Version 14 : September 2020

Features Tutorials

Proprietary Notice

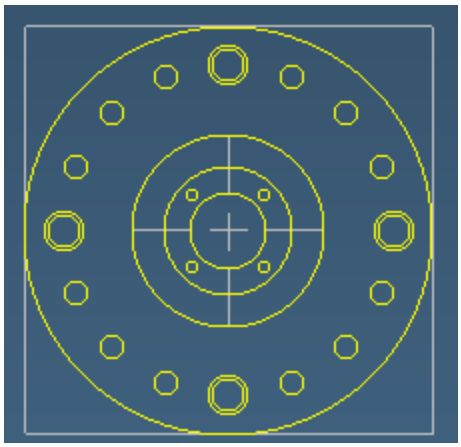
HOLE WIZARD

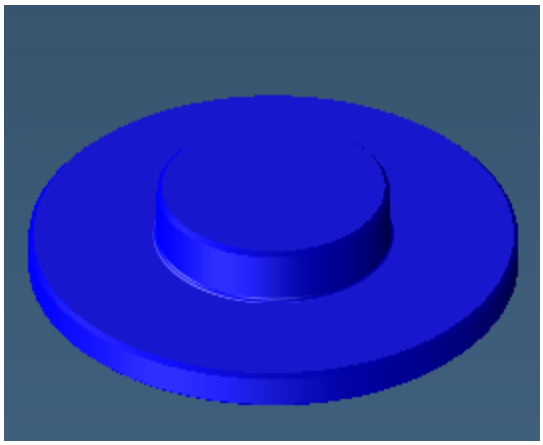
This tutorial provides step by step instructions on how to use the Hole Wizard. We will start this part with an existing stock condition and tools that are already in the crib. In this exercise we will create five hole types using the Hole Wizard: bolt holes, a bore hole, drill holes, reamed holes and tap holes. Additionally, we will need to create a tool to satisfy the Hole Wizard Checker and manually change a process from the Hole Wizard's recommendations.

Note that this tutorial is based on the Hole Wizard's default preferences. If you have modified the defaults you may have different results than shown in the tutorial.

Set Up

1. Open the part, "Hole Wizard tutorial.vnc".





The elements to be machined include twelve 10mm holes, four 16mm holes, one 50mm hole, four 5mm holes and eight points that need to be created. The pre-defined stock is a short 160mm cylinder with a 76mm boss in the center. The total depth of the stock is 32mm.

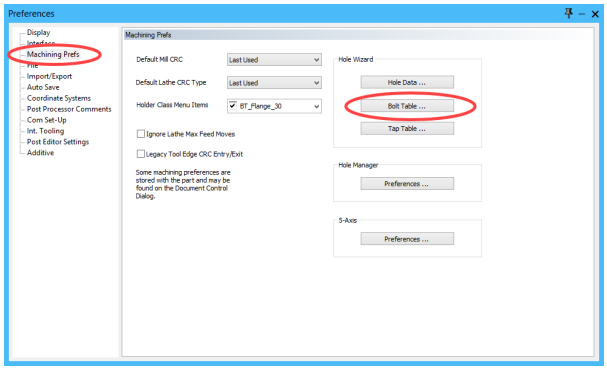
In this part we have 14 pre-loaded tools including end mills, drills, spot drills, center drills, counter sinks, a tap, and a reamer. There is also an empty space at tile #8 for a tool we will need to add later in this exercise.

New Bolt Table Entry

1. Select File >  Preferences.

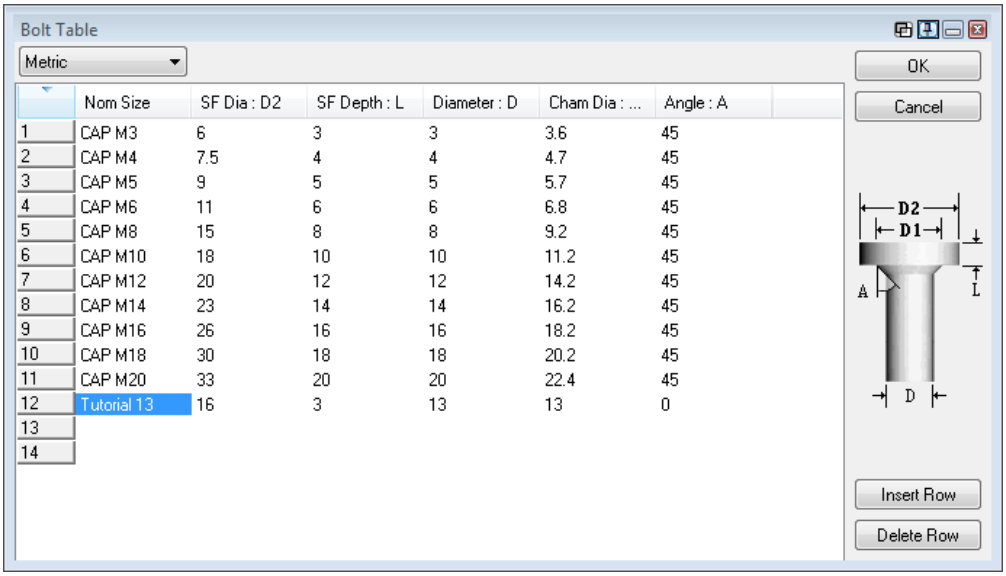
   The Preferences dialog box opens.

2. Click the Machining Prefs tab and then, under Hole Wizard, click Bolt Table.



Double-click in the next empty positions' Nom Size cell. Type "Tutorial 13". Add the bolt hole definition shown: D2 = 16, L = 3, D = 13, D1 = 13, A = 0.

We will be using this later in the tutorial when we create Bolt holes.



	Nom Size	SF Dia : D2	SF Depth : L	Diameter : D	Cham Dia : ...	Angle : A
1	CAP M3	6	3	3	3.6	45
2	CAP M4	7.5	4	4	4.7	45
3	CAP M5	9	5	5	5.7	45
4	CAP M6	11	6	6	6.8	45
5	CAP M8	15	8	8	9.2	45
6	CAP M10	18	10	10	11.2	45
7	CAP M12	20	12	12	14.2	45
8	CAP M14	23	14	14	16.2	45
9	CAP M16	26	16	16	18.2	45
10	CAP M18	30	18	18	20.2	45
11	CAP M20	33	20	20	22.4	45
12	Tutorial 13	16	3	13	13	0
13						
14						

Clicking the Tab key to move between fields is an easy way to navigate the Hole Wizard's tables.

3. Click the OK button to close the Preferences dialog.

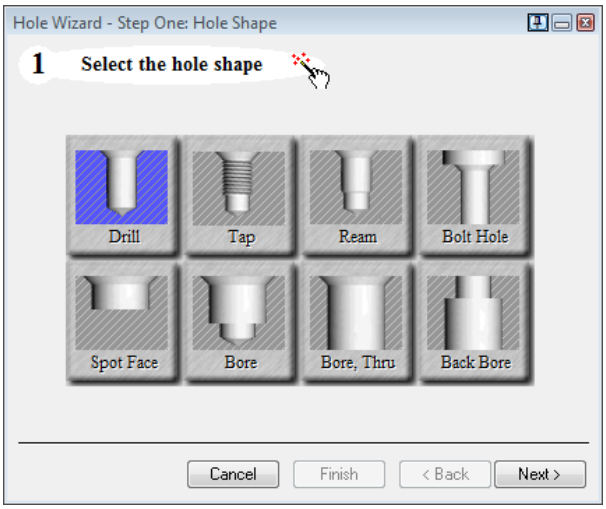
The new entry is saved.

Creating 10mm Holes

There are four steps to creating these holes: select the hole shape, define the hole size, select the hole position, and build the processes and operations.

Step 1. Select the Hole Shape

We will now make the first set of holes. These are basic drill holes that are 10mm wide with a 1mm chamfer. The Full Diameter Depth extends to the bottom of the existing stock condition, which is 32mm.



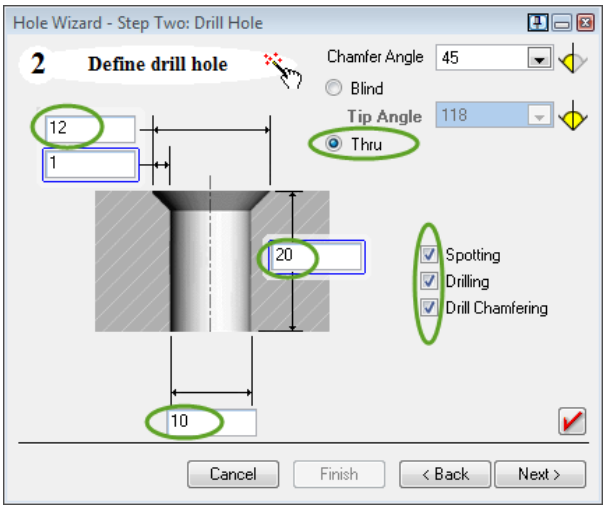
1. Open the Hole Wizard. To access the Hole Wizard from the Main menu bar,

   click Features >  Hole Wizard. You can leave the Hole wizard dialog open throughout the exercises. Click and hold the Title bar to drag the dialog to a convenient corner of the workspace.

2. Click the Drill button.

   The Hole Wizard - Step Two: Drill Hole dialog displays.

Step 2. Define the Hole Size



1. Enter the values in Step Two as shown.

   A stock depth of 20mm will take the drill through the part.

   Note that the Chamfer Angle should be set to 45. As this is a Thru hole, we do not need a Tip Angle. The available angles are derived from the tools in the Tool List.

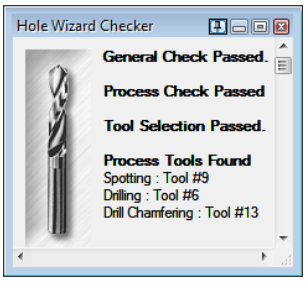
The Spotting, Drilling and Drill Chamfering processes have all been selected by the Hole Wizard. You may have noticed that as you input or changed numbers that these options were being selected or unselected. The Hole Wizard is comparing the size of the hole against the Knowledge Base to determine what is needed to make the hole. You may deselect any process you do not want to create.

2. Click the Next button.

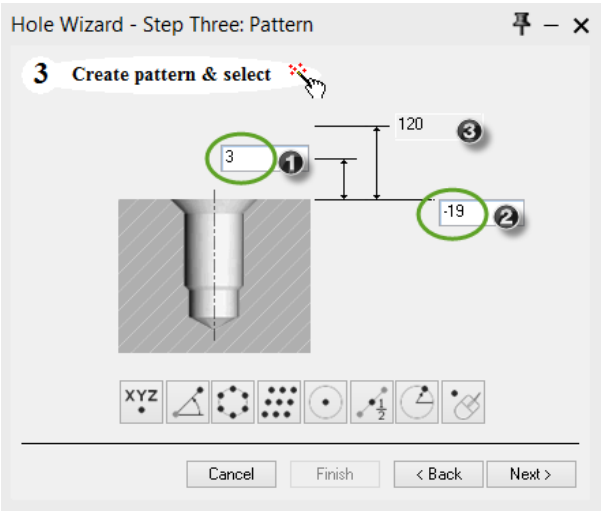
Step 3. Select the Hole Position

Notice that several tools in the tool list are highlighted. These are the tools the Hole Wizard has selected for the drill holes.

If the required tools were not found or if there was something wrong with the settings for this drilling process, the Hole Wizard Checker would alert you. If you were to go back to Step Two and open the Hole Wizard Checker by clicking the red check sign, it would look like the following image.



3. Enter the clearance values as shown.

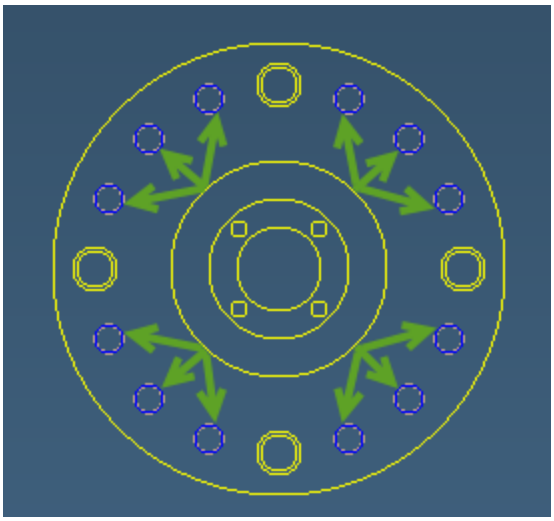


This will set the clearance to be 3mm above the defined surface (1) and the Surface Z to be at -19mm (2). On the previous dialog, we specified this operation's Full diameter Z to be "20", meaning 20mm from the Surface Z. The Surface Z value of -19mm minus the 20mm depth of the cut is -39mm. This ensures the full diameter of the drill is below the bottom of the existing stock. The value

at (3) is calculated incrementally from the top drill surface and the master clearance plane set up in the Document Control Dialog.

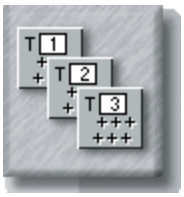
We need to designate which geometry to machine.

4. `Ctrl-click` the circles shown below in a clockwise order. This is advisable as the clearance is not set high enough to clear the central boss and toolpath travelling over it would cause gouging.



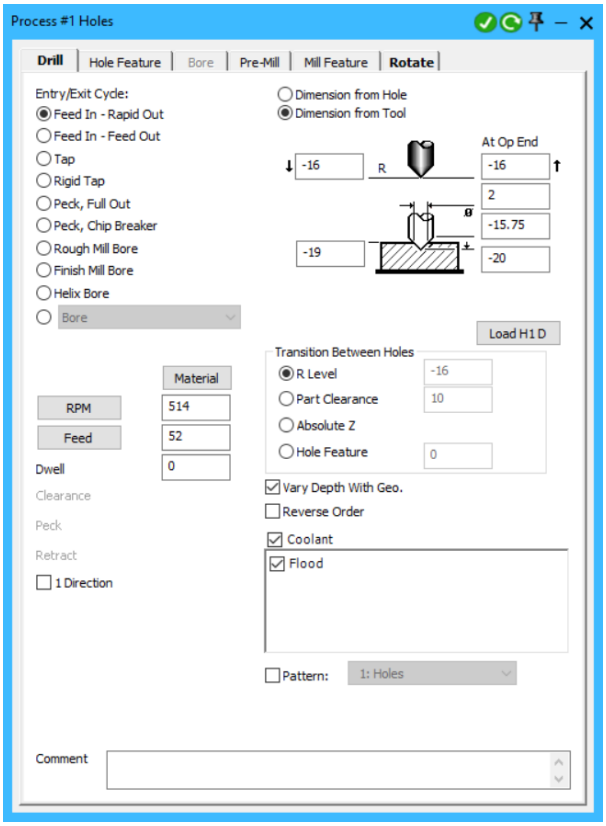
Step 4. Build the Processes and Operations

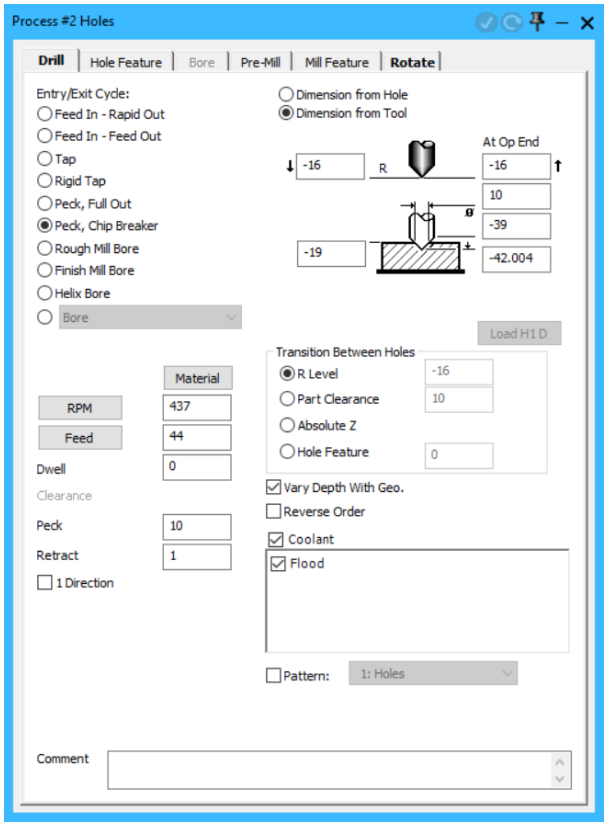
1. `Click` the Build Processes button.



You can verify the processes created by opening each one or skip to the next step in the tutorial.

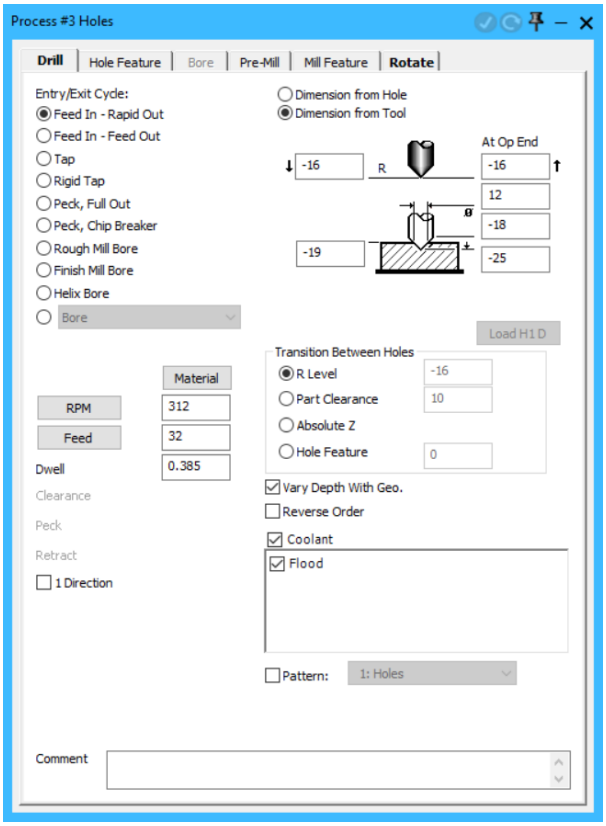
Clicking Build Processes sets up processes based strictly on the Hole Wizard's default preferences. Looking at the three processes created, we see the first drilling process is using the 8.5mm spot drill, cutting 1mm deep. On moves between holes the tool will retract to -16mm, or 3mm above the existing stock.





