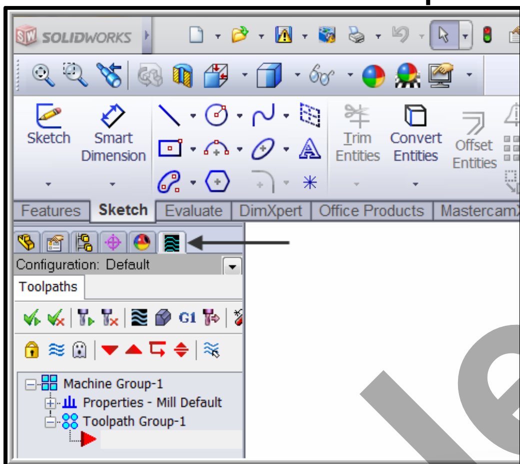
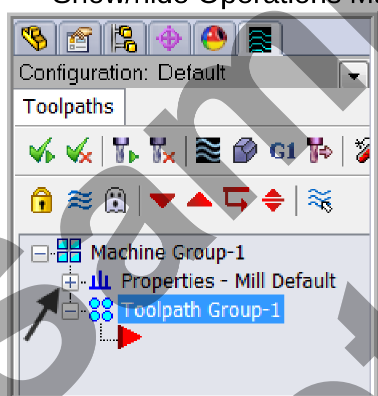
Toolpath Creation

TASK 5:
DEFINING THE ROUGH STOCK USING STOCK SETUP

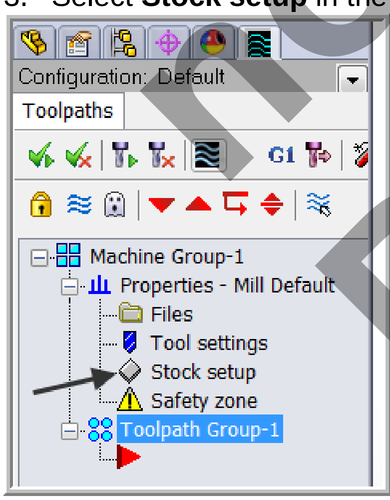
1. Click on the **Mastercam Toolpath Manager** tab as shown below.



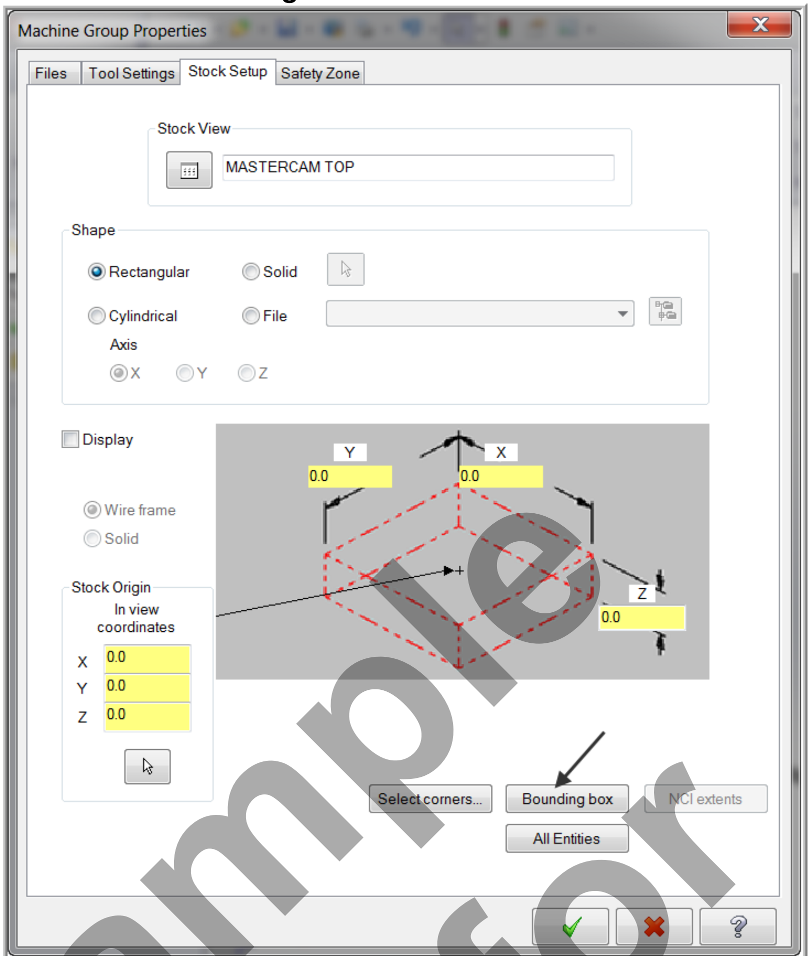
2. Select the plus in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will Show/hide Operations Manager pane.



3. Select **Stock setup** in the toolpath manager window.

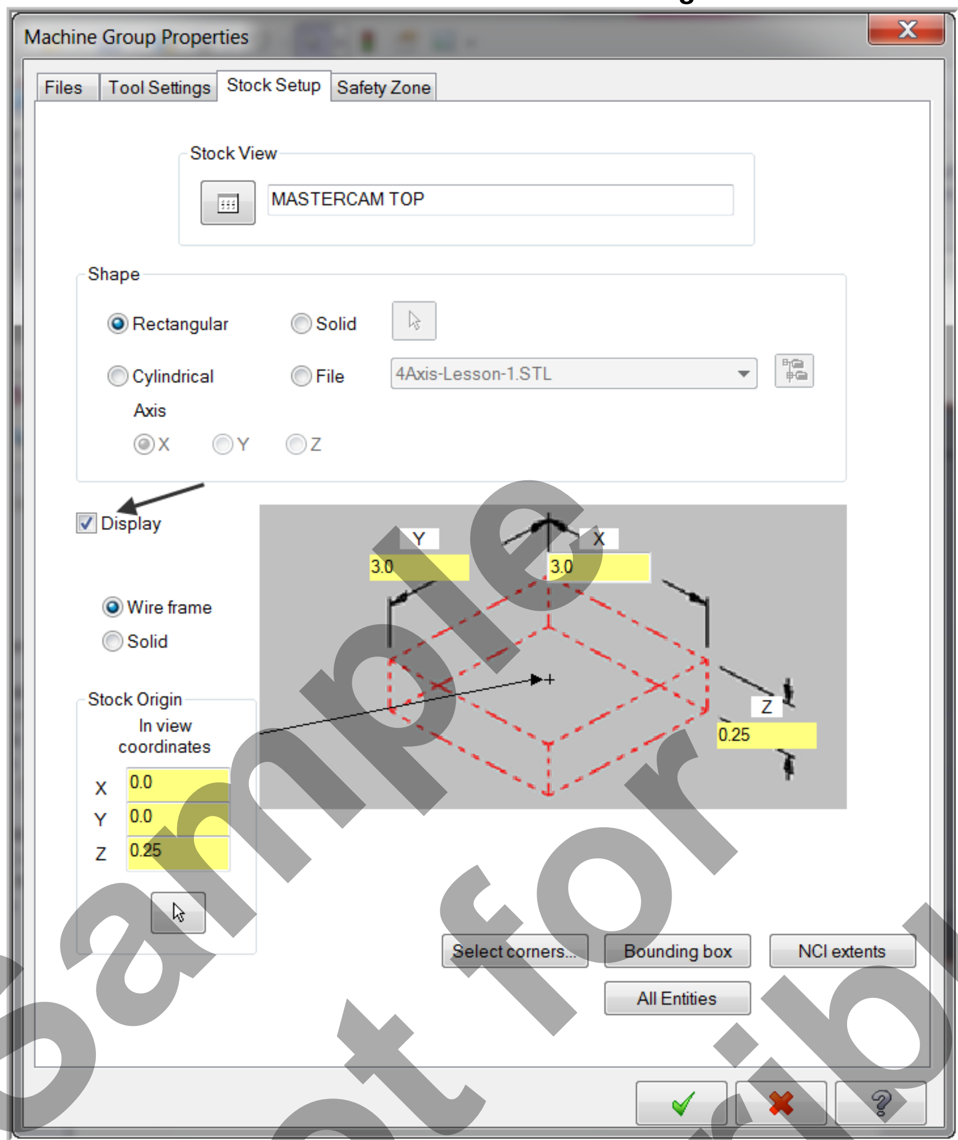


4. Click on **Bounding box** to set the Stock sizes.

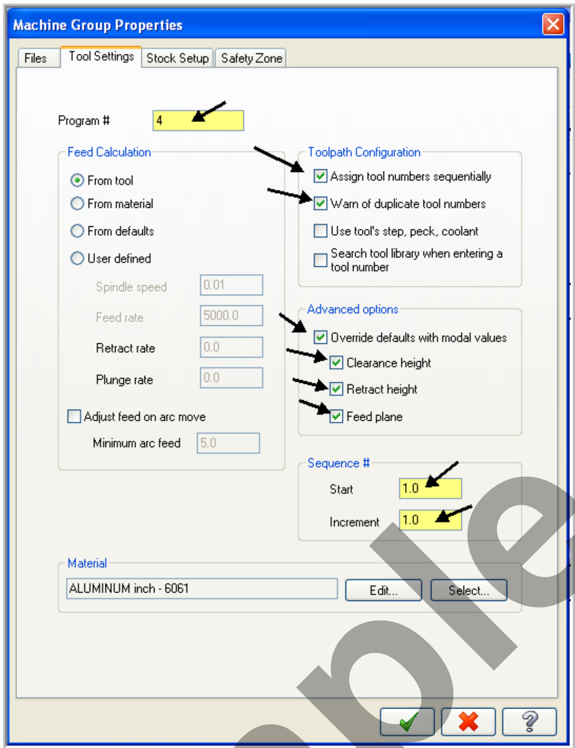


5. Nothing needs changing in the Bounding Box/Cylinder window, click on **OK** .

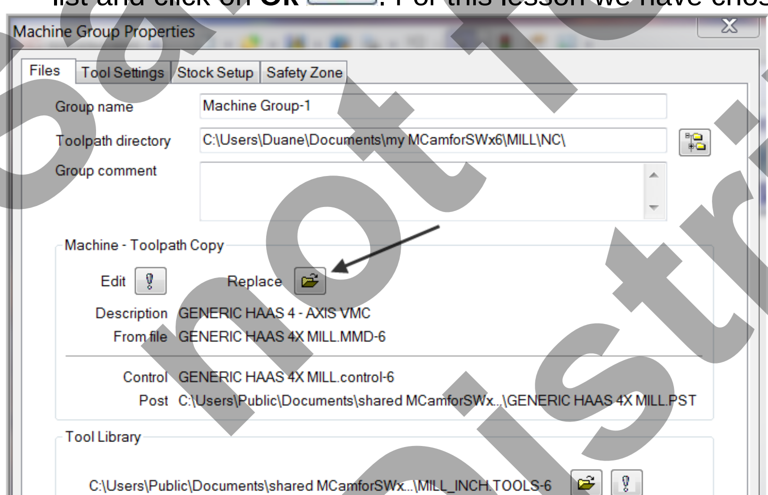
6. Click the box beside **Display** to turn on the Stock Setup display. Note the **Y, X and Z** coordinates are filled in as well as the **Stock Origin** coordinates.



7. Select the Tool Settings tab and change the parameters to match the **Tool Settings** screenshot below.



8. ***To select the machine that will be used to machine this part;*** Select the **Files** tab at the top of the **Machine Group Properties** window.
9. Click on the **Replace** icon to open the Machine folder. Select the proper machine from the list and click on **Ok** . For this lesson we have chosen the **Generic Haas 4X MILL.**

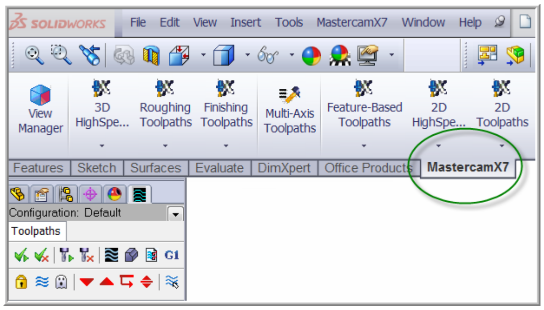


10. Select the OK button again to complete this Stock Setup function.

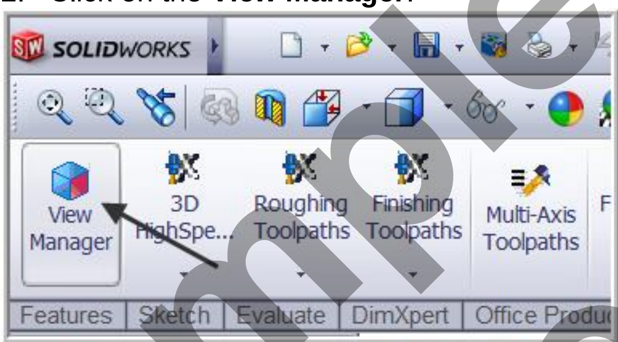
TASK 6:
DRILL THE 4 X .125" DIAMETER HOLES

➲ In this task you will drill the four .125" diameter holes through the part with a center cutting two flute end mill that is .125 " diameter.

