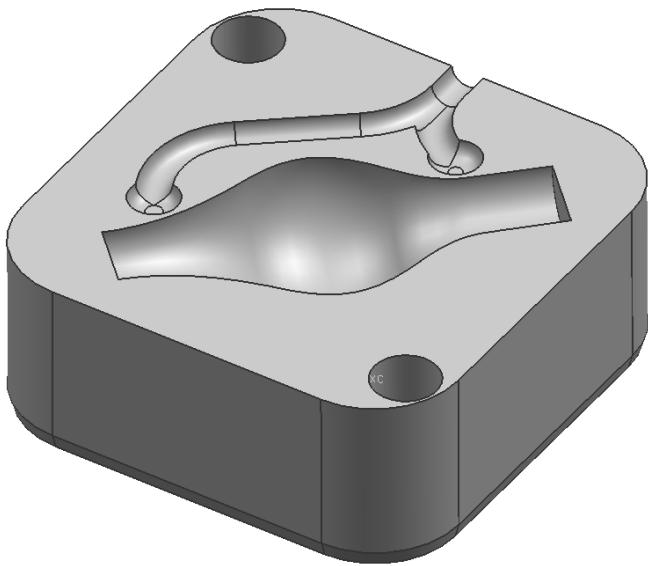


Written by: Mike Beltran, Department of Mechanical Engineering

Last Edited: January 6, 2020 (NX version: 1876)



This walkthrough will guide the student through the use of planar milling, surface milling, and drilling operations in NX manufacturing. Covered topics will include planar milling geometry selection, surface milling path option, drilling procedures and geometry selection, order of operations manipulation, and g-code generation.

This walkthrough assumes the student has completed the NX CAM Tutorial – Basic Manufacturing #1: Setup, Cavity, and Area Milling, and is familiar with creating manufacturing setups in NX. A student should not attempt to complete this walkthrough if they are not confident in using the NX modeling environment. This walkthrough is intended to be completed with instructor guidance.

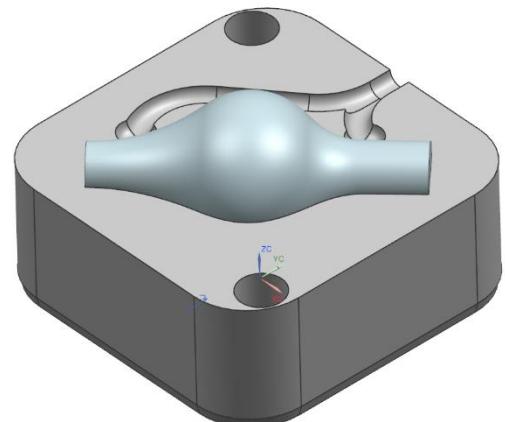
Parts for this walkthrough will be available on Canvas.

### Create the manufacturing assembly and enter the manufacturing environment.

We will create the same basic manufacturing setup as done in the previous walkthrough, by creating an assembly with the blank model overlaid on the part model.

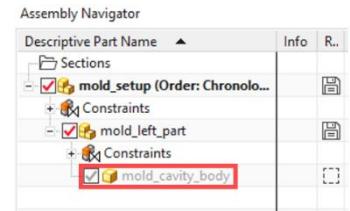
Recall that a proper NX manufacturing setup is an independent assembly file which calls on both the part and blank. A manufacturing setup file should never have part geometry within the file.

1. Create a new assembly file, in inches, and save the assembly to the same directory as the parts you have downloaded into your local user (H:) directory. Name the file mold\_setup.prt.

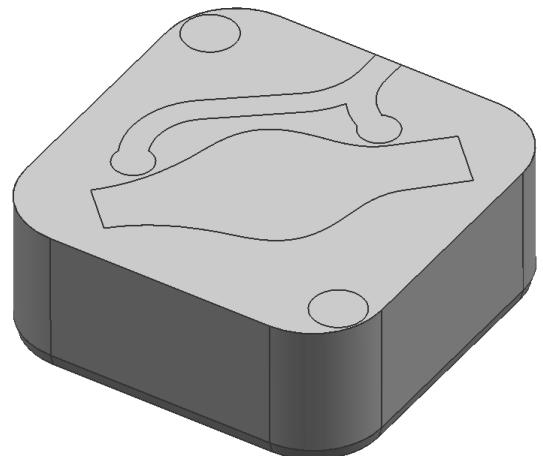
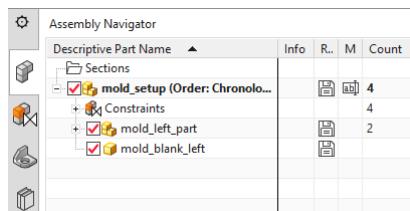


2. Add the part component “mold\_left\_part”, as shown at right, which includes model geometry. Add either a ‘fix’ assembly constraint, or three ‘touch align’ constraints to the assembly absolute CSYS.

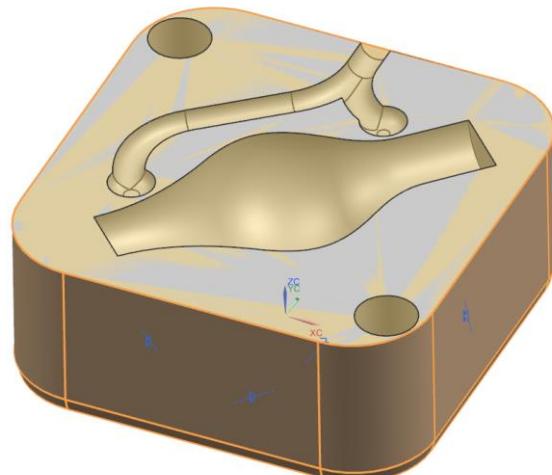
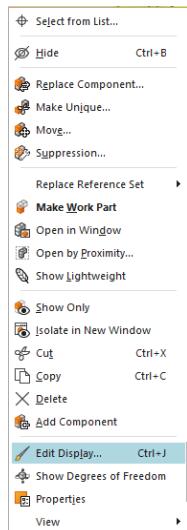
Note: you can hide the core part in the “Assembly Navigator” menu.



3. Add the blank component “mold\_blank\_left”, and align the blank to the part using three touch align assembly constraints, between the front, side, and top faces of each model. Your assembly tree should show something similar to the one shown, with the assembly you have saved shown at top, calling on the two independent models.

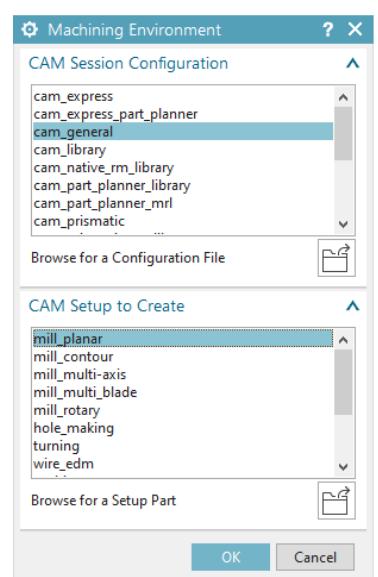


4. Right click the blank part, and go to ‘edit display’. Change the translucency to anywhere between 60% - 90%. Set it to a level you feel comfortable viewing.

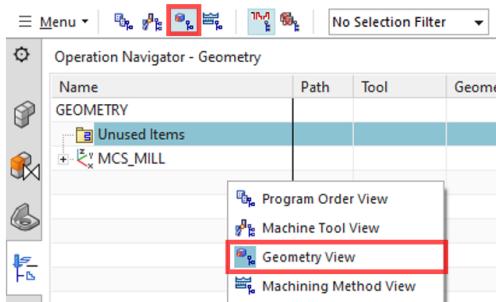


5. Enter the manufacturing environment by going to Application tab and selecting ‘manufacturing’. This will exit the modeling environment, and switch to the manufacturing environment.

6. Under the CAM configuration window, select the options to cam\_general, and mill\_planar. Recall that these are only initial configurations, and will not create any operations.



- Right click in the Operation Navigator, and select geometry view.



### Edit the MCS and create a clearance plane.

We will now edit the default MCS and rapid clearance plane created by NX. To restate, this is the most critical part of creating a manufacturing setup. A mismatch between this MCS and the MCS used in the machine setup can have disastrous consequences.

- Expand the MCS\_MILL item so you can see the Workpiece item. Right click the MCS\_MILL item, and select Edit.
- Before editing any parameters, we must make sure the selection filters in NX allow us to select any items in the model. Just above the modeling window, under the toolbar on the top section of the NX window, pull down the second dropdown menu and select 'entire assembly' to allow every item in the assembly to be selected.

3. In the MCS MILL window, under the Machine Coordinate System section, select the small CSYS Dialogue button to the right of the 'Specify MCS' item.

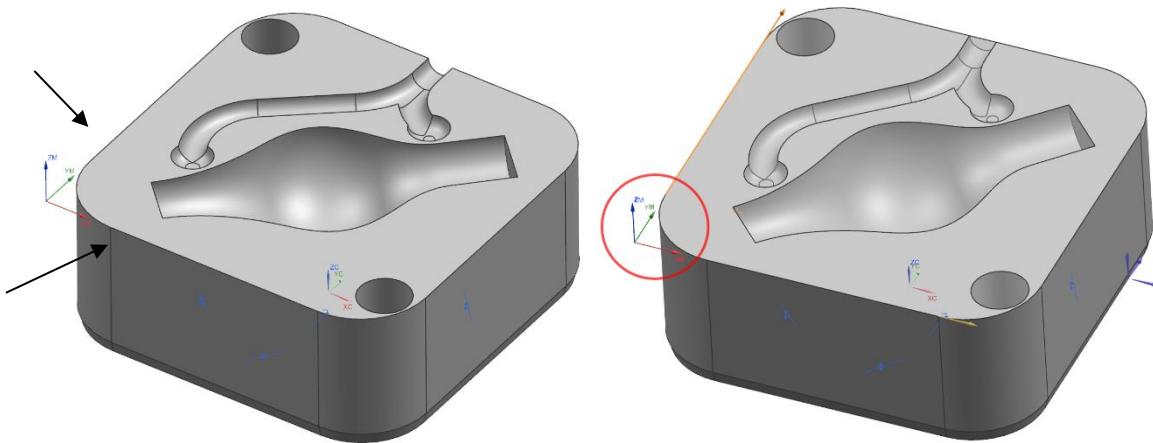
4. We wish to place the MCS on a corner of the top face of the model. However, notice how now there is no single point that exists where we would like to place the MCS, since that corner has been rounded. We must use a unique selection method to specify where we would like to place the MCS. Under the Type section, expand the pulldown menu and observe the various selection methods. Select 'X-Axis, Y-Axis' as the method.

5. We now will select the X-zero and Y-zero axis. Observe how the CSYS dialogue window now has X-Axis and Y-Axis sections, and has the X-Axis vector selection highlighted, indicating we can now select

what we want to be our XAxis. Select the edge of the model opposite the runner entrance on the top face, as shown. You can reverse the direction of the axes using the “Reverse Direction” button.

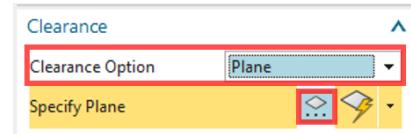


- Once the X-axis vector has been selected, the CSYS dialogue switches to the Y-Axis vector selection. Select the edge of the top face of the model perpendicular to the X-axis, closest to the runners, as shown.

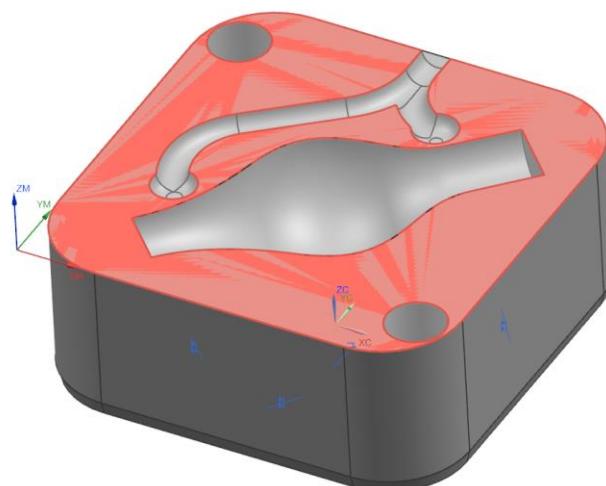
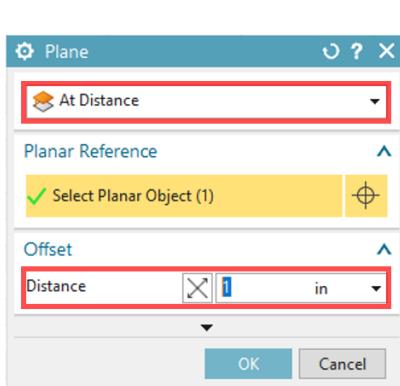


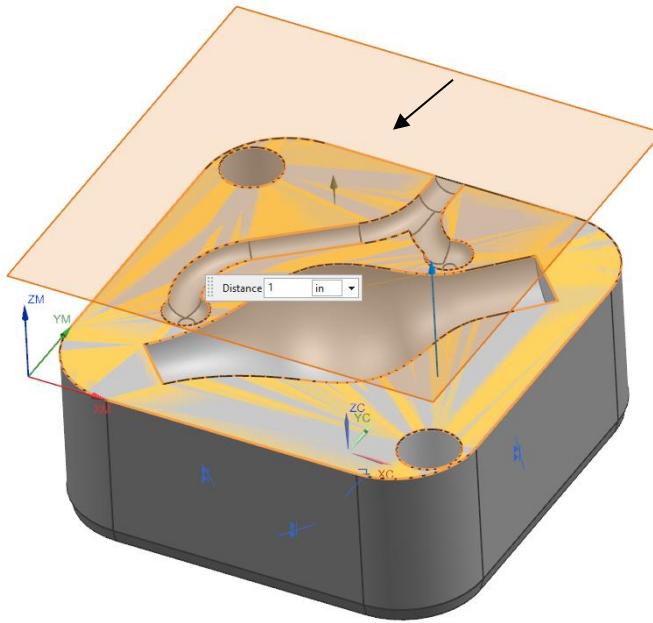
- After selecting the Y-Axis vector, the MCS should appear at the corner that would exist had the corner not been rounded. Select OK to exit the CSYS dialogue. Select OK to exit the CSYS dialogue.

- In the Mill Orient window, under the Clearance section, expand the dropdown menu to select the ‘Plane’ clearance option and then hit on “plane dialog” button.



- Change the plane mode from “Inferred” to “At Distance”. Select the top face of the model, and enter in a 1.00 inch offset. The clearance plane indicator should then appear 1 inch above the top face of the model. Select OK, and check the model from a side orientation to ensure that the clearance plane has been placed where desired.

