

CHAPTER 9 – MANUFACTURING

As we discussed in Chapter 1 about the product realization process, the models and drawings created by designers will undergo manufacturing processes to get to the finished products, which is the essence of CAD/CAM integration. The most widely and commonly used technique is to generate control codes for CNC machines to mill the desired part. This technique reduces the amount of human programming in creating CNC codes, which facilitates the designers to create products with complex geometries. In this chapter, we will cover the *Manufacturing Module* of NX 12 to generate CNC codes for *3-Axis Vertical Machining Centers*. This module allows us to program and do some post-processing on drilling, milling, turning and wire-cut EDM tool paths.

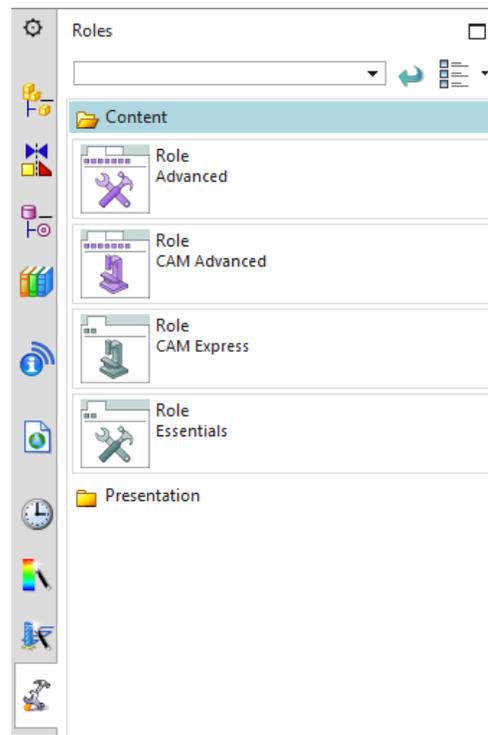
9.1 GETTING STARTED

A few preparatory steps need to be performed on every CAD model before moving it into the CAM environment. In this chapter, we are going to work with a model that was finished in the previous exercise problem. All the units are followed in **millimeters** in this model and manufacturing of the component.

Before getting started, it would be helpful if you can get into a *CAM Advanced Role*. To do this, go to the *Roles* menu on the *Resource Bar*. Click *Content* and a list of choices will pop up in which the *CAM Advanced* role can be seen as shown in the figure.

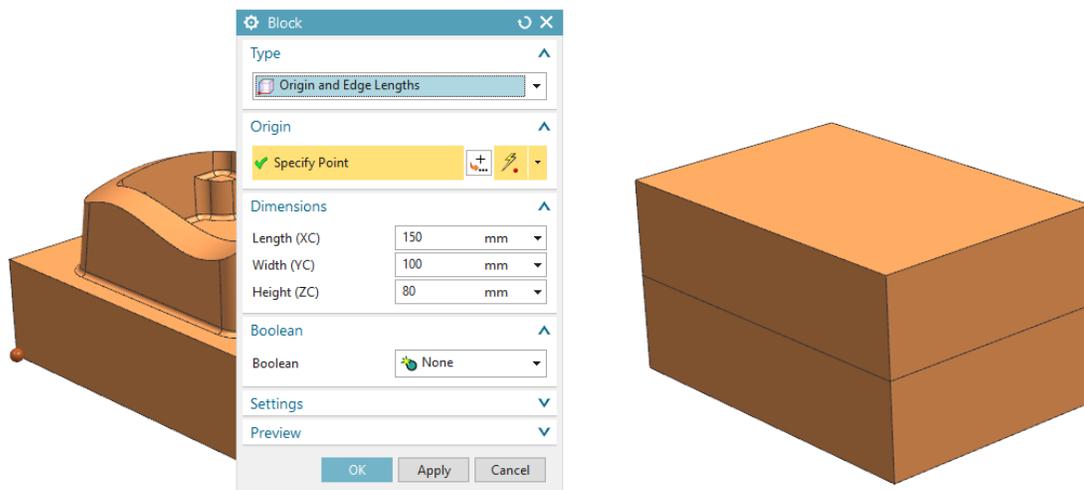
9.1.1 Creation of a Blank

After completing the modeling, you should decide upon the raw material shape and size that need to be loaded on the machine for the actual fabricating. This data has to be the input in NX 12. It can be achieved in two ways. The first method is creating or importing the model of the raw material as a separate solid in the same file and assigning that solid as the *Blank*. The second method is letting NX decide the extreme dimensions of the designed part and



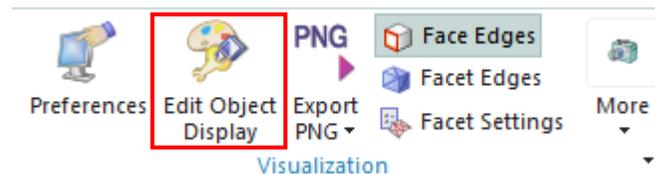
some offset values if needed. The later method allows a quick way of assigning the size details but it can only be used for prismatic shapes.

- Open the file **Die_cavity.prt** for the exercise problem in Chapter 4
- To create a *Blank*, **Insert** a block with the following dimensions
Length = **150** mm
Width = **100** mm
Height = **80** mm
- For the **Origin** option, choose the lower most corner of the base block, so that the new block created can wrap up the whole previous model as shown below

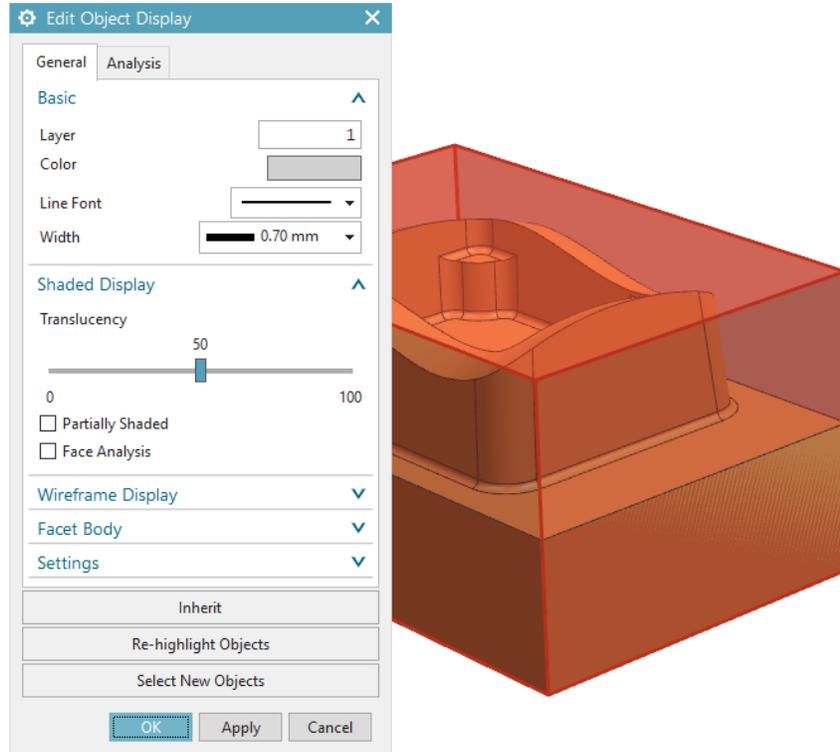


This block encloses the entire design part so we need to change the display properties of the block to have a better visualization.

- Click on the **Edit Object Display** icon in the **Visualization** group of the **View** tab



- Select the block you just created and click **OK**
- When a window pops up, change the display **Color** and change the **Translucency** to **50**
- click **OK**

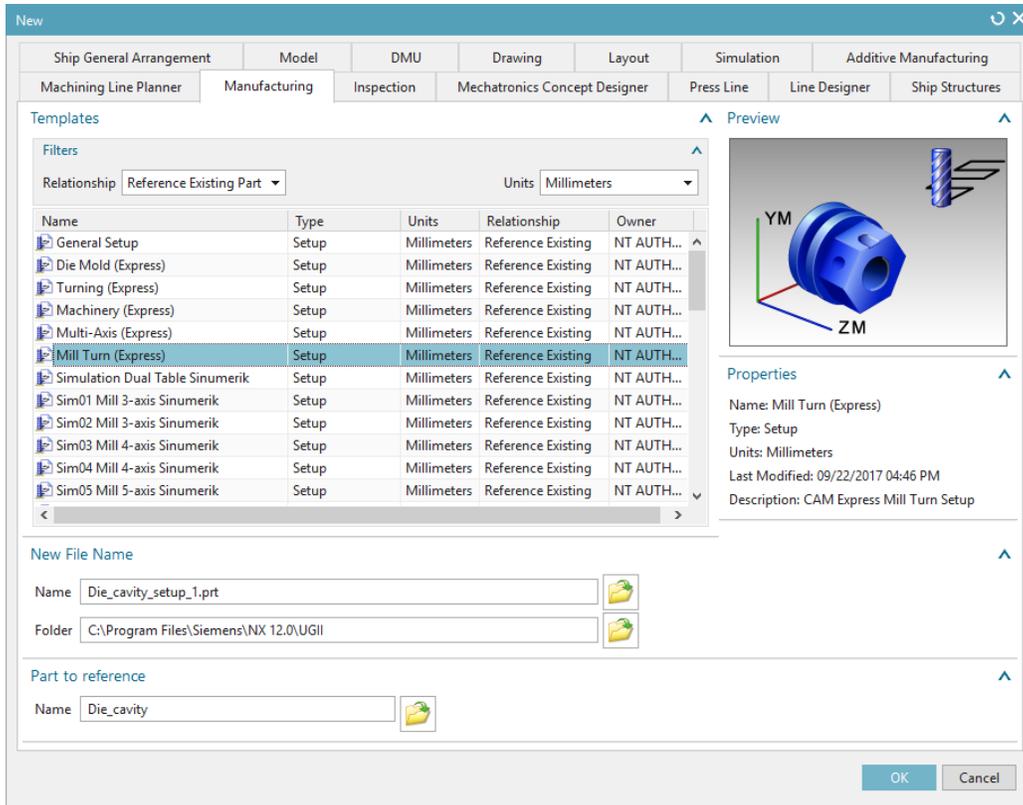


- **Hide** the block you just created by right clicking on the block in the **Part Navigator** and click **Hide** (you may need to use **Select from List...** to choose the blank). This will make the raw block disappear from the working environment.

9.1.2 Setting Machining Environment

Now we are set to get into the **Manufacturing** module.

- Select **File** → **New** → **Manufacturing** → **Mill Turn**
- There are many different customized CAM templates available for different machining operations. Here, we are only interested in the milling operation.



9.1.3 Operation Navigator

As soon as you get into the *Manufacturing* environment, you will notice many changes in the main screen such as new icons that are displayed.

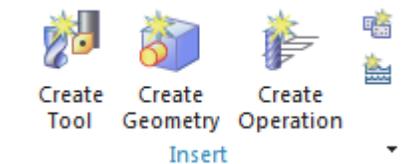
- Click on the **Operation Navigator** tab on the left on the **Resource Bar**

The *Operation Navigator* gives information about the programs created and corresponding information about the cutters, methods, and strategies. A list of programs can be viewed in different categorical lists. There are four ways of viewing the list of programs in the *Operation Navigator*, which are *Program Order view*, *Machine Tool view*, *Geometry view* and *Machining Method view*.