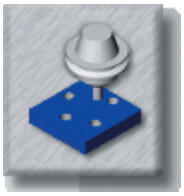
The second drilling process uses the 10mm drill. The drill will peck to -39mm to clear out the hole. It too will retract to -16mm to make moves between holes.

The third drilling process uses the 14mm counter sink to create a chamfer. The tip of counter sink will machine to -25mm to create the chamfers. It too will retract to -16mm to make moves between holes.



If you accidentally changed any of these settings while looking at them `click` the Rebuild Processes button.

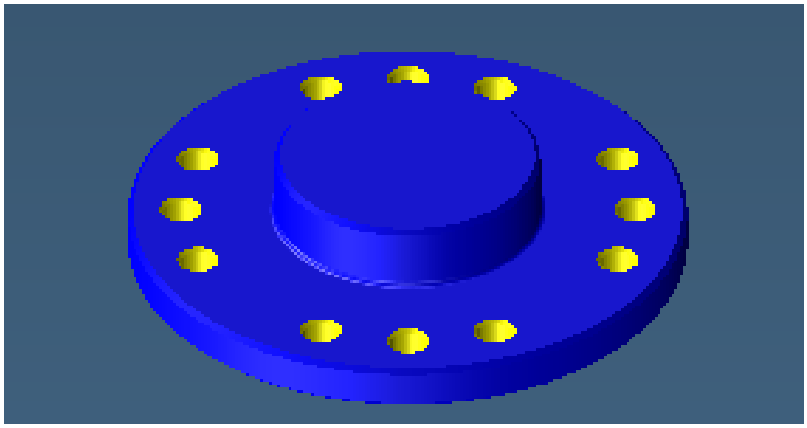
2. `Click` the Build Operations button.



Since the Machining palette is open we could have used the Do it button to create the drilling operations, however, we would have lost our associativity with the Hole Wizard.

3. `Click` the Finish button to close the Hole Wizard.

4. Render the Drilling operations.



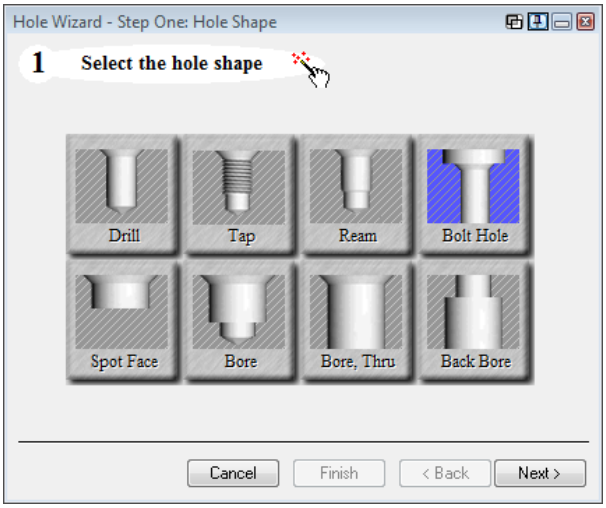
13mm Bolt Holes

There are three steps to creating these holes.

Step 1 Select Hole Shape

We will now make the bolt holes. Be sure to deselect the operations you just made and delete the processes from the process list. This way you will not overwrite any operations.

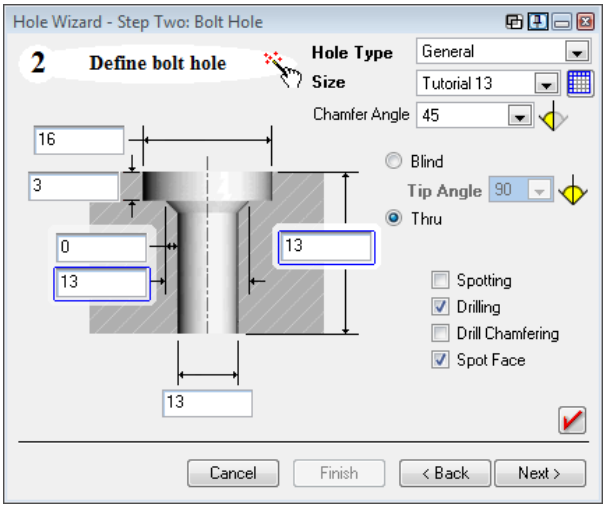
1. Click Features >  Hole Wizard to return to the Hole wizard.



2. Click Bolt Hole.

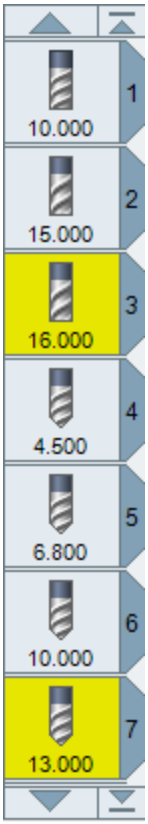
The Hole Wizard - Step Two: Bolt Hole dialog displays.

Step 2 Define Bolt Hole



1. Click the pull-down arrow on the Size dialog and select the entry we created at the beginning of the tutorial. Enter a 13mm stock depth.

2. Click the Next button. 

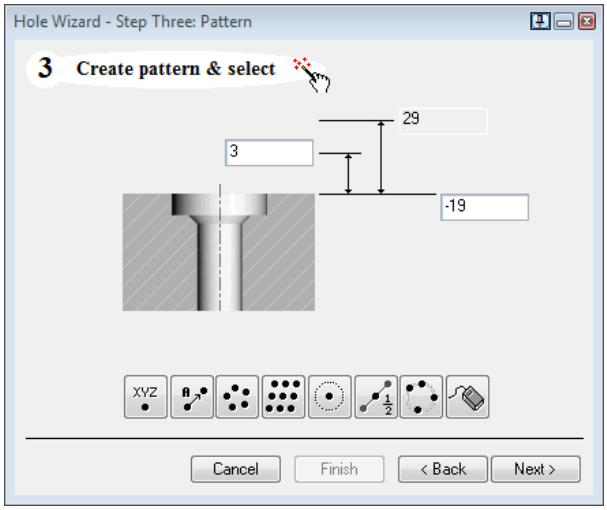


Two tools in the tool list are highlighted. These are the tools the Hole Wizard has selected for the drill holes.

If the required tools were not found or if there was something wrong with the settings for this drilling process the Hole Wizard Checker would alert you.

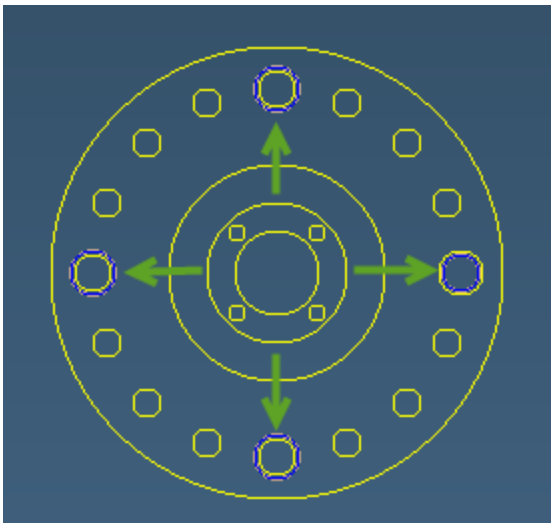
Step 3 Select the Hole Position

The Hole Wizard - Step Three: Pattern dialog appears. We will use the same clearance values as those used for the drill holes. The previously entered values are already the defaults.



We need to select which geometry to machine. There are two sets of circles for each hole. It doesn't matter which circle you choose because only the center point will be used.

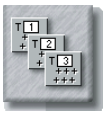
1. Select the circles shown.

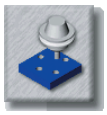


2. Click the Next button.

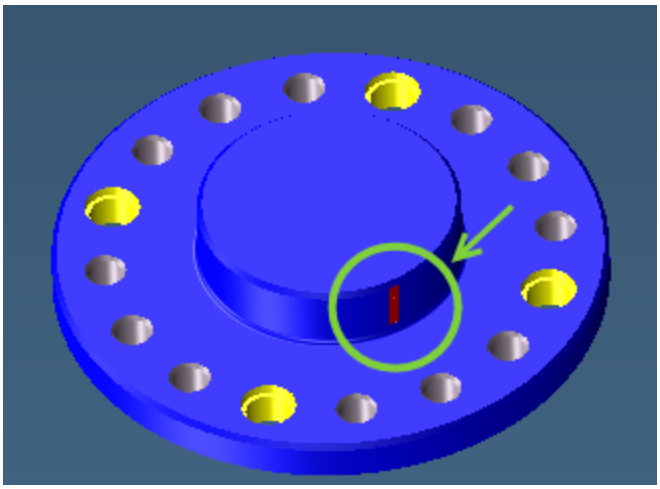
Step 4 Build the Processes and Operations

1. Click the Rebuild Processes and Build Operations buttons. If processes were in the list they have been replaced by the current Hole Wizard Process.





2. Render the Bolt holes. Notice there is a problem with the part.

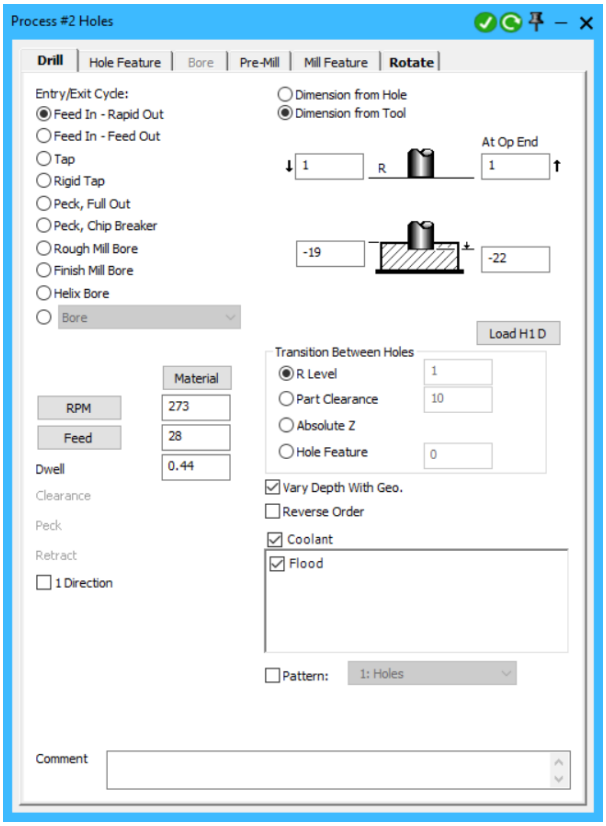


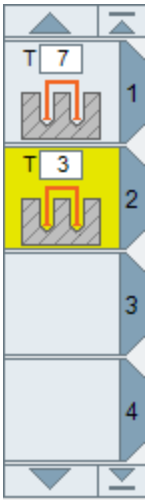
The 16mm end mill is scrapping the side of the boss during the move to each hole. We could modify the Entry/Exit Clearance Z specified in Step Three of the Hole Wizard. The second option is to modify the clearance value in the process itself. We will use the second option so the drilling retracts remain minimal.

Redoing the Operations

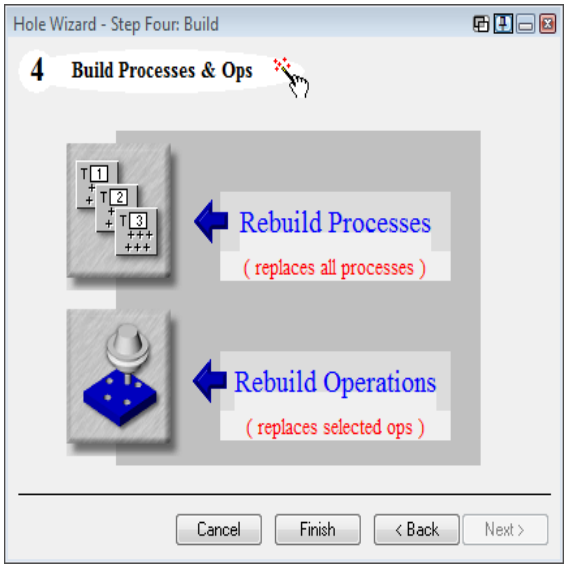
1. Reload the last operation or open Process #2.

2. Change the clearance planes to be 1mm, as shown.

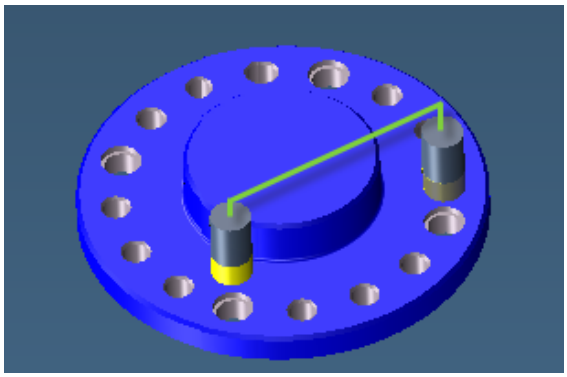




3. Click only the Rebuild Operations button.



If you click the Rebuild Processes, the data you just changed will be replaced with the Hole Wizard's recommendation. By clicking Rebuild Operations directly we keep the association between these processes and the Hole Wizard. This will make things easier if you need to modify this hole shape later.



Now, when the part is rendered, the end mill does not interfere with the part.

M8 Tap Holes

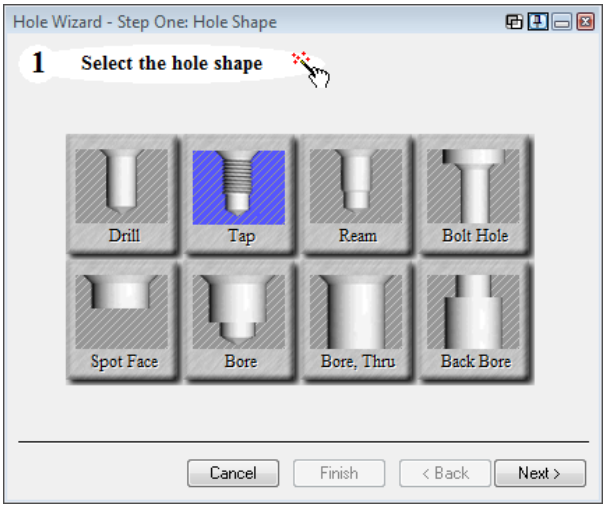
As before, we will complete the four steps of the Hole Wizard to create a circle of bolt tap holes.

Steps 1-4

We will now make tap holes on the top of the boss. Be sure to deselect the operations you just made and delete the processes from the process list. This way you will not overwrite any operations.

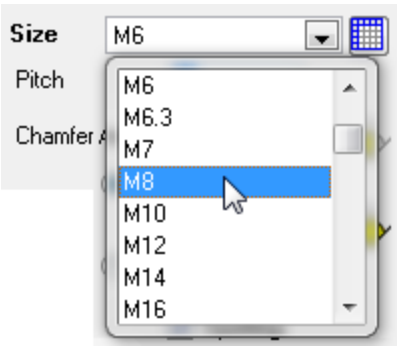
1. Deselect any operations.

2. Click the Back button to return to Step One and click the Tap shape.

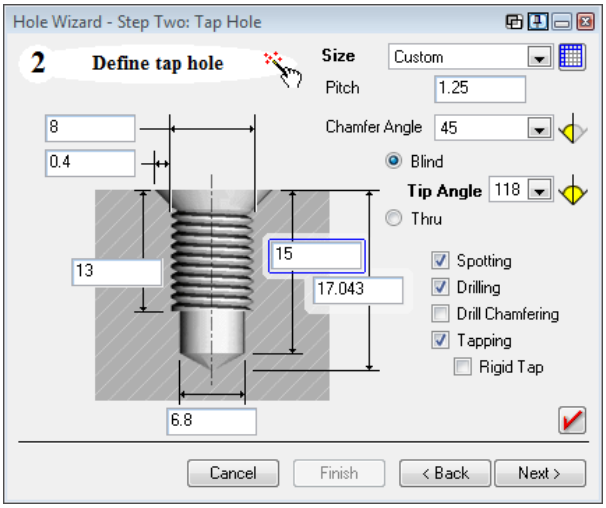


The Hole Wizard - Step Two: Tap Hole dialog appears.

3. Select M8 from the Size pull-down menu.



4. Enter the values as shown. Notice that M8 description is renamed "Custom".

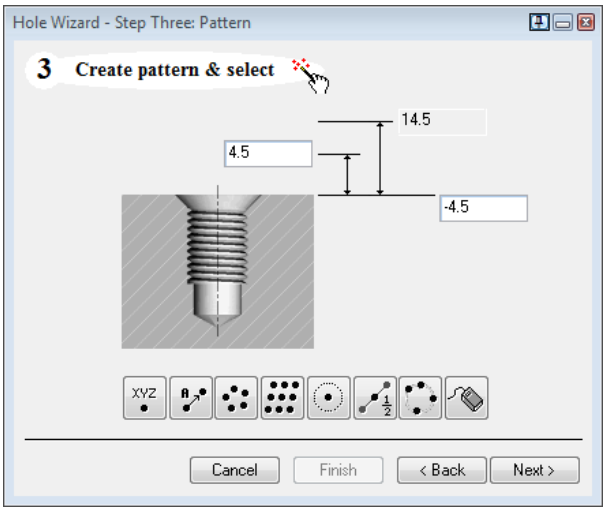


Selecting a pre-set size from the Tap table makes defining a tapping operation very easy.