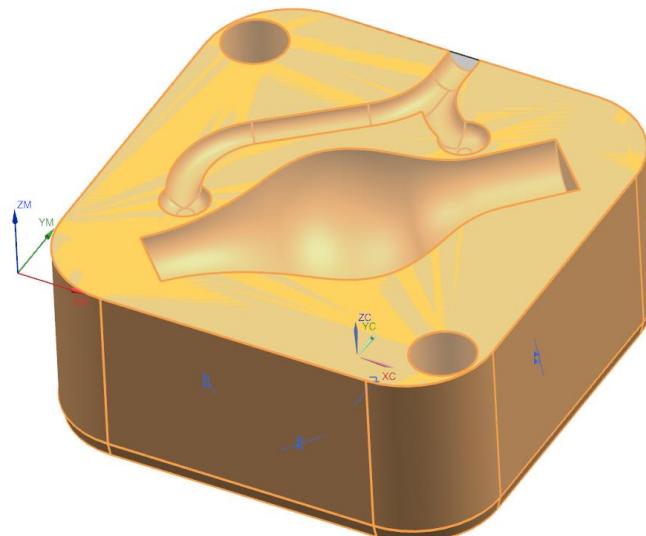
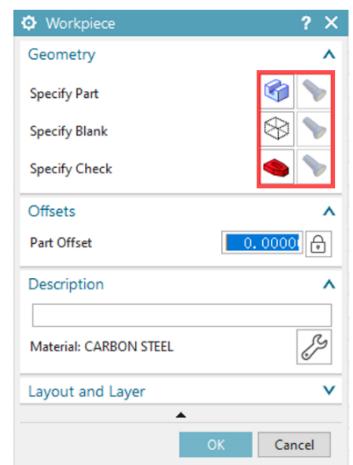




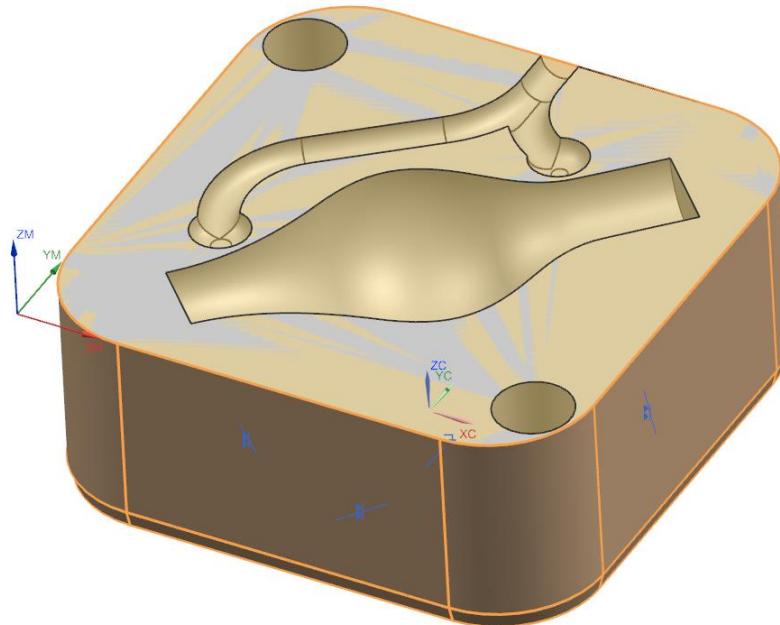
10. Select OK to exit the Mill Orient Dialogue.

#### Set the Workpiece Part and Blank Items

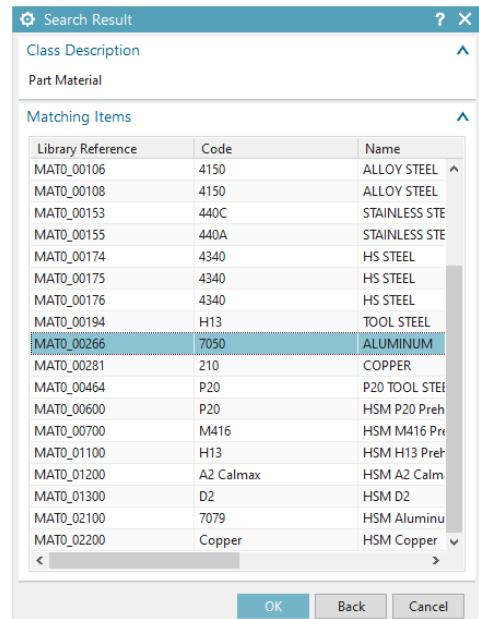
1. Right click the Workpiece item underneath the MCS\_MILL item in the Operation Navigator geometry view and click on edit to bring up the workpiece window.
2. Select the active button to the right of the 'Specify Part' item under the Geometry section. This will activate the Part Geometry window.
3. Select the model shown as the part. Ensure that this is the part with the model features, not the blank. If you have trouble selecting the part, use quick pick by holding your mouse over the model until the cursor changes to three dots, and then left click. The quick pick window will appear to allow you to choose the proper model.



4. Click on Specify Blank to assign the blank. Ensure that this is the model with no feature geometry, and use quick pick to allow selection of the proper model. You can also check to make sure you have selected the blank if you can see a highlighted region in the runner entrance.

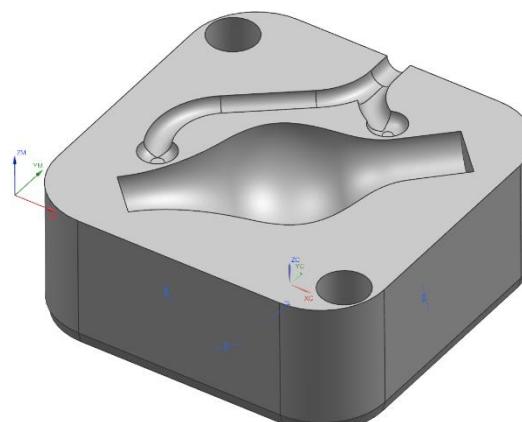
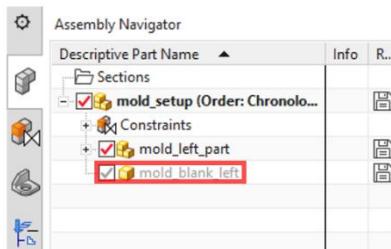


5. Set the part material in the Mill Geometry window Description section, and select Aluminum 7050 as the material of the manufacturing setup. Select OK once done.



6. Click OK in the Mill Geometry window to exit the workpiece editing dialogue.

7. Hide the blank part by going to the assembly navigator via the assembly tab on the far left of the NX window, and clicking the check box next to the mold blank part. This will make future selections of geometry simpler by eliminating the need to use quick pick every time you wish to select part geometry.



## Create a $\frac{1}{4}$ ball end mill tool

We will now create a tool to be used in multiple future operations, including the milling of injection runners, and the entire process of milling the main part body in the mold part. The  $\frac{1}{4}$  inch ball end mill is primarily chosen since it is the exact diameter of the channel used in the runners.

1. Select the 'Create Tool' item in the NX main toolbar. Select the following parameters for the tool to be created:

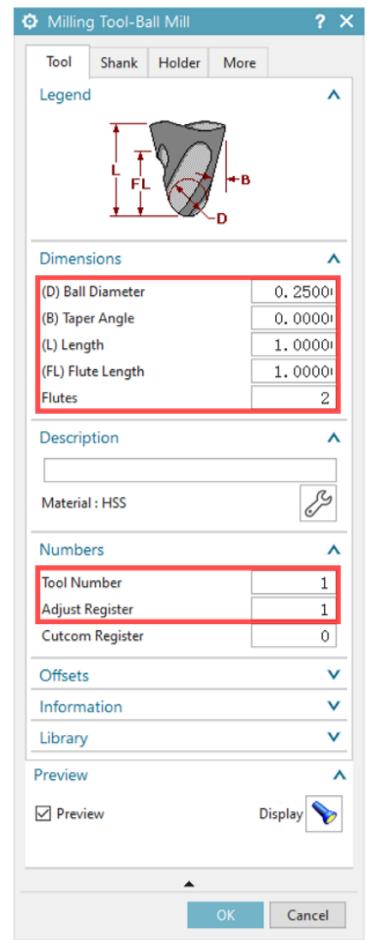
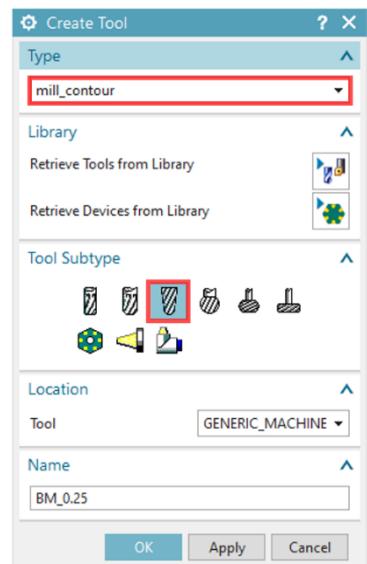
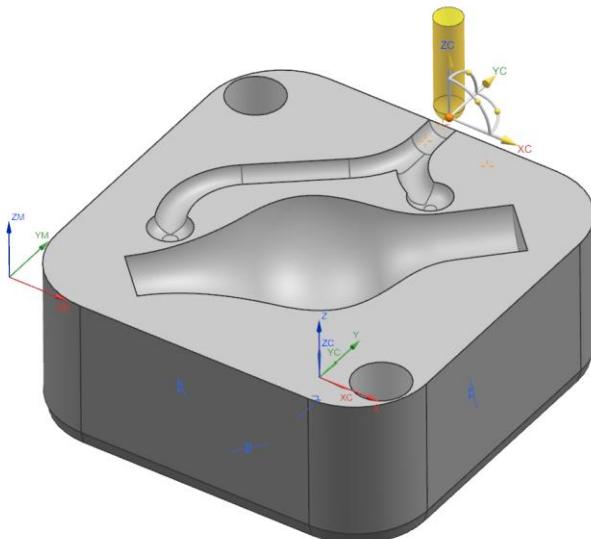
Type: mill\_planar  
Tool Subtype: Ball\_Mill  
Name: BM\_0.25

2. In the Milling Tool – Ball Mill Parameters window, enter the following parameters for the ball mill to be created:

Diameter: 0.25 in  
Length: 1.00 in  
Taper Angle: 0.00 in  
Flute Length: 1.00 in  
Flutes: 2  
Tool Number: 1  
Adjust Register: 1

We will not create a holder for this tool.

3. Select OK to create the tool.



## Create a Planar Profile Milling Operation.

We will now create a Planar Profile milling operation, which is a subtype of the Planar Mill type category. Planar milling is the most basic type of operation in NX, which operates based on 2D boundaries and depths, rather than the differences between a part and blank piece as in a contour milling operation.

You will begin to observe how planar milling operations, while more basic, can be used to target very specific sections or create unique methods of milling. This planar profile operation is one of the more basic and versatile uses of planar profile milling, which can often mill a feature in a significantly simpler method than a contour milling operation.

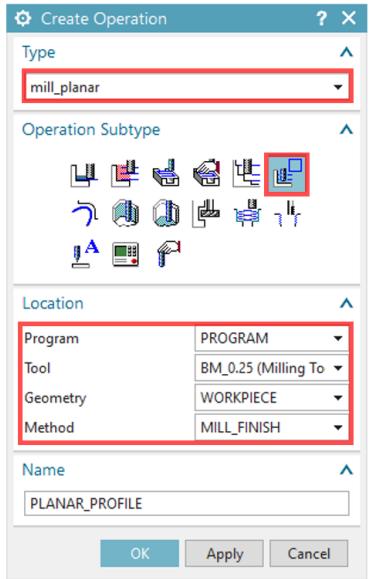
If time permits, attempt to mill out the runner feature in the following section using a cavity mill operation, and observe how unnecessarily complex the operation can become.

1. Select the 'Create Operation' item in the NX toolbar. Select the following parameters for the operation:

Type: mill\_planar  
Operation Subtype: Planar Profile  
Program: Program  
Tool: BM\_0.25  
Geometry: Workpiece  
Method: MILL\_FINISH

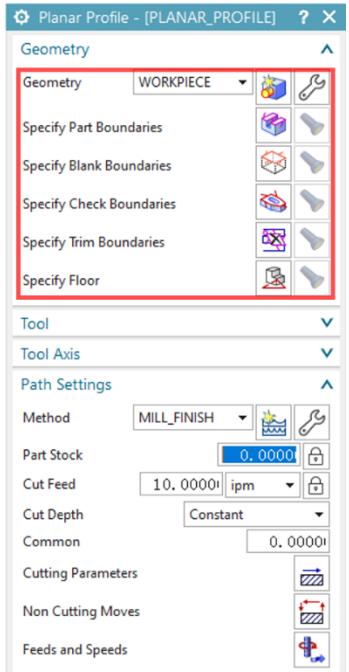
Notice how we are selecting a Mill Finishing operation, without first using a roughing operation. This methodology will become apparent in the following section.

2. Observe the Planar Profile dialogue window. Notice how in place of Part and Blank items in the Geometry section, there are now items designating boundaries instead of parts.

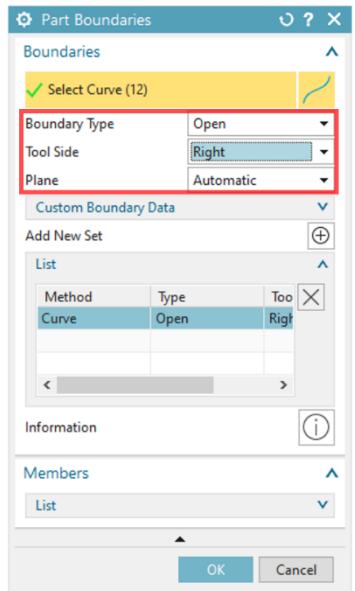


## Specify the Part Boundaries.

1. Select the button to the right of the 'Specify Part Boundaries' item. Here we will designate which section of the model we would like to mill.
2. When selecting a part boundary, there are a rules to consider:
  - a. NX will only mill below the plane of your selected boundaries. Thus, you must select the topmost boundary which defines the feature you would like to create.
  - b. The selection methods for boundaries are very particular. Pay close attention to the 'Mode', 'Material Side', and 'Type' items that we will review, as they will point NX to which areas you intend to mill.



3. The first window to appear is a 'Boundary Geometry' window. You can see how the first item, 'Mode' is defaulted to Face. We wish to designate our own custom boundary. Enable this by expanding the dropdown menu for 'Mode' and selecting 'Curves' and enter these parameters:



Type: Open

This designates the profile we will be creating will not be closer, and will have an open end. This will result in NX entering this feature through the open end, rather than entering from above the part.

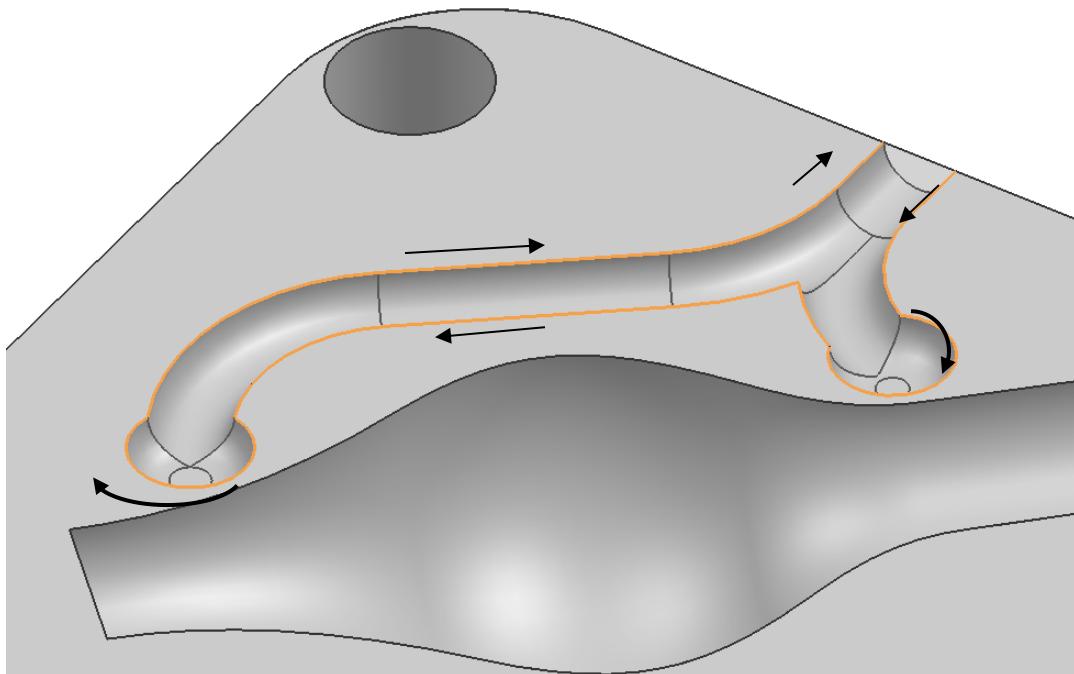
Plane: Automatic

This will assume the plane we want the profile to fall on is the same as the plane the existing geometry is on.

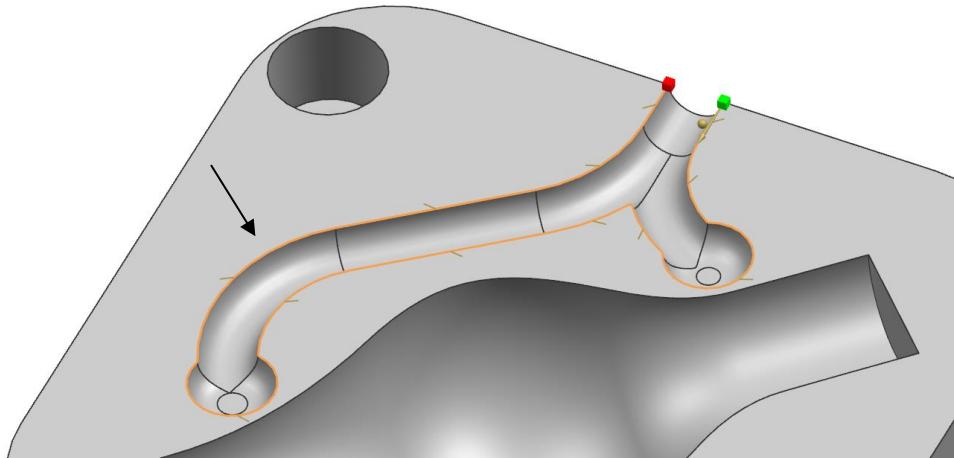
Tool Side: Right

This item designates which side the blank material will be, with respect to the direction we will be selecting the profile. This is important to remember, as this will dictate where you must start to select the profile.

