GIBBSCAM® 14

Version 14 : September 2020

Mill/Turn Tutorials

CAMBRIO

Proprietary Notice

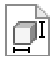
MILL/TURN TUTORIALS

The following tutorials will demonstrate how to create milling and rotary milling operations on Mill/Turn parts. There is little lathe/turning work included in these exercises. All lathe operations are exactly the same as the standard lathe interface for Mill/Turn parts. It is assumed that you have already completed the Mill and Turning tutorials and are therefore have a basic knowledge of GibbsCAM.

EXERCISE 1: CLUTCH BASKET

In order to complete this exercise, you will need to have installed the Sample parts.

1. Open the Sample parts/Production/Tutorial Parts - Required/MillTurn Parts/Clutch Basket.vnc file.

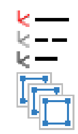


In the Document Control dialog, the Machine selected is a C Axis Horizontal Lathe - Generic Shank. When one of the C Axis MDDs is used, four coordinate systems are automatically created by the system: ZX plane, XY plane, HY backside plane and YZ plane. These coordinate systems are used to properly position geometry in order to cut a standard Mill/Turn part. More functionality is available if Advanced CS is available.

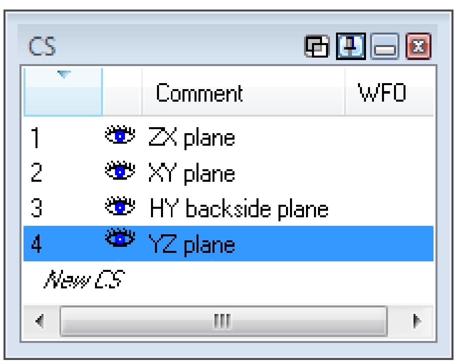
All the tools required have already been defined.

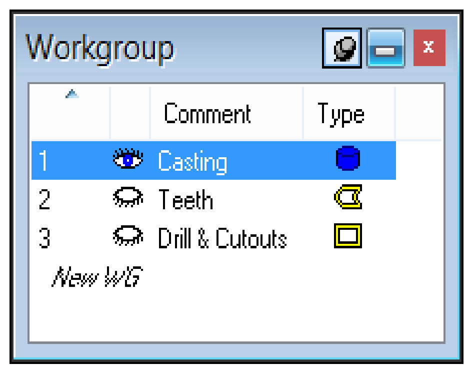
About the Part

1. Open the CS list and Workgroup list from the commands palette.



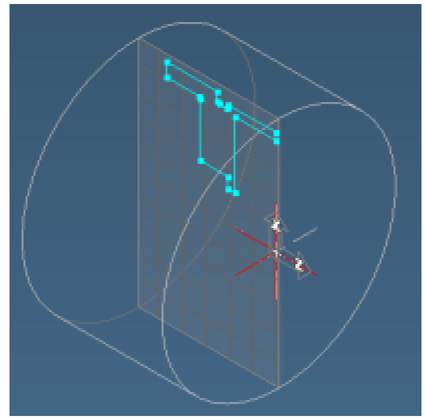
2. Or from the status bar.



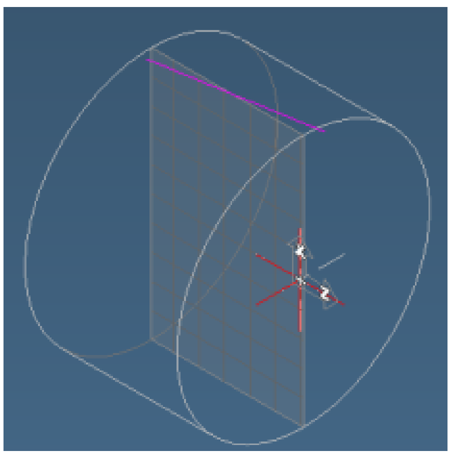


Workgroup 1

This workgroup is designated as revolved Part Stock in the WG info dialog. The geometry is defined in the ZX plane which is the standard plane for turning operations. This is the profile for the initial stock condition.



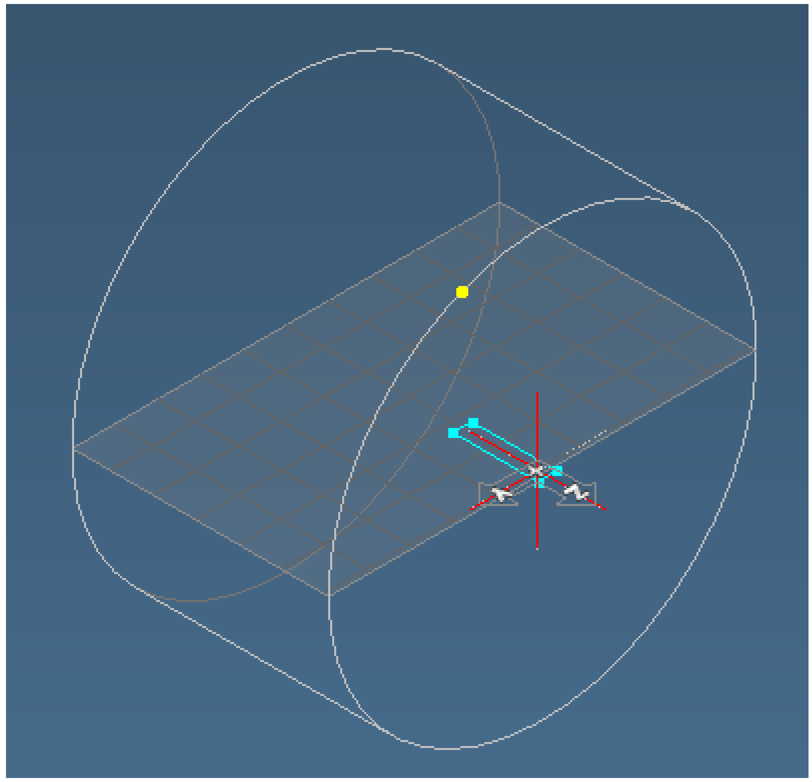
Workgroup 2



This workgroup contains geometry which will be used to create angled slots on the OD of the part.

Workgroup 3

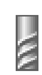
This workgroup contains geometry defined which will be used to cut other slots on the OD of the part. There is also a point which will be used to create bolt holes on the front of the part.



Machining the Part

#1 Main Slots

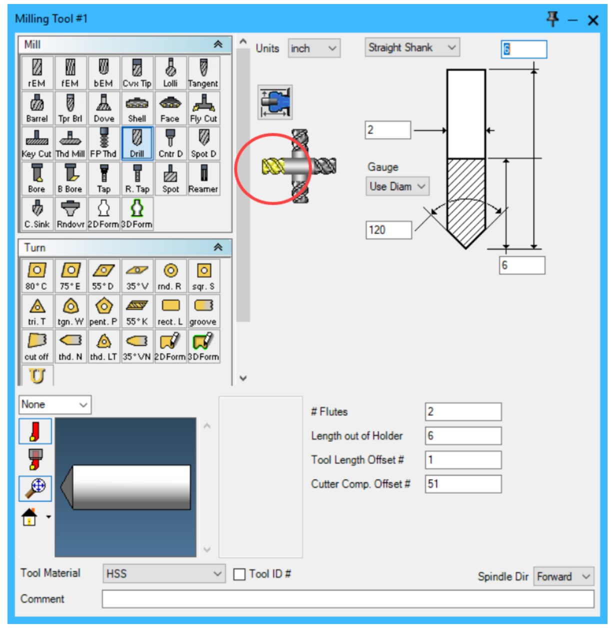
1. Open the Tool list.



There are five tools in the tool list including a 2" Drill, 0.25" Rough Endmill, 1" Roundover Tool, 0.625" Spot Drill and 0.375" Drill.

2. Open Tool #1 dialog.

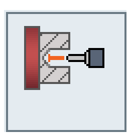
The first operation will be to drill out the center of the part so the tool orientation for the first four tools is set to approach along the Z Axis. The tool orientation diagram will look like the following image.

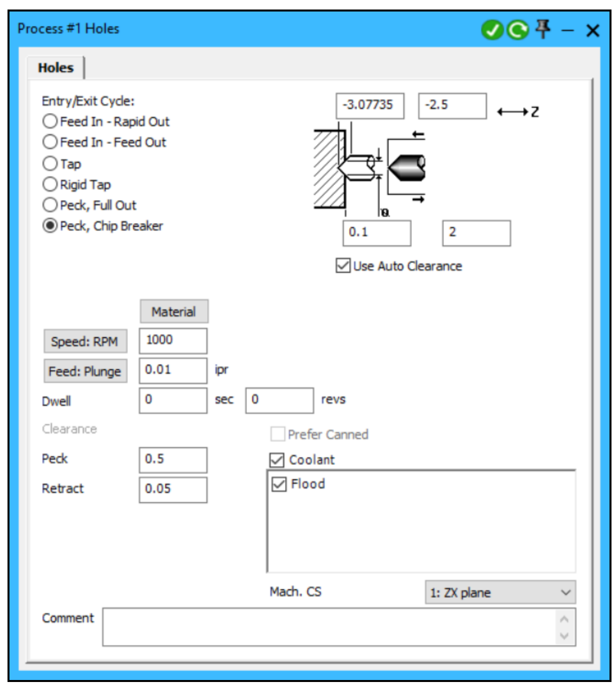


This orientation designates a Z Axis approach which is the necessary orientation for front face operations.

3. Click the CAM button and double-click a Process tile.

4. Create a Lathe Holes process with the 2in Drill (Tool #1).





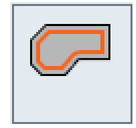
5. Create the toolpath.

The next operation to be created will be a Mill contour process.

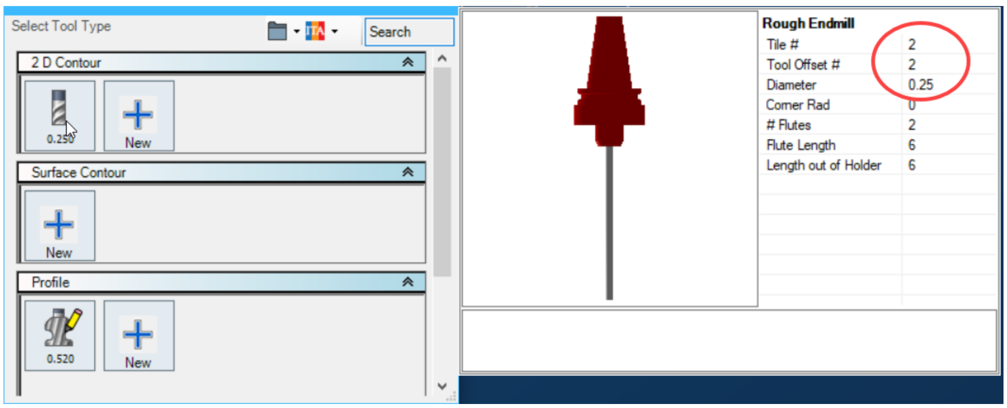
#2 Main Slots

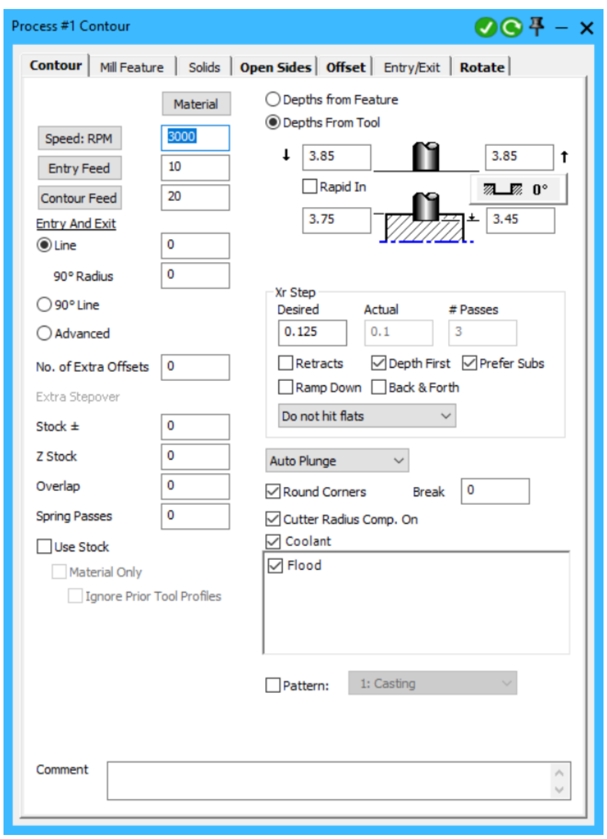
1. Click Clear to delete the holes process.

