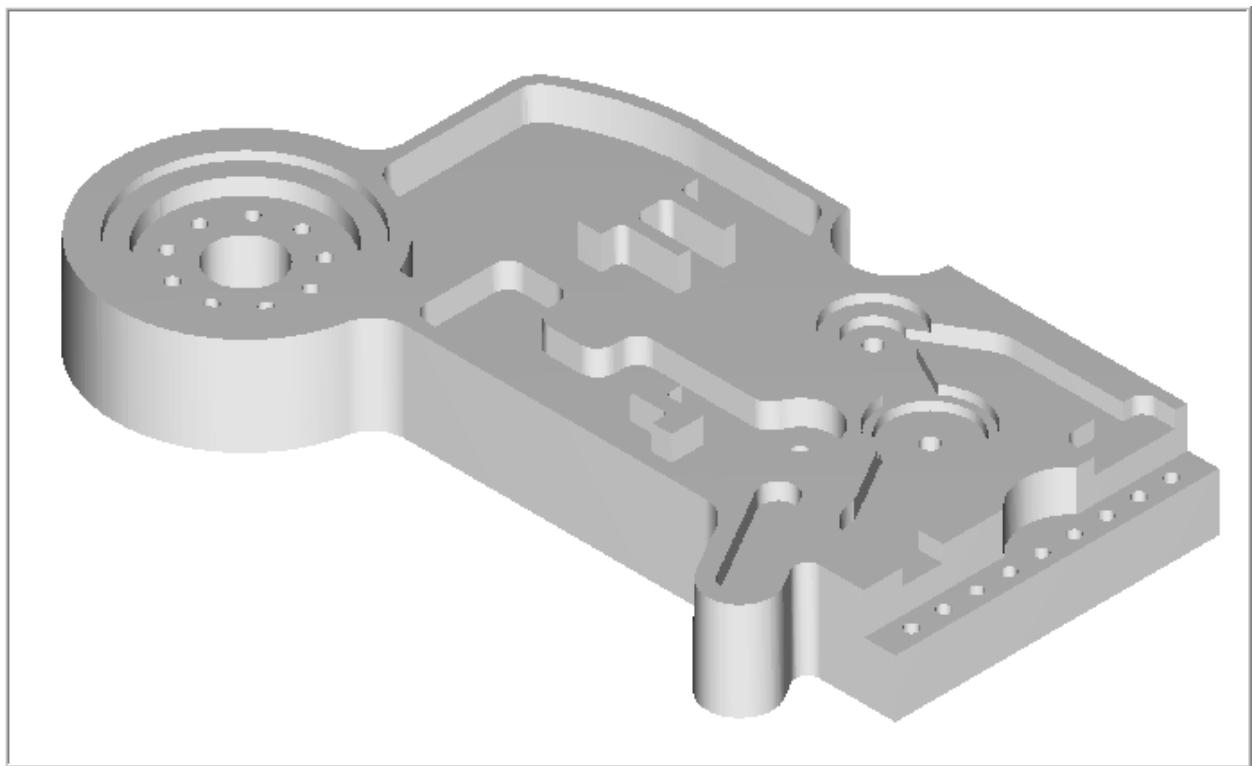


***Mastercam X<sup>3</sup>***

**TRAINING**

**GUIDE**



**MILL-LESSON-FBM-2**

**FBM MILL AND FBM DRILL**

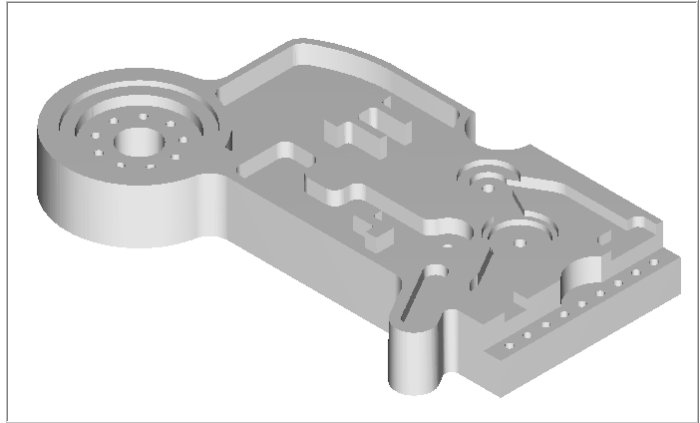
### Objectives

This lesson will use the same Feature Based Machining (FBM) methods used in Mill-Lesson-FBM-1, how ever this time applying FBM to much more complex geometry. Using similar configurations techniques as the previous lesson, you will see advantages in programming time using the FBM toolpaths.

#### ➤ **Feature Based Machining (FBM)**

FBM eliminates manual feature identification for programming milling and drilling operations on solid parts. The FBM Mill and FBM Drill operations automatically create the toolpaths needed to machine features that are identified using your specific criteria. FBM's interface is simple and intuitive.

You will generate the toolpaths for Mill-Lesson-FBM-2 to machine the part on a CNC vertical milling machine.



This Lesson covers the following topics:

#### ➤ **Establish Stock Setup settings:**

Stock size.

Material for the part.

Feed calculation.

#### ➤ **Generate a 2-dimensional milling toolpath consisting of:**

FBM Mill operation and child toolpath operations.

FBM Drill operation and child toolpath operations.

#### ➤ **Inspect the toolpath using Mastercam's Verify by:**

Launching the Verify function to machine the part on the screen.

Saving the FBM Mill results as an in-process STL in Verify.

Applying the FBM Drill results to the in-process stock definition.

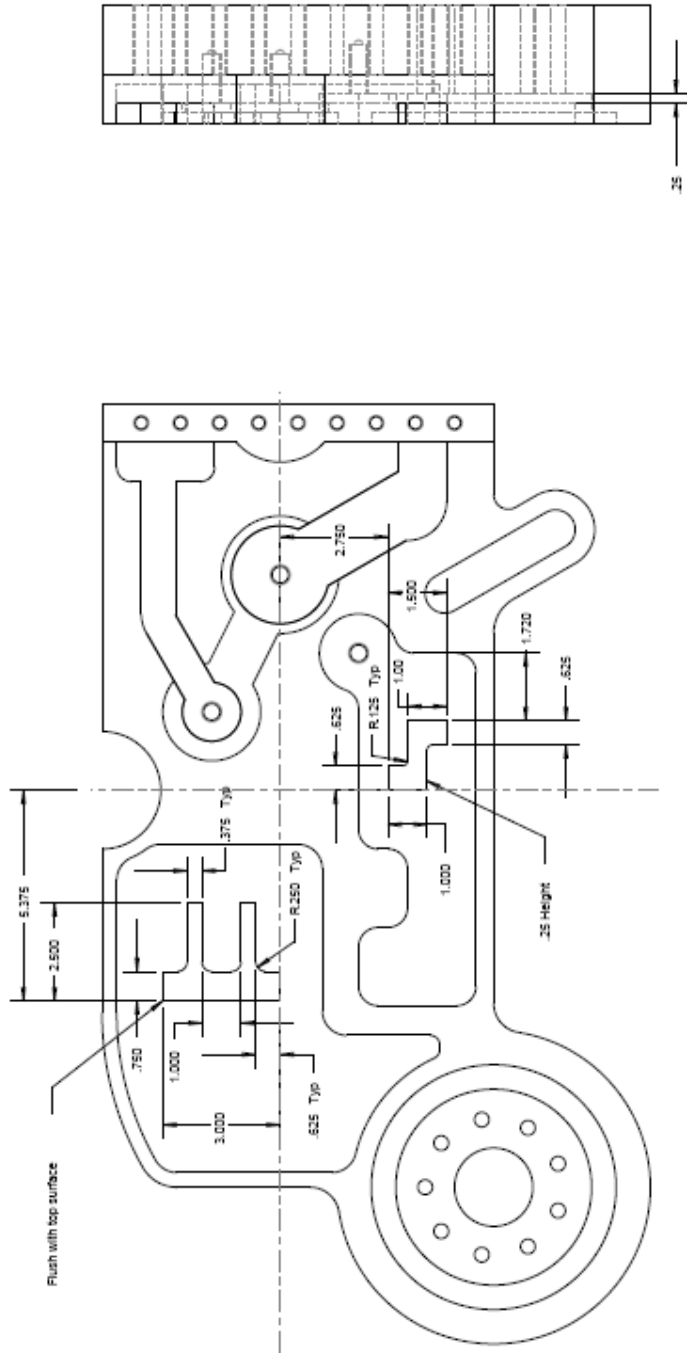
Comparing the machined stock to the part geometry.

Generating the NC- code.







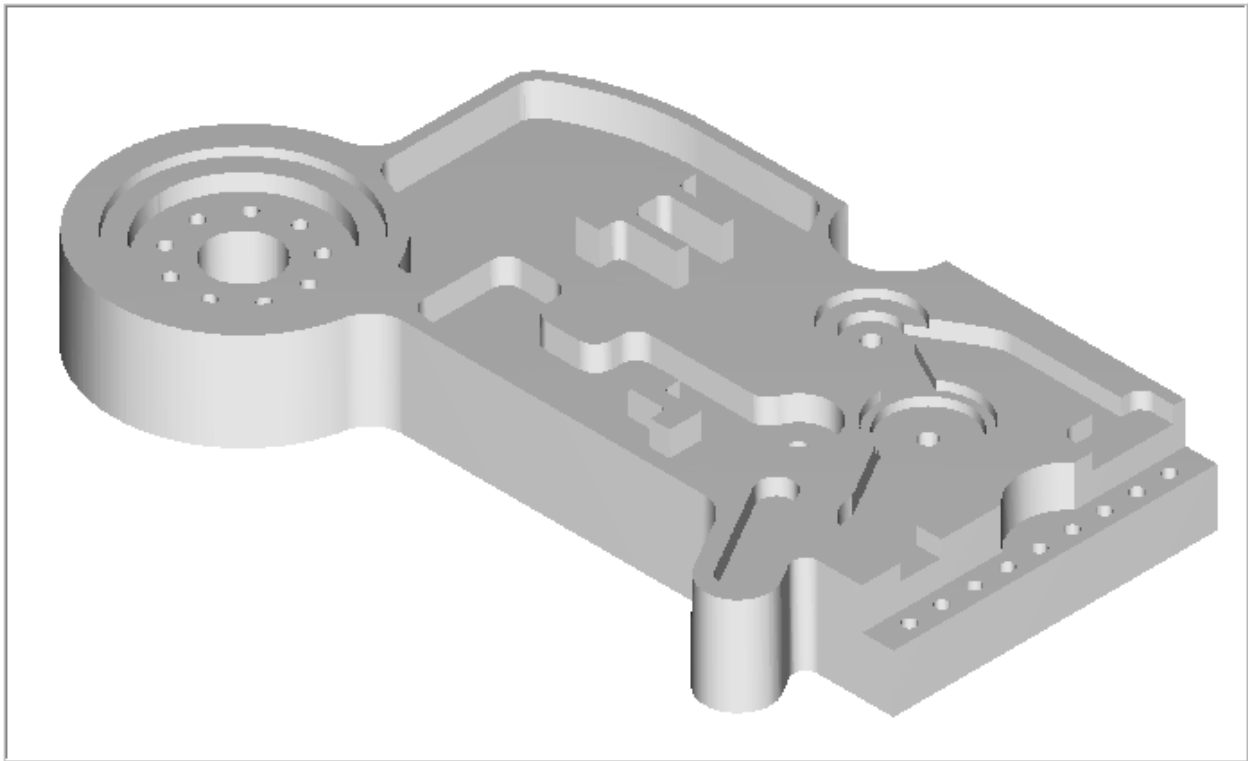


Advanced Exercise #3
Sheet 4 of 4
Material: Aluminum T6O61
All Dimensions in Inches

CAMInstructor.COM

**TOOL LIST**

- 6.0" face mill
- 1.0" diameter flat end mill
- 0.75" diameter flat end mill
- 0.625" diameter flat end mill
- 0.5" diameter flat end mill
- 0.375" diameter flat end mill
- 0.25" diameter flat end mill
- 0.125" diameter flat end mill
  
- 0.5" spot drill
- 1/2"-13 UNC tap
- 0.4219" drill
- 0.375" drill
- 3/8"-16 tap
- 0.3125" drill



# Mastercam Training Guide

Proj./Part No.: 0  
 Drawing No. : 1  
 Prog. No. : 0

Date : 10/01/08  
 Customer : -  
 Programmer : 1



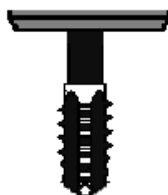
Tool type : 6 Face mill  
 Manufact.code :  
 Chuck :  
 Tool Number : 1  
 Diameter : 6 RPM : 254 Feedrate : 4.064  
 Corner radius : 0 Tip angle : 45 Plunge feed r.: 2.032  
 Flute length : 0.75 Material : ALUMINUM ... Length offset : 1  
 Overall length: 3 No flutes : 4



Tool type : 0.375 Drill Drill  
 Manufact.code :  
 Chuck :  
 Tool Number : 2  
 Diameter : 0.375 RPM : 2444 Feedrate : 23.4624  
 Corner radius : 0 Tip angle : 118 Plunge feed r.: 23.4624  
 Flute length : 2.5375 Material : ALUMINUM ... Length offset : 2  
 Overall length: 3.0704 No flutes : 2



Tool type : 0.5 Spot Drill 1/2 SPOTDRILL  
 Manufact.code :  
 Chuck :  
 Tool Number : 4  
 Diameter : 0.5 RPM : 1833 Feedrate : 7.332  
 Corner radius : 0 Tip angle : 90 Plunge feed r.: 7.332  
 Flute length : 2 Material : ALUMINUM ... Length offset : 4  
 Overall length: 3 No flutes : 2