4. Select the boundaries as shown below, in the direction shown. Be sure to select the edges directly next to the previous section. Selecting boundaries out of order will confuse NX. Start with the edge to the right of the runner entrance, and proceed around the runner feature till you arrive back at the runner entrance. Remember to leave the profile open.

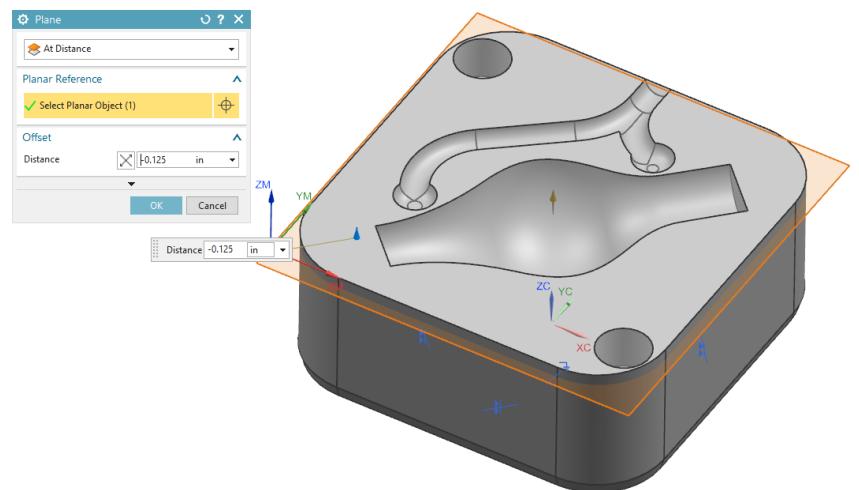
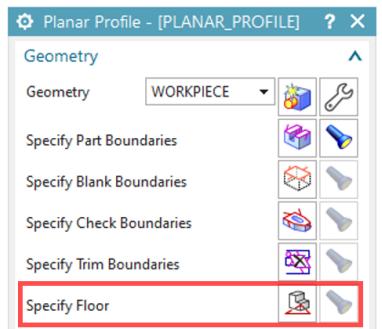


5. Select OK, and observe the lines extending outwards along each selected segment. This designates that the material to be left after machining is outside the profile, or to the left of the profile in the direction that we selected the profile.



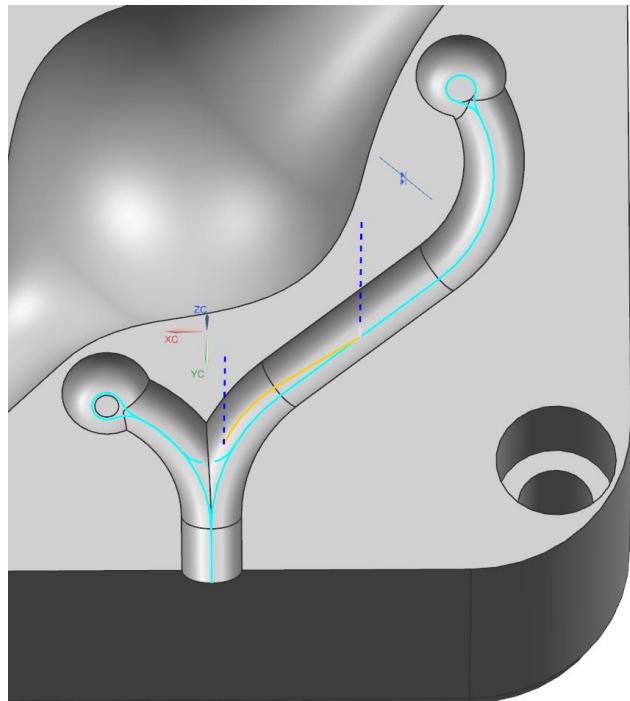
### Specify the Profile Depth.

1. We must now designate a depth for this profile. Recall, how in a planar profile operation, we are no longer operating with a part and blank piece. We have so far instructed NX which 2D profile to mill. We must now add the third dimension of depth.
2. Select the 'Specify Floor' active button. Note that for this planar operation, we do not need to designate a blank boundary. A planar milling operation can generate this path without a designated blank, since we have told NX to mill inside our profile. If the material to remove is outside, we must designate the outer limit, or the blank boundaries. The Workpiece geometry selection does not affect the geometry of the operation, but rather effects the verification of the operation.
3. A 'Plane' window now appears, similar to the clearance plane window we have previously used. However, now we will select the floor, or bottom limit the runner feature will be machined to. The channel for the runners has a semi-circle profile, of a circle of 0.25 inches in diameter. Thus, the depth of the channel is 0.125 inches, or one radius of the circle profile (Note, that this is identical to the tool diameter selection).
4. Select the top face of the model, and enter in an offset of -0.125 inches. This will place our floor plane at the bottom of the channel. Then select OK.

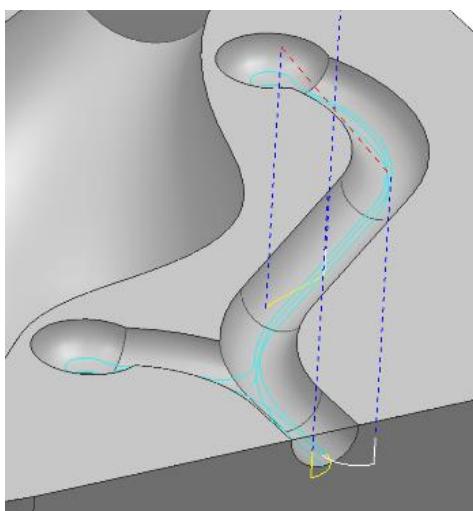
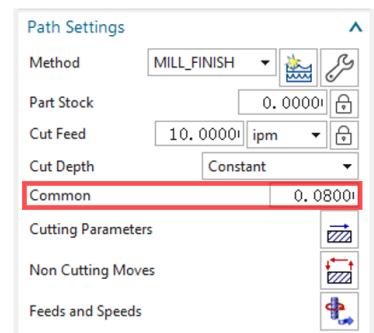
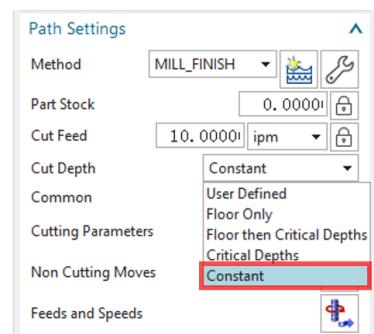


## Generate the Toolpath and modify the depth of cut.

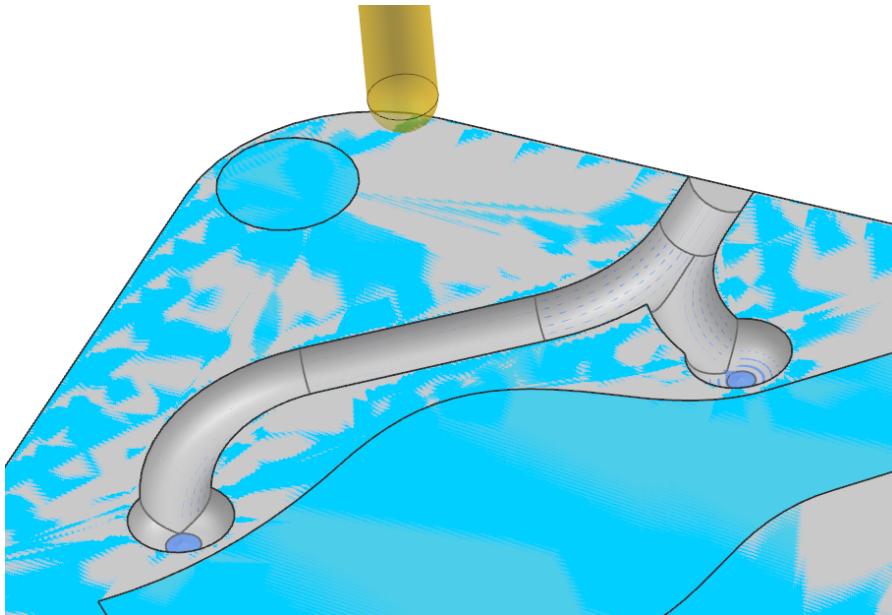
1. Create the toolpath by selecting Generate at the bottom of the Planar Profile Dialogue. There are various items to observe:
  - a. The path stays within the borders we have designated.
  - b. The path only hugs the floor plane, and does not take depths of cut. We still must alter this parameter. The use of a Mill\_Finish method also contributes to the final depth by not allowing any stock to remain along the floor.
  - c. Recall, that the tool we are using is a ball end mill. This will prove significant in the final shape of the feature. (Think what the final shape would be like with a flat endmill and this path).



2. As with all other types of operations, we must edit some parameters, such as depth of cut. In the Planar Profile dialogue, select the 'Cut Depth' dropdown menu under the Path Settings section, and observe the options. We will use a 'Constant' option.
3. Notice how the current number in the 'Common' item underneath Cut Depth is 0.0. This will cause the tool to default to the designated floor plane. Enter in a value of 0.08 inches, which is roughly 1/3 the diameter of the 0.25 inch tool.
4. Regenerate the path, and observe how the tool now takes multiple cuts to create the path.



5. Visualize the operation via the verification tool at the bottom of the Planar Profile dialogue. Use a 3D dynamic method to see how the channels will be cut. Observe the following:
- The pools at the end of each runner do not have flat ends, as in the model. This is due to the use of a ball end mill. For items like this, it is important to consider the implications of an incorrect model. This isn't part of the intended part to be manufactured, so is it fully necessary to have perfect geometry?
  - The first cut has two side by side paths. This is because the tool has not yet fully descended into the part, and since a section of the tool with a smaller diameter is in the part, NX has created a small stepover. You can observe this with the visualization if you set a slow replay speed.

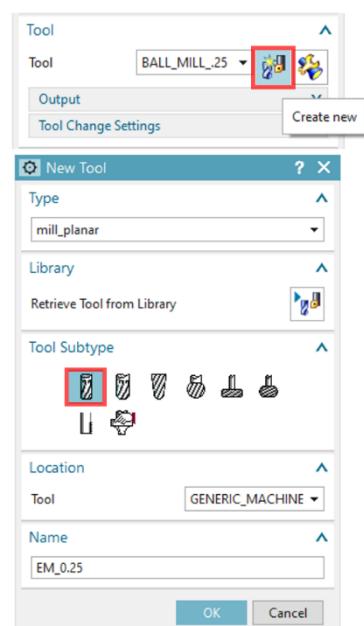


### Create a new tool to substitute into the Planar Profile Operation

As stated, the choice of tool in this operation as a direct effect on the final shape of the feature. We will now create a flat faced endmill to substitute into this operation, and observe the difference in final shape. This tool will also be saved and used for future operations.

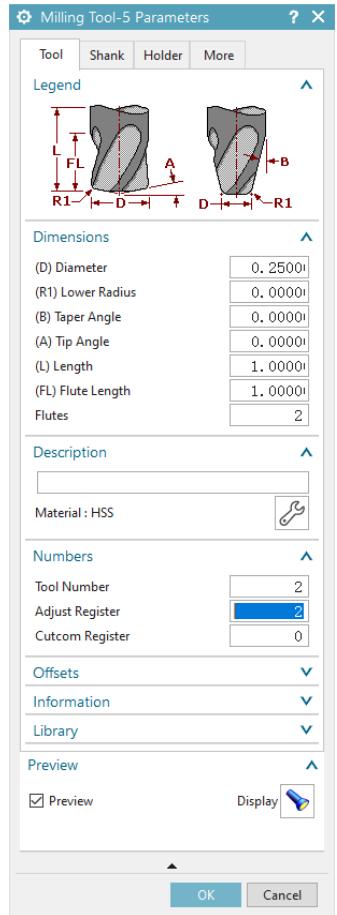
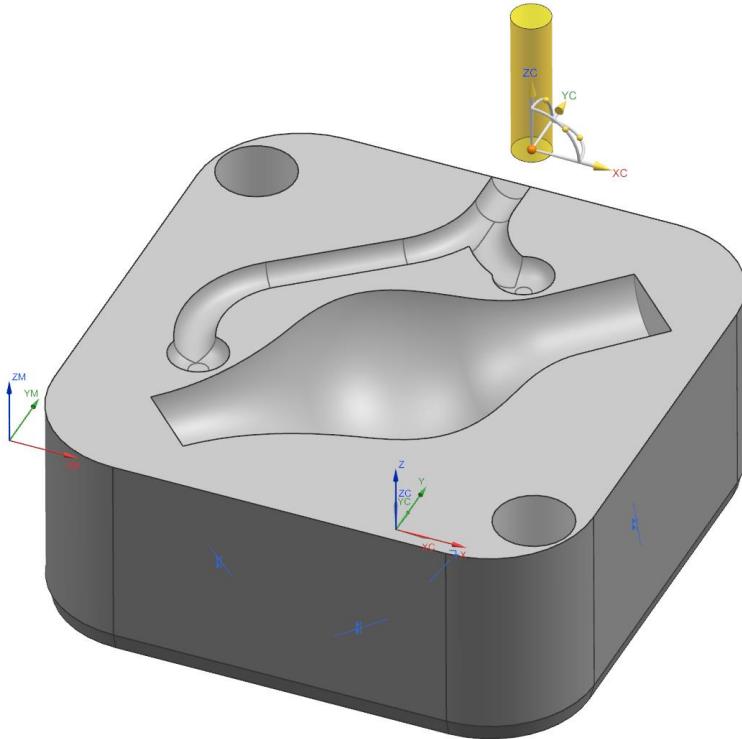
- Create a new tool via the new tool dialogue in the Planar Mill dialogue Tool section. Enter the following parameters for the tool to be created:

Type: mill\_planar  
 Tool Subtype: MILL  
 Name: EM\_0.25

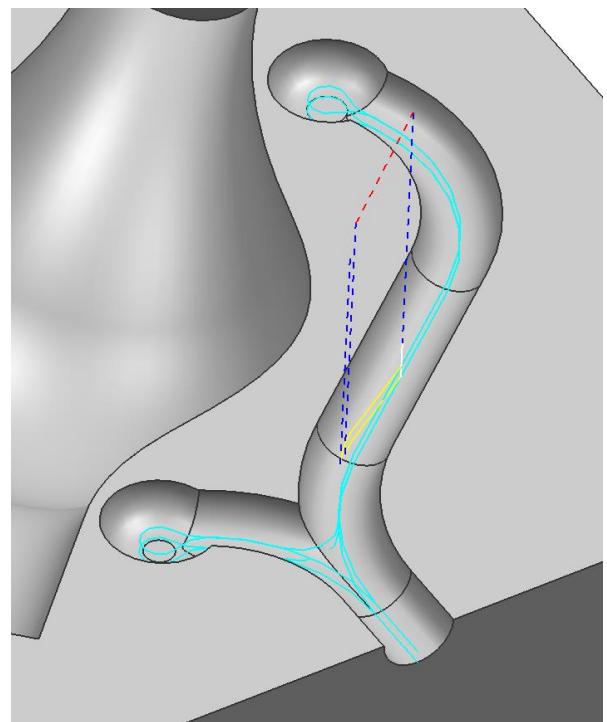


2. Enter the following parameters for the EM-0.25 tool:

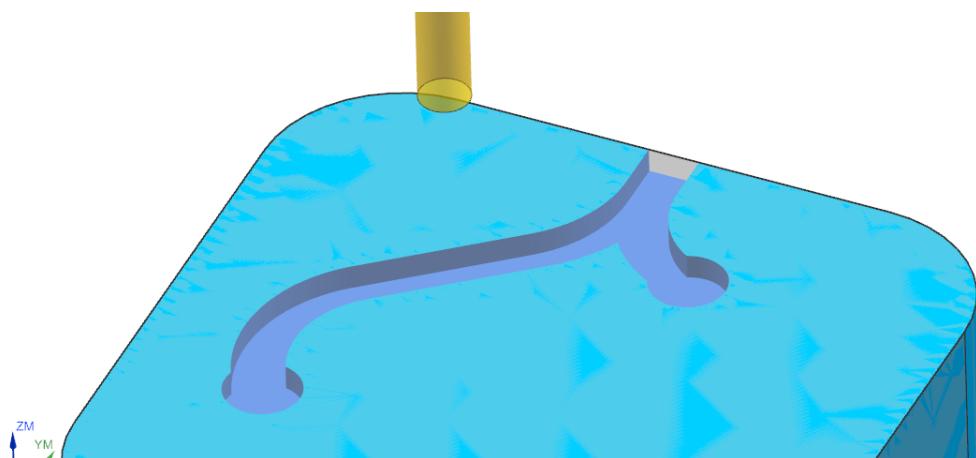
Diameter: 0.25 in  
 Lower Radius: 0 in  
 Length: 1.00 in  
 Flute Length: 1.00 in  
 Flutes: 2  
 Tool Number: 2  
 Adjust Register: 2



3. Click OK to create the tool. This tool will now be set to the default operation tool by NX, since the tool was created via the operation dialogue. Regenerate the toolpath. Nothing should appear different as compared to the ball mill toolpath. However, the final shape will reveal changes.