2. Double-click Process tile and choose a Mill Contour Process.



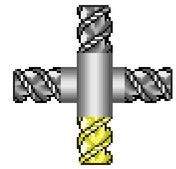
Choose the 1/4” Rough Endmill (Tool #2) If you hover over the suggested 2D Contour tool a brief tool summary is displayed enabling you to confirm this is indeed the correct tool.



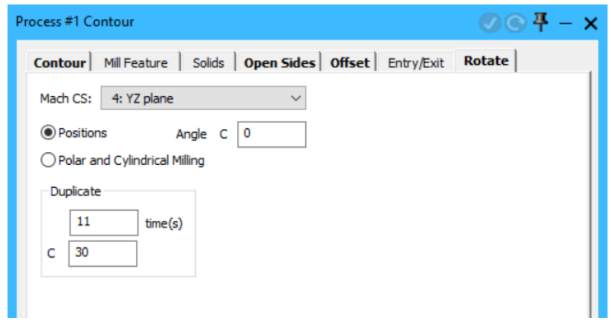


3. Use the parameters shown in the Process dialog.

The tool orientation designates an X Axis approach which is the necessary orientation for OD operations.

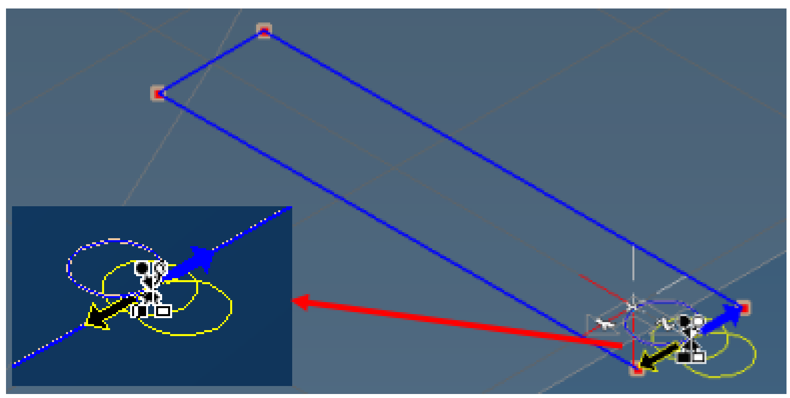


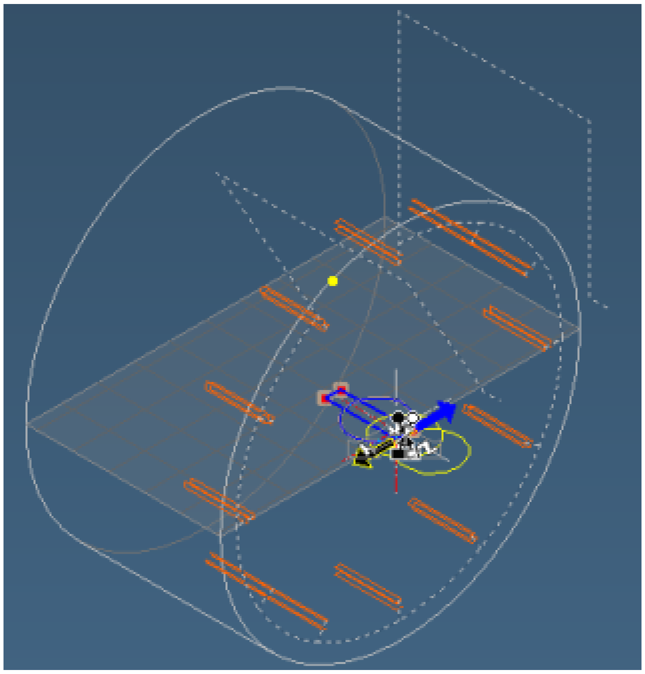
4. Switch to Workgroup 3: Drill & Cutouts.
5. Switch to CS4: YZ plane.
6. Enter this information in the Rotate tab of the Contour Process.



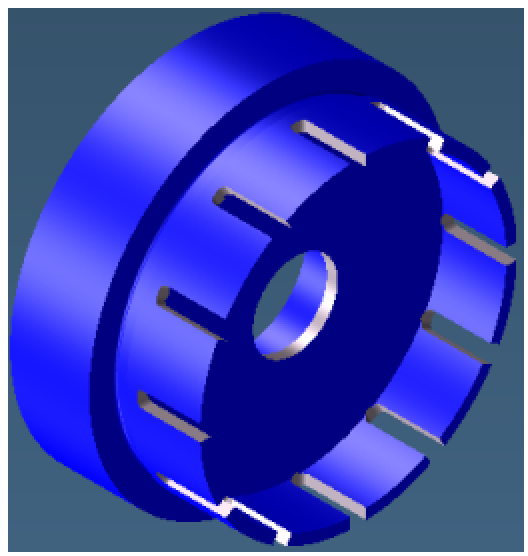
This will create an operation that starts at C0 then gets rotated clockwise by 30° and machined 11 more times.

7. Select the geometry shown.
8. Set the machining markers as shown.





9. Create the toolpath.

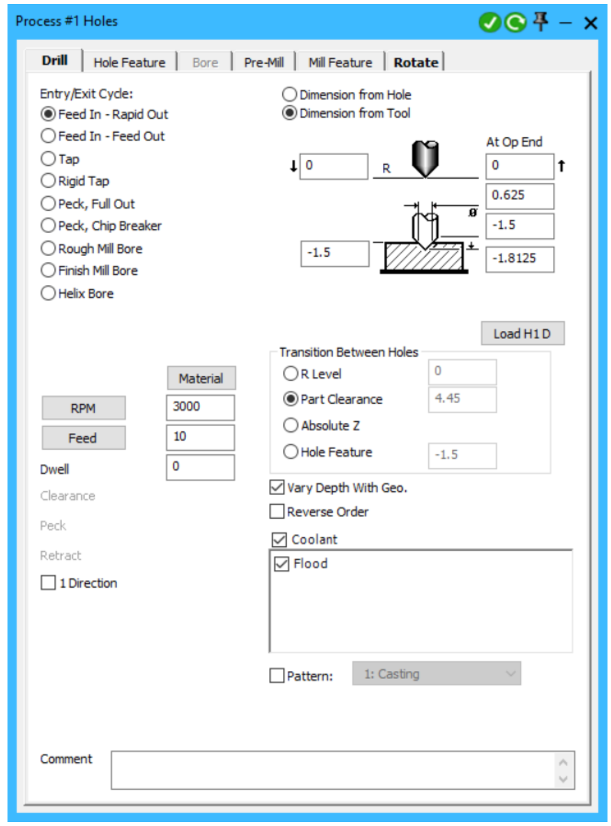


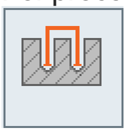
10. Render the operations, using Op Sim  then clear the process list Clear.

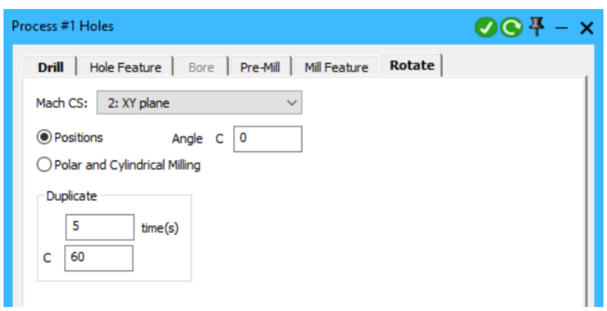
#3-4 Bolt Holes

The next two operations will be a Multi-process operation.

1. Switch to CS2: XY plane.

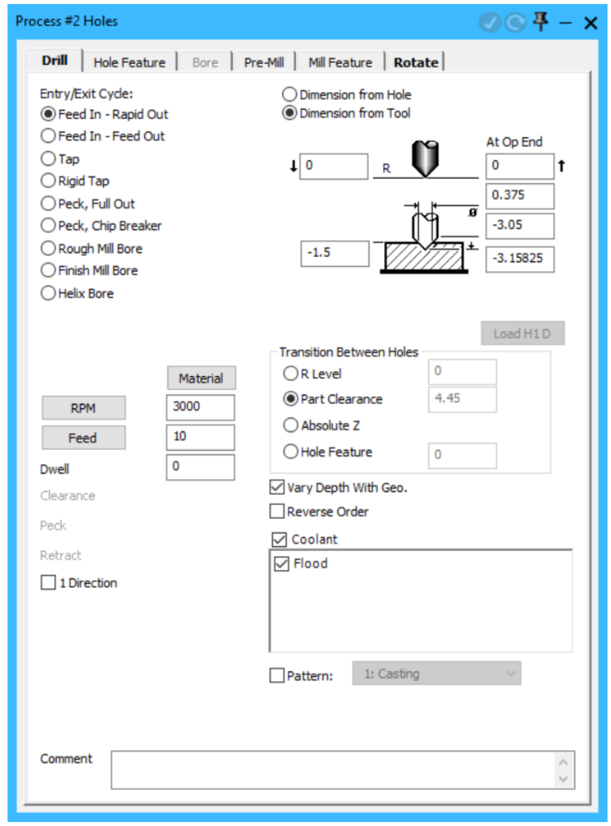


2. For process #1, create this Hole process  with the .625” Spot Drill (Tool #4).



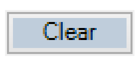
3. Enter this information in the Rotate tab.

4. For process #2, create this Holes process with the .375in Drill (Tool #5).



In a Multi-process operation, the Rotate tab will copy the last-entered values for the process type into the next process in the list of the same type.

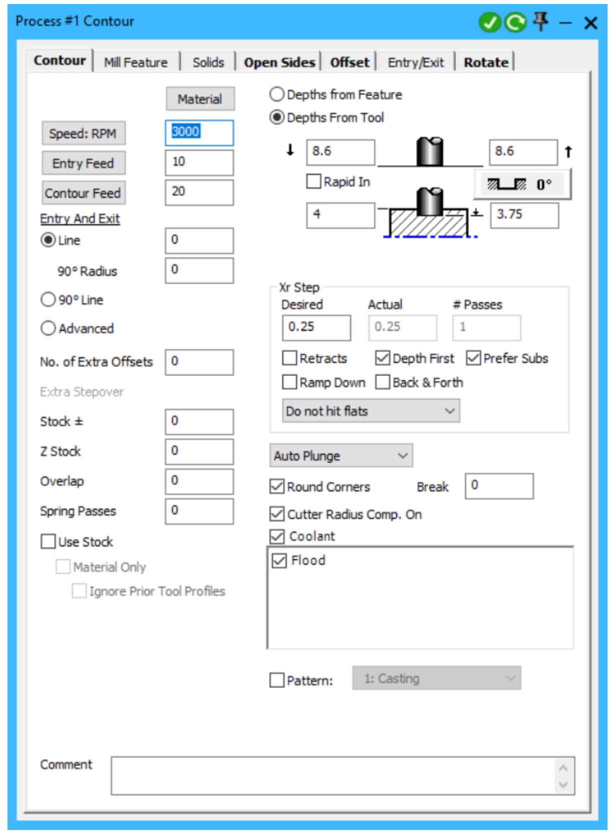
5. Select the point shown.

