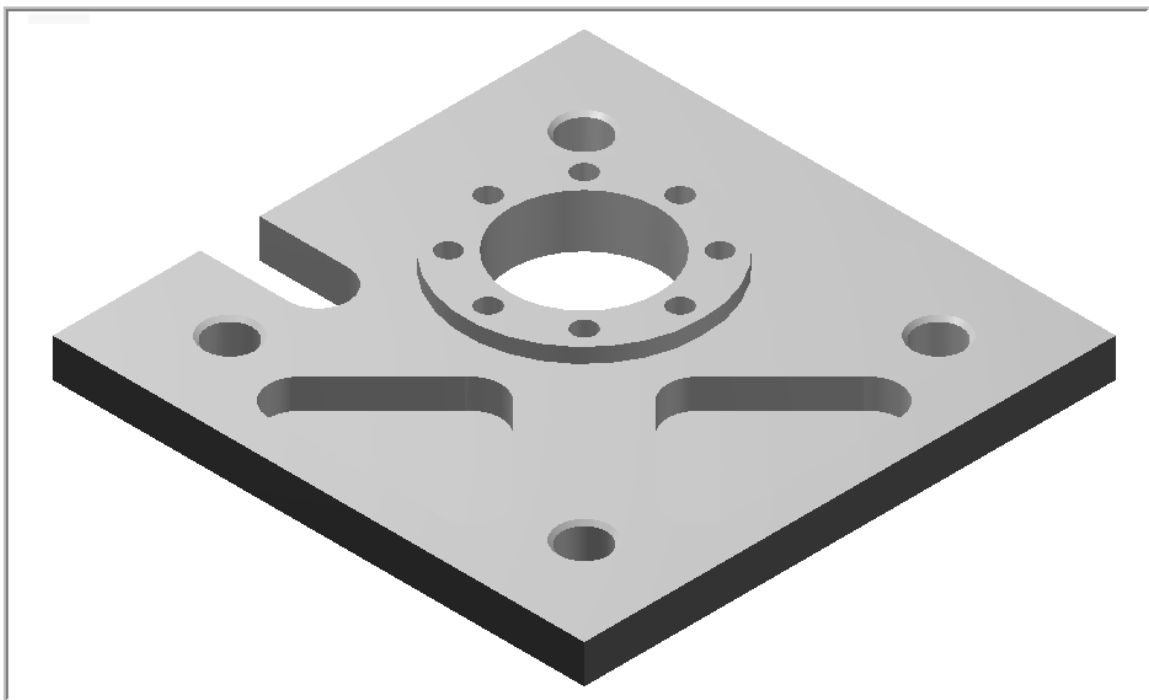


Mastercam X³

TRAINING

GUIDE



MILL-LESSON-FBM-1

FBM MILL AND FBM DRILL

Objectives

Previously in Mill-Lesson-6 and Mill-Lesson-7 geometry was created and machined using standard Mastercam methods. This lesson will apply Feature Based Machining (FBM) methods including FBM Mill and FBM Drill to a solid model previously created in SolidWorks.

➤ 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-1 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:

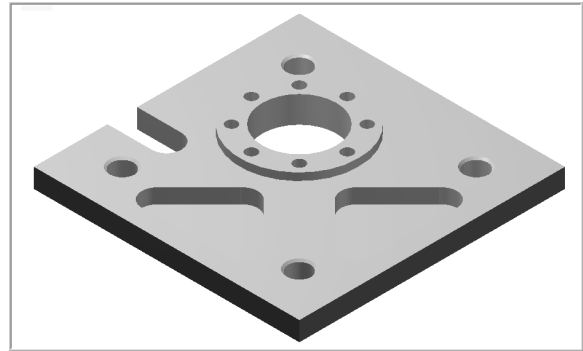
FBM Mill operation and child toolpath operations.

FBM Drill operation and child toolpath operations.

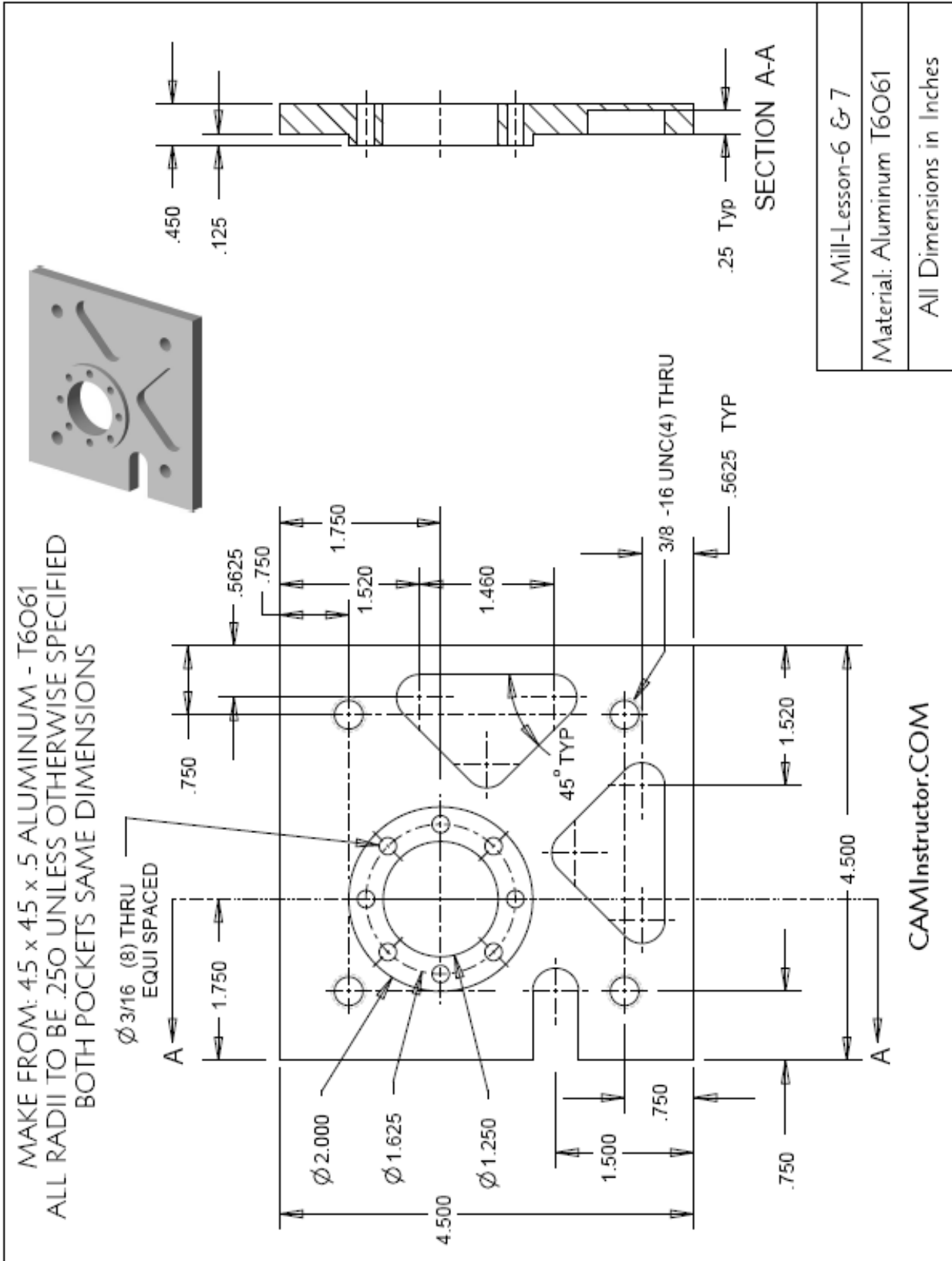
➤ Inspect the toolpath using Mastercam's Verify by:

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

Generating the NC- code.

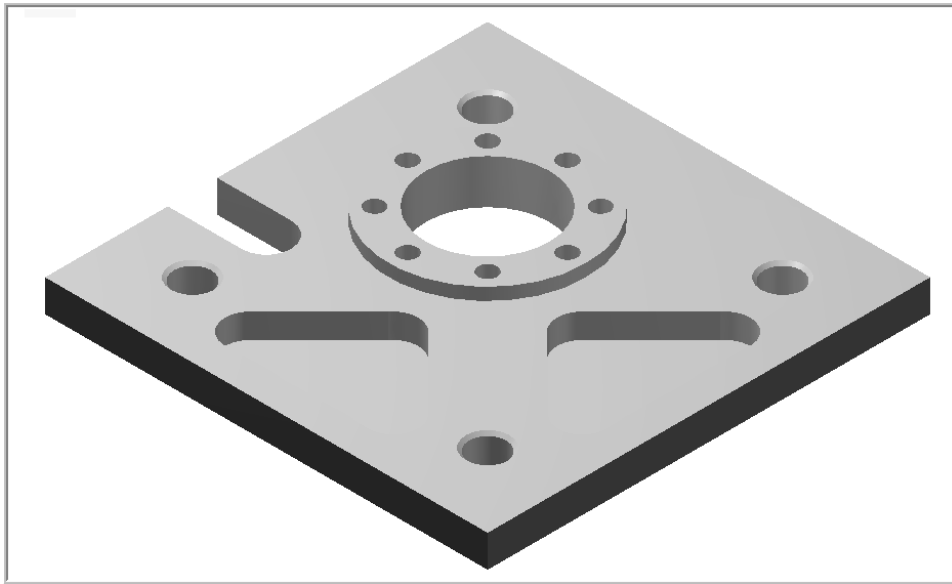


MILL-LESSON-FBM-1 DRAWING



TOOL LIST

- 3.0 face mill to face the part to thickness.
- 1.0 flat end mill to rough machine around 2.0" diameter boss.
- 0.5 diameter flat end mill to rough machine the pockets and 1.25 bore.
- 0.375 diameter flat end mill to rough machine the slot and finish all areas.
- 0.375 diameter spot drill all holes.
- 0.1875 diameter drill to drill the bolt hole circle.
- 0.3125 diameter drill to tap drill the 3/8-16 UNC tapped holes.
- 3/8-16 UNC tap to tap holes.
- 0.5 countersink to countersink the 3/8-16 UNC tapped holes.



Proj./Part No. : 0
 Drawing No. : 1
 Prog. No. : 9

Date : 09/24/08
 Customer : -
 Programmer : 1



Tool type : 3 Face mill 3" Face Mill
 Manufact.code :
 Chuck :
 Tool Number : 1 Feedrate : 20
 Diameter : 3 RPM : 1500 Plunge feed r.: 20
 Corner radius : 0 Tip angle : 45 Diam. offset : 1
 Flute length : 1.969 Material : ALUMINUM ... Length offset : 1
 Overall length: 2.1 No flutes : 4



Tool type : 1 Endmill1 Flat 1 INCH FLAT ENDMILL
 Manufact.code :
 Chuck :
 Tool Number : 2 Feedrate : 10
 Diameter : 1 RPM : 1300 Plunge feed r.: 10
 Corner radius : 0 Tip angle : 0 Diam. offset : 2
 Flute length : 2 Material : ALUMINUM ... Length offset : 2
 Overall length: 4 No flutes : 4



Tool type : 0.5 Endmill1 Flat 1/2 FLAT ENDMILL
 Manufact.code :
 Chuck :
 Tool Number : 3 Feedrate : 10
 Diameter : 0.5 RPM : 3000 Plunge feed r.: 10
 Corner radius : 0 Tip angle : 0 Diam. offset : 3
 Flute length : 1 Material : ALUMINUM ... Length offset : 3
 Overall length: 3 No flutes : 4



Tool type : 0.375 Endmill1 Flat 3/8 FLAT ENDMILL
 Manufact.code :
 Chuck :
 Tool Number : 4 Feedrate : 8
 Diameter : 0.375 RPM : 4750 Plunge feed r.: 8
 Corner radius : 0 Tip angle : 0 Diam. offset : 4
 Flute length : 0.75 Material : ALUMINUM ... Length offset : 4
 Overall length: 2.5 No flutes : 4



Tool type : 0.1875 Drill 3/16 DRILL
 Manufact.code :
 Chuck :
 Tool Number : 5 Feedrate : 4.1073
 Diameter : 0.1875 RPM : 1426 Plunge feed r.: 4.1073
 Corner radius : 0 Tip angle : 118 Diam. offset : 5
 Flute length : 2 Material : ALUMINUM ... Length offset : 5
 Overall length: 3 No flutes : 2



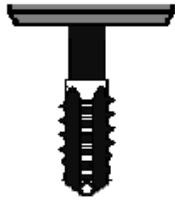
Tool type : 0.5 Csink 1/2 COUNTERSINK 90 Deg
 Manufact.code :
 Chuck :
 Tool Number : 7 Feedrate : 7.332
 Diameter : 0.5 RPM : 1833 Plunge feed r.: 7.332
 Corner radius : 0 Tip angle : 90 Diam. offset : 7
 Flute length : 2 Material : ALUMINUM ... Length offset : 7
 Overall length: 3 No flutes : 2



Tool type : 0.3125 Drill 5/16 DRILL
 Manufact.code :
 Chuck :
 Tool Number : 8 Feedrate : 4.2442
 Diameter : 0.3125 RPM : 855 Plunge feed r.: 4.2442
 Corner radius : 0 Tip angle : 118 Diam. offset : 8
 Flute length : 2 Material : ALUMINUM ... Length offset : 8
 Overall length: 3 No flutes : 2

Proj./Part No.: 0
Drawing No. : 1
Prog. No. : 9

Date : 09/24/08
Customer : -
Programmer : 1



Tool type : 0.375 Tap RH 3/8-16 TAPRH
Manufact.code :
Chuck :
Tool Number : 9 Feedrate : 44.5667
Diameter : 0.375 RPM : 713 Plunge feed r.: 44.5667
Corner radius : 0 Tip angle : 180 Diam. offset : 9
Flute length : 2 Material : ALUMINUM ... Length offset : 9
Overall length: 3 No flutes : 1



Tool type : 0.375 Spot Drill 3/8 SPOTDRILL
Manufact.code :
Chuck :
Tool Number : 10 Feedrate : 65.1947
Diameter : 0.375 RPM : 4074 Plunge feed r.: 65.1947
Corner radius : 0 Tip angle : 90 Diam. offset : 10
Flute length : 2 Material : ALUMINUM ... Length offset : 10
Overall length: 3 No flutes : 4

