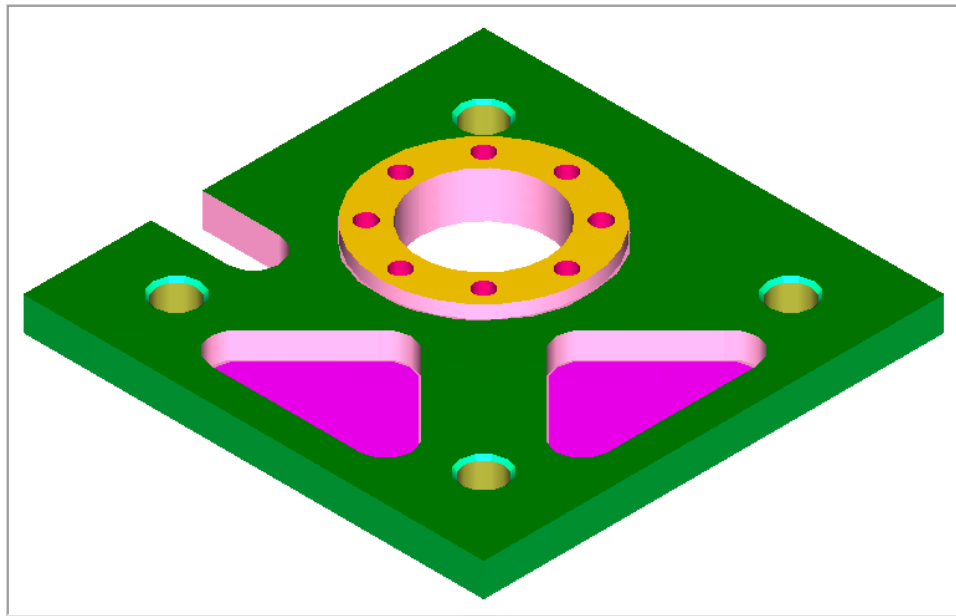


Mastercam 2017

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.

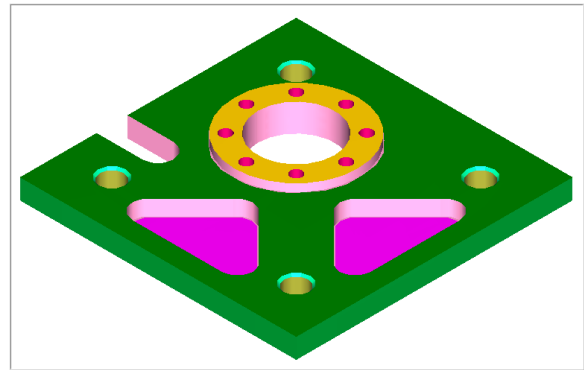
Material for the part.

Feed calculation.

➤ Generate a 2-dimensional milling toolpath consisting of:

FBM Mill operation and child toolpath operations.

FBM Drill operation and child toolpath operations.

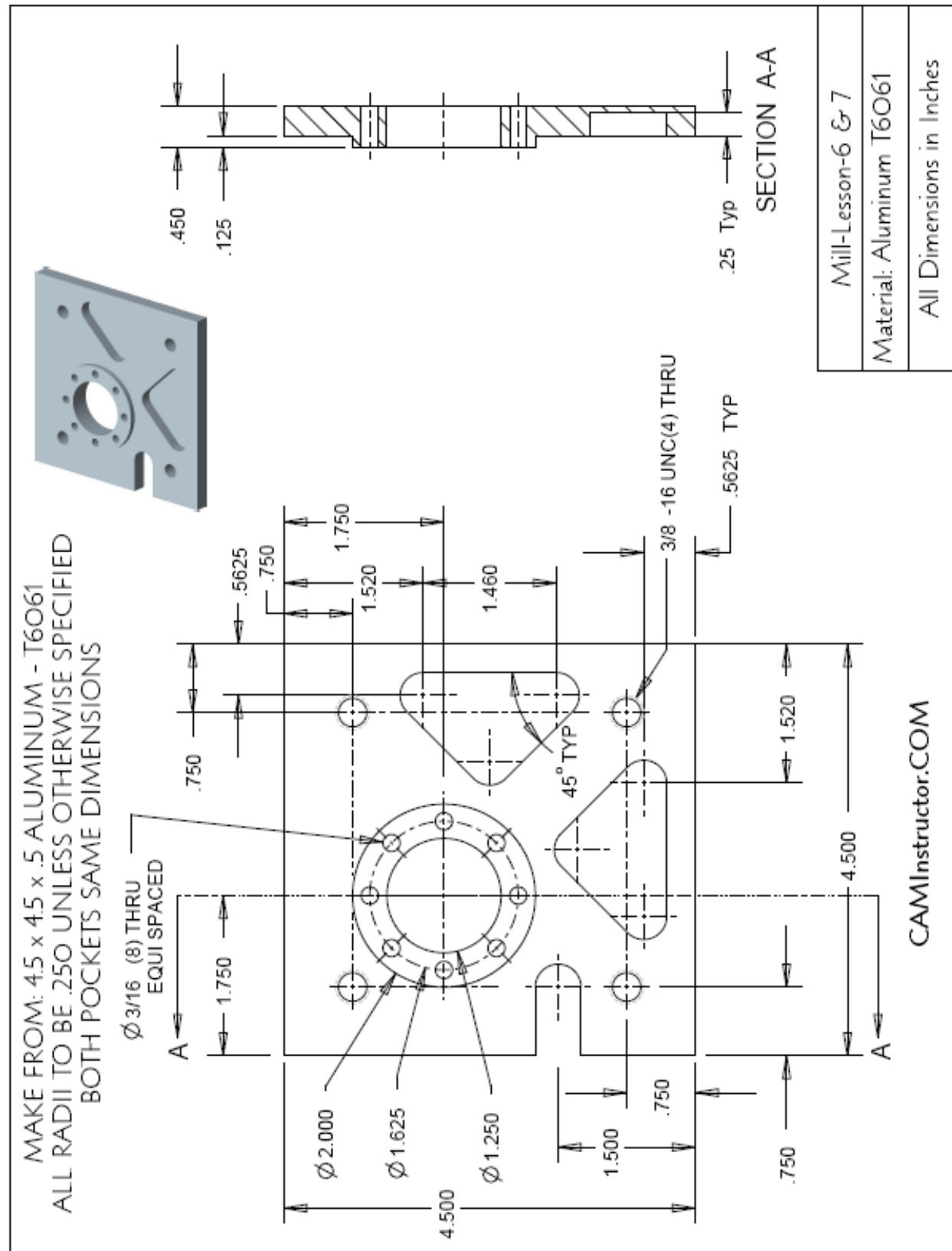


➤ Inspect the toolpath using Mastercam's Verify by:

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

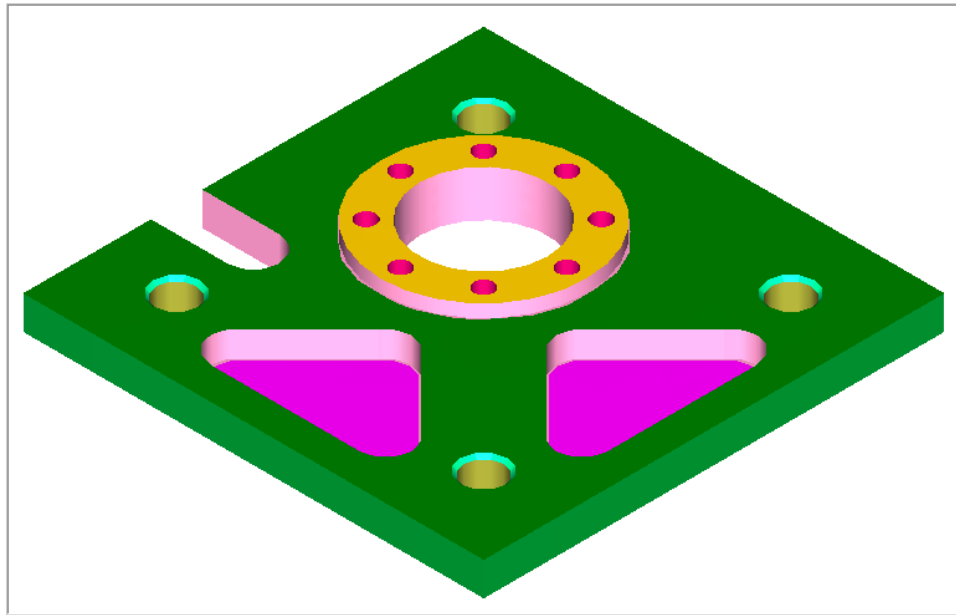
Generating the NC- code.

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.



MILL-LESSON-FBM-1 - THE PROCESS

Toolpath Creation

- TASK 1:** Open File
- 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 Mastercam file
- TASK 11:** Post and create the CNC code file

TASK 1: OPEN FILE

- In this lesson you will use a Mastercam solid model file that was previously created in SolidWorks.

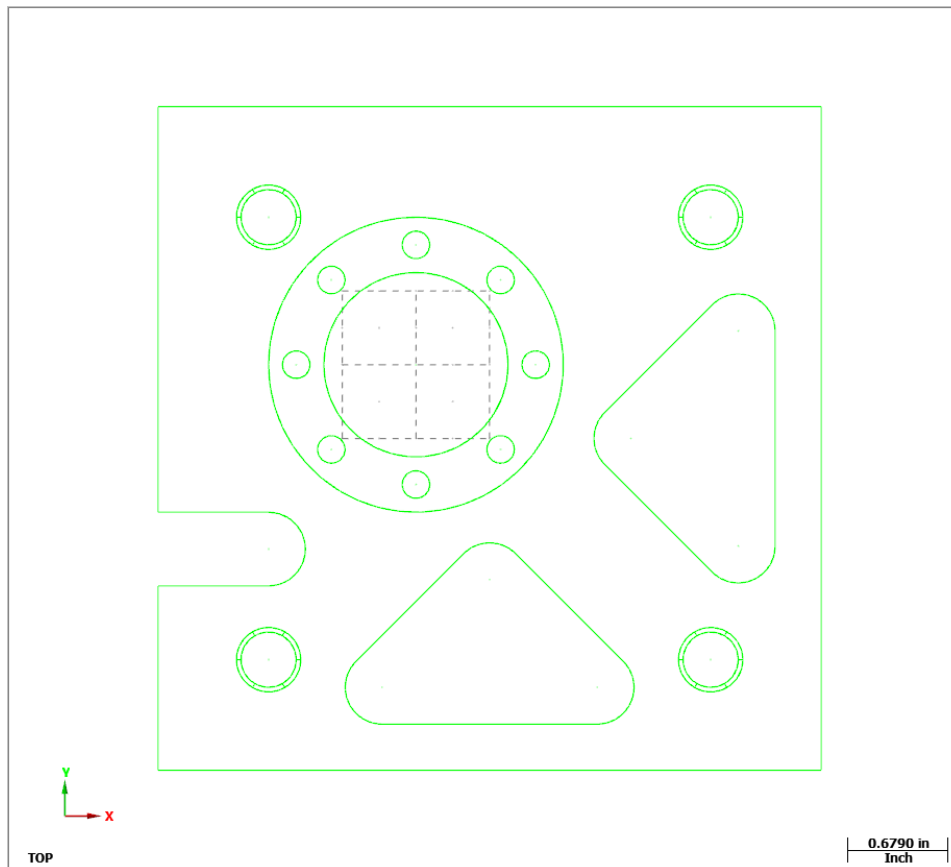
Feature Based Machining (FBM)

Features define a solid part's topology. They describe the part's physical properties including its faces, and the shape, size, and location of holes, slots, pockets, bosses and other characteristics.

Mastercam's feature based machining (FBM)—available with Mastercam Mill level 1 or Mastercam Router or higher—**eliminates the manual processes** involved in identifying features for programming milling and drilling operations on solid parts.

FBM analyzes the part for specific feature types and **automatically creates** the individual toolpaths needed to machine the selected features.

1. Click on **File>Open**
2. On the multimedia DVD that came with this text is **a folder called Mastercam-Files** browse to this location and select **Mill-Lesson-FBM-1**. If you are an on-line user you will find the file in this lesson.
3. Select the Open button to load the file and complete this function.
4. The screen should look like the one below:



5. Select **File>Save as...**
6. Save to an appropriate location.
7. Select the Save button to save the file and complete this function.

TASK 2: SETTING THE ENVIRONMENT

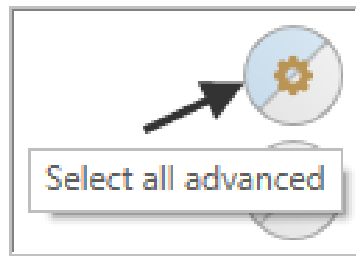
- Before moving on you should set up the grid and machine type as outlined in the **Setting the environment** section at the beginning of this text:

 1. Set up the Grid. This will help identify the location of the origin.
 2. Set the machine type to a Haas Vertical Spindle CNC machine.

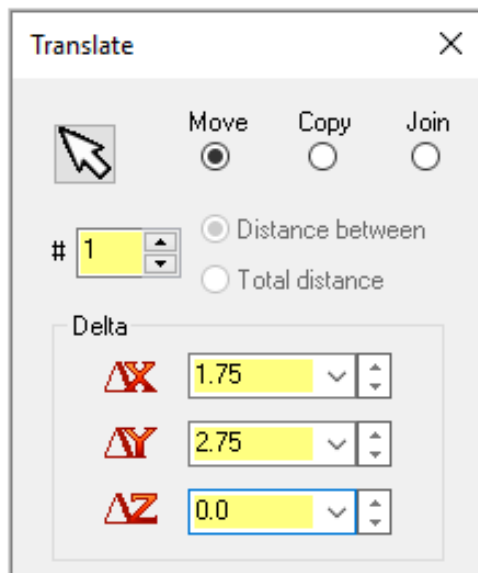
TASK 3: TRANSLATE PART GEOMETRY TO MACHINING ORIGIN

- Translating the geometry between 2 points.

 1. Select the **TRANSFORM** tab and in the **Position** section select **Translate**.
 2. You are prompted to "Translate: select entities to translate". First select the **Select all advanced** button on the right of your screen.