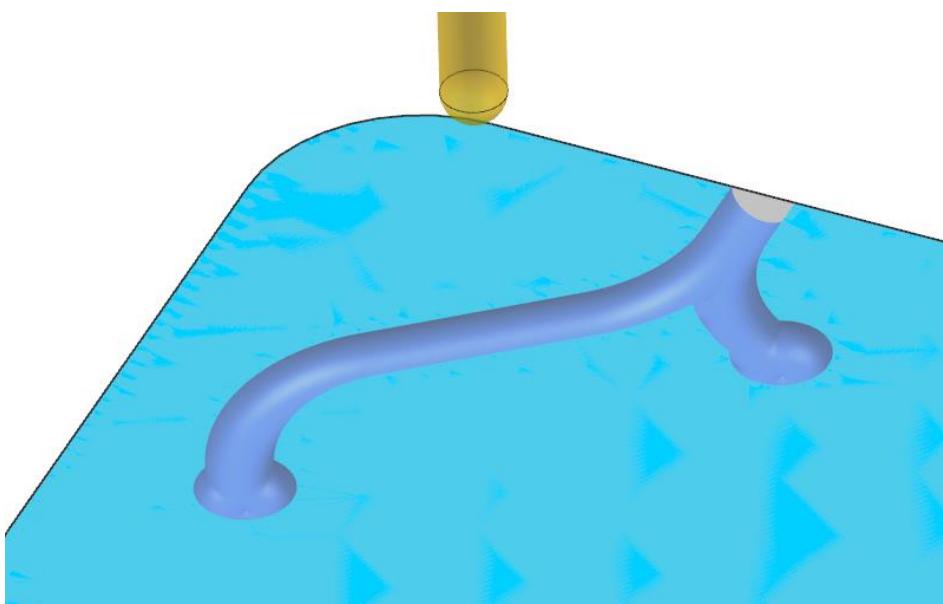
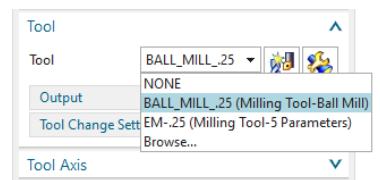


4. Verify the operation via the visualization environment, using the 3D dynamic method.



Observe how now the final profile has changed because of the flat endmill tool selection (hide mold\_left\_part under Assembly Navigator to see the result shown above). This is not how the part reflects this feature. Recall how in the geometry selection, we merely specified the outer border that the tool will follow, and nothing more. There was no data input to specify the round geometry of the channel, only the final depth of the channel. The final shape is now governed not only by the path and geometry selections, but by the tool selection. The tip of the tool when a ball endmill is used still hits the bottom geometry, but since we selected a tool of the same diameter as the channel, we create a feature identical to the one intended in the model.

5. Replace the newly created EM-0.25 with the original Ball\_Mill\_.25. Regenerate the path, and verify the operation to ensure you have the original result.



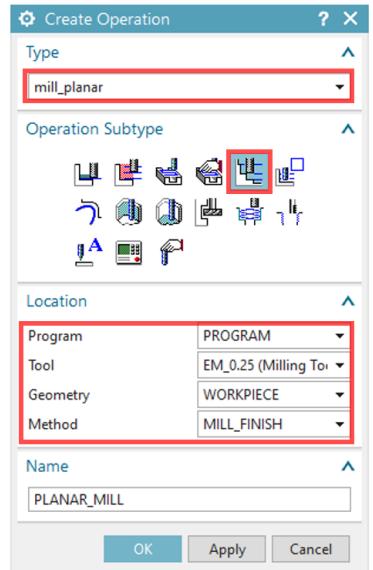
6. Select OK to create the operation and exit the Planar Profile Dialogue.

## Create a Planar Milling operation to create the bolt countersinks.

We can now use the flat faced end mill created in the previous planar profile operation to machine the countersink features for the bolts at the corners of the piece. This operation could also be accomplished using a cavity mill operation, However this example is to show the simple versatility of the planar milling operations.

1. Create a new operation via the ‘Create Operation’ item on the NX toolbar. Enter the following parameters for the operation to be created:

Type:	mill_planar
Operation Subtype:	Planar Mill
Program:	Program
Tool:	EM_0.25
Geometry:	Workpiece
Method:	Mill_Finish

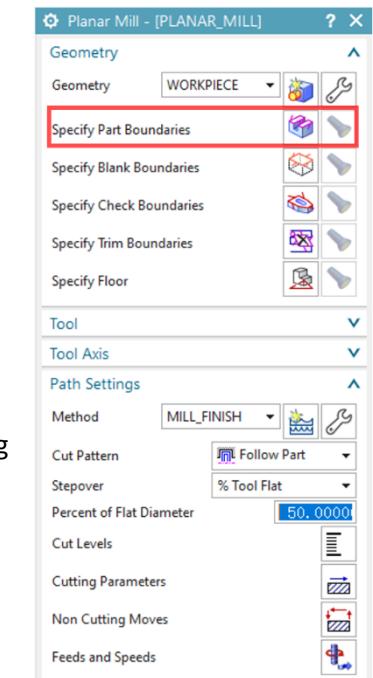


2. Observe the Planar Mill dialogue window. It is similar to the Planar Profile operation window we previously used. The differences will soon become apparent.

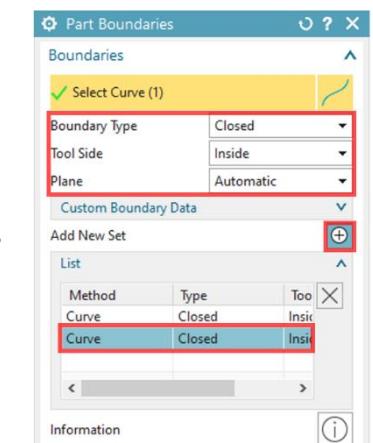
### Specify the Part Boundaries.

3. Select the ‘Specify Part Boundaries’ Button to bring up the Boundary Geometry window.
4. In the dropdown menu to specify the ‘Mode’, again select ‘Curves’.
5. The selection of parameters for the boundary definition is again crucial to creating a successful operation. We intend to select the circular edges defining the profile of the countersink, so we should select parameters accordingly. Enter the following parameters:

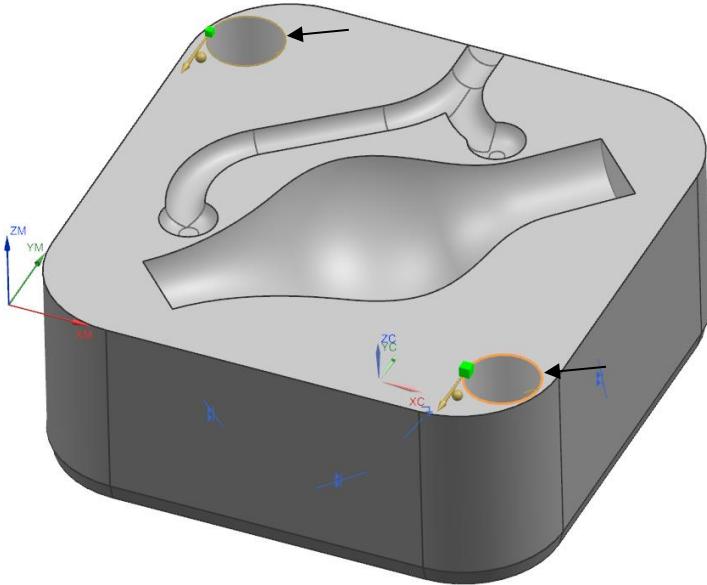
Type:	Closed
Plane:	Automatic
Tool Side:	Inside



This will tell NX we have a closed profile for the circles, and the blank is outside the selected profiles.



6. Since we will be creating two cutting regions, there is a specific sequence of items that must be selected to instruct NX to recognize both regions equally. First, select one of the edges of the countersink, then click on the “Add New Set” button and select the edge of the second countersink.

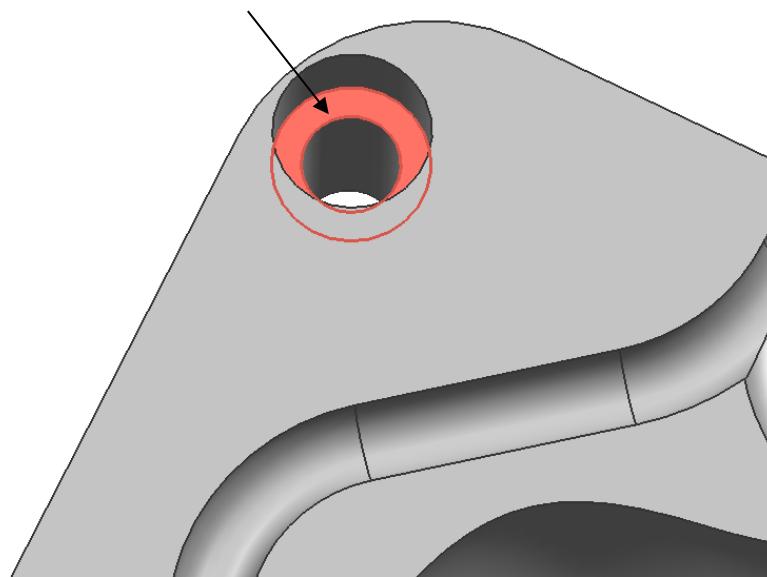
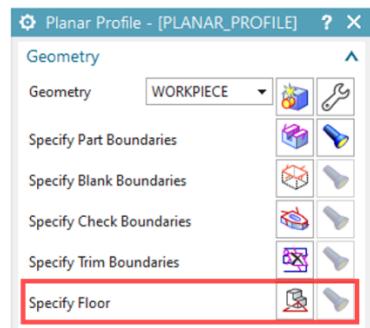


7. Select OK to exit back to the Planar Mill dialogue.

### Specify the Profile Depth.

As with the previous Planar Profile operation, we must specify the depth of the feature. This is again a required item for all planar milling operations. In this case, we have the benefit of an existing plane which we can use to define our floor, without having to indirectly set the floor by using an offset.

1. Select the active button next to the 'Specify Floor' item in the Planar Mill dialogue. This will activate a 'Plane' window.
2. Fortunately, we have the advantage of model geometry which can easily define our profile depth. Select the base of the countersink as shown for the profile floor.

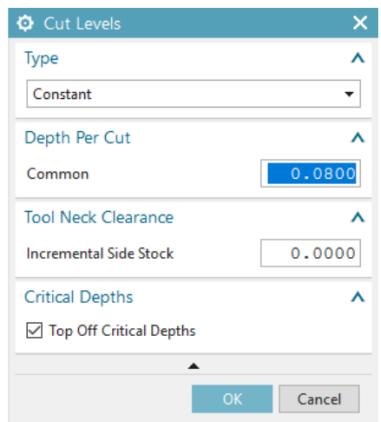
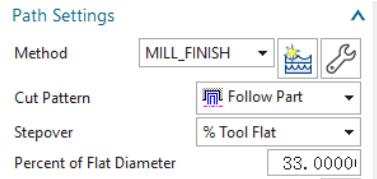


3. Select OK to exit back to the Planar Mill dialogue.

## Edit the Cut Pattern, Stepover and Cut Levels parameters.

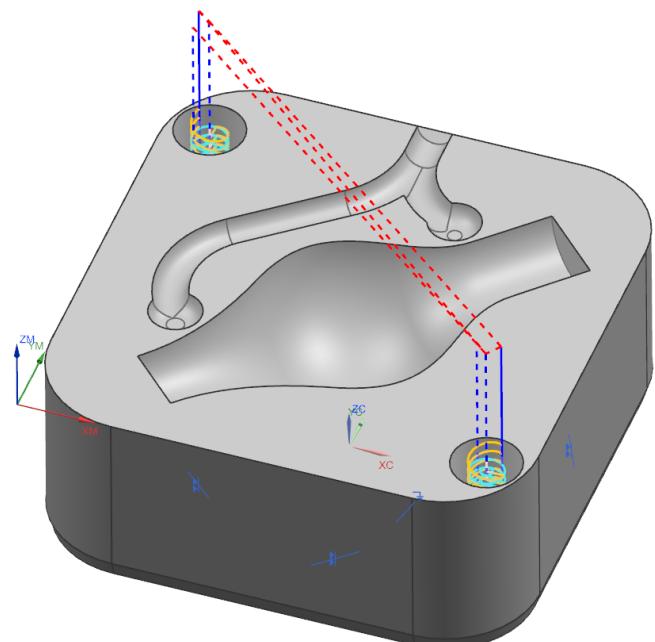
Similar to cavity milling operations, all of the typical cut pattern, stepover, and cut level values still must be set. If not addressed, items such as the cut levels will default to take one cut along the floor of the profile, which can easily break the tool.

1. Select the 'Follow Part' method from the Cut Pattern item in the Path Settings section of the Planar Mill dialogue.
2. Select the dropdown menu next to the 'Stepover' item in the Path Settings section, and ensure that it reads '% Tool Flat'.
3. Change the Percent of Flat Diameter value beneath the dropdown menu to 33%.
4. Select the active button next to the Cut Levels item in the Path Settings section. This will activate the 'Cut Levels' window.
5. We wish to select a constant value for the depths of cut. Expand the 'Type' dropdown menu and select 'Constant'.
6. Enter 0.080 inches for the Maximum value, which is approximately 1/3 the diameter of the  $\frac{1}{4}$  inch diameter endmill.
7. Select OK to exit the Depth of Cut Parameters window.



## Generate the toolpath, and verify the operation.

1. Generate the toolpath via the generate button at the bottom of the Planar Mill dialogue window. Observe the toolpath, and how it spirals along the cut profile. If you do not see complete circular cut patterns in each of the sections, you have not selected the part profiles correctly. Review the profile selection process if needed.



- Verify the operation through the Visualization environment.

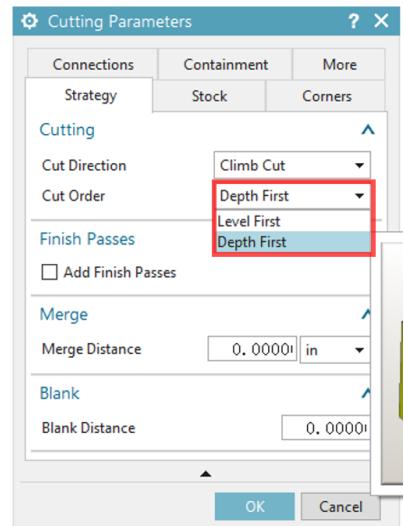


- Select OK to exit the visualization environment and return to the Planar Mill dialogue.

