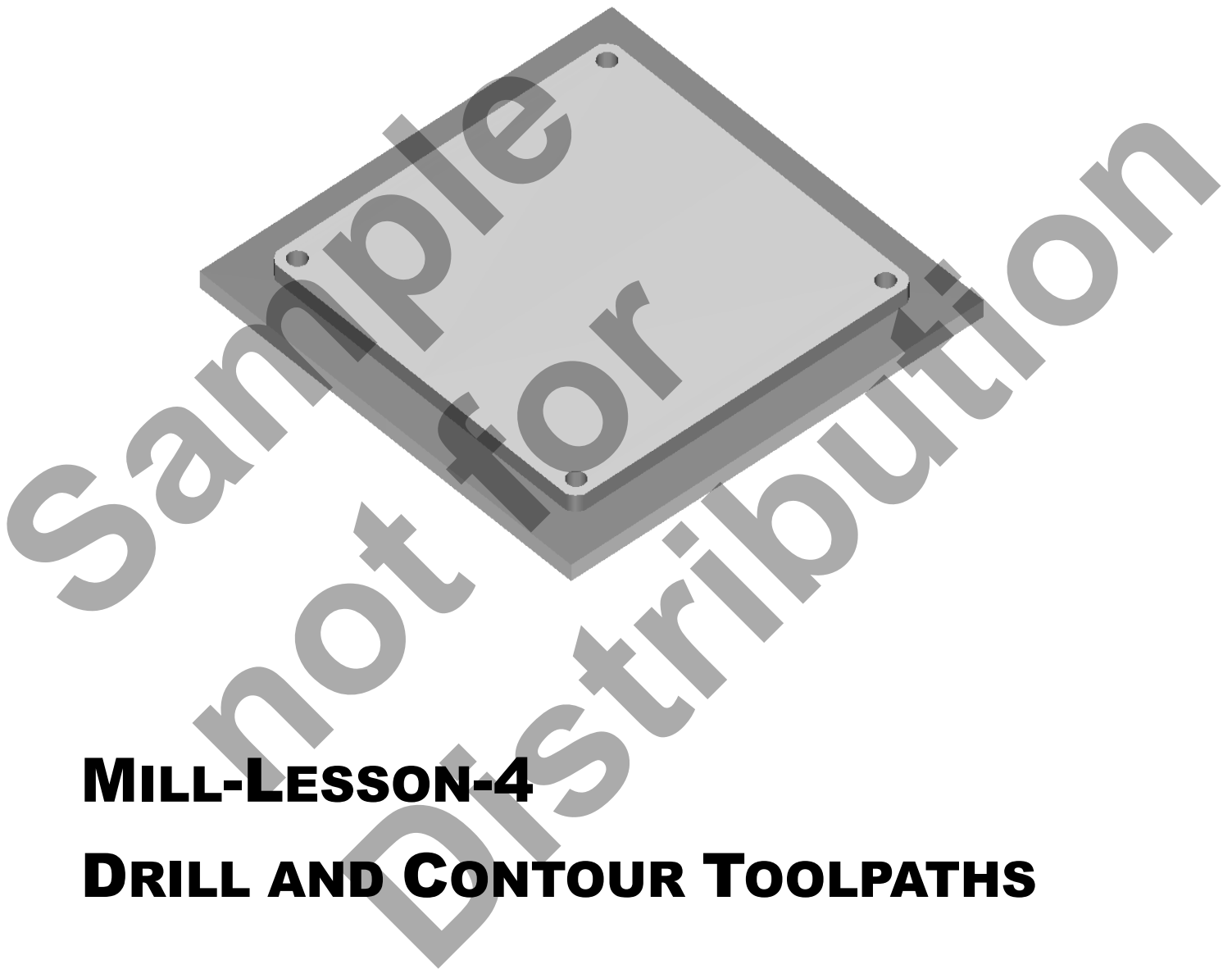


Mastercam X⁷

TRAINING

GUIDE



MILL-LESSON-4

DRILL AND CONTOUR TOOLPATHS

camInstructor

Objectives

You will create the geometry for Mill-Lesson-4, and then generate a toolpath to machine the part on a CNC vertical milling machine. This lesson covers the following topics:

➤ **Create a 2-dimensional drawing by:**

Creating rectangles.
Creating arcs.

➤ **Establish Stock Setup settings:**

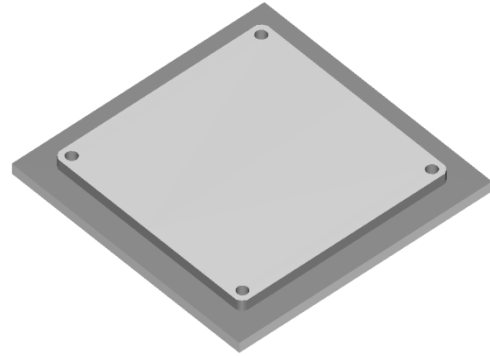
Stock size.
Material for the part.
Feed calculation.

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

Contour toolpath.
Drill toolpath.

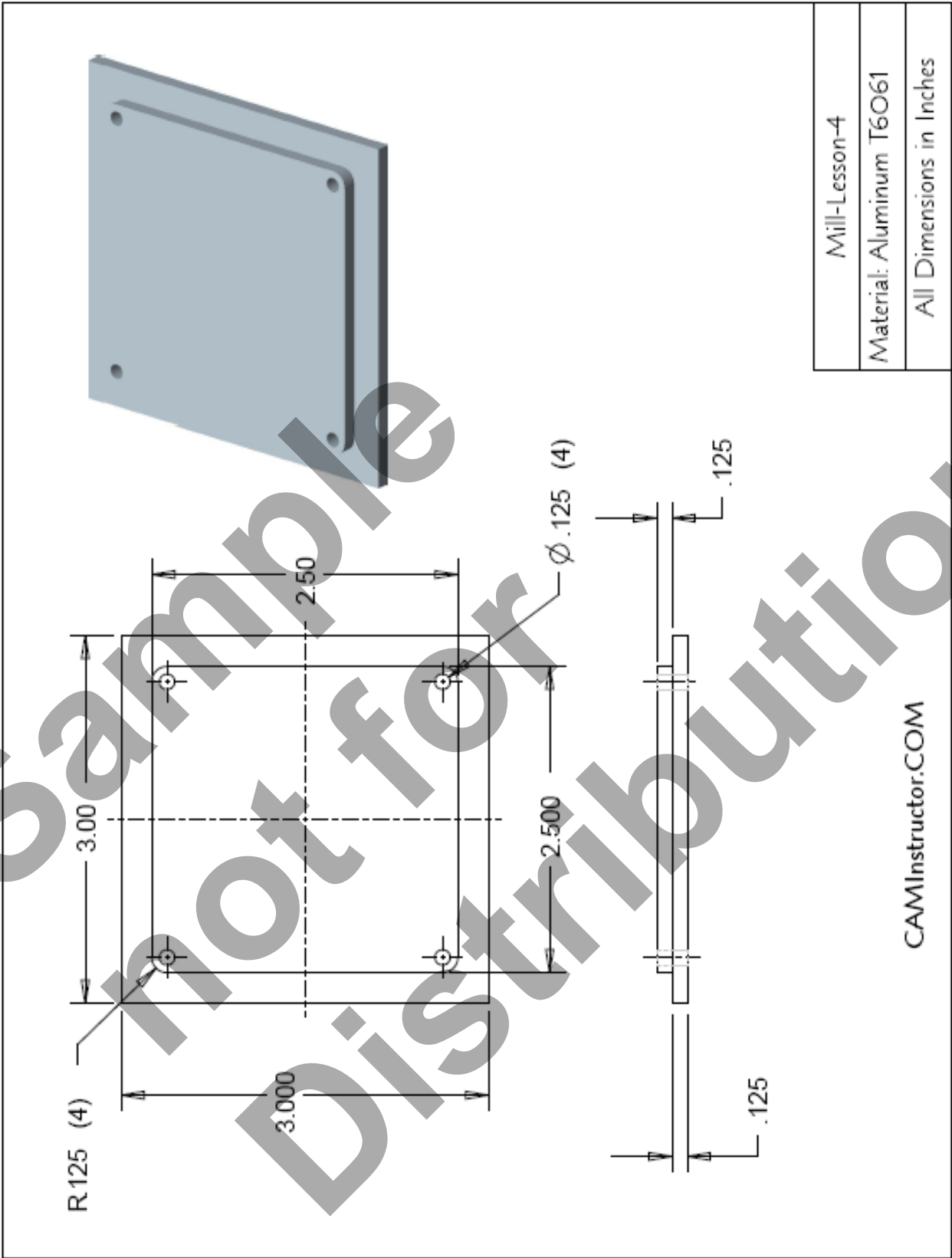
➤ **Inspect the toolpath using Mastercam's Verify and Backplot by:**

Launching the Verify function to machine the part on the screen.
Using Backplot to identify the correctness of the toolpaths.
Generating the NC- code.



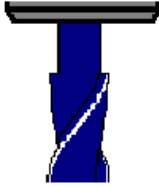

Sample
not for
Distribution

MILL-LESSON-4 DRAWING



TOOL LIST

- ☞ Two cutters will be used to create this part.
- ☞ The .125" diameter two flute center cutting flat end mill will be used to drill the four .125" diameter holes.
- ☞ The .5" diameter two flute flat end mill will be used to machine the contour.

Operation List of MILL-LESSON-4.MCX-5	
Proj./Part No. : 0	Date : 05/03/10
Drawing No. : 1	Customer : -
Prog. No. : 4	Programmer : 1
<hr/>	
	Tool type : 0.125 Endmill Flat 1/8 FLAT ENDMILL
	Manufact.code :
	Chuck :
	Tool Number : 1 Operation time: 0h 0m 11s Feedrate : 7
	Diameter : 0.125 RPM : 5000 Plunge feed r.: 6.1609
	Corner radius : 0 Tip angle : 0 Diam. offset : 1
	Flute length : 0.375 Material : ALUMINUM ... Length offset : 1
	Comment : Canned Cycle
	Tool type : 0.5 Endmill Flat 1/2 FLAT ENDMILL
	Manufact.code :
	Chuck :
	Tool Number : 2 Operation time: 0h 4m 5s Feedrate : 10
	Diameter : 0.5 RPM : 4500 Plunge feed r.: 10
	Corner radius : 0 Tip angle : 0 Diam. offset : 2
	Flute length : 1 Material : ALUMINUM ... Length offset : 2
	Comment : Contour

MILL-LESSON-4 - THE PROCESS

Geometry Creation

- TASK 1:** Set the environment
- TASK 2:** Create a rectangle 3" x 3" – the center is at X0 Y0
- TASK 3:** Create the inside entities
- TASK 4:** Create the four .125" fillet radii
- TASK 5:** Save the drawing

Toolpath Creation

- TASK 6:** Define the rough stock using stock setup
- TASK 7:** Drill the 4 x .125" diameter holes
- TASK 8:** Machine the contour
- TASK 9:** Backplot the toolpath
- TASK 10:** Modify the contour toolpath to add roughing cuts and a finish pass
- TASK 11:** Verify the toolpath
- TASK 12:** Save the updated Mastercam file
- TASK 13:** Post and create the CNC code file

Geometry Creation

TASK 1:

SETTING THE ENVIRONMENT

Before starting the geometry creation 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 2:

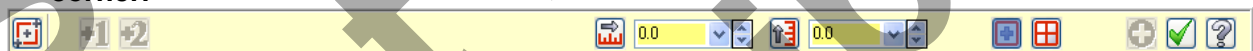
CREATE A RECTANGLE 3" X 3" – THE CENTRE IS AT X0 Y0.

☞ This task explains how to create the 3" square. These four lines could be created in many different ways, this is just one option.

1. Select from the pull down menu **Create>Rectangle...**



2. The Create Rectangle ribbon bar appears and you are prompted to **Select position of first corner.**



3. On the ribbon bar click in the space for **Width** and enter a value of **3.0**, hit the tab key and you will be moved over to the Height section.



4. In the **Height** section enter a value of **3.0** and then hit Enter.
5. Now select the **Anchor to Center option** - this option is shown above. To activate the **Anchor to Center** option the icon is pressed down as shown above.

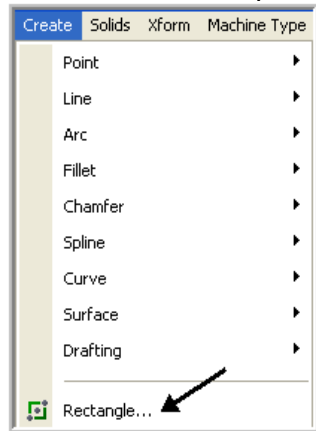
Anchor to center: Sets the base point of the rectangle to the center point, and draws the rectangle outward from the center.

6. Move the cursor to the center of the Grid and snap to the Origin for the base point.
7. Click on the OK icon to complete this feature.
8. Select the Screen Fit icon found at the top of the screen to fit the part to the screen .

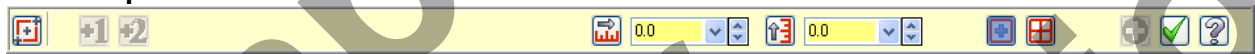
TASK 3: CREATE THE INSIDE ENTITIES

- In this task you will create the four inside lines. These lines form a 2.5" square.
- You will use **Create Rectangle** again to construct these four inside lines.

