GIBBSCAM® 14

Version 14 : September 2020

Mill Tutorials

CAMBRIO

Proprietary Notice

ABOUT THE MILL TUTORIALS

The easiest and most effective way to learn the system is to actually program a part, which is what we will do in this tutorial. The following step-by-step instructions guide you through setting up a part, creating tools, machining operations, and post processing. The geometry for the part was created in the Geometry Creation tutorial. For dimensions of the part, see "Part Print" on page 73.

BASIC MILL TUTORIAL

This tutorial uses a part we created in the Geometry Creation tutorial. We will go over the part setup, creating tools, processes and operations. We will then render the part to ensure we have what we want. Finally we will post the part.

Setting Up The Part

Open An Existing File

1. Click the Document Control Dialog button in the Command palette. 

2. Click the Open button. 

3. Locate and choose the Mill Tutorial.vnc part that was created in the Geometry Creation tutorial.

   If you have not created the geometry for the part, go back to the Geometry Creation tutorial and do so, or open it from the Sample parts folder.

   You might need to change the current drive and directory in order to locate the file.

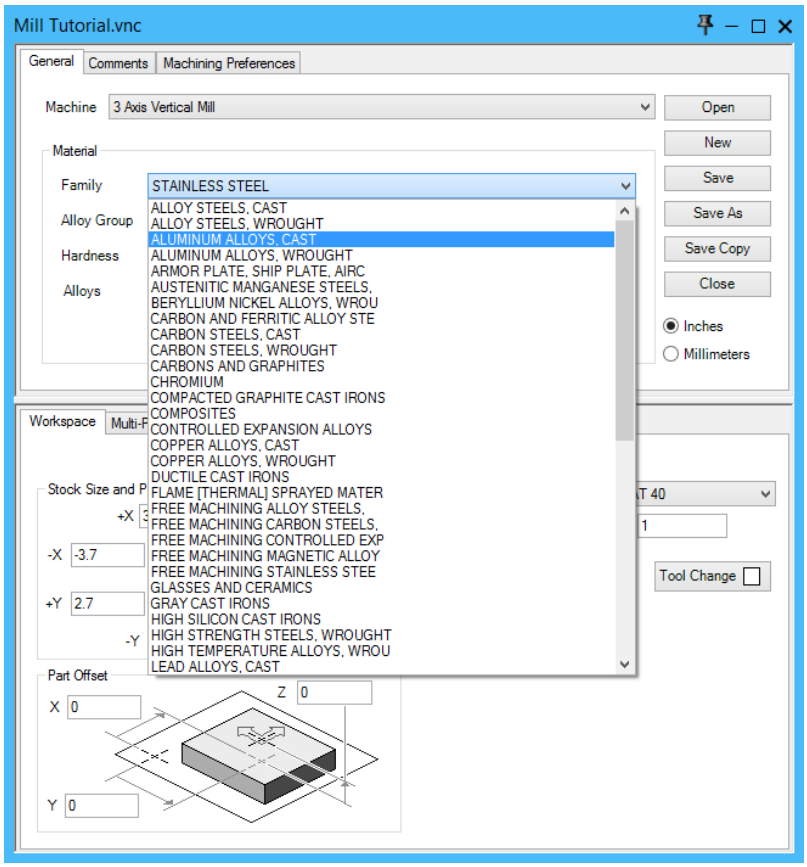
4. Click Open.

> Parts can also be opened by:
>
> - Double-clicking the associated VNC part file with the system icon.
> - Drag and drop the part file icon onto the application icon, shortcut or application Workspace.

About the Part

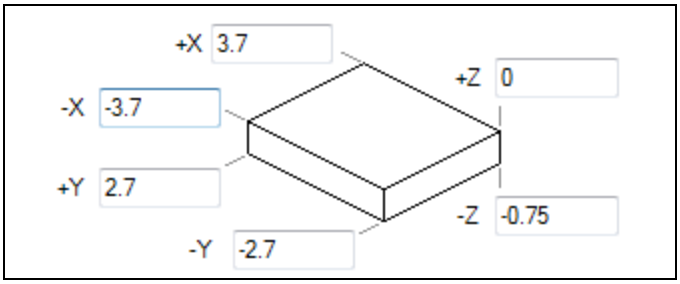
The Document Control dialog now contains the information for the Mill Tutorial part. The top dialog contains Material selections in a pop-up menu. The Material menu will have multiple selections available if you have purchased the CutDATA™ material library or created your own Material Database. Otherwise, there is only the default selection for materials. For more detailed information on the Material Database, refer to the Common Reference user guide, "Materials" chapter.

1. Set the material type to Aluminum Alloy, Cast.



If you have not purchased CutDATA you will not be able to perform this task. That is fine, just be aware that the feeds and speeds used in this tutorial will come from the material database and are for cutting aluminum which is the only material available to users who do not have CutDATA.

2. Verify the settings are as shown.



The stock size values affect how the geometry of the part is shown on the screen and Material Only calculation. It is recommended that the stock size values closely resemble the initial part stock dimensions.

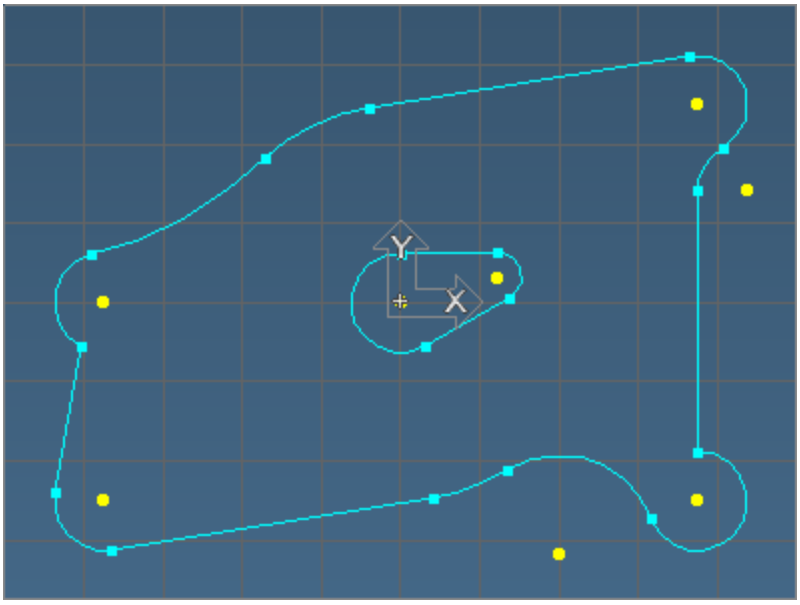
The Clearance Plane Z indicates where the tool will rapid to immediately after a tool change. It also specifies the last rapid move the tool will make before going to the Tool Change Position. Additionally, the tool will move to this position between drilling moves if the second Retract to Z option is selected in the Drilling Process dialog. For more information on Clearance Moves and Drill process dialog refer to the Mill user guide, "Machining" chapter.

The Tool Change Position is used on milling machines without automatic tool changers. It is also used when the table must be moved to avoid crashing during a tool change due to a tall part, fixtures, or a rotary table.

3. Close this dialog.

Familiarizing Yourself With The Part

The geometry previously created in the Geometry Creation manual is drawn in the workspace outline.



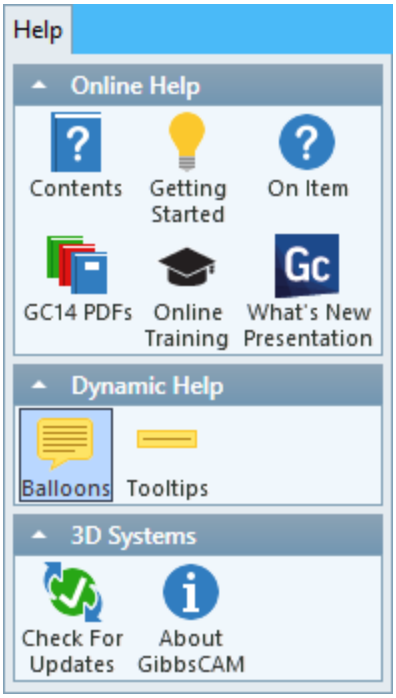
We are currently looking at the top view of the part, or the XY plane. In order to get a better visual of the part, we will change the view.

1. Open the View Control palette.

The View Control palette, also referred to as the Trackball, allows you to change views, Redraw, and Unzoom. You can select the different options by clicking on the corresponding buttons.

For an explanation of the various buttons and views, we will turn on the Balloons option.

2. Turn on Balloon help (Help > Dynamic Help > Balloons) or use the shortcut (Ctrl+B).



The Balloons option provides on-line reference information. Notice that the Balloons icon now displays within a blue box indicating that a menu item is in use.

To see the balloon help popups, move the cursor over the desired object.

3. Click the Isometric view button.

Isometric view allows you to see the wireframe of the stock. The Top view is recommended for creating geometry and toolpath for a 3-Axis part, while the Isometric view is excellent for rendering.

4. Choose Help > Balloons (Ctrl+B).

This will turn the Balloons off.

> ⓘ Turn on Balloons whenever you want online reference information on the various functions of the system.

5. Click the Top View button (Ctrl+E) .

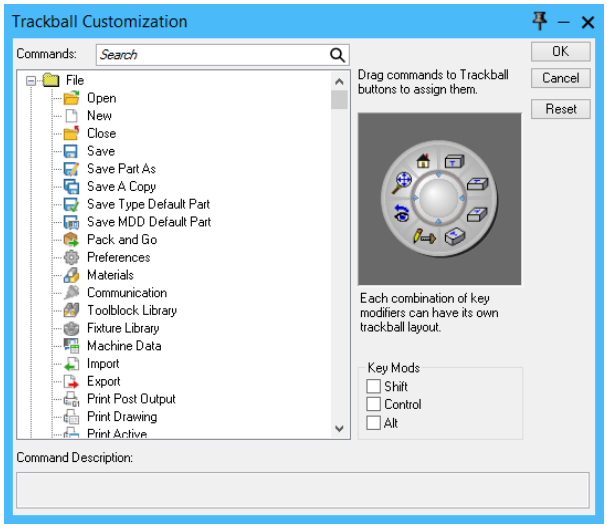
6. Click the View Control button to close the View palette.

The Main Menu View submenu also allows you to select all of the views available in the View palette. It also has additional commands that may be useful for viewing your part.

Now look at the bottom left of the screen. You will see an axis marker.  Hover over this marker and you will see a Trackball appear. This is very useful to enable quick view changes to be made.

You can change the position of this marker to any corner of the screen by editing the File > Preferences > Display > Edit Appearance Settings > Background tab and setting the Draw Axis location (or turning it off).

Now try right-clicking on the center of the trackball or view palette. A dropdown menu appears with a Customize option. This option enables you to add common commands to your trackball, simply by dragging them onto the required buttons. Extra commands can be added in combination with the Shift, Ctrl and Alt keys. Use the Reset option to delete the customization.



Look at the bottom right border of the screen. This area gives basic information about the part (Inch or metric). Current Coordinate System and Workgroup are shown (click them to open the dialogs).

The Trashcan and Progress bar are also situated here, where they are easily accessible. If your part is to be duplicated, this will also be indicated.

7. Double-click the CS button. The CS dialog opens. Use the ✕ to close the dialog.

Creating a Tool list

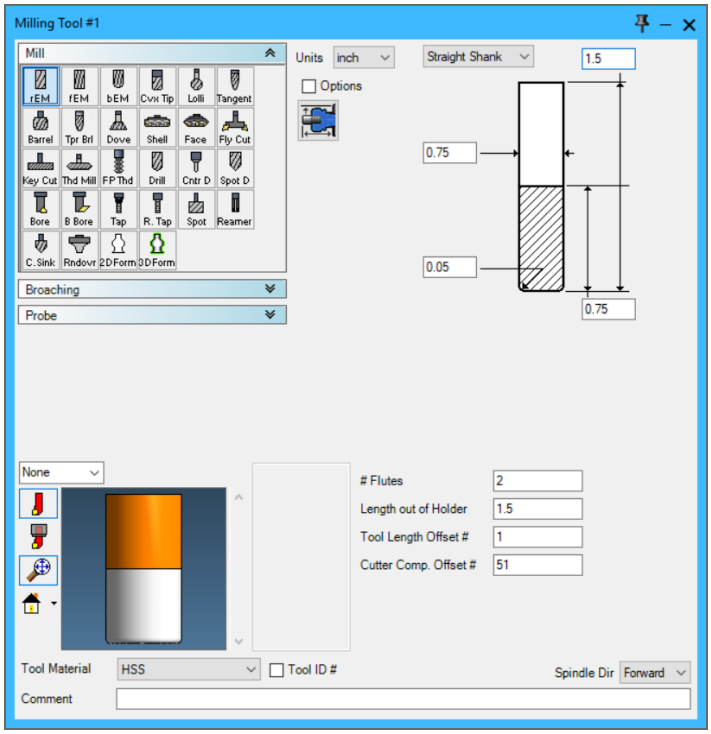
1. Click the Tool list button in the Main palette. 

When you click this button, the Tool tile list will appear in the upper left-hand side of the screen.

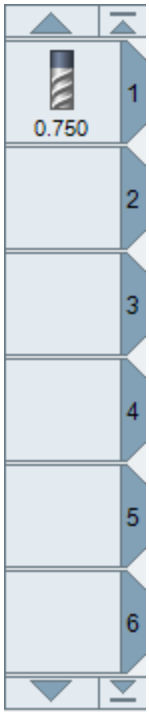
2. `Double-click` the empty tile position #1 in the Tool list.

   A Tool Creation dialog will appear on the screen when you `double-click` an empty Tool tile location. All tool specifications will be entered in a Tool dialog similar to the one shown. Information in this dialog can be edited at any time during part creation. If operations have already been created using the edited tool, you will need to reprocess the operations in order for the new tool specifications to take effect.

3. Select `rEM` (Rough Endmill) and enter the values shown.



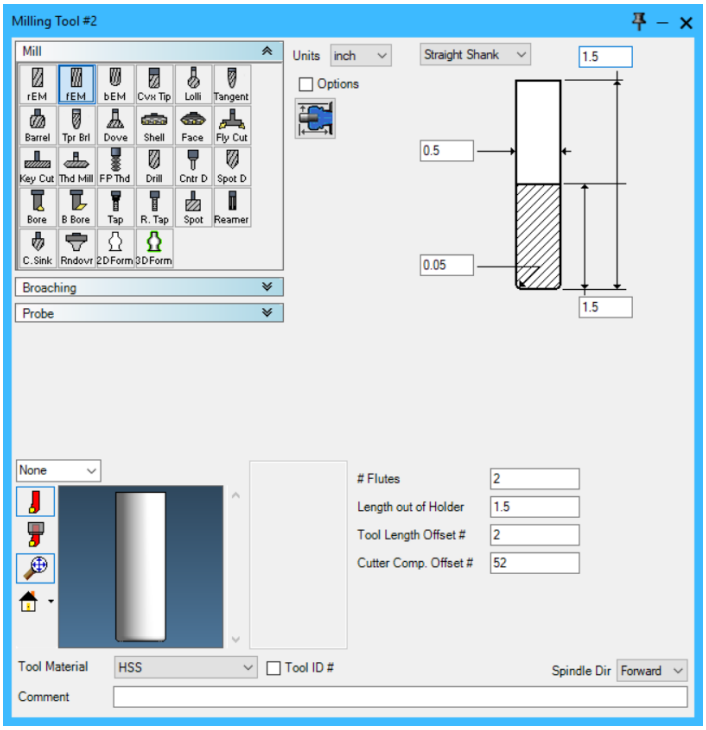
This is a 3/4" High Speed Steel flat-bottomed endmill and will be used for roughing the part. The particular holder you select is not important and unless you run Op Sim you will not see it.



4. `Double-click` tile position #2 in the Tool tile list.

   This closes the Tool #1 dialog and brings up the Tool # 2 dialog. A Tool tile displaying Tool 1 type and tool diameter appears in the first position of the Tool tile list.

5. Select `fEM` (Finish Endmill) and enter the tool information shown.

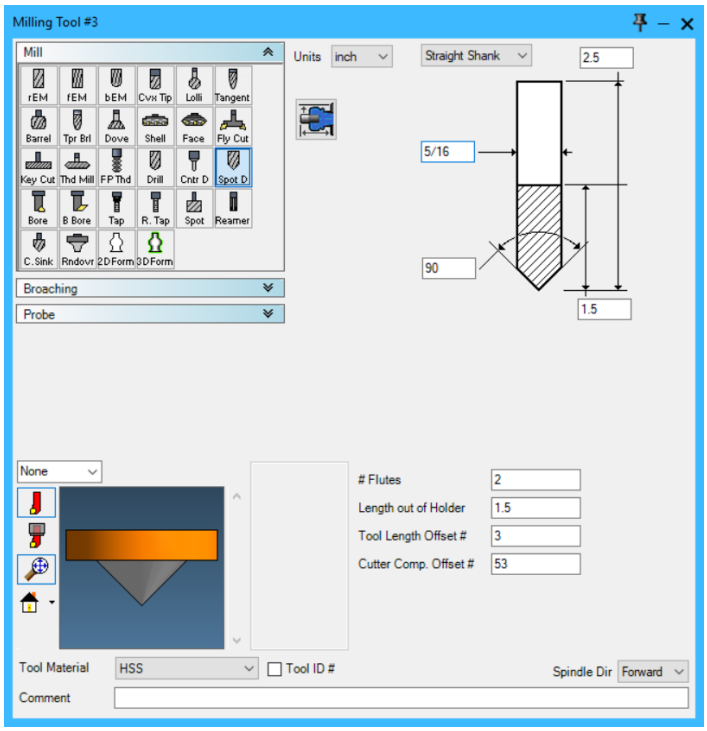


This is a High Speed Steel flat-bottomed finish endmill with a 0.05" bottom corner radius and will be used for finish contour passes on the part.

6. `Double-click` tile position #3 in the Tool list.

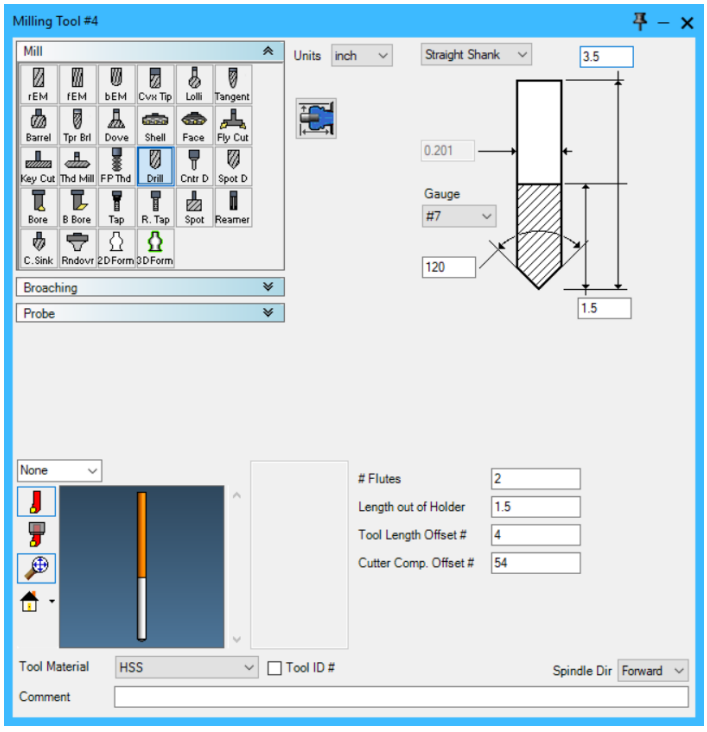
7. Create a Spot Drill as shown. By typing `5/16` in the diameter field and then pressing the `ENTER` key, GibbsCAM will calculate a diameter of `0.3125` for you.

   This is a fairly standard 5/16" spot drill.



8. `Double-click` tile position #4 in the Tool list.

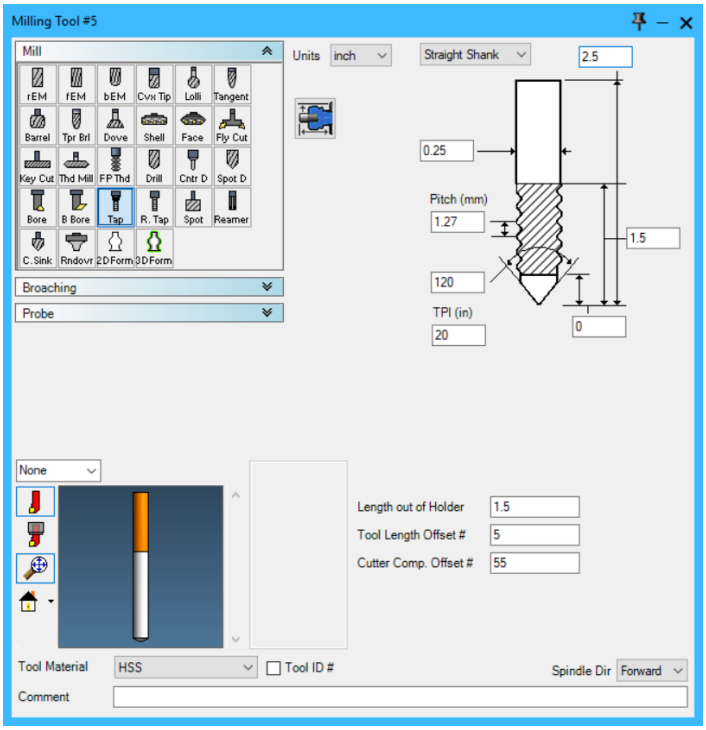
9. Create a Drill as shown.



This standard drill will be used for drilling out tap holes.

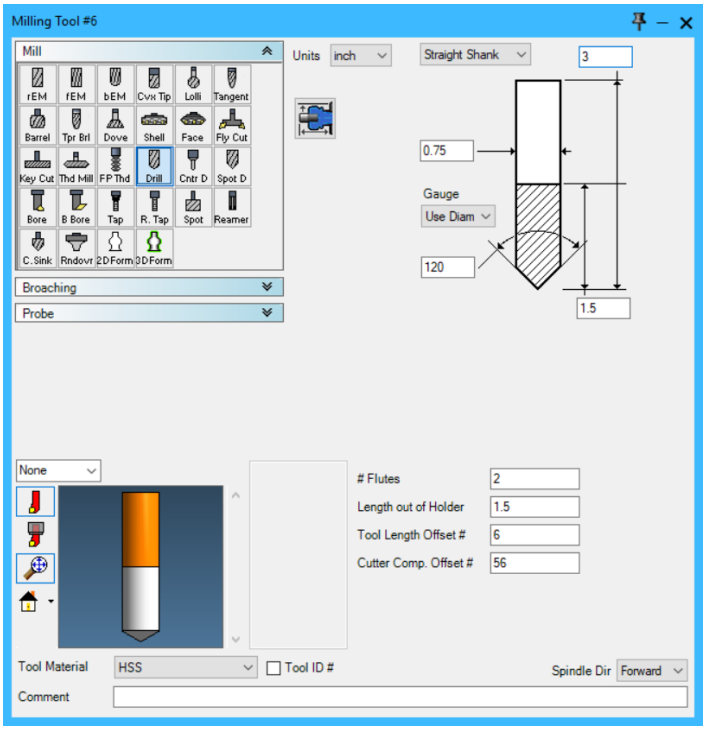
10. Create a Tap tool as shown.