- Click on **Geometry View**  |   

9.1.4 Machine Coordinate System (MCS)

- Click on the **Create Geometry** icon in the **Insert** group to initiate setup for programming

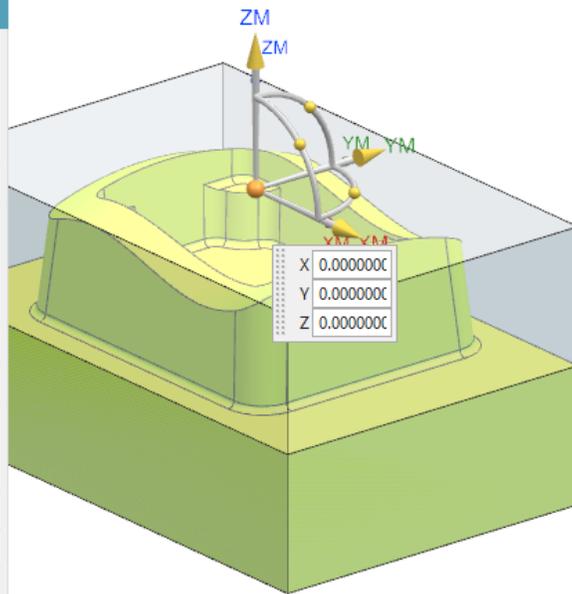
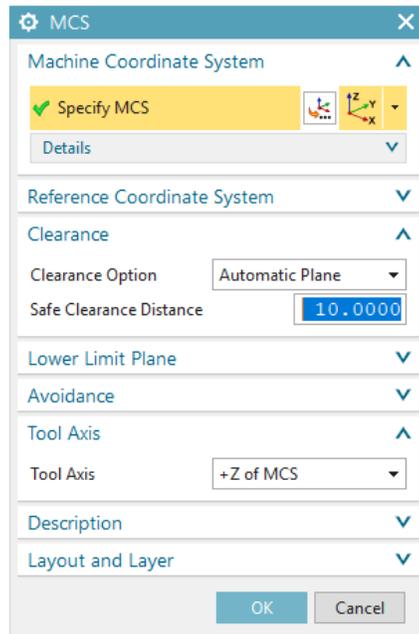
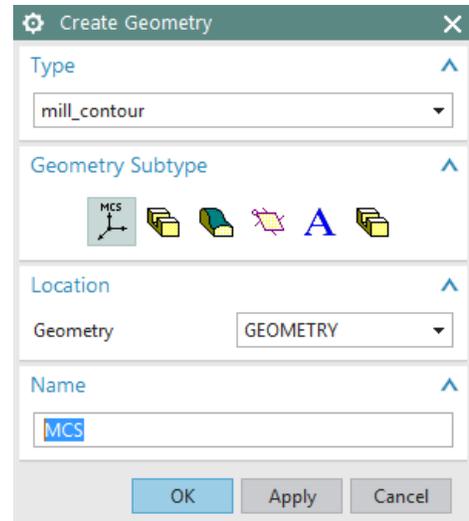


You will see a *Create Geometry* popup dialog.

- Click **OK**

Another popup window will allow you to set the *MCS* wherever you want. By default, NX 12 takes the original Absolute-CS as the MCS.

- Click **OK** to select the default choice as the **MCS**

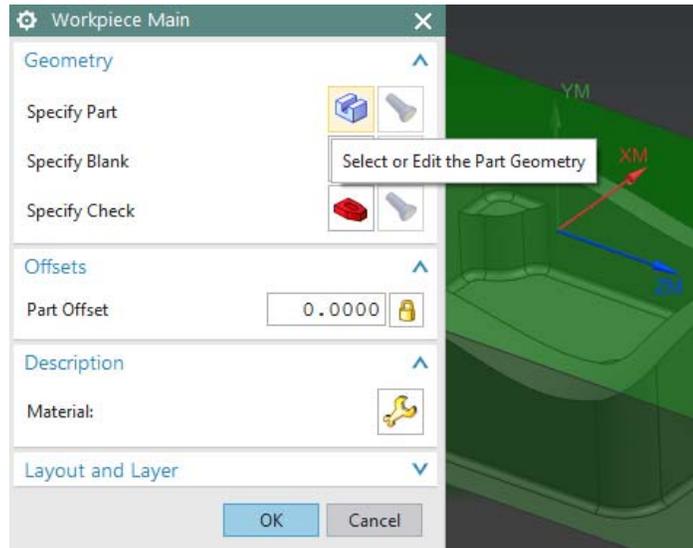


9.1.5 Geometry Definition

- Click on **Geometry View**
- Expand **MCS_MAIN_SPINDLE** by clicking on the '+' sign in the **Operation Navigator**

- Double click on **WORKPIECE_MAIN** in the **Operation Navigator**. If you do not see it, try clicking on related '+' signs

The popup window *Workpiece Main* appears. This is where you can assign the *Part* geometry, *Blank* geometry, and *Check* geometry.



- Click on the **Part** icon
- Select the design part and click **OK**

Now we will select the *Blank* Geometry.

- Click on the **Blank** icon

This will open the *Blank Geometry* Window. As mentioned earlier there are several ways to define the *Blank*. You can use a solid geometry as the *Blank* or can allow the software to assign a prismatic block with desired offsets in the X, Y, and Z directions. As we have already created a block, here we can use that as the *Blank* geometry.

- Click on the **Block** and press **OK**

Now we are finished assigning the *Part* and *Blank* geometries. Sometimes it may be required to assign *Check* geometry. This option is more useful for shapes that are more complex or 5-Axes milling operations where the tool cutters have a higher chance of dashing with the fixtures. In our case, it is not necessary to assign a *Check* geometry.

- Click **OK** to exit the **Workpiece Main** window

9.2 CREATING OPERATION

9.2.1 Creating a New Operation

The manufacturing setup is now ready for us to work further with *Programming Strategies*. There are many different manufacturing strategies involved in programming and it takes practice to know which one is optimal. Here, the basic guidelines are given for the most widely and frequently used

strategies. This chapter will also cover some important parameters that are to be set for the programs to function properly.

- Click on the **Create Operation** icon in the toolbar

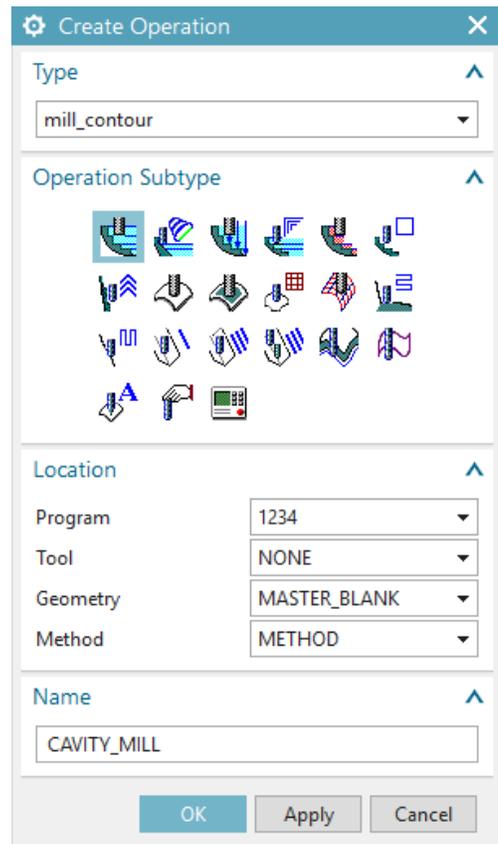


The *Create Operation* window will pop up.

- Make sure the **Type** of Operation is **Mill_Contour**

There are a set of different subtypes under *Mill-Contour*, namely *Cavity Mill*, *Z-Level Follow Cavity*, *Follow Core*, *Fixed Contour*, and so on. These different subtypes are used for different situations and profiles of the design part. As mentioned before, how you select a strategy for a certain situation depends on your knowledge and experience.

- Click on the **Cavity_Mill** icon at the top left as shown in the figure
- Choose the **Program** as **1234**
- Change the **Geometry** to **WORKPIECE_MAIN**
- Click **OK**



The program parameters window with *Cavity Mill* in the title bar will pop up. On this window, you can set all the parameters for this program. A brief introduction on every important parameter and terminology will be given as we go through the sequence.

9.2.2 Tool Creation and Selection

One of the most important decisions to make is to select a tool with the right shape and size to use. Before starting with the *Tool parameter* settings, we must first know about the types of *Tool cutters*. The *Milling Tool Cutters* are categorized into three forms of cutters. Hence, when selecting a cutter, it is important to take into consideration the size, shape, and profiles of the design parts. For example, if the corner radius of a pocket is 5 mm, the pocket should be finished by a cutter

with a diameter less than or equal to 10 mm. Otherwise it will leave material at the corners. There are other special forms of cutters available in markets that are manufactured to suit different needs.

Flat End Mill Cutters

These cutters have a sharp tip at the end of the cutter as shown in the figure below. These cutters are used for finishing parts that have flat vertical walls with sharp edges at the intersection of the floors and walls.



Ball End Mill

These cutters have the corner radii exactly equal to half the diameter of the shank. This forms the ball shaped profile at the end. These cutters are used for roughing and finishing operations of parts or surfaces with freeform features.



Bull Nose Cutters

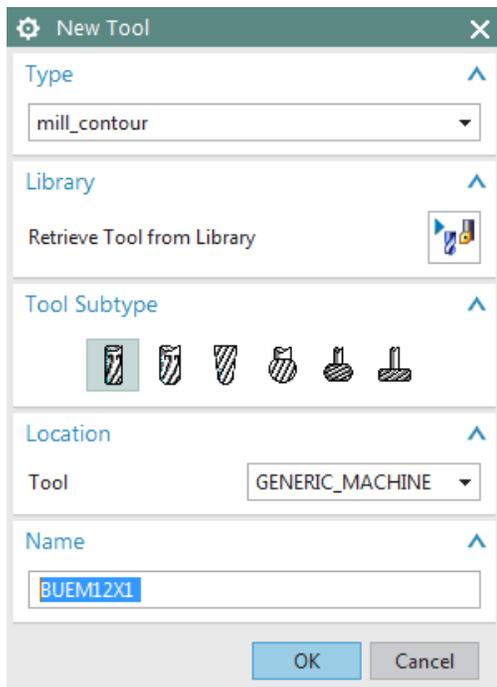
These cutters have small corner radii and are widely used for roughing and/or semi-finishing the parts as well as for finishing of inclined and tapered walls.



The cutter that we are going to use to rough out this huge volume is BUEM12X1 (Bullnose End Mill with 12 diameter and 1 corner radius).

- In the **Cavity Mill** popup menu click on the **Create New** button in the **Tool** dialog box

- Click **New**
- On the **New Tool** window, select the **Mill** icon

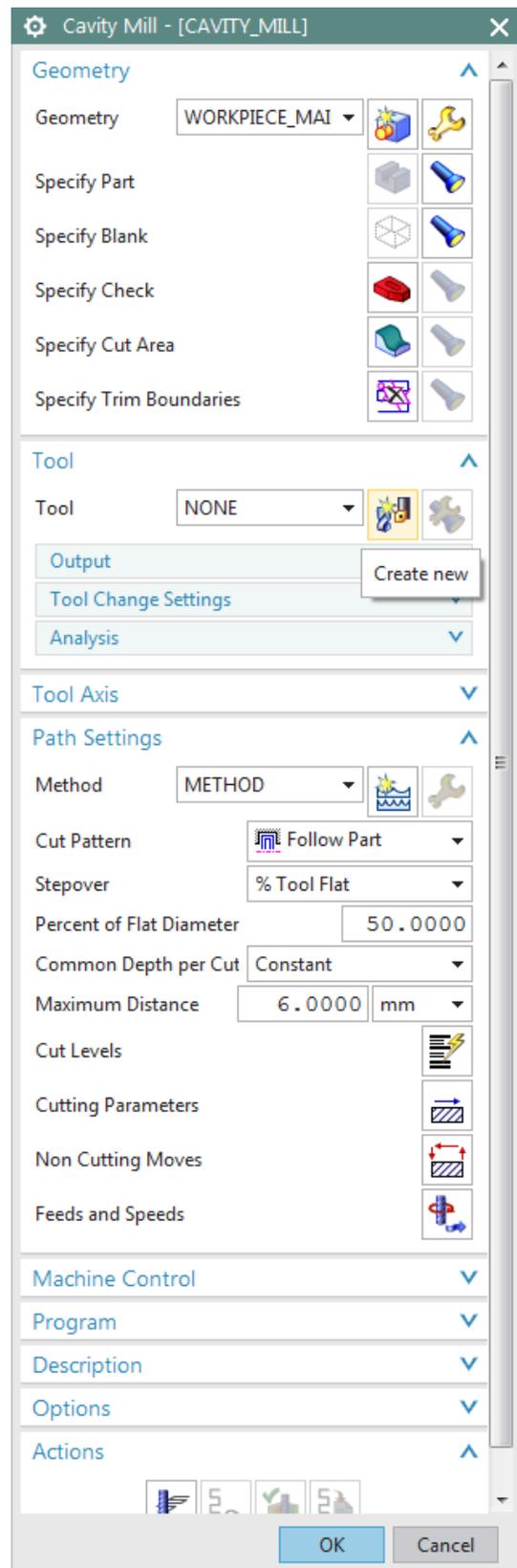


- Type in **BUEM12X1** as the **Name** and click **OK**

This will open another window to enter the cutter dimensions and parameters. You can also customize the list of tools that you would normally use and call the predefined cutters from the library.