Tool type : 0.375 Tap RH 3/8-16 UNC  
 Manufact.code :  
 Chuck :  
 Tool Number : 5  
 Diameter : 0.375 RPM : 2444 Feedrate : 152.75  
 Corner radius : 0 Tip angle : 118 Plunge feed r.: 152.75  
 Flute length : 2 Material : ALUMINUM ... Length offset : 5  
 Overall length: 2.42 No flutes : 1



Tool type : 0.5 Endmill1 Flat  
 Manufact.code :  
 Chuck :  
 Tool Number : 6  
 Diameter : 0.5 RPM : 3056 Feedrate : 48.896  
 Corner radius : 0 Tip angle : 0 Plunge feed r.: 24.448  
 Flute length : 3 Material : ALUMINUM ... Length offset : 6  
 Overall length: 3.63 No flutes : 4

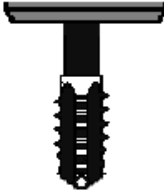

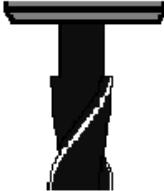






Tool type : 0.75 Endmill1 Flat  
 Manufact.code :  
 Chuck :  
 Tool Number : 7  
 Diameter : 0.75 RPM : 2037 Feedrate : 32.592  
 Corner radius : 0 Tip angle : 0 Plunge feed r.: 16.296  
 Flute length : 3 Material : ALUMINUM ... Length offset : 7  
 Overall length: 3.63 No flutes : 4



Tool type : 0.3125 Drill Drill  
 Manufact.code :  
 Chuck :  
 Tool Number : 8  
 Diameter : 0.3125 RPM : 2933 Feedrate : 28.1568  
 Corner radius : 0 Tip angle : 118 Plunge feed r.: 28.1568  
 Flute length : 1.9439 Material : ALUMINUM ... Length offset : 8  
 Overall length: 2.3521 No flutes : 2



	<p>Tool type : 0.5 Tap RH 1/2-13 UNC            Manufact.code :            Chuck :            Tool Number : 9            Diameter : 0.5 RPM : 1833            Corner radius : 0 Tip angle : 118            Flute length : 2 Material : ALUMINUM ...            Overall length: 2.42 No flutes : 1</p>	<p>Feedrate : 141            Plunge feed r.: 141            Diam. offset : 9            Length offset : 9</p>
	<p>Tool type : 0.4219 Drill Drill            Manufact.code :            Chuck :            Tool Number : 10            Diameter : 0.4219 RPM : 2173            Corner radius : 0 Tip angle : 118            Flute length : 2 Material : ALUMINUM ...            Overall length: 2.42 No flutes : 2</p>	<p>Feedrate : 20.8608            Plunge feed r.: 20.8608            Diam. offset : 10            Length offset : 10</p>
	<p>Tool type : 0.25 Endmill11 Flat            Manufact.code :            Chuck :            Tool Number : 24            Diameter : 0.25 RPM : 6112            Corner radius : 0 Tip angle : 0            Flute length : 0.75 Material : ALUMINUM ...            Overall length: 0.9075 No flutes : 4</p>	<p>Feedrate : 97.792            Plunge feed r.: 48.896            Diam. offset : 24            Length offset : 24</p>
	<p>Tool type : 0.125 Endmill11 Flat            Manufact.code :            Chuck :            Tool Number : 25            Diameter : 0.125 RPM : 12224            Corner radius : 0 Tip angle : 0            Flute length : 0.75 Material : ALUMINUM ...            Overall length: 0.9075 No flutes : 4</p>	<p>Feedrate : 195.584            Plunge feed r.: 97.792            Diam. offset : 25            Length offset : 25</p>
	<p>Tool type : 0.375 Endmill11 Flat            Manufact.code :            Chuck :            Tool Number : 31            Diameter : 0.375 RPM : 4074            Corner radius : 0 Tip angle : 0            Flute length : 1 Material : ALUMINUM ...            Overall length: 1.21 No flutes : 4</p>	<p>Feedrate : 65.184            Plunge feed r.: 32.592            Diam. offset : 31            Length offset : 31</p>
	<p>Tool type : 1 Endmill11 Flat            Manufact.code :            Chuck :            Tool Number : 33            Diameter : 1 RPM : 1528            Corner radius : 0 Tip angle : 0            Flute length : 3 Material : ALUMINUM ...            Overall length: 3.63 No flutes : 4</p>	<p>Feedrate : 24.448            Plunge feed r.: 12.224            Diam. offset : 33            Length offset : 33</p>
	<p>Tool type : 0.625 Endmill11 Flat            Manufact.code :            Chuck :            Tool Number : 35            Diameter : 0.625 RPM : 2444            Corner radius : 0 Tip angle : 0            Flute length : 3 Material : ALUMINUM ...            Overall length: 3.63 No flutes : 4</p>	<p>Feedrate : 39.104            Plunge feed r.: 19.552            Diam. offset : 35            Length offset : 35</p>

# MILL-LESSON-FBM-2 - THE PROCESS

## Toolpath Creation

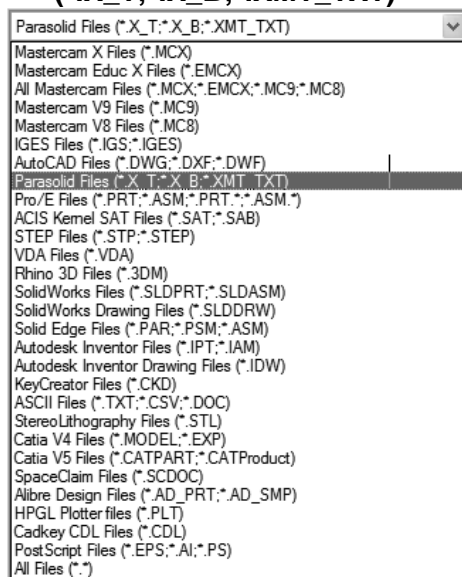
- TASK 1:** Import part geometry from Parasolid file
- TASK 2:** Setting the environment
- TASK 3:** Save part geometry for later inspection
- TASK 4:** Define the rough stock using stock setup
- TASK 5:** Run FBM Mill for all milling operations
- TASK 6:** Verify the toolpath, comparing results to part geometry, and saving out in-process stock
- TASK 7:** Run FBM Drill for all hole making operations
- TASK 8:** Verify the toolpath using saved stock
- TASK 9:** Save the updated MCX file
- TASK 10:** Post and create the CNC code file

### TASK 1:

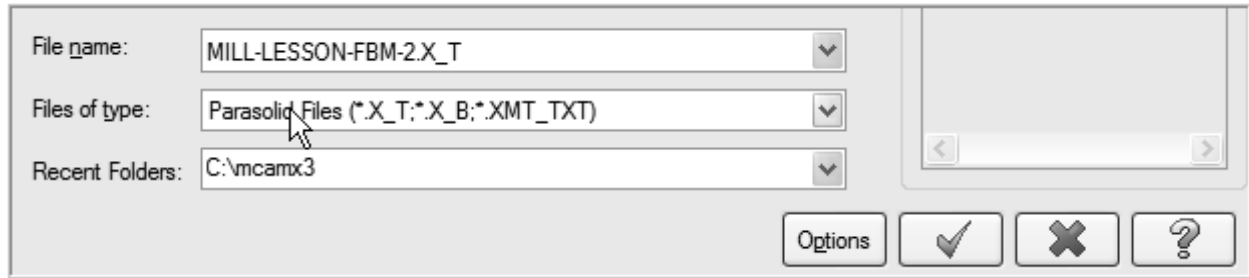
#### IMPORT PART GEOMETRY FROM PARASOLID




➔ In this lesson you will use an imported solid model.

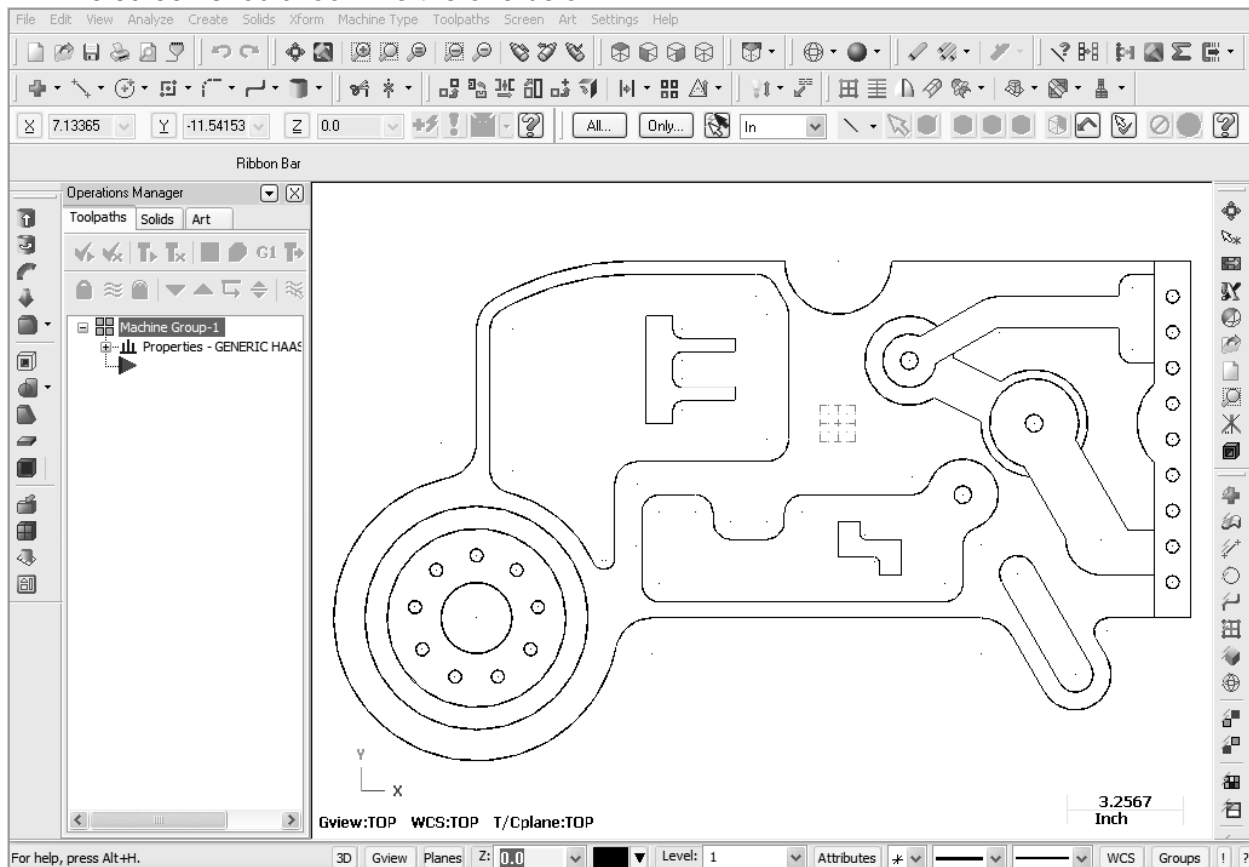
1. Click on **File>Open**
2. Beside **Files of Type** click on the drop down arrow and select **Parasolid Files (\*.X\_T;\*.X\_B;\*.XMT\_TXT)**



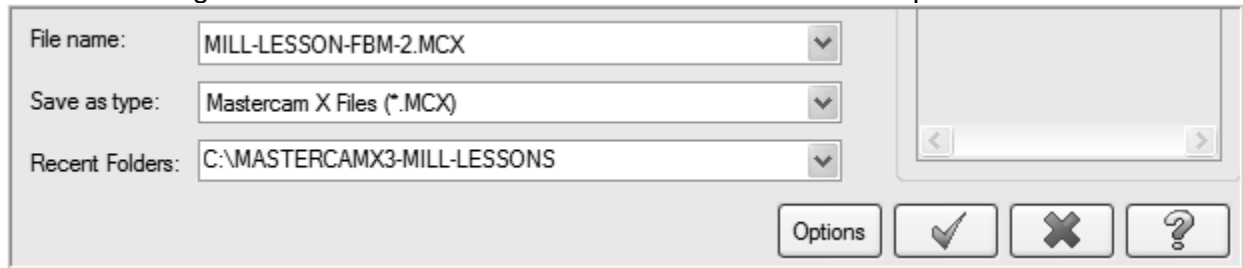
- On the multimedia CD that came with this text is a **folder called Mastercam-Files** browse to this location and select **MILL-LESSON-FBM-2.X\_T**



- Select the green check mark button  to load the file and complete this function.
- Press and hold the **Alt** key on your keyboards and type **s** to toggle shading in Mastercam. This Key mapping shortcut is often denoted as **<Alt+S>**.
- Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen .
- For a better view of the part use the toolbar at the top of the screen to change the graphics view to **Isometric**. Review the features that make up this part.
- Press and hold the **Alt** key on your keyboards and type **s** to toggle wireframe in Mastercam.
- Change the graphics view to the Top View by using the toolbar at the top of the screen.
- Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen .
- The screen should look like the one below:



12. Select **File>Save as...**
13. In the File name box, type **Mill-Lesson-FBM-2**.
14. Save to an appropriate location.
15. Select the green check mark button  to save the file and complete this function.

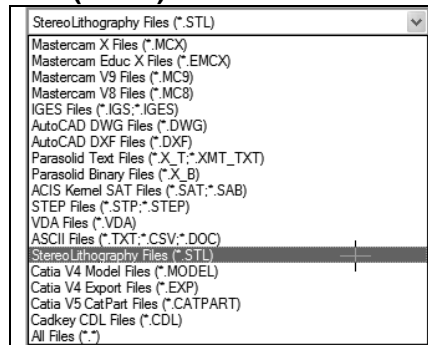


### TASK 2: SETTING THE ENVIRONMENT

- Before starting the geometry import you should set up the grid, toolbars and machine type as outlined in the **Setting the environment** section at the beginning of this text:
1. Set up the Grid. This will help identify the location of the origin.
  2. Customize the toolbars to machine a 2D part.
  3. Set the machine type to a Haas Vertical Spindle CNC machine.