    This is a standard 20 TPI tap.



11. Create a Drill as shown.

    This 3/4" drill will be used to make entry holes for the roughing operations.



12. Close the tool dialog (`Ctrl+W`). 

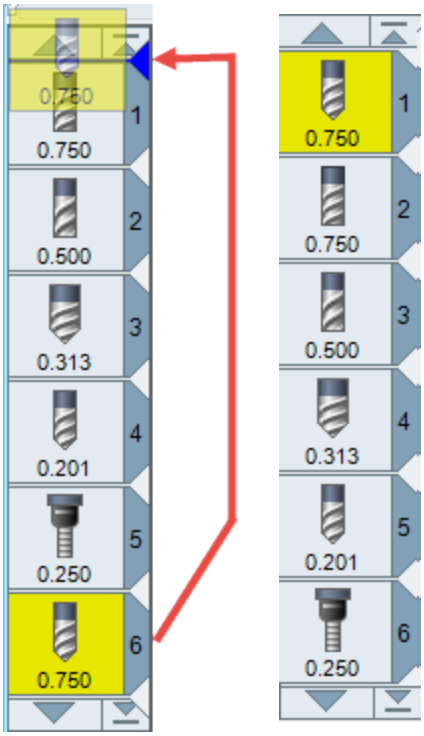
Our tool list is complete. We will now make a minor modification to the list and look at a tool that tells us about our chosen tools.

More About the Tools

Tool #6 is actually going to be the first tool we will use as it is going to make some entry holes for us. While the position of a tool in the list is not important, it is convenient to keep them in some sort of logical order, such as order of use.

1. Drag tool #6 up to the insertion point above tile position #1. As you drag the tile, the cursor changes to a hand symbol. Put the tip of the fingers on the line above where you want the tile to go.

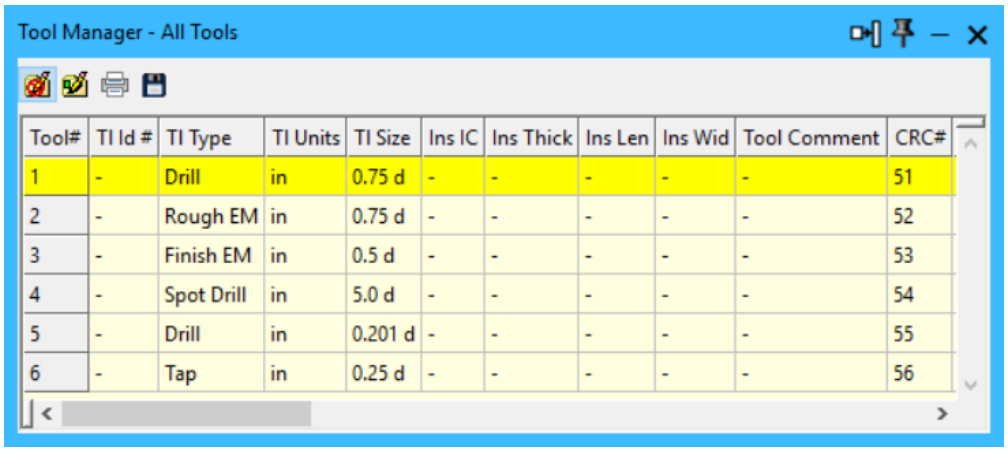
   A blue arrow appears indicating where the tool will be inserted. When you let go the mouse, the tile is put into position. As you can see, reordering tools is very easy.



The tools necessary to machine this part have been created. We will now review the tool specifications with the Tool Manager dialog.

2. Choose Window > Managers>  Tool Manager or right-click a tool tile.

   This dialog contains a list of all of the current tools and the tool specifications. The columns displayed can be customized by right-clicking the heading row. Print or save this information by selecting the appropriate icons.

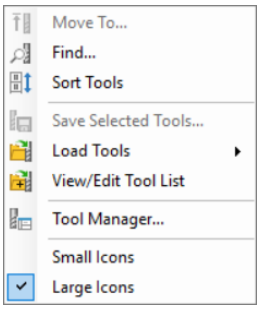


Tool#	TI Id #	TI Type	TI Units	TI Size	Ins IC	Ins Thick	Ins Len	Ins Wid	Tool Comment	CRC#
1	-	Drill	in	0.75 d	-	-	-	-	-	51
2	-	Rough EM	in	0.75 d	-	-	-	-	-	52
3	-	Finish EM	in	0.5 d	-	-	-	-	-	53
4	-	Spot Drill	in	5.0 d	-	-	-	-	-	54
5	-	Drill	in	0.201 d	-	-	-	-	-	55
6	-	Tap	in	0.25 d	-	-	-	-	-	56

Saving a tool list

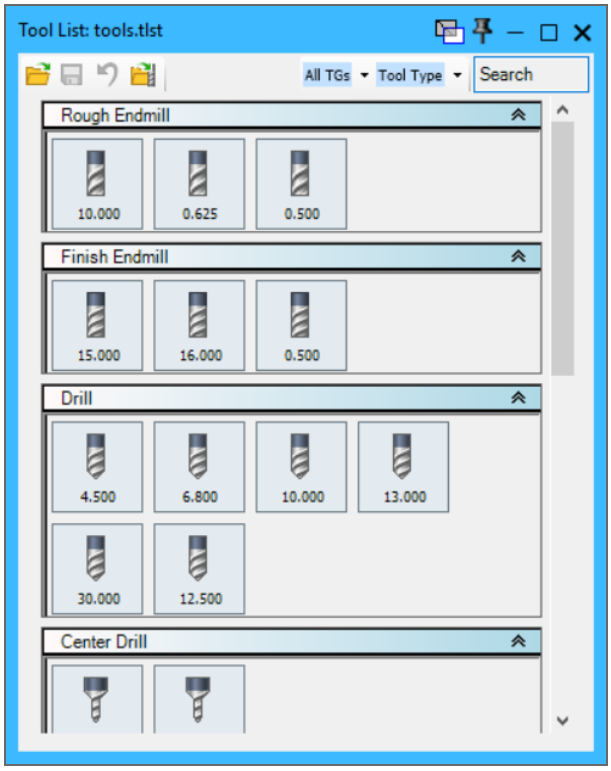
We will now add these tools to a tool list file so that we can use them in other parts.

1. Right-click on any Tool Tile to display the Tool list menu as shown.



2. Select the option View/Edit Tool List and navigate to the sample file tools.tlst.

3. Click Open. The tool list opens.



4. Click in a tool tile and `Ctrl-A` to select all the tools in your list.
5. Now drag the highlighted tools and drop them into the Tool list dialog.
6. The tools will automatically be sorted into the correct types.
7. Close the Tool list dialog and save your changes.

Creating Operations

About Processes and the Process List

We will now begin to make toolpath. A toolpath is contained within or defined by operations, such as a drilling operation or a roughing operation. The actual parameters of the operation are set by a processes. We will learn about making processes and operations in this section.

1. Open the CAM lists.

The Process list appears under the Tool list on the left of the screen and the Operations list appears on the right of the screen.

Processes

Process tiles are created in one of two ways:

The first is by choosing the required tool from the Tool tiles and dragging it to a Process list tile. The Process dialog then appears and you select the process type you require.

The other, more intuitive way is to double-click a process tile, select the process required and then choose a tool suitable for that process. When you create a process this way, the system will list the most suitable tools for the process first. If the correct tool is not found you can immediately create one. You are of course free to choose any suitable tool that in your tool list.

Operations

Operations must be created in order to machine any part. Operation tiles are automatically created from Process tiles when a cut shape is selected and the 

Do It or 

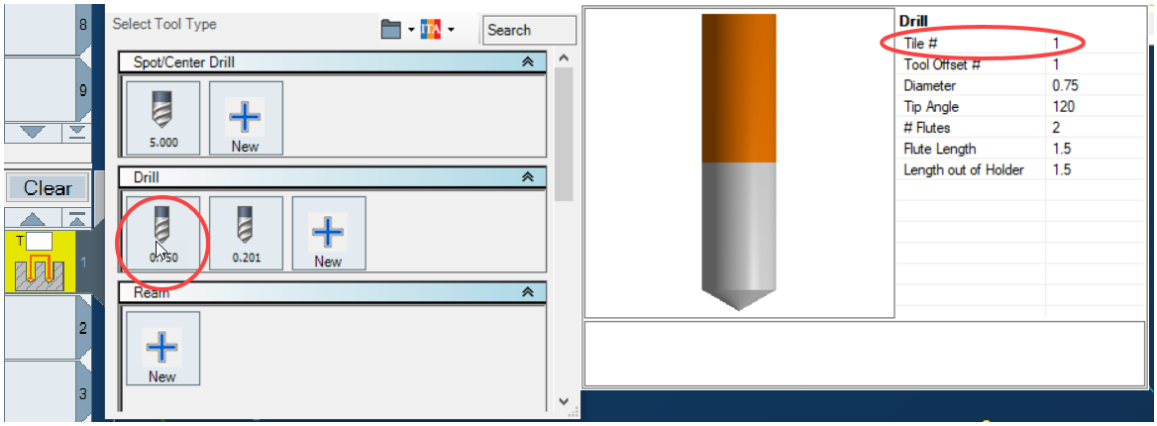
Redo button is clicked.

The First Processes - Drill, Rough and Contour

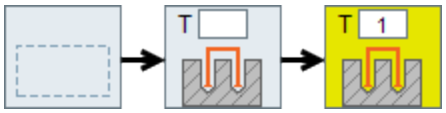
First, we will create a group of operations to rough and finish the inside of the contour. This will be accomplished using Multiple Process Programming which allows for multiple processes to be applied to one cut shape to create multiple operations.

1. Double-click process tile #1. The Process selection dialog opens. Select a Drill process.

The drill process icon now appears in the process list and you are presented with the tool choice dialog. You will notice that the spot drill appears at the top, as this would normally be used first to drill a pilot hole. We will choose the 3/4" standard Drill on this occasion. Hovering over the tool icon as shown below will reveal a snapshot of the tool specification. This will enable you to confirm it is indeed Tool #1.



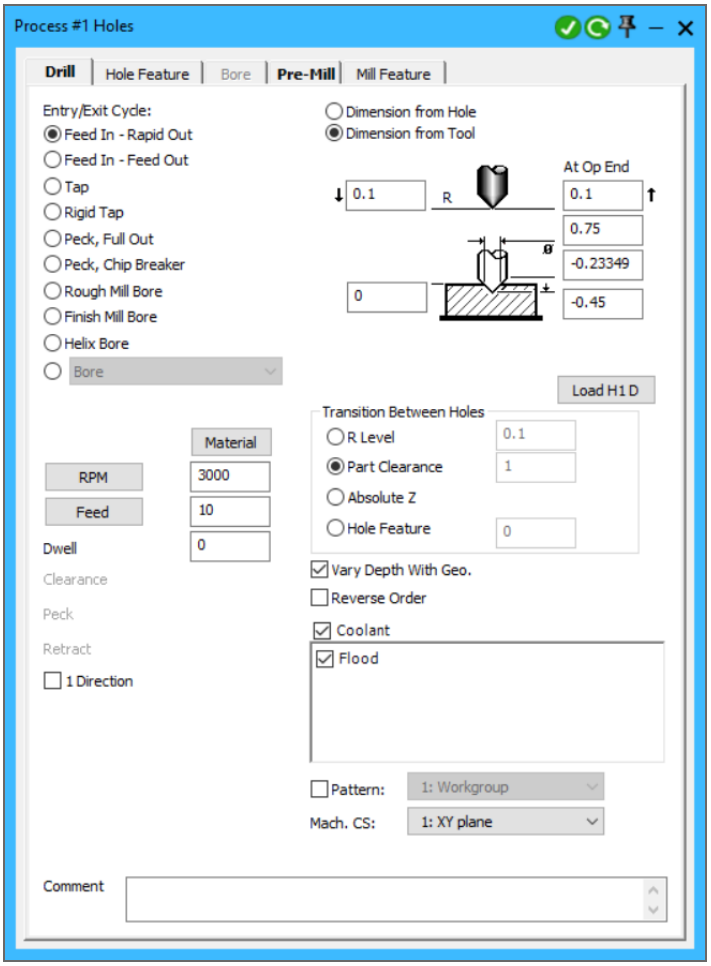
When you have selected the tool, the Tool number, which is #1 in this case, will be displayed in the small box on the Drilling process tile.

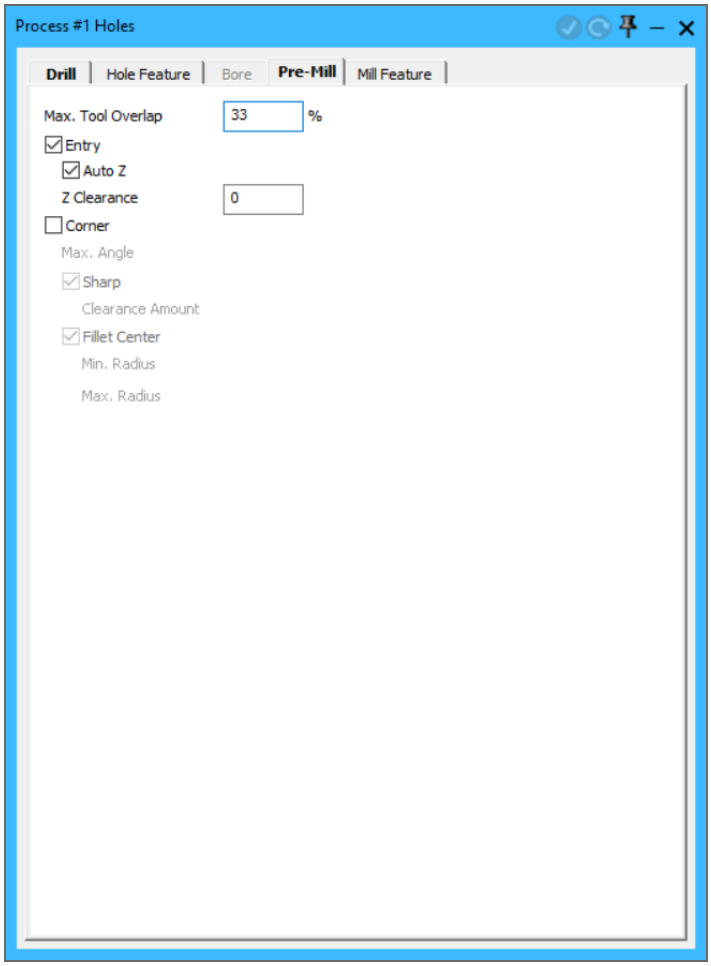


In addition, the Drilling process specification dialog will appear.

2. Enter the following into the Drill process shown.

   This process sets the clearances to 0.1" above the material which is at `Z0`. The tip of the drill will cut 0.45" deep and the tool will feed in and rapid out of the hole.





3. Click the Pre-Mill tab and set the parameters as shown.

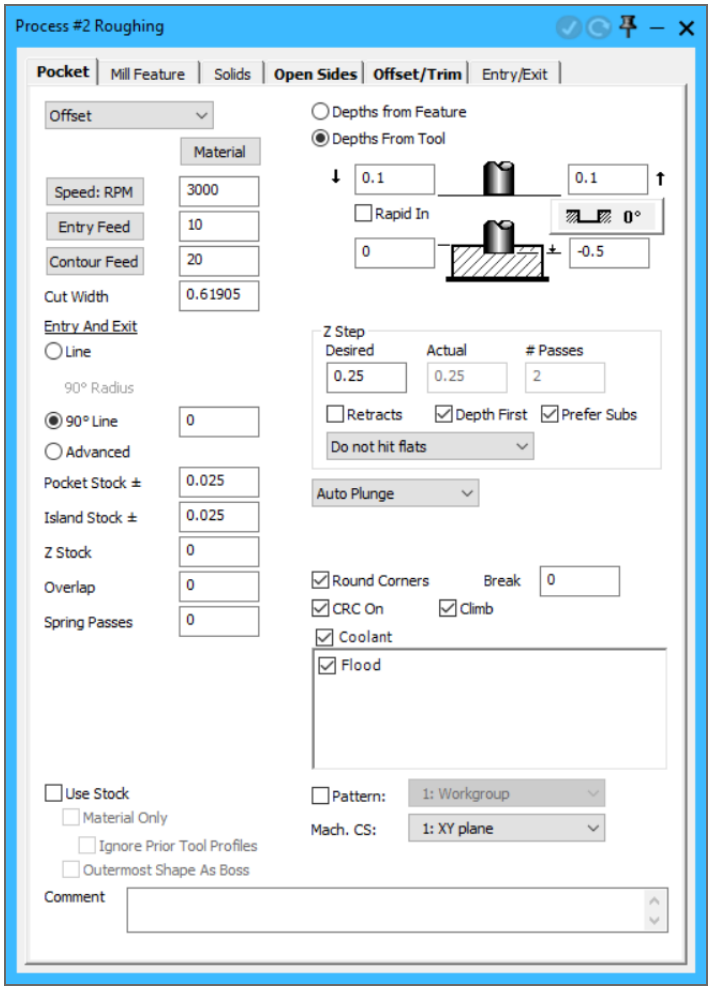
   These settings generate entry holes for the following tool.

4. Close the Process dialog (`Ctrl+W`). 

5. Double-click Process tile #2 in the Process list and click the Roughing process. 

6. Select the Rough Pocketing tool suggested (Tool #2).

7. Create the Roughing Process shown.



This is a standard Offset roughing process. The tool will cut to Z-0.5" in two steps of 0.25". We have an entry move of .05 perpendicular to the wall. Typically this is where your cutter comp will be applied. Then we have a radius move of .25 as we cut into the wall. This is a roughing operation so we aren't concerned with leaving a clean wall. In fact, we are leaving 0.025" of stock on the walls for the next operation to clean up. Different stock amounts can be specified for pocket and island walls. The value entered indicates how much material will be left on the walls (per side) after the toolpath is completed. In this case we are leaving the same amount of material.

The tool entry style selections determine how the tool will enter the material. When Auto Plunge is selected, the system will calculate the optimal location for the tool to plunge into the material to begin cutting.

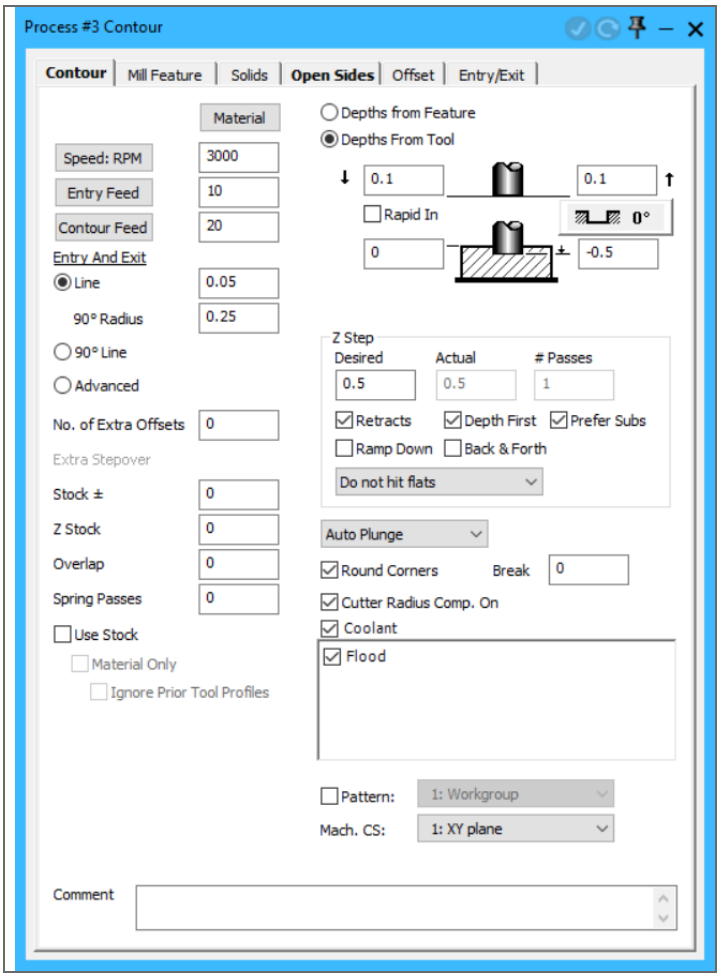
The Open Sides tab title is bold, which means it could have some bearing on the toolpath, but this operation will not have any open sides. You can ignore the Open Sides tab in this case.

8. Close the Roughing Process dialog (`Ctrl+W`) .

9. Double-click position #3 in the Process list and choose Contour.

10. Choose the .5” Finish Endmill (Tool #3).

11. Create the Contour Process as shown.



This operation will also cut to a final depth of Z-0.5" but will do so in a single step. As this operation is finishing walls we have specified that the tool should have an entry and exit line and radius. The radius will be added tangent to the start feature of the toolpath and the line will be tangent to the radius.

12. Close the Contour Process dialog (`Ctrl+W`).

The processes that will rough and finish the shape have now been completed. The order in which operations will be created is based on the order of processes in the Process list. Anytime a roughing process is created, all other processes in the same Process list will work in reference

to the roughing process. Drilling processes will drill entry holes for the pocket. Contouring processes will automatically finish the pocket. Combining processes in this manner make “multi-process operations”. Multi-process operations are very powerful and useful in that the processes have an associativity to each other, making your job easier.

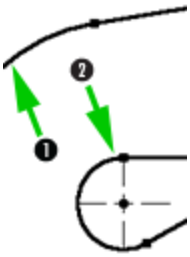
Basic Operations 1-3

With the Process list complete, we need to select the cut shape. Pocketing roughing operations can only be performed on closed shapes.

1. Click anywhere on the large, outer shape.

   When creating pocketing roughing operations, it is only necessary to select one feature of the contour to be pocketed. Because pocketing roughing processes can only be applied to closed shapes, you need only indicate which shape you wish to pocket and the system will do the rest.

2. While holding down the `Ctrl` key, click any point or feature of the boss.



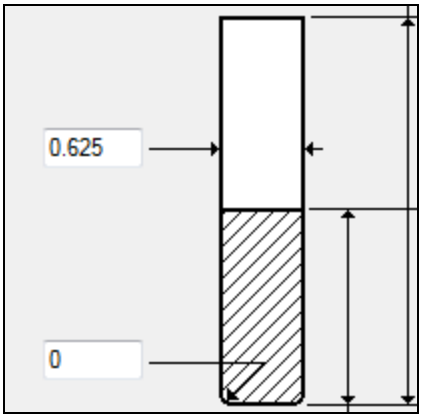
   Holding down the `Ctrl` key allows you to select more than one piece of geometry at a time. Once the pocket geometry is selected, the floating `Do It` button next to the Process tiles becomes active. Clicking on the `Do It` button applies the entire Process list to all selected geometry, creating as many operations as are required.

3. Click the `Do It` button (or press `Ctrl+.`).

   Five operations are created and placed in the Operation list.

   The first operation is a drilling operation that drills entry holes. The system determines the location of the entry holes based on the pocketing geometry selected. The next two operations rough out the pocket. Two drilling and roughing moves are created with retracts because the Rough Endmill cannot fit through the small opening in the corner. So an entry hole is created for the Endmill to use. We will correct this by changing the diameter of Tool #2 so that it will fit through the channel. The last operations contour the pocket wall and the island wall.