- Enter the values as shown in the figure below

Dimensions	
(D) Diameter	12.0000
(R1) Lower Radius	1.0000
(B) Taper Angle	0.0000
(A) Tip Angle	0.0000
(L) Length	75.0000
(FL) Flute Length	50.0000
Flutes	2

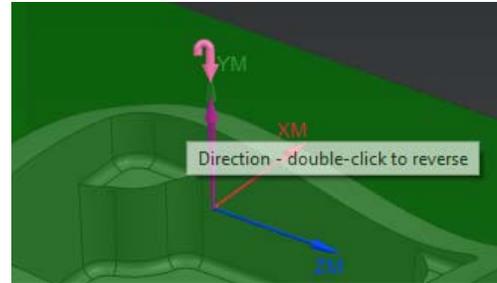
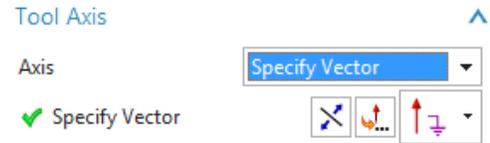


- Click **OK**

9.2.3 Tool Path Settings

Make sure that the *Tool Axis* is perpendicular to the top surface on the block.

- Click on **Tool Axis** and choose **Specify Vector**
- Select the appropriate axis as shown
- In the **Cavity Mill** menu click on the **Path Settings** option



There are different **Cut Patterns** in which the tool can move. The following is a description of each.



Follow Part: This is the most optimal strategy where the tool path is manipulated depending on the part geometry. If there are cores and cavities in the part, the computer intelligently considers them to remove the materials in an optimal way. This is widely used for roughing operations.



Follow Periphery: This takes the path depending upon the periphery profile. For example, if the outer periphery of our part is rectangular, the tool path will be generated such that it gradually cuts the material from outside to inside with a Stepover value. This option is mostly used for projections and cores rather than cavities.



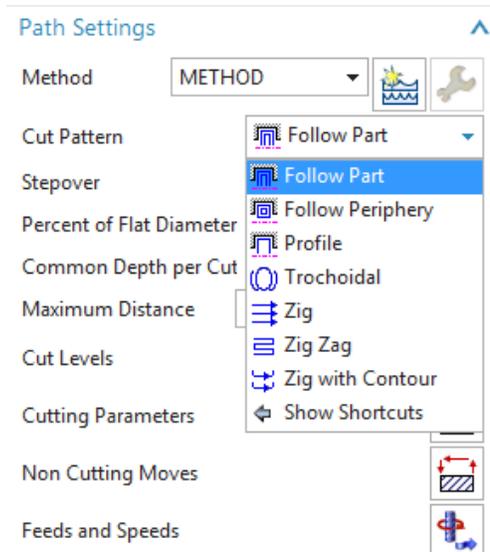
Profile: This takes the cut only along the profile of the part geometry. It is used for semi-finishing or finishing operations.



Trochoidal: This cutter is huge and is used for removing a large amount of material. The bulk of material is removed by gradual trochoidal movements. The depth of cut used will be very high for this strategy.



Zig: This takes a linear path in only one direction of flow.



 **Zig Zag:** This tool takes a zigzag path at every level of depth. It saves time by reducing amount of air cutting time (idle running). The climb and conventional cuts alternate.

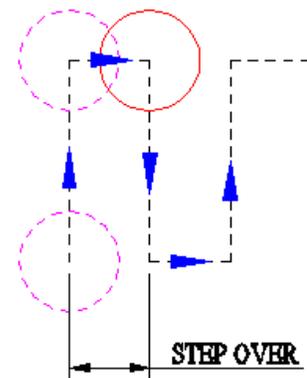
 **Zig with Contour:** This takes the path in one direction either climb or conventional. The unique thing is that it moves along the contour shape nonlinearly.

- For this exercise, select the **Follow Part** icon from the **Cut Pattern** drop-down menu since we have both projections and cavities in the part

9.2.4 Step Over and Scallop Height

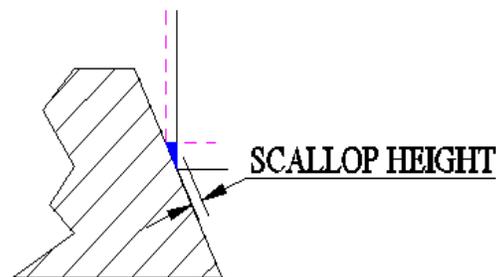
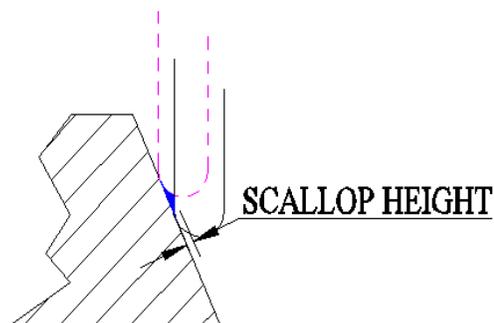
Step Over

This is the distance between the consecutive passes of milling. It can be given as a fixed value or the value in terms of cutter diameter. The *Stepover* should not be greater than the effective diameter of the cutter, otherwise it will leave extra material at every level of cut and result in an incomplete milling operation. The numeric value or values required to define the *Stepover* will vary depending on the *Stepover* option selected. These options include *Constant*, *Scallop*, *Tool Diameter*, etc. For example, *Constant* requires you to enter a distance value in the subsequent line.

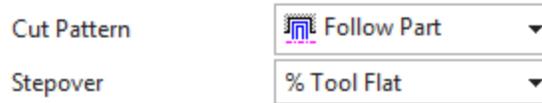


Scallop Height

Scallop Height controls the distance between parallel passes according to the maximum height of material (scallop) you specify to be left between passes. This is affected by the cutter definition and the curvature of the surface. *Scallop* allows the system to determine the *Stepover* distance based on the scallop height you enter.



- For the **Stepover**, select **%Tool Flat** and change the **Percent** to **70**



9.2.5 Depth Per Cut

This is the value to be given between levels to slice the geometry into layers and the tool path cuts as per the geometry at every layer. The cut depth value can vary for each level. Levels are horizontal planes parallel to the XY plane. If we do not give cut levels, the software will unnecessarily try to calculate slices for the entire part and machine areas that are not in our interest.

- Change the **Common Depth per Cut** value to **0.5**

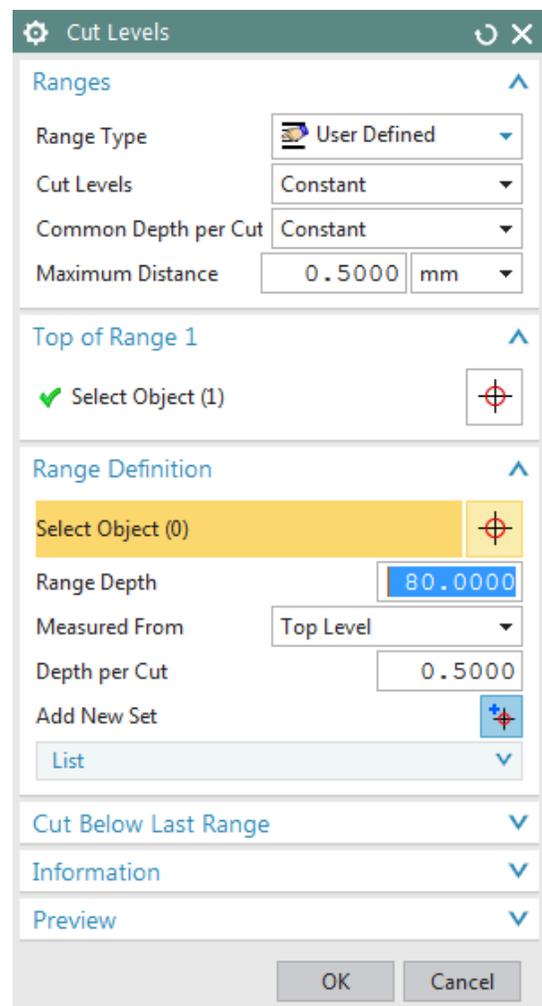


Now we will add the level ranges. This will split the part into different levels along the Z-direction to be machined.

- Click on **Cut Levels**

This will pop up a dialog box of *Cut Levels*. You need to set the level of the cut. You can either point to the object till which the cut level is or provide it as a *Range Depth* value. We are not going to mill up to the bottommost face of the part, but up to the floor at 40 mm from top. Therefore, we must delete the last level.

- Change the **Range Type** to **User Defined**
- Change the **Range Depth** to **80**
- Select **OK**



9.2.6 Cutting Parameters

- On the **Path Settings** menu, click **Cutting Parameters**
- Under the **Strategy** tab button, change the **Cut Order** from **Level First** to **Depth First**

Changing the cut order to *Depth First* orders the software to generate the tool path such that it will mill one island completely up to the bottom-most depth before jumping to another island. The *Depth First* strategy reduces the non-cutting time of the program due to unnecessary retracts and engages at each depth of cut.

- Click on the **Stock** tab
- Change the value of the **Part Side Stock** to **0.5**

This value is the allowance given to every side of the part. If you want to give different values to the floors (or the flat horizontal faces) uncheck the box next to *Use Floor Same As Side* and enter a different value for *Part Floor Stock*.

- Click **OK**

9.2.7 Avoidance

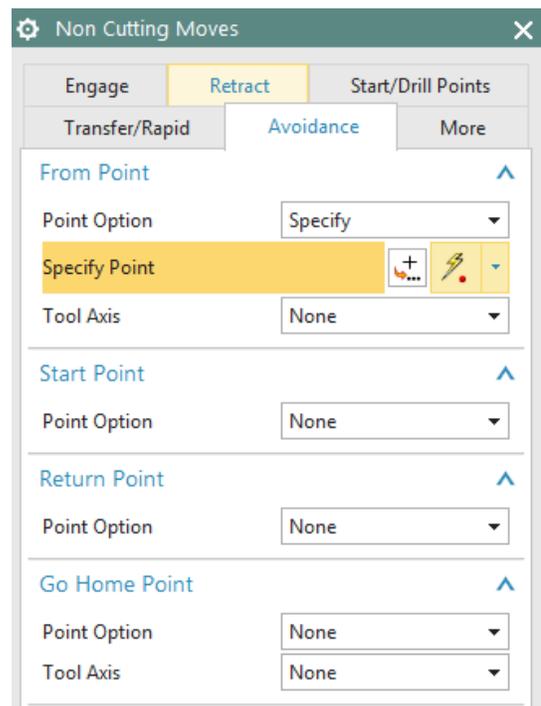
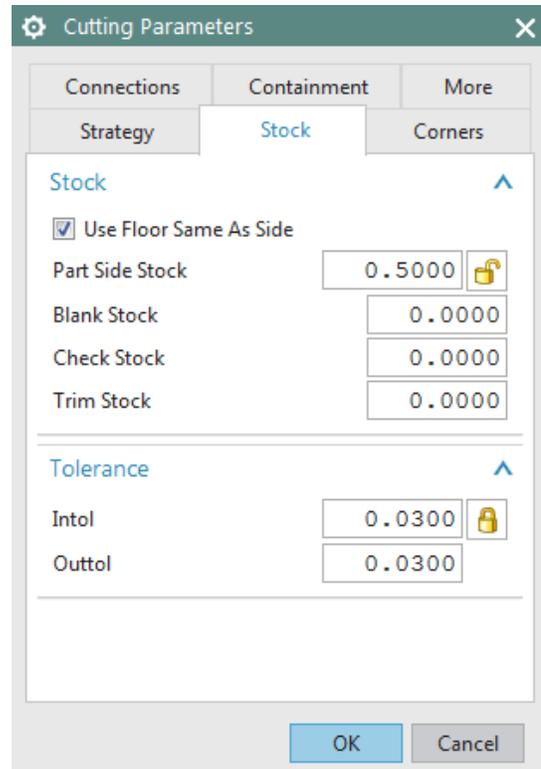
- Click the **Non Cutting Moves**
- Click the **Avoidance** tab

This window consists of several avoidance points of which we are concerned with the following points:

From Point

This is the point at which the tool change command will be carried out. The value is normally 50 or 100 mm above the Z=0 level to enhance the safety of the job when the cutter is changed by the Automatic Tool Changer (ATC).

- Click **From Point**
- Choose **Specify** in the **Point Option** field
- In the **Point Dialog**, enter the coordinates of XC, YC and ZC as **(0, 0, 50)**



- Click **OK**

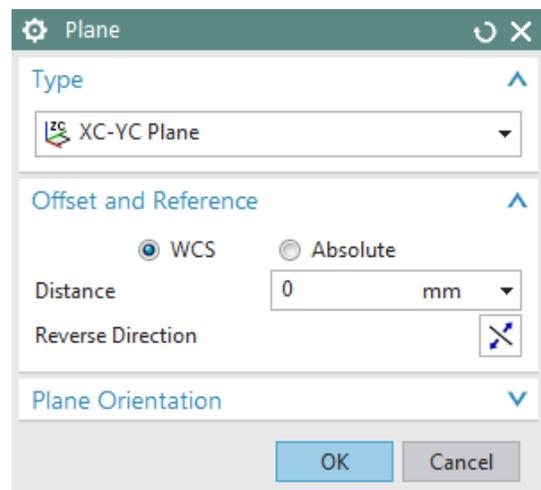
Start Point

This is the point at which the program starts and ends. This value is also 50 or 100 mm above the Z=0 level to enhance safety. It is also the point at which the machine operator checks the height of the tool mounted on the spindle with respect to the Z=0 level from the job. This cross checks the tool offset entered in the machine.