1. Select from the pull down menu **Create>Rectangle...**



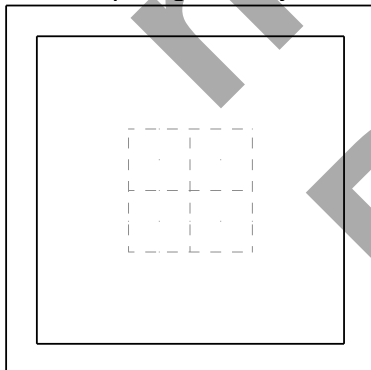
2. The Create Rectangle ribbon bar appears and you are prompted to **“Select position of base point”**.



3. On the ribbon bar click in the space for **Width** and enter a value of **2.5**, hit the tab key and you will be moved over to the Height section.



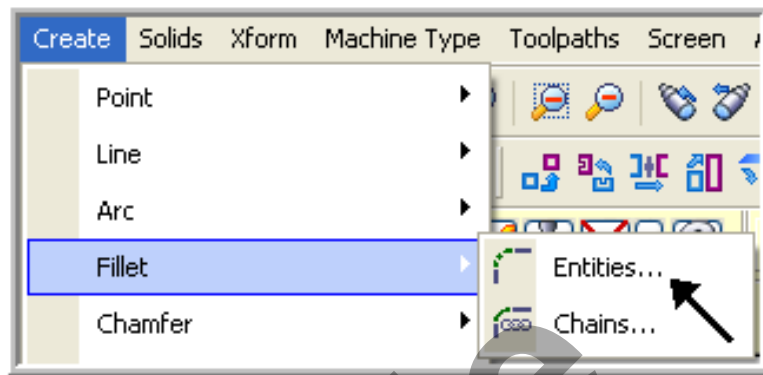
4. In the **Height** section enter a value of **2.5** and then hit Enter.
5. Select the **Anchor to Center option** - this option is shown above. To activate the **Anchor to Center** option the icon is pressed down as shown above.
6. Move the cursor to the center of the Grid and snap to the origin for the base point.
7. Click on the OK icon to complete this feature.
8. Your part geometry should look as below: 3.0" and 2.5" square.



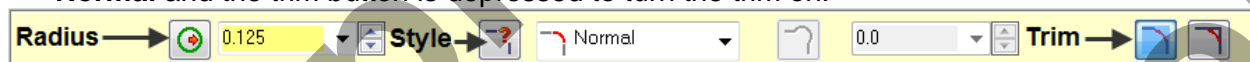
**TASK 4:
CREATE THE FOUR .125” FILLET RADII**

➤ In this task you will create the four .125 fillet radii on the corners of the 2.5” square you have just created.

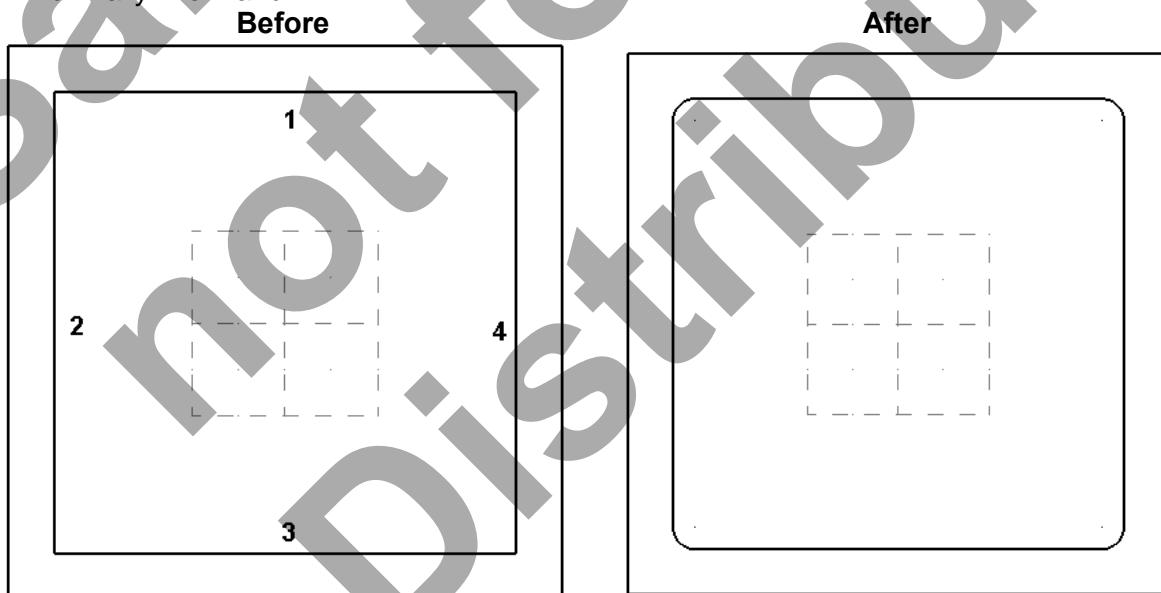
1. Select **Create>Fillet>Entities...**



2. On the Fillet ribbon bar enter **.125 for the radius**. Ensure the **Style** of radius is set to **Normal** and the trim button is depressed to turn the trim on.



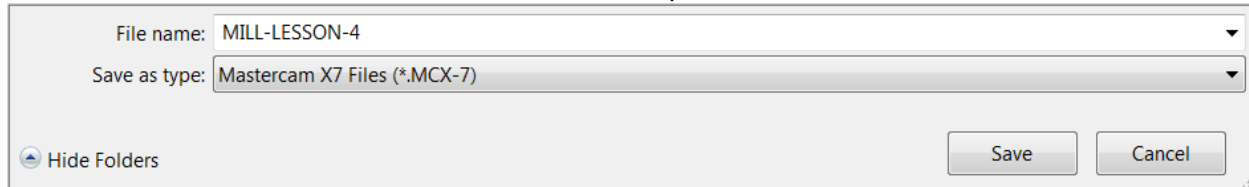
3. When prompted to “**Select an entity**”, select Line 1 and 2 as shown below. The fillet radius appears at the corner of line 1 and 2.
4. To complete the remaining three fillet radii, select:
5. Line 2 and 3.
6. Line 3 and 4.
7. And finally line 4 and 1.



8. Click on the OK icon to complete this feature.
9. Your completed part geometry should look like the figure on the right above.

**TASK 5:
SAVE THE DRAWING**

1. Select **File**.
2. Select **Save As...**
3. In the “**File name**” box, type “**Mill-Lesson-4**”.
4. Save to an appropriate location.
5. Select the **Save** button to save the file and complete this function.



Sample
not for
Distribution

Toolpath Creation

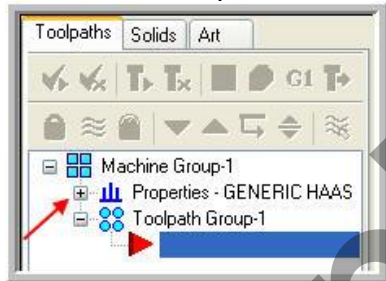
TASK 6:

DEFINING THE ROUGH STOCK USING STOCK SETUP

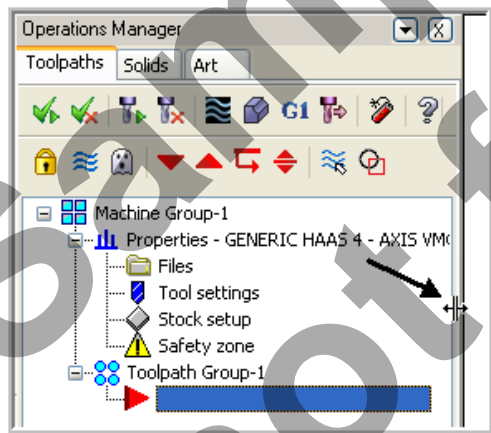
1. For a better view of the part use the toolbar at the top of the screen to change the graphics view to Isometric



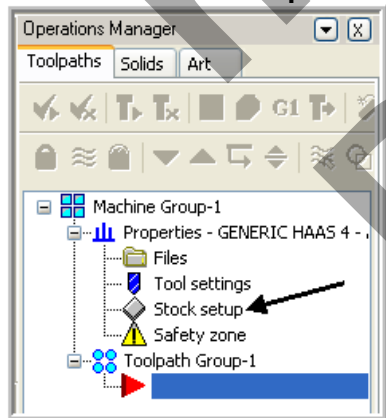
2. Now select the Fit to screen icon 
3. Select the plus in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will Show/hide Operations Manager pane.



4. You may need to extend the toolpaths manager window, if so click on the right hand pane, hold the mouse button down and drag to the right.

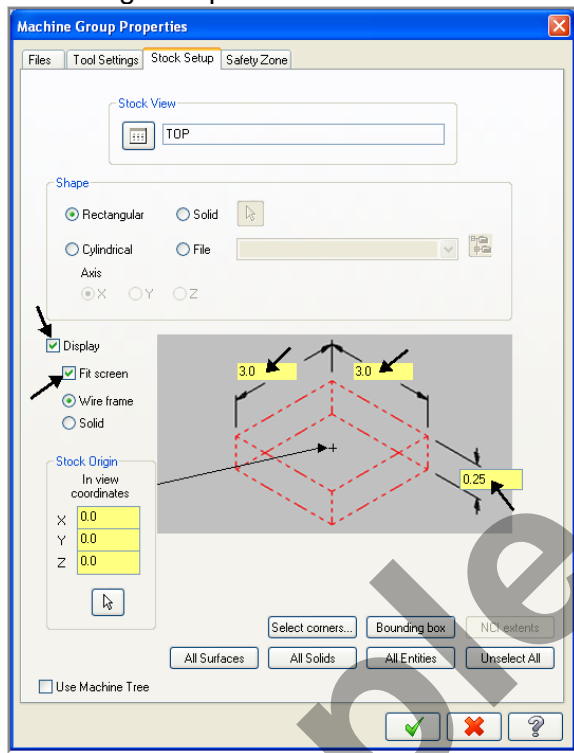


5. Select **Stock setup** in the Toolpath manager window.

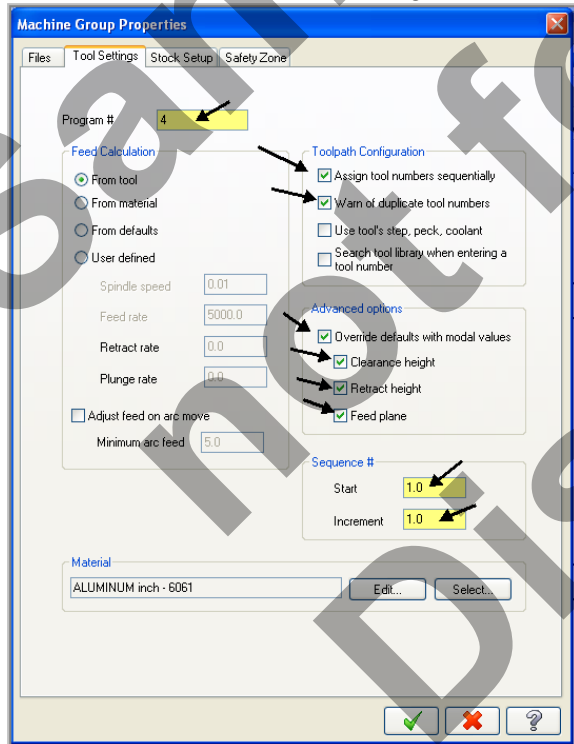


Mastercam Training Guide

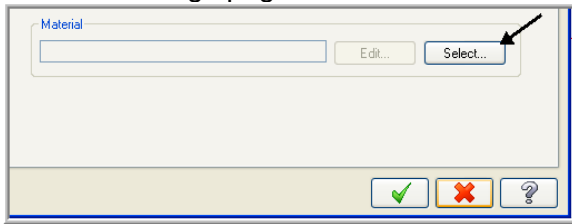
6. Change the parameters to match the Stock Setup screenshot below:



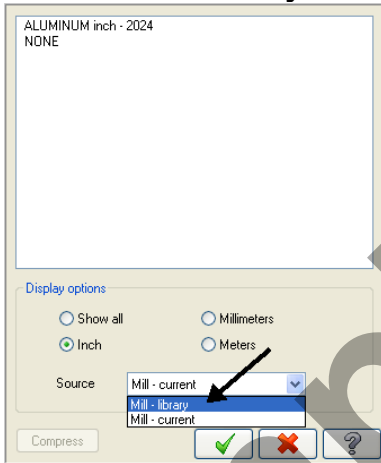
7. Select the Tool Settings tab and change the parameters to match the **Tool Settings** screenshot below. To change the **Material** type follow the next set of instructions.



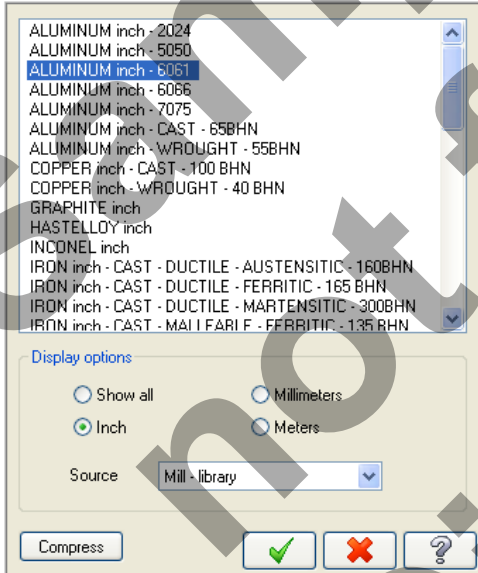
8. To change the **Material** type to **Aluminium 6061** pick the **Select** button at the bottom of the Tool Settings page.