4. Open Tool dialog #2 and edit the diameter as shown.



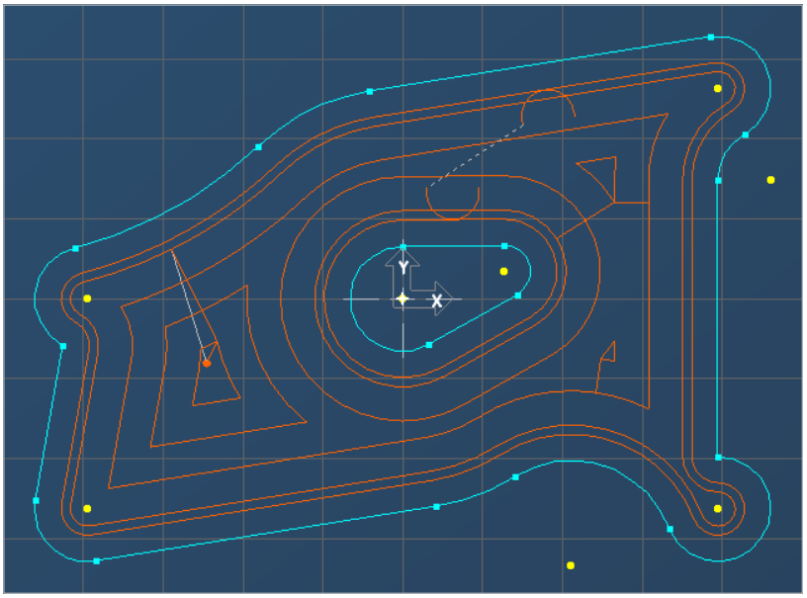
5. Close tool #2 dialog (`Ctrl+W`).

6. Reselect the geometry. Note that the `Redo` 

button is now selectable. If any changes are made to the information contained in the Process list, clicking on the `Redo` button will create new operations according to the new Process dialog information. If the old operations are still highlighted when the `Redo` button is clicked on, those operations will be deleted and replaced with the new operations created from the revised Process list.

7. Click the `Redo` button (or press `Ctrl+,`).

The toolpath will be recreated according to the new tool specifications. We now have four operations. The toolpath is drawn on the screen. Orange lines–both dashed and solid–represent the operation toolpath. Dashed orange lines signify rapid moves, while solid orange lines signify feed moves. The gray dashed lines represent inter-operational moves and approach/retract moves.



Rendering the Multi-Process Operations

Let's take a look at what we have created. We are going to render the part.

1. Click the View button (F5). 

2. Select the Isometric View  (Ctrl+I).

   You can choose whether you want to run one of the Simulations, or Legacy CPR which can be faster.

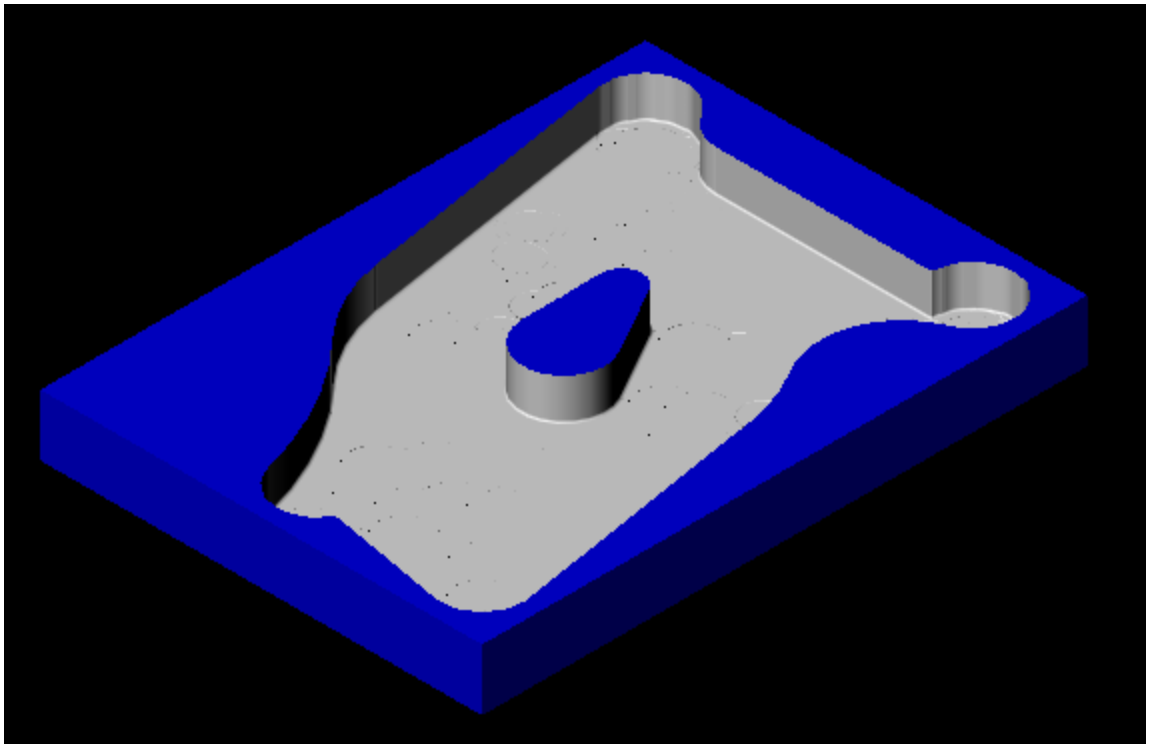
3. If you want Legacy CPR for instance, click on the arrow under the currently selected Render button in the Command Palette and scroll to Legacy CPR. Click to select and run. 

   When Cut Part Rendering is in operation, the part will no longer appear as a wire diagram but instead will be displayed as a 3D solid model. The Cut Part Rendering palette appears on the screen, this palette gives you control over the Rendering process.

4. Click the Run button. 

   Use the Rewind button to replay and if you need to, adjust the speed of replay using the slider control. A part can be rendered any time after operations have been created. The stock will

always be drawn in shades of blue. Highlighted operations will render in shades of yellow, while all other operations will render in shades of gray. Red will be displayed when non-cutting tool surfaces or rapid moves hit the material. For more information, refer to the Common Reference guide, "Cut Part Rendering" chapter.



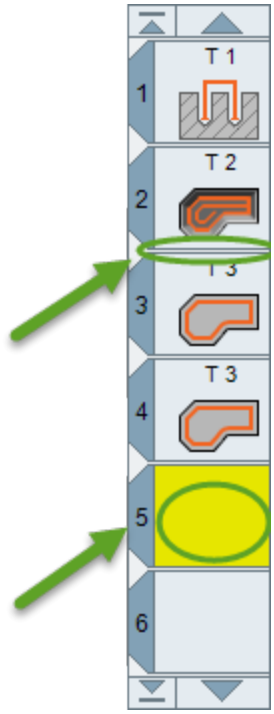
5. Click the close button to exit rendering mode. 

Basic Operations 5 and 6

We are now going to move on to the next operations but before we do we need to do a little cleaning.

1. Deselect the highlighted operations by clicking on an empty Operation location or by clicking on an insertion point between Operation locations.

When operations have been satisfactorily completed, it is very important that they be deselected before proceeding. By deselecting the completed operations, they become fixed in the Operations list and will not be affected by changes made to the Process list. Operations can be edited, but to do so they must be reselected.



2. Hover your mouse over the Axis block  in the lower left corner of the workspace.

   A temporary view palette appears. Move the mouse to Top view (just to the right of the "home" button) and click it.

   If you prefer, this can also be accomplished by selecting  Top View from the Main Menu View dropdown menu. All items available in the View Control Palette can be accessed from this View dropdown menu.

3. Click somewhere off the shape to deselect the part geometry.

Now, we will create contouring operations to finish the outside wall of the part. After operations have been created from the Process list, the entire list of Process tiles can be thrown away, and you can start creating a new Process list from scratch. However, sometimes it is more efficient to simply modify existing Process tiles rather than to create new ones. We will edit existing processes.

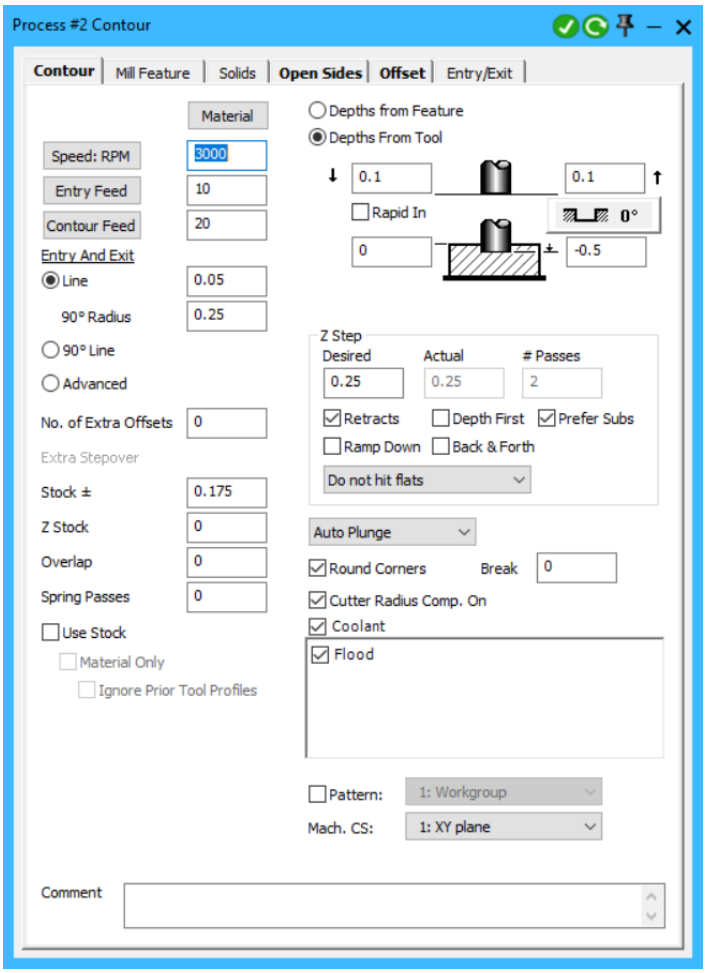
4. Click and Drag the Holes Process tile in position #1 to the Trashcan. (Mouse-drag the tile. When the hand symbol is over the trashcan the trashcan lid will open - let go the mouse button and the tile will disappear.)

5. Right-click the existing Roughing Process tile and choose Change Process from the menu. Select a Contour Process.

This will automatically change the existing roughing process to a contouring process. Tool #2 will be used again for the contouring process, so no tool changes need to be made.

6. Set the process parameters as shown.

   We are leaving behind 0.175" of stock to create the outer wall. We will cut the material in two steps of 0.25".



7. Double-click the Contour Process in position #3 (Process 2 will close automatically) and change the Stock± amount in this pass to `0.15`.

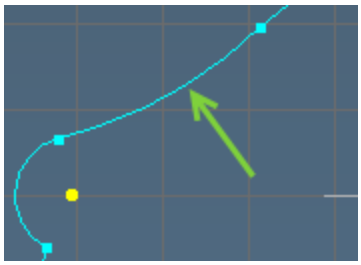
   This will cut off 0.025" of material, leaving a 0.15" thick wall.

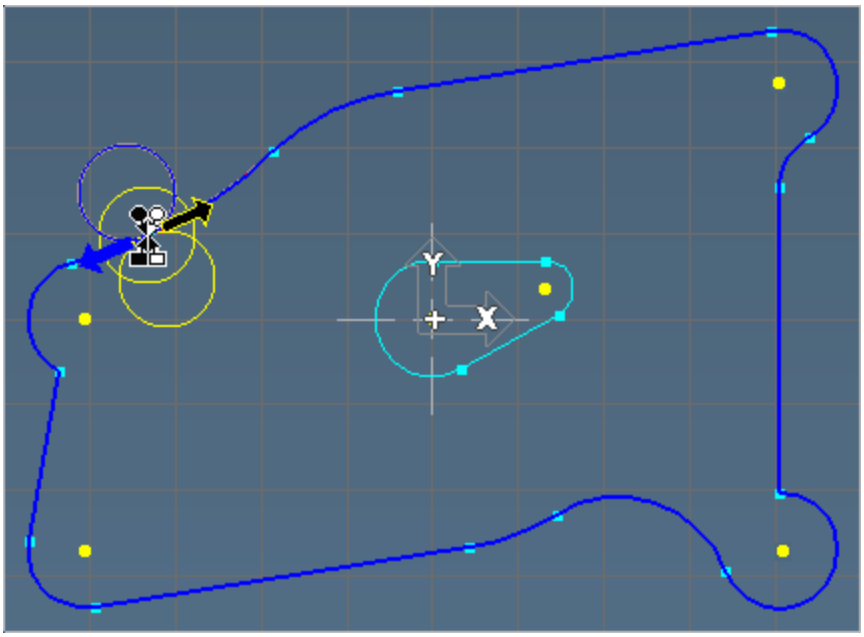
8. Close the Contour Process dialog (`Ctrl+W`) .

Contouring processes that are not associated with roughing processes use Machining Markers to designate the cut shape. These markers specify the Start and End feature, the Start and End point of the cut shape, the cut direction and which side of the geometry to cut. The arrows allow you to choose the cut direction. The three circles allow you to choose where the tool will cut in relation to the cut shape. When geometry is selected for contouring processes, the Machining Markers appear on the selected geometry. For more information on Machining Markers, refer to the Mill user guide, "Machining" chapter.

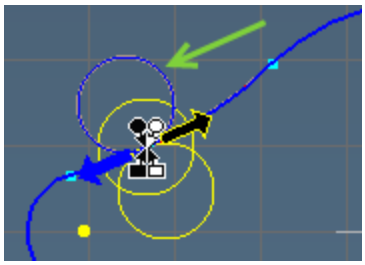
9. Click the outside shape in the position shown.

   The Machining Markers will appear at the location where the shape was selected, as shown below.

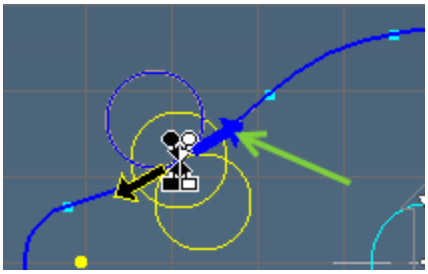


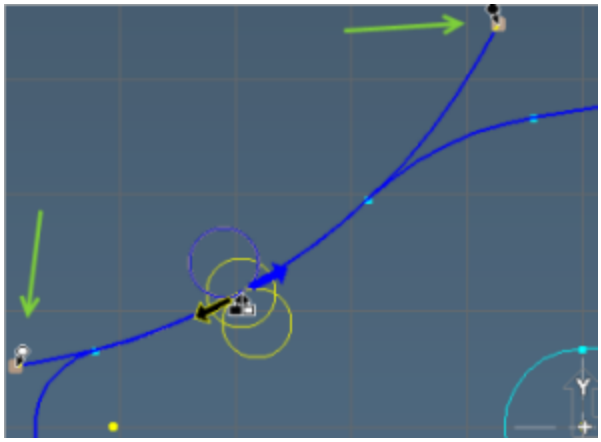


10. Now click the outside circle.



11. Click the arrow pointing right to create the toolpath for a clockwise contour.





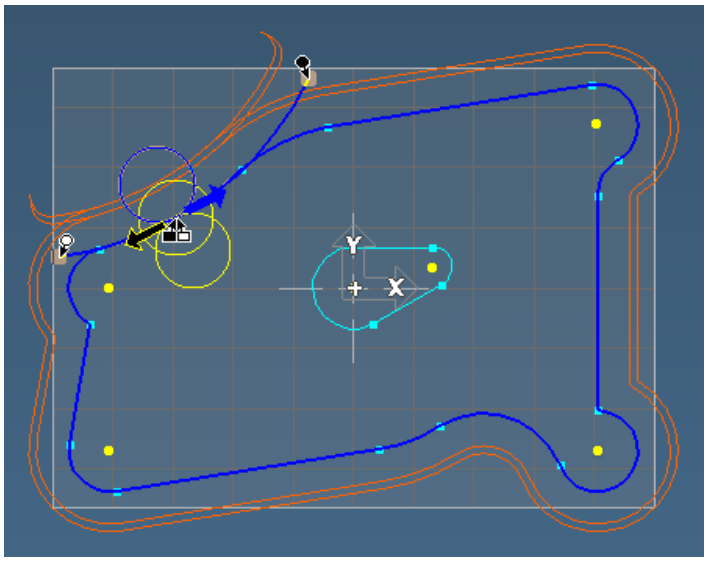
12. Drag the Start Point Marker (white circle) and End Point Marker (black circle) to the positions shown.

These positions lie on the large arc that we trimmed to create the arc that the Machining Markers are positioned on. When the Start Point Marker is dragged off the part, it automatically snaps to the nearest extension of the selected Start feature. The End Point Marker works in the same manner.

In this case, the start feature is the same as the end feature. Also, when we selected geometry for the contour, we clicked on the feature we wanted to start and stop the cut on. Because the Machining Markers appear at that point, it is not necessary to move the Start and End Feature Markers.

13. Click the Do It button (`Ctrl+.`).

The toolpath is drawn in orange. The dashed gray lines represent the approach and retract moves to and from the toolpath. In this case, they are all rapid moves.

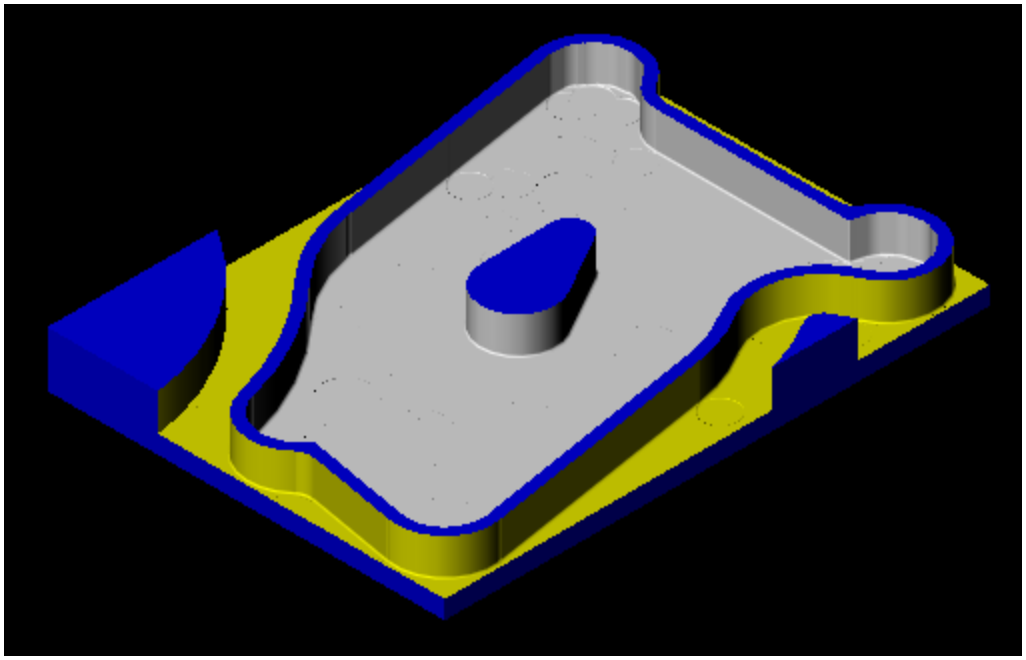


14. Select the Isometric View (`Ctrl+I`).

The rendering process does not restart whenever the view is changed, therefore you can change the view at any time during rendering to get a better view. Use the slider to slow or increase the speed or rendering at any time.

15. Click the Rendering button on the Command palette (F6).

The cut part rendered image should look like this picture.

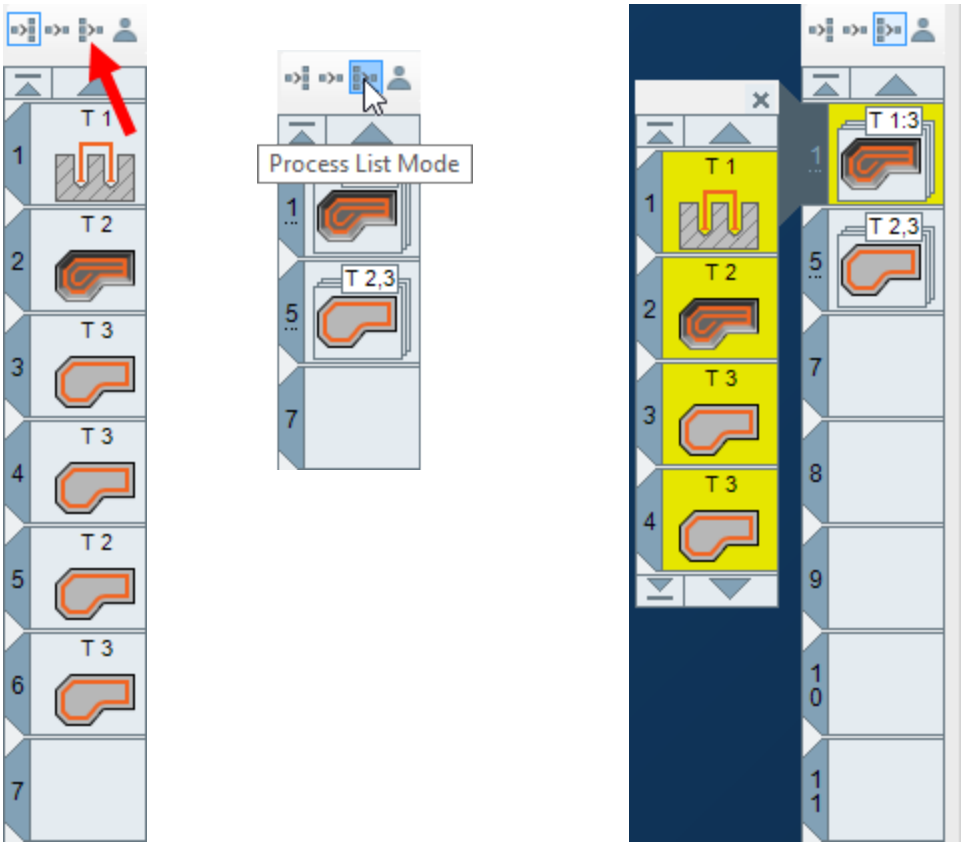


16. Deselect highlighted operations.
17. Select (Ctrl-click) then drag the Contour Processes in position #2 and #3 to the Trashcan. The Rendering will restart and reflect the change.

    There are two areas on the part stock that need to be roughed before the contouring operations that finish the outside wall are done. Geometry for this roughing operation has not yet been created. You can add geometry to a part at any time. We will put the roughing geometry in a different workgroup. Workgroups are used as layers to separate different sets of geometry to limit confusion when creating complex parts.
18. Close the Simulation palette.
19. Select the Top View.
20. Close the View Control Palette by clicking the View button again (F5).

Now look at the Operations list. Just above the list are a series of options.

21. Click the button as shown. This will enable Tile stacking, where consecutive operations, which machine the same geometry and belong to the same flow, can be stacked together. This can dramatically reduce the space operations would otherwise occupy on the operations tile list. Double-click a tile stack to open a flyout showing the contents.