- Click on **Start Point**
- Choose **Specify**
- Enter the coordinates **(0, 0, 50)** in the **Point Dialog**
- Click **OK** to exit the **Point Constructor**

Clearance Plane is the plane on which the tool cutter will retract before moving to the next region or island. This is also known as *Retract Plane*. Sometimes the *Clearance Plane* can be the previous cutting plane. However, when the tool has to move from one region to another, it is necessary to move to the *Clearance Plane* before doing so. The value of the *Clearance Plane* should be at least 2 mm above the topmost point of the workpiece or fixture or whichever is fixed to the machine bed.

- Click on the **Transfer/Rapid** tab
- Choose **Plane** in the **Clearance Option**
- Choose the **XC-YC Plane** from the drop-down menu in **Type** tab
- Under the **Offset and Reference** tab enter the value of **3** as the **Distance**
- Click **OK** twice to go back to the **Cavity Mill** parameters window



9.2.8 Speeds and Feeds

- Choose **Feeds and Speeds** to enter the feed and speed parameters

Speed

Speed normally specifies the revolutions per minute (rpm) of the spindle (spindle speed). However, technically the speed refers to the cutting speed of the tool (surface speed). It is the linear velocity of the cutting tip of the cutter. The relative parameters affecting this linear speed are rpm of the spindle and the diameter of the cutter (effective diameter).

- Enter the **Spindle Speed** value as **4500 rpm**

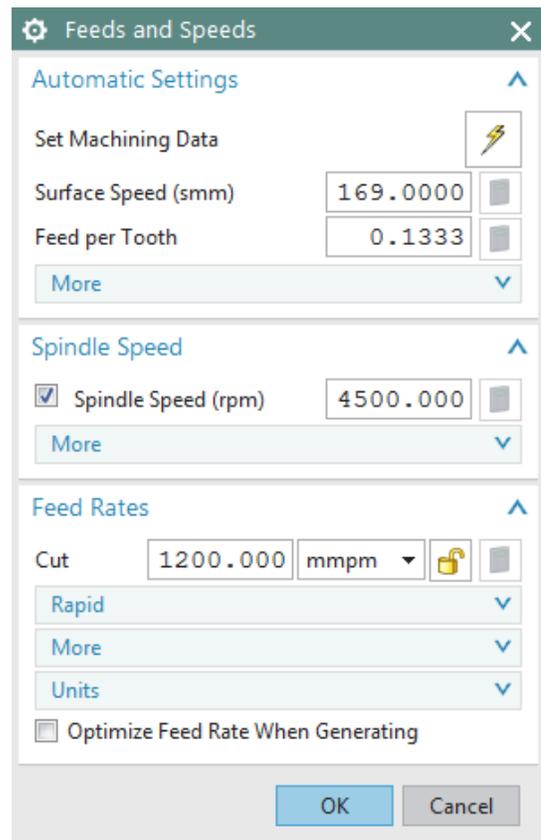
For the Surface Speed and the Feed per Tooth, you should enter the recommended values given by the manufacturers of the cutter (for this example, click on the calculator button near spindle speed). By entering these values, the software will automatically calculate the cutting feed rate and spindle speed. You can also enter your own values for feed rates and spindle speeds.

Feeds

There are many feeds involved in a single program. The most important is the *Cutting* feed. This is the feed at which, the tool will be in engagement with the raw work-piece and actually cutting the material off the work-piece. It is the relative linear velocity, at which the cutter moves with respect to the job.

The other feeds are optional. Some machine control systems use their default retracts and traverse feed. In those cases, even if you do not enter the values of other feeds, there would not be any problems. Some control systems may look for these feed rates from the program. It can be slightly less than the machine's maximum feed rate.

- Enter the **Cut** value as **1200 mmpm**
- Click **OK**



9.3 PROGRAM GENERATION AND VERIFICATION

9.3.1 Generating Program

Now we are done entering all the parameters required for the roughing program. It is time to generate the program.

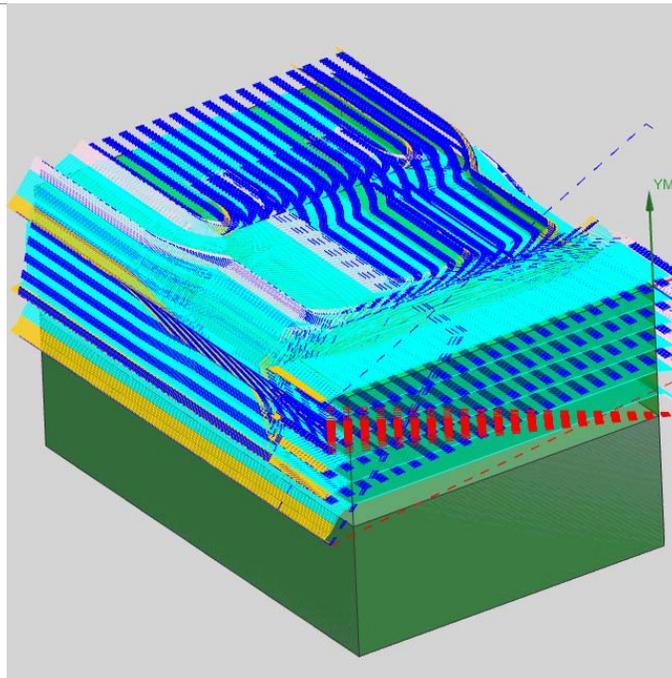
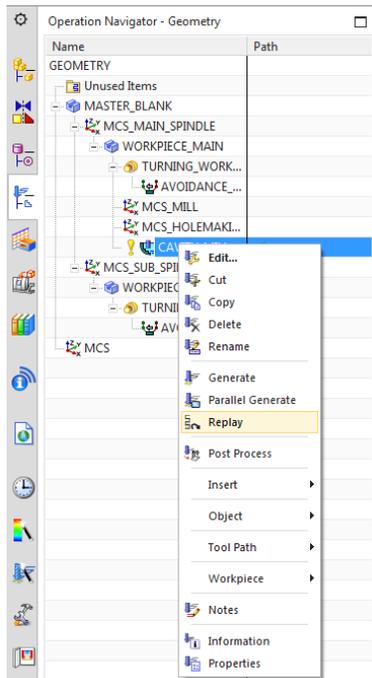
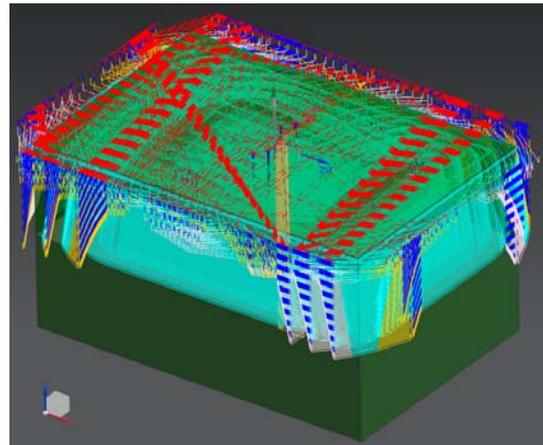
- Click on the **Generate** icon at the bottom of the window

You can now observe the software slicing the model into depths of cuts and creating tool-path at every level. You can find on the model cyan, blue, red and yellow lines as shown in the figure.



9.3.2 Tool Path Display

Whenever you want to view the entire tool-path of the program, right-click on the program in *Operation Navigator* and click *Replay*. It will give the display as shown in the Figure below.



You can now observe that next to the program in the *Operation Navigator* is a yellow exclamation point instead of a red mark. This means that program has been generated successfully but has not been post-processed. If any change is made in the model, the program will again have a red mark next to it. This implies that the program has to be generated again. However, there is no need to change any parameters in the program.

9.3.3 Tool Path Simulation

It is very important to check the programs you have created. This prevents any improper and dangerous motions from being made in the cutting path. It is possible that wrong parameters and settings will be given that cause costly damages to the work piece. To avoid such mistakes, NX 12 provides a *Tool-path* verification and a *Gouge* check.

The *Tool-Path verification* can be used to view the cutter motion in the entire program. You can observe how the tool is engaged and how it retracts after cutting. It also shows the actual material being removed through graphical simulation. You can also view the specific zone of interest by moving the line of the program.

- Right-click on the program in the **Operation Navigator** and choose **Tool Path** → **Verify** or click on the **Verify Tool Path** button in the toolbar

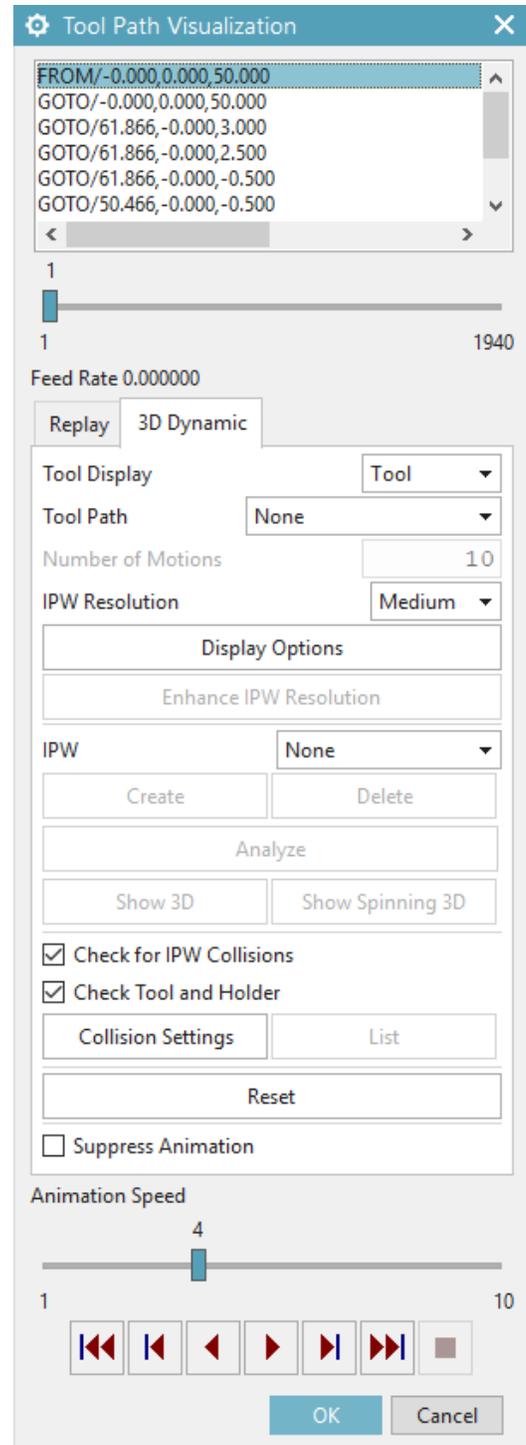
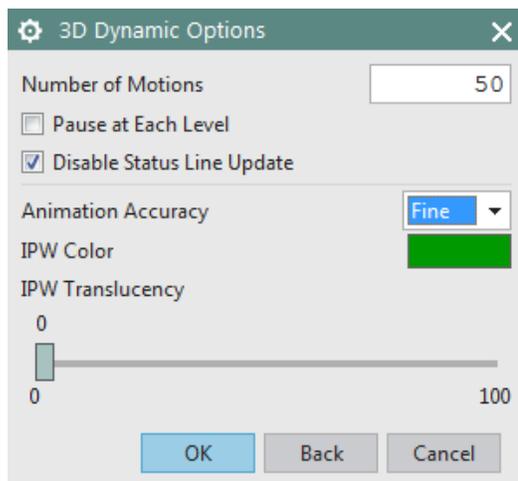


This will allow you to set the parameters for visualization of the *Tool-Path*.

- On the **Tool Path Visualization** window, click on the **Play** icon  to view the motion

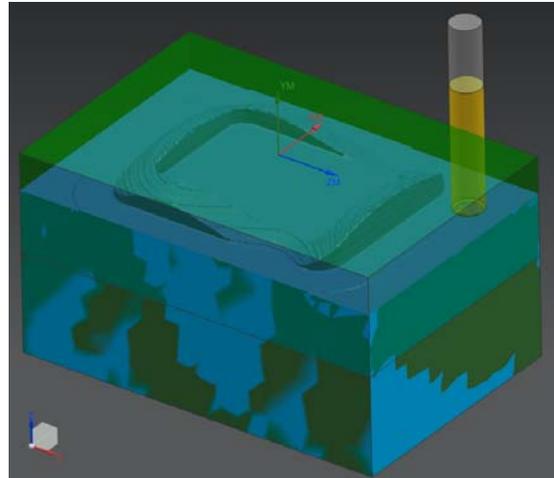
You can also view the visualization in different modes by changing the options in the drop-down menu next to *Display*.