MILL-LESSON-FBM-1 - THE PROCESS

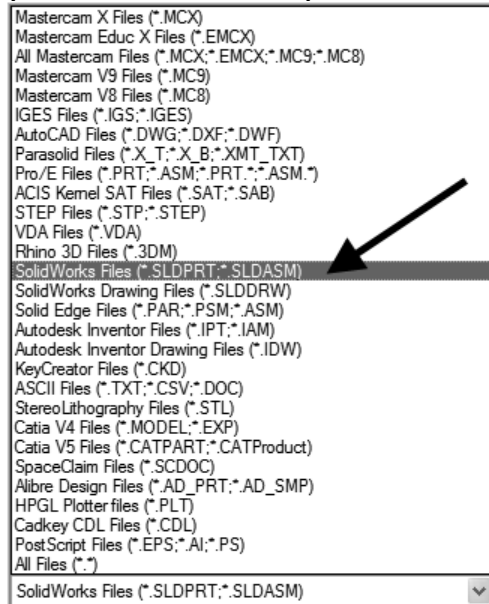
Toolpath Creation

- TASK 1:** Import part geometry from SolidWorks
- TASK 2:** Setting the environment
- TASK 3:** Translate part geometry to machining origin
- TASK 4:** Define the rough stock using stock setup
- TASK 5:** Run FBM Mill for all milling operations
- TASK 6:** Verify the toolpath
- TASK 7:** Run FBM Drill for all hole making operations
- TASK 8:** Verify the toolpath
- TASK 9:** Remove the Outside Finish Pass
- TASK 10:** Save the updated MCX file
- TASK 11:** Post and create the CNC code file

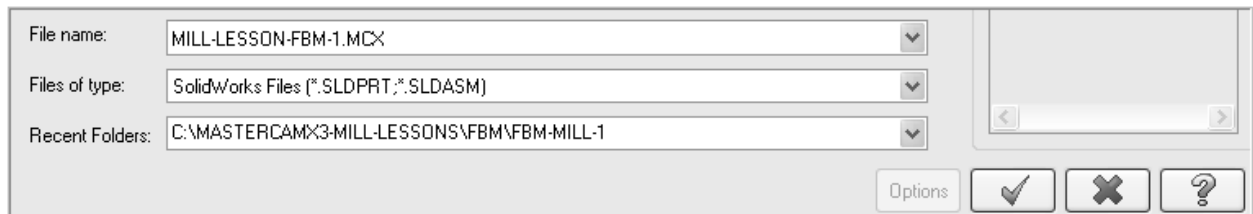
TASK 1: IMPORT PART GEOMETRY FROM SOLIDWORKS

☞ 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 **SolidWorks Files (*.SLDPRT;*.SLDASM)**

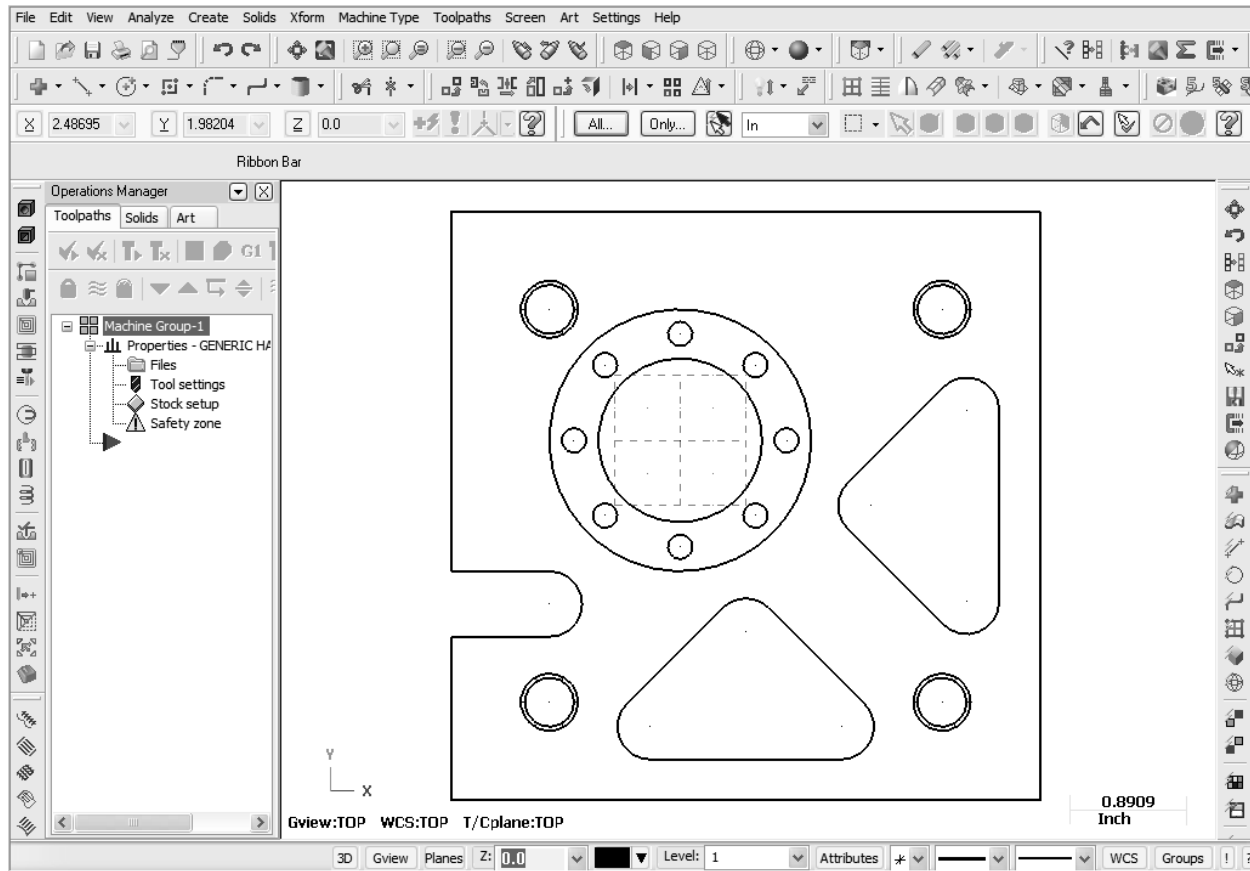


3. On the multimedia CD that came with this text is a **folder called Mastercam-Files** browse to this location and select Mill-Lesson-FBM-1.SLDPRT



4. Select the green check mark button to load the file and complete this function.

5. The screen should look like the one below:



6. Select **File>Save as...**

7. In the File name box, type **Mill-Lesson-FBM-1**.

8. Save to an appropriate location.

9. 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.

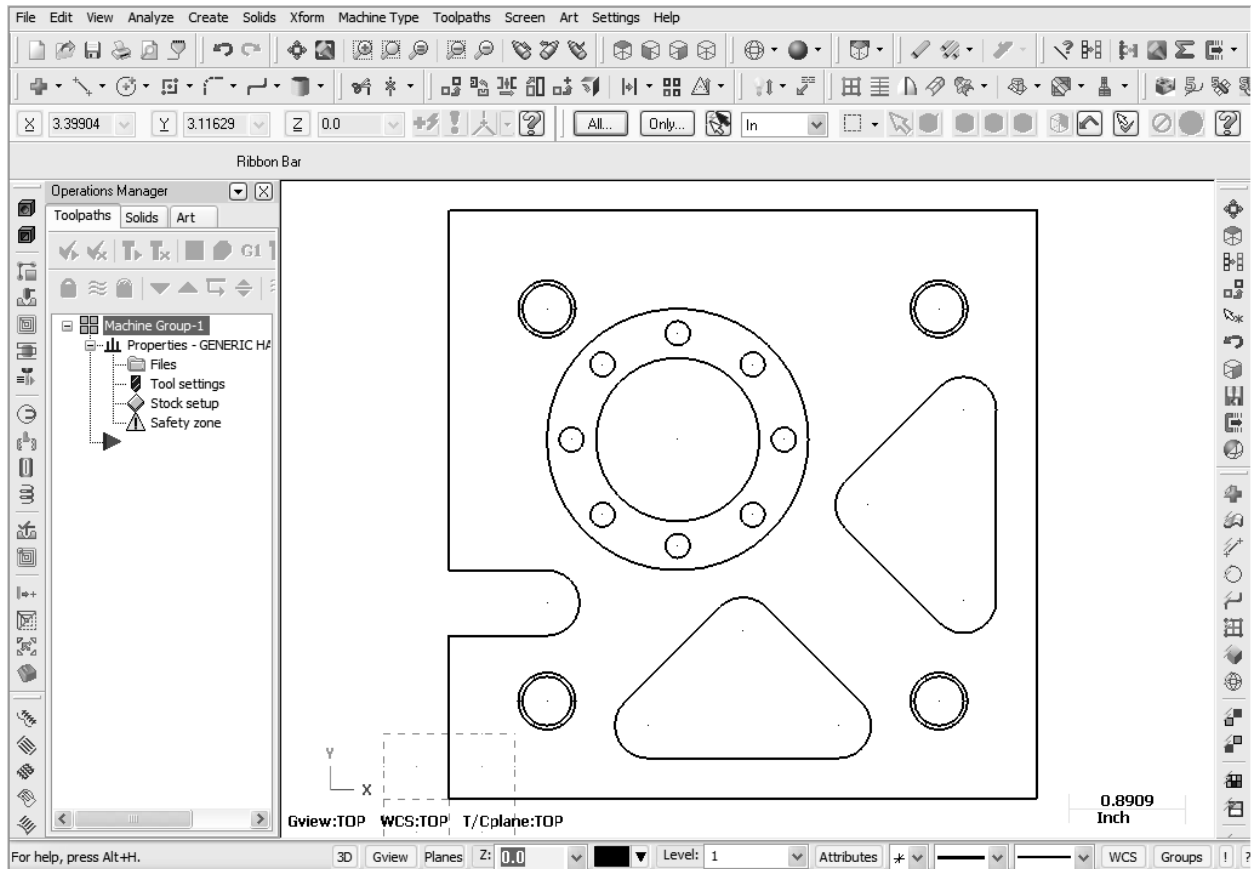
TASK 3: TRANSLATE PART GEOMETRY TO MACHINING ORIGIN

➔ Translating the geometry between 2 points.

1. Select the **Xform Translate...** icon .
2. Select the **All...** button  from the **General Selection Ribbon Bar**.
3. Select the **All Entities** button  from the **Select All** dialog Box.
4. Pick the **End Selection** icon .
5. Select the **Move** radio button and enter Delta vales of ΔX 1.75, ΔY 2.75, and ΔZ 0.
6. Select the OK button  to complete the geometry translation.



7. Click on the **Clear Colors**.
8. Select the **Fit to screen** icon .
9. The screen should look like the one below:




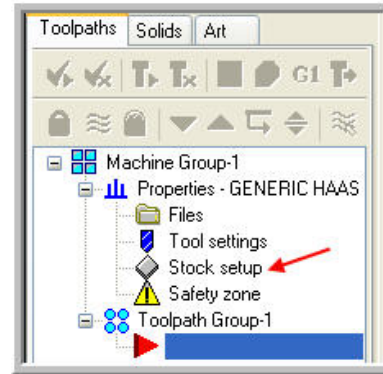
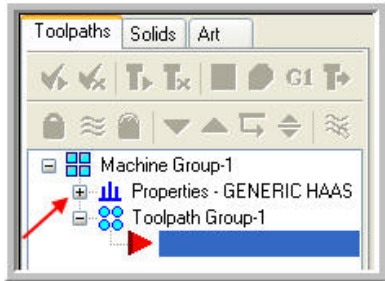
Toolpath Creation

TASK 4:

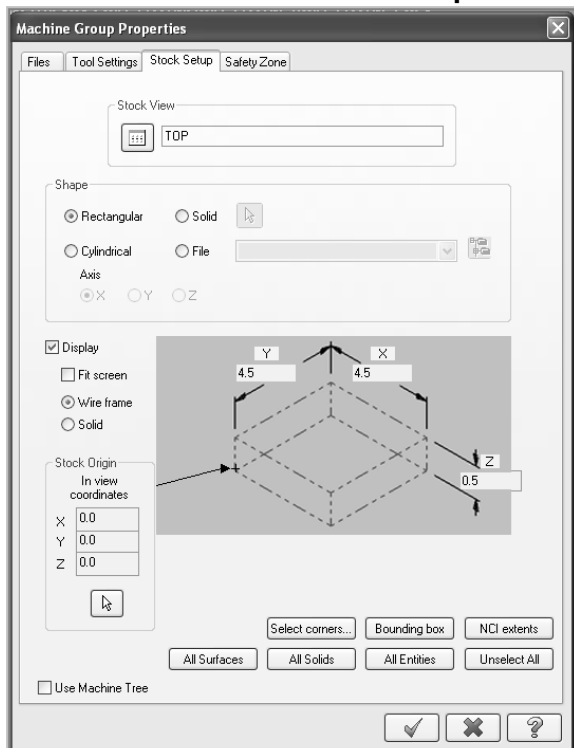
DEFINE THE ROUGH STOCK USING STOCK SETUP

➤ Defining the Rough Stock using stock setup.

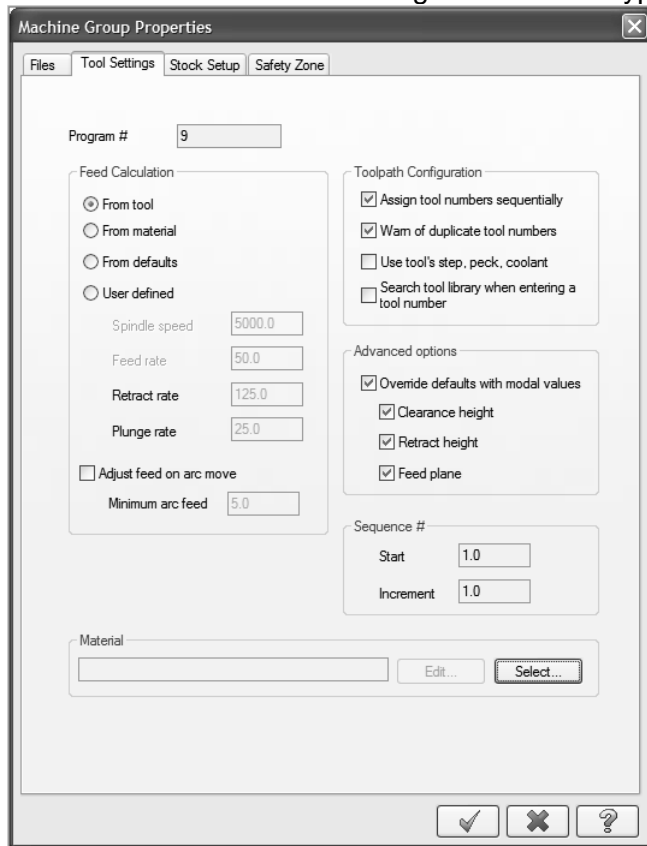
1. Select the **Fit to screen** icon .
2. Select the **plus** in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will **Show/hide** Operations Manager pane.



