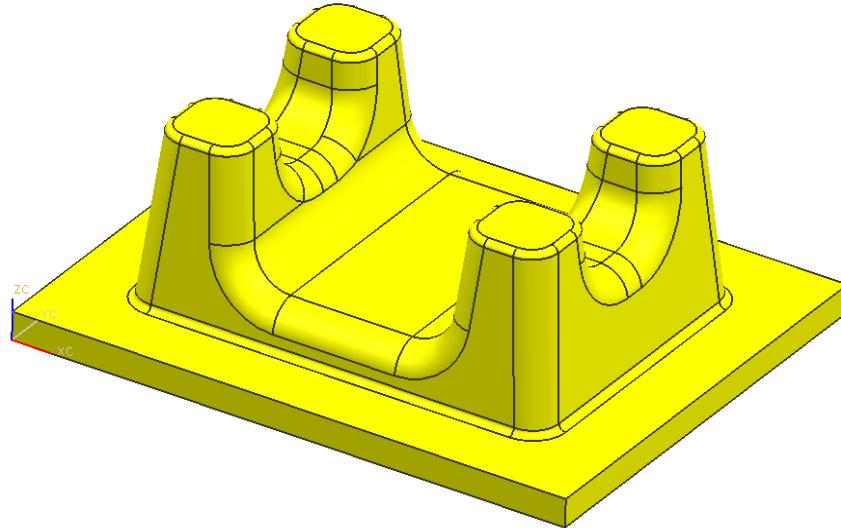


# NX CAM Tutorial – Basic Manufacturing #1: Setup, Cavity, and Area Milling

Written by: Mike Beltran, Department of Mechanical Engineering

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



This walkthrough will guide the student through the creation of a basic manufacturing setup in NX. The walkthrough will begin with the creation of assembly setup file and general machining setup, and be followed by the addition of various milling operations, covering rough v. finishing operations, stepover and cut levels, cut patterns, tool feed and speed rates, path generation, to generate a manufactured part.

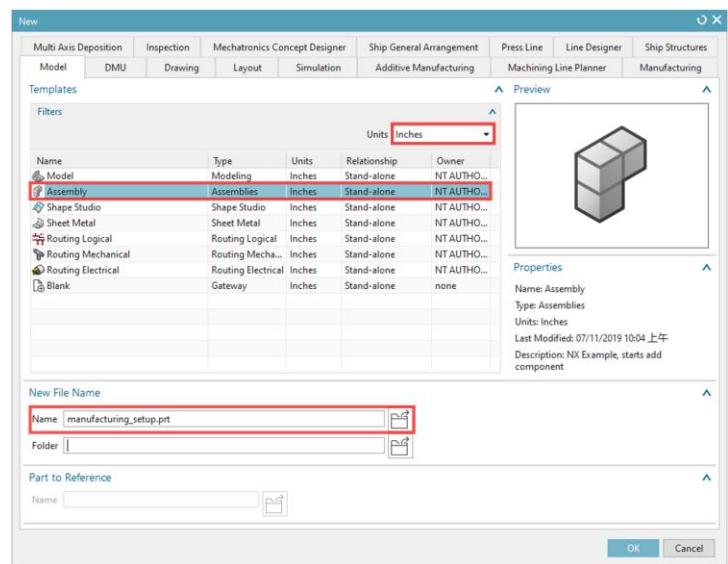
This walkthrough assumes a student is familiar with the basic workings of NX. A student should not attempt to complete this walkthrough if they are not confident in using the NX modeling environment, and creating assemblies.

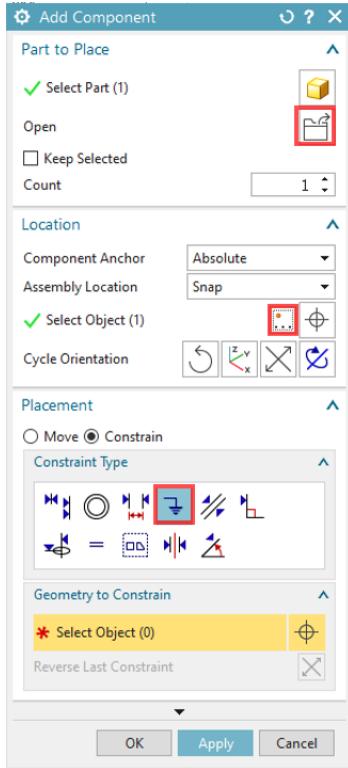
Parts for this walkthrough will be available on Canvas.

## Create an assembly and enter the manufacturing environment.

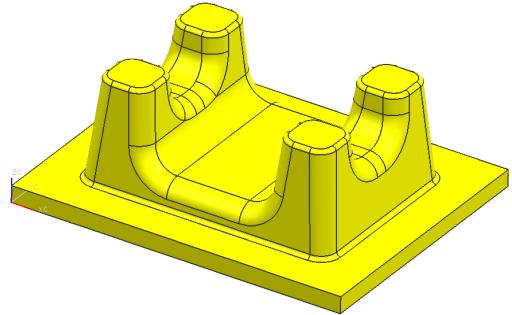
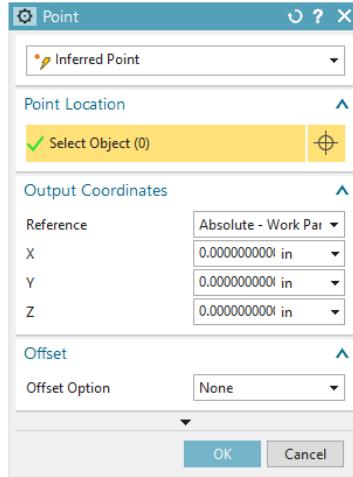
The beginning part of generating a manufacturing program is the creation of assembly using the part intended to be machined (part), and stock the part will be machined from (blank). The blank will be overlaid on the part we intend to create. This is the basis for ‘smart’ operations within NX. NX can recognize what material is there, and what you intend to create, and act accordingly.

1. Create a new assembly file, and save the assembly to the same directory as the parts you have downloaded. Name the file manufacturing\_setup.prt. Choose units to Inches. Click OK.



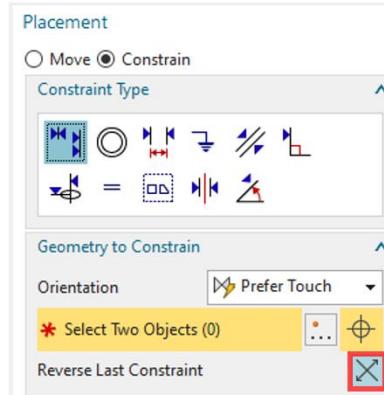
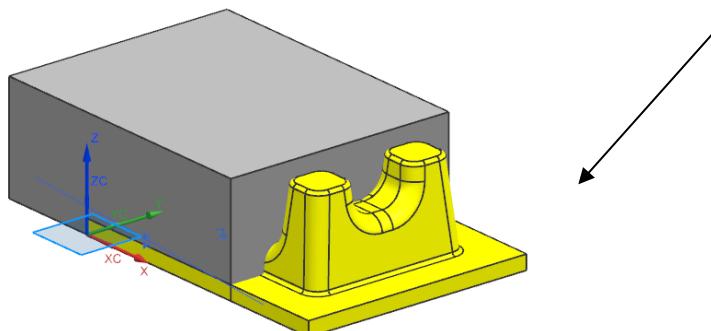
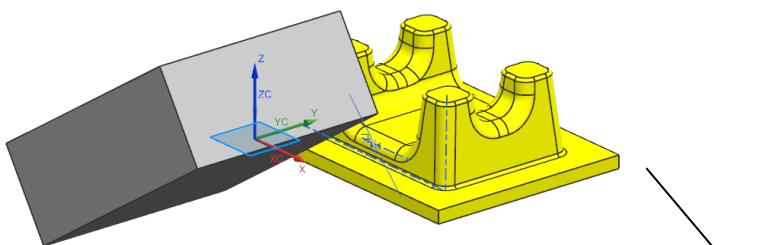
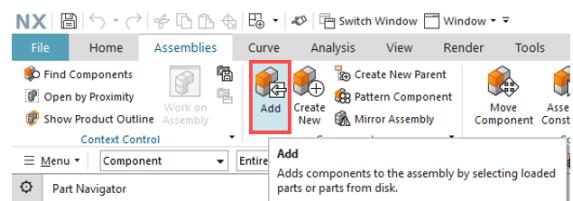


2. An 'Add Component' window should appear for us to add the part component. Open part 'mfg\_base\_1\_x\_t.prt'. To specify the placement location, click the Point Dialog button next to Select Object. Output Coordinate should be set to 0, 0, 0 by default. Click OK to confirm the location. Finally add a 'fix' constraint to prevent the part from moving.

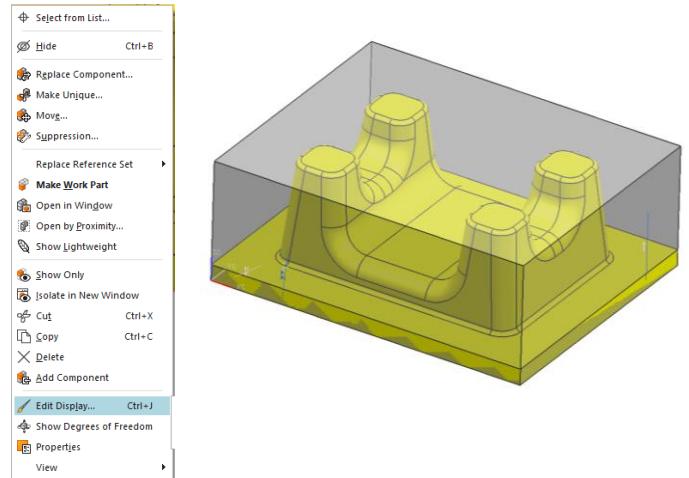


- Click 'Add' on the toolbar to add the blank component, 'mfg\_stock\_1\_x\_t.prt'. Align the blank so that there are touch align constraints between the front, side, and bottom of the blank square, and the corresponding sides of the rectangle section at the bottom of the part to be manufactured. Pay attention to the direction of the alignment.

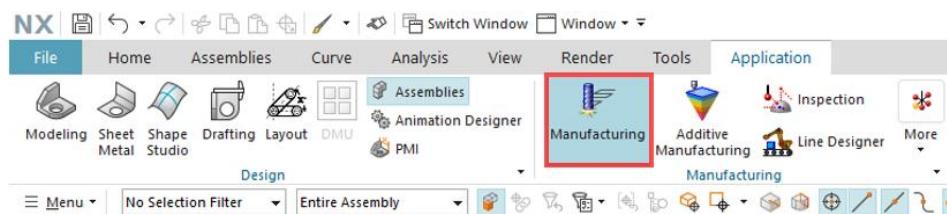
Note: You may need to use the 'Reverse Last Constraint' feature to align the parts properly.



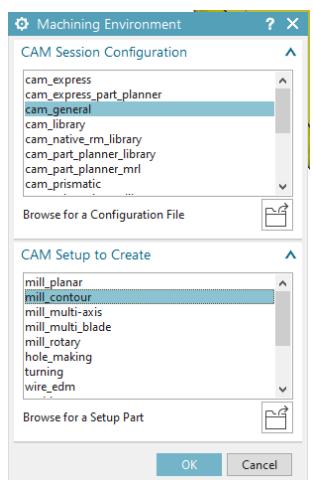
4. Right click the blank part, and go to 'edit display'.  
 Change the translucency to anywhere between 60% - 90%. Set it to a level you feel comfortable viewing.



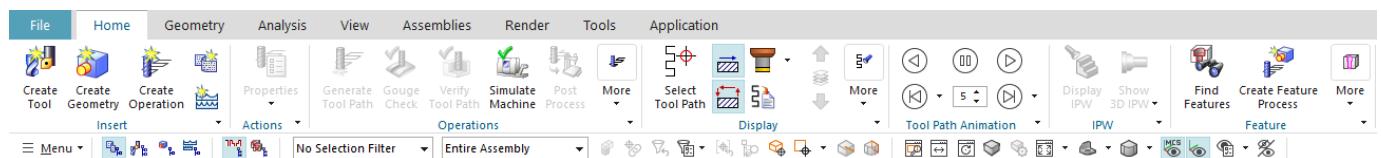
5. Enter the manufacturing environment by going to the Application tab and select Manufacturing button. This will exit the modeling environment, and switch to the manufacturing environment.



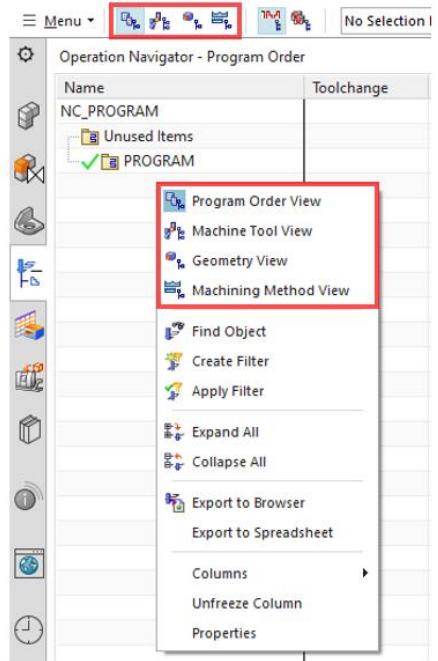
6. A CAM configuration window will pop up once you have entered the manufacturing environment. This will attempt to guide you along the path to creating new manufacturing operations. Select 'cam\_general' for the CAM session configuration, and 'mill\_contour' for the CAM setup to create. These are merely configuration items, and will not create any operations yet.



7. Observe the manufacturing environment toolbar. Like other areas of NX, we will be working generally from left to right.



8. Observe how the assembly tree has now switched to a new tab, unique to the manufacturing environment. This is called the Operation Navigator. Four different views are available to view all the manufacturing operations you have created. They are the Program order, Machine Tool order, Geometry order, and Machining Method order. The differences between these views will become more apparent once we have added multiple manufacturing operations.



9. Right click on the toolbar, and a long menu will appear with the available shortcut layouts that may be placed on the toolbar. You can also access the layouts through the buttons next to 'Menu'. Click the different views to see what they show.
10. For the time being, select the 'geometry view' option. This will give you a straightforward method of creating all the items necessary for your setup.

Operation Navigator - Geometry		
Name	Path	Tool
GEOOMETRY		
Unused Items		
MCS_MILL		

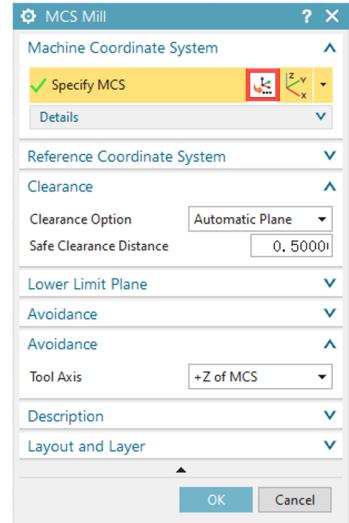
### Create and Edit the MCS and Clearance Plane

When a new manufacturing setup is created, there are defaults that are put in place, just like the existing planes in a typical part modeling file. In Manufacturing, a new coordinate system called the machining coordinate system (MCS) is created. This is THE MOST IMPORTANT item you must set in any manufacturing file. This is what determines the X Y Z zero points in your setup, and it is what you must match to when zeroing your part in the actual machine. If this is mismatched between the NX manufacturing setup and your machine setup, you can cause significant damage to the machine, particularly if you place your coordinate system in the machine, below where it is in your NX setup.