- Click on the **3D Dynamic** tab on the same window
- Click on the **Display Options** button on the same window
- Change the **Number of Motions** to **50**
- Change the **Animation Accuracy** to **Fine**
- Change the **IPW Color** to **Green**
- Click **OK**



- Click on the **Play button**  again

The simulation will look as shown in the figure on the right. With this option, you will be able to view the actual cutting simulation and material removal through computer graphics. It is *3D Dynamic*, where you can rotate, pan and zoom the simulation when it is playing.



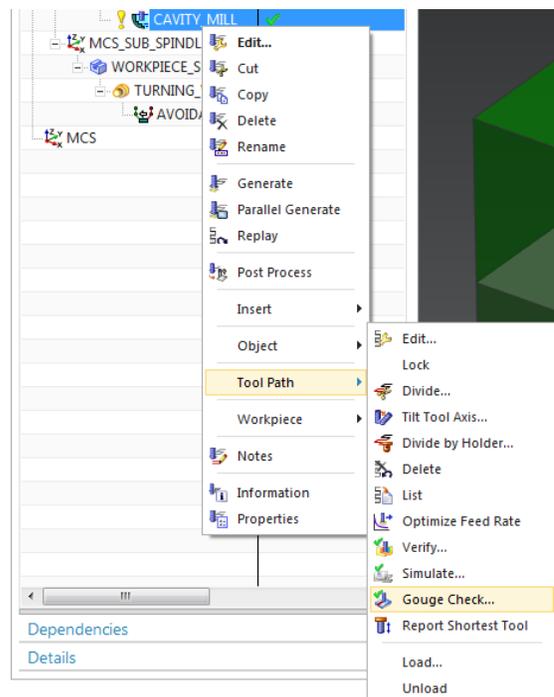
9.3.4 Gouge Check

Gouge Check is used to verify whether the tool is removing any excess material from the workpiece with respect to *Part Geometry*. Considering a *Design Tolerance*, any manufacturing process may produce defective parts by two ways. One is removing excess material, which is also called *Less Material Condition*. The other one is leaving materials that are supposed to be removed which is *More Material Condition*. In most cases, the former is more dangerous since it is impossible to rework the design part. The latter is safer since the leftover material can be removed by reworking the part. The gouge check option checks for the former case where the excess removal of material will be identified.

- Right Click the program in the **Operation Navigator**
- Choose **Tool Path** → **Gouge Check**
- Click **OK**

After the gouge check is completed, a **Tool Path Report** window will pop up. If in case there are any gouges found, it is necessary to correct the program. Otherwise,

- Click **OK** or directly close the popup window



9.4 OPERATION METHODS

9.4.1 Roughing

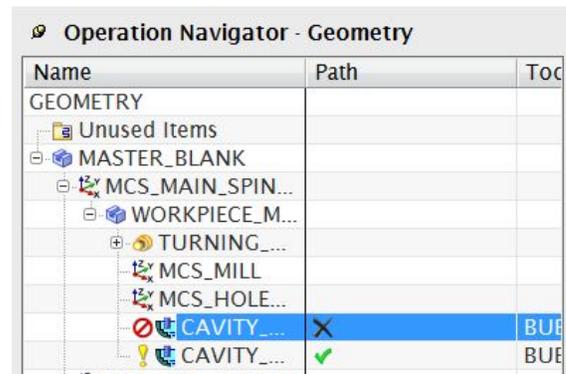
In case of milling operation, the first operation should be rough milling before finishing the job. The main purpose of roughing is to remove bulk material at a faster rate, without affecting the accuracy and finish of the job. Stock allowances are given to provide enough material for the finishing operation to get an accurate and good finish job. What we did in the previous part of this chapter is generating a roughing program. Now we have to moderately remove all the uneven material left over from the previous program.

9.4.2 Semi-Finishing

Semi-Finishing programs are intended to remove the unevenness due to the roughing operation and keep even part stock allowance for the *Finishing* operations. Once we are done with the first roughing program, semi-finishing is always easier and simpler to perform.

Now we will copy and paste the first program in the *Operation Navigator*. In the new program, you only have to change a few parameters and cutting tool dimensions and just regenerate the program.

- Right click **CAVITY_MILL** program in the **Operation Navigator** and click **Copy**
- Right-click **CAVITY_MILL** again and choose **Paste**
- Right-click the second **CAVITY_MILL_COPY** you just made and click **Rename**
- Rename the second program as **CAVITY_MILL_1**



You can see that next to the newly created **CAVITY_MILL_1** is a red mark, which indicates that the program is not generated.

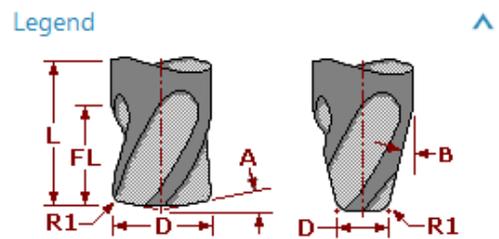
Let us now set the parameters that need to be changed for the second program. Before we start, we should analyze the part geometry to figure out the minimum corner radius for the cutter diameter. In our model, it is 5 mm and at the floor edges, it is 1 mm. Therefore, the cutter diameter can be

anything less than 10 mm. For optimal output and rigidity, we will choose a *Bull Nose Cutter* with a diameter of 10 and a lower radius of 1.

- Double click **CAVITY_MILL_1** on **Operation Navigator** to open the parameters window

Just as we did in the previous program, we will create a new cutter. In the *Tool* tab, you will see the cutter you first chose. It will show *BUEM12X1* as the current tool.

- Create a new **Mill** and name it **BUEM10X1**
- It should have a **Diameter** of **10**, a **Lower Radius** of **1** and a **Flute Length** of **38**
- Click **OK**
- Click the **Common Depth per Cut** as **0.25** in the **Path Settings**



Dimensions	
(D) Diameter	10.0000
(R1) Lower Radius	1.0000
(B) Taper Angle	0.0000
(A) Tip Angle	0.0000
(L) Length	75.0000
(FL) Flute Length	38.0000
Flutes	2

Path Settings

Method: METHOD

Cut Pattern: Follow Part

Stepover: % Tool Flat

Percent of Flat Diameter: 70.0000

Common Depth per Cut: Constant

Maximum Distance: 0.2500 mm

- Then click on **Cutting Parameters** button
- Click on the **Stock** tab
- Uncheck the box next to **Use Floor Same As Side**
- Enter **0.25** for **Part Side Stock**
- Enter **0.1** for **Part Floor Stock**
- Click on the **Containment** tab button

Cutting Parameters

Strategy: Stock

Use Floor Same As Side

Part Side Stock: 0.2500

Part Floor Stock: 0.1000

Blank Stock: 0.0000

Check Stock: 0.0000

Trim Stock: 0.0000

- In the drop-down menu next to **In Process Workpiece**, choose **Use 3D**

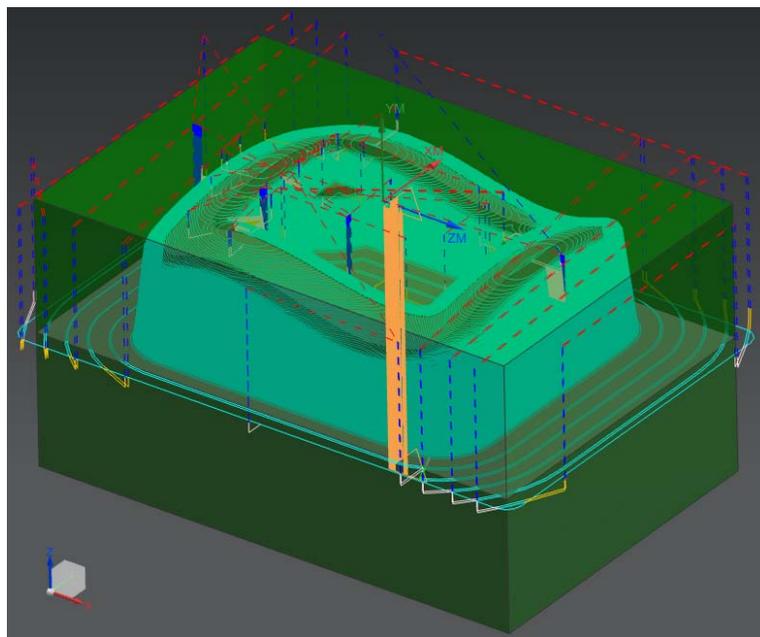
In Process Workpiece is a very useful option in NX. The software considers the previous program and generates the current program such that there is no unnecessary cutting motion in the *No-material* zone. This strategy reduces the cutting time and air cutting motion drastically. The algorithm will force the cutter to only remove that material, which is left from the previous program and maintain the current part stock allowance.

- Choose **OK** to return to the **Parameters** window
- Click **Feeds and Speeds**
- Enter the **Spindle Speed** as **5000**
- Then click **OK**

The parameters and settings are finished for the semi-finishing program.

- Regenerate the program by clicking on the **Generate** icon 
- After the software finishes generating click **OK**

Then replay the *Tool Path Visualization*. The overall *Tool Path* generated in the second program will look like the following figure. You can replay it or check for the gouging in a similar way.



9.4.3 Finishing Profile

So far, we are done with the roughing and semi-finishing programs for the part. There is a small amount of material left in the *Workpiece* to be removed in the finishing programs to obtain the desired accurate part geometry. The finishing programs should be generated such that every surface in the part should be properly machined. Therefore, it is better to create more than one program to uniquely machine sets of surfaces with relevant cutting parameters and strategies rather than make one program for all the surfaces. The following illustrates how to group the profiles and surfaces and create the finishing programs.

9.4.3.1 Outer Profile

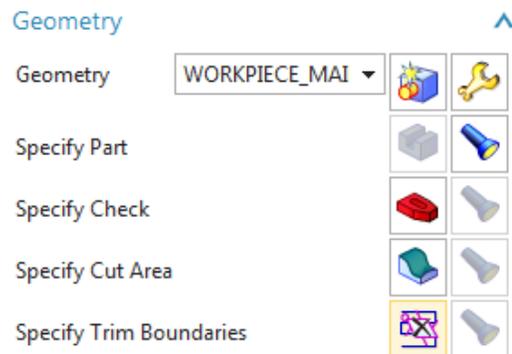
This program is intended to finish the outer inclined walls onto the bottom of the floor. Because the program should not touch the contour surface on the top, we will give *Check* and *Trim* boundaries in the program.

- Repeat the same procedure as before to copy and paste **CAVITY_MILL_1** on **Operation Navigator**
- Rename the program **CAVITY_MILL_2**
- Double click **CAVITY_MILL_2** to make parameter changes
- In the popup parameters window, change the **Cut Pattern** to **Profile** and the **Stepover percentage** to **40**
- Click on the **Specify Trim Boundaries** tab

The *Trim Boundaries* window will pop up. Make sure to carry out the following procedure in the right sequence. Keep the default setting of *Trim Side* to *Inside*. This tells the software that the cutter should not cut material anywhere inside the boundary. Trim allows

you to specify boundaries that will further constrain the cut regions at each cut level.

- Change the **Selection Method** to **Curves**
- Change the **Plane** from **Automatic** to **Specify** and click on the **Plane Dialog**

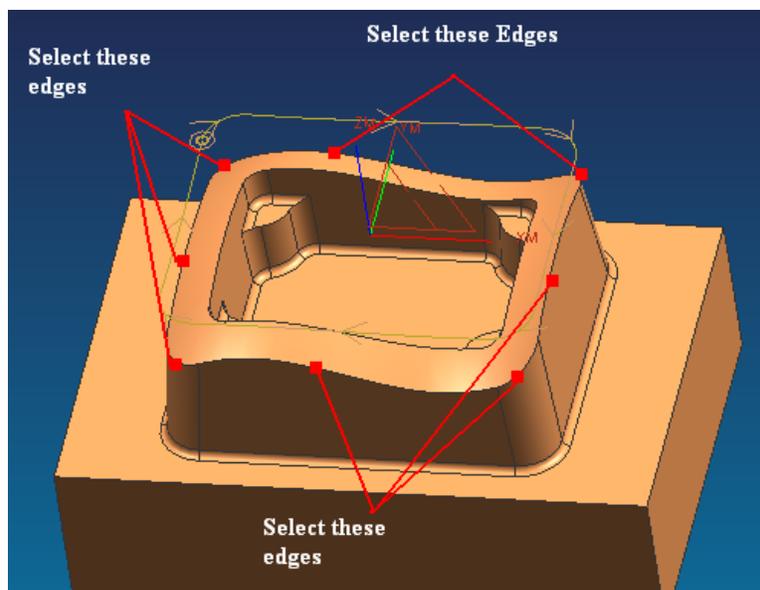


A new window will pop up. The window will ask for the mode of selection of the plane on which the curves should be projected. This should normally be over the topmost point of the part geometry. Precisely, it should be over the *MCS*.