9. Select **Mill - library** at the Material List dialog box open the Source drop down list.



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



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

TASK 7:

DRILL THE 4 X .125" DIAMETER HOLES


- In this task you will drill the four .125" diameter holes through the part with a centre cutting two flute end mill that is .125 " diameter.
- As the four holes will be drilled at the centre of the .125" fillet radii you need to create points at the centre of these arcs, when creating the drilling toolpath you can snap to the centre of the arcs.

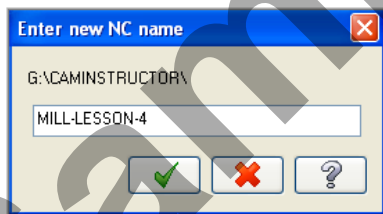
1. Change the graphics view to a **Top View** by using the toolbar at the top of the screen.



2. From the menu bar select **Toolpaths>Drill...**



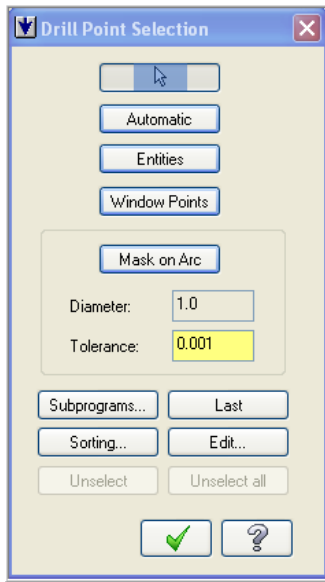
3. When prompted to "**Enter new NC name**" Input **Mill-Lesson-4** as shown below and then select the OK button .



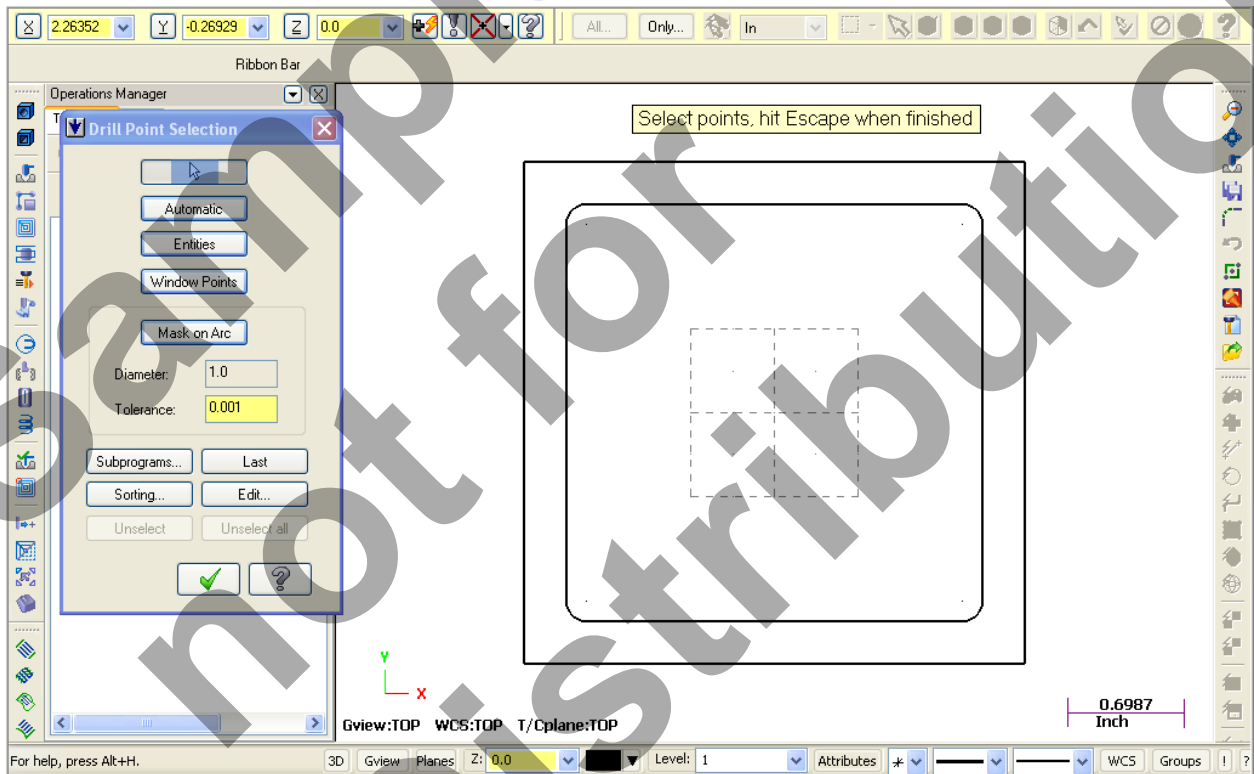
4. Now you are prompted to:

Select points, hit Escape when finished

5. The **Drill Point Selection** dialog box appears:

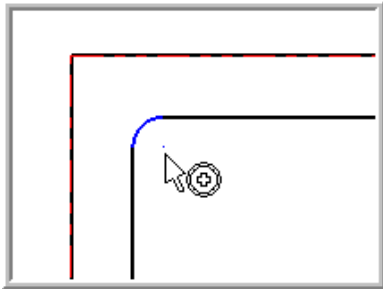


6. Your screen will look similar to the screenshot below:

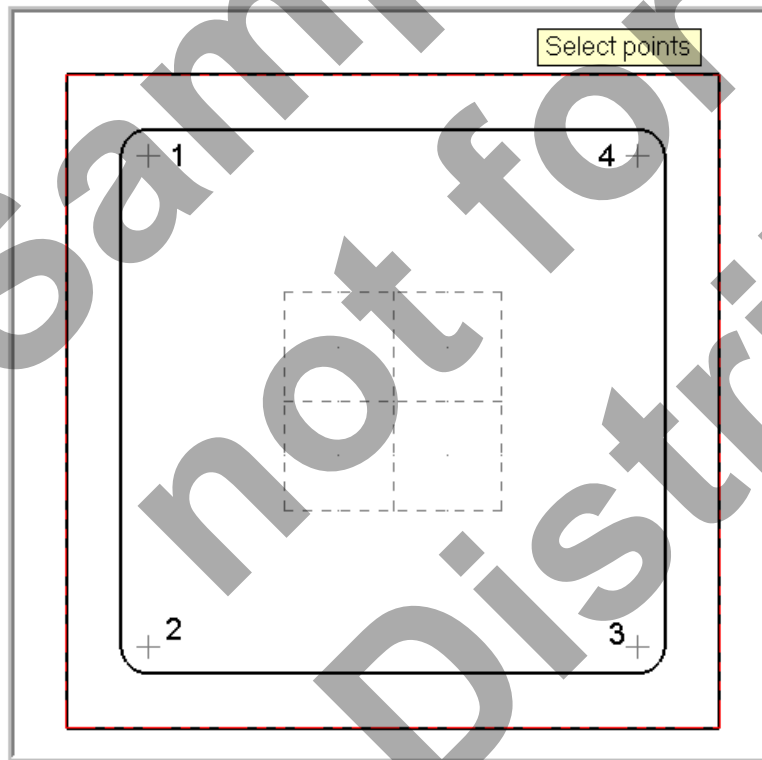


Mastercam Training Guide

- As you need to drill the four holes at the centre of the four fillet radii all you need to do now is snap to the centre of the fillet radius.
- 7. Move the mouse and position the cursor over the **center of the top left hand fillet radius**, you will notice a circle appears as you move closer to the center of the radius, this is a visual cue.
- 8. This circle demotes you are snapping the centre of this arc. Use the left button of your mouse to pick this center point.



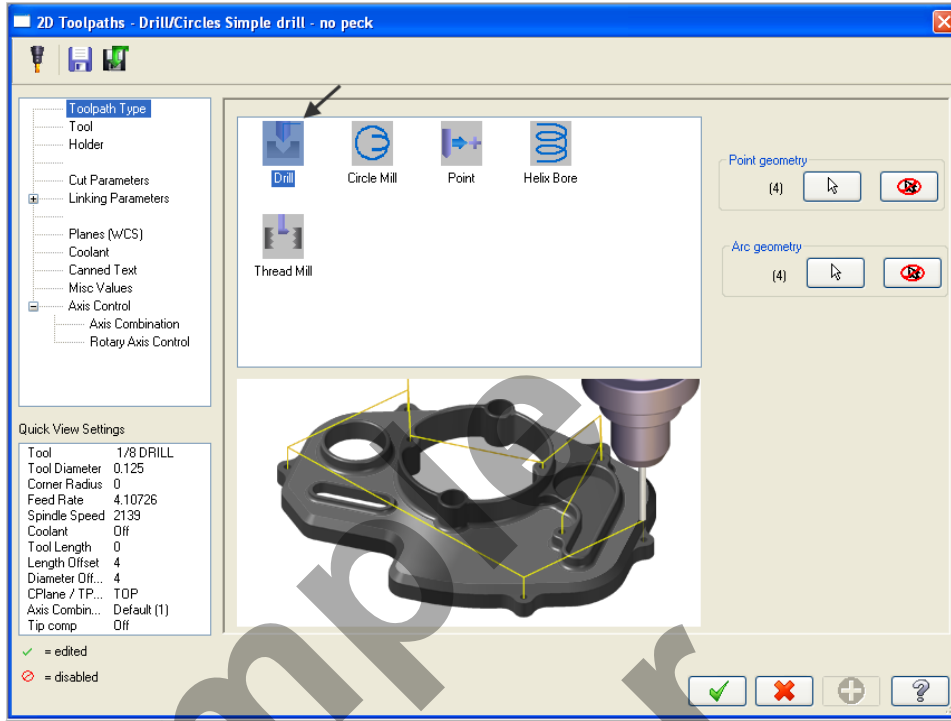
- 9. Using the method outlined above continue on and pick the remaining three center points, in this order:
 - 10. Lower left.
 - 11. Lower right.
 - 12. Upper right.
13. Your screen should look like the screenshot below after selecting the four center points:



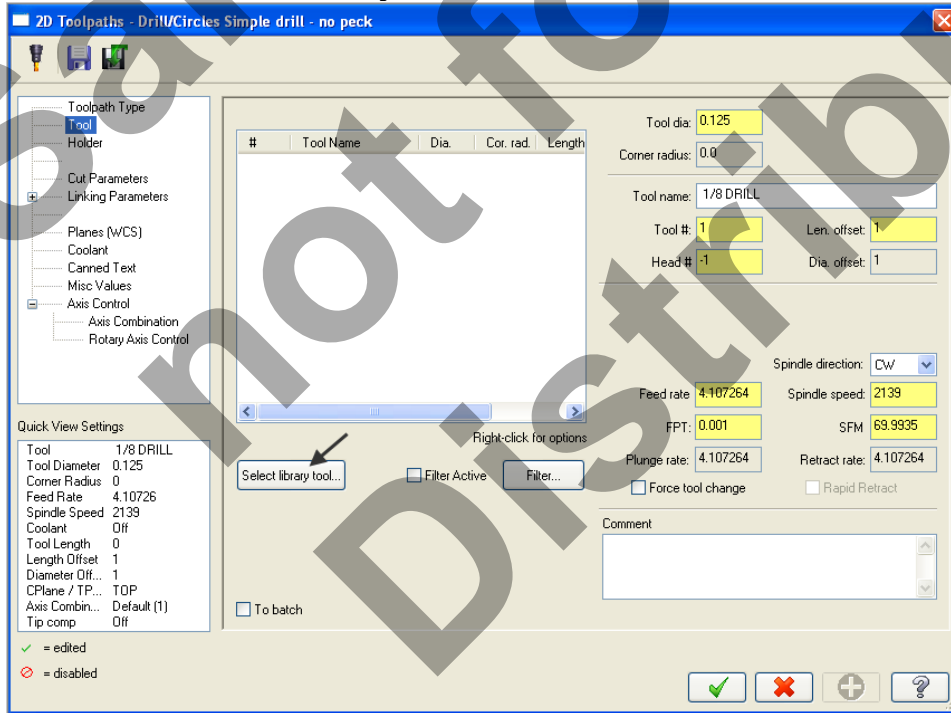
14. Select the OK button in the **Drill Point Selection** dialog box.

➤ After selecting the OK button, you are confronted with the Drill Toolpath Type page. The first task here will be to select a **.125" diameter end mill**.