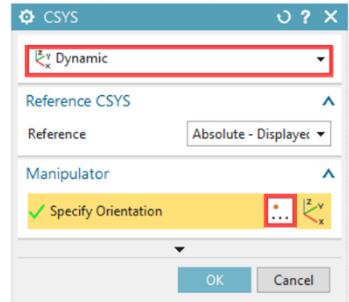
We will edit the existing MCS to a simple existing point on our part and blank assembly, and create a clearance plane, which is the plane a tool will retract to when moving in a 'rapid' movement across a workpiece.

1. Make sure you are viewing the operation manager in geometry view. Expand the item called 'MCS\_MILL', so you can see the 'Workpiece' item underneath.

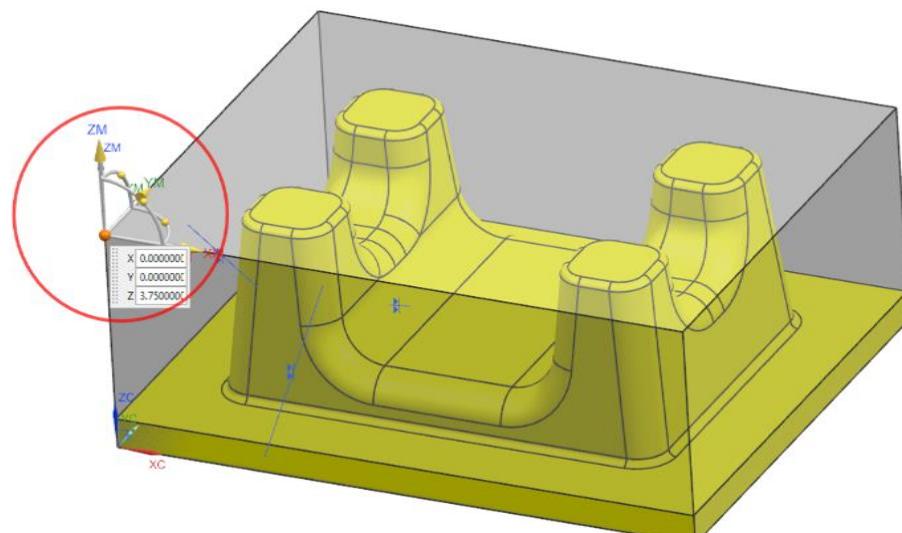
- Right click the MCS\_MILL item, and select 'Edit'. A window titled 'MCS Mill' will appear. Observe how items exist to edit the Machine Coordinate System, and the Clearance plane.
- Under Machine Coordinate System, click the smaller button, which displays the CSYS Dialogue when the mouse pointer is over it. This will bring up a CSYS window where we can choose where to place our MCS.



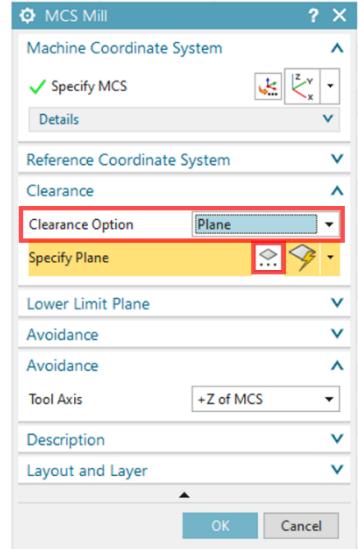
- Expand the first pulldown menu. Observe how all the same methods of creating a new coordinate system are the same from the modeling environment dialogue of creating datum geometry. From the pulldown menu, select the Dynamic item and then click on point dialog button.
- Select the bottom left corner of the blank as the zero point. If you have problem selecting the corner, make sure you have 'No Selection Filter', can select from the 'Entire Assembly', and 'Enable Snap Point' is activated.



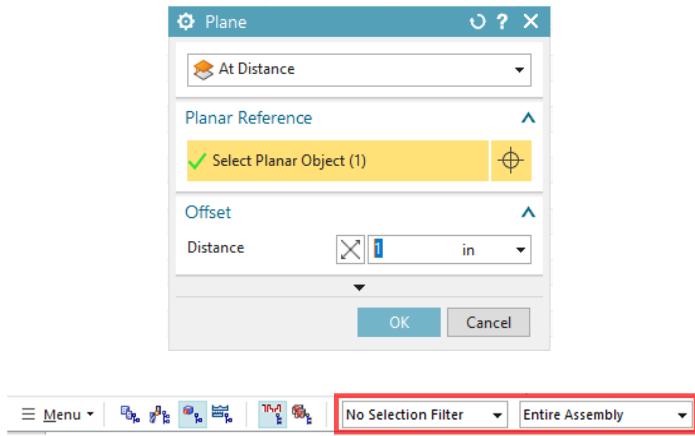
- Exit out of the menus to the MCS\_MILL window, and observe the new coordinate system you have placed. Observe how the axis are labeled XM, YM, and ZM, inferring that this is the 'M', or machine coordinate system.



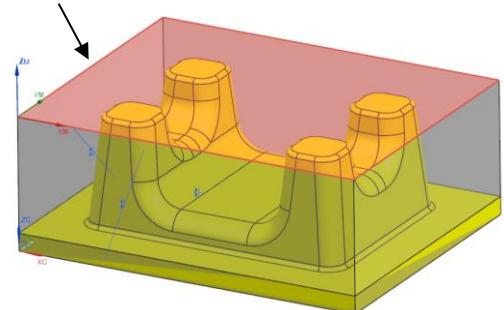
7. In the MCS\_Mill window, change the Clearance Option to Plane underneath the ‘clearance’ menu. Click on plane dialog button from Specify Plane option.



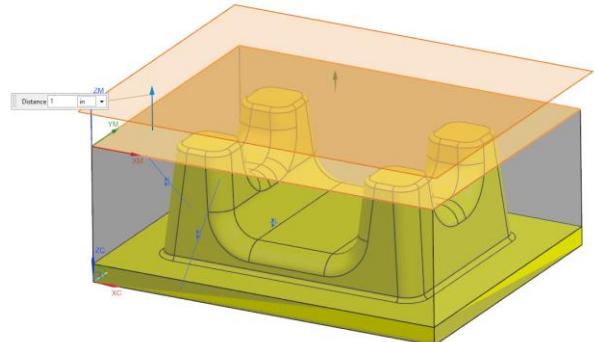
8. We wish to place a plane 1 inch above the top face of the blank part. Change “Inferred” to “At Distance”. Enter in 1.000 in the offset dialogue box. Once again make sure your selection filters allow you to select items from the ‘Entire Assembly’.



9. You should now be able to select the top face of the blank part.



10. Exit the menus and return to the main modeling window. When at the MCS\_MILL window, you should be able to see the clearance plane above the top face of the blank. This represents where we have placed the clearance plane, and you will see how this selection effects machining operations.



## Set the Part and Blank Items

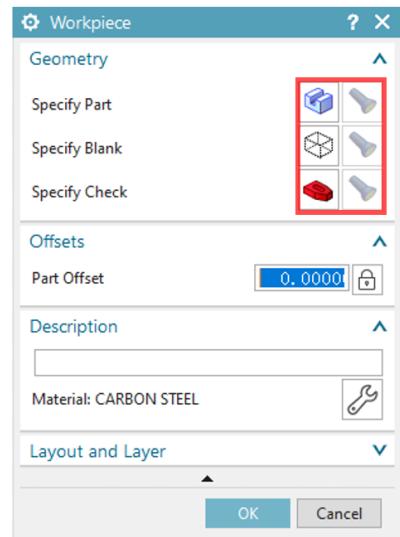
As described at the beginning of the walkthrough, we have placed two models into our assembly: One is the part we intend to create, and one is the part that we will begin with when machining. Now we must tell NX which part is which, so that it can later interpret these items when creating manufacturing operations.

NX has already created a default workpiece, which is the combination of part and blank, just as it has created a default MCS. We must edit the existing workpiece and specify our part and blank.

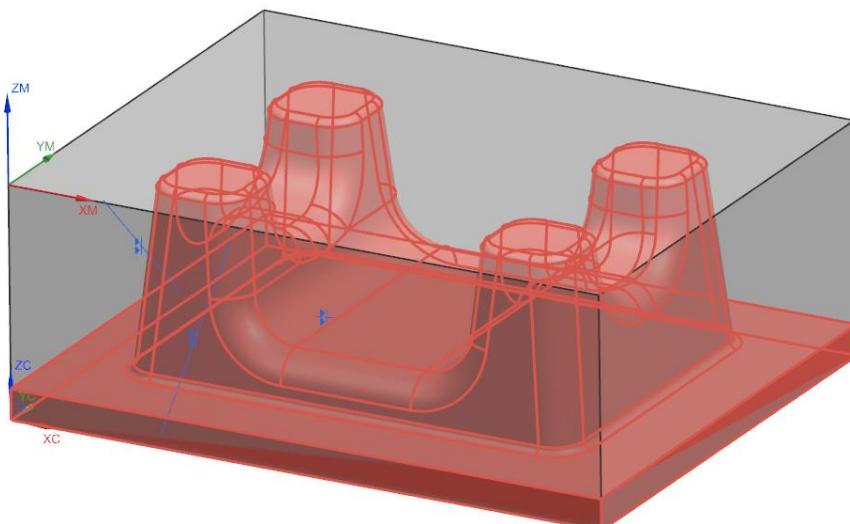
- Under the MCS\_MILL item in the operation navigator in the geometry view, there is an item called ‘Workpiece’. Right click and select ‘edit’.

- You can see the three items that exist under Geometry. Part, Blank, and Check. You must specify a part for any manufacturing setup. A Blank and Check part add additional functionality, primarily for cavity milling. Check geometry allows you to place in models which represent items you are using to hold down your workpiece, and wish to avoid. For this class, we will excluded using check geometry.

- Click the active button to specify a part. This should open the 'Part Geometry' window.

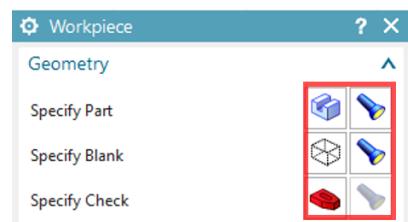


- Select the model as shown at right as the part. If you cannot select the piece as a result of the blank part preventing selection, you can hold your mouse still for a few seconds on the part until the cursor shape changes, then by clicking on the part a QuickPick pop-up will show up. Select the part (not the blank) from the list.



- Select OK once the part has been selected, and return to the Geometry window. Select the active button to specify a blank. A 'Blank Geometry' window should activate.

- Select the rectangular block as the blank part, and exit back to the Mill Geometry window. Notice now that you have selected parts, the flashlight buttons are now active next to the Specify Part and Specify Blank items. You can click these buttons to have NX highlight the part you have selected.



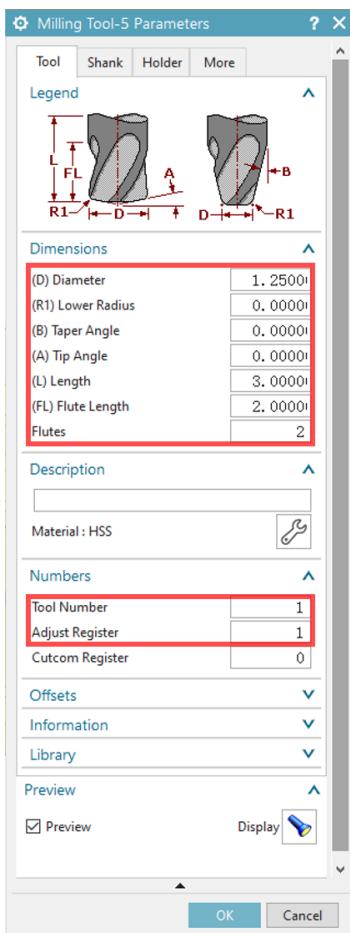
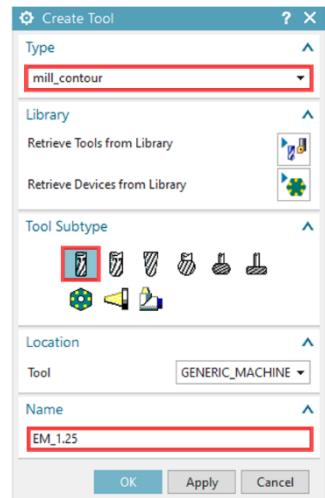
- Select OK and exit the Mill Geometry workpiece. You have now set the workpiece to our part and blank bodies.

## Create a tool

The next step in creating a manufacturing setup is to add the tools you will be using to manufacture the part. This can be done before creating a manufacturing operation, as we will do now, or while creating a new operation, which we will review further into this walkthrough. Both methods bring up the same dialogues, and create tools that you can use for the current operation, and any other operations. Tool creation also allows you to add geometry such as toolholders, which are the equivalent of check geometry for tools. This can allow you to model the entire tool in NX, and check for any possible collisions between the tool and workpiece.

NX does have an existing library of tools, however we will first learn to create our own tools. This class will not review adding tools from the NX library.

1. Click the ‘Create Tool’ button on the toolbar at the top of the NX environment.  
This will bring up a ‘Create Tool’ window where you will choose what type of tool to create, and have the option to name the tool.
2. Select the first Tool Subtype, Mill, and name the tool EM\_1.25. Leave the tool location to the default “Generic\_Machine”. Select OK to enter the Milling Tool Parameters window.



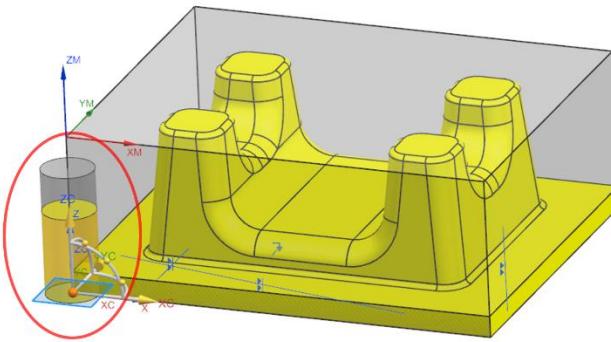
3. The Milling Parameters window allows you to edit any parameter of a tool. The two main items to add should be the diameter and length of the tool, as these will give you the basic representation of the tool. You also need to give the tool a number and an adjust register. The adjust register is used by the CNC machine to perform tool changes, and should have the same value as the tool number. Create the milling tool with the following parameters:

Diameter:	1.25 in
Length:	3.00 in
Flute Length:	2.00 in
Tool Number:	1
Adjust Register:	1

Remember, when selecting a tool, you must choose a tool which can reach the absolute bottom of your features. 3 inches is the height of our largest feature, so our tool has been appropriately sized. For this exercise, we will not be adding a holder feature.

4. You may choose the material of the cutter under the description section. There are many different types of material, which NX will attempt to use to guess a feed and speed rate during operations using this tool. This class will generally use High Speed Steel tools, so you may select HSS.

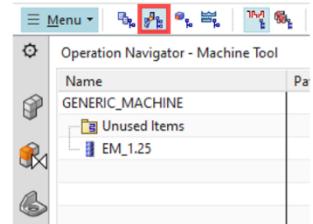
5. If you observe the modeling window, a model of the tool has appeared at the absolute origin of the part. This is to give you a visual representation of the tool you will be using. NX will also use this information to create XYZ coordinates which have been appropriately offset to your chosen tool diameter.