- Choose the **XC-YC Plane** from the drop-down menu under **Type**
- Enter a value of **3** next to **Distance**
- Click **OK**

Now we will start selecting edges from the part. These selected edges will be projected on the $Z = 3$ plane as curves and used as the boundary.

- Select all the top outer edges on the wall along the contour surface as shown in the figure. Make sure to select all **8 edges** and in a **continuous order**
- Choose **OK**

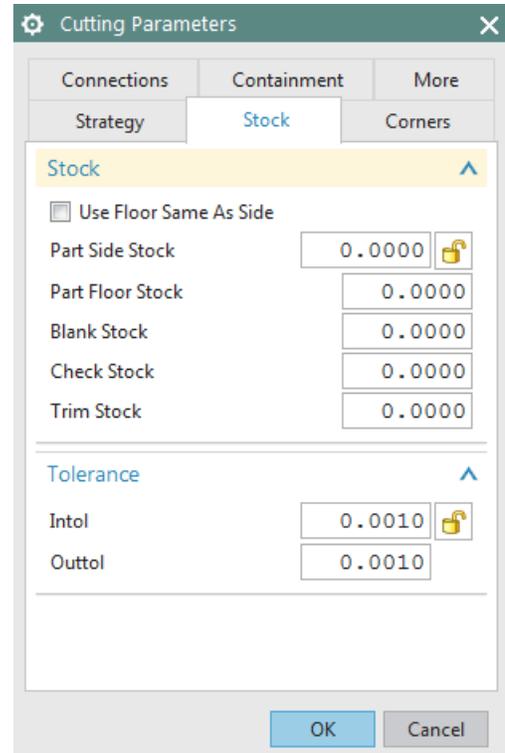


- Enter the **Common Depth per Cut** as **0.2**
- Click **Cutting Parameters**
- In the popup dialog box, click on **Stock** tab
- Enter the **Part Side Stock** and **Part Floor Stock** values to be **0.00**

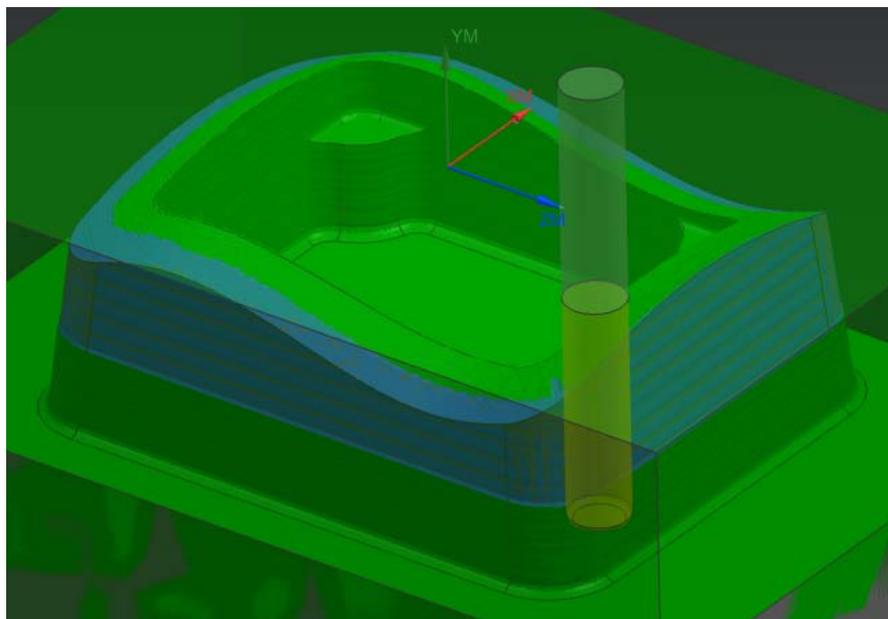
Intol allows you to specify the maximum distance that a cutter can deviate from the intended path into the workpiece.

Outtol allows you to specify the maximum distance that a cutter can deviate from the intended path away from the workpiece.

- Enter the **Intol** and **Outtol** values to be **0.001** as shown in the figure
- Click **OK**
- Click on the **Generate** icon  to generate the program in the **Main Parameters** window
- Click **OK** on the parameters window when the program generation is completed



The finishing program for the outer profile is now ready. You can observe while replaying the tool path that the cutter never crosses the boundary that has been given for trim and check. The cutter retracts to the Z=3 plane for relocation.

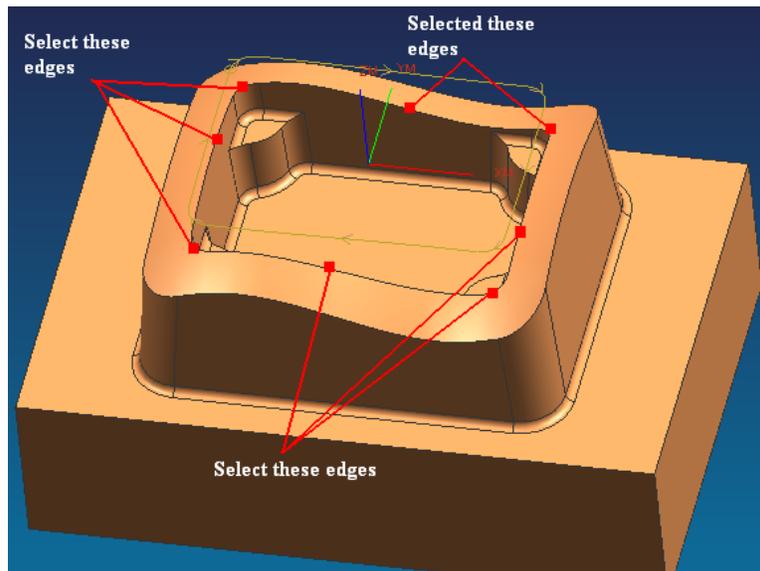


9.4.3.2 Inner profile

- Repeat the same procedure as before to copy and paste **CAVITY_MILL_2** on **Operation Navigator** and rename it as **CAVITY_MILL_3**
- Double click **CAVITY_MILL_3** to edit the parameters or right click on it and choose **Edit**
- Select the **Specify Trim Boundaries** tab and choose **Trim Side** to be **Outside** in the popup dialog box

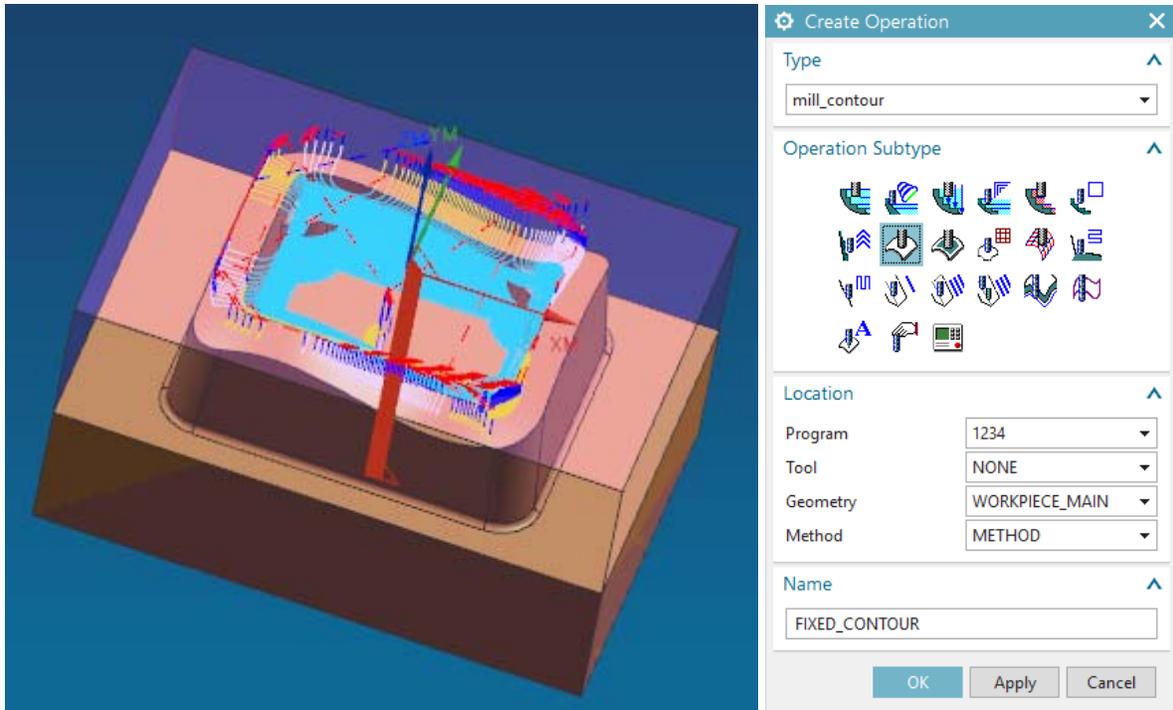
This will prevent the cutter from passing outside the boundary.

- Change **Selection Method** to **Curves**
- Change the plane manually to be the **XC-YC** plane and enter the offset distance as **3**
- Click **OK**
- Select all the top inner edges along the contour surface as shown in the figure. Again, make sure all 8 edges are selected in a continuous order (using Shift+click to de-select a curve).



- Choose **OK** to return to the parameters window
- Generate the program
- Click **OK** when the generation is finished

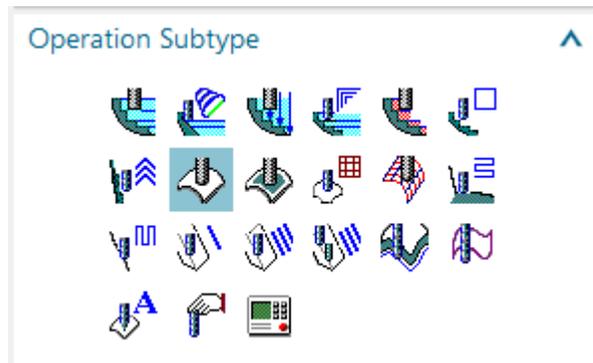
The finishing program for the inner profile is now ready. By replaying the tool path, you can observe that the cutter never crosses the boundary that has been given for trim and check.



9.4.4 Finishing Contour Surface

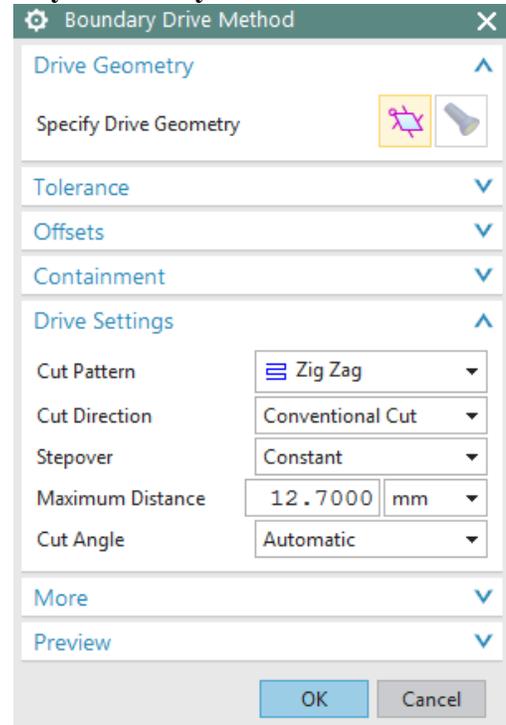
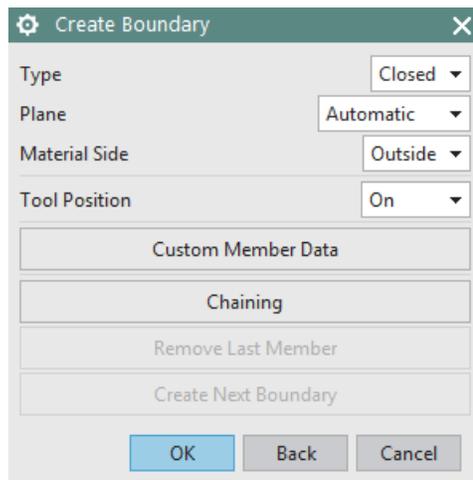
Now we will use a different type of strategy to finish the top freeform surface.