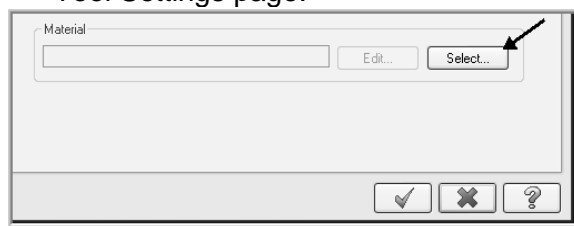
3. Select **Stock setup** in the toolpath manager window as shown above on the right:
4. Change the parameters to match the **Stock Setup** screenshot below:
 - The bottom face of the material has been pre-machined and is not part of the toolpath operations for this lesson.
 - The material has also been pre-machined to 4.5 x 4.5 and is not part of the toolpath operations for this lesson.
 - **Z zero** is at the **bottom of the part**.



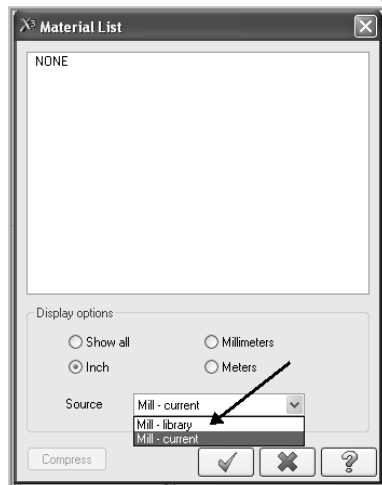
5. Select the **Tool Settings** tab and change the parameters to match the Tool Settings screenshot below. To change the Material type follow the next set of instructions.



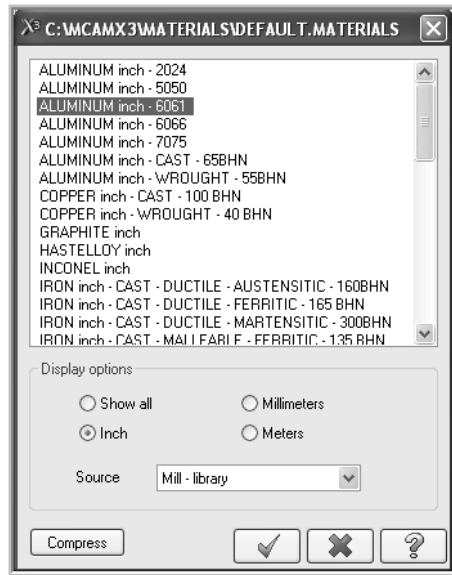
6. To change the **Material type to Aluminium 6061** pick the **Select** button at the bottom of the Tool Settings page.



7. At the **Material List** dialog box open the **Source** drop down list and select **Mill - library**.



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

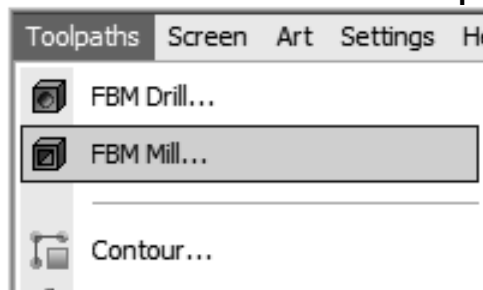


9. Select the OK button again to complete this Stock Setup function.

TASK 5: RUN FBM MILL FOR ALL MILLING OPERATIONS

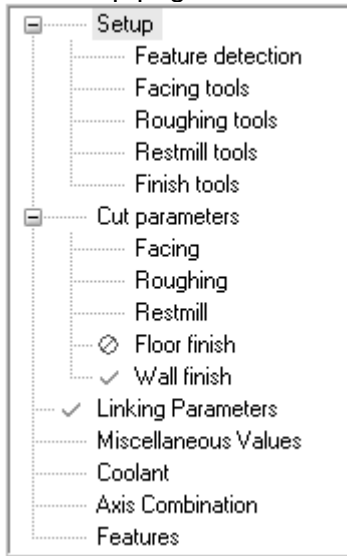
➤ In this task you configure settings in FBM Mill to set up tools and interrogate the solid to create machining regions and apply toolpath strategies.

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

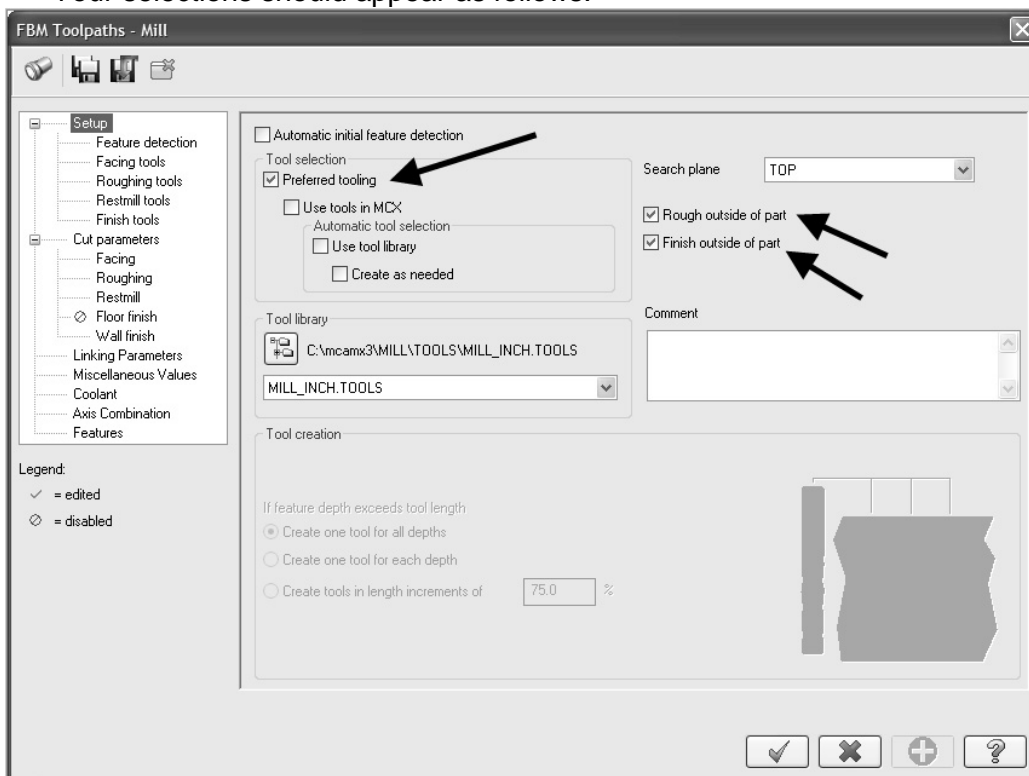


FBM Mill is a new method for quickly producing the equivalent of standard Face, Pocket, and Contour toolpaths. A single Parent operation is used to interrogate the solid model, detecting the part's features and determining machining zones. A single set of configuration settings will address facing, roughing, rest milling, floor and wall finishing, and outside contouring. Child operations are created using 2D High Speed, Face, Pocket, and Contour operations. The parameters of Child operations can be edited for further customization.

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



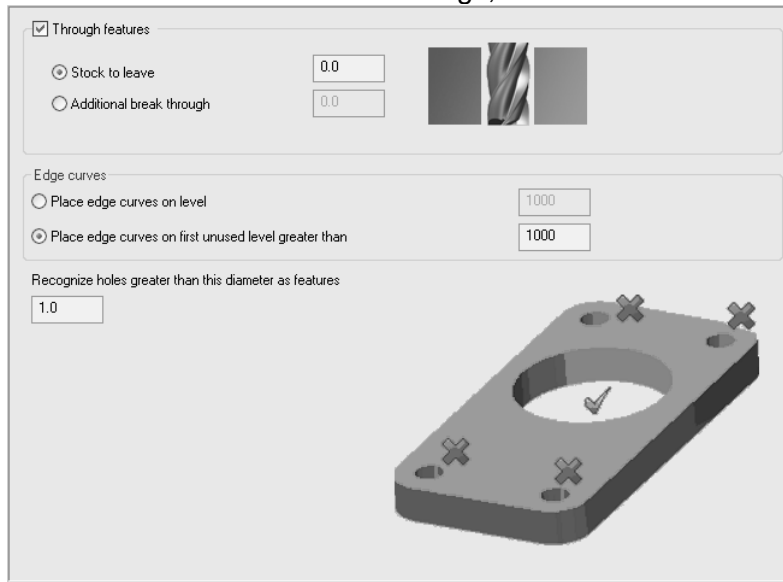
- From this page you will select **Preferred tooling** which will be set up in a future step. You will also elect to rough and finish the outside of this part, to ensure the 1/2" slot is address. Your selections should appear as follows:



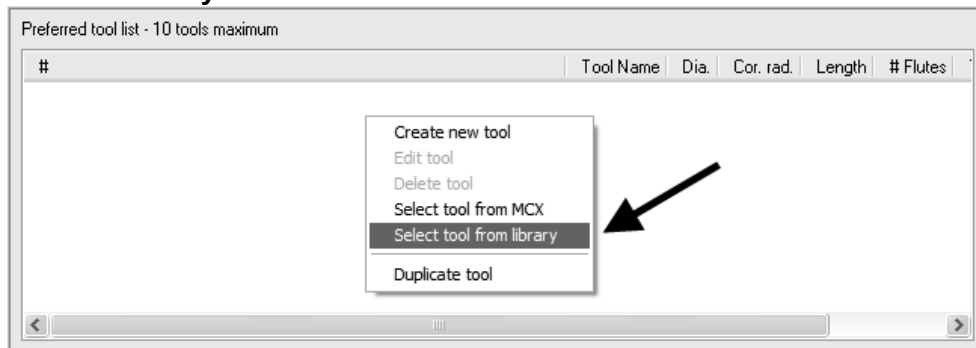
- Click on **Feature detection** in the Tree Control menu on the left of the dialog box..

Mastercam Training Guide

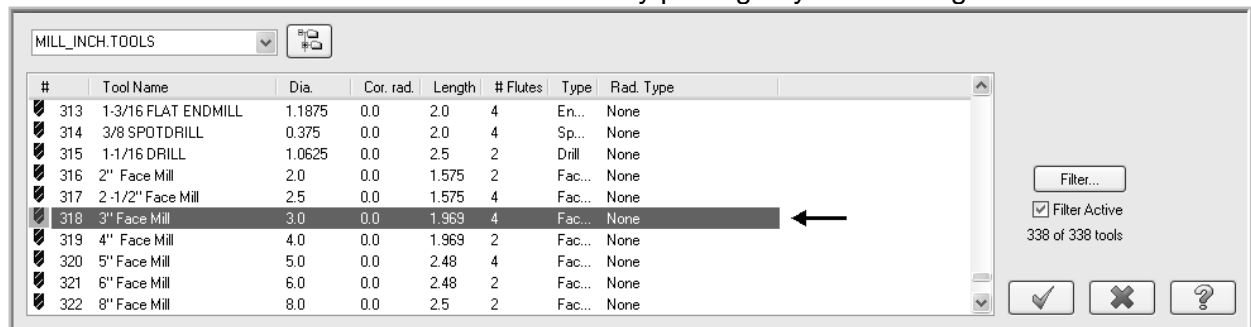
5. You will leave the default settings, which should be as follows:




6. Click on **Facing tools** in the Tree Control menu. You will select a Facing tool from the Tool Library.
7. Right click in the empty white space in the Preferred tools list and left click on **Select tool from library**.

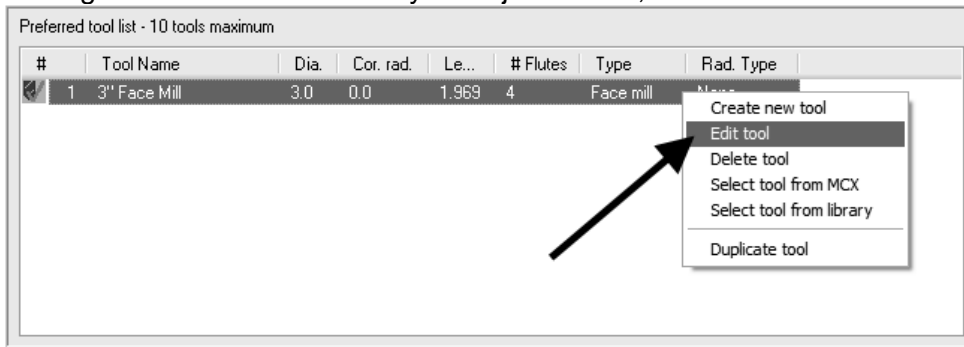


8. Use the slider bar on the right of this dialog box to scroll down and locate a **3.0 diameter face mill**. Select the 3.0 diameter face mill by picking anywhere along the row.