1. Ensure the **Toolpath Type** is set to **Drill** as shown below and then select **Tool** from the list on the left.

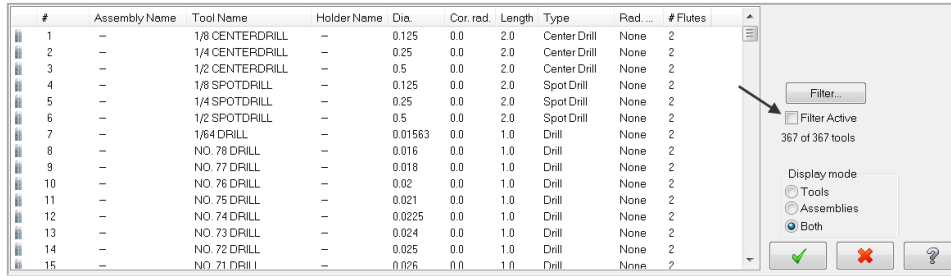


2. Click on the **Select library tool** button in the lower left corner.



Mastercam Training Guide

- On the right hand side of the Tool Selection dialog box remove the green check mark from the **Filter Active** to review all the tools.



- Use the slider bar on the right of this dialog box to scroll down and locate a .125" diameter flat end mill.
- Select the **.125" diameter flat end mill** by picking anywhere along the .125 end mill row, as shown below:


The screenshot shows the Tool Selection dialog box with a list of tools. The row for the .125 inch diameter flat end mill is highlighted in blue. An arrow points to the selected row.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	Rad. ...	# Flutes
229	-	1/32 FLAT ENDMILL	-	0.03125	0.0	0.375	Endmill1 Flat	None	4
230	-	1/16 FLAT ENDMILL	-	0.0625	0.0	0.375	Endmill1 Flat	None	4
231	-	3/32 FLAT ENDMILL	-	0.09375	0.0	0.375	Endmill1 Flat	None	4
232	-	1/8 FLAT ENDMILL	-	0.125	0.0	0.375	Endmill1 Flat	None	4
233	-	5/32 FLAT ENDMILL	-	0.15625	0.0	0.375	Endmill1 Flat	None	4
234	-	3/16 FLAT ENDMILL	-	0.1875	0.0	0.4375	Endmill1 Flat	None	4

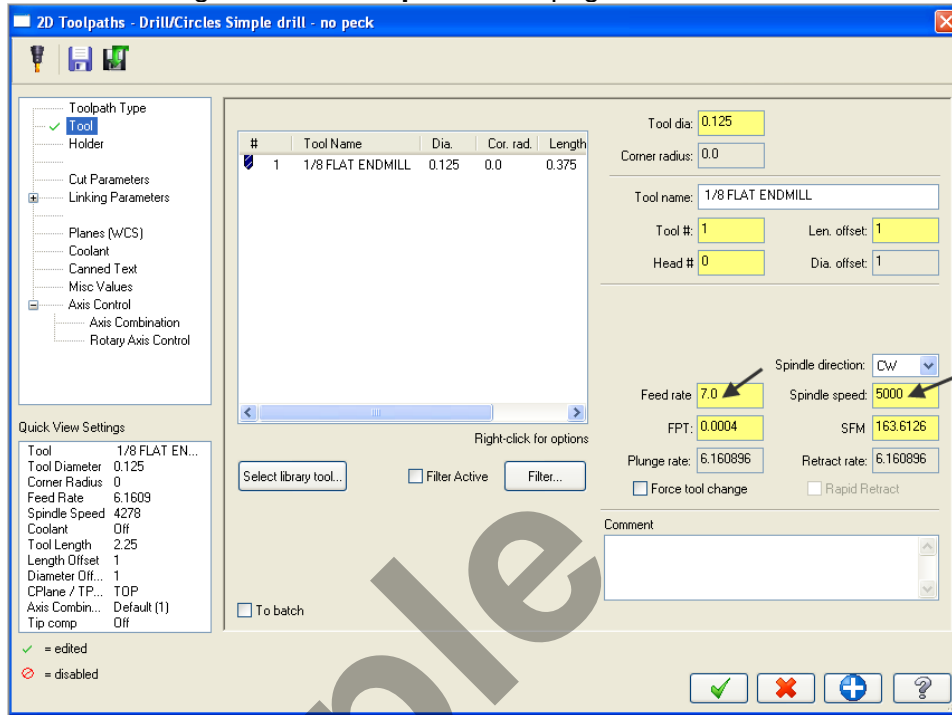
- To resize a column in the **Tool Selection dialog box**, click on the divider between the columns with your left mouse button, as shown below, hold the left mouse button down and move to the right or left.

The screenshot shows the Tool Selection dialog box with a list of tools. The 'Tool Name' column is highlighted in blue, and an arrow points to the divider between the 'Tool Name' and 'Holder Name' columns.

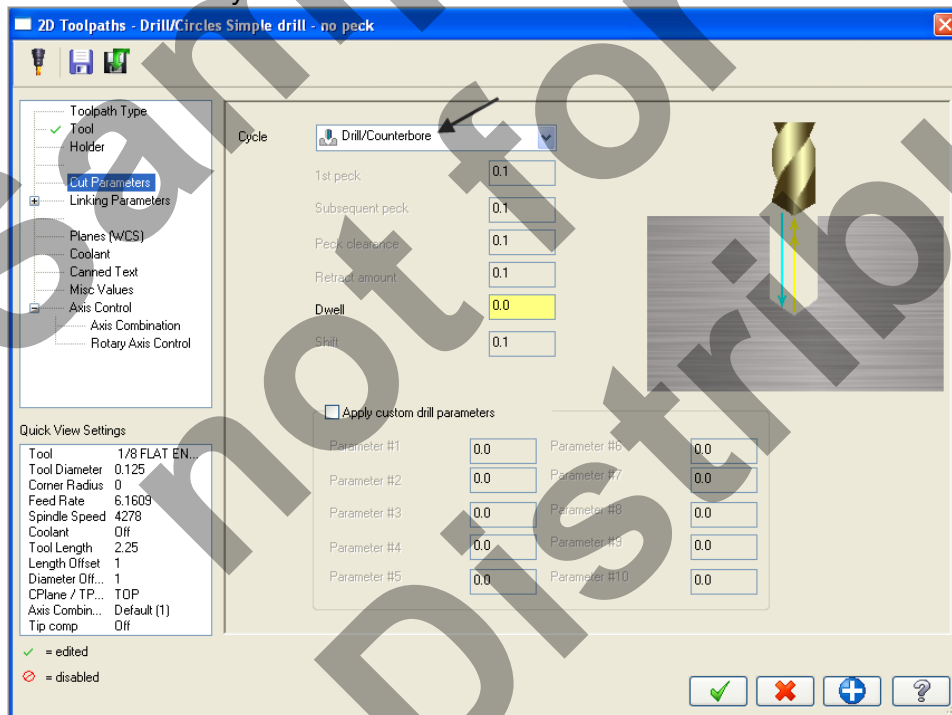
#	Assembly Name	Tool Name	Holder Name
229	-	1/32 FLAT ENDMILL	-
230	-	1/16 FLAT ENDMILL	-
231	-	3/32 FLAT ENDMILL	-
232	-	1/8 FLAT ENDMILL	-

- Select the **OK** button  to complete the selection of this tool.

8. Make changes to the **Tool parameter** page as shown below:

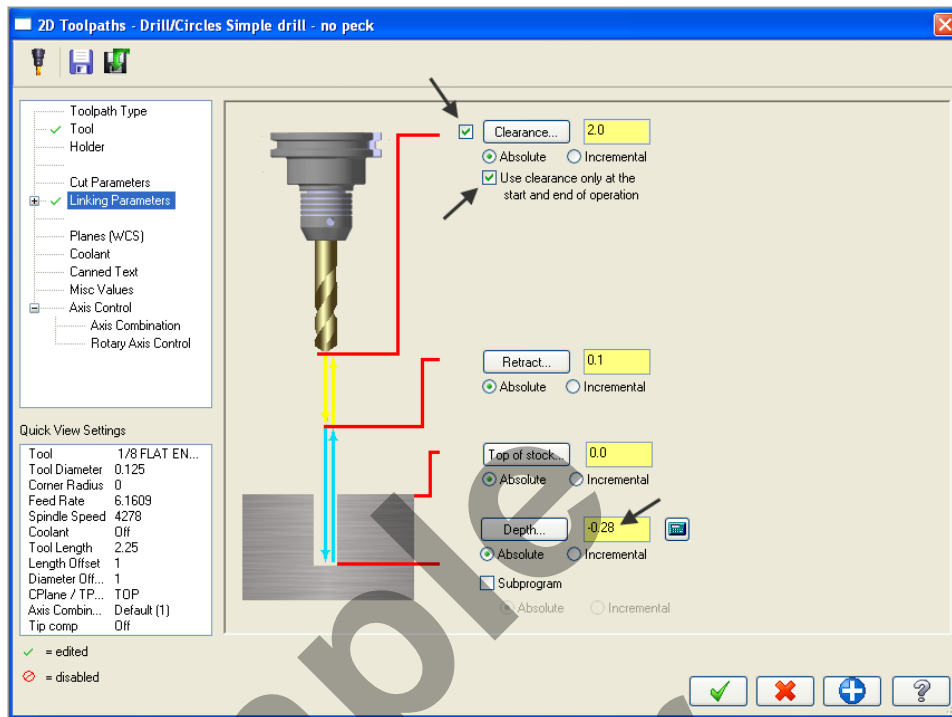


9. Select **Cut Parameters** from the list on the left and make changes to this page as shown below. The Cycle should be set to Drill/Counterbore.

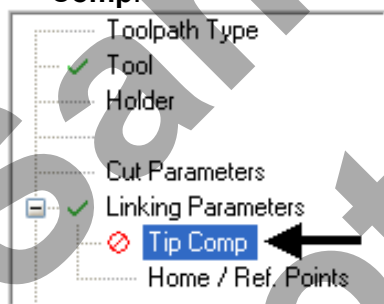


Mastercam Training Guide

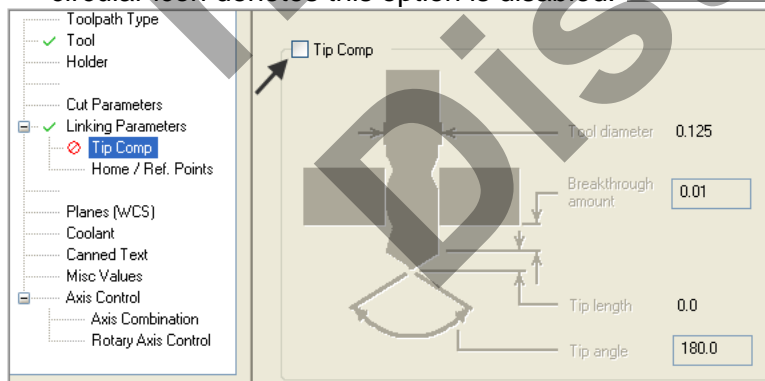
10. Select **Linking Parameters** from the list on the left and make changes to this page as shown below. Input the **depth of -0.28** and the other values as shown below. **Note** all the values are set to **Absolute**.



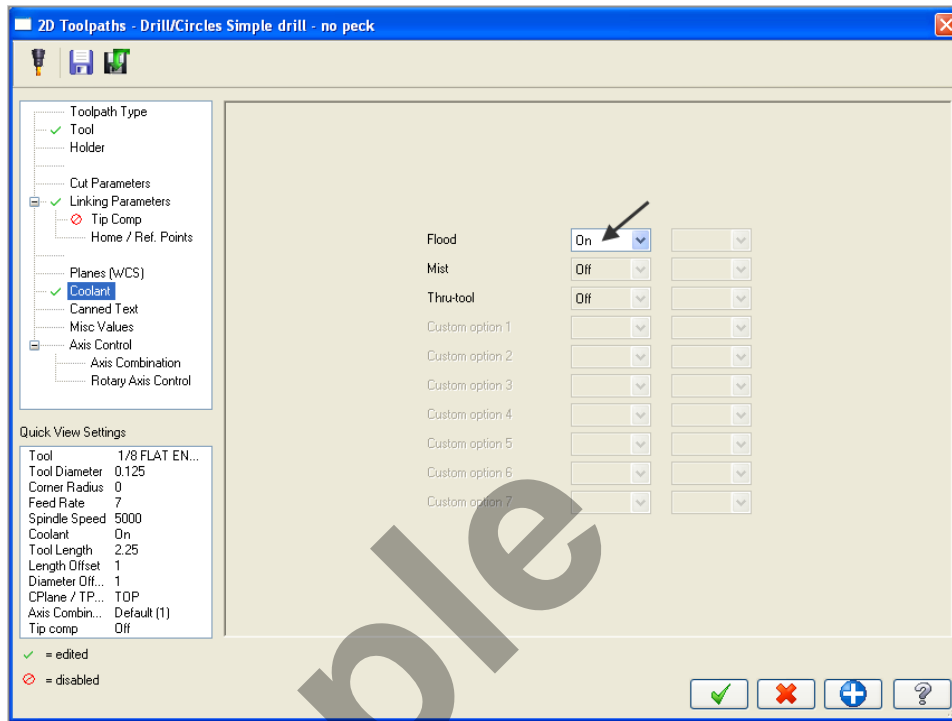
11. Select the plus sign to the left of **Linking Parameters** to expand the list and click on **Tip Comp**.



12. Ensure **Tip Comp** is **not activated** as shown below. The **Tip Comp** box is empty. The red circular icon denotes this option is disabled.