22. Click the button again to return to normal view.

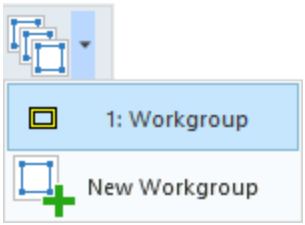
Basic Operation 7: Manual Toolpath

The contour operations left some material behind outside the pocket. We are going to remove that material.

1. Open the Geometry palette (F2).

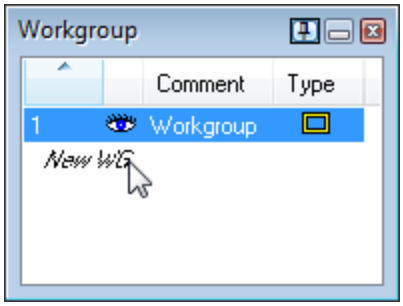
2. Click the small down arrow to the right of the Workgroup Selection button.

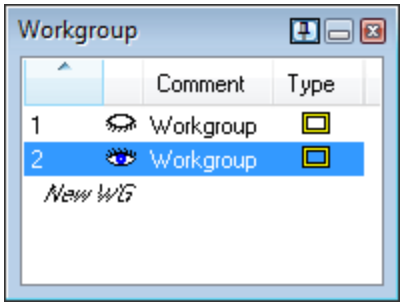
This displays all the workgroups associated with the part and enables you to easily switch back and forth between them. The current Workgroup is always highlighted in blue.



3. Move the mouse to New Workgroup and click it.

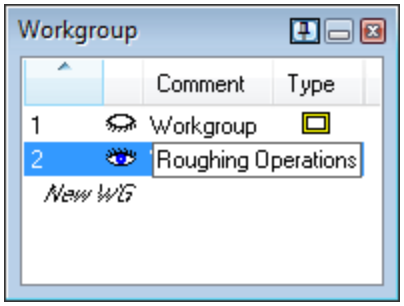
   Another workgroup is created, and the Workgroup dialog opens. This second workgroup is now the current workgroup, highlighted in blue and indicated by an open eye. Also, you will notice that the screen now displays an empty stock outline.





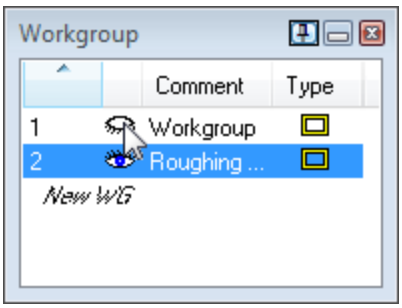
   It is advisable to label the workgroups with appropriate names so that it is easy to tell what each workgroup contains.

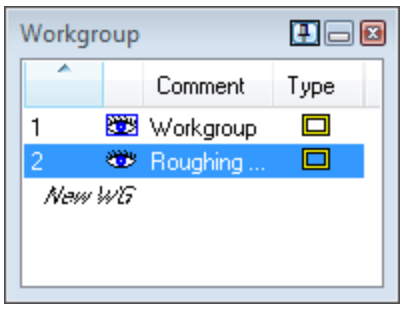
4. Enter "Roughing Operations" in the Comment box as shown. Double-click in the comment box to enter the text.



   In order to create the geometry for the necessary roughing operations, we will need to view the part geometry that we have already created. You will notice that the geometry on the screen has disappeared. This is because the active workgroup (Workgroup 2) has no geometry.

5. Double-click inside the eye of Workgroup 1 as shown below. An open eye appears. The geometry from that workgroup now appears in gray as a background element. Any geometry created in the current Roughing Operations Workgroup will be drawn in the standard blue and yellow.





6. Close the Workgroup dialog.

The part geometry previously created is drawn on the screen in gray and cannot be selected or changed without switching workgroups.

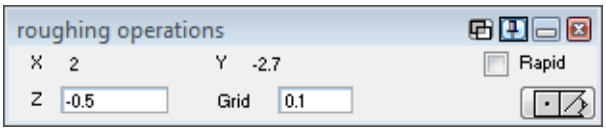
Now, we will create the roughing geometry. Since the roughing operations do not require geometry that must adhere to any blueprint specifications, we will simply create points and lines using the mouse. While this method does not provide for exact precision, it is the fastest way to create continuous, connected lines.

7. Click the Line button. 

8. Select the Mouse Line button. 

A geometry dialog will appear when the Mouse Line button is clicked. The X and Y values are not entered, but are determined by the location of the mouse. As you move the mouse, those values automatically change. The Z value specifies the depth at which the line(s) will be created. The Grid value specifies the grid spacing that the points created by clicking on the mouse will snap to.

9. Enter Z value and Grid value shown.



When using the Mouse Line method to create lines, you can select a tool and an outline of that tool's diameter will be shaded around all lines as they are created.

10. Select tool #2.

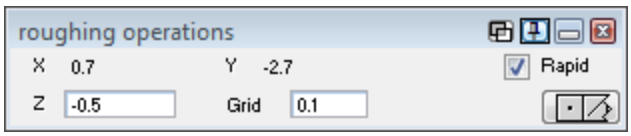
    Double-clicking on a Tool tile will bring up the Tool Creation dialog. A single click will select the tool and highlight it.

11. Click positions 1, 2, and 3 as shown.

    Points are created wherever you click the mouse, and a line will be drawn between the points created. All geometry created using this method is fully connected; the endpoints of the contour are terminators and the points between lines are connectors. You want to make sure the path of the tool covers the area of stock we need to rough.

    Because we are only going to make one contour to remove all of the excess material, we will need to rapid the tool over the part. In order to do that we will change the Z value in the geometry dialog.

12. Change the Z value and click the Rapid checkbox as shown.

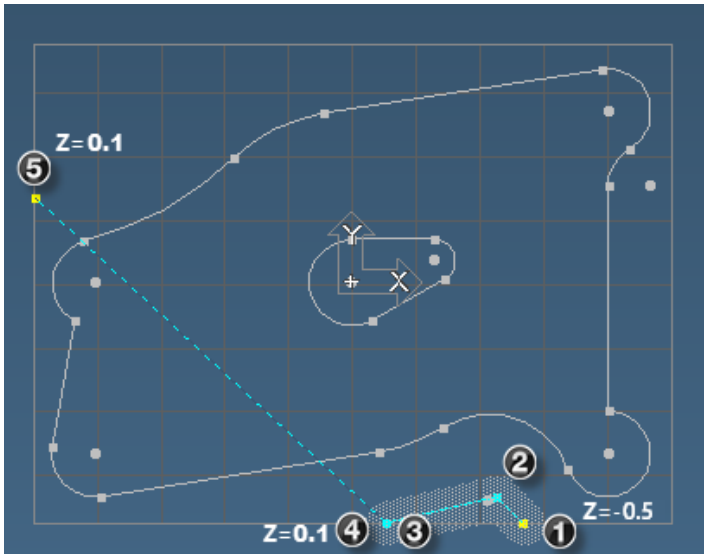


    This geometry dialog is unique in that it allows you to create rapid geometry. It is especially useful to efficiently move around the part and avoid tool interference.

13. Click the last point created again.

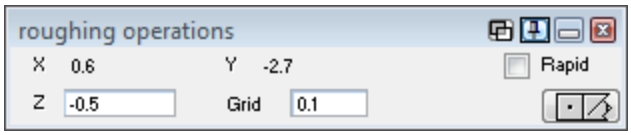
    This will create a rapid line above the stock and ready to rapid over the part. This next step demonstrates how to create a rapid move over a part to a desired position.

14. Click in the location of point #5, as shown.



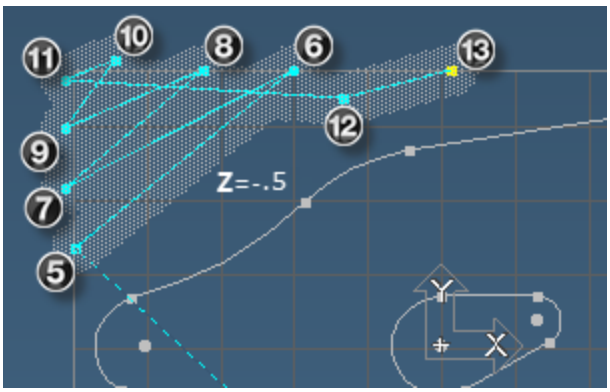
The line created is dashed and blue, indicating that it is a rapid and connected line, and the shaded tool diameter is not drawn. If the part was rotated, it would clearly show the rapid up to point #4.

15. Change the Z value and turn off the Rapid option as shown.



16. Click again on point #5 (to feed back down into the part) and create a zig-zag pattern similar to the one shown to clean up the corner.

The actual pattern is not important, just be sure to cover the area with the tool.



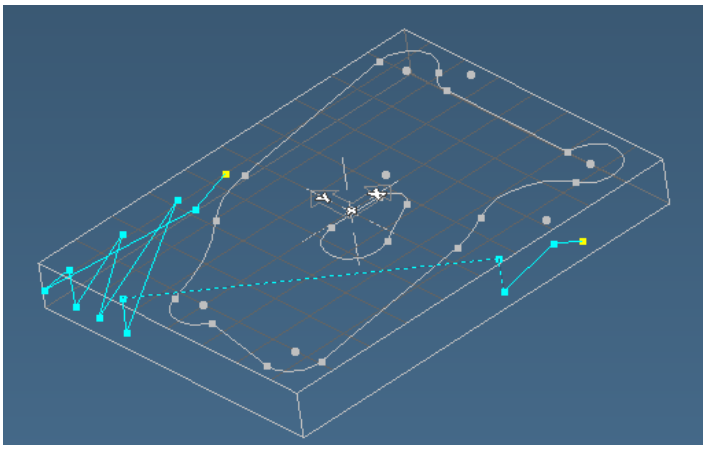
17. Click the Return to Geometry Creation palette button.

18. Close the Geometry Creation palette.

19. Main Menu View > Redraw (Ctrl+R).

To confirm that the lines are at the correct Z depths and that the rapid line over the part was done correctly, we will switch to a view that allows us to see the depth dimension of our drawing.

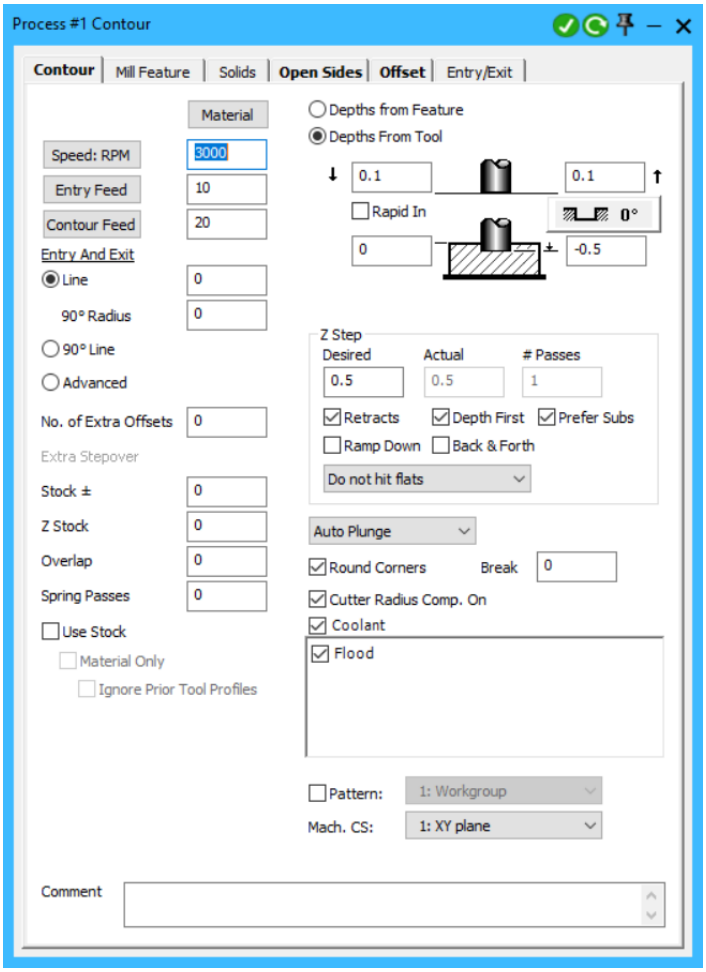
20. Switch to the Isometric view (`Ctrl+I`).



The most common problem encountered at this point involves the rapid line. If you did not reselect any points after changing the information in the geometry dialog, the rapid line will either not be above the part or will be slanting up. If this is the case, delete the mouse lines and start over, making sure that you click all the necessary points.

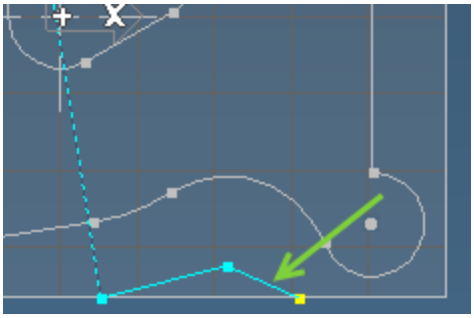
21. Return to the Top view (`Ctrl+E`).

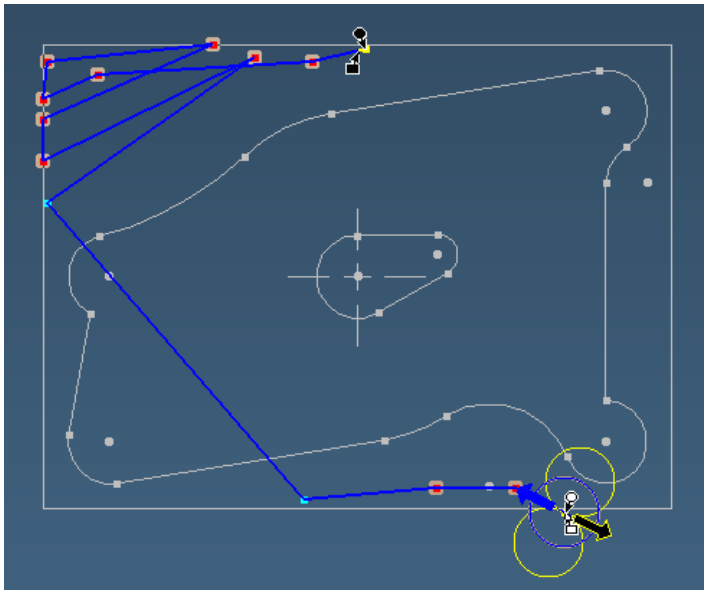
22. Create this Contour Process with the 5/8" Rough Endmill (Tool #2).



23. Click the part geometry as shown.

    Machining Markers will appear on the geometry.





24. Set the Machining Markers as shown, ensuring that you select the center circle.

    The Start and End Point Markers and the Start and End Feature Markers will move to the correct positions.

    In this particular example the Machining Markers automatically snapped to the correct position. If this were a closed shape, however, that would not be the case.

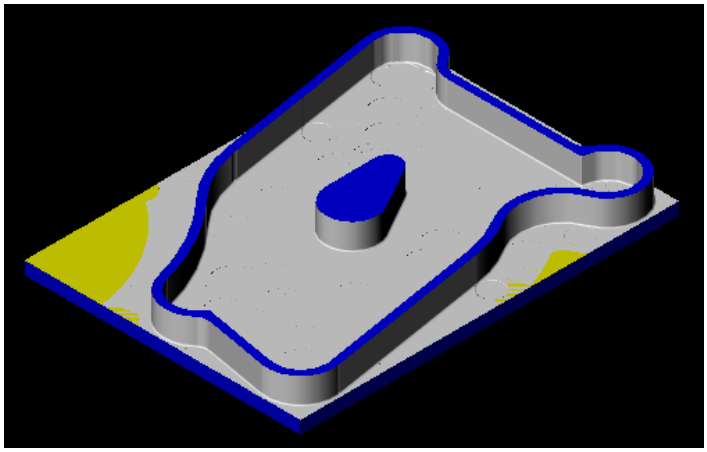
25. Create the toolpath for this Process (`Ctrl+.`).

> ⓘ To quickly position the End Feature and End Point Markers, hold down `Shift+Ctrl` and click the desired end feature. Then position the End Point Marker.

26. Switch to the Isometric view (`Ctrl+I`).

27. Render the Operations.

    The excess material should now be removed.



28. Exit Rendering.
29. Deselect any highlighted operations.
30. Empty the Process list.

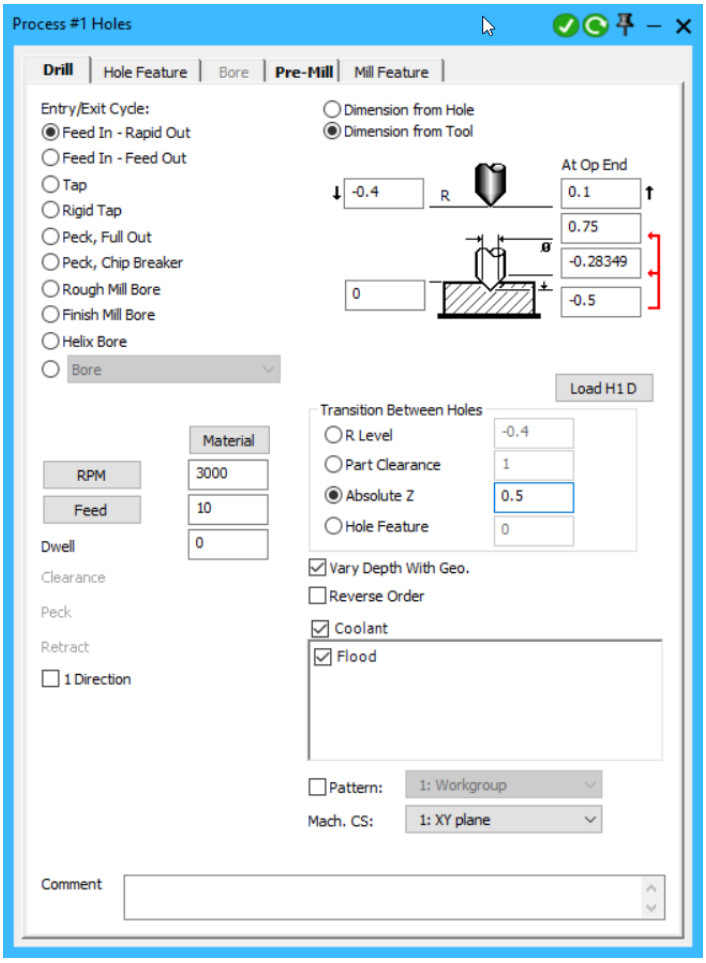
Basic Operations 8-10: Tapped Holes

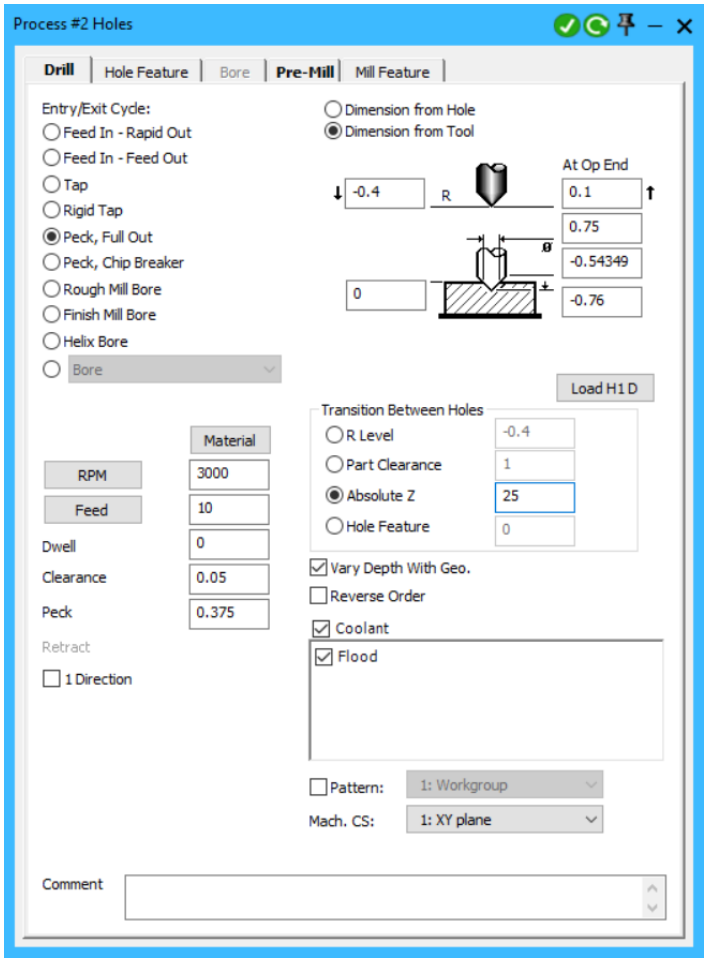
Finally, we will create operations to drill and tap four holes. To do this, we will need to switch back to the first workgroup so that we can select the points to drill.

1. Select `Workgroup 1` in the Workgroup Selection pop-up menu.
2. Return to the Top View (`Ctrl+E`).

3. Create a Holes Process as shown using the 1/4” Spot drill (Tool #4).

   This process will drill a chamfer for the tapped holes. The spot drill goes to its full diameter at Z = -0.5.



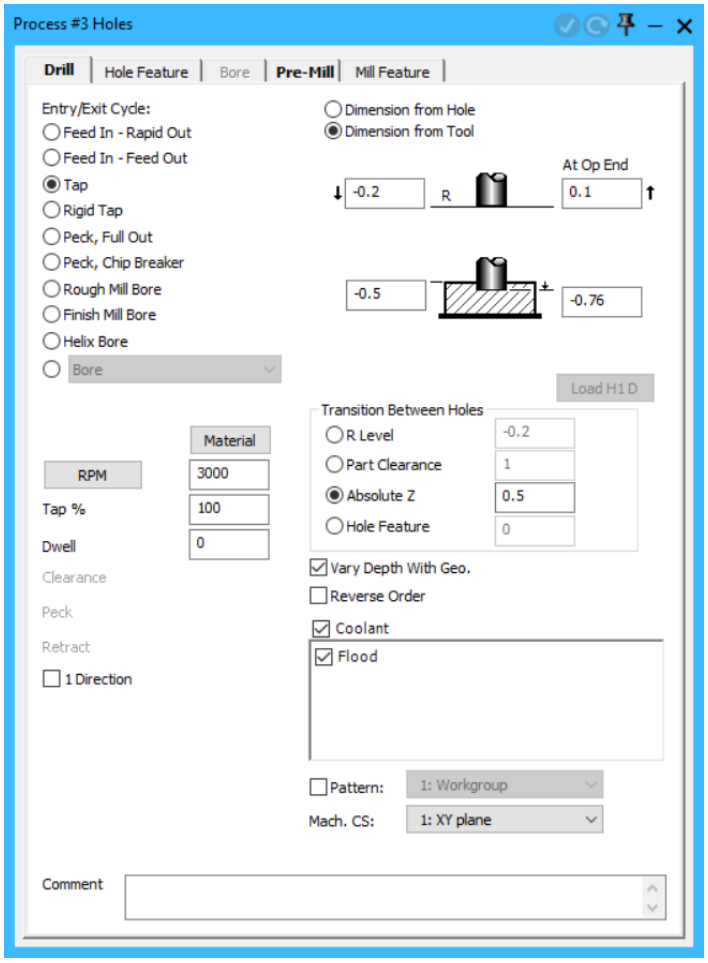


4. For the second process in the multi-process group, create a Holes Process as shown using the .201" Drill (Tool #5).

   This process drills the tap hole. We are sure to send the full diameter of the tool all the way through the part to Z = -0.76".