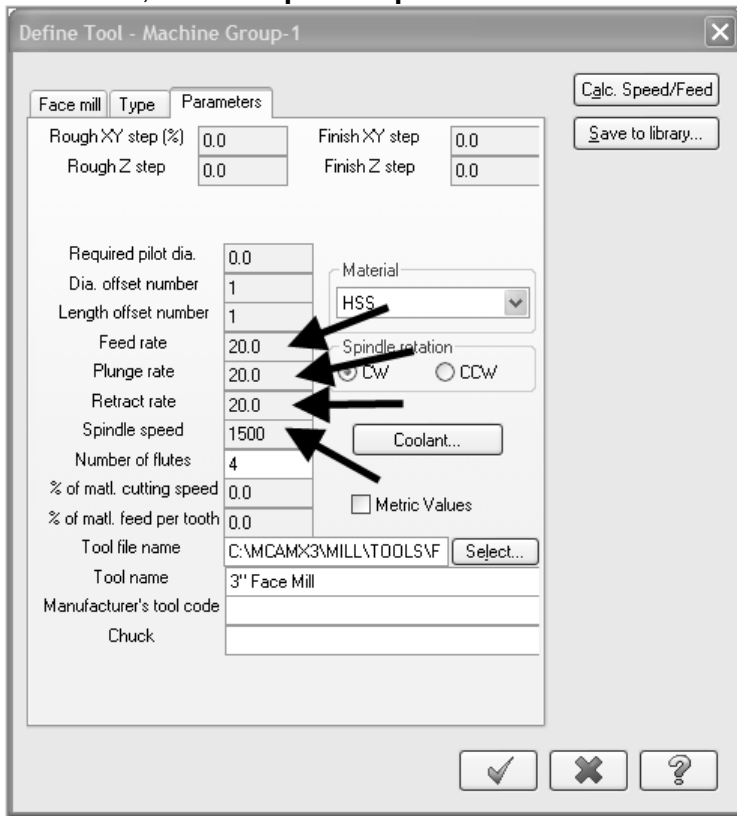


9. Select the OK button  to complete the selection of this tool.

10. Right click on the Face mill you've just added, and select **Edit tool**.



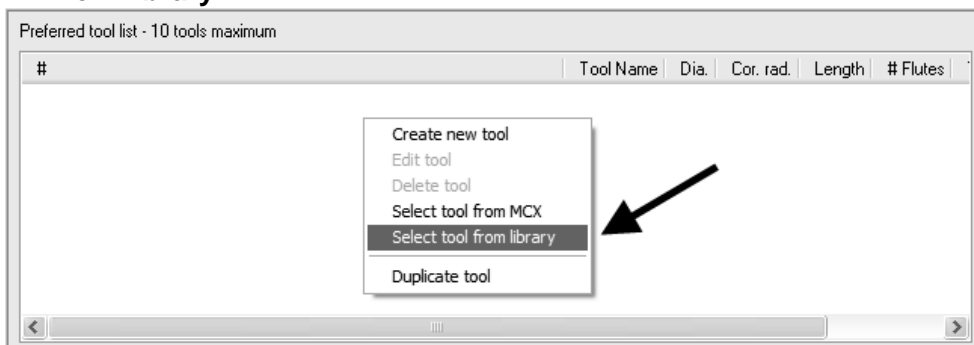
11. Left click on the **Parameters** tab and change the **Feed rate**, **Plunge rate**, and **Retract rate** to 20.0, and the **Spindle speed** to 1500.



12. Select the OK button to complete these changes in Parameters.

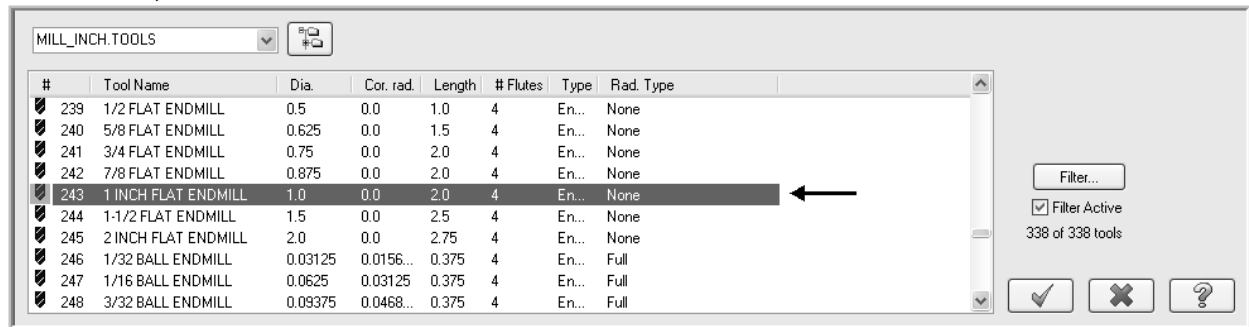
13. Click on **Roughing tools** in the Tree Control menu. You will select Roughing tools from the Tool Library.

14. Right click in the empty white space in the Preferred tools list and left click on **Select tool from library**.

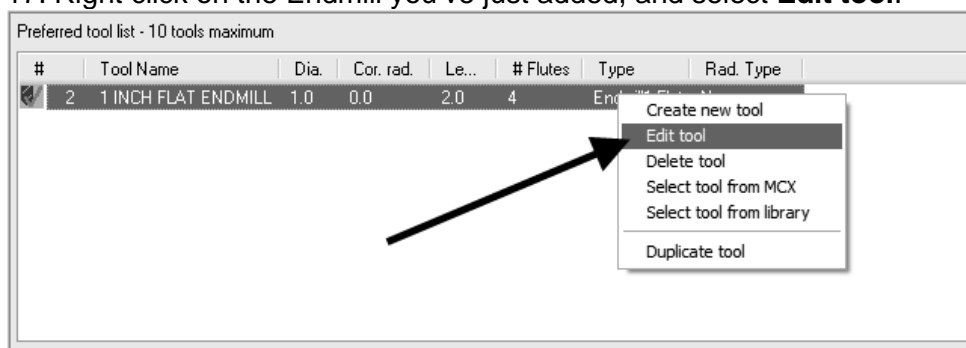


Mastercam Training Guide

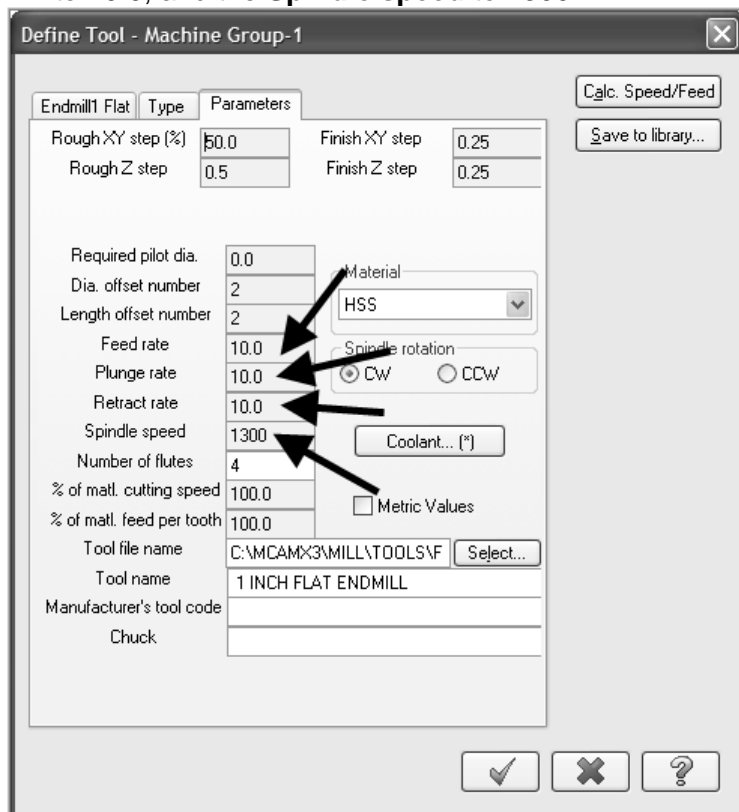
15. Use the slider bar on the right of this dialog box to scroll down and locate a **1.0 diameter flat end mill**. Select the 1.0 diameter flat end mill by picking anywhere along the 1.0 end mill row, as shown below:



16. Select the OK button to complete the selection of this tool.
17. Right click on the Endmill you've just added, and select **Edit tool**.

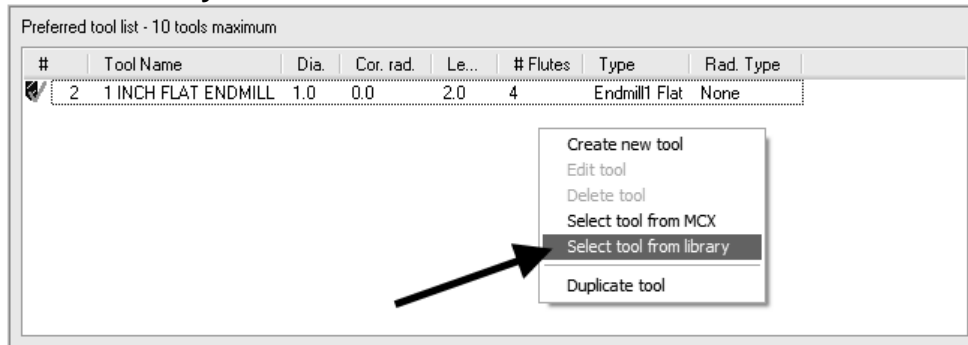


18. Left click on the **Parameters** tab and change the **Feed rate**, **Plunge rate**, and **Retract rate** to 10.0, and the **Spindle speed** to 1300.

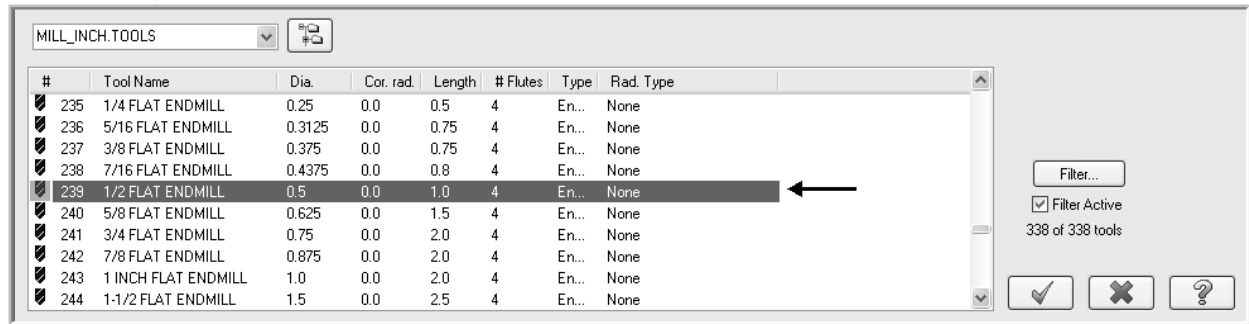


19. Select the OK button to complete these changes in Parameters.

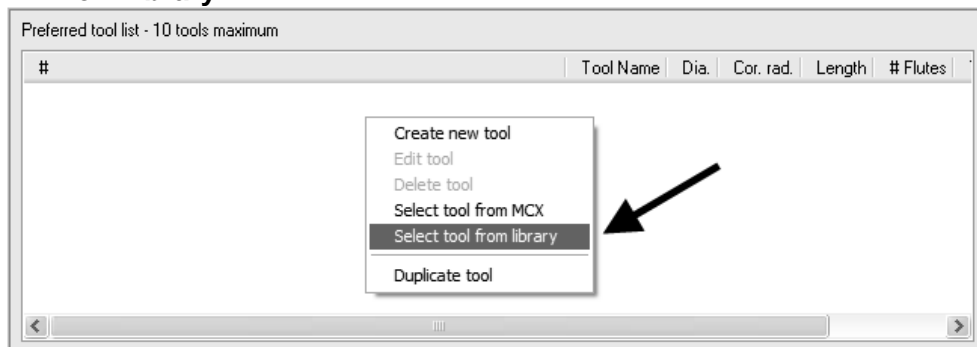
20. Right click in the empty white space in the Preferred tools list and left click on **Select tool from library**.



21. Use the slider bar on the right of this dialog box to scroll down and locate a **0.5 diameter flat end mill**. Select the 0.5 diameter flat end mill by picking anywhere along the 0.5 end mill row, as shown below:

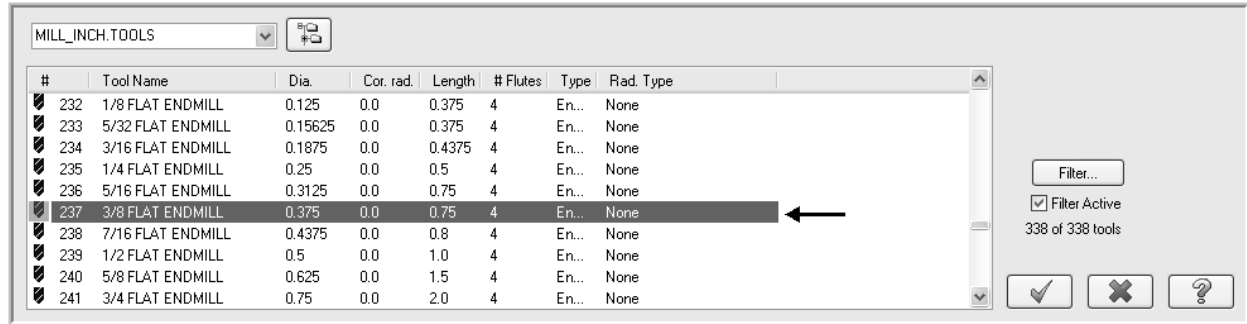


- 22. Select the OK button to complete the selection of this tool.
- 23. Set the feeds and speeds for this tool to 10"/min and 3000 RPM by repeating the steps used on the previous tool.
- 24. Click on **Restmill tools** in the Tree Control menu. You will select a Restmill tool from the Tool Library.
- 25. Right click in the empty white space in the Preferred tools list and left click on **Select tool from library**.