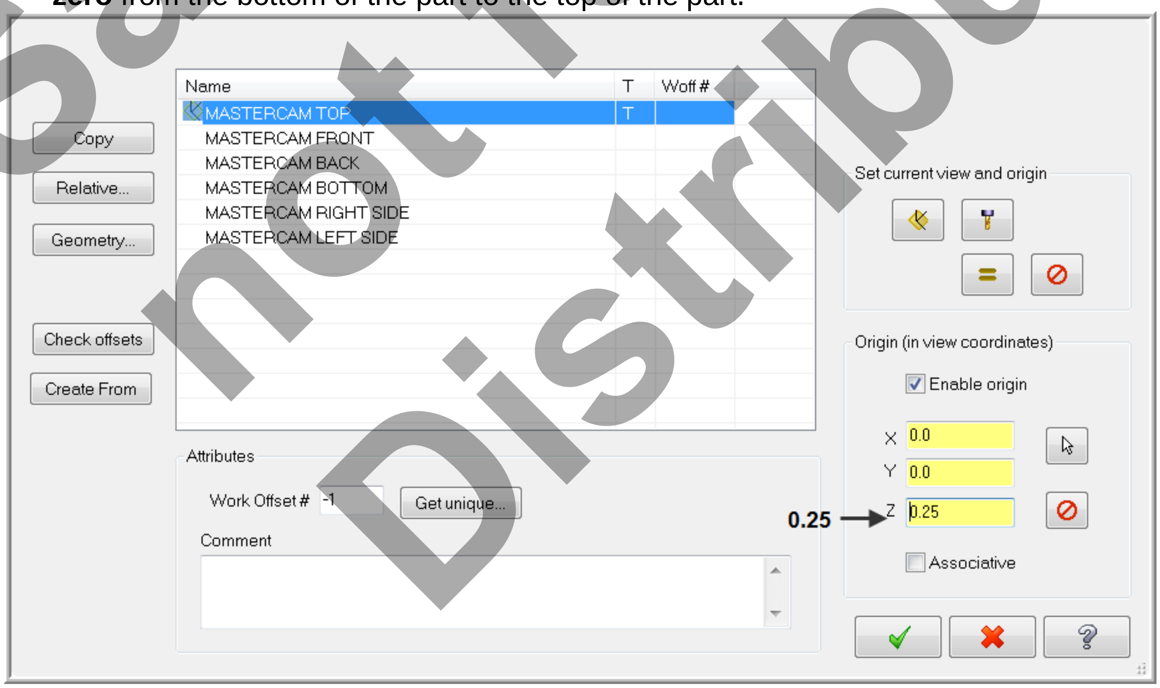
1. From the menu bar click on the **Mastercam Tab** as shown below.



2. Click on the **View Manager**.

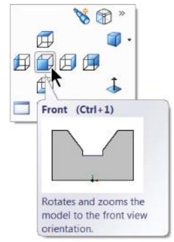


3. Enter **0.250** in the **Z coordinate window in Origin** as shown below. This will change the **Z zero** from the bottom of the part to the top of the part.

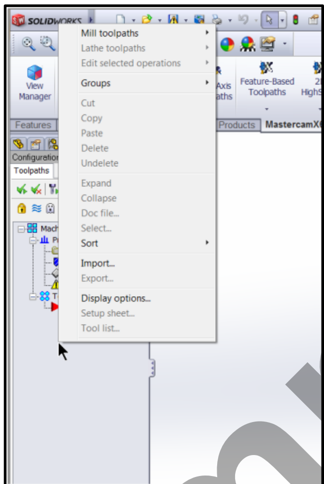


4. Click on the **OK** button  in the **View Manager** window.

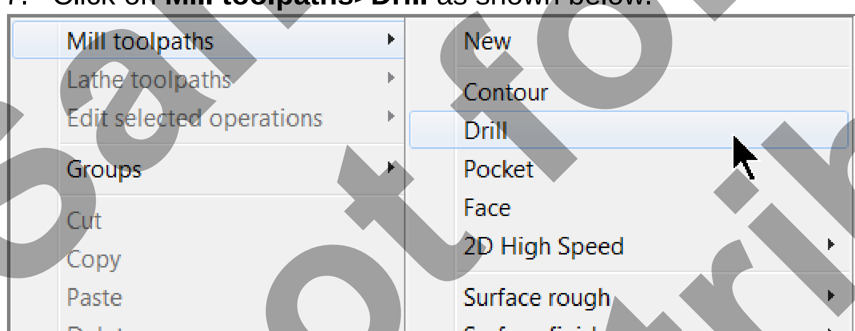
5. Change the **Graphics View** to **Front** by pressing the **space bar** and the **Orientation** window will appear. Click on **Front**.



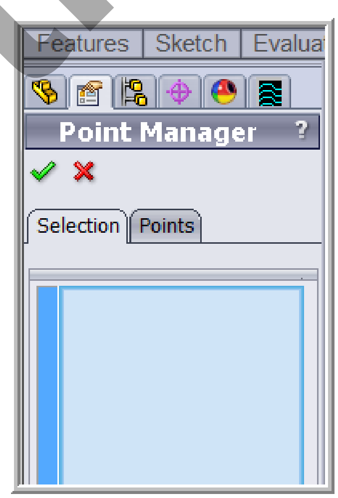
6. Position the cursor in the **Mastercam Toolpath Manager** and **right click the mouse button**. A new window will appear as shown below:



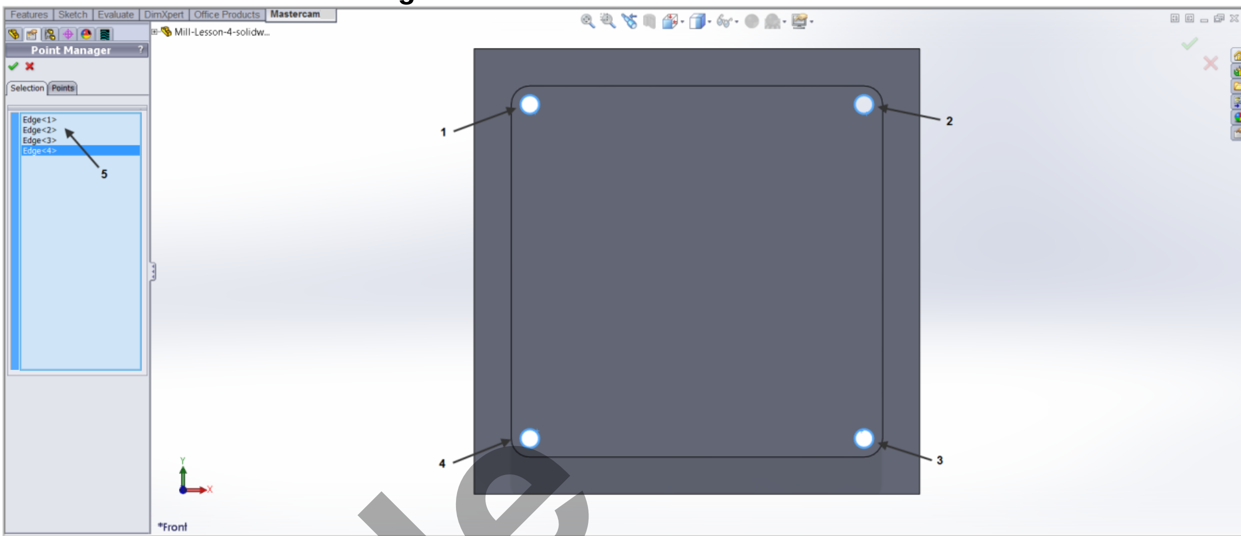
7. Click on **Mill toolpaths>Drill** as shown below.



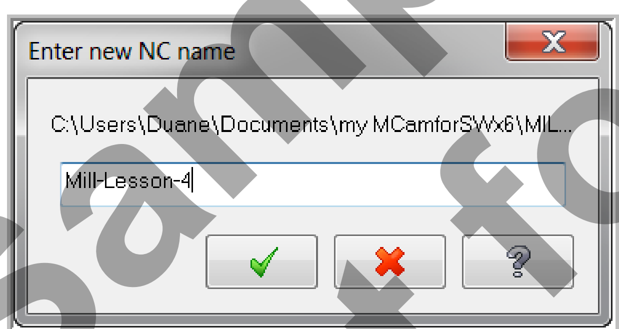
8. The **Point Manager** window appears.



9. Select the **4** holes in the order shown below by clicking on the edge of each hole. Note the **4 edges** listed in the **Point Manager (5)**.
10. Click **OK** in the **Point Manager** window.