6. Create the toolpath then clear the process list .

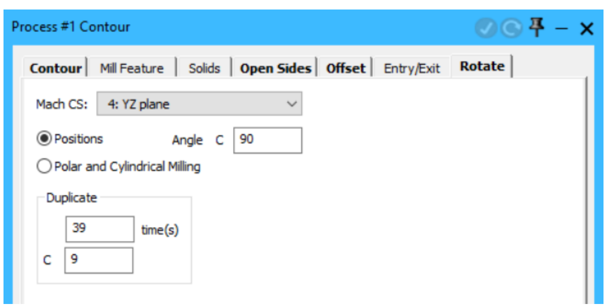
#5 Teeth

1. Switch to CS4: YZ plane.

2. Switch to WG2: Teeth.

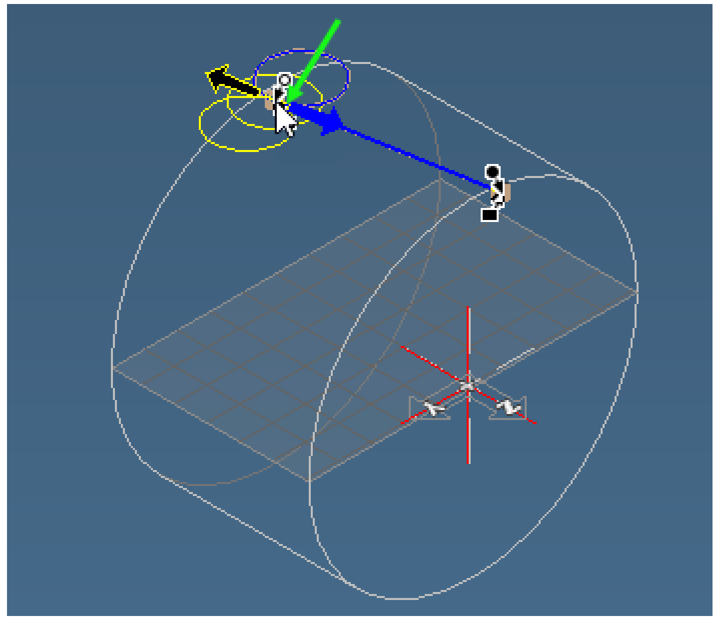


3. Create this Mill Contour process with the 1” Roundover Tool (Tool #3).



4. Enter this information in the Rotate tab.

This will create a contour operation that starts at C90° and repeats 39 times in 9° intervals.

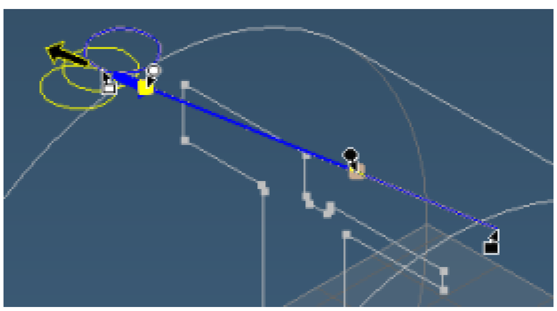


5. Click the line as shown

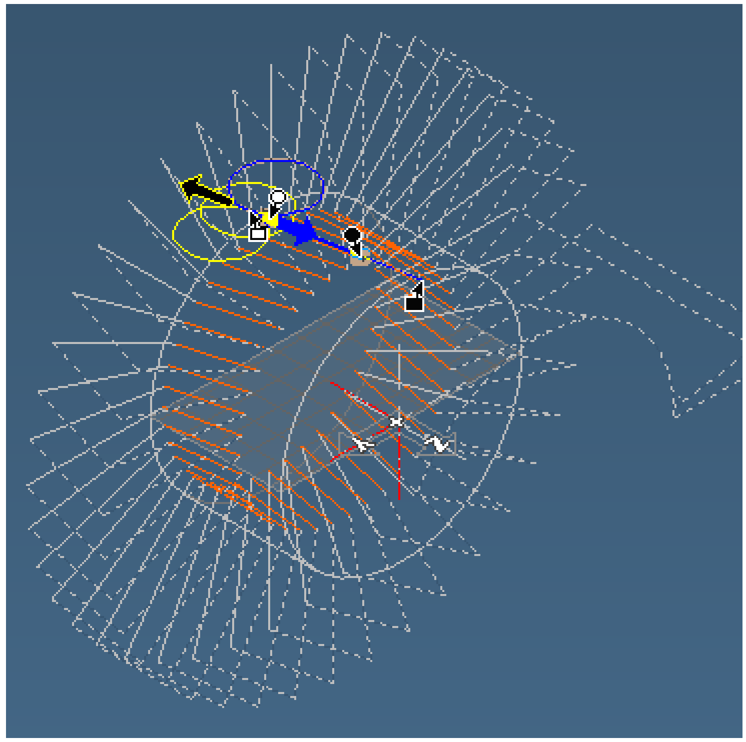
6. The markers should extend past the stock geometry.

7. `Double-click` the eye icon for the WG1.

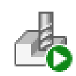
This will help you to properly place the markers.

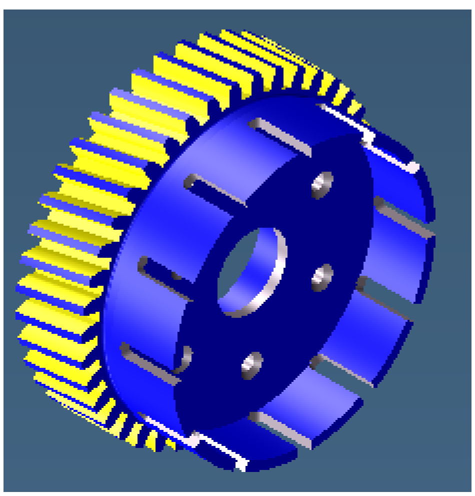


8. Create the toolpath.



9. Render the operations.





10. Save the part.

EXERCISE 2: COUPLING

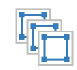
This exercise gives examples for using front face milling and rotary milling operations on mill/turn parts.

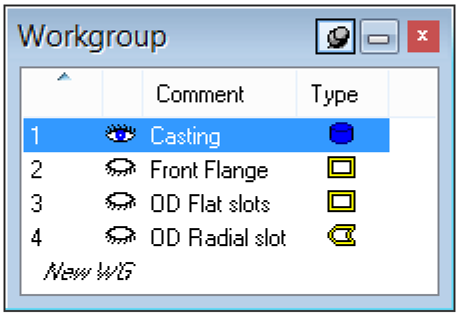
1. Open the Coupling.vnc file.

   The Machine selected is a C Axis Horizontal Lathe - Generic Shank. The file contains all the necessary geometry and tools to machine this part.

About the Part

1. Open the Workgroup list.

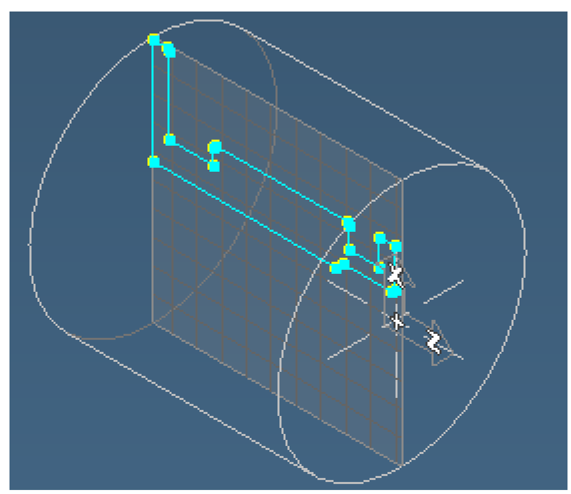




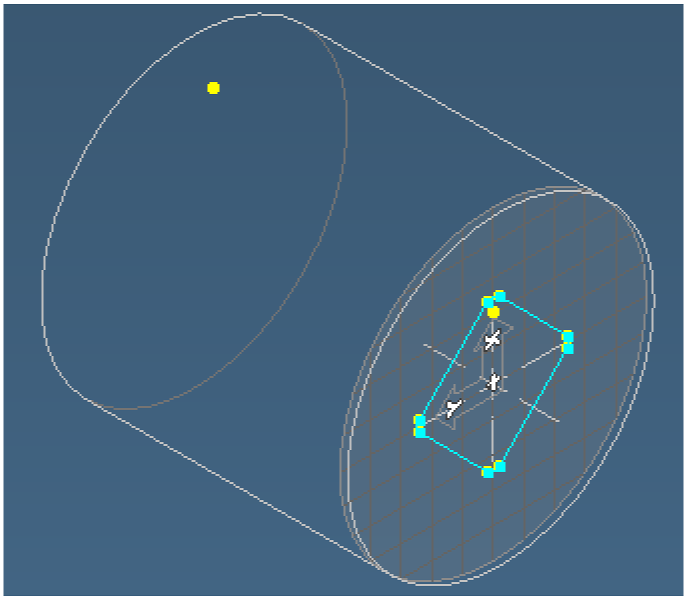
The Workgroup list has four workgroups which contain the geometry for different elements of the part. The Type column shows images of the type of geometry that is contained in each workgroup.

Workgroup 1

The first workgroup, Casting, (designated as Part Stock - revolved) in the WG info dialog. When the part is rendered the initial stock condition will be based on this geometry.



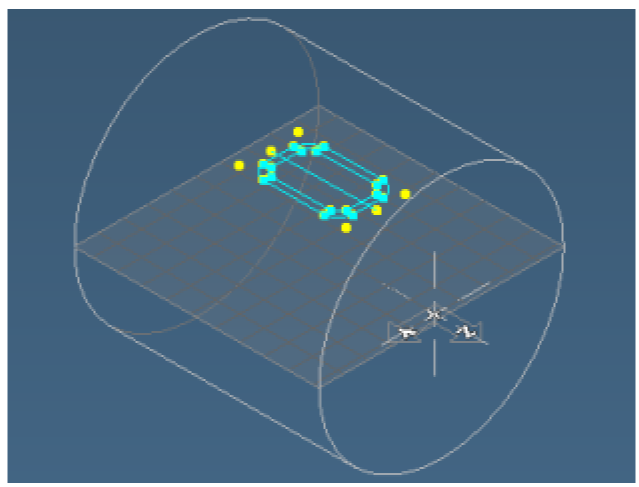
Workgroup 2



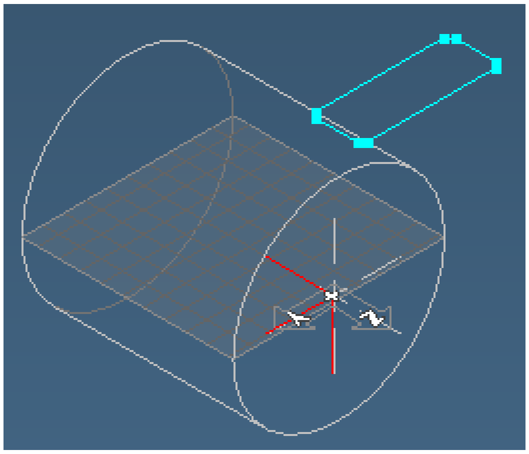
This geometry (designated as geometry, not wrapped) will be used to contour a flange on the front face of the part. This workgroup also contains points which will be used to create bolt holes on the front and back flanges of the part.

Workgroup 3

This geometry will be used to create flat slots and holes on the OD of the part.



Workgroup 4



This geometry (designated as geometry, wrapped) is defined in the YZ plane, which will be used to create a radial slot on the OD of the part.