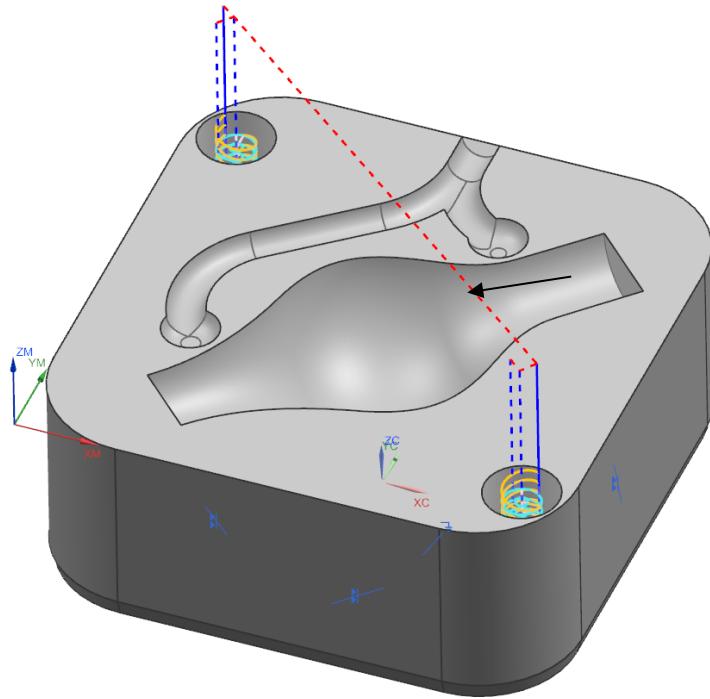
### Edit the Cut Order

You may notice during visualization, that the tool is jumping back and forward between the features, since it is cutting only one level at a time. This seems to be an inefficient use of time, and we can instruct NX to complete one feature before starting on another. This is called instructing NX to machine in a 'Level First', or 'Depth First' order.

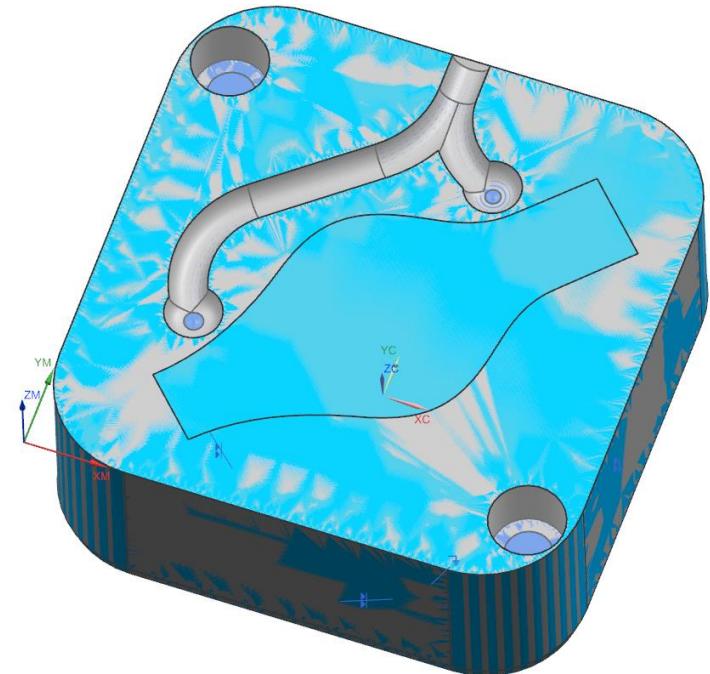
- In the Path Settings section of the Planar Mill dialogue, select the active button next to the Cutting Parameters item. This will activate the Cutting Parameters dialogue box.
- Under the first 'Strategy' tab, this is where we can change the overall cutting method used by the operation. Under the 'Cutting' section, there is a 'Cut Order' item, which has the 'Level First' option selected. This is what is causing NX to cut one depth level at a time, across all features, rather than complete one feature before moving on to another. Select 'Depth First' from this dialogue, and select OK to exit back to the Planar Mill dialogue.



3. Regenerate the toolpath. You can now see how there is only one single red dashed line between the two regions, indicating the tool will only traverse between the two features a single time.



4. Verify the operation, and observe how the tool cuts one feature completely, before moving to another. Notice the final result does not change, only the cutting method.



5. Select OK to exit the visualization environment, and select OK in the Planar Mill dialogue to create the operation.

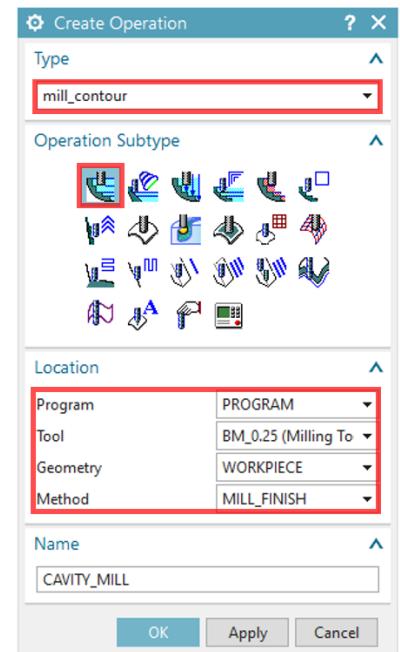
## Create a Contour Milling Operation, and specify the Cut Area.

We will now focus on machining the main mold feature. This feature is a complex surface, which will require the use of advanced features to achieve a proper result in the piece. For this feature, it should be obvious to the student that a planar milling operation cannot achieve this type of feature, and contour milling operations must be used. We will also be required to use a surface milling operation to finalize the feature.

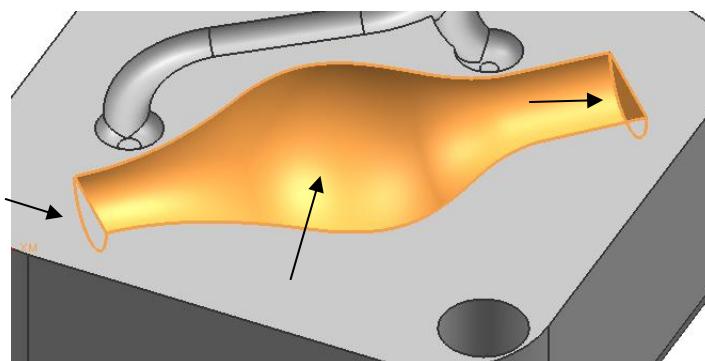
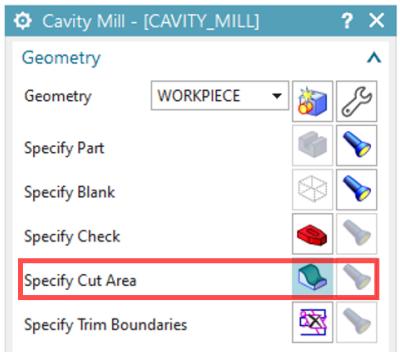
We will be creating an initial contour milling operation to remove the majority of the material. However, for this operation, we will not use the MILL\_ROUGH method, but rather use the MILL\_FINISH method to remove material right down to the intended part surface. The reason for skipping directly to a finishing operation, is that this actual operation will not create the final desired surface. The following surface milling operation will function as the finishing operation, and that operation will benefit the removal of as much material as possible from the feature before being executed. Thus, using a MILL\_FINISH option will allow the surface milling operation to function more effectively, since it will be required to remove less material.

1. Create a new operation with the following parameters:

Type: mill\_contour  
Operation Subtype: Cavity Mill  
Program: Program  
Tool: BM\_0.25  
Geometry: Workpiece  
Method: Mill Finish



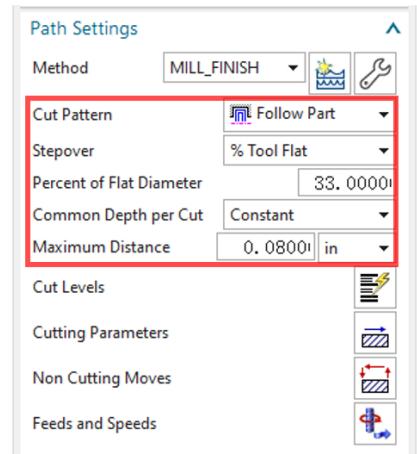
2. A cavity milling operation, under normal circumstances, will look for any differences between the part and blank, and attempt to remove as much material as possible. However, as we have previously reviewed, a cavity milling operation can be narrowed down to focus on a specific area. In this case, we will instruct the cavity milling operation to only focus on the main mold feature through the use of a specified Cut Area.
3. In the Geometry section of the Cavity Mill dialogue, select the active button next to the 'Specify Cut Area' item. This will activate the 'Cut Area' dialogue.
4. Select the three faces of the main mold feature, including the complex surface, and the two end cap sections. Select OK in the cut area box once the three faces have been selected.



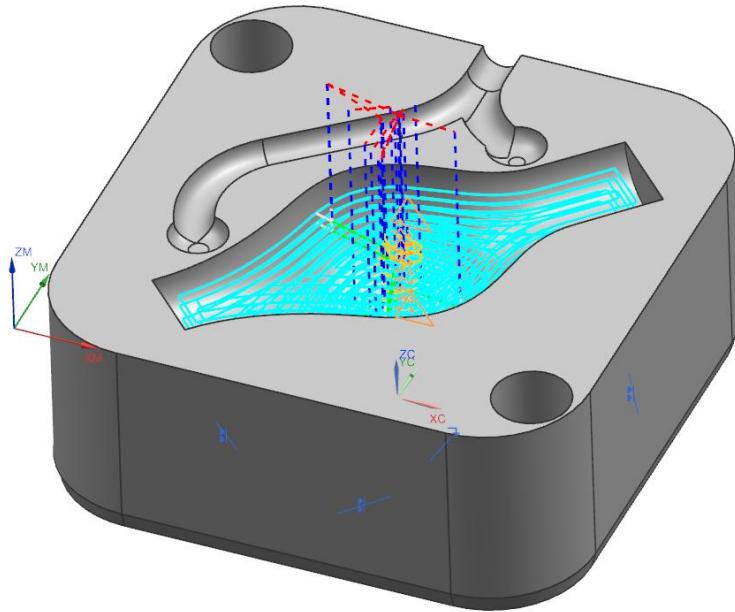
## Edit the Cut Pattern, Stepover, and Depth of Cut.

1. Change the following items under the Path Settings in the Cavity Mill dialogue.

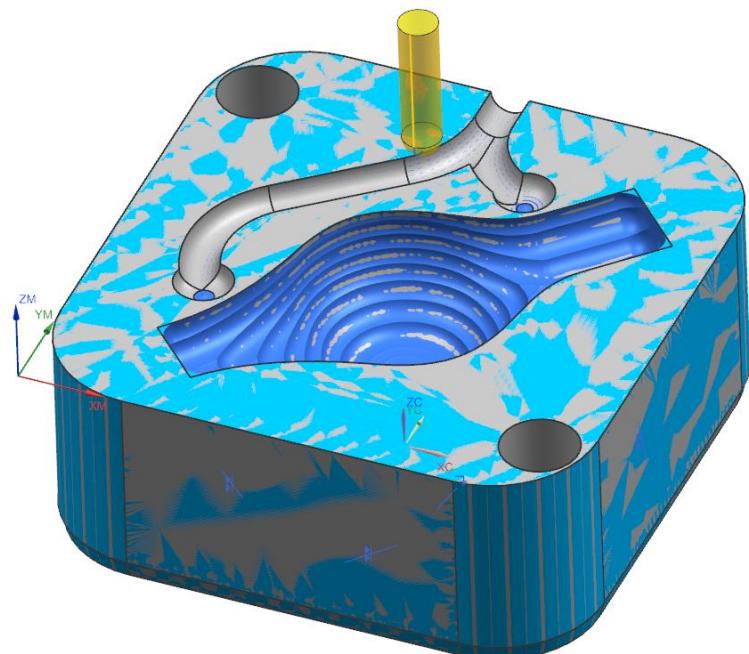
Cut pattern: Follow Part  
Stepover: % Tool Flat  
Percent of Flat Diameter: 33.0%  
Common Depth of Cut: Constant  
Maximum Distance: 0.080 inches



2. Generate the toolpath, and verify in the visualization environment.



3. If we observe into the main cavity, since a Finishing method has been used, the tool has attempted to remove as much material as possible. However, the finish along the surface is far from perfect, due to the large depth of cut taken. We will need to address this with a finishing operation, where we will use a surface milling operation.



4. Select OK to exit the visualization environment, and select OK in the cavity mill dialogue to create the operation.

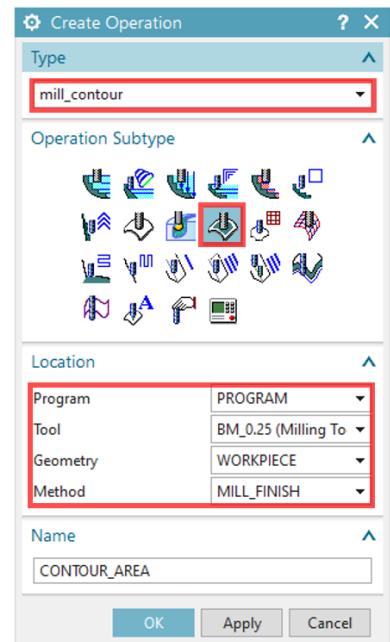
## Create a Surface Milling Operation, and select the cut area.

A surface milling operation is one of the more advanced features in NX manufacturing, which is used in the machining of complex surfaces. In these types of operations, we will focus heavily on the type of Drive Method, which is the surface milling equivalent of Cut Pattern. We will also heavily rely on the visualization tool to see how the choice of drive parameters affects the final surface finish of the part.

Note: This is one of the most complex areas of NX to learn, as well as to teach. You will most likely spend the greatest amount of time adjusting parameters for surface milling operations to achieve your desired results.

1. Create a new operation with the following parameters:

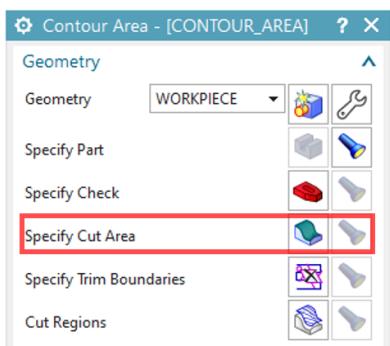
Type: mill\_contour  
Operation Subtype: Contour\_Area  
Program: Program  
Tool: BM\_0.25  
Geometry: Workpiece  
Method: Mill Finish



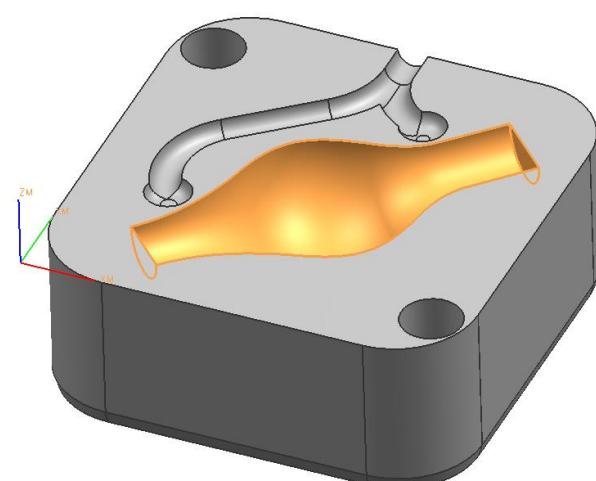
The other types of surface milling operations, such as ‘Fixed Contour’ and ‘Contour Surface Area’, can also be selected. You can interchange the subtype here by altering the drive method, which you will see next.

2. Observe the Contour Area dialogue. There is a new section entitled ‘Drive Method’, where the option has defaulted to Area Milling. The other subtypes of surface milling will change this default to the other options, so the proper selection of an area milling operation is not critical. It can be changed in this dialogue as well.

3. We must first select the Cut Area for the area milling operation to function. This is a required geometry input for surface milling operations. Under the Geometry Section of the Contour Area dialogue, select the active button next to Cut Area. This will bring up the Cut Area window.



4. Select the same three faces as shown, the same three faces as previously selected for the cavity milling operation. Select OK once done to exit back to the Contour Area dialogue.



## Edit the Area Milling Drive Method, and observe various surface finish results.

1. Select the wrench button in the Drive Method section next to the 'Area Milling' option in the dropdown menu. This will activate the Area Milling Drive Method window.