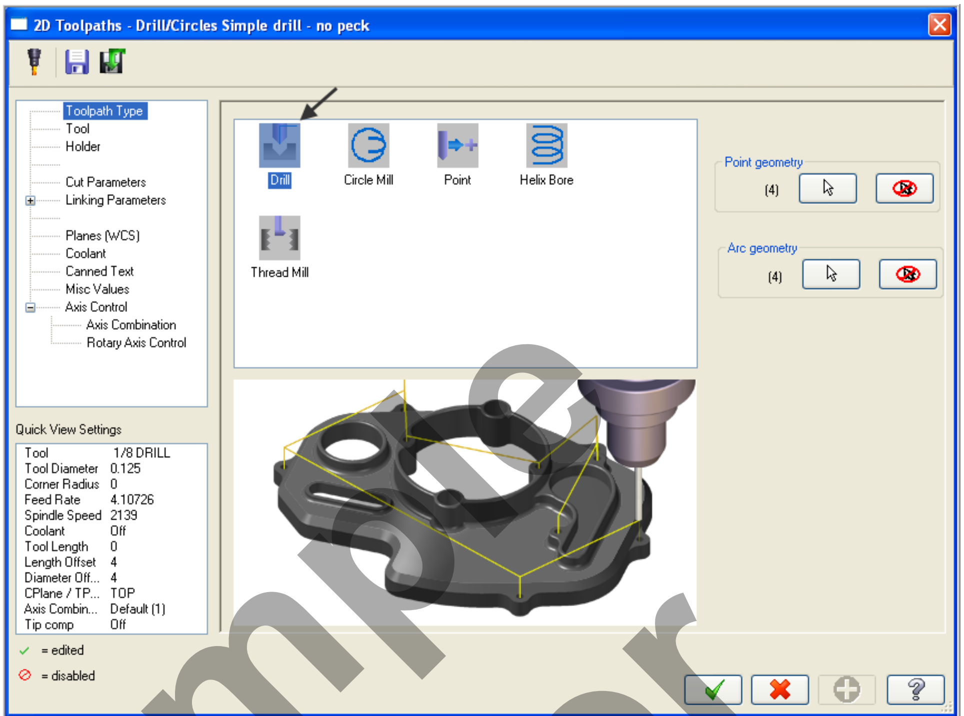


11. When prompted to **"Enter new NC name"**input **Mill-Lesson-4 as** shown below and then select the **OK** button .

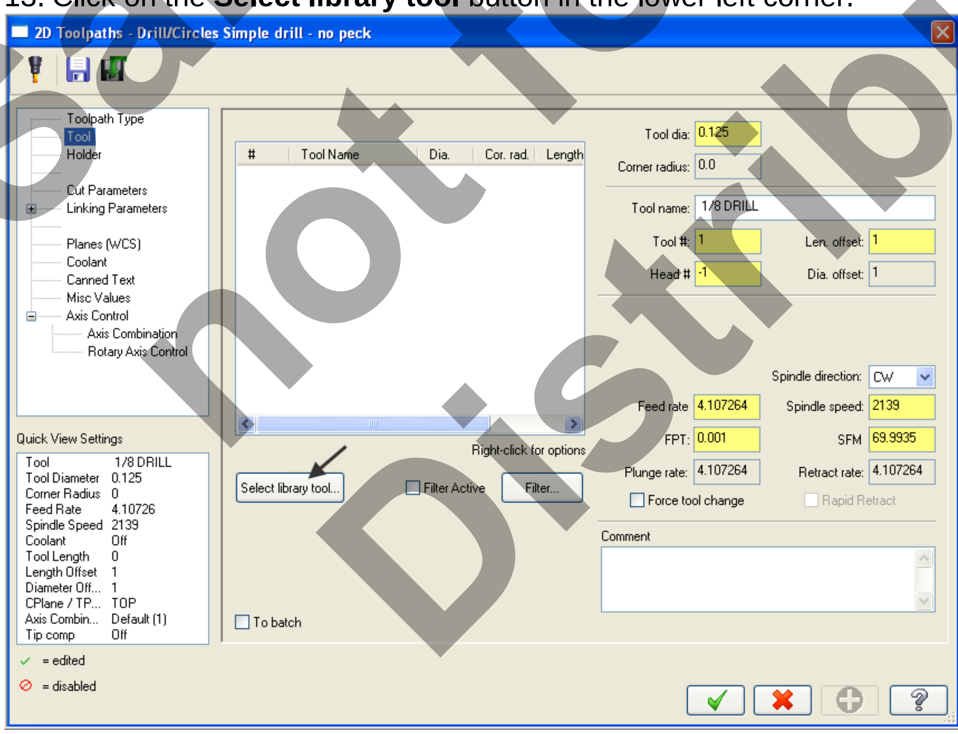


- After selecting the **OK** button, you are confronted with the Drill Toolpath Type page. The first task here will be to select a **.125" diameter end mill.**

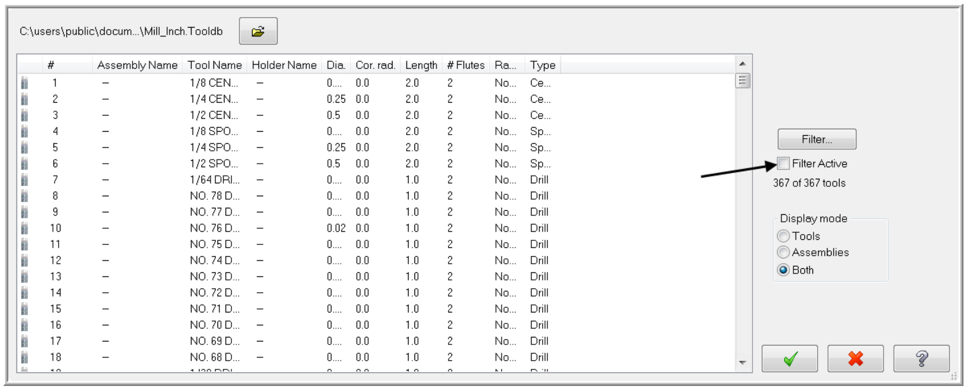
12. Ensure the **Toolpath Type** is set to **Drill** as shown below and then select **Tool** from the list on the left.



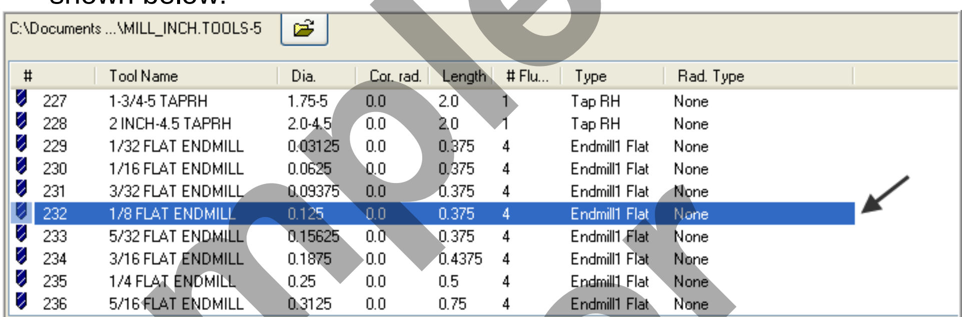
13. Click on the **Select library tool** button in the lower left corner.



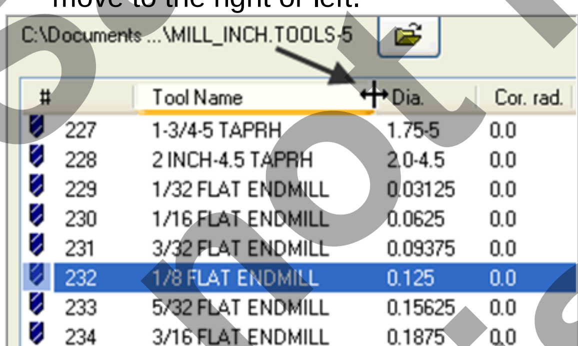
14. On the right hand side of the Tool Selection dialog box remove the green check mark from the **Filter Active** to review all the tools.



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

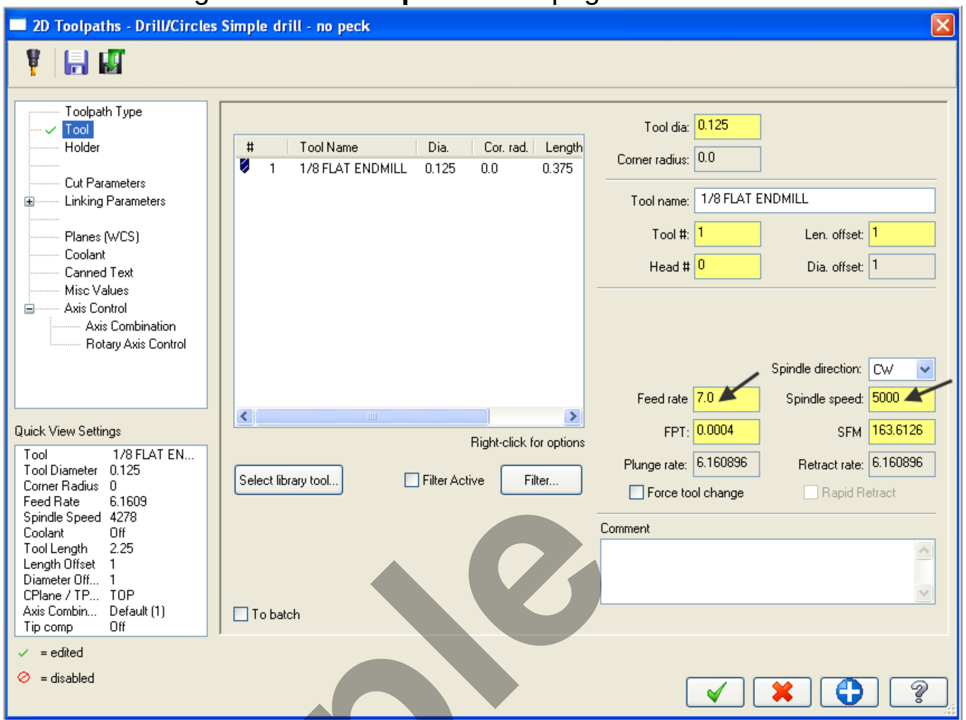


➢ To resize a column in the **Tool Selection dialog box**, click on the divider between the columns with your left mouse button, as shown below, hold the left mouse button down and move to the right or left.

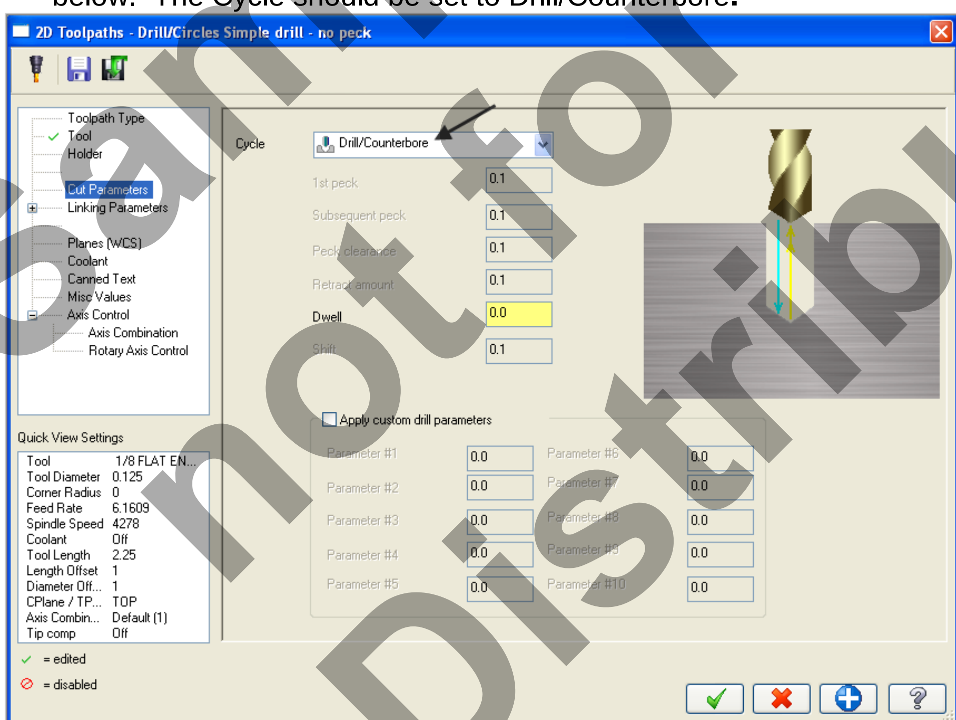


17. Select the **OK** button  to complete the selection of this tool.

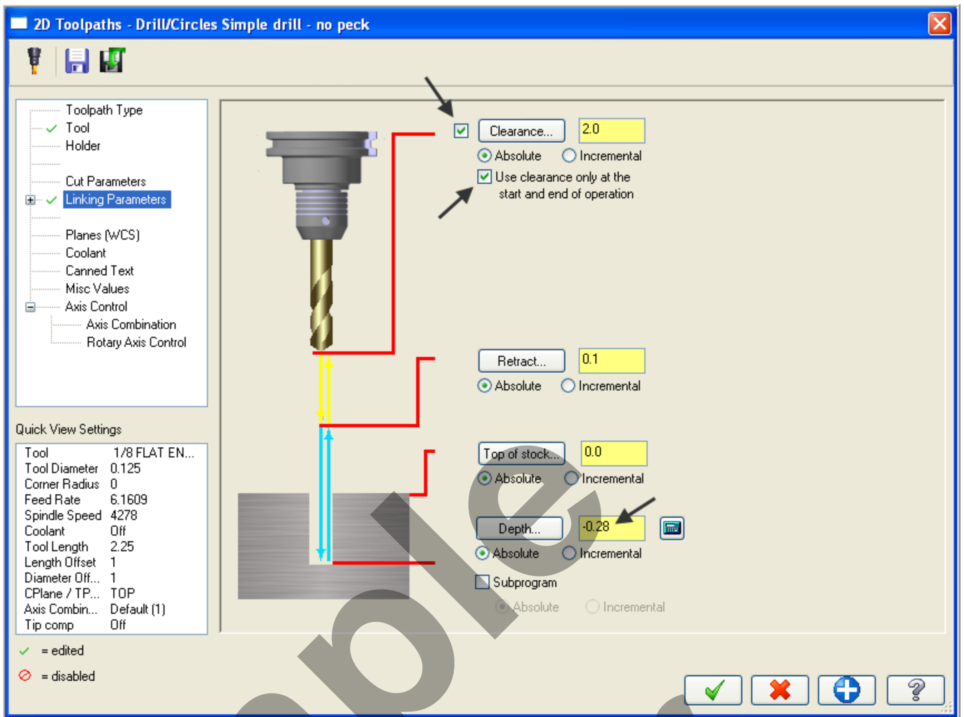
18. Make changes to the **Tool parameter** page as shown below:



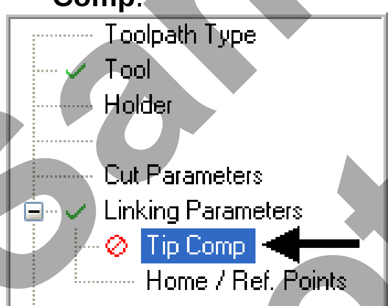
19. Select **Cut Parameters** from the list on the left and make changes to this page as shown below. The Cycle should be set to Drill/Counterbore**.**



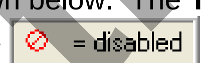
20. Select **Linking Parameters** from the list on the left and make changes to this page as shown below. Input the **depth of -0.28** and the other values as shown below. **Note** all the values are set to **Absolute.**

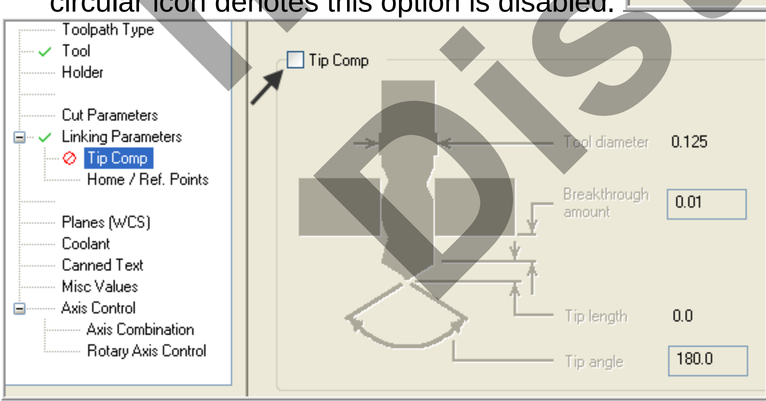


21. Select the plus sign to the left of **Linking Parameters** to expand the list and click on **Tip Comp**.

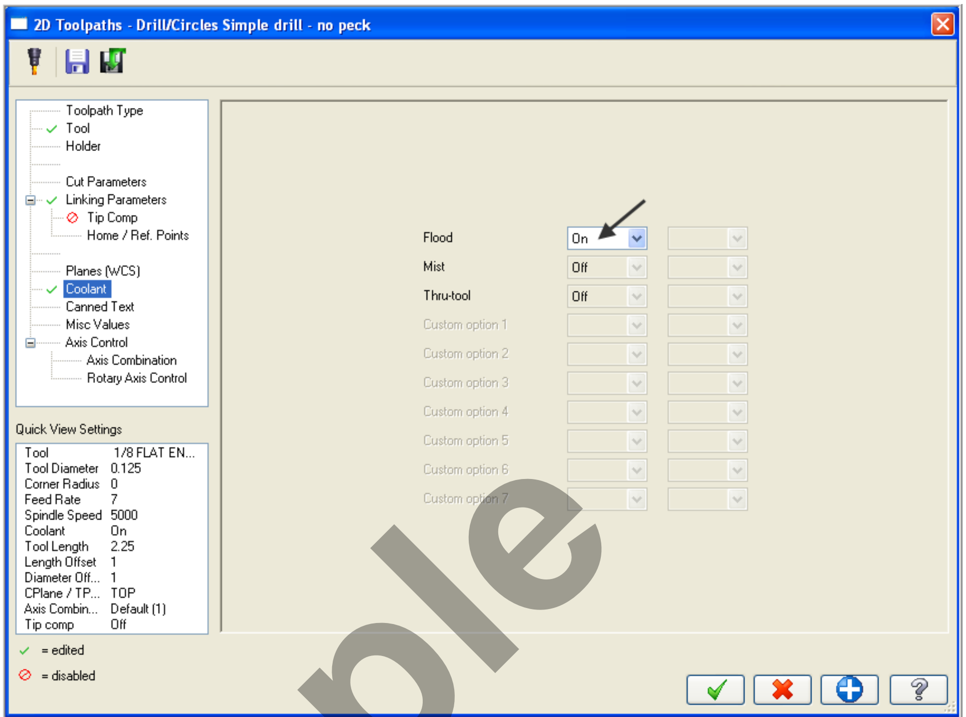


22. Ensure **Tip Comp** is **not activated** as shown below. The **Tip Comp** box is empty. The red circular icon denotes this option is disabled.

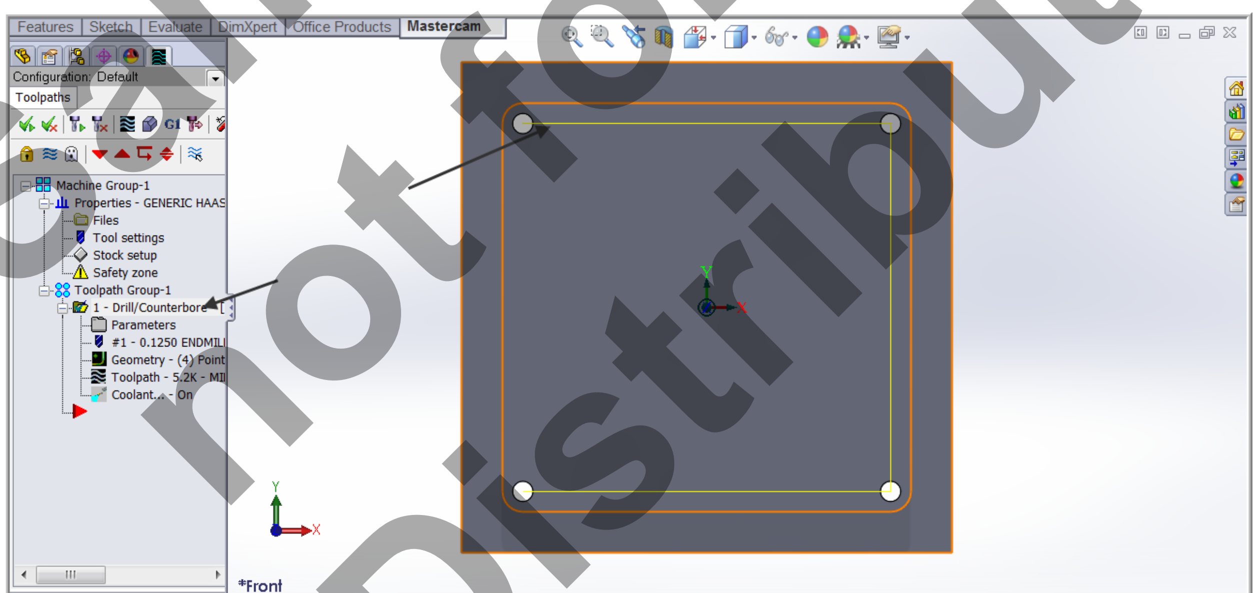




23. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.



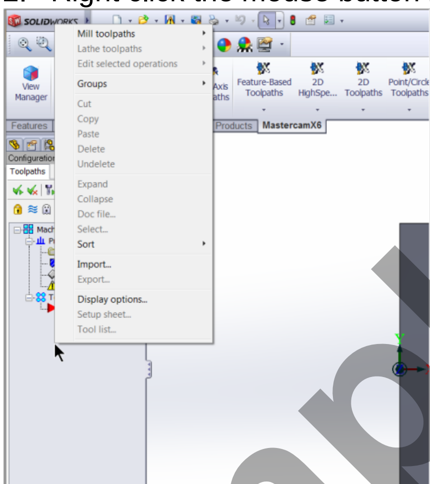
24. Select the **OK** button to complete this function.
25. Your part should look like the screenshot below:

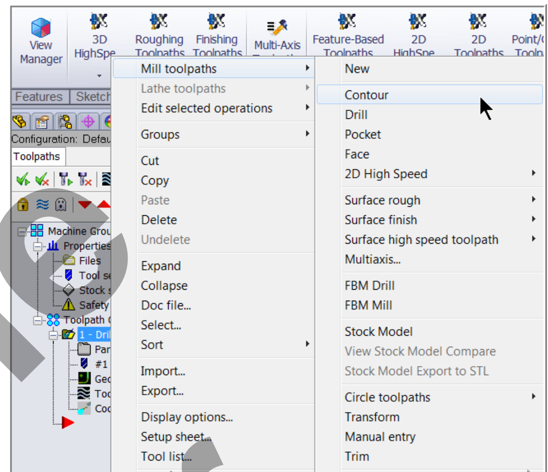


TASK 7:
MACHINE THE CONTOUR.

- 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.

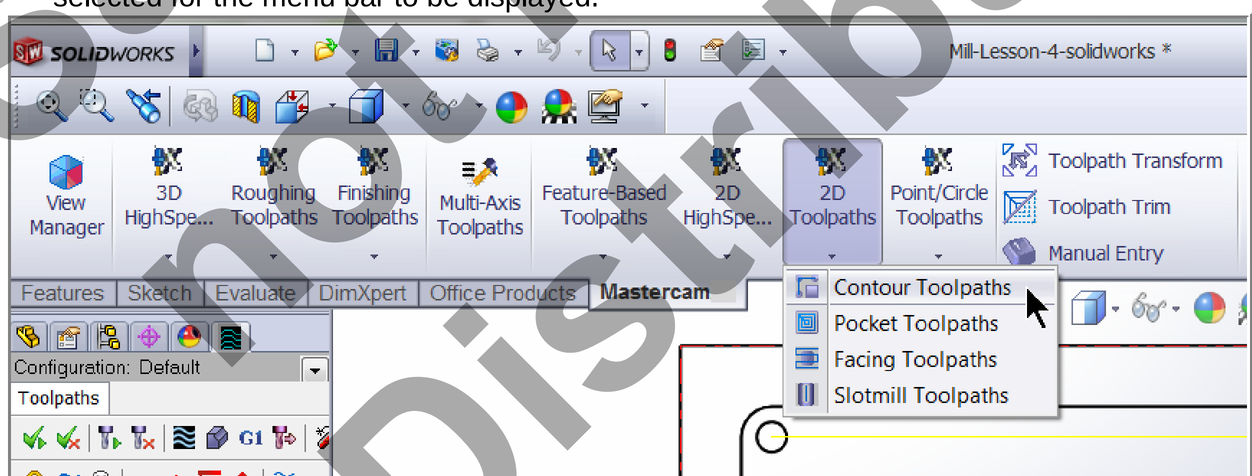
1. Change the view to T**rimetric** by pressing the **space bar** and then Clicking on T**rimetric**  .
2. Right click the mouse button in the **Toolpath Manager** and select **Mill toolpaths>Contour.**





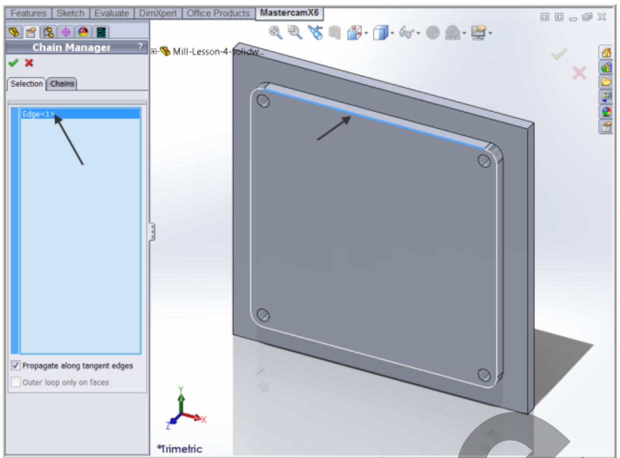
OR

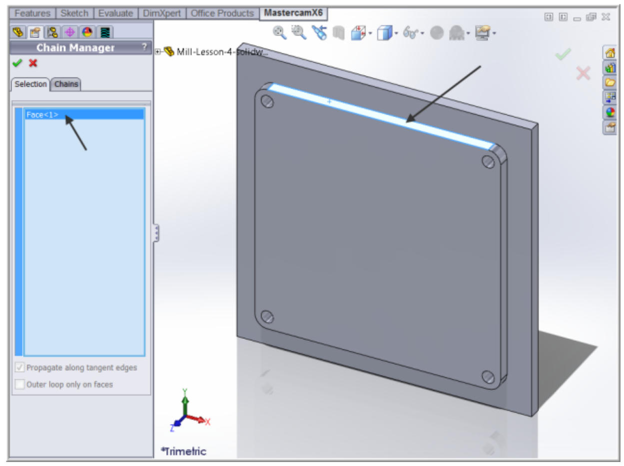
3. Another way to select the **Contour Toolpath** is to click on **2D Toolpaths>Contour Toolpaths** on the menu bar as shown below. Note, the **MastercamX7** Tab needs to be selected for the menu bar to be displayed.



