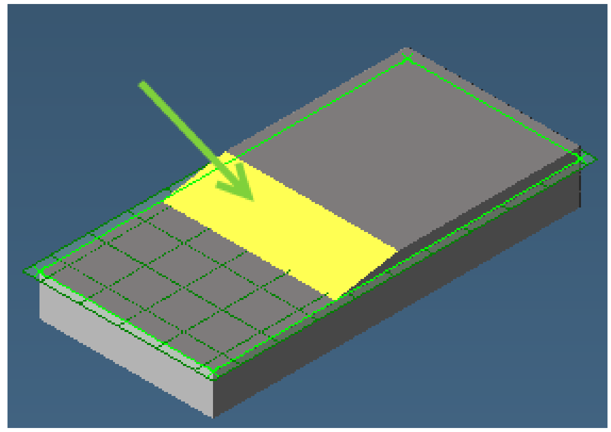
The Cap body will be brought back to the workspace. You will need to 2D rotate and translate this body, in the same manner that we originally did with the cavity, in order to get it in the appropriate machining position. Once this is done, the workspace should contain a part identical to the image on the left.

#2: Contour with Profiler

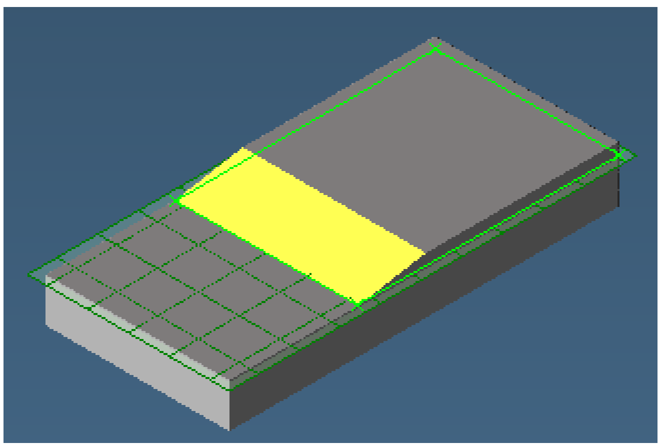
The next operation will face off the lower flat of the cavity parting line. We will use a contour process with the Profiler to accomplish this.

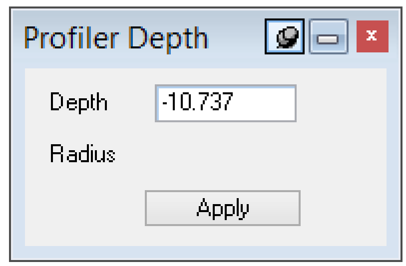
1. Turn on the Profiler. Right-click the green grid and choose Profiler Depth.

2. Interrogate the face shown.

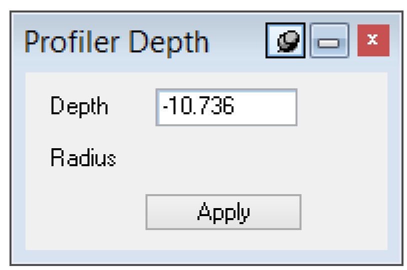


The Profiler now has the depth of the face. If you click Apply your part will look like the image shown here. We are not going to use this value. While the Profiler gives us the correct data (the lowest point of this face is Z-10.737mm), we need the Profiler to be on the lowest part of the angled face. We will adjust the depth data.



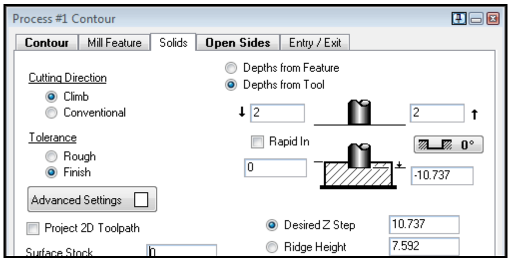


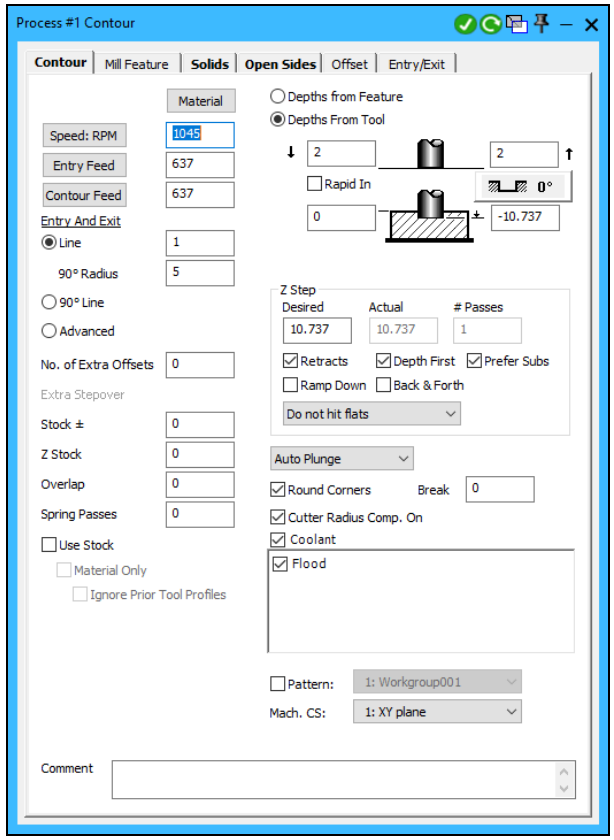
3. Modify the Profiler Depth to -10.736 mm and clickApply.



The 0.001mm difference moves the Profiler so that it actually lies around the area that we need, not the actual bottom boundary of the face. Switch off Profiler.

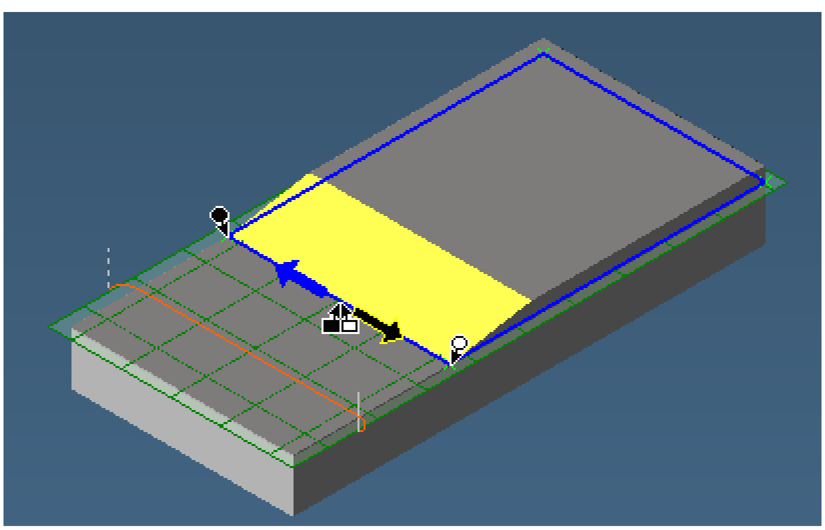
4. Create this Contour process with the 65mm Face Mill (Tool #1).





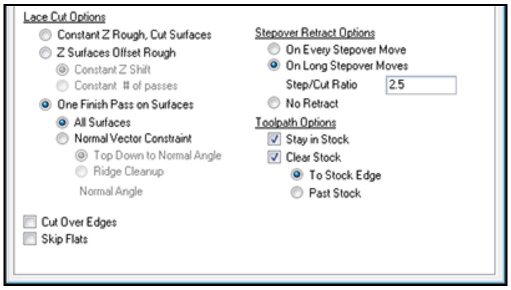
We need to set the machining markers on the Profiler and select the angled face to provide the cut shape process.

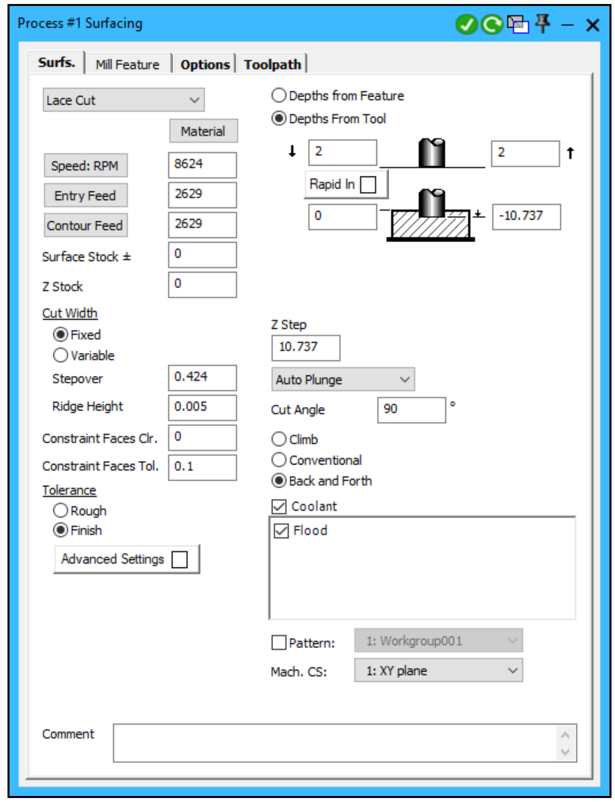
5. Set the machining markers as shown, `Ctrl-click` the angled face and create the toolpath. Switch off the profiler.



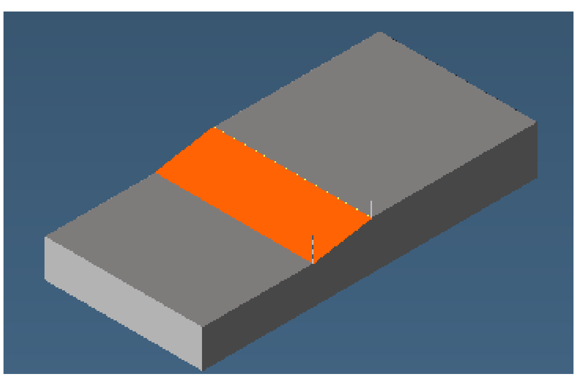
#3: Lace Cut on the Angled Face

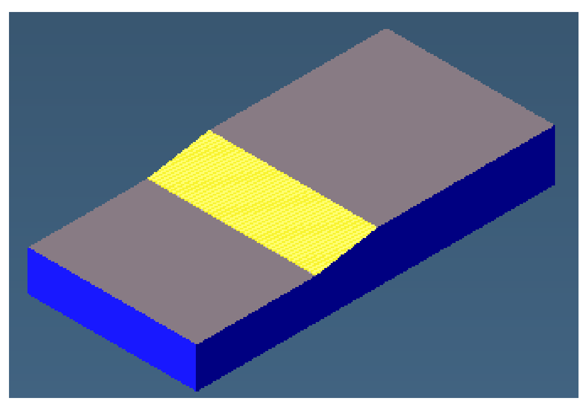
1. Create this Surfacing process with the 9mm Ball EM (Tool #3).





2. In face selection mode, select the angled face of the Cap body.
3. Create the toolpath and render the operations.



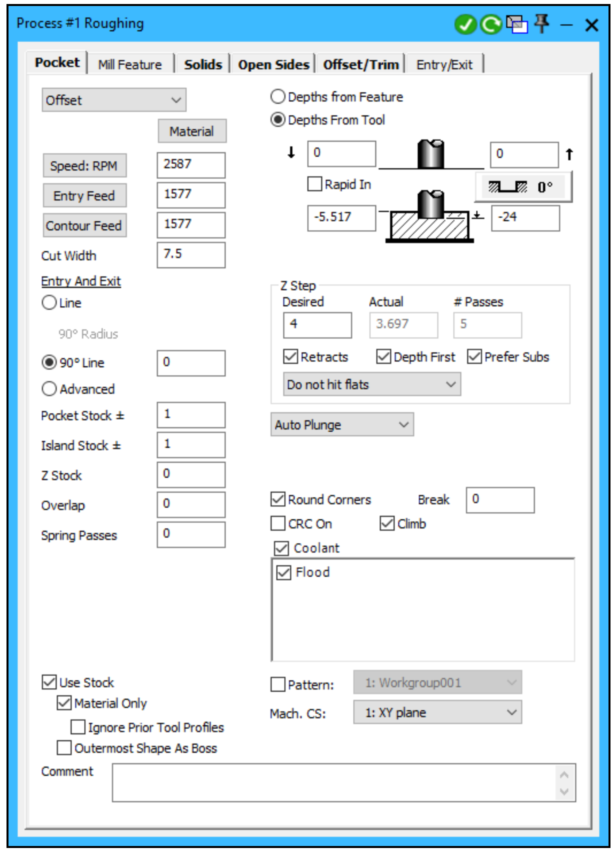


4. Place the Cavity Base body into the Body Bag as it will be used in a future operation.

#4: Roughing the Pocket

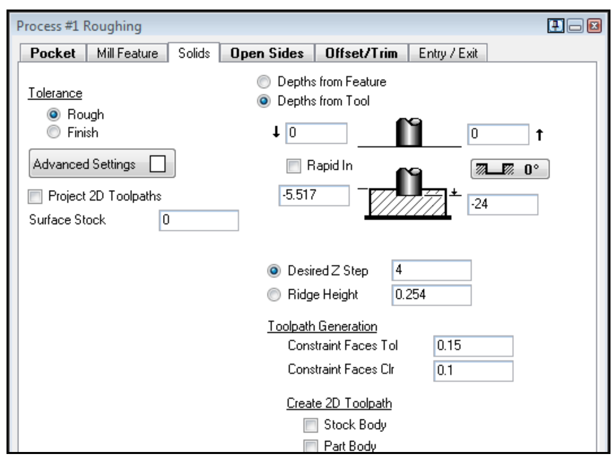
Now, we will rough out the cavity using a roughing operation.

1. Create this Roughing process with the 15mm Rough EM (Tool #2).

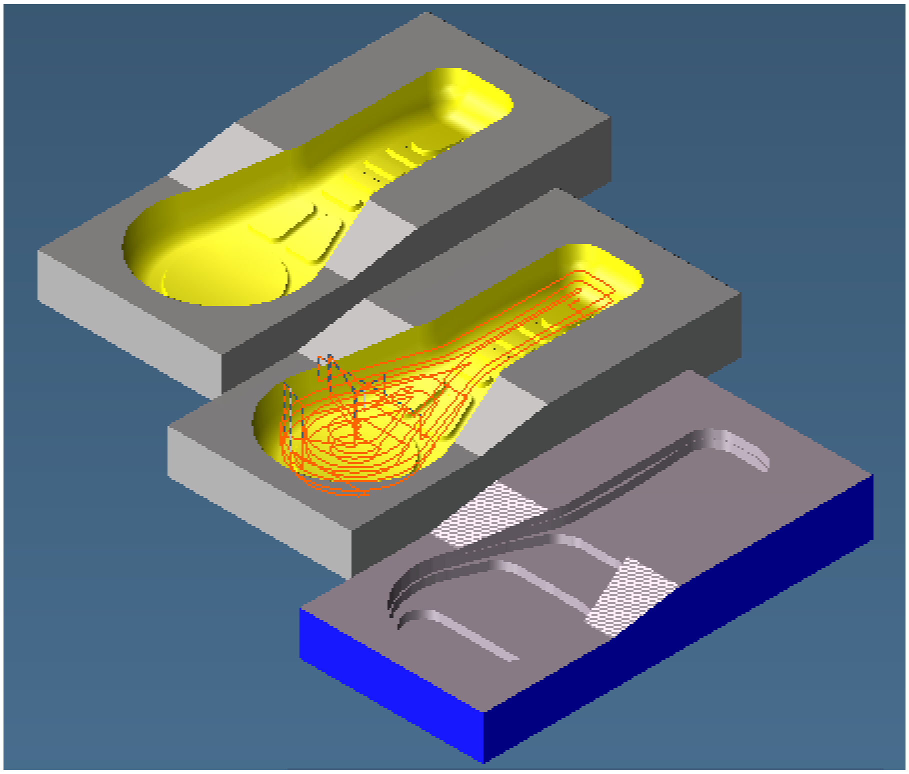


Notice that we have specified Pocket Stock and Island Stock in the Roughing process dialog. This will leave stock in X and Y only, not in Z. The Surface Stock specification, found in the Solids tab, will leave stock in all three dimensions.

2. Enter the following information in the Solids tab.



3. Select the faces that compose the inside of the cavity.



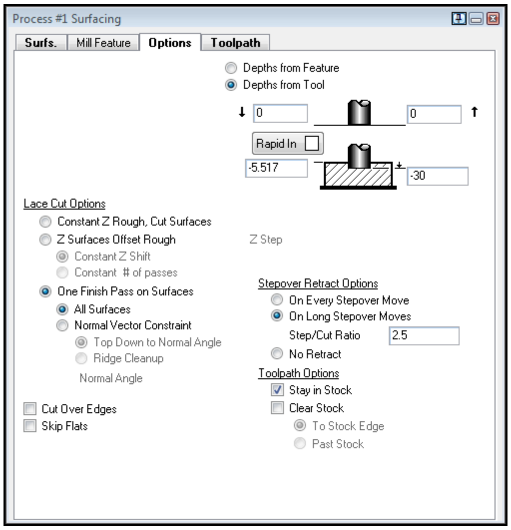
It may be easier to select the entire part model and then deselect the surfaces not to be included.

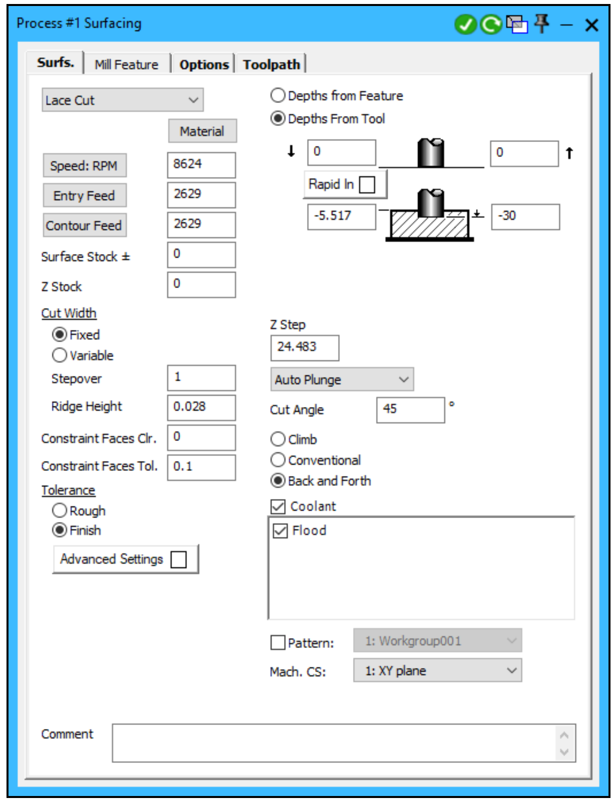
4. Create the toolpath.
5. Render the operation.

#5: Semi-Finishing the Pocket

We will now create a semi-finishing operation for the cavity.

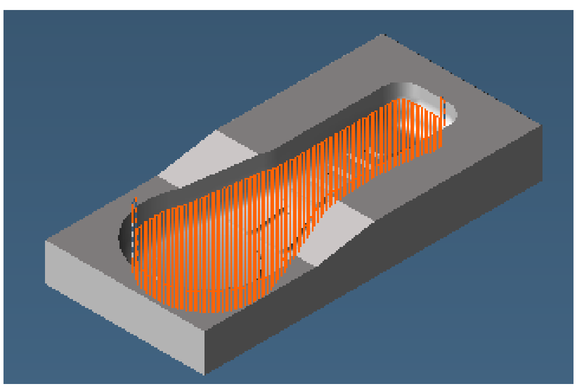
1. Create a Surfacing process with the 9mm Ball EM (Tool #3).

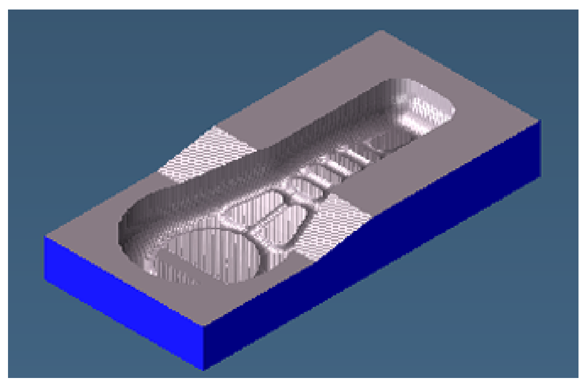




2. Select the faces that compose the inside of the cavity.

3. Create the toolpath and render the operation.

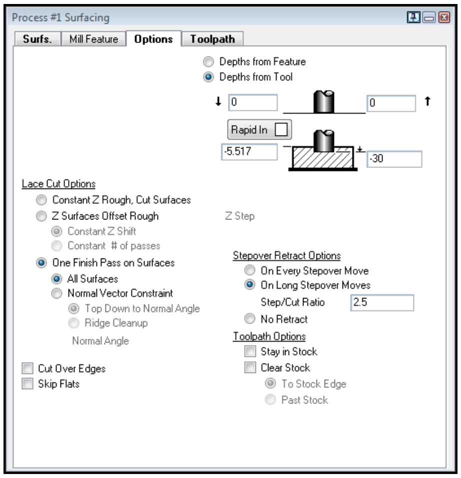


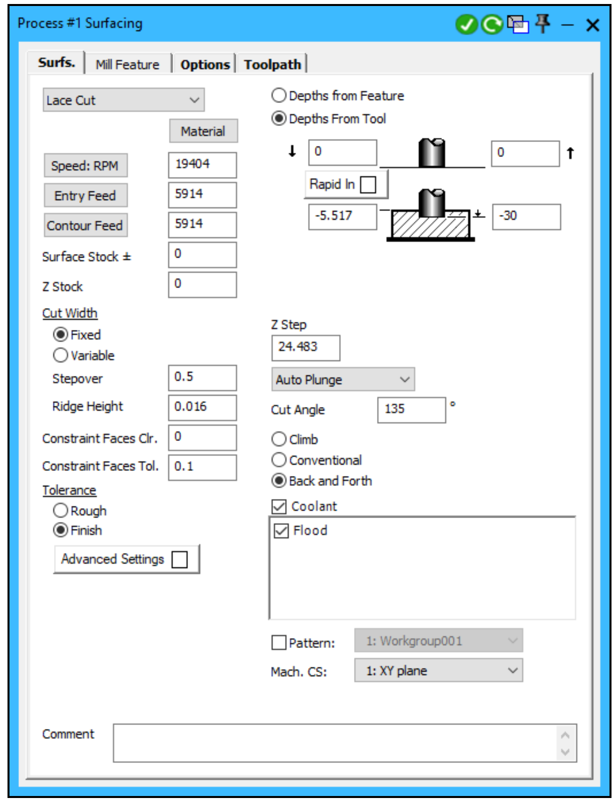


#6: Finishing the Pocket.

The next toolpath we will generate will finish the inside of the cavity. We will use the same selection as the last operation.

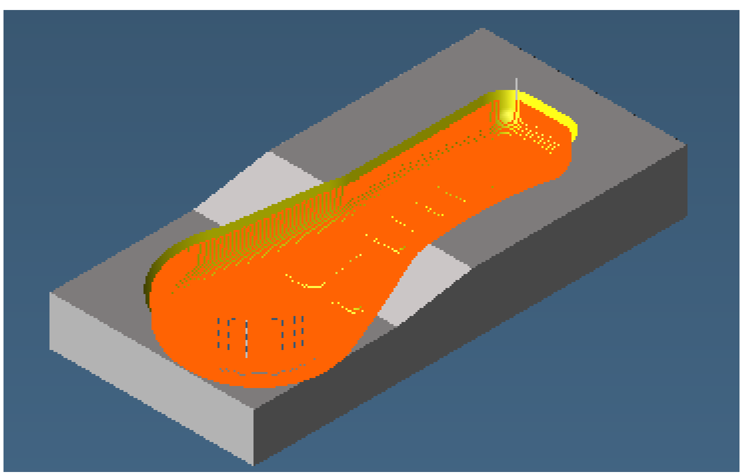
1. Create a Surfacing process with the 4mm Ball EM (Tool #4).





This operation will create a lace cut with a very small stepover amount to give us a nice smooth finish on the inside of the cavity.

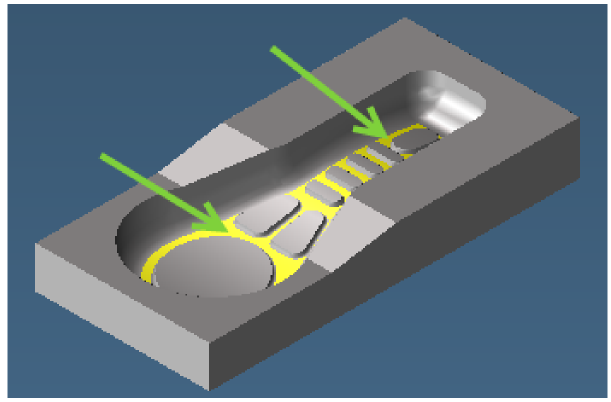
2. Create the toolpath.



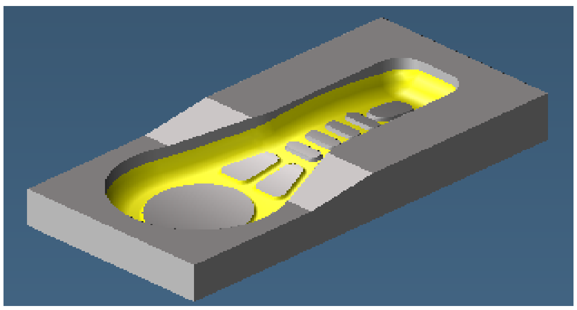
#7: The Buttons

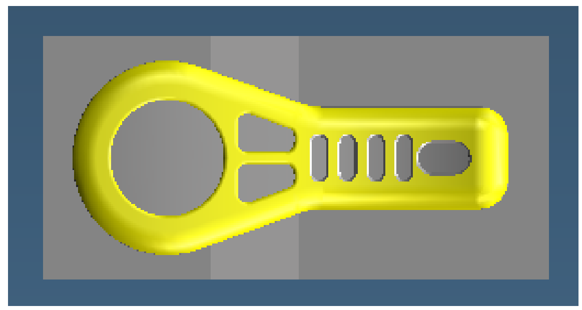
The final operation will finish around each of the buttons in the cavity. To accomplish this, we will use an intersection cut on the bottom faces of the cavity.

1. In face selection mode, select the two bottom faces of the cavity, `Right-click` the faces and choose Select>Select Tangent Faces.

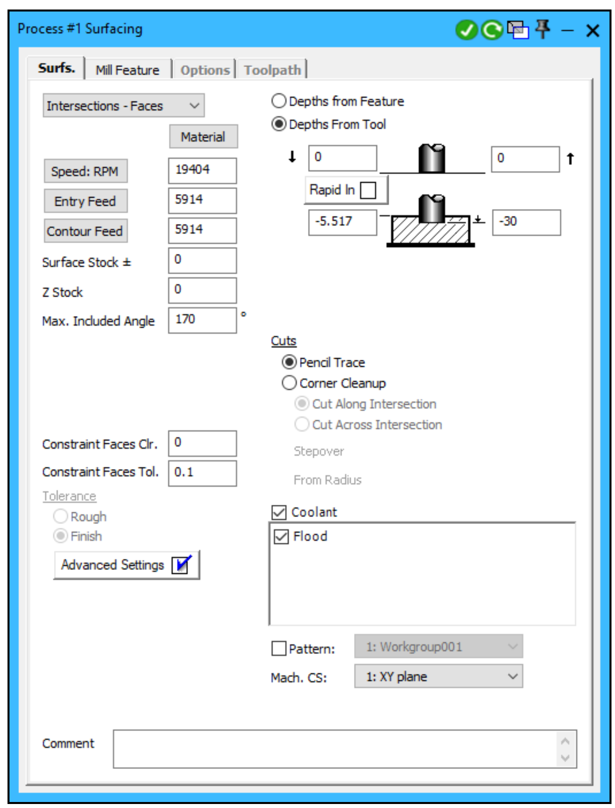


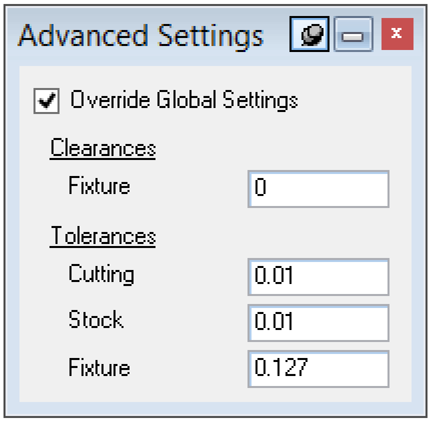
The resulting selected faces should be all of the faces in the cavity except for the vertical walls and the tops (and some edges) of the buttons, as seen below.





2. Create this Surfacing process with the 4mm Ball EM (Tool #4).

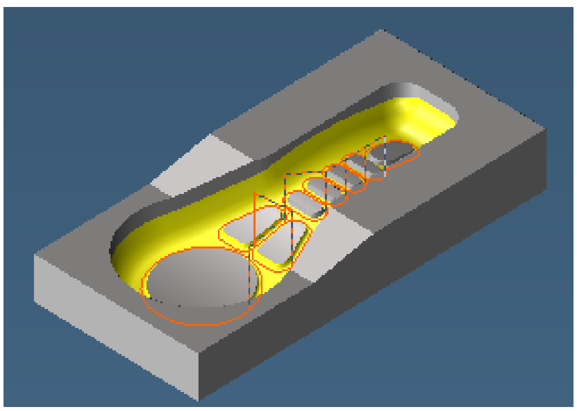


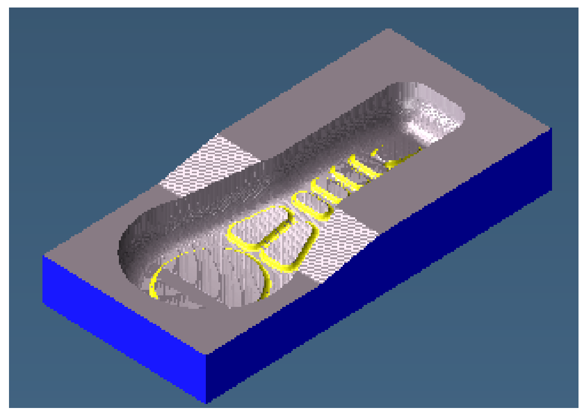


To get the toolpath to wrap around all of the button islands we need a very large included angle and we need to override the default tolerance settings. Click the Advanced Settings checkbox.