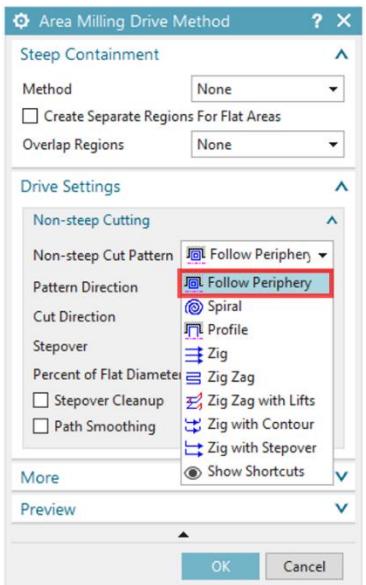
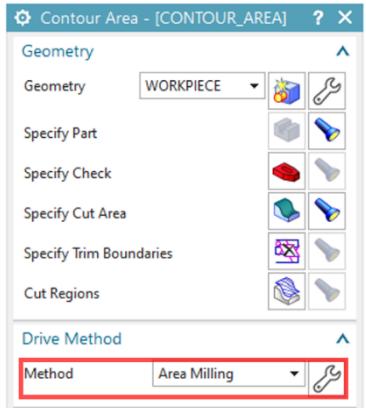
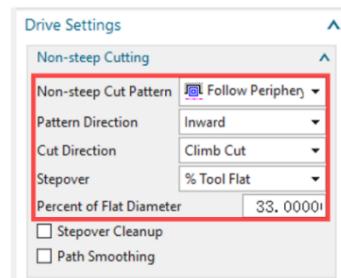
2. You can observe how many of the parameters you would find in a normal contour milling operation now exist here, such as cut pattern and stepover.

3. We will first edit the cut pattern. Expand the dropdown menu under the Drive Settings section, and select 'Follow Periphery'.

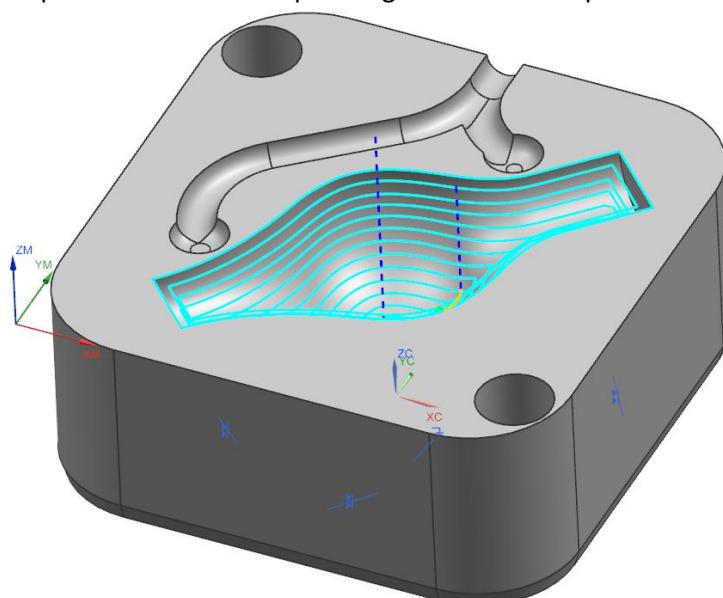
4. The Pattern direction and Cut direction should read Inward, and Conventional cut, respectively.

5. We will now focus primarily on the stepover parameters. This is what will mainly dictate the resulting surface of our part. For the initial attempt, set the following parameters:

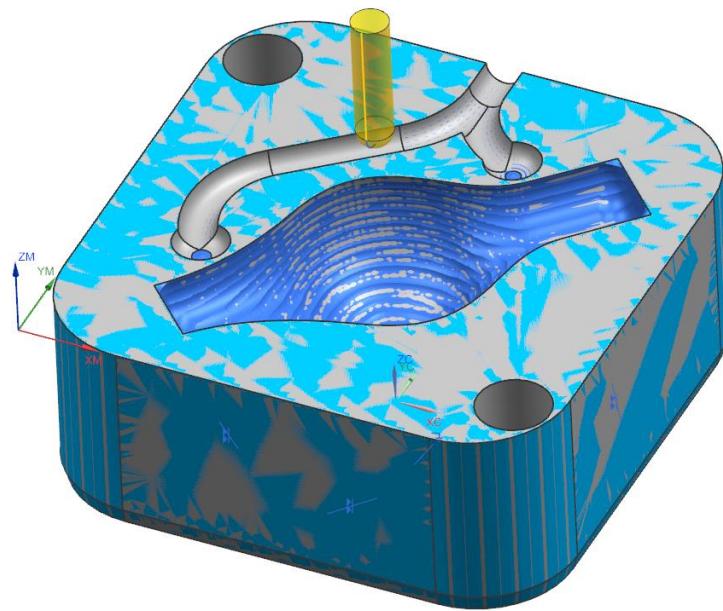
Stepover: Follow Periphery  
 Percent of Flat Diameter: 33%



6. Select OK, and exit back to the Contour Area dialogue, and generate the toolpath. You should immediately be able to recognize that this toolpath is spaced far too much apart to generate an adequate surface finish.

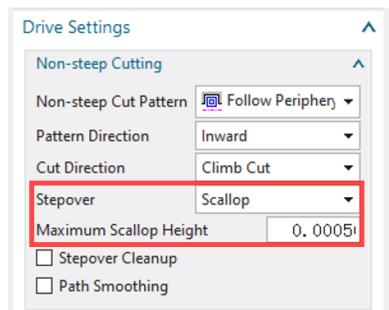


7. Verify the toolpath using 3D dynamic mode. You can see how the surface has only marginally improved. Select OK to exit the visualization environment.



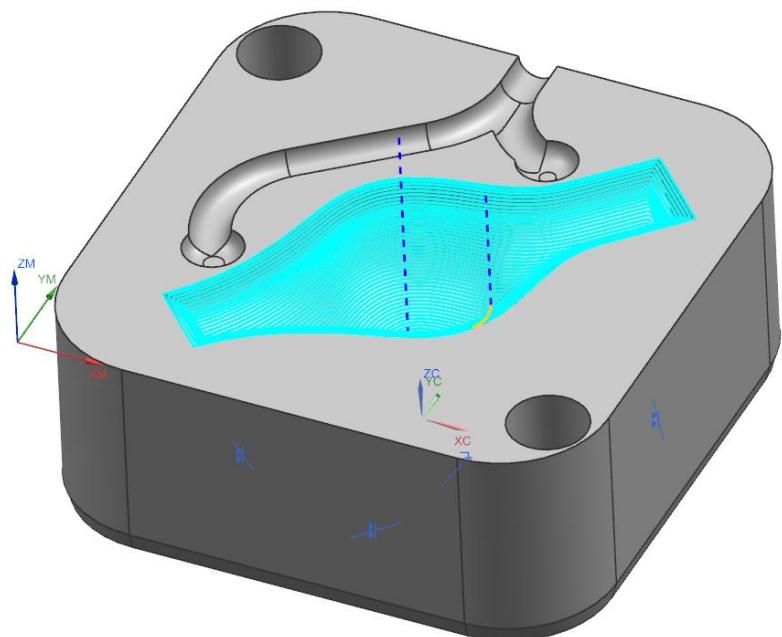
8. Re-enter the Area Milling Drive Method dialogue through the Drive Method section of the Contour Area dialogue. Select the following parameters for stepover options:

Stepover: Scallop  
Scallop Height: 0.0005 inches

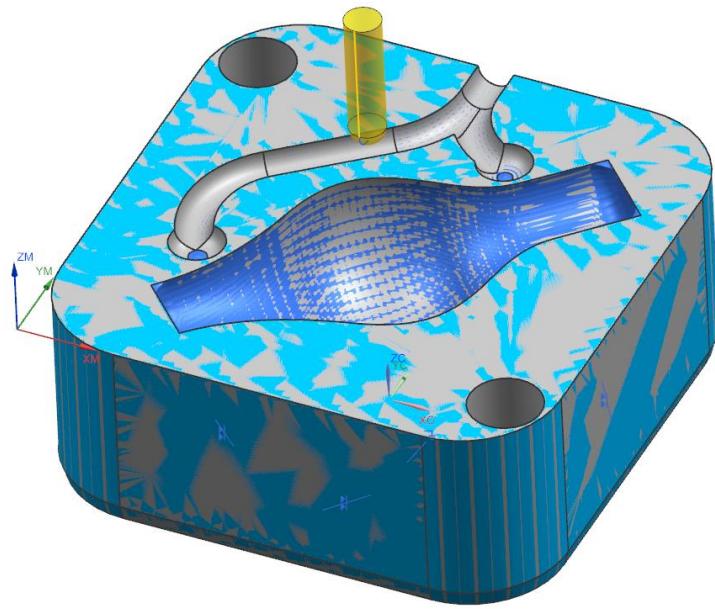


Scallop height is the height of the sections left behind when passing a ball endmill over a relatively flatter surface. This is what we have been trying to eliminate from the start, so NX has integrated an option to allow us to specify how large we want these scallops to be. Unfortunately, a perfect surface can never be completely achieved using just a ball endmill, but we can get relatively close by selecting a small height for our scallops.

9. Select OK to exit the Area Milling Drive Method window, and regenerate the toolpath. We can now observe how the distance between passes has been reduced, so the surface finish should improve.



10. Verify the part using the 3D dynamic visualization method. The surface has significantly improved, however there are still large un-machined regions on the upper regions of the desired shape.



11. Exit the visualization environment, and select OK to accept and create the Contour Area operation.

## Spot Drilling.

The standard procedure to drill a hole is to spot drill the proper position first, to prevent the drill from ‘walking’, or bending out of position when first contacting the part. Thus, we must use two operations when drilling one or multiple holes. With a CNC controlled machine, without a spot drilled hole, the drill will almost always break since the machine is powerful enough to cause the drill to bend. Caution should always be taken to spot drill a hole before drilling.

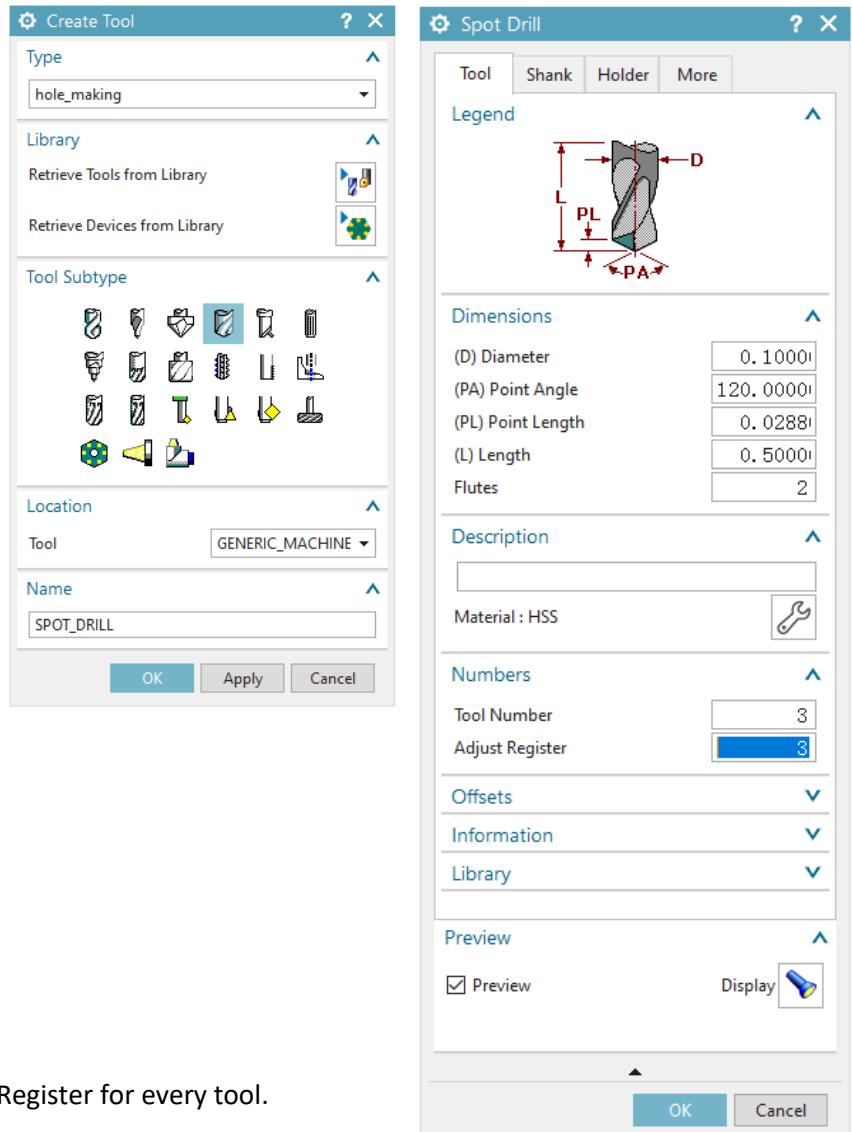
### Create the spot drilling tool.

1. In the NX toolbar, select the ‘Create Tool’ button to activate the new tool window. Enter the following parameters for the tool to be created:

Type:	hole_making
Tool Subtype:	SPOT_DRILL
Tool:	Generic_Machine
Name:	SPOT_DRILL

2. Enter in the following parameters for the spot drilling tool.

Diameter:	0.1 in
Length:	0.5 in
Point Angle:	120 deg
Flute Length:	0.5 in
Pitch:	0
Flutes:	2
Tool Number:	3
Adjust Register:	3



Be sure that the Tool Number matches the Adjust Register for every tool.

3. Select OK to create the tool.

## Create a spot drilling operation.

In NX 11, more information is automatically brought in from the part geometry. We must specify geometry, and then specify calculated depths.

1. Create a new operation with the following parameters:

Type: hole\_making  
Operation Subtype: Spot Drilling  
Program: Program  
Tool: SPOT\_DRILL (Spot Drill)  
Geometry: Workpiece  
Method: Drill\_Method

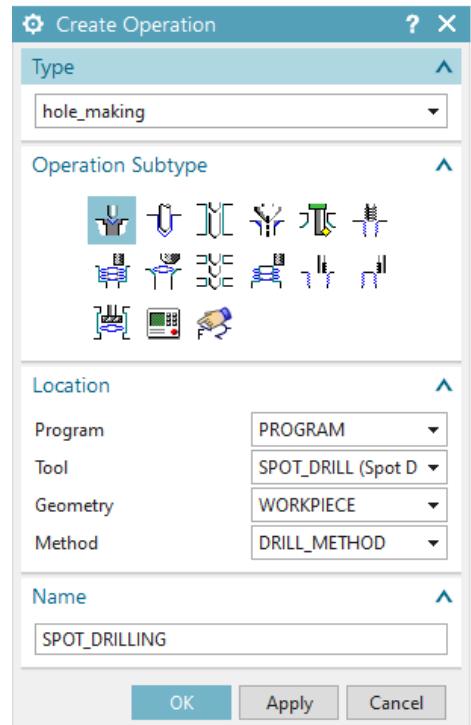
2. Observe the Spot Drilling dialogue window. We need only specify the holes and part surface from feature geometry.

## Specify the hole locations.

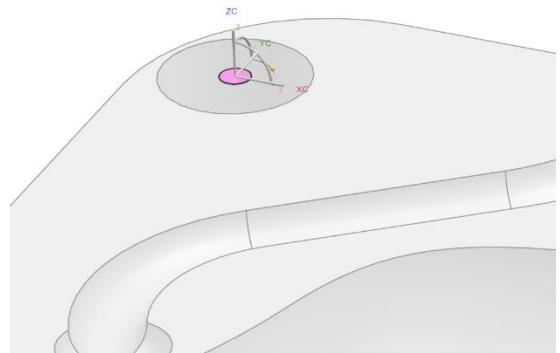
1. Select the active button next to 'Specify Feature Geometry' in the geometry section. This will bring up a window that allows us to specify where the holes will be that we would like to spot drill.

2. We need to tell NX where the holes are that we would like to spot drill. To do this, we will select the outer edge at the top of each hole. In the Centered Hole section, click the rightmost button next to 'Select Object.'

4. Select the edge at the top of the first hole to be spot drilled. If you are having trouble clicking the edge, hover the mouse over it and then use the QuickPick feature when three squares appear next to your cursor. With the edge selected, coordinate axes should appear at the center point of the top of the hole.