Search Result		
Class Description		
Tool Material		
Matching Items		
Library Reference	Material Name	Material Descrip
TMC0_00001	HSS	High Speed Steel
TMC0_00002	Carbide	Carbide, Uncoate
TMC0_00003	Carbide	Carbide, Uncoate
TMC0_00004	Carbide Coated	Carbide, Coated (
TMC0_00006	HSS Coated	High Speed Steel
TMC0_00021	HSM Ball Mill TiAIN Coat...	HSM Carbide Ball
TMC0_00022	HSM End Mill TiAIN Coat...	HSM Carbide End
TMC0_00023	HSM Bull Nose Inserted	HSM Inserted Bul
TMC0_00025	HSM End Mill Inserted	HSM Inserted Enc
TMC0_00026	HSM Ball Mill Inserted	HSM Inserted Ball
TMC0_00027	HSM Bull Nose TiAIN Co...	HSM Carbide Bull
TMC0_00028	HSM End Mill Extra Long	HSM HardCut Ins
TMC0_00041	Ruby	Ruby
TMC0_00042	Silicon Nitride	Silicon Nitride
TMC0_00043	Zirconia	Zirconia
TMC0_00051	Brass Wire	Brass Wire
TMC0_00052	Zinc Coated Brass Wire	Zinc Coated Brass

OK Back Cancel

6. Select OK, and create the tool. Switch to the Machine Tool view on the operation manager via the buttons on the toolbar. Notice how the tool you have just created is now in the tree. Take note, that the tool is not viewable in the other trees at this time. Switch back to the Geometry view once you are done.



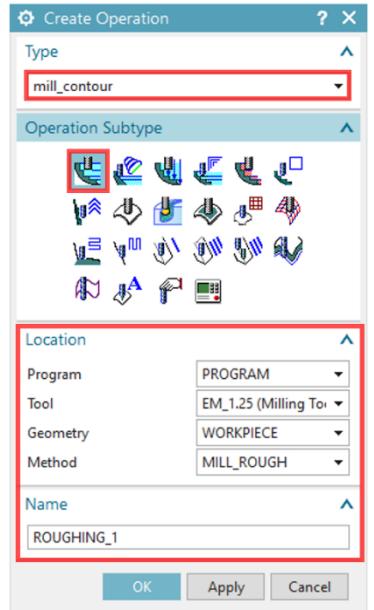
## Create a Roughing, Cavity Milling Operation

We will now create a cavity milling operation. The cavity milling operation is one of the more advanced and powerful manufacturing operations NX has, since it can recognize between the part and blank piece, and act accordingly based on your specifications. More basic operations exist, and we will review them at a later time.

The first operation we will be creating is the roughing operation, which will remove the bulk of the material from the piece. The end mill with a flat end geometry (like the one we just created) is ideal for this type of operation. Successive operation will focus on the details of the part.

The options available to edit the parameters of an operation are so numerous, it will be primarily up to the student to review all the methods available to them. We will review a general approach to creating this piece, and review a few of the most important items to consider when creating a milling operation.

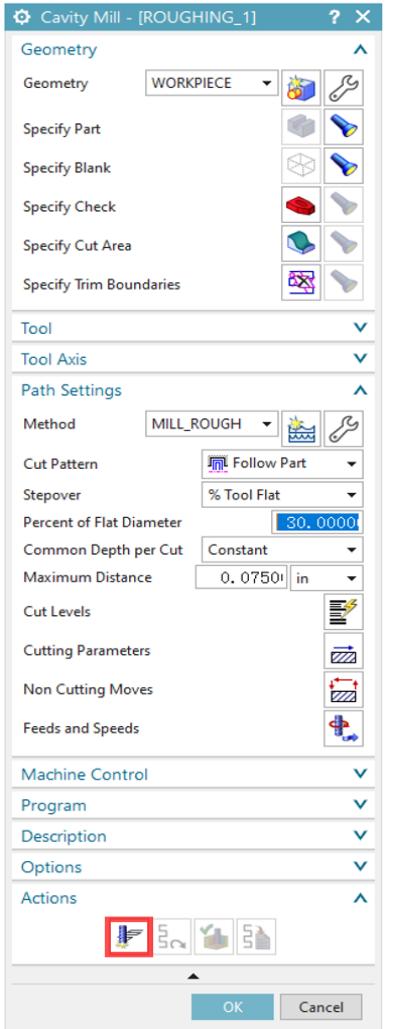
- Create a new operation by selecting the 'Create Operation' button on the left side of the toolbar. This will bring up the Create Operation window. As you can see, there are a huge number of operations to select from. The dropdown menu under the Type section shows the general categories of operations, beginning with mill\_planar, the most basic of operations. Mill\_contour contains more advanced operations, with the following categories increasing in complexity and specialization.



- Select the mill\_contour category under the Type section, and choose the first option, which should be a basic Cavity\_Mill operation. The options listed below allow us to inherit some options that we have previously set up, such as geometry and tools. Select the following items under each pulldown menu:

Program:	Program	(This is the default created program)
Tool:	EM_1.25	(This operation will default to our tool)
Geometry:	Workpiece	(This will inherit our part and blank)
Method:	MILL_ROUGH	
Name:	ROUGHING_1	

- Once the above items have been entered, select OK and enter the Cavity Mill Dialogue. This is the general layout for all types of manufacturing operations, listing required geometry, tool, tool axis, path settings, and other additional variables that can be set. We have previously set up the only two required items for a cavity mill, the geometry and tool to be used. The rest of the settings have set to NX default settings, which we successively modify to observe the how each changes our operation.

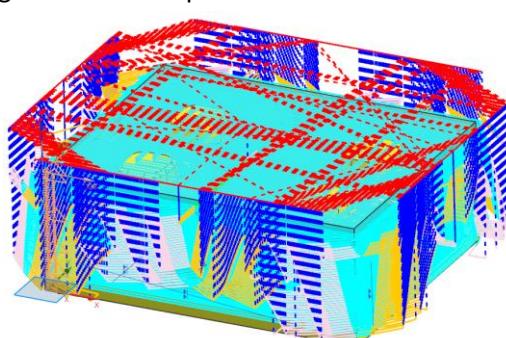


Note: Do NOT press Esc while working with any dialogue. Pressing Esc will close the dialogue without saving.

- We will proceed directly to generate a toolpath to observe the defaults that NX uses. Scroll to the bottom of the Cavity Mill window, and select the leftmost and only active 'Generate' button. This will create a toolpath as shown below:

Colors represent the following:

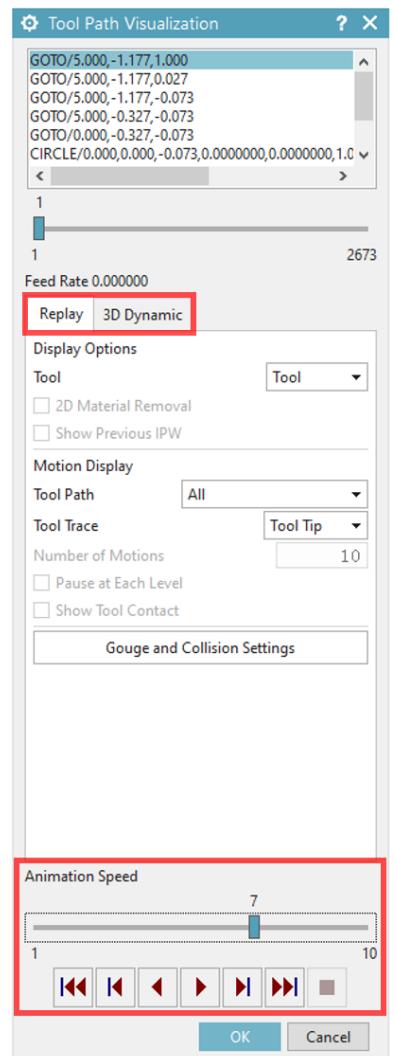
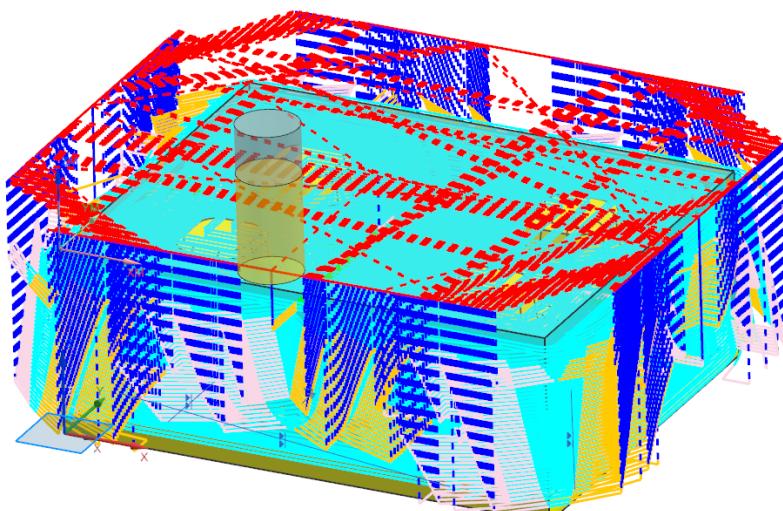
<b>Red:</b>	Rapid movement along the clearance plane
<b>Yellow:</b>	Lead-in movement
<b>Blue:</b>	Lead-out movement
<b>Turquoise:</b>	Cutting movement



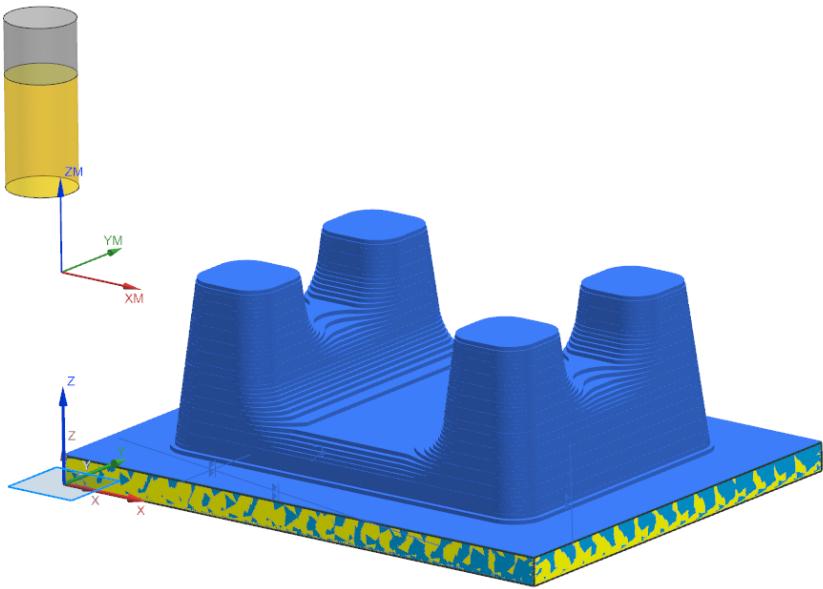
## Preview the Cavity Milling Operation

- Once the toolpath has generated, select the 'Verify' button, 3<sup>rd</sup> from left at the bottom of the Cavity Mill Dialogue. This will enter a preview verification method where you can observe how the operation will proceed in 3D. The toolpath view is far too cluttered to observe what is actually happening, so the verify option allows you to see the process in real and accelerated time. This is a significantly important tool, as it will allow you to see what your part will look like after a certain operation, and thus be able to modify aspects of your operation to obtain your desired result.
- When you enter the verification environment, a Tool Path Visualization window will appear. The first window displays commands within NX which represent the toolpath as NX stores it in its memory. This is not machine gcode, which we will create later. Below this exists a section with 2 tabs: Replay and 3D Dynamic. Replay is the default option every time the visualization environment is opened.
- Below the tabbed window is a slider tool to control the animation speed. Use this tool to adjust the speed of the preview so you can see what is actually happening.
- Hit the play button on the bottom to observe what 'Replay' displays as visualization, and then switch to successive tabs to observe how they display the tool preview, and the options under the Visualization dialogue.

Replay: A simple observation of the tool moving along the toolpath.



**3D Dynamic:** The part can be manipulated as normal during the preview, and material is removed from the blank as the tool moves along the toolpath. 3D Dynamic is an essential tool for reviewing operations. The simulated toolpath, material removal, and surface finish is a very accurate presentation of the real machining process.



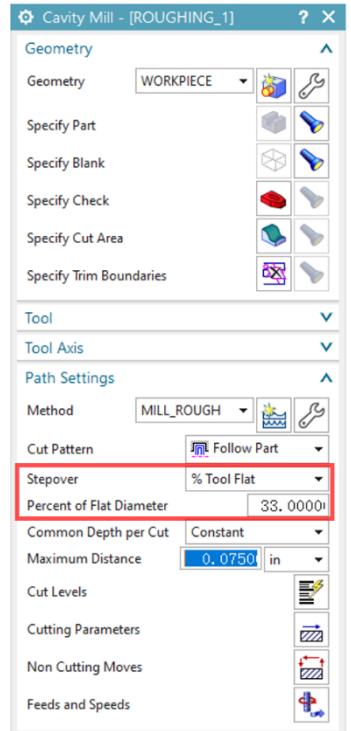
- When finished, select OK. Do not exit the Cavity Mill dialogue.

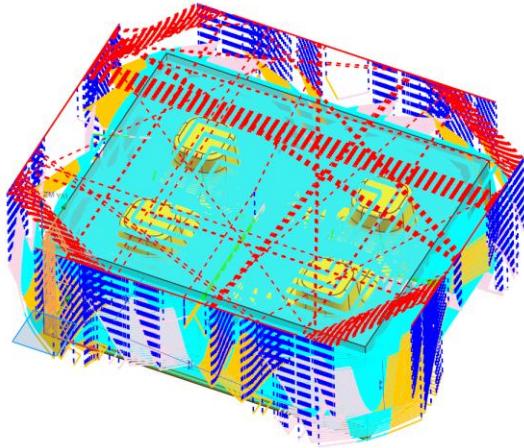
#### Change the tool step-over, and re-generate the toolpath.

When programming operations into NX, the operator still must follow the basic machining rules regarding the amount of cut a single tool may take. This is a common source of error and forgetfulness in programming toolpaths, and this cannot be adjusted during machining.

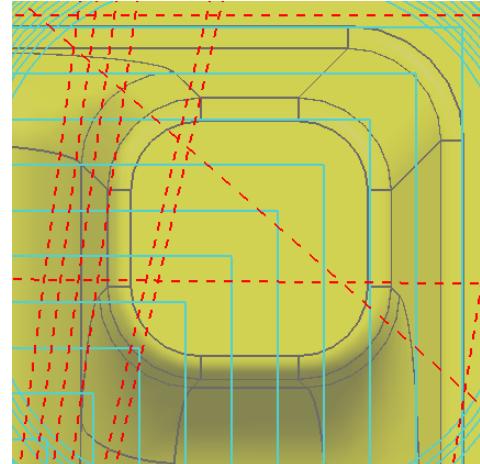
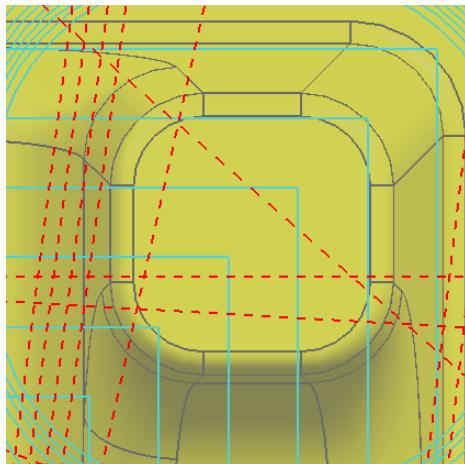
As a general rule, the largest cut a single endmill should take is 1/3 the diameter of the tool, in both depth of cut, and stepover. This will prevent tool damage, and pre-mature wear on the endmill.