5. Click the Next button. 

We have the tools needed to make this tap hole so the Hole Wizard Checker has not appeared.

6. In Step Three, set the clearance values as shown.

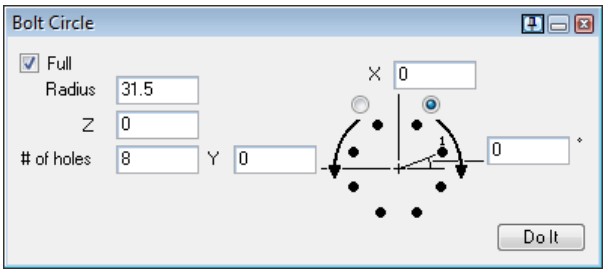


We now need to make the geometry to be machined. We will use the Hole Wizard's point creation capability to make a bolt circle of points on the boss.

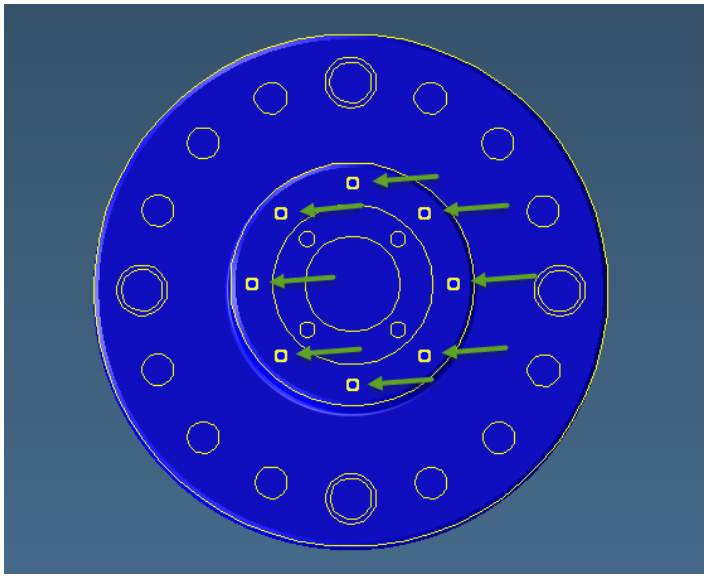
7. Click the Bolt Circle button  at the bottom of Step Three dialog.

The Hole Wizard is now hidden. The Holes palette and the Bolt Circle dialog have appeared.

8. Create the Bolt Circle shown.



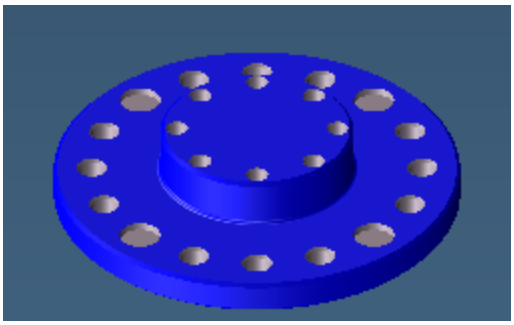
9. Select the points in the Workspace.



10. `Click` close on the Geometry Creation palette to return to the Hole Wizard.

11. `Click` Next and build the processes and operations for the tap holes.

12. Render the tap operations.



50mm Bore Hole

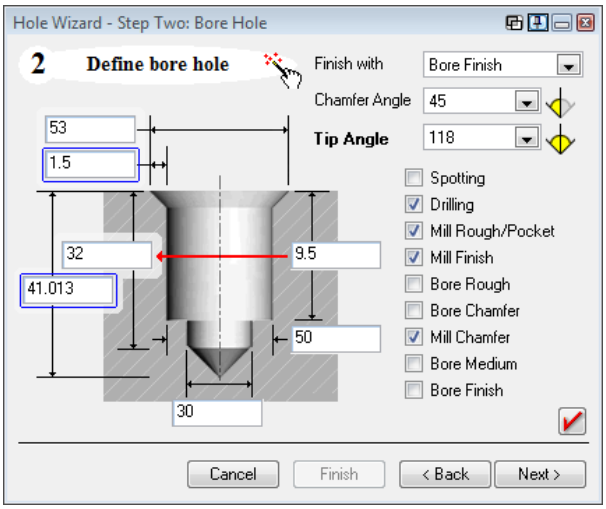
Again, we will complete the four steps of the Hole Wizard to create the bore hole.

Steps 1-4

We will now make a bore hole in the center of the boss. Remember to deselect operations and delete processes.

1. `Click` the Back button until you are at Step One and select the Bore shape. < Back

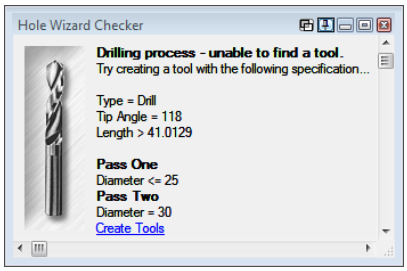
2. Enter the values in Step Two as shown.



Note that we are making the Full Diameter Depth (the 32mm entry) deeper than the Hole Wizard default. Every time the cursor enters the Bore Full Depth box (the 9.5mm entry), the Hole Wizard updates the Full Diameter Depth. thus you will need to enter the Full Diameter Depth after the Bore Full Depth.

3. Click the Next button to move to Step Three. 

The Hole Wizard Checker opens because we don't have a drill defined that is sufficiently large enough to meet the specified Bore Drill Hole Diameter.

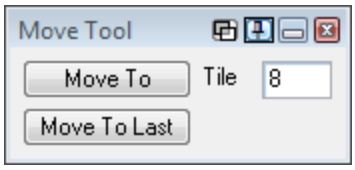


4. Click the Create Tools link.

The system created a tool for us, but we need to move it to a different position.

5. Deselect the other tools that were selected for this process.

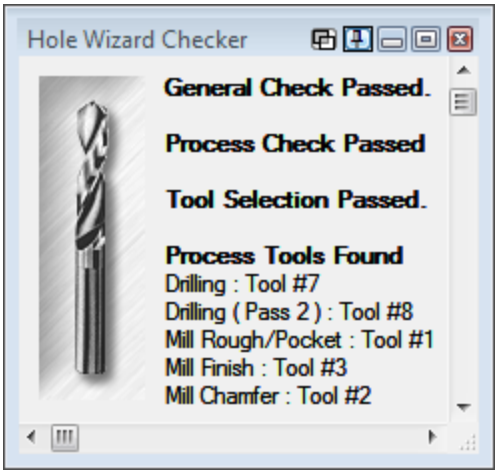
6. Right-click tool #17 that was created and choose Move To.



7. Enter 8 and click the Move To button.

This will place the tool in the empty slot.

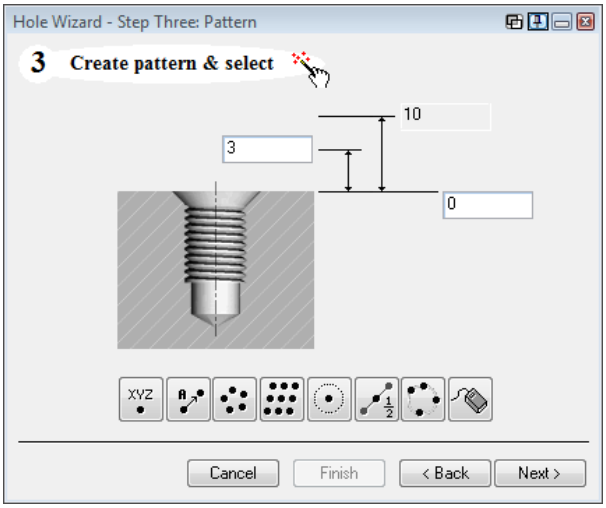
8. Close the Move tool dialog.



9. Click the Back button then click the Hole Wizard Checker button.

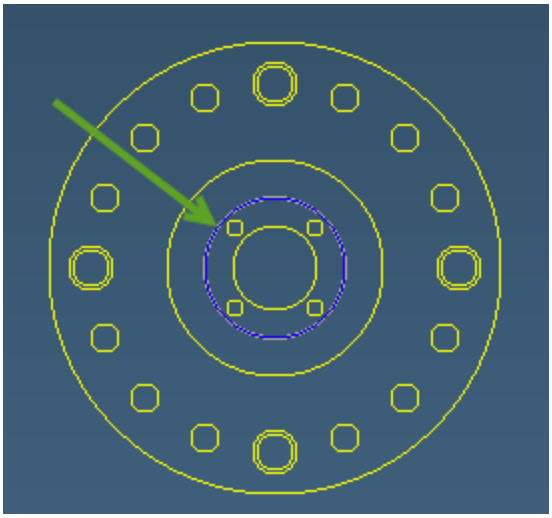
The 30mm drill has satisfied the needs of the Hole Wizard and a smaller drill has been chosen for the first drill pass.

10. Click Next to move to Step Three.



11. Enter the clearance values shown.

12. Roll up  the Hole Wizard and select the circle shown.

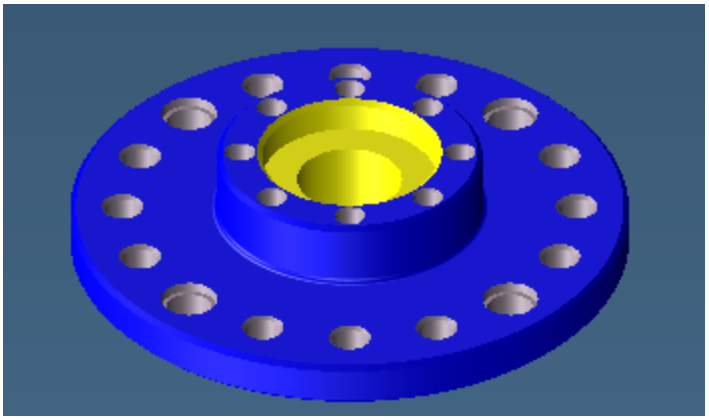


This is the 50mm hole we will mill bore. The inner circle represents the 30mm drill hole.

13. Unroll the Hole Wizard and go to Step Four.

14. Build the processes and operations for the bore hole.

15. Render the Bore operation.



5mm Reamed Holes

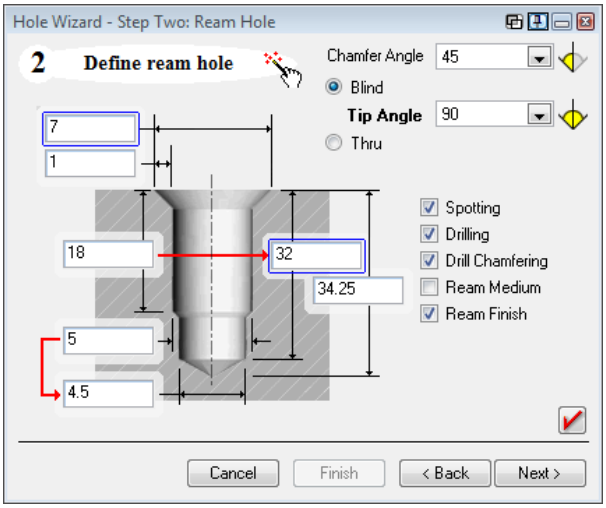
There are four steps to creating these holes.

Steps 1-4

We will now make a reamed hole shape in the bore hole.

1. Select the Ream hole shape button.

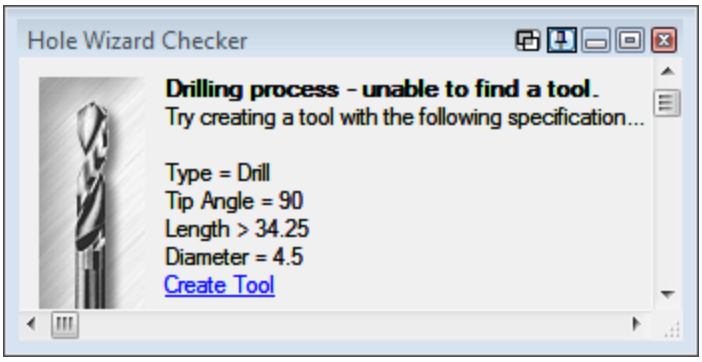
2. Enter the values for the Ream hole as shown.



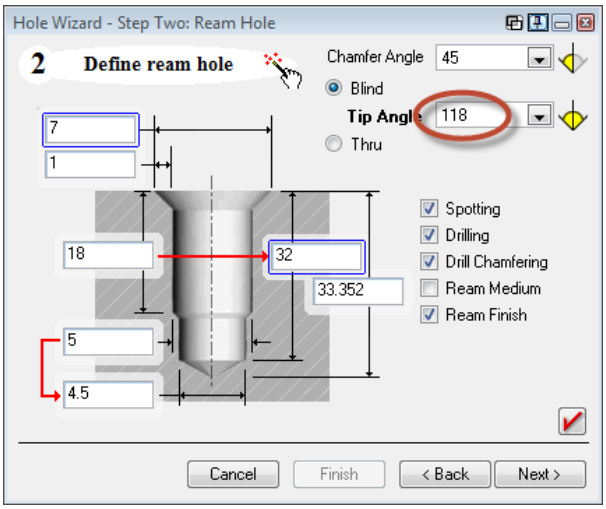
3. For this hole shape we want the full diameter to be deeper than the automatic calculation. This drill needs to go completely through the part. Override the depth value with 32mm.

4. Click the Next button. 

The Hole Wizard Checker has not been able to find a suitable tool.



5. Click the Back button and make the Tip Angle selection 118.

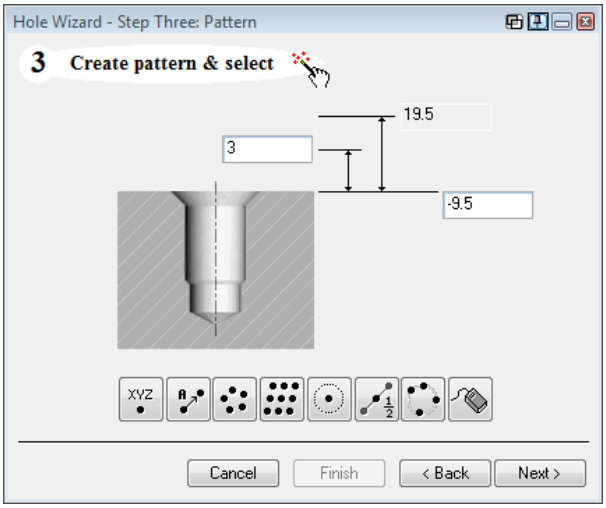


A new drill does not need to be defined. There is a 4.5mm drill in the tool list that has a 118° tip. Since this tool will be used to drill through the bottom of the part, the tip angle used should not make much difference.

6. Click the Next button. 

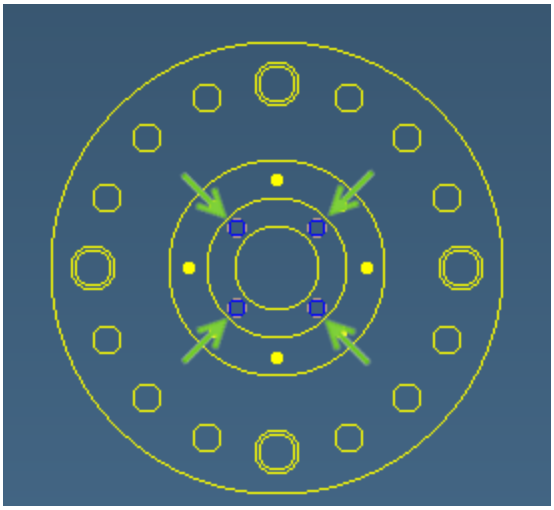
The Hole Wizard Checker has no missing tools now.

7. Enter the clearance values shown.



This will start the reamed holes at the bottom of the 50mm bore hole.

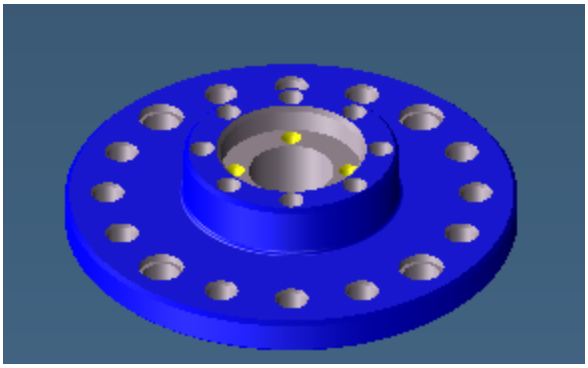
8. Roll up  the Hole Wizard and select the circles shown.



9. Unroll the Hole Wizard and go to Step Four. 

10. Build the processes and operations for the reamed holes.

11. Render the reamed holes.



Save the part as Hole Wizard Tutorial Complete.vnc.

HOLE MANAGER

In this exercise we will use an existing part file to learn about the Hole manager's capabilities. The part file is based on an industry standard automatic feature recognition model. If you have read through the Solids Import manual, you may be familiar with the part. The part has been simplified to remove unneeded shapes and the holes have been made more complex. The part file has a stock body to simplify the visualization when rendering.

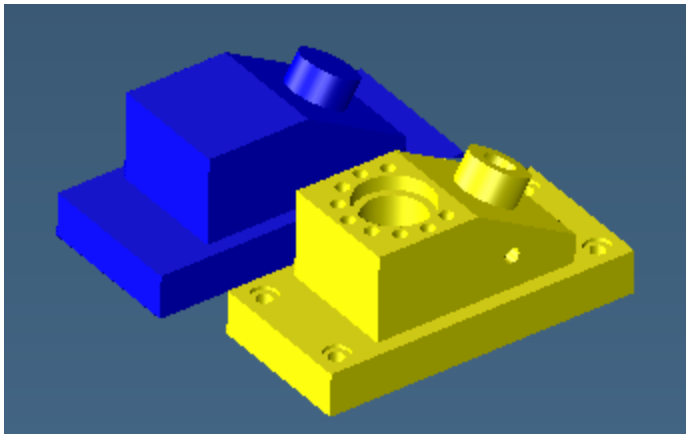
Using the Hole Manager

In this section of the tutorial we will learn how to set up the part, run the Hole Manager, group and sort holes and edit hole data.

