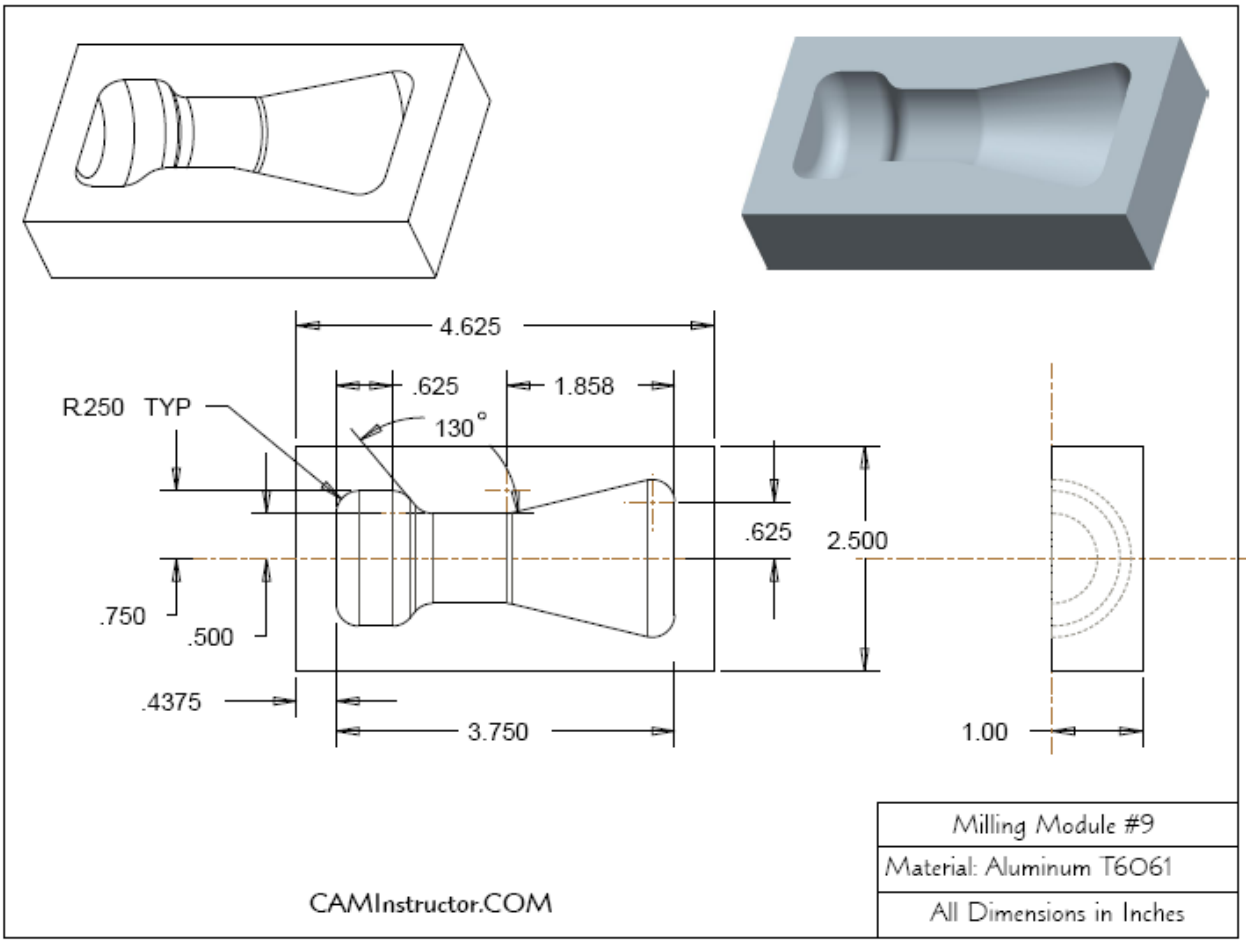
Mastercam X6
for Solidworks

TRAINING GUIDE



LESSON-9-CAD

camInstructor

Objectives

You will create the geometry for SolidWorks-Lesson-9 using **SolidWorks 3D CAD software**. You will be working through the process to machine parts in later Lessons; SolidWorks-Lesson-9 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.