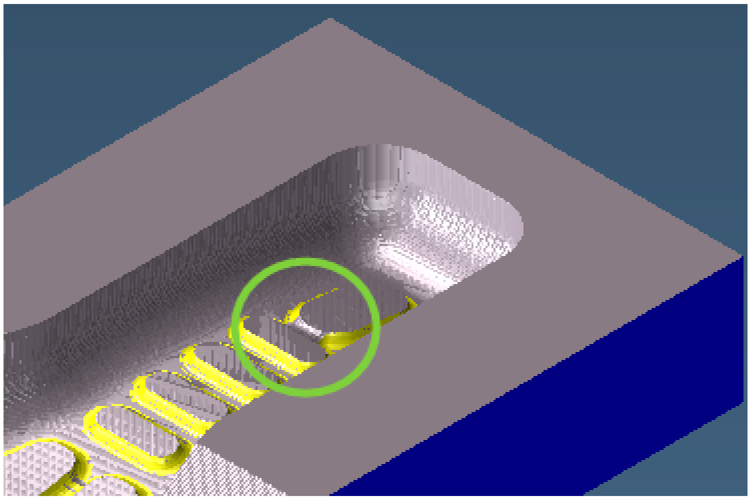
This process will create a single operation which finish around each of the button islands.

3. Create the toolpath and render the operation.





When rendered if you look at the results closely you will see that toolpath is not fully cutting around two of the shutoffs. This is because of the tool size and how close the shutoffs are. This area could be machined in several ways or could just be ground out. For this tutorial we won't worry about it, but there is a lesson in this – always check the rendered part before sending G-code to the control.



ADVANCED 3D EXERCISES

The part models for all of the exercises are supplied on the installation CD. We assume you are familiar with the interface and the principles used in Mill and Advanced CS.

All parts in these exercises are set to be made of cast aluminum alloys. The feeds and speeds are all system defaults. 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 you will be using the default material – stainless steel. The feeds and speeds may be set by clicking on the calculation buttons.

It is advisable that all rendering be done using Op Sim and that the Skip Unselected Ops  icon option is enabled (it will turn blue). These tutorials are designed to get you familiar with the functions, not to actually machine the part. As such, stepovers and tolerances will be loose to speed up processing and to allow you to more easily see the toolpath.

#1: The Mold Cavity

- Part Setup
- Roughing the Mold Cavity
- Finishing the Mold Cavity

Part Setup

In this exercise, we will explore the balance between toolpath processing time and toolpath quality. First, we will use Advanced 3D Machining to rough the part. Next, we will see various strategies of roughing and finishing using Advanced 3D Machining. Finally, we will use simple contouring to finish the part.

1. Open the Adv3D#1 Mold Cavity.vnc part supplied on the installation CD.
2. Verify the following tool list.

#	Type	Length	Flute Length	Diameter	Bottom Radius	# Flutes
1	Rough EM	3.0	1.75	1.0	0.015625	3
2	Rough EM	2.5	1.5	0.5	0.015625	3
3	Finish EM	2.5	1.5	0.375	0.015625	3

Roughing the Mold Cavity

The sample part to be used for this tutorial is Adv3d#1Mold Cavity.vnc and can be found in the sample parts folder. Completed files are also supplied in order to demonstrate the various strategies employed.

The Steps

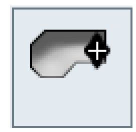
To rough out the cavity we will use several methods and strategies. We will start with a simple Advanced 3D Roughing technique. From there we will expand the number of processes and techniques to produce different results.

- Strategy A will add a second process to help to rough machining areas that were missed.
- Strategy B will increase the amount of processing time but reduce the amount of toolpath by doing Rest Material operations.
- Strategy C (Operations #3-4) will decrease the amount of processing time but increase the amount of toolpath by using a Bounding Box definition.
- Strategy D (Operations #5-7) will use either Strategy B or C to semi-finish the part.

Operation #1: Roughing

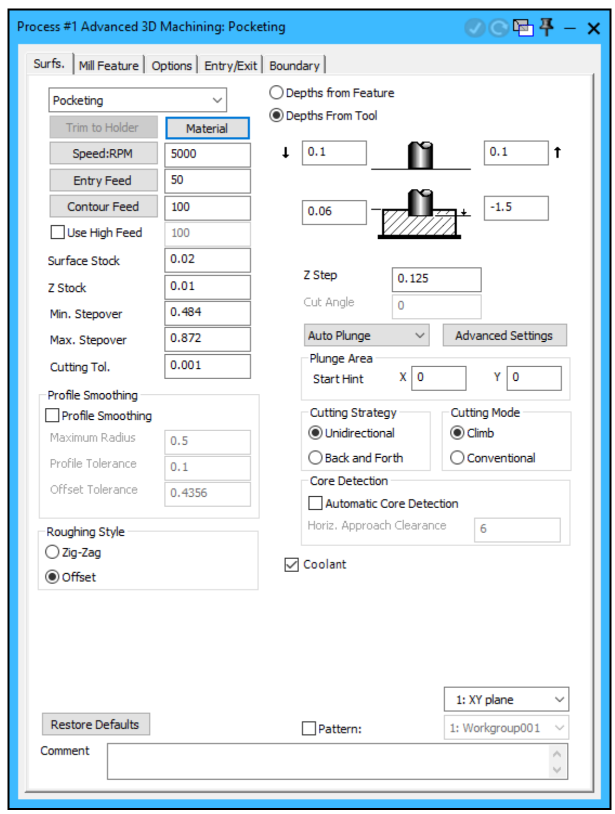
First, we will create an Operation to rough the cavity.

1. Create a new Advanced 3D Machining Process with the 1.0 Rough EM (Tool #1).



For now, we will use mostly default values.

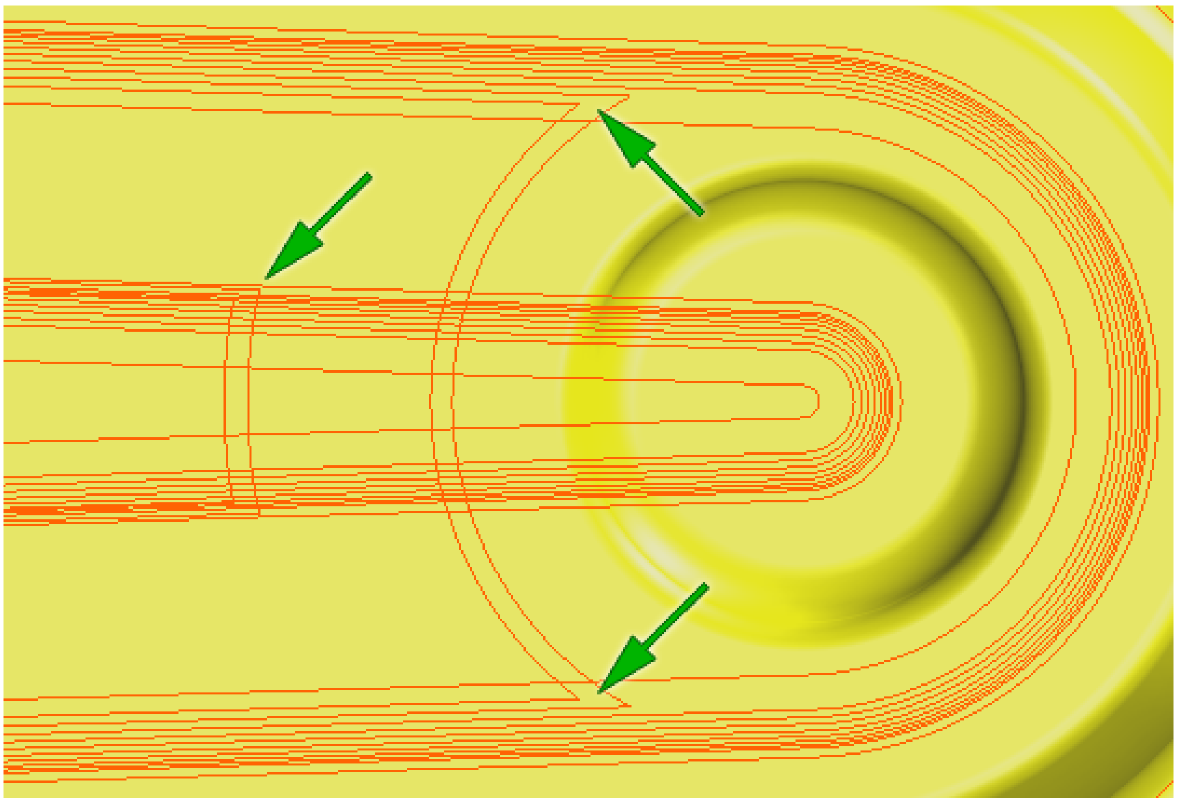
2. Click the Restore Defaults button, and then enter values as shown. The Advanced Settings do not need to be changed.



Clicking the Restore Defaults button ensures that we are going to get correct automatic values, particularly for the stepovers and cutting tolerance, based on our tool size as we enter data. The Min. Stepover and Max. Stepover values shown are 50 percent and 90 percent of the tool's flat bottom diameter, and Z-1.5 is the cavity floor depth.

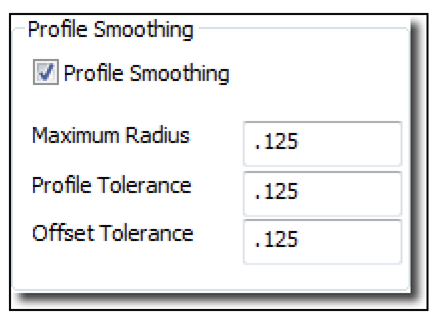
3. Close the Process dialog.
4. Select all (`Ctrl+A`) faces of the part body.
5. `Click` the Do It button to generate Operation #1.

   In the Top view, if you zoom in on the area near the right-side boss as shown, you will notice sharp corners. Advanced 3D lets you smooth those corners for high speed machining. We will adjust the process settings to add corner rounding.



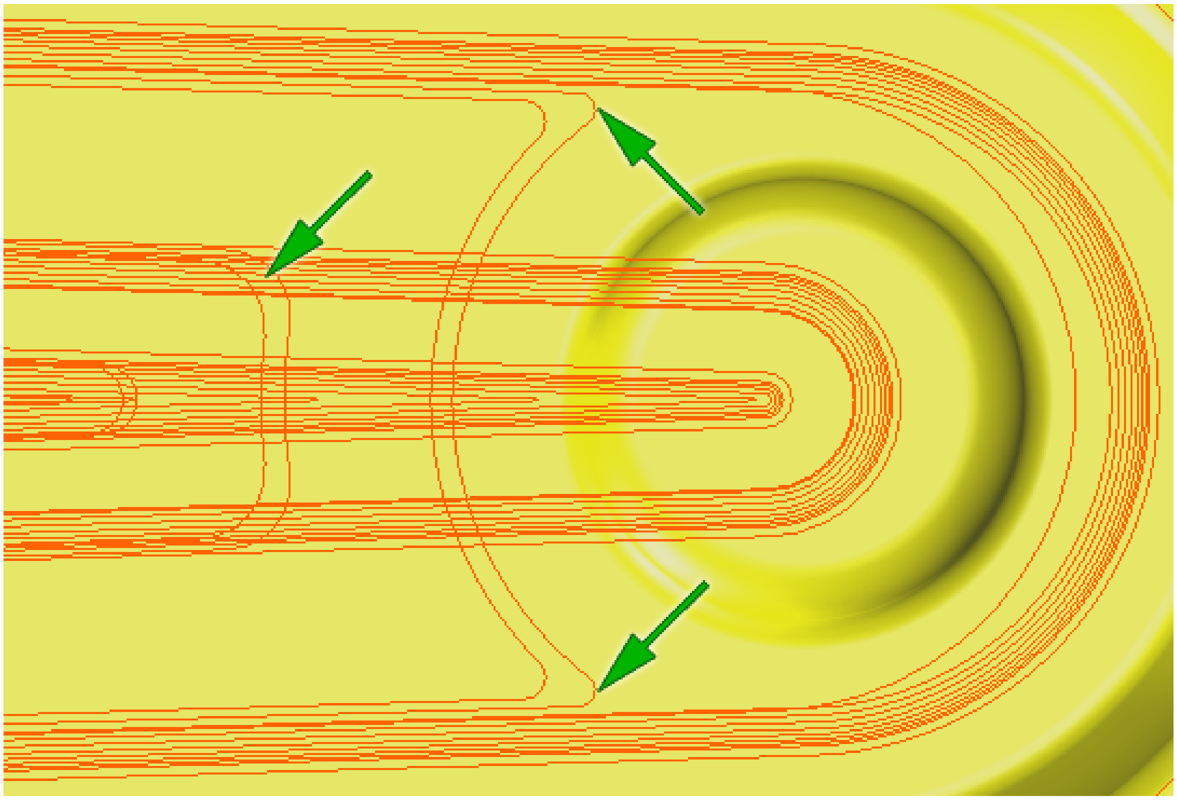
An example of Sharp Corners in the toolpath

6. Open the Process.
7. Select Profile Smoothing and enter values shown to the right.



8. `Click` Redo to recalculate the toolpath.

Notice in the area near the right-side boss that the formerly sharp corners are now rounded, as shown.



An example of Rounded Corners in the toolpath.

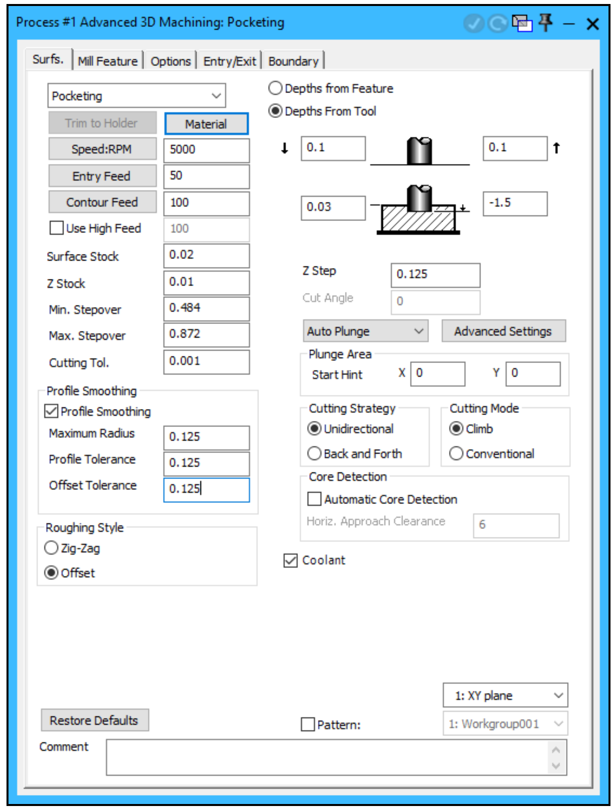
9. Render the part.

   Notice that material remains near the right-side boss. This is because the 1.0 tool diameter is too large to fit between the boss and the cavity wall. To correct this, we will use a smaller-diameter tool, experimenting with several strategies.

Roughing, Strategy A

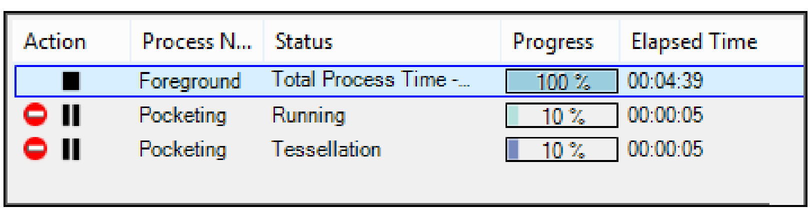
1. Leave the Process tile for Operation #1 on the process list as we will use this again.
2. Create a new Adv3D Operation using the 0.5 Rough EM (Tool #2) onto the new Process.

3. Enter the values as shown for the Surfs tab. The Advanced Settings do not need to be changed.



4. Close Process #2 and ensure that the part is still selected.

5. Ensure that the Operation #1 tile is still selected and then click Redo to recalculate the toolpath.

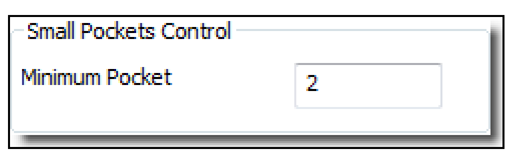


If your computer has multi-core processors, notice that both processes are calculating simultaneously, as shown. Keep the Task Manager open for later comparison, by clicking the pin symbol.

6. Select Operation #2 and redraw (Ctrl+R) or (View> Redraw) it.

Notice that toolpath is now calculated inside the four through holes and in the corners of the part.

7. Open Process #2, click the Options tab, and enter the value 2.0 for Minimum Pocket, as shown.



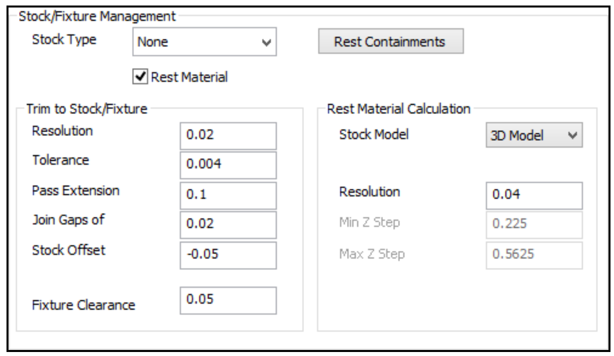
8. Close Process #2 and ensure that the part is still selected.
9. Ensure that both Operation #1 and #2 tiles are highlighted and then click Redo to recalculate the toolpath.

   Notice that there is no longer any toolpath for the four through holes and in the corners of the part. This is because the holes are 2.0 in diameter and Small Pockets Control has not allowed the toolpath engine to machine any pocket equal to or smaller than 2.

   Operation #2 is “cutting air” in the cavity. Because this is undesirable, we will further adjust Process #2, experimenting with two different Boundary / Stock Management strategies.

Roughing Strategy B - Rest Material

1. Open Process #2, Boundary tab, and enter the values as shown, with Stock ManagementRest Material checkbox enabled.



   Using a Pass Extension value of 0.1 forces the tool to start cutting 0.1 away from the rest material. Using a Stock Offset value of -0.05 forces the tool to not cut anything with less than 0.05 remaining (rest) material. Using the Stock Model option 3D Model forces the tool to look primarily at remaining (rest) material in the Z-axis.

2. Close Process #2.

   If the 3D Task Manager is still open, take note of the Elapsed Time of Operation #2 from the previous toolpath calculation. Strategy B (Rest Material), will require more processor time to analyze the toolpath. Also, during the calculation, the status of Operation #2 will be Blocked. This is because Operation #2 must now wait for the preceding operation to complete before it can begin to analyze the Rest Material condition.

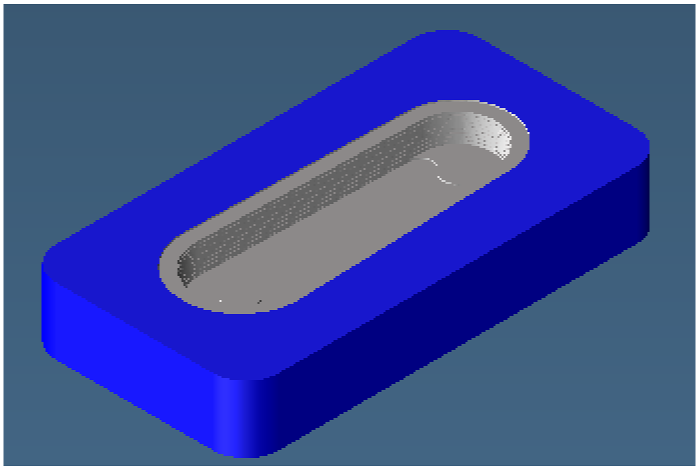
3. Click Redo to recalculate the toolpath.

   In the 3D Task Manager, notice that the Elapsed Time is greater than for Strategy A.

4. Change to Isometric View (Ctrl+I), click Operation #2 and redraw (Ctrl+R) it.

   Notice the toolpath has been automatically trimmed to the Rest Material that Operation #1 did not cut.

5. Render the results.



Notice the superior results compared to Strategy A alone, but also the greater elapsed time.

Action	Process N...	Status	Progress	Elapsed Time
■	Foreground	Total Process Time -...	100 %	00:00:01
⛔ ■	Pocketing	Completed	100 %	00:00:05
⛔ ■	Pocketing	Completed	100 %	00:00:09

Strategy A

Action	Process N...	Status	Progress	Elapsed Time
■	Foreground	Total Process Time -...	100 %	00:00:00
⛔ ■	Pocketing	Completed	100 %	00:00:05
⛔ ■	Pocketing	Completed	100 %	00:00:05

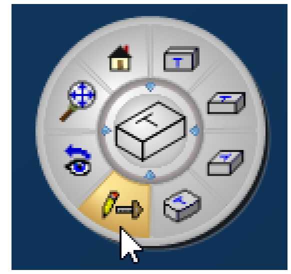
Strategy B

Roughing Strategy C - Bounding Box

1. Close the rendering palette.
2. `Double-click` Operation #2 to select both operations.
3. Open Process #1 and change the finish Z-depth value to `-1.25` (the boss tops).
4. Verify that the Z Step value is `0.125` and then close Process #1.
5. Open Process #2 and change the start Z-depth to `-1.25` (the boss tops) and the finish Z-depth to `-1.5` (the floor).
6. Verify that the Z Step value is `0.125`.
7. `Click` the Boundary tab and in Stock Management, Stock Type uncheck the Rest Material checkbox.
8. Close Process #2.
9. Select all (`Ctrl+A`) again and `click` Do It to calculate a new toolpath.

Notice in the Task Manager that Operations #3 and #4 are being created simultaneously, saving programming time. Process #2 may finish calculating before Process #1.

10. Select Process #3 and redraw it.



Notice that the toolpath stops above the boss tops.

11. Select Process #4 and redraw it.

Notice that the toolpath begins at the boss tops and machines down to the cavity floor. However, it also machines 100% of the floor, requiring more machining time.

12. Render the toolpath.

If you interrogate the depths of the floor, boss tops, and top flat (you can use the plug-in named Show Position), the values are not Z-1.5, Z-1.25, and Z-0.1. The following processes, strategy D, correct this.

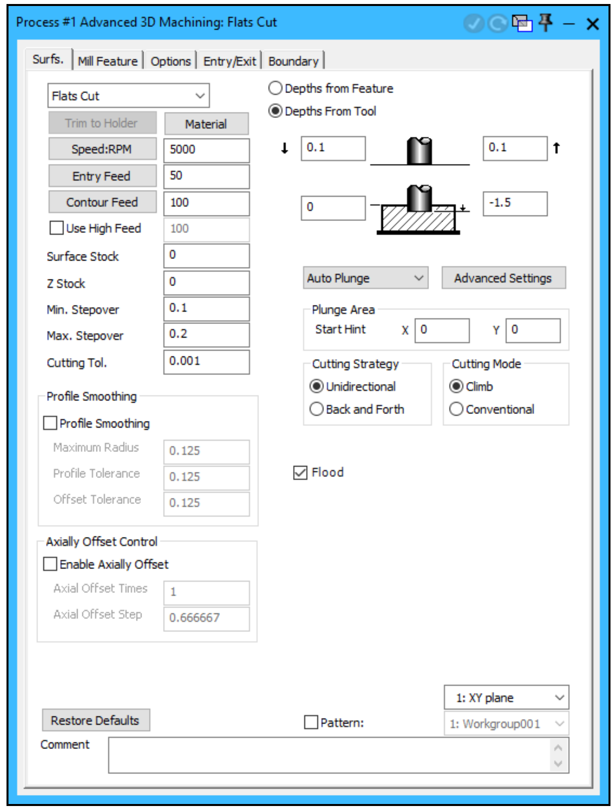
Semi-Finishing Via Flats Cut, Strategy D

1. `Double-click` roughing Operations #1 and #2.

Alternatively, if you preferred Strategy C to Strategy B, you could select roughing Operations #3 and #4. Strategy B (#1-2) used Rest Material to reduce the amount of toolpath. Strategy C (#3-4) used Bounding Box to decrease the processing time for dual-core or quad-core processors.

2. Create a new Advanced 3D Machining Process #3 with Tool #3.

3. Change machining strategy to Flats Cut and enter values as shown.



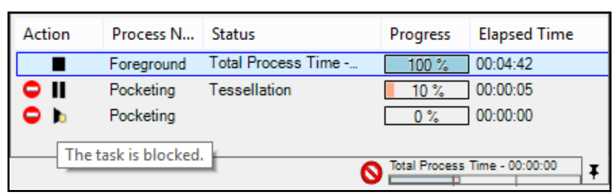
4. Close Process #3.

5. Click Do It to calculate the toolpath.

Let's look at the Task Manager for a moment, specifically the Status column. What it displays will depend upon which strategy you are using and also your computer's central processor unit (CPU).

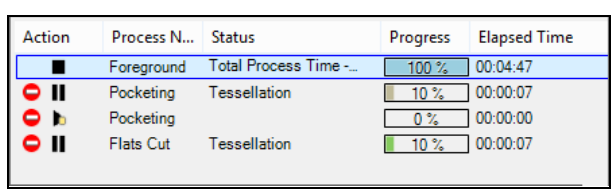
Strategy B

If you are calculating the toolpath using strategy B (Ops #1-2), you will see that Processes #1 and #2 are running simultaneously, but Process #3 is Blocked. This is because it is using Rest Material and it is dependent on the results of Operation #2 and thus cannot begin running until Process #2 is completed.



Strategy C, Dual Core Processor

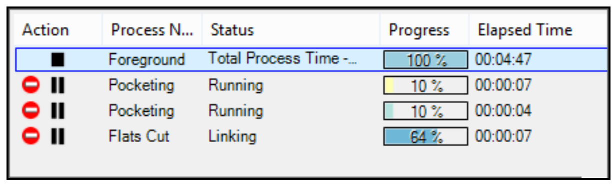
If you are calculating the toolpath using strategy C (Ops #3-4) on a machine with a dual-core processor you see that Processes #1



and #3 run simultaneously while Process #2 is Queued. This is because the computer has only two processors available. Process #2 must wait in a queue until one of the processors becomes available. Once Process #1 or #3 is complete, Process #2 begins running.

Strategy C, Quad Core Processor

If you are calculating the toolpath using strategy C (Ops #3-4) on a machine with a quad-core processor, you see that Processes #1, 2, and 3 all run simultaneously. With four processors available, and with no process dependent on another process, a quad-core machine is able to run all three processes simultaneously.



As a result of the calculation, new Operations #5-7 are created. However, the new toolpath has several problems, most notably the cutting on the top face of the part at Z-zero. Other problems are less noticeable because the toolpath is shown for all three Operations.

6. Select Operation #7 and redraw (`Ctrl+R`) it.

   Notice that the boss tops at Z-1.25 have not been cut. Also, it is clearly visible that Flats Cut is machining the top of the part at Z-zero, which is undesirable. First, we will correct only the boss tops at Z-1.25.

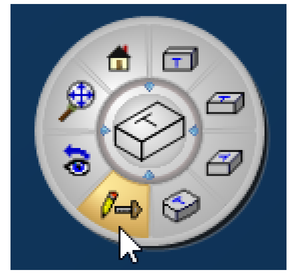
7. `Double-click` Operation #7 to select Operations #5-7.

8. Open Process #3, Options tab, and change the value of Minimum Pocket to `0.0`.

9. Close Process #3.

10. `Click` Redo and wait for the toolpath to recalculate.

11. Click Operation #7 and redraw it.



Notice that the toolpath now machines the boss tops. This is because the boss tops have diameter less than 2.0 but greater than zer0.

12. Render the toolpath.

If you use the Show Position plug-in to interrogate the depths of the floor, boss tops, you find the values are now correctly Z-1.5, Z-1.25, and Z-0.1.

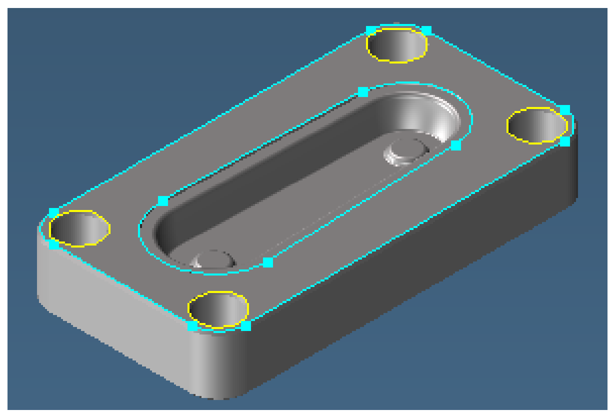
However, the toolpath is still machining the top of the part. We will correct this next.

Refining Toolpath, Strategy A - Selected Curves

We will correct the machining of the top of the part by containing the toolpath within the cavity area. We will experiment with two of the possible strategies.

1. Select the top face and extract the geometry.

   This yields a closed geometry loop around the top edge of the cavity.



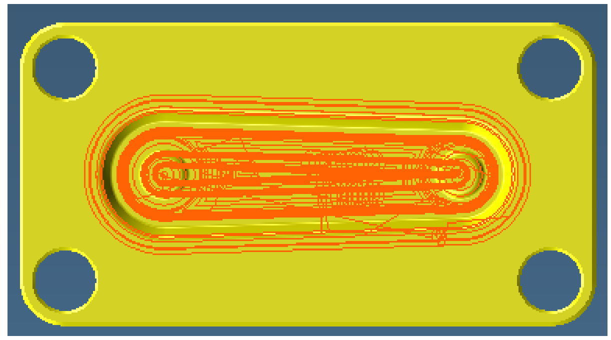
2. Double-click Operation #7 to select Operations #5-7.
3. (Ctrl+click) the closed geometry loop around the top edge of the cavity.
4. Open Process #1, Boundary tab.

   In Boundary Style, Boundary Type, notice that Selected Curves is active. If you now open Processes #2 and #3, you would notice that they automatically default to Selected Curves and are disabled. This is because all three processes in the group must use the same Selected Curves for the Boundary Type.