Machining the Part

#1-2 Front Flange Bolt Holes

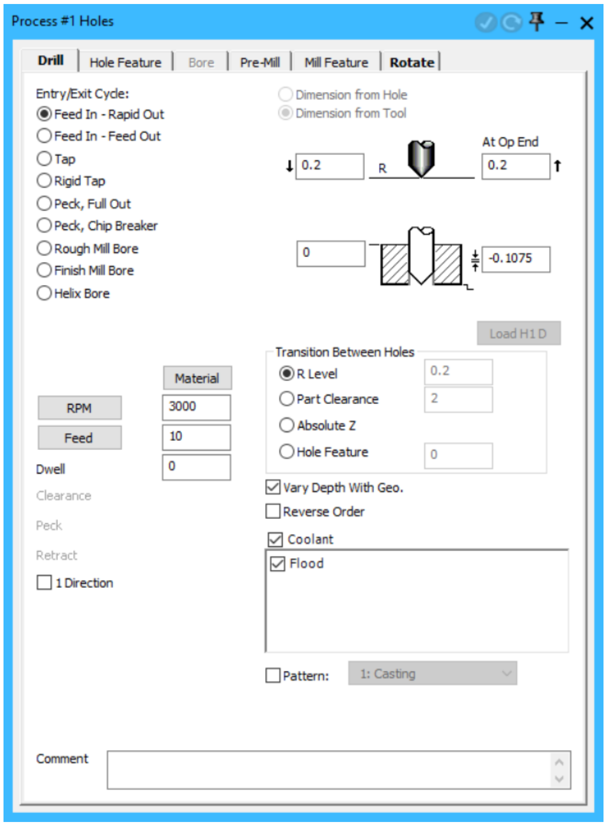
The first set of machining operations will create bolt holes on the front face of the part.

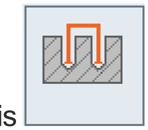
1. Switch to WG2: Front Flange.
2. Switch to CS 2: XY plane.
3. Open the Tool list.

   There are a total of eleven tools in the Tool list.

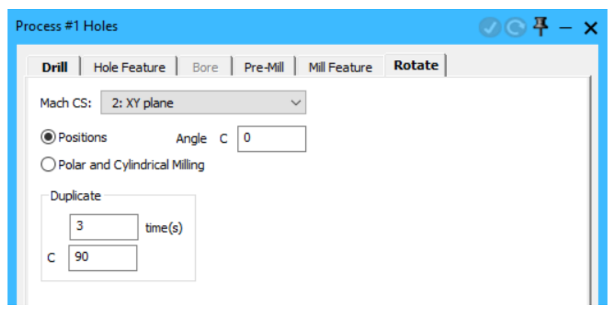
   The first set of operations will be machining the face of the part so the tool orientation for the first four tools is set to approach along the Z Axis.
4. Open Tool #1.

   Note the tool orientation diagram. This orientation designates a Z Axis approach which is the necessary orientation for front face operations.
5. Open the Process and Operation tiles.

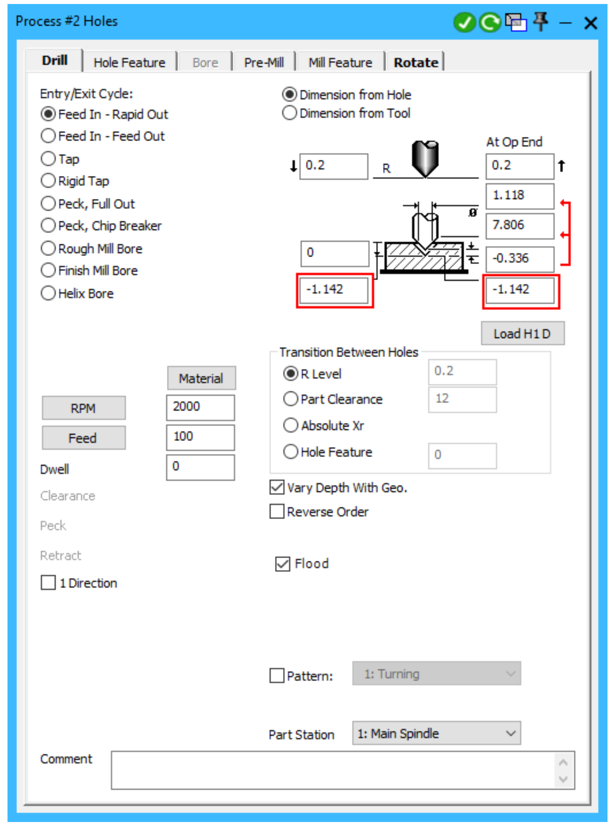


6. For process #1, create this  Holes process with the .375” Spot Drill (Tool #1).

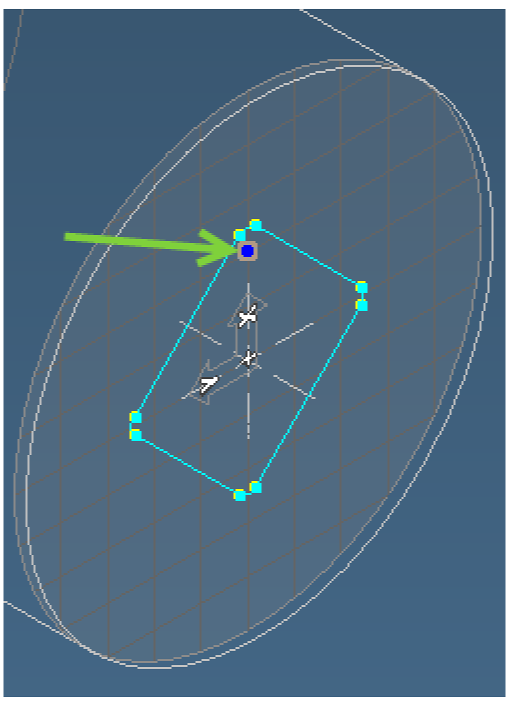
7. Enter this information in the Rotate tab.



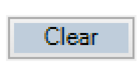
If you have Advanced CS, use the Mach CS list to designate the XY plane.



8. For process #2, create this Holes process with the .1875" Drill (Tool #2).

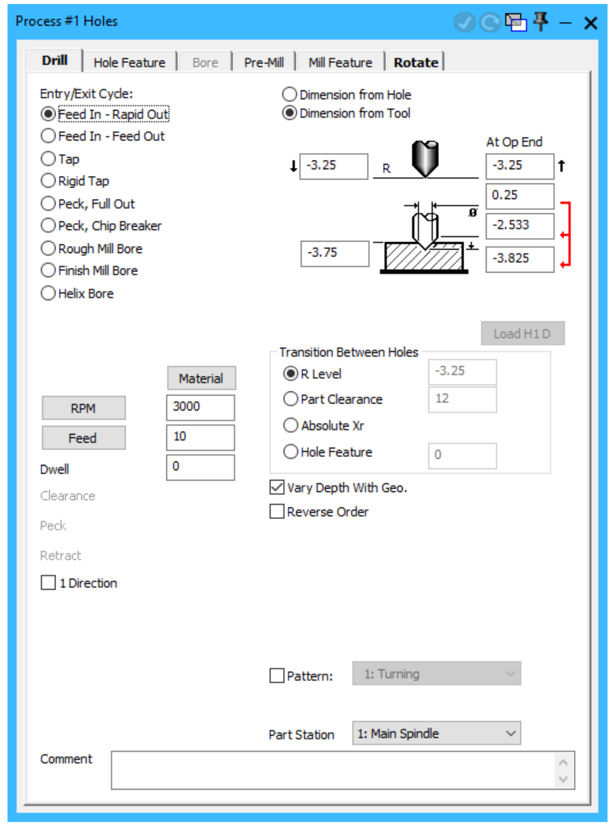


9. Select the point shown.

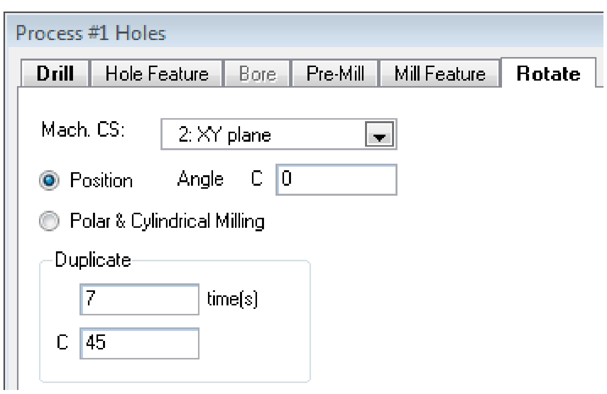
10. Create the toolpath then clear the process list 

.

#3-5 Back Flange Bolt Holes

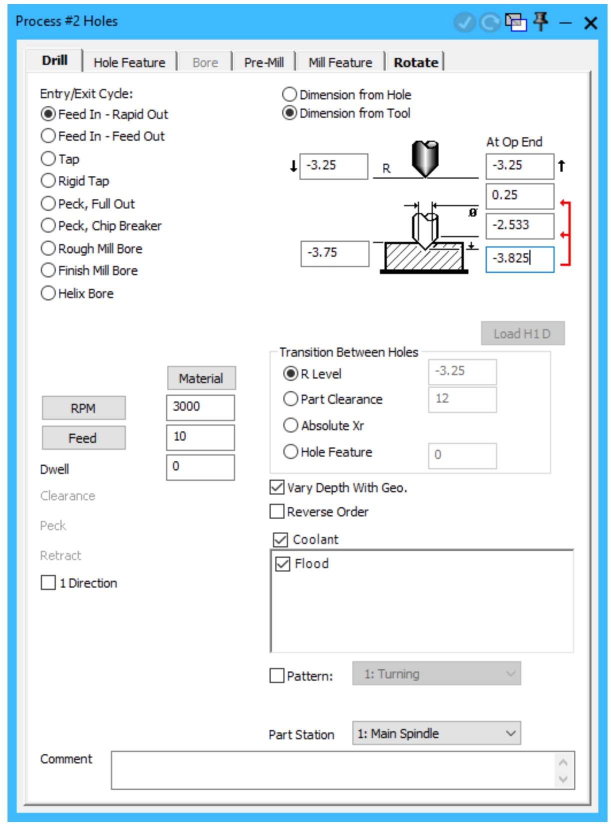


1. For process #1, create this Holesprocess with the .375" Spot Drill (Tool #1).

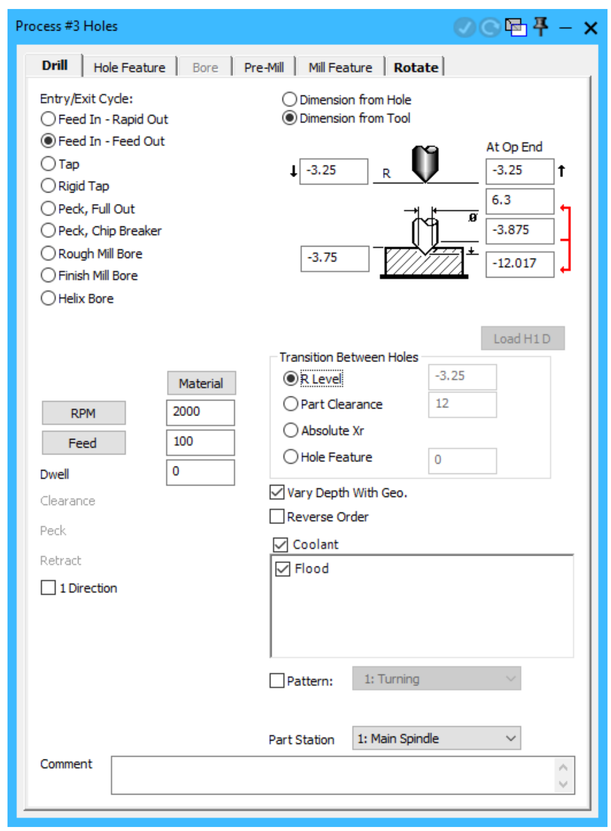


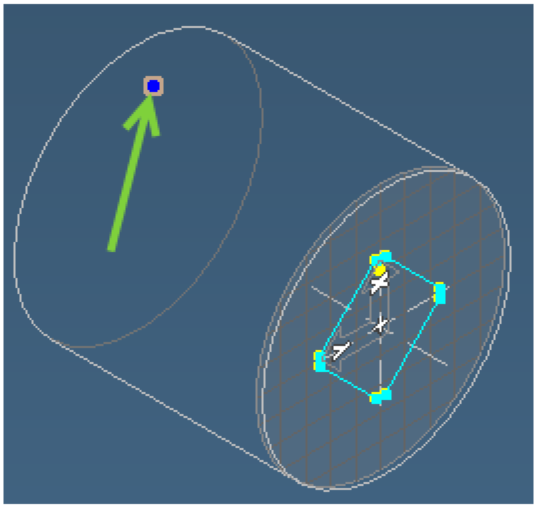
2. Enter this information in the Rotate tab.