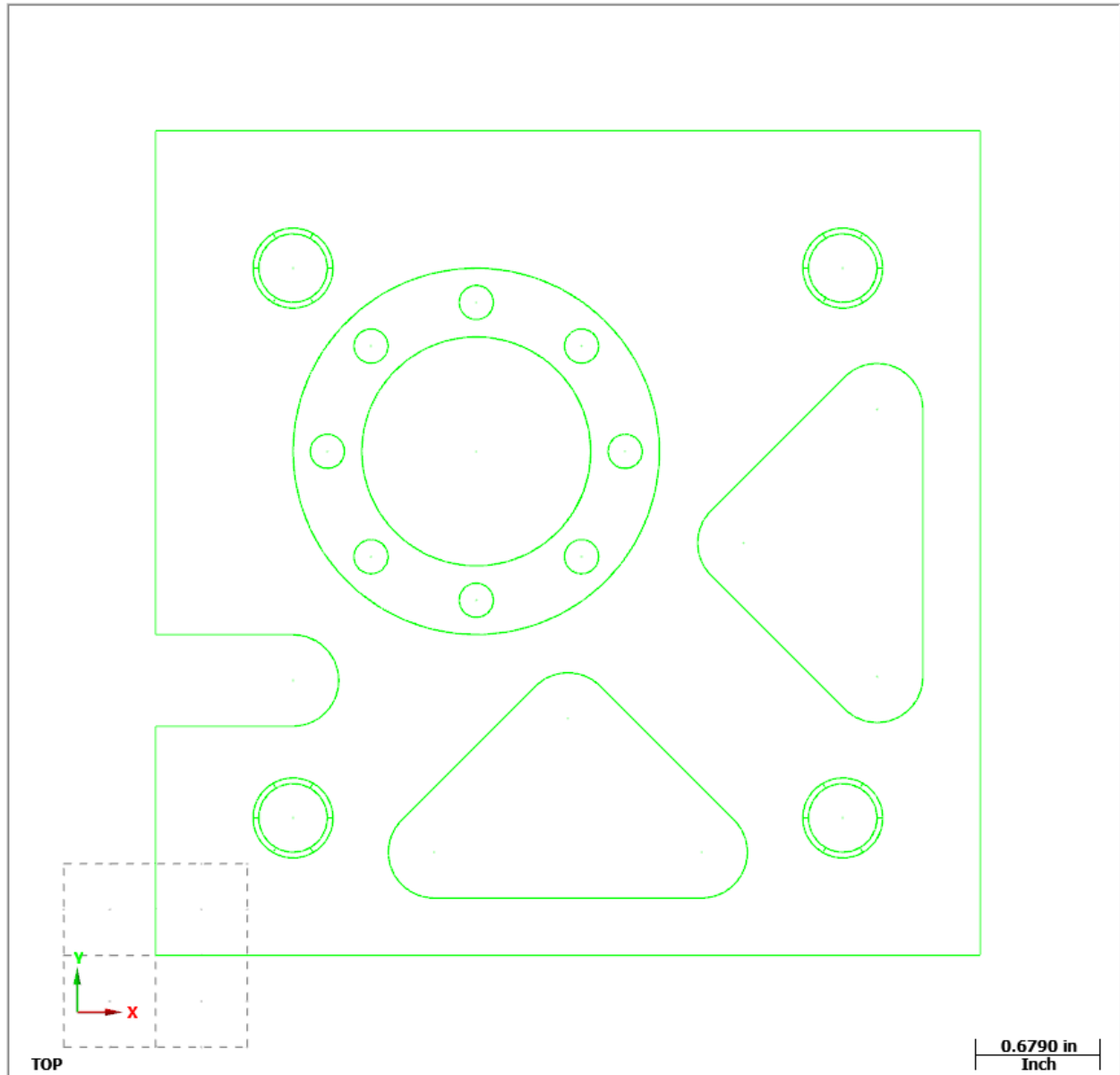
3. Select the **All Entities** button  from the **Select All** dialog Box.
4. Pick the **End Selection** icon .
5. Select the **Move** radio button and enter Delta values of ΔX 1.75, ΔY 2.75, and ΔZ 0.



6. Select the OK button  in the Translate dialog box.
7. Right mouse click in the graphics area and click on the **Clear Colors** icon .

8. Right mouse click in the graphics area and click on the **Fit** icon .
9. Right mouse click in the graphics area and click on **Un-Zoom 80%**.

☞ The screen should look like the one below:



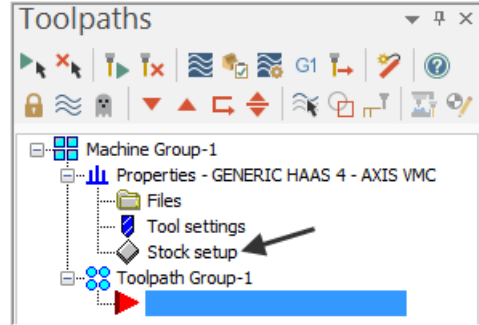
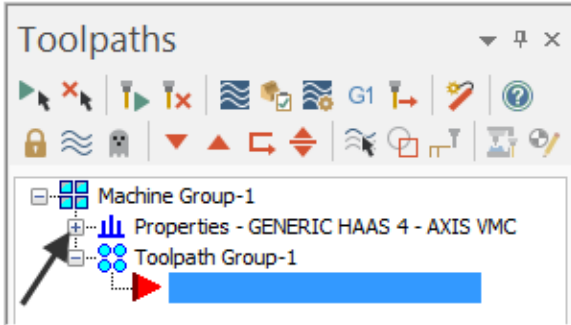
Toolpath Creation

TASK 4:

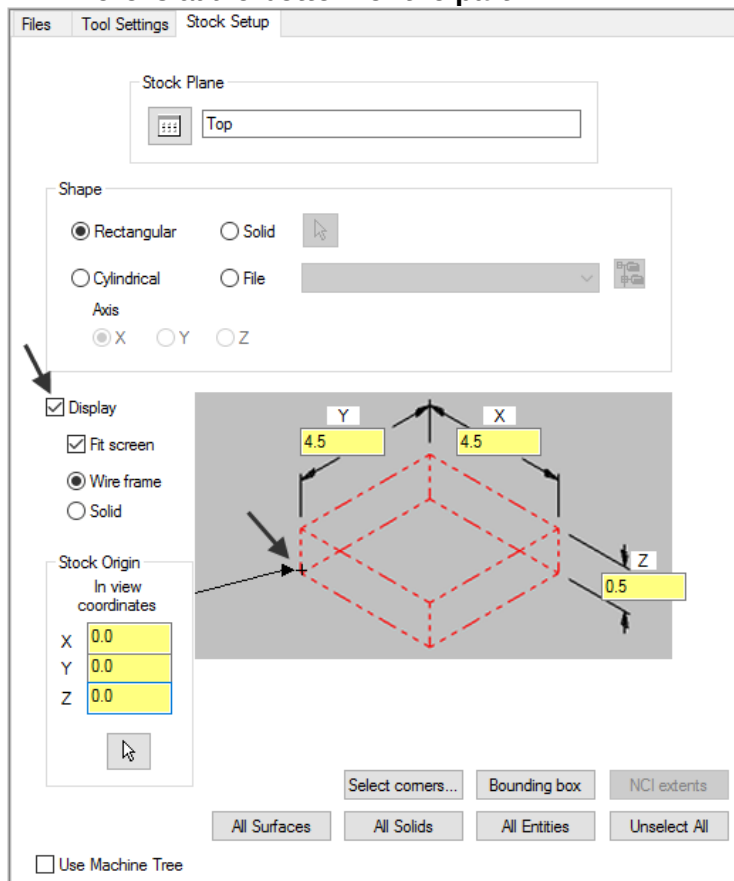
DEFINE THE ROUGH STOCK USING STOCK SETUP

➤ Defining the Rough Stock using stock setup.

1. Right mouse click in the graphics area and click on the **Fit** icon .
2. Select the **plus** in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will **Show/Hide** Toolpaths Manager pane.



3. Select **Stock setup** in the Toolpaths 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**.



Mastercam Training Guide

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.

Files | Tool Settings | Stock Setup

Program # 9

Feed Calculation

- ☒ From tool
- ☐ From material
- ☐ From defaults
- ☐ User defined

Spindle speed 5000.0

Feed rate 50.0

Retract rate 125.0

Plunge rate 25.0

☐ Adjust feed on arc move

Minimum arc feed 5.0

Toolpath Configuration

- ☒ Assign tool numbers sequentially
- ☒ Warn of duplicate tool numbers
- ☐ Use tool's step, peck, coolant
- ☐ Search tool library when entering a tool number

Advanced options

- ☒ Override defaults with modal values
- ☒ Clearance height
- ☒ Retract height
- ☒ Feed plane

Sequence #

Start 1.0

Increment 1.0

Material

ALUMINUM inch - 2024

Edit... Select...

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

Material

ALUMINUM inch - 2024

Edit... Select...

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

Display options

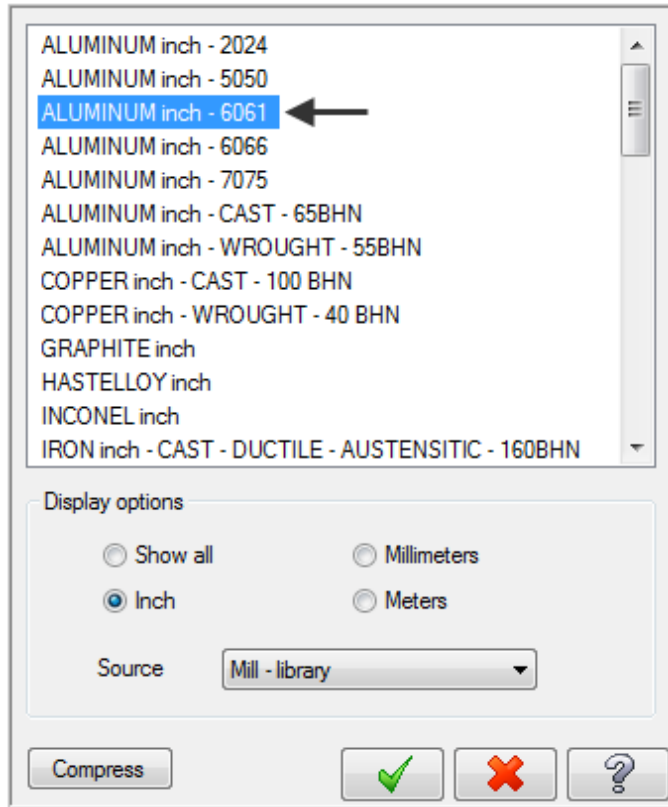
- ☐ Show all
- ☒ Inch
- ☐ Millimeters
- ☐ Meters

Source

Mill - library

Compress ?

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



ALUMINUM inch - 2024
ALUMINUM inch - 5050
ALUMINUM inch - 6061
ALUMINUM inch - 6066
ALUMINUM inch - 7075
ALUMINUM inch - CAST - 65BHN
ALUMINUM inch - WROUGHT - 55BHN
COPPER inch - CAST - 100 BHN
COPPER inch - WROUGHT - 40 BHN
GRAPHITE inch
HASTELLOY inch
INCONEL inch
IRON inch - CAST - DUCTILE - AUSTENSITIC - 160BHN

Display options

☐ Show all ☐ Millimeters
☒ Inch ☐ Meters

Source

Compress   

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. Select the **TOOLPATHS** tab at the top right side of the screen.
2. Select **FBM Mill** in the **2D Milling** toolpath section.

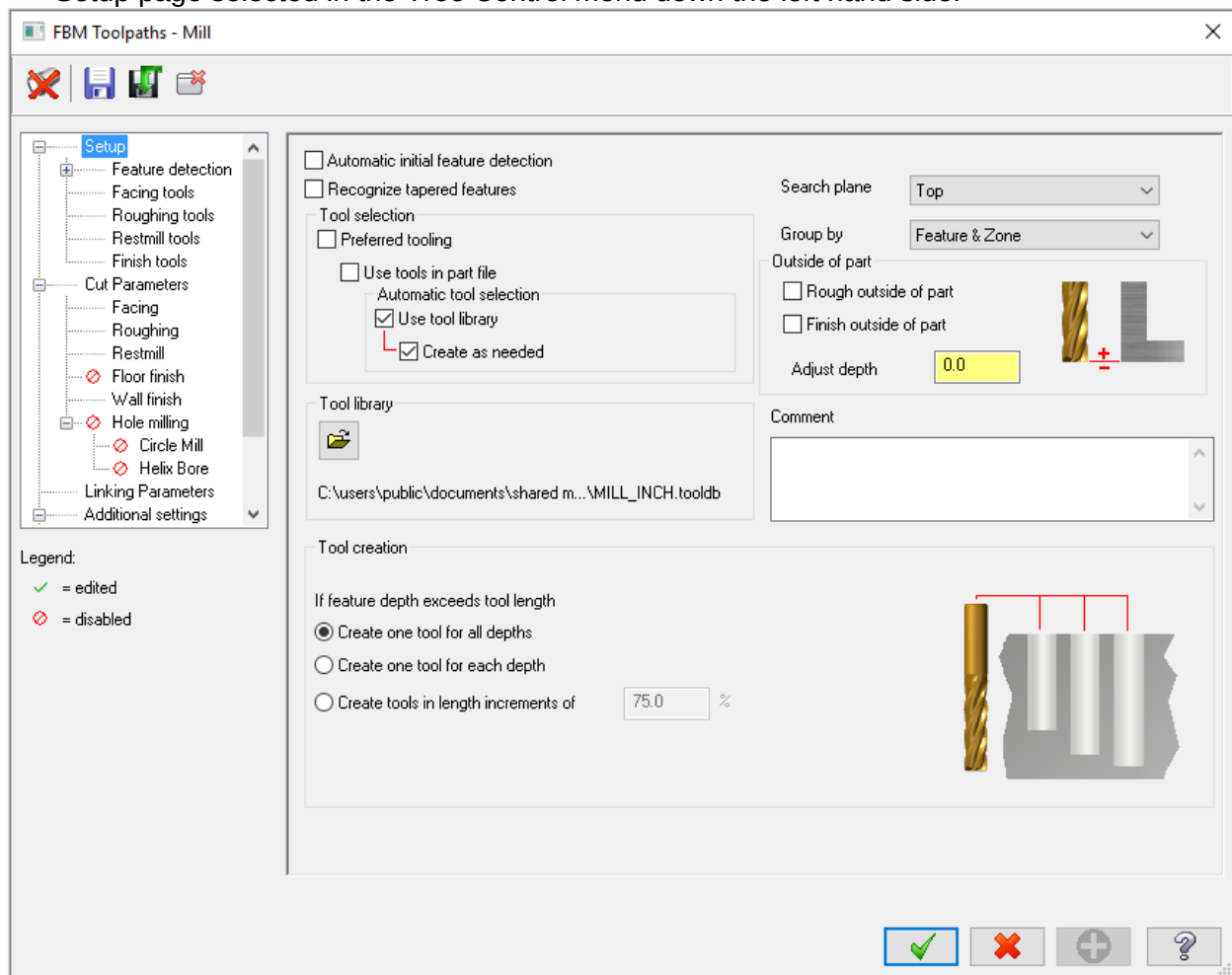
FBM Mill is a method for quickly producing the equivalent of standard Face, Pocket, and Contour toolpaths.