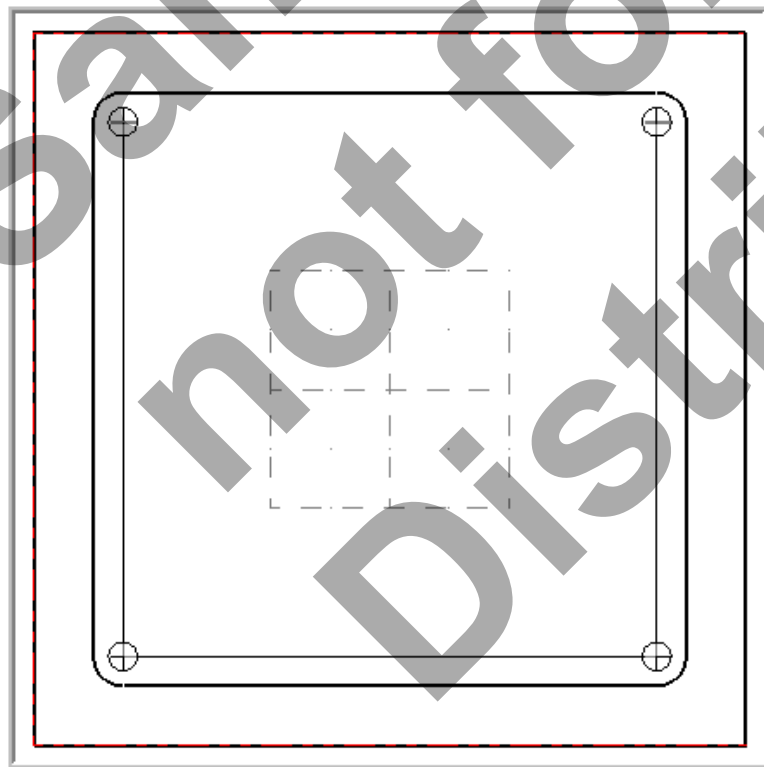


13. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.



14. Select the **OK** button to complete this function.

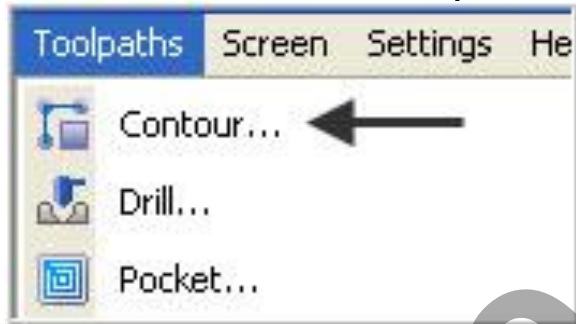
15. Your part should look like the screenshot below:



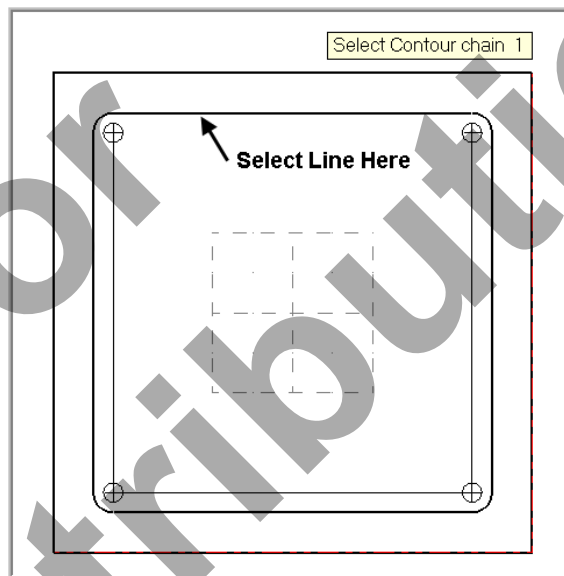
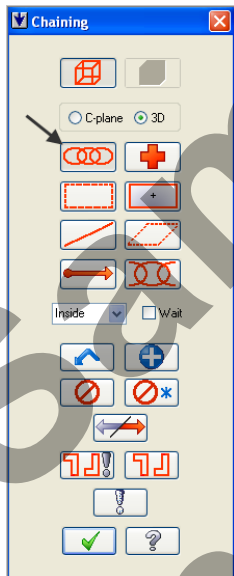
TASK 8: MACHINE THE CONTOUR.

- In this task you machine the contour with a .5" diameter 2 flute end mill.
- Initially you will machine the contour in one cut at a depth of -.125" and then later in this Lesson add roughing and finishing cuts using Depth of Cuts and Multi Passes.

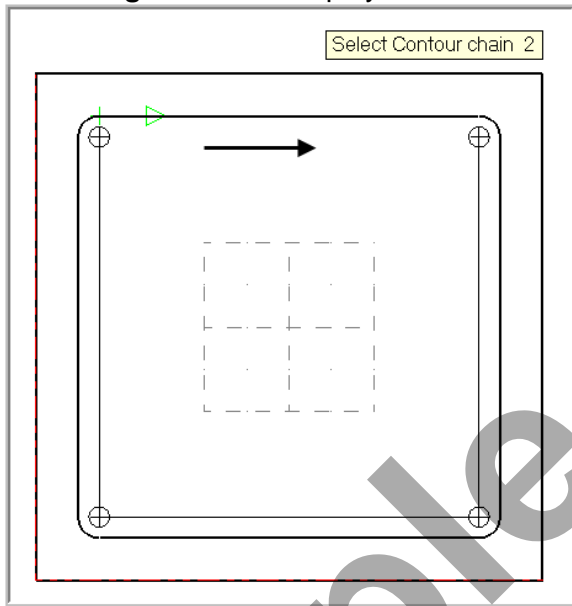
1. From the menu bar select **Toolpaths>Contour...**



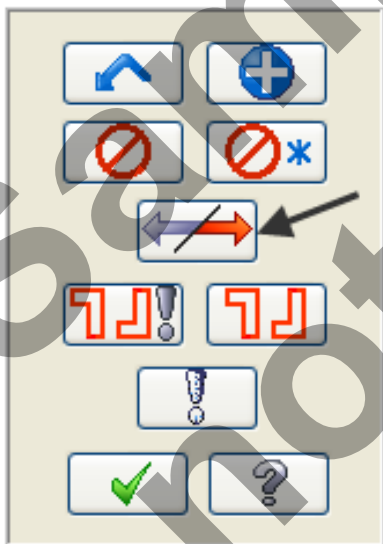
2. On the screen you will now see the **Chaining** dialog box with **Chain** set and in the graphics screen a prompt to "**Select Contour chain 1**". Select the top horizontal line as shown below:



3. After selecting the top horizontal line your graphics screen should look like the screenshot below, with the **green arrow pointing to the right**, clockwise direction.
 - The material for this part is aluminium so to attain a good finish when contouring **climb milling** should be employed.



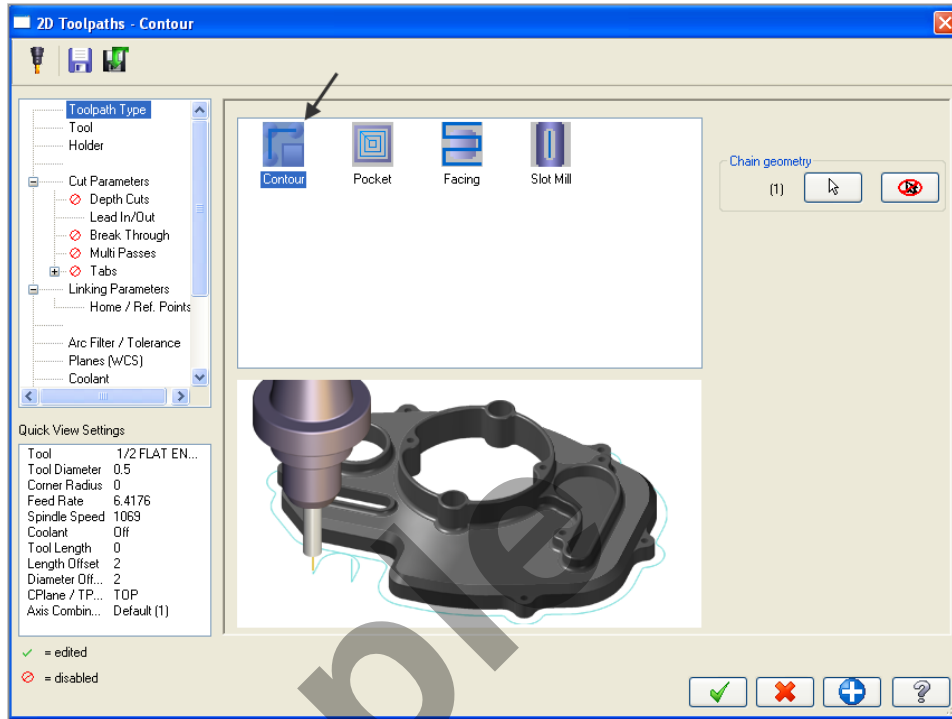
4. If the arrow is not pointing to the right select the arrow from the **Chaining** dialog box shown below to reverse the direction:



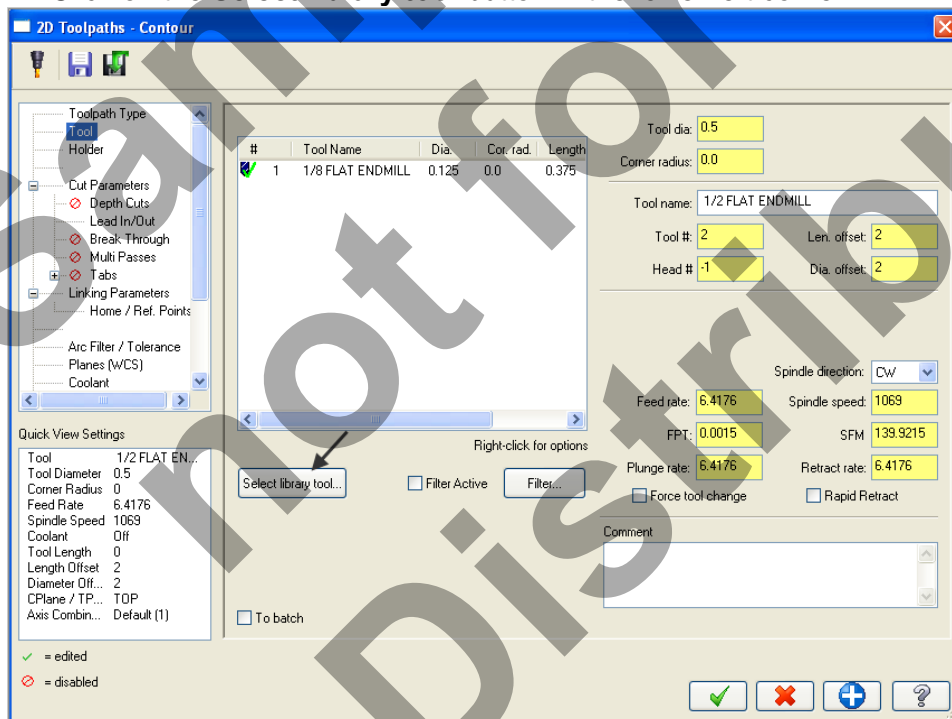
5. After the contour has been successfully chained select the **OK** button  at the bottom of the Chaining dialog box.

Mastercam Training Guide

6. Ensure the **Toolpath Type** is set to **Contour** as shown below and then select **Tool** from the list on the left.



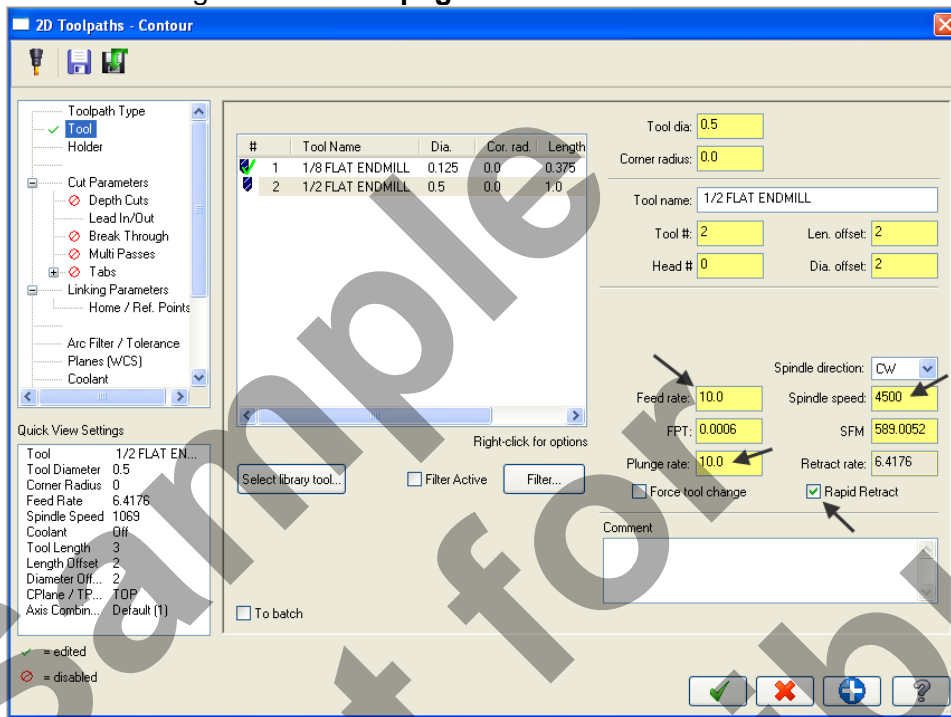
7. Click on the **Select library tool** button in the lower left corner.