- In the Cavity Mill Dialogue, under Path Settings, you will find a stepover item. The default method of data entry is '% of tool diameter'. The pulldown menu shows the other options, including a constant, scallop, and multiple type of data entry. You may choose any of those to see the data type required underneath the dropdown menu.
- For the % of tool diameter option, a 'Percent of Flat Diameter' variable is directly below the dropdown menu, and has a default value of 30.0. Change this value to 33.0.
- Regenerate the toolpath by selecting the 'generate' button at the bottom of the Cavity Mill Dialogue. Regenerating the toolpath is necessary after every change of variable to show the change of that variable on the toolpath.

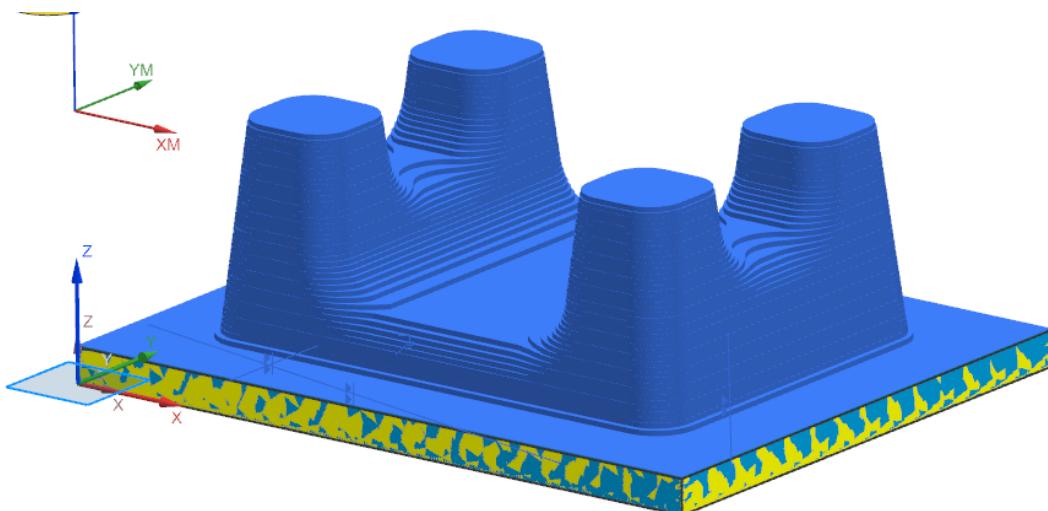




4. At first glance, nothing appears very different. Select the top view of the model from the View Orientation dropdown menu. If you look at the spacing between the Turquoise lines, you can see they are now closer together.



5. Enter the visualization environment, and preview the part using 3D dynamic. You can see how the operation now takes slightly longer to complete, since the tool is taking less of a cut during every pass. You may slow down the visualization to see how the stepover cut is less than before. The final part does not appear any different, since the variable edited has no effect on the final part, only the path taken to get there.



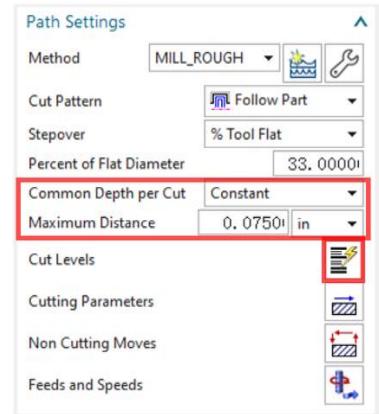
6. Exit the visualization environment by selecting OK. Remain in the Cavity Mill dialogue.

## Change the depth of cut, and re-generate the toolpath.

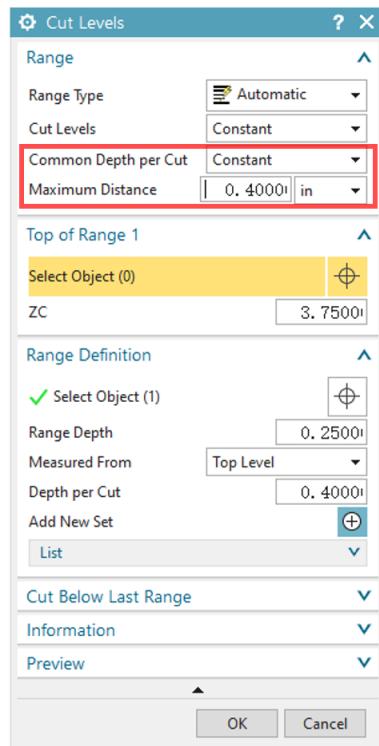
As previously stated, we must follow the 1/3 rule of both stepover, and depth of cut. We will now edit the depth of cut for this operation.

NX refers to depth of cut by 'Cut Levels', which creates different regions of the part, where depths of cut can be changed. Depth of cut is a relatively un-important variable in roughing operations, but becomes a very significant variable in finishing operations, since it is the primary factor in determining a surface finish on angled surfaces, and overall accuracy of the final part.

- Under the Path Settings section of the Cavity Mill dialogue, you will find an item labeled 'Cut Levels'. Select the button to the right to enter the cut levels dialogue.

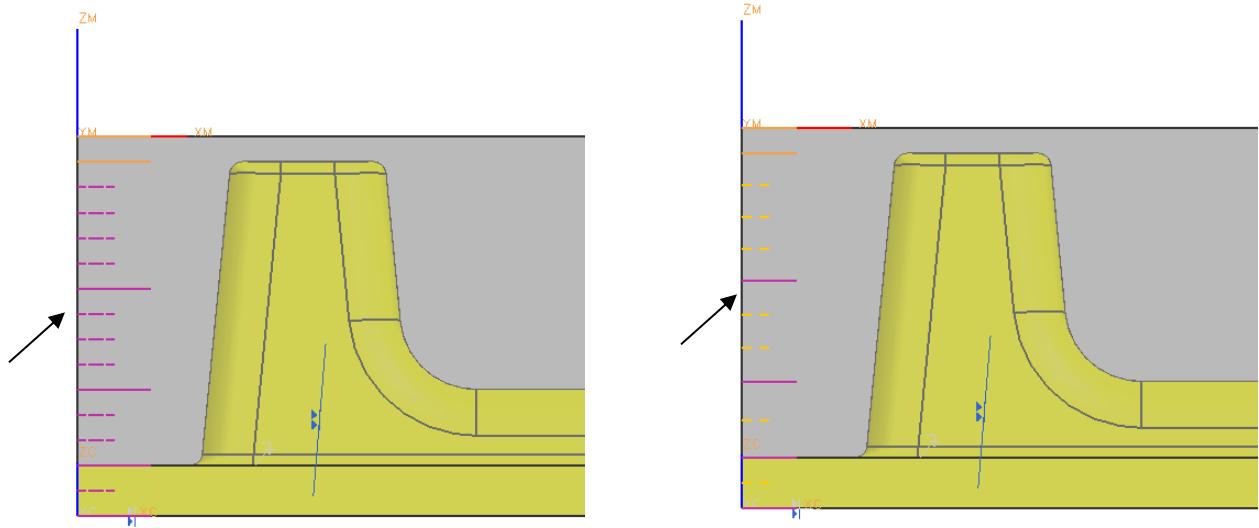


- The Cut Levels dialogue displays a selection of Range Type buttons at the top, which are Automatic, User Defined, and Single. The default Automatic method has created various ranges, as shown by in the modeling window by the large triangles on the scale that has appeared underneath the MCS. NX recognizes different ranges each time there is a new flat section of the model. As you can see, there are 4 flat depths on our model, plus the top and bottom of the blank part. For each of these sections you can set the depth of cut by hitting the green up and down arrows to select the appropriate range. We will be changing Range Types in later operations. For this operation, we will remain with Automatic.

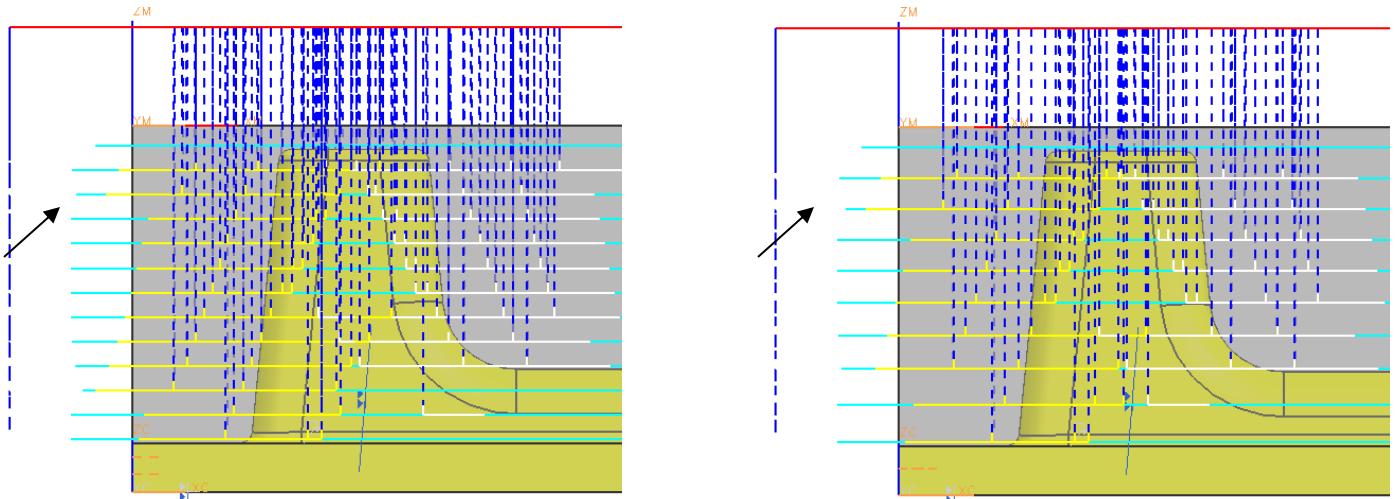


- We wish to change the depth of cut for every range. The simplest method to do so is to change the Common Depth per Cut option at the top of the Cut Levels dialogue. This will change the value for every range. Set the depth to 0.4 inches, which is approximately 1/3 of the 1.25 inch diameter of our tool.

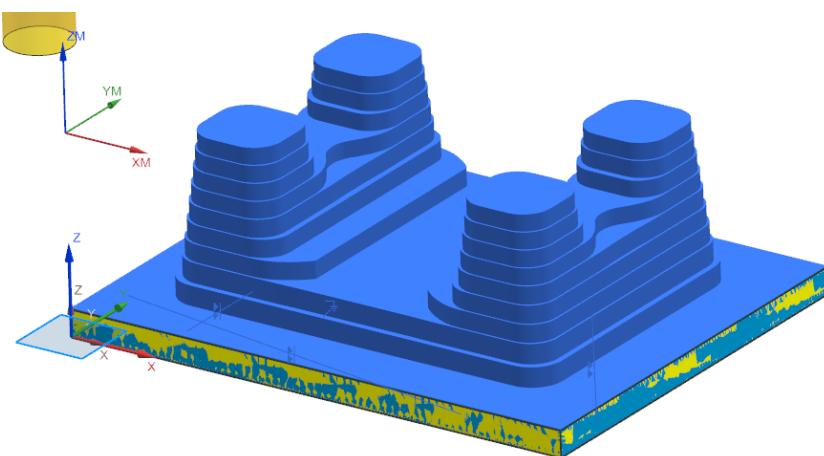
- Observe how the increments in the scale displaying cut levels now grow larger. This reflects the increase in cut the tool will take on each pass.



5. Regenerate the toolpath with the 'generate' button. Again, at a simple glance, it is difficult to observe any differences. View the part from the front with the view orientation tool, and observe the spacing between turquoise lines.



6. Verify the path using 3D Dynamic. If you view the part from the front again, you can observe how the tool is taking smaller cuts. Notice how this time, the final part has changed slightly.



7. Exit the visualization environment, and remain in the Cavity Mill dialogue.

## Change the tool feed and speed rates

The proper tool feed and speed rates are another critical input to a milling operation. Incorrect tool feed and speed rates are one of the most common causes of tool damage, part damage, and machine damage. A feed rate too large could break an endmill, damage your part, and possibly damage the machine you are using. An incorrect tool speed may prematurely wear, or even break a tool, and either result in a poor surface finish, or significantly damage your part.

In order to ensure you have a proper tool feed and speed rate, there are two equations which can be used to determine a proper feed and speed rate.

$$RPM = \frac{Cutting\_Speed \times 4}{Tool\_Diameter}$$

$$Feedrate = \frac{Feed}{Flute} \times RPM \times \# \text{ flutes}$$

The Feed/Tooth input and Cutting Speed can be found in any Handbook of Mechanical Engineering. For rough inputs, consider the following:

Feed/Flute: 0.0005" - 0.003" per flute.

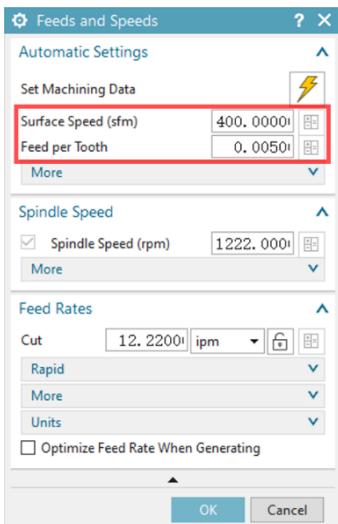
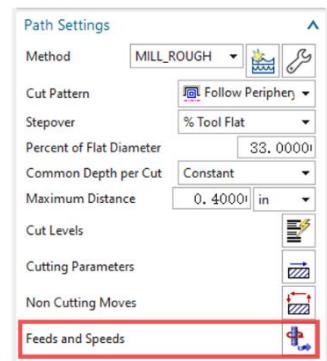
Cutting Speed: Steel: 50-100 in/min  
Aluminum: 400 in/min

The results that these equations give you are a good starting point. However, be aware that other factors, such as tool length, significantly small end mills, and other factors that may require you to change your feed or speed rates.

In order to err on the side of safety, always input feed and speed rates on the low end. Far less damage can occur with slower feed and speed rates than with higher ones.

Additionally, many machines feature override functions which can increase your federate during machining. If you're unsure about your feed and speed rates, always feel free to ask someone who has more experience with machining.

- Under the Path Settings section of the Cavity Mill Dialogue, the last option is the Feeds and Speeds item. Click the button to the right to enter the Feeds and Speeds dialogue.
- There are many ways to input information to obtain a feed and speed rate for an operation. The first button to 'set machining data' allows us to specify items for a library of feed and speed combinations with certain materials and tools. For now, we will take a more direct approach.
- You can see under Automatic Settings, NX is asking for the same inputs we have put into the previously listed equations. Enter 400 for our surface speed, and .005 for the Feed per tooth and hit the calculate button on their right. This causes the Spindle Speed and federate to set themselves to 1222 RPM and 12.22 IPM, respectively.



- Run the equations we have given to you, using the same inputs. Notice how the results are very similar. They are most likely different because NX uses a different method of calculating spindle speed and feed rate. You may choose either method to input a feed and speed rate.
- Hit OK and exit back to the Cavity Mill dialogue. Nothing we have edited in this section will alter the toolpath, but generating the toolpath now refreshes the ‘Time’ value under ‘Program Order View’. Because we have entered feed and speed, this value is now a good estimation of the machining time for this operation.

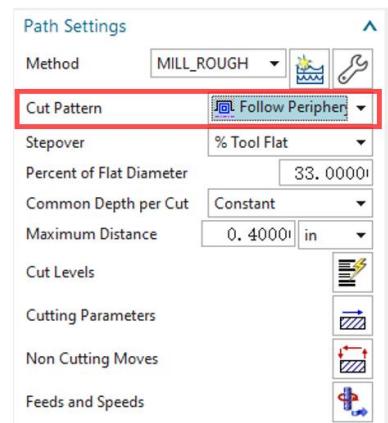
Name	Toolchange	Path	Tool	Tool Number	Time
NC_PROGRAM					02:51:08
Unused Items					00:00:00
PROGRAM					02:51:08
ROUGHING_1			EM_1.25	1	02:50:56

### Change the Cut Pattern and re-generate the toolpath.

The cut pattern in NX defines a general path the tool will attempt to follow when removing material. There are a huge number of ways NX has pre-programmed to attempt to remove material from a part, ranging from simple zig-zag patterns to much more complicated user defined paths.

