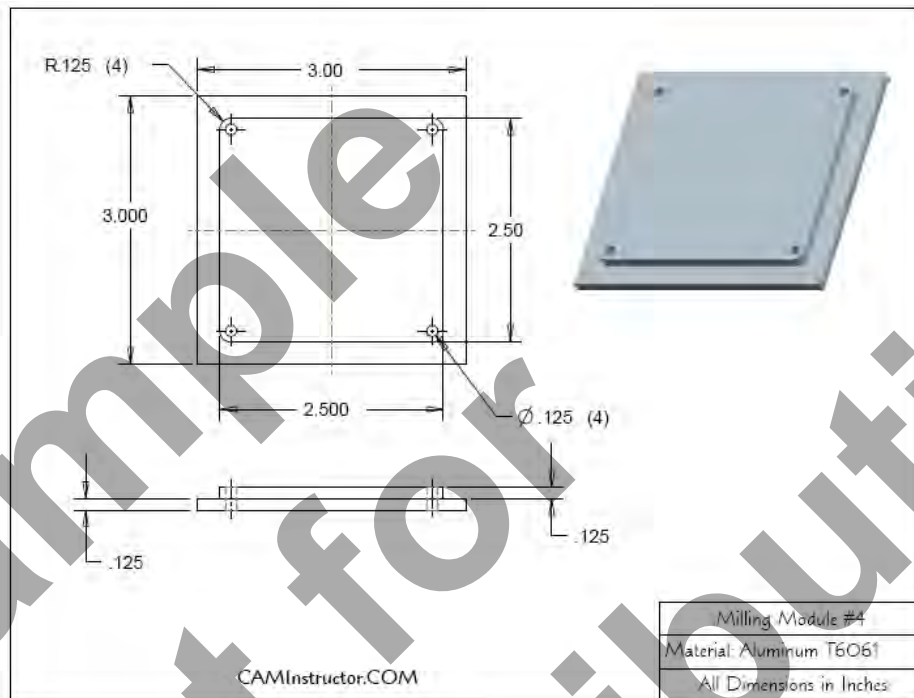


SolidWorks and *Mastercam X⁶* TRAINING GUIDE



LESSON-4-CAD

Objectives

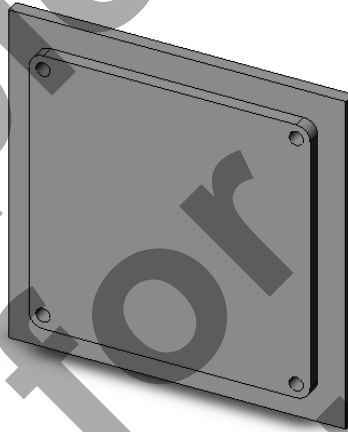
You will create the geometry for Mill-Lesson-4 using **SolidWorks 3D CAD software**. You will be working through the process to machine parts in later Lessons; Mill-Lesson-4 will just cover geometry creation. This Lesson covers the following topics:

➤ **Create 2-dimensional sketches by:**

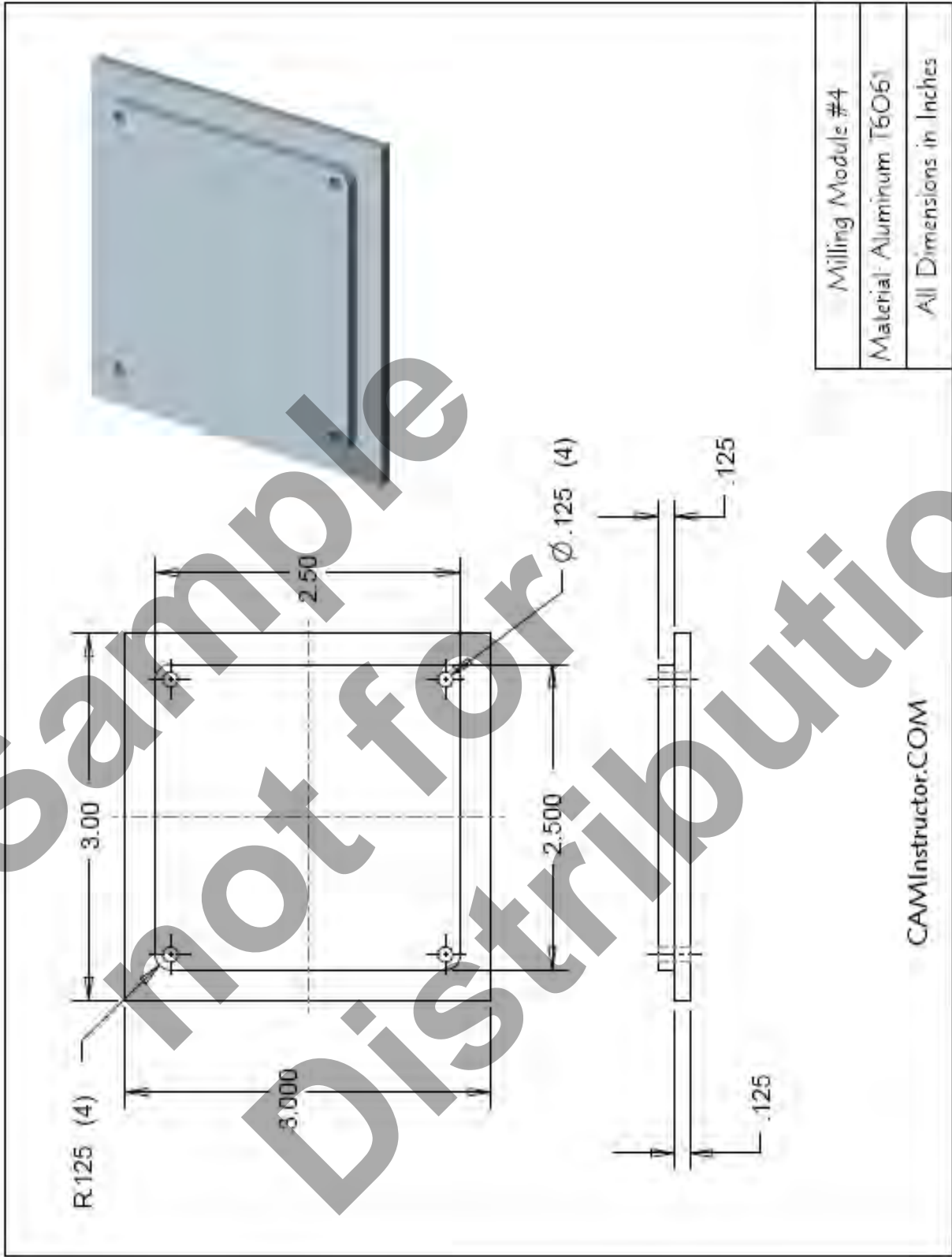
Creating the geometry by using features found in the sketch toolbar.
Setting up relations and fully defining the sketch by means of dimensions.

➤ **Extruding or Cut-Extrude 2-dimensional sketches by:**

Using the extrude tool to extrude a base or extrude and cut fully defined sketches to build geometry.

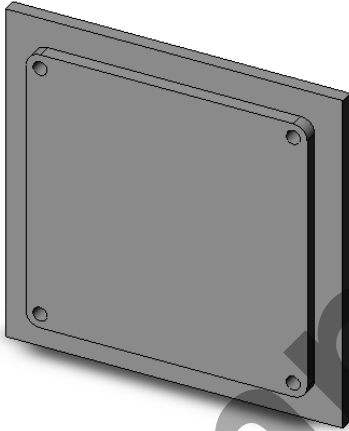


LESSON-4 DRAWING



LESSON - 4 - THE CAD PROCESS


- TASK 1:** Start a new part and choose the sketch plane.
- TASK 2:** Create the fully defined base shape sketch and extrude.
- TASK 3:** Sketch the raised portion and extrude to complete the shapes.
- TASK 4:** Sketch the four holes and cut-extrude.



Sample
not for
Distribution

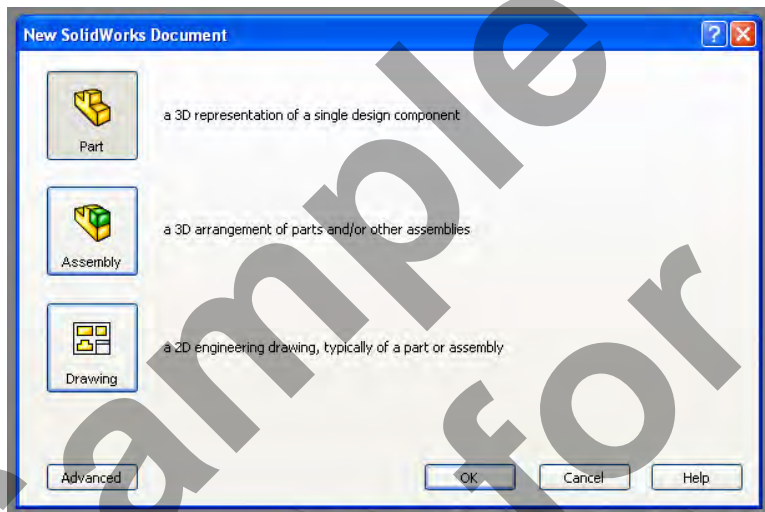
CAD TASK 1: START A NEW PART AND CHOOSE THE SKETCH PLANE

➔ Create a new part in SolidWorks

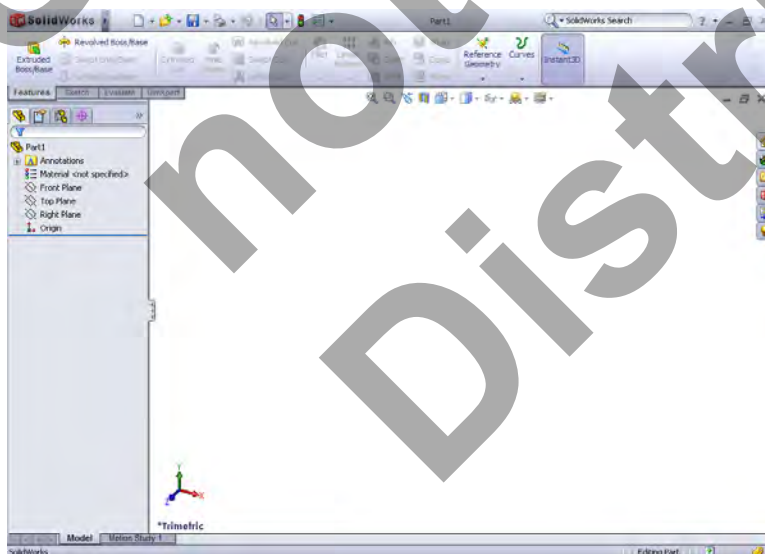
1. Click on **File>New...** from the menu bar or the **File New** icon . This will create a new document.



2. Click on **Part** (a 3D representation of a single design component) and select OK.



3. Your screen will look like the one below:

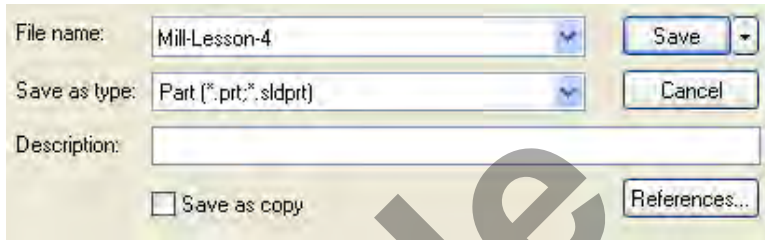


SolidWorks and Mastercam Training Guide

- Click on the save icon on the menu bar (will perform a save as) or on the arrow beside to expand and click on **Save As**.



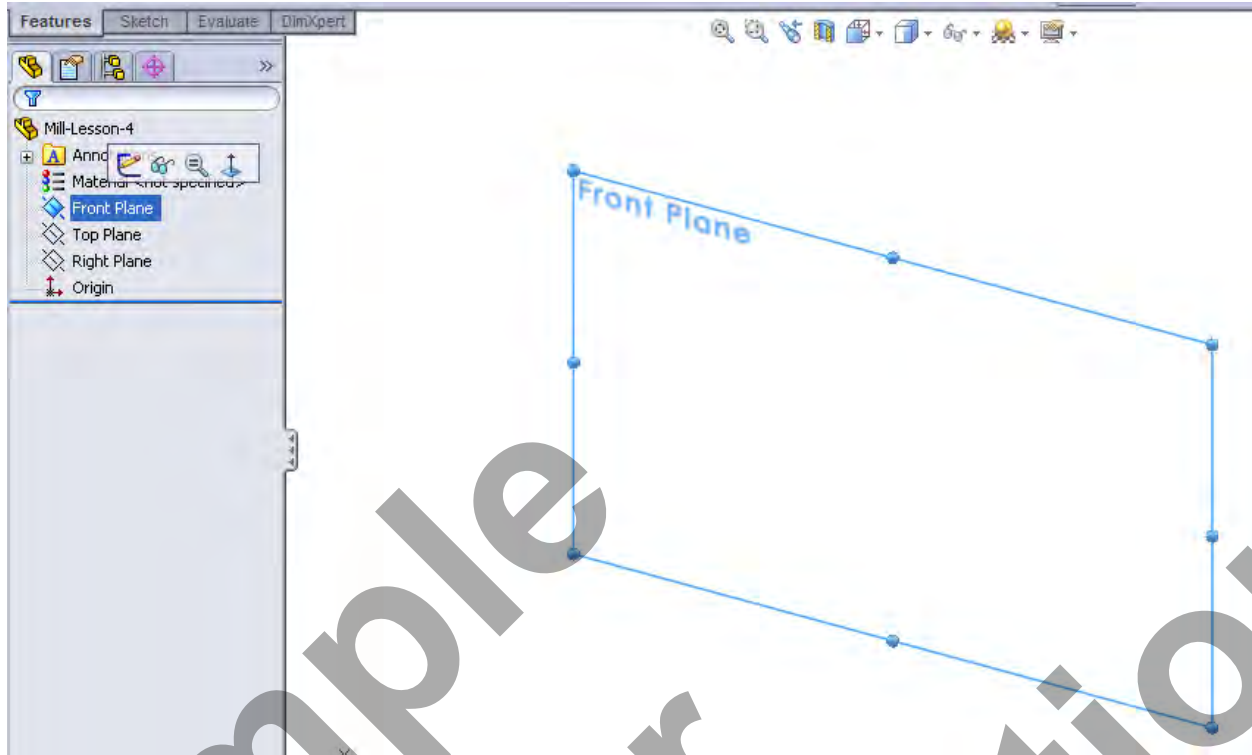
- Save Part1 as **Mill-Lesson-4** in the appropriate directory location you are saving.



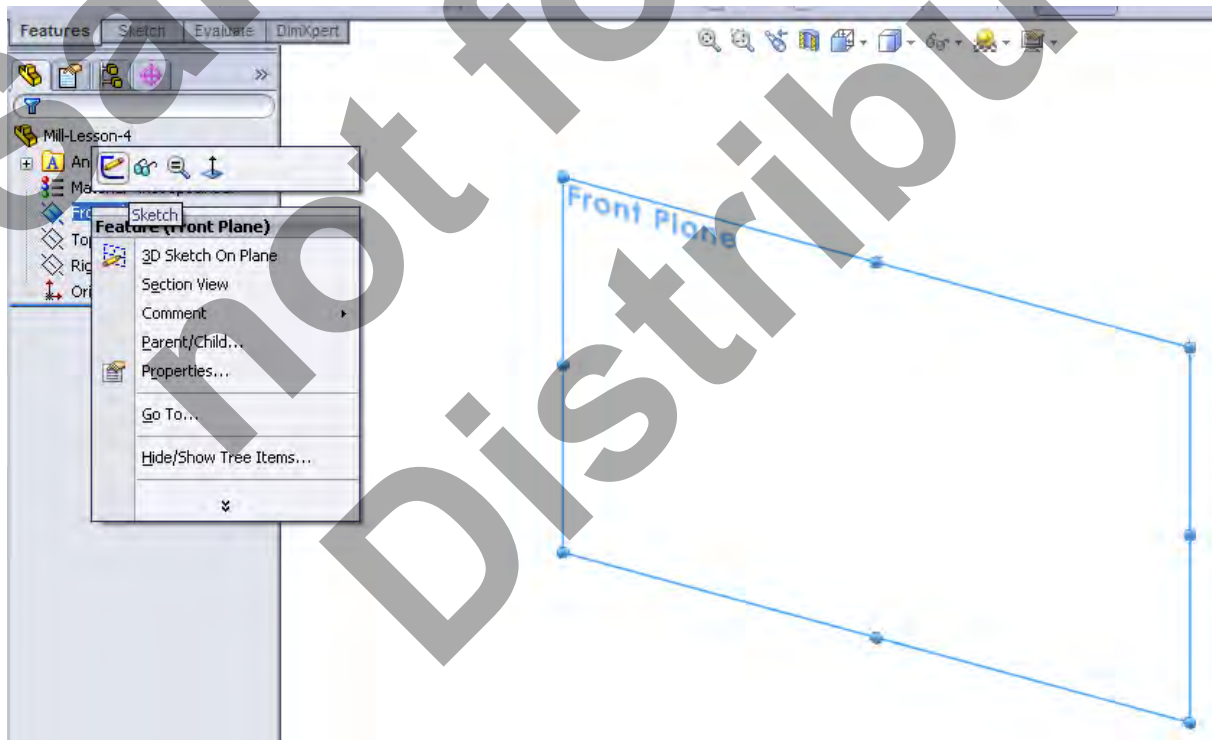
- Note the part name change in the feature manager design tree from **Part1** to **Mill-Lesson-4** once saved. The part name will also display at the top beside the bar pull-downs.



7. Create a sketch using the front plane. **Click on Front Plane** in the Feature Manager as shown below: Note the rectangular image named Front Plane shown in the drawing area.

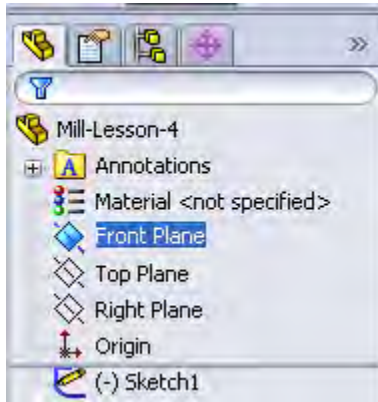


8. Right click on the highlighted plane and select the sketch icon as shown below:

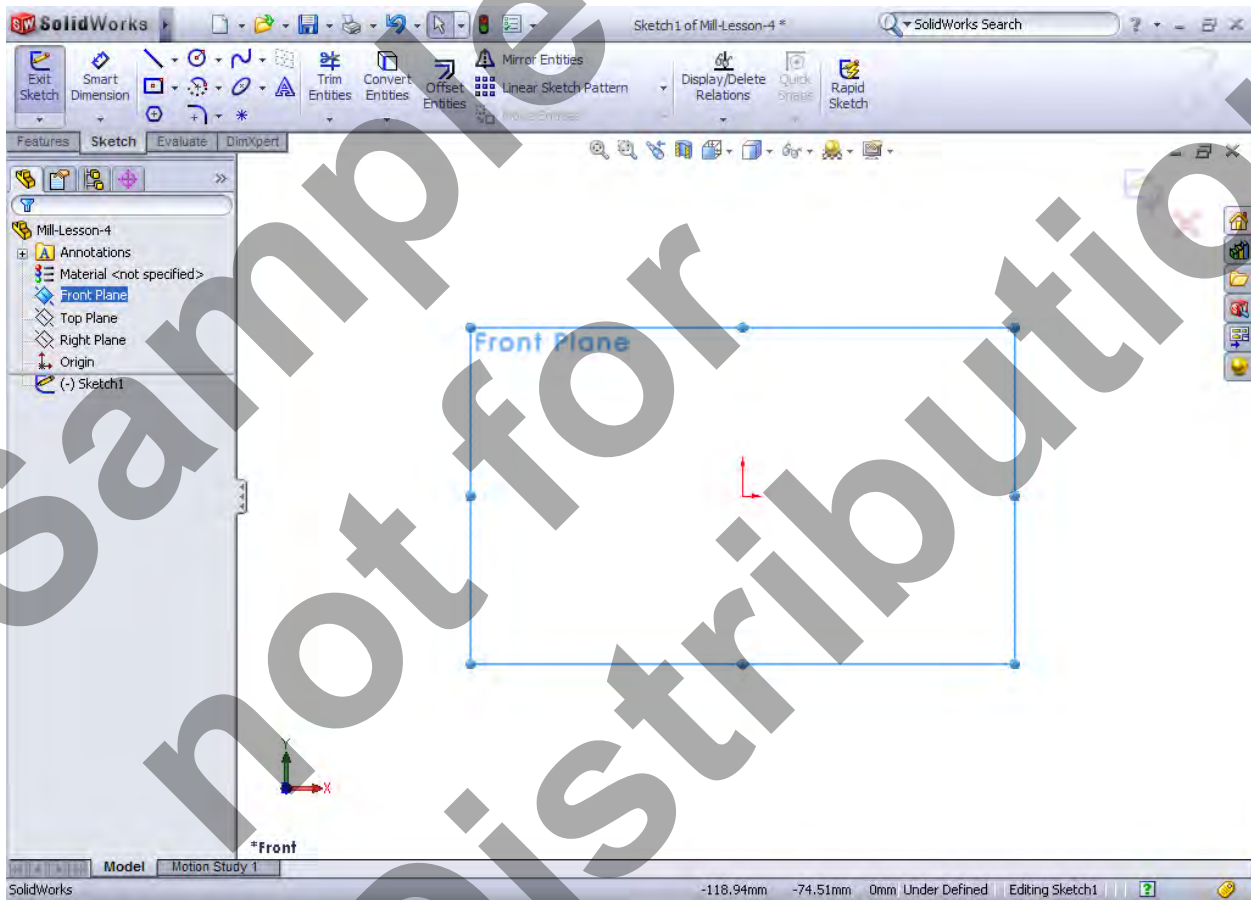


9. A new sketch will be started in the tree and in the working area.

Editing Sketch1



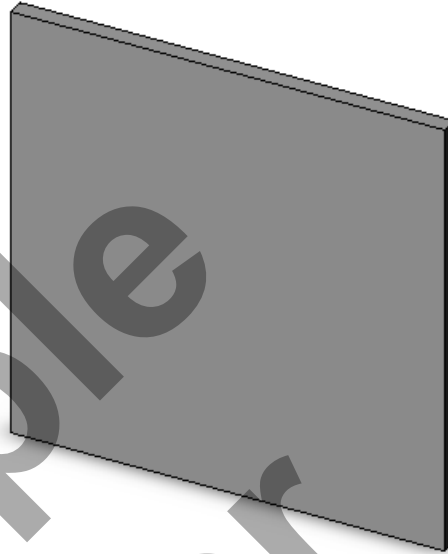
10. You are now ready to start creating geometry.