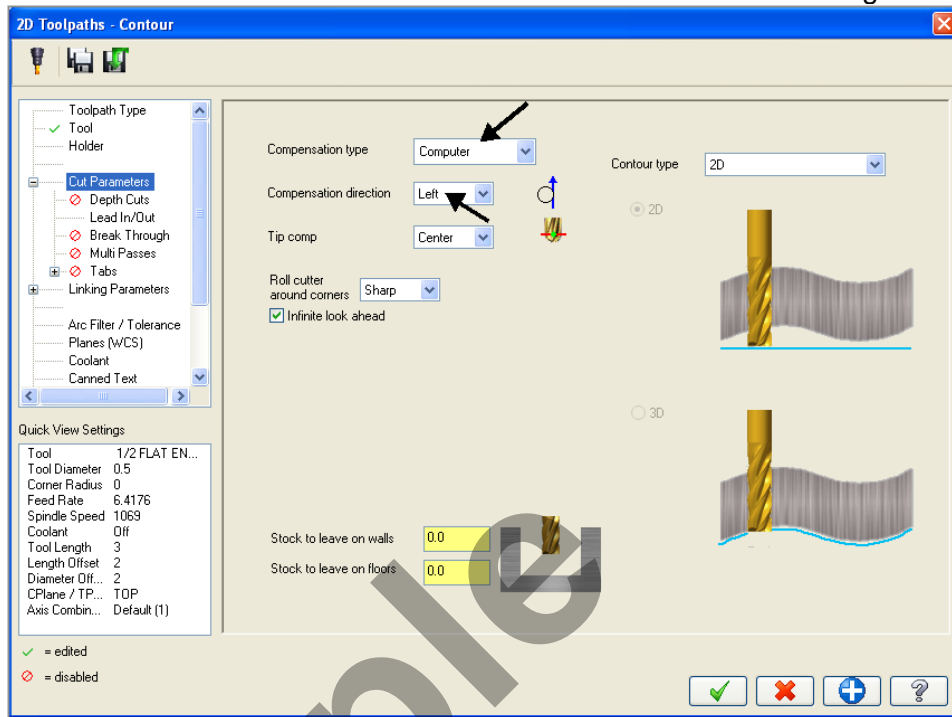
8. Use the slider bar on the right of this dialog box to scroll down and locate a **.5" diameter flat end mill**. Select the .5" diameter flat end mill by picking anywhere along the .5 end mill row, as shown below:

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes
236	—	5/16 FLAT ENDMILL	—	0.3125	0.0	0.75	4
237	—	3/8 FLAT ENDMILL	—	0.375	0.0	0.75	4
238	—	7/16 FLAT ENDMILL	—	0.4375	0.0	0.8	4
239	—	1/2 FLAT ENDMILL	—	0.5	0.0	1.0	4
240	—	5/8 FLAT ENDMILL	—	0.625	0.0	1.5	4
241	—	3/4 FLAT ENDMILL	—	0.75	0.0	2.0	4

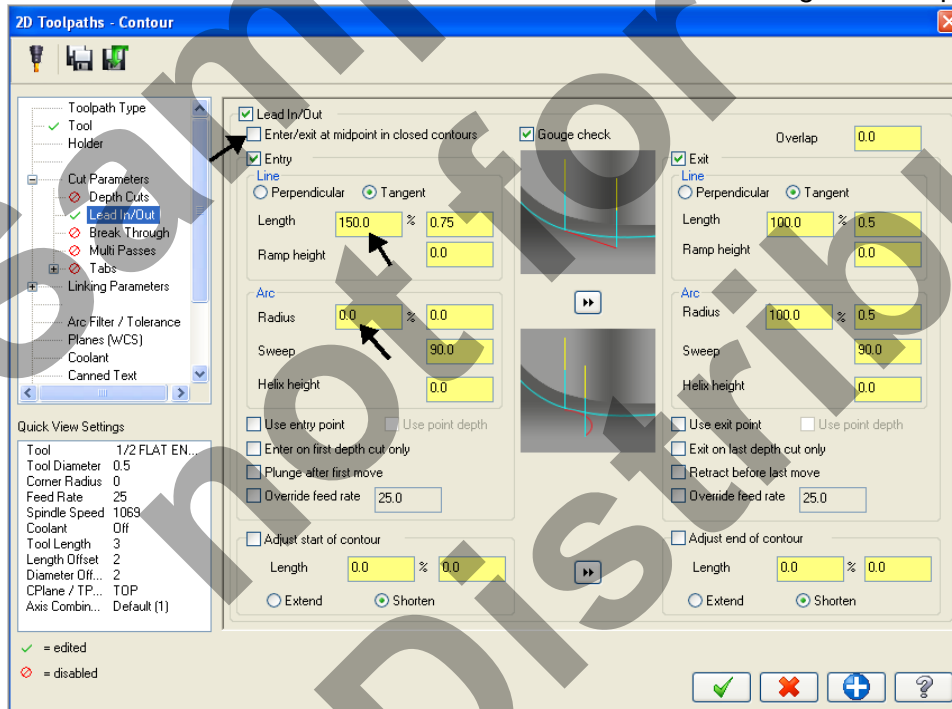
9. Select the **OK** button  to complete the selection of this tool.
 10. Make changes to the **Tool** page as shown below:



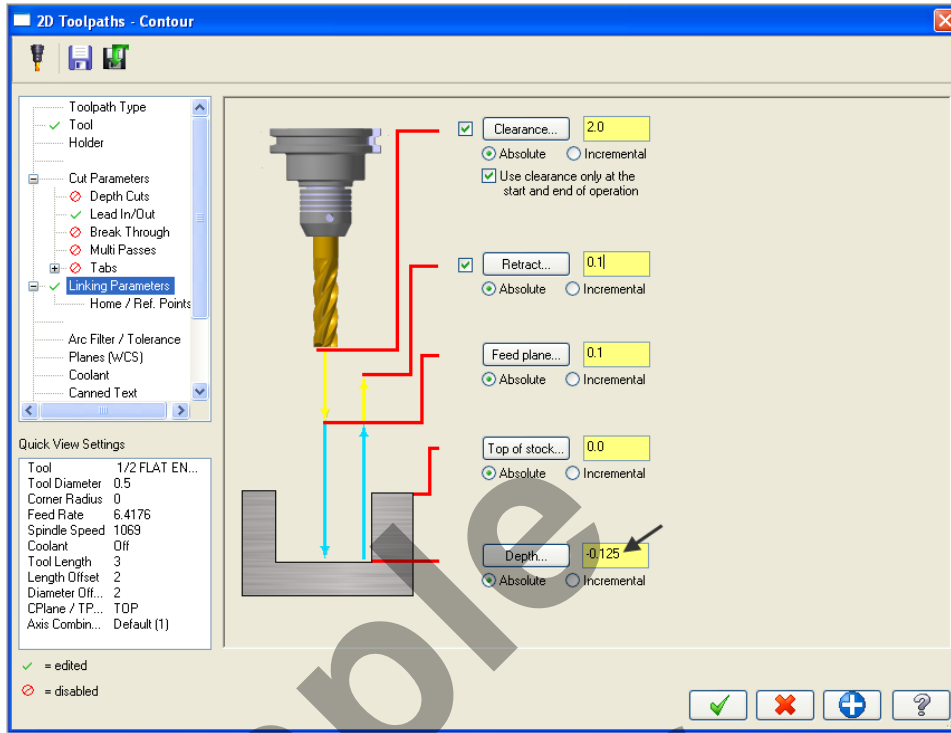
11. Select **Cut Parameters** from the list on the left and make changes to this page if required.



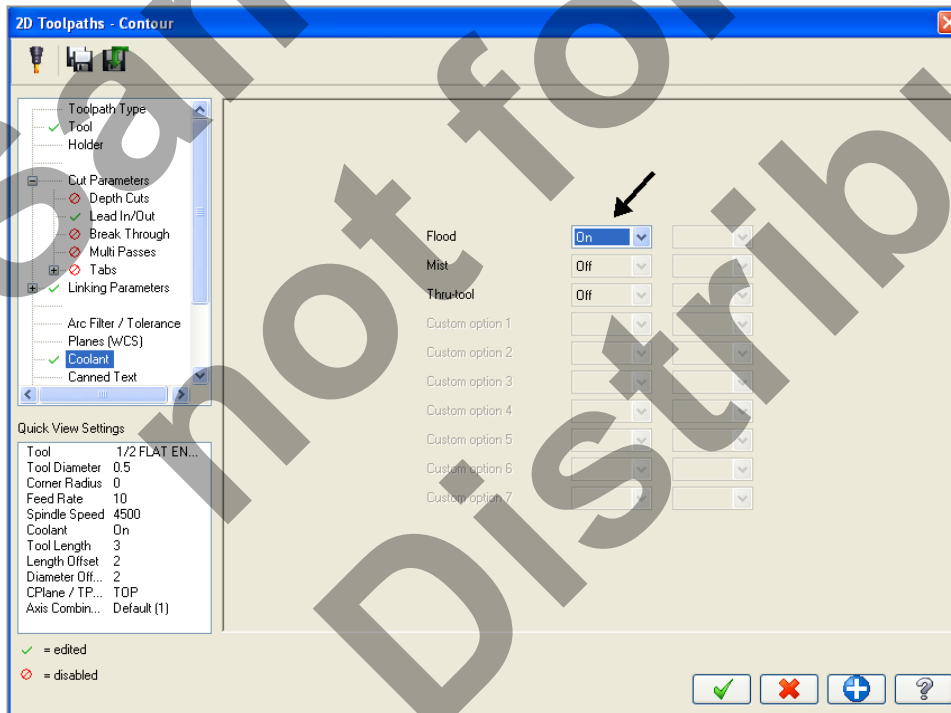
12. Select **LeadIn/Out** from the list on the left and make changes to this page.



13. Select **Linking Parameters**. Input the **depth of -0.125** and the other values as shown below. **Note** all the values are set to **Absolute**.




14. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.

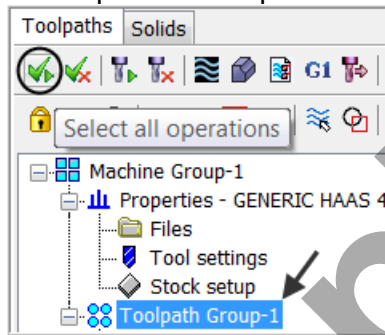


15. Select the **OK** button  to complete this function

TASK 9: BACKPLOT THE TOOLPATH

- In this task you will use Mastercam's Backplot function to view the path the tools take to cut this part.
- Backplot will enable you to review the cutting motions and identify any problem areas when cutting the part.
- When the toolpaths are being Backplotted Mastercam displays tool path information on the right of the screen. Information such as the current tool position in X, Y, and Z coordinates.
- **For more information on Backplot see the Tips and Techniques section on the multimedia DVD supplied with this text.**

1. To pick all the operations to backplot pick the Select All icon  circled below:



- Another method to **Select all** the operations is by clicking on the **Toolpath Group-1** in the Tool Manager as shown by the arrow above.

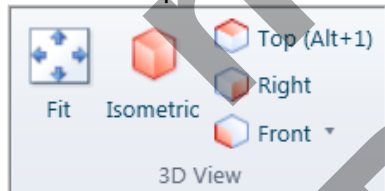
2. The next step is to select the **Backplot selected operations** icon shown below:



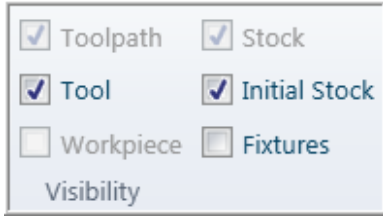
- 3. **Maximize** the Backplot/Verify window if required.
- 4. Select the **Home** Tab if required.



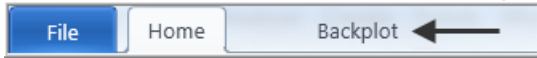
5. At the top of the screen select the **Isometric** icon and then select **Fit**.



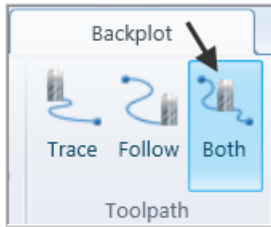
4. Activate the options shown below in the **Visibility** section of the Home tab.