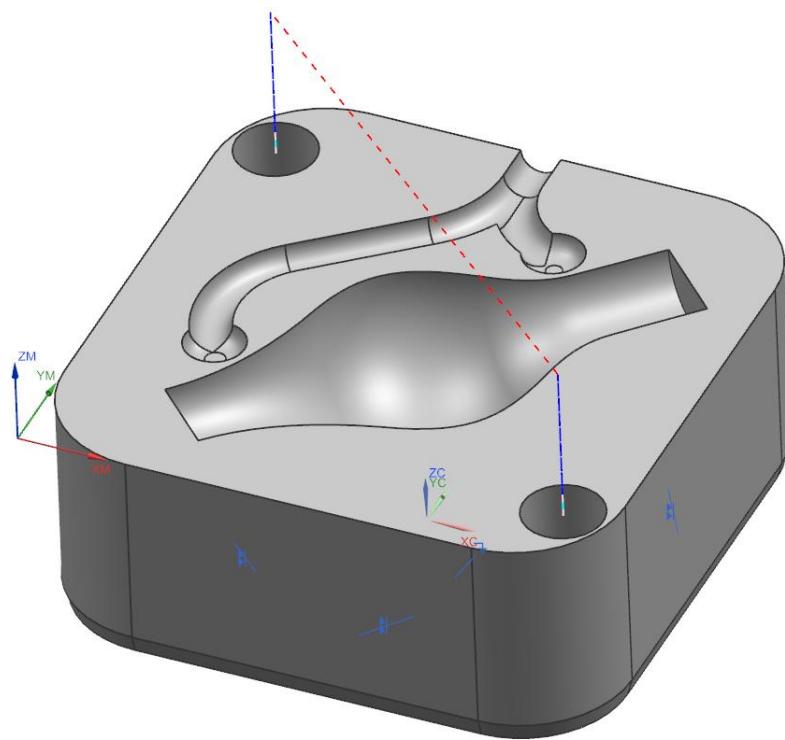


4. Notice that the hole appears as item 1 in the list further down in the Centered Hole section. We will now tell NX how deep we want the spot drill to go. Click the green lock next to the Depth field and select ‘User Defined.’ Enter 0.03 inches.



5. Repeat this procedure for the second hole on the other side of the mold. Be sure to click the outer edge at the top of the hole. This hole should appear as item 2 in the list. The depths of the spot drilling operations can also be seen in this list, along with other information. Note that because we have selected the upper edges for each hole, NX automatically recognizes the part surface on which we are drilling.

6. Select OK at the bottom of the Feature Geometry window to return to the Spot Drilling dialogue window. Generate the toolpath:



You may notice here how we are drilling into previously removed material. We could instruct the operation to drill to a lower depth, which would be just as effective. However, for purposes of teaching other features of NX CAM, leave this operation as it exists for the time being.

7. You may verify the path, but it will not display any additional features because of previously stated reasons. Select OK to exit the spot drilling dialogue and create the operation.

## Drilling operation

Since the Specify Feature Geometry now allows more information (such as the hole diameter) to be taken from the selected part geometry, we will first create the drilling tool to ensure the correct hole diameter.

### Create the drilling tool

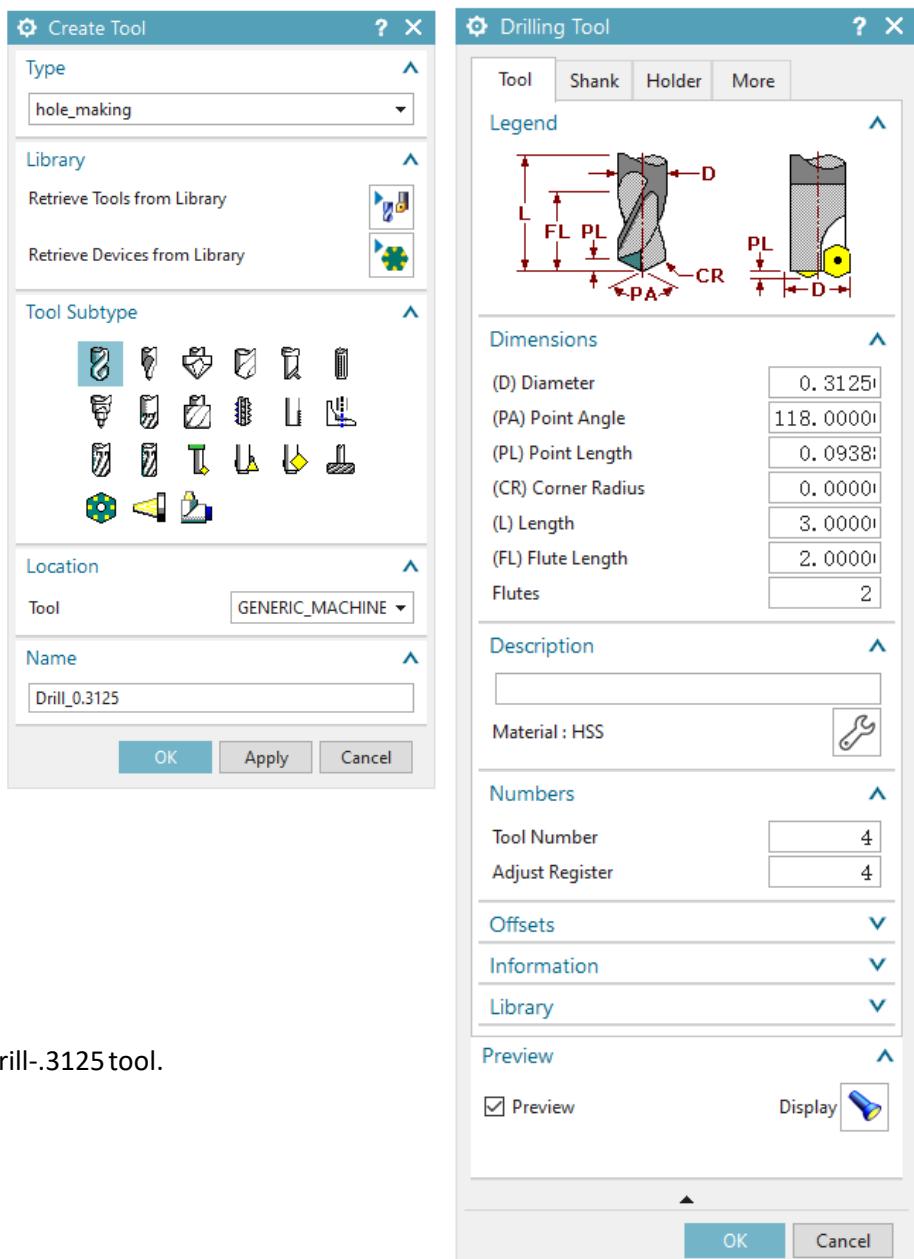
1. Expand the tool section of the Drill dialogue, and select the 'create new tool' button to activate the New Tool window. Enter the following parameters for the tool to be created:

Type: hole\_making

Tool Subtype: drilling\_tool

Tool: Generic\_Machine

Name: Drill\_0.3125



2. Enter the following parameters for the Drill-.3125 tool.

Diameter: 0.3125 in

Length: 3.00 in

Point Angle: 118 deg

Flute Length: 2.00 in

Flutes: 2

Tool Number: 4

Adjust Register: 4

Make sure that the tool number matches the adjust register for each tool. Select OK to create the tool, and return to the Drill dialogue.

## Create the drilling operation

We now need to add the actual drilling operation, which will create the hole. We will continue with assuming we will begin drilling from the top of the part.

1. Create a new operation with the following parameters:

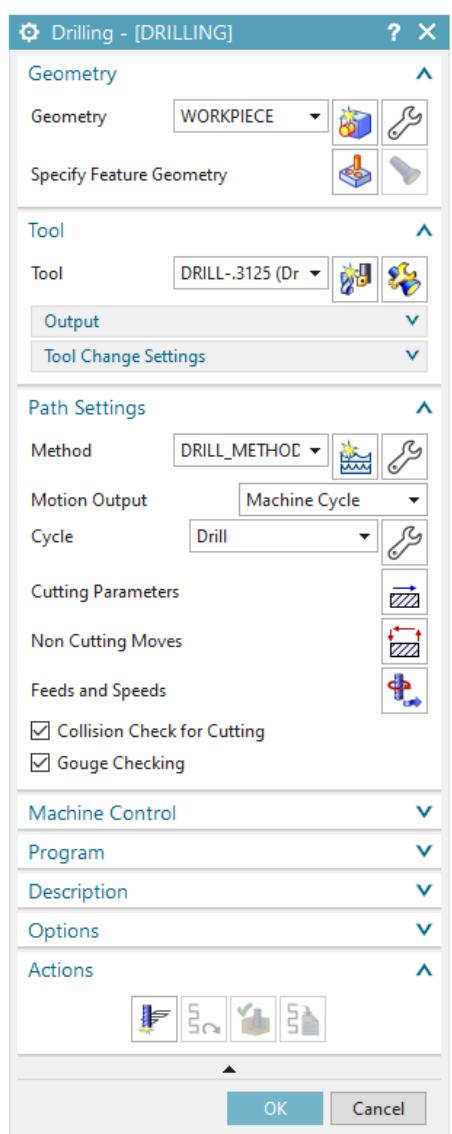
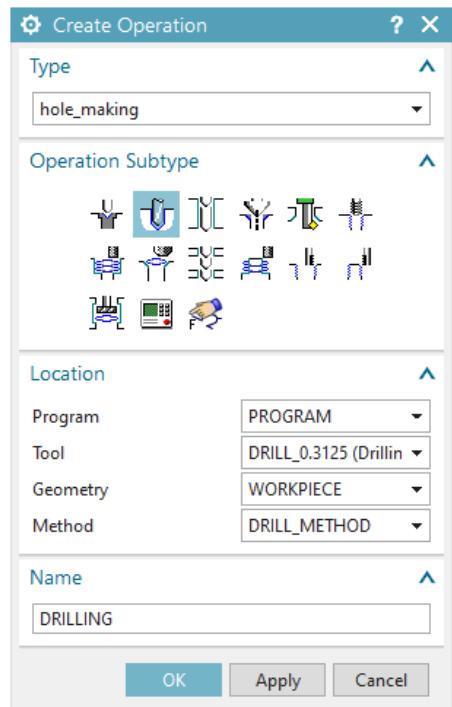
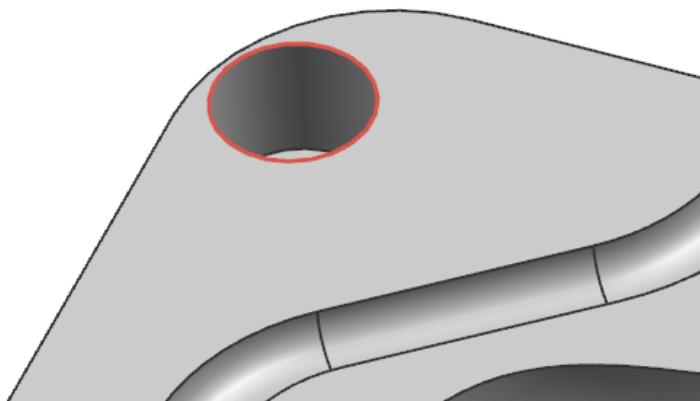
Type: hole\_making  
Operation Subtype: drilling  
Program: Program  
Tool: Drill\_0.3125  
Geometry: Workpiece  
Method: Drill\_Method  
Name: Drilling

2. Observe the Drilling dialogue

This dialog box is very similar to the one used for spot drilling. In this operation we will be using Specify Feature Geometry for the majority of the inputs.

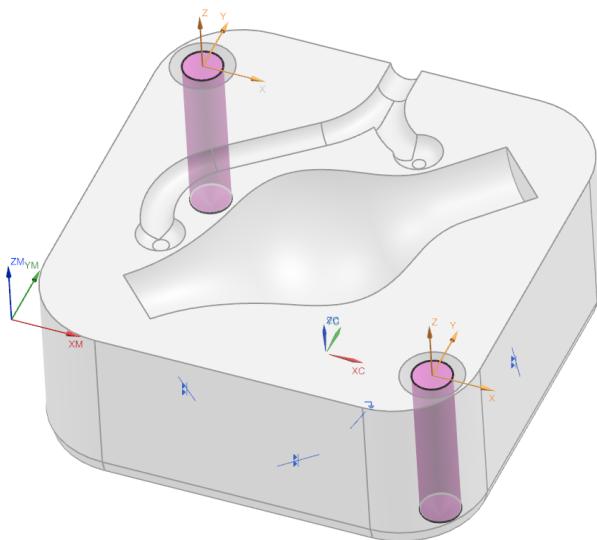
3. Select the hole locations

The procedure for selecting the hole location is identical to the previous spot drilling operation. Select the Specify Feature Geometry button which brings up the Feature geometry dialog. The select object box will be active. Use quick pick to select the edge of the hole.



#### 4. Enter the depths in the Feature Geometry Window

In the new hole making operation, NX attempts to gather more information about the geometry to be drilled. By selecting an edge next to the counterbore, NX thinks we want to drill a hole to the depth of the counterbore. We instead want to drill all the way through the part. Click the green lock symbol next to the depth input and select user defined to unlock the input. Enter 1.375 as the cutting depth.



**Feature Geometry**

**Common Parameters**

- In Process Workpiece: Local
- Machining Area: FACES\_CYLINDER\_1

**Cutting Parameters**

- Control Point: In-Process Feature
- Use Predefined Depth
- Bottom Stock: 0.0000
- Tool Drive Point: SYS\_CL\_TIP
- Intersection Strategy: None

**Feature**

- Select Object (1)
- Specify Orientation
- Diameter: 0.3125
- Depth: 1.3750 (highlighted with a red box)
- Start Diameter: 0.0000
- Depth Limit: Blind
- Reverse Direction

**List**

Item	Diameter	Depth	Start
1	0.313	1.375	0.0000
2	0.313	1.375	0.0000

**Preview**

**Display**

**Sequence**

- Optimization: Closest
- Reorder List
- Reverse List

OK Cancel

#### 5. Set cutting parameters

Check the top offset to ensure that drill starts its cutting move before contacting the surface. Check the bottom offset to ensure the drill moves all the way through the bottom before stopping.

**Cutting Parameters**

**Strategy** **Stock** **More**

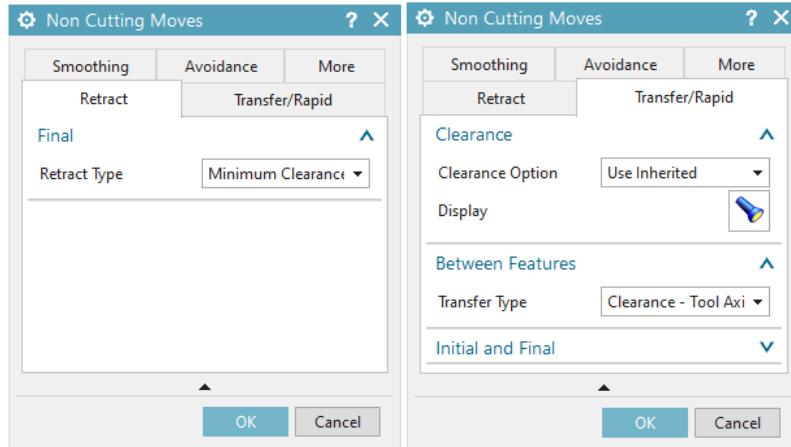
**Extend Path**

- Top Offset: Distance 0.1250
- Rapto Offset: Distance 0.0000
- Bottom Offset: Distance 0.1000

OK Cancel

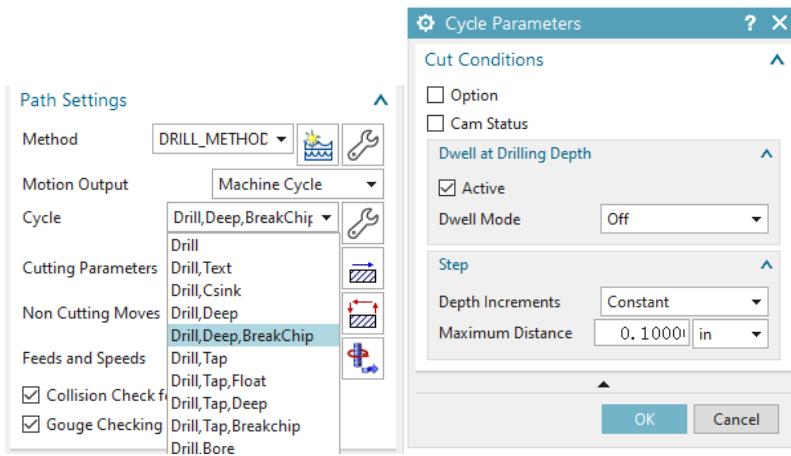
## 6. Set non cutting moves

Check the Non Cutting Moves dialog box to ensure that the clearance plane set in the beginning is inherited. By default, the retract type should be set to Minimum Clearance and the Clearance Option should be set to Use Inherited. Check to ensure these options are selected.