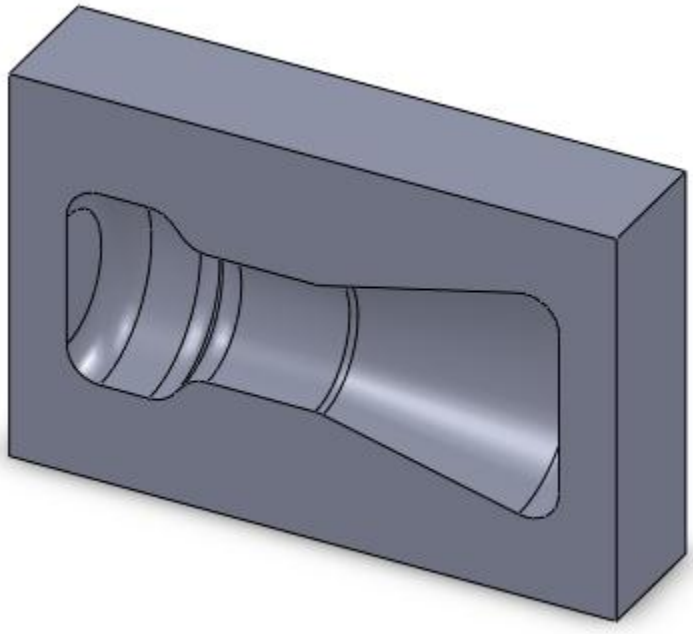
➲ **Extruding 2-dimensional sketches by:**
Using the extrude tool to extrude a base using fully defined sketches to build geometry.

➲ **Revolving and Cut Extrude by:**
Using the revolved cut tool to revolve the fully defined sketch and produce the 3D cavity completing the model.

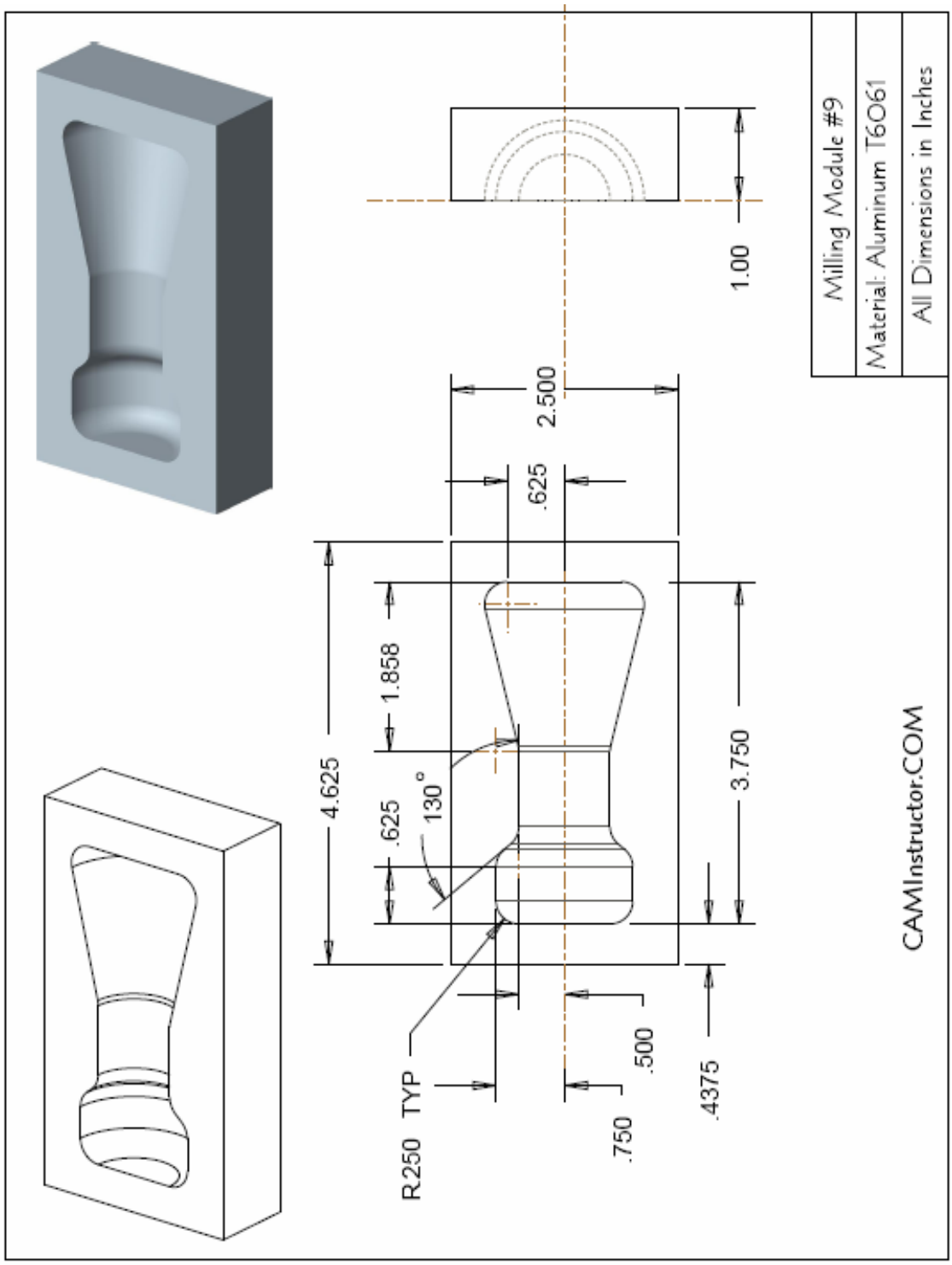
➲ **Establish Stock Setup settings:**
Stock size using Bounding Box.
Material for the part.
Feed calculation.