3. For process #2, create this Holes process with the 1/4” Spot Drill (Tool #3).

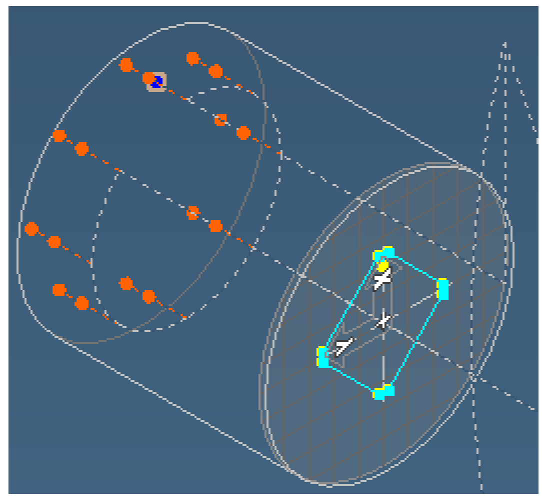


4. For process #3, create this Holes process with the .375” Rough Endmill (Tool #4).



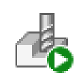


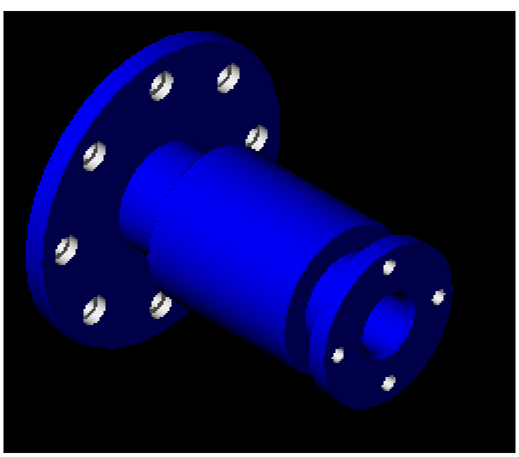
5. Select the point shown.



6. Create the toolpath.

7. Render the operations.

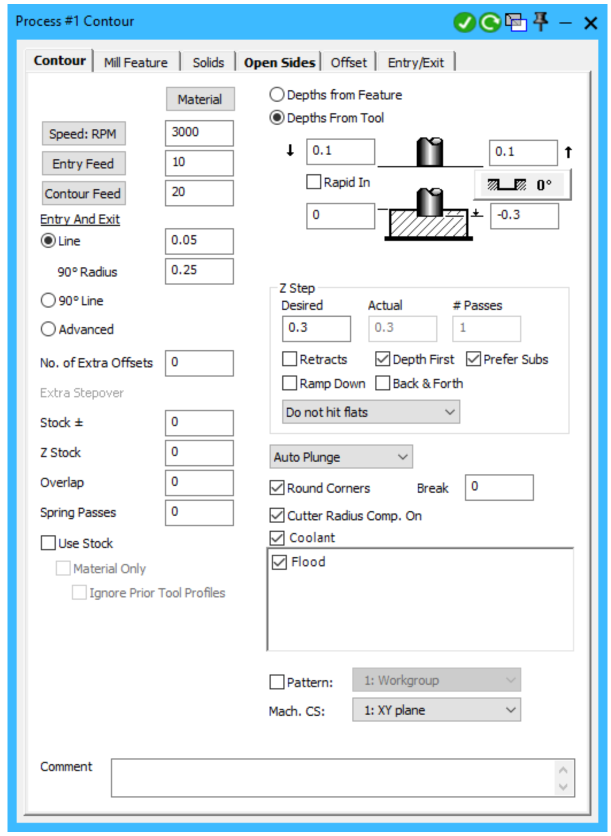


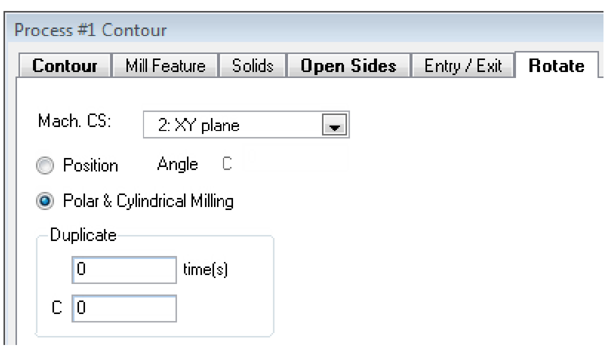


If your rendered image has a cutaway section removed, click the Stock Cutaway icon on the render palette until it is a complete cylinder. 

#6 Front Flange Contour

1. Create this Contour process with the 1/2” Face Mill (Tool #4).





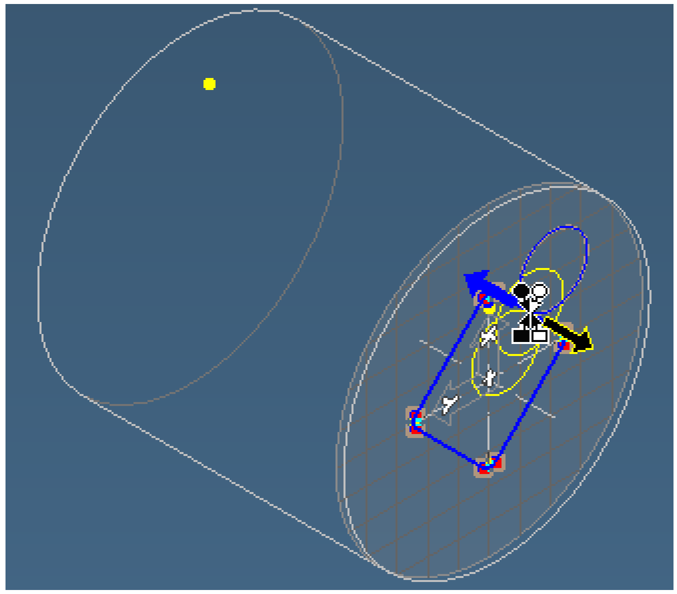
2. Enter this information in the Rotate tab.

The Polar & Cylindrical Milling selection is only available when this option is installed. The Polar & Cylindrical Milling option provides for C Axis rotary interpolation.

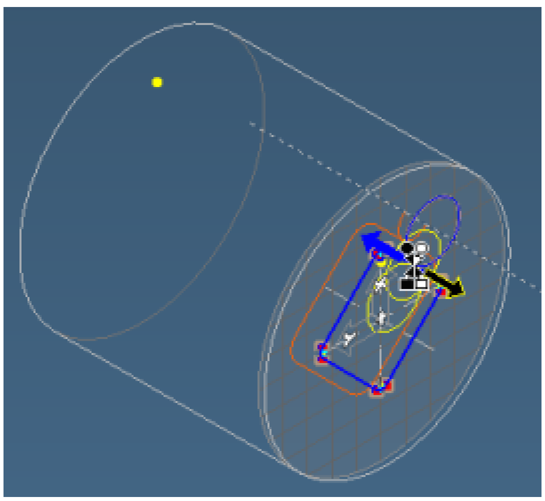
3. If Polar & Cylindrical Milling is unavailable, select Position.

This will create the identical toolpath, although it will require Y Axis moves. Not all Mill/Turn machines support the Y Axis; your machine may not be able to cut this toolpath.

4. Place the Machining Markers as shown.



5. Create the toolpath.



OD Milling

#7-8 Flats

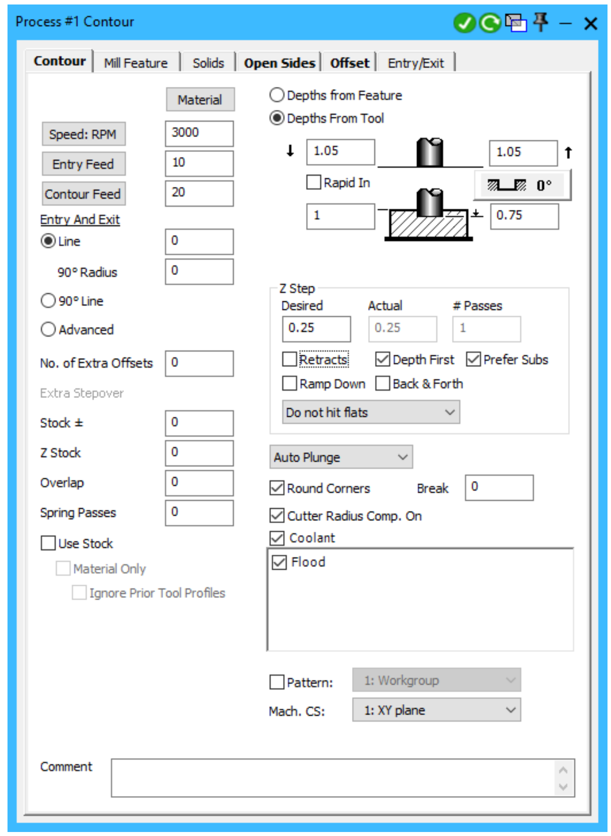
The first set of milling operations will face mill a flat at 45° angles on both sides of the 2.0" diameter of the casting.

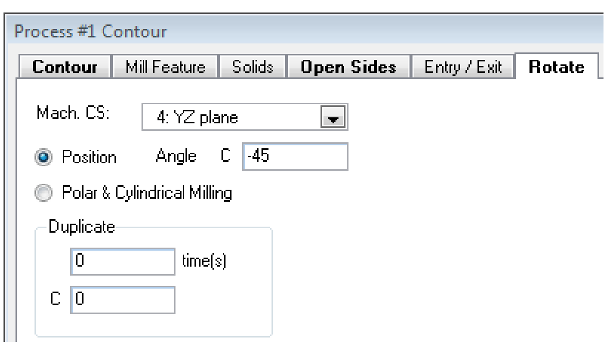
1. Switch to the CS4: YZ plane.
2. Switch to WG3: OD Flat Slots.

Notice that the geometry for these operations is created flat in the YZ plane. These shapes will actually be cut at 45° planes, but are positioned as if they will cut without any rotation. When creating cut shape geometry for OD operations, the geometry should be created as if it were being machined at a position C0 (no rotation). The rotation is accomplished when the toolpath is generated.

You should also note that the remaining tools in the tool list are all set up to cut OD operations. Therefore, the tool orientation diagram designates an X Axis approach, which is used to machine the OD.