The benefit to altering cut patterns is to simplify the toolpath and attempt to minimize the number of unnecessary movements by the tool, thus saving machining time. Altering cut patterns can also change the final shape of the part after the operation. Until now, we have been using the default option of ‘Follow Part’ method. We will change this to minimize the number of unnecessary tool movement.

- Under the Path Settings section of the Cavity Mill dialogue, there is an item labeled Cut Pattern. Under the dropdown menu, select ‘Follow Periphery’, rather than the default ‘Follow Part’.



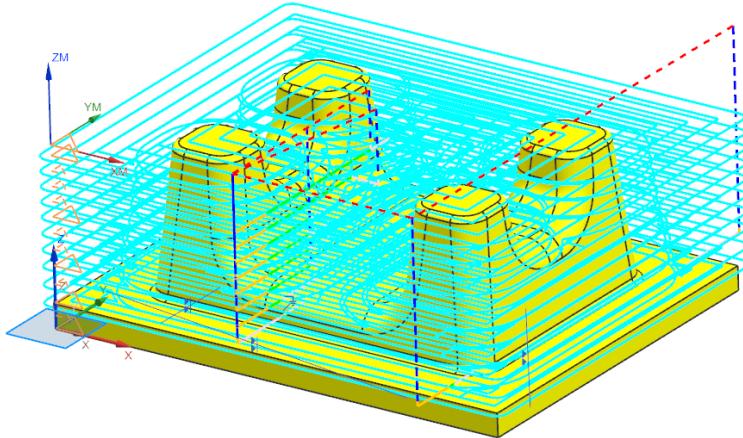
- Regenerate the toolpath using the Verify button. Now, the toolpath begins by following the periphery, or outer bounds of the blank, rather than tracing the path of the part during the entire milling operation. There are much fewer red lines, indicating the tool is retracting to the clearance plane far less than before, eliminating unnecessary movements. Also notice the estimated machining time for the new toolpath is about 20 minutes shorter.

The improved toolpath can be much more easily observed during 3D dynamic visualization.

Name	Toolchange	Path	Tool	Tool Number	Time
NC_PROGRAM					02:51:08
Unused Items					00:00:00
PROGRAM					02:51:08
ROUGHING_1			EM_1.25	1	02:50:56



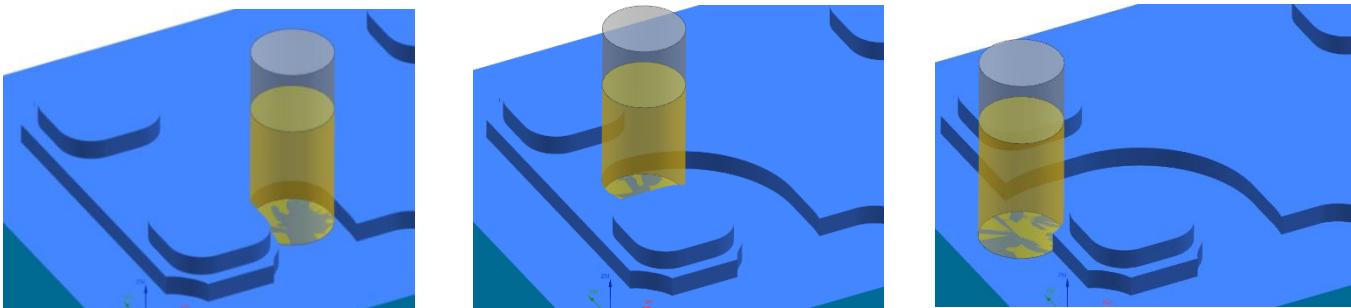
Name	Toolchange	Path	Tool	Tool Number	Time
NC_PROGRAM					02:29:29
Unused Items					00:00:00
PROGRAM					02:29:29
ROUGHING_1			EM_1.25	1	02:29:17



3. Enter the visualization environment, and preview the part using 3D dynamic mode.
  
  
  
4. Exit the visualization environment. Hit OK on the Cavity Mill dialogue. Observe how now the ROUGING\_1 operation appears in the Geometry View Operation Manager as a child of the MCS\_MILL and WORKPIECE items. This, like any other NX item, can be double clicked to edit the item.

#### Alternative Adaptive Milling Operation

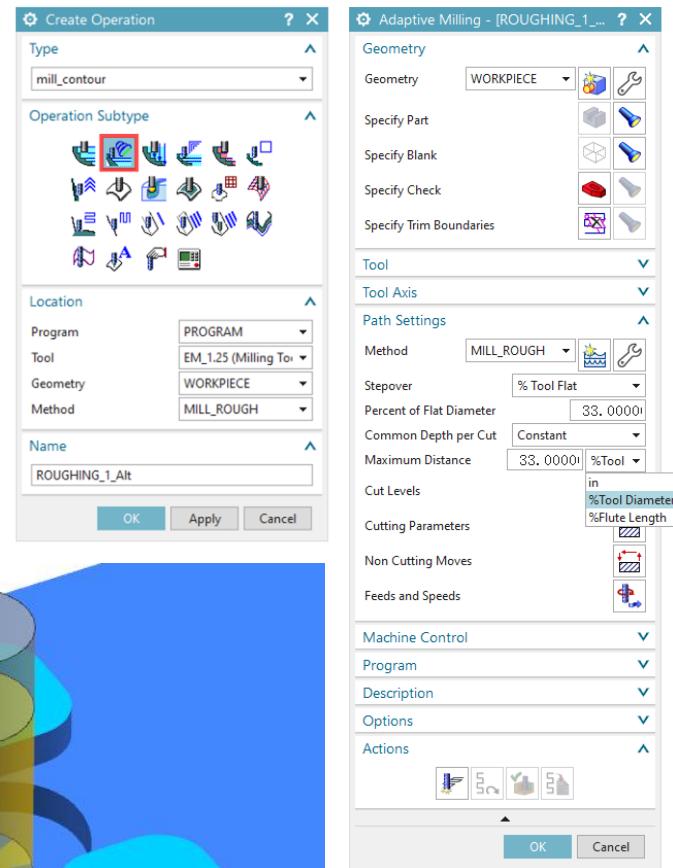
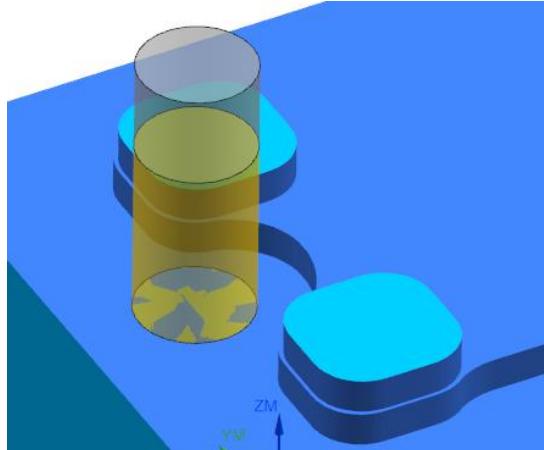
One important thing to note about the stepover setting in NX is that the percent of flat diameter set by the user is not absolute. Even though we have set the value to 33%, the tool path can sometimes result in movements with greater stepover (even 100% engagement!). If you pay close attention in 3D Dynamic, you can catch this potentially tool breaking mistake:



This issue often occurs when large tools have to move in between two extrusions. Sometimes changing the cut pattern can resolve this issue. However, with this specific model, you may notice that none of the available cut patterns can completely eliminate the 100% tool engagement. We can also reduce the feed rate and depth of cut to reduce the load on the tool, which is not ideal as it adds machining time.

Another solution is to use a different milling operation. ‘Adaptive Milling’, unlike ‘Cavity Mill’, maintains a consistent cutting engagement.

- Keep the ROUGHING\_1 operation for now, but delete its toolpath by right clicking the operation under Operation Navigator, select 'Tool Path' > 'Delete'. This tool path has already removed the blank material, so we need to delete it in order to see material removal in 3D Dynamics for the new operation.
- Create a new 'Adaptive Milling' operation with the same settings as the previous cavity mill operation.
- Set stepover to 33% tool flat, and depth of cut to 33% of the tool diameter. Also set the feed and speed to 400 sfm, 0.005 feed per tooth similar to the previous operation.
- There is no option for cut pattern. Generate the tool path. In 3D Dynamic, notice how the tool cuts the area between the extrusions in a circular pattern in order to maintain a consistent stepover.



- We will move forward with adaptive milling to avoid damaging the end mill. Exit Verify and click OK to save the operation. Delete the previous cavity milling operation.

We will now proceed to the addition of finishing operations. These will be the operations that will create the fine detail in the part, bringing us as close as possible to the final product.

## Create a Cavity Milling Finishing Operation.

Finishing operations are operations which cut material down to the final desired part, rather than leave material behind, as in a roughing operation. Unlike roughing operations, there will typically be more than one finishing operation since each will target a specific area of the part.

Part of the learning curve of NX Manufacturing is the ability to identify areas of the part which need to be further finished, and select the proper milling operation to remove that area of material. This walkthrough will guide you through a pre-determined path which has been shortened. You should anticipate spending a good deal of time examining your verified parts and picking out areas to machine further, then choosing the proper method.

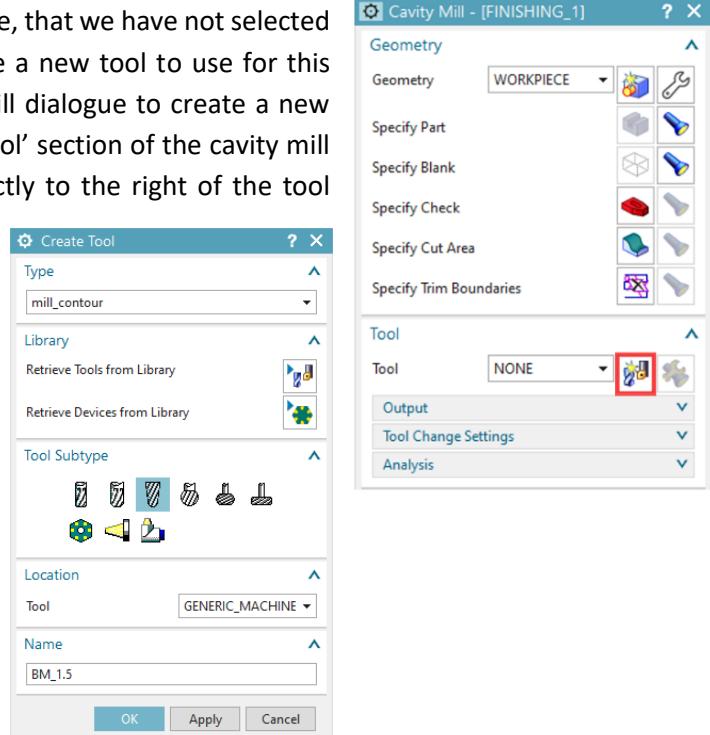
We will first attempt to remove the remaining material along the part, and choosing parameters to get us as close as possible to the final shape. After this operation, we will then be able to target specific areas with further finishing operations.

1. Create an operation by clicking the 'Create Operation' button in the toolbar. We will be creating a cavity milling finish operation which will remove the bulk of the remaining material. When the Create Operation window is displayed, choose the following options:

Type:	mill_contour
Operation Subtype:	Cavity Mill
Program:	Program
Tool:	None
Geometry	Workpiece
Method:	MILL_FINISH
Name:	FINISHING_1

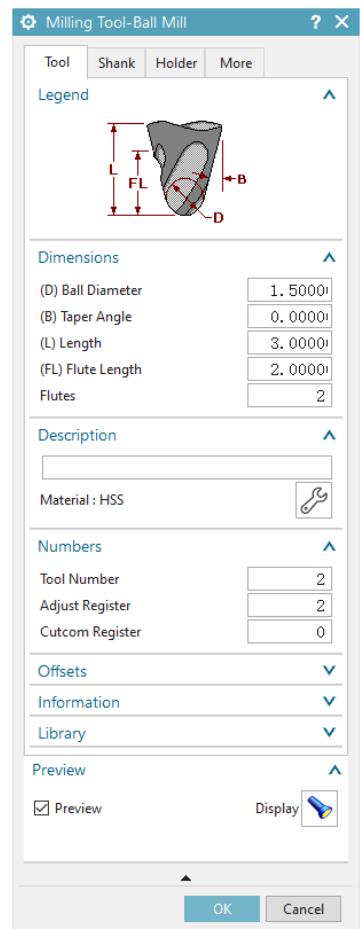
## Create a new tool within the Cavity Mill Dialogue

1. Click OK, and enter the Cavity Mill Dialogue. Take note, that we have not selected a tool to be used in this operation. We must create a new tool to use for this operation. However, we need not exit the cavity mill dialogue to create a new tool. We can create a new tool by expanding the 'Tool' section of the cavity mill dialogue, and selecting the only active button directly to the right of the tool dropdown menu.
2. The new tool dialogue now appears, just as if we had used the 'create tool' item on the toolbar. Select the third tool subtype to create a ball endmill, and name the tool BM\_1.5 and press ok.

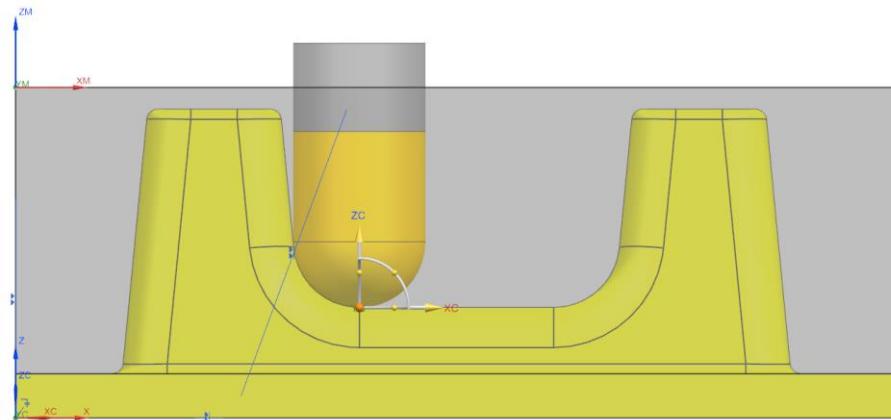


3. Enter the following parameters for the new tool:

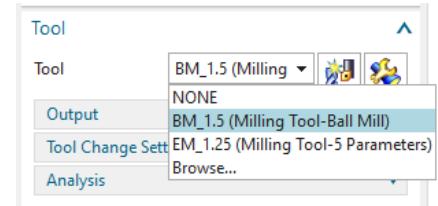
Diameter:	1.5 in
Length:	3.00 in
Flute Length:	2.00 in
Tool Number:	2
Adjust Register:	2



4. The choice of tool may not be apparent immediately. When selecting a tool finish a large surface which is not vertical, it is typically best practice to use a ball endmill of the largest diameter possible. This will put the flattest possible surface up against the sidewalls of our part, and create the flattest surface possible. (Imagine moving both a small and large diameter ball end mill around the walls of our part. The small end mill must take much smaller depths of cut to achieve a smooth surface, compared to the large tool.) You can check this by moving the tool close to the existing curves of the part. You can see the end mill matches the inner curves of the part, and will give the best finish along the steep walls.