### TASK 3: SAVE PART GEOMETRY FOR LATER INSPECTION

1. Select **File>Save as...**
2. Beside **Save as type:** click on the drop down arrow and select **StereoLithography Files (\*.STL)**

	<p><b>STL files</b> are used with StereoLithography systems. An STL file comprises triangular facets of data that represent surface and solid models.</p> <p>Mastercam can read and write both binary and ASCII STL files.</p>
---	--

3. In the File name box it should read **Mill-Lesson-FBM-2**.
4. Save to an appropriate location – typically **C:\Mcamx\data** for STL files
5. Select the green check mark button  to save the file and complete this function.

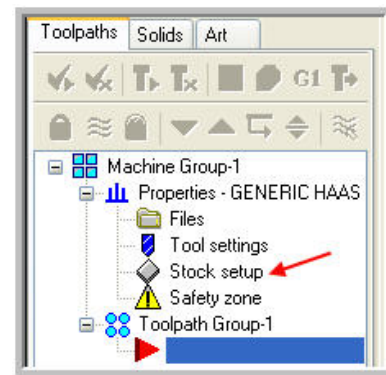
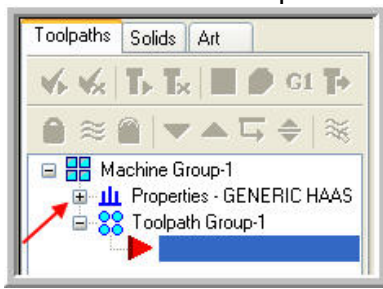
# Toolpath Creation

## TASK 4:

### DEFINE THE ROUGH STOCK USING STOCK SETUP

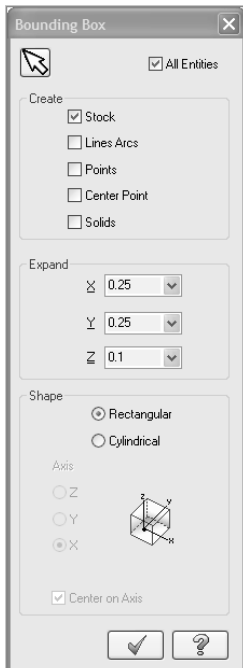
➤ Defining the Rough Stock using stock setup.

1. **Fit to screen** using the Key mapping shortcut **<Alt+F1>**
2. Select the **plus** in front of **Properties** to expand the Toolpaths Group Properties. **<Alt+O>** will **Show/Hide** Operations Manager pane.



3. Select **Stock setup** in the toolpath manager window as shown above on the right:
  - The bottom face of the material has been pre-machined and is not part of the toolpath operations for this lesson.
  - The instructions for this lesson require plus 0.1" stock on thickness, and additional stock in length and width is left to our discretion.
  - **Z zero** is at the **bottom of the part**.
4. From the Machine Group Properties Stock Setup tab, select the **Bonding box** button
 

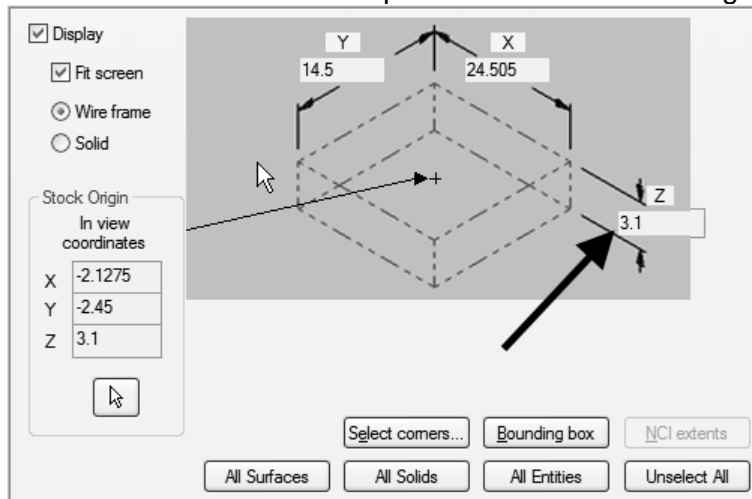
Bonding box
5. You will create stock from **All Entities** with **Expand** entries of 0.25 in X and Y, and .1 in Z.



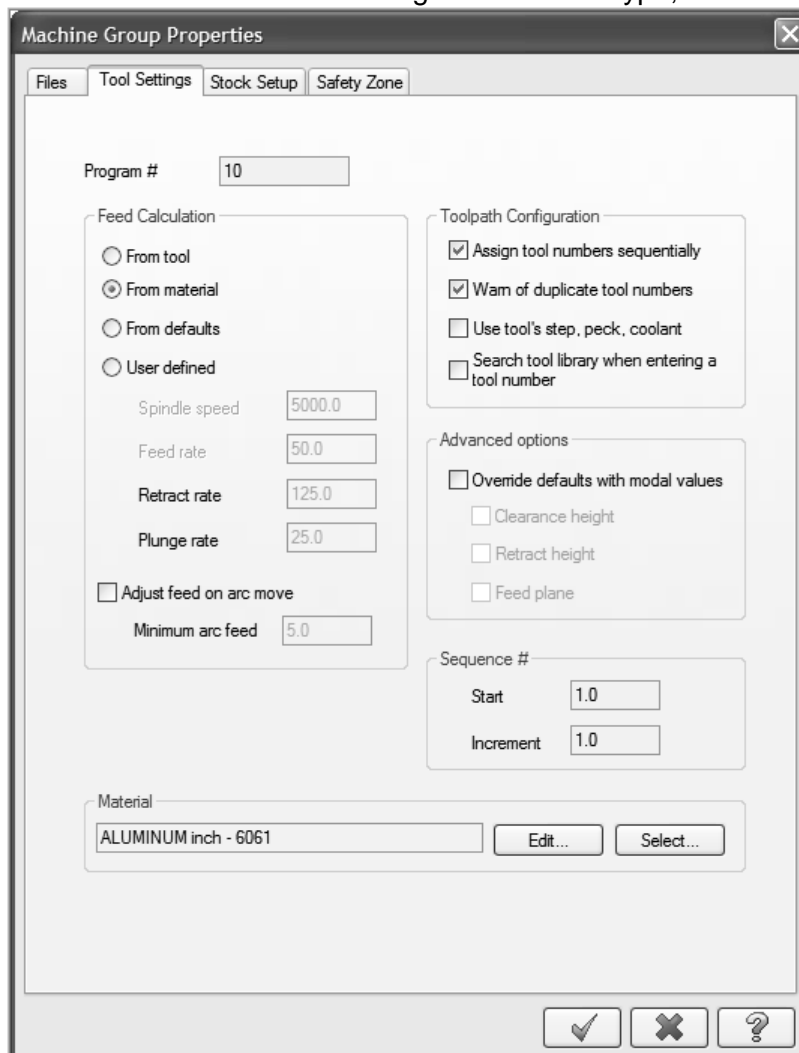
6. Select the green check mark button  to close this dialog box.

## Mastercam Training Guide

7. **Expand** adds stock values to each face. In order to remove the additional .1” from the bottom of the stock – the pre-machined face – change the Z thickness value to 3.1”

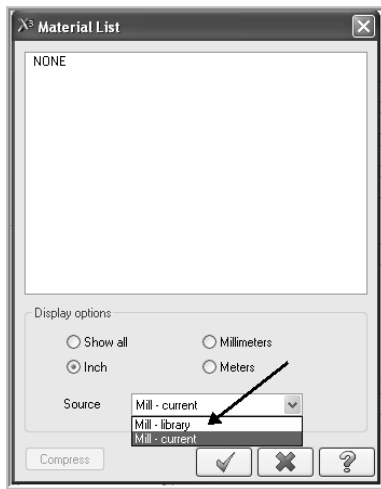


8. Select the **Tool Settings** tab and change the parameters to match the Tool Settings screenshot below. To change the Material type, follow the next set of instructions.

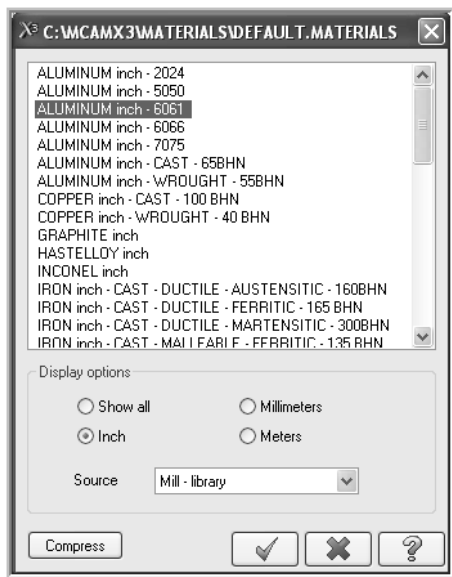


9. To change the **Material type to Aluminium 6061** pick the **Select** button at the bottom of the Tool Settings page

10. As you change the **Feed Calculation** to **From Material** you will be prompted to pick a material from the **Material List** dialog box. Change the **Source** to **Mill - library**



11. From the Default Materials list select **ALUMINIUM inch - 6061** and then select .



With the selection of **Feed Calculation From material** in the **Machine Group Properties Tool Settings** tab, all feeds and speeds are set based on calculated values.

Many users take the time to create a custom tool library with only their tools, with desired feeds and speeds set in the library.

Feeds and speeds can be edited in the resulting FBM child operations if desired.

12. Select the OK button  again to complete this Stock Setup function.

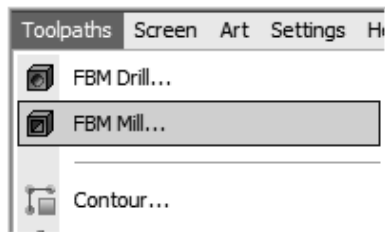
## TASK 5: RUN FBM MILL FOR ALL MILLING OPERATIONS

- In this task you configure settings in FBM Mill to set up tools and interrogate the solid to create machining regions and apply toolpath strategies.

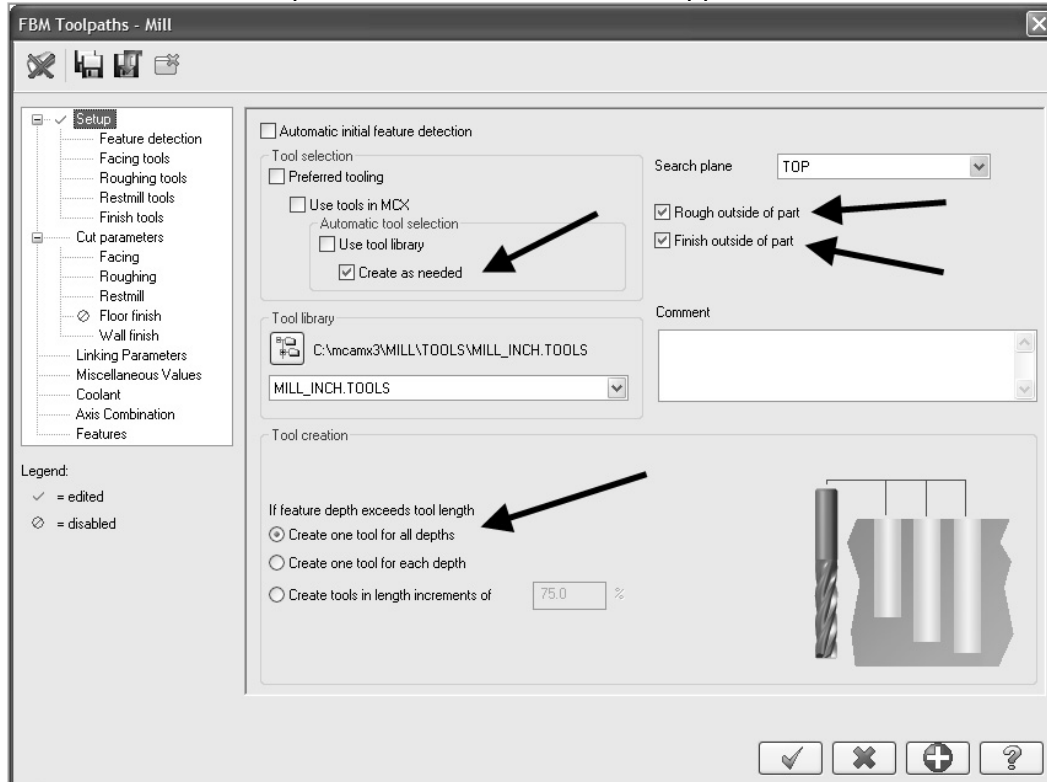
FBM Mill pays close attention to flute length during automatic tool selection. If allowing the use of Tools from library, shorter library tools will be selected for shallow pockets. When the pocket depth exceeds the flute length of the library tool, a longer tool with the same diameter will be created to achieve the required depth of cut.

In this exercise we will allow FBM Mill to generate all tools as required to machine the part geometry. In order to avoid multiple same diameter tools, we will avoid the use of library tools.