3. Create this Mill Contour process with the 1.5” Face Milling tool (Tool #5).



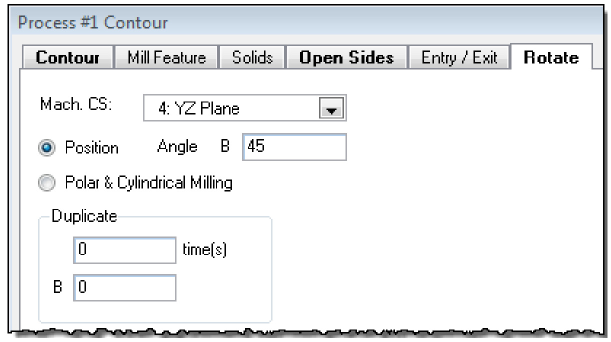


4. Enter this information in the Rotate tab.

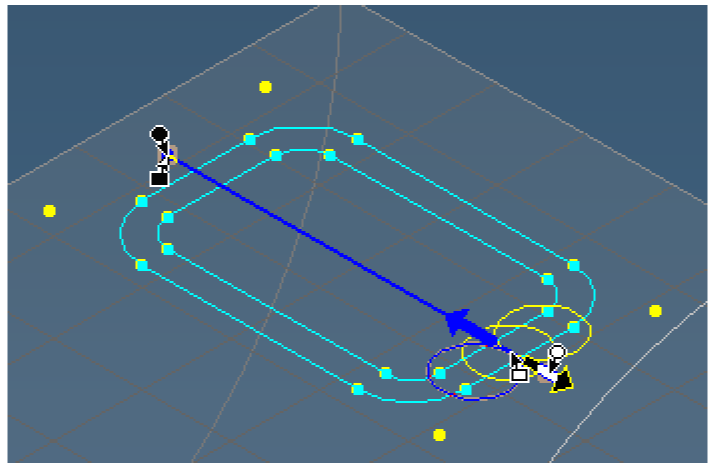
The toolpath will be created at a 45° angle from the C0 position.

5. Now create an identical operation #2 to machine the opposite 45° plane. The only change that we need to make is in the rotation angle.

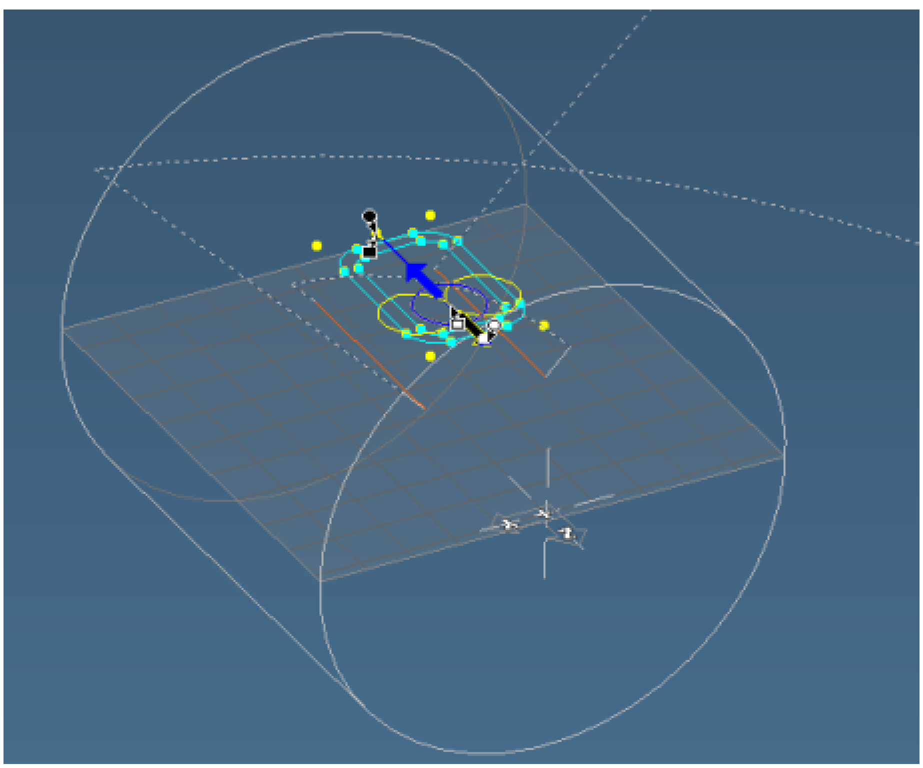
6. Change the Position value in the Rotate tab.



7. Place the Machining Markers as shown.

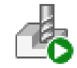


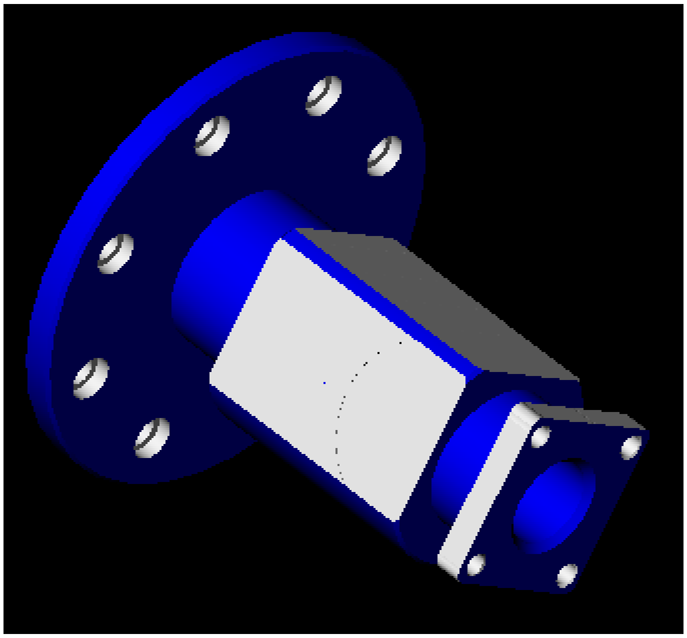
8. Create the toolpath.



.

9. Render the operations.



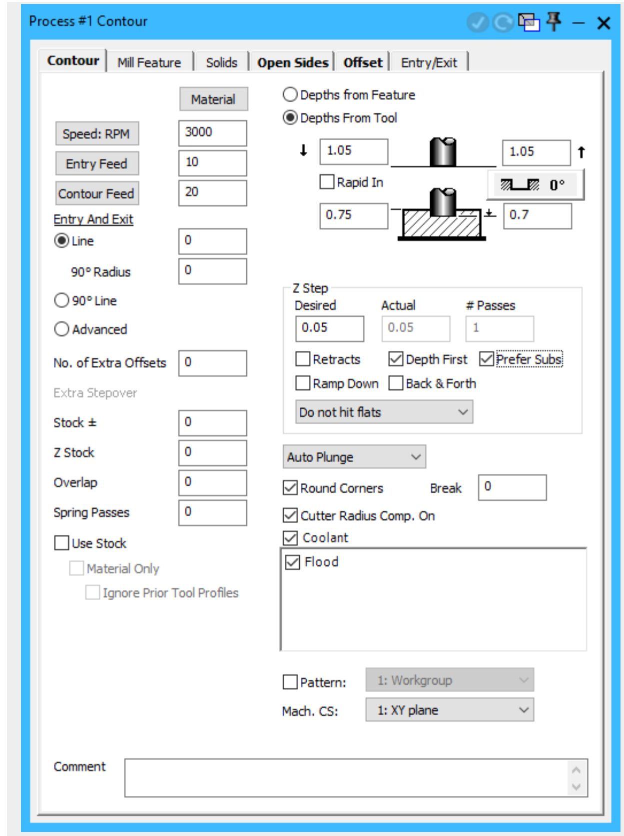


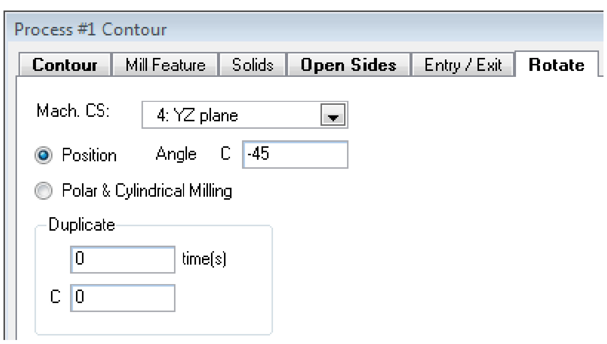
Y-Axis Machines

#9-10 Gasket

The next operations will contour a gasket on the flats. The machining of the gasket and the subsequent pockets require Y Axis tool moves.

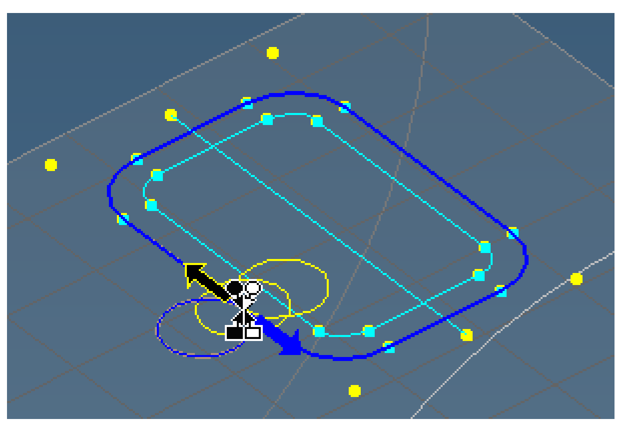
1. Create this Contour process with the 0.0625” Ball Endmill (Tool #6).



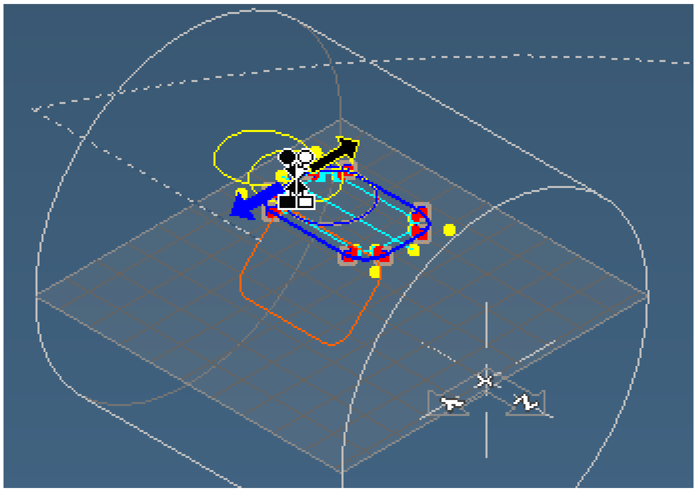


2. Enter this information in the Rotate tab.

3. Place the Machining Markers as shown.

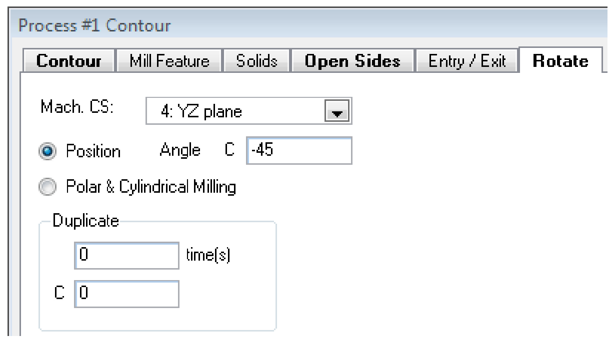


4. Create the toolpath.



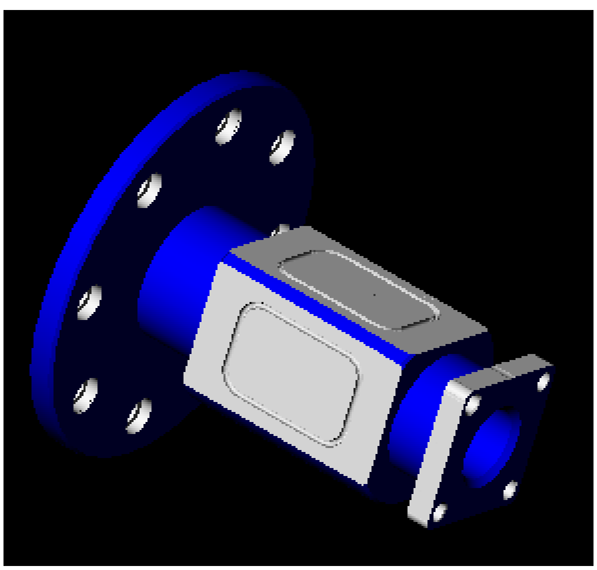
Again, we will perform an identical operation on the opposing 45° flat.