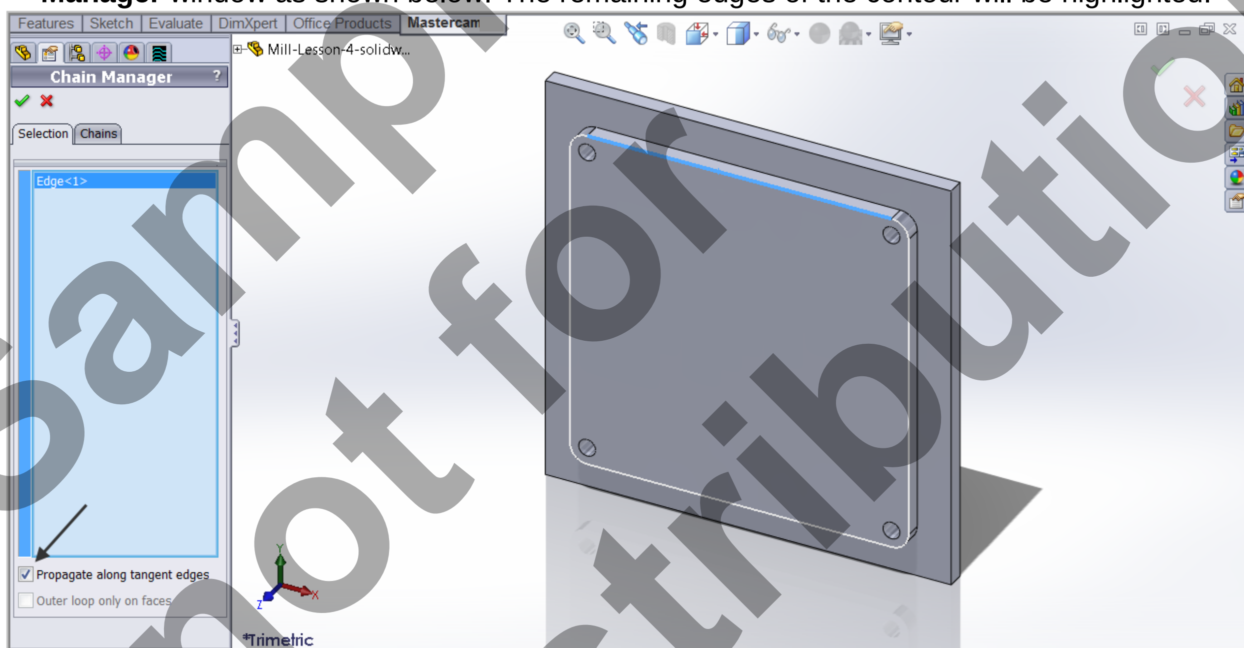
4. The **Chain Manager** window will be displayed. Click on the top Edge of the **Contour** as shown below – Be sure that you click on the **Edge** and NOT the **Face**. **Edge<1>** will be displayed in the Chain Manager window.

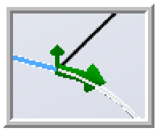
Correct **Incorrect**



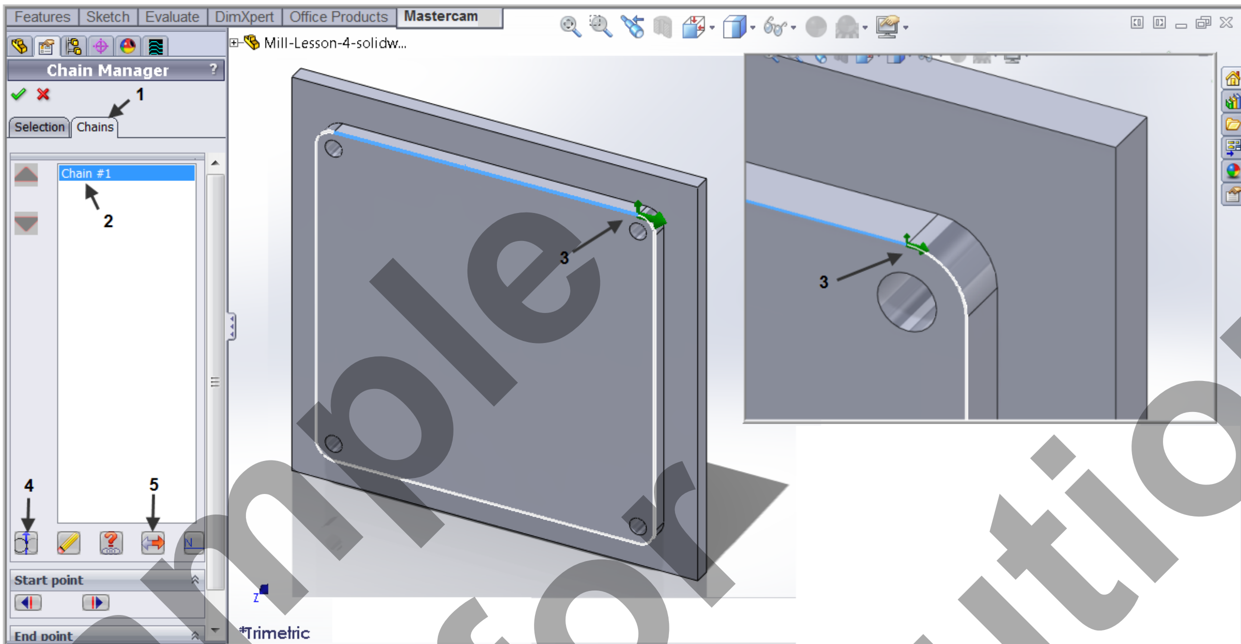


5. Click on the **Propogate along tangent edges** check box at the bottom of the **Chain Manager** window as shown below. The remaining edges of the contour will be highlighted.



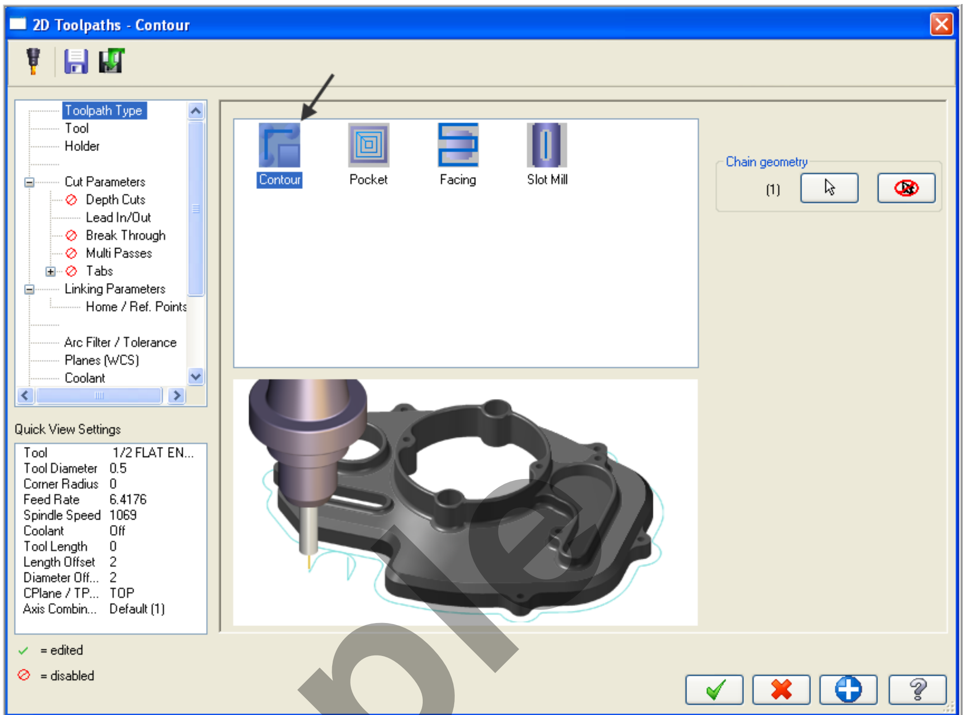
6. Click on the **Chains** tab in the **Chain Manager** window and click on **Chain#1**.
7. Your graphics screen should look like the screenshot below, with the **arrow pointing to the right and up**,  in a clockwise direction.

➲ The material for this part is aluminium so to attain a good finish when contouring **climb milling** should be employed.

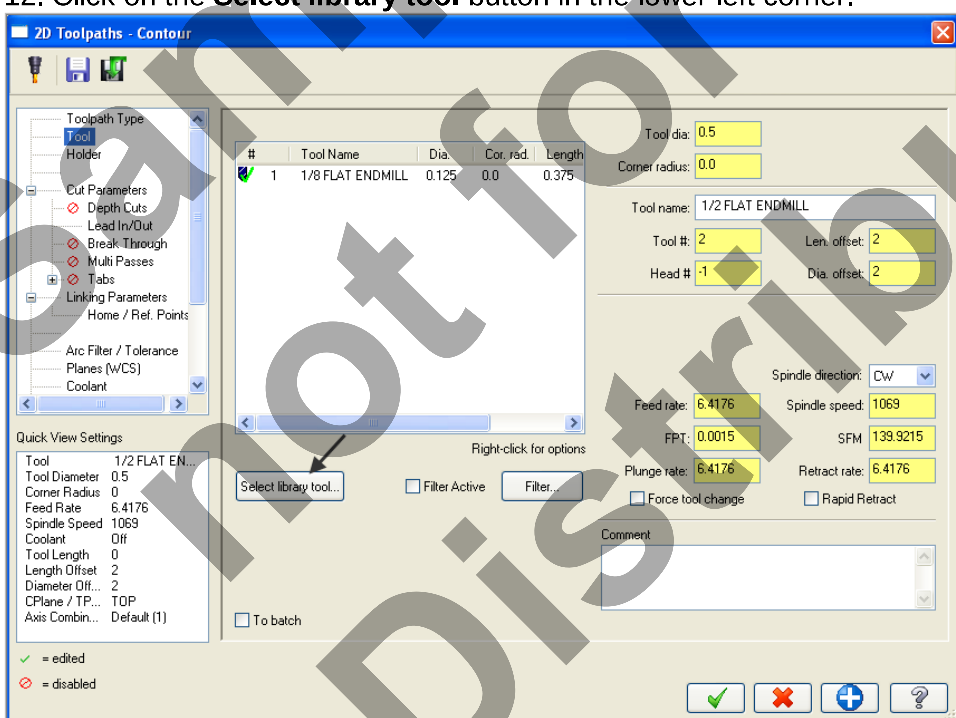


8. If the arrow is not pointing to the **RIGHT** click the **reverse arrow (5)** from the **Chaining** dialog box shown above to reverse the direction.
9. If the arrow is not pointing **UP** click the **Offset icon (4)** to change the direction. This will ensure the cutter will travel along the outside of the contour and not the inside of the contour.
10. After the contour has been successfully chained select the **OK** button  at the top of the **Chaining** dialog box.

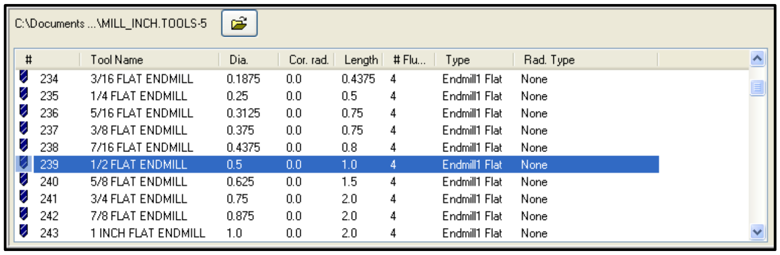
11. Ensure the **Toolpath Type** is set to **Contour** as shown below and then select **Tool** from the list on the left.