5. Close the process.

6. Click Do It to calculate a new toolpath.

   This creates three new Operations: #8, #9, #10. When the calculation has completed, notice that the toolpath for each operation is now contained within the geometry loop.



Alternatively, we can employ a different strategy, Silhouette, to refine the toolpath.

Refining Toolpath, Strategy B - Silhouette

1. Deselect the highlighted operations and then redraw.
2. Activate Face Selection.
3. Right-click the cavity floor (Z-1.5) and choose Select>Select Faces Above.
4. (Ctrl+click) the boss tops (Z-1.25).

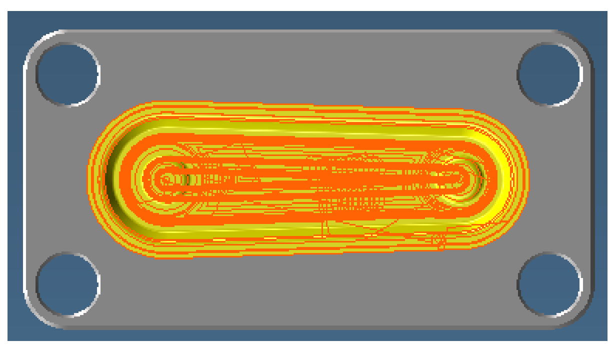
   At this point, all cavity faces are selected, but the faces outside the cavity are not selected and the geometry is not selected.

5. Open Process #1, Boundary tab, and in Boundary Style, Boundary Type, select Silhouette.

   If you were to open Processes #2 and #3 now, you would notice that they automatically default to Silhouette and are disabled. This is because all three processes in the group must use the same Silhouette for the Boundary Type.

6. Close the process.
7. Click Do It to calculate a new toolpath.

   This creates three new Operations: #11, #12, and #13. When the calculation has completed, notice that the toolpath is now contained within the silhouette boundary of the cavity.



It is now time to use the finishing tools. First we will use Contour to finish the main cavity. Then we will use several different optimization techniques.

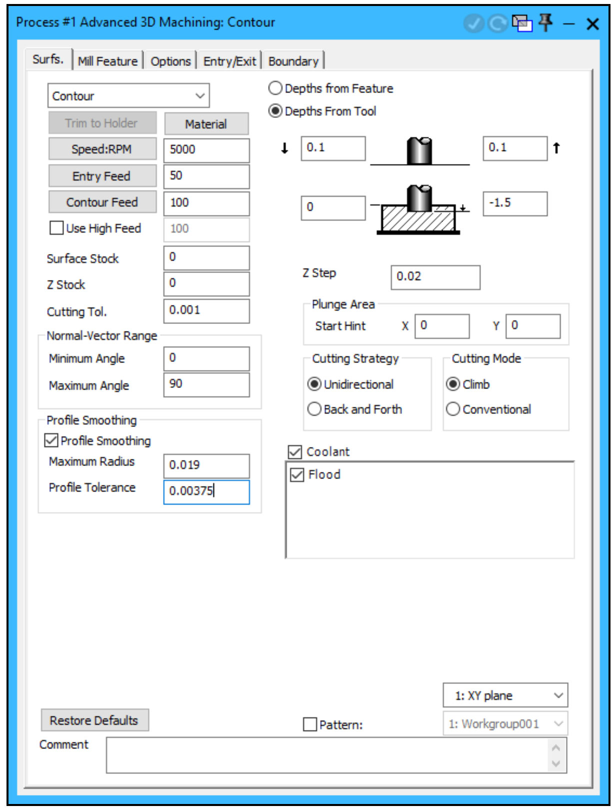
Finishing the Mold Cavity

Finishing and Optimizing

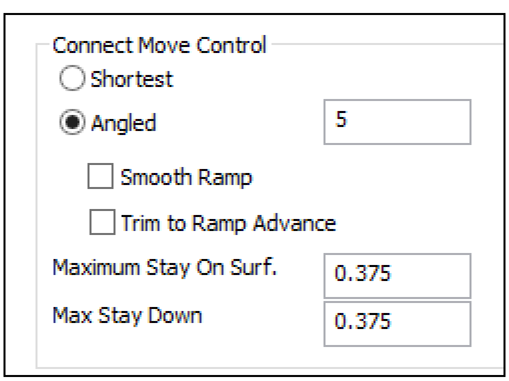
1. Delete the three existing processes.
2. Create a new Advanced 3D Machining process with the 2.5 Rough EM (Tool #3).
3. Change the machining strategy to Contour.

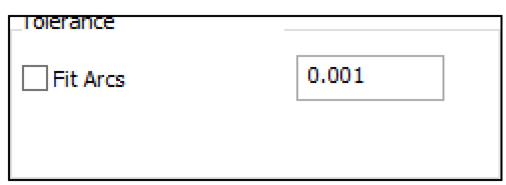
For now, we will use mostly default values.

4. Click Restore Defaults, and then enter values as shown.

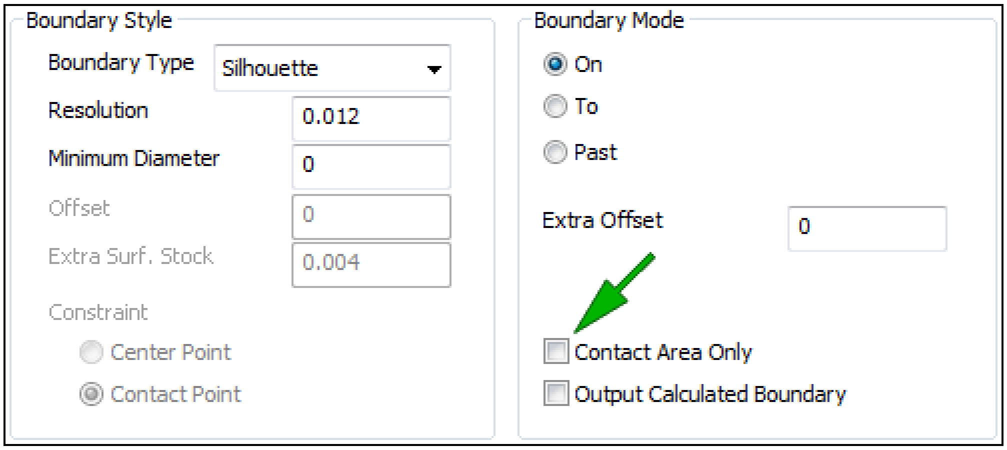


5. Click the Options tab, and then make changes to Connect Move Control and Point Reduction as shown.



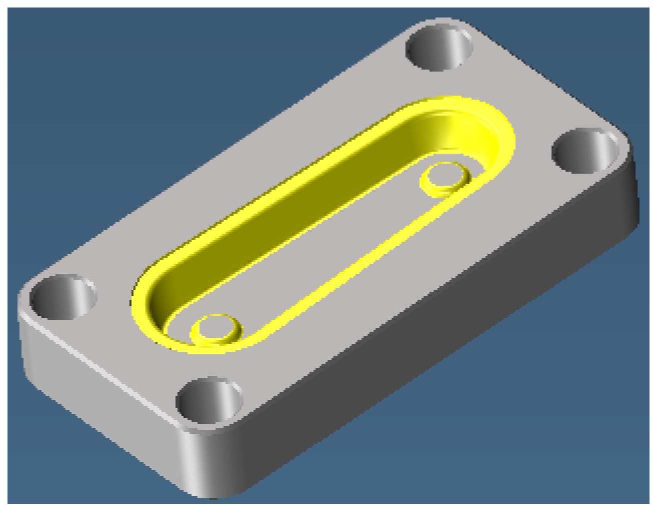


6. Click the Boundary tab, and then make changes to Boundary Style and Boundary Mode as shown.



7. Close the process.

8. Select the faces to be machined as shown.



Be sure to select both bosses. Each boss has two small radii (top and bottom) and a small wall between the radii. This can be accomplished using the selection we used earlier and deselecting the floor.

9. Click Do It to calculate a new toolpath and generate Operation #14.

In this new Operation (#14), you will notice several problems, such as air-cutting moves just above the edge of each silhouette boundary.

In the following steps, we will optimize the toolpath to remove the air-cuts and make a better quality surface finish. We will make one change at a time to see the resulting changes individually.

Optimization, Step 1

1. Double-click Operation #14 to select the silhouette faces.

2. Open the process, Boundary tab, and select the checkbox for Contact Area Only.

3. Close the process, and then click Redo.

   After the calculation has completed, notice that the toolpath is no longer making air-cuts. This is because of the Contact Area Only option, which causes the toolpath engine to treat Silhouette as a 3D-type boundary that recognizes and trims the 3D material. (In contrast, when Contact Area Only was disabled, the toolpath engine treated Silhouette as a 2D-type boundary that did not use 3D material recognition.)

Optimization, Step 2A

1. Double-click Operation #14 to select the silhouette faces.
2. Open the process, Boundary tab, and enter 0.1875 as the value for Extra Offset.
3. Close the process and then click Redo.

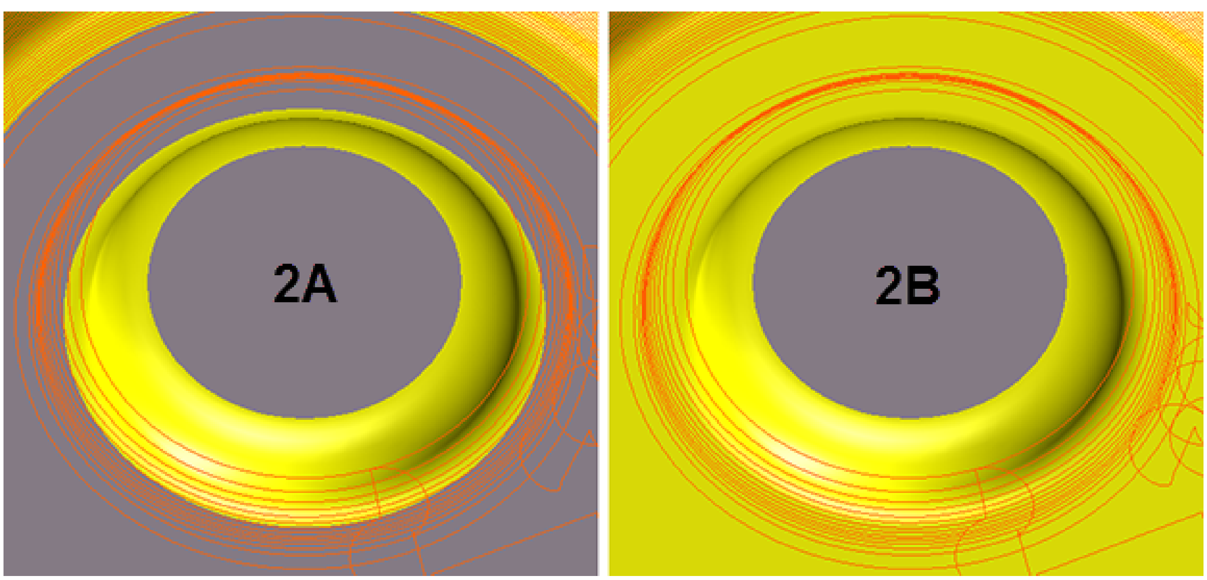
   After the calculation has completed, notice that the toolpath now extends an extra amount (the tool's radius) past the edges of the selected faces.

Optimization, Step 2B

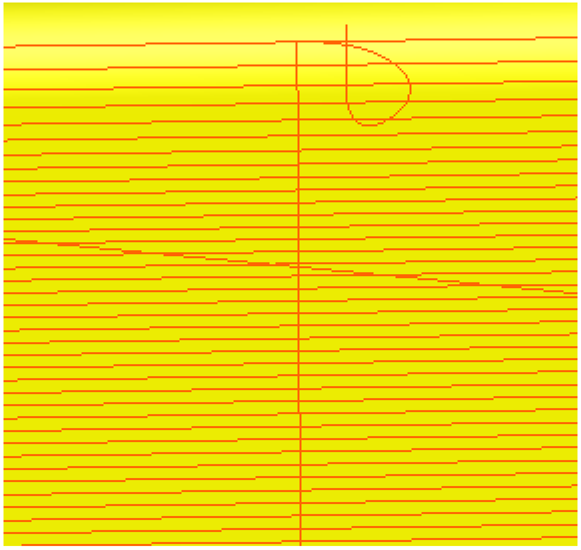
1. Double-click Operation #14 to select the silhouette faces.
2. Open the process, Boundary tab, and enter 0.0 as the value for Extra Offset.
3. Close the process.
4. (Ctrl+click) the cavity floor and then click Redo.

   After the calculation has completed, notice that the toolpath now extends an extra amount (the tool's radius) past the edges of the selected faces.

   In these examples, both 2A and 2B produced the desired results with virtually identical results. However, on other parts, one strategy might work better than the other.



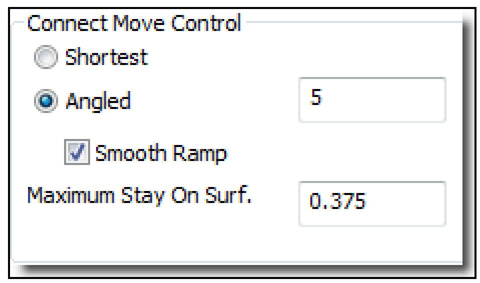
Optimization, Step 3



Find the lead-in helix on the pocket wall and look closely at the connection moves, as shown. Notice that the connect moves are vertical. In this step, we will change the connect method, to produce a better surface finish with reduced tool pressure and tool wear.

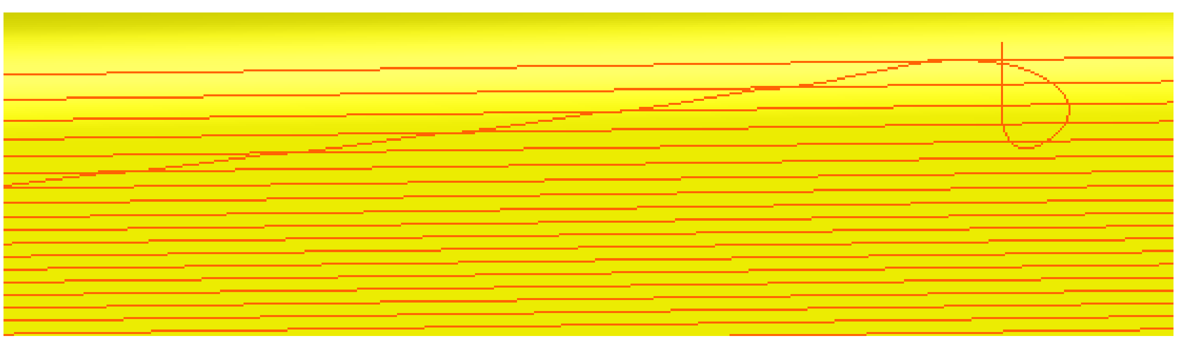
1. Double-click Operation #14 to select the silhouette faces.

2. Open the process, Options tab, and make changes as shown below.



3. Close the process and then click Redo.

   After the calculation has completed, notice that the toolpath connect moves are now angled, with a smooth ramp from one Z-level to the next, as shown.



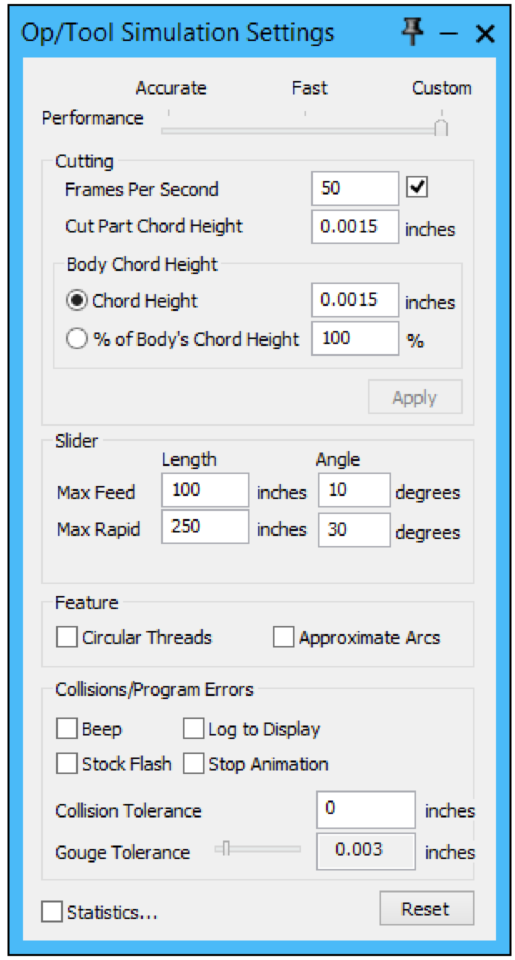
4. Save the file.

5. Select Operations #11-14, and then render the toolpath using Op Sim.

6. Right-click the title bar of Op Sim and make sure Skip Unselected Ops is checked.

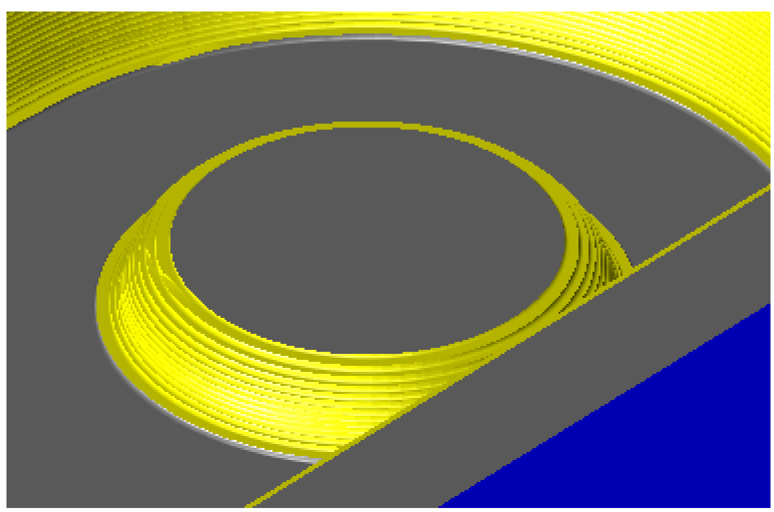
7. Right-click the title bar of Op Sim and choose Settings.

8. Enter values as shown.



9. Close the Settings dialog and then run Op Sim.

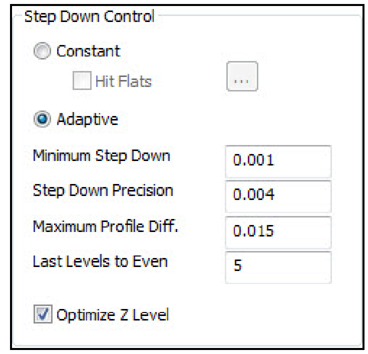
   After the rendering has completed, mouse-zoom from an isometric view so that you can see the top radius and the floor and boss radii. Notice that the floor has some material remaining and also notice that the top radius is rough-looking, with large steps. We will fix both problems in the next step.



Optimization, Step 4

1. Double-click Operation #14 to select the silhouette faces.

2. Open the process, Options tab, and make changes as shown.



In the Step Down Control area, the Adaptive option is selected, and the step down values are 0.001, 0.004, and 0.015.

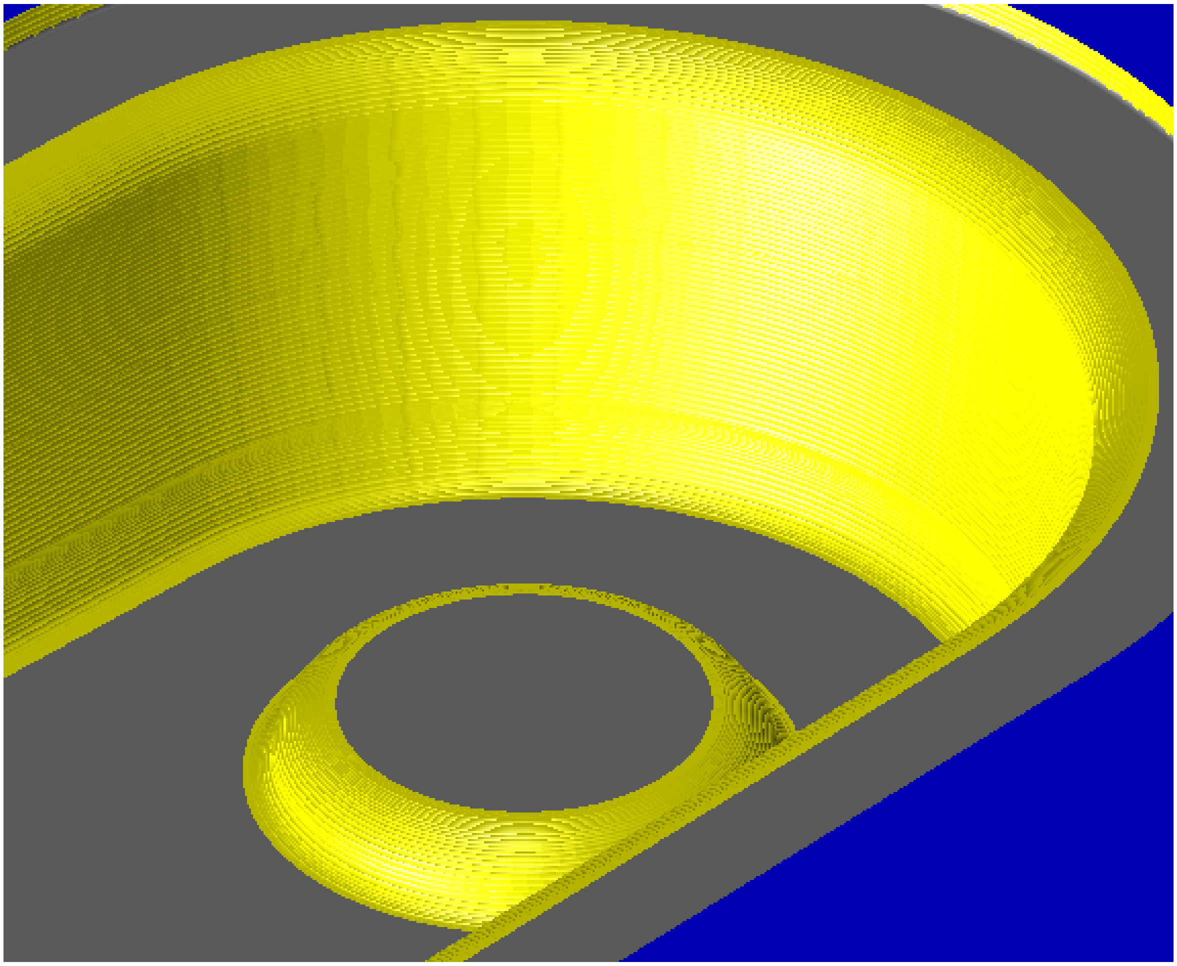
3. Close the process, and then click Redo.

After the calculation has completed, notice that the toolpath has many more step-down passes at the fillets.

4. Save the file.

5. Select all four of Operations #11-14, and then render the toolpath using Op Sim.

After the rendering has completed, mouse-zoom from an isometric view so that you can see the top radius and the floor and boss radii. Notice that there is no material remaining at the floor and that the top radius is smooth-looking, with small steps.



6. Save the file.

#2: The Food Bin

In this exercise, we will explore five different types of toolpath for a simple mold core:

1) Tool Holder Goug Checker

2) Flats Cut

3) Lace Cut

4) Toolpath Splitter; and

5) N-Curve Flow

The goal of this tutorial is to familiarize you with various toolpath options.

Part Setup

1. Open the Adv3D#2 Food Bin.vnc part supplied on the GibbsCAM installation CD.
2. Verify the following tool list.

#	Type	Length	Flute Length	Diameter	Bottom Radius	Length Out of Holder
1	Rough EM	100	11	25	1.0	12
2	Rough EM	100	22	25	1.0	23
3	Rough EM	100	32	25	1.0	33
4	Finish EM	100	32	25	0	33
5	Finish EM	50	25	6	.75	27.5
6	Finish EM	50	25	6	.75	27.5
7	Finish EM	50	25	6	0	27.5
8	Ball EM	75	28	6	n/a	30
9	Ball EM	75	28	6	n/a	30
10	Ball EM	50	15	6	n/a	20

Machining the Food Bin

#1-4: Roughing

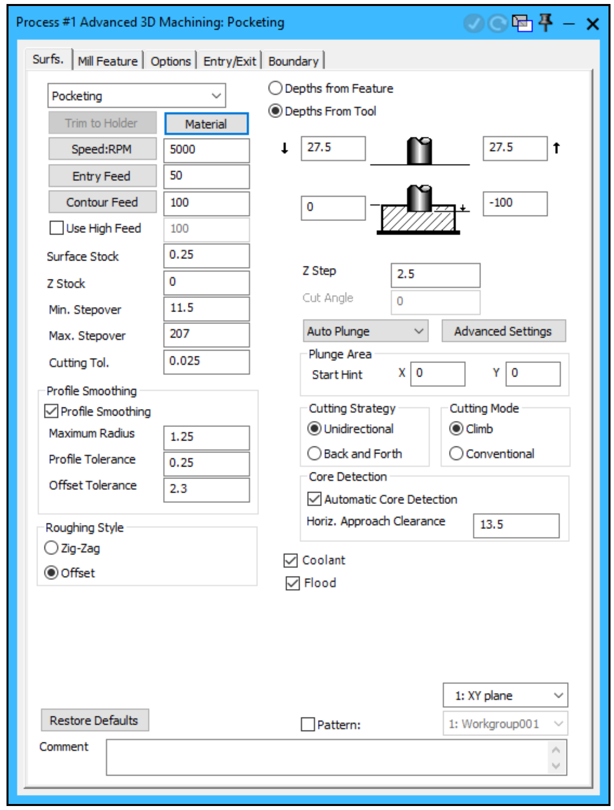
For roughing the part we will use Tools #1-4, the 25mm end mills. In Processes #1-3, we rough the part with progressively longer lengths out of the holder. Then in Process #4 we will use Tool #4 to make a Flats Cut.

1. (`Click-Shift+select`) Tools #1-4 and drag them to the process list.

   The four tool tiles populate the first four list positions.

2. (`Double-click` Process Tile #1 and choose Advanced 3D Machining.
3. `Click` Restore Defaults.
4. Verify that the Operation Type is set to Pocketing.

5. Enter values for the Surfs tab as shown.



6. Click the Entry/Exit tab.

7. Change the Retract Style to Minimal Vertical, and set the value of Clear Surface By to 10.

This will cause the tool to maintain a distance of at least 10mm above the surface of the part when making rapid links.

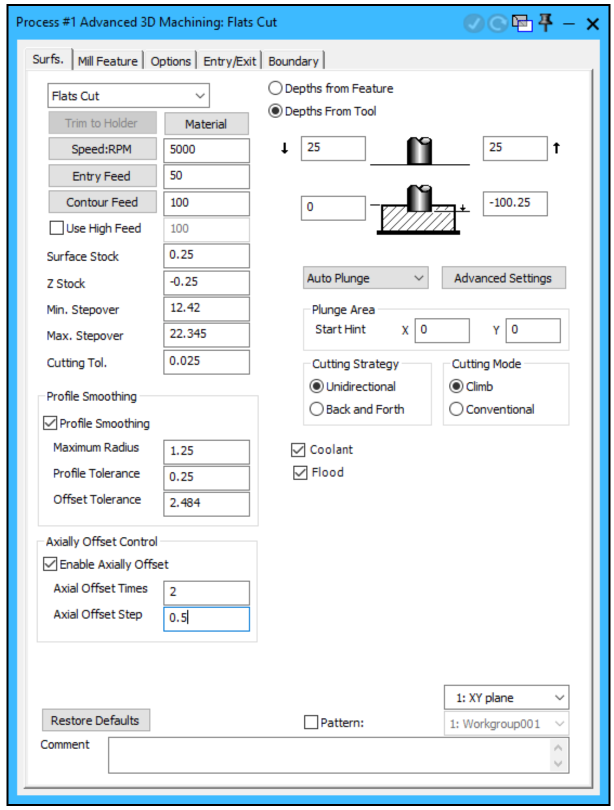
8. Close Process #1.

9. (Double-click Process Tile #2 and choose Advanced 3D Machining. Use the same settings as in Process #1.

10. (Double-click Process Tile #3 and choose Advanced 3D Machining. Again, use the same settings.

11. (Double-click Process Tile #4 and choose Advanced 3D Machining. Change the Operation Type to Flats Cut.

12. Click Restore Defaults and then enter values for the Surfs tab as shown.



You will make a Flats Cut to finish the floor at Z-31.75mm. You set the Surface Stock value to 0.25 to prevent the tool from touching the core's walls. However, because this prevents the tool from cutting the floor, you set the Z Stock value to -0.25, which will shift the tool down to the floor depth.

Also, since Operation #3 will leave approximately 1.75mm of material on the floor, the Axially Offset Control is used to make multiple Z-steps. In this example, the Axially Offset Times value of 2 will add two additional Z-steps above the final Z-step, for a total of three Z-steps. The Axially Offset Step value of 0.5mm will make the first pass 0.5mm above the finish floor depth, leaving that amount for the finish pass at Z-31.75.

13. Click the Options tab, change the Smooth Radius value to 12.5, and change the Maximum Stay On Surf value to 320.

Setting the Smooth Radius value to 12.5 will cause the connect moves from one stepover to the next to be blended with a 12.5-mm radius, allowing for higher feedrates. Setting the Maximum Stay On Surf value to 320 will cause the tool to cut the entire floor area without retracting until the end of each Z-step.

14. Close Process #4.

15. Select the solid (the entire part) and then click Do It to generate the operations.

After all four processes have run, you have Operations #1-4. We will use Op Sim in Op Stop mode to clearly see the results of each operation.

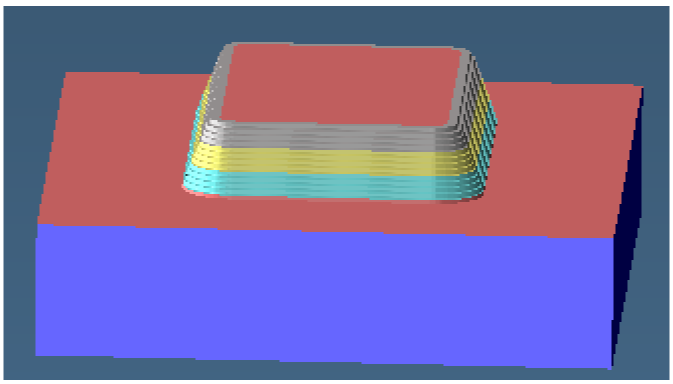
16. In the Top taskbar, ensure that Tool Holder Display is enabled.