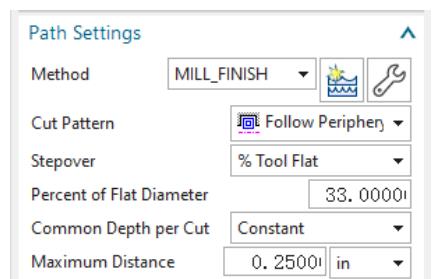
5. Click OK, and create the tool. You will re-enter the Cavity Mill Tool dialogue. You can now see that the tool we have just created is now set as the tool in the tool dropdown menu. You can expand this menu to see the other options available, including the tool used in the previous cavity milling roughing operation.



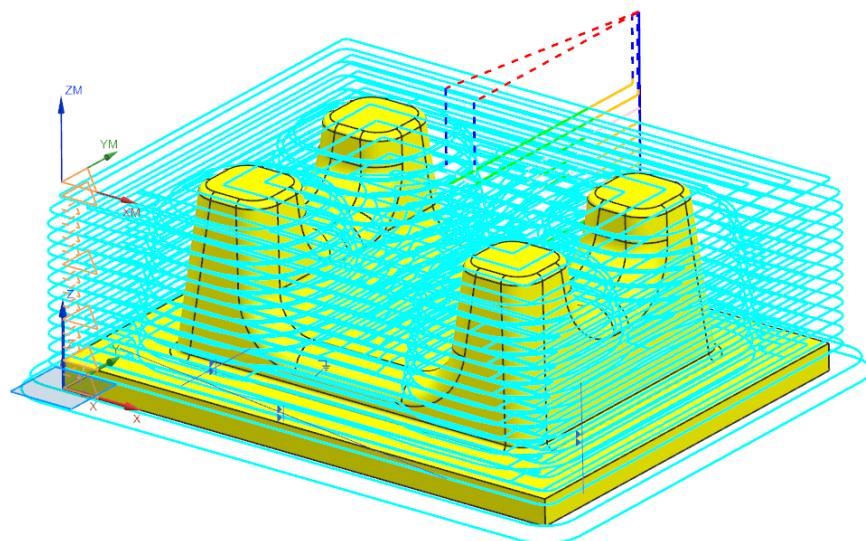
#### Change the stepover and cut pattern, and generate the toolpath

1. In the cavity mill dialogue, you can immediately change the stepover and cut pattern without entering any submenus. Change the following items:

Cut Pattern:	Follow Periphery
Percent of Flat Diameter:	33.0%
Common depth per cut:	Constant
Maximum Distance:	0.25 in



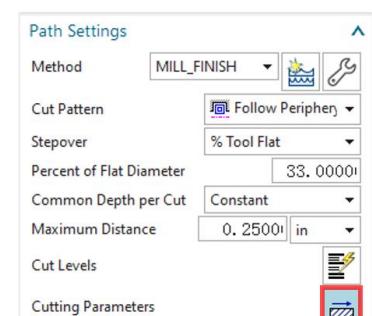
2. Generate the toolpath with the 'generate' button on at the bottom of the cavity mill dialogue. You can observe how the 'follow periphery' method has limited the number of tool retractions, and the stepover has been set to a small amount. However, there are two things that are incorrect.
  - a. The tool is cutting material which we should have already removed during our roughing operation. We must instruct NX to look to previous operations and inherit the part as it exists when this operation is executed.
  - b. The tool is attempting to cut around the base of the part, which is something we do not want. We must prevent the tool from ever passing below the lower face of our part.



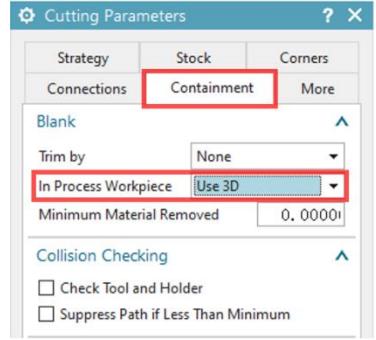
### Enable the In-Process Workpiece

While NX is a very advanced piece of software, it will not take certain actions unless told to. We must select the proper option in the cavity mill dialogue to tell NX to set the part blank as the workpiece as it exists up to that point in the manufacturing process.

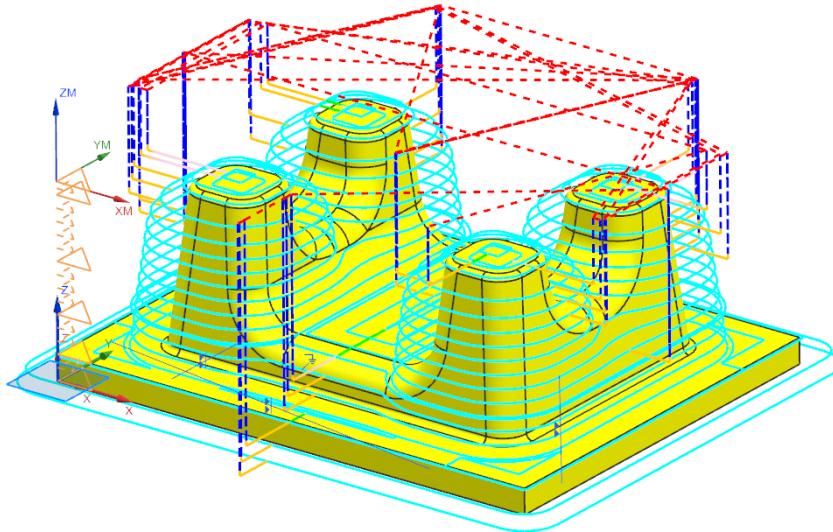
1. Enter the cutting parameters dialogue, which is under the path setting section in the Cavity Milling dialogue.
2. Go to the tab labeled 'containment'. This will contain the options to enable the in-process workpiece.



- Under the 'blank' section, the second item is the In-Process Workpiece option. Select 'Use 3D' to enable the in-process workpiece blank.



- Select OK, and re-generate (click generate) the toolpath. Observe now how the toolpath generation takes longer, since NX is running through all the previous operations to obtain the in-process workpiece. Once the toolpath has generated, you can now see how the tool only moves along the part, and does not attempt to cut the bulk of the material as it would in a roughing operation.

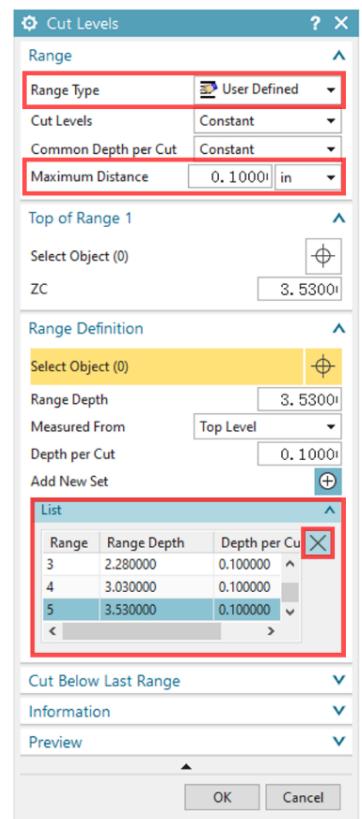


### Correct the base of the part, and set the depth of cut

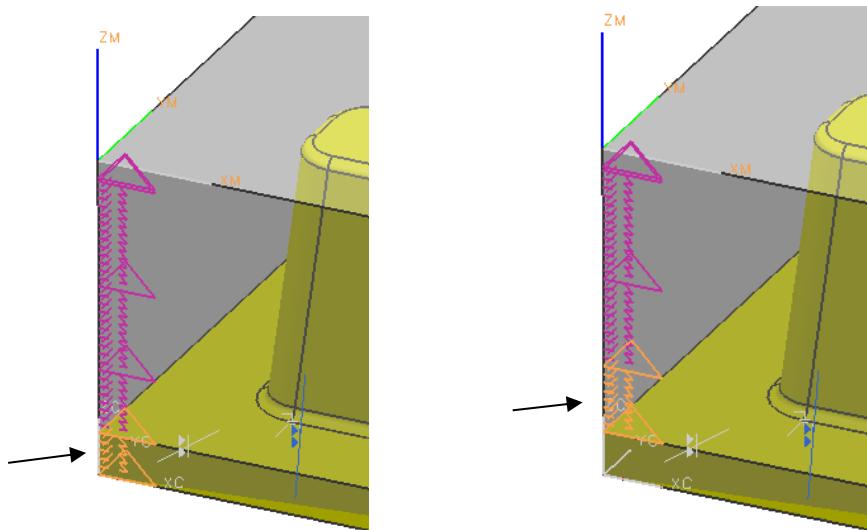
We will now eliminate the toolpaths below the lower flat of the part, and edit the depth of cut at the same time. This can all be accomplished via the cut levels dialogue.

Cut levels can be used to specify only a certain height that a tool will cut. We will use this operation later on to further narrow down the area an operation will cut.

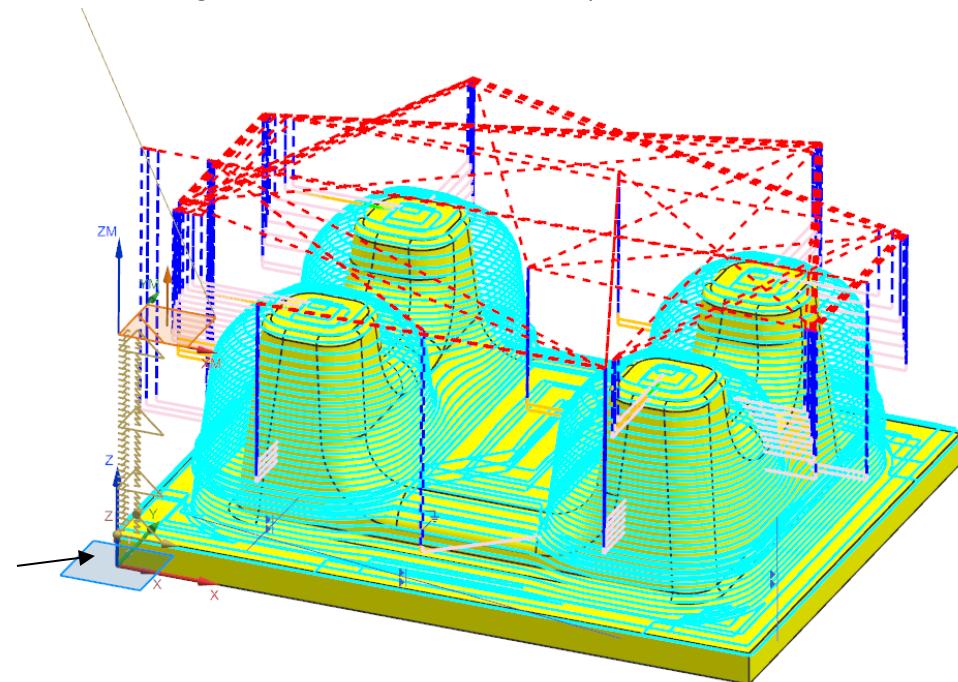
- Enter the Cut Levels dialogue under the Path Settings section of the Cavity Mill dialogue.
- Change the Maximum Distance to 0.1 in the 'Maximum Distance' parameter input.
- Select the second range type, the user-defined ranges. This will allow us to modify the ranges which have been created using the Auto Generate cut levels.



4. Under the Range Definition section, click List and scroll down to Range 5, the lower most level. Once selected, hit the black X button underneath the green arrows. This will delete this range. The highlighted range should now be above the bottom face of the part.

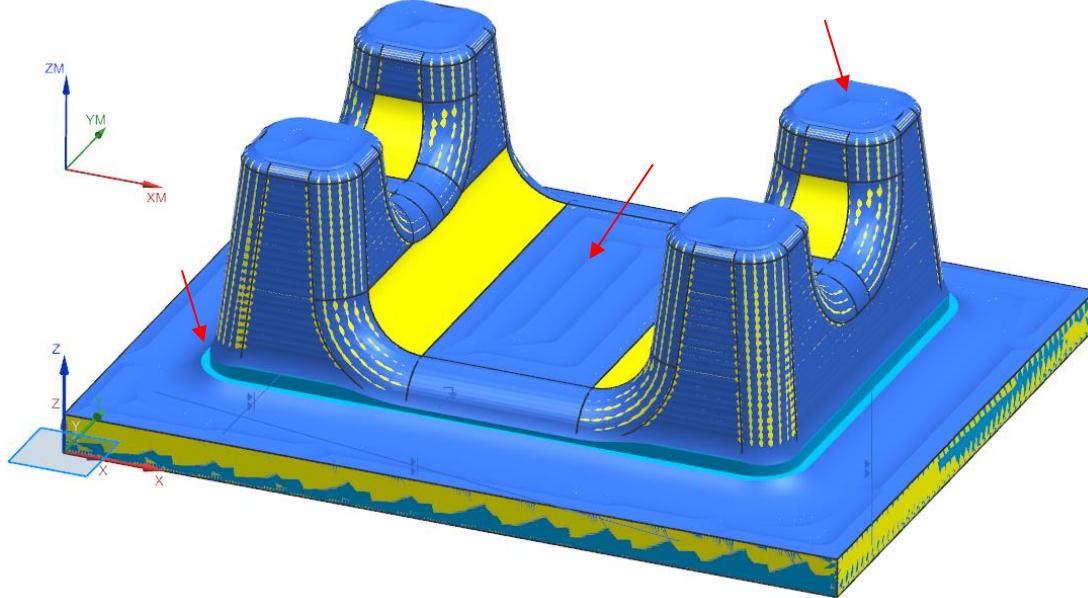


5. Hit OK on the Cut Levels dialogue to exit back to the cavity mill dialogue, and re-generate the part. Observe how now the toolpaths now try to not go below the bottom face of the part.



## Verify the toolpath and check for desired results

1. Verify the generated toolpath, and observe how the starting point is the blank from the previous operation. There are a few items to observe when looking at this generated part



- a. The walls appear to achieve a good result.
- b. The flat areas are not finished. This is a result of the ball endmill on a flat surface. We will need to add an additional finishing operation to address this.
- c. The bottom corner has not been fully machined. This is because the radius of the tool is too large to reach that corner. We will again need an additional operation to address this.

Note: You can explore different cut pattern options and make a decision based on surface finish and estimated machining time.

2. Hit OK on the visualization dialogue, and select OK on the cavity mill dialogue to create the operation.

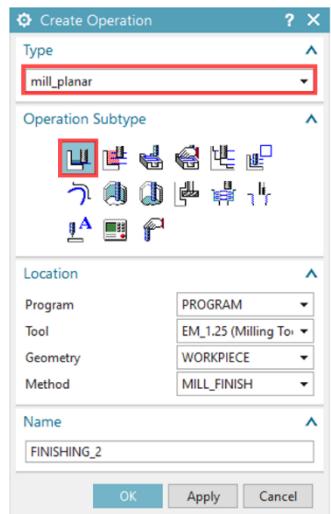
## Create an Area Milling Operation, and specify Cut Areas

So far, we have been using contour mill operations which have been using the part and blank to determine where to cut. We will now use an area milling operation, which falls under the category of planar milling. Planar milling is a more basic operation category, which only operates based on input from the part, and ignores the blank part completely. You will see how we now need to select the geometry we wish to machine, since we will not input the blank into the operation parameters.

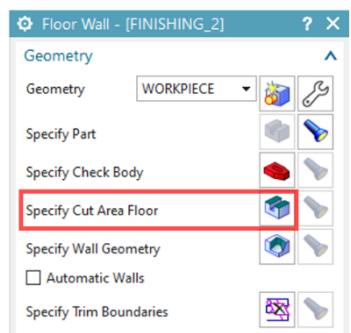
The area milling operation will be used to flatten the areas of the part which need it, including the center, edge, and top of the 4 posts.