- Click on the **Create Operation** icon  in the Toolbar
- Click on the **Fixed Contour** icon as shown in the figure
- Choose **1234** for Program
- Choose **WORKPIECE_MAIN** for **Geometry**
- Keep the default name of program
- Click **OK**
- On the **Parameters** window, under **Drive Method**, make sure that **Boundary** is selected



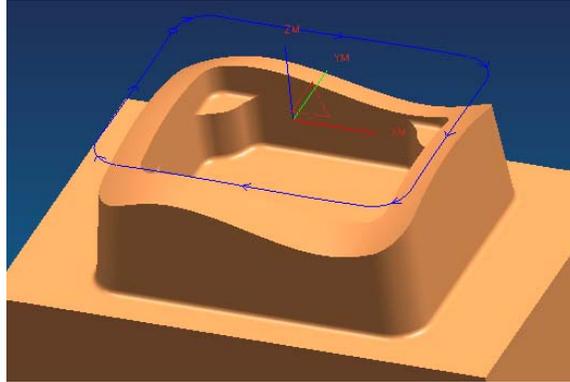
- Click  icon, which will pop up the **Boundary Drive Method** window, click on the  icon as shown in the figure to open the **Boundary Geometry** menu

- Change the **Mode** to **Curves/Edges**
- Select the **Material Side** to be **Outside**
- Select the **Tool Position** to be **On**



The *Tool Position* determines how the tool will position itself when it approaches the *Boundary Member*. *Boundary Members* may be assigned one of three tool positions: *On*, *Tanto*, or *Contact*.

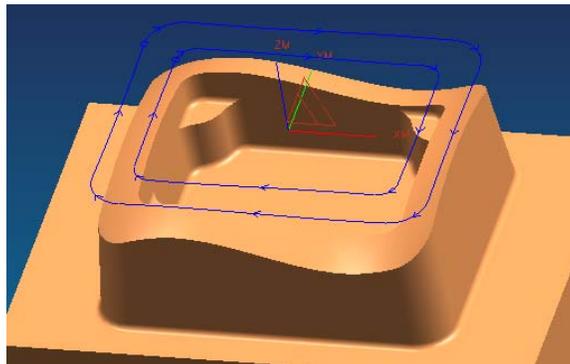
- In **On** position, the center point of the tool aligns with the boundary along the tool axis or projection vector.
- In **Tanto** position, the side of the tool aligns with the boundary.
- In **Contact** position, the tool contacts the boundary.
- For the **Plane**, choose **User-Defined**
- Again, set the plane to be **XC-YC** with a **Distance** of **3**
- Click **OK**
- Select the outer loop of the top contour surface as shown in the figure. Remember to select the edges in a continuous order



- Click **OK**

We have trimmed the geometry outside the loop. Now we have to trim the geometry inside the inner loop so that the only geometry left will be the area between the two loops.

- Choose the **Mode** to be **Curves/Edges**
- Choose the **Material Side** to be **Inside** and **Tool Position** to be **On**
- Choose the plane to be user-defined at **XC-YC** with a **Distance** of **3**
- Select the inner edges of the contour surface as shown
- Click **OK** to return to the **Boundary Drive Method** window



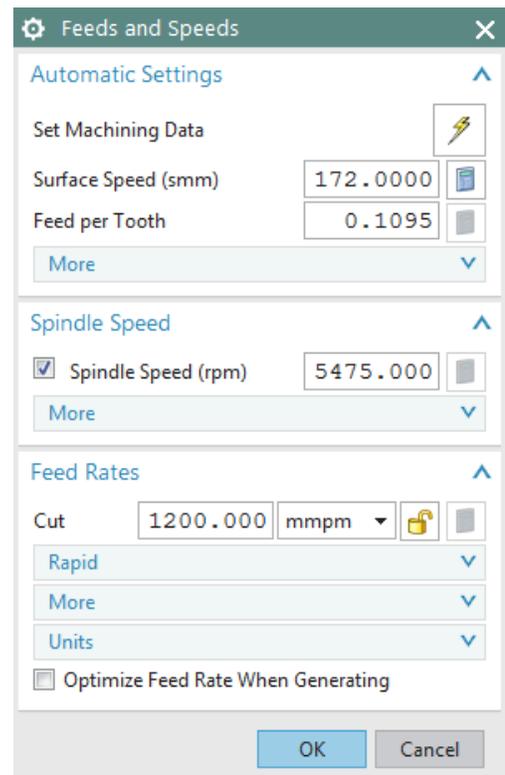
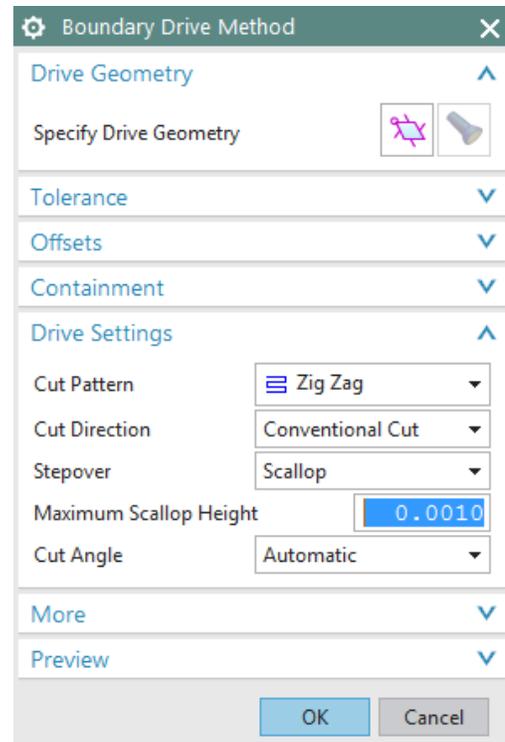
- Change the **Stepover** method to **Scallop** and enter the height to be **0.001** and click **OK**

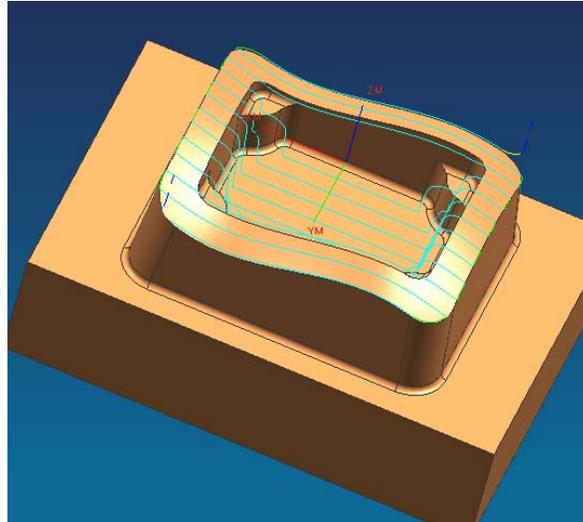
- Click on **Cutting Parameters**
- Change the **Tolerance** values in the **Stock** tab so that the **Part Intol** and **Part Outtol** is **0.001**
- Click on the **More** tab button and enter the value of **Max Step** as **1.0**
- Click **OK**

In the main parameters window,

- Create a new tool and name it **BEM10**
- Change the diameter to be **10 mm** and the lower radius to be **5 mm**.
- Click **OK**
- Click on the **Feeds and Speeds** icon on the parameters window
- Enter the **Spindle Speed, Feed Rates (Cut)** and calculate the others as shown in the figure on right
- Click **OK**
- **Generate** the program

The contour surface is now finished and you can view the simulation by *Tool Path Verification*.

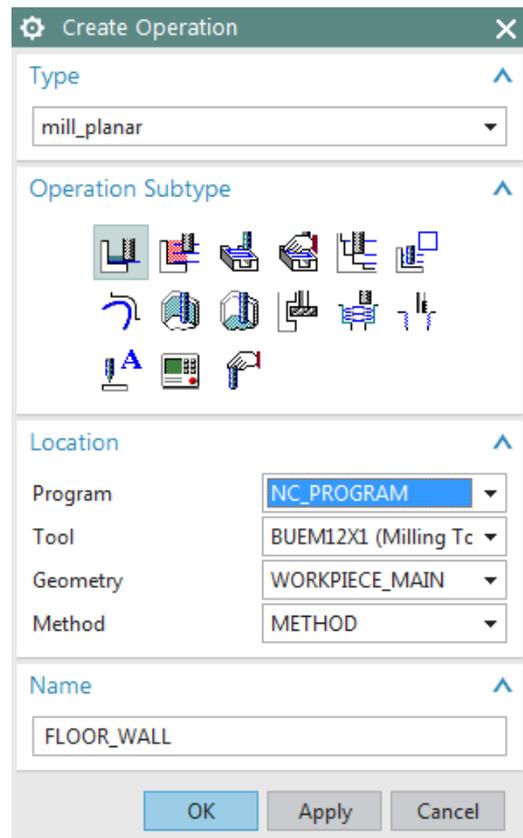




9.4.5 Flooring

Flooring is the finishing operation performed on the horizontal flat surfaces (*Floors*) of the part. In most of the milling processes, flooring will be the final operation of them. All the horizontal surfaces will be finished. This planar operation runs the cutter in a single pass on every face.

- Click on the **Create Operation** icon 
- Change the **Type** to be **mill_planar** at the top of the window
- Change all the options as shown in the figure
- Click **OK**
- In the parameters window, change the **Cut Pattern** to be **Follow Part**



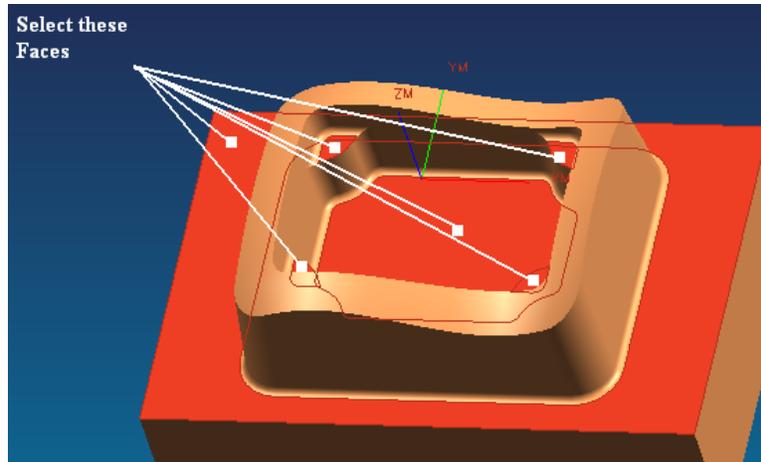
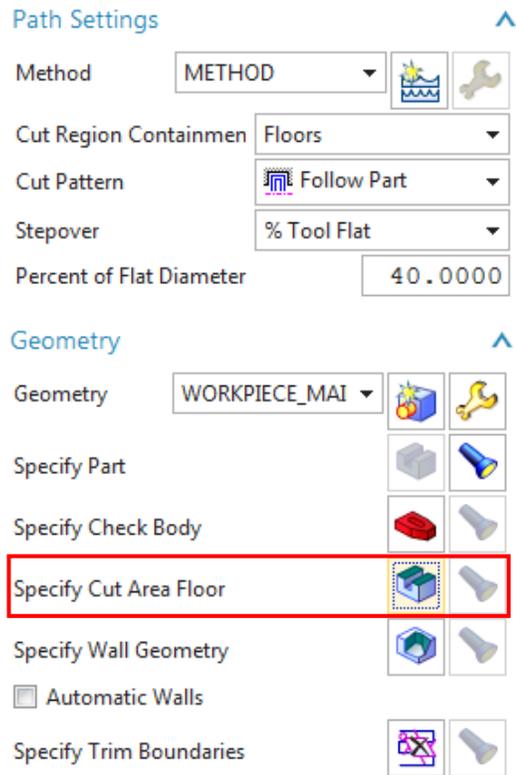
- Change the percent of the tool diameter for **Stepover** to be **40**

In flooring operations, it is always better to keep the *Stepover* value to be less than half of the diameter of the cutter in order to achieve more flatness on the planar surfaces.

Unlike previous programs, we have to select a cut area.

- Click on the **Specify Cut Area Floor** as shown
- Select the highlighted surfaces shown in the figure below

In case you are not able to select the surfaces as shown go to *Part Navigator* and *Hide the Blank* (to choose the blank, you can right click on the blank, use *Select from List...*), select the surfaces and *Unhide the Blank* again.