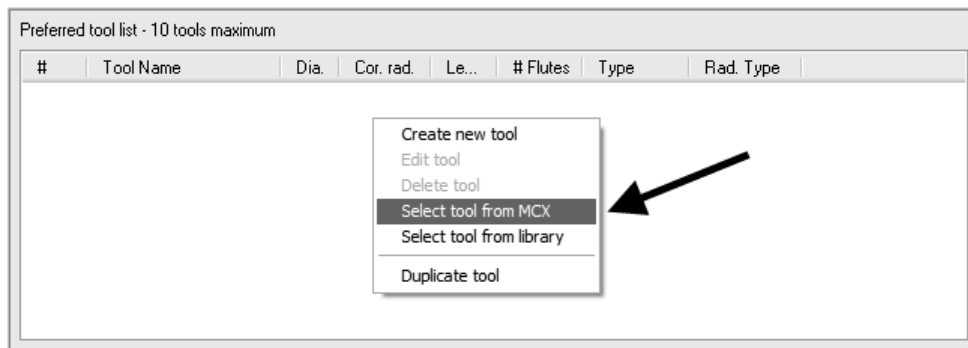


Mastercam Training Guide

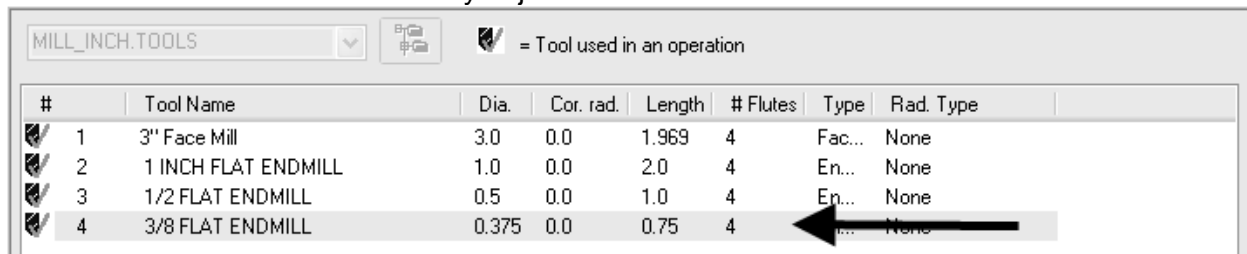
26. Use the slider bar on the right of this dialog box to scroll down and locate a **0.375 diameter flat end mill**. Select the 0.375 diameter flat end mill by picking anywhere along the row.



27. Select the OK button to complete the selection of this tool.
28. Set the feeds and speeds for this tool to 8⁷/min and 4750 RPM by repeating the steps used on the previous tool.
29. Click on **Finish tools** in the Tree Control menu. You will select a Finish tool from the list of tools you have just added.
30. Right click in the empty white space in the Preferred tools list and left click on **Select tool from MCX**.

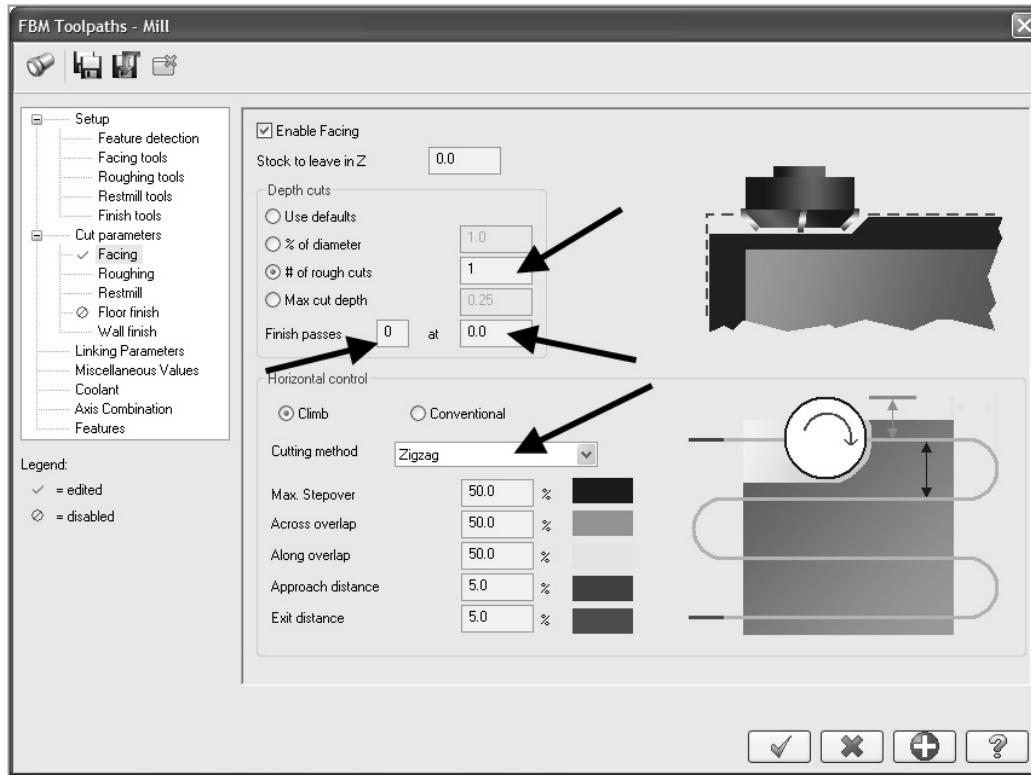


31. Select the **3/8 FLAT ENDMILL** you just added as a Restmill tool.

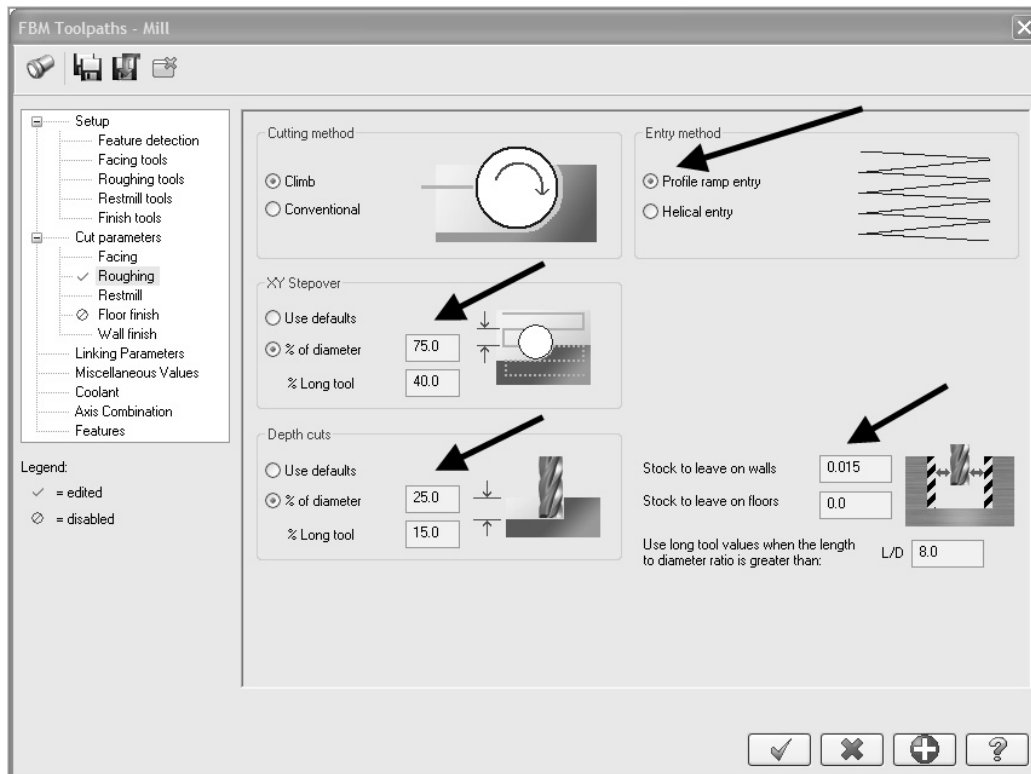


32. Select the OK button to complete the selection of this tool.

33. 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**. Your selections should be set as follows:

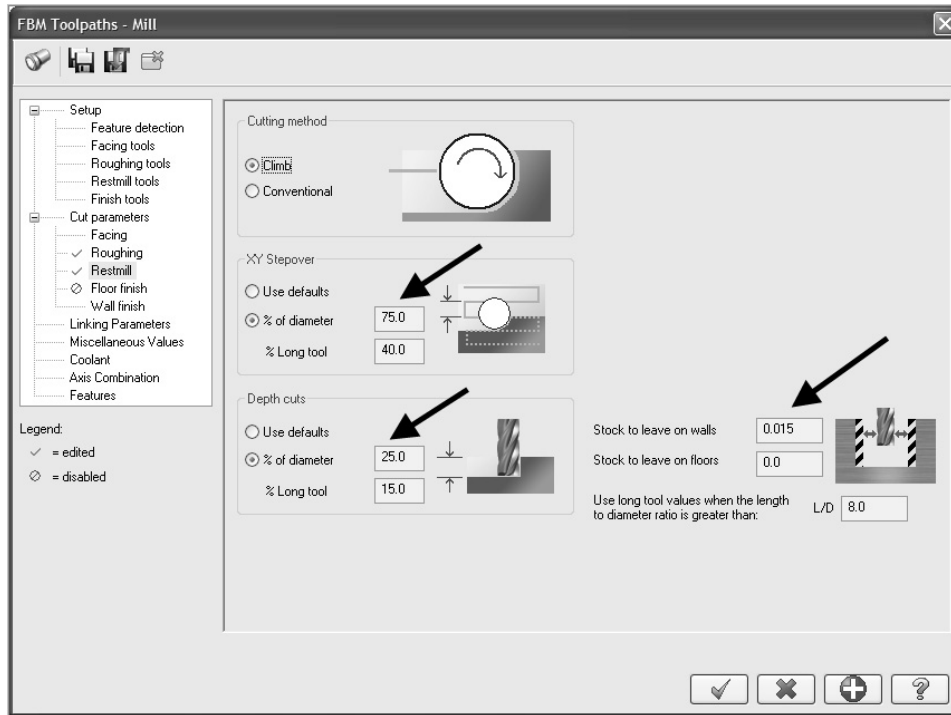


34. Next you will set Roughing cut parameters, selecting **Roughing** from the Tree Control dialog box, and adjusting settings for **Profile ramp entry**, 75% of diameter **XY Stepover**, 25% of diameter **Depth Cuts**, and 0.015" **Stock to leave on walls**. Your selections should be set as follows:



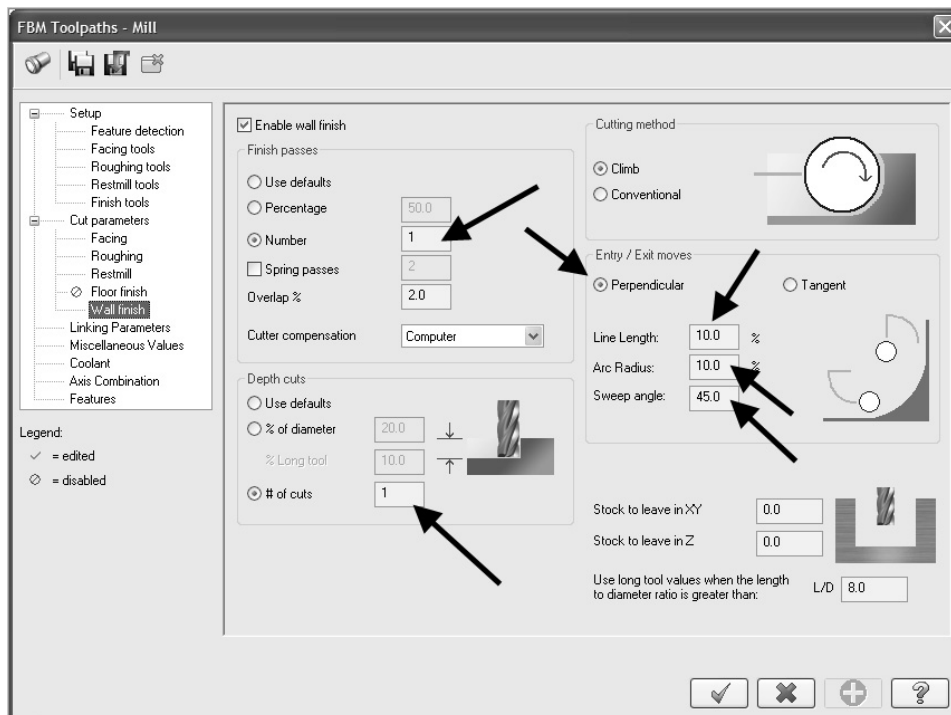
Mastercam Training Guide

35. 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:

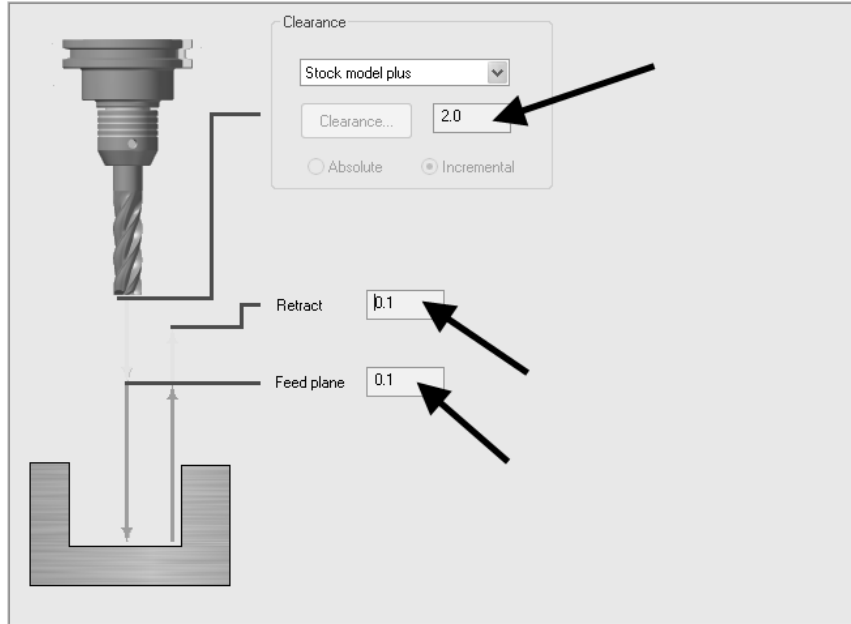


36. Select **Floor Finish** from the Tree Control dialog box and uncheck **Enable floor finish** **Enable floor finish** in order to disable floor finishing.

37. 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 **Perpendicular Entry / Exit moves** with lines and arcs that are 10% of the tool's diameter.. Your selections should be set as follows:



38. 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:



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



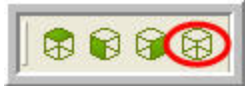
40. Select the OK button to process all milling operations for this part.

41. When prompted to **Enter new NC name** ensure **MILL-LESSON-FBM-1** is displayed and then select the OK button .

TASK 6: VERIFY THE TOOLPATH

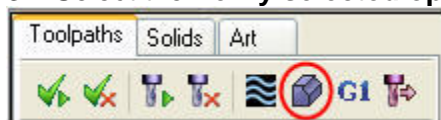
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.