A single Parent operation is used to interrogate the solid model, detecting the part's features and determining machining zones.

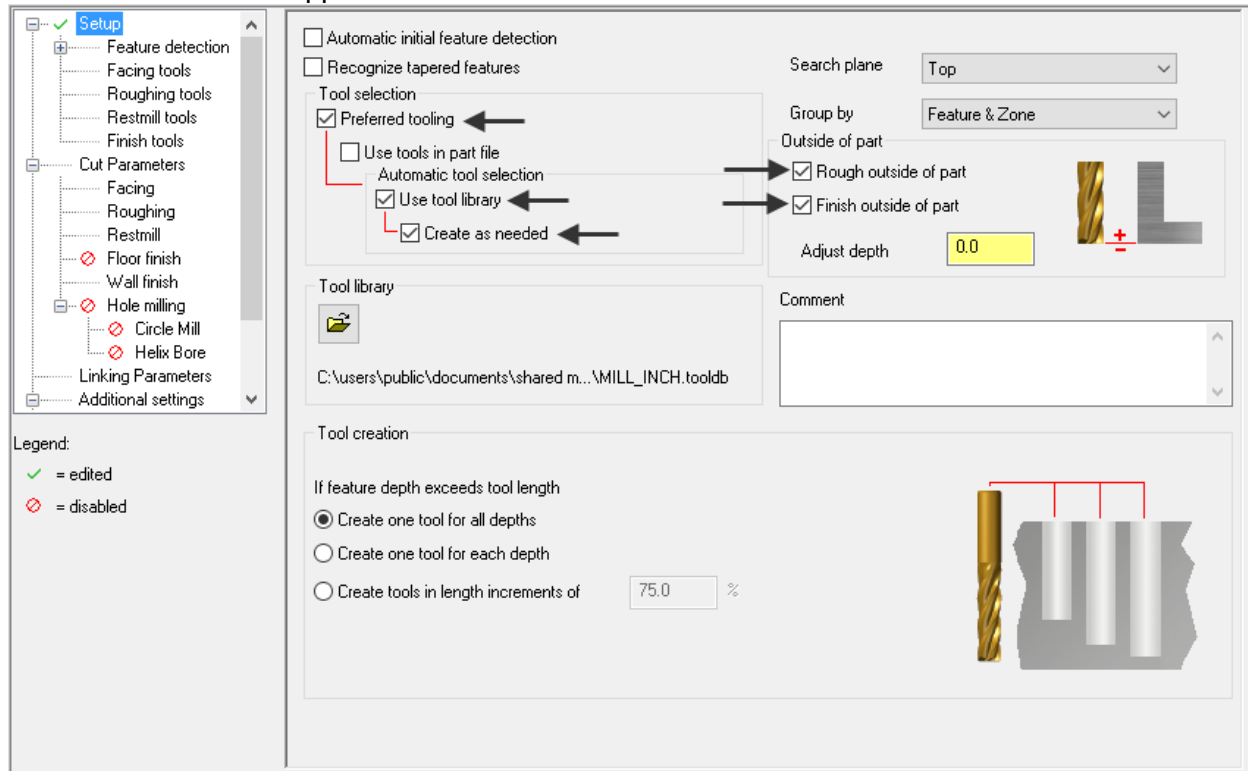
A single set of configuration settings will address facing, roughing, rest milling, floor and wall finishing, and outside contouring.

Child operations are created using 2D High Speed, Face, Pocket, and Contour operations. The parameters of Child operations can be edited for further customization.

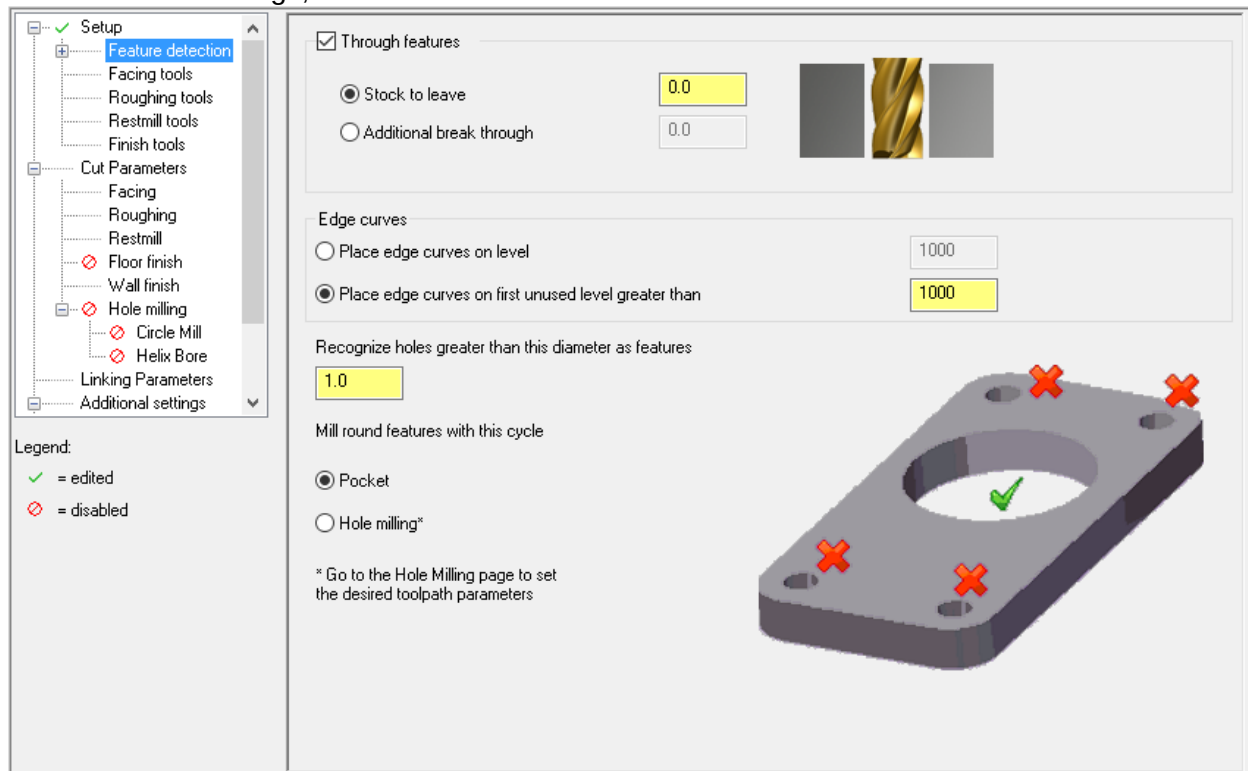
3. On the screen you will now see the **FBM Toolpaths – Mill** dialog box, starting with the Setup page selected in the Tree Control menu down the left hand side:



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

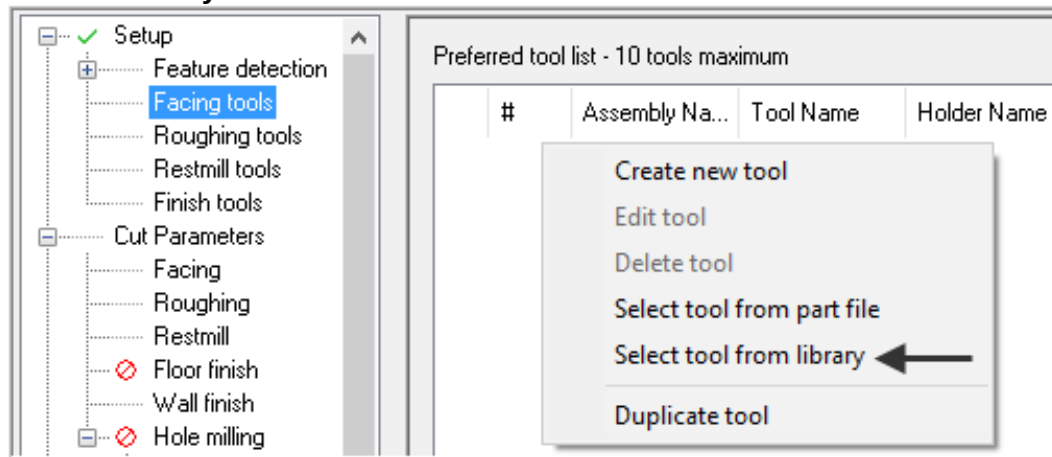


5. Click on **Feature detection** in the Tree Control menu on the left of the dialog box and leave the default settings, which should be as follows:



Mastercam Training Guide

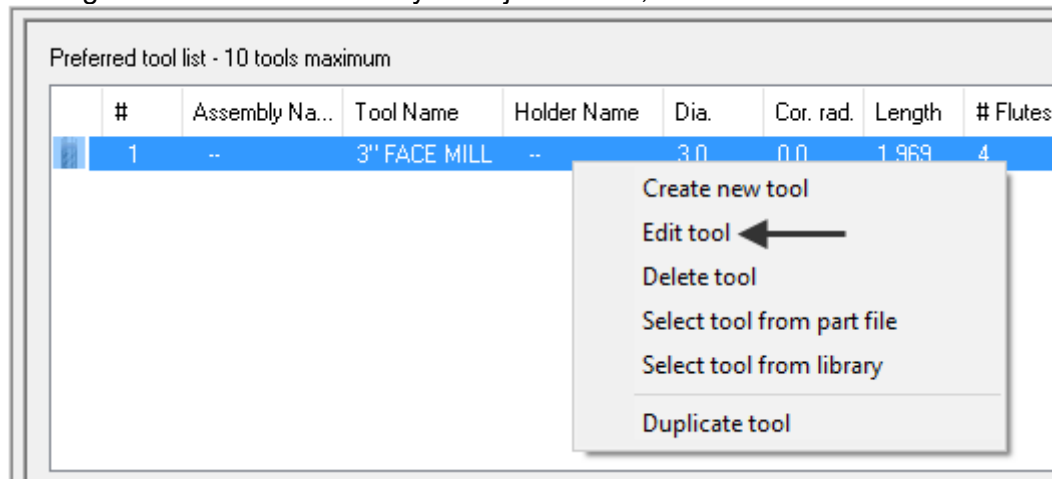
- Click on **Facing tools** in the Tree Control menu. You will now select a Facing tool from the Tool Library.
- Right click in the empty white space in the **Preferred tools list** and left click on **Select tool from library**.

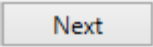


- Use the slider bar on the right of this dialog box to scroll down and locate a **3.0 diameter face mill**. Select the 3.0 diameter face mill by picking anywhere along the row.

#	Tool Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. Type
323	2 -1/2" FACE MILL	2.5	0.0	1.575	4	Face mill	None
324	3" FACE MILL	3.0	0.0	1.969	4	Face mill	None
325	4" FACE MILL	4.0	0.0	1.969	2	Face mill	None
326	5" FACE MILL	5.0	0.0	2.48	4	Face mill	None

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



11. Click on the Next button .
12. Change the **Feed rate**, **Plunge rate**, and **Retract rate** to **20.0**, and the **Spindle speed** to **1500**.

Finalize miscellaneous properties.

Adjust any miscellaneous properties before finalizing tool creation.

Operation

Tool number: 1

Length offset: 1

Diameter offset: 1

Head number: -1

SFM: 1178.01047

FPT: 0.00333

Number of flutes: 4

Feed rate: 20

Plunge rate: 20

Retract rate: 20

Spindle speed: 1500

Spindle direction: Clockwise

Material: HSS

General

Name: 3" FACE MILL

Description:

Manufacturer name:

Manufacturer's tool code:

Tool Grade:

Milling

☒ Rough Tool

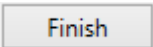
☒ Finish Tool

Rough XY step (%): 0

Rough Z step (%): 0

Finish XY step (%): 0

Finish Z step (%): 0

13. Select the Finish button  to complete these changes in Parameters.
14. Click on **Roughing tools** in the Tree Control menu. You will now select Roughing tools from the Tool Library.
15. Right click in the empty white space in the Preferred tools list and left click on **Select tool from library**.