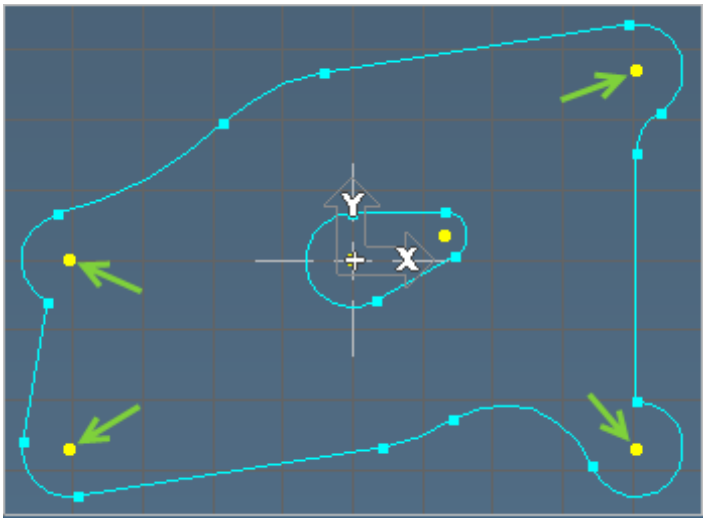
5. For the third process in the multi-process group, create a Holes Process as shown using the Tapping Tool (Tool #6).

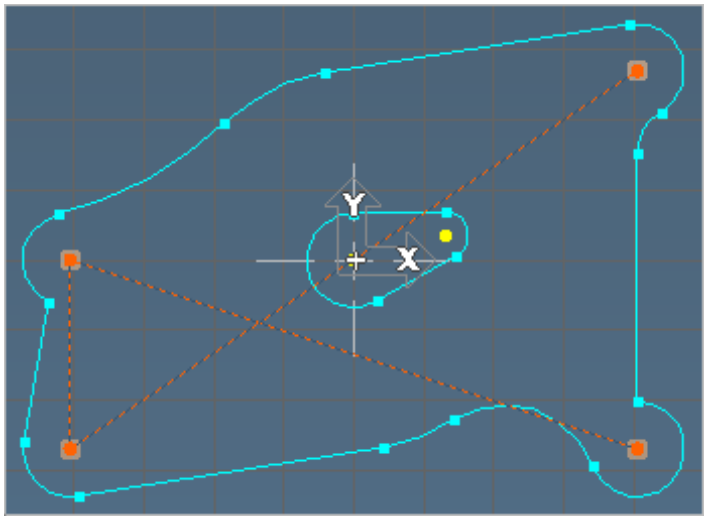
   The process taps the hole. Be sure to select `Tap` for the Entry/Exit Cycle.



We will now select the points to be drilled and tapped.

6. `Ctrl+click` the four points shown below.





7. Create the toolpath.

Three drilling operations will be created and placed in the Operations list. The holes will be drilled in the order they were selected, so the toolpath on your screen may vary slightly from the one shown here.

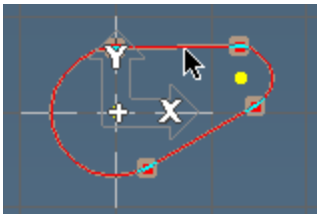
Multiple drilling operations created from one Process list generate a subprogram of the selected pattern of points to be drilled in the posted output. Each of the operations that uses that pattern of points calls that subprogram in the posted code.

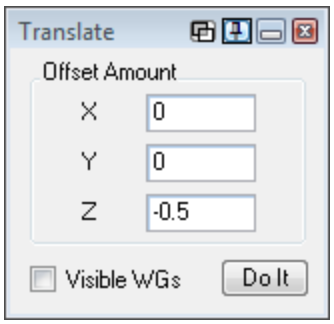
8. Switch to the Isometric View.
9. Render the Operations.

Basic Operation 11: Tapered Boss

Now we will create a tapered wall on the boss. The boss geometry must be drawn at the correct Z depth in order for the taper to be created correctly. The system calculates the taper from the Z level of the geometry, not the Surface Z value entered in the Entry/Exit Clearance Diagram.

1. `Double-click` the boss geometry so that it becomes selected.





2. Choose Modify >  Translate.

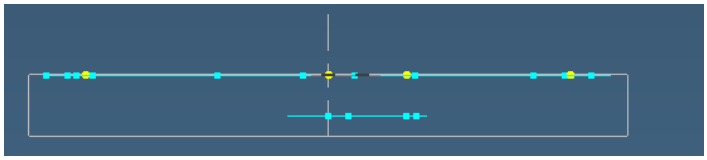
3. Enter the data shown.

4. Click the Do It button (`Ctrl+.`) and close the Translate dialog.

This will move the selected geometry down in Z to -0.5 which is the bottom of the boss.

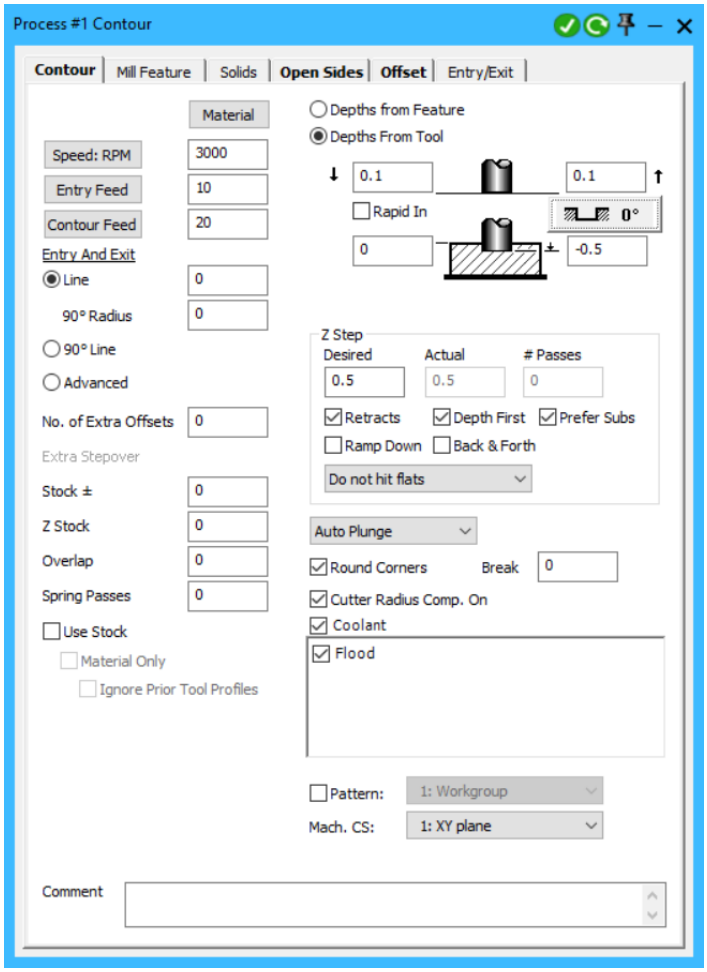
5. Switch to the Front view (`Ctrl+F`).

This allows you to see the geometry at a different depth level.



6. Switch to the Top View (`Ctrl+E`).

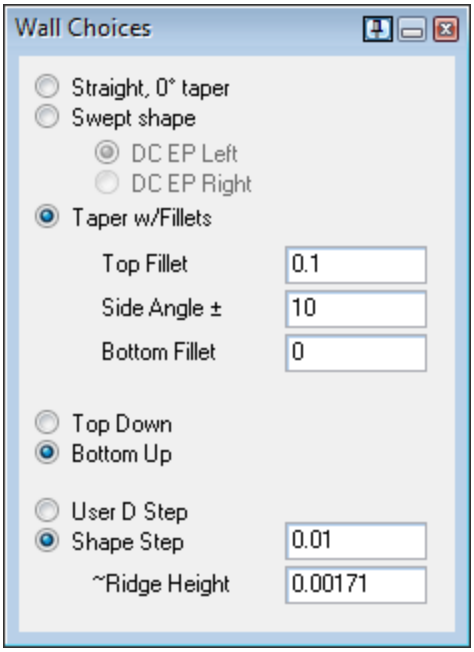
7. Create this Contour Process with the 1/2" Finish Endmill (Tool #3).



8. `Click` the Wall Control button.

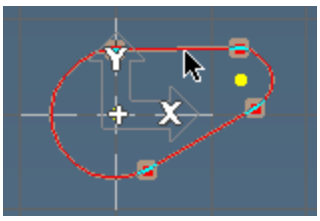
This brings up the Wall Choices dialog which provides options for creating tapered and swept walls.

9. Select the Taper w/Fillets option and enter the information shown below in the Wall Choices dialog.



10. Close the Wall Choices dialog.

11. Select the boss geometry.



The machining markers will come up on the selected shape. Make sure the outside circle and right arrow are selected.

The Start and End Markers do not need to be moved.

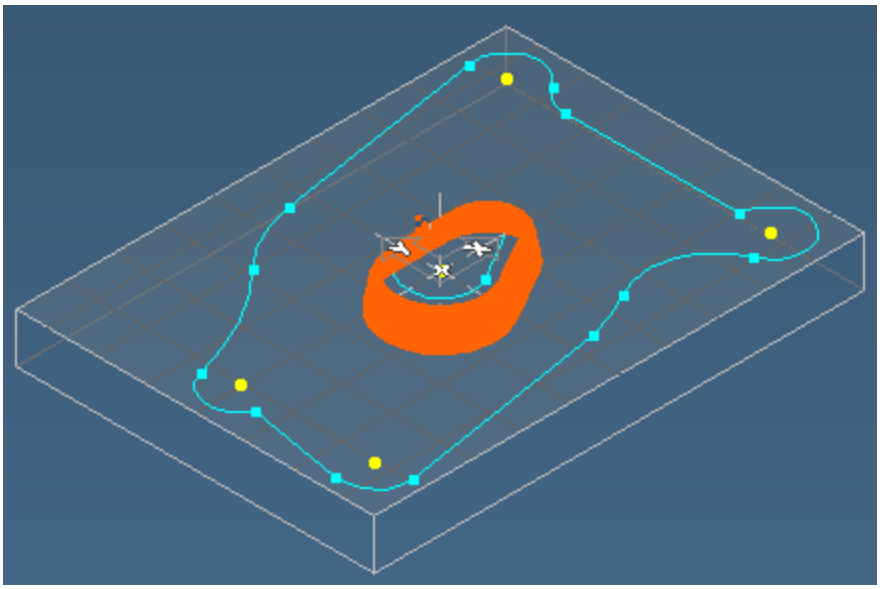
12. Select Operation tile #5.

This is the operation created from the first Process list that finished the boss with a straight wall. By selecting just this operation we can replace it with the new tapering operation.

The Redo button becomes active indicating that an operation can be reprocessed.

13. Click the Redo button (Ctrl+,).

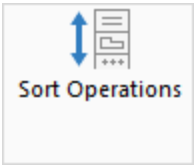
A new toolpath will be generated that creates a tapered wall on the boss.



Organizing the Operations

All the operations to machine the part have been created. All that remains is to organize the operations in the Operations list to provide for the most efficient machining of the part. This will be accomplished using Sort Operations.

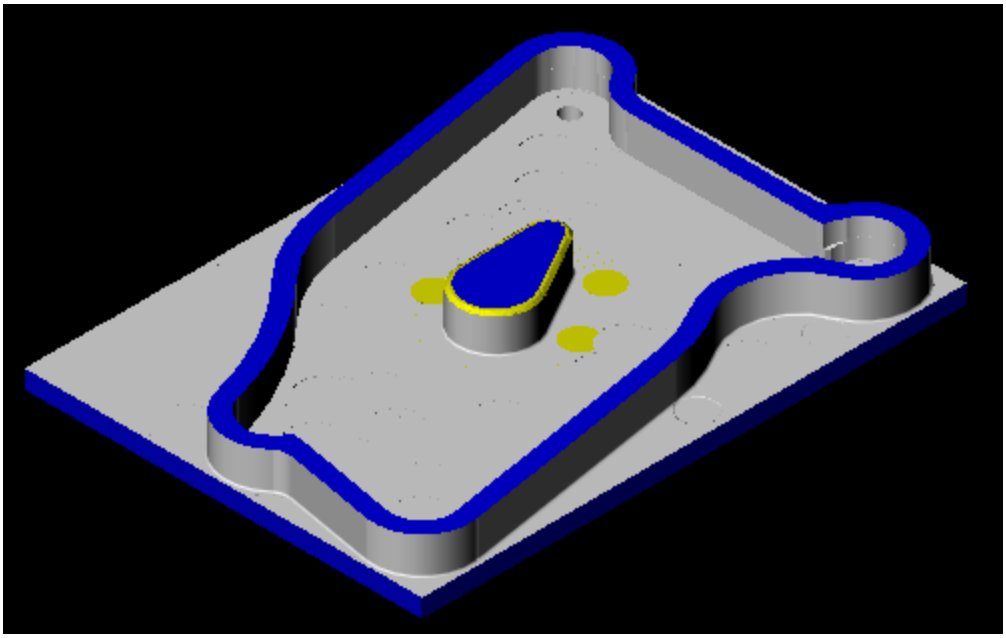
1. Right-click an Operation Tile and choose Sort Operations from the menu.



Sort Operations reorganizes the Operations list according to tool number and creation order. Operations created from the same Process list will never be moved in front of each other, regardless of tool number. For example, the first Process list created four operations. The drilling operation created first in the Process list will never be moved below the roughing or contouring operations created from that same Process list, even if the tool number for the drill was higher than the numbers of the other tools used in that Process list.

The contouring operations using the finish endmill, Tool #3, will be moved down in the Operations list after the roughing operations that use Tool #2. This minimizes tool changes and provides for a shorter cut time.

2. Render the Operations.



The machining and rendering of the part is complete. We have a good part and there are no errors.

Final Steps

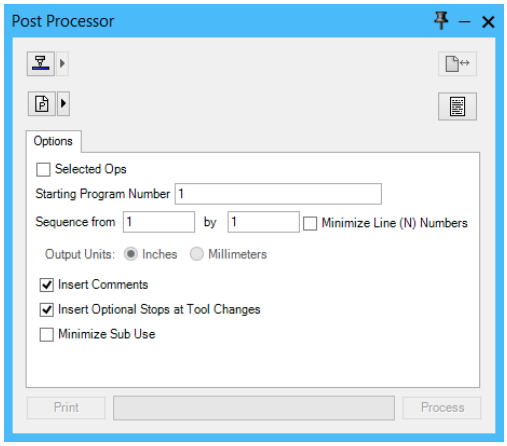
Post Processing

Now that all of the operations to machine the part have been satisfactorily created and verified, we need to post process the file. Post processing will create a text file that the CNC machine will use to cut the part.

1. Click the Post Processor button in the Command palette. 

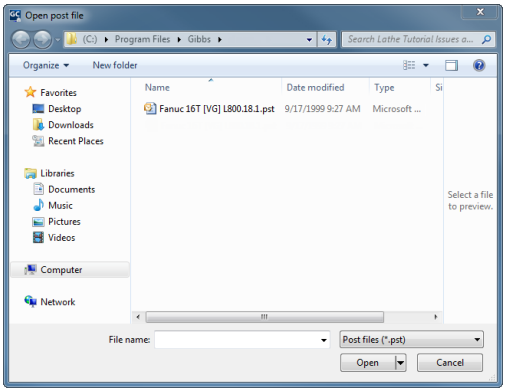
   The Post Processor dialog shown below opens up. This dialog allows you to select a post processor, specify a program name and the format of the output.

2. Click the Post Processor Selection button. 



Now select a post processor. Change drives, directories, or folders in order to locate the post processors on your system. Typically, post processors are found in a directory/folder called Posts. Once the Post folder is chosen in this dialog, it will remain as the default folder when selecting post processors.

3. Locate the post processors on your system and select one that is appropriate, i.e. a 3 (or more) axis Mill post. If you do not have one, there is one located in the Sample parts folder for use only with this tutorial for learning purposes.

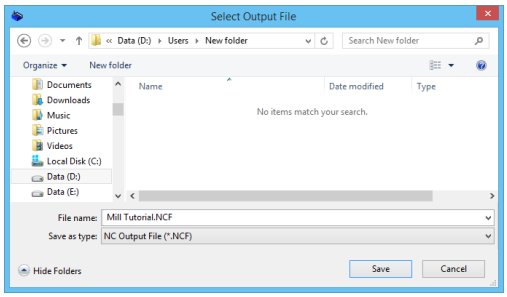


4. Click the Open button.

The name of the selected post processor will appear in the dialog next to the  button.

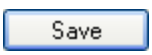
5. Click the Program Name button.

This reveals the dialog shown. Now name the posted file and designate where it will be saved.



The Program Name defaults to the name of the part file. You can change the name or leave it as it is. Part files have a *.VNC extension while posted output files have an *.NCF extension.

6. Click the Save button.



Once you have selected a post processor and named and saved the program, the Process button at the bottom of the dialog becomes active.

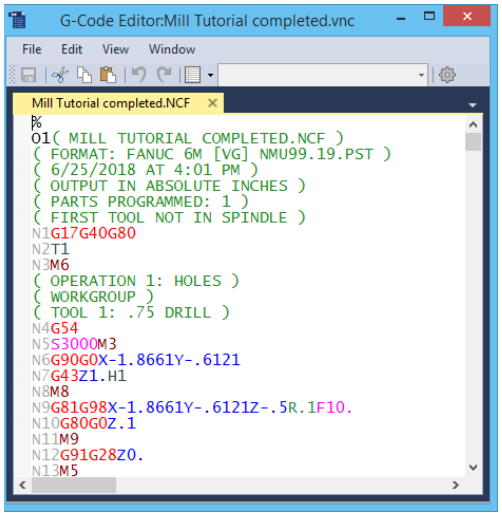
When you click the Process button, the system generates a text file that can then be communicated to the CNC machine to cut the part.

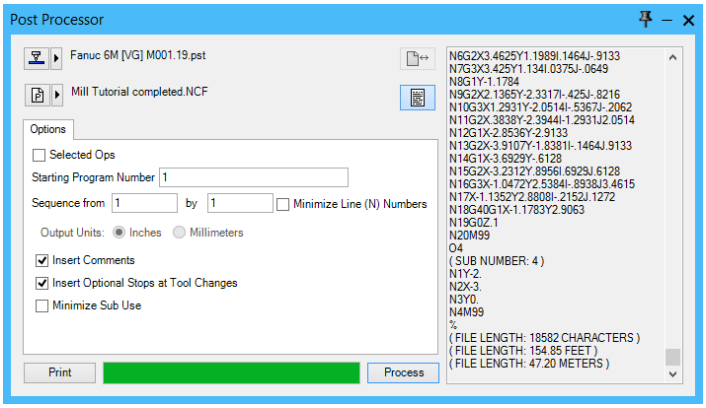
7. Click the Text Window button. 

This will extend the dialog, allowing you to view the posted code as it is processing.

The actual dialog that appears might be different. If you have chosen to use the Internal Post Editor in the  Preferences > G-Code Editor Settings dialog, then that will also open in another window. This customizable, color-enhanced editor allows you to directly edit the program using cut/copy/paste etc., with multiple subs and flows clearly distinguished. You can also find a link to this internal editor under the Plugins >Main tools menu  , which will enable you to edit existing .ncf files without having to open the .vnc file.

8. Click the Process button. 





You can click the Pause button on your keyboard to momentarily stop the processing so that you can read the program. Once the program has finished processing, the Print button becomes active and the posted code can be printed. Now that the program has been generated, it can be used to cut the part on the CNC machine. Any necessary changes can be made using the internal post editor

9. Save the part file.

This tutorial is complete.

Hole Manager and Hole Feature

Hole feature is a tab located in the Drilling dialog. It can be a powerful tool for machining multiple hole segments when used with Hole Manager.

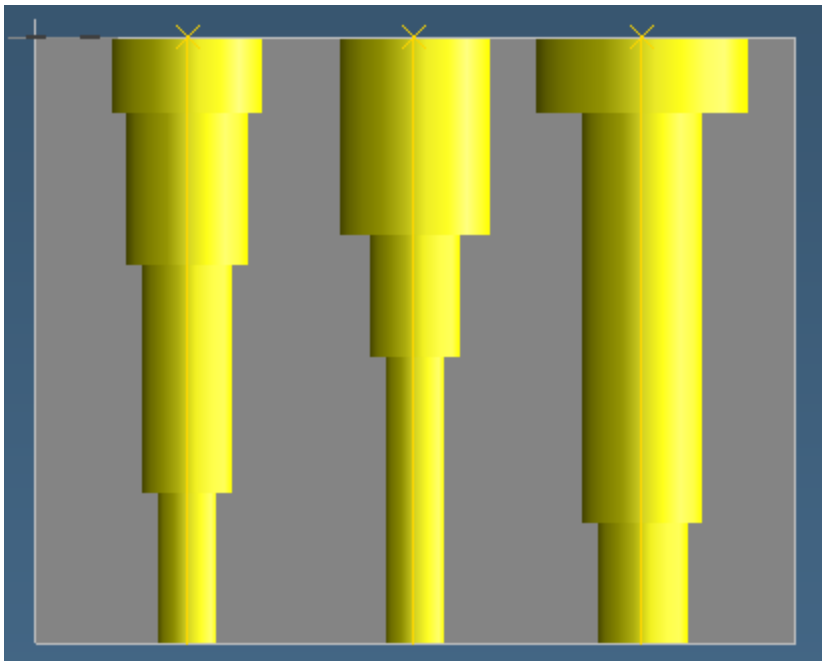
In this exercise we will machine a part using the Match Segment by Properties/Index in the Hole Feature Tab.

1. Open part file “Hole Match.vnc” located in the Sample Parts > Production > Tutorial Parts - Required folder.

   Looking at this part, you can see there are three holes, with varying length and number of segments. Four tools are provided in the tool list.

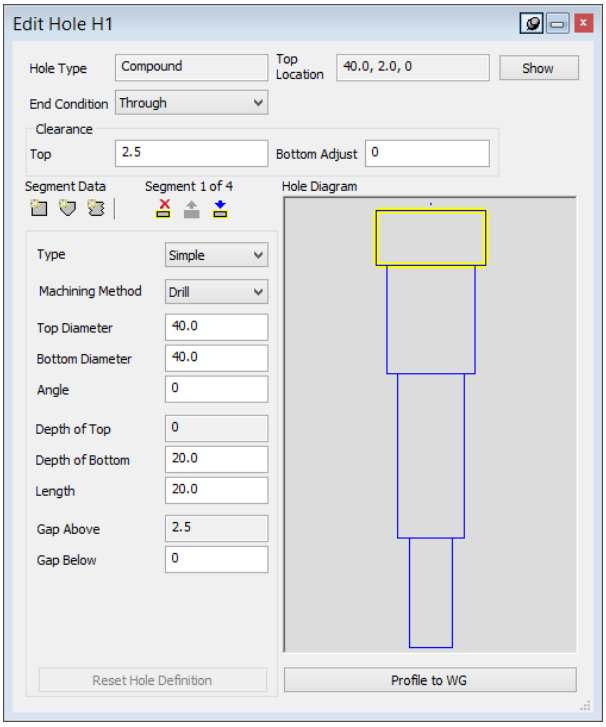
2. Select the part by clicking anywhere on it (it will turn yellow).