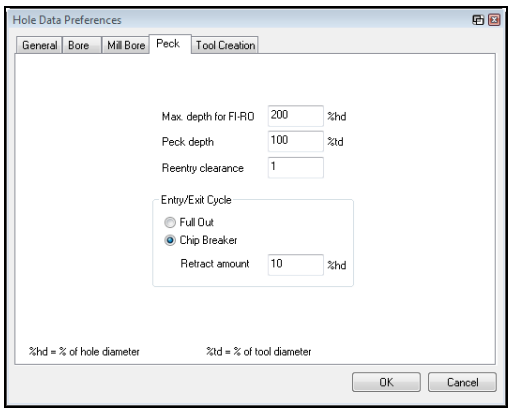
Part Set Up

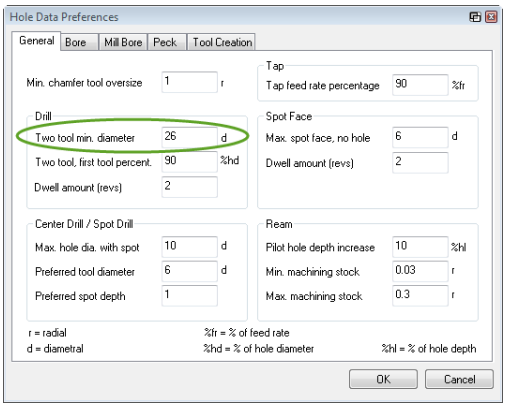
1. Open the part file "Hole Manager.vnc" located in the Sample Parts > Production > Tutorial Parts - Required folder.

The only settings this metric part has is a stock definition located in the body bag. Otherwise this part could have just been imported. The Master Clearance Plane is set to 250mm. All of the tools and processes will be created by the Hole Manager and the Hole Wizard.



2. Ensure the default metric values for the File > Preferences > Machining Prefs > Hole wizard > Hole Data are the same as the following including changing the Two tool minimum diameter to 26mm.





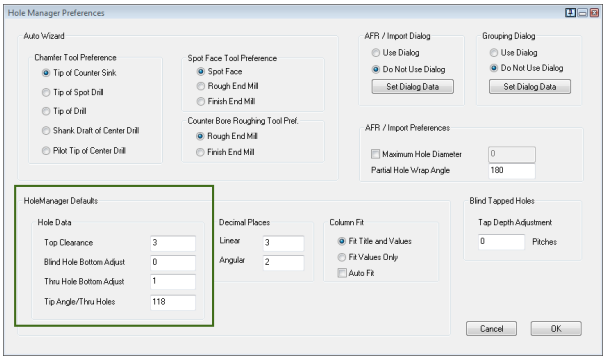
Running the Hole Manager

1. Select the model.

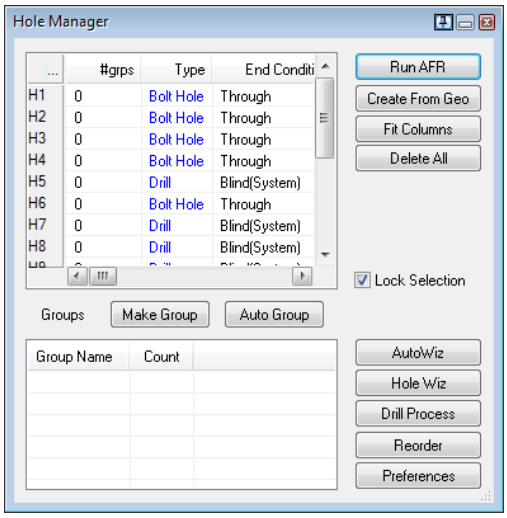
2. Choose Features >  Hole Manager and click the Preferences button.

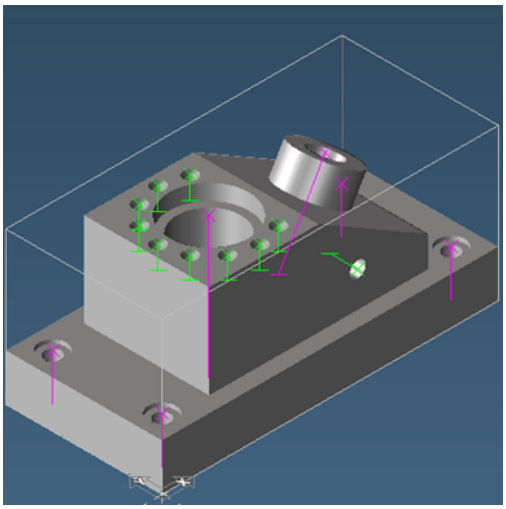
   Use the Hole Manager Defaults as shown, but we will change Top Clearance to 3mm, to automatically add 3mm clearance to the top of all holes. We will also send the tool past the bottom of through holes by 1mm, by changing Thru Hole Bottom Adjust.

   Click OK.



3. In the Hole Manager Dialog click the Run AFR button.





The Hole Manager will now have sixteen entries (ten drill holes and six bolt holes). The workspace will have lines indicating the hole orientation, with an “X” indicating the top of each hole. Through holes end at a point, Blind Holes are shown with a line and a point at the bottom. Holes are color coded (Drill -Green, Bolt Hole - Magenta. The full list of hole color codes can be found in Common Reference>Miscellaneous>Colors>Hole Manager.

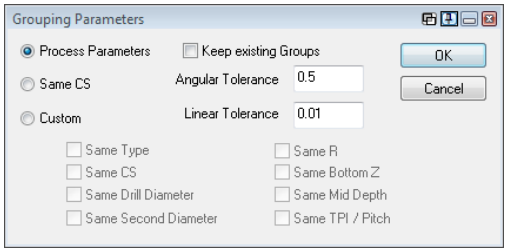
Notice that no extra coordinate systems are added as holes are not associated with particular CS's.

Grouping the Holes

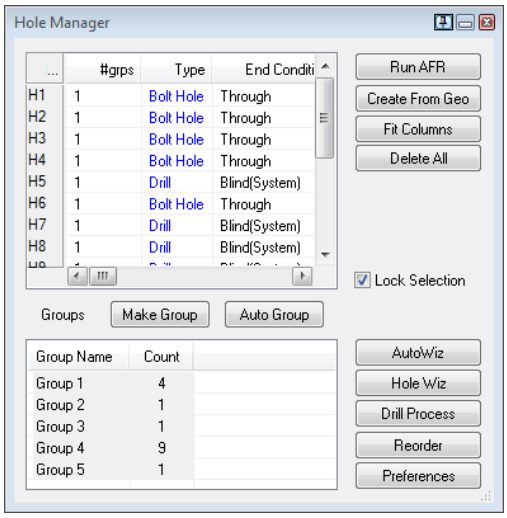
Rather than looking at each hole individually, let’s group the holes and then take a closer look. Go to the Hole Manager Preferences.

1. Click Set Dialog Data in the Grouping Dialog section. Ensure the settings are as shown and click OK.

   Click OK again to close the Preferences dialog.



2. Click the Auto Group button.

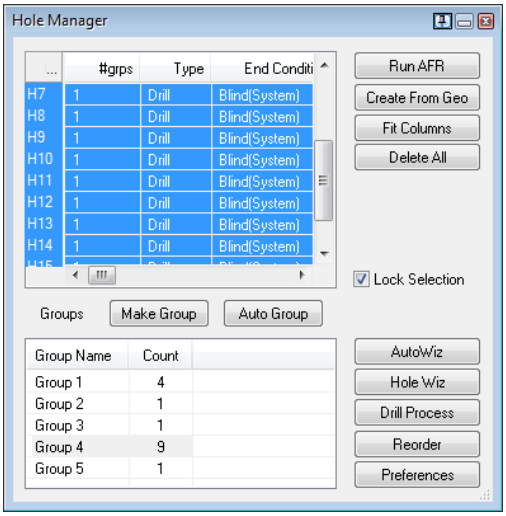


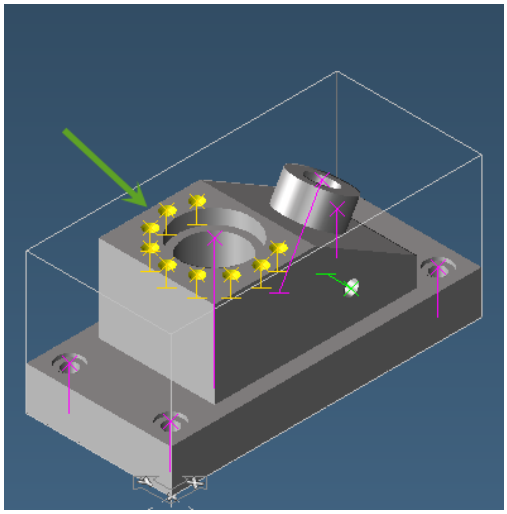
We now have five groups, including two groups of similar holes and three groups of a single hole each. Let's have a look at these groups.

3. Move the Hole Manager dialog so you can see the model.

4. Double-click Group 4, the group with nine holes.

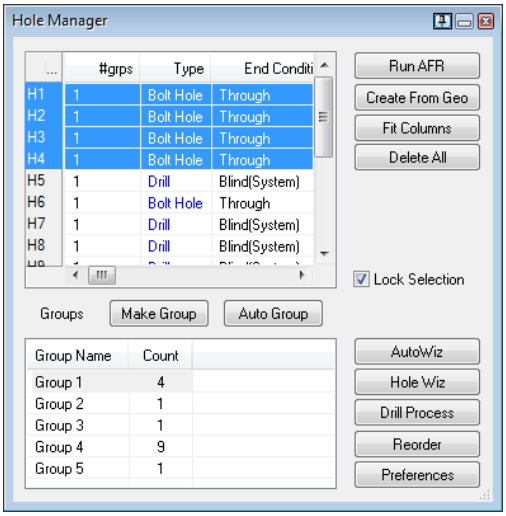
   The Hole Manager highlights the nine hole entries in the list and the model faces that define the nine holes and the point at the bottom of each hole is selected. These nine holes are simple drill holes with a chamfer.

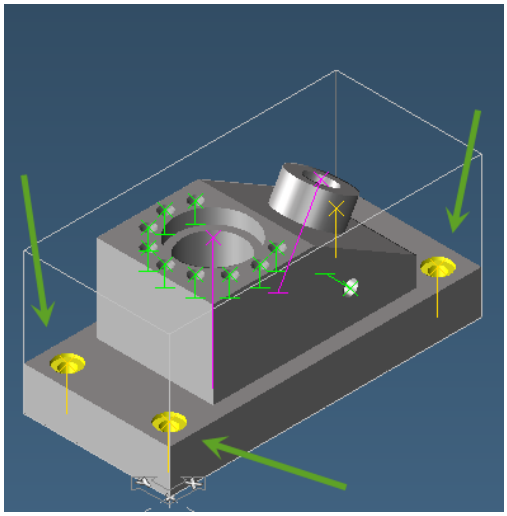




5. Double-click Group 1, the group with four holes.

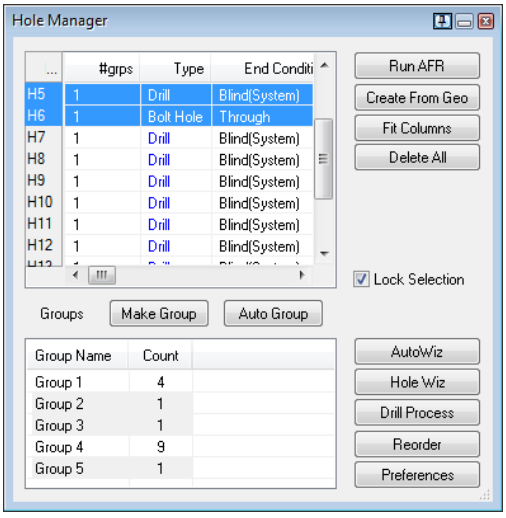
   The Hole Manager highlights the four holes. These holes are bolt holes.

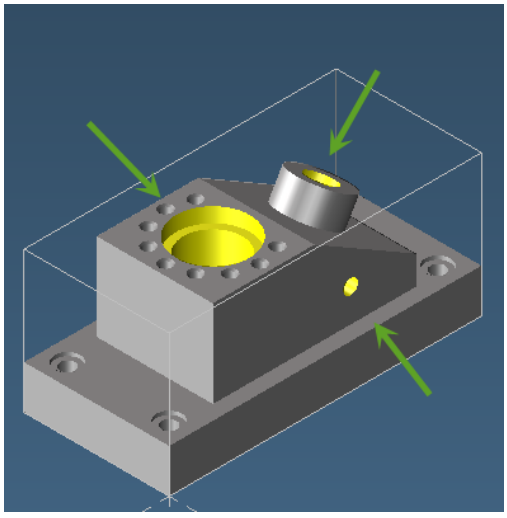




6. `Ctrl+click` Group 2 and 3 and `Ctrl+Double-click` Group 5.

   These groups comprise the three individual holes shown.





Editing Hole Data

Now that we are familiar with the holes and the groups, let's get some work done. We are going to modify the bottom angle of a group of holes to ensure we drive a drill deep enough.

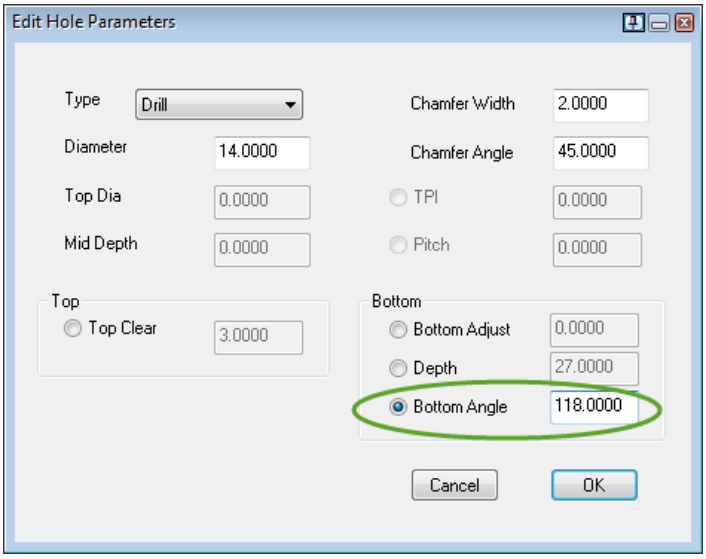
1. `Double-click` Group 4.

   If you widen the Hole Manager dialog and scroll over to the Bottom Angle column (10) in the Hole Manager list, you will see that these nine holes have a bottom angle of 180°. The part was modeled with these holes having a flat bottom. We are going to change these entries to have a Bottom Angle of 118°.

2. Right-click on one of the highlighted holes in the list and choose Edit from the dropdown menu.

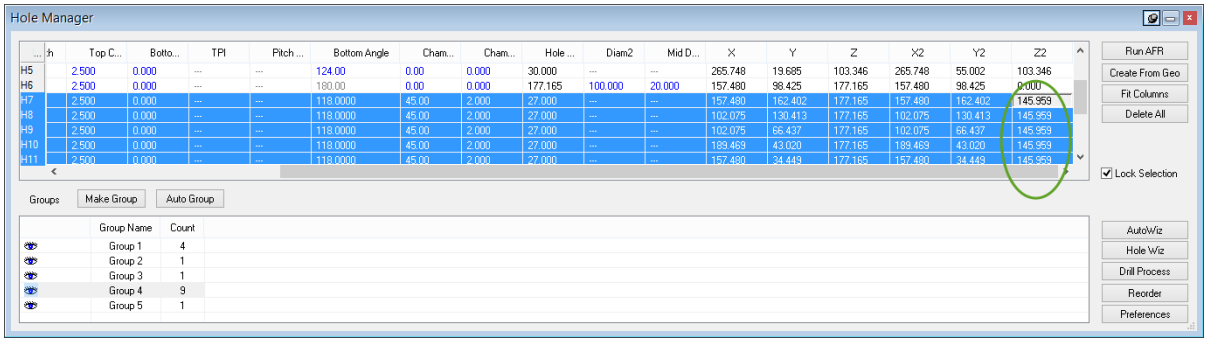
3. Check the Bottom Angle button and change the value to 118.

   Click OK.



By making this change we will drive the full diameter of the drill to the full diameter of the hole when creating toolpath.

Looking at the data in the list we see that Z2 (the final depth of the hole) has been updated to 145.959 due to the change in the Bottom Angle. The Part Top (Z) value is derived directly from the model, thus it will not change with basic edits. Bottom Adjust is used to account for stock.



Sorting Holes

Before we generate operations, let's check the order of the holes in the groups we will be machining. When a group is created, the holes are placed in numerical order, based on hole number. The number assigned to the hole may not be in an order you consider appropriate for optimized toolpath. Therefore, we will sort the tools and re-order the holes in a group.

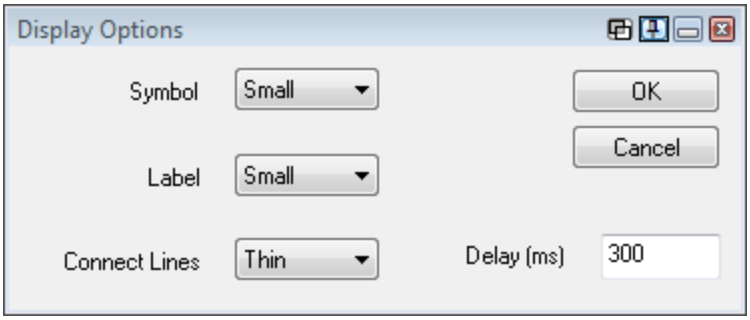
1. With Group 4 selected, click the Reorder button.

   This brings up the Reorder Group dialog. This allows us to view the hole order and sort holes in a number of ways including:

a. Manually by drag and drop
b. Automatically by a planar sort, either an “S” pattern or by finding the next closest hole
c. Rotary sort
d. Reversing the current order
e. Sort by order between two points
f. Following a connected shape

2. Click the Options button, change Connect Lines to Thin, and the Delay (ms) to 300.

   Click OK.