CAD TASK 2:

CREATE THE BASE SHAPE SKETCH AND EXTRUDE

- This task explains how to create the base rectangle portion of the geometry for Mill-Lesson-4 by means of a fully defined sketch.
- Note the location of the origin in relation to the geometry.
- You will start by creating the solid part to satisfy design intent.

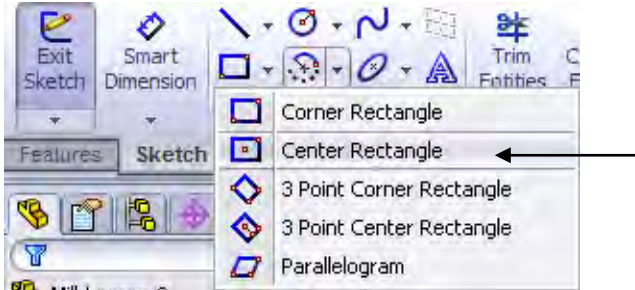


Sample
not for
Distribution

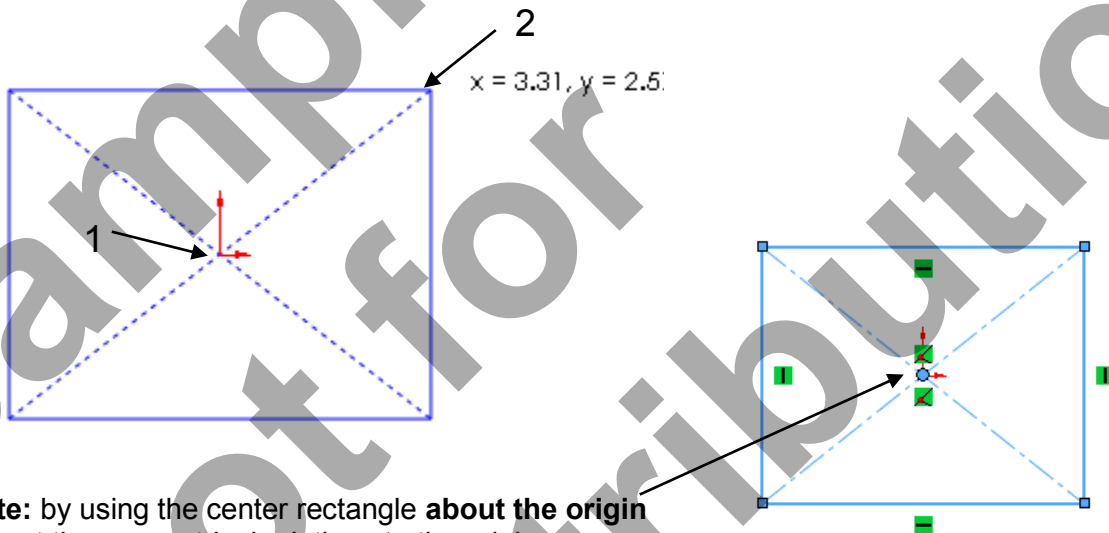
➤ Create Rectangle around the origin

1. Click on pull down arrow for the rectangle icon to display the different type of rectangles.

Select the **center rectangle icon**  in sketch toolbar.



2. **Create a rectangle** as shown below by clicking in the area shown by **number 1** below (**origin**) and dragging the mouse to the area shown by **number 2** below. Release left mouse button to create rectangle (Do not worry about size for now). Click on **check mark** to accept and end rectangle tool.



Note: by using the center rectangle **about the origin** we get the symmetrical relations to the origin automatically so when we dimension in the next step the rectangle will move about the center origin.

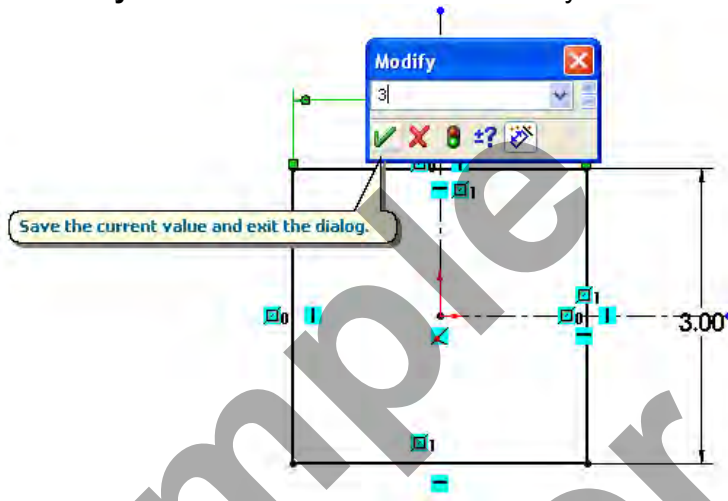
➤ Add dimensions to fully defined rectangle

- Referring to the dimensions provided from the **Mill-Lesson-4** drawing click on **smart**

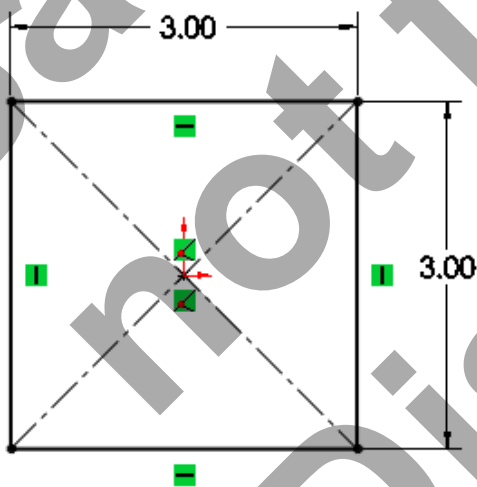


dimension from the sketch tool bar. Click on the **right vertical line** and drag the mouse to the right clicking again to place the dimension. Enter **3.00** in the **Modify** window and click **OK check mark**.

- Next click on the **top horizontal line** and drag up to place the dimension. Enter **3.0** in the **Modify** window. Notice how it remains symmetrical about the origin.



- Click on the **check mark** to accept.



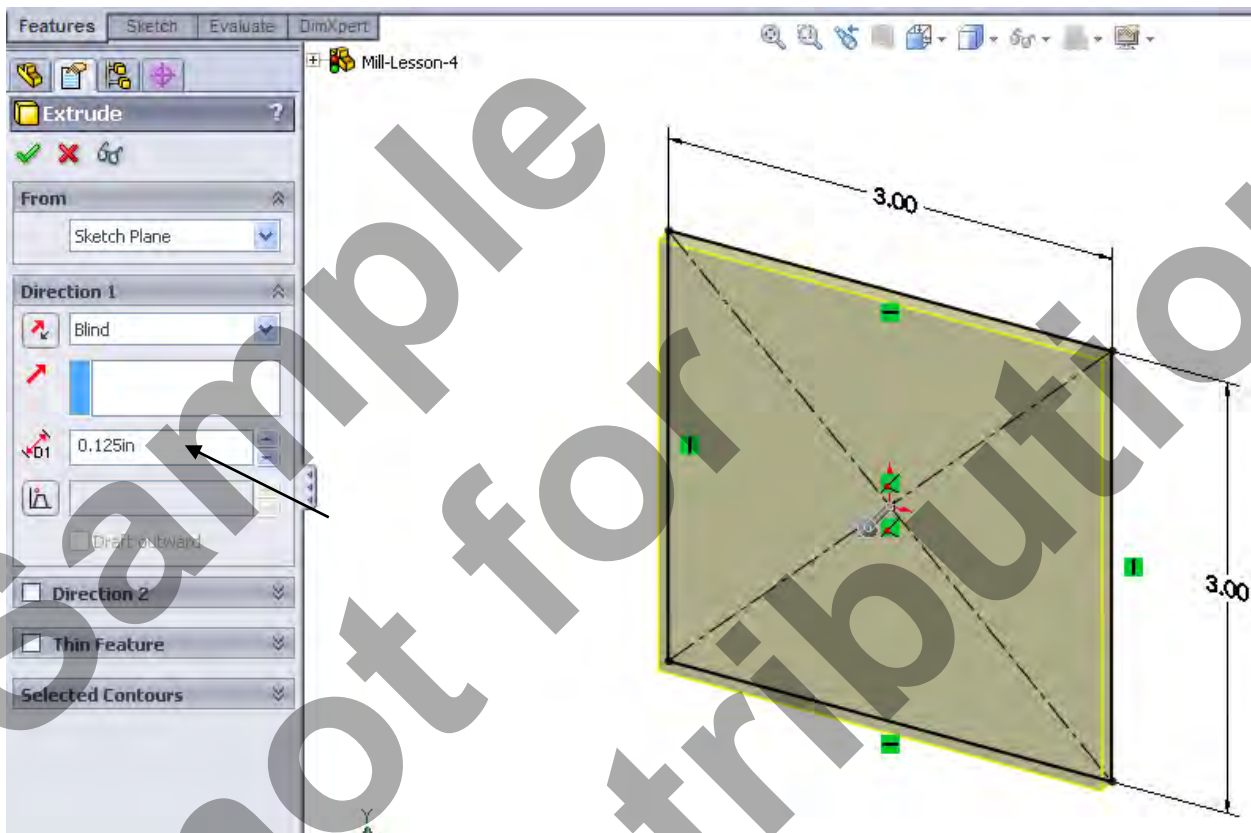
SolidWorks and Mastercam Training Guide

➤ Extrude the fully defined sketch

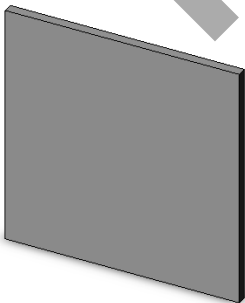
6. Select **Features Toolbar** tab. Select the **Extruded Boss/Base** icon.



7. Under **Direction 1** in the Extrude Properties, select **Blind** from the drop down list. Then enter **0.125** in the **Distance** window as shown below:



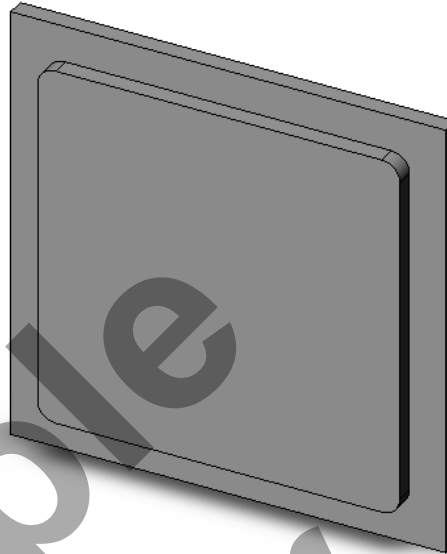
8. Click on the **check mark** to accept.




CAD TASK 3:

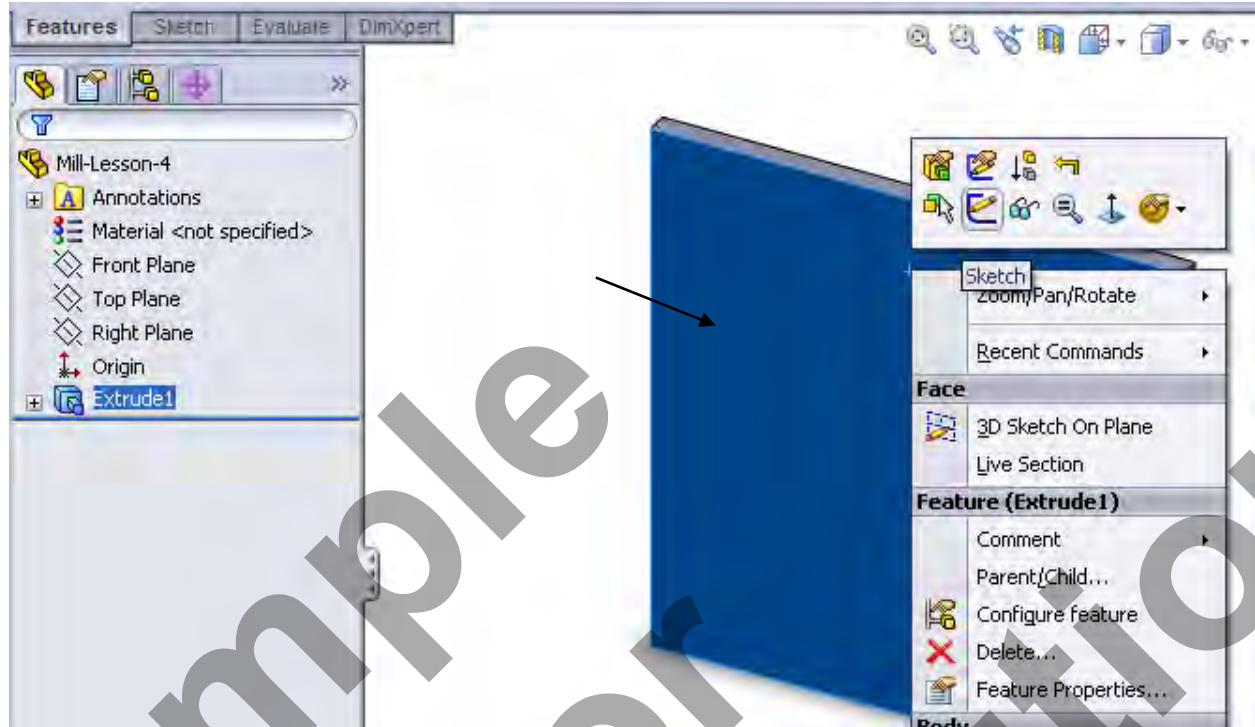
SKETCH RAISED PORTION AND EXTRUDE

- This task explains how to add a new sketch to create the raised portion for Mill-Lesson-4. The sketch entities will turn black and the sketch will indicate fully defined.
- Sketch must be fully defined in order for us to extrude into the 3D model.

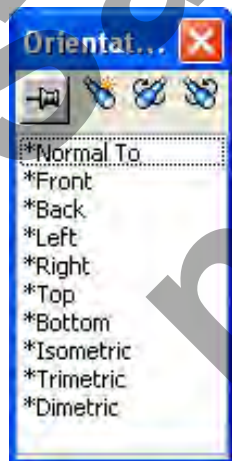


➤ Create new sketch


1. **Right click** on the front surface as shown below (indicated by arrow) and select **Sketch** Icon . This will create a new sketch on this surface.

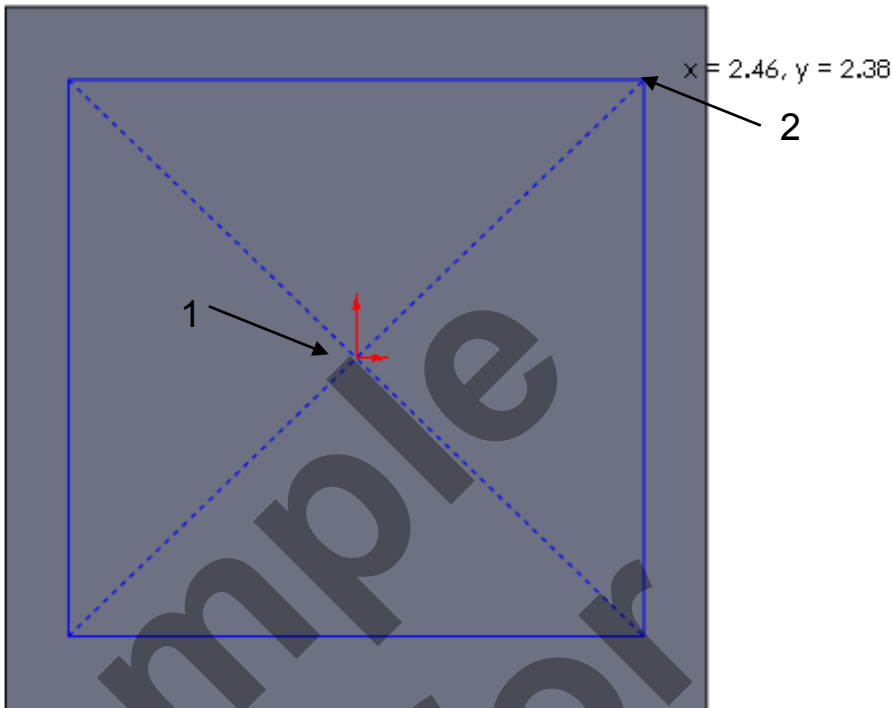



2. Press the space bar and the view orientation will appear. Double click on ***Normal To** to change the orientation to be looking straight on the sketch plane.

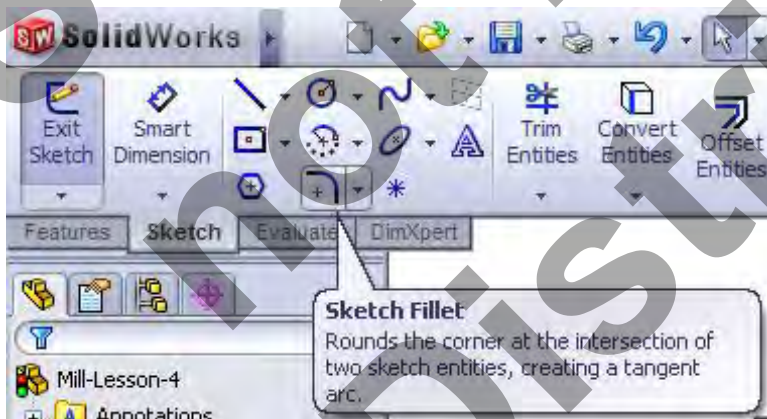


➤ Create shape of the raised portion

3. Select the **center rectangle icon**  as before in task 2 and place a rectangle **about the origin** (shown by number 1 below and dragging the mouse to the area shown by number 2 below). Click on the **check mark** to accept.