17. Open Op Sim and then click Op Color mode button. 
18. Set the Op Color Mode option to Op Number.
19. Select the Stops option and set the Stop before Operation number to 2. 
20. Run Op Sim.

    Notice that Operation #1 cuts as close as possible to the core, but leaves material where the holder would gouge the part.

21. Resume the rendering.

    Operation #2 cuts remaining (rest) material that Operation #1 could not, and likewise with Operation #3.



22. Delete the four processes.
23. Save the interim part, which now has Operations #1-4.

#5-7: Roughing and Finishing

In this section we will use Tools #5-7 (6mm-diameter end mills) to rough and finish the lid area. Tool #5, a rough end mill with a 0.75mm tool nose radius, will rough the entire lid area. Tool #6, a finish end mill with a 0.75mm tool nose radius, will finish the areas that have fillet radii between the floor and wall. Tool #7 is a finish end mill without a tool nose radius (zero). We will build three processes for this group of operations.

1. (`Click-Shift+select`) Tools #5-7 and drag them to the process list.

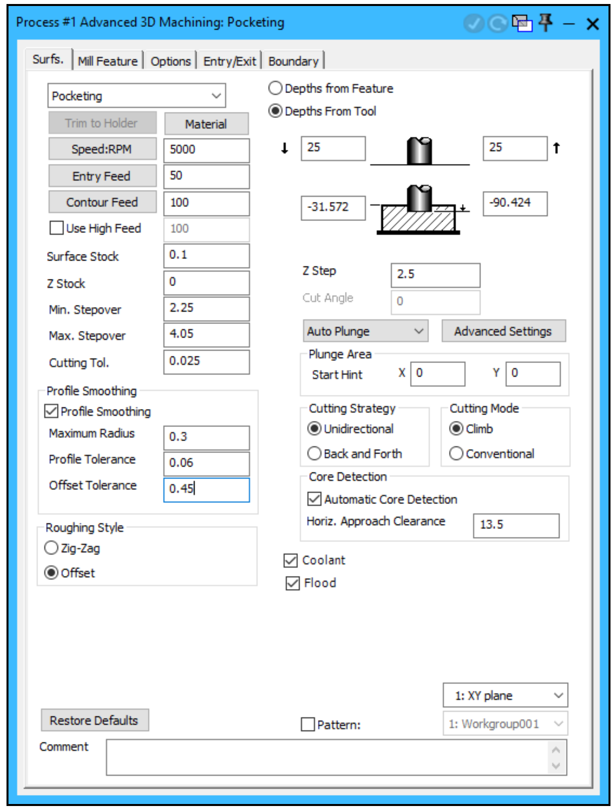
   The three tool tiles populate the first three process list positions.

2. (`Double-click` Process Tile #1 and choose Advanced 3D Machining.
3. Ensure that the Operation Type is set to Pocketing and then `click` Restore Defaults.
4. `Click` Advanced Settings and then select the option Profile Ramping.

   Profile Ramping causes all entrance moves to be done with a ramp move instead of a helix.

5. Close the Advanced Settings dialog box.

6. Enter values as shown for the Surfs tab.



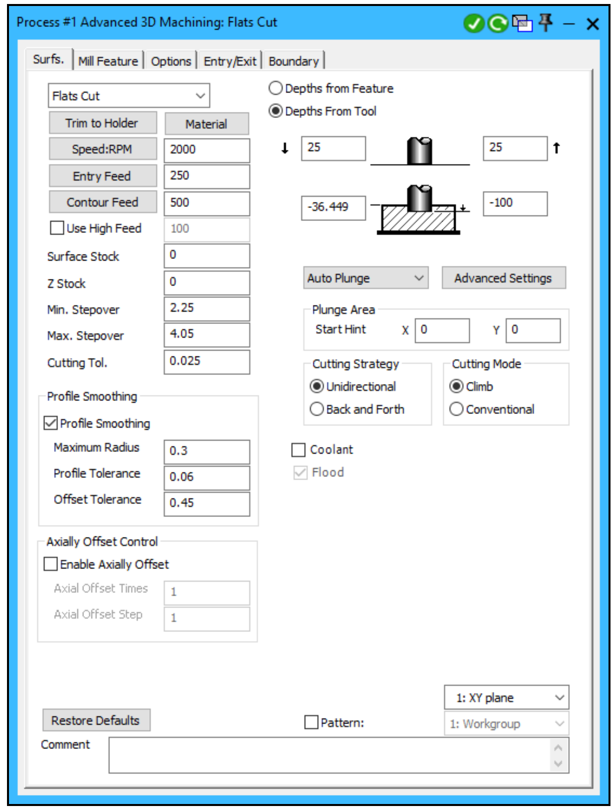
The Z start depth value at the top of the lid area allows you to select the entire solid, not faces or geometry for containment, and it does not require the use of Rest Material from previous operations. As a result, when this strategy is used with a group of processes, the calculation time is much faster by allowing the processes to calculate simultaneously. The benefits from using this strategy are the ease of model selection and the increased speed at which the processes are calculated.

7. Click the Entry/Exit tab and choose the Vertical option for the Entry and Exit styles.

8. Click the Boundary tab and ensure that Boundary Type is set to Part Bounding Box and that Stock Type is set to None.

9. Close Process #1 and (Double-click Process Tile #2 and choose Advanced 3D Machining.

10. Click the Surfs tab and enter values as shown.



This Flats Cut process will finish the floors and walls that have a 0.75mm radius. You set the Z start depth to -36.449, which is equal to the floor of the second step. This also allows you to select the entire solid, not faces or geometry for containment, and does not require the use of Rest Material from previous operations.

11. Ensure Operation Type is set to Flats Cut and then click Advanced Settings.

12. Select the option Profile Ramping.

13. Close the Advanced Settings dialog box.

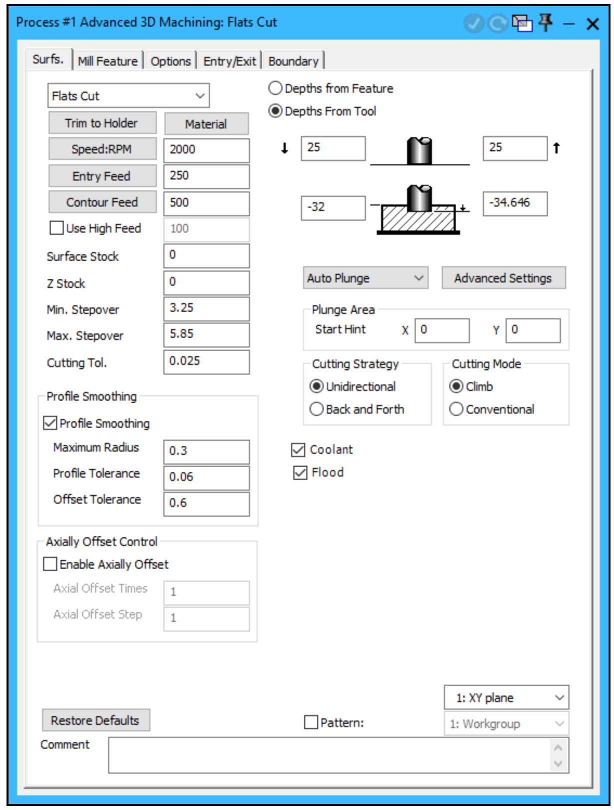
14. Click the Entry/Exit tab and choose the Vertical option for the entry and exit styles.

15. Click the Boundary tab and ensure that Stock Type is set to None.

    Notice that Boundary Type is grayed out, because Process #2 must use the same Boundary Style as Process #1.

16. Close Process #2 and (Double-click Process Tile #3 and choose Advanced 3D Machining.

17. Click the Surfs tab and enter values as shown.



This Flats Cut process will finish the floor and wall that has a sharp corner. By setting the start Z depth to just below the large flat area we can select the entire part. Also, Z final depth is set to -34.6456, which is equal to the floor of the first step.

18. Ensure Operation Type is set to Flats Cut and then click Advanced Settings.

19. Select the option Profile Ramping.

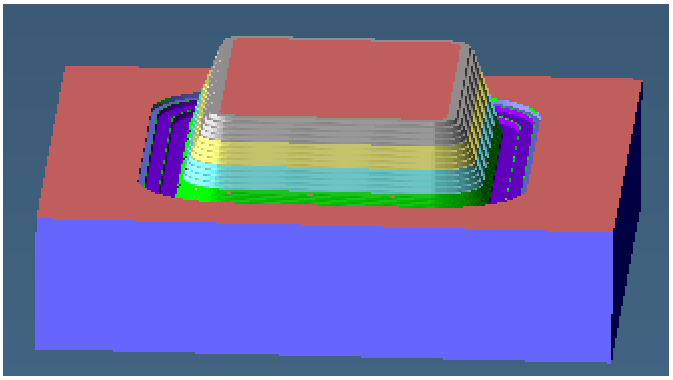
20. Close the Advanced Settings dialog box.

21. Close Process #3. Click the empty tile below the first 4 operations.

22. Select the solid (the entire part) and then click Do It to generate the operations.

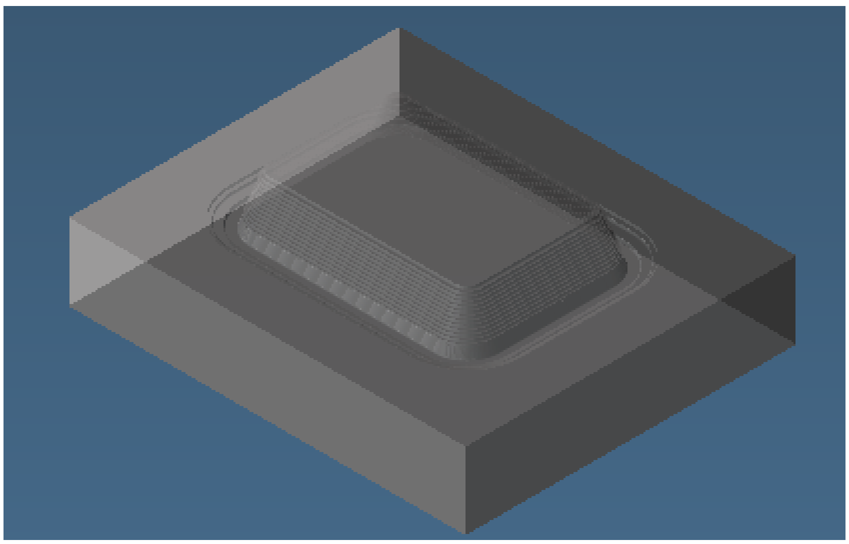
Notice in the Task Manager that Processes #2 and #3 finish calculating before Process #1. This is a result of the strategy used to make the processes. After all three processes have run, you have Operations #5-7.

23. Run Op Sim and verify the results when it completes.



24. Right-click the Op Sim title bar and choose Create Facet Body.

25. Close the rendering palette.

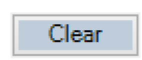


The facet body we just created is shown in the workspace. It is partially transparent. You may need to place the part model in the Body Bag to see the facet body

26. Place the facet body into the Body Bag and set its properties to Stock – Display Only.

You can use this body as the starting material condition for each subsequent Sim rendering. Note that if the facet body properties were set to Stock, then each subsequent toolpath process would set the Stock Management, Stock Type to Solid (in other words, a material-only starting condition).

27. Delete the three processes.



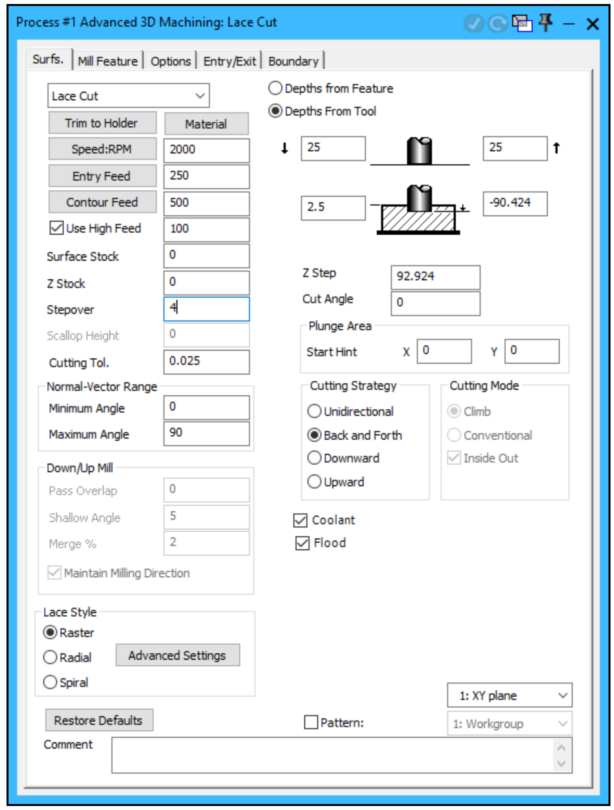
28. Save the interim part, which now has Operations #1-7.

#8: Lace Cut

We will finish the “flat” walls of the core with lace cuts.

1. Create an Advanced 3D Machining process the 6mm Ball EM (Tool #8).

2. `Click` on Restore Defaults and enter the values as shown.



This will create toolpath parallel to the X-axis. We have set the stepover very loose on purpose so that the processing time is very quick and we can easily see the results.

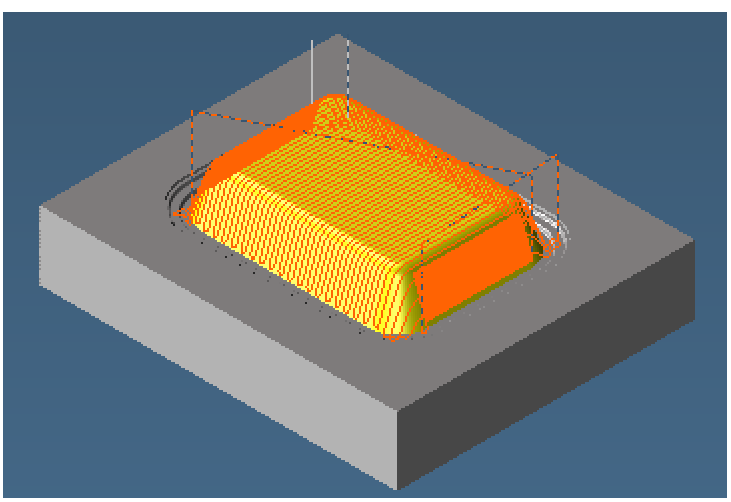
3. Close the process.

4. Ensure Face Selection mode is enabled.

5. `Right-click` one of the angled walls on the core and choose Select>Select 3D Faces.

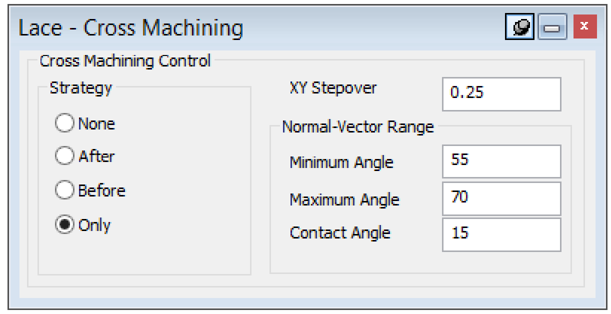
6. (`Ctrl+click`) the top face of the core.

7. `Click` Do It to generate the new operation.



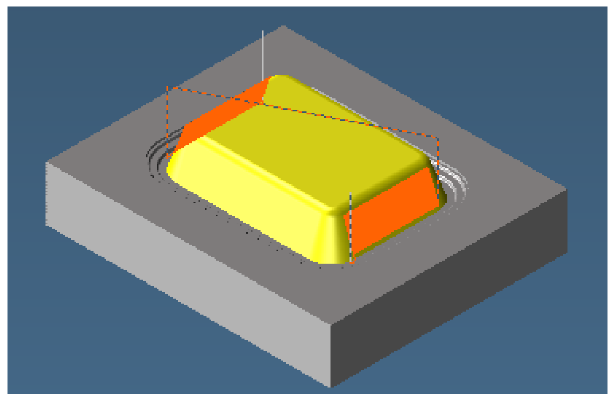
We can see that default lace cut settings are going to try to machine all of the walls. We want these walls to be machined up and down, not across the faces.

8. Open the process dialog and `click` on the Advanced Settings button. Set the values as shown.



Cross machining creates toolpath perpendicular to the cutting direction. By setting the Cross Machining Control to "Only" we have eliminated all of the toolpath that is parallel to our Cut Angle.

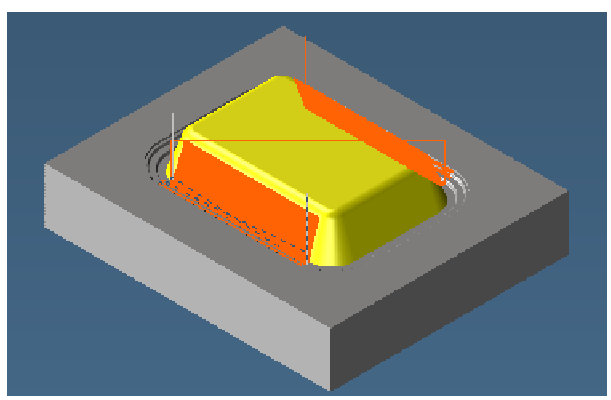
9. `Click` Redo to recreate the operation.



#9: Lace Cut on the Long Walls

We will continue what we did on the walls parallel to the X-axis but modify the process to cut the long walls.

1. Open the process for Operation #8 and change the cut angle to `90`. `Click` Do It to create a new operation.

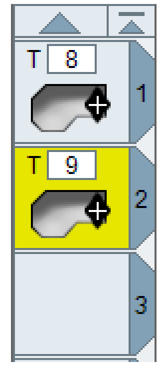


This operation is good but it creates too much wear on the tool.

#10: Lace Cut Toolpath Splitter

We will now split the toolpath of Operation #9 into two operations with two separate tools. When using Toolpath Splitter, be sure to use two or more identical tools (in this case, Tool #8 and Tool #9). When the toolpath is split, notice that the split will be created at a safe area in the toolpath.

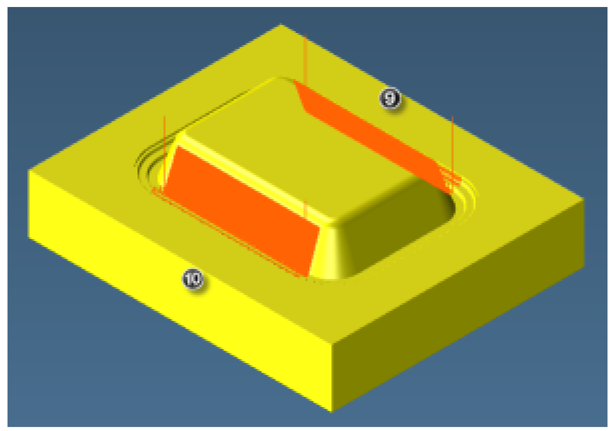
1. Create an Advanced 3D process with the second 6mm Ball EM (Tool #9) in Process tile #2.



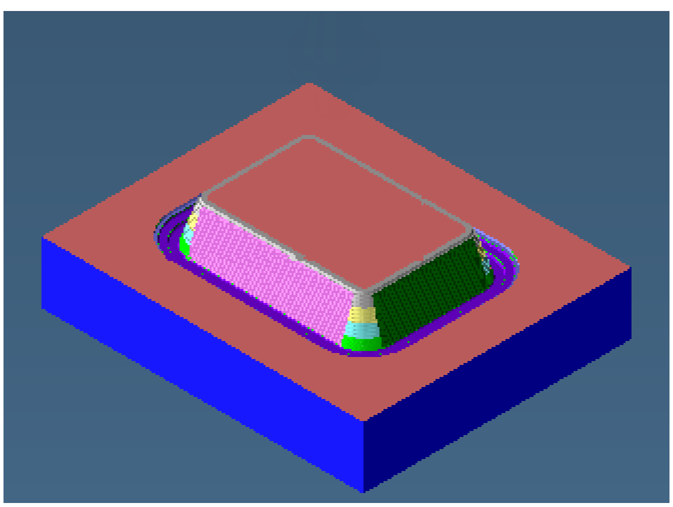
2. Set the cut type to Toolpath Splitter.

3. Click Redo.

   The resulting operations split the toolpath in half.



If Operations #8-10 are rendered, the part will look as below.

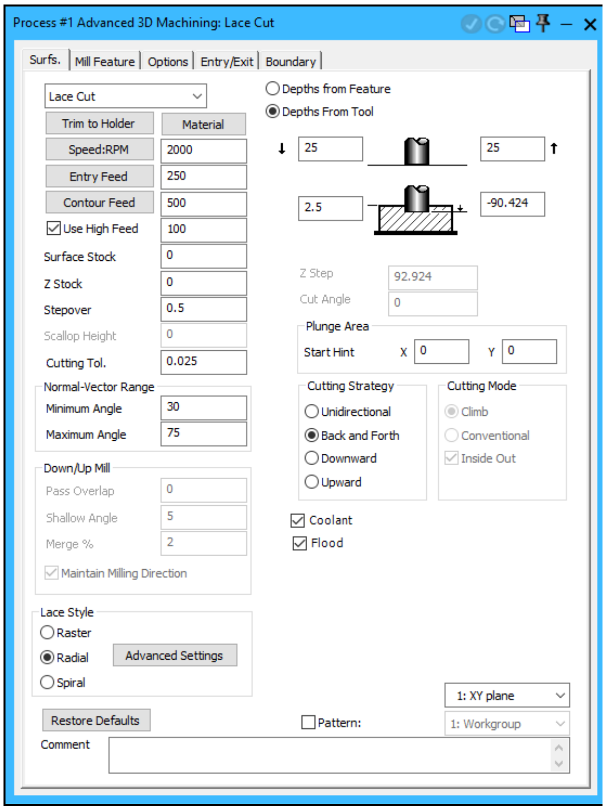


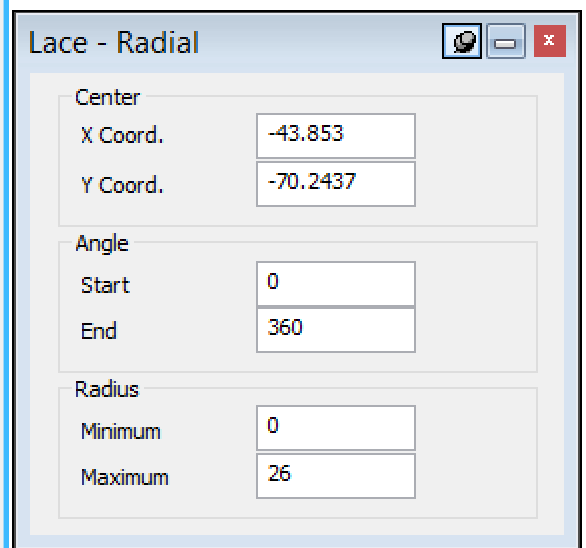
4. Delete the process tiles.

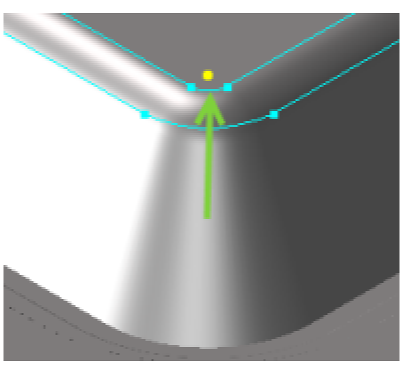
#11-14: Lace Cut - Radial with Up Machining

We will now create toolpath on the 3D corners. We will accomplish this using a radial lace cut.

1. Create an Advanced 3D Machining Lace Cut process using the 6mm Ball EM (Tool #8).
2. Click the Restore Defaults button and set the values as shown. Click Advanced Settings to input the Lace-Radial settings.







The primary changes we make here are the Normal-Vector Range values and the Lace Style. The center coordinates for the radius can be easily acquired by interrogating the point in the X-Y- quadrant of the part (the point closest to you when the part is in isometric view.)

3. Click the Options tab and set Minimum Pass Length to 6.0.

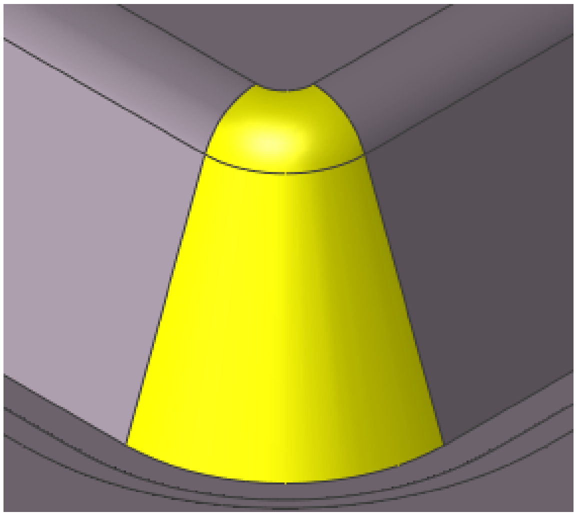
Using a Minimum Pass Length of 6mm will prevent the tool from machining the floor of the lid area, when the faces are selected in the next few steps.

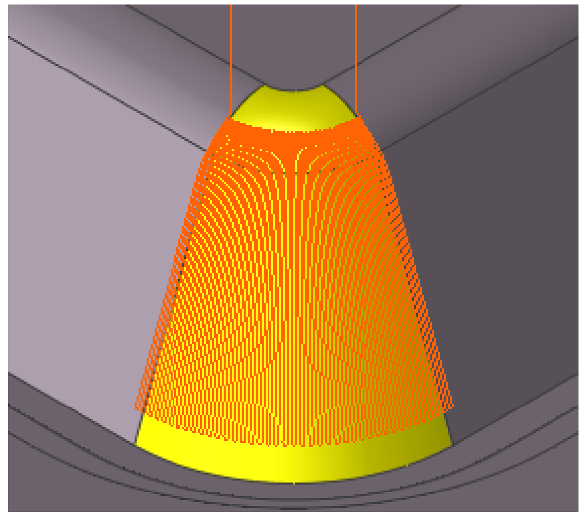
4. Click the Entry/Exit tab and set the Entry Style and Exit Style to Vertical. Use the default value of 1.5.

5. Click the Boundary tab and verify that the Boundary Type is Silhouette.

6. Close the process.

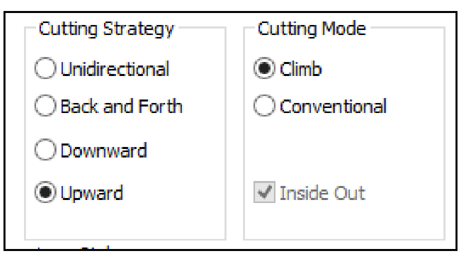
7. Select the faces as shown and click Do It to generate the new Operation (#11).

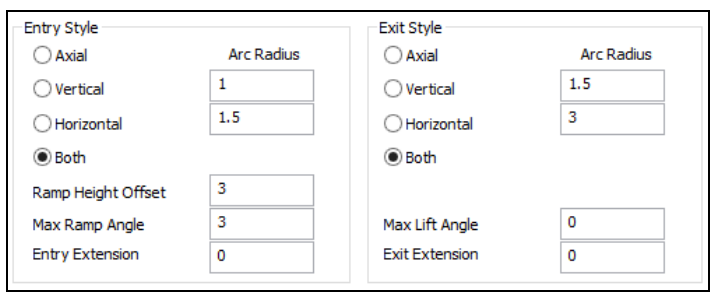




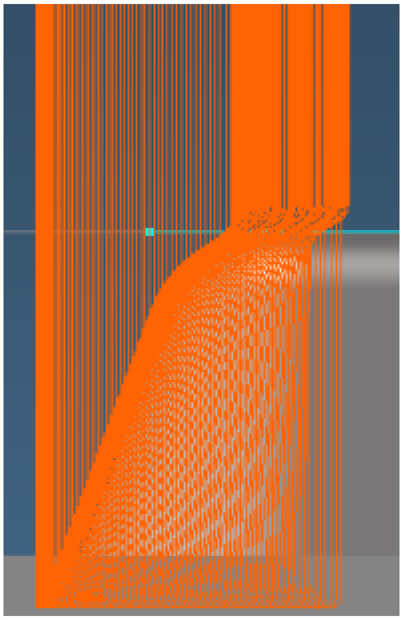
That is acceptable toolpath but we want to machine uphill only and we wanted toolpath that arcs on and off the part.

8. Open the process dialog and change the Cutting Strategy to Upward. Also change the Entry/Exit values as shown.





9. Click Redo to recreate the toolpath.

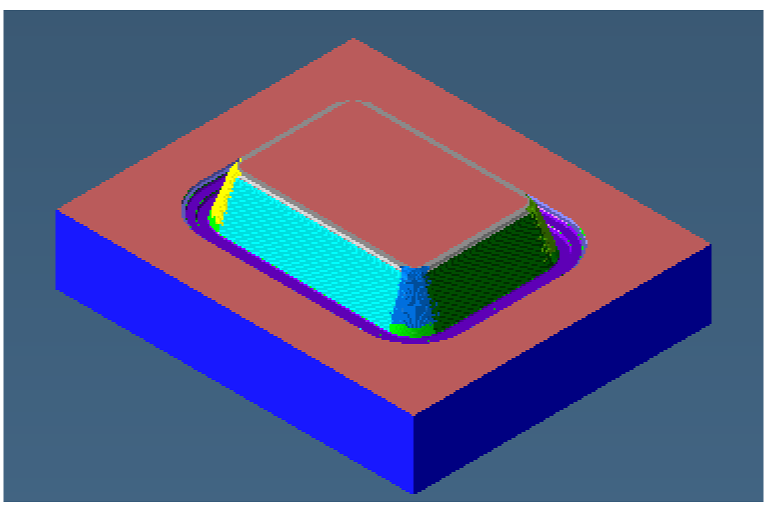


Looking at the part from the side we can clearly see the arcing motion on and off of the part.