Setup

Feature detection

Facing tools

Roughing tools

Restmill tools

Finish tools

Cut Parameters

Facing

Roughing

Restmill

Floor finish

Wall finish

Hole milling

Preferred tool list - 10 tools maximum

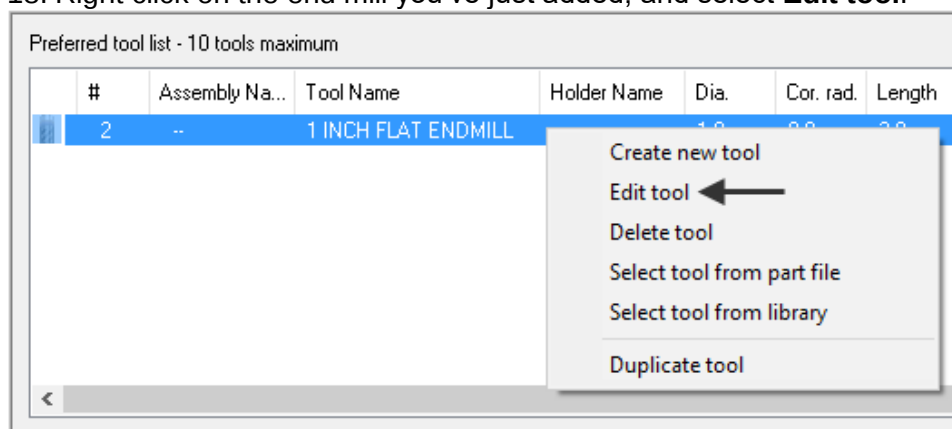
#	Assembly Na...	Tool Name	Holder Name
Create new tool Edit tool Delete tool Select tool from part file Select tool from library Duplicate tool			

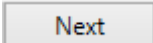
Mastercam Training Guide

16. 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:

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ra...	Type
294	--	3/4 FLAT ENDMILL	--	0.75	0.0	2.0	4	No...	Endmill1 Flat
295	--	13/16 FLAT ENDMILL	--	0.8125	0.0	2.0	4	No...	Endmill1 Flat
296	--	7/8 FLAT ENDMILL	--	0.875	0.0	2.0	4	No...	Endmill1 Flat
297	--	1 INCH FLAT ENDMILL	--	1.0	0.0	2.0	4	No...	Endmill1 Flat
298	--	1-3/16 FLAT ENDMILL	--	1.1875	0.0	2.0	4	No...	Endmill1 Flat

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



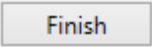
19. Click on the Next button .
20. Change the **Feed rate**, **Plunge rate**, and **Retract rate** to **10.0**, and the **Spindle speed** to **1300**.

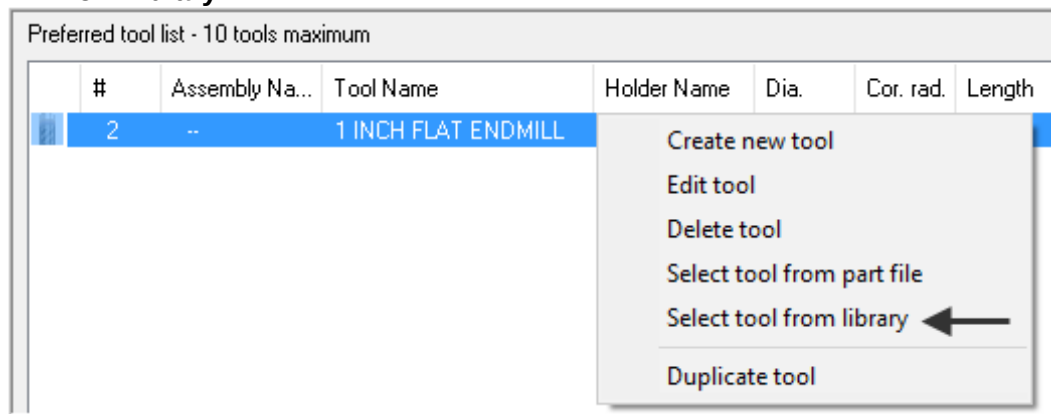
Finalize miscellaneous properties.

Adjust any miscellaneous properties before finalizing tool creation.

Operation		General	
Tool number:	2	Name:	1 INCH FLAT ENDMILL
Length offset:	2	Description:	
Diameter offset:	2	Manufacturer name:	+
Head number:	0	Manufacturer's tool code:	
SFM:	340.31414	Tool Grade:	+
FPT:	0.00192		
Number of flutes:	4		
Feed rate:	10		
Plunge rate:	10		
Retract rate:	10		
Spindle speed:	1300		
Spindle direction:	Clockwise		
Material:	HSS		

Milling	
☒ Rough Tool	
☒ Finish Tool	
Rough XY step (%):	50
Rough Z step (%):	50
Finish XY step (%):	25
Finish Z step (%):	25

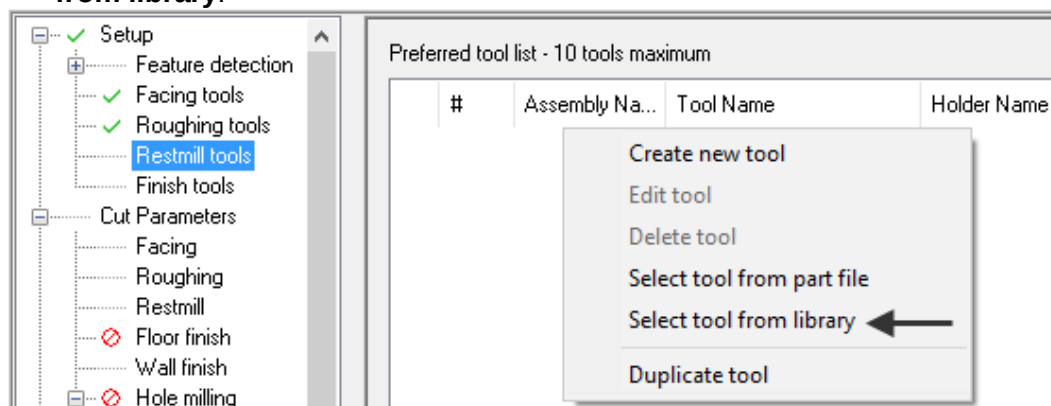
21. Select the Finish button  to complete these changes in Parameters.
22. Right click in the empty white space in the **Preferred tools list** and left click on **Select tool from library**.



23. 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:

#	Assem...	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	# Flutes	Rad. Type
287	--	3/8 FLAT ENDMILL	--	0.375	0.0	0.75	Endmill1 Flat	4	None
288	--	13/32 FLAT ENDMILL	--	0.40625	0.0	0.8	Endmill1 Flat	4	None
289	--	7/16 FLAT ENDMILL	--	0.4375	0.0	0.8	Endmill1 Flat	4	None
290	--	1/2 FLAT ENDMILL	--	0.5	0.0	1.0	Endmill1 Flat	4	None
291	--	17/32 FLAT ENDMILL	--	0.5312	0.0	1.0	Endmill1 Flat	4	None

24. Select the OK button  to complete the selection of this tool.
25. Set the **Feed rate**, **Plunge rate**, and **Retract rate** to **10.0**, and the **Spindle speed** to **3000** by repeating the steps used on the previous tool.
26. Click on **Restmill tools** in the Tree Control menu. You will now select a Restmill tool from the Tool Library.
27. Right click in the empty white space in the Preferred tools list and left click on **Select tool from library**.



Mastercam Training Guide

28. 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.

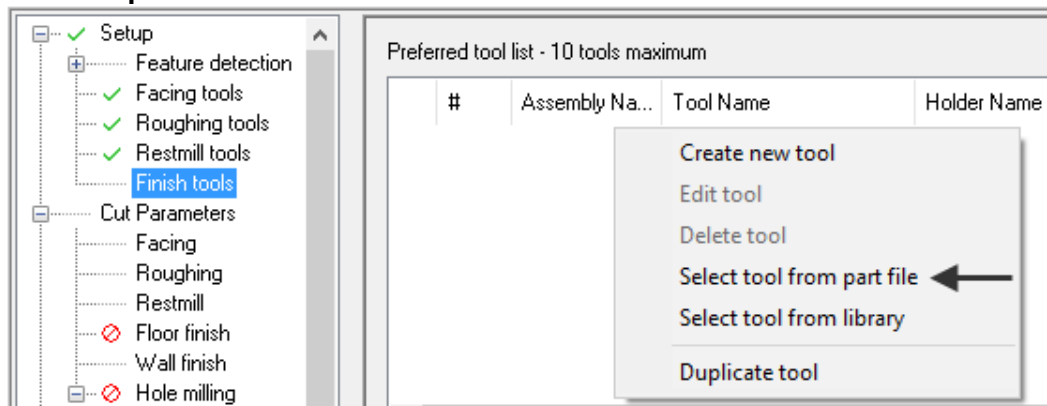
#	Assem...	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	Rad. Type
285	--	1/4 FLAT ENDMILL	--	0.25	0.0	0.5	Endmill1 Flat	None
286	--	5/16 FLAT ENDMILL	--	0.3125	0.0	0.75	Endmill1 Flat	None
287	--	3/8 FLAT ENDMILL	--	0.375	0.0	0.75	Endmill1 Flat	None
288	--	13/32 FLAT ENDMILL	--	0.40625	0.0	0.8	Endmill1 Flat	None

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

30. Set the **Feed rate**, **Plunge rate**, and **Retract rate** to **8.0**, and the **Spindle speed** to **4750** by repeating the steps used on the previous tool.

31. Click on **Finish tools** in the Tree Control menu. You will select a Finish tool from the list of tools you have just added.

32. Right click in the empty white space in the Preferred tools list and left click on **Select tool from part file**.

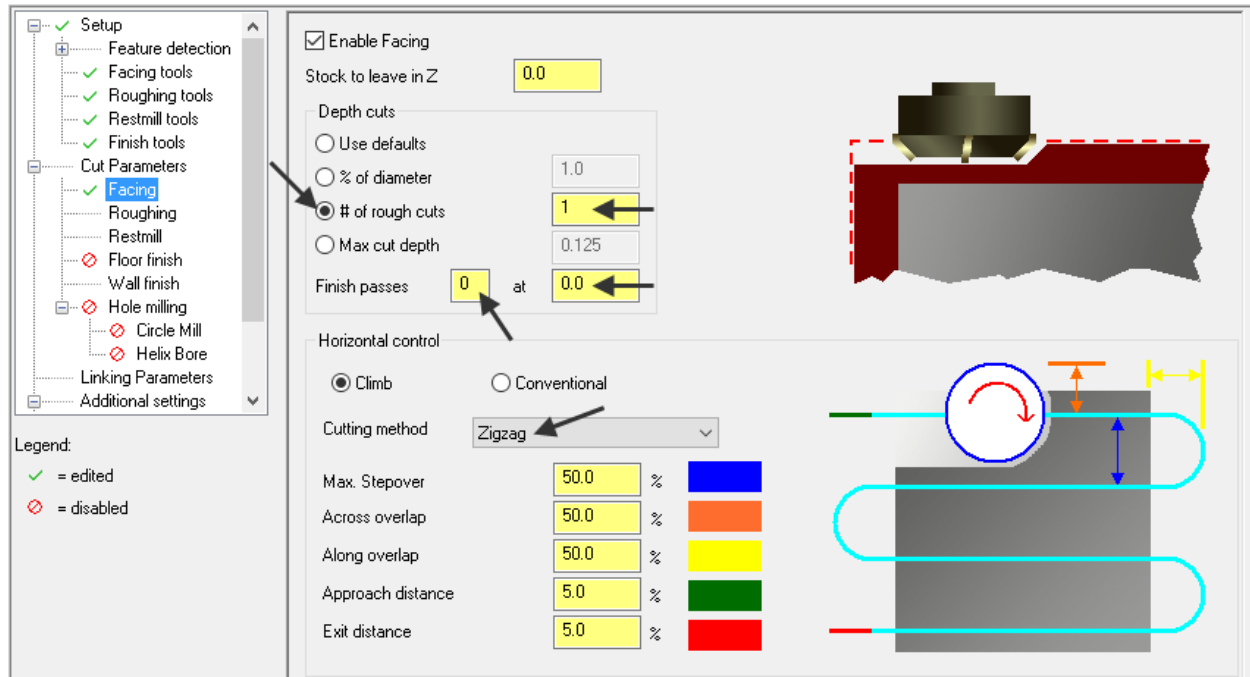


33. Select the **3/8 FLAT ENDMILL** you just added as a Restmill tool.

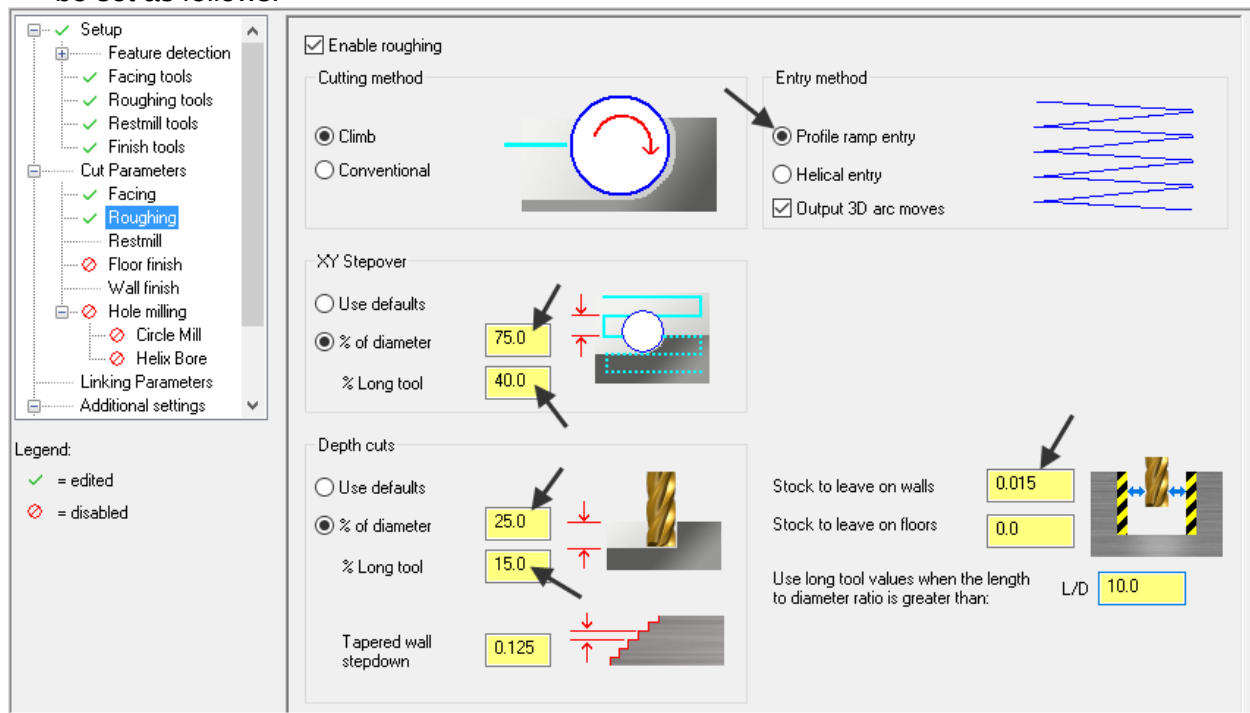
#	Assem...	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. Type
1	--	3" FACE MILL	--	3.0	0.0	1.969	4	Face mill	None
2	--	1 INCH FLAT ENDMILL	--	1.0	0.0	2.0	4	Endmill1 Flat	None
3	--	1/2 FLAT ENDMILL	--	0.5	0.0	1.0	4	Endmill1 Flat	None
4	--	3/8 FLAT ENDMILL	--	0.375	0.0	0.75	4	Endmill1 Flat	None