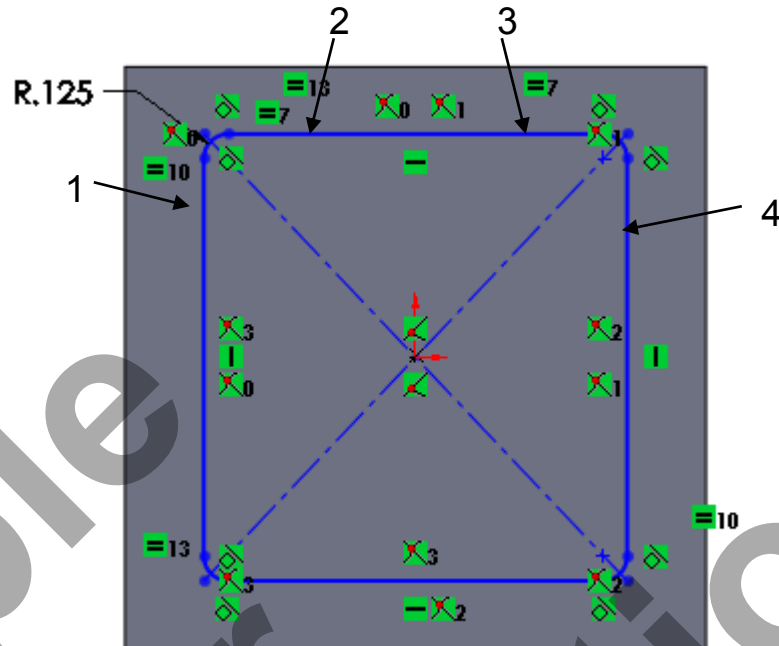
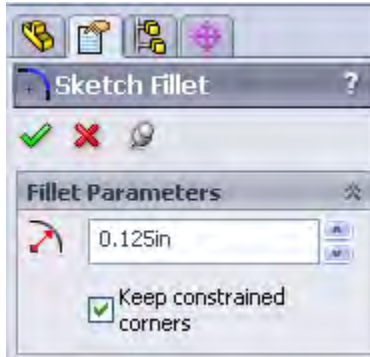


4. Add the **.125 inch fillets** to your sketch in each corner by the **sketch fillet icon**  on the sketch toolbar.

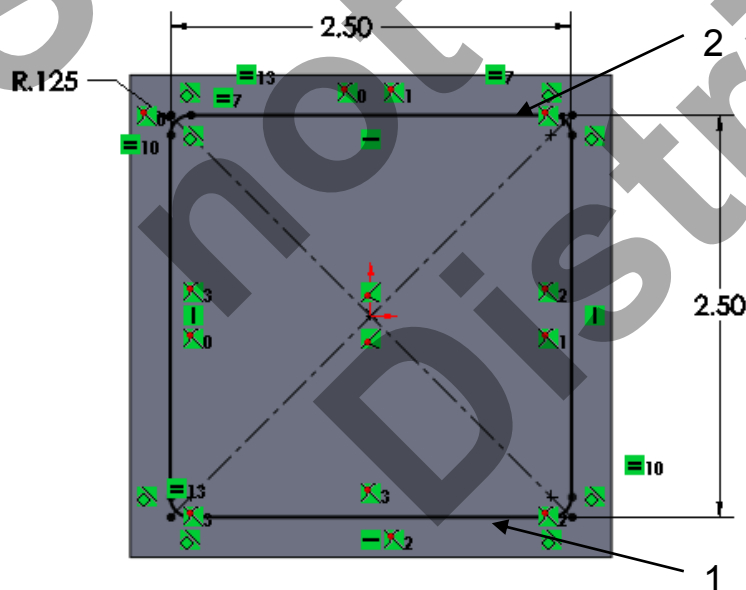


SolidWorks and Mastercam Training Guide

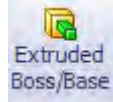
5. **Setting the radius to .125.** Click on a line (1) then the adjacent line (2); the fillet will appear. Continue working your way around each corner adding the fillets (3) (4) etc. Click on the **check mark to accept when all corners have fillets** (Note: you could dimension and extrude and add the fillets on the solid model later like we did in exercise 2).



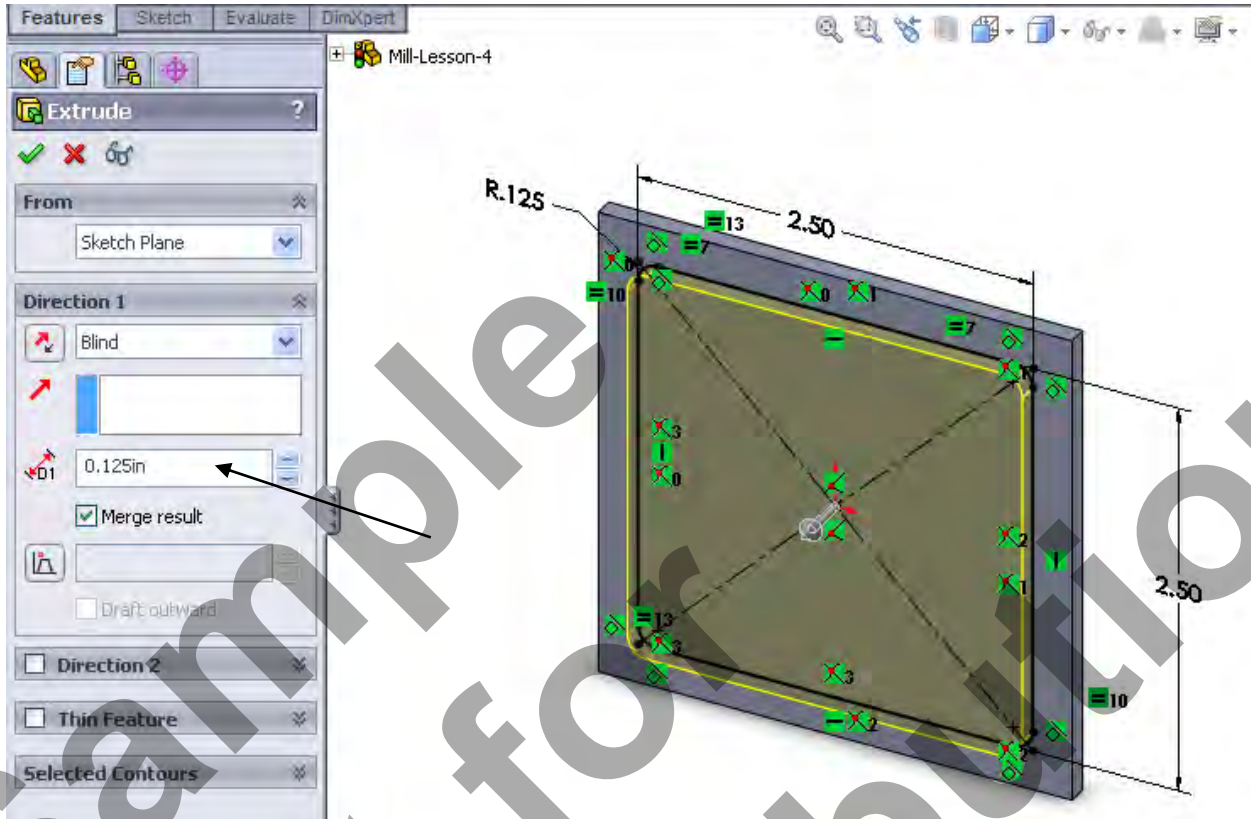
6. Dimension to fully define the sketch. From the toolbar click on **smart dimension**. **Dimension 2.5 x 2.5 rectangle.** Clicking on the **lower horizontal line (1)**, move up and click on the **upper horizontal line (2)**. Drag to the right and click to place the dimension. Enter **2.5** and click on the **check mark**. Do the same for the **vertical lines** with a **2.5** inch dimension. The part will turn black indicating **fully defined**.



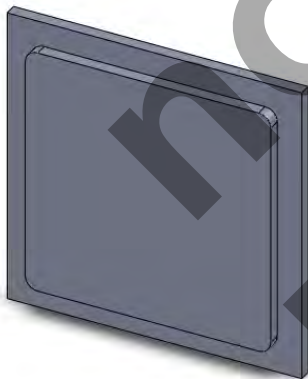
7. You are now ready to extrude the sketch. Select **Features Toolbar** tab. Select the



Extruded Boss/Base icon. **Change your view** by pressing the space bar and double clicking on ***trimetric** so you can see what will happen and verify the direction. Leave the direction **Blind** and set the D1 to **.125** inch.



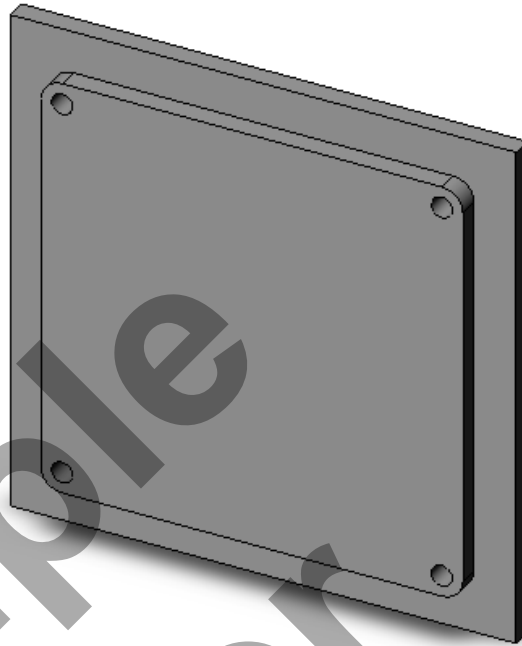
8. Click on the **check mark** to accept.



CAD TASK 4:


SKETCH THE FOUR HOLES AND CUT-EXTRUDE...

- This task explains how to sketch and produce the holes for Mill-Lesson-4.
- This task will create a new sketch.
- This task will also use the Cut-Extrude command.



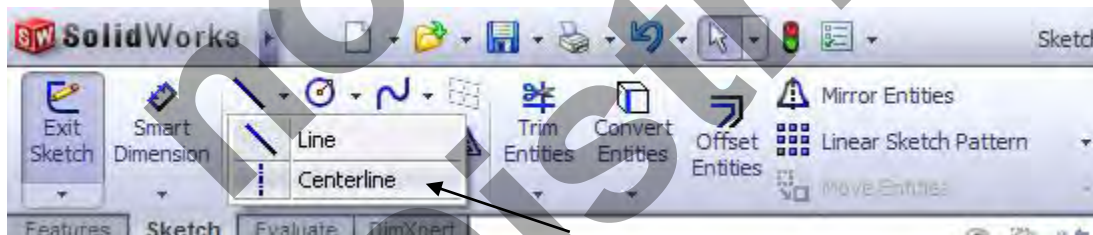
Sample
not for
Distribution


➤ Create a new sketch on front surface.

1. **Right click** on the front surface as shown below (indicated by arrow) and select **Sketch** Icon . This will create a new sketch on this surface.





2. Press the space bar and the view orientation will appear. Double click on ***Normal To** to change the orientation to be looking straight on the sketch plane.
3. Select from sketch toolbar: click on the arrow pull down beside the line icon to click on **Centerline**.

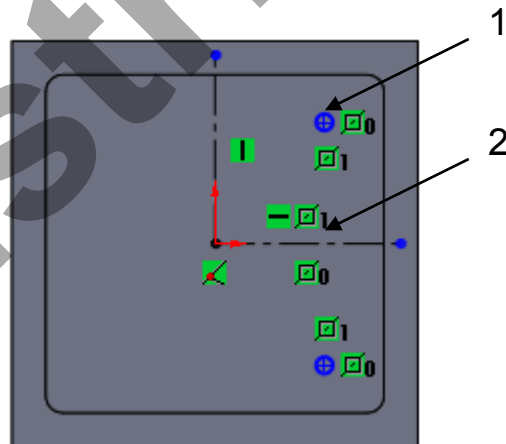
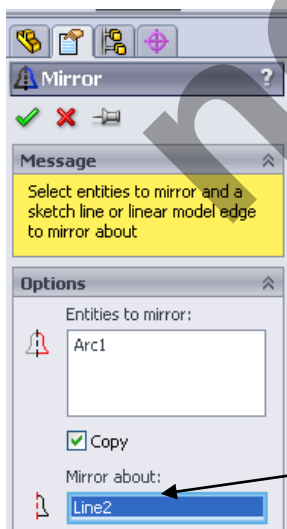


4. Hover over the **origin**  of the sketch area. **Click on the origin and drag vertically** clicking again to produce the vertical centerline. Now while still in centerline mode create the horizontal centerline by **clicking on the origin** again and dragging (same method as vertical centerline). Click on the **check mark** to accept the centerlines created followed by the **Esc key** on the keyboard.

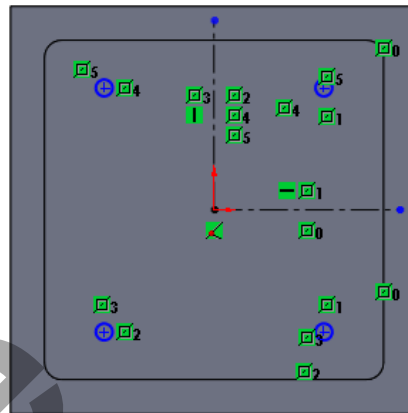
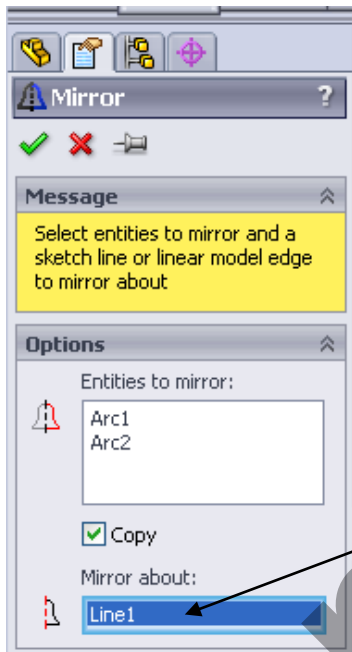


➤ Sketch the first hole of pattern


5. Select Circle icon  and sketch a **.125 diameter circle in the top right area of the surface/square** (within the inner square). Do not worry about location.
6. **Mirror the hole about the horizontal centerline** using the **Mirror Entities** command 
7. Click on the circle (1) as shown below: Note that Arc1 is displayed in the **Entities to mirror** window. Verify **copy** is checked in properties box.
8. Select the Mirror about box (turns blue) and click on the Horizontal Centerline (2) as shown below, note Line2 is displayed under **Mirror about**:
9. Click on the **Check mark** to accept.

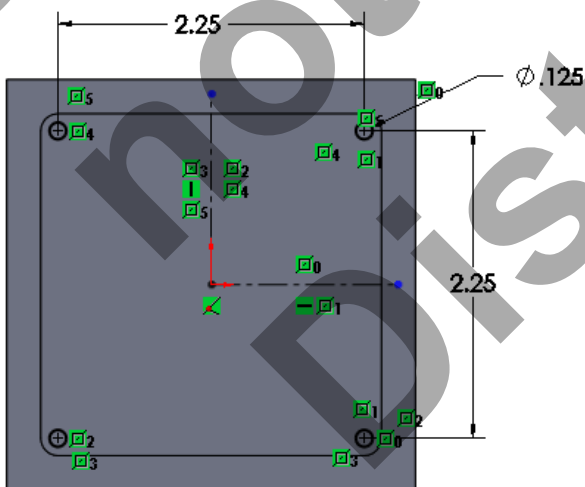


10. Click on **Mirror Entities** and click on **both circles**. Then click on the **Mirror about** area in the **Mirror** properties box and this time select the **vertical centerline**. Click on the **check mark** to accept the mirror. You now have the four circles.

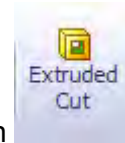


11. Dimension the sketch using smart dimension. Click on **smart dimension** . Click on the **upper right circle** and drag to right. Enter **.125** in modify box and click on **check mark** to accept.

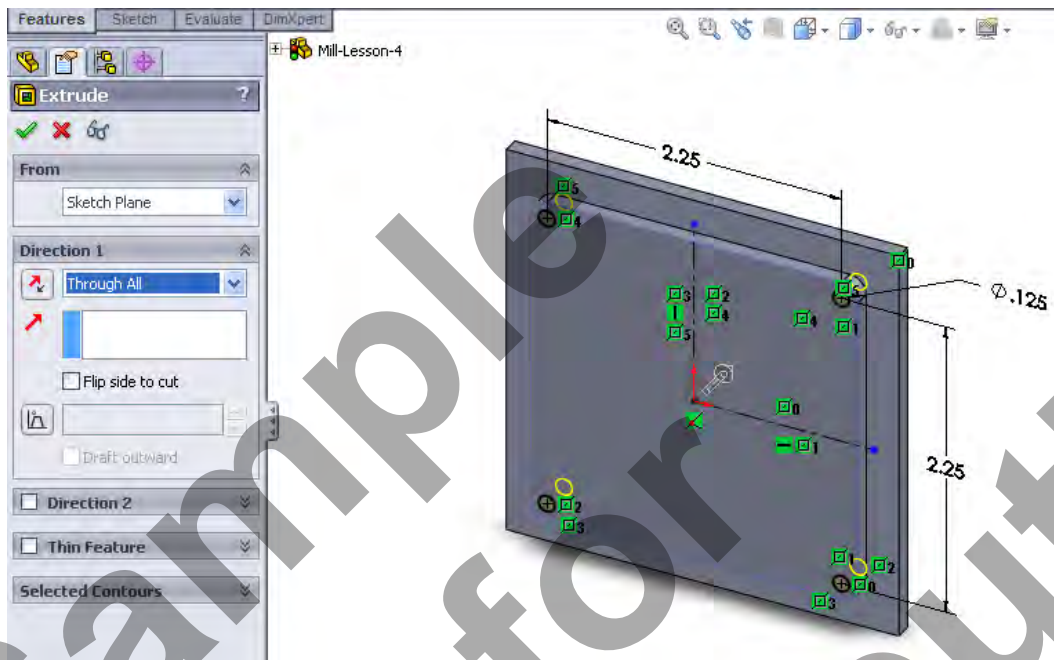
12. Place the **2.25 horizontal and vertical dimensions** shown below by clicking on two upper circles and dragging followed by the two vertical circles. Verify that the sketch is **fully defined** .



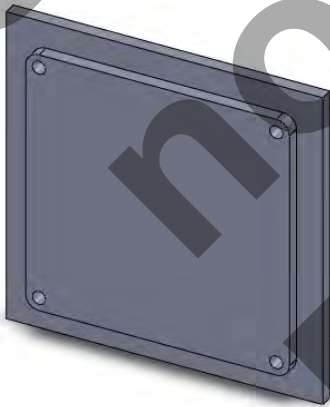
➤ Cut-Extrude the hole pattern



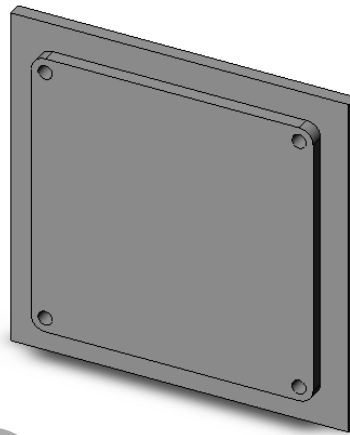
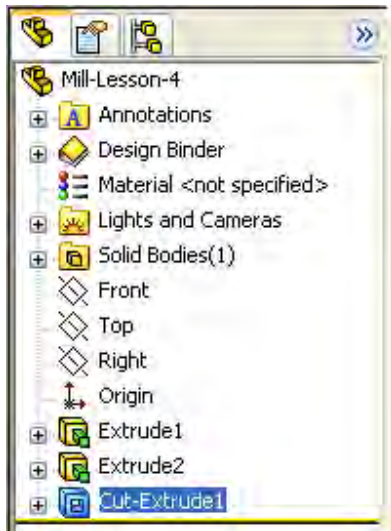
- From the **Features Toolbar** tab select the **Extruded Cut** icon. **Change your view** by pressing the space bar and double clicking on ***trimetric** so you can see how the drawing will change and verify the direction.
- Change the direction to **Through All** in the **Cut-Extrude** properties to cut through regardless of thickness.



- Click on the **check mark** to accept the cut.