5. With the previous Contour process change the Position angle in the Rotate tab.



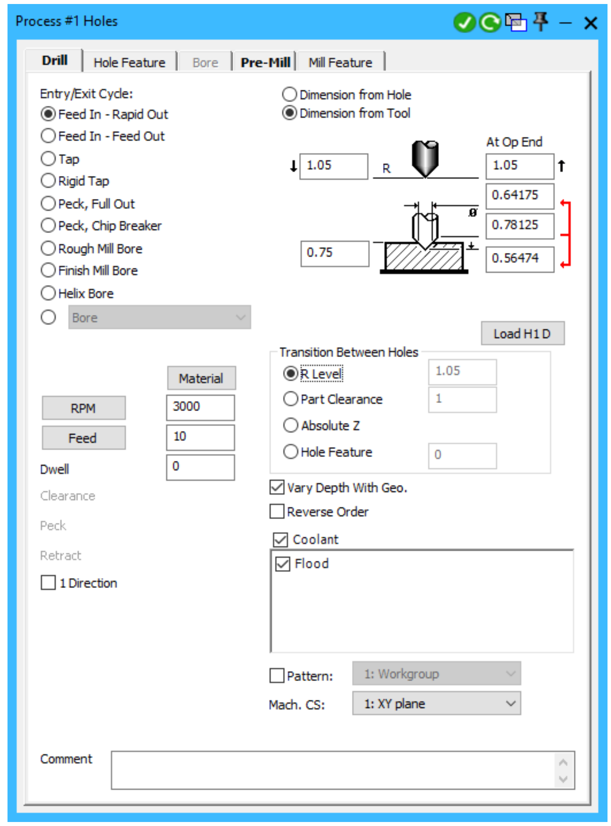
The machining markers should be positioned exactly as they were for the previous operation.

6. Create the toolpath.

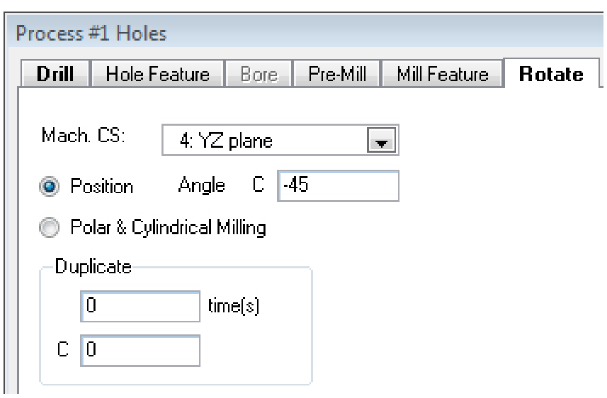


#11-16 OD Drilling

The next group of operations are drilling operations which will spot drill, drill and tap the four holes surrounding the gasket. These drilling operations will be performed on both 45° flats like the previous sets of operations.

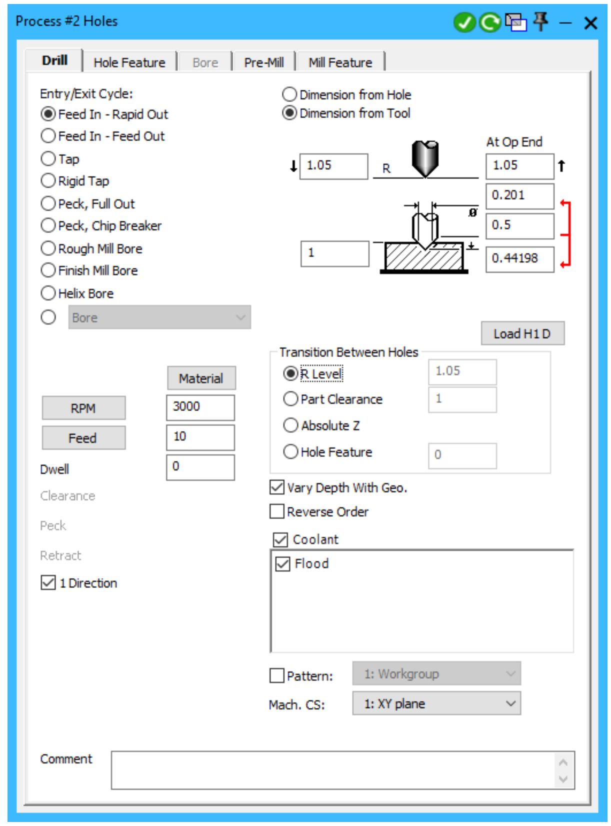


1. For process #1, create this Mill Holes process with the 1/4” Spot Drill (Tool #9).

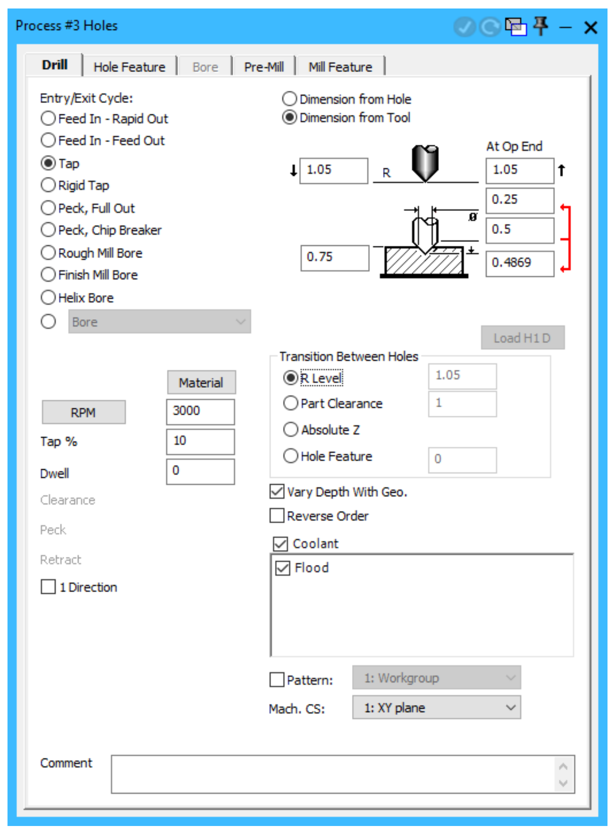


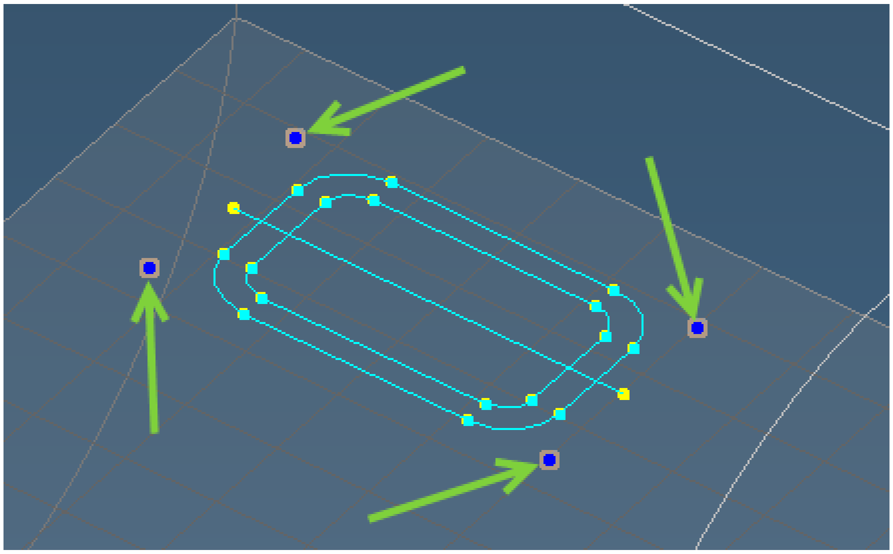
2. Enter this information in the Rotate tab.

3. For process #2, create this Holes process with the 0.1875” Drill (Tool #7).



4. For process #3, create this Holes process with the 0.1875” Tap (Tool #8).

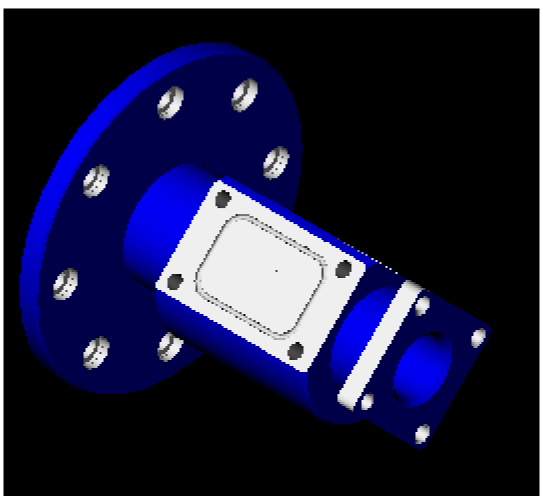




5. Select the four holes shown.

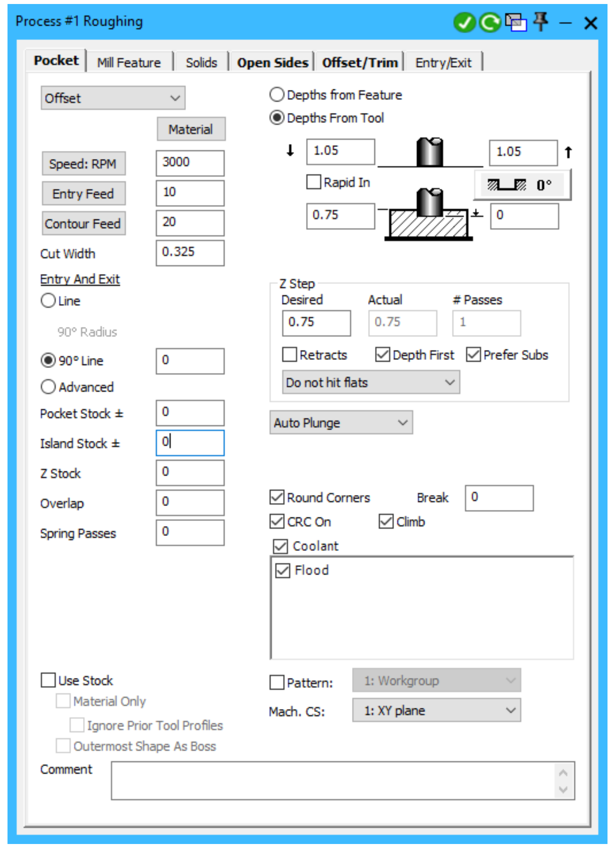
6. Create the toolpath.
7. Change the Position to 45° in all the Rotate tabs.

   Make sure you do this for all three processes.
8. Create the toolpath.
9. Render the operations.

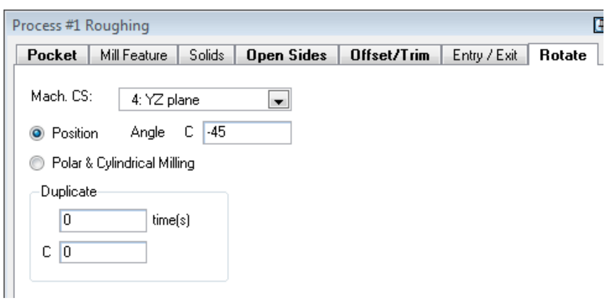


#17 -18 Pockets

The final set of operations that will machine this area of the part will pocket out the area inside the gasket. Again, this set of operations requires Y Axis interpolation, which may not be supported by your Mill/Turn machine.