1. From the menu bar select **Toolpaths>FBM Mill...**



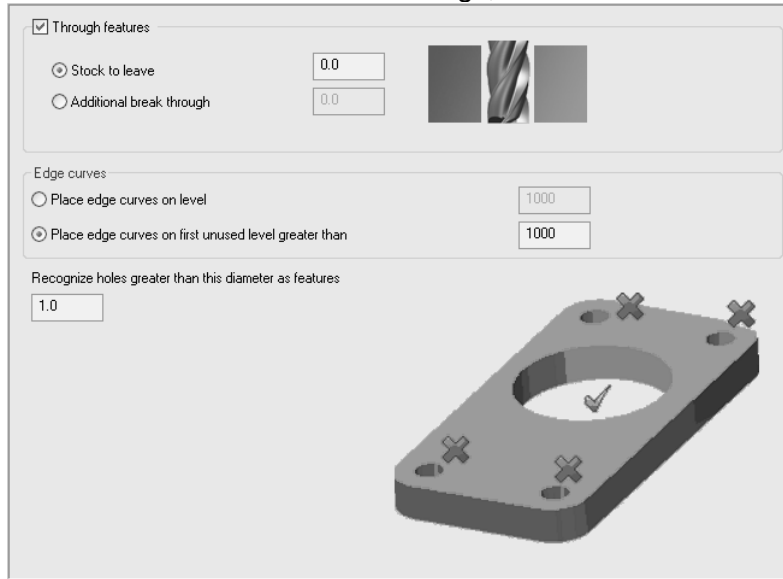
2. On the screen you will now see the **FBM Toolpaths – Mill** dialog box, starting with the Setup page. From this page you will select **Create as needed** to have FBM Mill inform us of the tools required to complete machining of the part. You will also elect to rough and finish the outside of this part. Your selections should appear as follows:



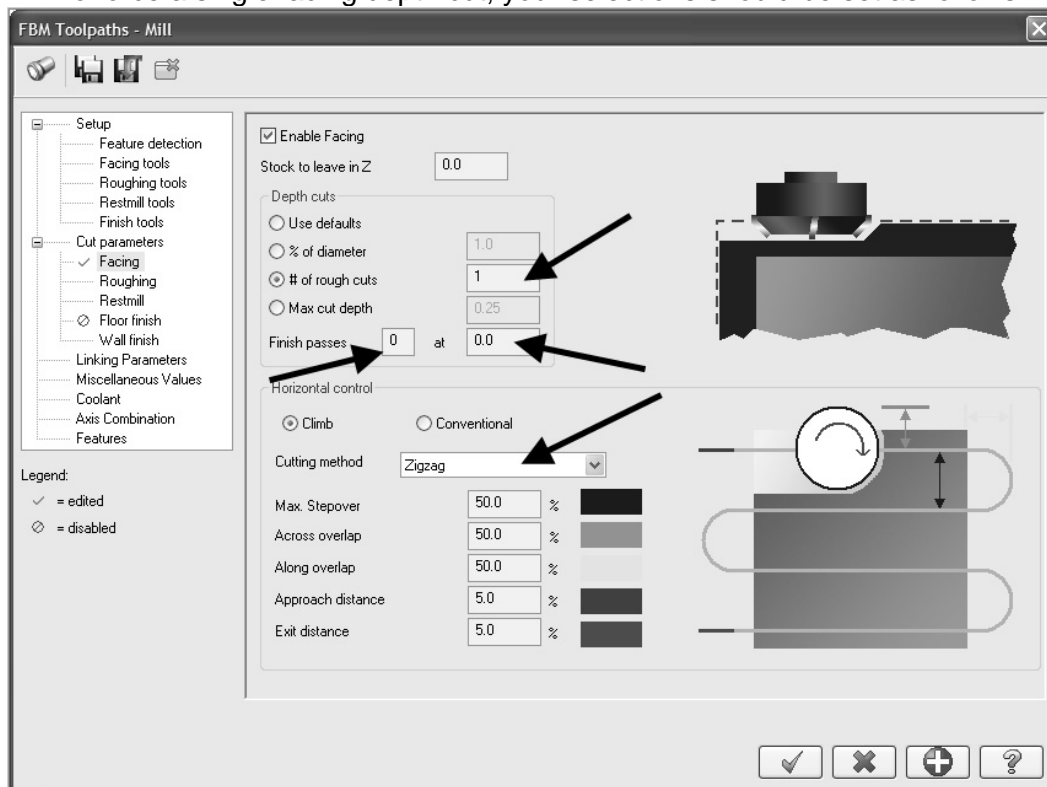
Tool numbering will appear to be non-sequential. Tools are created with the length required for the pocket being processed. If an identical diameter tool is required to be longer for a deeper pocket, the original tool will be replaced with a longer tool, but will have a higher tool number than the original.



3. Click on **Feature detection** in the Tree Control menu on the left hand side of the dialog box.
4. You will leave the default settings, which should be as follows:

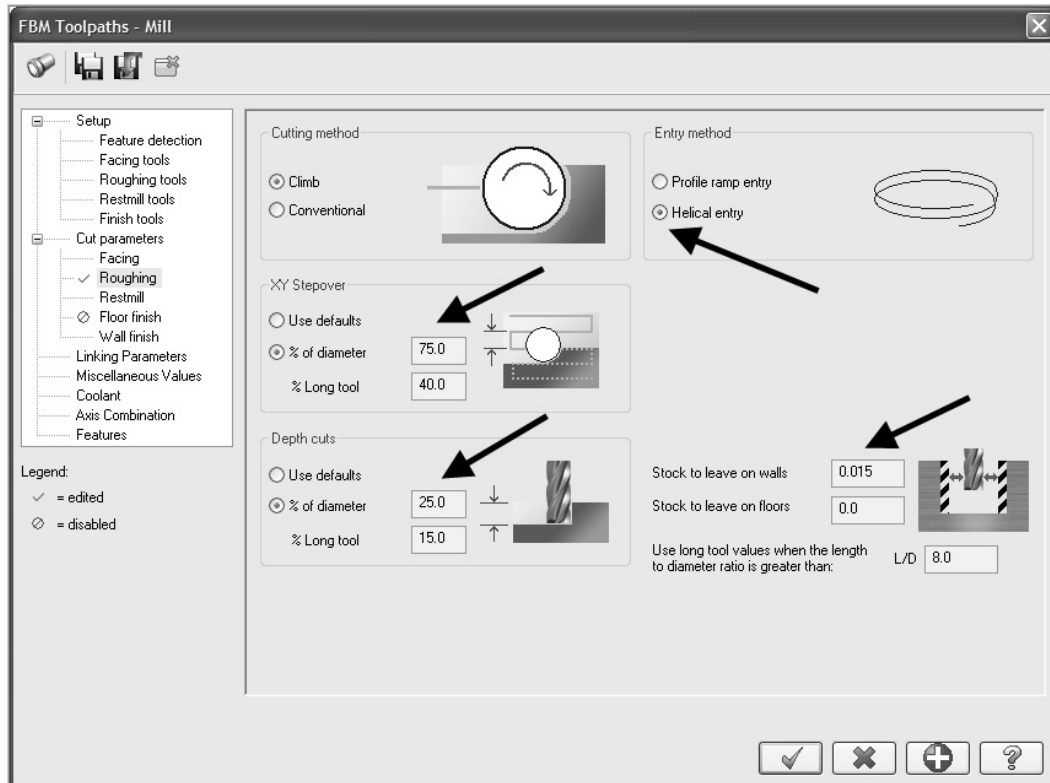


5. You will skip past the **Preferred tool list for Facing tools, Roughing tools, Restmill tools, and Finish tools.**
6. Next you will set Facing cut parameters, selecting **Facing** from the Tree Control dialog box, and adjusting settings for **1 rough cut**, and selecting **Zigzag**.  
To force a single facing depth cut, your selections should be set as follows:

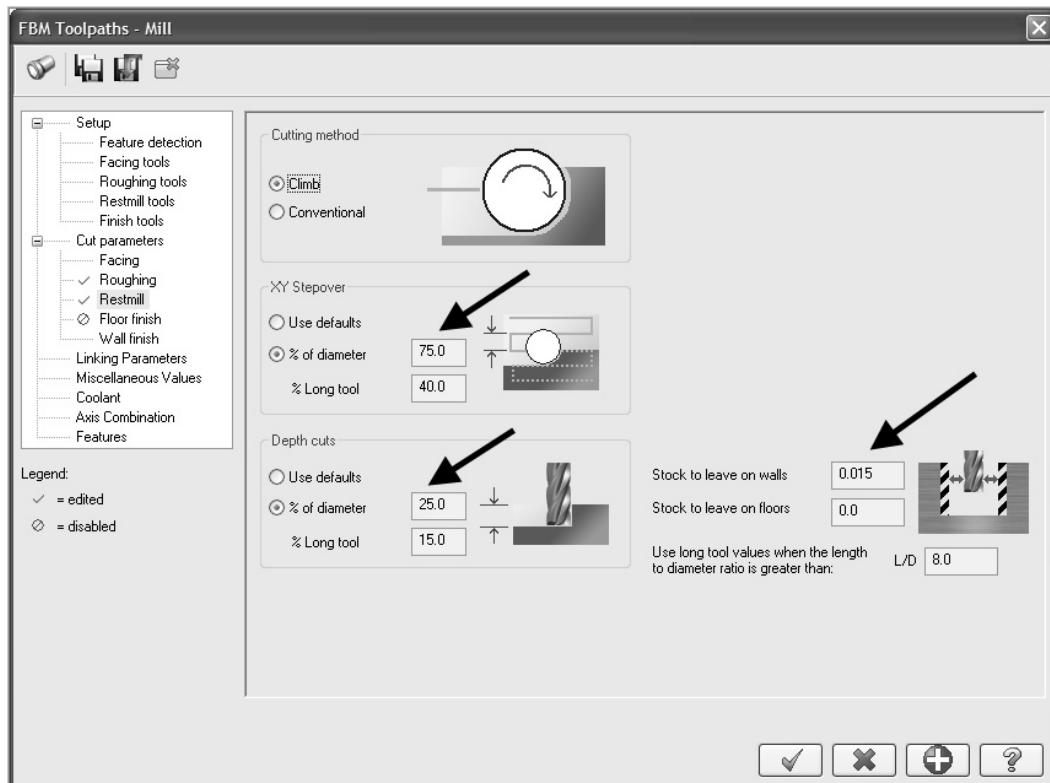


## Mastercam Training Guide

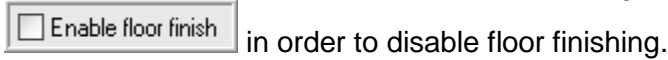
7. Next you will set Roughing cut parameters, selecting **Roughing** from the Tree Control dialog box, and adjusting settings for **Helical entry**, 75% of diameter **XY Stepover**, 25% of diameter **Depth Cuts**, and 0.015" **Stock to leave on walls**. Your selections should be set as follows:



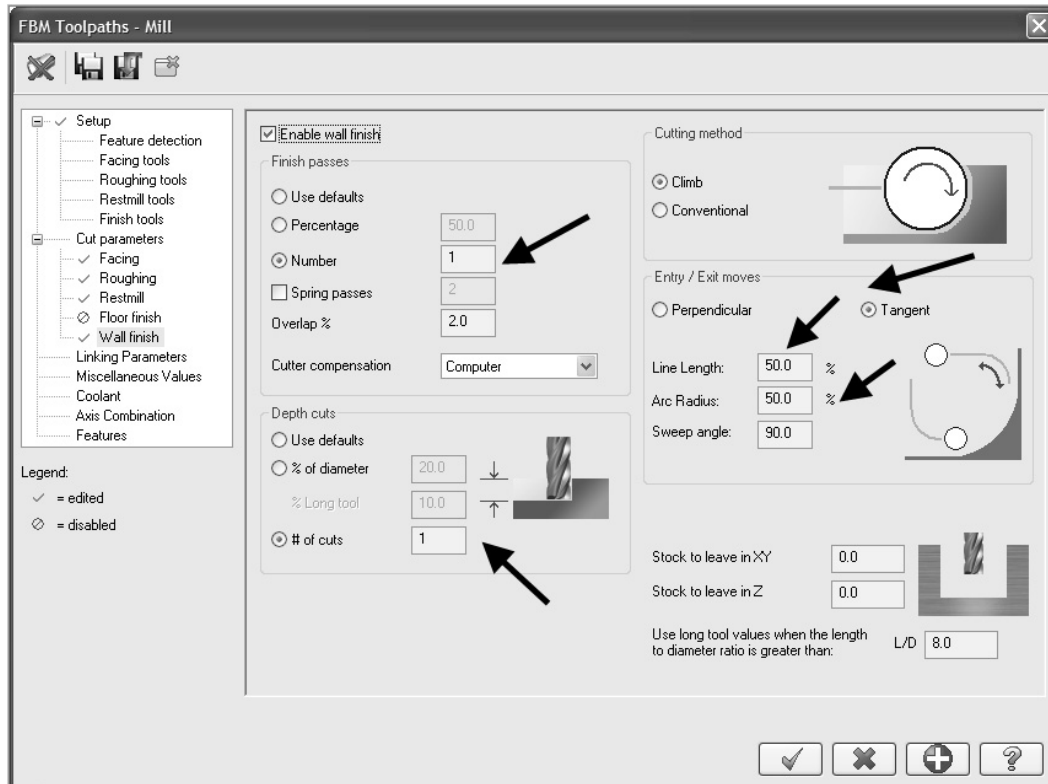
8. Next you will set Restmill cut parameters, selecting **Restmill** from the Tree Control dialog box, and adjusting settings for 75% of diameter **XY Stepover**, 25% of diameter **Depth Cuts**, and 0.015" **Stock to leave on walls**. Your selections should be set as follows:



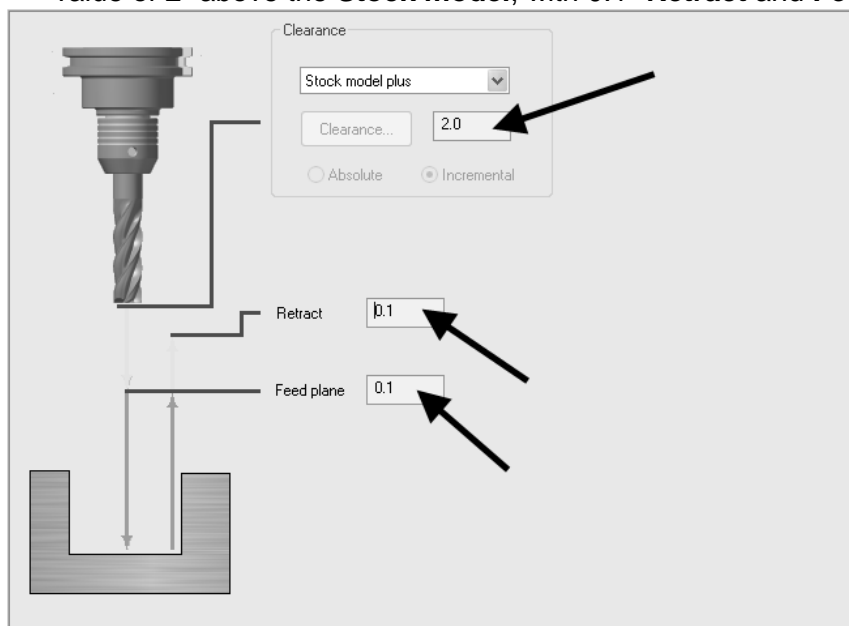
9. Select **Floor Finish** from the Tree Control dialog box and uncheck **Enable floor finish**



10. Next you will set Finish cut parameters, selecting **Wall Finish** from the Tree Control dialog box, and adjusting settings for 1 **Finish Pass**, 1 **Depth Cut**, and **Tangent Entry / Exit moves** with lines and arcs that are 50% of the tool's diameter.. Your selections should be set as follows:



11. Select **Linking Parameters** from the Tree Control dialog box. You will use a **Clearance** value of 2" above the **Stock model**, with 0.1" **Retract** and **Feed plane** values:



# Mastercam Training Guide

12. Select **Coolant** from the Tree Control dialog box. Set **Flood** to **On**:

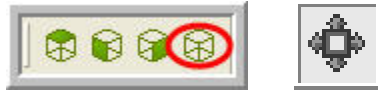


13. Select the OK button  to process all milling operations for this part.

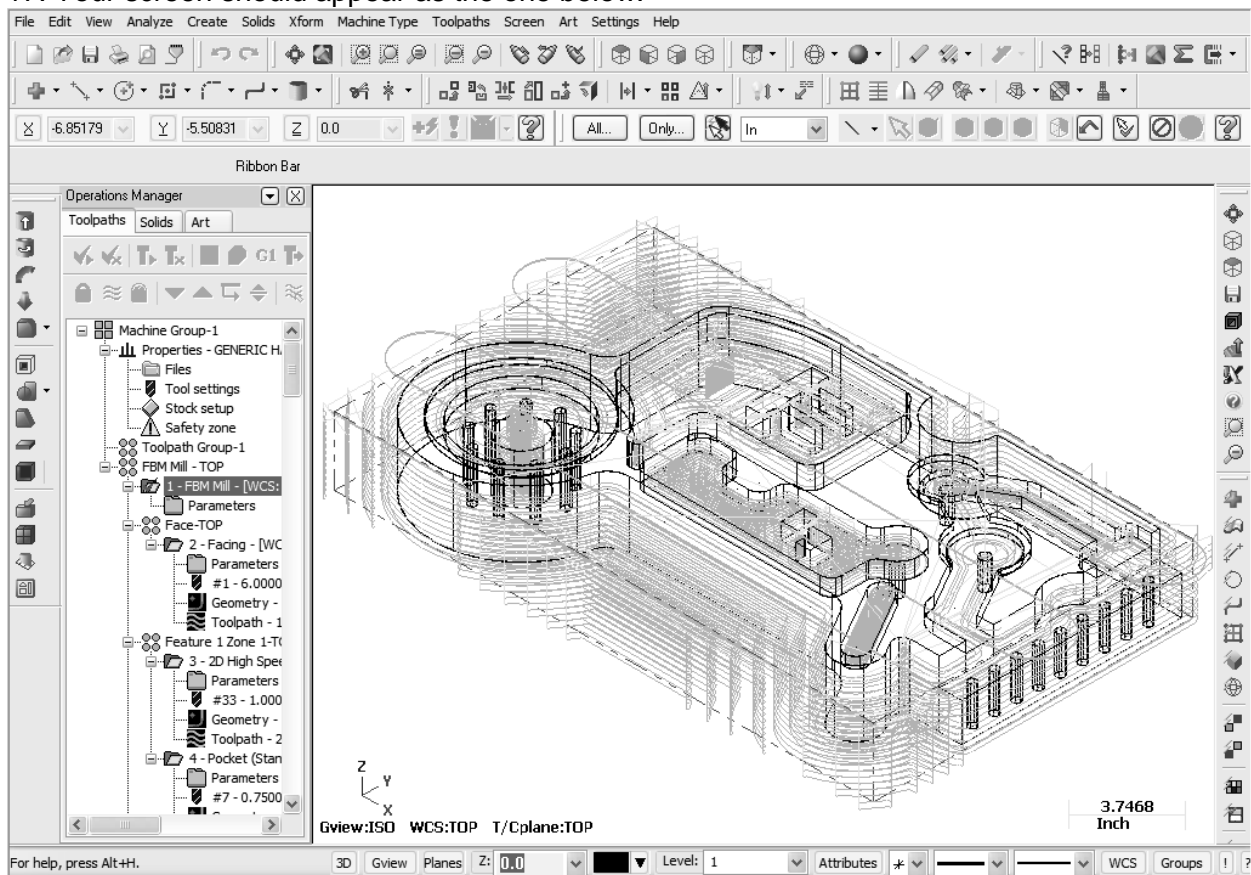
14. When prompted to **Enter new NC name** ensure **Mill-Lesson-FBM-2** is displayed and then select the OK button .

15. Process times can be significant for complete geometry, so be patient.

16. For a better view of the part use the toolbar at the top of the screen to change the graphics view to Isometric and activate a screen fit.




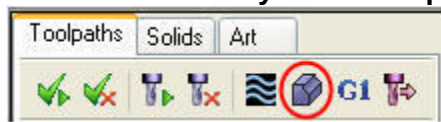
17. Your screen should appear as the one below:



## TASK 6: VERIFY THE TOOLPATH, COMPARING RESULTS TO PART GEOMETRY, AND SAVING OUT IN-PROCESS STOCK

- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.

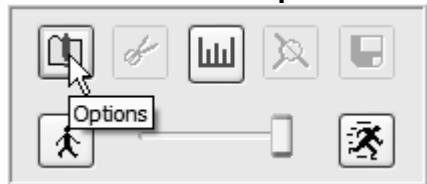
1. In the Toolpath Manager pick all the operations to verify by picking the **Select All** icon .
2. Select the **Verify selected operations** button circled below:



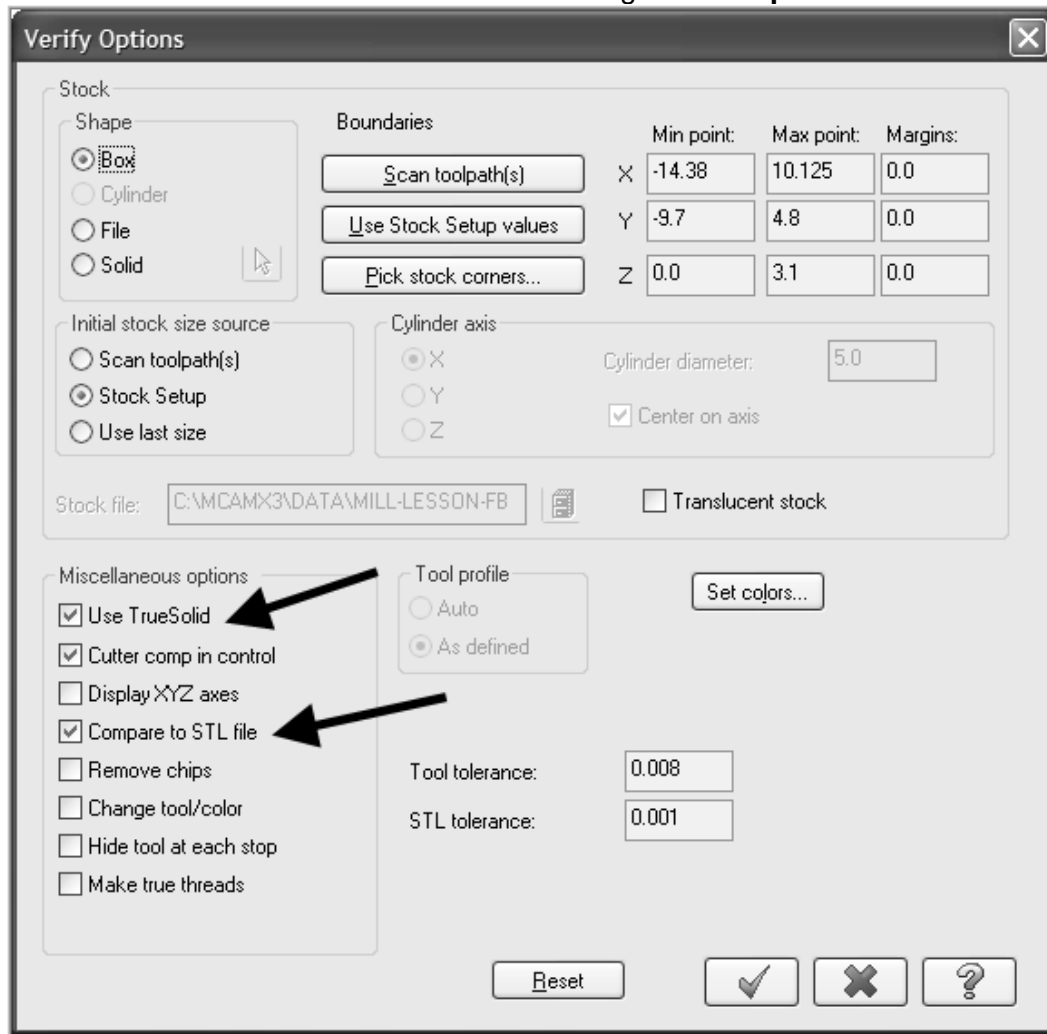
3. Select the Turbo mode.




4. Adjust the speed to the far right along the speed control bar.
5. Now select the **Options** button:




6. Ensure that **Use TrueSolid** is selected along with **Compare to STL file**.




7. Select the green check mark button  to close this dialog box.

8. Select the **Fast Forward** button  to verify the toolpaths.

9. When Verification is complete, the **STL Compare** window will open.

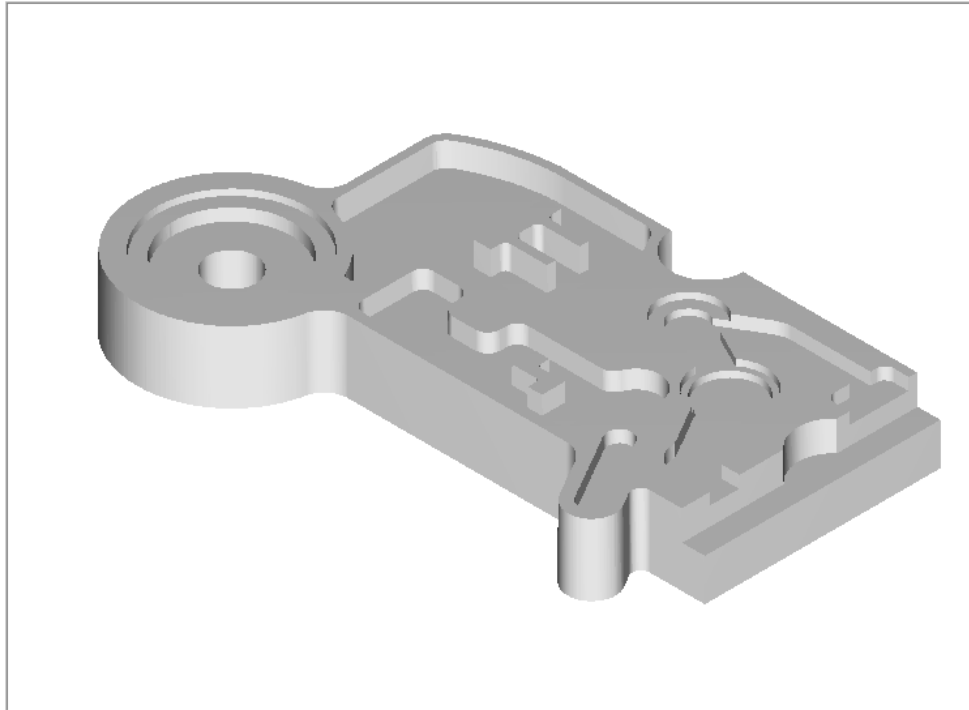
10. Click  and select the file MILL-LESSON-FBM-2.STL

11. Click  to **Compare the machined stock and the STL File**.

12. Cuts within tolerance are displayed in Green. The holes will appear as Blue as you have yet to process them. This check demonstrates that FBM Mill has addressed the features of this part.

13. Select the OK button  to return to the **Verify** dialog box.

☞ The verified toolpaths are shown below.



14. Select the **Save stock as file** button  using the filename **-MILL-LESSON-FBM-2\_MILL.STL**

15. Select the OK button  to save the in-process part geometry.

16. Select the OK button  to exit Verify.

## **TASK 7:**

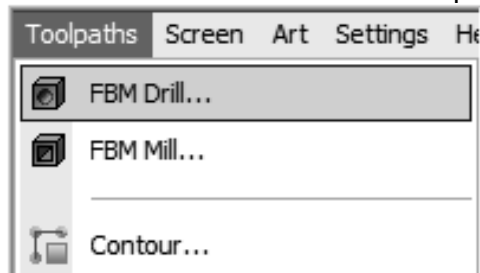
### **RUN FBM DRILL FOR ALL HOLE MAKING OPERATIONS**

☞ In this task you configure settings in FBM Drill to interrogate the solid, identify hole features, and apply holemaking strategies.