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

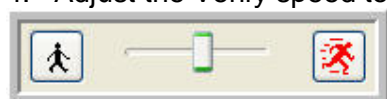


2. In the Toolpath Manager pick all the operations to verify by picking the **Select All** icon .

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



4. Adjust the Verify speed to midway along the speed control bar.

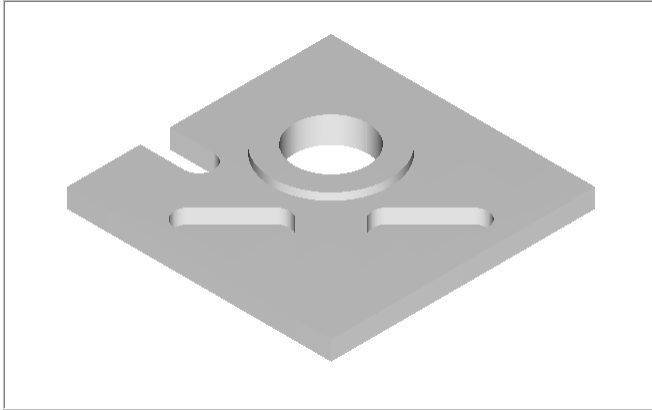


Mastercam Training Guide

5. Select the play button to verify the toolpaths.



➤ The verified toolpaths are shown below.

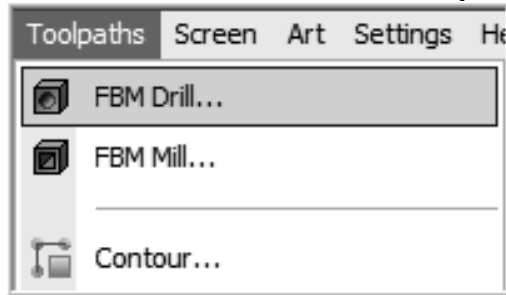


6. Select the OK button to exit Verify.

TASK 7: RUN FBM DRILL FOR ALL HOLE MAKING OPERATIONS

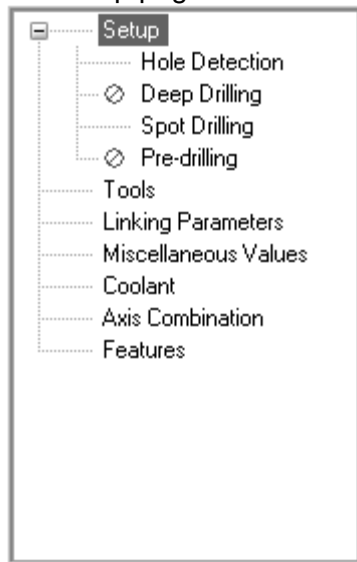
➤ In this task you configure settings in FBM Drill to interrogate the solid, identify hole features, and apply holemaking strategies.

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

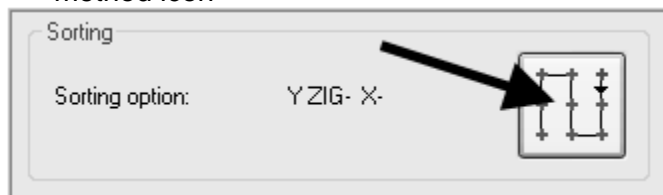


FBM Drill is a new method for quickly producing the equivalent of standard Drill toolpaths. A single Parent operation is used to interrogate the solid model, detecting the part's hole features and apply the necessarily holemaking operations. A single set of configuration settings will address spot drilling, drilling, tapping, and boring. Child operations are created using standard Drill operations. The parameters of Child operations can be edited for further customization.

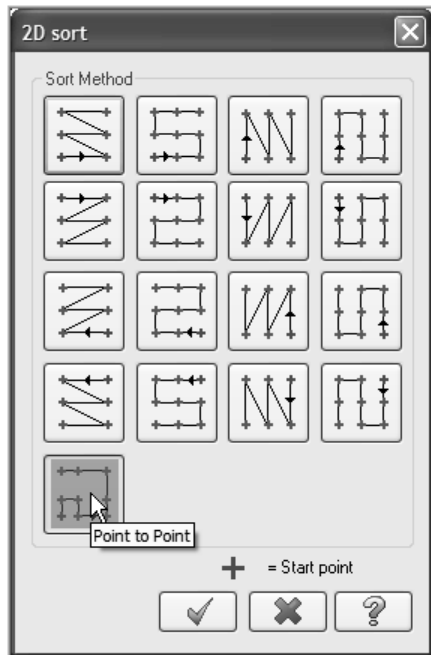
2. On the screen you will now see the **FBM Toolpaths – Drill** dialog box, starting with the Setup page selected in the Tree Control menu down the left hand side:



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




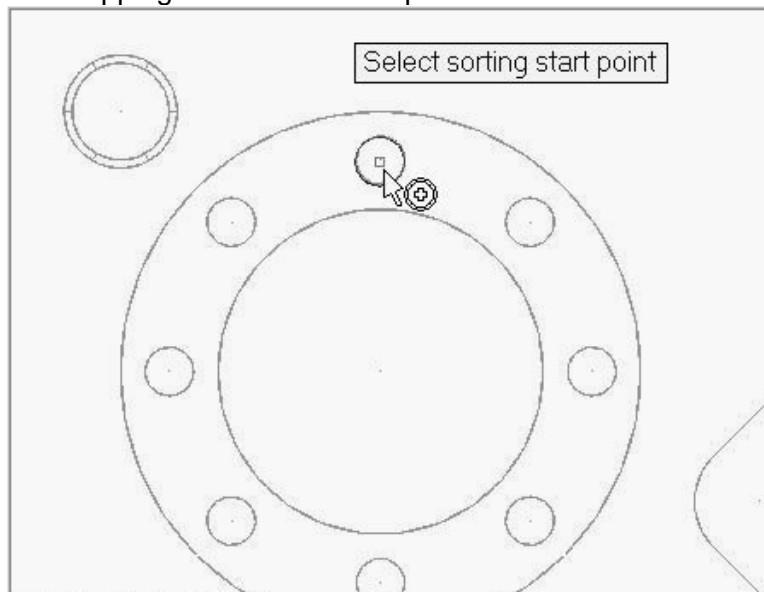
4. Select the **Point to Point** 2D Sort Method



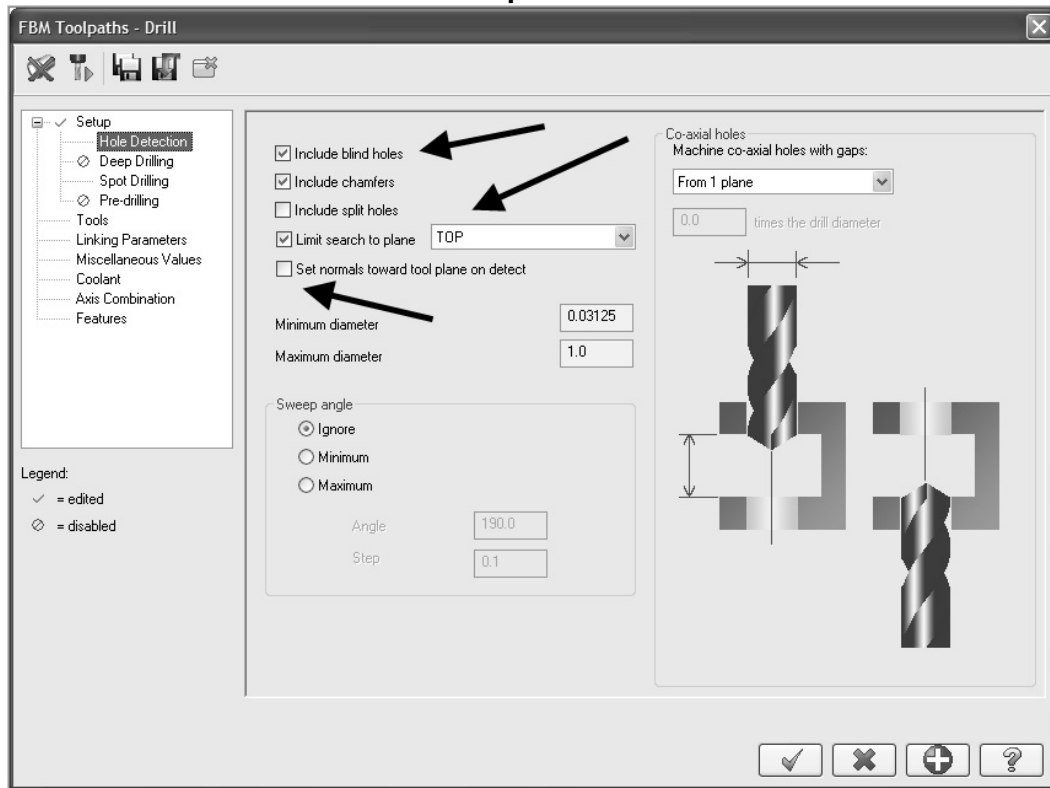
5. Select the OK button in the **2D sort** page.

6. The prompt now changes to **Select sorting start point**. Move the mouse and position the

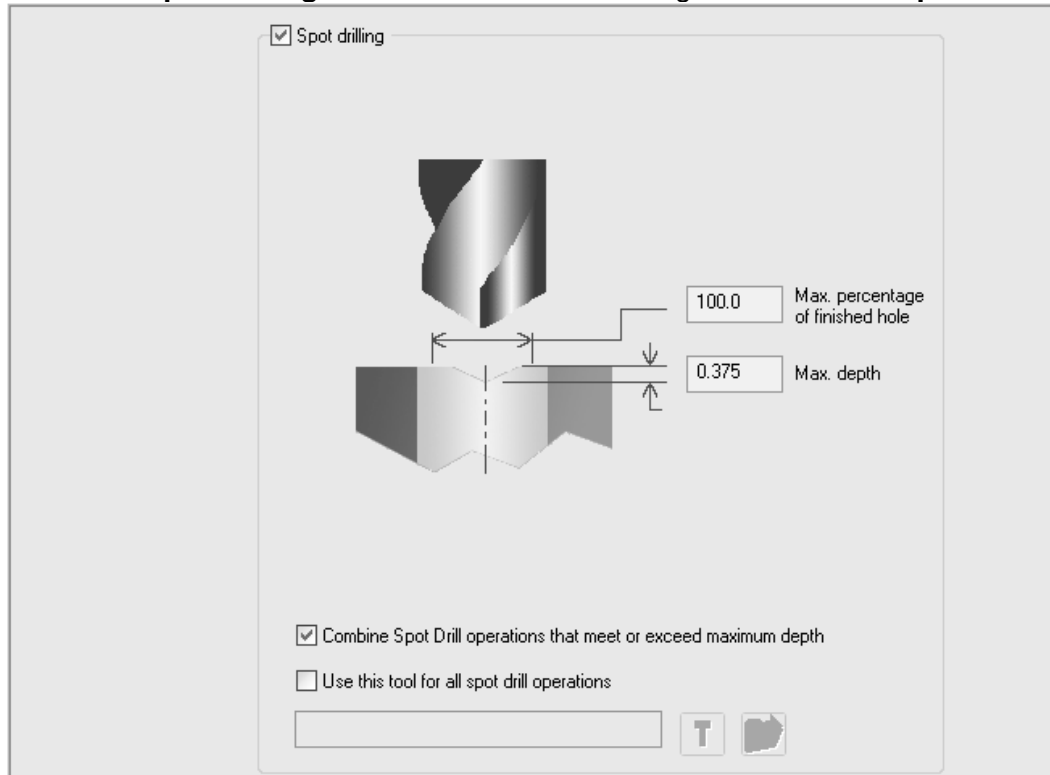
cursor over the point at the center of circle shown below, you will notice this icon  appears as you move over the point at the center of this circle. This cross denotes you are snapping to a circle center point. Use the left button of your mouse to pick this point.



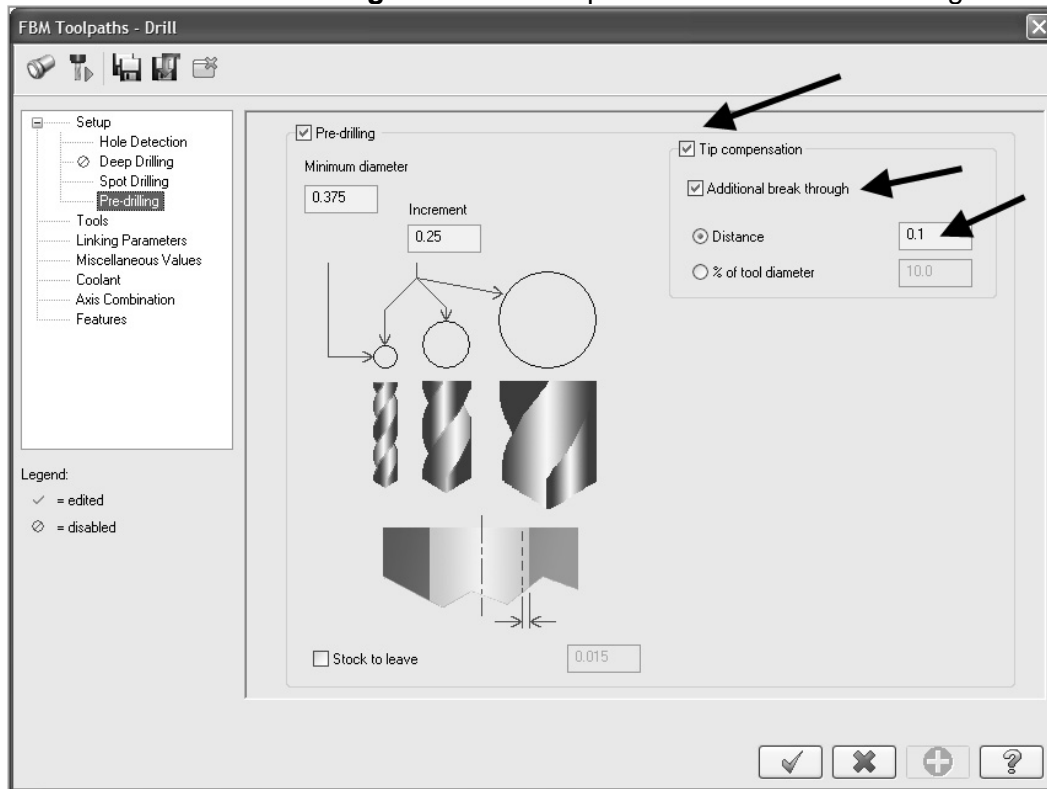
7. Select **Hole Detection** from the Tree Control dialog box and set parameters to **Include blind hole** and to **Limit search to plane TOP** as follows:



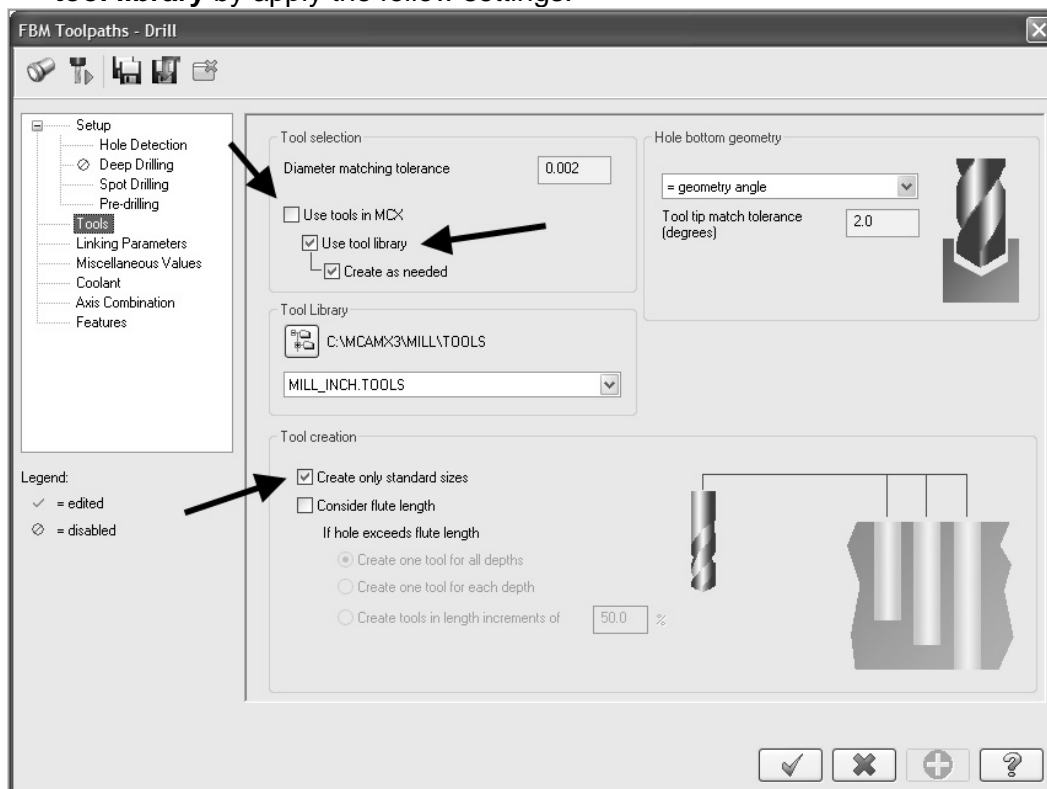
8. Select **Deep Drilling** from the Tree Control dialog box and uncheck **Deep drilling** Deep drilling.
9. Select **Spot Drilling** from the Tree Control dialog box and check **Spot drilling**.



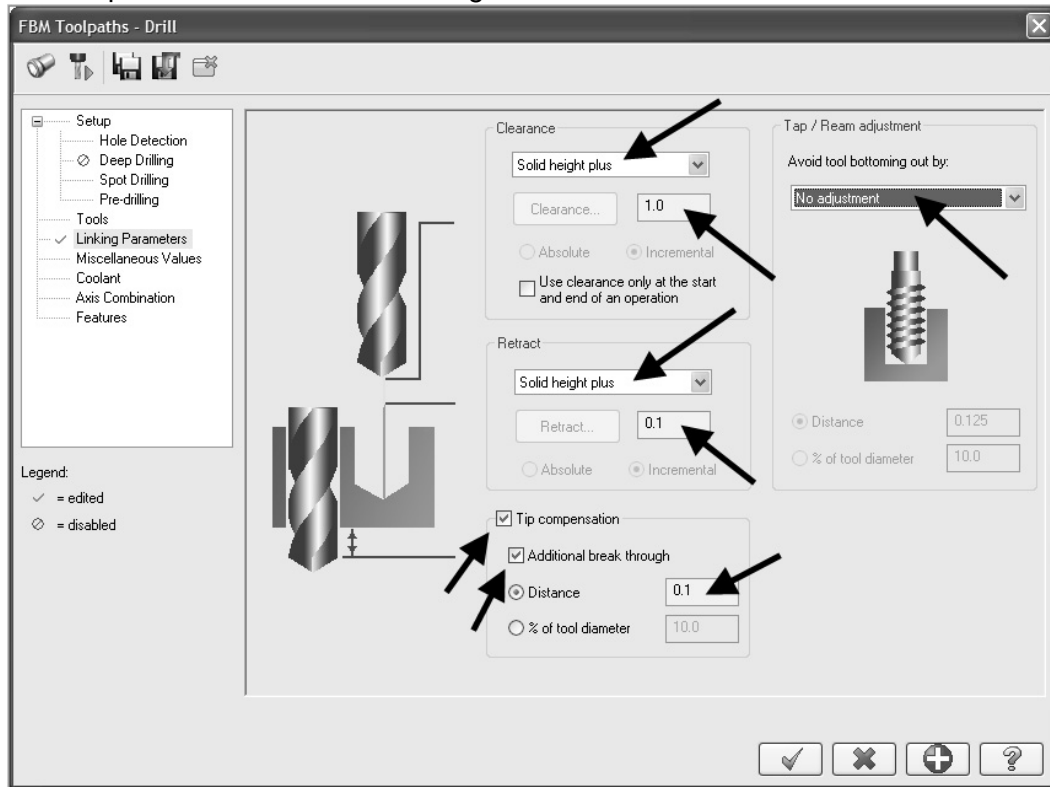
10. Select **Pre-drilling** from the Tree Control dialog box and check **Pre-drilling** Pre-drilling. The 3/8-16 UNC tapped holes will require Pre-drilling with a 5/16" drill. You will use an **Additional break through** of .1" for this part which features all through holes.



11. Select **Tools** from the Tree Control dialog box. You will allow FBM Drill to automatically **Use tool library** by apply the follow settings:



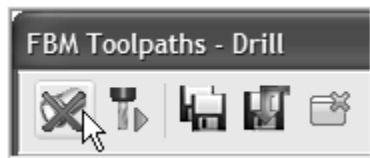
12. Select **Linking Parameters** from the Tree Control dialog box. This dialog box will be used to set our **Clearance** at **Solid height plus 1."**, **Retract** at **Solid height plus 0.1"**, an **Additional break through** of **.1"**, and **No adjustment** for the **Tap / Ream adjustment** for this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



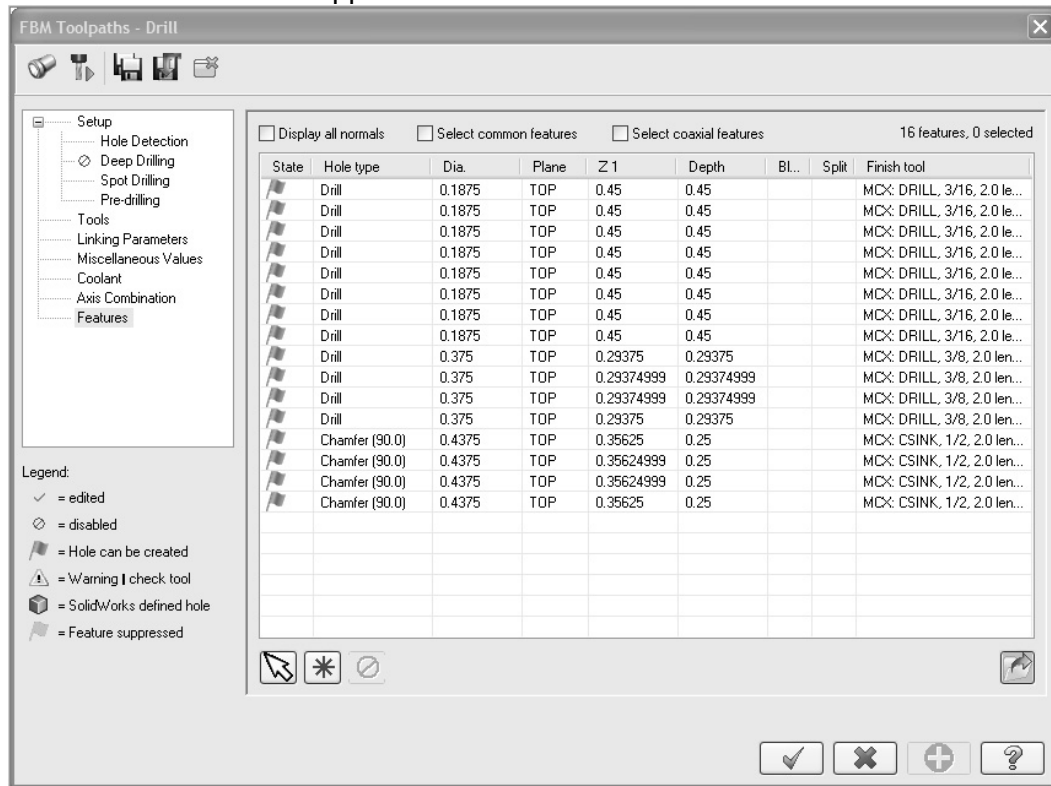
13. Select **Coolant** from the Tree Control dialog box. Set **Flood** to **On**:



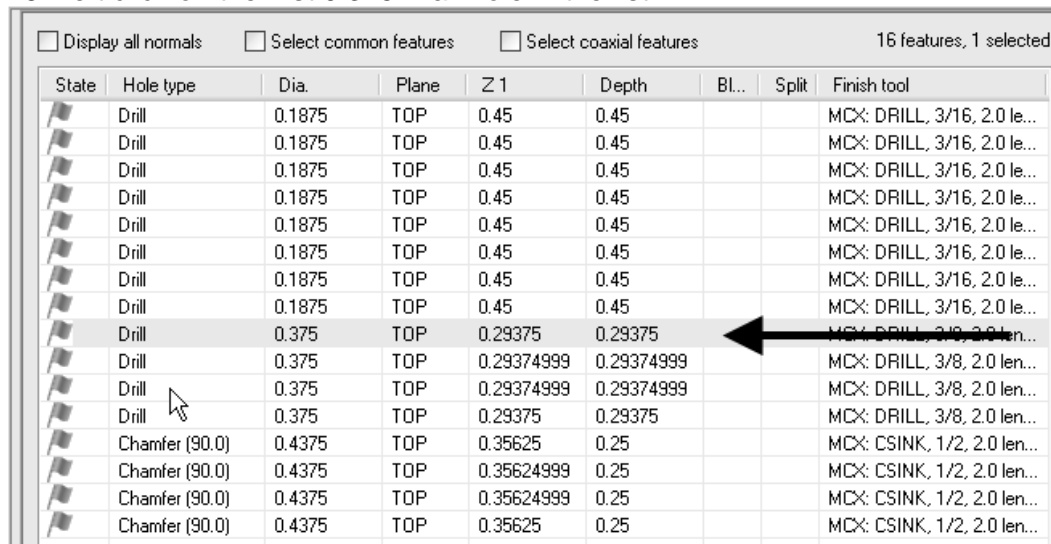
14. Click the **Detect...** button at the top of the FBM Toolpaths – Drill dialog box.



15. Your screen should appear as follows:

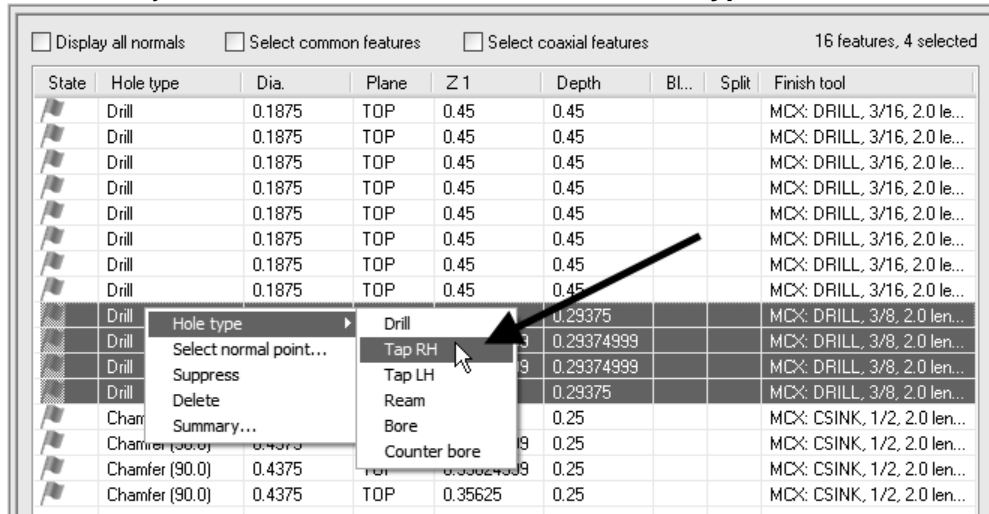


16. Left click on the first 0.375 Dia. Hole in the list.



17. Press and hold the **Ctrl** key on your keyboard. Left click on the next three 0.375 Dia. Holes in the list. With **Ctrl** still held, you can select and unselect any mistakes. When the four correct holes are selected, release the **Ctrl** key.

18. Right click on any of these four 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.



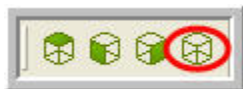
19. Select the OK button  to process all holmaking operations for this part.

Drill cycles used are selected based on the cycle set within each tool stored in the tool library. With the selection of **Feed Calculation From Tool** in the **Machine Group Properties Tool Settings** tab, all feeds and speeds are set based on the values set in the Tool Libraries. Many users take the time to create a custom tool library with only their tools, with desired feeds and speeds set in the library.

TASK 8: VERIFY THE TOOLPATH

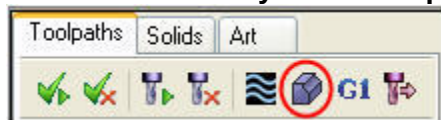
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.

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. In the Toolpath Manager pick all the operations to verify by picking the **Select All** icon .