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

35. Next you will set **Facing** cut parameters, selecting **Facing** from the Tree Control dialog box, and adjusting the settings for 1 **rough cut**, and selecting **Zigzag**. Your selections should be set as follows:

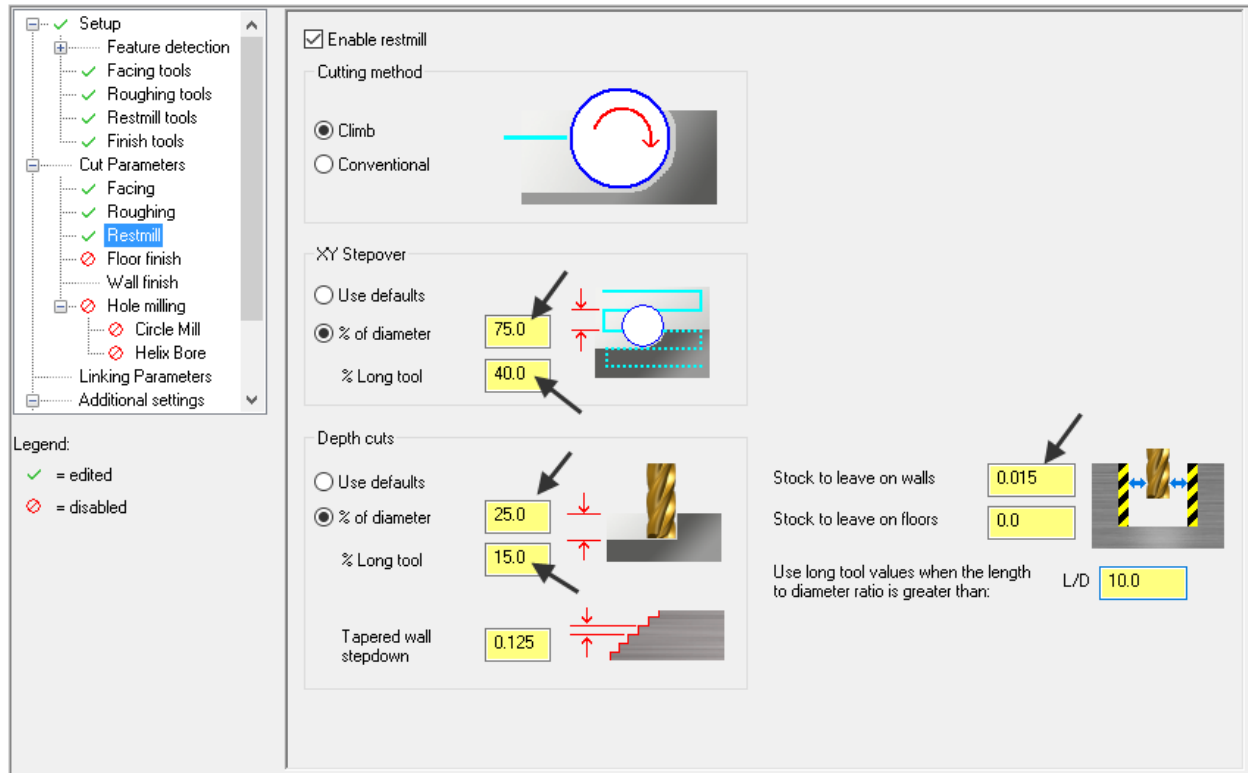


36. Next you will set **Roughing** cut parameters, selecting **Roughing** from the Tree Control dialog box, and adjusting settings for **Profile ramp entry**, 75% of diameter **XY Stepover**, 25% of diameter **Depth Cuts**, and **0.015" Stock to leave on walls**. Your selections should be set as follows:

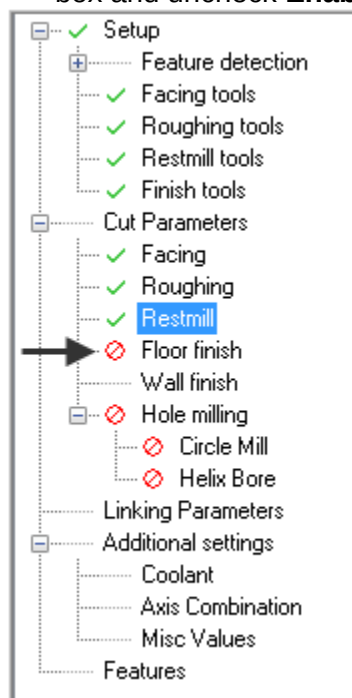


% of diameter (FBM) Determines the maximum depth and number of cuts made in the FBM-generated toolpath by multiplying the diameter of the tool by the diameter percentage you enter in this field. **For example**, if you enter 10%, a 2 inch tool takes .02 passes.

37. 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:

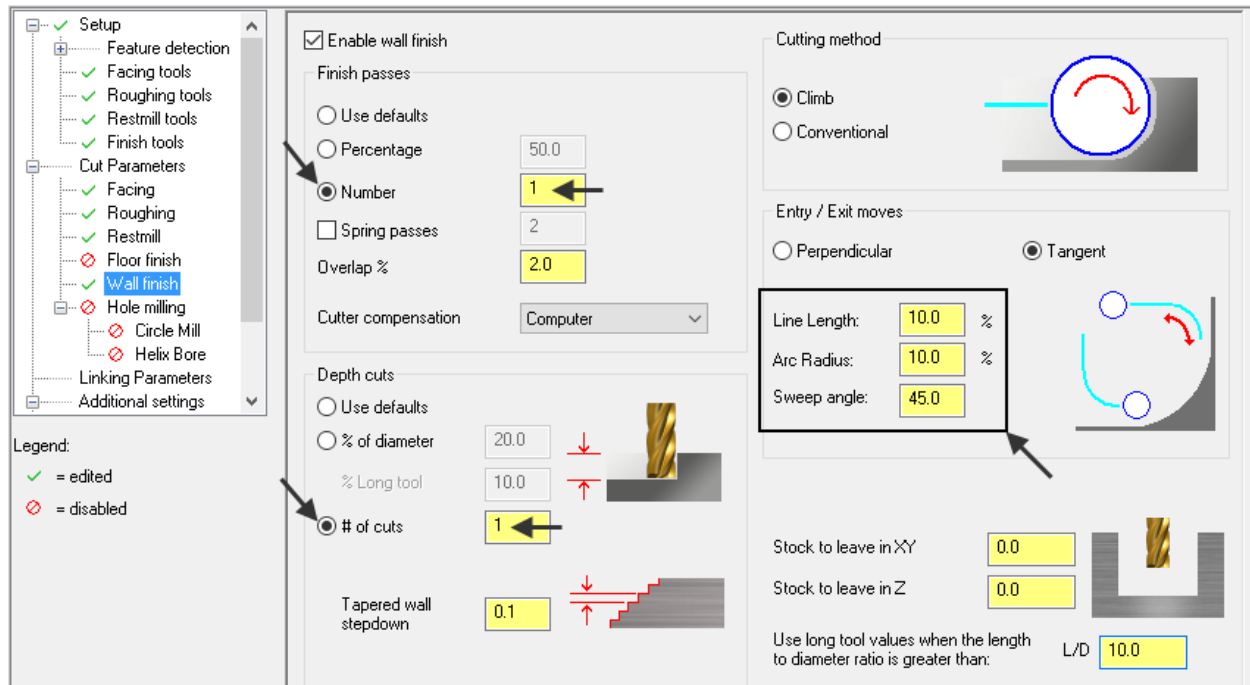


38. Ensure **Floor Finish** is **disabled**. If not, select **Floor Finish** from the Tree Control dialog box and uncheck **Enable floor finish** ☐ Enable floor finish in order to disable floor finish.

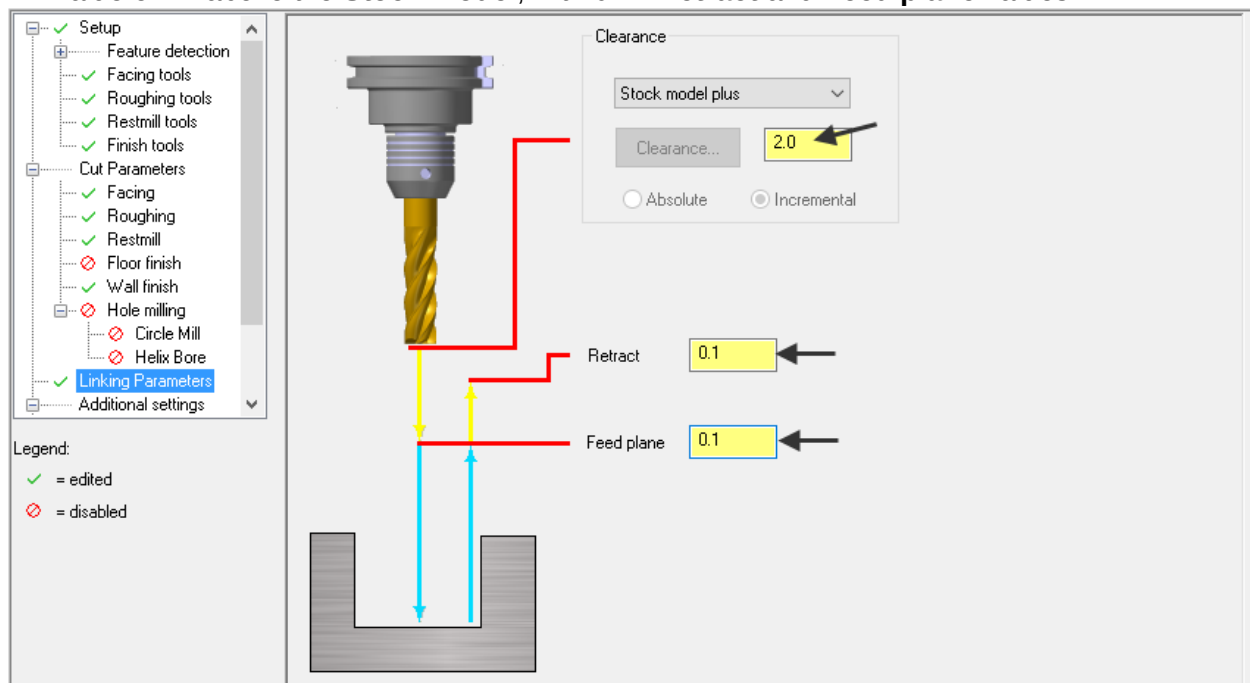


Floor finish page Select Enable floor finish to add finish passes for the floor of each pocket feature in the FBM Mill operation. When this option is deselected, Mastercam creates only roughing and restmill operations for pocket floors. Finish toolpaths remove material left from the restmill operations to the Stock to leave depth you define in this page.

39. 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:



40. 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:



41. Expand **Additional settings** and set **Coolant Flood** to **On**.
 42. Select the OK button to process all milling operations for this part.
 43. When prompted to **Enter new NC name** ensure **MILL-LESSON-FBM-1** is displayed and then select the OK button .

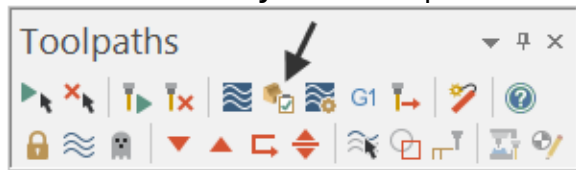
TASK 6: VERIFY THE TOOLPATH

- 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. Right mouse click in the graphics area and click on **Isometric**.
2. In the Toolpaths Manager pick all the operations to verify by picking the **Select All** icon



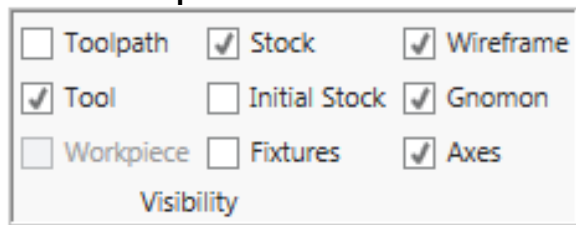
3. Select the **Verify selected** operations button shown below:



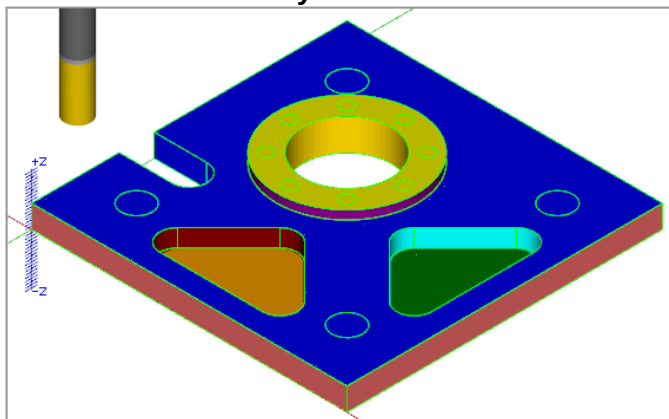
4. **Maximize** the Backplot/Verify window if required.
5. Activate the **Both** option in the **Toolpath** section of the **Verify** tab.
6. Now select the **Home** Tab.



7. Activate the options shown below in the **Visibility** section of the Home tab. **Initial Stock** and **Toolpath** not activated.



8. Activate the **Color Loop** to change the color of the tools for the verified part.
9. In the lower right corner of the screen now set the run **Speed** to midway along the slider bar.
10. Now select the **Play Simulation** button to review the toolpaths.