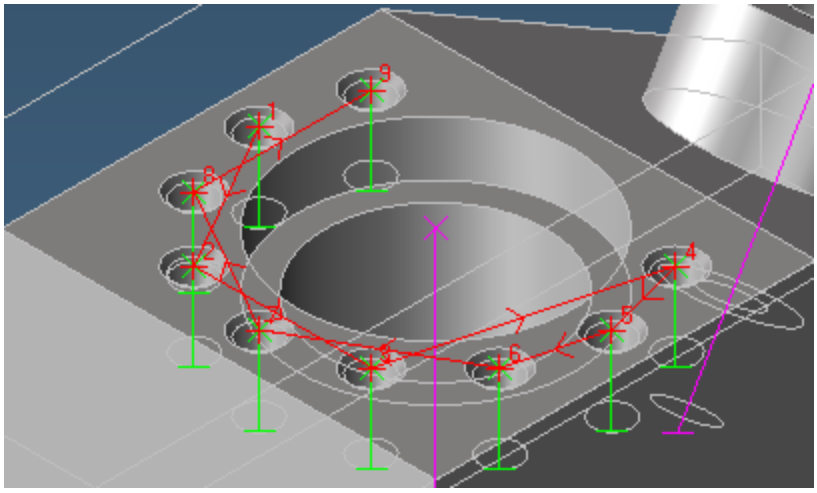


By changing these items, the order in which the holes will be drilled will be easier to see.

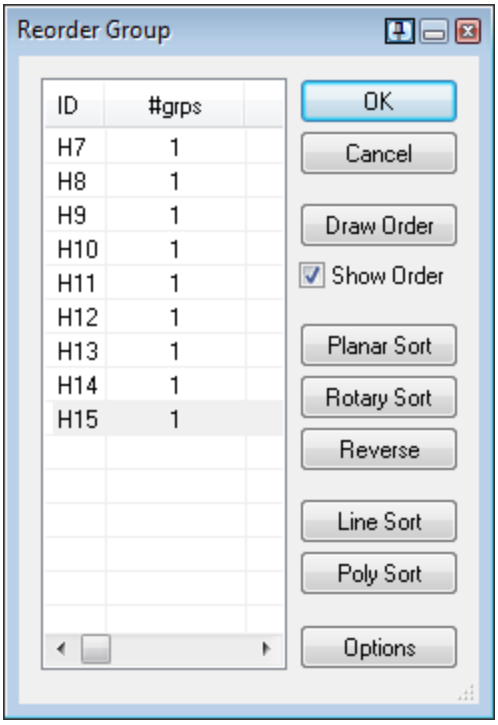
3. Click the Draw Order button.

   The current drilling order is redrawn in red with a slight pause between each hole. The lines show the numbered drilling order, with arrow heads indicating the direction of travel. The current order is not optimal, so we will change the order of the holes.

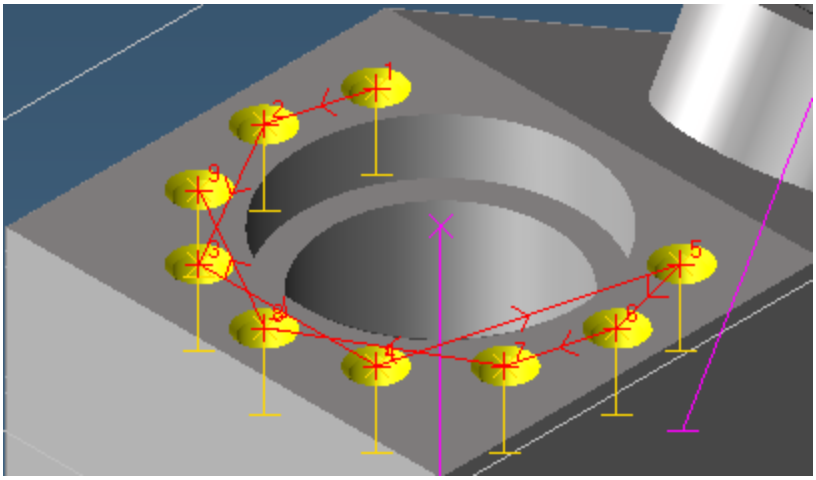
4. Click and hold the mouse over the Hole 15 entry (H15), then drag the position arrow up to just above the Hole 7 entry (H7) and release the mouse.



5. Check the Show Order checkbox.



We have not improved the order. We could drag several times to get the order right. Instead, let's try a different method to reorder the holes.



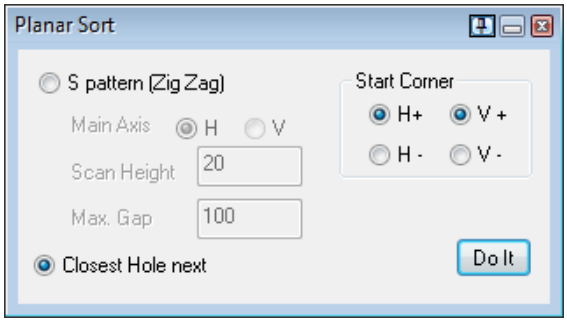
6. Click the Planar Sort button.

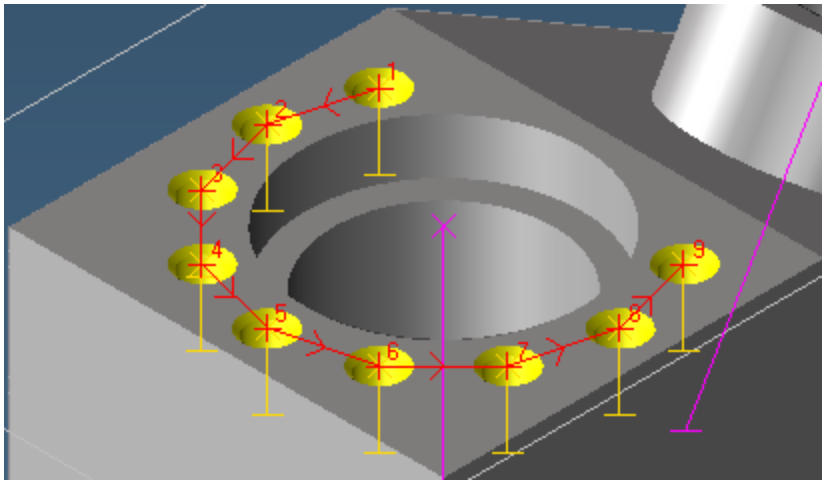
   This dialog allows us to automatically sort points in either an "S" shaped pattern or by finding the next closest hole.

7. Set the dialog as shown and click the Do It button.

We instructed the system to sort the holes by starting at the upper right corner then find the next closest hole. Since this is a circular pattern, the system just followed the bolt hole circle around, counter clockwise, sorting the points.

If you `click` the Draw Order button your display will look like the image shown.





8. `Click` OK to close the Reorder Group dialog and save the new hole order.

   Lets take a look at the other group with multiple holes, Group 1

9. `Double-click` Group 1 to ensure you've got the correct group.

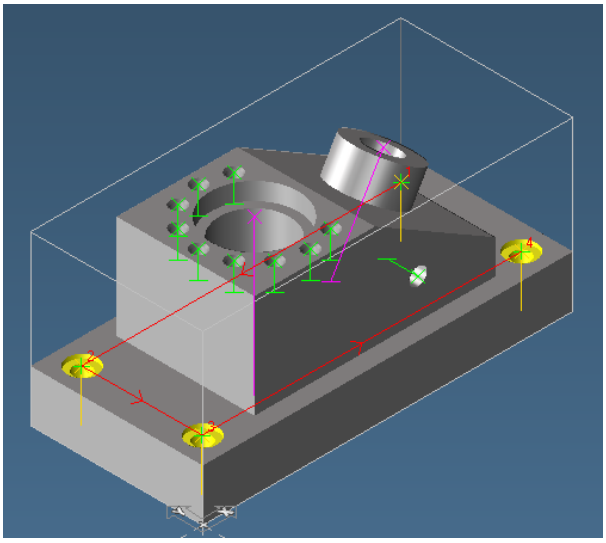
   The four identical bolt holes will be highlighted.

10. `Click` the Reorder button and then the Draw Order button.

    These holes are in a logical order, but we want to set them to machine in the same direction as the previous group – starting at the top-right corner, progressing counter clockwise.

11. `Click` the Reverse button and then the Draw order button.

The holes will now be sorted as shown.



12. Click OK to save the changes to the hole order.

Machining Using the Hole Manager

In this section of the tutorial, we will apply machining processes to five sets of holes.

Nine 14mm Drill Holes

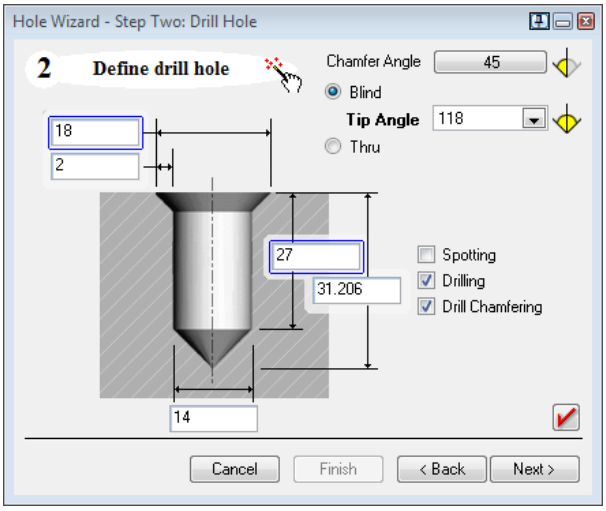
We will generate operations on Group 4 using the Hole Wizard.

1. Select Group 4 and click the Hole Wiz button.

   The Hole wizard opens at Step 2. Step 1 is not needed because the type of hole is already known and is displayed in Type Column #2 of the Hole Manager list.

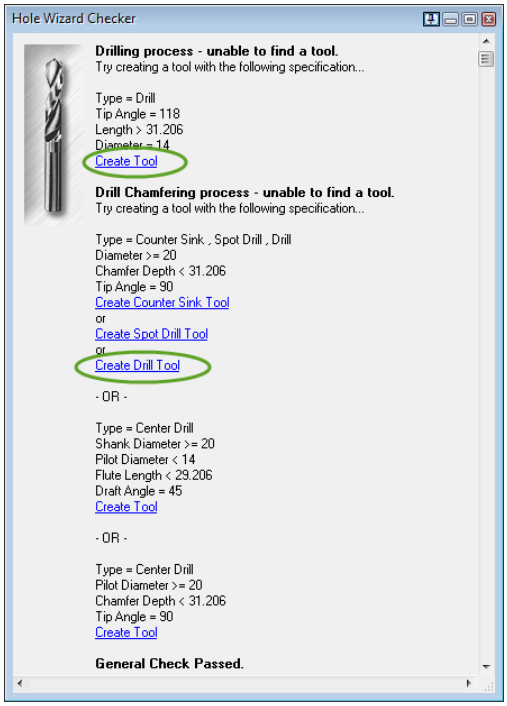
2. Change the tip angle to 118 as shown. Note that Spotting is deselected in favor of Drill Chamfering.

   Click Next.



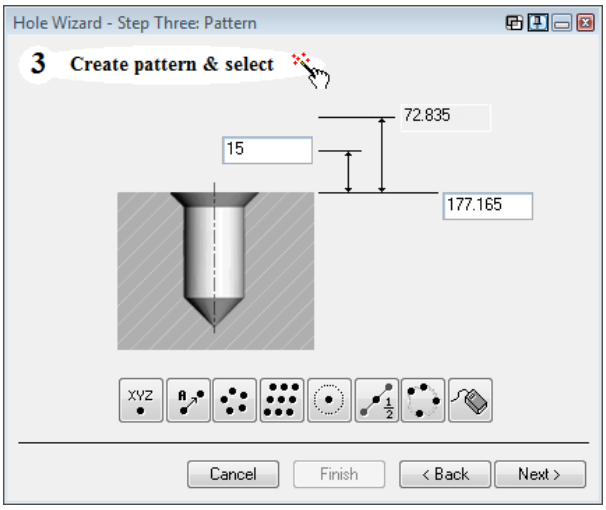
The Hole Wizard Checker runs, prompting you to create tools.

3. Click the Create Tool link in the Hole Wizard Checker for the Drilling process and click the Create Drill Tool link for the Drill Chamfering process.

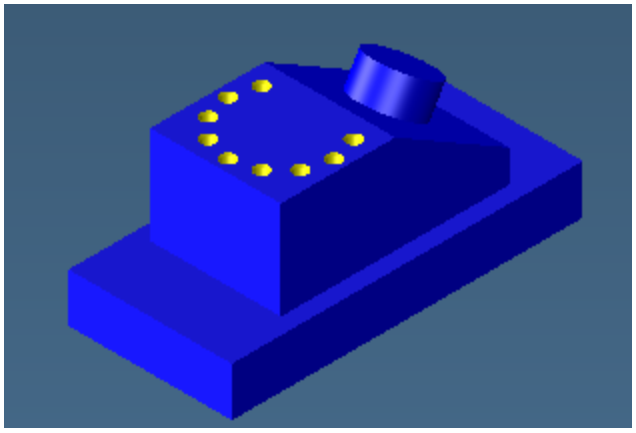


This provides us with maximum tool reuse, minimizing the number of required tools.

4. Change the clearance to 15mm, as shown.



Create the processes and operations then render.



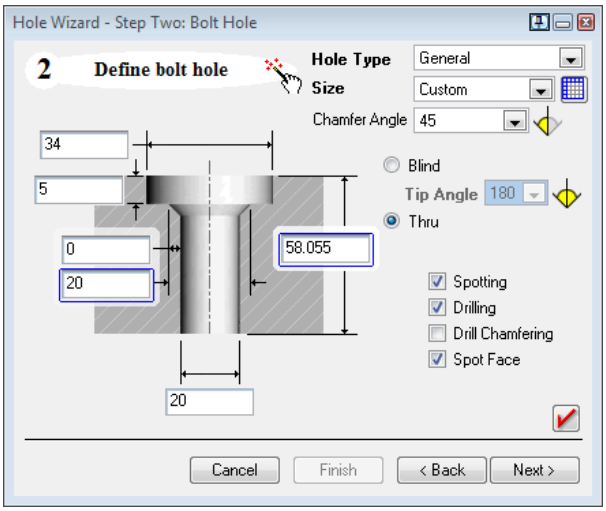
Be sure to deselect the operations before moving on to the next set of holes.

Four 20mm Bolt Holes

We will generate operations on Group 1 using the Hole Wizard.

5. Select Group 1 and click the Hole Wiz button.

   No changes in Step 2 are required.



6. Click Next.

   The Hole Wizard Checker opens, unable to find tools.

7. Click the Create Spot Drill Tool & Create Rough End Mill Tool links.

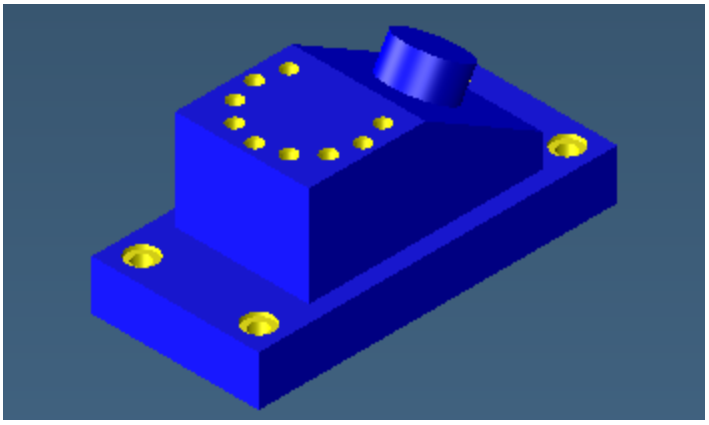
8. Close the Hole Wizard Checker.

   We need to change the clearances. The default entered by the Hole Wizard (3mm above the surface) will not raise the tool over the central boss with the nine drill holes.

9. Change the clearance value to 150mm and click Next.

10. Build the processes and operations, then click Finish.

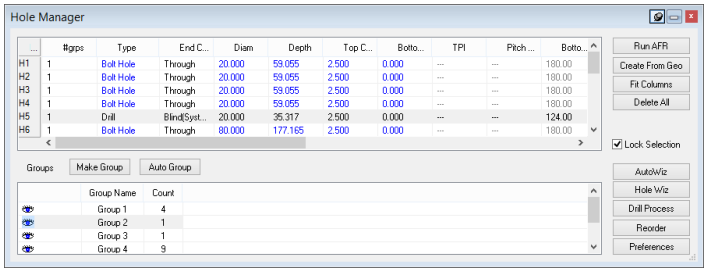
11. Render the operations.



The 20mm Drill Hole

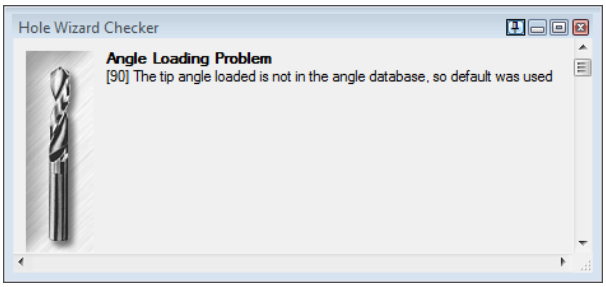
The next group we will generate operations for is the 20mm hole on the side of the part. This hole is the only item in Group 2. Normally, we would not machine this hole next, because of its position. However, we can create the operations and change the machining order later.

If we look at the hole's data, we can see that the Bottom Angle was defined as 124°. The Bottom Angle of the hole isn't particularly important to the model or the machined results. If we wanted to, we could simply select the Bottom Angle entry and change the value to 118, as shown. However, we do not need to do this. We will let the Hole Wizard handle it.