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



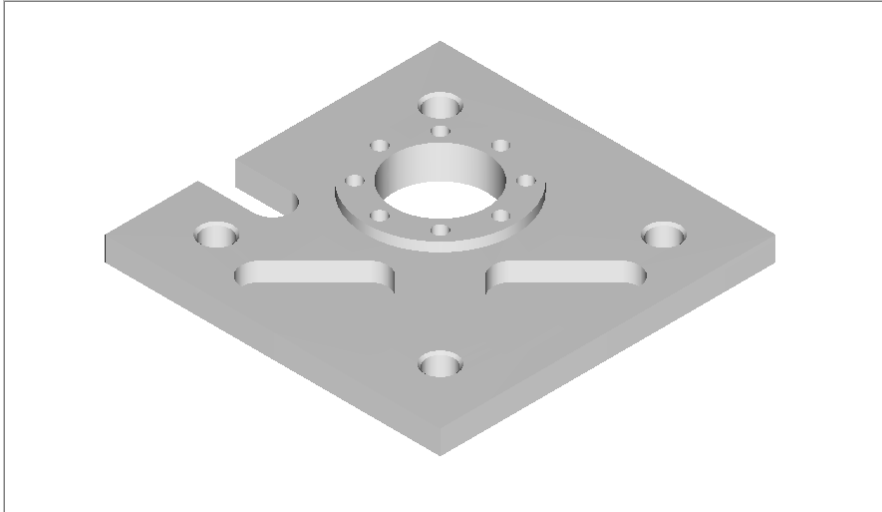
4. Adjust the Verify speed to midway along the speed control bar.



5. Select the play button to verify the toolpaths.



- ☞ The verified toolpaths are shown below.

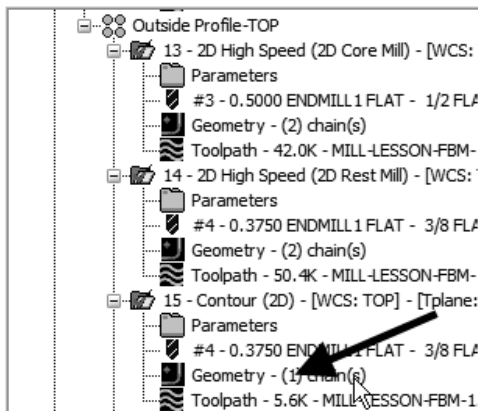


6. Select the OK button to exit Verify.

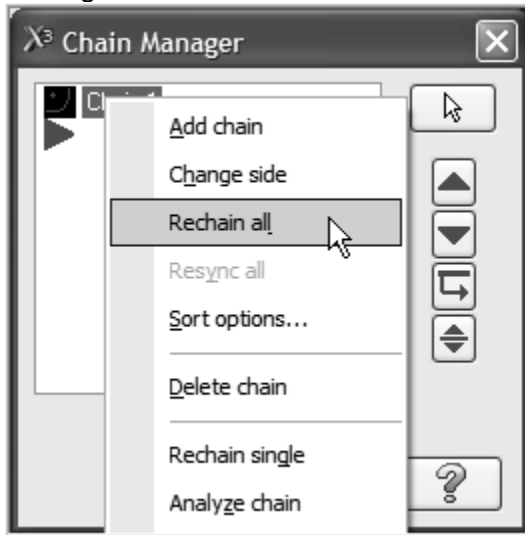
TASK 9: REMOVE THE OUTSIDE FINISH PASS

The drawing for this lesson noted that the stock size is equal to the outside dimensions of the part. We configured **FBM Mill** to **Rough outside of part** and to **Finish outside of part** in order to address the slot. However, the complete outside finish pass around the part is undesirable for work holding. FBM child operations can be edited independently of the parent parameters. In this task you will edit the outside profile **Chain** to address only the slot, and modify the **Lead in/out** parameters.

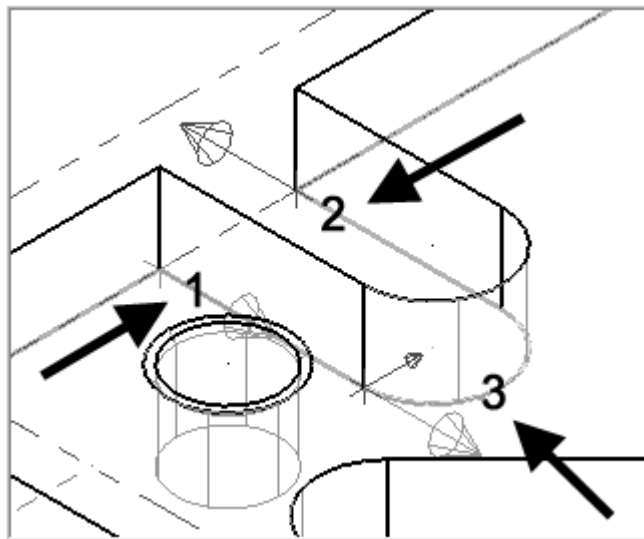
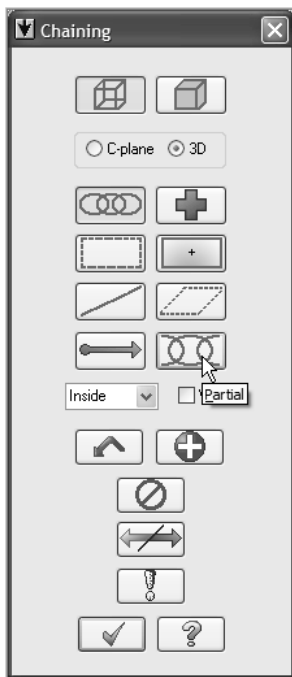
1. Use the **Operations Manager** on the left side of the Mastercam interface Locate **Operation 15 – Contour (2D)** within the **Outside Profile-TOP** Toolpath Group. Click on the **Geometry** icon.



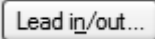
- Right click on Chain 1 and select **Rechain all**

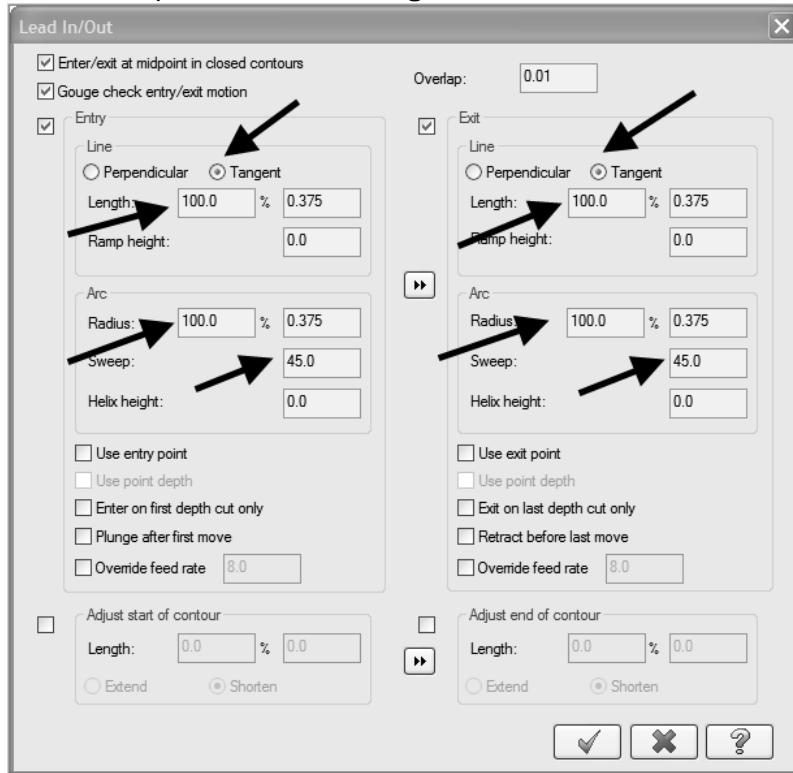




- Right click in the Geometry window and select the **Isometric GView**. Right click again and select **Fit**.
- Press and hold the **Alt** key on your keyboards and type **s** to toggle shading in Mastercam. This Key mapping shortcut is often denoted as **<Alt+S>**. Toggle shading until it is Off and the full wireframe is visible.
- Select **Partial** chaining. When prompted for **Select the first entity**, pick line 1. Next prompt **Select the last entity** pick line 2. Final prompt **Branch point reached. Select next branch** pick the slot radius 3.

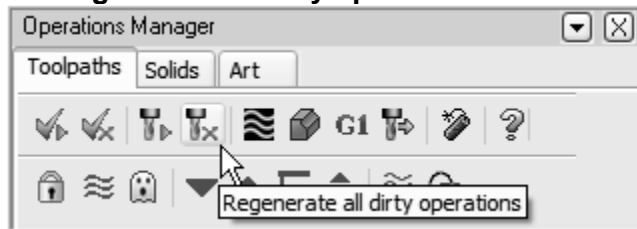


- Select the OK button to complete the **Chain** selection.
- Select the OK button to complete the **Chain Manager**.
- Still focusing on **Operation 15 – Contour (2D)** in the **Operations Manager** click on the **Parameters** icon.

9. In the **Contour Parameters Tab** select the **Lead in/out** button 
10. Set the parameters for **Tangent** lines and arcs, sized at 100% of the tool's diameter.




11. Select the OK button  to close the **Lead in/out** dialog box.
12. Select the OK button  to close the **Contour 2D** dialog box.
13. **Regenerate all dirty operations** to refresh this modified Operation.

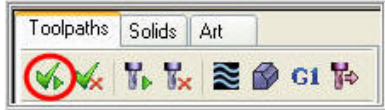


TASK 10: SAVE THE UPDATED MCX FILE

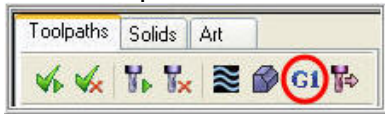
1. Select **File**.
2. Select **Save**.

TASK 11: POST AND CREATE THE CNC CODE FILE

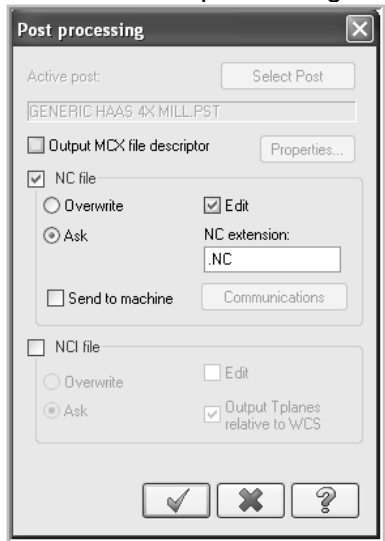
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.




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 manger window and expand the window to the right.



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

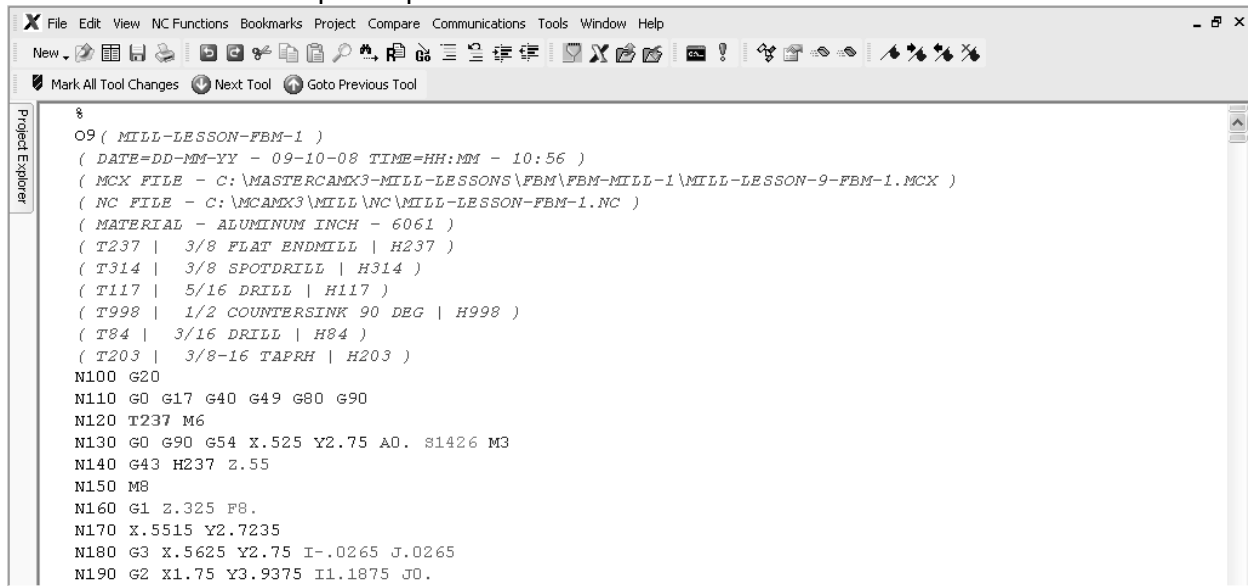


5. Select the OK button  to continue.
6. Ensure the same name as your Mastercam part file name is displayed in the NC File name field as shown below:




7. Select the Save button.

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



```
O9 ( MILL-LESSON-FBM-1 )
( DATE=DD-MM-YY - 09-10-08 TIME=HH:MM - 10:56 )
( MCX FILE - C:\MASTERCAMX3\MILL-LESSONS\FBM\FBM-MILL-1\MILL-LESSON-9-FBM-1.MCX )
( NC FILE - C:\MASTERCAMX3\MILL\NC\MILL-LESSON-FBM-1.NC )
( MATERIAL - ALUMINUM INCH - 6061 )
( T237 | 3/8 FLAT ENDMILL | H237 )
( T314 | 3/8 SPOTDRILL | H314 )
( T117 | 5/16 DRILL | H117 )
( T998 | 1/2 COUNTERSINK 90 DEG | H998 )
( T84 | 3/16 DRILL | H84 )
( T203 | 3/8-16 TAPRH | H203 )
N100 G20
N110 G0 G17 G40 G49 G80 G90
N120 T237 M6
N130 G0 G90 G54 X.525 Y2.75 A0. S1426 M3
N140 G43 H237 Z.55
N150 M8
N160 G1 Z.325 F8.
N170 X.5515 Y2.7235
N180 G3 X.5625 Y2.75 I-.0265 J.0265
N190 G2 X1.75 Y3.9375 I1.1875 J0.
```

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

This completes Mill-Lesson-FBM-1.