➲ **Generate a 3-dimensional milling toolpath consisting of:**
Surface Pocket.
Surface High Speed Waterline.

➲ **Inspect the toolpath using Mastercam's Verify and Backplot by:**
Launching the Verify function to machine the part on the screen.
Generating the NC- code.



MASTERCAM FOR SOLIDWORKS-LESSON-9 DRAWING

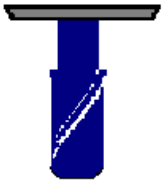


TOOL LIST

- 0.500 diameter bull end mill with a 0.125 corner radius to rough machine the pocket.
- 0.500 diameter ball end mill to finish machine the pocket.

Tool List of MILL-LESSON-9.MCX-5

Proj./Part No.: 0			Date	: 05/25/10	
Drawing No. : 1			Customer	: -	
Prog. No. : 9			Programmer	: 1	



Tool type	:	0.5 Endmill3 Bull 1/2 BULL ENDMILL 0.125 RAD					
Manufact.code	:						
Chuck	:						
Tool Number	:	1				Feedrate	: 12
Diameter	:	0.5	RPM	: 2500		Plunge feed r.	: 12
Corner radius	:	0.125	Tip angle	: 0		Diam. offset	: 1
Flute length	:	1	Material	: ALUMINUM ...		Length offset	: 1
Overall length	:	3	No flutes	: 4			

Tool type	:	0.5 Endmill2 Sphere 1/2 BALL ENDMILL					
Manufact.code	:						
Chuck	:						
Tool Number	:	2				Feedrate	: 15
Diameter	:	0.5	RPM	: 2500		Plunge feed r.	: 15
Corner radius	:	0.25	Tip angle	: 0		Diam. offset	: 2
Flute length	:	1	Material	: ALUMINUM ...		Length offset	: 2
Overall length	:	3	No flutes	: 4			