6. Click on the **Backplot** tab at the top left of the screen.



7. Activate the **Both** option in the Toolpath section of the Backplot tab.



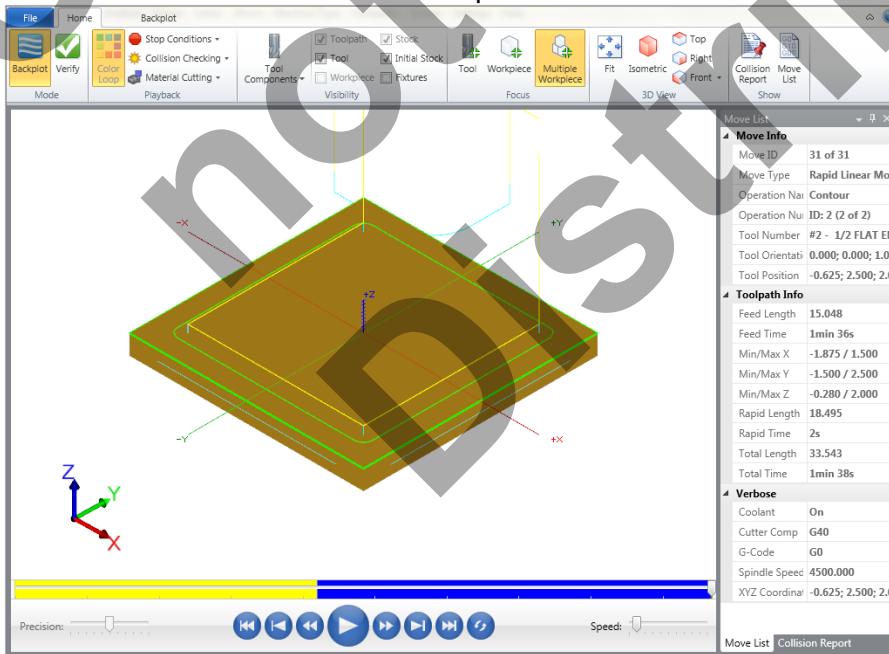
8. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.



9. Now select the **Play Simulation** button to review the toolpaths.



10. After reviewing the backplot of the two toolpaths using a .125" and .5" end mill select the **Close** button  to exit Backplot.

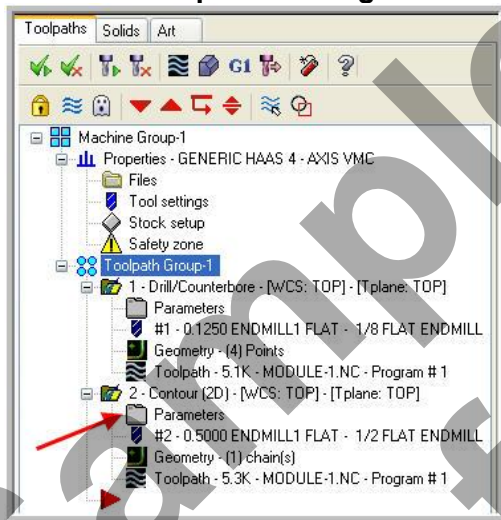


TASK 10:

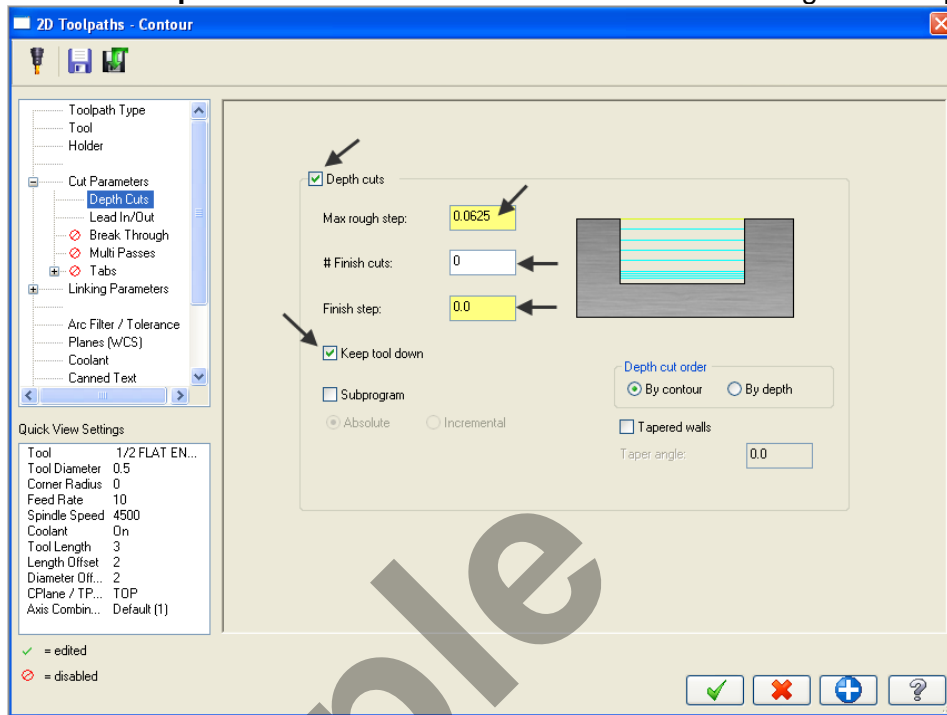
MODIFY THE CONTOUR TOOLPATH TO ADD ROUGHING CUTS AND A FINISH PASS

- In this task you will use Mastercam's **Multi Passes** and **Depth of cuts** to perform a roughing and finishing operation for the contour toolpath.
- **Multi Passes** will let the tool approach the part geometry at the cutting depth in steps instead of cutting right to the part geometry.
- **Depth of cuts** can be used to set the number of depth cuts, you can enter a maximum rough step and Mastercam divides the total depth into equal steps. Or you can enter the exact number of finish steps and the size of each finish step. Mastercam never creates unequal rough depth cuts.
- For more information on Multi Passes and Depth of cuts see the **Tips and Techniques** section on the multimedia CD supplied with this text.

1. In the **Toolpaths Manager** select **Parameters** from the contour toolpath as shown below:



2. Select **Depth Cuts** from the list on the left and make changes to this page as shown below:



About the Depth cuts dialog box

Max rough step:

Sets the maximum amount of material removed in the Z axis with each rough cut.

Finish cuts:

Sets the number of finish cuts for the contour toolpath. This number multiplied by the finish step value equals the total amount of stock cut by the finish passes. Setting the number of finish cuts to 0 creates no finish cuts.

Finish step:

Sets the amount of material removed in the Z axis with each finish cut. This number multiplied by the number of finish passes equals the total amount of stock cut by the finish passes.

Keep Tool Down:

Determines if the tool should retract between multi passes.

Depth cut order:

By pocket/contour

Performs all depth cuts in a contour or region before moving to the next contour or region.

By depth

Creates depth cuts at the same level in every contour or region and then descends to the next depth cut level.

In this example you will perform:

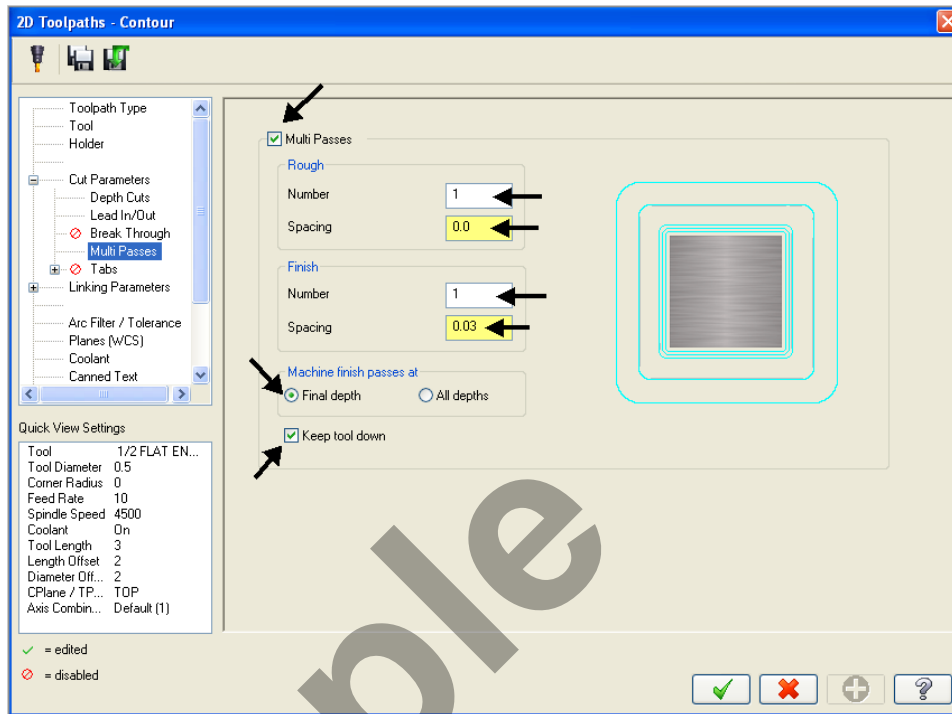
Each depth of cut will not exceed .0625", therefore as your final depth is -.125" you will perform only two rough cuts. The first at -0.0625 and the second rough cut at -.125

Only one finish pass at the final depth

The finish pass will only take place at the final depth, this final cut will machine the .030" from the contour that you set up using Multi Passes

In between passes the tool will be kept down

3. Select **Multi Passes** from the list on the left and make changes to this page as shown below:



About the Multi Passes dialog box

Roughing passes:

Number: Enter the number of cutting passes you want Mastercam to create.

Spacing: Enter the amount of stock to remove with each cut.

Finishing passes:

Number: Enter the number of cutting passes you want Mastercam to create.

Spacing: Enter the amount of stock to remove with each cut.

Machine finish Passes at:

Final Depth: Performs a single finish pass at the final depth.

Keep Tool Down: Determines whatever the tool should retract between multi passes.

In this example you will perform:

No roughing cuts in the XY plane.


Only one finish pass at the final depth.

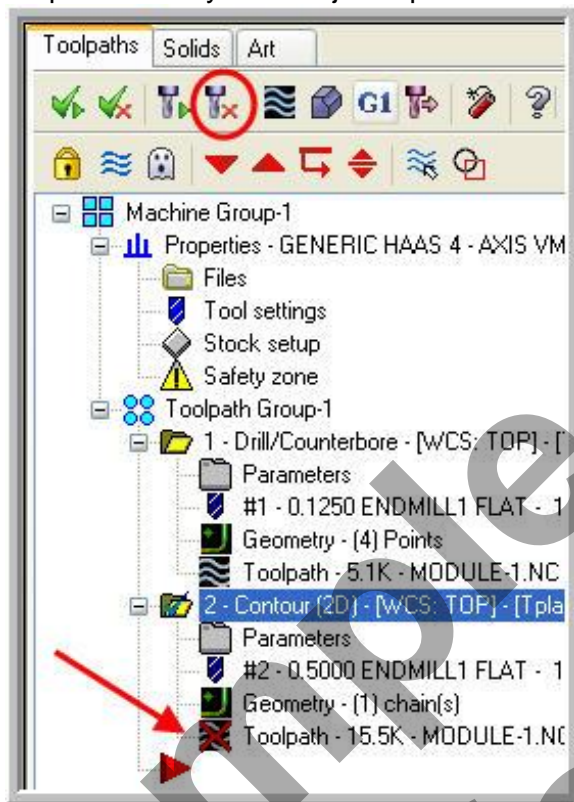
While cutting at the various depths you will stay .030" away from the contour.

The finish pass will take place at the final depth.

In between passes the tool will be kept down.

4. After reviewing the values input in the **Multi Passes** dialog box select the OK button  to exit.

5. Select the **Regenerate all dirty operations** icon  to remove the red X from the contouring operation you have just edited. You need to update the toolpath with the new parameters you have just input.



Dirty toolpath

This happens if you have changed certain parameters of the underlying geometry, or in this example you have updated the contour toolpaths to use Depth of cuts and multi passes.

Toolpaths can be regenerated by clicking the Regenerate button at the top of the Toolpaths Manager circled above.

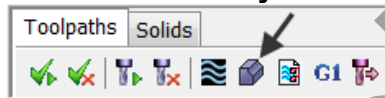
TASK 11: VERIFY THE TOOLPATH

- ☞ 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.
- ☞ Backplot and Verify are very similar. The differences between these two functions are that Backplot offers basic simulation options. Whereas Verify offers material removal, collision checking and precision control.
- ☞ **For more information on Verify see the Tips and Techniques section on the multimedia DVD supplied with this text**

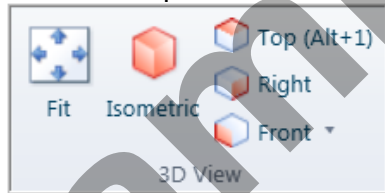
1. In the **Toolpaths Manager** pick all the operations to verify by picking the **Select All** icon



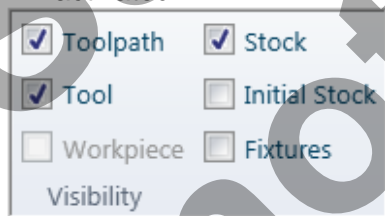
2. Select the **Verify selected operations** icon shown below:



3. **Maximize** the Backplot/Verify window if required.
4. At the top of the screen select the **Isometric** icon and then select **Fit**.



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




6. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.