1. Change the graphics view to the Top View by using the toolbar at the top of the screen.

2. Select the Screen Fit icon found at the top of the screen to fit the part to the screen 

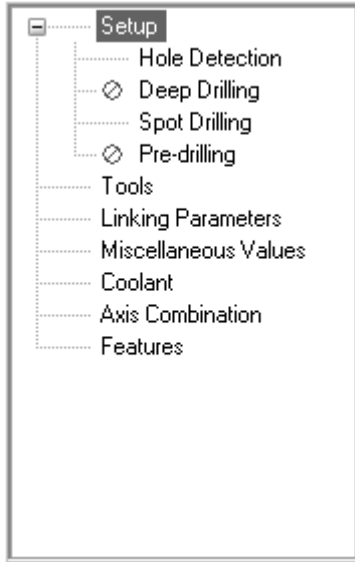
3. From the menu bar select Toolpaths>FBM Drill...



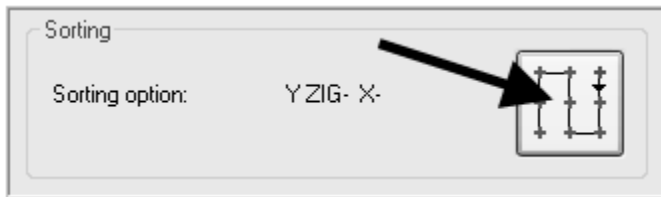
## Mastercam Training Guide

---

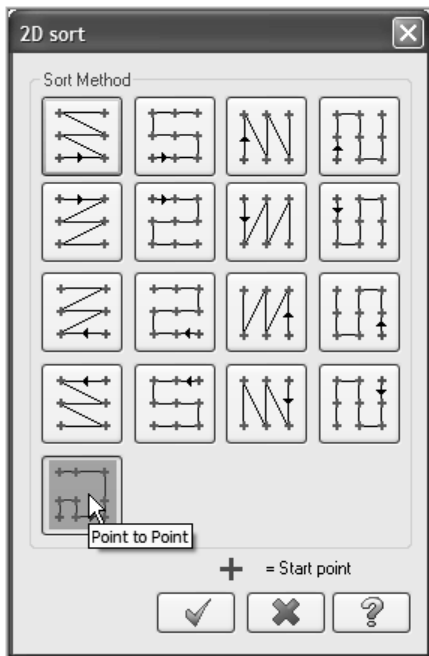
4. On the screen you will now see the **FBM Toolpaths – Drill** dialog box, starting with the Setup page selected in the Tree Control menu down the left hand side:



5. From this page you will change the **Sorting option** by click on the current Sorting method icon




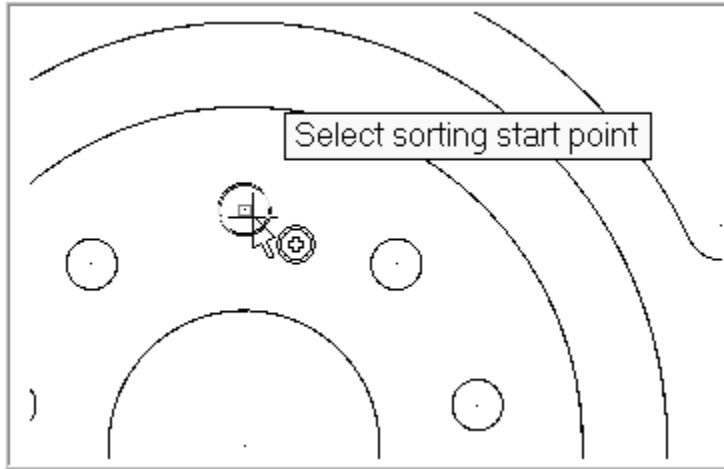
6. Select the **Point to Point 2D Sort Method**



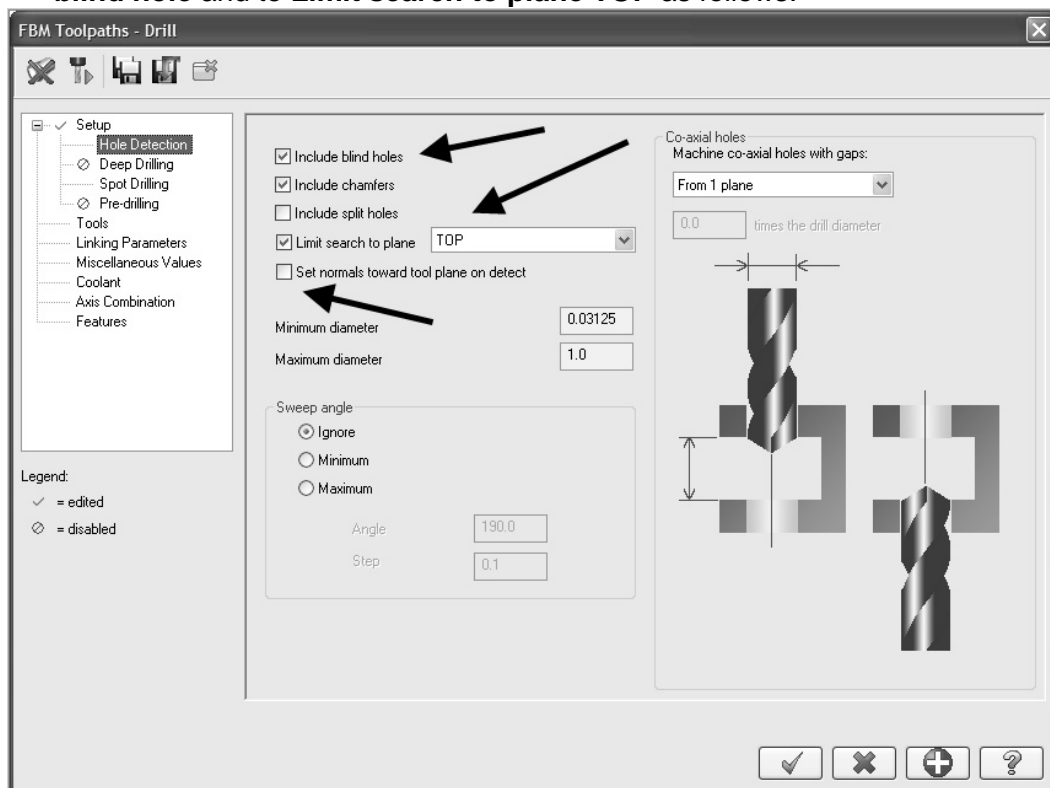
7. Select the OK button  in the **2D sort** page.



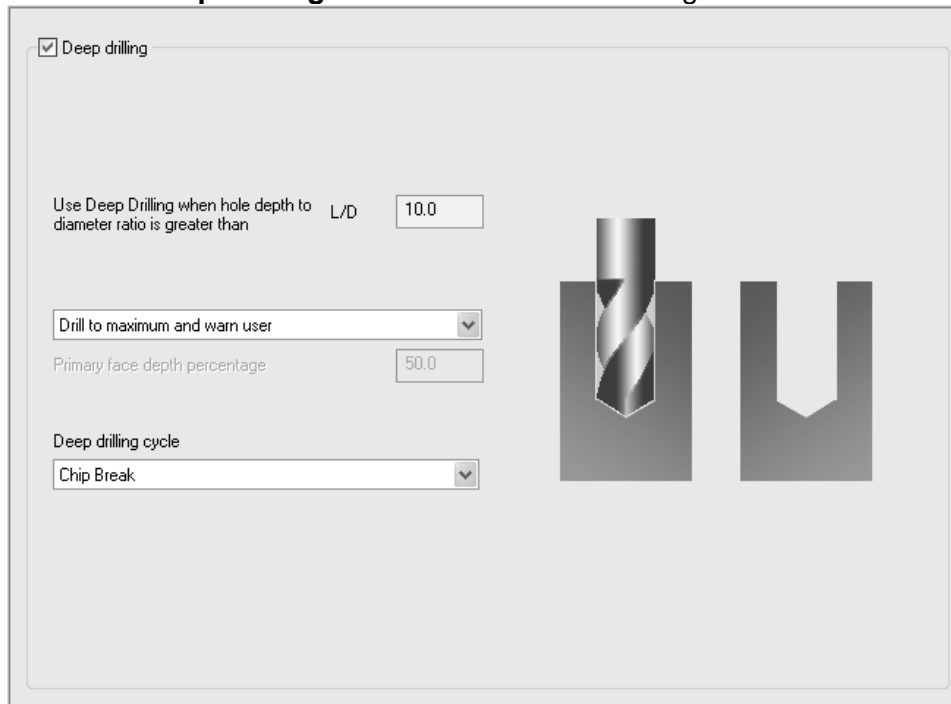
8. The prompt now changes to **Select sorting start point**. Move the mouse and position the cursor over the point at the center of circle shown below, you will notice this icon  appears as you move over the point at the center of this circle. This cross denotes you are snapping to a circle center point. Use the left button of your mouse to pick this point.



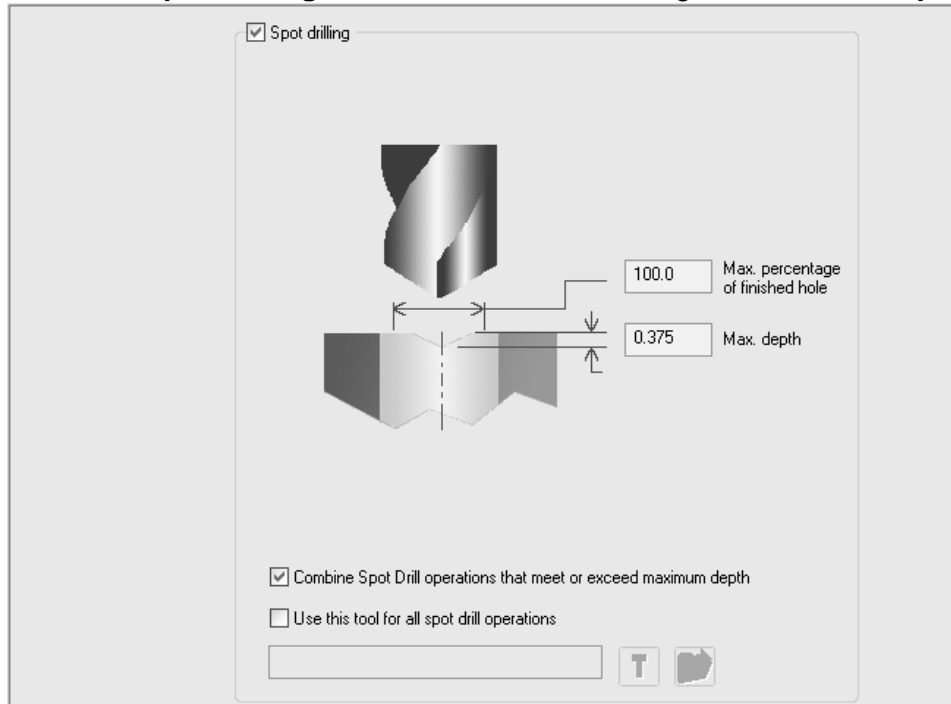
9. Select **Hole Detection** from the Tree Control dialog box and set parameters to **Include blind hole** and to **Limit search to plane TOP** as follows:



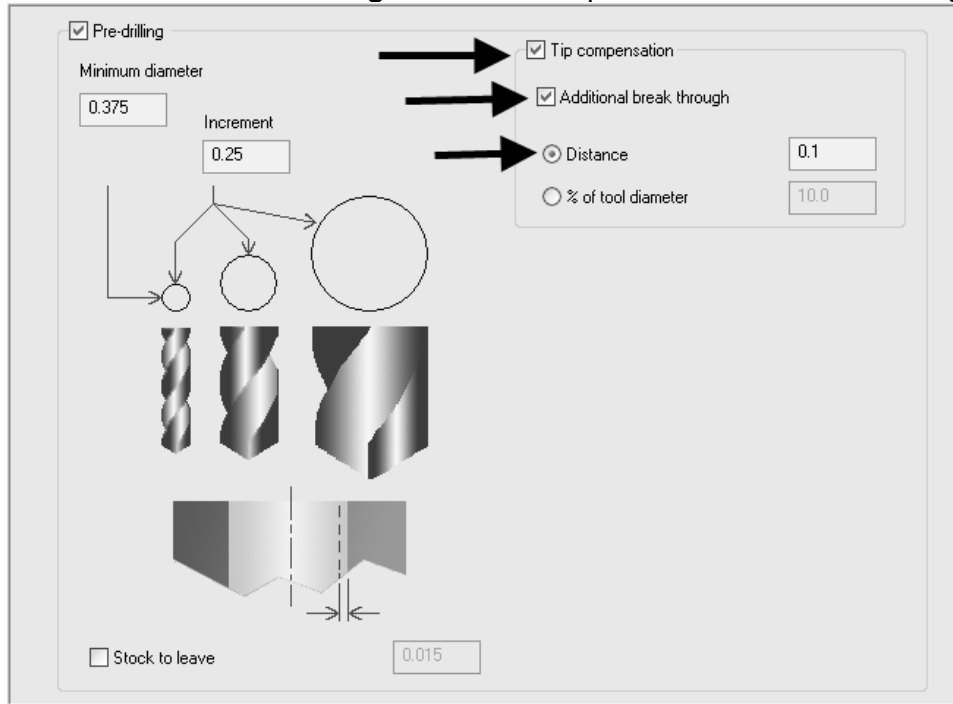
## 10. Select **Deep Drilling** from the Tree Control dialog box and check **Deep drilling**