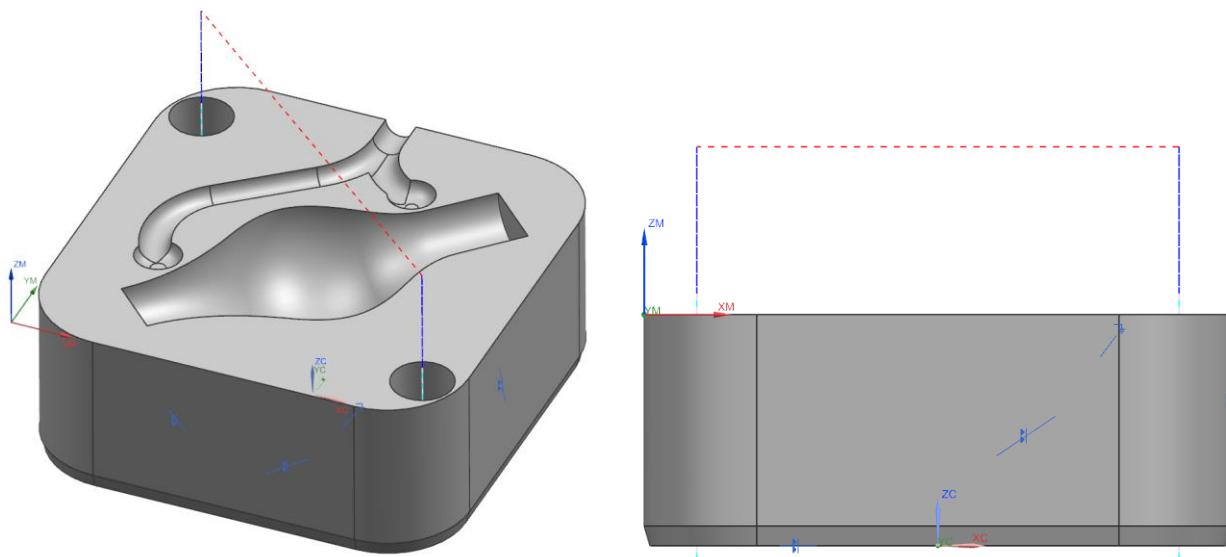
## 7. Set the Machine Cycle

Expand the dropdown menu for the Cycle under the Cycle Type section of the Drill dialogue. Select ‘Drill,Deep,Break Chip’ as the cycle. This cycle will call on the proper commands when we translate this operation to the CNC machine. Select the wrench icon to set the cycle parameters. Peck drilling is a type of drilling which does not drill the entire hole at once, but rather drills a small increment, and then retracts out of the hole. This is done for two reasons: To clear the chips formed within the drill, and to allow coolant and air to circulate around the drill, preventing overheating. For our peck drilling operation we wish to only use 1 constant step. Set the value next to Maximum distance to 0.1 inches and select OK.



## 8. Generate the toolpath

Verify the toolpath and make sure that the path includes drilling moves through the part geometry and retracts to the clearance plane. The visualization will not show the pecking motion of the drill, However, when the G-Code is generated this cycle will call a G83 instruction which is correct for break chip drilling.



## Reorder the operations.

For the two drilling operations created, we have worked on the basis that we will begin drilling on the top face, even though we have already created the operation which will machine the counterbore. However, it is possible to rearrange the operations so that the drilling occurs before the counterboring operation, and the drill operations do in fact machine material.

This is beneficial for the counterboring operation as well, since the tool will not be forced to remove as much material.

1. Observe the order of operations in the geometry view of the operation navigator. Click and drag the operations to the new order, as shown at below right.



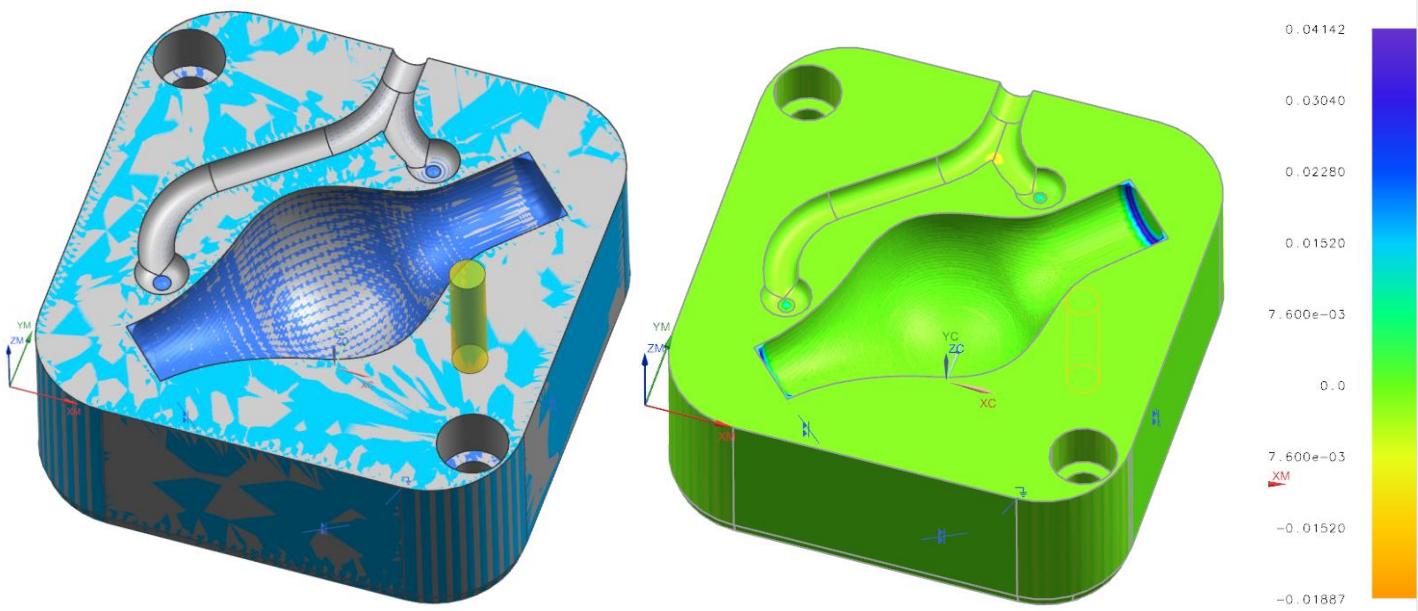
Operation Navigator - Geometry				
Name	Path	Tool	Geometry	Method
GEOMETRY				
Unused Items				
MCS_MILL				
WORKPIECE				
PLANAR_PROFILE	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
PLANAR_MILL	✓	EM-25	WORKPIECE	MILL_FINISH
CAVITY_MILL	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
CONTOUR_AREA	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
SPOT_DRILLING	✓	SPOT_DRILL	WORKPIECE	DRILL_METHOD
DRILLING	✓	DRILL-3125	WORKPIECE	DRILL_METHOD

Operation Navigator - Geometry				
Name	Path	Tool	Geometry	Method
GEOMETRY				
Unused Items				
MCS_MILL				
WORKPIECE				
PLANAR_PROFILE	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
BALL_MILL...	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
CAVITY_MILL	✓	CAVITY_MILL	WORKPIECE	MILL_FINISH
CONTOUR_AREA	✓	CONTOUR_AREA	WORKPIECE	MILL_FINISH
SPOT_DRILLING	✓	SPOT_DRILL	WORKPIECE	DRILL_METHOD
DRILLING	✓	DRILL-3125	WORKPIECE	DRILL_METHOD
PLANAR_MILL	✓	EM-25	WORKPIECE	MILL_FINISH

This new operation minimizes tool changes, and places each operation in a location where it is performing minimal work. When you are creating your own manufacturing programs, keep this option in mind, so that you may program your operations, and rearrange them to optimize your program.

2. Select the workpiece item on the operation navigator, and select the verify button from the NX toolbar. Verify the entire manufacturing setup using the 3D Dynamic method. Also review the final surface finish with the 'Analyze' tool within 3D Dynamic.



## Edit the tools, and feeds and speeds.

As a result of rearranging operations, the tools numbers we have designated may be placed out of order. The best practice is to use tools in a numerical order, and place them in the machine in the same way, whenever possible. Thus, the tools we are using should occur in 1,2,3,4 order.

1. Right click the operation navigator, and select ‘Machine Tool’ view. This will display all the tools we have created, and the operations that call on them.

Name	Path	Tool	Description	Tool Number
GENERIC_MACHINE			Generic Machine	
Unused Items			mill_planar	
BALL_MILL_25		BALL_MILL_25	Milling Tool-Ball Mill	1
PLANAR_PROFILE		PLANAR_PROFILE	1	
CAVITY_MILL		BALL_MILL_25	CAVITY_MILL	1
CONTOUR_AREA		BALL_MILL_25	CONTOUR_AREA	1
EM-.25		EM-.25	Milling Tool-5 Parameters	2
PLANAR_MILL		PLANAR_MILL	2	
SPOT_DRILL		SPOT_DRILL	Spot Drill	3
DRILL-.3125		DRILL-.3125	SPOT_DRILLING	3
DRILLING		DRILLING	Drilling Tool	4
			DRILLING	4

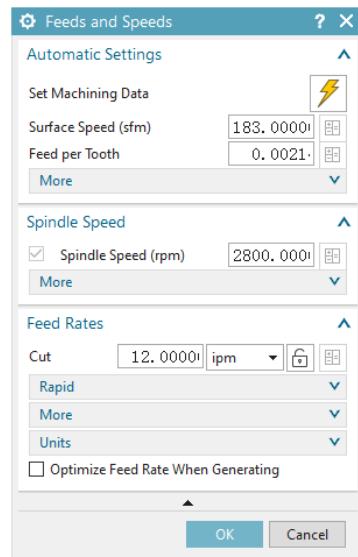
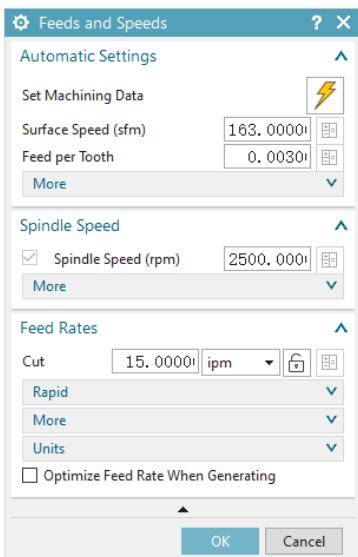
2. Right click each tool, and select ‘Edit’. Check the tools numbers to adhere to the following parameters:

¼ inch ball endmill:	1	Spot drilling tool:	3
¼ inch flat endmill:	2	0.3125 inch drill bit:	4

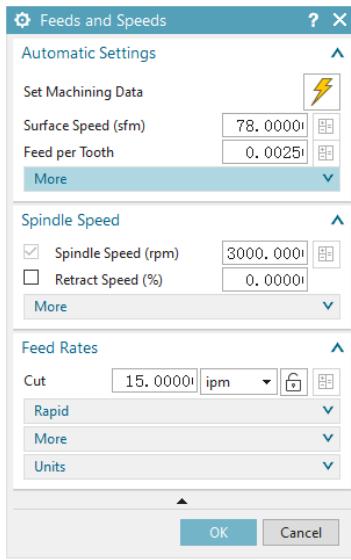
3. To edit feed and speed rates, we need to return to geometry view in the operation navigator. Do so by right clicking in the blank area in the operation navigator, and selecting ‘geometry view’.
4. Go through each operation, and calculate the proper feeds and speeds based on the equations given to you in the previous walkthrough. When you are calculating the values, you will end up with numbers such as RPM in the +6000 vicinity. Few machines have this capability, so we must reduce the feeds and speeds to acceptable machine levels. The screenshots below show some acceptable values for the various operations.

¼ endmill (ball and flat) roughing operations:

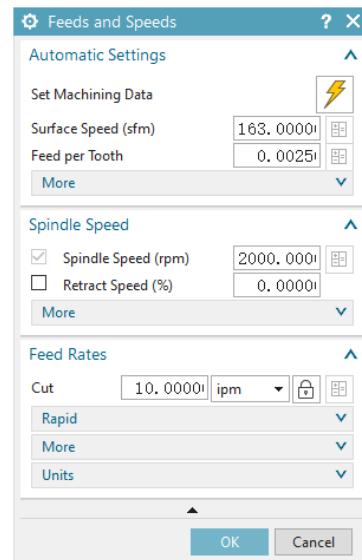
¼ endmill (ball and flat) finishing operations:



## Spot drilling operation:



## Drilling operation:



## Generate G-code

The final step in a manufacturing operation is to generate g-code. G-code is the programming language that CNC machines can read, which is different from the ‘programs’ we have been creating in NX. The NX CAM program creates these operations using its own internal designations for all the operations and items. We must ‘translate’ these operations into G-code for a machine.

To do this, we require the use of a post processor. A post processor is a set of three files which allows a CAM program to translate its operations to g-code. Unfortunately, there are a large number of CAM programs in existence, and even more types of CNC machine controllers in each type of CNC machines. Thus, there are unique post processors for every pairing of CAM program and CNC controller. Hence, there are a huge number of post processors available, and choosing the correct one is critical.

An incorrect post processor can leave out certain items, or enter incorrect parameters in G-code which can cause severe problems in manufacturing. Always ensure that you have correct post processors, or are using post processors which have been verified in the machine you are using, and have been looked over by shop experts.

1. To create the g-code for our manufacturing program, you must select the whole operation, and not an individual operation. Select the Workpiece item in geometry view to make sure all the operations have been selected.

Alternatively, we may change the operation navigator view to program view. Program view is the master generation method, and will display the same operations. However, when using this method, be sure to check that the order of operations is the same as have been created in the geometry view.

Operation Navigator - Geometry				
Name	Path	Tool	Geometry	Method
GEOMETRY				
Unused Items				
MCS MILL				
WORKPIECE				
PLANAR_PROFILE	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
CAVITY_MILL	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
CONTOUR_AREA	✓	BALL_MILL...	WORKPIECE	MILL_FINISH
SPOT_DRILLING	✓	SPOT_DRILL	WORKPIECE	DRILL_METHOD
DRILLING	✓	DRILL-.3125	WORKPIECE	DRILL_METHOD
PLANAR_MILL	✓	EM-25	WORKPIECE	MILL_FINISH

2. Find the Post Process button on the NX toolbar to the right of the toolpath action buttons.
3. This will open the Postprocess window. This is where we will select the postprocessor file to use. For this purpose, select the generic Mill\_3\_Axis postprocessor. This is a default NX processor which is not machine specific.

For future g-code generation, we will use a postprocessor unique to NX and the machines we will be using.

4. Under the settings section, make sure the Units are 'Post Defined'.

5. Select OK to generate the g-code file. This will bring up a text file window within NX which will display the g-code. The instructor will review the points within the g-code which translate to the items we have specified in NX.

```

N12 T01 M6
N14 G54
N16 G17 G0 G90 X1.8661 Y3.7406 S2800 M3
N18 G43 Z1. H1
N20 Z0.02
N22 G94 G1 Z-0.08 F12.
N24 Y3.624
N26 G3 X1.7411 Y3.499 R0.125
N28 G1 Y3.249
N30 G2 X1.4501 Y2.6425 R0.7776
N32 G1 X0.7882 Y1.1115
N34 G3 X0.6195 Y1.333 R0.6323
N36 G2 X0.6277 Y1.2487 R0.1166
N38 G3 X0.7083 Y1.2992 R-0.0709
N40 G2 X0.6222 Y1.3663 R0.1166
N42 X0.7987 Y2.0984 R0.6155
N44 G1 X1.4606 Y2.6294
N46 G3 X1.6647 Y2.8756 R0.7943
N48 G2 X1.8608 Y2.8909 R0.1166
N50 G3 X2.0587 Y2.7223 R0.5956
N52 G2 X2.1139 Y2.6588 R0.1166
N54 G3 X2.15 Y2.7469 R-0.0709
N56 G2 X2.04 Y2.752 R0.1166
N58 X1.7579 Y3.249 R0.5788
N60 G1 Y3.499
N62 G3 X1.6329 Y3.624 R0.125
N64 G1 Y3.7406

```

6. It is always best practice to go over generated g-code to ensure the machine will do what you want it to do.