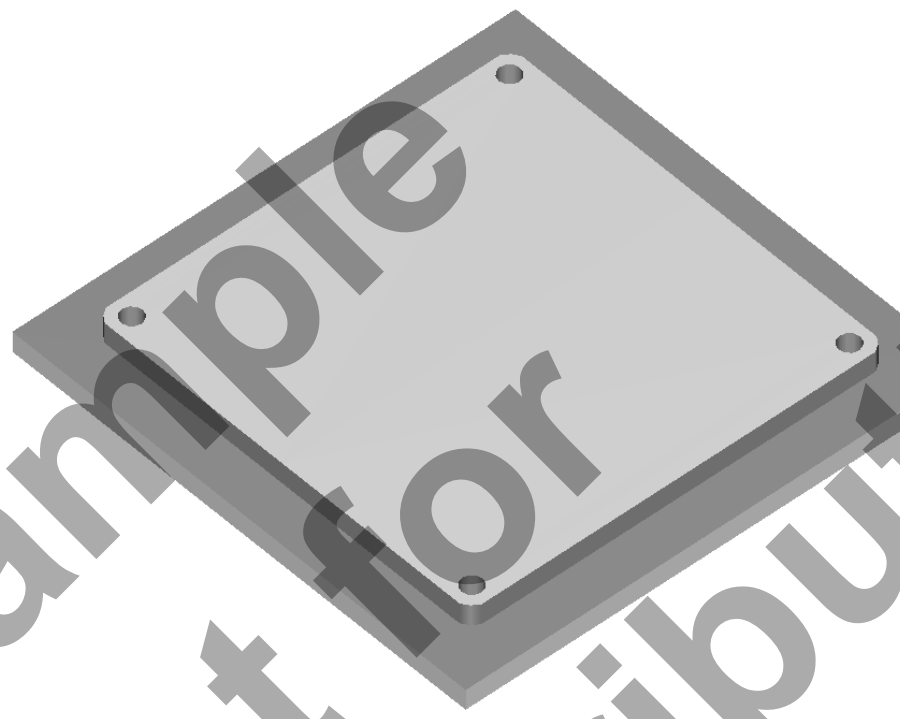
16. Your part Mill-lesson-4 is now complete. **Remember to save.**



Note: The dimensions and relations established are to define the sketch and do not necessarily represent the placement of the dimensions on the finished drawing. This lesson was dimensioned this way and used the mirror command to establish symmetry about the middle of the part (origin). The design intent is to allow the hole pattern and size of the raised rectangle to remain centered as the feature dimensions change.

Sample
not for
Distribution

SolidWorks and ***Mastercam X⁶*** **TRAINING GUIDE**



MILL-LESSON-4-TOOLPATHS **DRILL AND CONTOUR**

Objectives

You will generate a toolpath to machine the part on a CNC vertical milling machine. This lesson covers the following topics:

➤ **Import and Position the Part in Mastercam**

➤ **Position the Part in Mastercam**

➤ **Establish Stock Setup settings:**

Stock size.

Material for the part.

Feed calculation.

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

Contour toolpath.

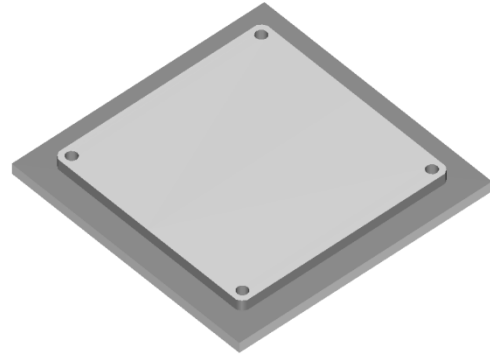
Drill toolpath.

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

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

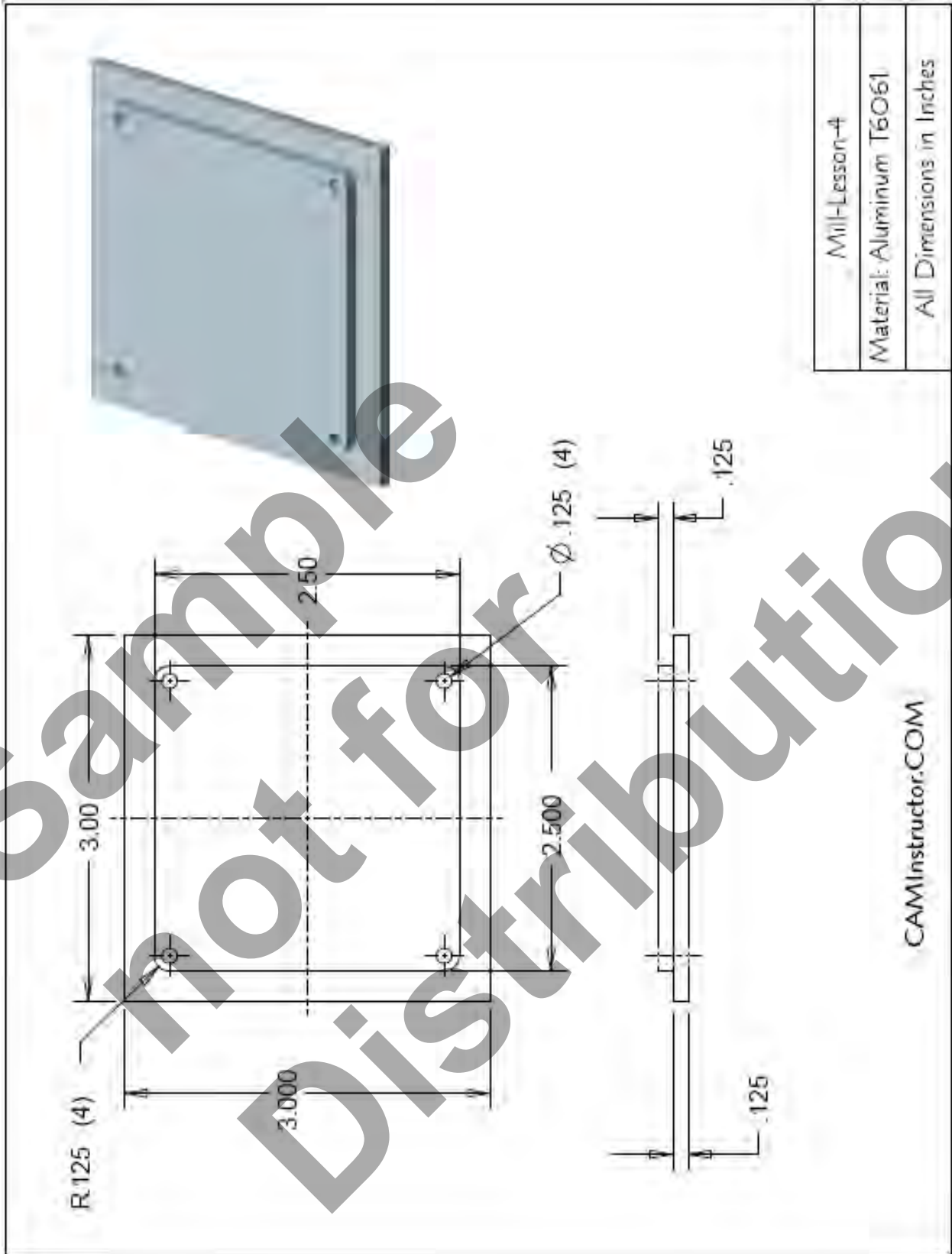
Using Backplot to identify the correctness of the toolpaths.

Generating the NC- code.



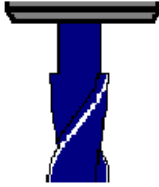

Sample
not for
Distribution

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

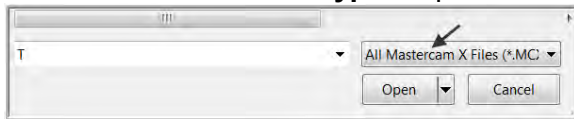
LESSON-4 - THE CAM PROCESS

Toolpath Creation

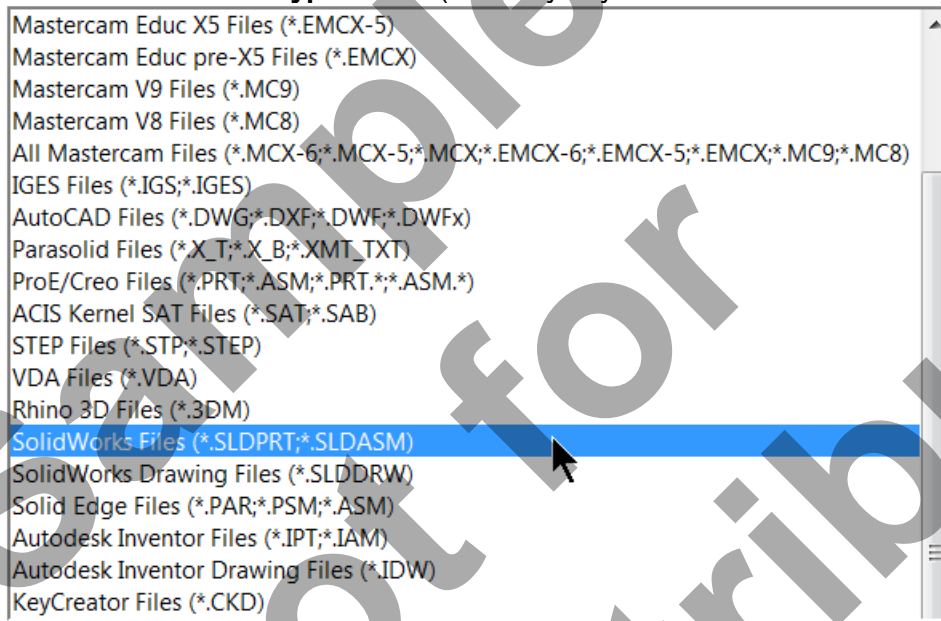
- TASK 1:** Import the CAD file into Mastercam
- TASK 2:** Position the part in Mastercam
- TASK 3:** Define the rough stock using stock setup
- TASK 4:** Drill the 4 x .125" diameter holes
- TASK 5:** Machine the contour
- TASK 6:** Backplot the toolpath
- TASK 7:** Modify the contour toolpath to add roughing cuts and a finish pass
- TASK 8:** Verify the toolpath
- TASK 9:** Post and create the CNC code file

TOOLPATH TASK 1: IMPORT THE CAD FILE IN MASTERCAM

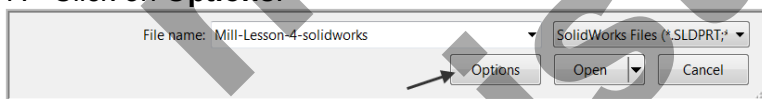
1. Start **Mastercam**.
2. Before starting the geometry creation you should set up the grid, toolbars and machine type as outlined in the **Setting the environment** section at the beginning of this text:
 - a. Set up the **Grid**. This will help identify the location of the origin.
 - b. Customize the toolbars to **Machine a 2D Part**.
 - c. Set the machine type to a **Haas Vertical Spindle CNC machine**.
3. Click on **File>Open...**
4. Click on the **Files of type** drop down list.



5. Click on the **File Type** button (currently says **All Mastercam X Files**) as shown below.

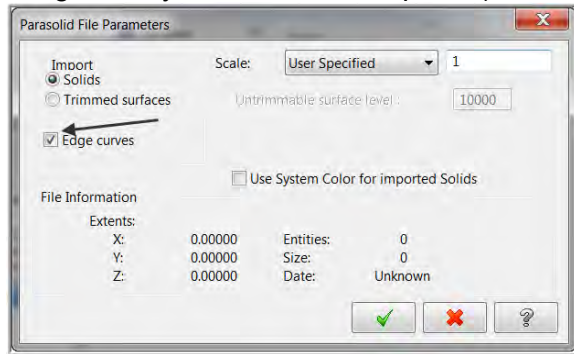


6. Find the location of the **Solidworks-Mill-Lesson-4.sldprt** file and click on the file.
7. Click on **Options**.



SolidWorks and Mastercam Training Guide

8. Click on the check box to enable **Edge Curves** as shown below. (This will apply wireframe geometry to the solid when it is imported into Mastercam, which will allow you to select the geometry to create the toolpaths.)



Edge Curves

Turning on Edge Curves tells Mastercam to add Lines and Arcs to all of the Edges on the Solid.

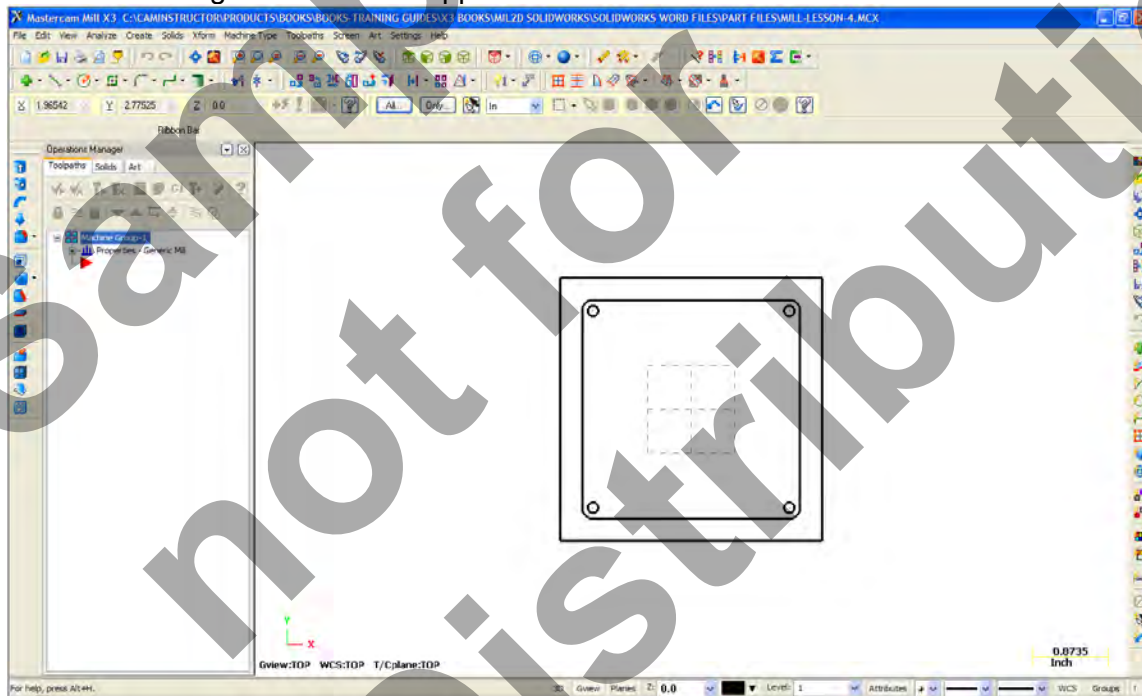
This is necessary so that you can select the geometry on the part to create the Toolpaths.

9. Click on **OK** .

10. Click on **Open**.



11. The following screen should appear:

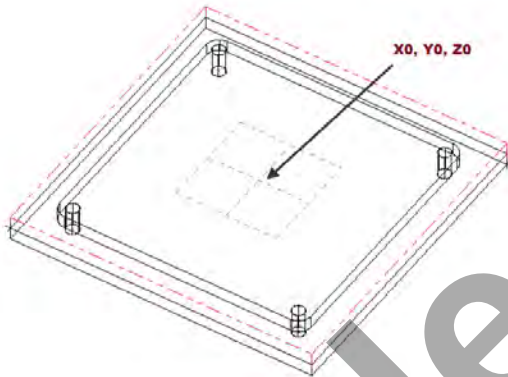


12. Click on the **Isometric** icon  and then the **Fit**  icon.

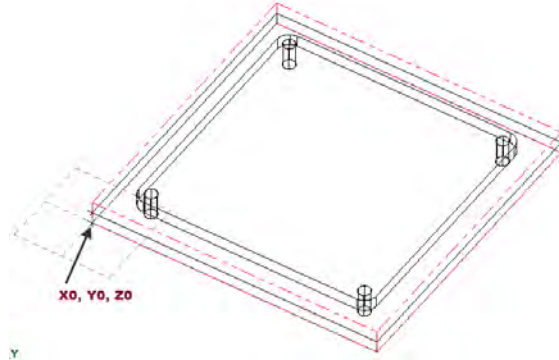
TOOLPATH TASK 2: POSITION THE PART IN MASTERCAM

- ☞ We want the Work Offset to be set to the lower left corner and top of the part. Currently it is at the center of the part and at the bottom.

Current location of X0, Y0, Z0



Required location of X0, Y0, Z0

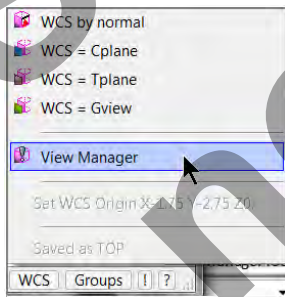


- ☞ There are a couple of options to change the **Work Offset** Location.
 - Option 1** is to move the part to the required location, however that means if any changes are made to the **CAD** model then the location will be different.
 - Option 2** is to change the **WCS** which tells **Mastercam** that we want the **Work Offset** location to be different than it's current location – without moving the part.
- ☞ For this Lesson we will use the WCS method.

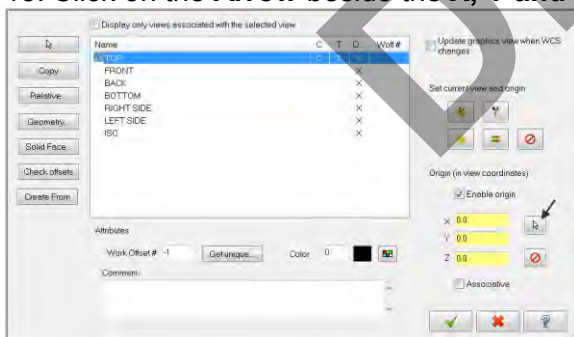
13. Click on **WCS>View Manager** located at the bottom of the screen as shown below.



14. Click on **View Manager**.

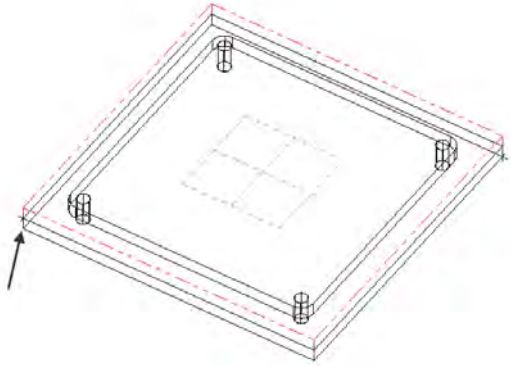


15. Click on the **Arrow** beside the X, Y and Z Origin windows as shown below.

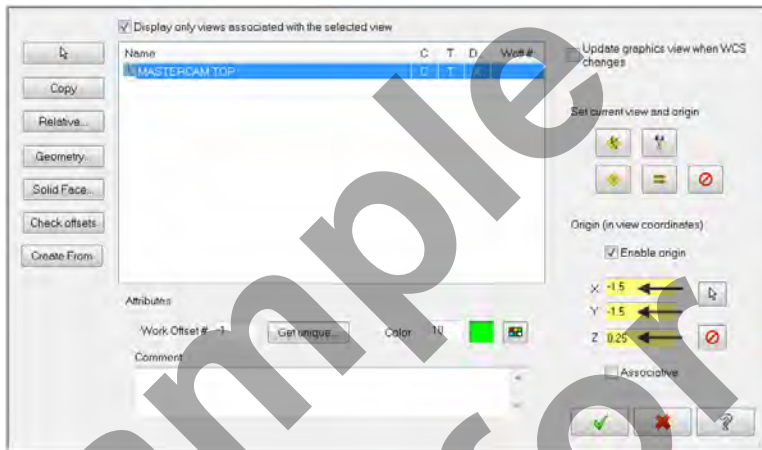


SolidWorks and Mastercam Training Guide

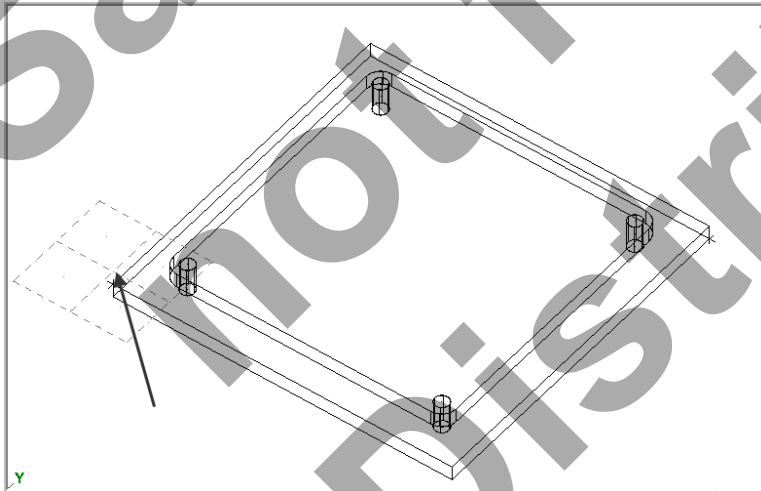
16. Click on the **lower left corner** of the model as shown below.



17. Notice the changes to the **X and Y dimensions**. We need to manually type in the new **Z dimension to 0.25** as shown below.