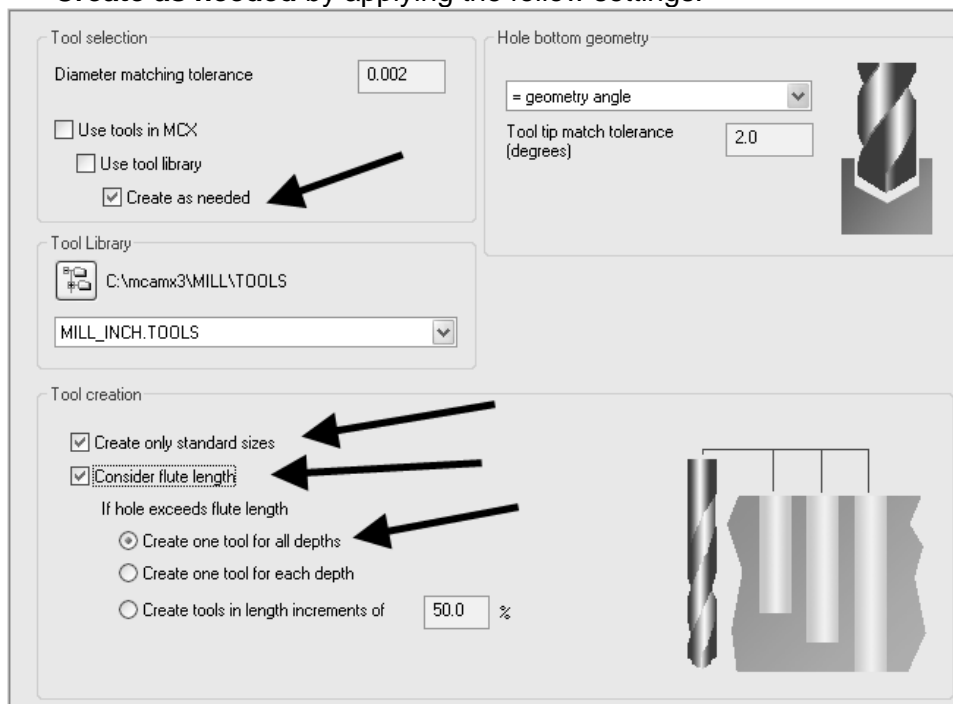
## 11. Select **Spot Drilling** from the Tree Control dialog box and check **Spot drilling**.



12. Select **Pre-drilling** from the Tree Control dialog box and check **Pre-drilling**  Pre-drilling. The 3/8-16 UNC tapped holes will require Pre-drilling with a 5/16" drill. You will use an **Additional break through** of .1" for this part which features all through holes.

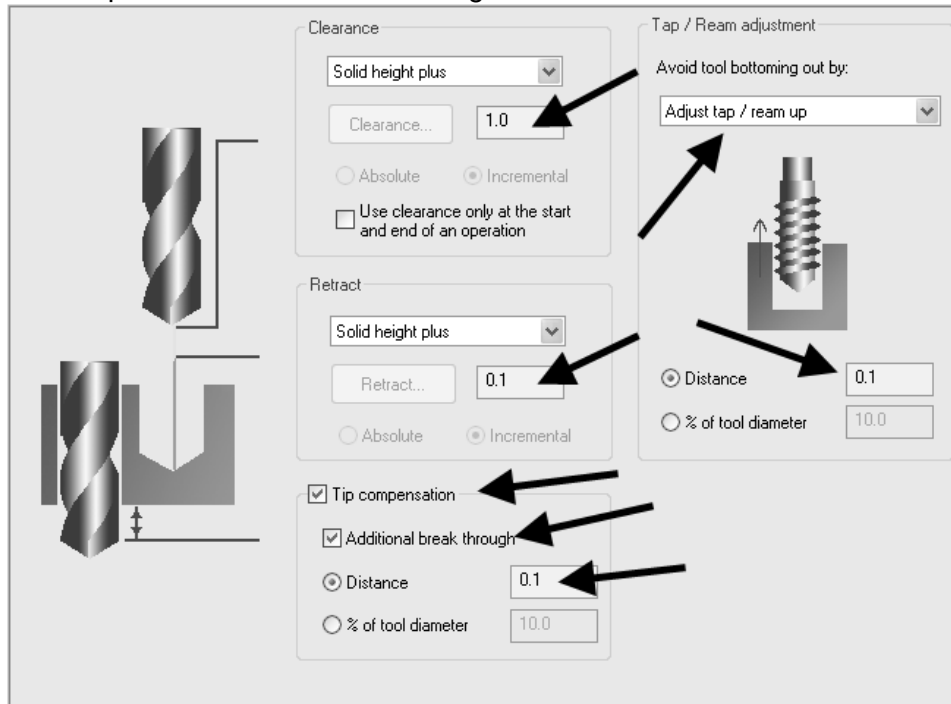


13. Select **Tools** from the Tree Control dialog box. You will allow FBM Drill to automatically **Create as needed** by applying the following settings:

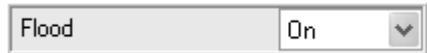


## Mastercam Training Guide

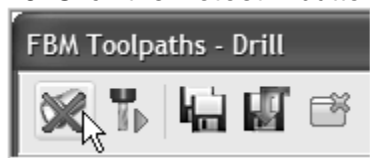
14. Select **Linking Parameters** from the Tree Control dialog box. This dialog box will be used to set our **Clearance at Solid height plus 1."**, **Retract at Solid height plus 0.1"**, an **Additional break through of .1"**, and **No adjustment** for the **Tap / Ream adjustment** for this part which features all through holes.



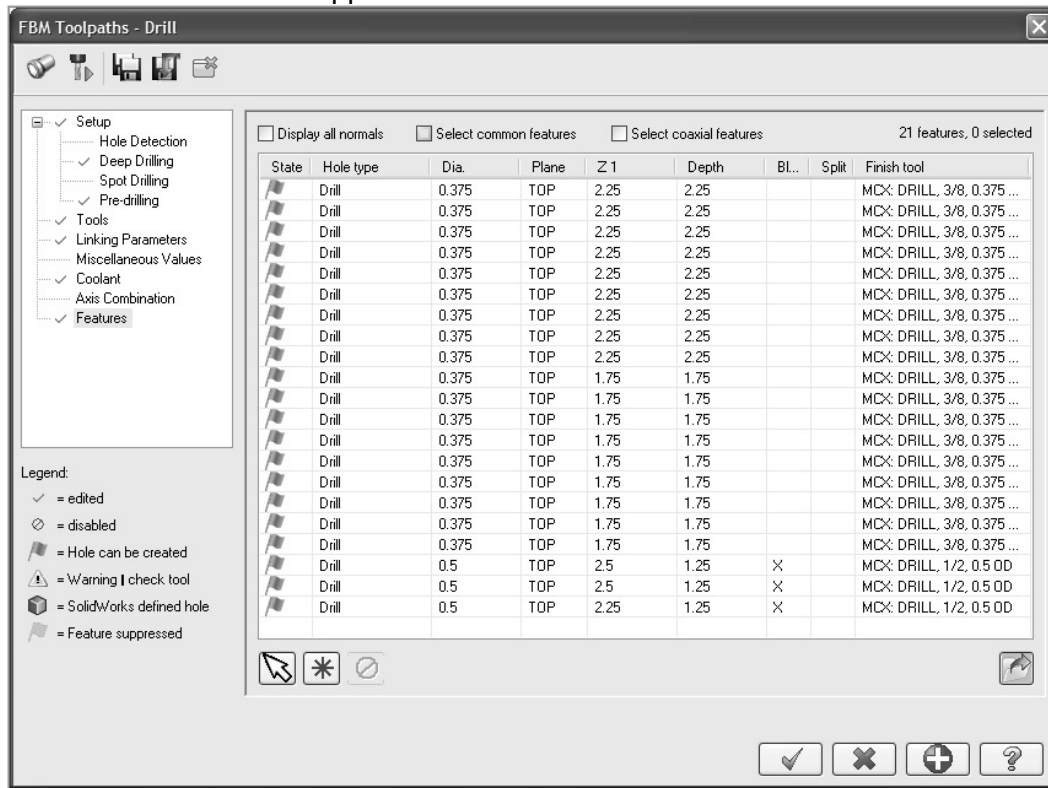
15. Select **Coolant** from the Tree Control dialog box. Set **Flood** to **On**:



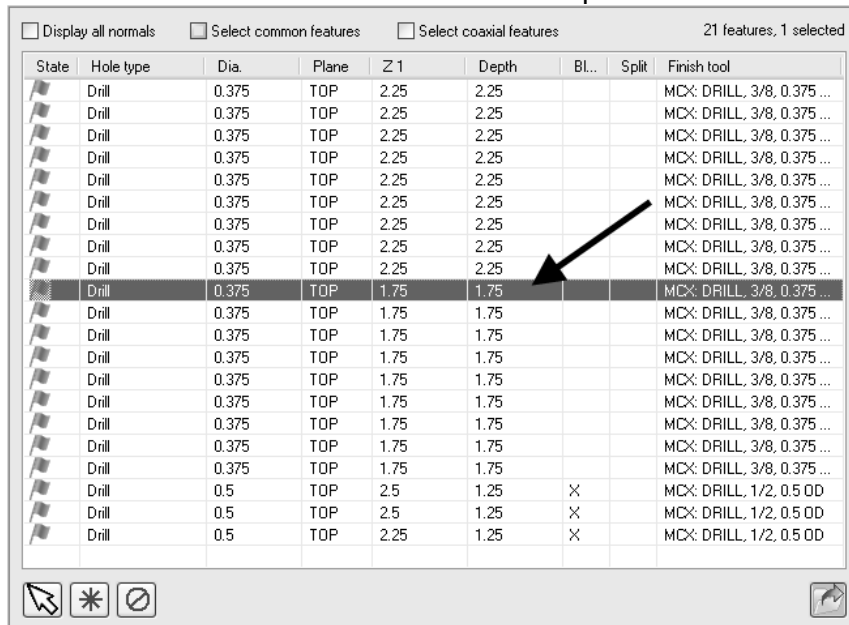
16. Click the **Detect...** button at the top of the FBM Toolpaths – Drill dialog box.



17. Your screen should appear as follows:



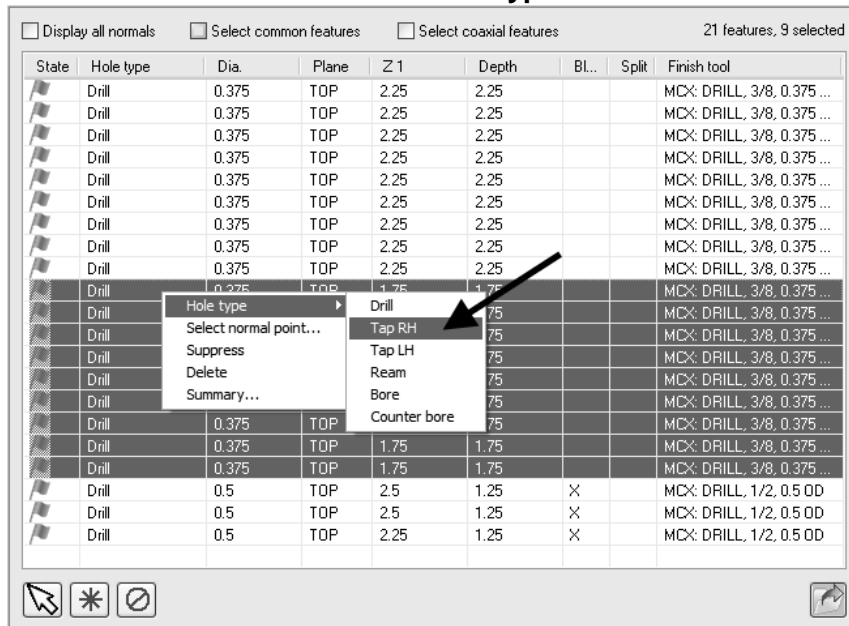
18. Left click on the first 0.375 Dia. / 1.75 Depth Hole the list.



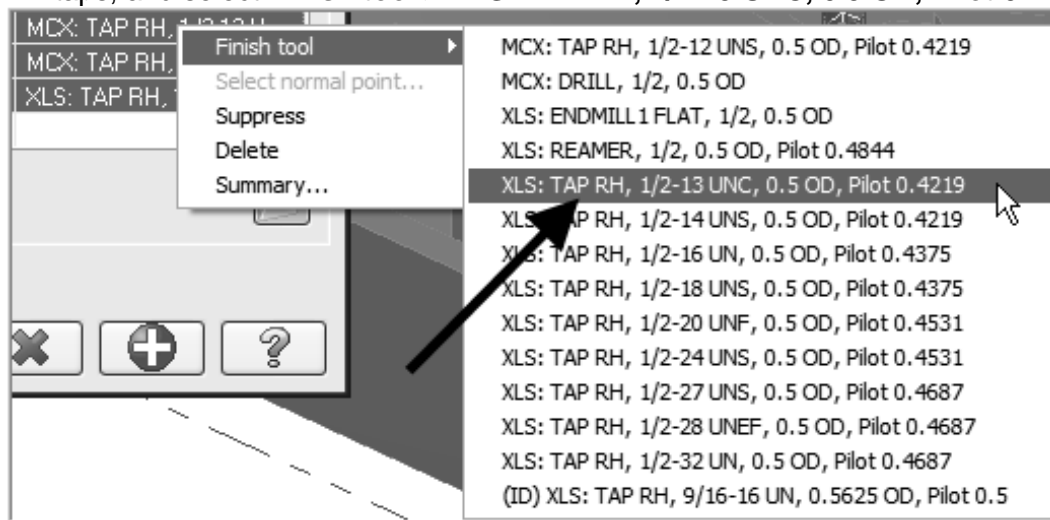
19. Press and hold the **Shift** key on your keyboard. Left click on the last 0.375 Dia. / 1,75 Depth Hole in the list. Release the **Shift** key. This will select all rows from the first mouse selection through the second mouse selection.