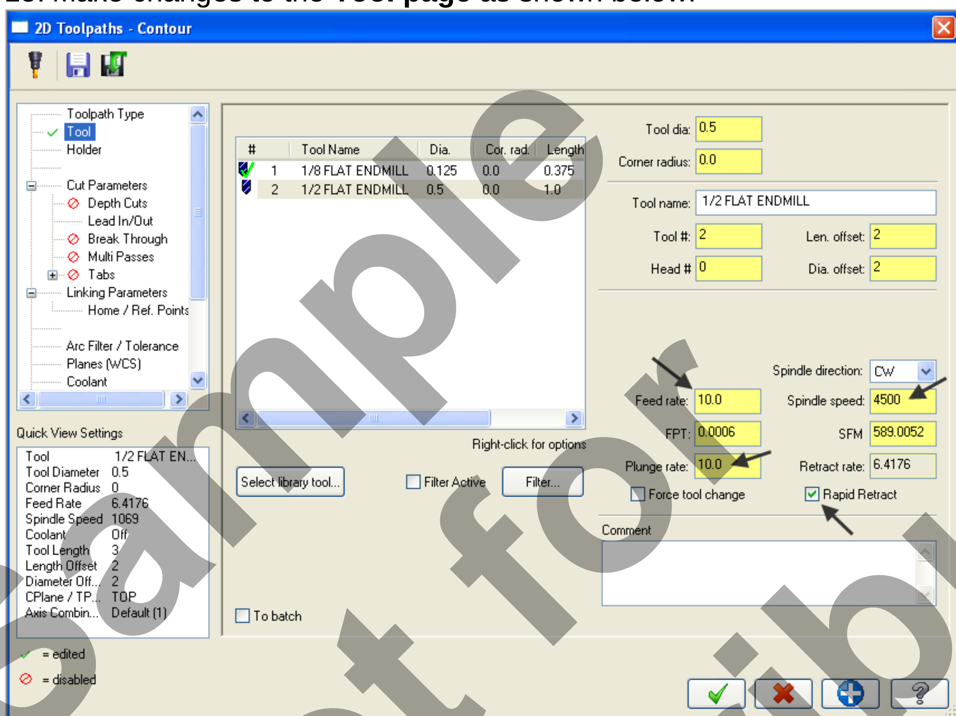
12. Click on the **Select library tool** button in the lower left corner.



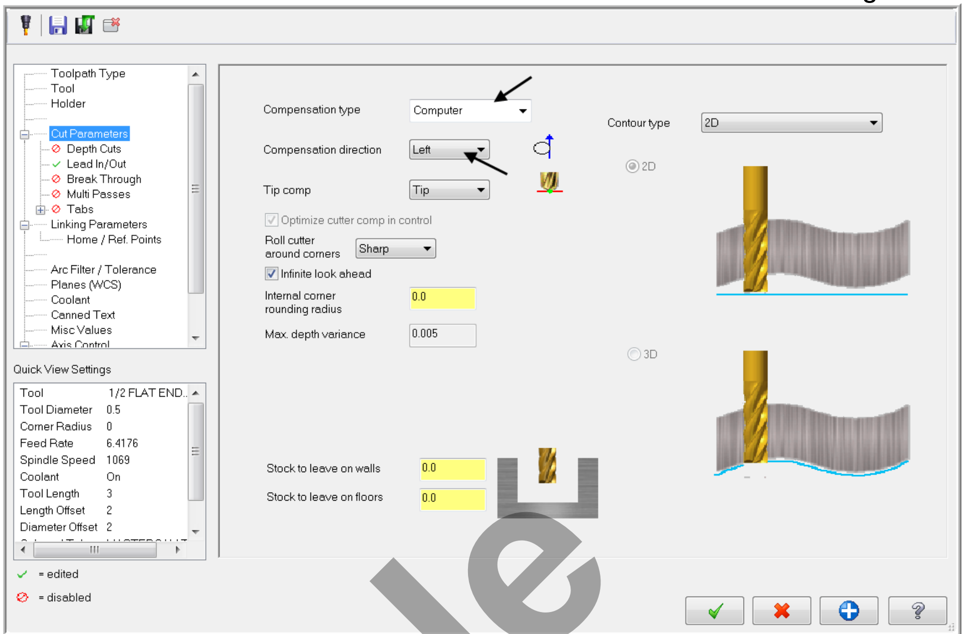
13. Use the slider bar on the right of this dialog box to scroll down and locate a **.5" diameter flat end mill**. Select the .5" diameter flat end mill by picking anywhere along the .5 end mill row, as shown below:



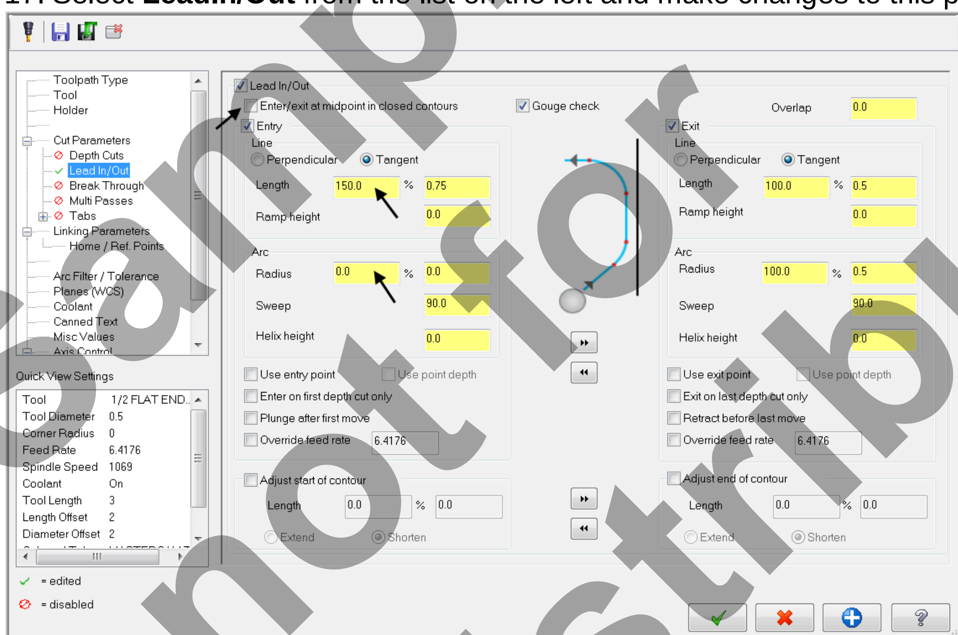
14. Select the **OK** button to complete the selection of this tool.
15. Make changes to the **Tool page** as shown below:



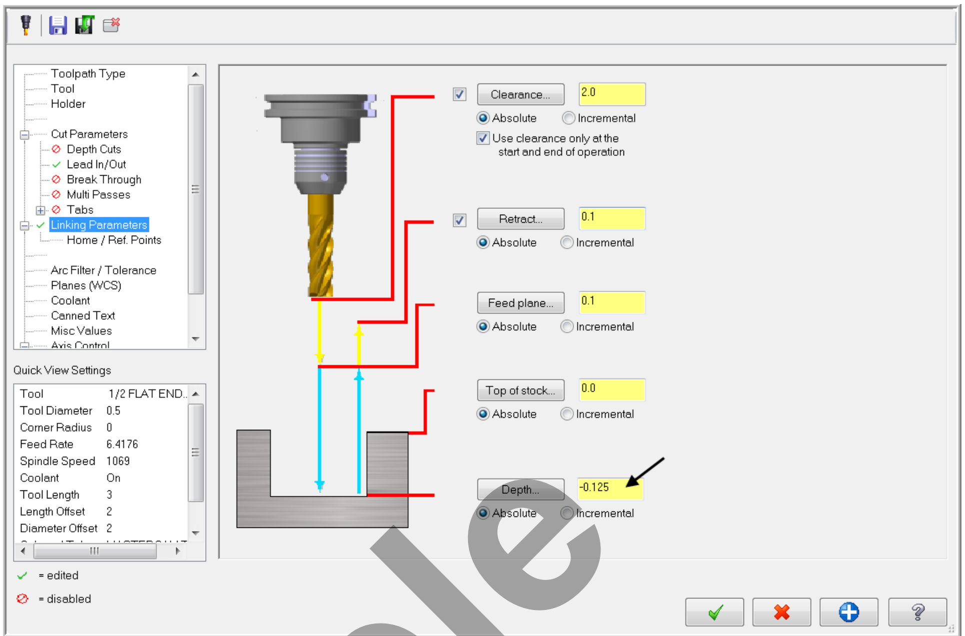
16. Select **Cut Parameters** from the list on the left and make changes to this page if required.



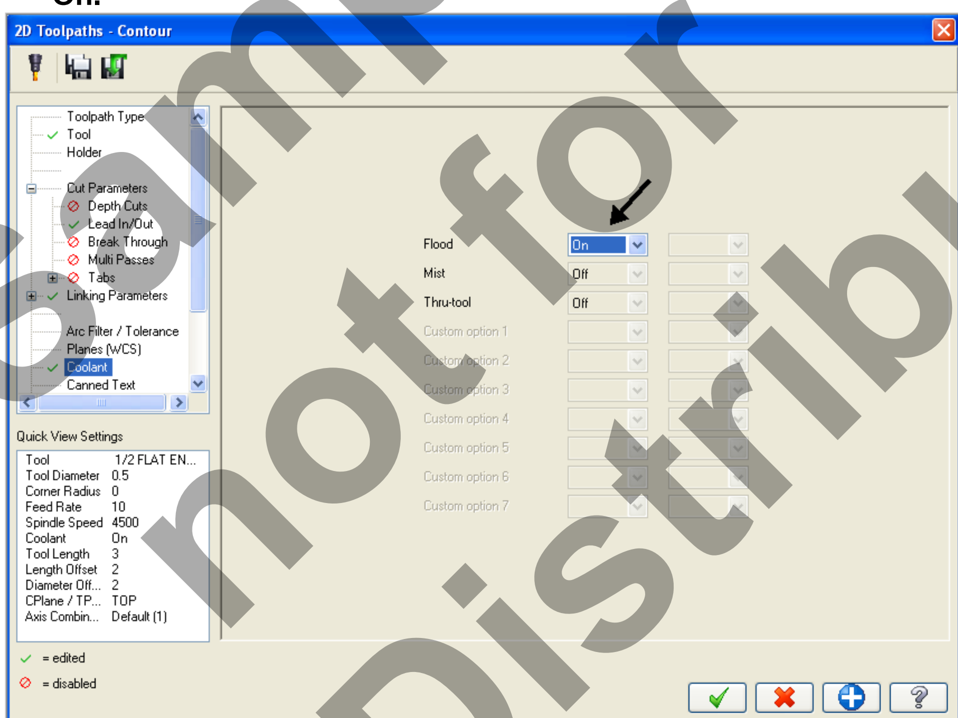
17. Select **LeadIn/Out** from the list on the left and make changes to this page.



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



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

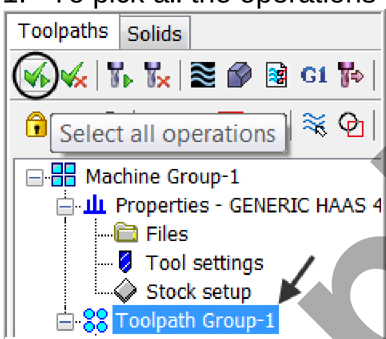


20. Select the **OK** button to complete this function

TASK 8:
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 us to review the cutting motions and identify any problem areas when cutting the part.
- When the toolpath is being Backplotted Mastercam displays the current X, Y, and Z coordinates in the left side of the status bar at the lower left corner of the screen.
- **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:

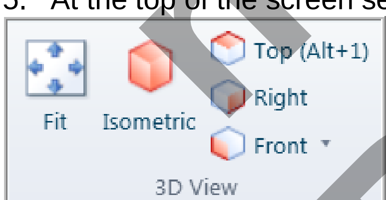


- Another method to **Select all** the operations is by clicking on the **Toolpath Group-1** in the Tool Manager as shown by the arrow above.

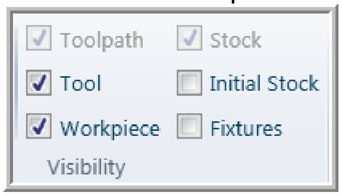
2. The next step is to select the **Backplot selected operations** icon shown below:

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**.