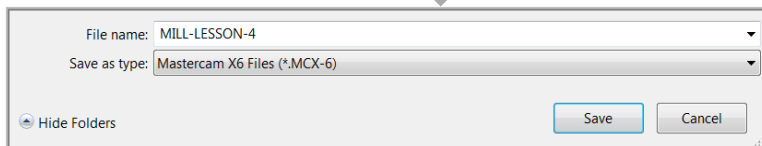


18. Click on **OK**. Notice the Work Offset Location has changed as shown below.



19. Click on **File>Save As...**

20. Name the file **MILL-LESSON-4** as shown below and click on **Save**.



Toolpath Creation

TOOLPATH TASK 3:

DEFINING THE ROUGH STOCK USING STOCK SETUP

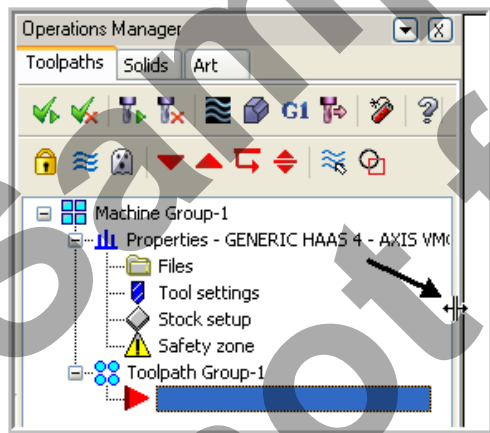
1. For a better view of the part use the toolbar at the top of the screen to change the graphics view to Isometric



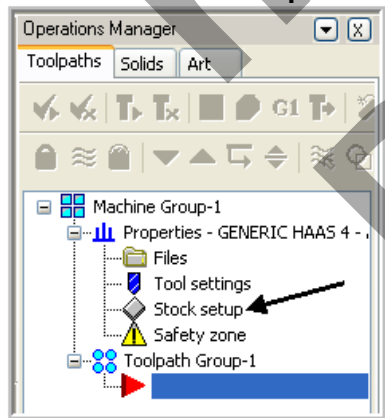
2. Now select the Fit to screen icon 
3. Select the plus in front of **Properties** to expand the Toolpaths Group Properties. **Alt-O** will Show/hide Operations Manager pane.



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.

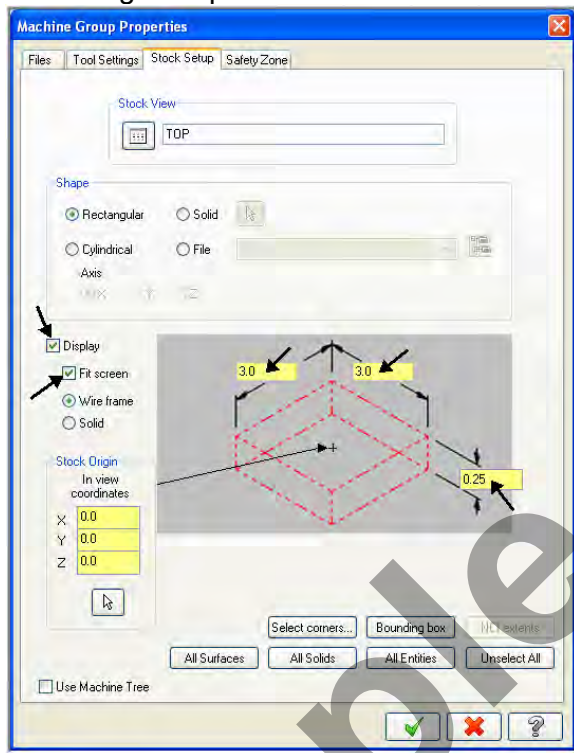


5. Select **Stock setup** in the toolpath manager window.

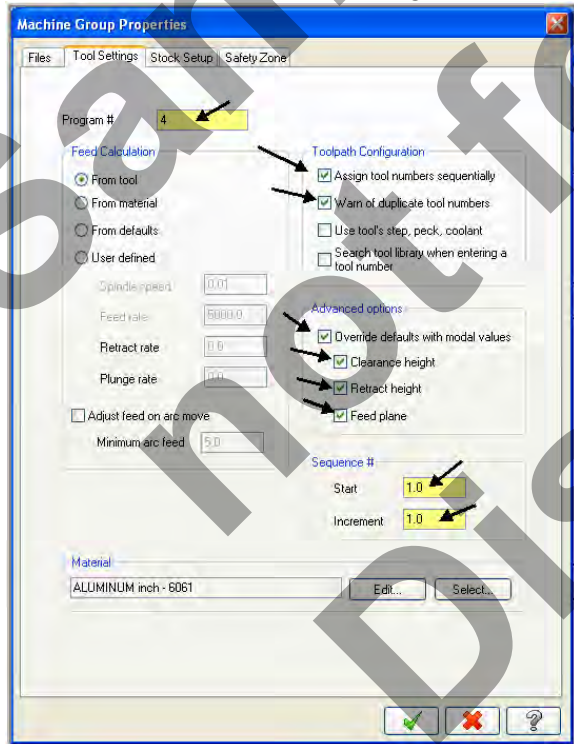


SolidWorks and Mastercam Training Guide

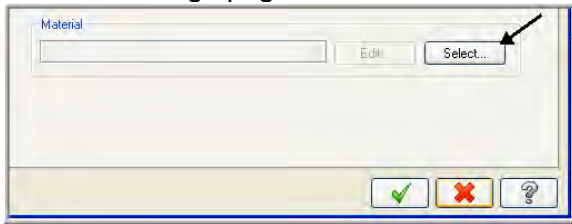
6. Change the parameters to match the Stock Setup screenshot below:



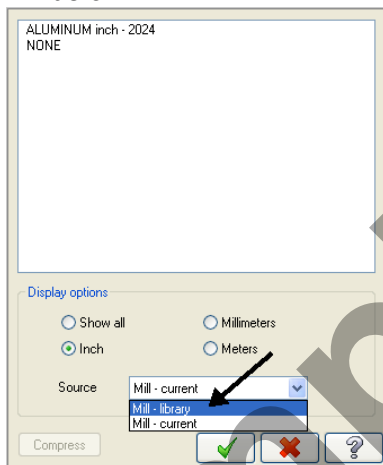
7. Select the Tool Settings tab and change the parameters to match the **Tool Settings** screenshot below. To change the **Material** type follow the next set of instructions.



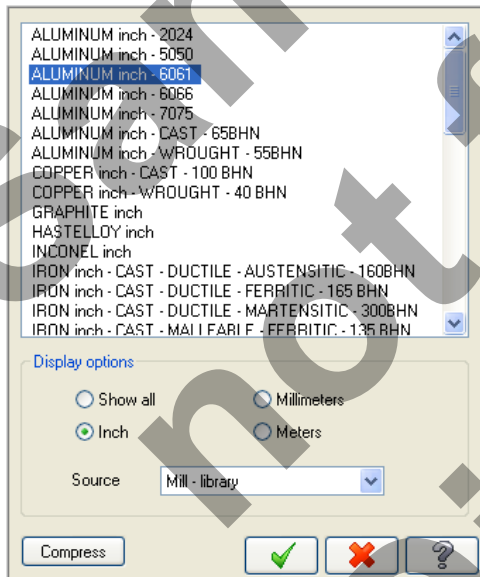
- To change the **Material** type to **Aluminium 6061** pick the **Select** button at the bottom of the Tool Settings page.



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



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



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

TOOLPATH TASK 4: 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.

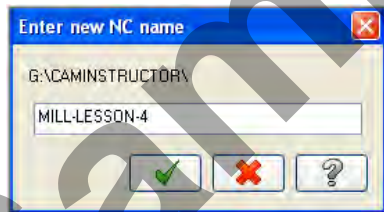
1. Change the graphics view to a **Top View** by using the toolbar at the top of the screen.



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



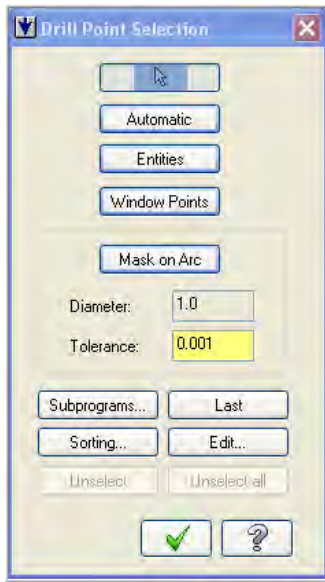
3. When prompted to "**Enter new NC name**" Input **Mill-Lesson-4** as shown below and then select the OK button .



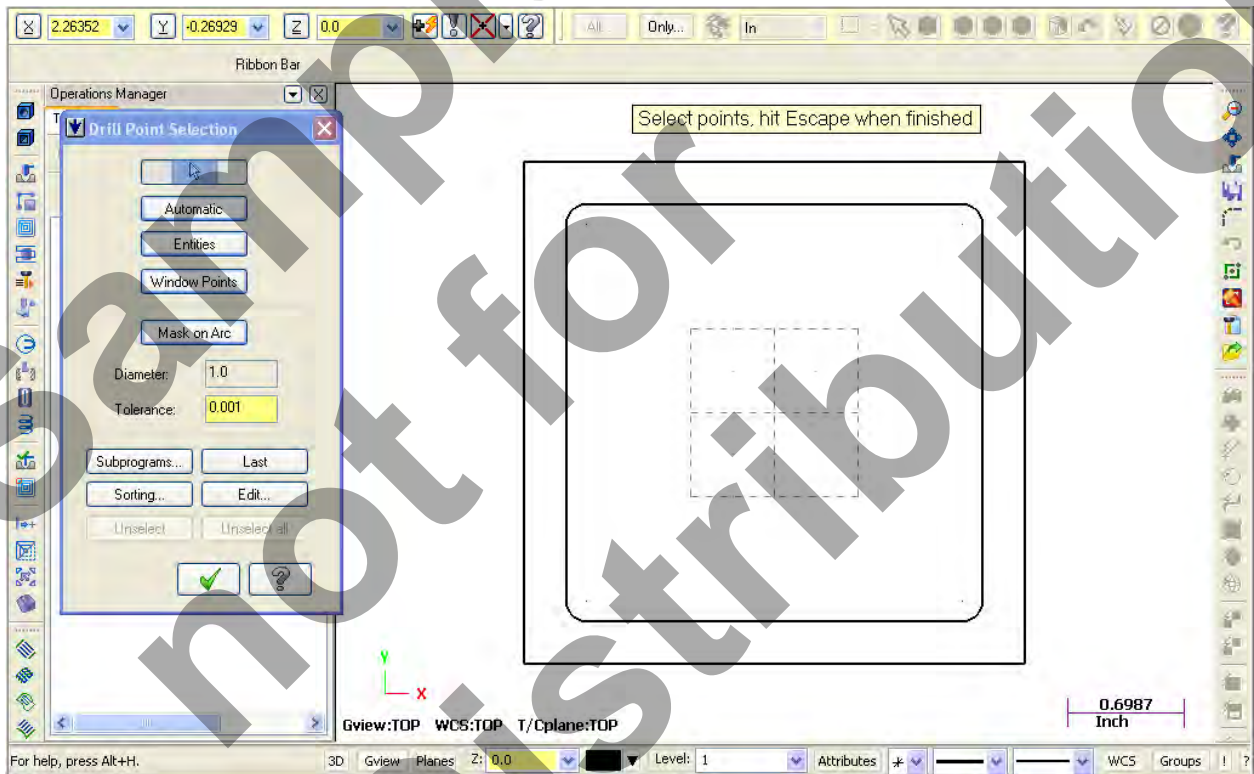
4. Now you are prompted to:

Select points, hit Escape when finished

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

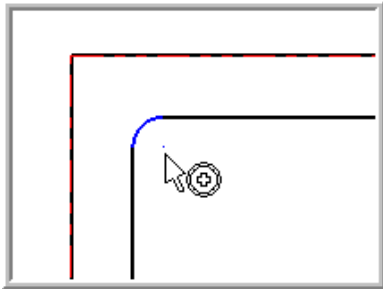


6. Your screen will look similar to the screenshot below:

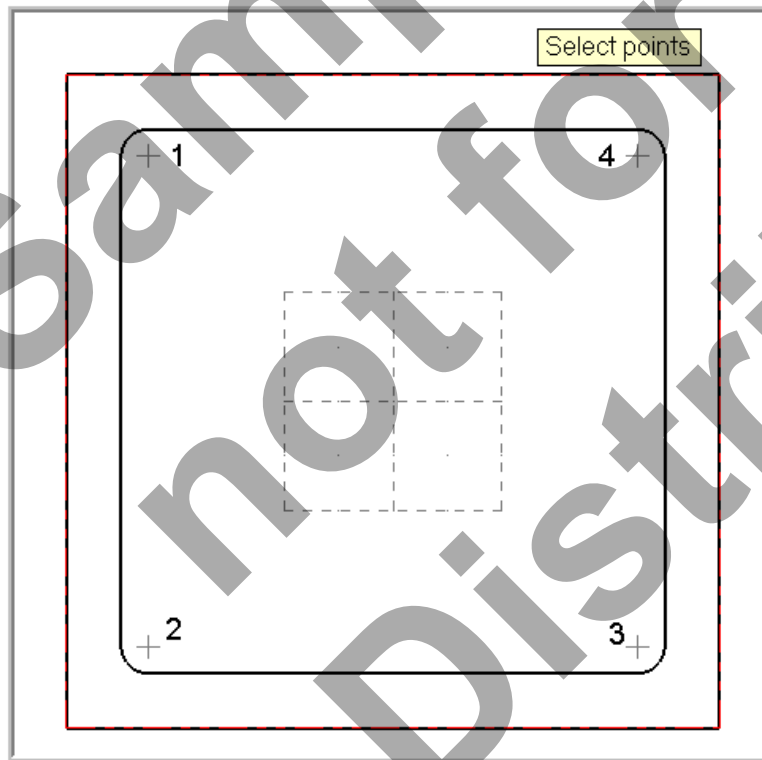


SolidWorks and Mastercam Training Guide

- 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 denotes you are snapping the centre of this arc. Use the left button of your mouse to pick this center point.



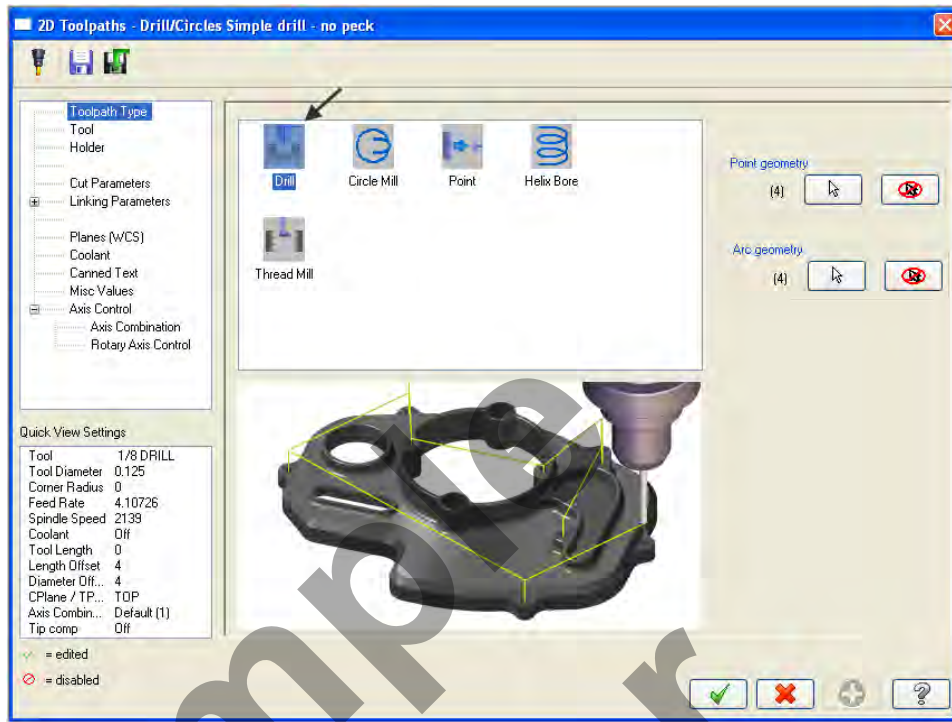
- 9. Using the method outlined above continue on and pick the remaining three center points, in this order:
 - 10. Lower left.
 - 11. Lower right.
 - 12. Upper right.
13. Your screen should look like the screenshot below after selecting the four center points:



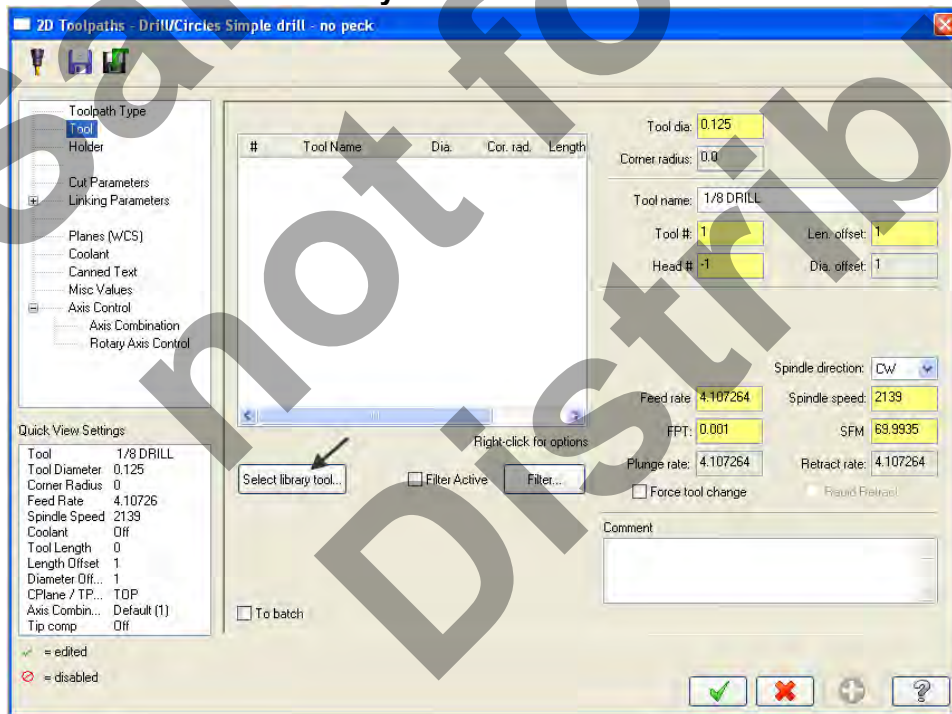
14. Select the OK button  in the **Drill Point Selection** dialog box.

➤ 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**.

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

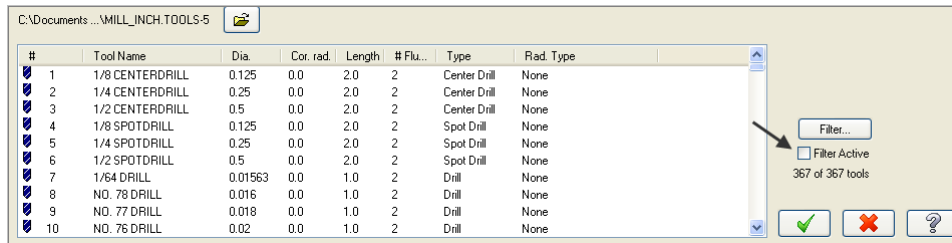


22. Click on the **Select library tool** button in the lower left corner.

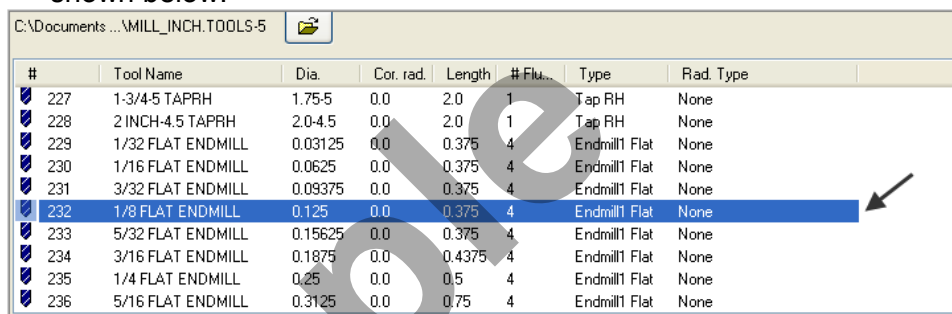


SolidWorks and Mastercam Training Guide

23. 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.