11. Select the Close button  in the top right hand corner to exit Verify.

TASK 7:

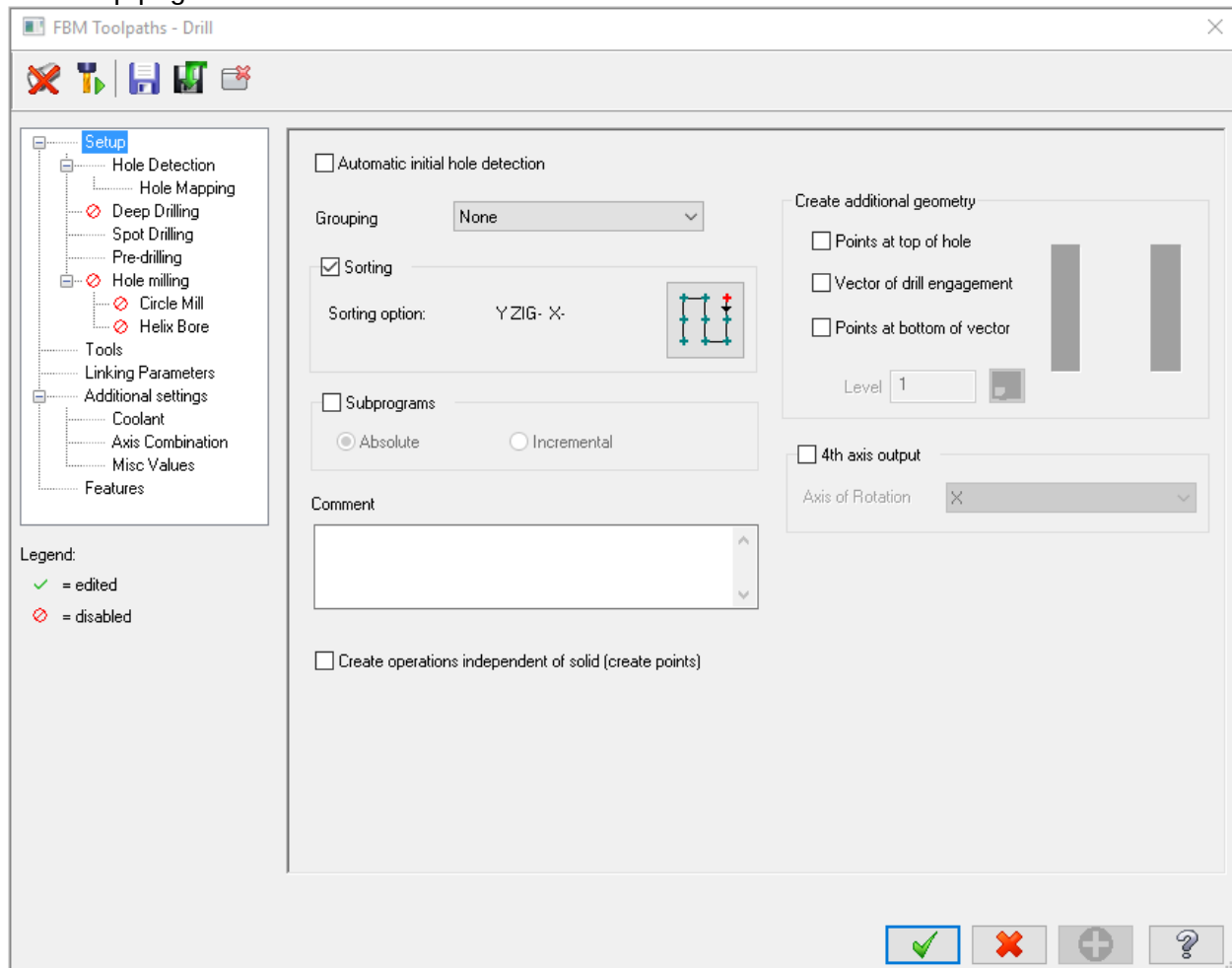
RUN FBM DRILL FOR ALL HOLE MAKING OPERATIONS

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

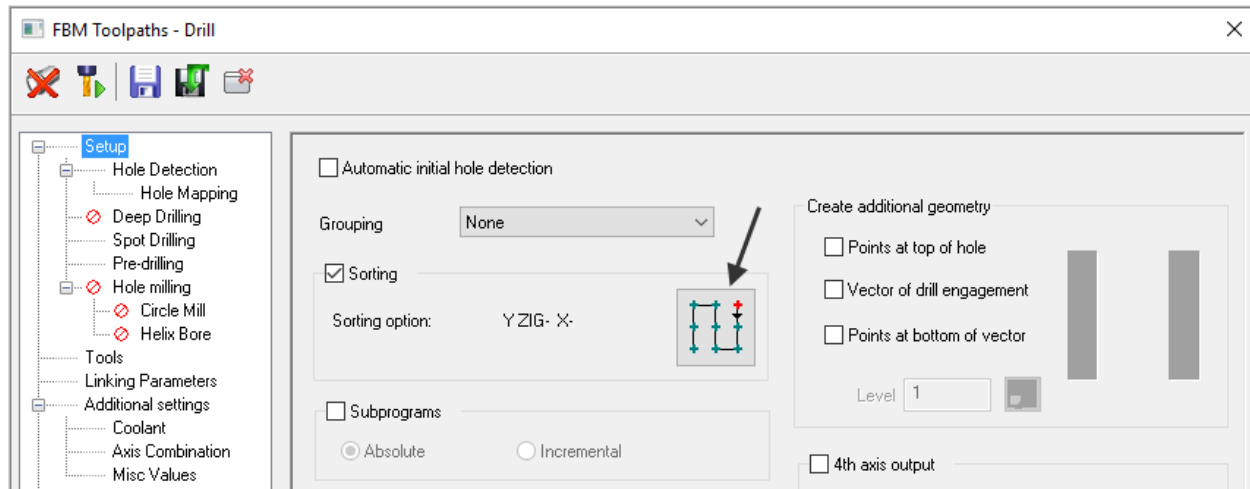
1. Right mouse click in the graphics area and click on **Top**.
2. Select the **TOOLPATHS** tab at the top right side of the screen.
3. Select **FBM Drill** in the **2D Holmaking** toolpath section.

FBM Drill is a 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 holmaking 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.

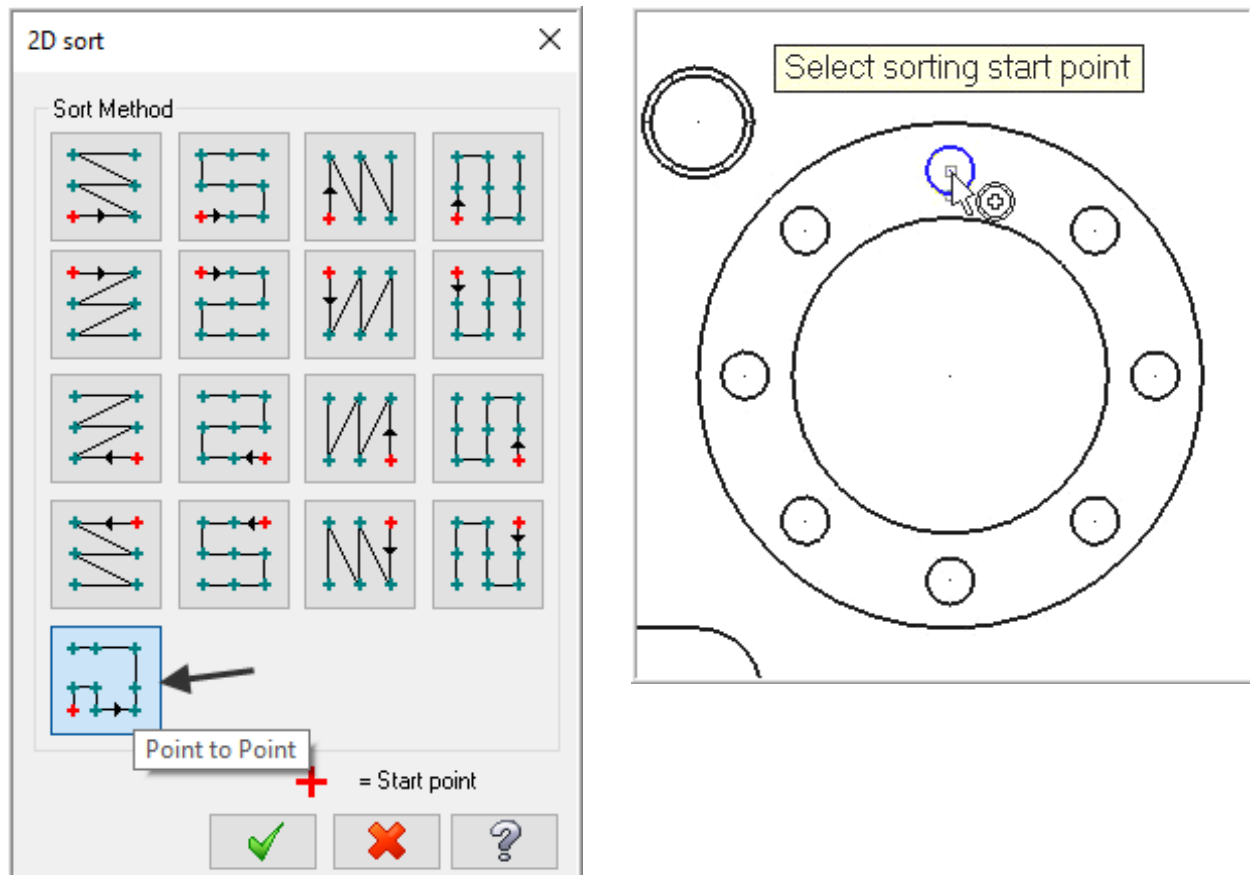
4. On the screen you will now see the **FBM Toolpaths – Drill** dialog box, starting with the Setup page selected in the Tree Control menu down the left hand side:



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

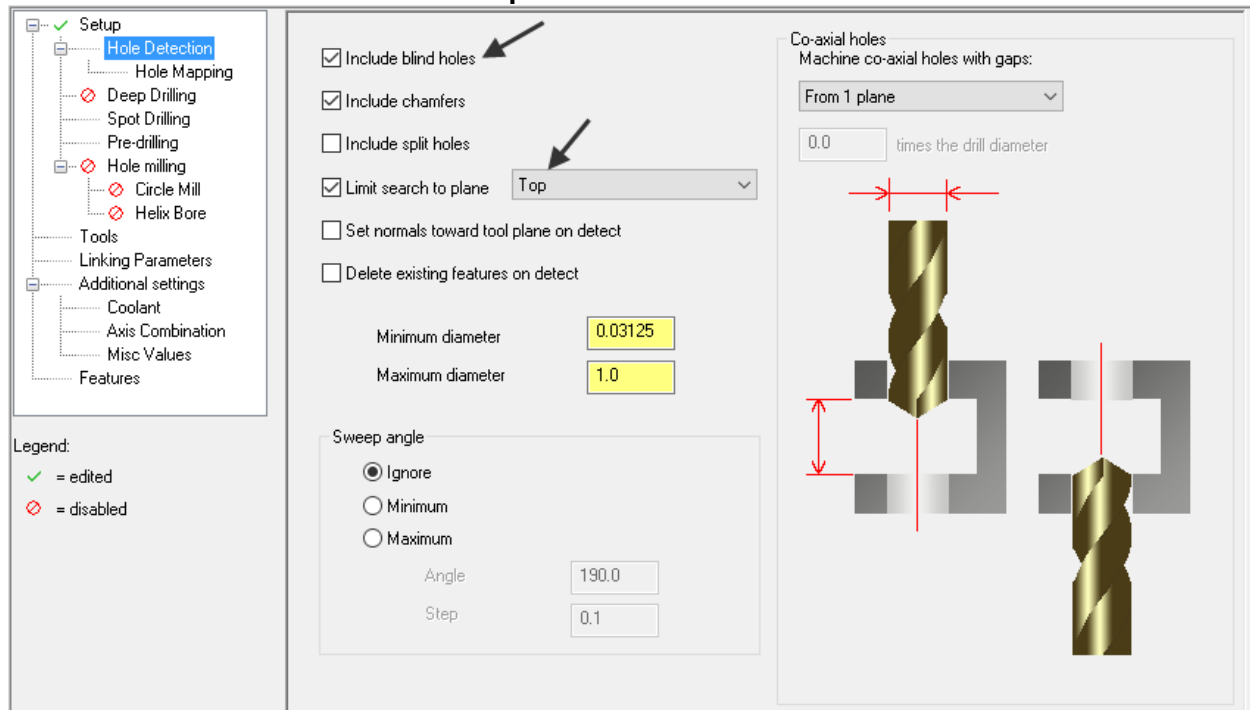


- Select the **Point to Point 2D Sort Method**.

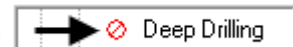


- Select the OK button  in the **2D sort** page.
- The prompt now changes to **Select sorting start point**. Move the mouse and position the cursor over the point at the center of circle shown above right; you will notice this icon  appears as you move over the point at the center of this circle. This cross denotes you are snapping to a circle center point. Use the left button of your mouse to pick this point.