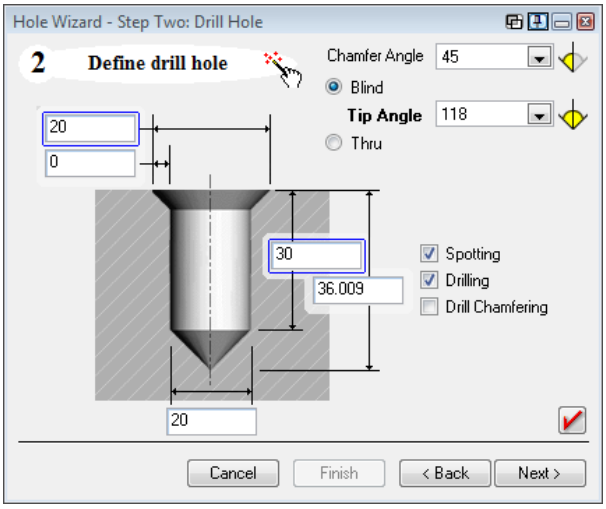
12. Select Group 2 and click the Hole Wiz button.

    A warning appears about the 124° Bottom Angle used for this hole.



13. Ignore the Hole Wizard Checker message and close it.

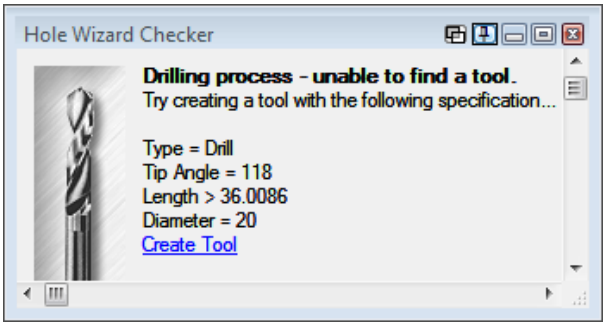
14. Enter the hole definition as shown.



You will likely have to change the Tip Angle from 90° to 118°, as 90° is the default, unless you have changed the Hole Wizard preferences. The only other change you will need to make is to set the Wizard to perform a Spotting then a Drilling operation.

15. Click the Next button.

    The Hole Wizard Checker displays again, prompting you to make a tool.



16. Click the Create Tool link and create the Processes and the Operations.

    Be sure not to overwrite any existing operations.

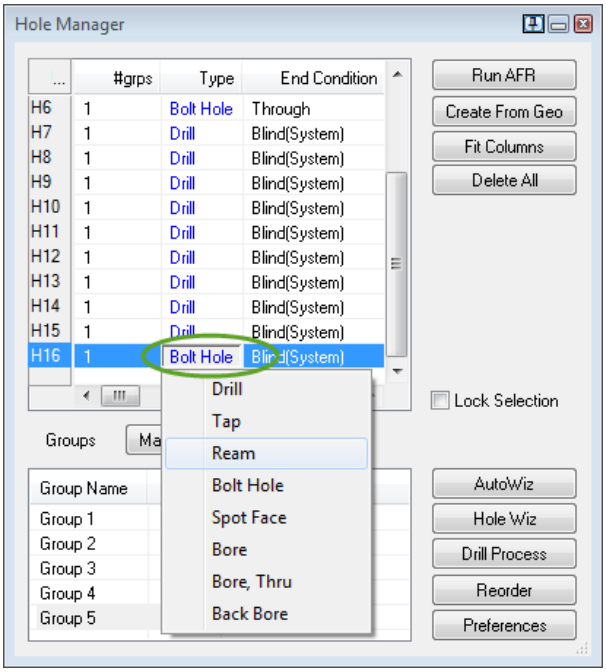
    The Hole Wizard selected the Spot Drill created for the prior hole and the clearances are adequate.

The 40-20mm Reamed Hole

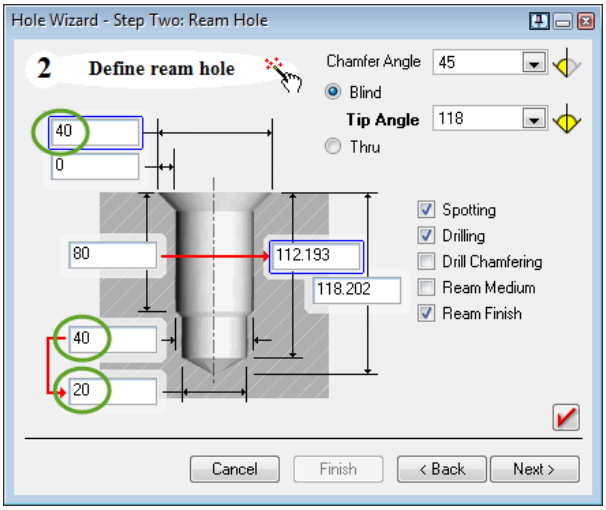
The next hole we will create toolpath for is the hole on the angled boss. The Hole Manager has identified this hole as a Bolt Hole, we are going to change it to be a Ream Hole. First, to minimize the number of tools the system will try to generate, we need to adjust the length of Tool #5.

1. Open Tool #5 and change the length to 120.

2. Double-click Group 5, and click the hole Type in the highlighted entry as shown. Change the type to Ream.



3. Click the Hole Wiz button and set the process parameters as shown.



4. Click the Next button.

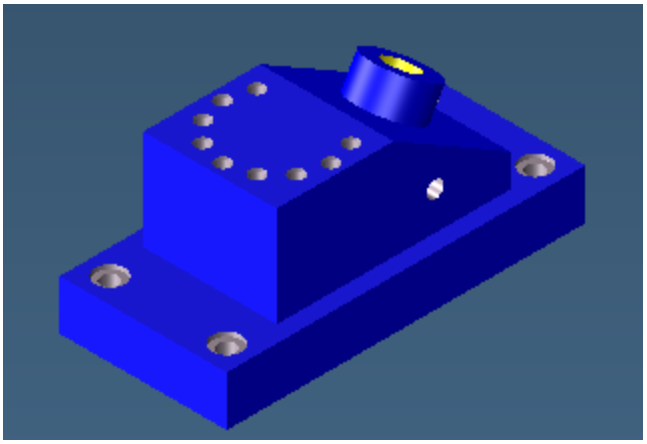
   The Hole Wizard Checker opens, prompting us for a 40mm Reamer.

5. Click the Create Tool link.

   The default clearances are acceptable.

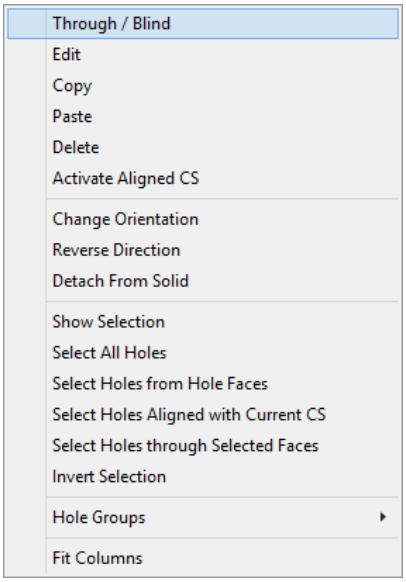
6. Click Next and create the processes and operations.

7. Create the toolpath and render the operations.



The 100-80mm Bore Hole

The remaining hole, 100mm at the top with an 80mm through hole will be cut with a boring tool. Because of the method we have chosen to machine the hole, it needs to be cut in two groups of operations. Classifying the hole as a Bolt Hole provides limited machining options (Spotting, Drilling, Drill Chamfering, and Spot Face) that do not include bore cycles. The boring options in the Hole Wizard do not provide for a multi-depth hole. Therefore, we will first cut the 80mm portion of the hole and then the 100mm portion.



1. Double-click Group 3. This will highlight the Bolt hole in the Holes list.

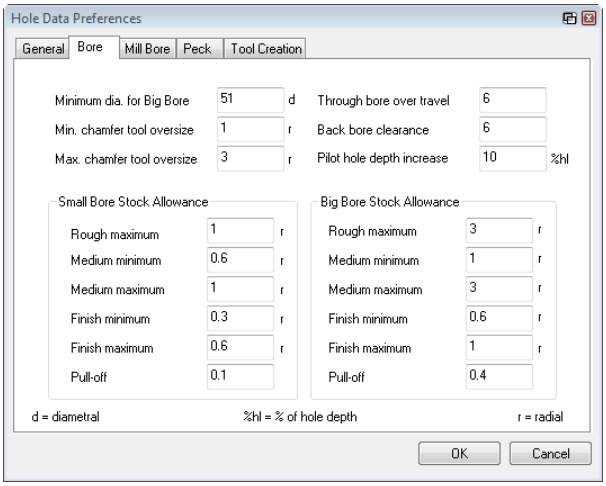
   Right-click the highlighted entry and change the hole entry type to Bore, Thru by clicking the Through/Blind toggle option in the right-click menu.

2. Click the Hole Wiz button in the Hole Manager dialog.

   Before we begin creating these operations, we need to check the Hole Wizard's Bore and Mill Bore preferences. We will change one of these items and examine others to understand what we are doing.

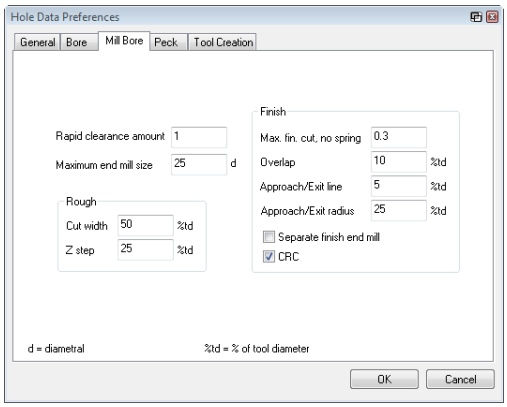
3. Right-click the title bar of the Hole Wizard and choose Hole Data.

In the Bore tab of the Preferences dialog, the Minimum diameter for Big Bore setting of 51mm indicates that this hole (80mm and 100mm diameters) qualifies for the Big Bore Stock Allowances. The Through bore over travel setting tells us the bore will go through the bottom of the hole by 6mm. The maximum amount of material the bore will remove is 3mm, 3mm or 1mm, depending upon whether the pass is a rough, medium or finish pass.

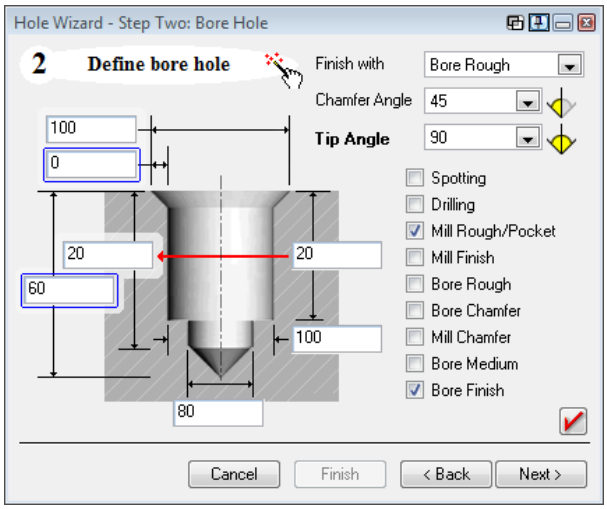


4. In the Mill Bore tab, make sure the Maximum end mill size is set to 25.

The Maximum end mill size states that the system will not define a mill tool for a mill boring process that is larger than 25mm. Cuts are made at the tools radius. You may recall that we changed a two drill setting to 26mm. That setting will allow us to use a single 26mm drill as an entry for a 25mm endmill. If the value was the smaller default the system would use a smaller drill as an entry for the 26mm drill.



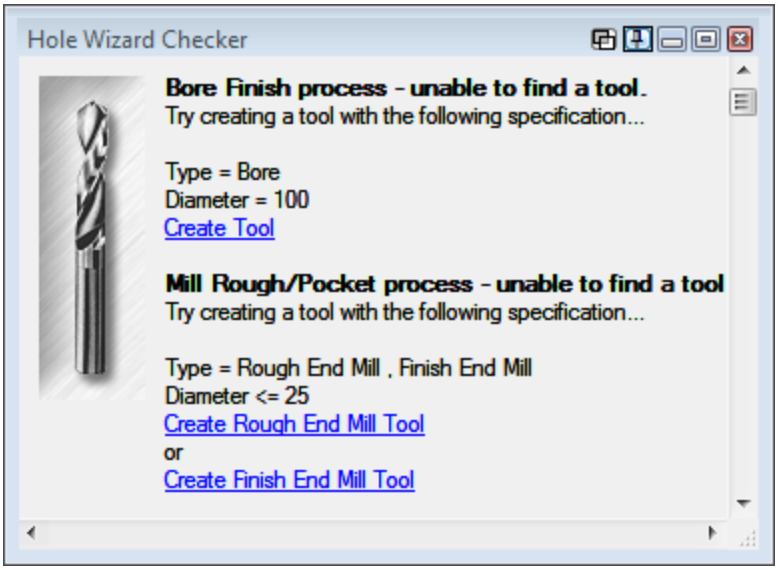
5. Close the Hole Data Preferences dialog and set the Hole Wizard data as shown, starting with the bore drill hole value.



The bore drill hole value will toggle many other settings, so it is a good place to start. With the Chamfer and Finish Diameters being identical, we indicate this will be a straight 80mm hole through the part, ignoring the 100mm section of the hole for now. We will spot the hole, then send a 26mm drill through the part, then pocket it with a rough endmill and finish the hole with a finish bore. We can predict that the end mill will have a 25mm diameter, due to the preference setting we changed.

6. Click Next and create the tools the Hole Wizard Checker is looking for.

   Choose the Create Rough End Mill Tool link.



7. Create the processes and operations.

   We will now cut the 100mm wide section of the hole.

8. If it is not already selected, select Group 3.

9. Click the Hole Wiz button and click the Back button.

   We need to change the process from a thru bore to a blind bore process.

10. In Step One click the Bore hole shape.

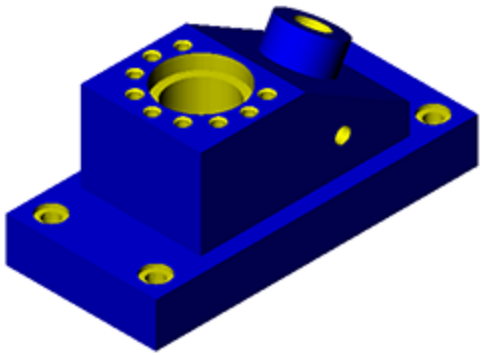
11. Enter the parameters as shown and click the Next button.

The drill diameter of 80mm is rather important. Without it, the endmill would cut air, trying to remove material that is not there. The 80mm value tells the system that, when it gets to the mill rough process, we removed 80mm of material in a drilling process. We did not make the process because we did not need to.

12. Create the tool suggested.

13. Create the toolpath and render the operations.

    This completes the part.

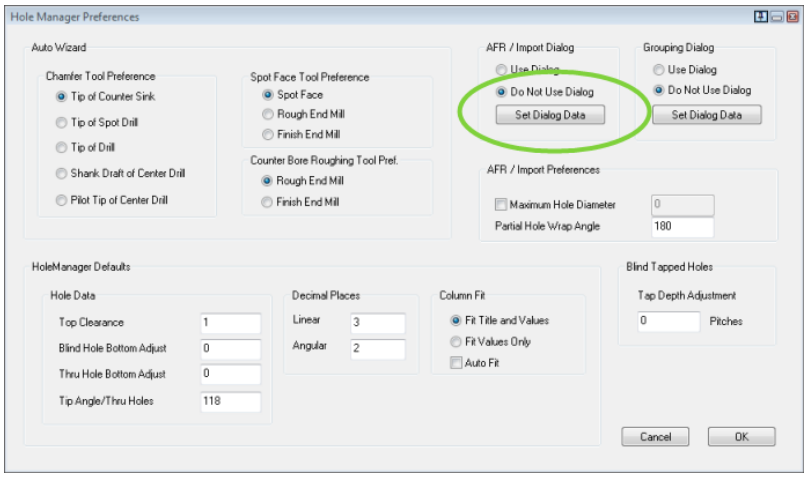


Using Hole Manager with AutoWiz

In this section of the tutorial, we will perform the same machining operations on the same part, this time using Auto Wizard to automate the process. Although Autowiz will work perfectly well for simple parts, in more complex parts it may need slight adjustment.

1. Open the part file “Hole Manager.vnc” located in the Sample Parts/ Production/ Tutorial Parts - Required folder.
2. Select the model.
3. Choose Features > Hole Manager. You can leave this dialog open thoughout the tutorial.
4. Click Preferences
5. Under AFR / Import Dialog make sure Do not use Dialog is checked.

By selecting this option, the Hole Wizard will use the current settings in dialog boxes. The dialog boxes will appear during Hole Wizard operation, but will require no input. In this way, you can determine the amount of automation versus control of individual settings throughout the Hole Wizard process.

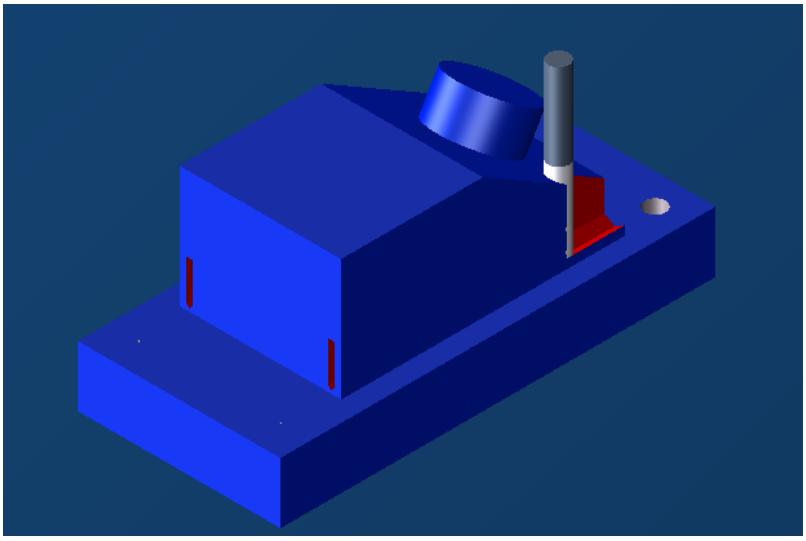


6. Click OK.
7. ClickRun AFR.
8. Click the Auto Group button.

We now have five groups including two groups of similar holes and three groups of a single hole each.