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

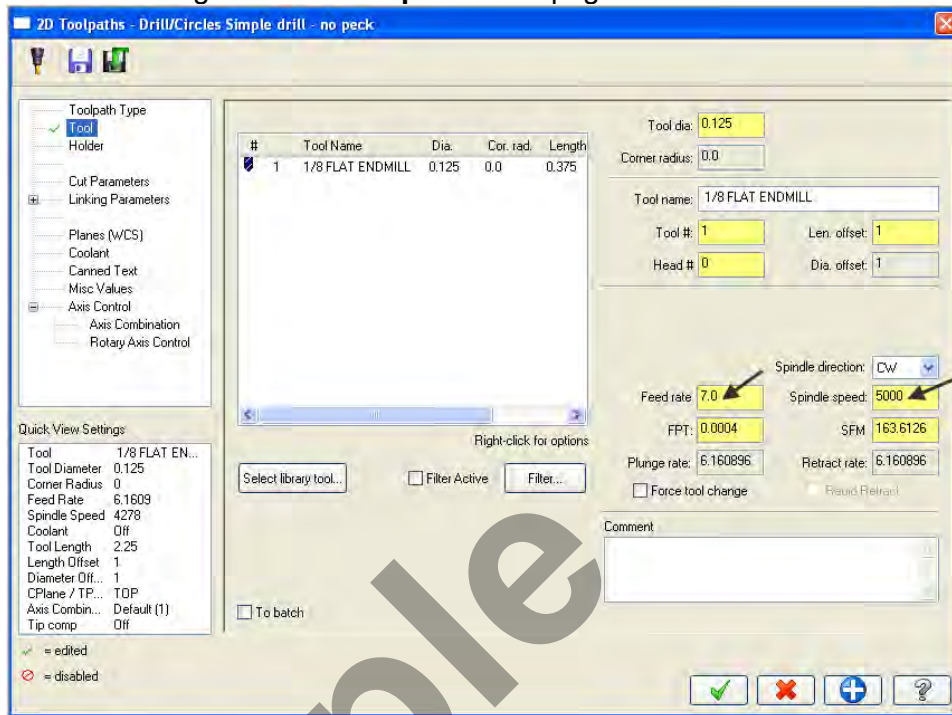


26. To resize a column in the **Tool Selection dialog box**, click on the divider between the columns with your left mouse button, as shown below, hold the left mouse button down and move to the right or left.

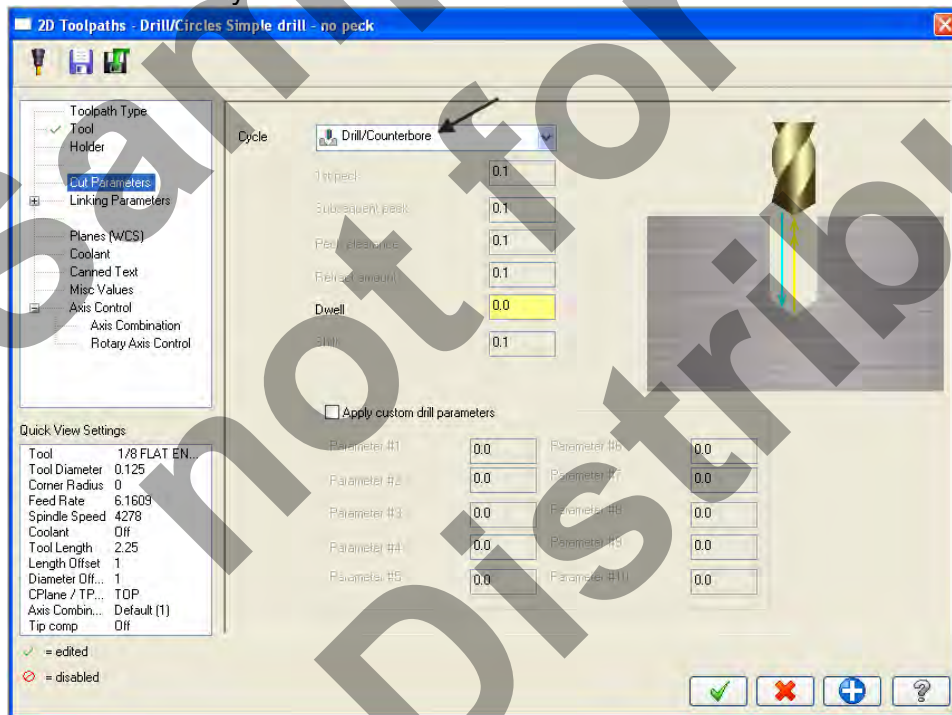


27. Select the **OK** button  to complete the selection of this tool.

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

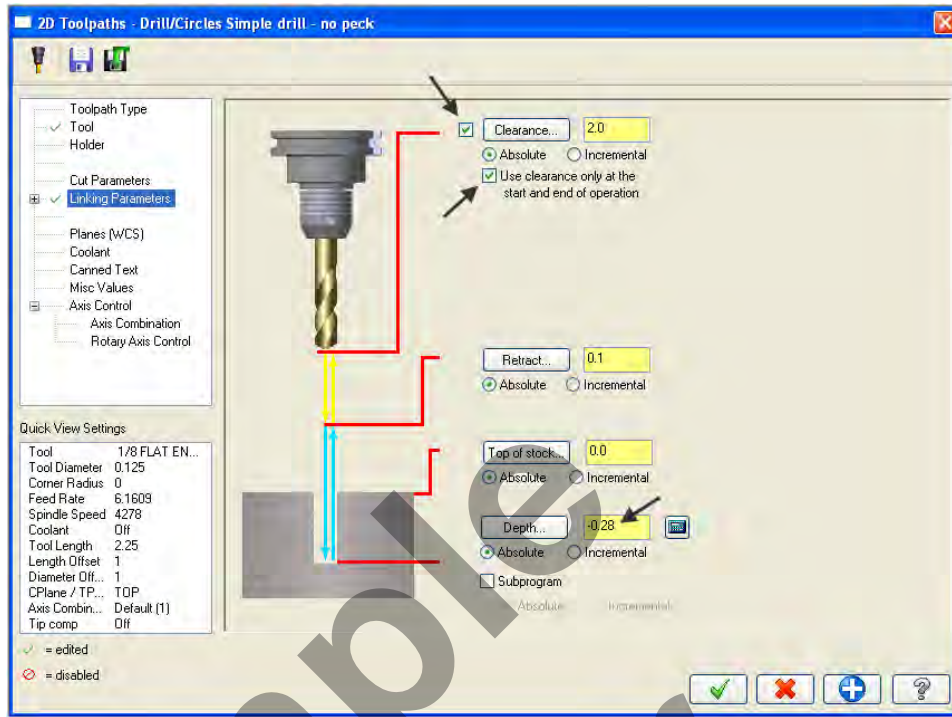


29. Select **Cut Parameters** from the list on the left and make changes to this page as shown below. The Cycle should be set to Drill/Counterbore.

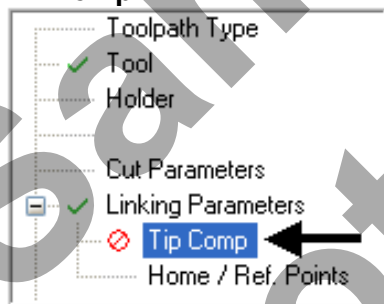



SolidWorks and Mastercam Training Guide

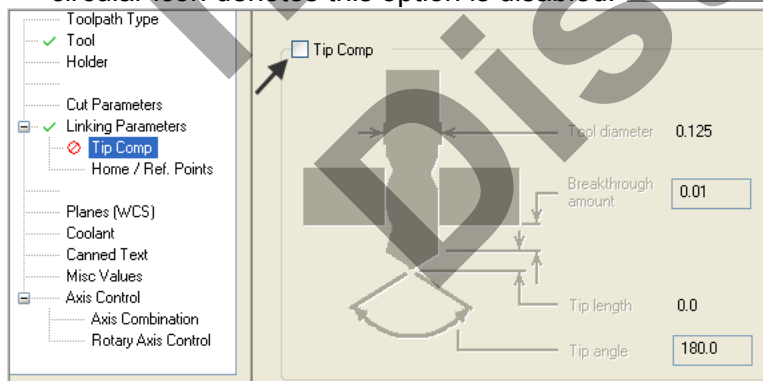
30. Select **Linking Parameters** from the list on the left and make changes to this page as shown below. Input the **depth of -0.28** and the other values as shown below. **Note** all the values are set to **Absolute**.



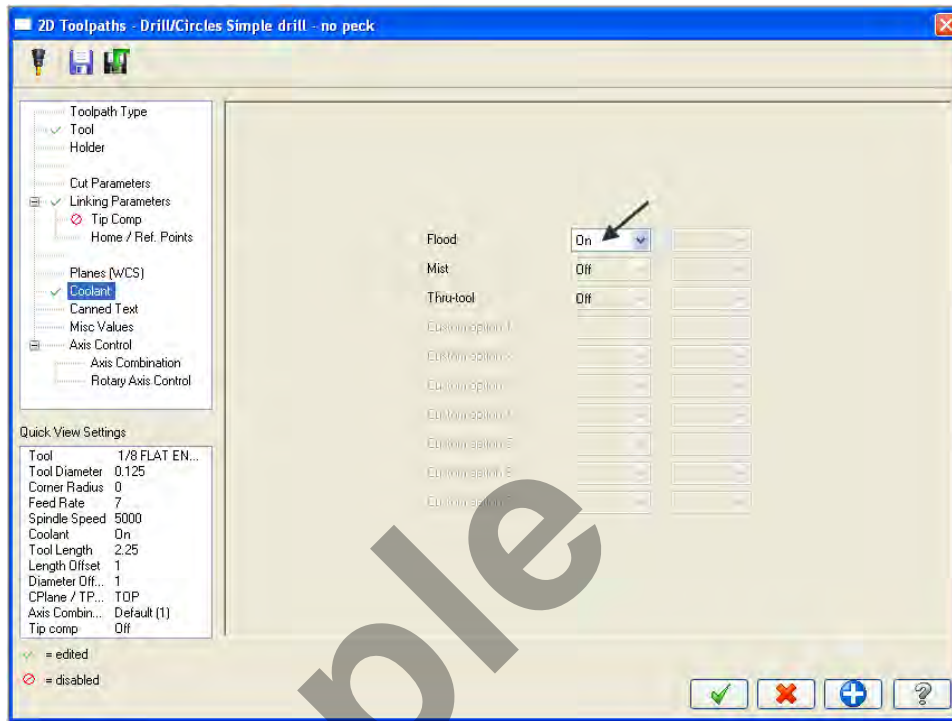
31. Select the plus sign to the left of **Linking Parameters** to expand the list and click on **Tip Comp**.



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

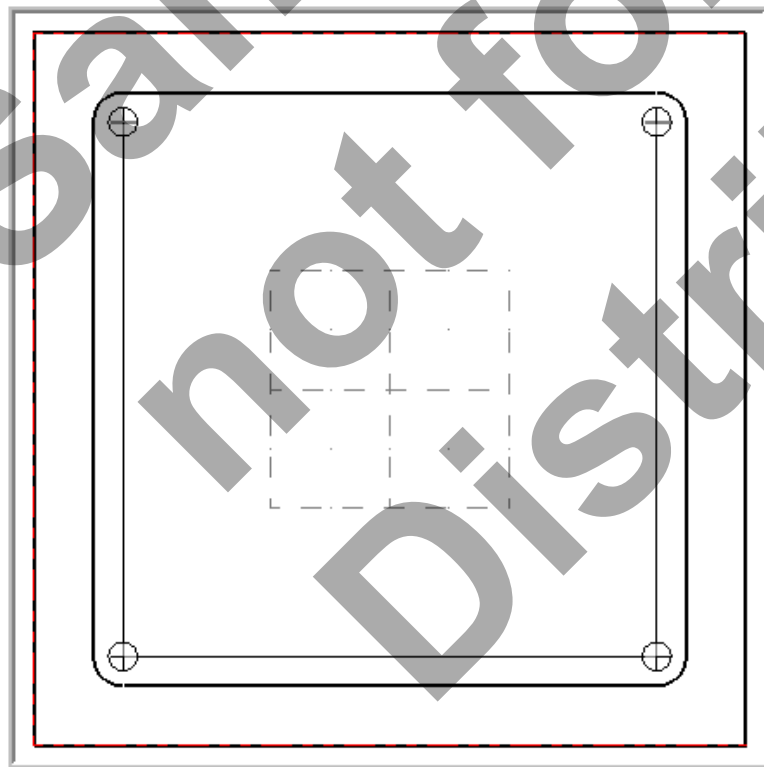


33. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.



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

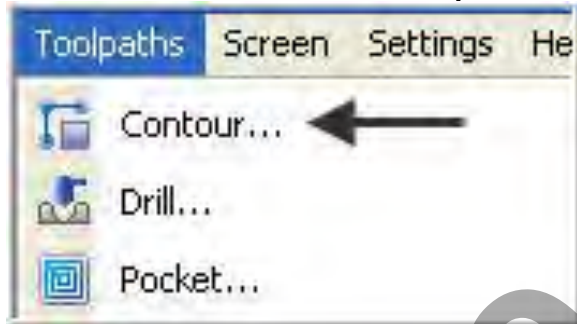
35. Your part should look like the screenshot below:



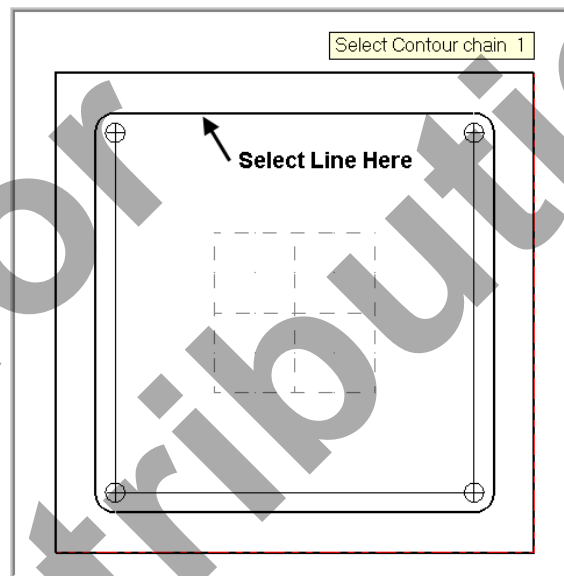
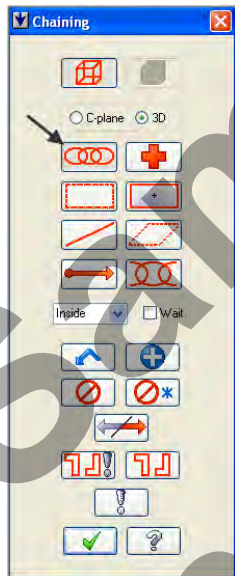
TOOLPATH TASK 5: MACHINE THE CONTOUR.

- 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.

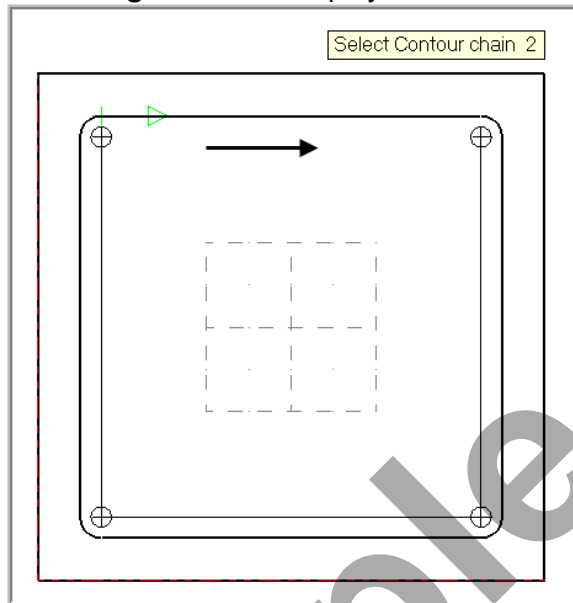
1. From the menu bar select **Toolpaths>Contour...**



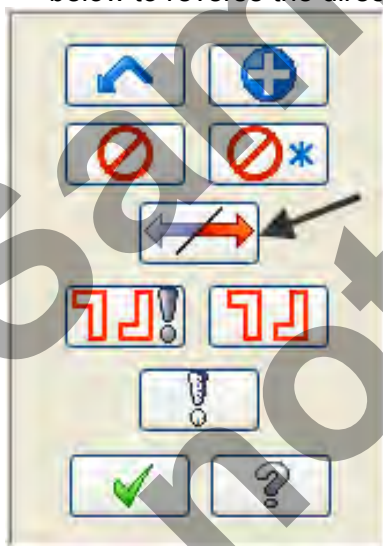
2. On the screen you will now see the **Chaining** dialog box with **Chain** set and in the graphics screen a prompt to "**Select Contour chain 1**". Select the top horizontal line as shown below:



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.



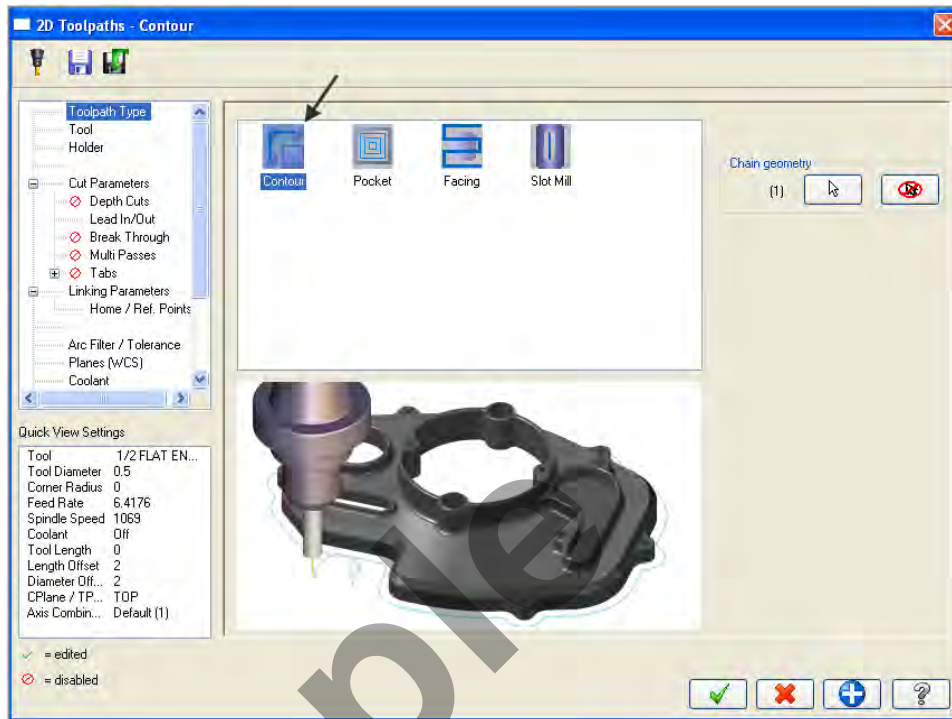
4. If the arrow is not pointing to the right select the arrow from the **Chaining** dialog box shown below to reverse the direction:



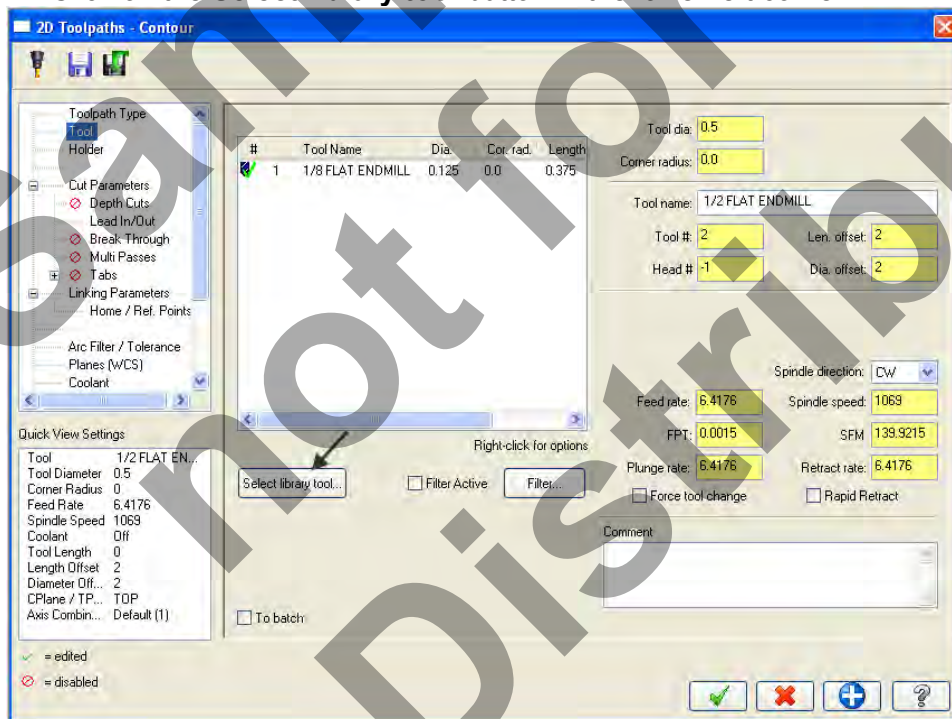
5. After the contour has been successfully chained select the **OK** button  at the bottom of the Chaining dialog box.