3. Open Hole Manager (Features > Hole Manager) and click the Run AFR button.



We will now create operations to machine these holes, using the data from the Hole Manager. You will note that the leftmost hole (with four segments) is the first hole in the list. This is our **Reference hole** the details of which are used in the Hole Feature Tab. Right-click the word Compound on the first hole (H1). Select Edit from the menu. Examine the hole profile by clicking within the Hole Diagram but do not change any values.

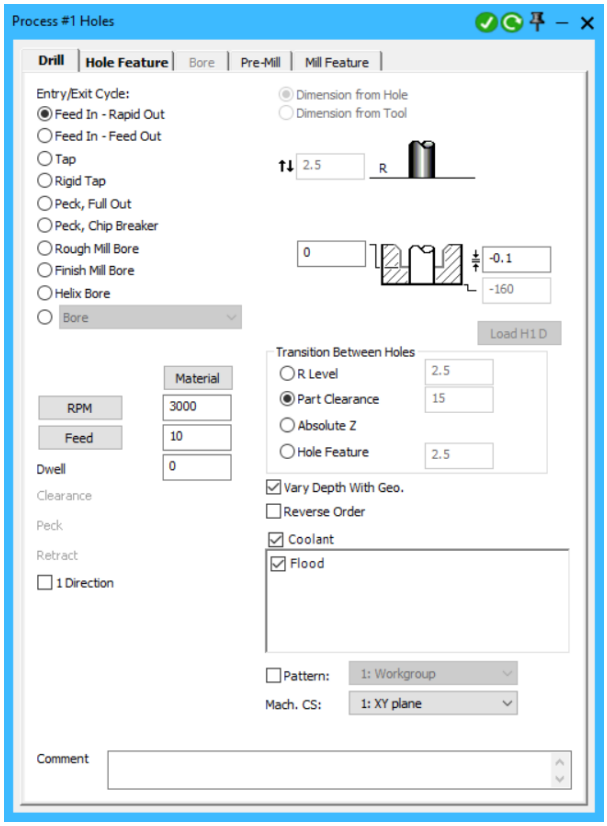
4. Close the dialog.



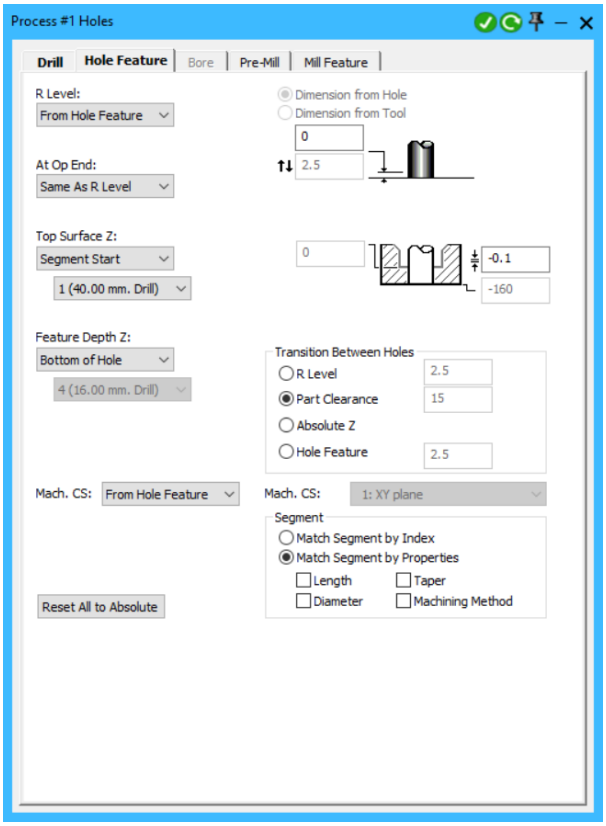
Click on the segments in the Hole Diagram window to examine the segment data.

5. `Click` (`Ctrl+A`) in the Hole List to select all the holes.

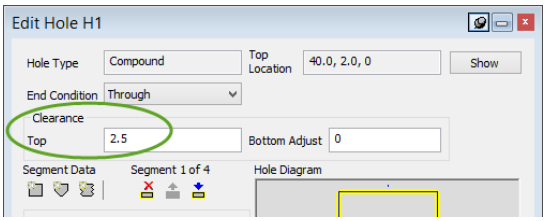
6. Create a Holes process by double-clicking a process tile and selecting the 16mm Endmill (Tool #1). We will be using this tool to drill the bottom segment.



7. Click on the Hole Feature tab.

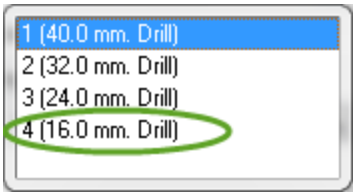


Set the R level to From Hole Feature which means the R level data will be taken from the Hole Manager. (Top of hole plus Top Clear value).



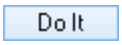
At Op End should be set to Same as R level.

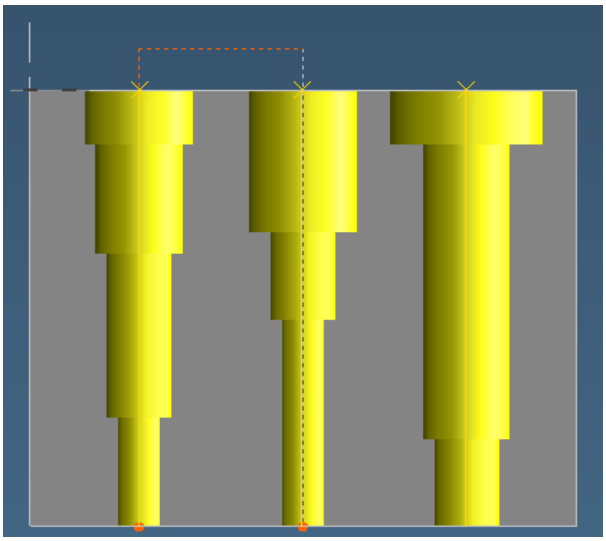
8. Choose the Segment Start option in Start Depth dropdown. This opens a further dropdown, listing all segments that are part of the **Reference Hole** (H1). Select Segment #4, the bottom segment. Choose Segment End and Segment #4 for Feature Depth Z.



9. In the Segment section (bottom right of the dialog), choose the Match Segment by Properties Radio button and check Diameter. This means that for all other holes, it will drill the first segment that is the same diameter as the 4th segment of the reference hole. (16mm)

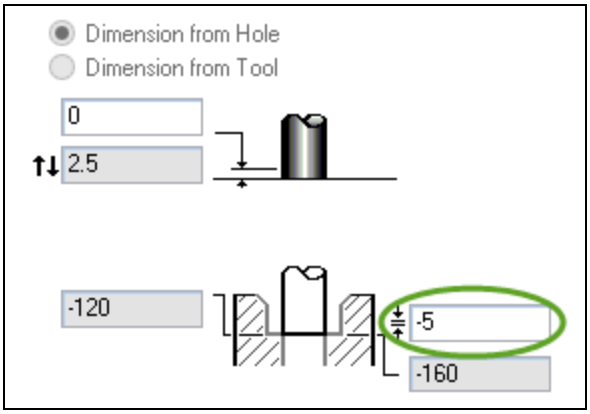
10. Click Do it .



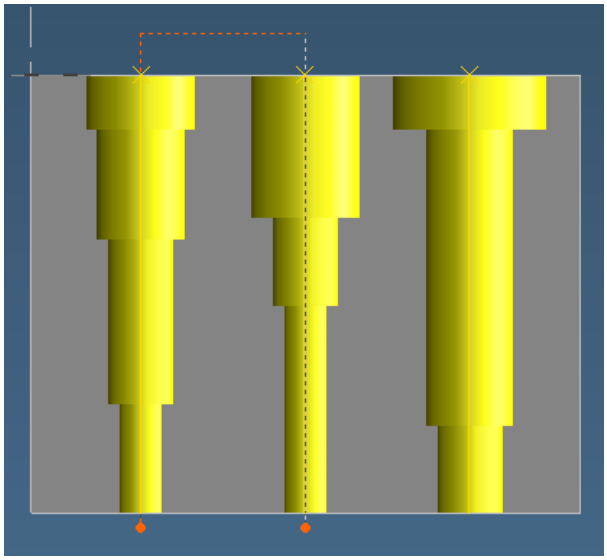


Holes 1 and 2 are machined. You will notice that the toolpath stops at the bottom of the hole. In this case, as this is a through hole, we want to plunge deeper to clear the bottom. Hole 3 is not machined because it has no 16mm diameter segment.

`Double-click` the Holes Process and add an adjustment value of `-5` to the Process dialog as shown.

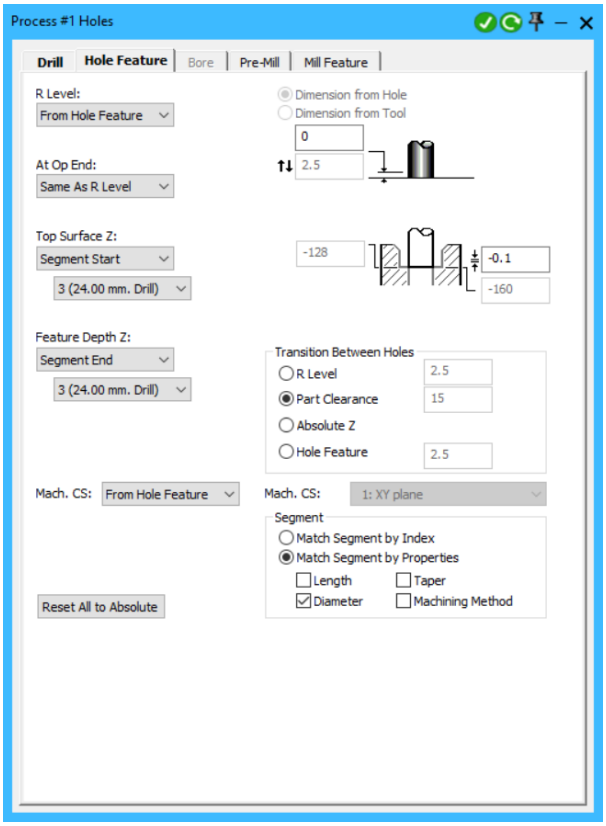


11. Click Redo .



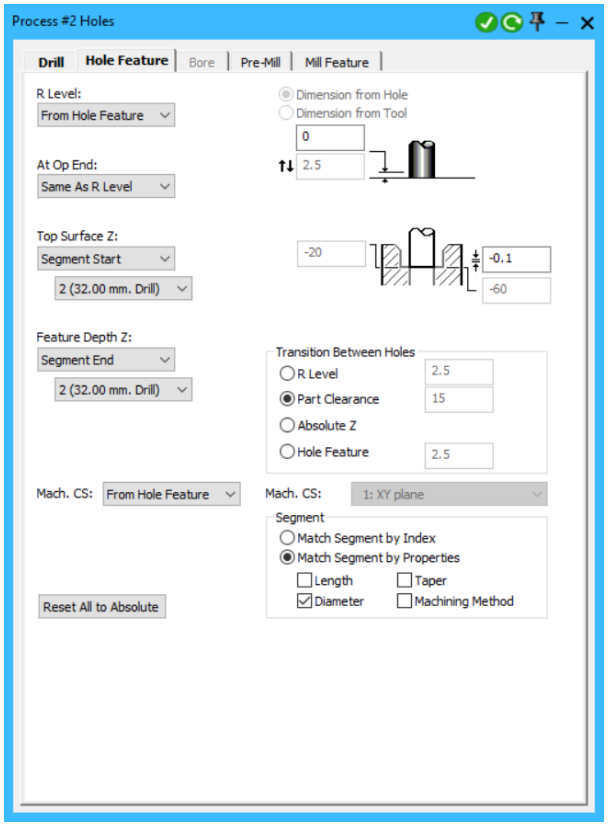
This time the tool plunges all the way through the hole.

12. Deselect the Operation tile and drop Tool #2 onto the existing Process #1. The Drill Tab is OK, but in the Hole Feature Tab enter the following:

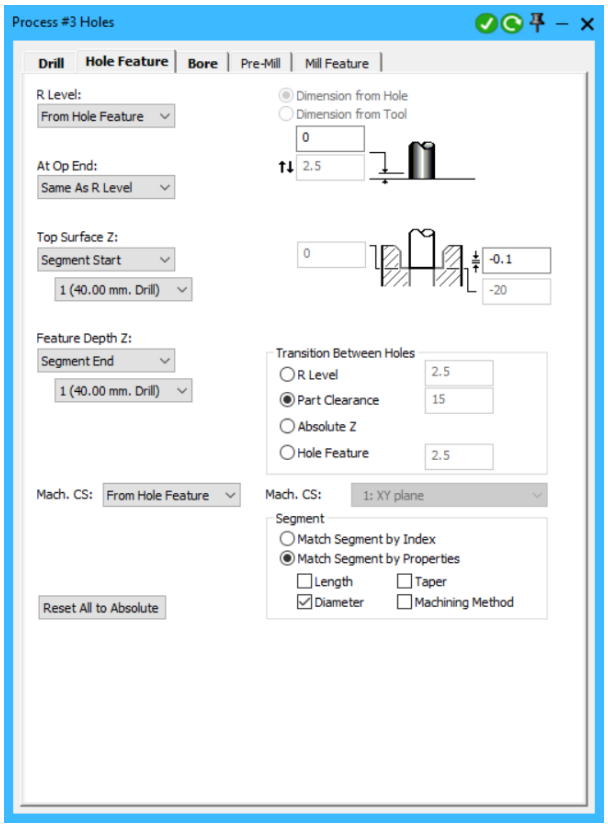


We will machine Segment 3 and any other segments that match the diameter of segment 3 regardless of their length.

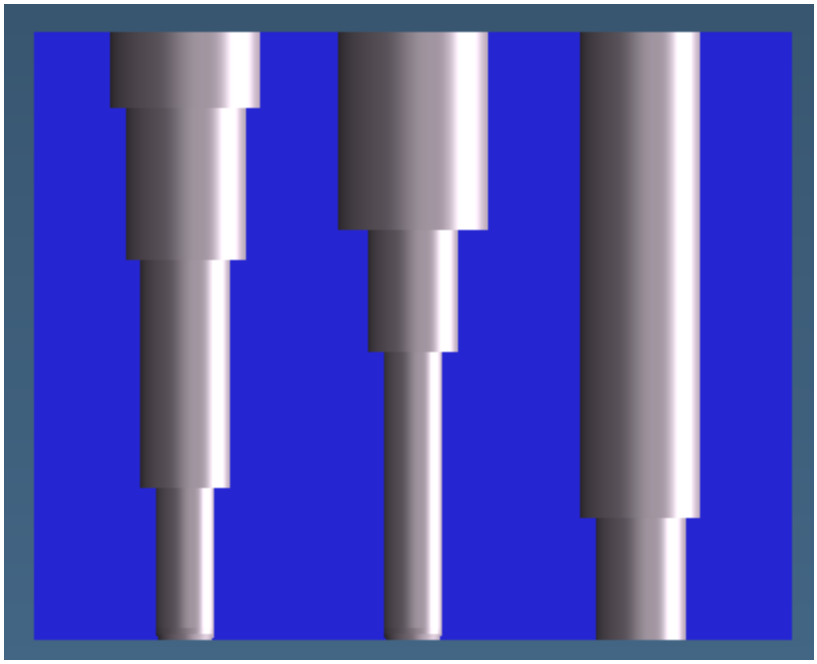
13. Create another Holes Process at position #2 using Tool #3, the 32mm REM. Enter the following into the Hole Feature Tab.



14. Create a Holes Process #3 using Tool#4, the 40mm REM. Enter the following into the Hole Feature Tab.

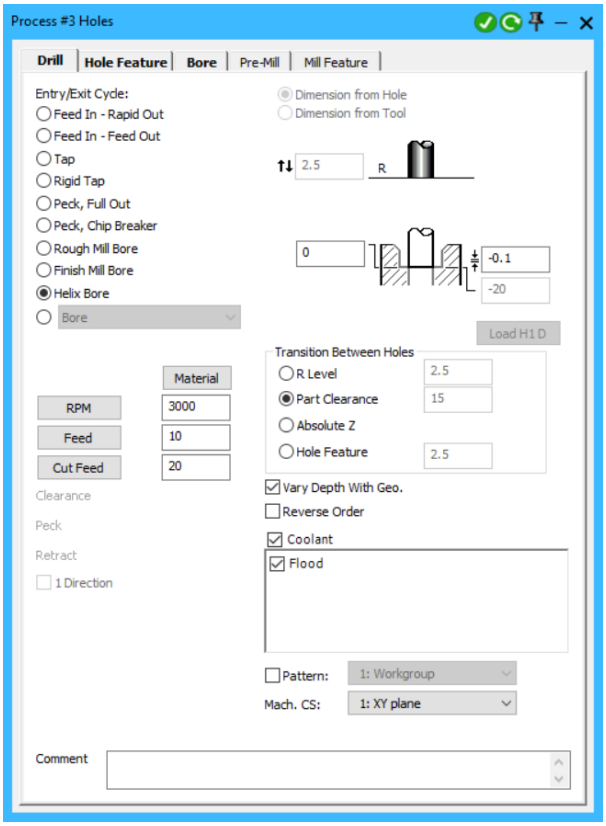


15. Render the Operations.



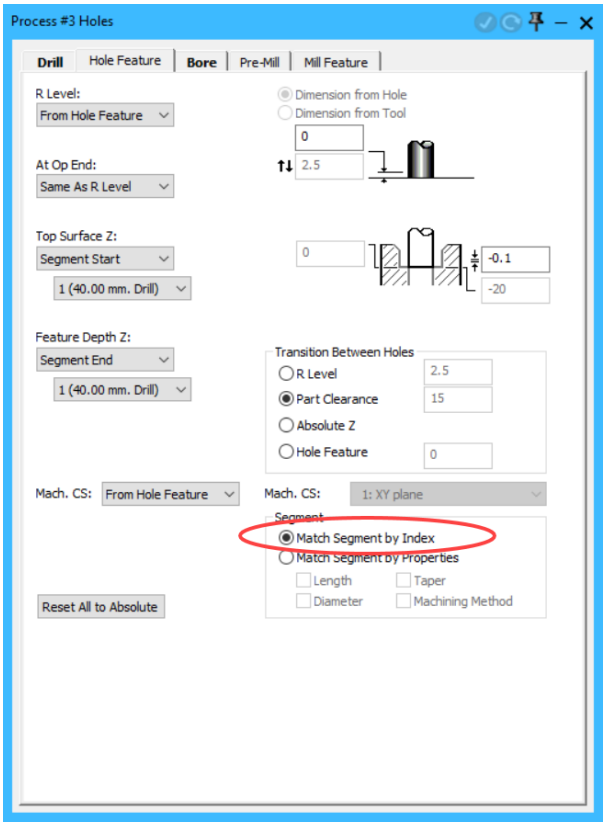
The top segment of the last hole has not been drilled - if you look more closely at the part you can see that it is wider than the others.

16. Double-click Operation #4 and drop tool #2 onto it. We will Helix bore the first segments instead.

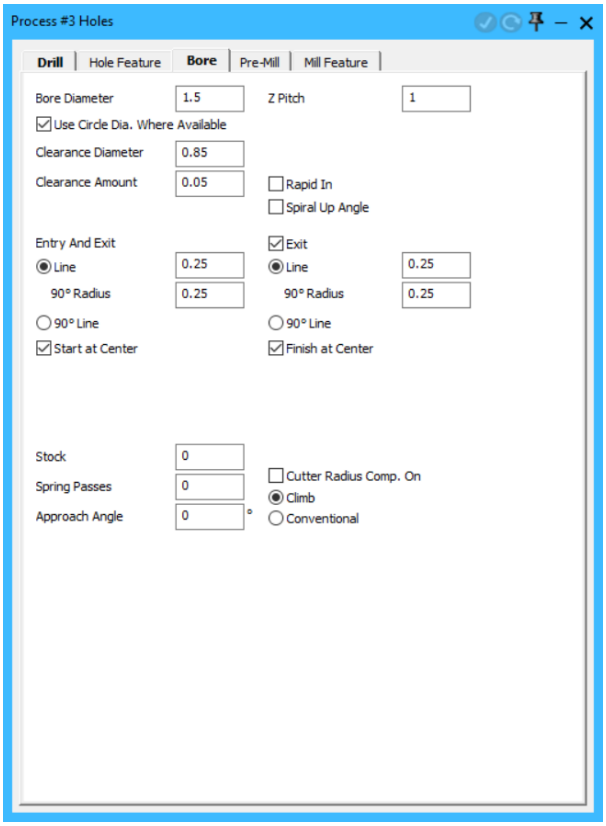


You will notice that the Bore tab has now also become active.

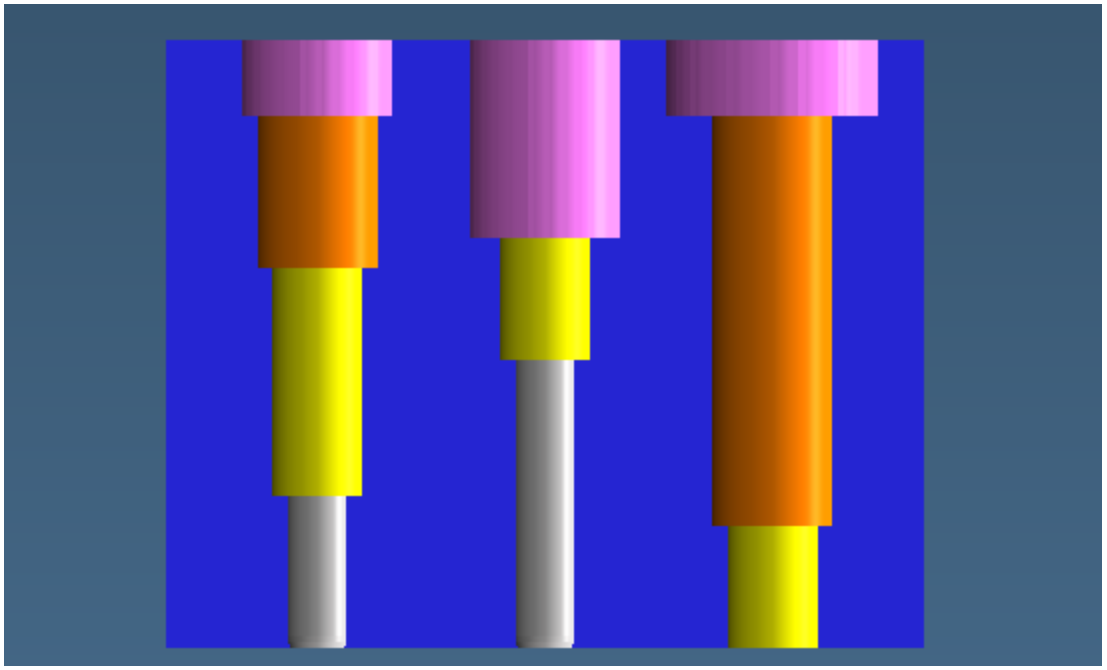
17. This time we will not match the hole by diameter, as we wish to helix into the top of all holes, regardless of diameter. Choose the Match Segment by Index Radio button.



18. In the Bore tab enter the following:



19. Render the operations.



20. Save the part.

Feature Manager

In this exercise we will add features to a part using the Feature Manager and then use the features to machine the part.