9. Click in the Group box and ensure all groups are selected. (Ctrl-A).
10. Click Autowiz.

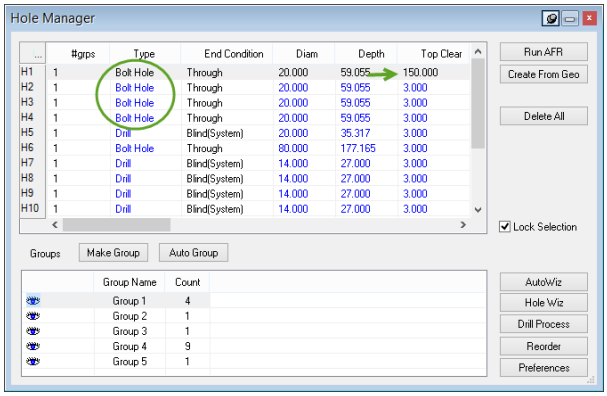
11. Fourteen Operations are successfully created, however when we render the operations there is a problem. The first operations are scrapping the part.



We need to change the clearance level in the operations that machine the 4 bolt holes (H1-4). The default entered by the Hole Wizard (3mm above the surface) will not raise the tool over the central boss with the nine drill holes. This is because Hole Wizard addresses individual holes and cannot take into account what else might be surrounding them. There are two ways to approach fixing this. Either you can edit the clearances within the Hole Manager, or you can directly edit the processes.

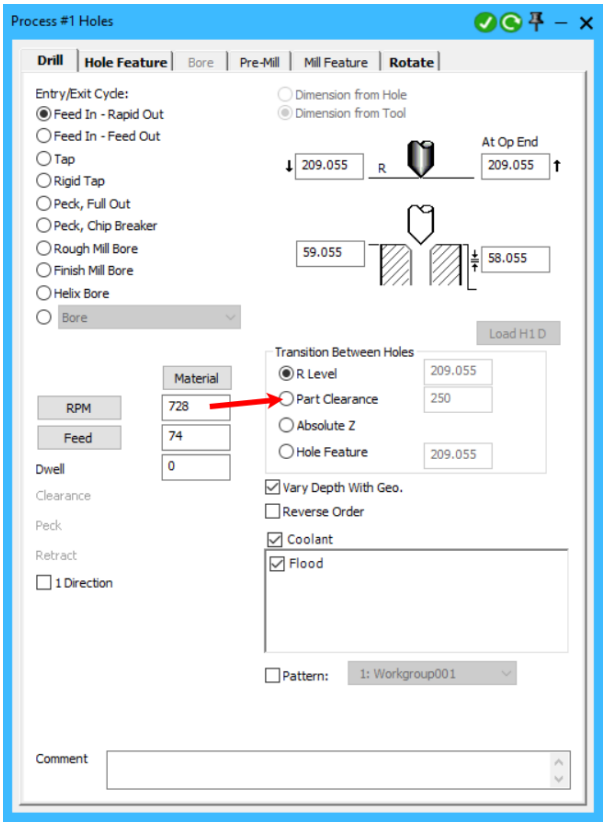
Edit Hole Manager

1. In the Hole Manager dialog (which should still be open), change the Top clearance in the Group One Bolt Holes from `3` to `150`.

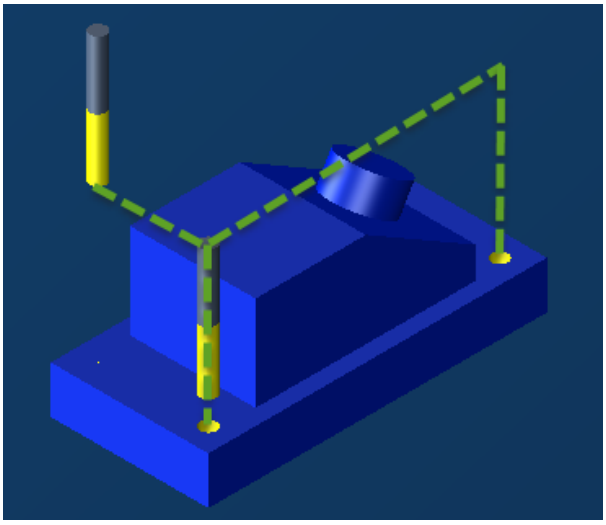


2. `Double-click` Operation tile #1. The Hole Wizard opens at Step Four. Click the Rebuild Processes button then the Rebuild Operations button.
3. Render the part.

Edit Hole Processes directly



1. `Double-click` Operation tile #1. The Hole Wizard opens and the 3 Processes associated with it appear in the process list. Close the Hole Wizard as we do not need it.
2. `Click` on the first Process Tile. You will see that Transition between Holes is set to R level. In order to clear the part we need to change this to Part clearance.
3. `Click` the Part Clearance option and close the dialog.
4. Open the other two processes in the process tile list and change Transition Between Holes to Part Clearance for these in the same way.
5. In the Machining Dialog `click` Redo. 



6. Render the part.

The clearance is now adequate and the tool no longer scraps the part.

The part is complete.

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.

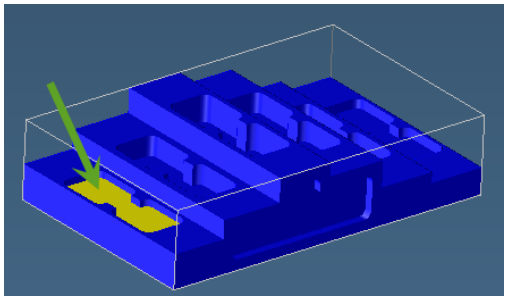
   If you look at this part you will see that there are pockets on three of the surfaces.

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

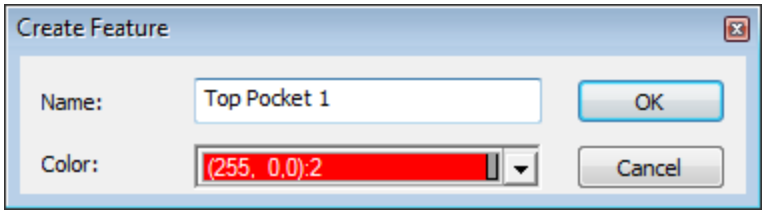
   Keep this dialog open throughout the tutorial.

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 .

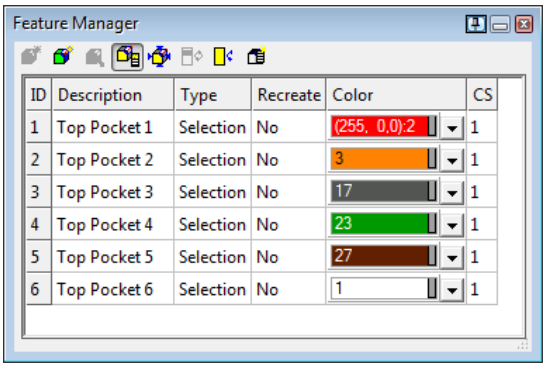


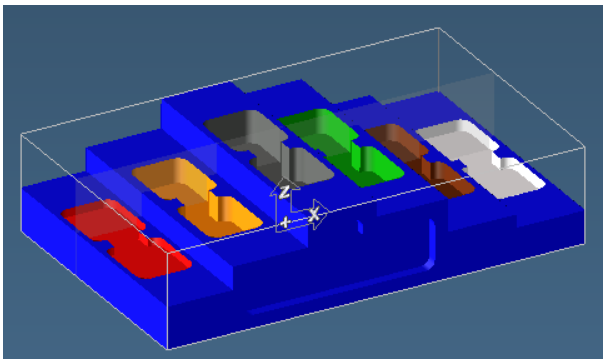
5. In the Feature Manager dialog, click the Create Feature from Selected button and enter details as follows.



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:



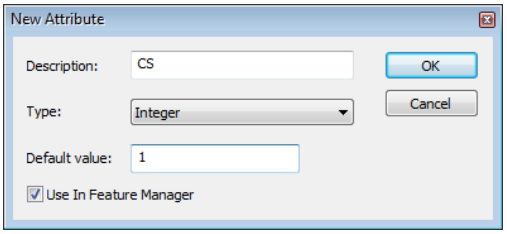


   (The choice of color is not important.)

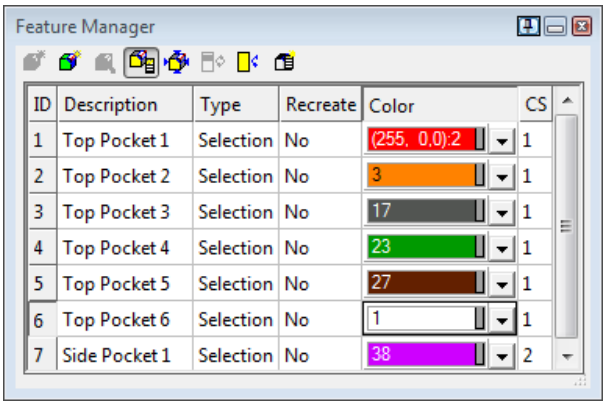
7. Now select the side pocket, making sure to look all around the part to ensure all surfaces are selected. Save the feature as Side Pocket 1.

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.

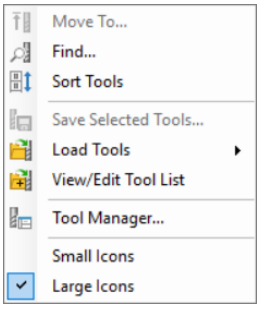


9. We need to edit the Features to add this new attribute. Click in the newly created column and add the following CS values:



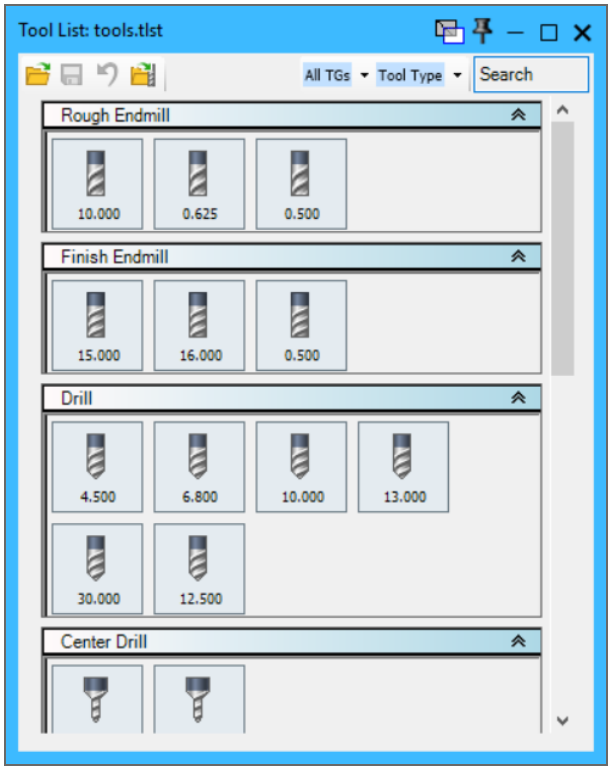
The part is now ready to be machined. First we will load some tools from the Tool list we set up in the Mill tutorial.

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

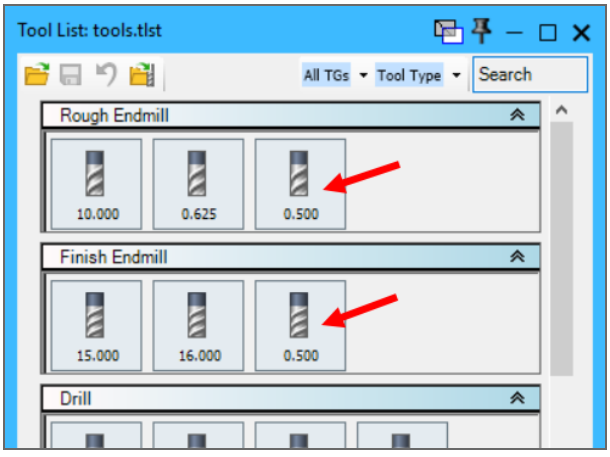


2. Select the option View/Edit Tool List and navigate to the sample file tools.tlst.

3. Click Open. The tool list opens.



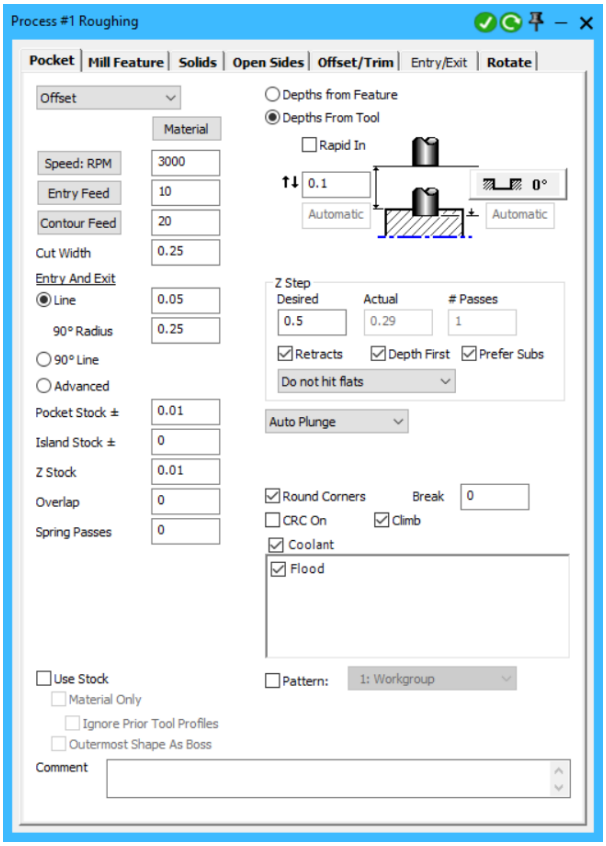
4. Ctrl-click the tools as shown to select them, then right-click and choose the option Import to part.



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.

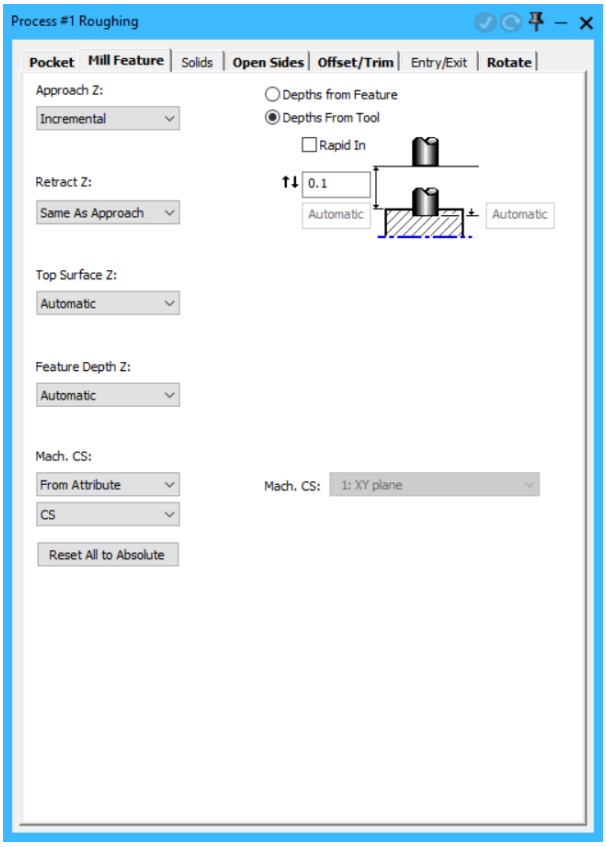
5. Close the Tool List dialog.

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.

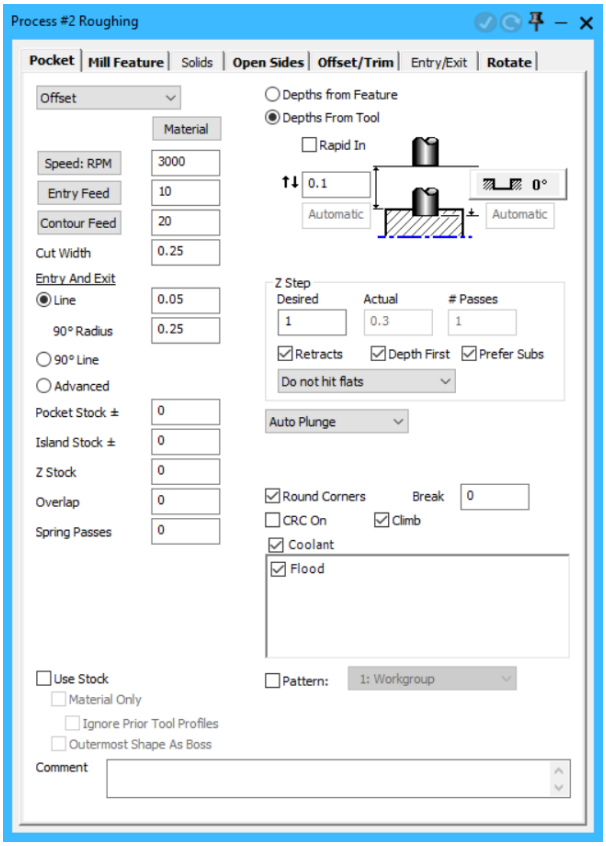


Approach and Retract Z are set to Incremental. The distance between the Clearance Plane and the Top Surface will be updated for each hole.