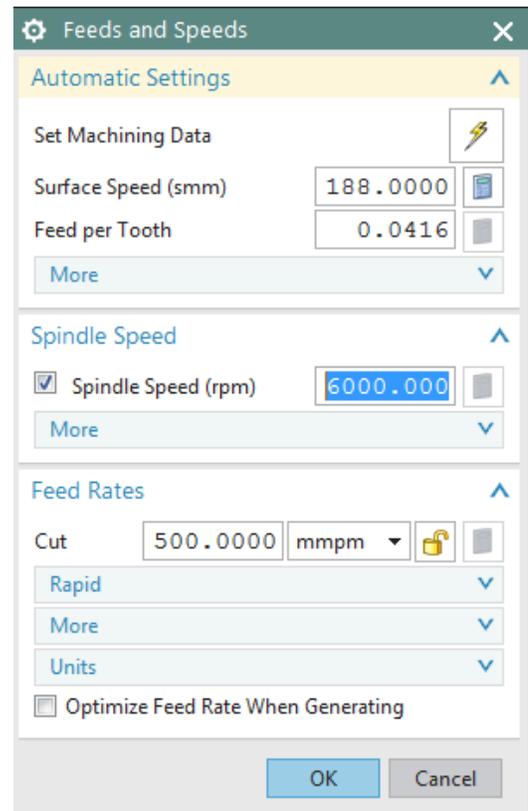
- Click **OK**
- Click on **Cutting Parameters** in the main parameter window
- Choose the **Stock** tab button and enter the **Intol** and **Outtol** values as **0.001**
- Click **OK**
- Click on **Feeds And Speeds**

In the main Parameters window,

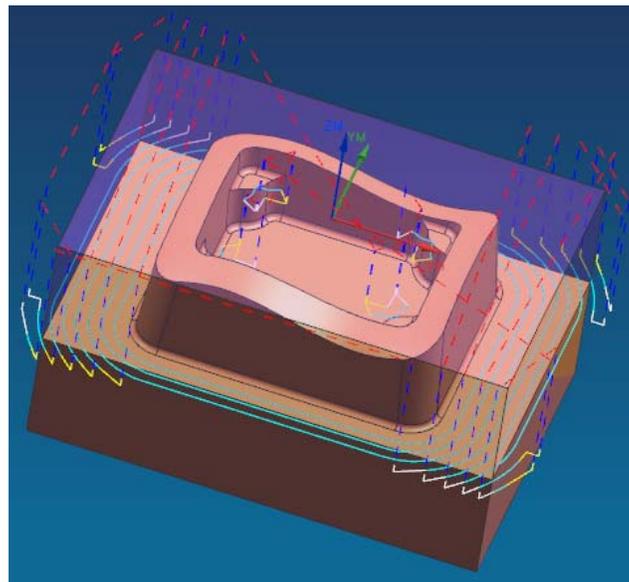
- Create a new tool and name it **BEF105**
- Change the diameter to be **10 mm** and the lower radius to be **5 mm**
- Click **OK**

Because this is a *Flooring* operation, it is better to make the spindle speed high and the feed rates low compared to the previous operations.

- Enter the values for **Spindle Speed**, **Feed Rates (Cut)** and calculate the others as shown in the figure
- Choose **OK**
- Generate the program. Then replay and verify the cutter path



The following figure shows the *Tool Path* display for the flooring.



9.5 POST PROCESSING

The primary use of the *Manufacturing Application* is to generate tool paths for manufacturing parts. Generally, we cannot just send an unmodified tool path file to a machine and start cutting because there are many different types of machines. Each type of machine has unique hardware capabilities, requirements and control systems. For instance, the machine may have a vertical or a horizontal spindle; it can cut while moving several axes simultaneously, etc. The controller accepts a tool path file and directs tool motion and other machine activity (such as turning the coolant or air on and off).

Naturally, as each type of machine has unique hardware characteristics; controllers also differ in software characteristics. For example, most controllers require that the instruction for turning the coolant on be given in a particular code. Some controllers also restrict the number of M codes that are allowed in one line of output. This kind of information is not in the initial NX tool path. Therefore, the tool path must be modified to suit the unique parameters of each different machine/controller combination. The modification process is called *Post Processing*. The result is a *Post Processed* tool path.

There are two steps involved in generating the final post-processed tool path.

1. Create the tool path data file, or called *CLSF (Cutter Location Source File)*.
2. Post process the *CLSF* into machine CNC code (*Post Processed* file). This program reads the tool path data and reformats it for use with a particular machine and its accompanying controller.

9.5.1 Creating CLSF

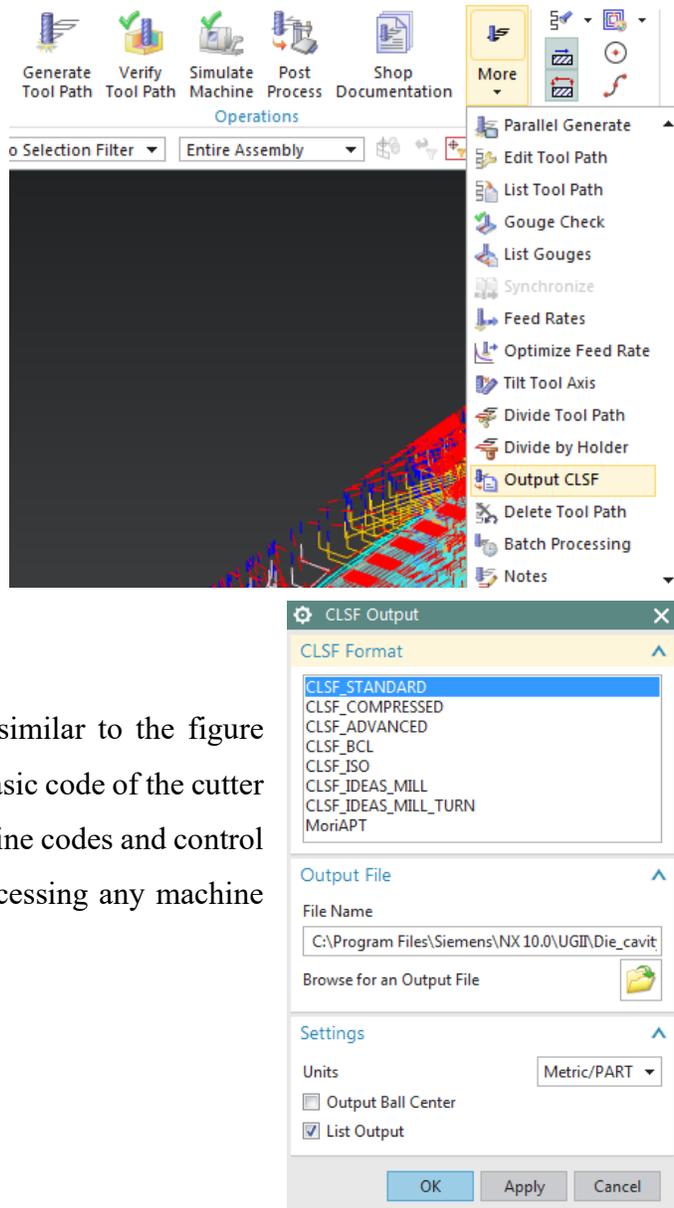
After an operation is generated and saved, the resulting tool path is stored as part of the operation within the part file. *CLSF* (*Cutter Location Source File*) provides methods to copy these internal paths from the operations in the part file to tool paths within the *CLSF*, which is a text file. The *GOTO* values are a "snapshot" of the current tool path. The values exported are referenced from the MCS stored in the operation. The CLS file is the required input for some subsequent programs, such as postprocessors.

- Click on one of the programs that you want to post process in the **Operation Navigator**
- Click on **Output CLSF** in the **Operations** toolbar

A window will pop up to select the *CLSF* Format.

- Choose **CLSF_STANDARD** and enter a location for the file
- Choose **OK**

The *CLSF* file will be created. It will be similar to the figure below. The contents of the file contain the basic code of the cutter motion without any information about machine codes and control systems. This file can be used for post-processing any machine control. The extension of the file is **.cls**.



```

SA\TA5763_2017Fall\chapters\files_ch9\Die_cavity_setup_1.cls - Notepad++
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
Die_cavity_setup_1.cls
1 TOOL PATH/FLOOR_WALL,TOOL,BEF105
2 TLDDATA/MILL,10.0000,5.0000,75.0000,0.0000,0.0000
3 MSYS/0.0000,0.0000,0.0000,0.0000000,-1.0000000,0.0000000,1.0000000,0.0000000,0.0000000
4 $$ centerline data
5 PAINT/PATH
6 PAINT/SPEED,10
7 PAINT/COLOR,186
8 RAPID
9 GOTO/18.3123,20.3584,-12.0000,0.0000000,0.0000000,1.0000000
10 PAINT/COLOR,42
11 FEDRAT/MMPM,500.0000
12 GOTO/18.3123,20.3584,-15.0000
13 GOTO/19.5080,25.2685,-15.0000
14 GOTO/19.5080,28.2685,-15.0000
15 PAINT/COLOR,31
16 GOTO/19.5080,29.5080,-15.0000
17 GOTO/18.2685,29.5080,-15.0000
18 PAINT/COLOR,37
19 GOTO/15.2685,29.5080,-15.0000
20 GOTO/10.3584,28.3123,-15.0000
21 GOTO/10.3584,28.3123,-12.0000
22 PAINT/COLOR,211
23 RAPID
24 GOTO/10.3584,28.3123,3.0000
25 PAINT/COLOR,186
26 RAPID
27 GOTO/-10.3584,28.3123,3.0000
28 PAINT/COLOR,211
29 RAPID
30 GOTO/-10.3584,28.3123,-12.0000
31 PAINT/COLOR,42
32 GOTO/-10.3584,28.3123,-15.0000
33 GOTO/-15.2685,29.5080,-15.0000
34 GOTO/-18.2685,29.5080,-15.0000
Normal text length: 34,961 lines: 959 Ln: 1 Col: 33 Sel: 0|0 Windows (CR LF) UTF-8 INS

```

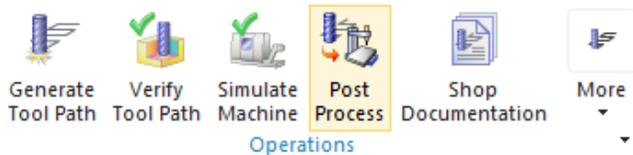
Any program that has been output to *CLSF* or *Post Processed* will have a green checkmark next to it in the *Operation Navigator*.

9.5.2 Post Processing

- Click on a program in the **Operation Navigator** that you want to post process

- Click **Menu** → **Tools** → **Operation**

Navigator → **Output** → **Postprocess** or
from the **Home** tab as shown



- Select the **MILL_3_AXIS** machine and enter a location for the file

- Select **OK**

This will create the *Post Processed* file for the target machine. You can find the block numbers with *G* and *M* codes concerning the machine controller type. The extension of the file is **.ptp**.

```
1 |
2 | N0010 G40 G17 G90 G70
3 | N0020 G91 G28 Z0.0
4 | N0030 T00 M06
5 | N0040 G00 G90 X.721 Y.8015 S6000 M03
6 | N0050 G43 Z-.4724 H00
7 | N0060 G01 Z-.5906 F19.7 M08
8 | N0070 X.768 Y.9948
9 | N0080 Y1.1129
10 | N0090 Y1.1617
11 | N0100 X.7192
12 | N0110 X.6011
13 | N0120 X.4078 Y1.1147
14 | N0130 Z-.4724
15 | N0140 G00 Z.1181
16 | N0150 X-.4078
17 | N0160 Z-.4724
18 | N0170 G01 Z-.5906
19 | N0180 X-.6011 Y1.1617
20 | N0190 X-.7192
21 | N0200 X-.768
22 | N0210 Y1.1129
23 | N0220 Y.9948
24 | N0230 X-.721 Y.8015
25 | N0240 Z-.4724
26 | N0250 G00 Z.1181
27 | N0260 X1.5131 Y3.2677
28 | N0270 Z-1.4567
29 | N0280 G01 Z-1.5748
30 | N0290 Y2.9889
31 | N0300 X1.6255 Y2.9528
32 | N0310 X1.6387 Y2.9485
33 | N0320 X1.672 Y2.9369
34 | N0330 X1.705 Y2.9245
35 | N0340 X1.7377 Y2.9113
```

Normal text length: 17,950 lines: 749 Ln: 1 Col: 1 Sel: 0|0 Windows (CR LF) UTF-8 INS

The final output (*filename.ptp*) file can be transferred to a machine for conducting an actual milling operation. The entire sequence starting from the transfer of the model into the *Manufacturing* module to the transfer of the files to the machine and fabricating the raw piece into the final part is called *Computer Aided Manufacturing*.