10. Create this operation three more times. Each time select a new value for the radius center point by interrogating the point and selecting the 3D faces on the corners. Work counter clockwise to create a total of four individual Operations (#11-14). No other process data needs to be changed than the radius center.

11. Select Operation tiles #8-14. Enable the Skip unselected ops function  (it turns blue). Run the rendering.

When rendered, the part will look similar to the image below.



12. Delete the process tile.

#15: N-Curve Flow

The last operation will machine the rounded top edges. It is a challenge to make a good contour-style toolpath without retracts for the 8.5mm radius around the top of the core. The best strategy to accomplish this is N Curve Flow, as described in the following steps.

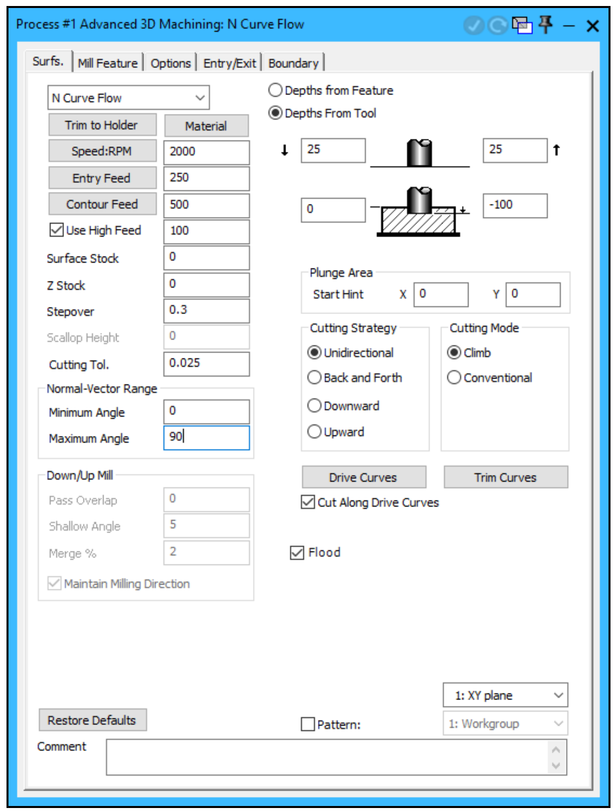
1. Deselect all operations so you can clearly see the part and geometry.

   The workgroup in this part contains two closed geometry profiles. Both profiles were extracted from the model's 8.5mm radius using a Force Depth to Z-zero. Next, the top profile was offset inward a small amount to allow for some overlap of the toolpath onto the top surface. The lower profile was offset outward by 4.5mm (75%) of the tool's radius to allow the contact point to reach the bottom of the 8.5mm radius.

   When using this strategy, the amount that the curves are offset may be different from one part to the next depending upon the feature angles at which the tool must make contact. For example, more-vertical angles may require a larger offset; less-vertical angles may require a smaller offset.

2. Create a new Advanced 3D Machining process using the 6mm Ball EM (Tool #10) and set the Operation Type to N Curve Flow.

3. Click Restore Defaults and then enter values as shown.



   Notice the change of Cutting Strategy option to Unidirectional.

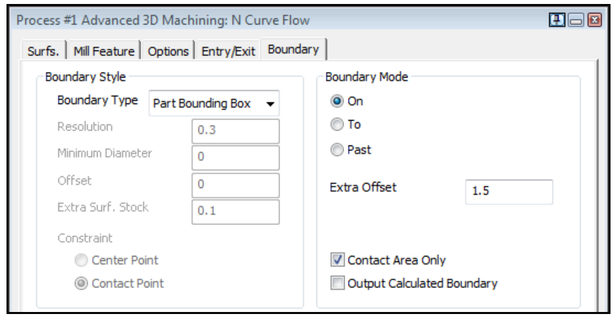
4. Click the Drive Curves button to open the Select Drive Curves dialog box.

5. Select both of the drive curves in the workgroup, making sure that the drive curve arrows point in the same direction.

   When you select a connector point, the arrow will first point in the default direction. To reverse the arrow direction, (Ctrl+click) the element on the opposite side of the connector point. To reverse the arrow back to its original direction, (Ctrl+click) the element in the original direction. (When you (Ctrl+click) the element in the original direction, you are actually deselecting and removing the element from the selection dialog, thus causing the arrow to return back to the default direction.)

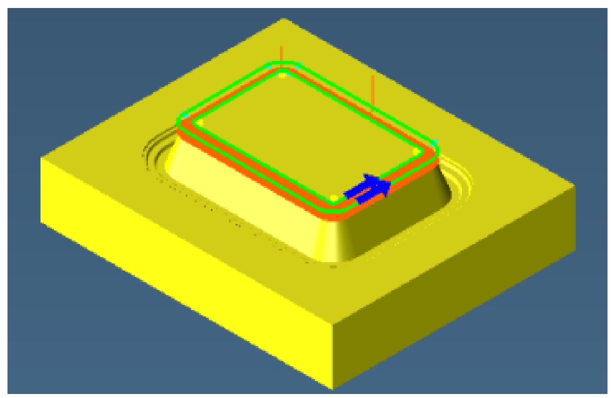
6. Click OK to close the Select Drive Curves dialog box.

7. Click the Boundary tab and enter values as shown.



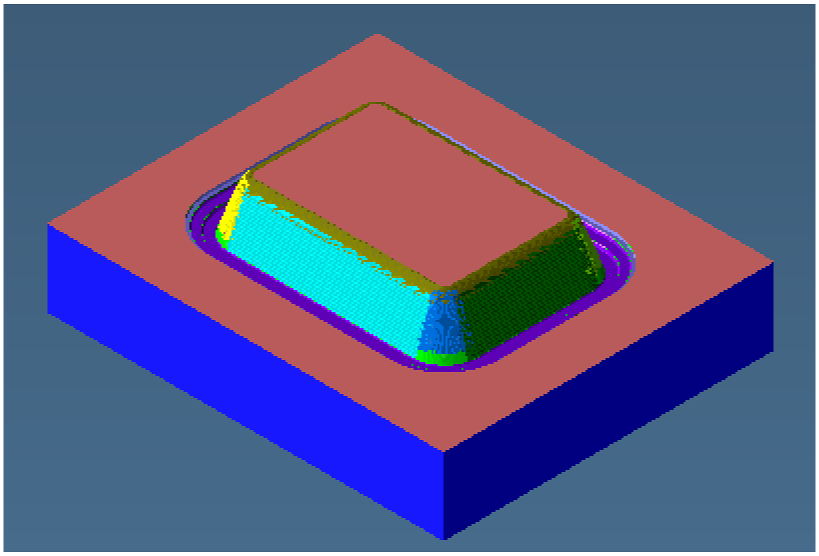
8. Close the process, select the body and then click Do It to generate Operation #15.

   When the calculation is complete, you will see a 3D contour style of toolpath with only one retract.



9. Ensure that Operations #8-15 are selected and that Skip Unselected Ops is still active. Render the part.

   When rendered, the part will look similar to the following.



10. Save the final part.

#3: The Mold Core

In this exercise, we will explore four different 3D toolpath options: 1) Constant Stepover; 2) Steep Shallow Cut; 3) Intersections; and 4) Intersections Rest. The goal of this tutorial is to familiarize you with various toolpath options. Although other operation types will also be used, they will not be explained in detail, as they were covered in previous Advanced 3D Machining tutorials.

Part Setup

1. Open the Adv3D#3 Mold Core.vnc part supplied on the GibbsCAM installation CD.

2. Verify the following tool list.

#	Type	Length	Flute Length	Diameter	Bottom Radius	Length out of holder
1	Rough EM	60	25	25.00	0	35
2	Rough EM	50	25	12.50	1mm	25
3	Ball EM	75	25	12.50	n/a	35
4	Ball EM	75	25	6.00	n/a	40
5	Ball EM	50	10	2.50	n/a	22

Roughing the Part

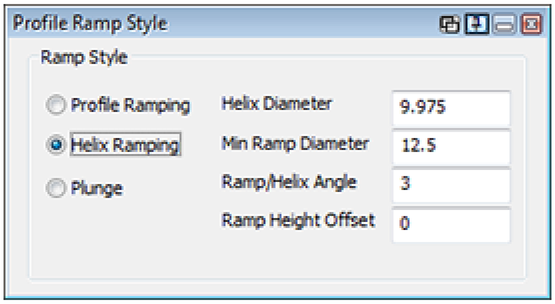
#1: Roughing the Pocket

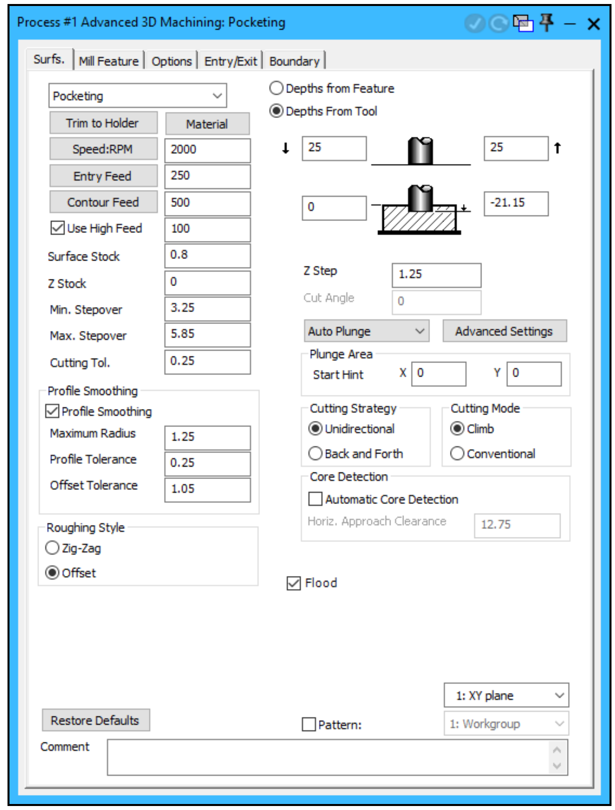
For Operation #1, we will use Tool #2 to rough the pocket at the top of the part.

1. Create an Advanced 3D Machining process with Tool #2.

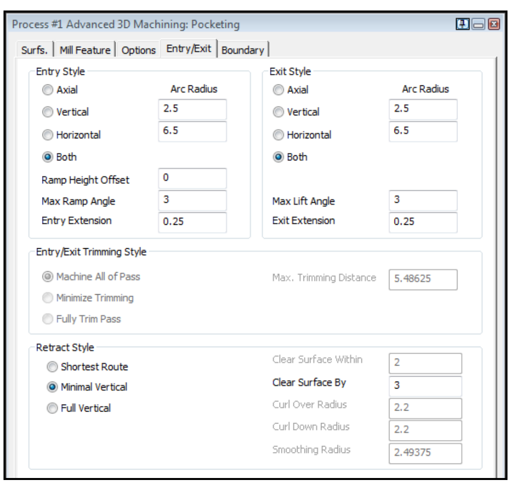
2. Verify that Operation Type is set to Pocketing and then click Restore Defaults.

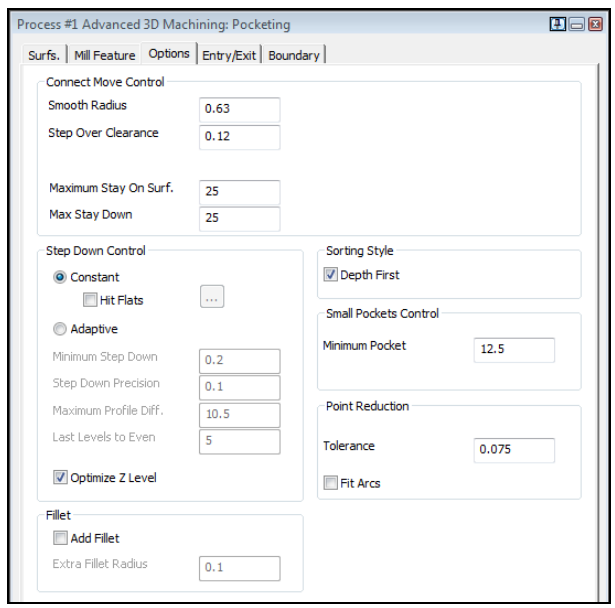
3. Enter values as shown for the Surfs tab and for Advanced Settings.





4. Set the values for the Options and Entry/Exit tabs as shown.

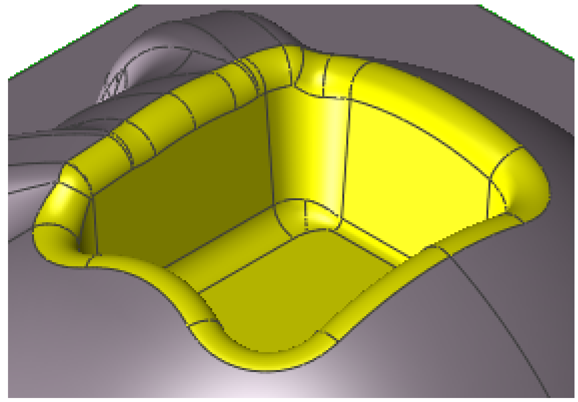




5. Click the Boundary tab and set the Boundary Type to Silhouette.

6. Close the process.

7. Ensure that Face Selection is turned on.
8. Right-click the pocket floor and choose Select>Select Faces Above.
9. Verify that all faces inside the pocket are selected, as shown.



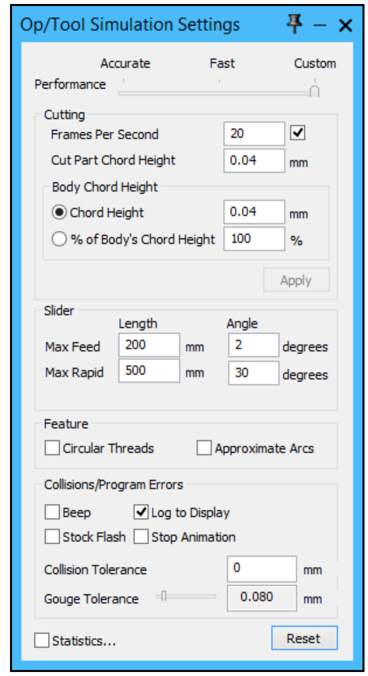
10. Click Do It to generate Operation #1.

    Next, you will adjust several settings that will allow you to run Op Sim more rapidly and to detect collisions between the tools/holders and the part.

11. Open Op Sim.
12. Click the Op Color Mode button. Ensure that Op Number is selected and then close the dialog box.

    This setting will allow you to see more clearly what each operation is cutting.

13. Click the Collsion checking icon  it will turn blue when enabled.
14. Right-click the title bar and choose Settings.



15. Ensure that the values shown are set, these are the system defaults. If they are not set, change the values and click Apply before closing the dialog box.

16. Run Op Sim and wait until the pocketing operation is rendered.

17. Right-click the Op Sim title bar again and choose Create Facet Body.

    You will use this facet body as stock when creating the next two operations. This technique allows you to create material-only toolpath calculation without having to create dependent processes.

18. Close the Op Sim dialog box.

19. Right-click the facet body and choose Show Properties.

20. Click the option for Stock.

    You choose Stock rather than Stock - Display Only because you will want Operations #2 and #3 to calculate toolpath based on this stock condition.

21. Close the Facet Body Properties dialog box.

22. Place the facet body into the Body Bag.

23. Delete Process 1.

24. Save the interim part, which now has Operation #1.

#2-3: Pocketing

For Operations #2 and #3, you will use Tools #1 and #3 to rough the outside of the part.

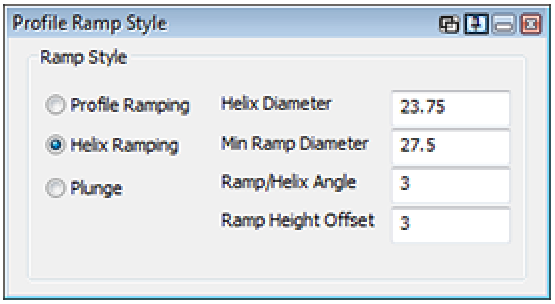
1. (Ctrl+click) Tools #1 and #3 and drag both tool tiles to position 1 on the process list.

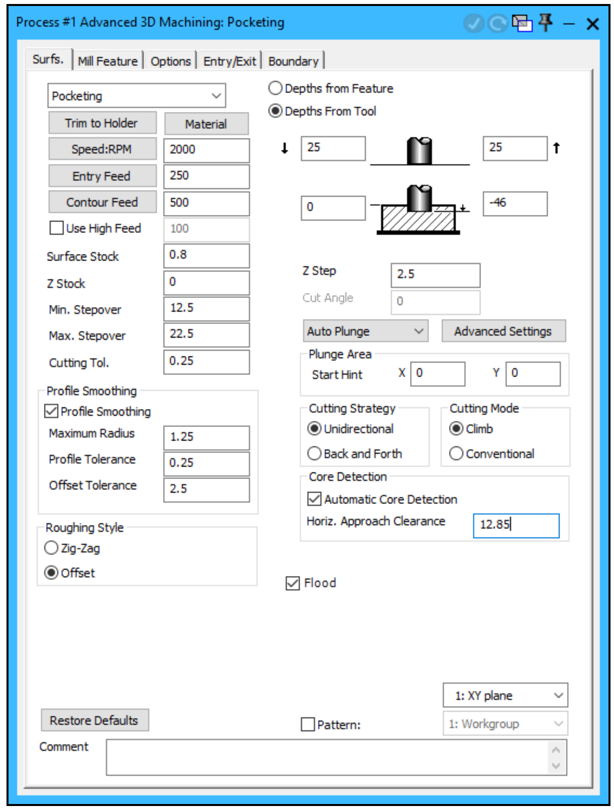
2. Choose the Advanced 3D Machining tile.

   Both processes are populated and Process #1 opens.

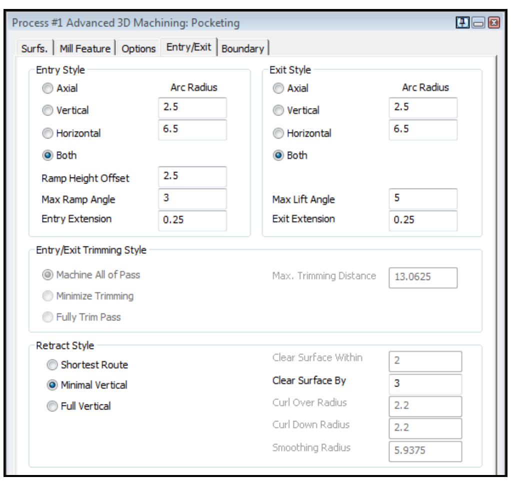
3. Verify that Operation Type is set to Pocketing and then click Restore Defaults.

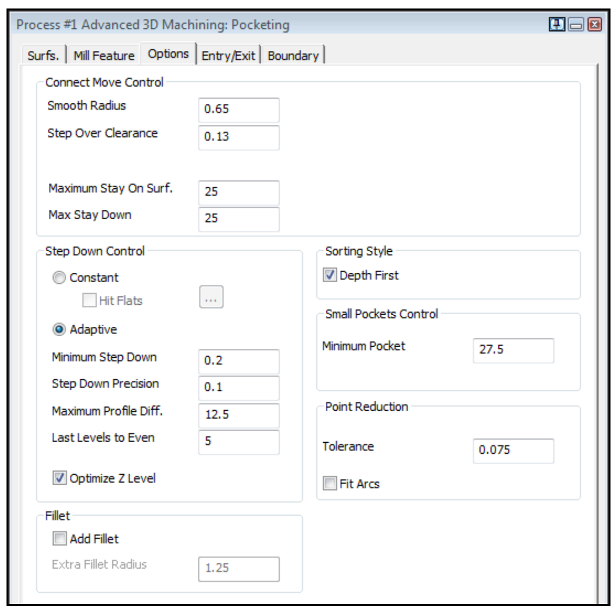
4. Enter values for the Surfs tab and Advanced Settings as shown.





5. Set the values for the Options and Entry/Exit tabs as shown.





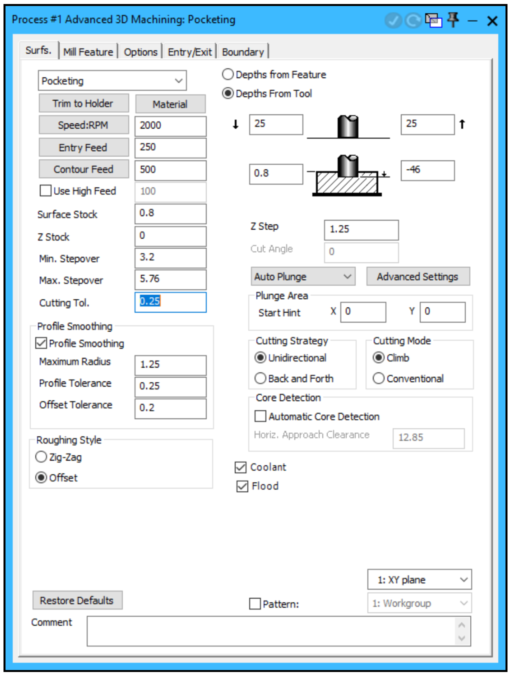
6. Click the Boundary tab and ensure the Boundary Type is set to Part Bounding Box.

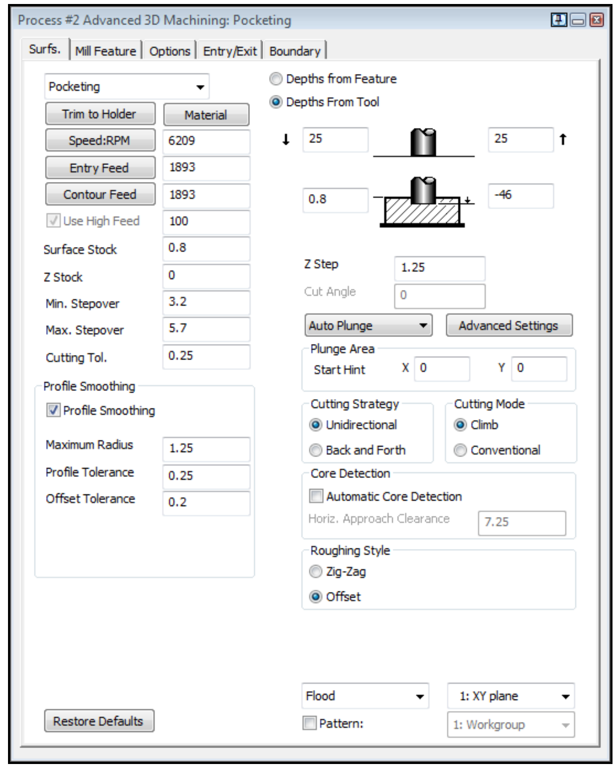
7. Double-click Process #2.

Process #1 closes and Process #2 opens. The goal of Process #2 (which will create Operation #3) is to smooth the large Z-steps left by Operation #2 with its 25mm rough tool.

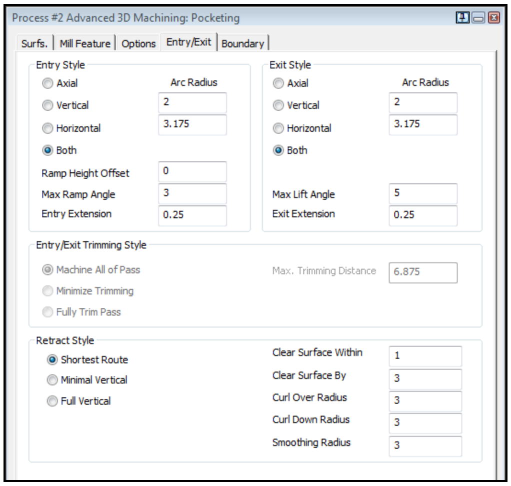
8. Click the Surfs tab, verify that Operation Type is set to Pocketing and then click Restore Defaults.

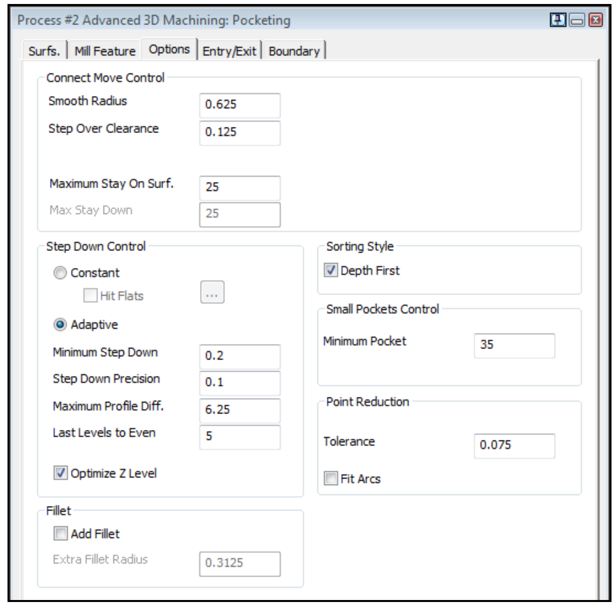
9. Enter values for the Surfs tab and Advanced Settings as shown.



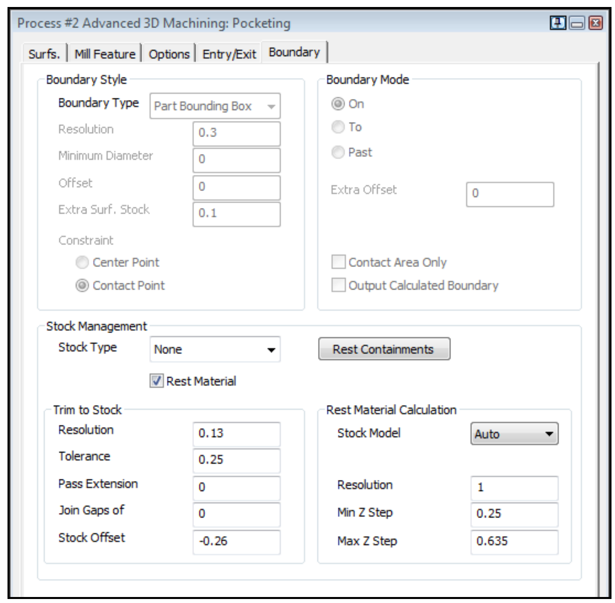


10. Set the values for the Options and Entry/Exit tabs as shown.





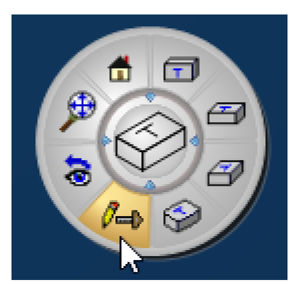
11. Click the Boundary tab and enter values as shown.

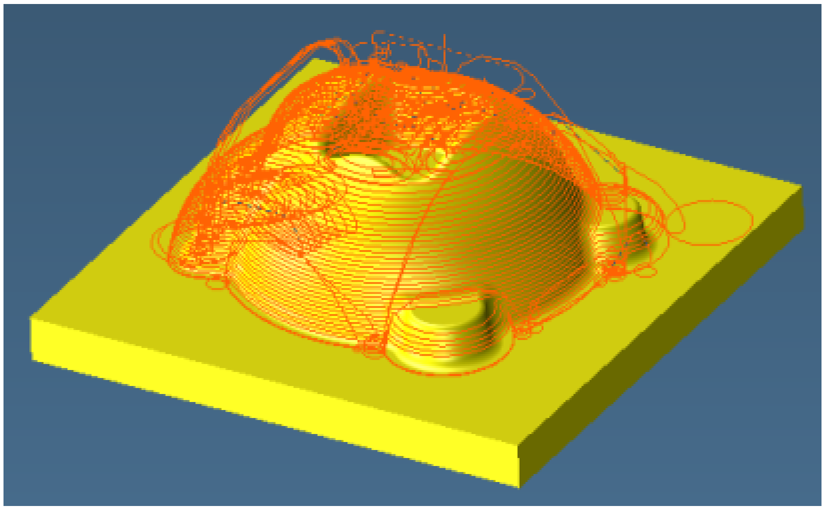


12. Close the process.

13. Turn off Face Selection and `click` the part body, thus selecting all faces then `click` Do It and wait for Operations #2 and #3 to be generated.

14. `Click` Operation #3 and redraw it (`Ctrl+R`).





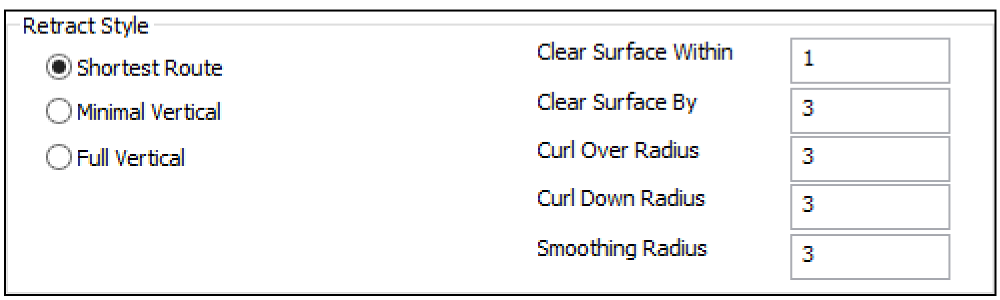
The toolpath for Operation #3 should be as shown.