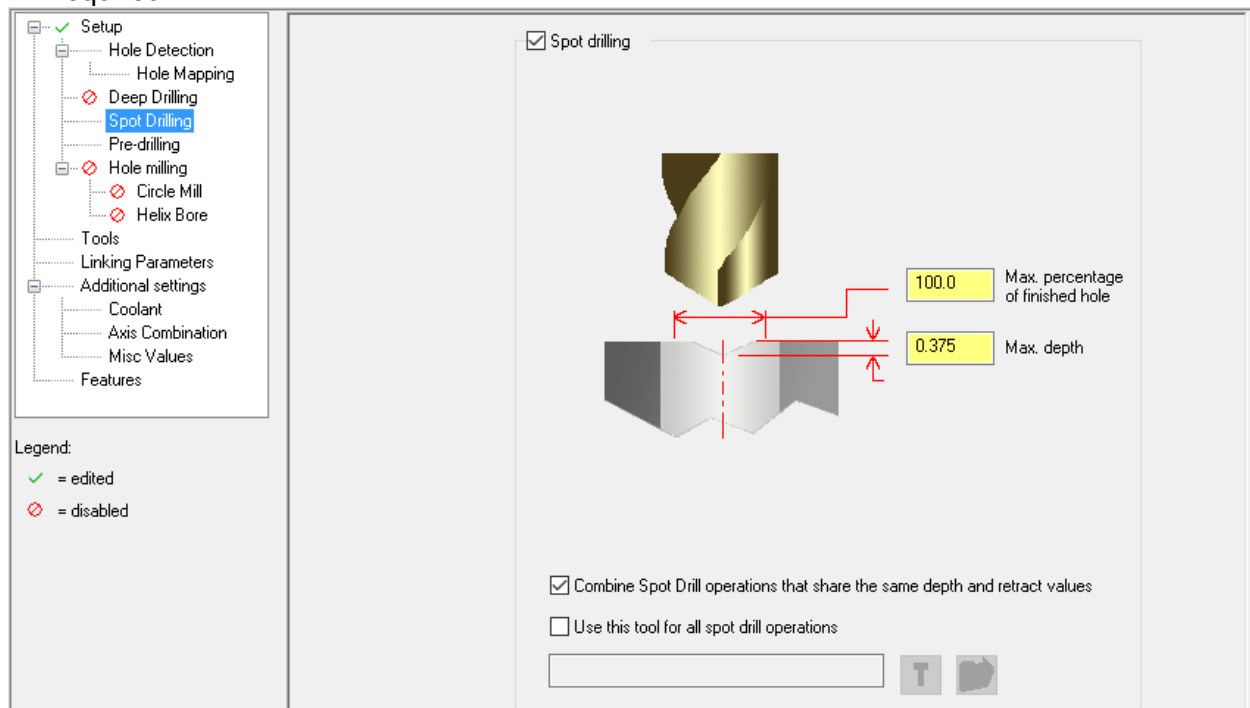
9. Select **Hole Detection** from the Tree Control dialog box and set parameters to **Include blind hole** and to **Limit search to plane TOP** as follows:



10. Ensure **Deep Drilling** is disabled from the Tree Control dialog box



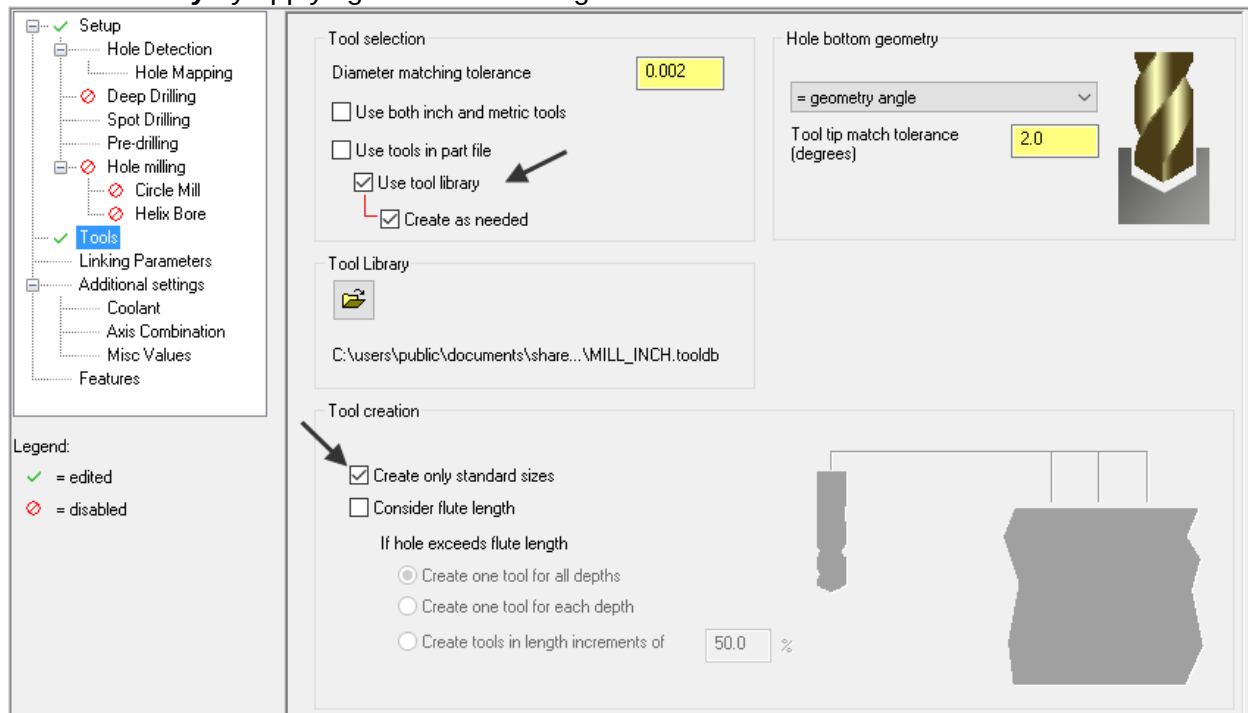
11. Select **Spot Drilling** from the Tree Control dialog box and make changes to this page if required.



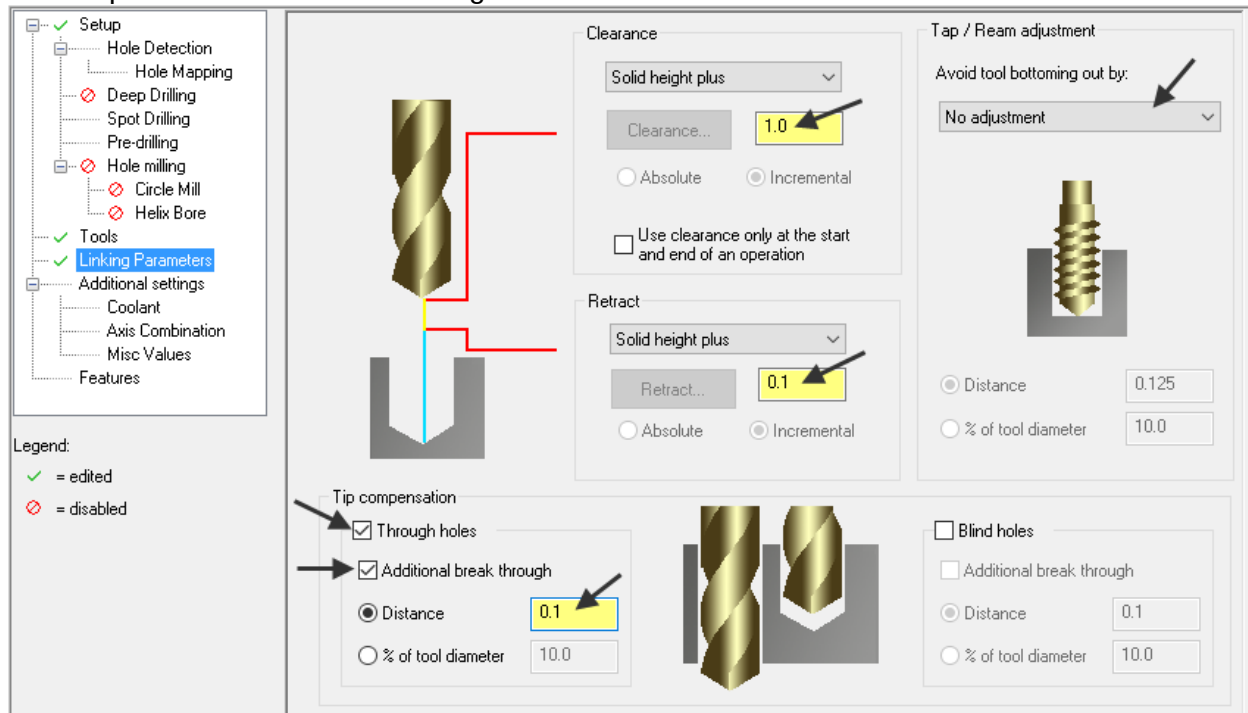
12. Select **Pre-drilling** from the Tree Control dialog box and make changes to this page if required.



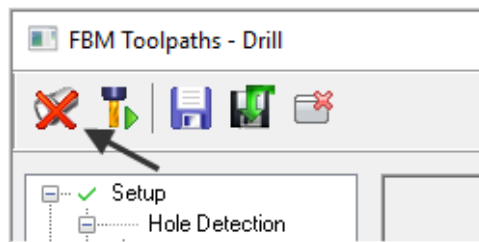
13. Select **Tools** from the Tree Control dialog box. You will allow FBM Drill to automatically **Use tool library** by applying the follow settings:



14. Select **Linking Parameters** from the Tree Control dialog box. This dialog box will be used to set our **Clearance** at **Solid height plus 1.0**, **Retract** at **Solid height plus 0.1**, an **Additional break through** of **0.1**, and **No adjustment** for the **Tap / Ream adjustment** for this part which features all through holes.

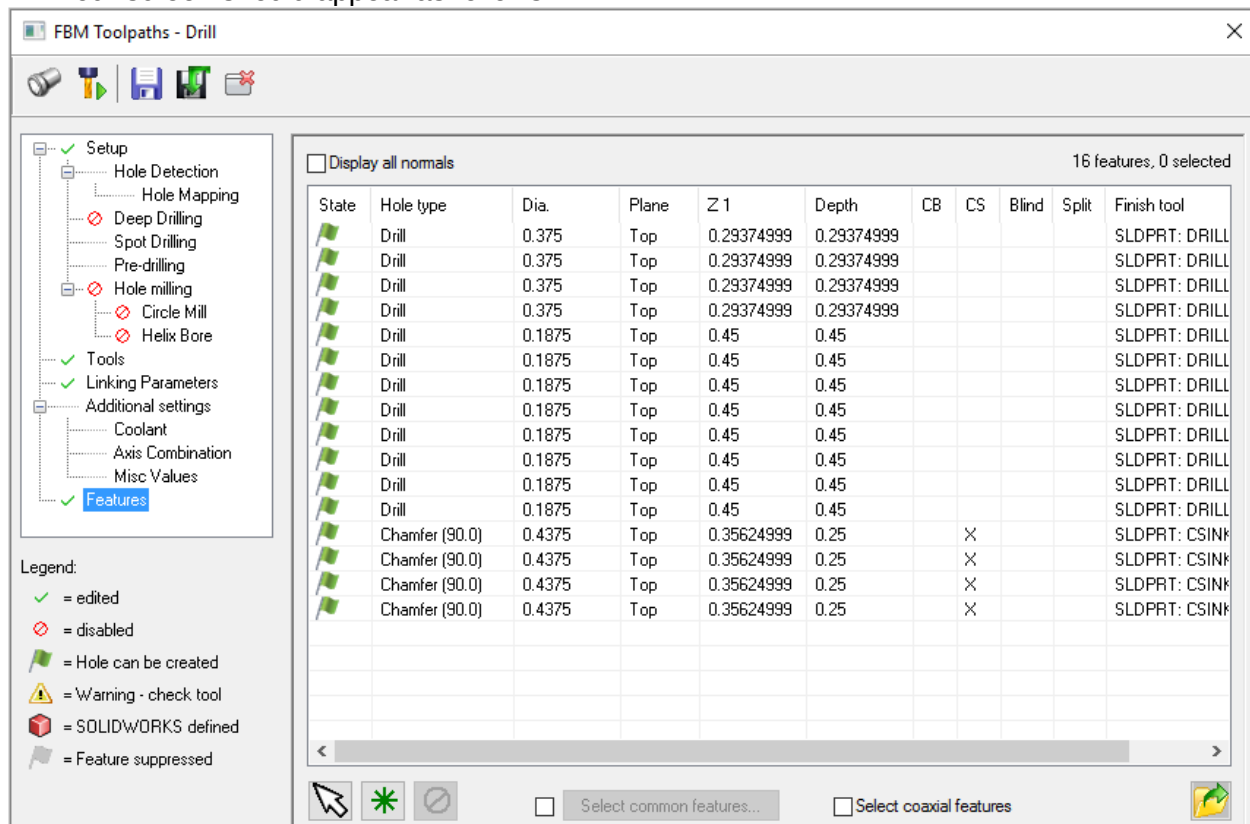


15. Expand **Additional settings** and set **Coolant Flood** to **On**.
 16. Click the **Detect...** button at the top of the FBM Toolpaths – Drill dialog box.



Mastercam Training Guide

17. Your screen should appear as follows:



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

State	Hole type	Dia.	Plane	Z 1	Depth	CB	CS	Blind	Split	Finish tool
	Drill	0.375	TOP	0.29374999	0.29374999					SLDPRT: DRILL, 3/8
	Drill	0.375	TOP	0.29374999	0.29374999					SLDPRT: DRILL, 3/8
	Drill	0.375	TOP	0.29374999	0.29374999					SLDPRT: DRILL, 3/8
	Drill	0.375	TOP	0.29374999	0.29374999					SLDPRT: DRILL, 3/8
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2

19. Press and hold the **Ctrl** key on your keyboard. Left click on the next three 0.375 Dia. Holes in the list. With **Ctrl** key still held, you can select and unselect any mistakes. When the four correct holes are selected, release the **Ctrl** key.
20. Right click on anywhere in Hole type column of these four selected holes. In the **Hole type** column their value currently reads **Drill**. Hover the cursor over **Hole type** and select **Tap RH** from the list.

State	Hole type	Dia.	Plane	Z 1	Depth	CB	CS	Blind	Split	Finish tool
	Drill	0.375	TOP	0.29374999	0.29374999					SLDPRT: DRILL, 3/8
	Drill									SLDPRT: DRILL, 3/8
	Drill									SLDPRT: DRILL, 3/8
	Drill									SLDPRT: DRILL, 3/8
	Drill									SLDPRT: DRILL, 3/1
	Drill									SLDPRT: DRILL, 3/1
	Drill									SLDPRT: DRILL, 3/1
	Drill									SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.4						SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.4						SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Drill	0.1875	TOP	0.45	0.45					SLDPRT: DRILL, 3/1
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2
	Chamfer (90.0)	0.4375	TOP	0.35624999	0.25		X			SLDPRT: CSINK, 1/2

21. 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.