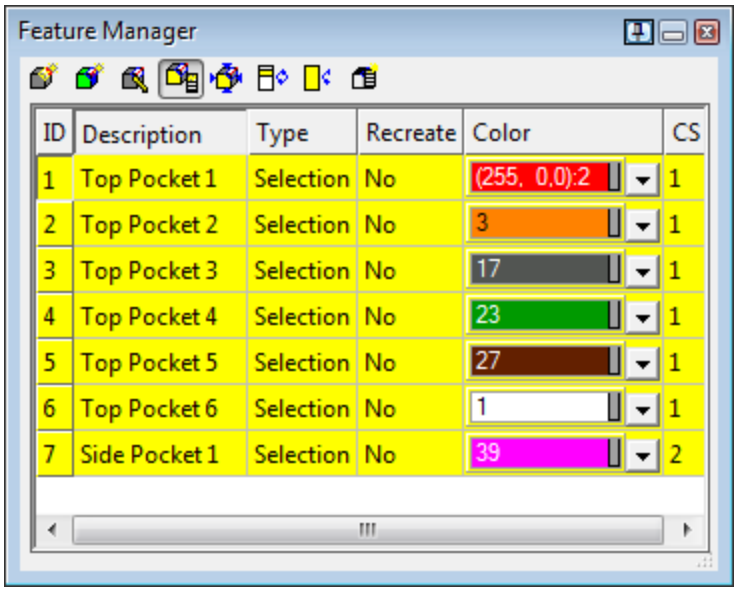
Top Surface and Depth Z are set to Automatic. The value will come directly from the geometry of the user feature.

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.

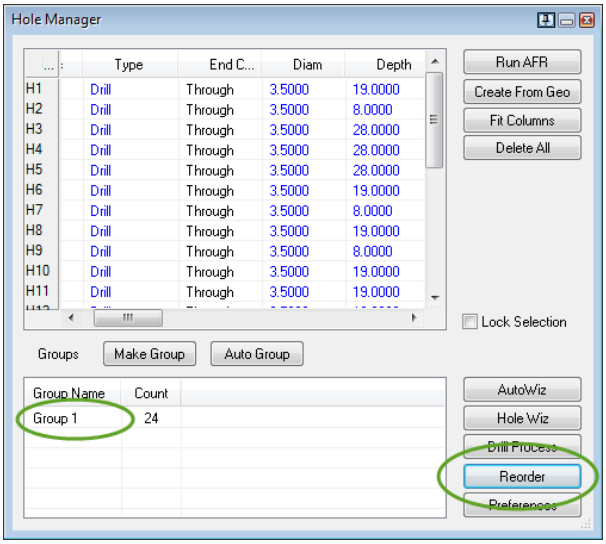
   First we need to select the holes to machine.

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

   24 Holes are found.

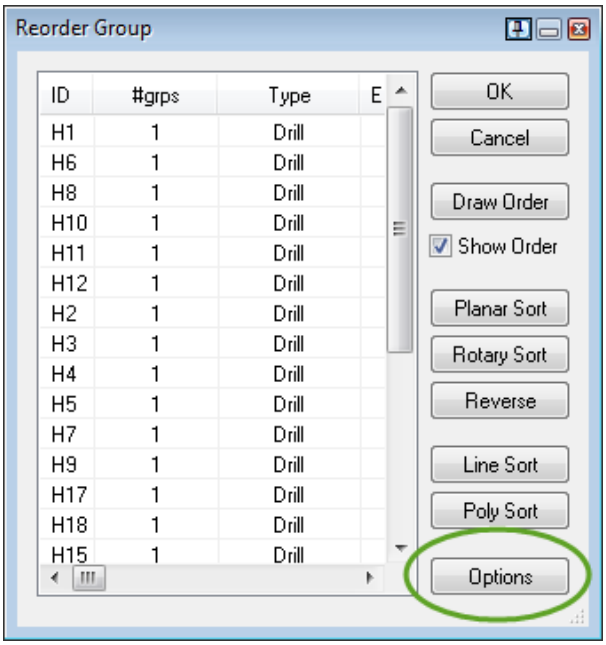
3. Select all the holes (Click into the Hole Manager and (Ctrl-A)). Then click the Make Group button.

   A single Group is created with 24 holes.



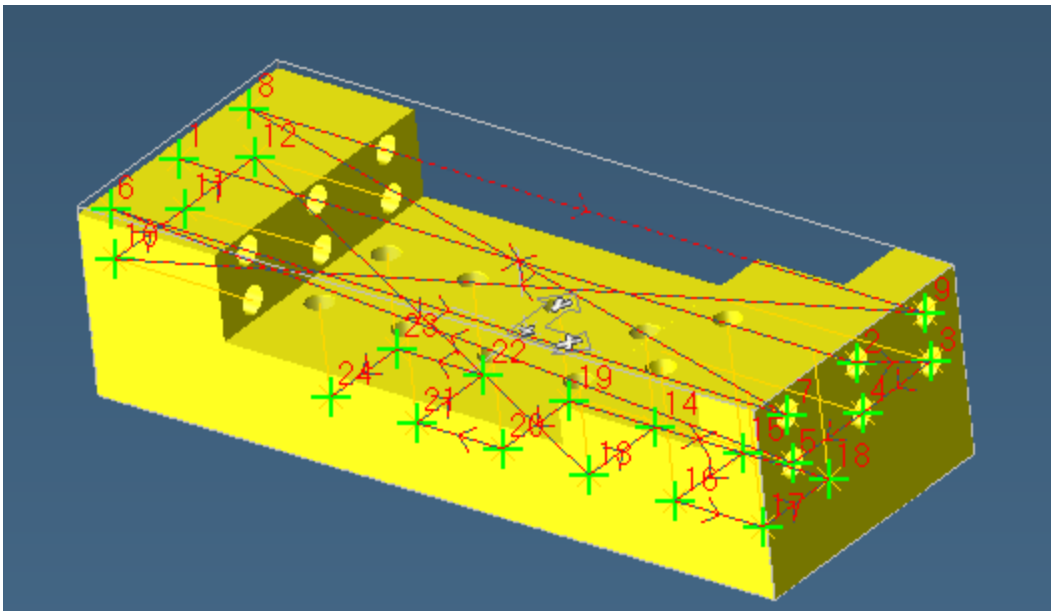
4. Click on Group 1 to select it, then click Reorder.

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:



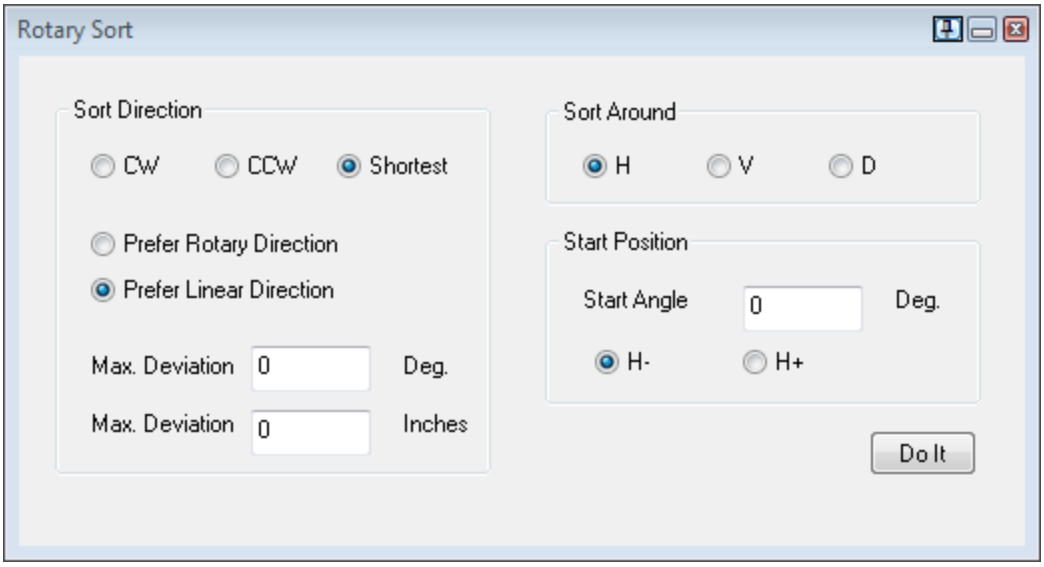
6. Click on Group 1 and click Reorder. This opens the Reorder Group dialog.

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.

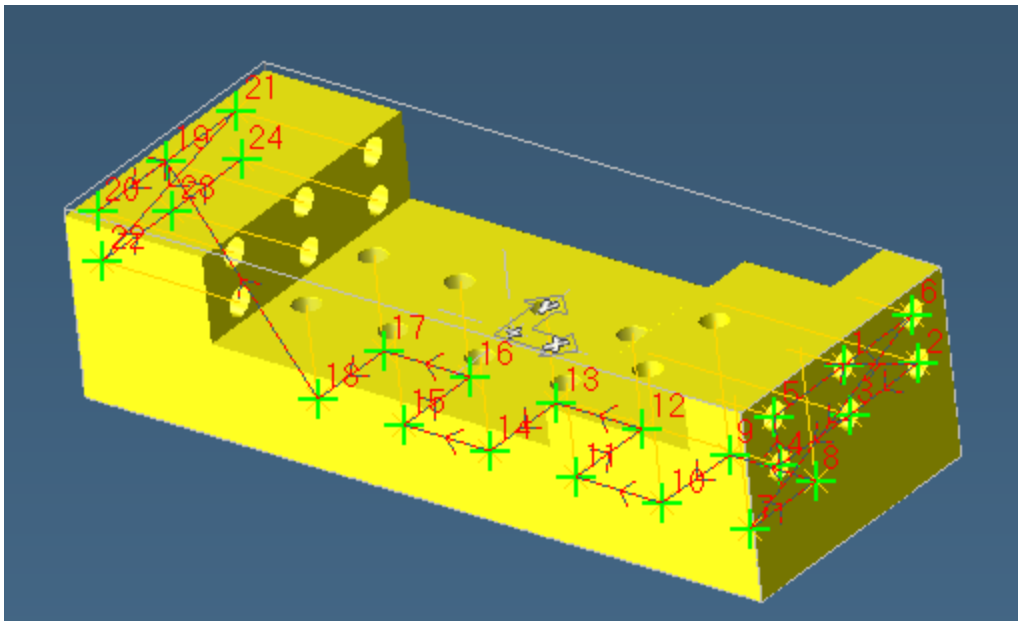


8. Now Click Rotary Sort, enter details as shown below then click Do it .

Do It



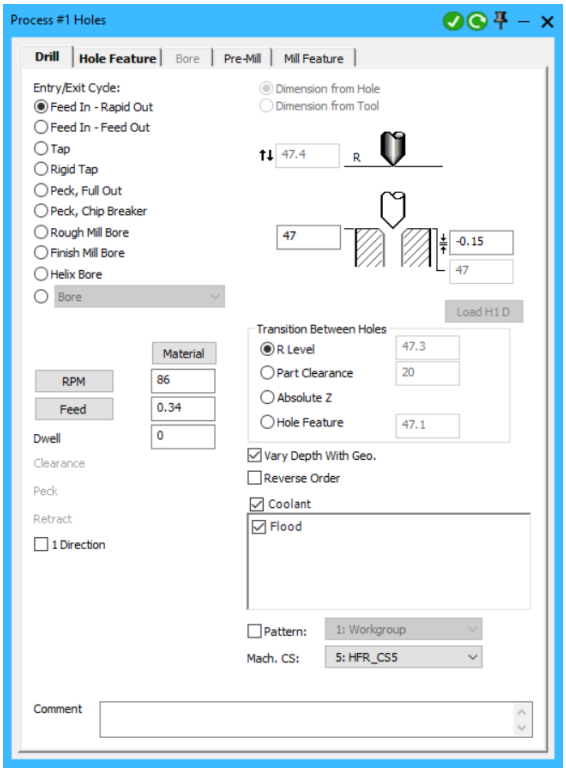
The toolpath is sorted.

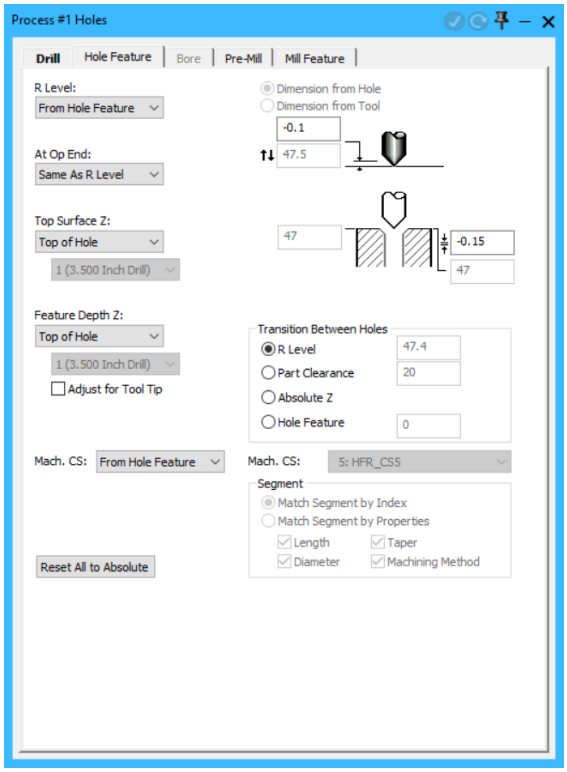


9. Click OK and return to Hole Manager. Group 1 should remain selected. Any holes selected within Hole Manager will be the ones used in the operations. Make sure the Hole Manager dialog is left open, this ensures that the Hole Feature Tab can use the values contained within Hole Manager.

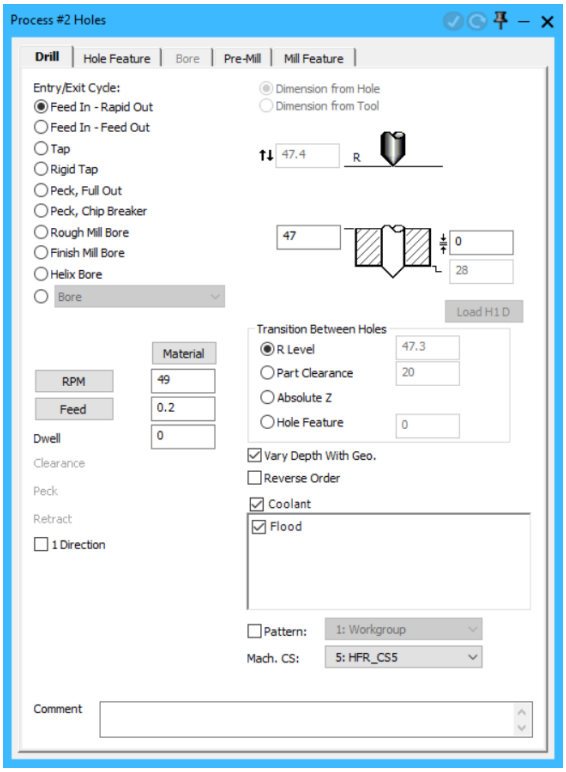
Now we will create two processes, one to Spot drill each hole and the other to Drill them.

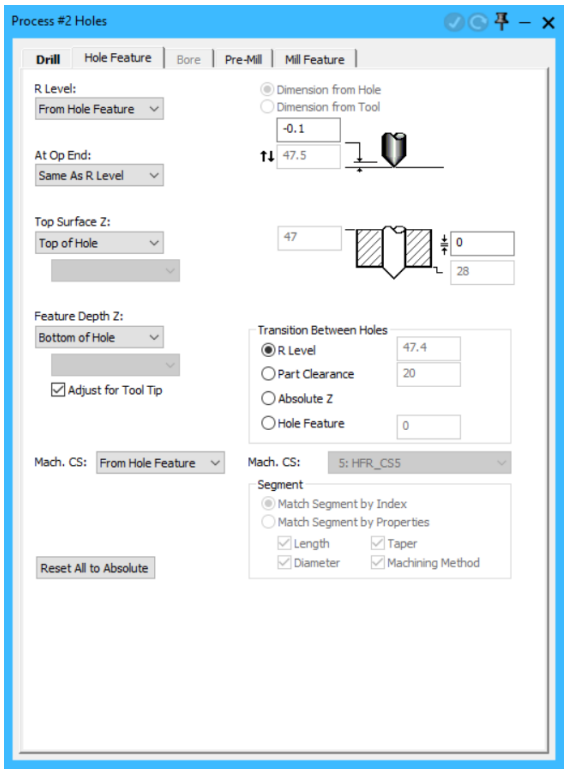
10. Create a Holes Process using the 2.00 Spot tool (Tool #1).





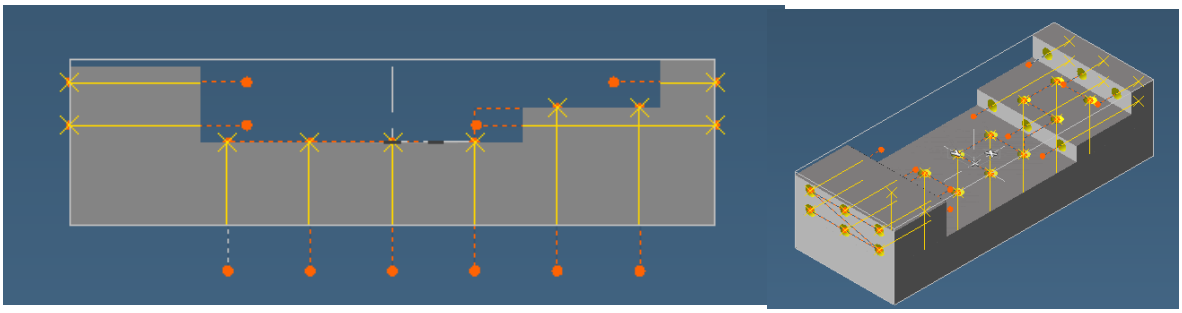
11. Create another Holes Process using the 3.5 Drill (Tool #2).





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.