SolidWorks and Mastercam Training Guide

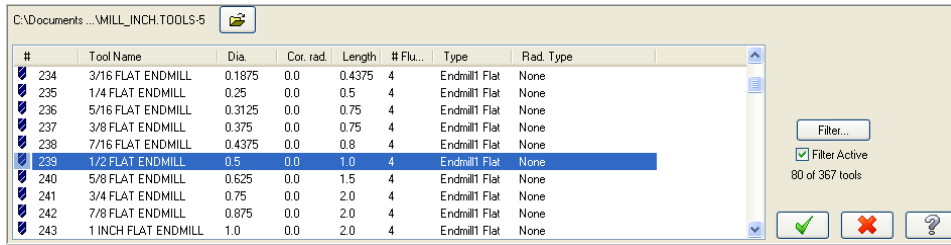
6. Ensure the **Toolpath Type** is set to **Contour** as shown below and then select **Tool** from the list on the left.



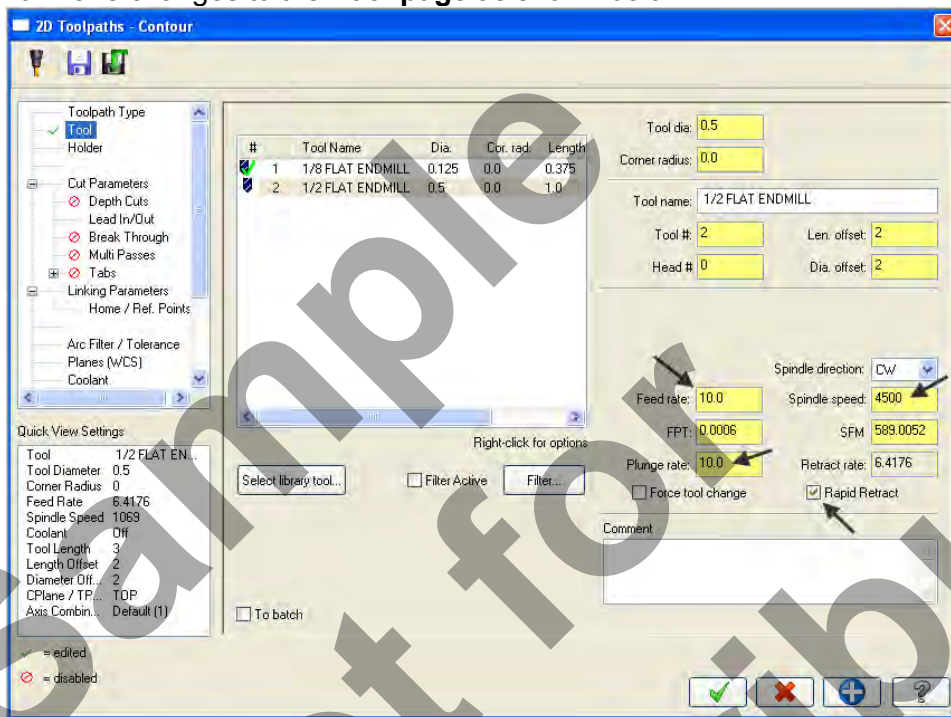
7. Click on the **Select library tool** button in the lower left corner.



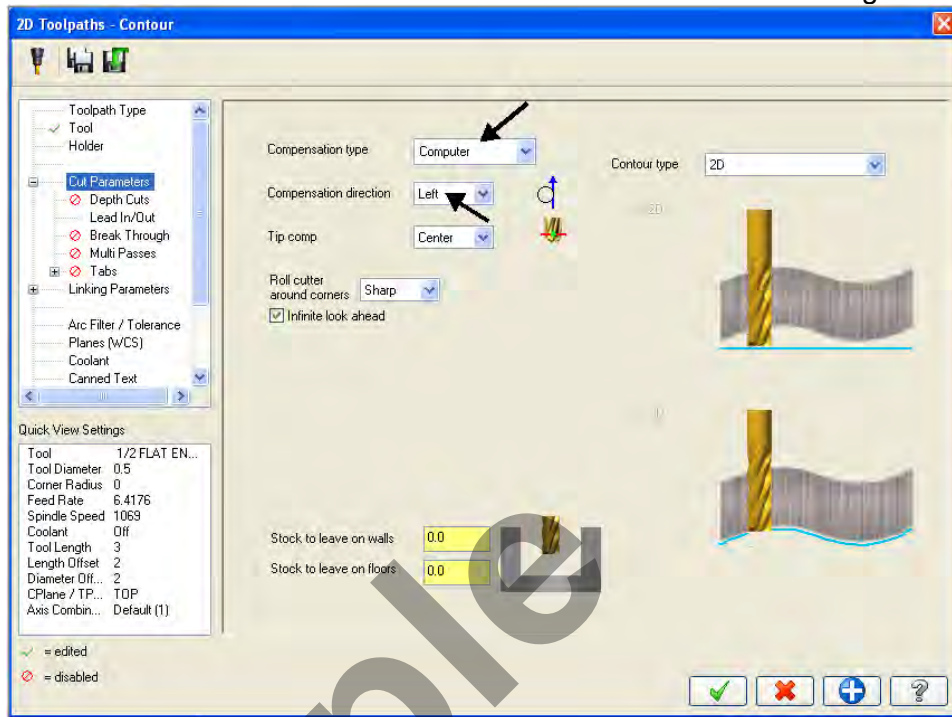
- Use the slider bar on the right of this dialog box to scroll down and locate a **.5" diameter flat end mill**. Select the .5" diameter flat end mill by picking anywhere along the .5 end mill row, as shown below:



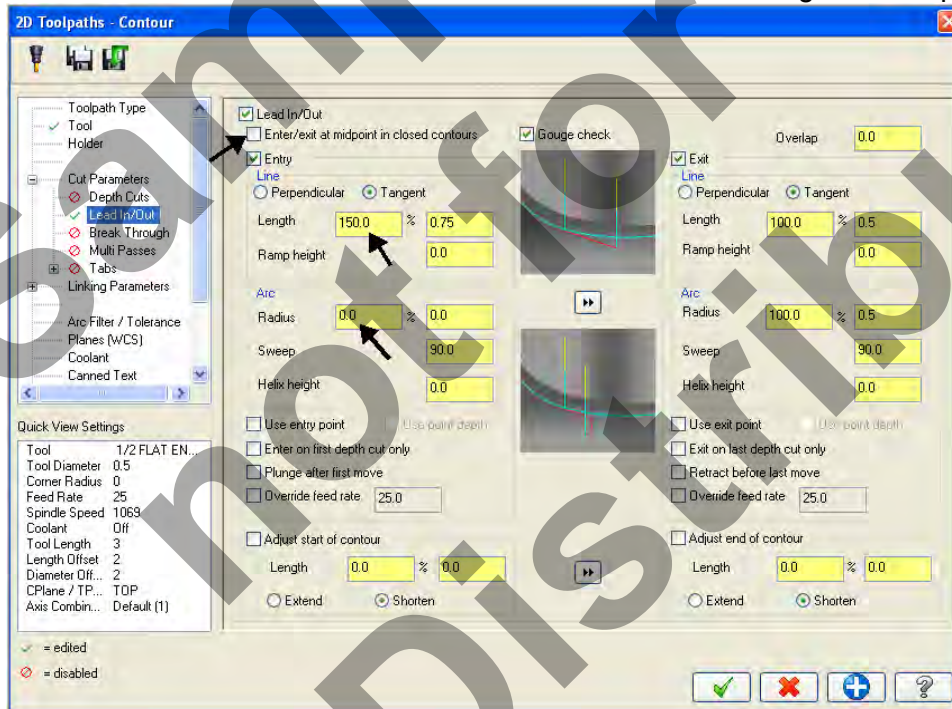
- Select the **OK** button to complete the selection of this tool.
- Make changes to the **Tool** page as shown below:



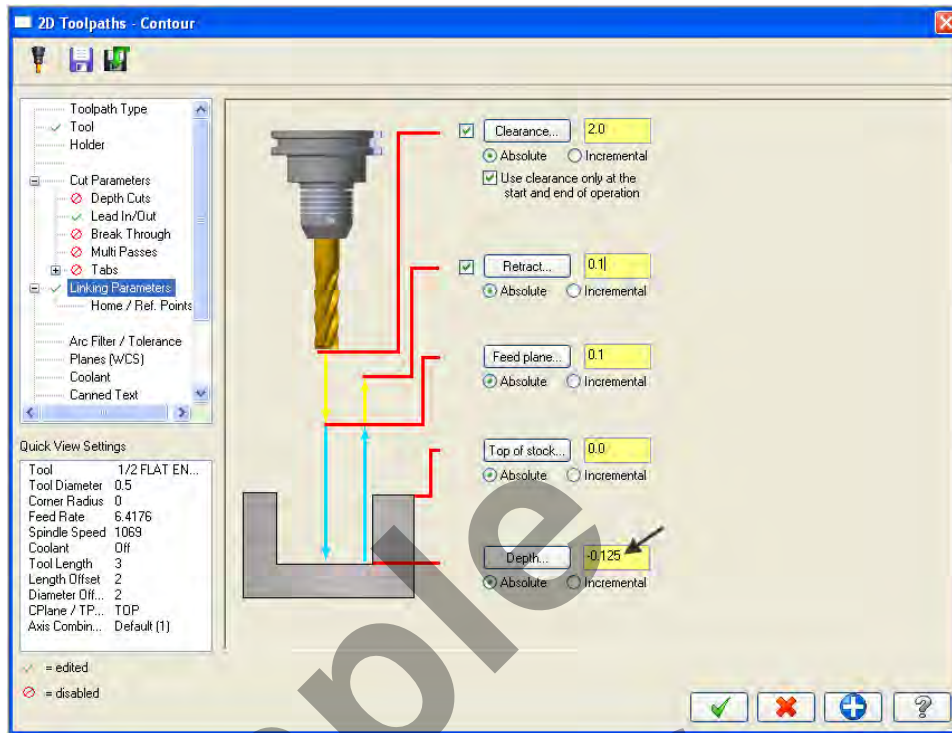
11. Select **Cut Parameters** from the list on the left and make changes to this page if required.



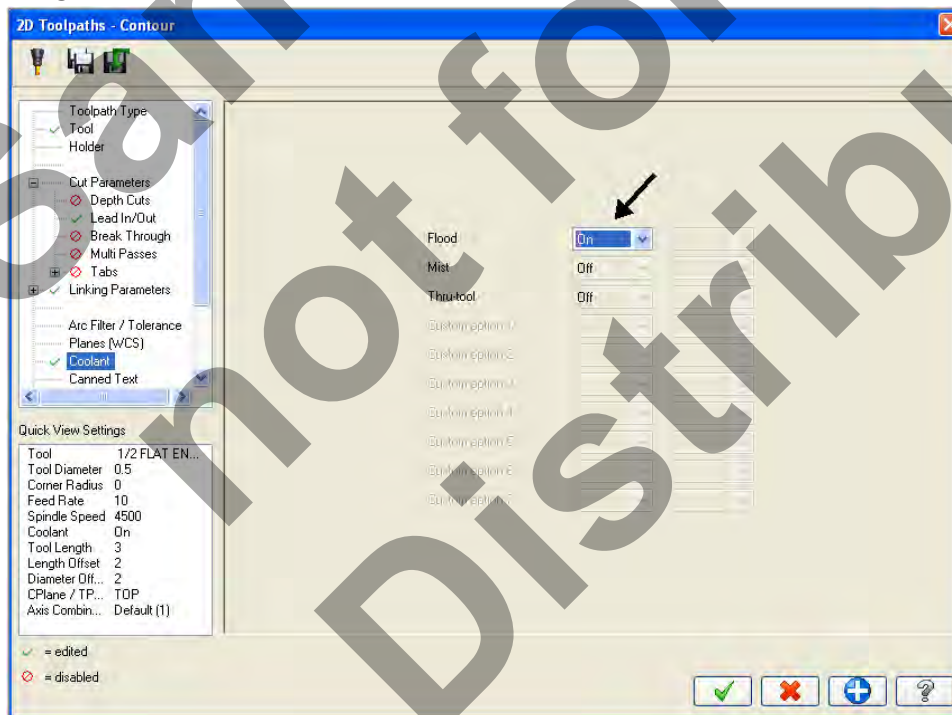
12. Select **LeadIn/Out** from the list on the left and make changes to this page.



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**.

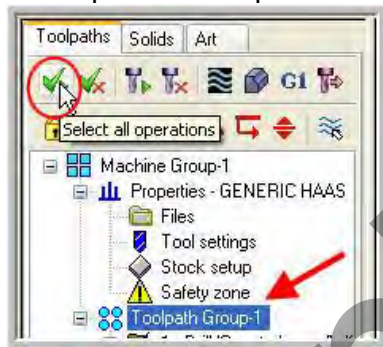


15. Select the **OK** button  to complete this function

TOOLPATH TASK 6: BACKPLOT THE TOOLPATH

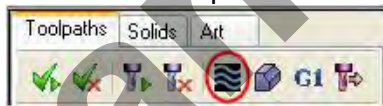
- In this task you will use Mastercam's Backplot function to view the path the tools take to cut this part.
- Backplot will enable us to review the cutting motions and identify any problem areas when cutting the part.
- When the toolpath is being Backplotted Mastercam displays the current X, Y, and Z coordinates in the left side of the status bar at the lower left corner of the screen.

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

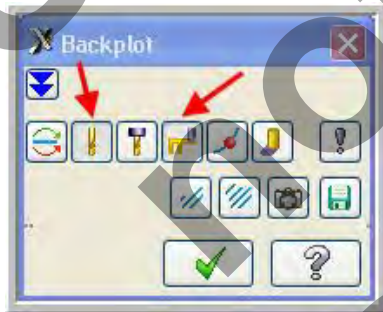


- Another method to **Select all** the operations is by clicking on the **Toolpath Group-1** in the Tool Manager as shown by the arrow above.

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



3. Before you Backplot the toolpath ensure the two buttons shown below are activated. The option on the left will **Display Tool** and the option on the right will **Display rapid moves**. These buttons act like a toggle switch, pressed in activates the function.

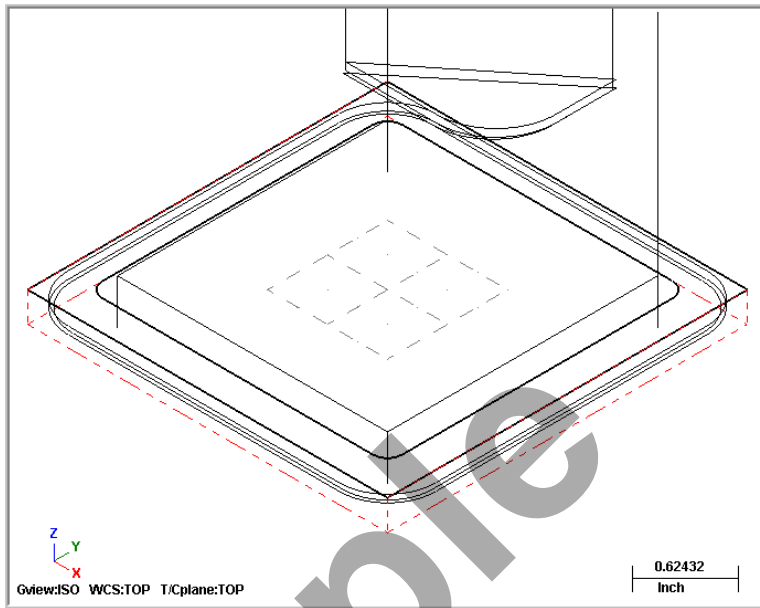


4. Select the Isometric view from the view toolbar.



5. Select the Screen Fit icon to fit the part to the screen .

6. Set the run speed on the Backplot controls midway along the speed bar as shown by the arrow below and then select the play button:

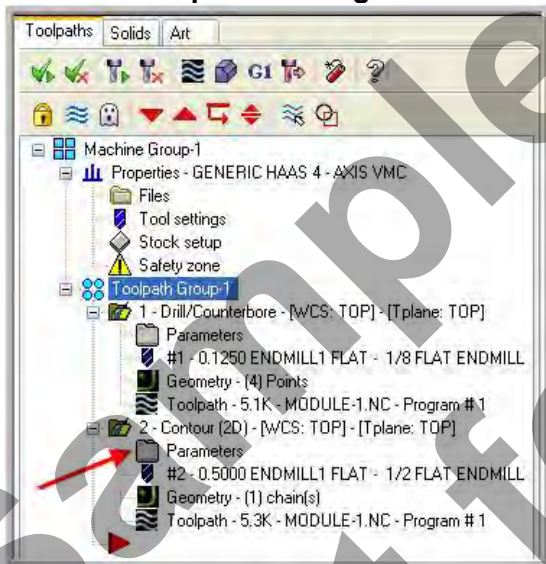


7. After reviewing the backplot of the two toolpaths using a .125" and .5" end mill select the OK button to exit Backplot.

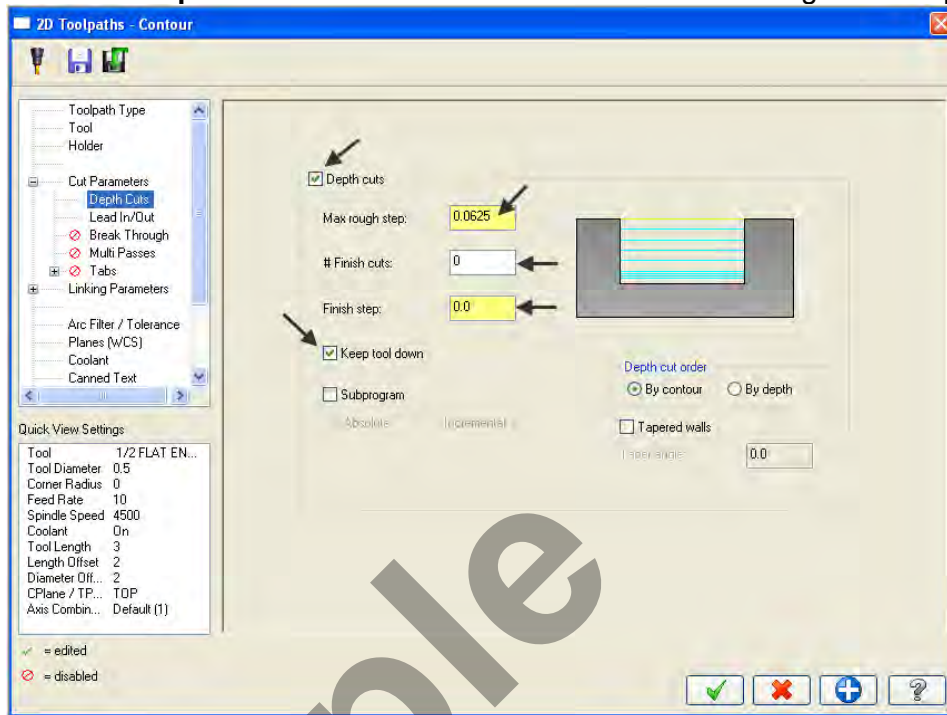
TOOLPATH TASK 7: MODIFY THE CONTOUR TOOLPATH TO ADD ROUGHING CUTS AND A FINISH PASS

- 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:

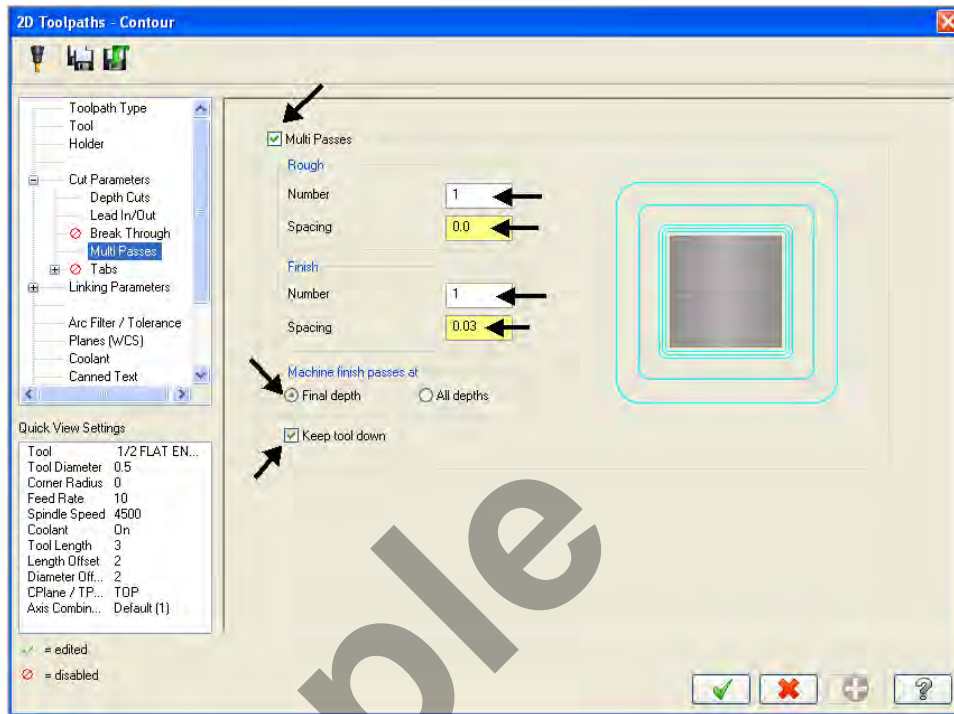
Each depth of cut will not exceed .0625", therefore as your final depth is -.125" you will perform only two rough cuts. The first at -.0625 and the second rough cut at -.125

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.