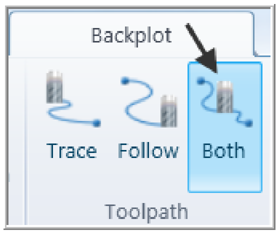


6. Activate the options shown below in the **Visibility** section of the Home tab.

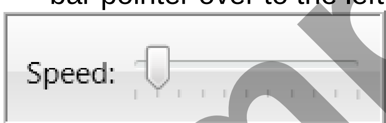


7. Click on the **Backplot** tab at the top left of the screen.

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

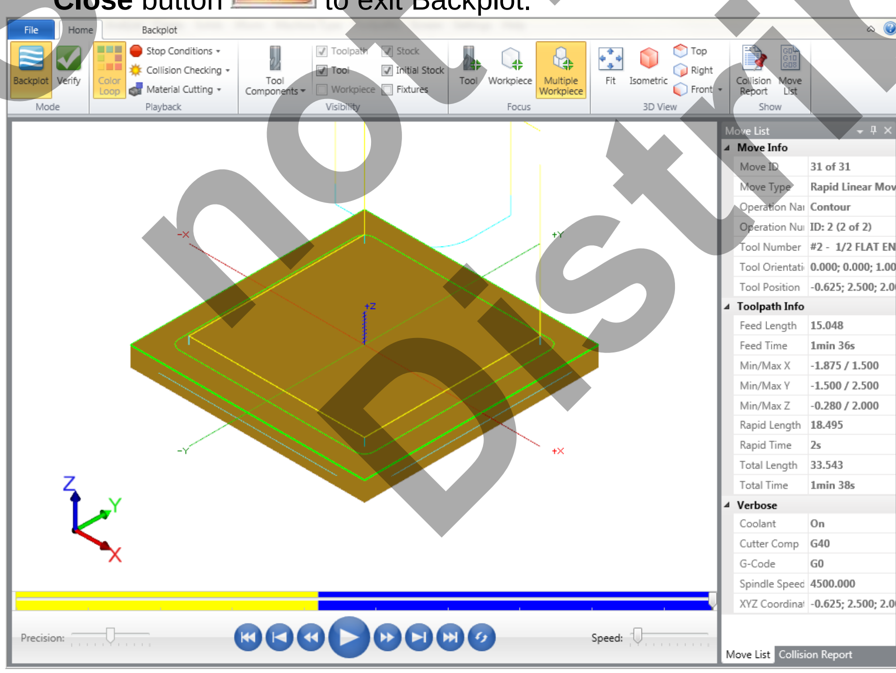


9. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.



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

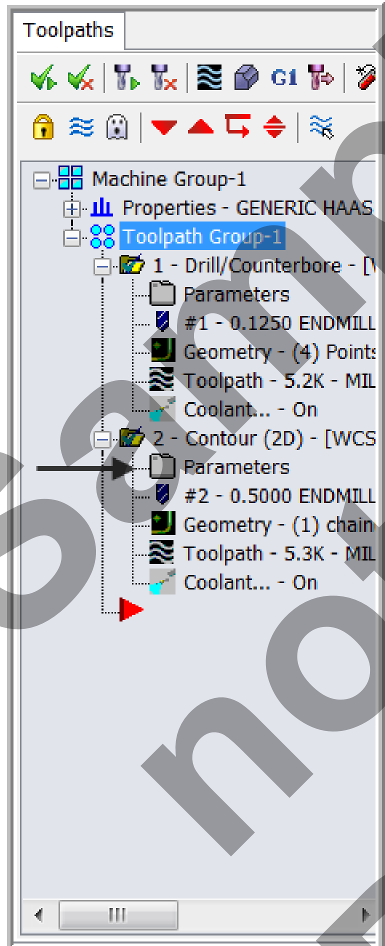
11. After reviewing the backplot of the two toolpaths using a .125" and .5" end mill select the **Close** button  to exit Backplot.



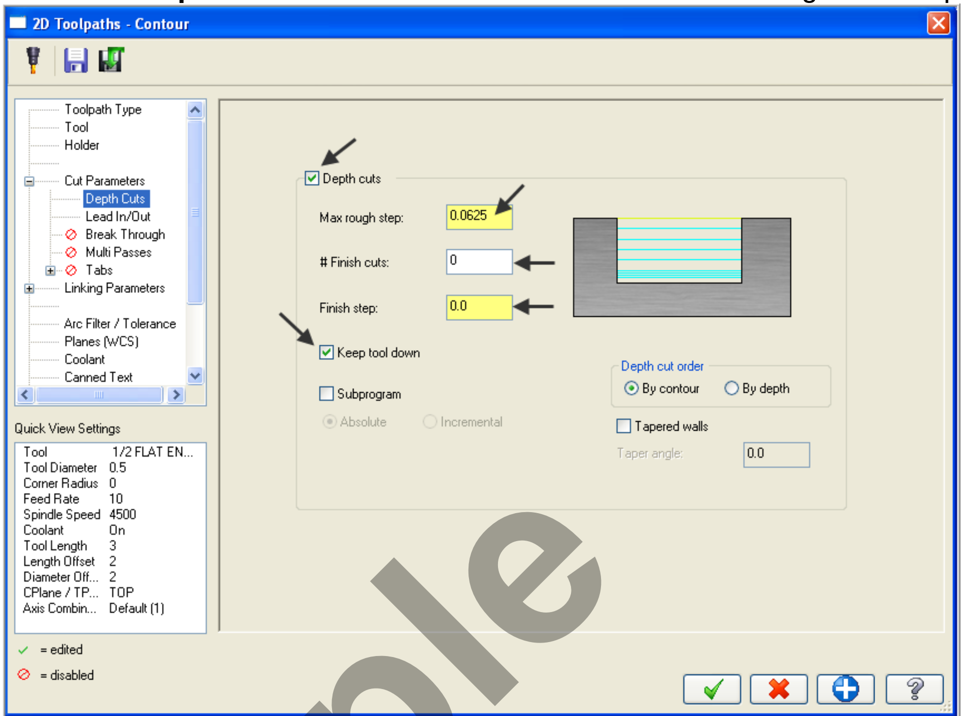
TASK 9:
MODIFY THE CONTOUR TOOLPATH TO ADD ROUGHING CUTS AND A FINISH PASS

- 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** click on the **Parameters** folder 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:

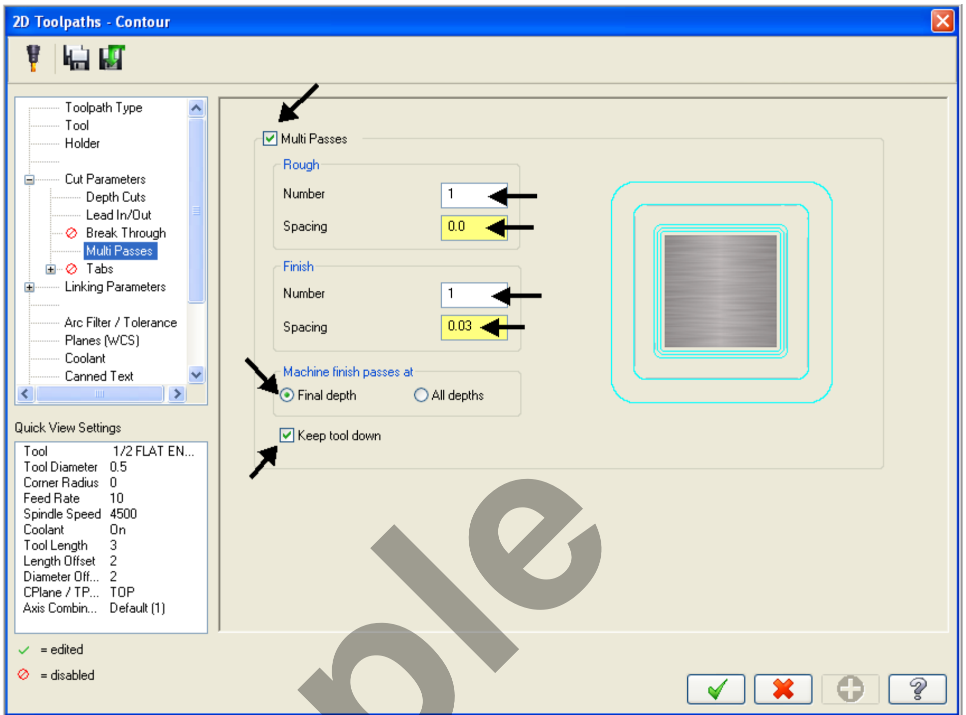


About the Depth cuts dialog box

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.

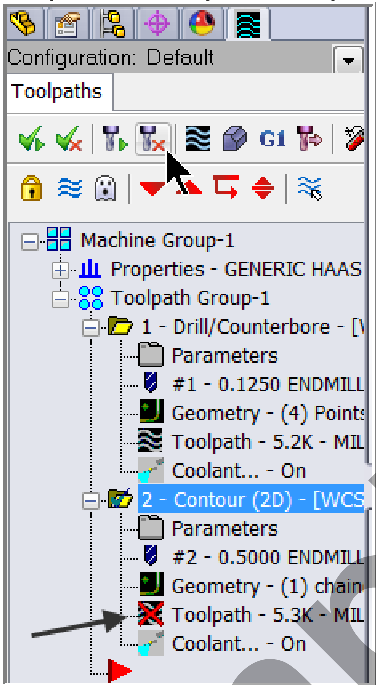
Finishing passes:
Number: Enter the number of cutting passes you want Mastercam to create.
Spacing: Enter the amount of stock to remove with each cut.

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 pane.
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 toolpath to use depth of cuts and multi passes.

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

TASK 10:
VERIFY THE TOOLPATH

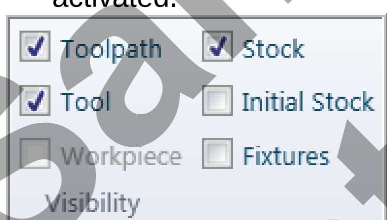
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part and shows collisions, if any exist.
- This allows you to identify and correct program errors ***before*** they reach the shop floor.
- **For more information on Verify see the Tips and Techniques section on 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:

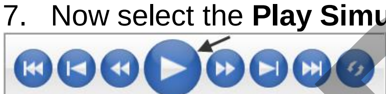
3. **Maximize** the Backplot/Verify window if required.
4. At the top of the screen select the **Isometric** icon and then select **Fit**.

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

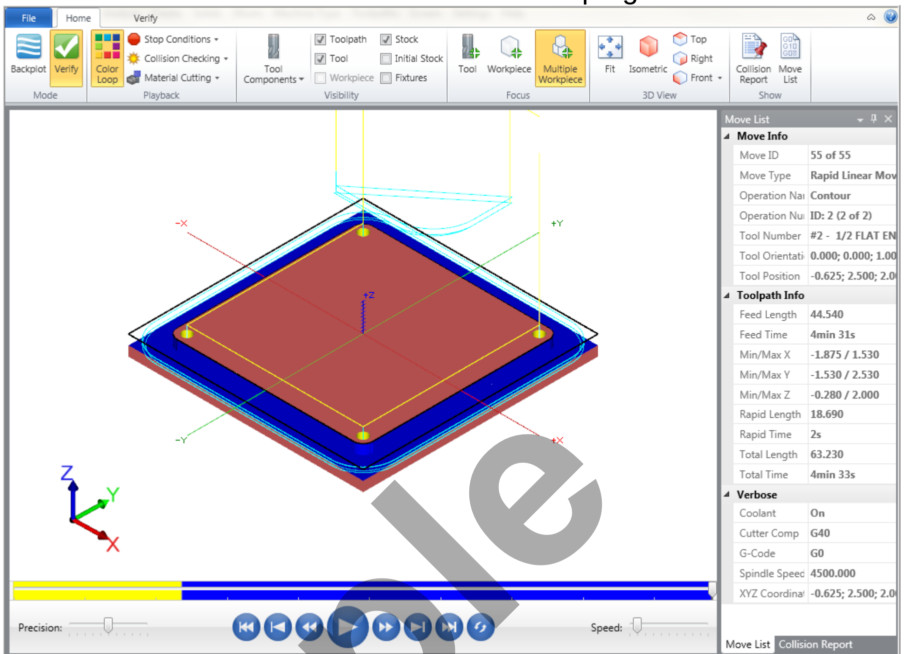


6. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.