1. Open part file “Mill Feature.vnc” located in the Sample Parts > Production > Tutorial Parts - Required folder.

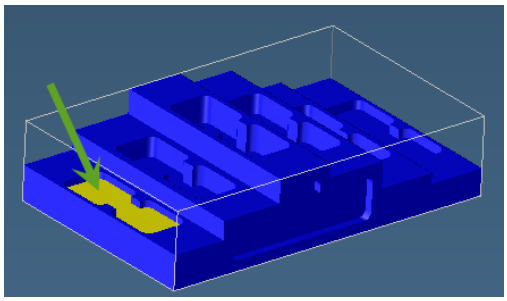
   If you look at this part you will see that there are pockets on three of the surfaces.

2. Click Features > Feature Manager on the Main Menu, to open the Feature Manager Dialog.

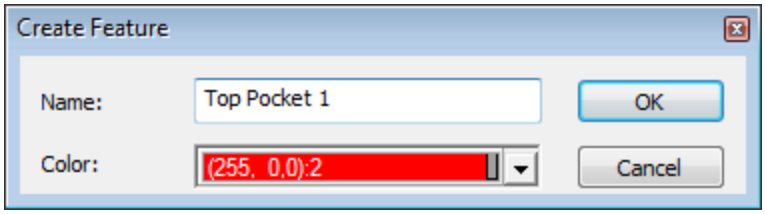
   Keep this dialog open throughout the tutorial.

3. Select the pocket shown below. (Click the bottom of the pocket, then right-click it and choose Select > Select Wall Faces.)

4. Ensure that Face Selection (on the floating Taskbar) is enabled .

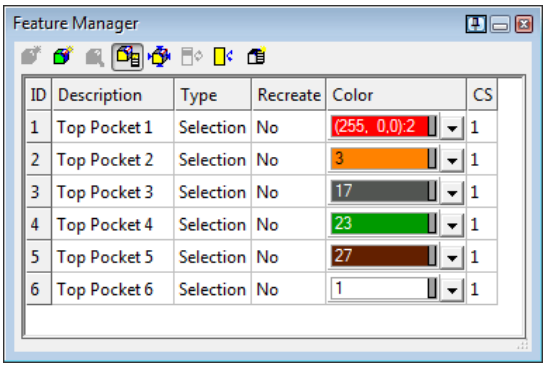


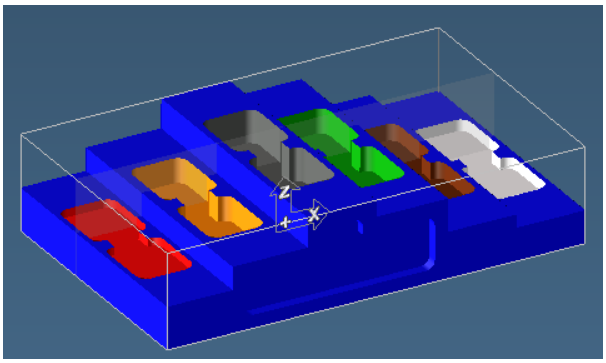
5. In the Feature Manager dialog, click the Create Feature from Selected button and enter details as follows.



If you cannot see the color displayed on your screen, ensure that the Feature Color display mode is on. This is located in the Floating taskbar.

6. Repeat the last two steps, until all pockets on the top surface have been added to the Feature Manager.
   Your Feature Manager dialog and screen will look as follows:



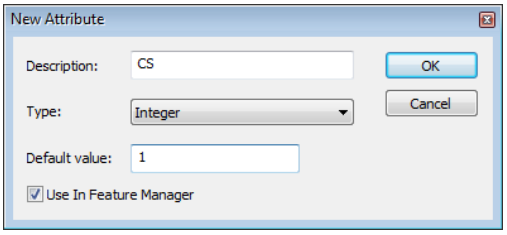


(The choice of color is not important.)

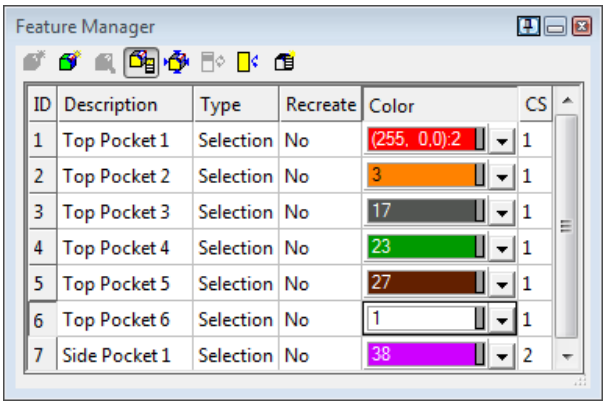
7. Now select the side pocket, making sure to look all around the part to ensure all surfaces are selected. Save the feature as Side Pocket 1.

We will now create an attribute for the CS.

8. In the Feature Manager click the Create new Attribute button  . Enter the following in the dialog.

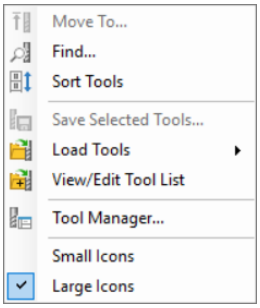


9. We need to edit the Features to add this new attribute. Click in the newly created column and add the following CS values:



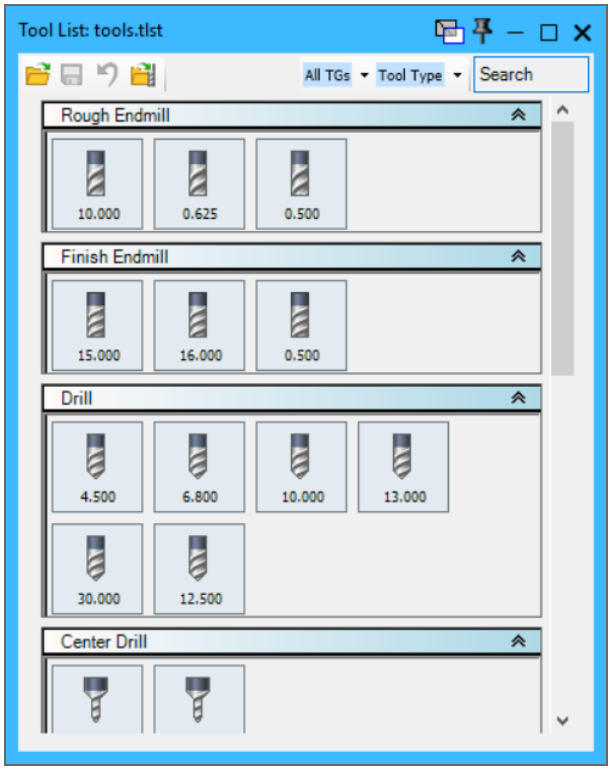
The part is now ready to be machined. First we will load some tools from the Tool list we set up in the Mill tutorial.

1. Right-click on any Tool Tile to display the Tool list menu as shown.

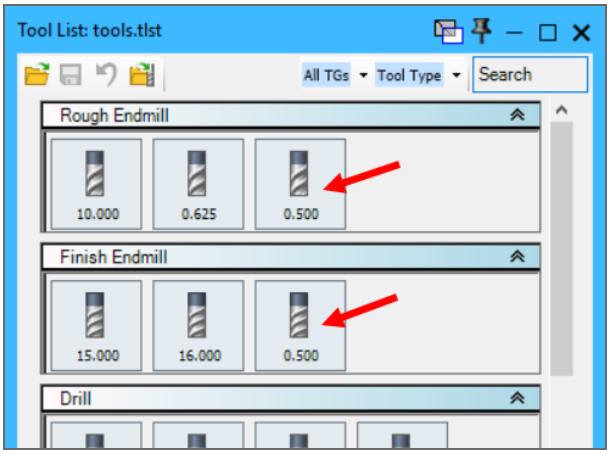


2. Select the option View/Edit Tool List and navigate to the sample file tools.tlst.

3. Click Open. The tool list opens.



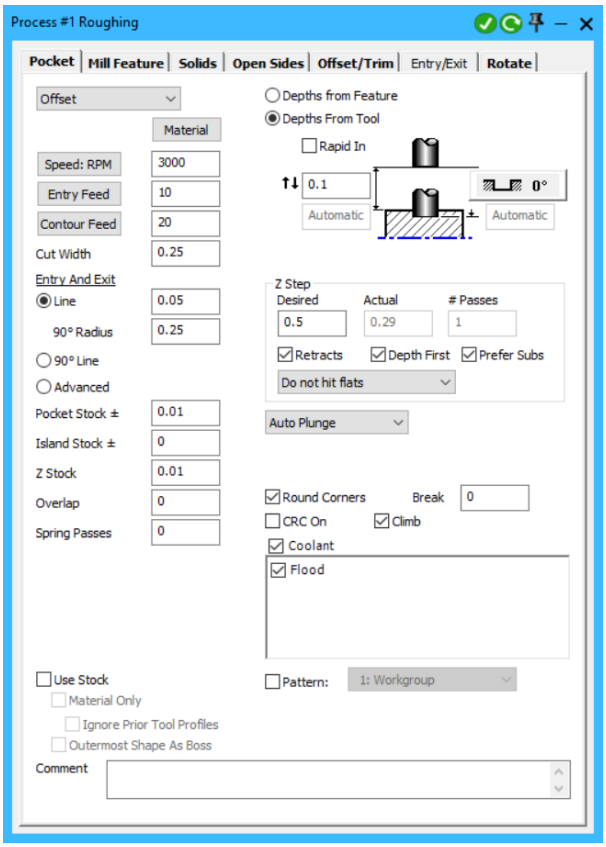
4. Ctrl-click the tools as shown to select them, then right-click and choose the option Import to part.



The tools will be inserted into the tool tiles. You can also drag the tools individually onto the tool tiles - Rough Endmill to Tile #1, Finish Endmill to Tile #2.

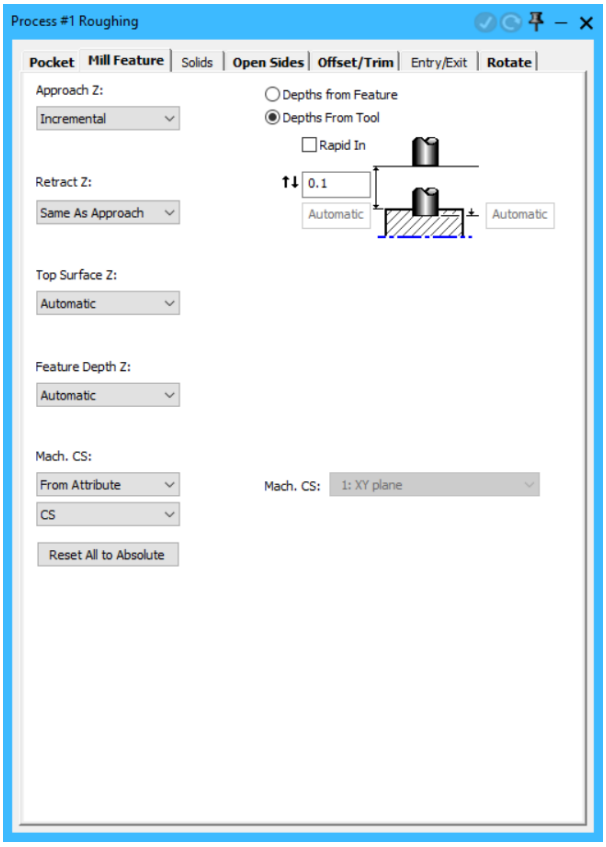
5. Close the Tool List dialog.

6. With Tool #1 create the following Pocketing Process.



You will notice the Mill Feature Tab is bold, meaning that settings will be applied.

7. Enter the settings shown below into the tab.

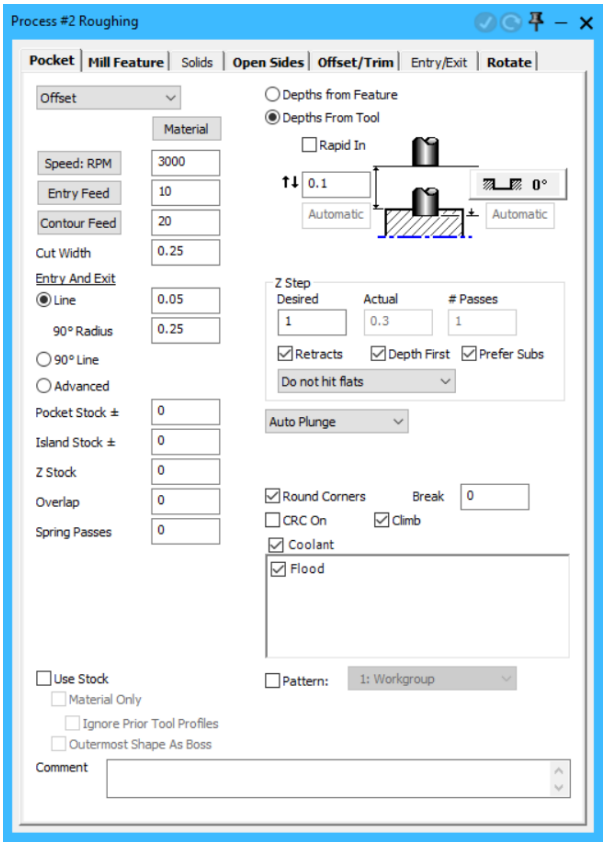


Approach and Retract Z are set to Incremental. The distance between the Clearance Plane and the Top Surface will be updated for each hole.

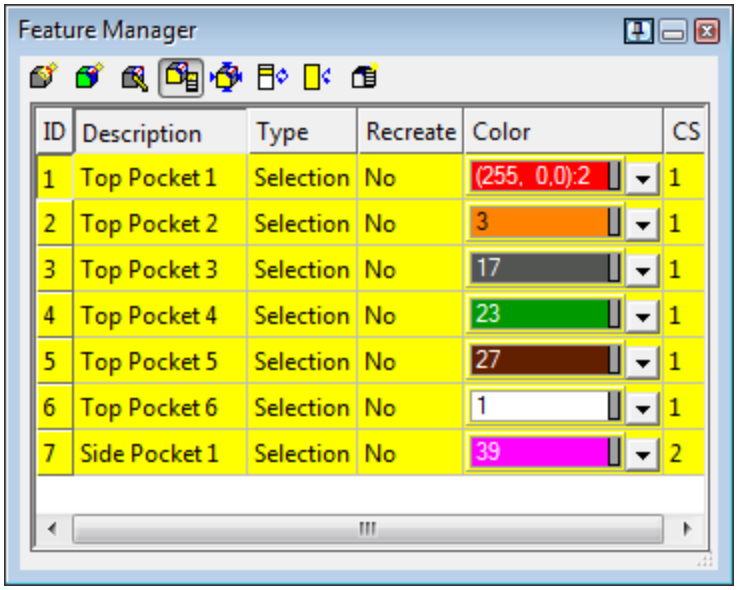
Top Surface and Depth Z are set to Automatic. The value will come directly from the geometry of the user feature.

Notice the Mach CS Dropdown is no longer available. This is because the Machining CS details are taken from the attribute set in the Feature Manager.

8. With Tool 2, create the following Finish Pocketing Process. The Mill Feature Tab will already be populated with the values set in the previous process. Leave these as set.



9. In the Feature Manager dialog, select all Features (Cursor on first Feature, then Shift-click bottom Feature, or Ctrl-click each of the rows).



10. Click Do it on the Machining dialog.

    Fourteen operations are created., machining each of the pockets selected in the Feature Manager.

11. Render the Operations.

Try adding the surfaces located on the bottom of the part to the Feature Manager and then machine them also.

Drilling Multiple Orientations using Hole Feature

This is a short exercise to demonstrate how the Hole feature tab, used in combination with Hole Manager can create operations for holes in multiple Orientations.

Please note that Mach. CS "From Hole Feature" in the Hole Feature Tab means that holes will be grouped together according to their orientation. One operation will be created for each orientation. For each of these operations, if there is already a CS that is aligned with hole depth axis, that then will be the CS assigned to that operation. If there is no CS that matches the hole depth axis, then a new CS will be created and assigned to the operation.

1. Open part "Hole Feature CS.vnc". This already has two tools set up.

   First we need to select the holes to machine.

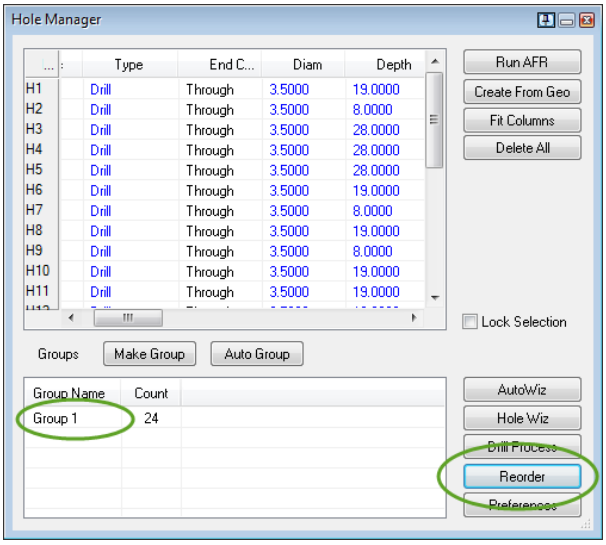
2. Open Hole Manager, select the part and Run AFR.

   24 Holes are found.

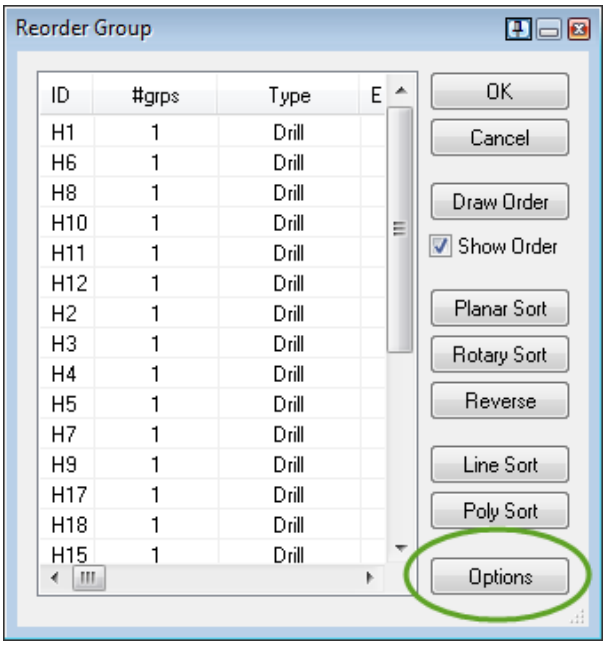
3. Select all the holes (Click into the Hole Manager and (Ctrl-A)). Then click the Make Group button.

   A single Group is created with 24 holes.

4. Click on Group 1 to select it, then click Reorder.

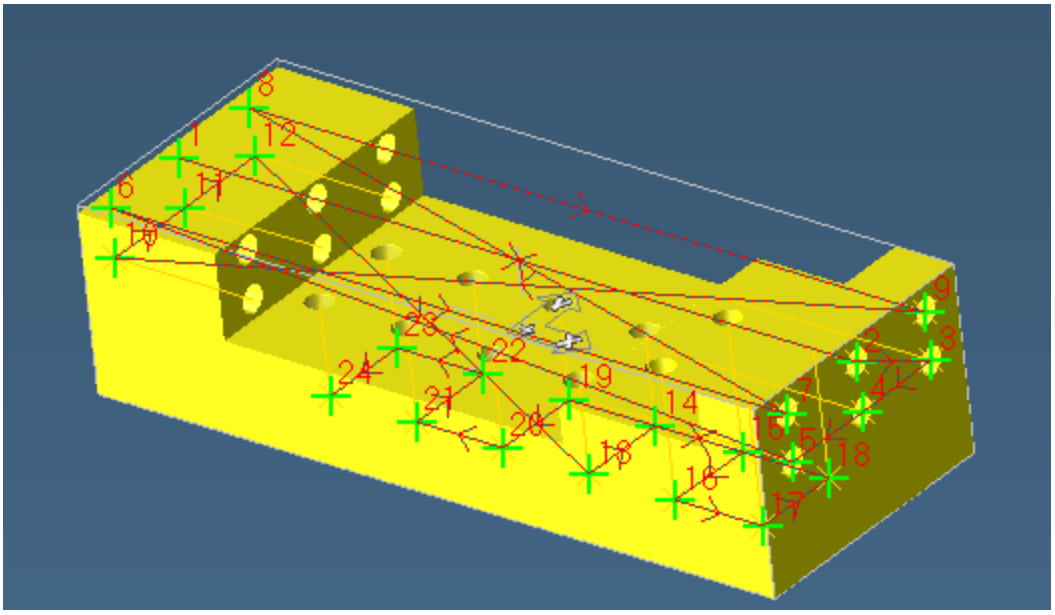


5. In order to see the toolpath correctly we will change the Display Options. Click the Options button at the bottom right of the Reorder Group dialog. Enter the following:

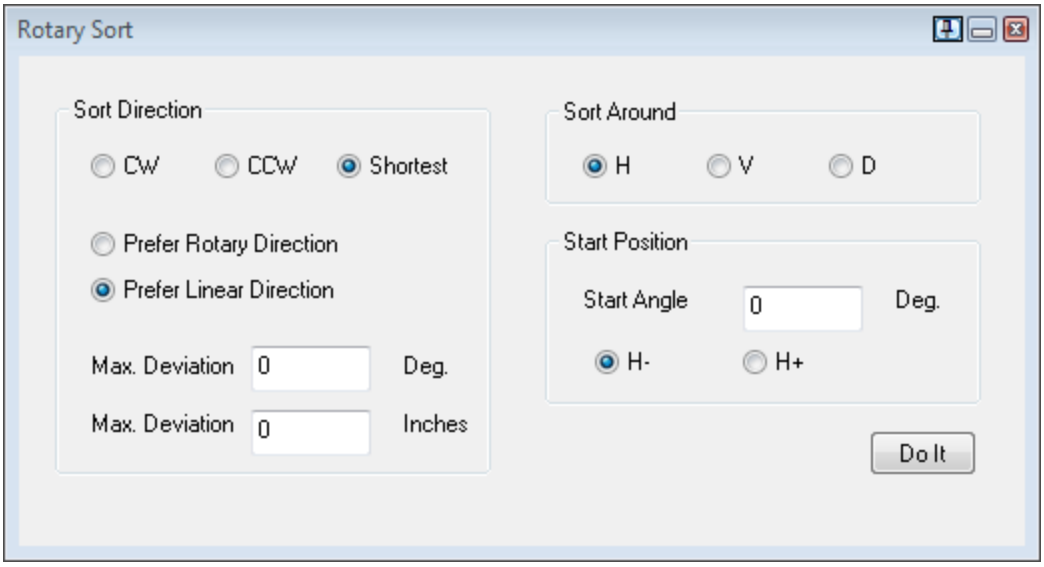


6. Click on Group 1 and click Reorder. This opens the Reorder Group dialog.

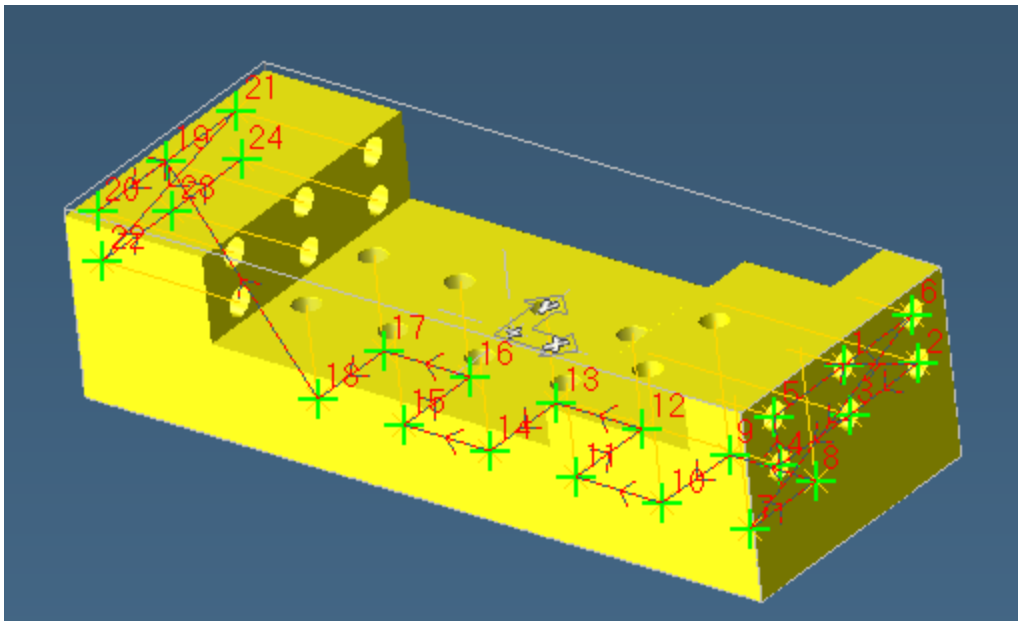
7. Now click the Draw Order button. This will highlight and number the machining order. You will see that the toolpath is all over the place. If this is not working, ensure the Show order checkbox is checked.