## Mastercam Training Guide

20. Right click on any of the selected rows, in the **Hole type** column who's value currently reads **Drill**. Hover the cursor over **Hole type** and select **TAP RH** from the list.



21. Using the same selection method, pick the three .5 Dia. Holes and set to **Tap RH**.  
22. A 1/2"-12 UNS tap is selected. Right click in the **Finish tool** column over one of three 1/2" taps, and select **Finish tool > XLS: TAP RH, 1/2-13 UNC, 0.5 OD, Pilot 0.4218**.



FBM Drill hole features are controlled by the file FbmToolTable.xls in the Mcamx\common\FBM directory. This file can be edited to offer additional selections.

If you have SolidWorks® installed on your system and have imported a .SLDPRT file, you can reselect the file from the FBM Drill Features page and update the features list with imported SolidWorks Hole Wizard® data.


23. Select the OK button  to process all holmaking operations for this part.

**TASK 8:  
VERIFY THE TOOLPATH USING SAVED STOCK**

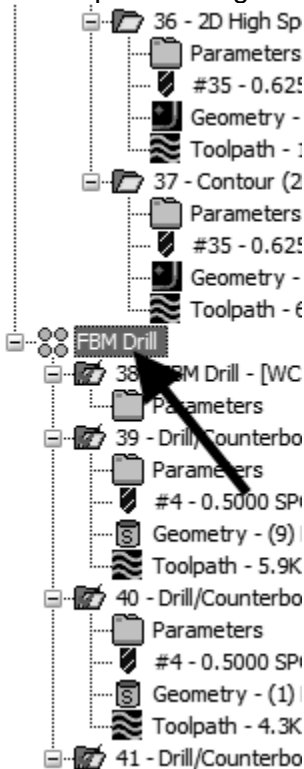
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.

1. For a better view of the part use the toolbar at the top of the screen to change the graphics view to Isometric.

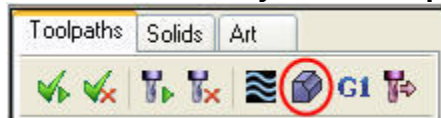


2. Select the **Screen Fit** icon found at the top of the screen to fit the part to the screen 

In the Toolpath Manager left click *only* on the **FBM Drill Toolpath Group**.



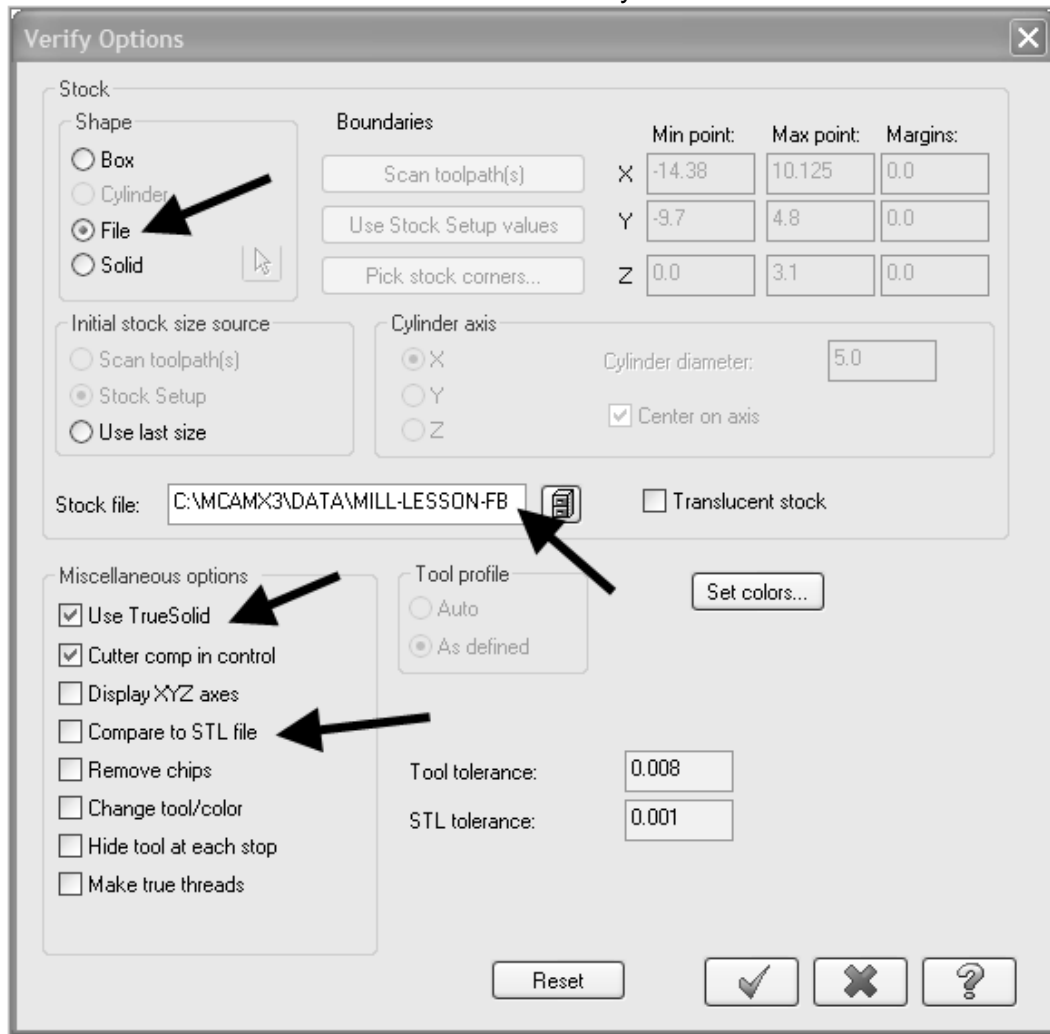
3. Select the **Verify selected operations** button circled below:




4. Now select the **Options** button:



5. In the **Verify Options** change the **Stock** selection to **File**. Also disable **Compare to STL file**. Select **MILL-LESSON-FBM-2.STL** as your **Stock file**.



6. Select the green check mark button  to close this dialog box.
7. Adjust the Verify speed to midway along the speed control bar.

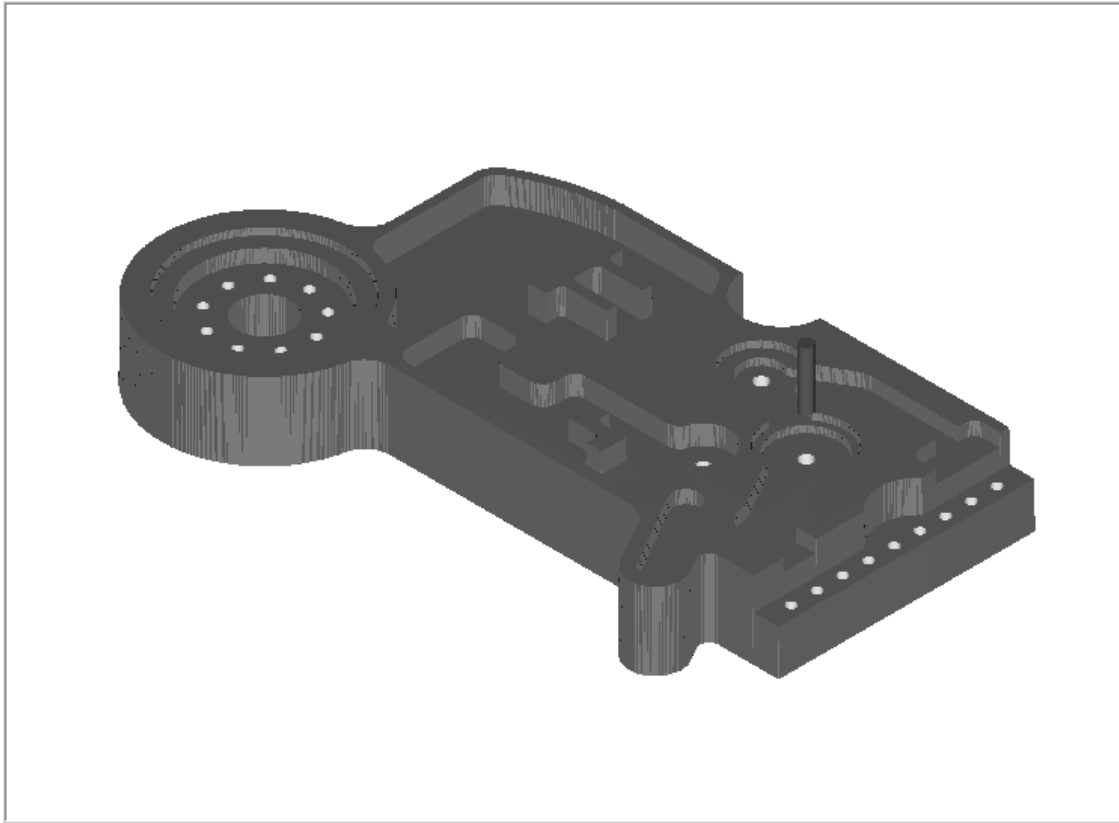


8. Select the play button to verify the toolpaths.





☞ The verified toolpaths are shown below.




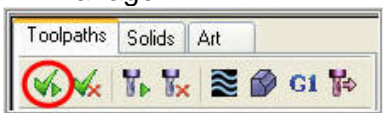
9. Select the OK button  to exit Verify.

### TASK 9: SAVE THE UPDATED MCX FILE

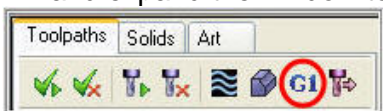
1. Select **File**.
2. Select **Save**.

### TASK 10: POST AND CREATE THE CNC CODE FILE

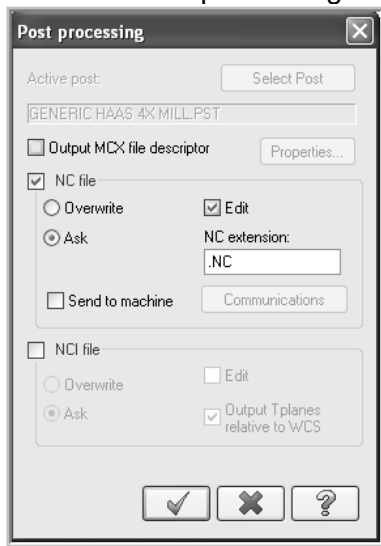
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.




2. Select the **Post selected operations** button from the Toolpath manager.
3. **Please Note:** If you cannot see **G1** click on the right pane of the Toolpath manager window and expand the window to the right.



4. In the Post processing window, make the necessary changes as shown below:



5. Select the OK button  to continue.
6. Ensure the same name as your Mastercam part file name is displayed in the NC File name field as shown below:



7. Select the Save button.

8. The CNC code file opens up in the default editor:

```

%
O0 ( MILL-LESSON-FBM-2 )
( DATE=DD-MM-YY - 01-10-08 TIME=HH:MM - 09:45 )
( MCX FILE - C:\MASTERCAMX3-MILL-LESSONS\MILL-LESSON-FBM-21.MCX )
( NC FILE - C:\MCAMX3\MILL\NC\MILL-LESSON-FBM-2.NC )
( MATERIAL - ALUMINUM INCH - 6061 )
( T1 | | H1 )
( T33 | | H33 | XY STOCK TO LEAVE - .015 | Z STOCK TO LEAVE - 0. )
( T7 | | H7 )
( T6 | | H6 | XY STOCK TO LEAVE - .015 | Z STOCK TO LEAVE - 0. )
( T35 | | H35 | XY STOCK TO LEAVE - .02 | Z STOCK TO LEAVE - 0. )
( T24 | | H24 | XY STOCK TO LEAVE - .015 | Z STOCK TO LEAVE - 0. )
( T31 | | H31 | XY STOCK TO LEAVE - .02 | Z STOCK TO LEAVE - 0. )
( T25 | | H25 | XY STOCK TO LEAVE - .02 | Z STOCK TO LEAVE - 0. )
( T4 | | 1/2 SPOTDRILL | H4 )
( T8 | DRILL | H8 )
( T2 | DRILL | H2 )
( T10 | DRILL | H10 )
( T5 | 3/8-16 UNC | H5 )
( T9 | 1/2-13 UNC | H9 )
N1 G20
N2 G0 G17 G40 G49 G80 G90
N3 T1 M6
N4 G0 G90 G54 X-14.68 Y4.7999 A0. S254 M3
N5 G43 H1 Z3.2
N6 M8
N7 G1 Z3. F2.
N8 X10.125 F4.1
N9 G2 X11.575 Y3.3499 I0. J-1.45
N10 X10.125 Y1.8999 I-1.45 J0.
N11 G1 X-14.38
N12 G3 X-15.8299 Y.45 I0. J-1.4499
N13 X-14.38 Y-1. I1.4499 J0.
N14 G1 X10.125
N15 G2 X11.575 Y-2.45 I0. J-1.45
N16 X10.125 Y-3.9 I-1.45 J0.
N17 G1 X-14.38
N18 G3 X-15.83 Y-5.35 I0. J-1.45
N19 X-14.38 Y-6.7999 I1.45 J0.
N20 G1 X10.125
N21 G2 X11.575 Y-8.2499 I0. J-1.45
    
```

9. Select the  in the top right corner to exit the CNC editor.

This completes Mill-Lesson-FBM-2.