15. Delete Processes 1 and 2.
16. Save the interim part, which now has Operations #1-3.

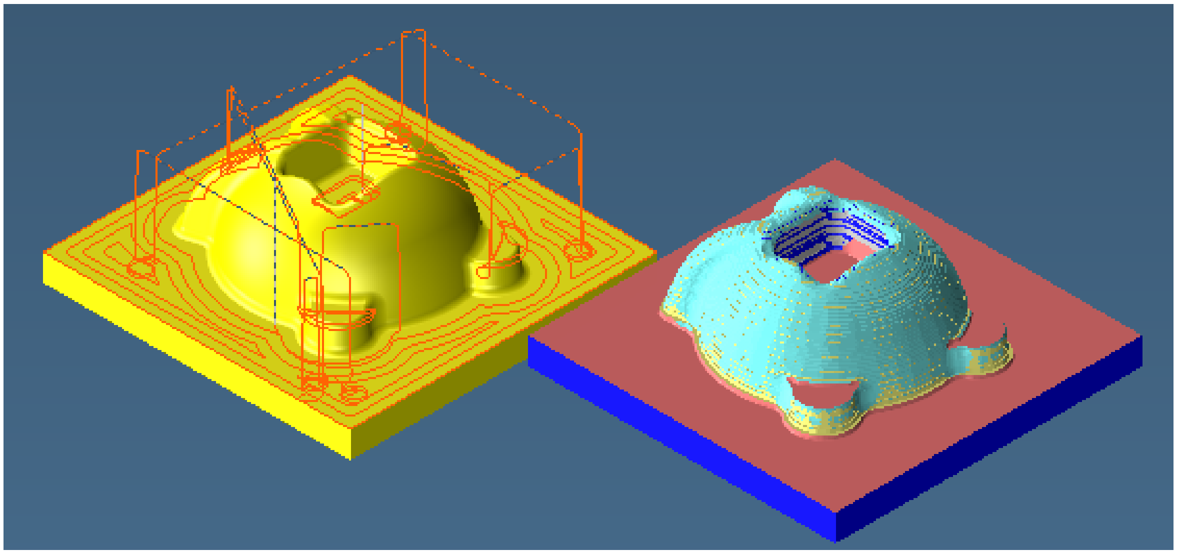
#4: Flats Cut

We will now clean up the flat areas of the part. This is very easy with Advanced 3D.

1. Create an Advanced 3D Machining process with Tool #2.
2. Change the Operation Type to Flats Cut and then click Restore Defaults.
3. Click the Entry/Exit tab and enter values as shown.



4. Click the Boundary tab, change the Boundary Type to Part Bounding Box, and verify that Stock Type is set to Solid.
5. Close the process.
6. Verify that all faces of the part body are selected and then click Do It to generate Operation #4.



7. Delete process 1 and save the interim part, which now has Operations #1-4.

#5: Semi-finishing, Using Constant Stepover Cut

1. Create an Advanced 3D Machining process with Tool #3.
2. Click Restore Defaults and then change the Operation Type to Constant Stepover Cut.
3. Change the value for Surface Stock from zer0 to 0.13.
4. Click the Boundary tab and verify that Boundary Type is set to Part Bounding Box and that Stock Type is set to Solid.
5. Click the Options tab and set Max Stay down to 12.5.
6. Close the process.
7. Select the part body and click Do It to generate Operation #5.

   When the toolpath has finished calculating, you notice that it is cutting all faces. This is undesirable, and so you will modify the process to get a better result.

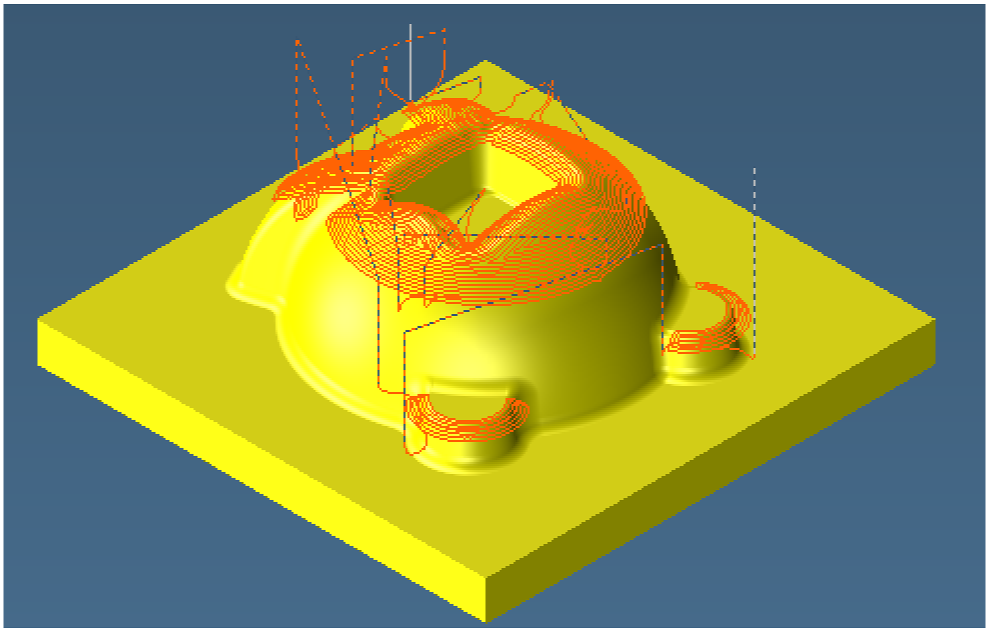
8. Open Process 1, Surfs tab.
9. Change Minimum Angle to 0.1 and Maximum Angle to 46.

   The Minimum Angle of 0.01 prevents the toolpath from machining flat areas (at zero degrees), and the Maximum Angle of 46 prevents the toolpath from machining areas that are steeper than 46 degrees. Later, in Operation #6, you will use a different Operation Type to semi-finish the steeper areas.

10. Click the Boundary tab, change the Boundary Type to Shallow Areas, and verify that Stock Type is set to Solid.
11. Close the process.

12. Verify that all faces of the part body are selected and then click Redo to regenerate Operation #5.

    The toolpath for Operation #5 will be similar to the image below.



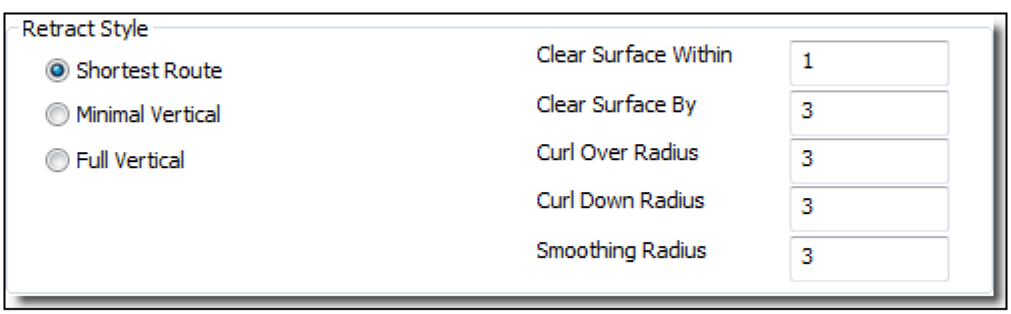
13. Retain the process and save the interim part, which now has Operations #1-5.

#6: Semi-finishing, Using Contour

1. Open process 1, Surfs tab.
2. Change Operation Type to Contour and then click Restore Defaults.
3. Enter a value of 0.13 for Surface Stock.
4. Change Minimum Angle to 45 and change Maximum Angle to 90.

   The Minimum Angle of 45 will cause the new toolpath to overlap that of Operation #5 and the Maximum Angle of 90 will cause the toolpath to machine all walls between 45 degrees and vertical.

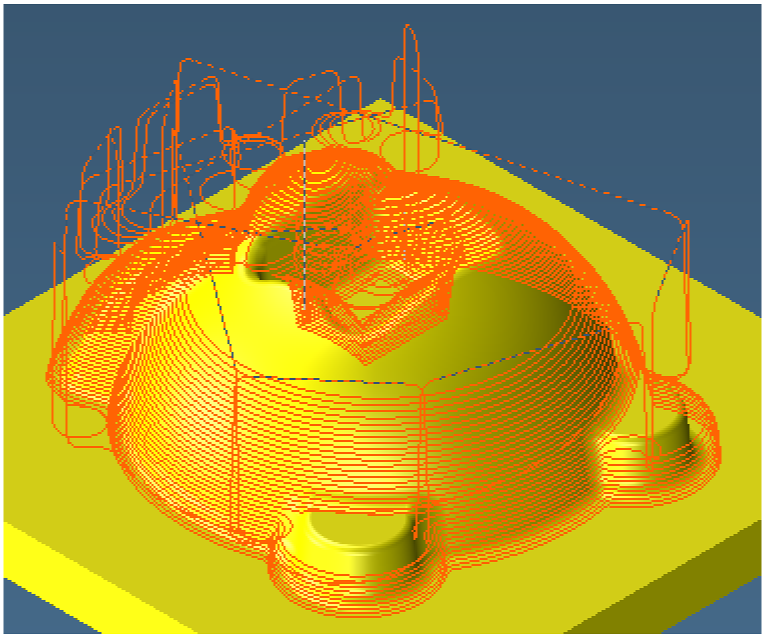
5. Change Z Step to 1.
6. Click the Entry/Exit tab and enter values as shown.



7. Click the Boundary tab, change the Boundary Type to Part Bounding Box, and verify that Stock Type is set to Solid.

8. Close the process.

9. Verify that all faces of the part body are selected and then click Do It to generate Operation #6.

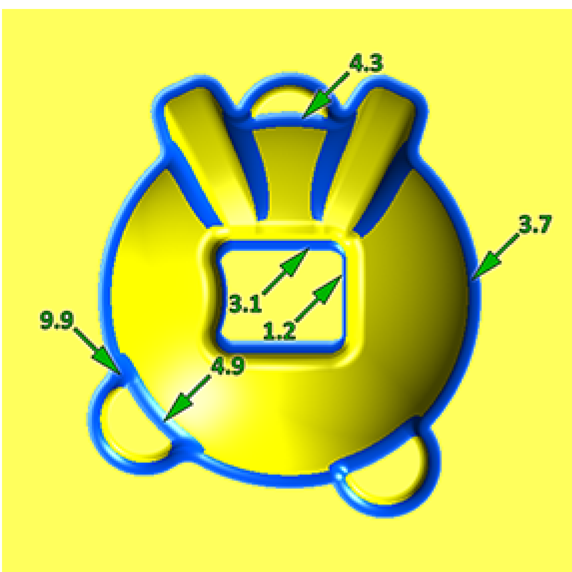


When the calculation is complete, you notice that the toolpath is cutting only the areas (walls) that were not cut by Operation #5, the previous operation.

10. Delete the process.

11. Save the interim part, which now has Operations #1-6.

#7: Semi-finishing, Using Intersections



Because of the many different radii used on this part (see illustration), it is not really possible to get a single Intersections operation that will get all of the edges. We will make an Intersections operation that will get most of the edges.

1. Create an Advanced 3D Machining process with Tool #4.

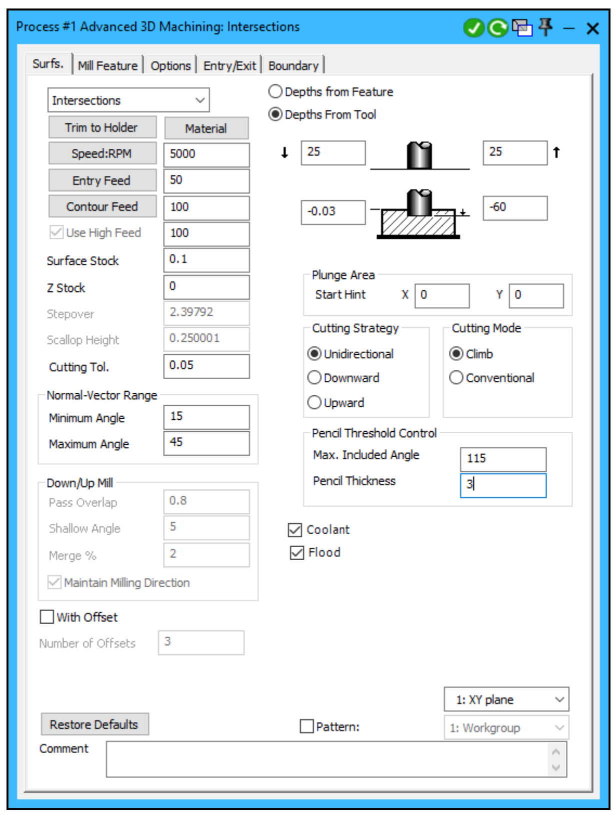
2. Change the Operation Type to Intersections and then click Restore Defaults.

3. Change the value for Surface Stock from zer0 to 0.1.
4. Click the Boundary tab and verify that Boundary Type is set to Part Bounding Box and that Stock Type is set to Solid.
5. Close the process.
6. Verify that all faces of the part body are selected and then click Do It to generate Operation #7.

   When the toolpath has finished calculating, you notice that it is cutting only some of the intersections in and around the pocket area. This is undesirable, and so you will modify the process to get a better result.
7. Open Process 1, Surfs tab.
8. Enter a value of 6.5 for Pencil Thickness.

   This will cause the toolpath to cut part radii that are larger than the radius of Tool #4.
9. Close the process.
10. Verify that all faces of the part body are selected and then click Redo to regenerate Operation #7.

    When the toolpath has finished calculating, you see that it is now cutting too many of the intersections. This is undesirable, and so you need to further modify the process settings.
11. Open Process 1, Surfs tab.
12. Set the process values as shown.

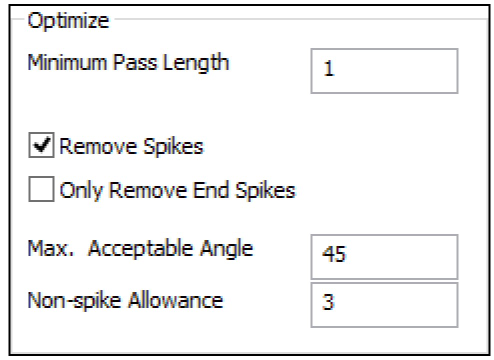


- The higher Cutting Tolerance will speed up the operation generation.

- The modified Normal-Vector Range will limit which faces are eligible for the Intersections toolpath.
- The lower Max. Included Angle and Pencil Thickness values will cause the toolpath to cut fewer part radii.

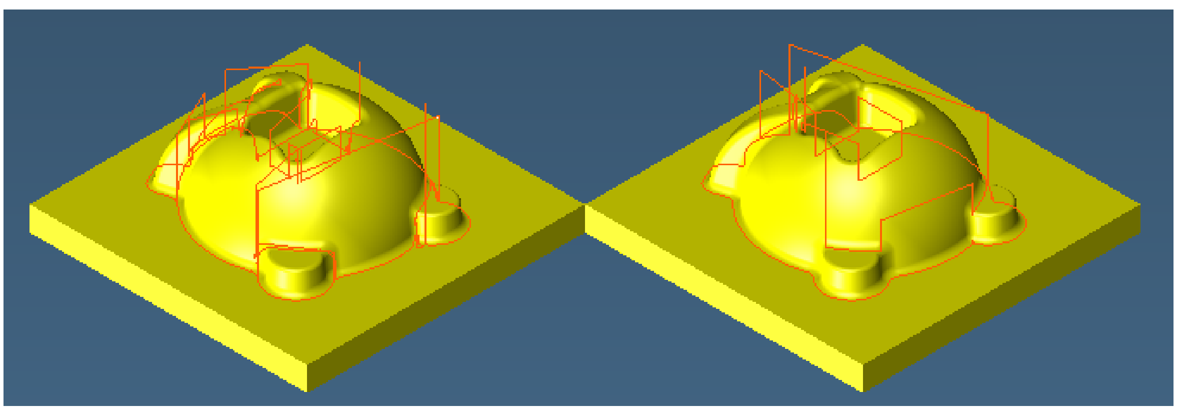
13. Click on the Options tab and set the items in the Optimize group as shown.

    This will smooth out the toolpath and remove some toolpath with moves in Z+.



14. Close the process.
15. Verify that all faces of the part body are selected and then click Redo to regenerate Operation #7.

    When the toolpath has finished calculating, you see that it is now cutting an acceptable number of intersections. This is more desirable than the previous results.



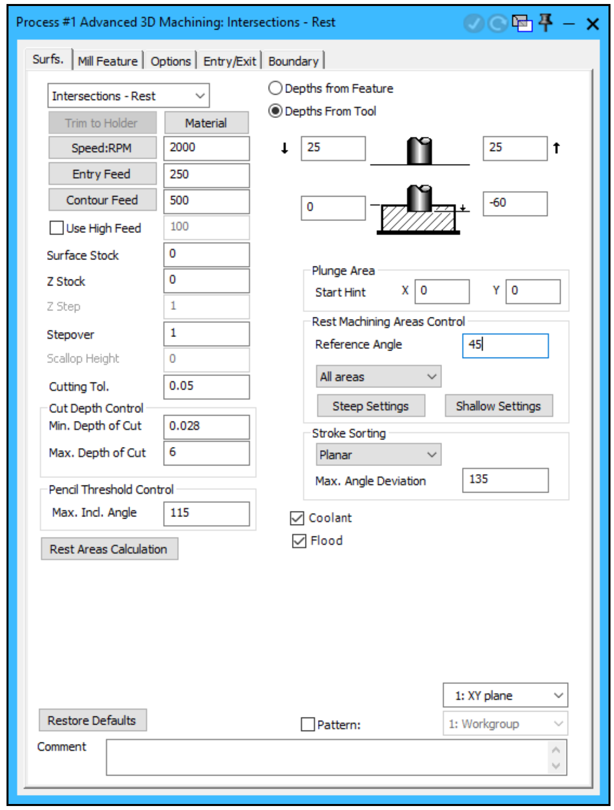
16. Retain the process and save the interim part, which now has Operations #1-7.

#8: Finishing, Using Intersections-Rest

The next three operations will finish the part.

1. Open the process, Surfs tab.
2. Change Operation Type to Intersections-Rest and then click Restore Defaults.

3. Enter values as shown.



In the Reference Tool area, you change Diameter to 12.5 so that the toolpath will make more offset loops, ensuring a more thorough cleanup of the part radii.

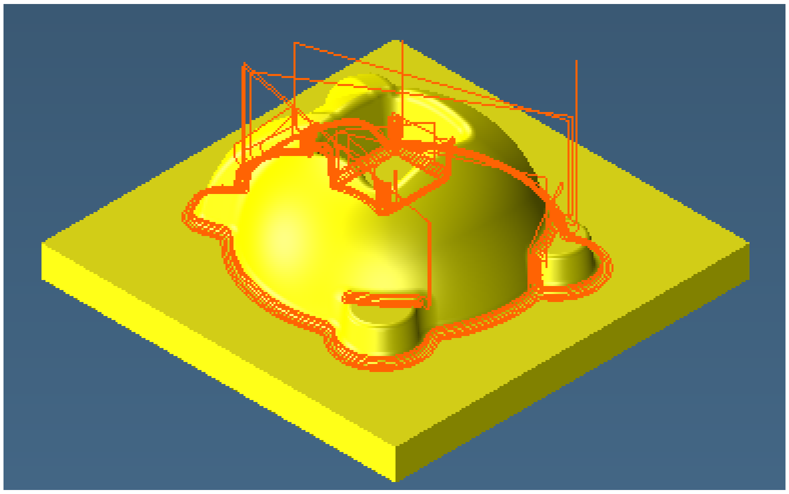
In the Rest Machining Areas Control area, you change Reference Angle to 45 so as to define which part features are steep and which are shallow. When used with the setting of All Areas it causes the toolpath to cut steep walls like a contour cut and shallow walls like a pocketing cut. In the dropdown list, if you were to choose Steep Areas Only (for example), it would restrict the toolpath to cut only steep areas.

In the Stroke Sorting area, you retain the option Planar (vs. Angular) to specify that the toolpath will first make cuts that are planar to the tool's depth-axis (as opposed to first making cuts that are angular to the tool's depth-axis). The choice of None would disable the sorting.

4. Click the Boundary tab and verify that the Boundary Type is set to Part Bounding Box and that Stock Type is set to Solid.

5. Close the process.

6. Verify that all faces of the part body are selected and then click Do It to generate Operation #8.



7. Retain the process and save the interim part, which now has Operations #1-8.

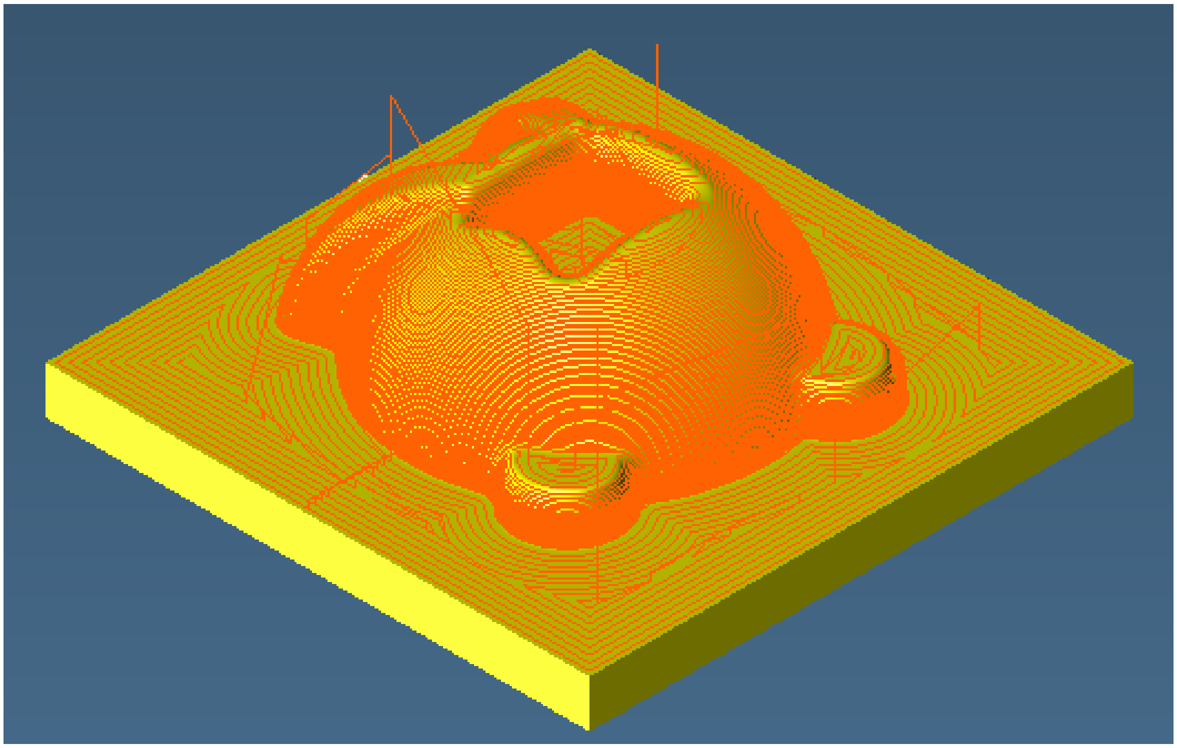
#9: Finishing, Using Steep Shallow Cut

A Steep Shallow Cut operation is designed to be single step finishing routine. Even complex parts can be finished with just this single operation because “steep” areas and “shallow” areas are cut differently. Typically you do not even need to change the default values to get acceptable results. The one downside to this operation type is that it is not fast to generate.

1. Open the process, Surfs tab.
2. Change Operation Type to Steep Shallow Cut and then click Restore Defaults.
3. Change the Z Step value to 0.5 and the Stepover to 2.
4. Click on the Options tab and change both the Steep and Shallow Maximum Stay of Surf. values to 20.

   This is done to minimize retracts.
5. Deselect the Depth First option.
6. Click the Boundary tab, ensure that the Boundary Type is set to Silhouette, and verify that Stock Type is set to Solid. Also set the Boundary Mode to On.

   This will take the tool center to the edge of the part and help minimize retracts.
7. Close the process.
8. Verify that all faces of the part body are selected and then click Do It to generate Operation #9.
9. The process will take at least several minutes to complete.



You can see how the XY moves are offset by 2mm while the Z moves are 0.5mm.

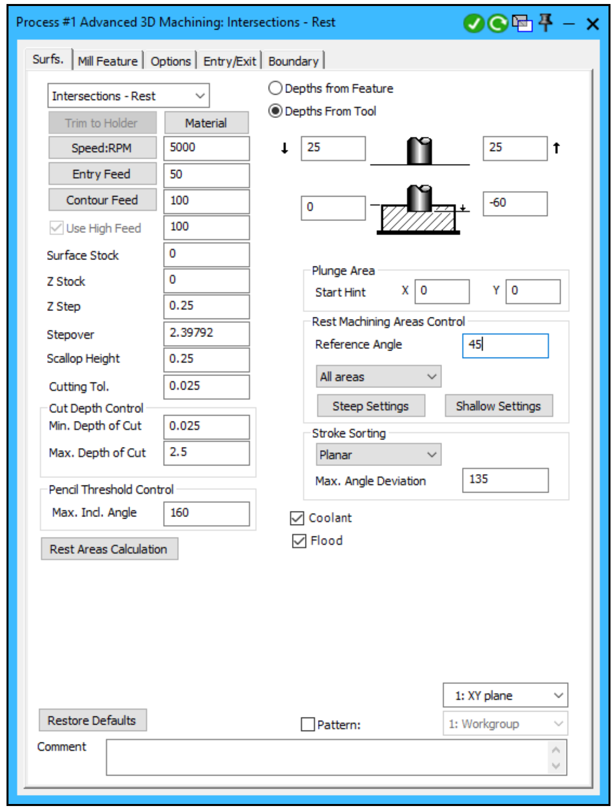
10. Retain the process and save the interim part, which now has Operations #1-9.

#10: Finishing, Using Intersections-Rest

The last thing we are going to do is finish the small rounded edges inside the pocket.

1. Drag Tool #5 onto the existing process.
2. Click the Surfs tab, change the Operation Type to Intersections-Rest, and then click Restore Defaults.

3. Enter values as shown.



4. Click the Options tab and change Minimum Pass Length to 5.

   By increasing this value, we are preventing some small segments from being cut because they do not need further machining. This is most notable in the transition area from the small radius to the larger radius edges and the top, outer edges.

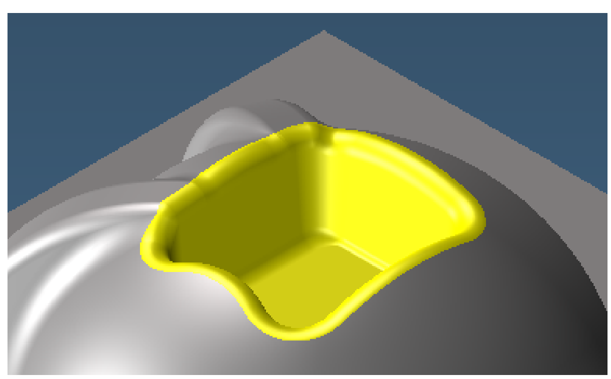
5. Click the Boundary tab, change Boundary Type to Silhouette, and verify that Stock Type is set to Solid.

6. Close the process.

7. Turn Face Selection on.

8. Right-click the pocket floor and choose Select Faces Above.

   Verify that all faces inside the pocket are selected, as shown.

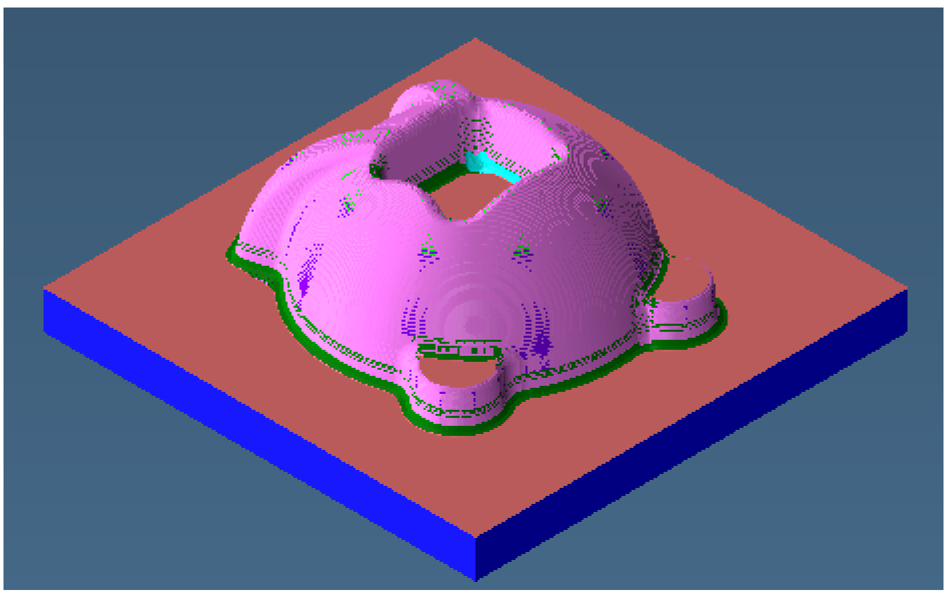


9. Click Do It to generate Operation #10.

   We can see that the toolpath is limited to the small radius edges.

10. Delete the process and save the final part, which now has Operations #1-10.

    When rendered, the part will look similar to the following image.



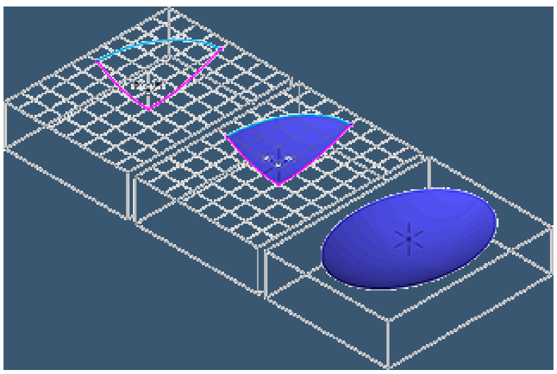
11. Save the part as the tutorial is now complete.

QUICK EXERCISES

- Building a Spherical Ellipse
- Replace History
- 2D Contour
- Advanced 3D, Using Curves

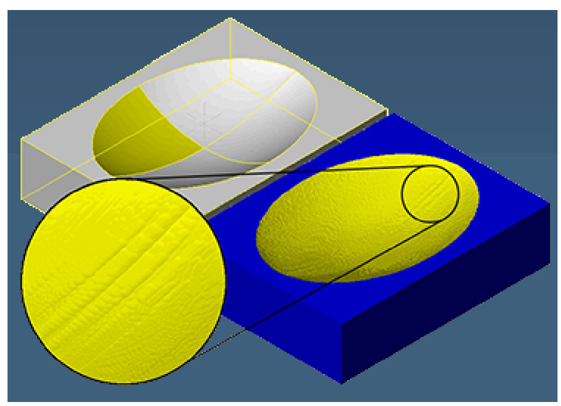
Building a Spherical Ellipse

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



The problem is that 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 creating a single continuous 3D flowing shape is to 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. Follow the steps below:

Step #1

1. Create an XZ plane.

2. Create points at X-35, Y0, Z0 then at X0, Y0, Z-8 and lastly X+35, Y0, 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.