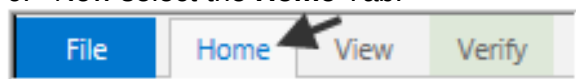
Many users take the time to create a custom tool library with only their tools, with desired feeds and speeds set in the library.

TASK 8: VERIFY THE TOOLPATH

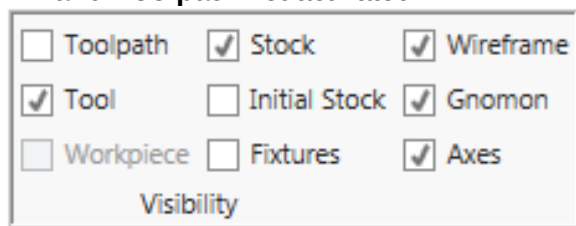
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
 - This allows you to identify and correct program errors before they reach the shop floor.
1. Right mouse click in the graphics area and click on **Isometric**.
 2. In the Toolpaths Manager pick all the operations to verify by picking the **Select All** icon.
 3. Select the **Verify selected** operations button shown below:



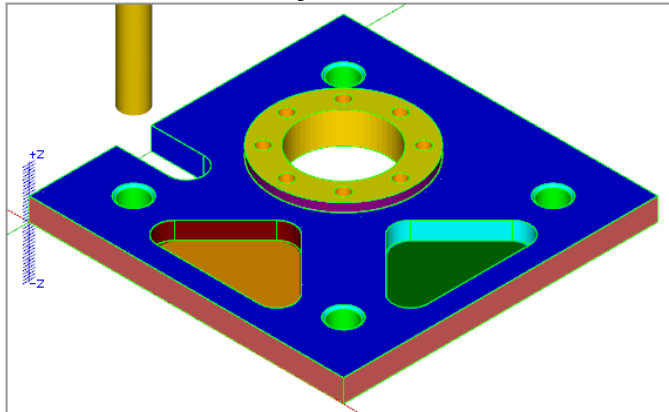
4. **Maximize** the Backplot/Verify window if required.
5. Activate the **Both** option in the **Toolpath** section of the **Verify** tab.
6. Now select the **Home** Tab.



7. Activate the options shown below in the **Visibility** section of the Home tab. **Initial Stock** and **Toolpath** not activated.



8. Activate the **Color Loop** to change the color of the tools for the verified part.
9. At the top of the screen select the **View** tab, the **Isometric** icon and then select **Fit**.
10. In the lower right corner of the screen now set the run **Speed** to midway along the slider bar.
11. Now select the **Play Simulation** button to review the toolpaths.

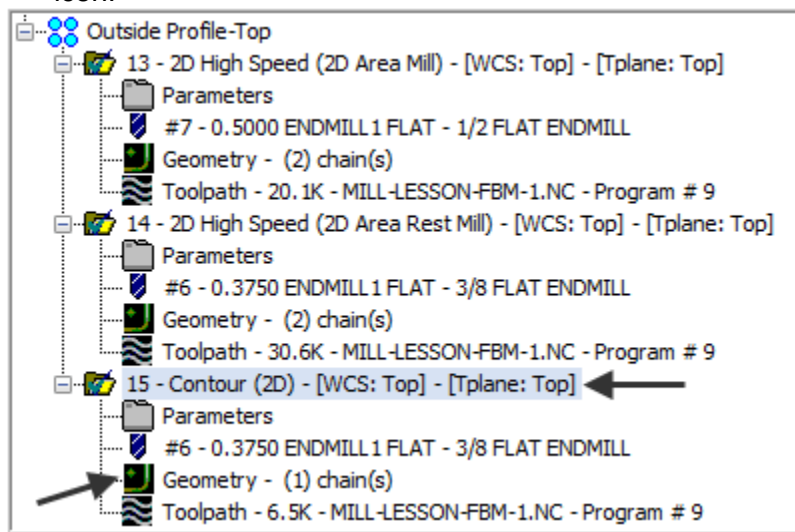


12. Select the Close button in the top right hand corner to exit Verify.

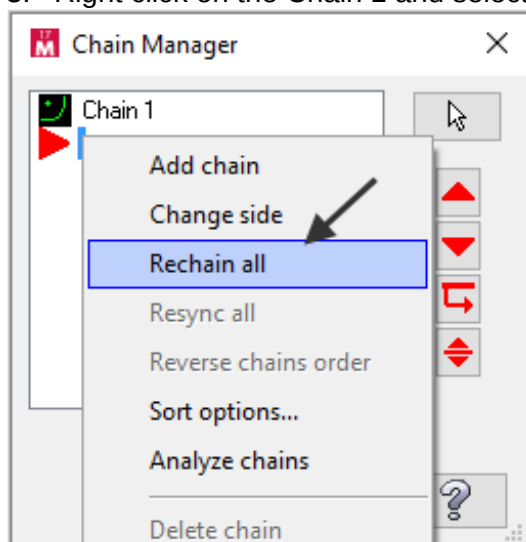
TASK 9: REMOVE THE OUTSIDE FINISH PASS

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. In the **View** tab click on **Wireframe** in the **Appearance** section.
2. Use the **Toolpaths 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.



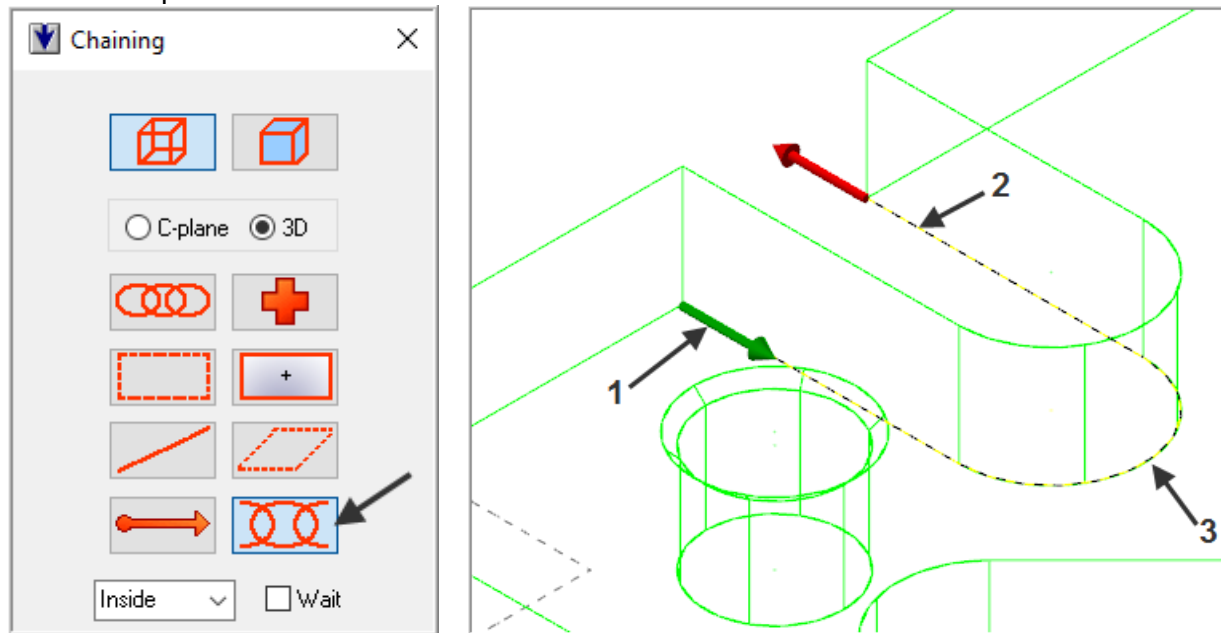
3. Right click on the Chain 1 and select **Rechain all**



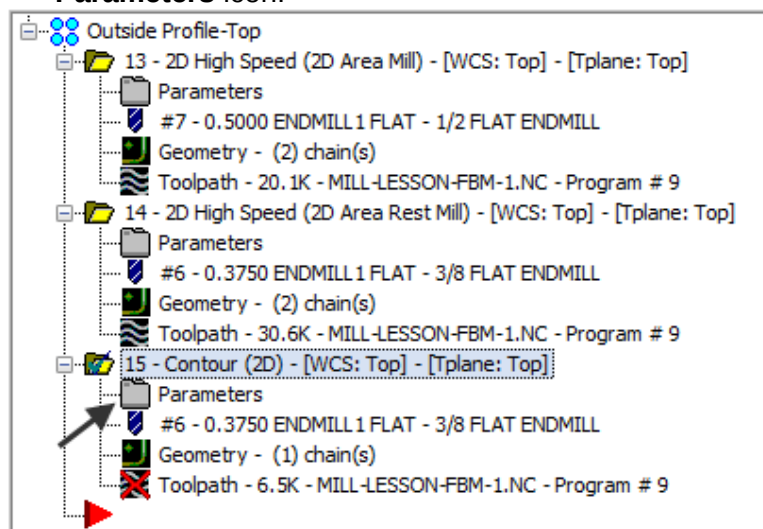
4. Right mouse click in the graphics area and click on **Isometric**.
5. Right mouse click in the graphics area and click on the **Fit** icon .

Mastercam Training Guide

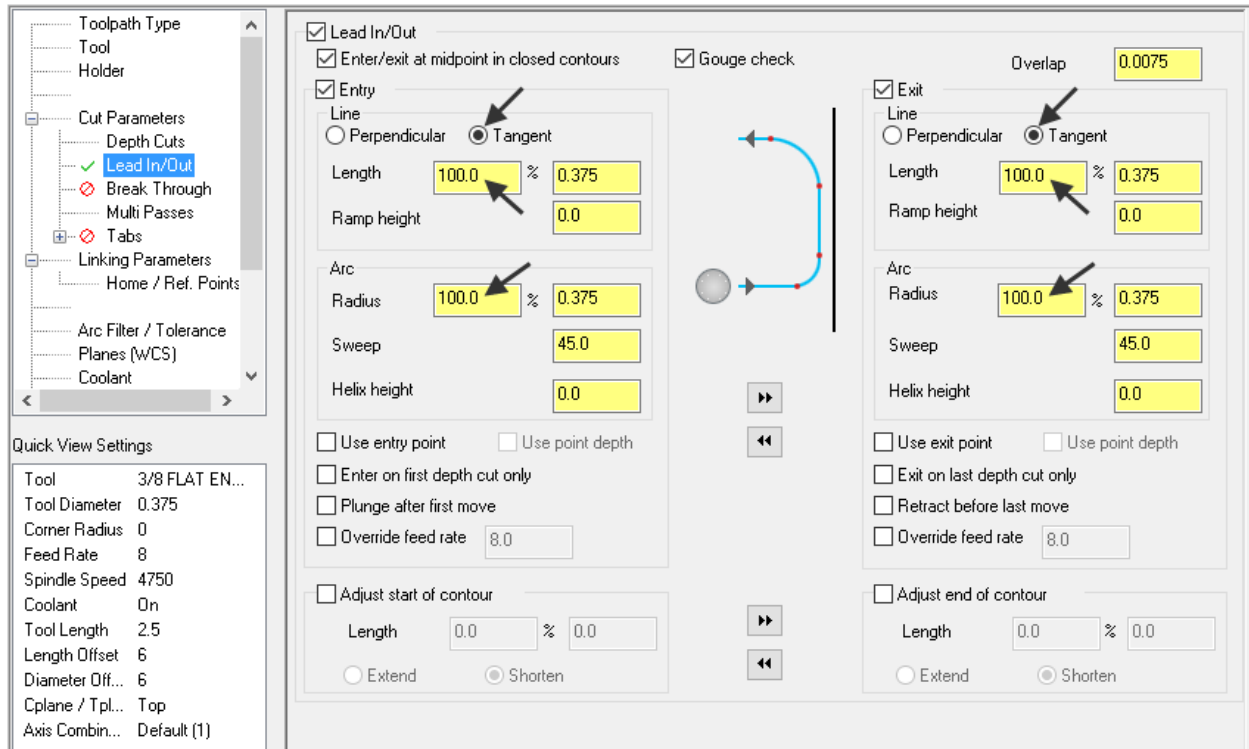
6. Select **Partial** chaining. When prompted for **Select the first entity**, pick line **1**. Next prompt **Select the last entity** pick line **2**. Final prompt **Branch point reached. Select next branch** pick the slot radius **3**.



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



10. Select the **Lead In/Out** from this list on the left and make changes to this page as shown below:



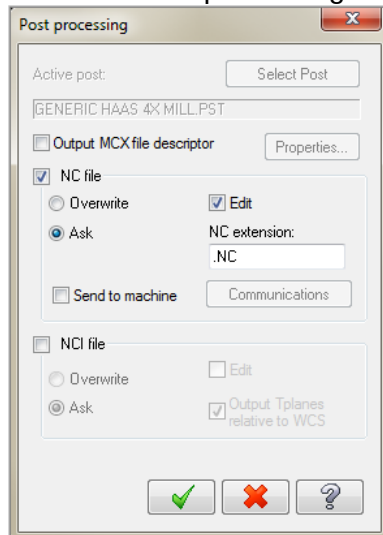
11. Select the OK button  to complete Contour operation.
12. Select **Regenerate all dirty operations** button  to refresh this modified Operation.

TASK 10: SAVE THE UPDATED MASTERCAM FILE

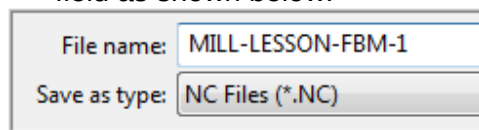
1. Select the **Save** icon from the **Quick Access** toolbar at the top left of the screen.

TASK 11: POST AND CREATE THE CNC CODE FILE

1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpaths Manager.
 2. Select the **Post selected operations** button from the Toolpaths Manager.
- Please Note:** If you cannot see **G1** click on the right pane of the Toolpath manger window and expand the window to the right.
3. In the Post processing window, make the necessary changes as shown below:



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



6. Select the Save button.
 7. The CNC code file opens up in the default editor:
 8. Select the  in the top right corner to exit the CNC editor.
- This completes Mill-Lesson-FBM-1.