1. Create a new operation via the button on the toolbar. On the Create Operation window, enter the following parameters:

Type: mill\_planar  
Operation Subtype: Floor and Wall  
Program: Program  
Tool: EM\_1.25  
Geometry: Workpiece  
Method: MILL\_FINISH  
Name: FINISHING\_2

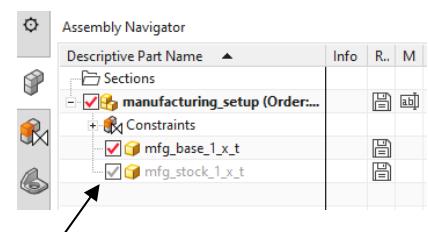


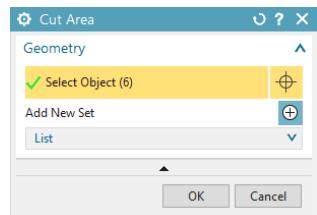
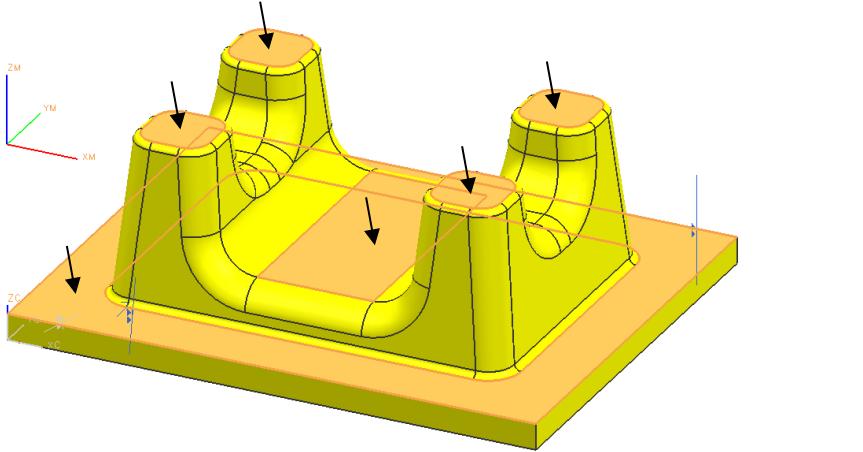
2. Select OK, and enter the Face Milling Area dialogue. The geometry section now lacks an area where you can specify the blank, and has been replaced by a 'Specify Cut Area' item. Click the button to the right of Specify Cut Area to instruct NX where we will be facing.



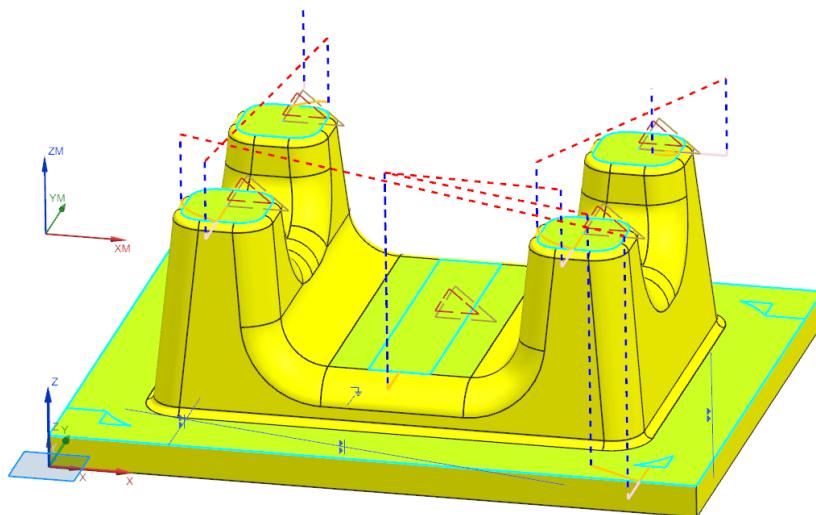
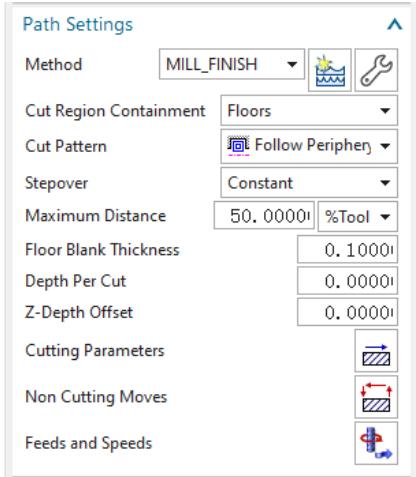
3. The Cut Area dialogue box will now appear. This is where we need to specify which areas of the part we will cut. You will need to select the flat faces of the part as shown below.

- a. If you cannot select these faces because the blank part is in the way, we can hide the blank to allow us to select these faces. Click the assemblies tab to the right of the operation navigator to switch to the assembly tree, and hide the blank by un-checking the box next to your blank part. You should then be able to switch back to the manufacturing tab and select the proper faces.

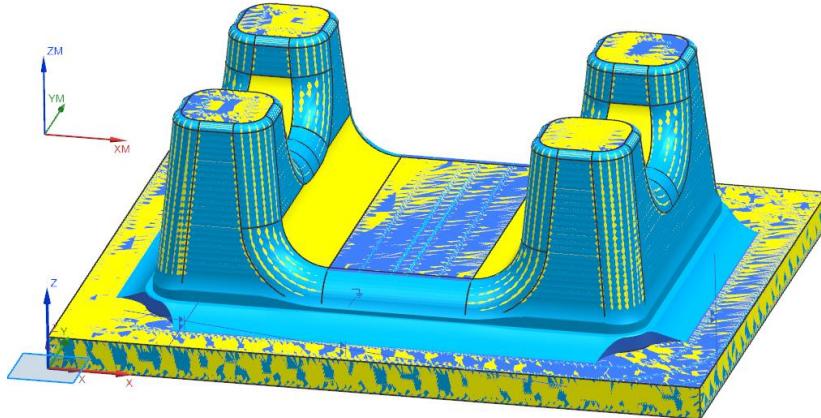




4. Click OK to exit the Cut Area dialogue. Change the stepover – Maximum Distance variable to 50% and Cut Pattern to Follow Periphery. Recall how we will be using the same tool as in our first roughing operation, since this tool has a flat end. We can take a larger stepover cut since we will be removing such a small depth of material.
  
5. Generate the toolpath with the button at the bottom of the Face Milling Area dialogue. You can now see how the tool only cuts along the faces we have specified, and ignores any other features of the part. However, you may notice how the paths still recognize where the rest of the part is, and avoids collisions. Planar milling operations are less advanced than cavity milling, but they can still recognize other areas of the part.



6. Preview the path using 3D dynamic visualization. You can now observe how NX uses different colors to show different operations acting on the part.



- Click OK to exit the visualization environment, and click OK to create the face milling operation.

### Finish the lower corner of the part

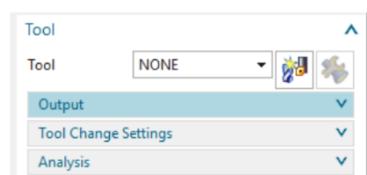
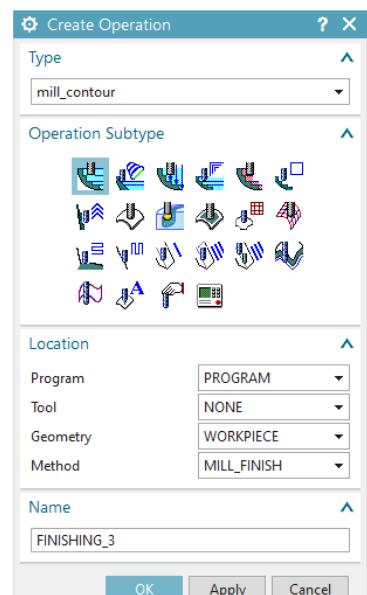
We will now add an operation to finish the lower corner of the part. In this operation, we will heavily modify a large operation to specify where it will machine, using a combination of cut levels, and in-process workpiece specifications.

For this operation, we will need to add a new tool with a radius small enough to fit into this corner. We will also attempt to create a tool which can fit into future areas which require finishing.

- Create a new operation with the 'Create Operation' button on the toolbar. We will be adding a finishing cavity mill operation, which we will narrow down to the required area. Enter the following parameters:

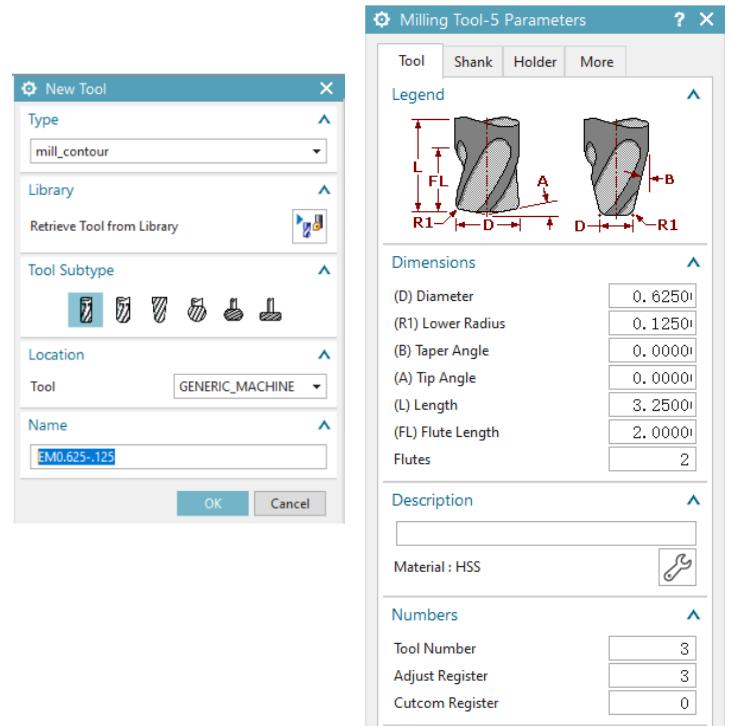
Type:	mill_contour
Operation Subtype:	Cavity Mill
Program:	Program
Tool:	None
Geometry:	Workpiece
Method:	MILL_FINISH
Name:	FINISHING_3

- We now need to add a tool which will fit into the lower edge of the part. You may use the measure tool in NX to query the radius of the corner, which you can see is 0.125". Create a new tool via the new tool button in the tool section of the Cavity Mill Dialogue.



3. Select the first tool subtype to create a flat endmill. Name the tool EM0.625-.125. Press ok. Enter the following parameters for the new tool:

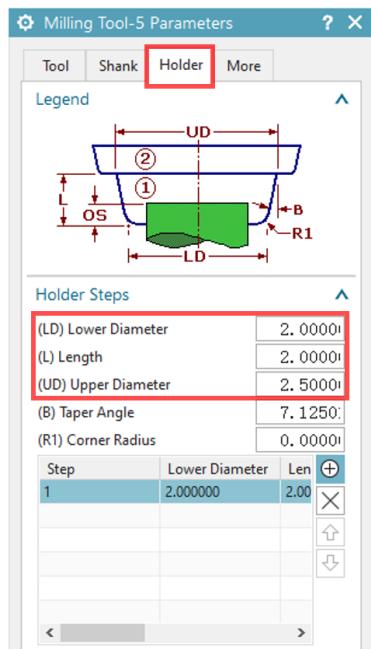
Diameter: 0.625 in  
 Lower Radius: 0.125 in  
 Length: 3.25 in  
 Flute Length: 2.00 in  
 Tool Number: 3  
 Adjust Register: 3



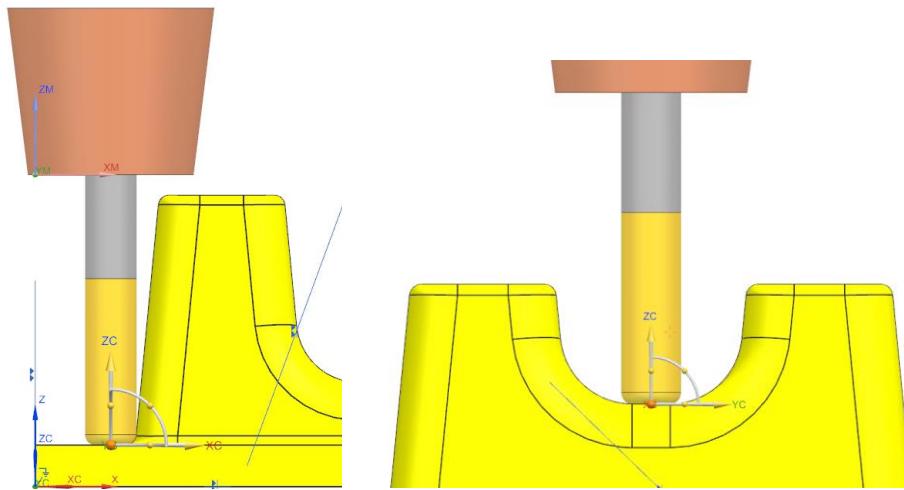
4. We will now add a holder section to our tool. We will mainly put this in since we are using a smaller diameter tool, which will require the tool length to be larger than normal when using a tool this diameter. By placing tool holder geometry in the operation, we can check to make sure we do not collide with the part at any point. This addition of geometry is done through the same create tool dialogue.

5. Select the third tab on the top of the Milling Tool-5 Parameters window, named 'Holder'. This will bring us to a new dialogue to enter a holder geometry. Enter in the following parameters to the section labeled 'Holder Steps'.

Diameter: 2.00 in  
 Length: 2.00 in  
 Upper Diameter: 2.50 in



6. You should see the holder section appear above the tool in the modeling window. We can continue to add steps by clicking the green highlighted button under the Holder Steps parameters, next to the large window with Step and Diameter headers.
7. While still in this dialogue, move the tool to the two points shown below, with these orientations. You can see how this tool should be able to machine both the lower corner of the part, and the flat section between the pairs of posts.

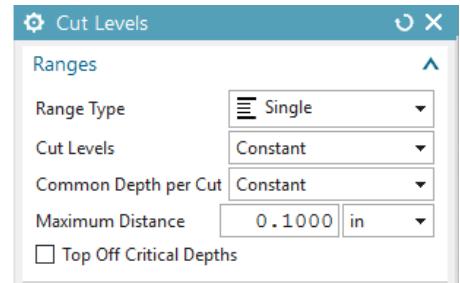


- Click OK to create the tool, and exit back to the Cavity Mill Dialogue.

#### Specify the desired area using cut levels and the in-process workpiece

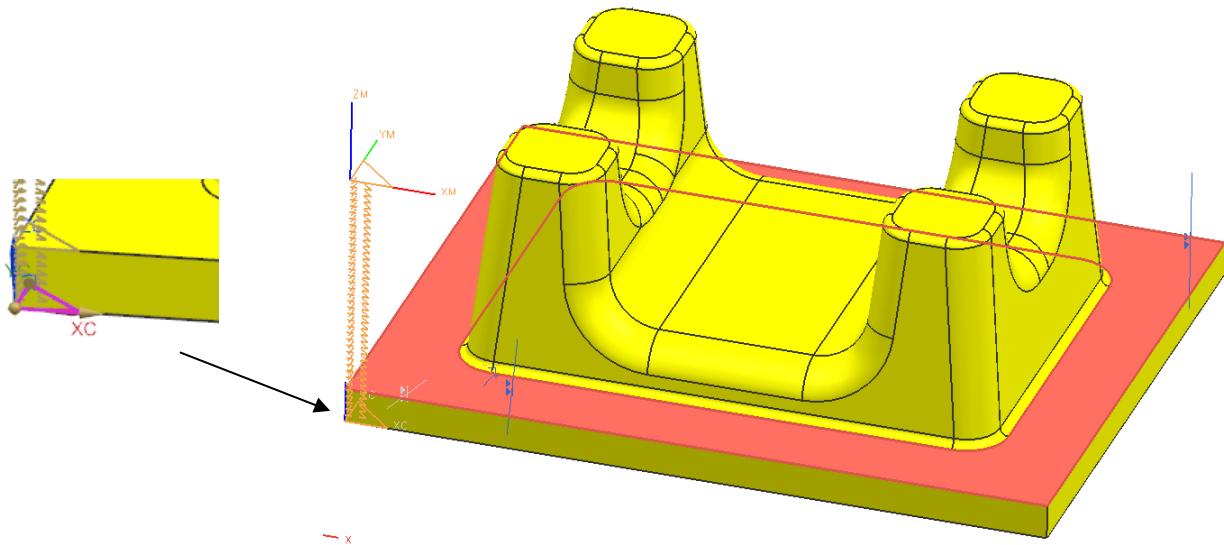
We will now set the operation to cut only at a certain level by narrowing the cut levels to only a certain range using the single range input type.