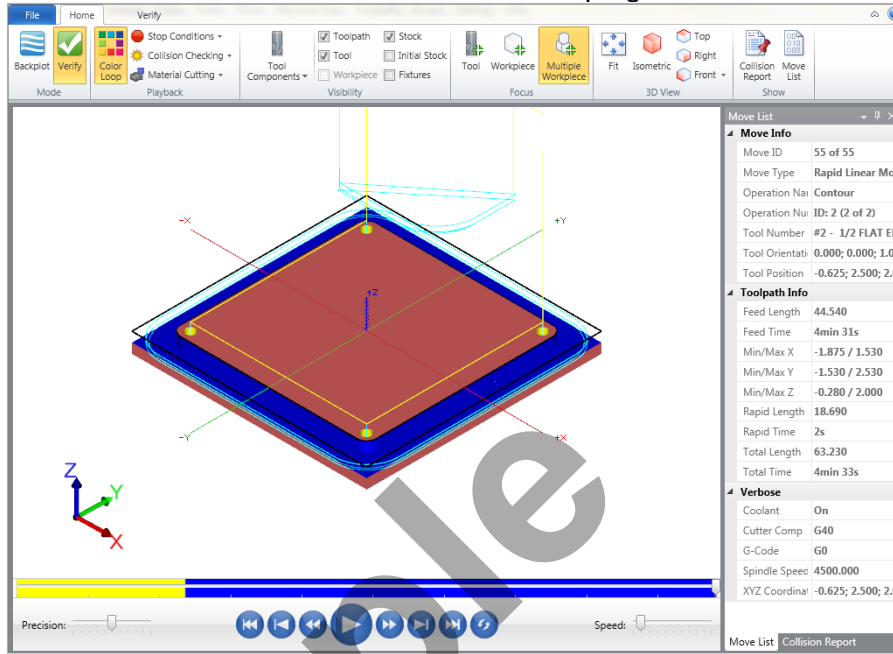


7. Now select the **Play Simulation** button to review the toolpaths.



8. After reviewing the two toolpaths the verified toolpaths should appear as in the picture below.

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




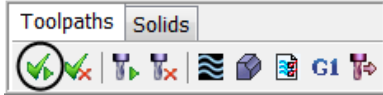
TASK 12: SAVE THE UPDATED MASTERCAM FILE

1. Select the **save** icon from the toolbar

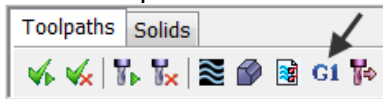


TASK 13: POST AND CREATE THE CNC CODE FILE

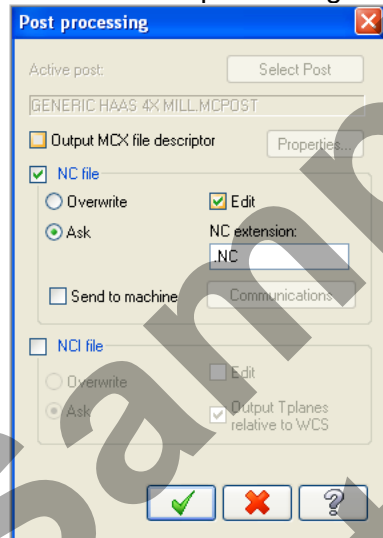
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpaths manager.



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 Toolpaths manager window and expand the window to the right.



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



About Post Processing

NC file:

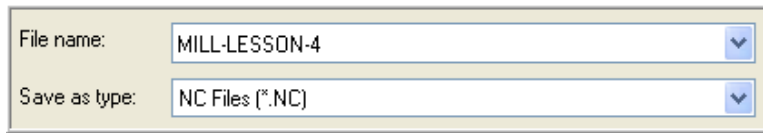
Select this option to save the NC file. The file name and extension are stored in the machine group properties for the selected operation. If you are posting operations from different machine groups or Mastercam files, or batch processing, Mastercam will create several files according to the settings for each machine group.

Edit:

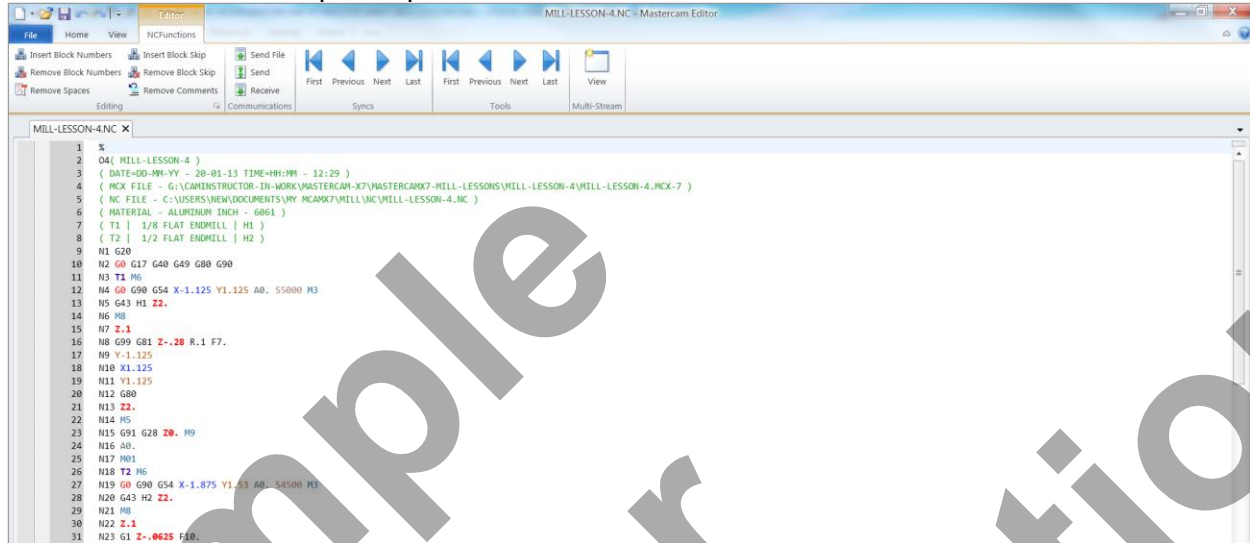
When checked, automatically launches the default text editor with the file displayed so that you can review or modify it.

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:

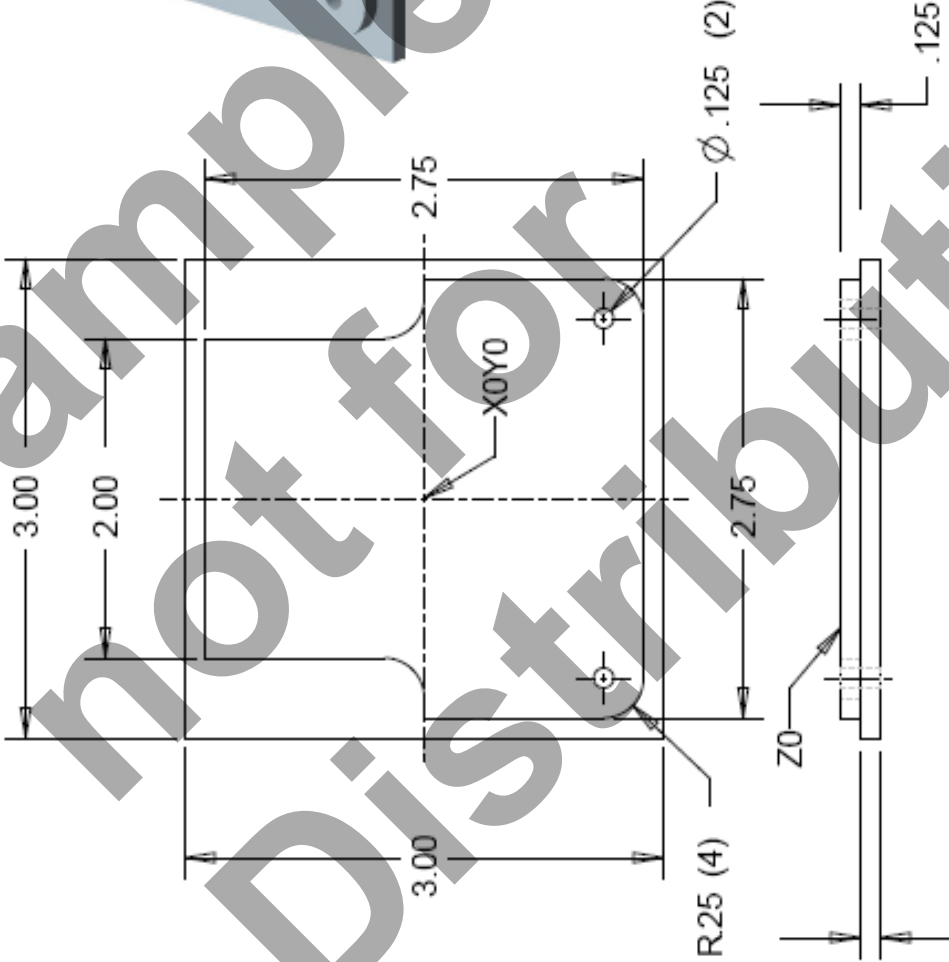
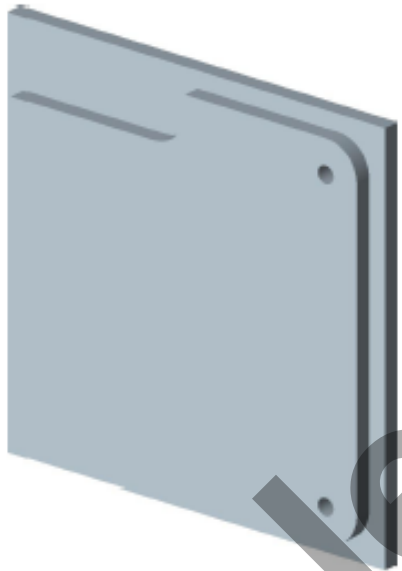


6. Select the **Save** button.
7. The CNC code file opens up in the default editor.



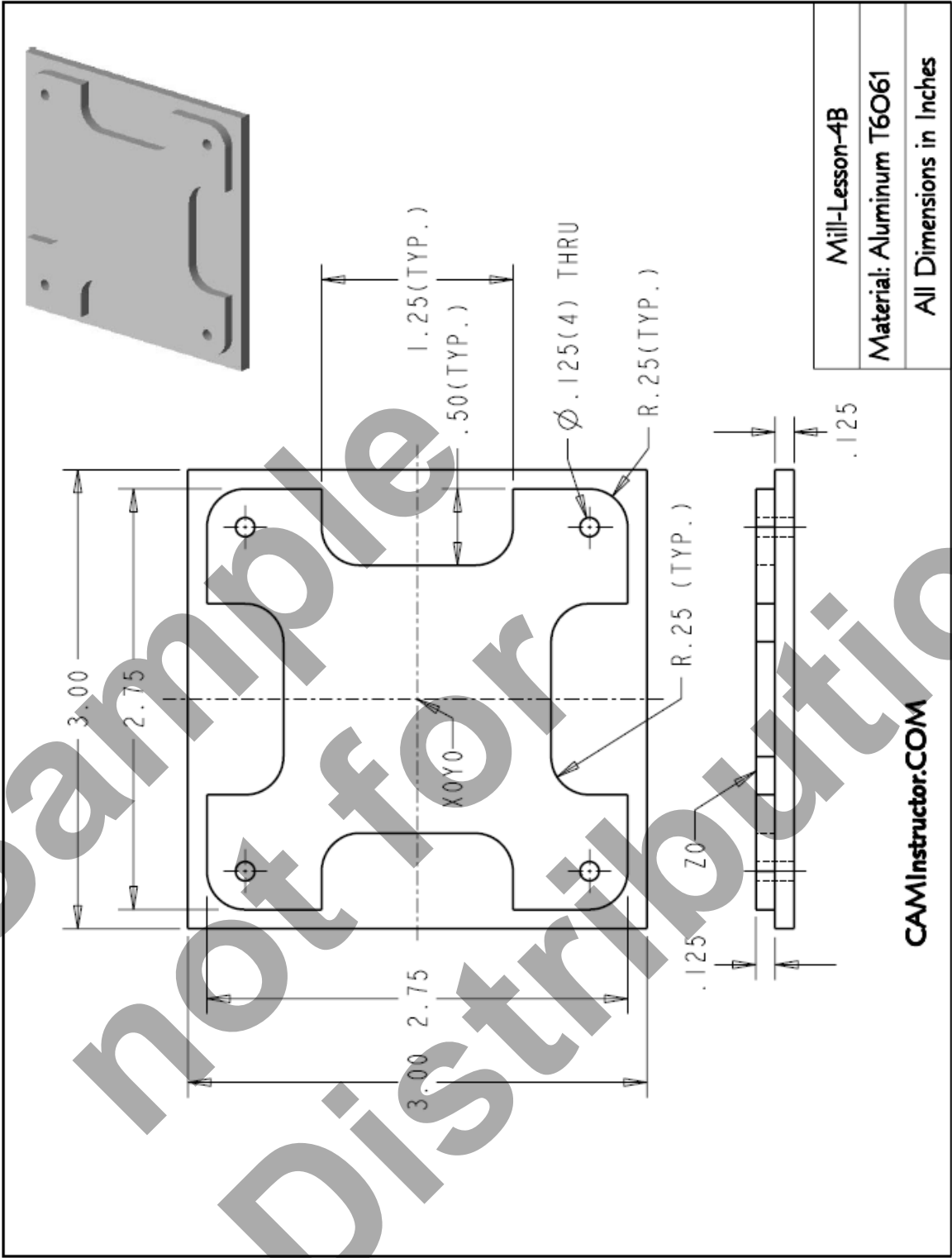
8. Select the  in the top right corner to exit the CNC editor.
9. This completes Mill-Lesson-4.

MILL-LESSON-4 EXERCISES



Mill-Lesson-4 Exercise
Material: Aluminum T6O61
All Dimensions in Inches

CAMInstructor.COM



Mill-Lesson-4B
Material: Aluminum T6O61
All Dimensions in Inches

CAMInstructor.COM

Sample
not for
Distribution