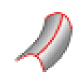
Step #2

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.

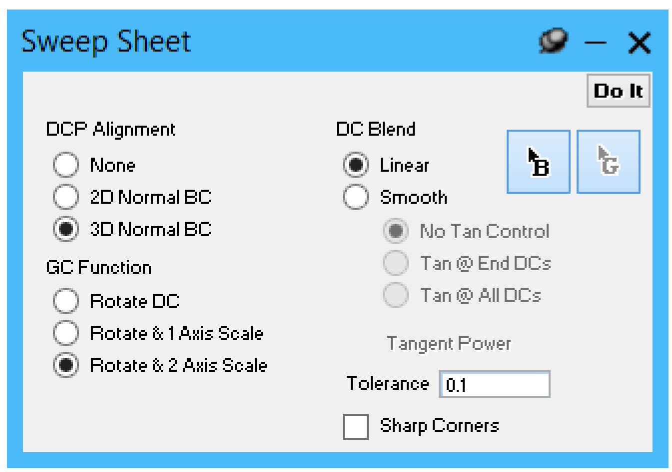
Step #3

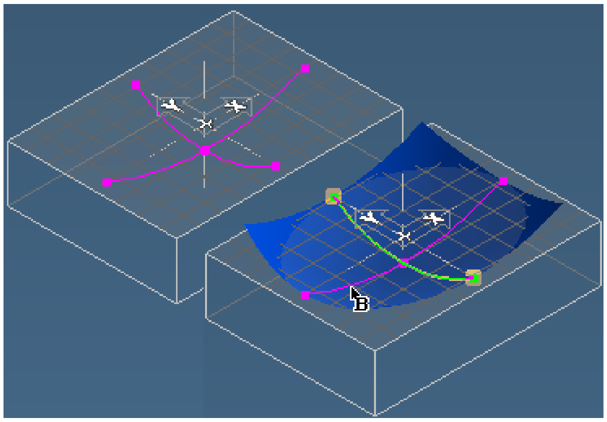
1. Switch to the XY plane.
2. Open the Surface Modeling palette.

3. Open the Sweep Sheet dialog.



4. Set the sweep options as shown.



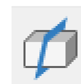


5. Place the base curve pointer on the curve shown and `Double-click` the other curve as the drive curve.

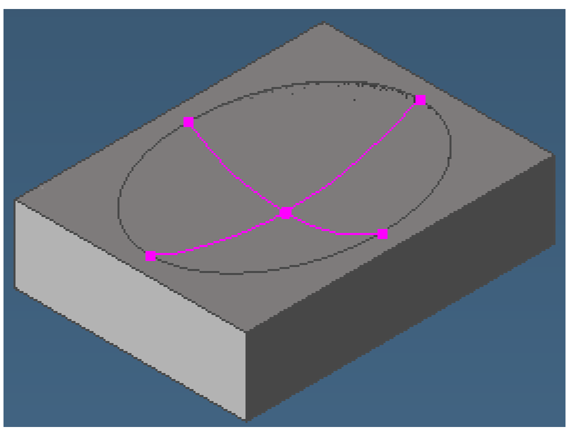
6. `Click` the Do It button to create the sheet.

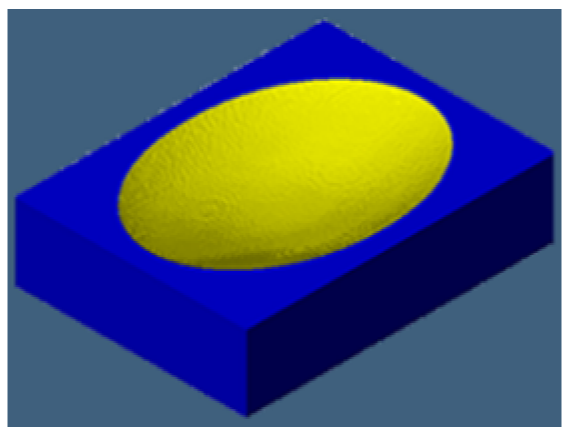
7. Create a cube based on the workspace stock size.

8. Slice the cube with the swept sheet.



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.

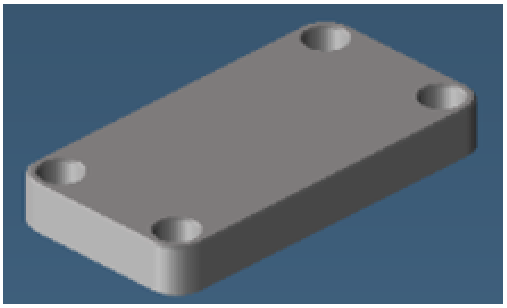
Replace History

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.

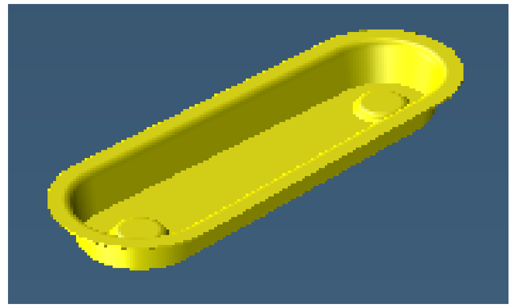
Part Creation

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

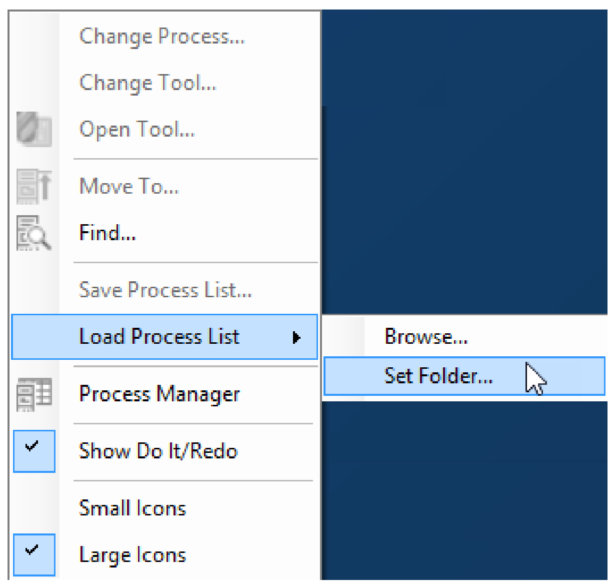
The first thing we will do is make the saved processes available for use.

2. Open the Sample Parts\Solids\Required\Swap Example folder and copy the Swap Processes folder to My Documents.

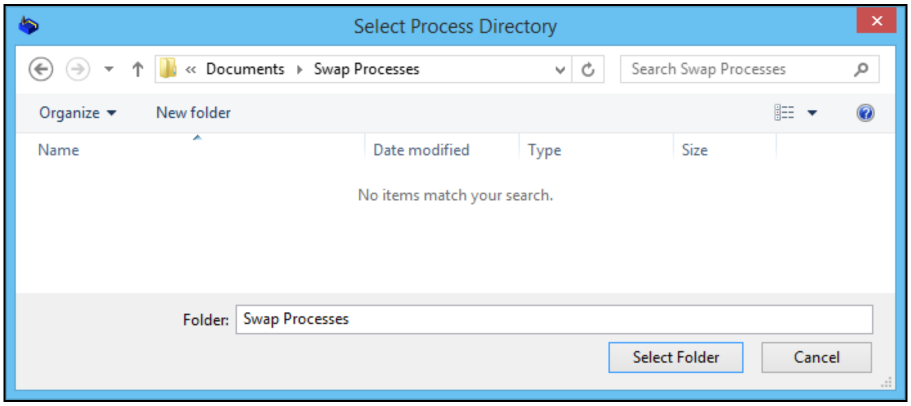
   Placing the saved processes here will make it easy to find them when we need them.

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.

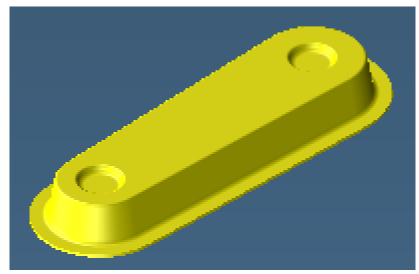


2. Navigate to the My Documents/ Swap Process directory and click Select Folder.



Unstitching and Subtracting "Pocket"

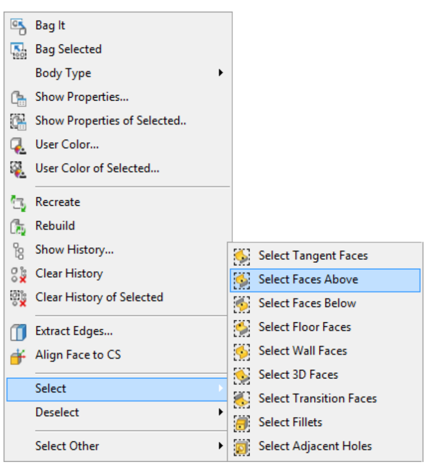
We are going to unstitch the body named "Pocket" to close off the shape, thereby creating a core. The interior topology is not important; it is the exterior shape we will be subtracting from the Block.

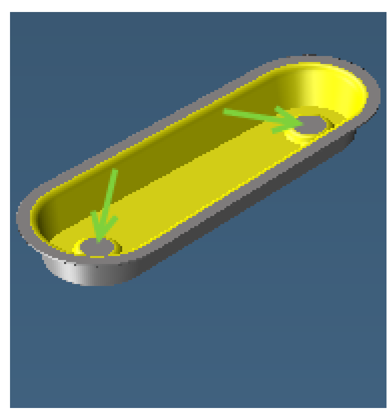


Opposite isometric view (`Ctrl+Alt+I`)

1. Turn on face selection and `Right-click` the bottom flat face of the pocket body.
2. Choose the Select Faces Above option.

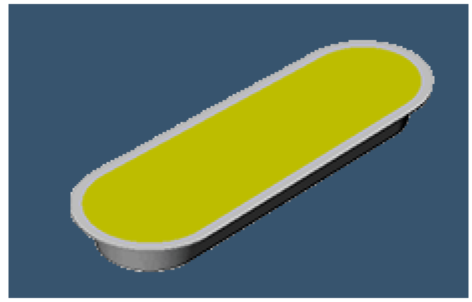
   All faces inside the pocket except the flats on top of the bosses should be selected.





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

4. Click the Unstitch button. 

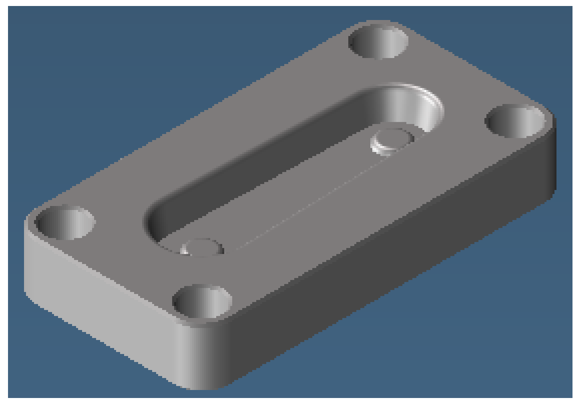


This results in two solids.

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.



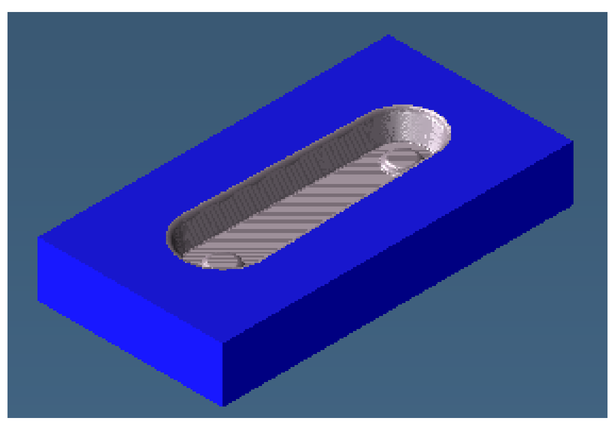
Machining the Part

Loading Processes.

1. From the process right-click menu, select Load Process List and select 1) Rough Pocket.prc.

This creates a tool and a process that will rough the pocket.

2. Select the faces in the pocket the same way we selected the faces for the unstitch and create the toolpath.



3. Deselect the operation and Delete the process tile.

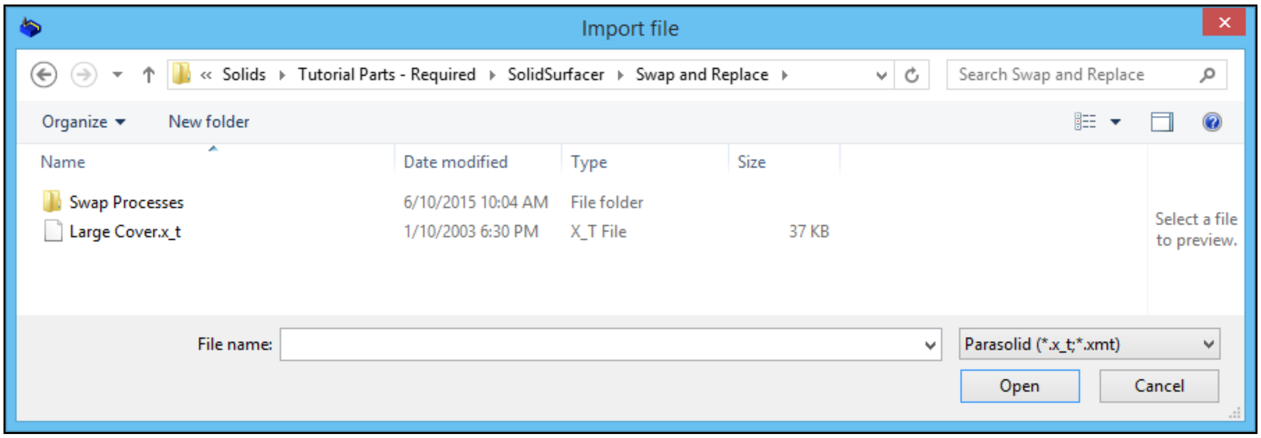
4. From the process right-click menu select Load Process List and select 2) Finish Pocket.prc and create the toolpath.
5. Render the operation.
6. Deselect the operations and place the model in the Body Bag.

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.

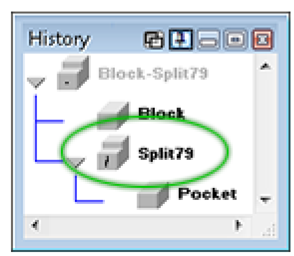
1. Import the file Large Cover.x_t from the Sample Parts/Solids/Tutorial Parts - Required/SolidSurfacer/Swap & Replace folder.



Replacing the Model History

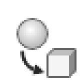
1. As with the smaller pocket shape, Right-click the bottom face and choose Select Faces Above and then select the tops of the two bosses.

2. Unstitch the solid to fill it and Delete the smaller solid.



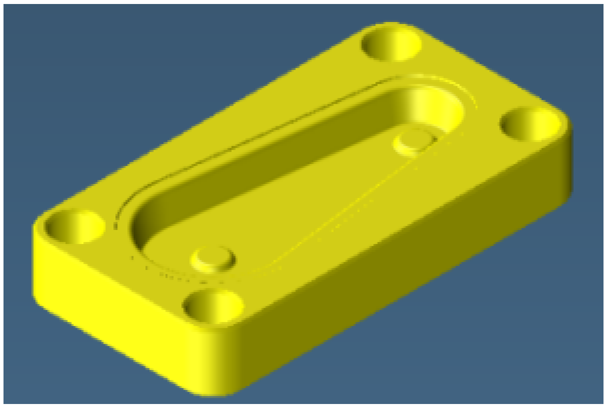
3. Extract the smaller filled pocket from the History of the Block.

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, etc.) we could just select the two original bodies and perform a swap or replace.

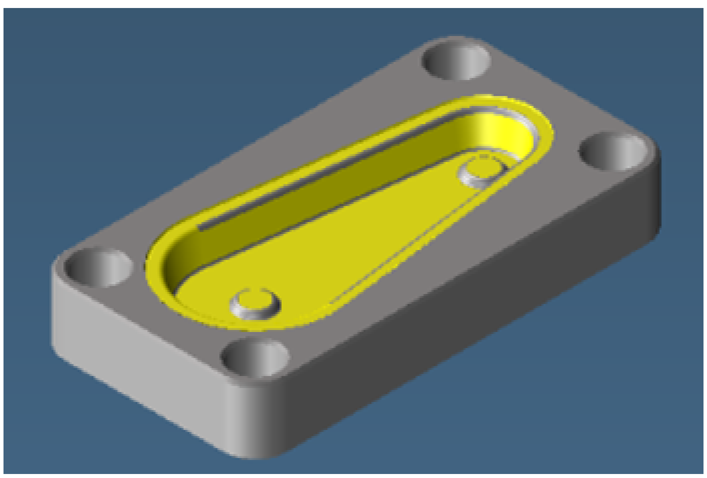
5. `Right-click` the Block model and choose Rebuild.



Redoing the Operations

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

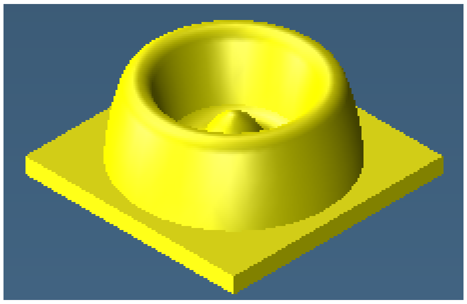
1. Select Edit > Redo All Ops and render the part.

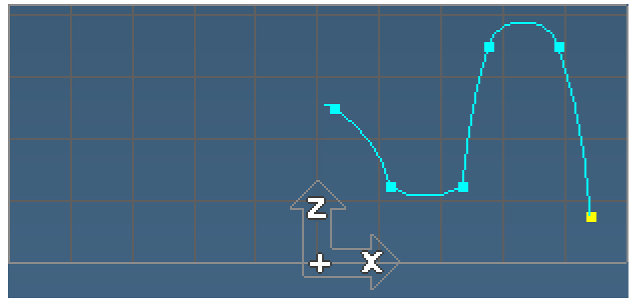


2. Save the file; it is complete.

2D Contour

This exercise shows the 2D optimize capability of the system which is for use on bodies which have 2D areas that we want to machine with arcs rather than several line segments. We will also use another surfacing corner cleanup option in this exercise. The part we will be making is shown here.





Model Creation

1. Create a new metric part with the following workspace stock size: X= 60, -X= -60; Y= 60, -Y= -60; Z= 50, -Z= 0. and a Clearance of 65.

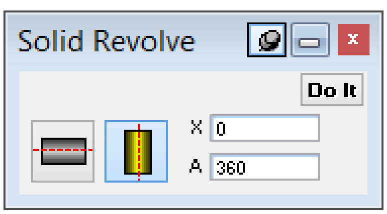
2. In the XZ plane we will create a contour. For the dimension and the part print for this, see “2D Contour” on page 167.

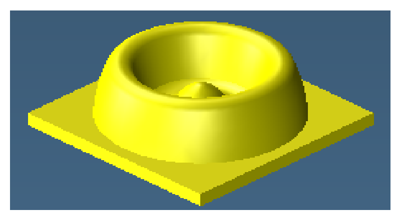
   We do not want to create the fillets on the 13mm arc because we will be using a tool and intersection machining to create those fillets. The intersection surfacing option can only be used on sharp corners which means that you do not want to draw any fillets that you wish to create with the tool.

Solid Model

1. Revolve the contour in the XZ plane as shown.

   Make sure that you are in the XZ plane and that you are revolving about the vertical axis. The resulting solid body will look like the picture shown here.





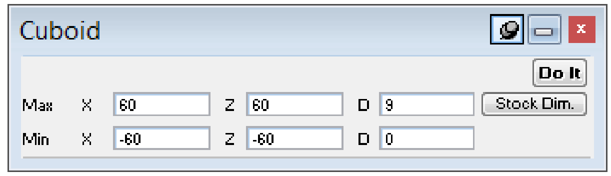
Base and Stock Bodies

Now, we will create a stock body by offsetting the revolved body.

1. Open the Offset/Shell dialog and enter the information shown.

2. Select the revolve and apply the Offset.
3. Place the offset in the Body Bag.

4. In the XY plane, create a cube based on the workspace stock dimensions but change the Max D value to 9mm.



This cube will be the base for the part. The same cuboid will be used for the stock body and the part body.

5. Select the cube and choose Modify >  Duplicate.

6. Extract the original revolve from the History list of the offset solid.

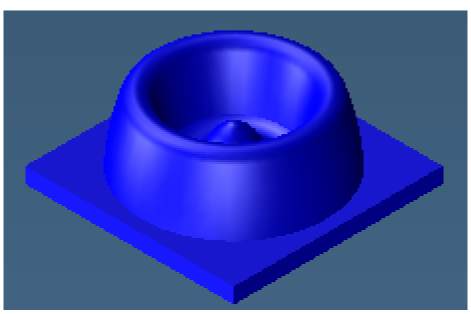
7. Add one of the cubes to the original revolve and add the other cube to the offset.

8. Open the Properties dialog and rename the offset `Stock`.

   The stock body should be only slightly larger.

9. In the Properties dialog select the Stock option.

   The stock model is now blue.



Creating Operations

Tool List

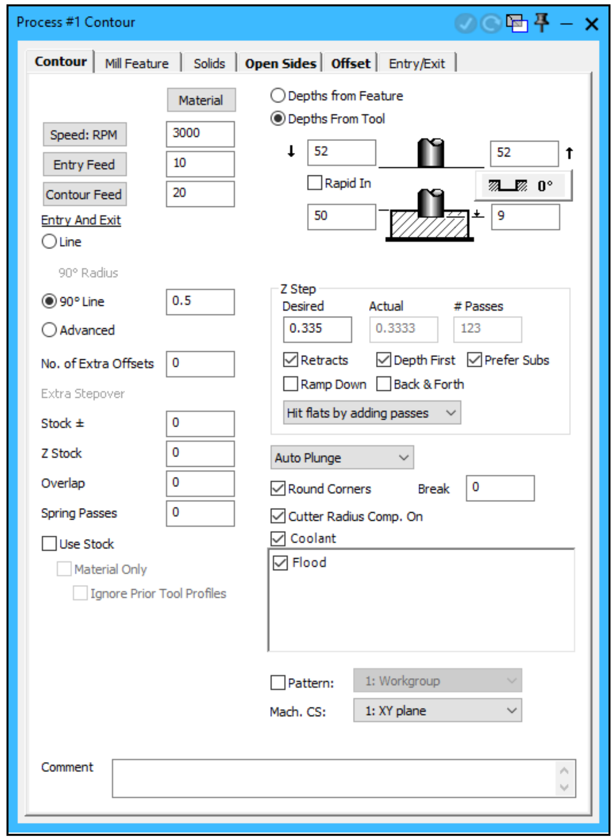
With the part and stock defined, we will create the machining operations which are intended to finish the part from the casting that is designated by our stock body.

1. Create the following tool list.

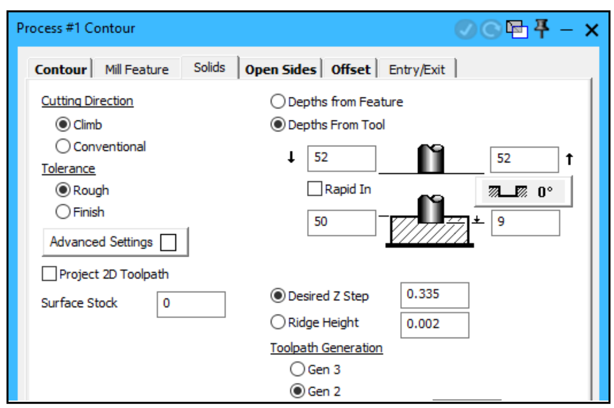
#	Type	Total Length	Diameter	Bottom Radius	# Flutes	Flute Length	Material
1	Ball EM	89mm	14mm	n/a	4	32mm	TiN Coated
2	Ball EM	63mm	8mm	n/a	4	20mm	TiN Coated

1-3, Contouring

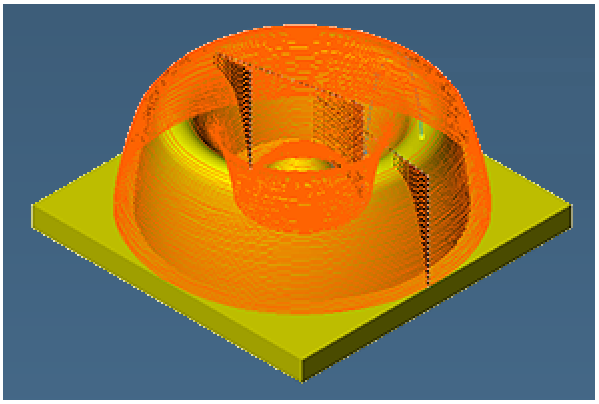
1. Create this Contour process with the 14mm Ball EM (Tool #1).



2. Click the Solids tab and enter the information shown.



3. Select the model and create the toolpath.



The system analyzes the selected body and identifies 2D and 2.5D areas of the solid that occur at the various Z depths. A 2D area on a part is defined as a face whose surface normal at every point lies in the XY plane, such as a vertical wall. A 2.5D face is defined as a face where all the surface normals along a constant Z line maintain the same angle with the tool axis. In this case, the faces of the part are all 2.5D and will be machined with arcs.

Three operations will be created which contour the part.

To verify the toolpath the Selected OP to Screen Points will draw points at the beginning of each tool motion in the toolpath.

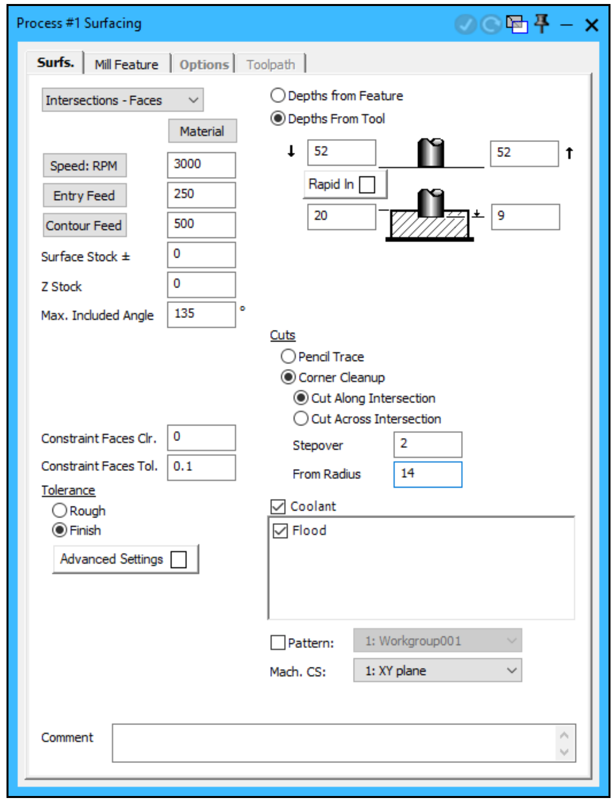
4. Choose the Solids > Tools > Selected OP to Screen Points item.

   Notice that a single string of points is created along the body indicating that the toolpath is composed of arc moves rather than several small line moves. If non-2D toolpath was generated, you would see several hundred points covering the toolpath, because it is composed of lines.

4, Surfacing

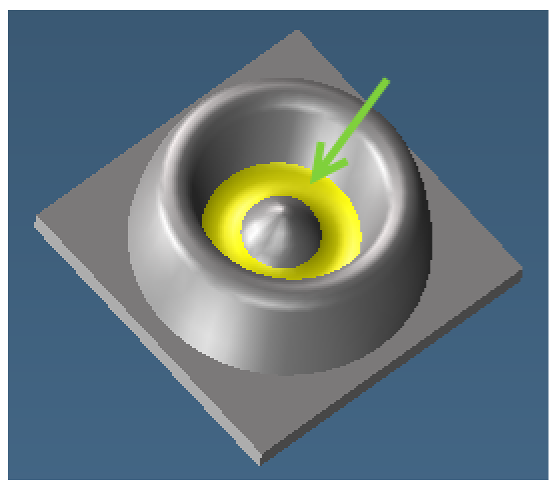
Now we will create the fillets in the cavity of the part using an Intersections operation. We will generate the operation from a face of the part.

1. Create this Surfacing process with the 8mm Ball EM (Tool #2).

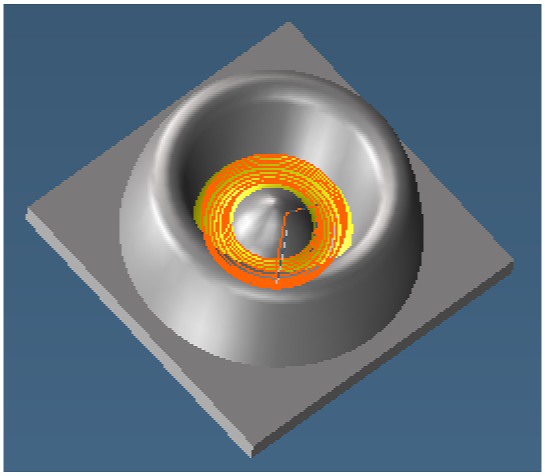


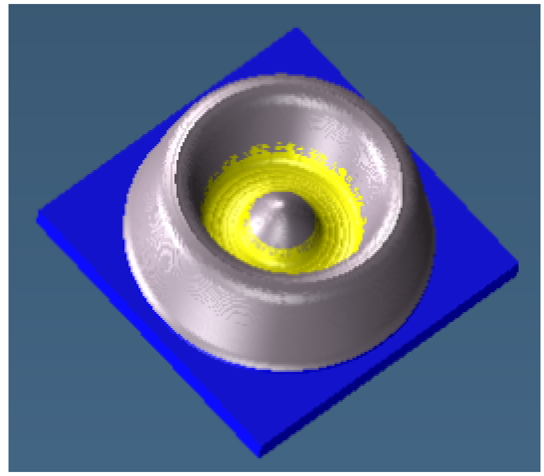
Corner Cleanup and Cut Along Intersection will create toolpath that runs along the edges of any selected faces, cleaning up the intersection between the selected and neighboring faces. The radius of the previous tool is entered (14mm) and a stepover is set.

2. Select the face shown and create the toolpath.



The resulting toolpath and the rendered part will look like the following images.