- Enter the cut levels dialogue via the button underneath the path settings in the cavity mill dialogue.

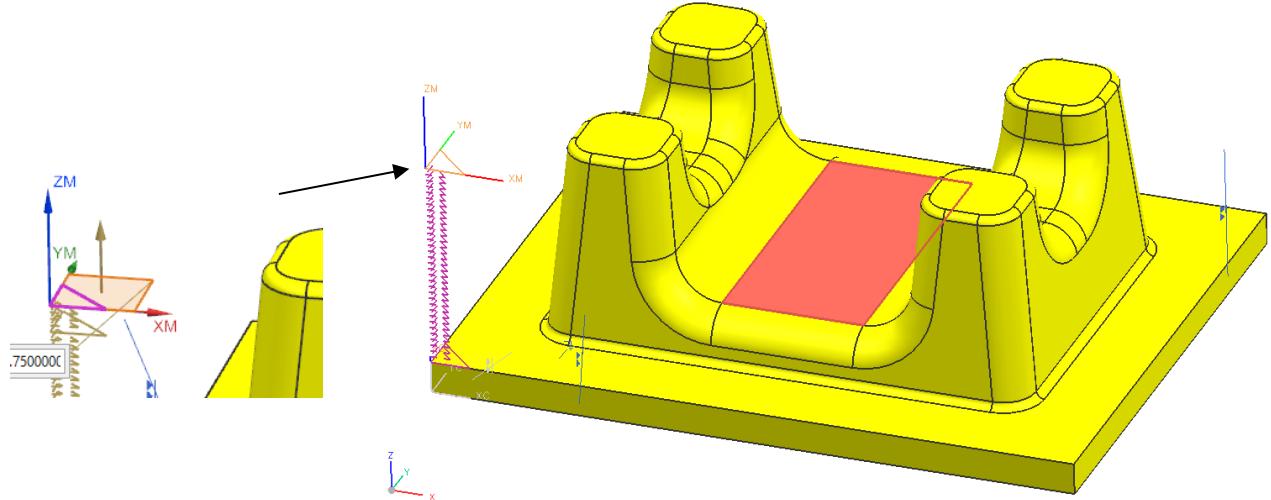


- Select the third range type, Single. Input 0.1 inches as the Maximum Distance.

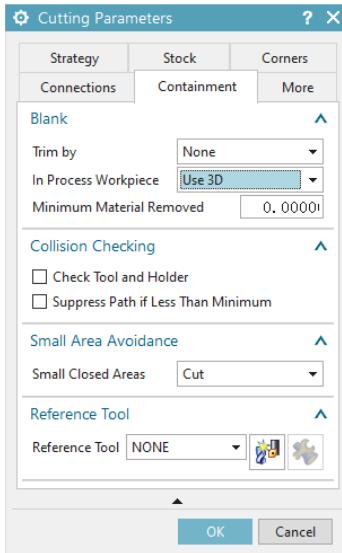
- Click the cut triangle and select the face as shown below. This should move the bottom of the cut level indicator up to this level.



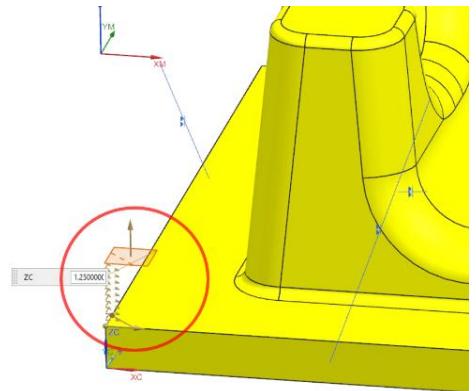
4. Click the appropriate triangle. Click the face shown below. This will bring the top of the cut level indicator down to this plane.



5. Observe how now the cut level indicator only shows cut levels between the two selected planes. This indicates we will only create toolpaths in this range. Select OK to accept the cut levels, and exit back to the Cavity Mill dialogue.

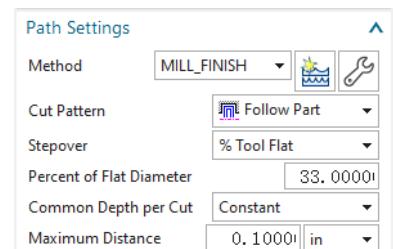


6. Activate the in-process workpiece by entering the cutting parameters dialogue under the path settings in the Cavity Mill dialogue. Go to the tab labeled 'Containment', and select the In-Process Workpiece dropdown menu variable to 'Use 3D'.

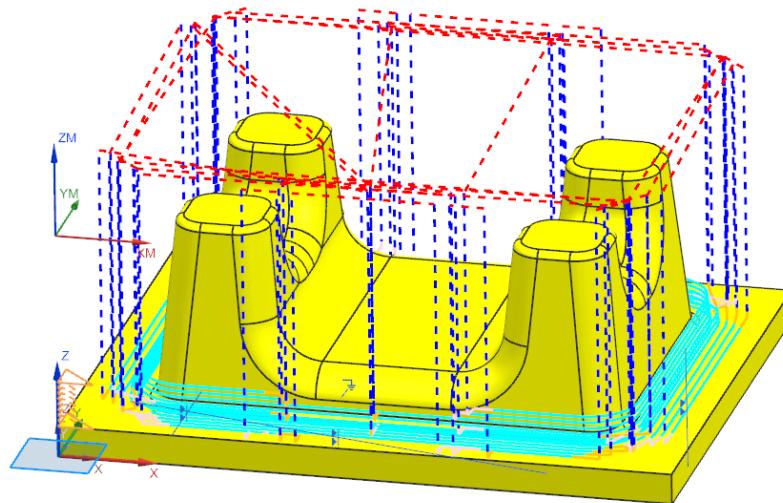


7. Click OK to exit back to the Cavity Mill dialogue. Change the following parameters in Path Settings. We want the depth of cut to match that of FINISHING\_1 operation for a consistent surface finish.

Cut Pattern:	Follow Part
Percent of Flat Diameter:	33%
Common Depth per Cut:	Constant
Maximum Distance:	0.1 in



8. Generate the toolpath with the 'Generate' button at the bottom of the Cavity Mill dialogue. Click OK to create the operation.

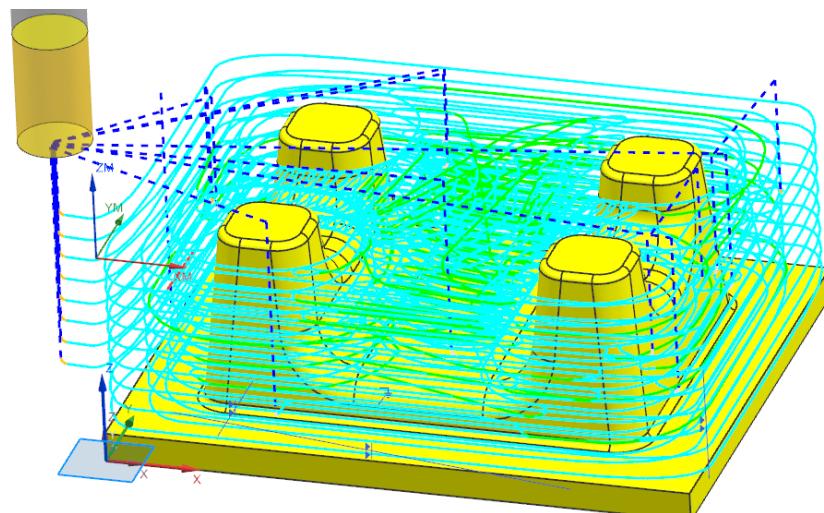
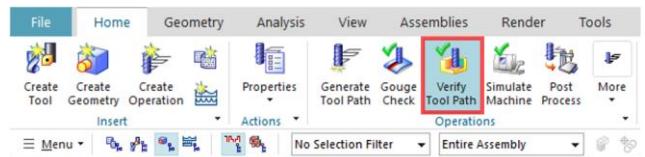


### Verify the entire manufacturing program

We have been using the visualization verification tool to so far observe only specific operations. We can also preview the entire operation using the same tool, but opened in a different manner.

1. With no dialogue windows open, select the 'Workpiece' item in the operation manager geometry view. By selecting this item, we are indirectly selecting all the operations that are this item's children. If we were to select an actual operation, we would only be acting on that one operation.
2. Click the Verify Tool Path button on the toolbar. This opens the same verification tool we have been using previously, however since we have entered the tool with the 'Workpiece' item selected, we will preview the entire part.

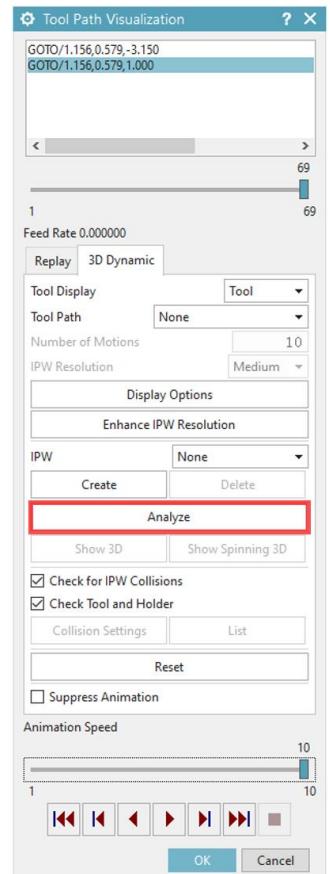
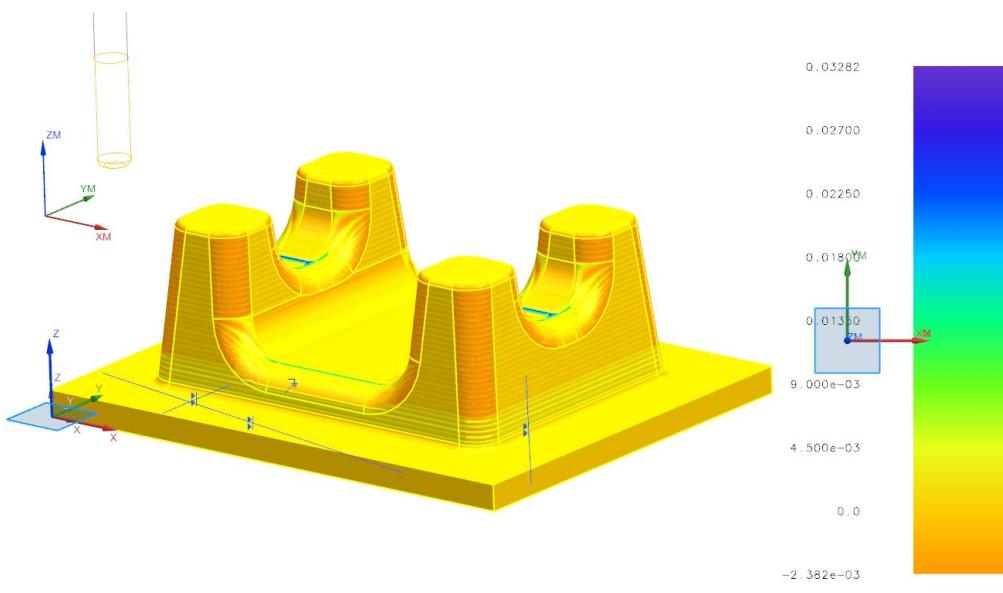
Name	Path	Tool
GEOOMETRY		
Unused Items		
MCS_MILL		
WORKPIECE		
ROUGHING_1	✓	EM1.25-.25
FINISHING_1	✓	BALL_NILL...
FINISHING_2	✓	EM1.25-.25
FINISHING_3	✓	EM0.625-1...



3. Preview the entire manufacturing program using the 3D Dynamic Preview. While work on your own project later on, you should use this feature to review all the toolpaths to make sure they are safe for manufacturing.

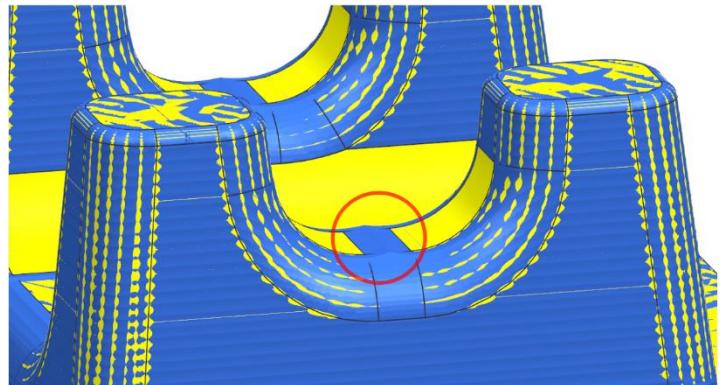
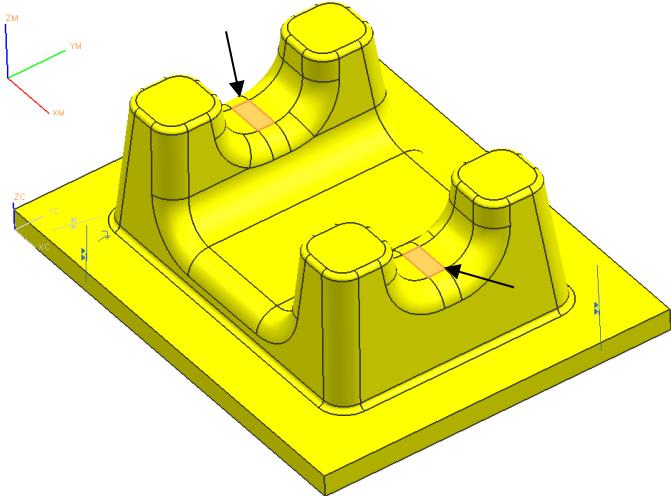


4. Another powerful tool you can use under the 3D Dynamic tab is 'Analyze'. 'Analyze' allows you to analyze the difference in thickness between the machined part and the original CAD. Use this feature to review the final surface finish. Notice how we have almost no leftover blank material except the two sections in between the extrusions.



### **Homework: Add the last finishing operation.**

The final remaining area to be finished is the area in between the pairs of posts. You should add an operation to finish this section of the part, and regenerate the entire toolpath to complete the entire part. You may need to add a new tool to fit into these areas.



### **Homework: Set all the Feed and Speed Rates**

You may have noticed, that other than the first operation, we have not set any feed or speed rates. You must use the equations given to you to fill in the feed and speed rates for each of the operations we have created, including the previous homework item to finish the last area of the part.

$$RPM = \frac{Cutting\_Speed \times 4}{Tool\_Diameter}$$

$$Feedrate = \frac{Feed}{Flute} \times RPM \times \# flutes$$