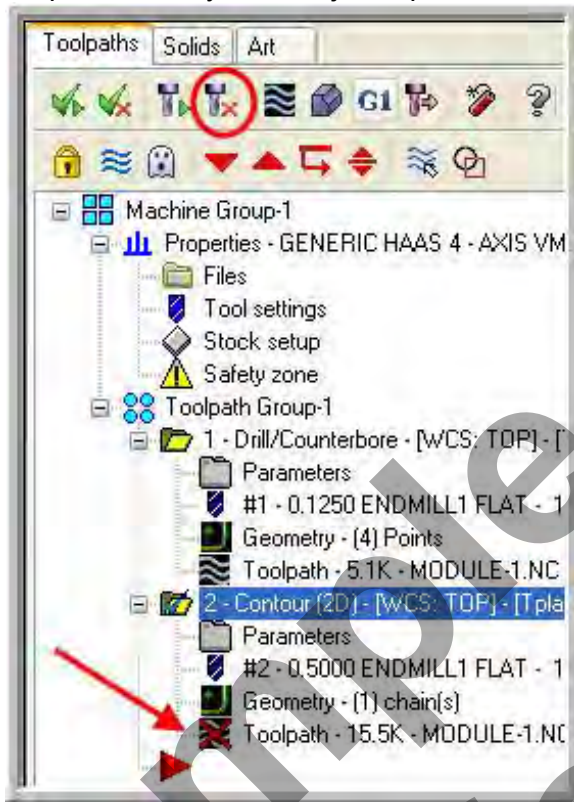
While cutting at the various depths you will stay .030" away from the contour.

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

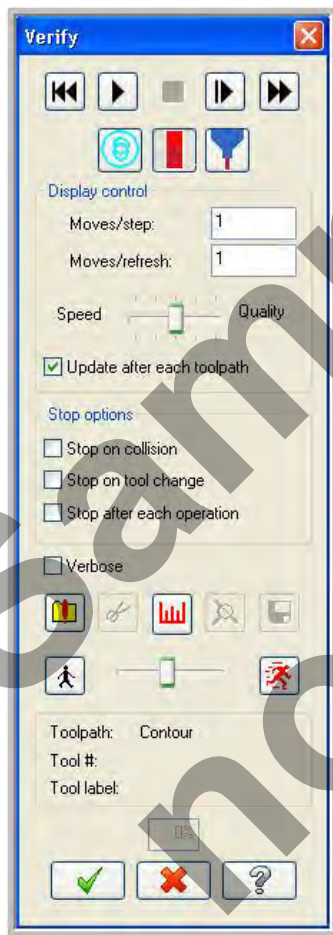
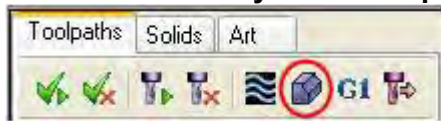
This happens if you have changed certain parameters of the underlying geometry, or in this example you have updated the contour toolpath to use Depth of cuts and multi passes.

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

TOOLPATH TASK 8: VERIFY THE TOOLPATH

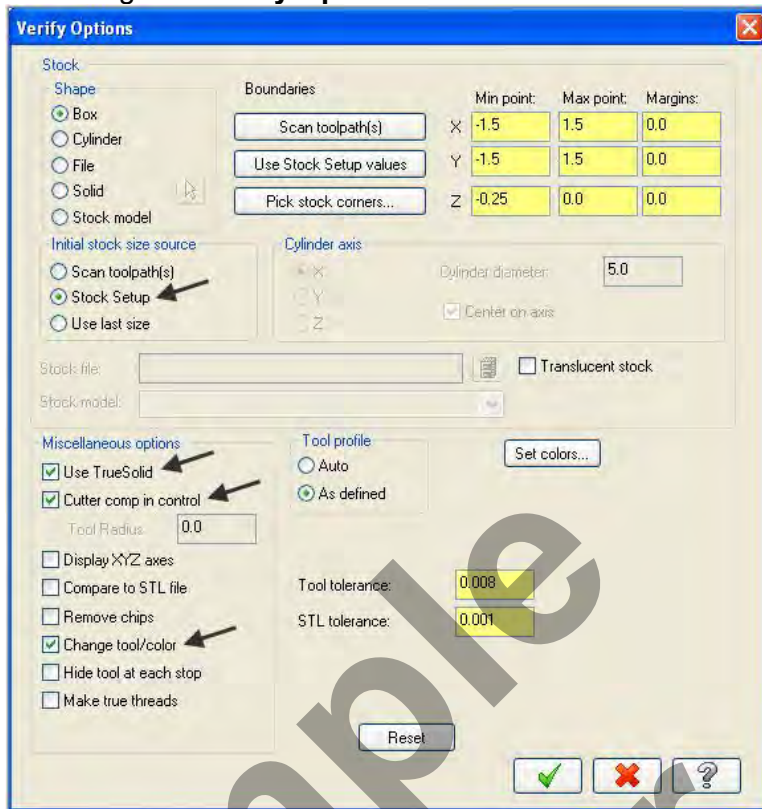
- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part and shows collisions, if any exist.
- This allows you to identify and correct program errors **before** they reach the shop floor.

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



3. Select the **Options** button 

4. Change the **Verify Options** to those shown below:



About the Verify Options dialog box

Initial stock size source:

Stock Setup:

Uses any available stock boundary information from the Job Setup dialog box to determine the stock boundaries. This is the default option.

Miscellaneous options

Use TrueSolid:

Switches between Standard mode and TrueSolid mode. Standard mode is pixel-based, while TrueSolid uses advanced solid modeling technology to create and manipulate complete and accurate solid models for toolpath simulation. TrueSolid also uses OpenGL graphics for dynamic 3D solid rendering and animation.

Cutter compensation in control:

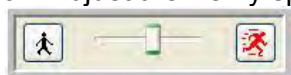
Allows you to view simulated cutter compensation during the verification process, if you selected cutter compensation in control in the toolpath.

Change tool/color:

Changes the color of the cut stock to indicate tool changes in the toolpath. You can set these colors by choosing the Set colors button. Once Mastercam has reached the last color, all subsequent tool changes remain in the last cut stock color.

5. Select the **OK** button  to exit **Verify Options**.

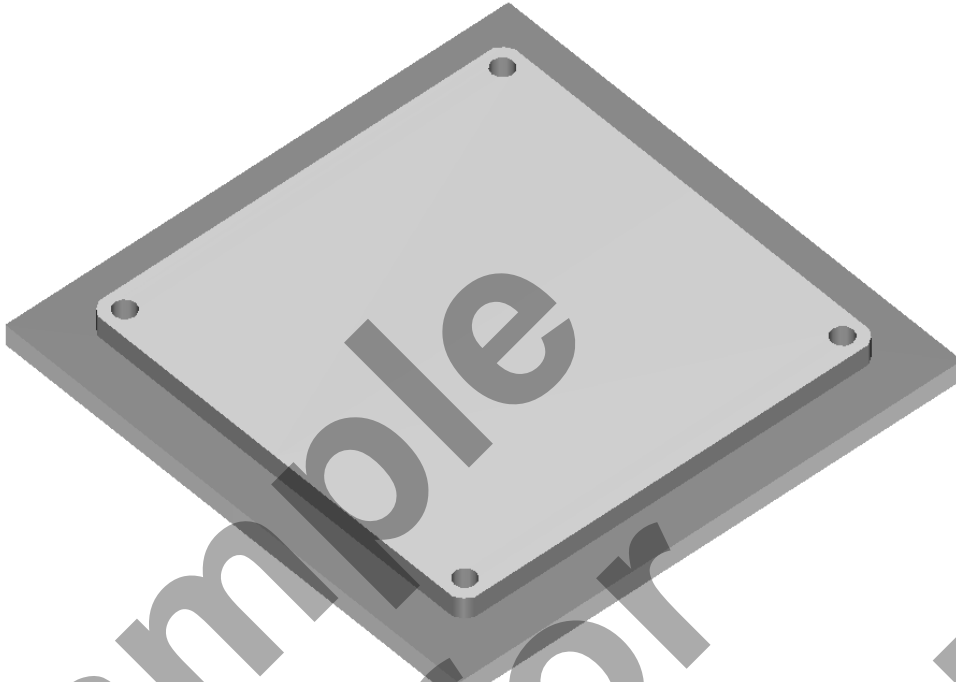
6. Adjust the Verify speed to **midway** along the speed control bar.



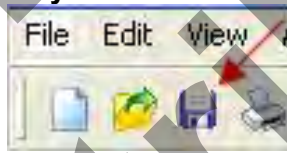
7. Select the **play** button to verify the two toolpaths.



8. The verified toolpaths should appear as in the picture below:




9. Select the OK button to exit **Verify**.



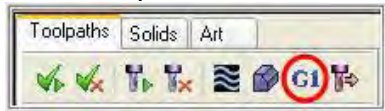
10. Select the **save** icon from the toolbar

TOOLPATH TASK 9: POST AND CREATE THE CNC CODE FILE

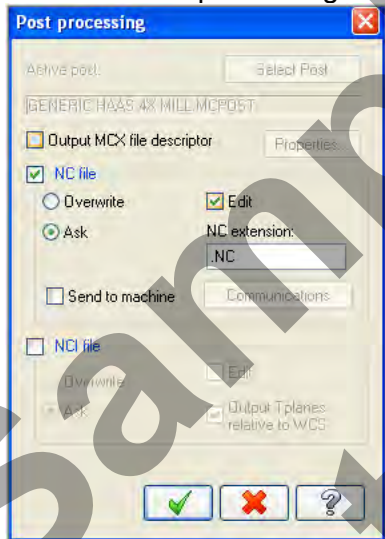
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.



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 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.

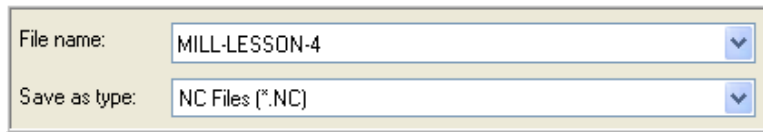
Edit:

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.

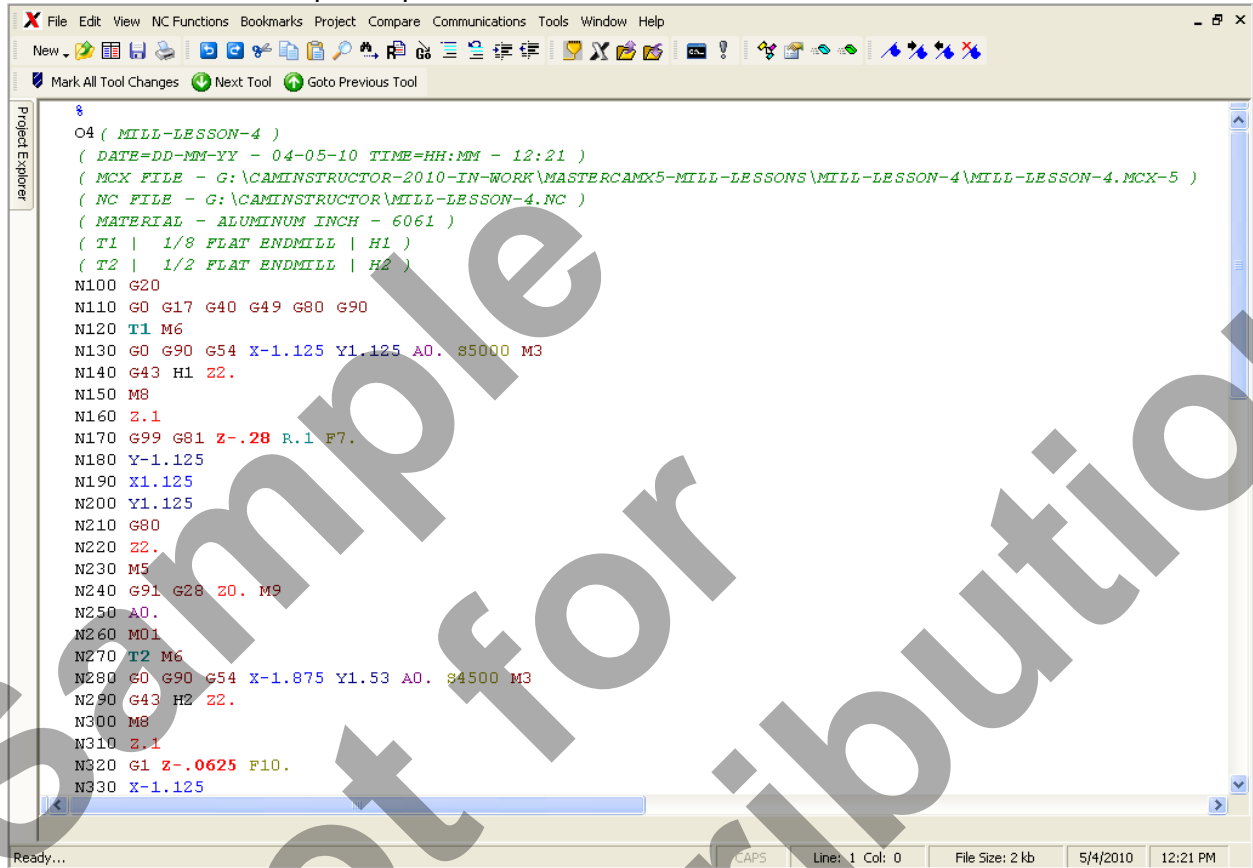
SolidWorks and Mastercam Training Guide

5. Ensure the same name as your Mastercam part file name is displayed in the **NC File name** field as shown below:



File name: MILL-LESSON-4
Save as type: NC Files (*.NC)

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



```
O4 ( MILL-LESSON-4 )
( DATE=DD-MM-YY - 04-05-10 TIME=HH:MM - 12:21 )
( MCX FILE - G:\CAMINSTRUCTOR-2010-IN-WORK\MASTERCAMX5-MILL-LESSONS\MILL-LESSON-4\MILL-LESSON-4.MCX-5 )
( NC FILE - G:\CAMINSTRUCTOR\MILL-LESSON-4.NC )
( MATERIAL - ALUMINUM INCH - 6061 )
( T1 | 1/8 FLAT ENDMILL | H1 )
( T2 | 1/2 FLAT ENDMILL | H2 )
N100 G20
N110 G0 G17 G40 G49 G80 G90
N120 T1 M6
N130 G0 G90 G54 X-1.125 Y1.125 A0. S5000 M3
N140 G43 H1 Z2.
N150 M8
N160 Z.1
N170 G99 G81 Z-.28 R.1 F7.
N180 Y-1.125
N190 X1.125
N200 Y1.125
N210 G80
N220 Z2.
N230 M5
N240 G91 G28 Z0. M9
N250 A0.
N260 M01
N270 T2 M6
N280 G0 G90 G54 X-1.875 Y1.53 A0. S4500 M3
N290 G43 H2 Z2.
N300 M8
N310 Z.1
N320 G1 Z-.0625 F10.
N330 X-1.125
```

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

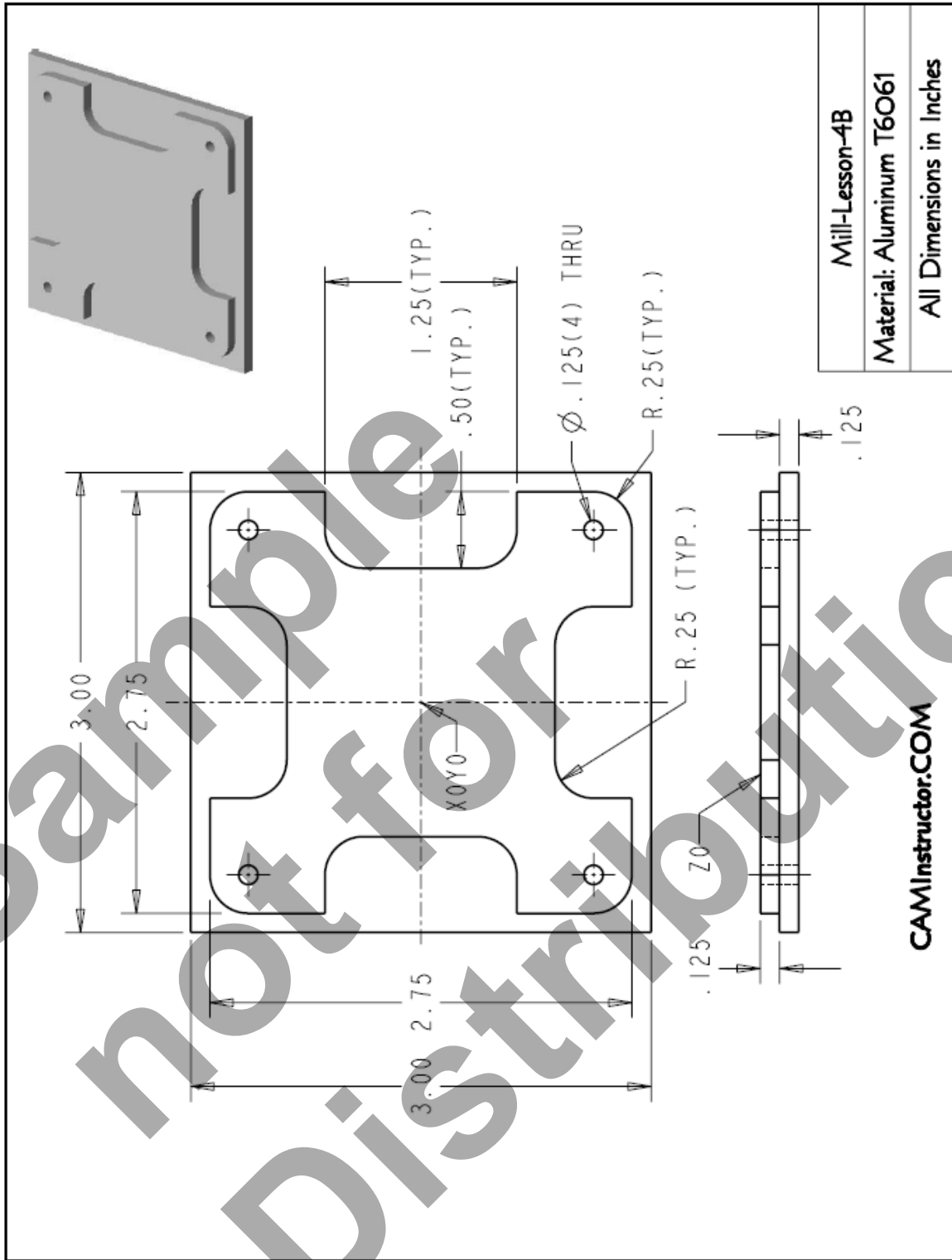
MILL-LESSON-4 EXERCISES

The technical drawing shows a rectangular plate with the following specifications:

- Overall dimensions: 3.00 inches by 3.00 inches.
- Internal dimensions: 2.00 inches by 2.75 inches.
- Four corner radii: R.25.
- Two circular holes: $\phi .125$ (2).
- Coordinate system: X0Y0 at the center.
- Thickness: .125 inches.
- Reference planes: Z0 at the top and bottom surfaces.

Mill-Lesson-4 Exercise
Material: Aluminum T6061
All Dimensions in Inches

CAMInstructor.COM



CAMInstructor.COM