3. Save this file as `2D Contour.vnc`.

Advanced 3D, Using Curves

- #1: N-Curve Flow
- #2: Curve Projection

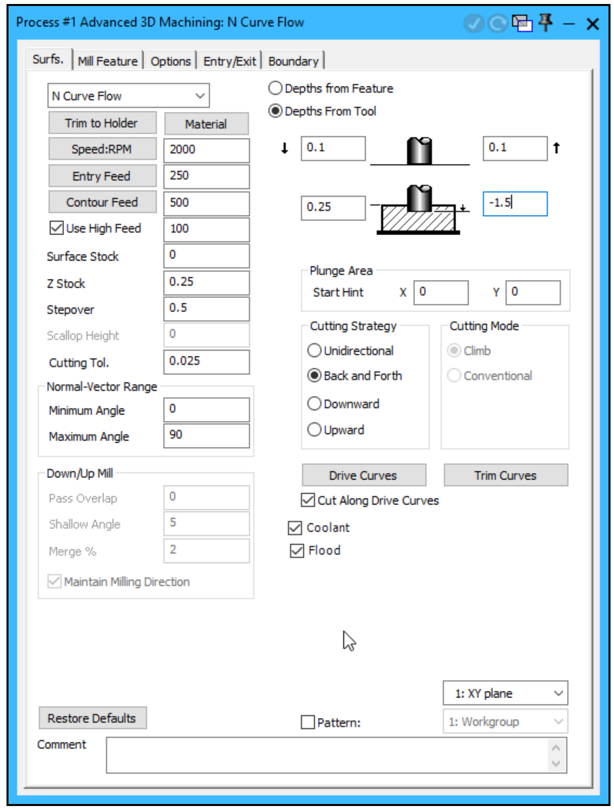
#1: N-Curve Flow

1. Open the file Adv3DExtras.vnc

   The geometry in Workgroup #1 consists of four open shapes positioned on the top surface of the core.

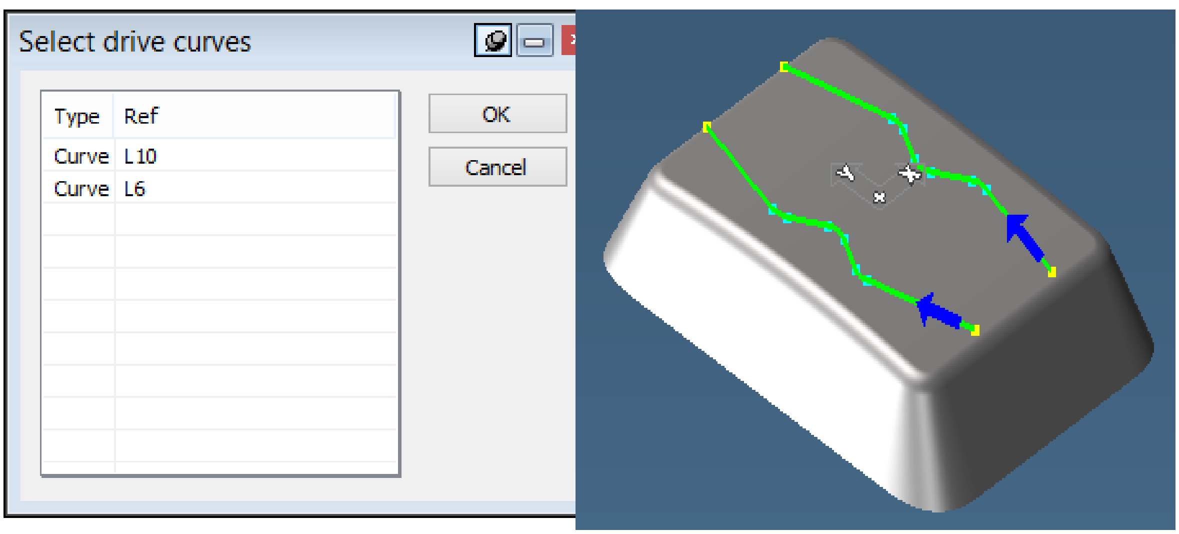
2. Create an Advanced 3D Machining process using the 12.5mm Ball EM (Tool #1) and set the Operation Type to N Curve Flow.

3. Click Restore Defaults and then enter values as shown.



4. Click the Drive Curves button to open the Select Drive Curves window.

5. Click one of the drive curves and then (Ctrl+click) the other, as shown.



6. Click OK to close the Select Drive Curves window.

7. If necessary, deselect the Face Selection button.

8. (Ctrl+click) the solid model so as to select all faces.

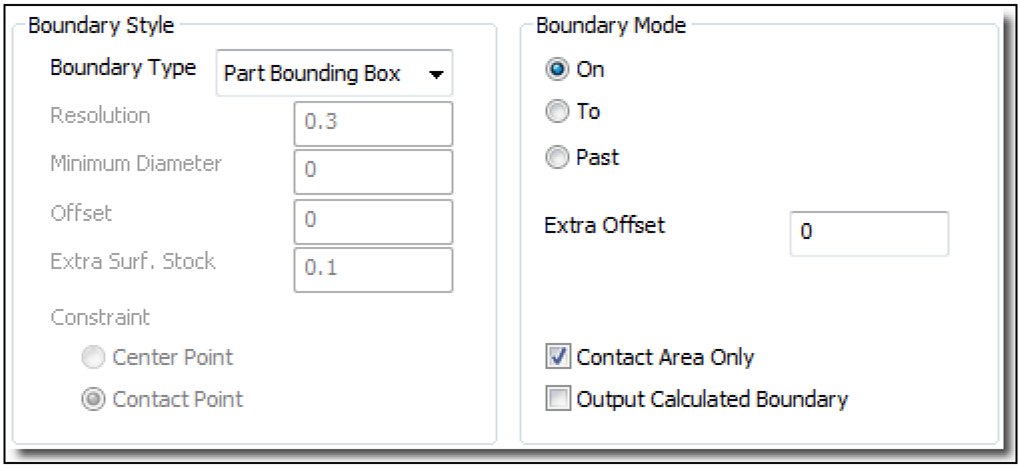
9. Click the Options tab and verify that Connect Move Control is set to Smooth.

10. Click Entry/Exit and ensure that Entry Style and Exit Style are set to Vertical.

11. Click the Boundary tab.

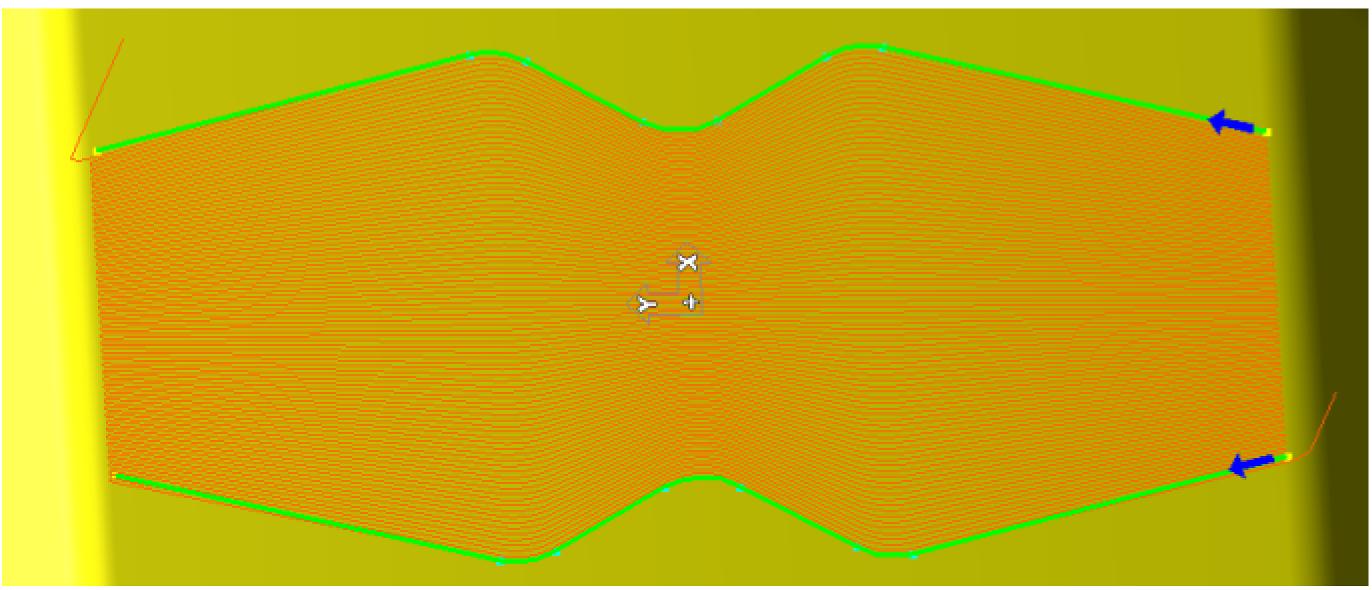
    Notice under Boundary Style that the Boundary Type is set to Part Bounding Box.

12. Set the Boundary Mode option to On and enter a value of 0 for Extra Offset.



13. Close the process and then click Do It to generate the operation.

    Notice that the resulting toolpath has morphed between the drive curves. In the Surfs tab, the Cut Along Drive Curves checkbox was selected, causing the toolpath to run approximately parallel to the drive curves. If the checkbox Cut Along Drive Curves were not selected, the toolpath would run generally perpendicular to the drive curves.

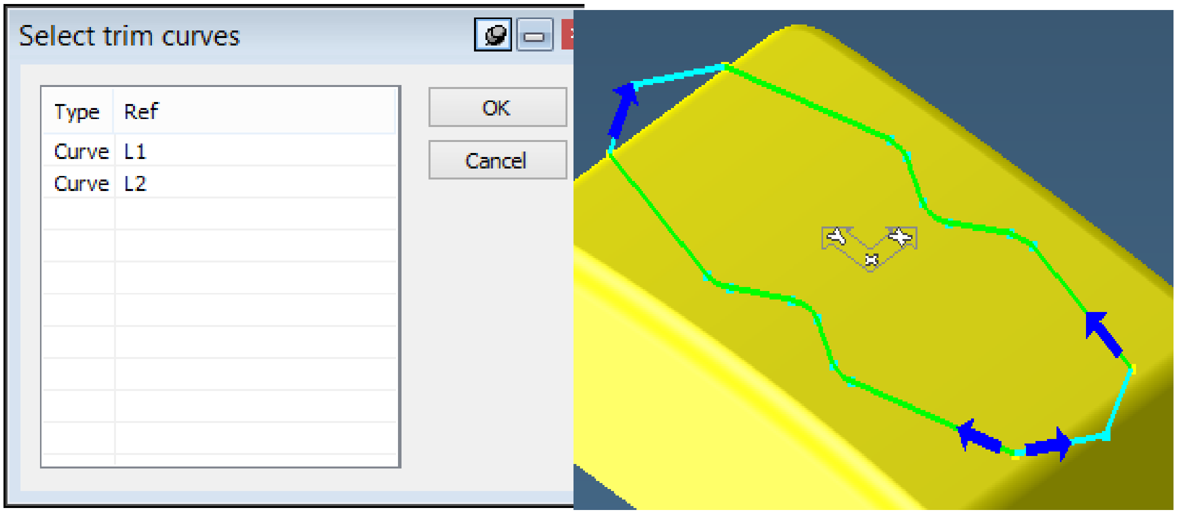


Now you will change this operation to use both drive curves and trim curves. When working with both Drive Curves and Trim Curves, each curve must be a single, open shape, and the endpoints of each trim curve must be at the exact same location (XYZ) of the endpoints of each drive curve.

14. Open the process, Surfs tab.

15. Click the Trim Curves button to open the Select Trim Curves window.

16. Click one of the trim curves and then (Ctrl+click) the other, as shown.

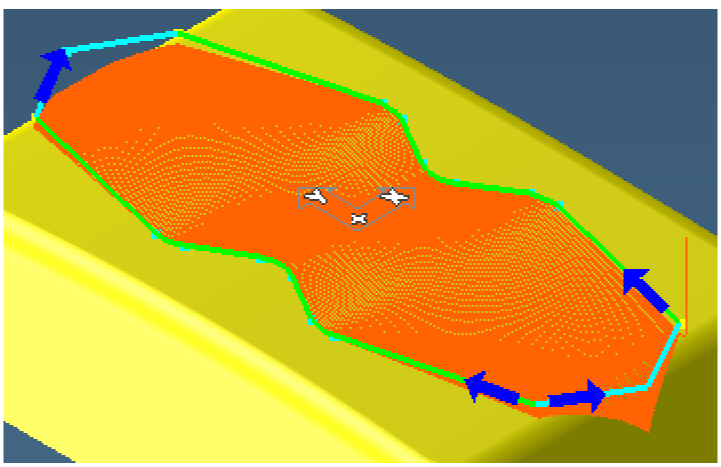


17. Click OK to close the Select Trim Curves window.

18. Select the body if needed.

19. Close the process and then click Redo to regenerate the operation.

    Notice the resulting toolpath has extended to the trim curves.



20. Save the part.

#2: Curve Projection

1. Select Workgroup #2.

   The geometry consists of two open shapes positioned on the top surface of the core.

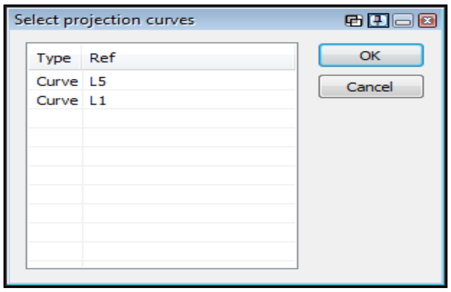
2. Open the process, Surfs tab, and set the operation type to Curve Projection.

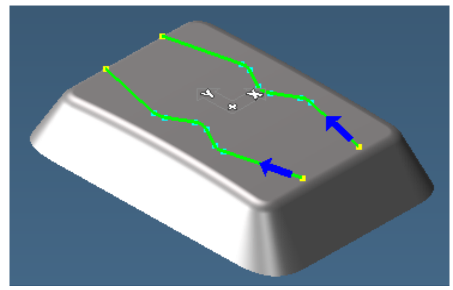
3. Click Restore Defaults.

   Notice that Stepover is grayed out; you cannot enter a stepover value.

4. Click the Projection Curves button to open the Select Projection Curves window.

5. Click one of the geometry projection curves and then (Ctrl+click) the other, as shown.



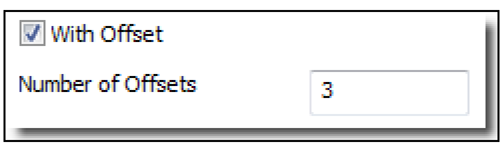


6. Click OK to close the Select Projection Curves window.

7. Select the With Offset checkbox and enter the value 3 for Number of Offsets.

   Notice that Stepover is now available.

8. Enter the value 0.125 for Stepover.



   If the With Offset checkbox were not selected, Curve Projection would make one single pass centered on the selected projection curves. When the With Offset checkbox is selected, Curve Projection will make a number of passes on each side of the projection curve that is equal to the number of offsets specified, and each of the offsets will be spaced by the Stepover value. Also, Curve Projection will make one single pass centered on the selected projection curve. In this case, with Number of Offsets equal to 3, the process will create seven toolpaths: one on the center and three on each side, all of which will be spaced by the Stepover value.

9. If necessary, deselect the Face Selection button.

10. (Ctrl+click) the solid model so as to select all faces, as shown above.

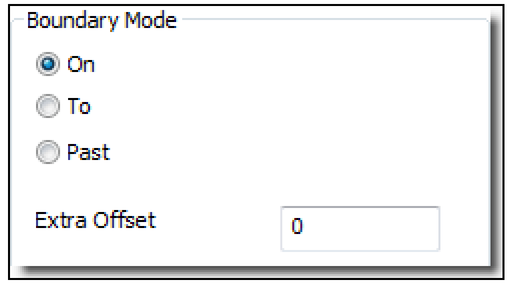
11. Click Options and verify that Connect Move Control is set to Smooth.

12. Click Entry/Exit and ensure that Entry Style and Exit Style are set to Vertical.

13. Click the Boundary tab.

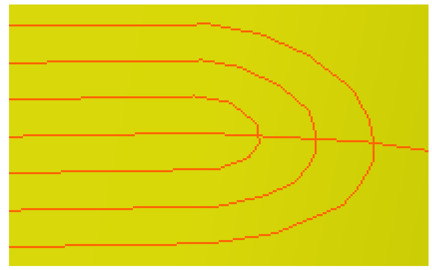
   Notice under Boundary Style that the Boundary Type is set to Part Bounding Box.

14. Set the Boundary Mode option to On and enter a value of 0 for Extra Offset.



15. Close the process and then click Do It to generate a new operation.

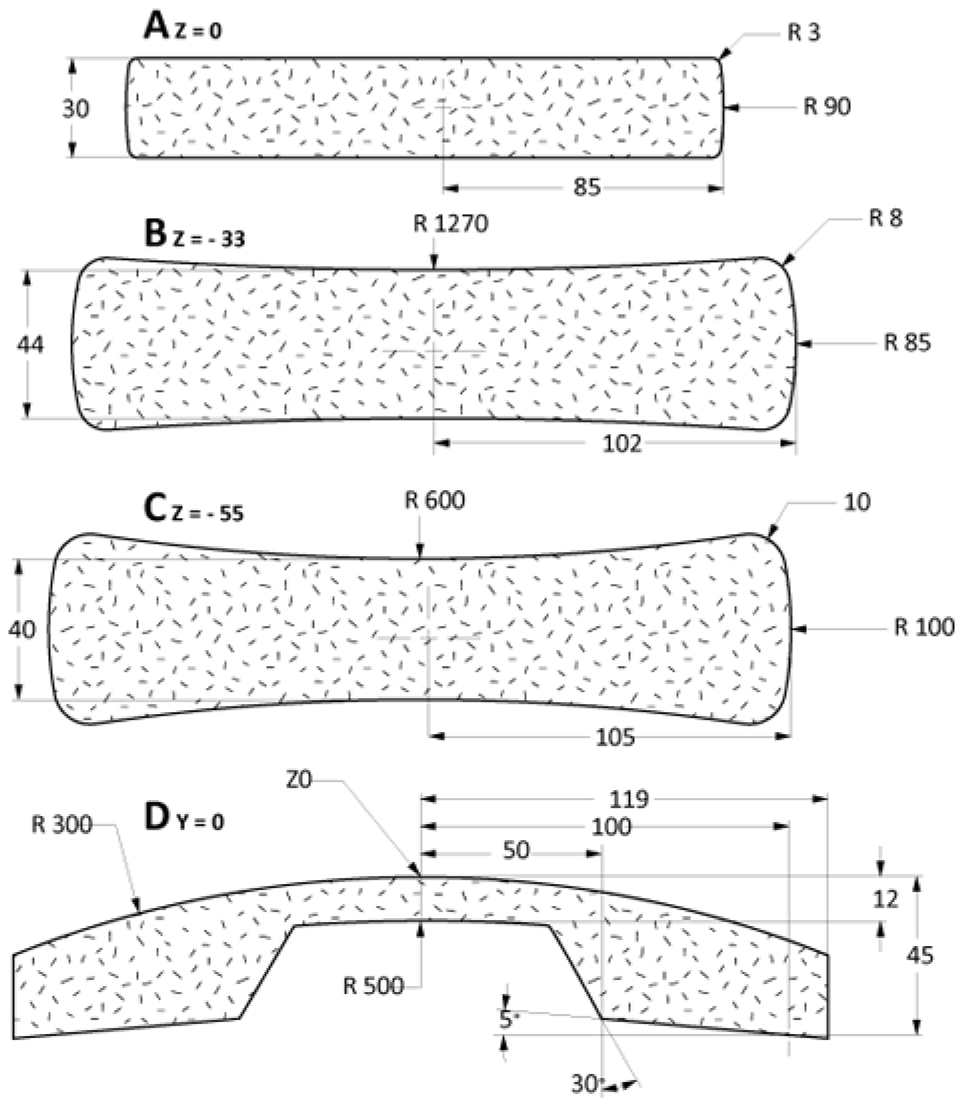
    If you zoom in, you can see that the calculation has generated seven toolpaths for each projection curve.

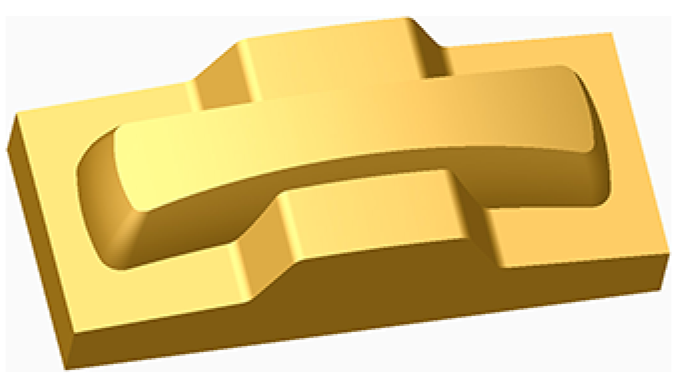


16. Save the part.

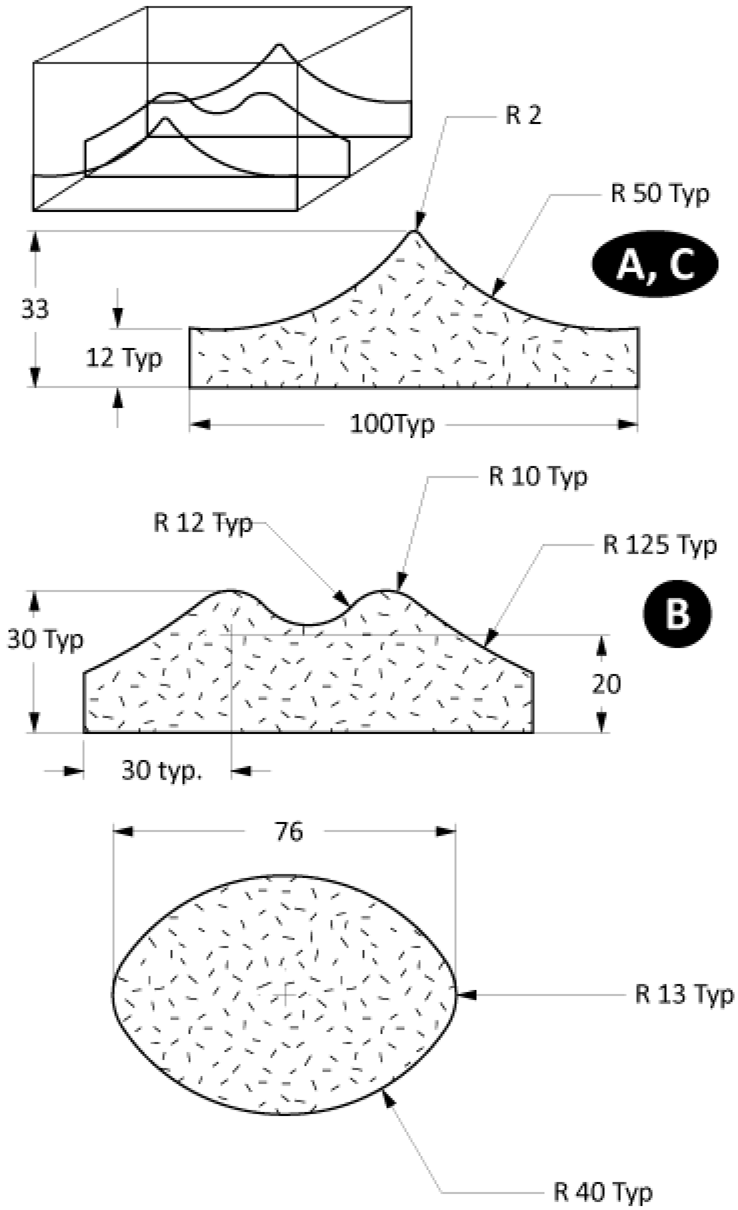
PART PRINTS

Phone

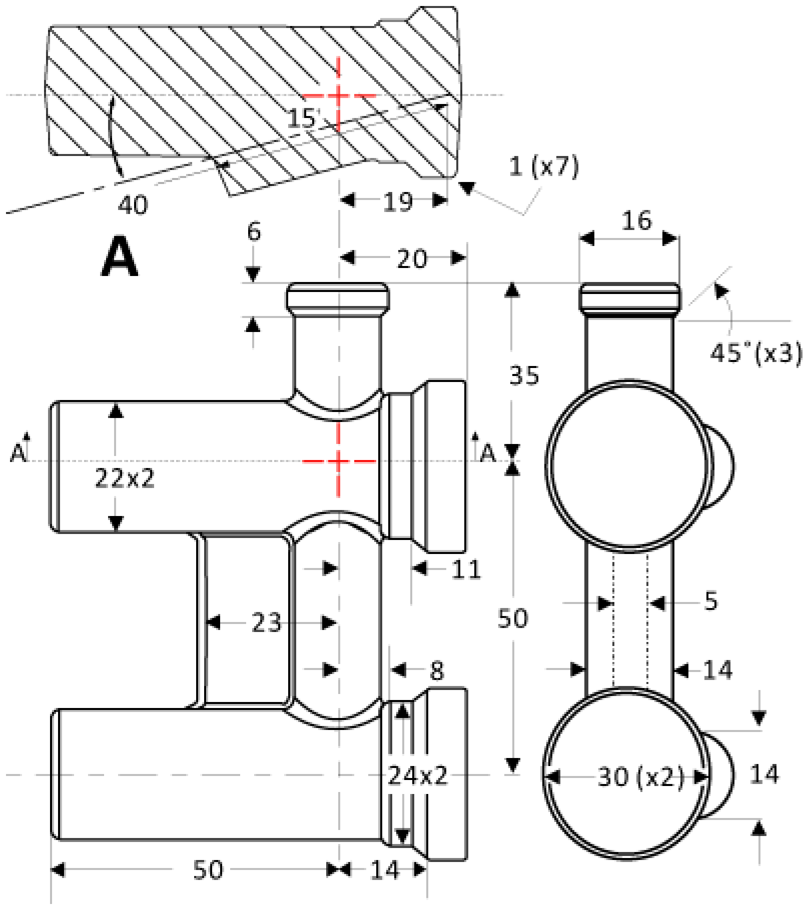


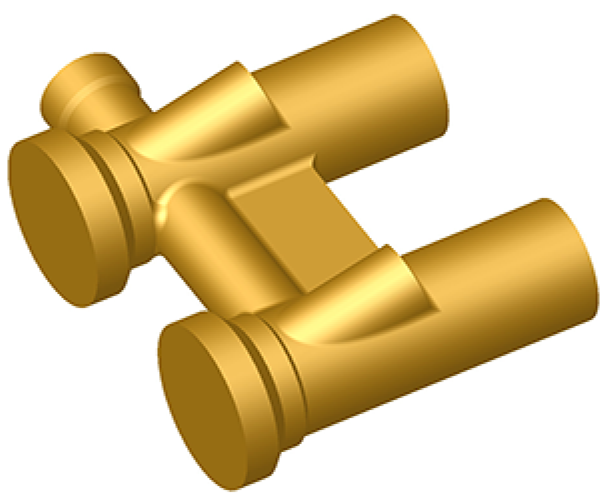


Hot Punch

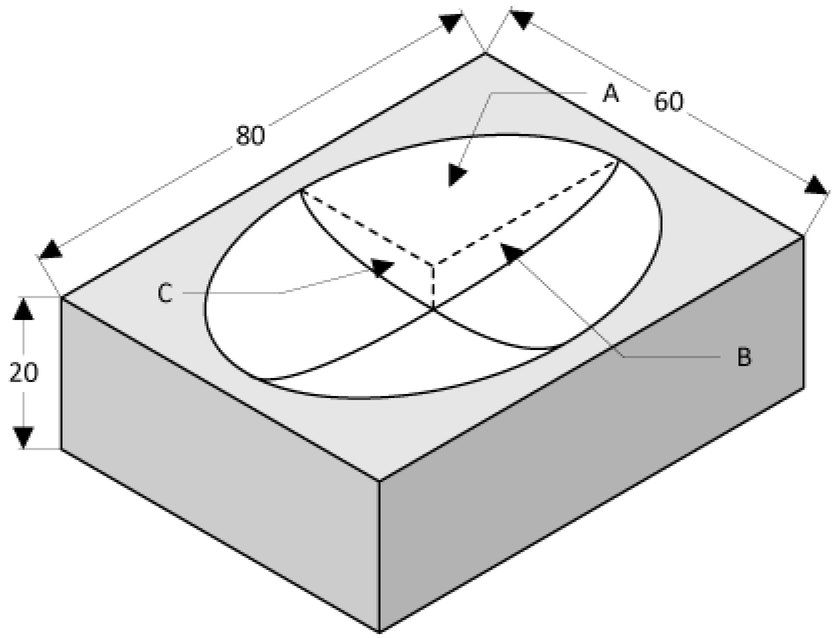

R 2
R 50 Typ
A, C
33
12 Typ
100Typ
R 10 Typ
R 12 Typ
R 125 Typ
B
30 Typ
20
30 typ.
76
R 13 Typ
R 40 Typ

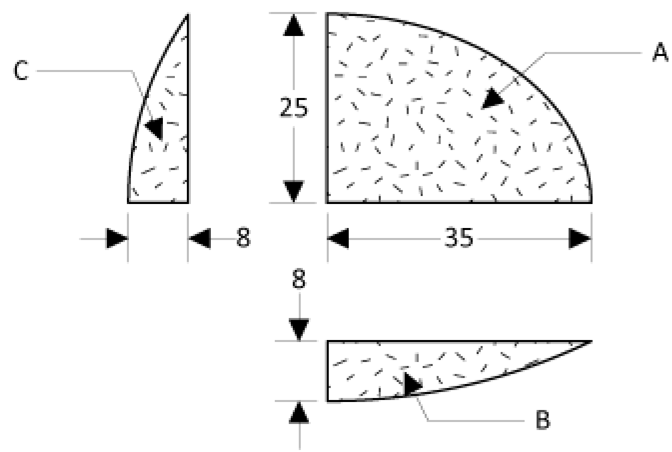
Plumbing


15'
1 (x7)
40
19
A
6
20
16
45°(x3)
35
A
A
22x2
11
5
50
23
14
8
24x2
30 (x2)
14
50
14



Spherical Ellipse





2D Contour