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



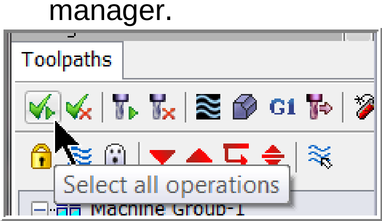
8. After reviewing the two toolpaths the verified toolpaths should appear as in the picture below.
9. Select the **Close** button  in the top right hand corner to exit Verify.



10. **Save** the updated Mastercam file.

TASK 11:
POST AND CREATE THE CNC CODE FILE

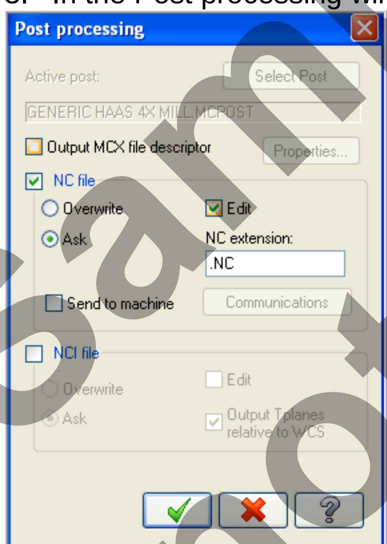
1. Ensure all the operations are selected by picking the **Select All** icon from the Toolpath manager.



2. Select the **Post selected operations** button from the Toolpath manager.

➲ **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:



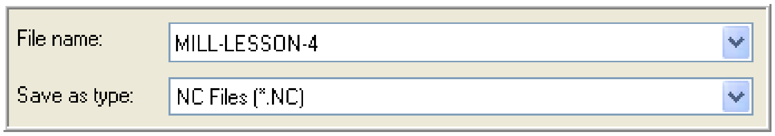
About Post Processing

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.

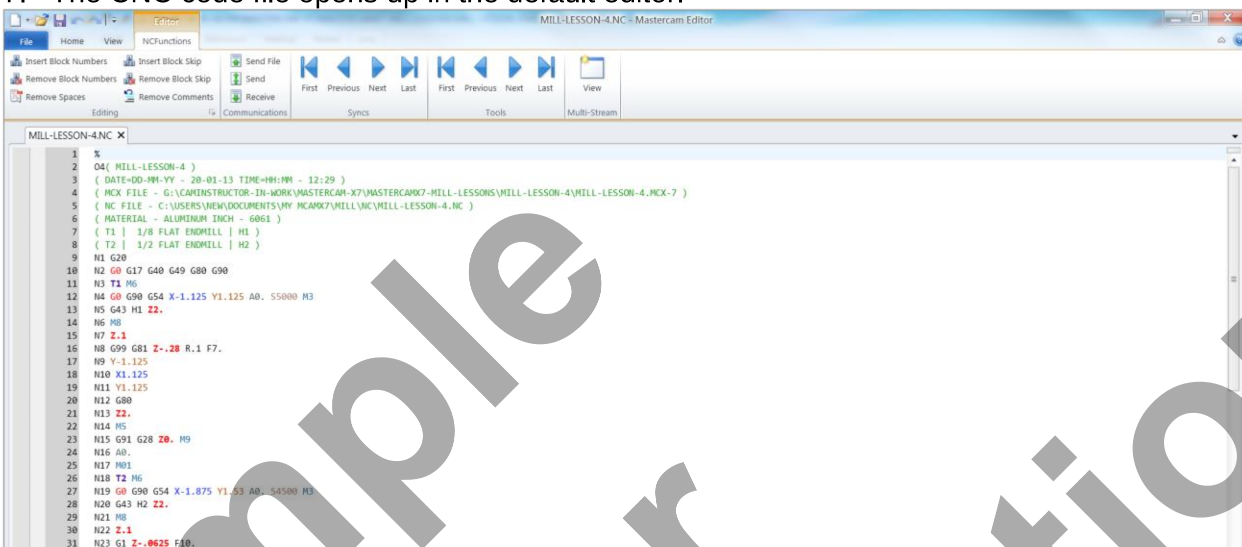
Edit:
When checked, automatically launches the default text editor with the file displayed so that you can review or modify it.

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:

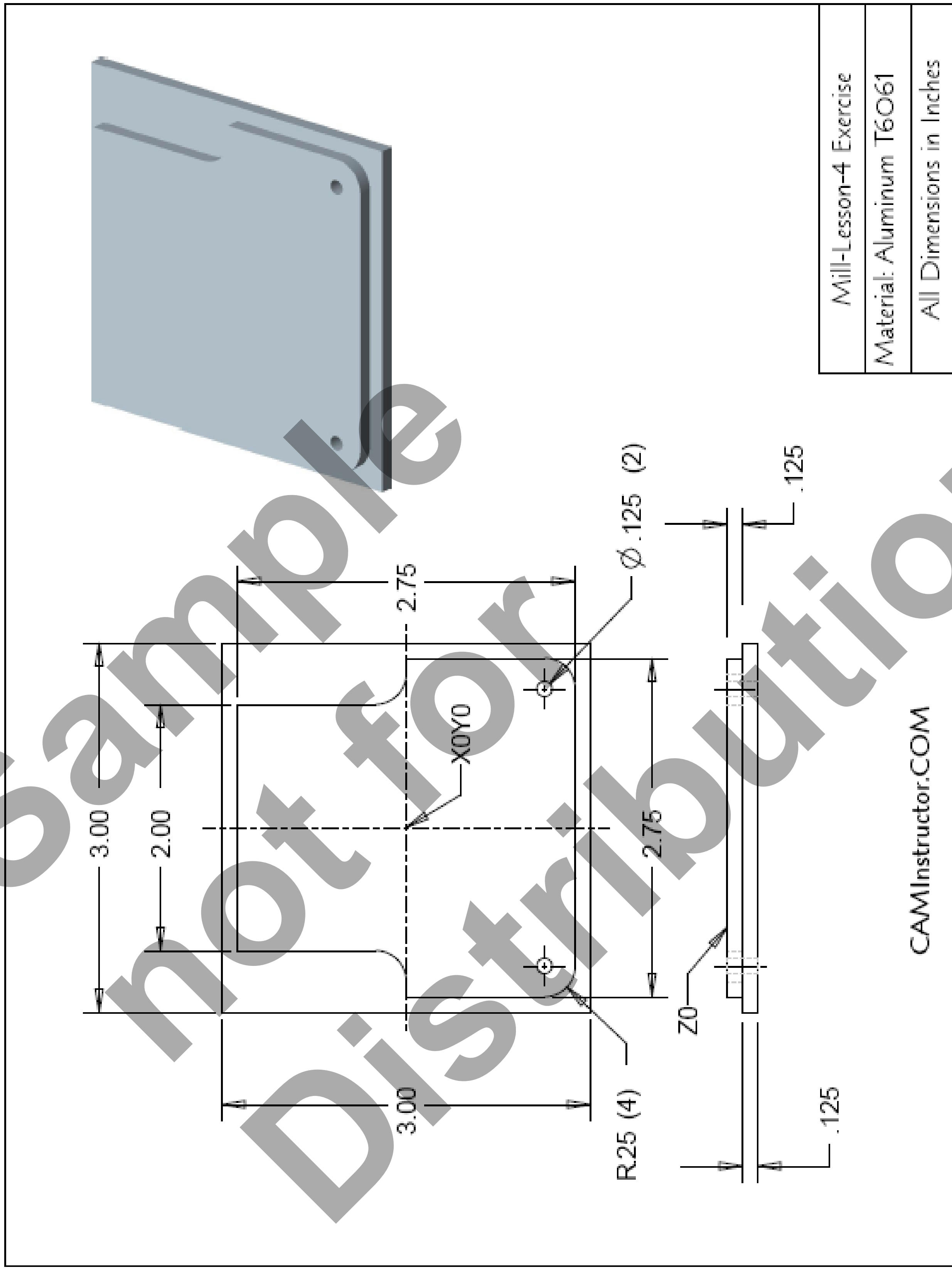


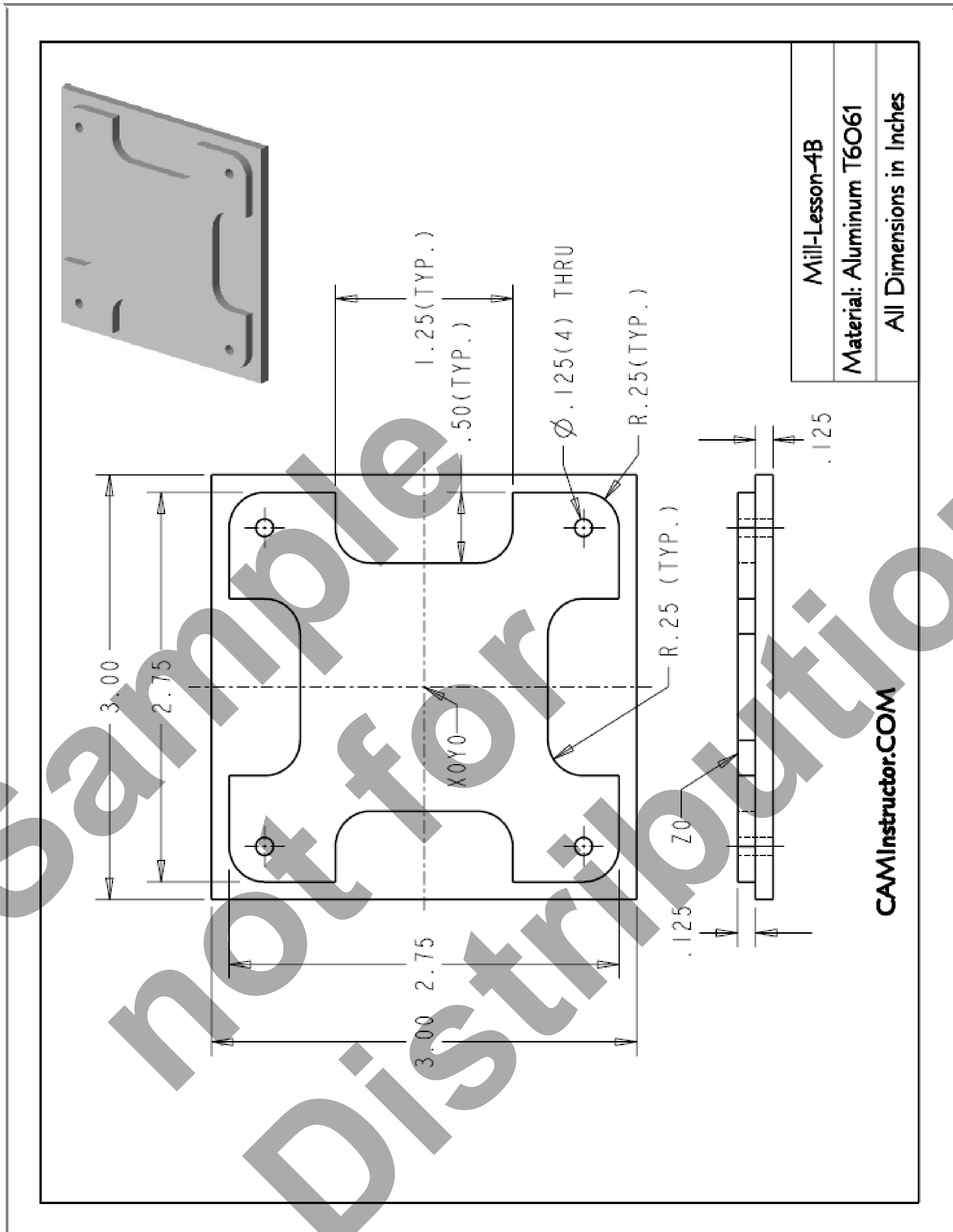
6. Select the **Save** button.
7. The CNC code file opens up in the default editor.



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-4B
Material: Aluminum T6061
All Dimensions in Inches
CAMInstructor.COM
1.25(TYP.)
.50(TYP.)
Ø.125(4) THRU
R.25(TYP.)
R.25 (TYP.)
3.00
2.75
X0Y0
Z0
.125