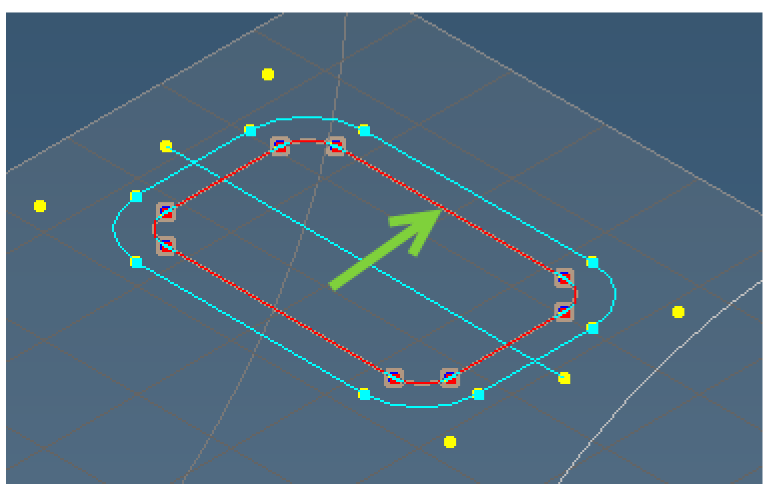


1. Create this Roughing process with the 1/4" Finish Endmill (Tool #10).

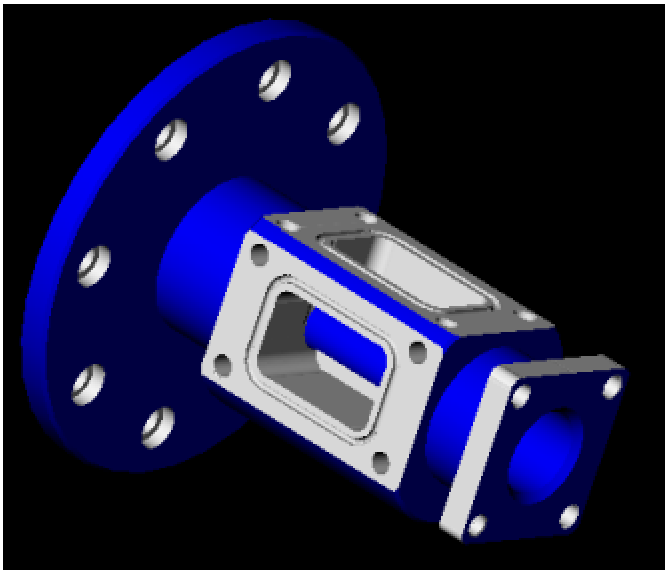


2. Enter this information in the Rotate tab.

3. Select the inner shape.



4. Create the toolpath.
5. Change the Position to 45° in the Rotate tab.
6. Create the toolpath.
7. Render the operations.



Rotary Milling

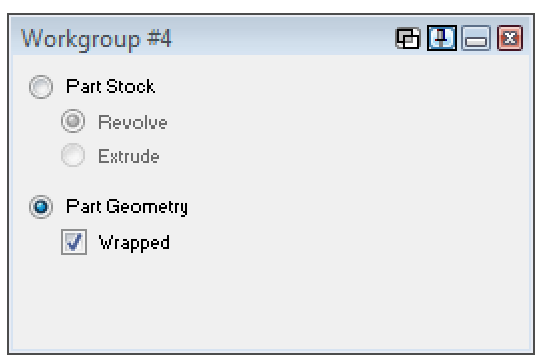
#19-20 Radial Slot

This final operation will complete the machining of this part is a radial slot that will be machined on the bottom side of the part. In order to create accurate toolpath you will need the Rotary Milling option.

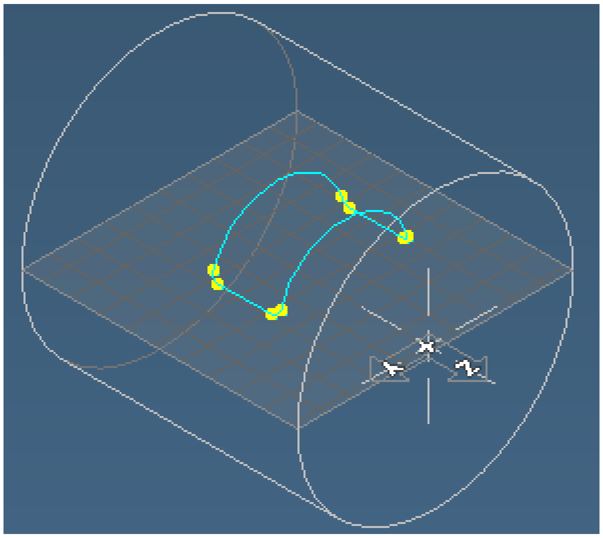
1. Change to WG 4: OD Radial slot.

   First, we need to wrap the geometry and then rotate it into position on the bottom of the part.

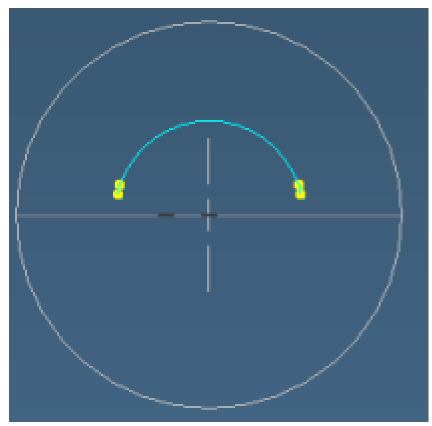
2. Right-click WG #4 and choose WG Info.
3. Select Wrapped.



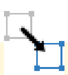
4. In the floating menu, turn on Wrap WGs. 



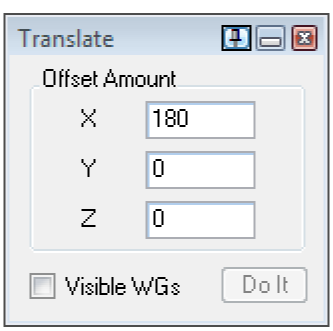
Viewing the geometry from the front face will clearly show how the geometry is wrapped at a given diameter.



5. Select the wrapped shape.

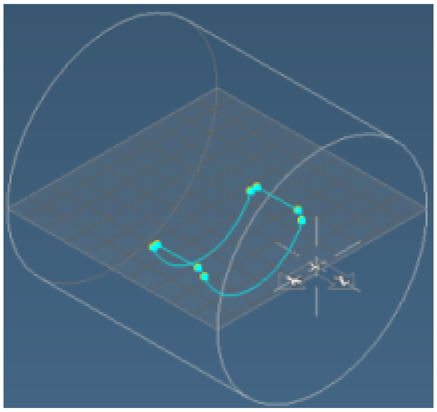
6. Choose Modify >  Translate.

When the system is in radial mode, some of the Modify dialogs allow for radial input--an angle of rotation and a radius value.

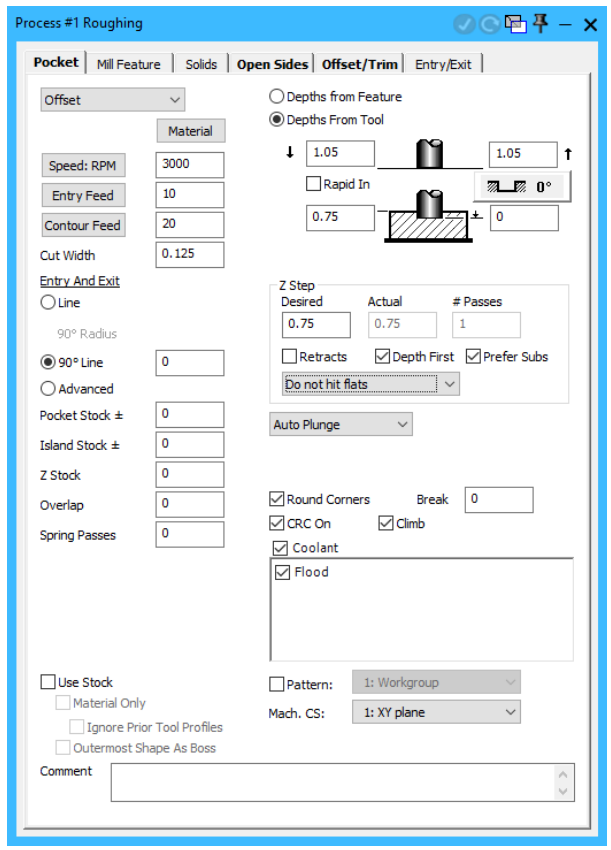


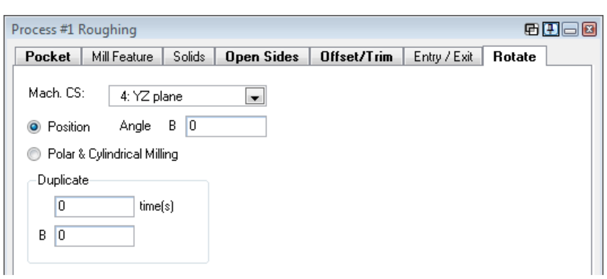
7. Translate the geometry as shown.

The geometry should be positioned radially at the bottom of the part as shown in the following picture.



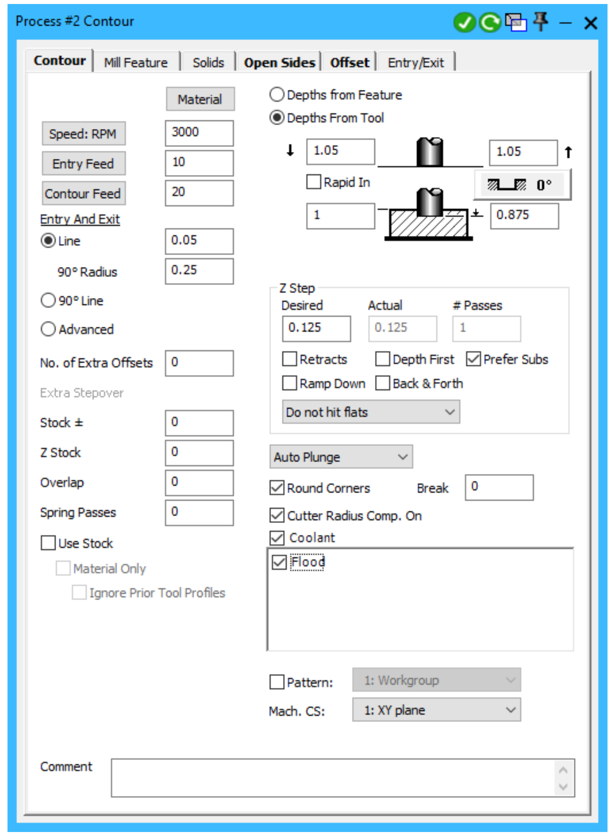
8. For process #1, create this Mill Roughing process with the 1/4” Finish Endmill (Tool #10).

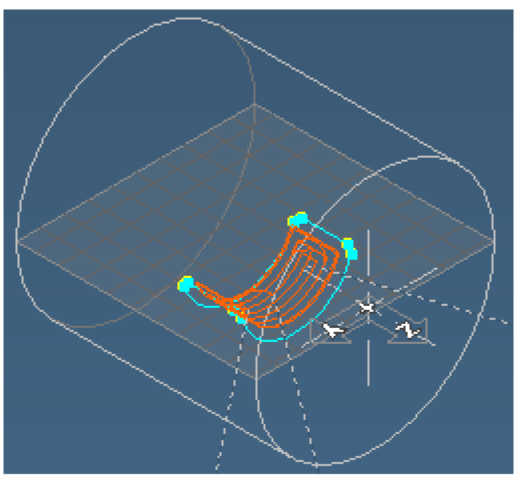




9. Enter this information in the Rotate tab.

10. For process #2, create this Mill contour process with the 0.1875” Finish Endmill (Tool #11).





11. Select the wrapped geometry and create the toolpath.

12. Render the operations.