8. Now Click Rotary Sort, enter details as shown below then click Do it .



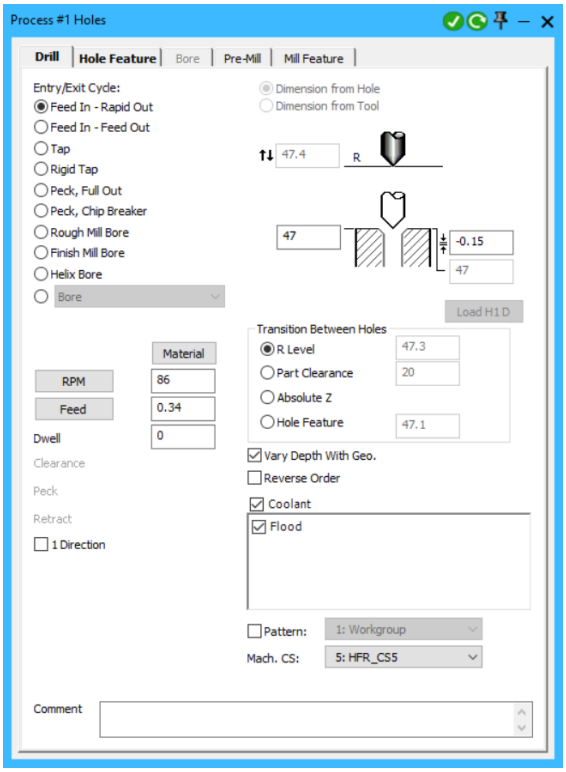
The toolpath is sorted.

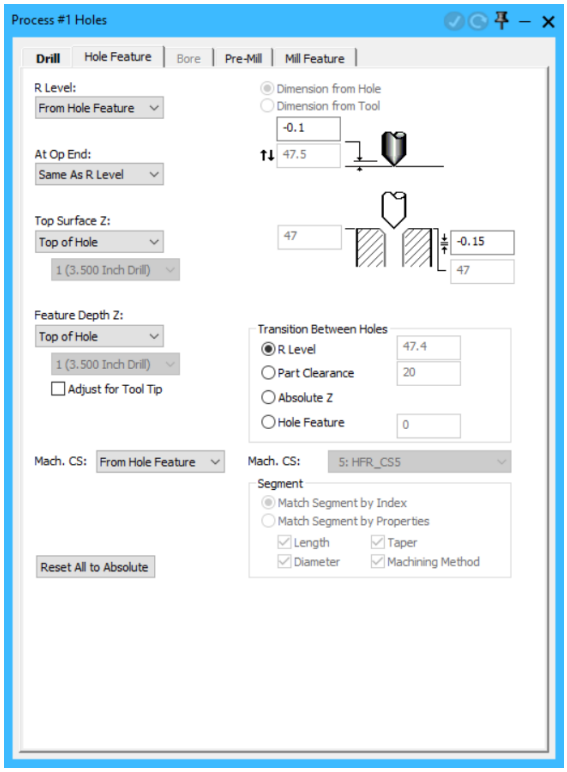


9. Click OK and return to Hole Manager. Group 1 should remain selected. Any holes selected within Hole Manager will be the ones used in the operations. Make sure the Hole Manager dialog is left open, this ensures that the Hole Feature Tab can use the values contained within Hole Manager.

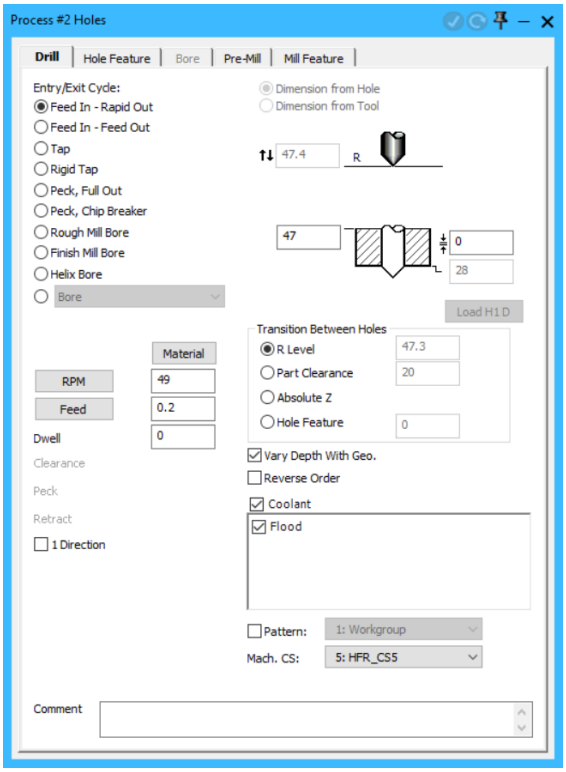
Now we will create two processes, one to Spot drill each hole and the other to Drill them.

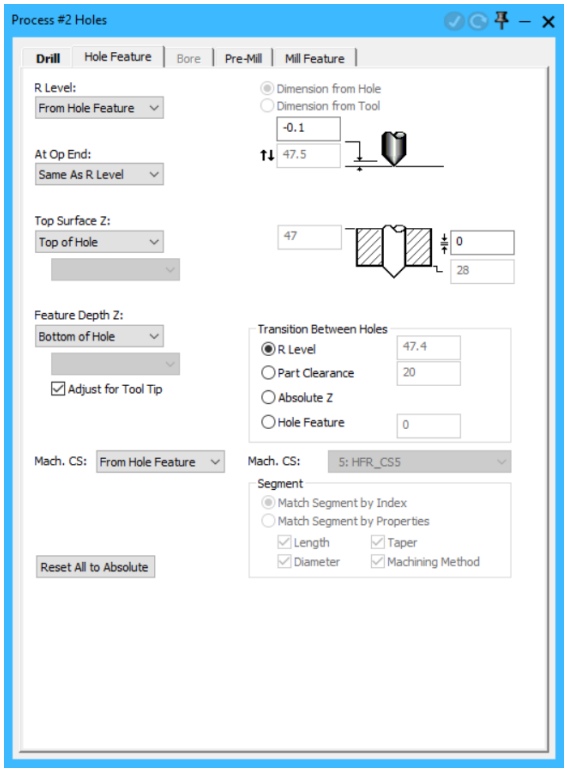
10. Create a Holes Process using the 2.00 Spot tool (Tool #1).





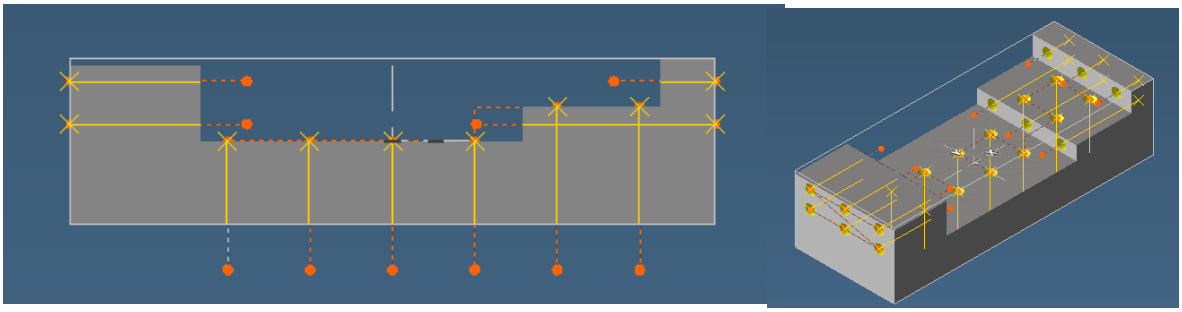
11. Create another Holes Process using the 3.5 Drill (Tool #2).





12. Click Do it on the Machining dialog.

A total of six operations are created, two for each hole orientation. You will note that hole depth, orientation and top clearance values are provided by the Hole Manager and require no user input.



13. Save the part.

Part Duplication

We will use the completed Mill tutorial part for this exercise.

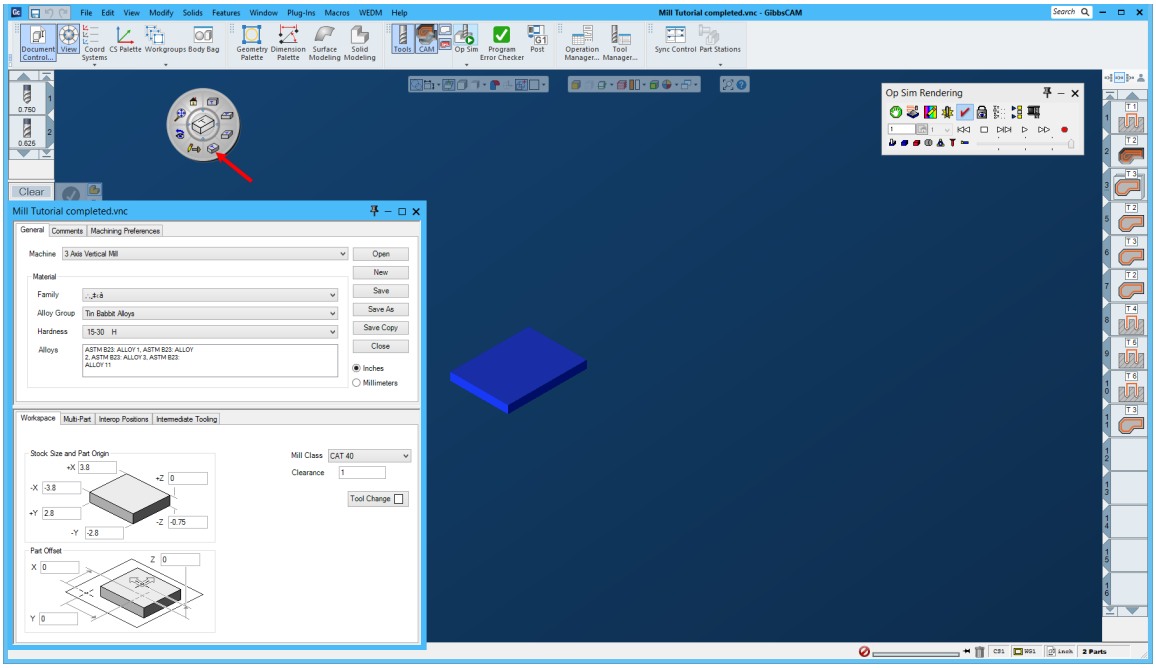
1. Open your saved Mill Tutorial.vnc file.

2. In order to view part duplication, you must to open the Op Sim Rendering palette.

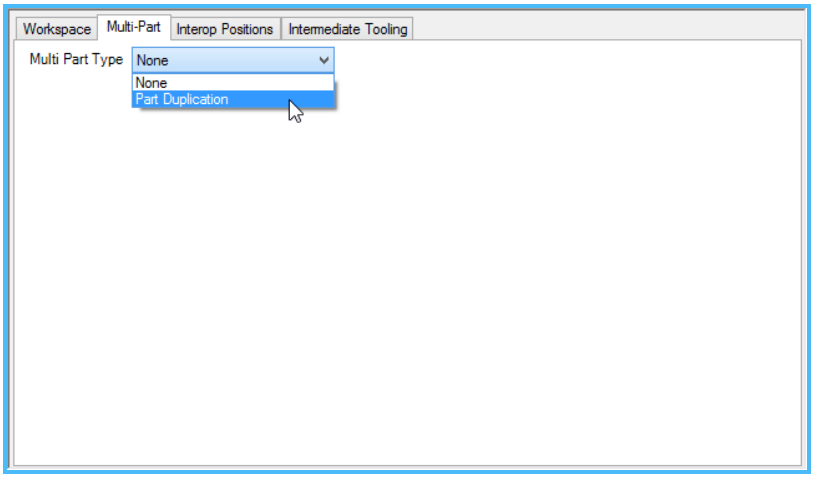
   Leave the Rendering palette on the screen during the entire tutorial. This will enable you to immediately see the duplication pattern as you make changes and then play/rewind the operations. In the mean time, just move the palette to the side of the screen.

3. Open the Document Control Dialog.

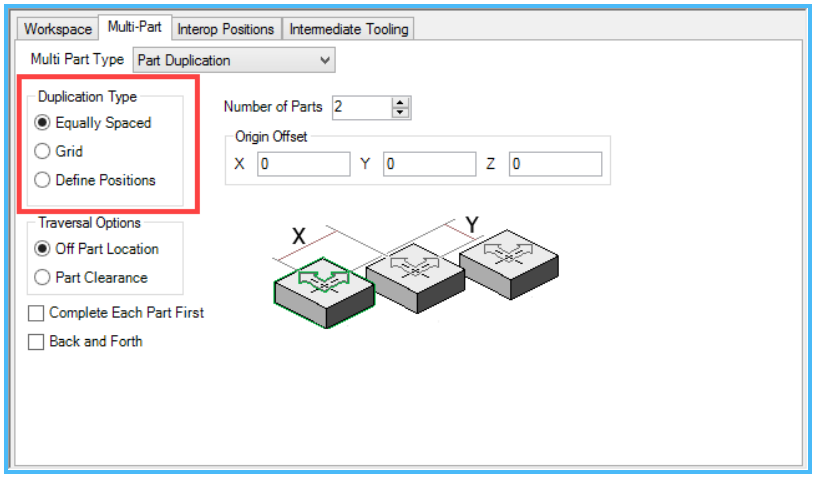
4. Change your workspace view to Isometric and adjust your screen so that the DCD, the part and the Op Sim palette are all visible.



5. From the bottom section of the Document Control Dialog, click to open the Multi-Part tab. Now choose Part Duplication from the dropdown menu as shown below.



The Multi-part dialog now opens. First we will turn our attention to the Duplication Type. (Area highlighted below). This is where we specify the grid pattern.

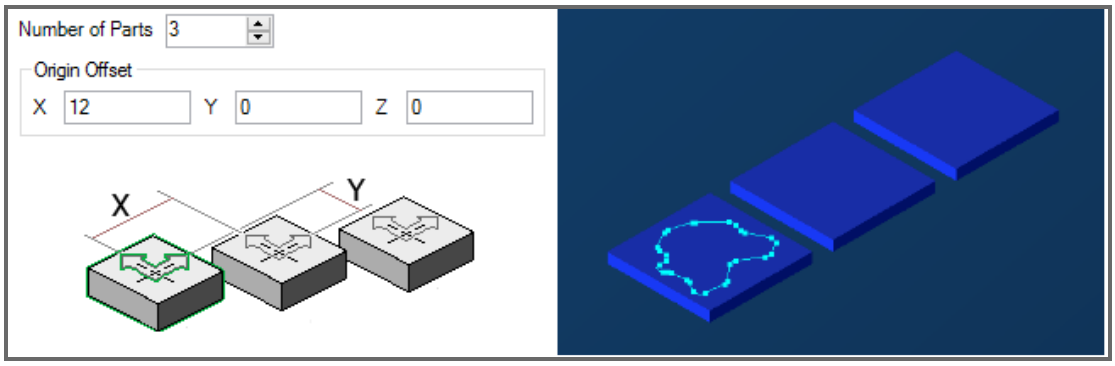


Equally Spaced

1. We will start by setting up an Equally spaced pattern.

   Equally spaced will duplicate in one direction specified by the Origin Offset.

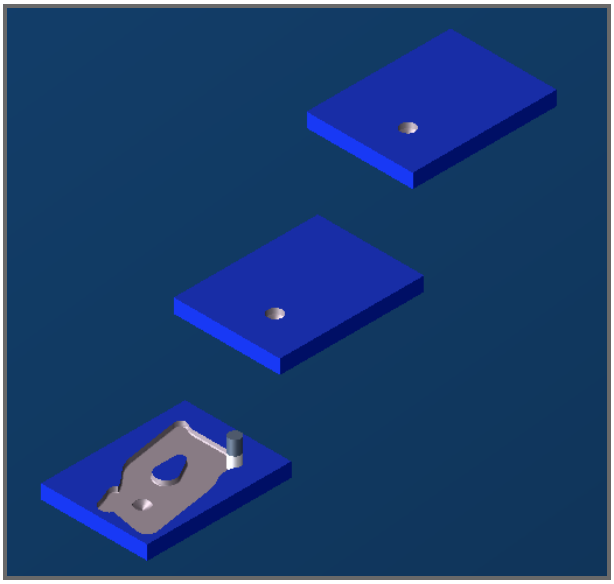
2. Change the number of parts to 3 and the origin offset to 12 as shown below.



Immediately you will notice that 3 parts are displayed on the screen, extending in one line.

3. Render the operations.

The first operation is performed on each part in turn, returning to the first part to start the next operation.



4. Change the Y offset to 5. Immediately the parts are also moved in the Y direction.

There are three other useful options on the Multi-Part tab which are common to all duplication methods.

- The Transversal Options buttons allow you to choose the tool retract between moves. It can either be to the Off Part Location specified in your MDD, or to the Tool Clearance amount you specified in the Workspace tab of the DCD.
- The checkbox Complete Each Part Firstmeans that all operations will be completed on a part before moving onto the next.
- If Back and Forth is checked, the next operation will not return to the first part, it will instead perform the next operation on the last part and work back to the beginning.

Feel free to experiment with different Number of Parts, Origin Offset amounts and other options before moving onto the next duplication type.

Grid Duplication

We will now try Grid duplication.

Grid will duplicate in a grid pattern. You need to specify the number of parts in each direction and the fixed grid offset.

1. Change the Duplication Type to Grid.
2. Change the number of parts in X to 4 and number of parts in Y to 4. This will create a parts grid of 4x4, or 16 parts.

You will notice that the display does not seem to change. This is because all the parts are on top of each other. You need to set the offset distance to separate them.

3. Enter 12 in the X and Y direction fields as shown below. You will now see all 16 parts in an even grid pattern.

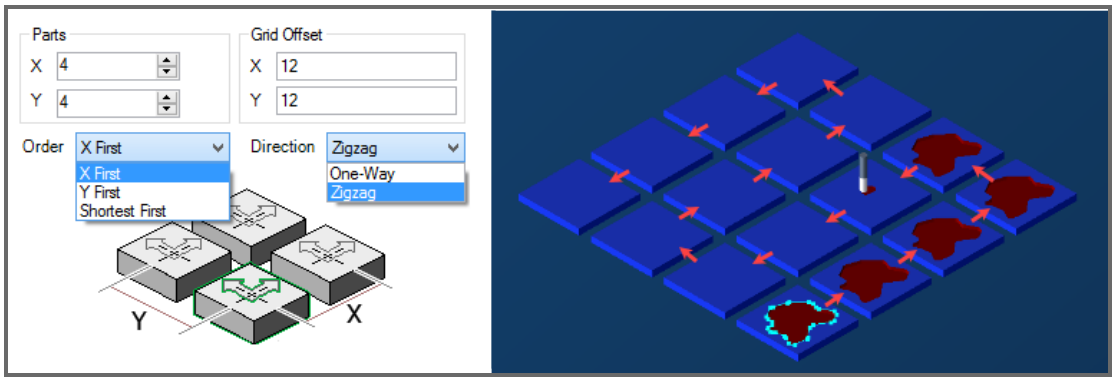
You now have two more dropdown options, Order and Direction.

The Order determines in which axis the tools move. You can choose to move in X or Y or whichever is shortest. The “shortest” means that if the grid is X=4, Y=3, the direction of the cut will be in Y. The first cut is always at 0,0,

4. For this tutorial we will choose X First.

Next we must choose Direction. If we choose One-Way the tool will start at 0,0, move along the X axis to the end of the row, then return to the beginning of the next row. Once the first operation is completed on all parts, the next operation then continues in reverse order from the last position.

5. Choose Zigzag pattern as shown below, this will serve to minimize tool travel.



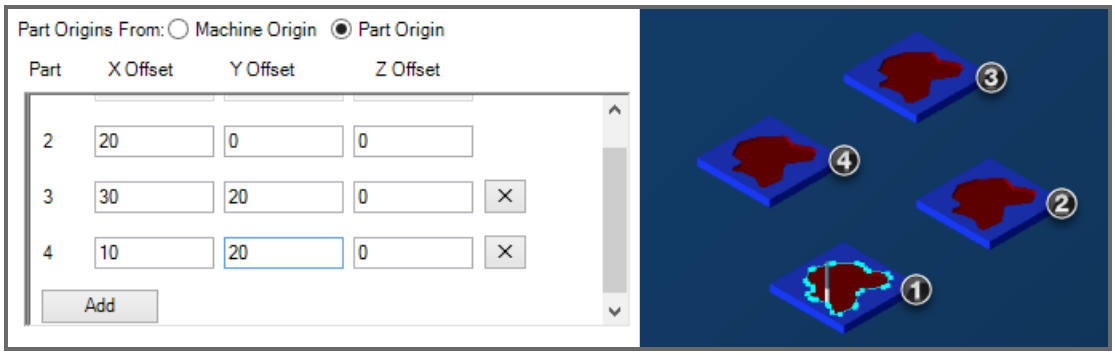
6. Render the operations.

Again feel free to experiment with different numbers of Parts, Origin Offset amounts and other options before moving onto the next duplication type.

Define Positions

Define Positions allows you to specify your own specific grid positions for each part and enter them in machining order.

Once again the first part is automatically set at 0,0, but you can specify this to be either from the Machine or Part Origin. Each line in the dialog is another part location, with the line number defining the machining order. As you type the location into the fields, the part is displayed on the workspace. Click Add to start the next entry. X will remove an entry and the remaining parts will be renumbered. Continue adding grid locations as shown below.



Render the operations and save the part.

PART PRINT

Mill