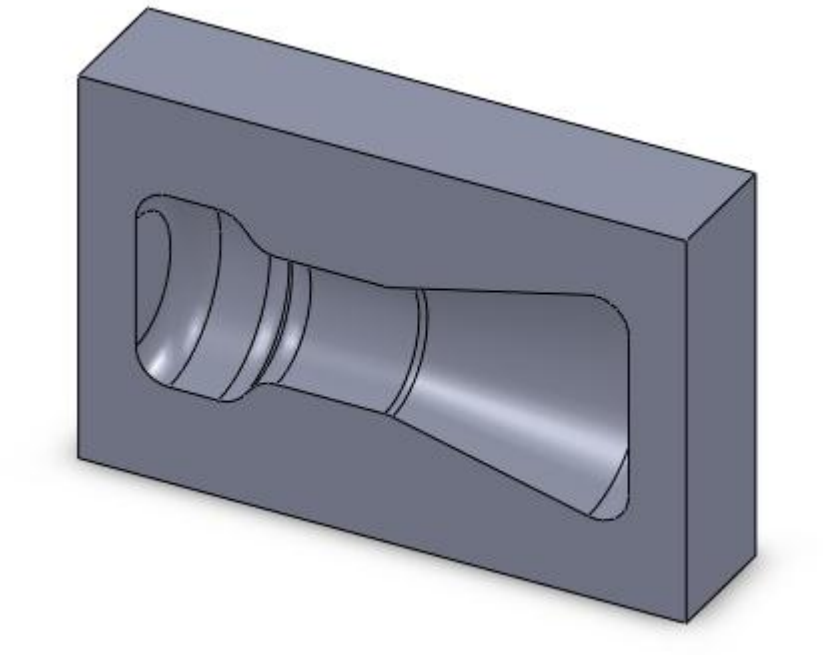
LESSON -9 - THE PROCESS

CAD

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 internal profile and perform a revolved cut to produce the cavity.

TOOLPATH

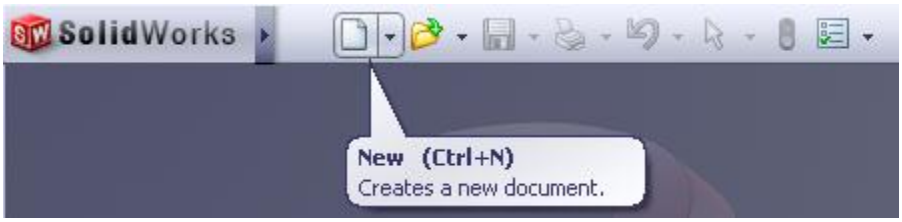
TASK 1: Define the Rough Stock.
TASK 2: Rough the Pocket using Surface Pocket.
TASK 3: Finish the Pocket using Surface High Speed Waterline.
TASK 4: Verify the Toolpath.
TASK 5: Post and create the CNC Code File.



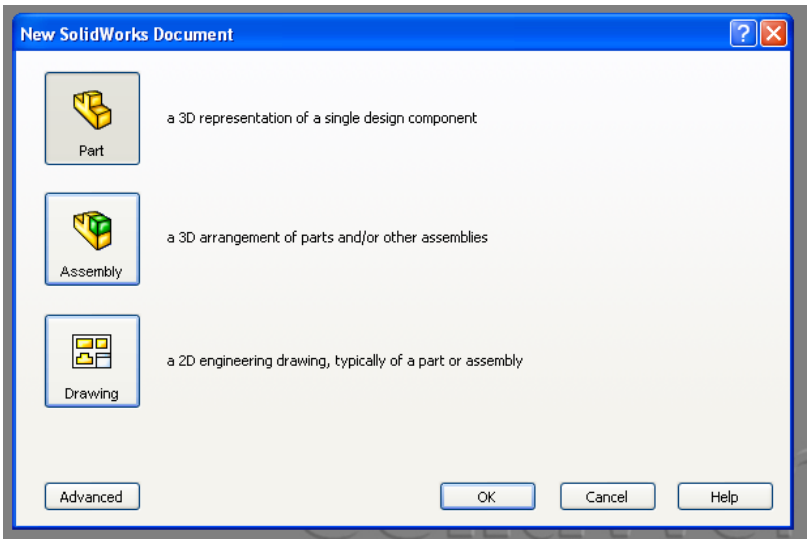
CAD TASK 1:
START A NEW PART AND CHOOSE THE SKETCH PLANE

Create a new part in SolidWorks:

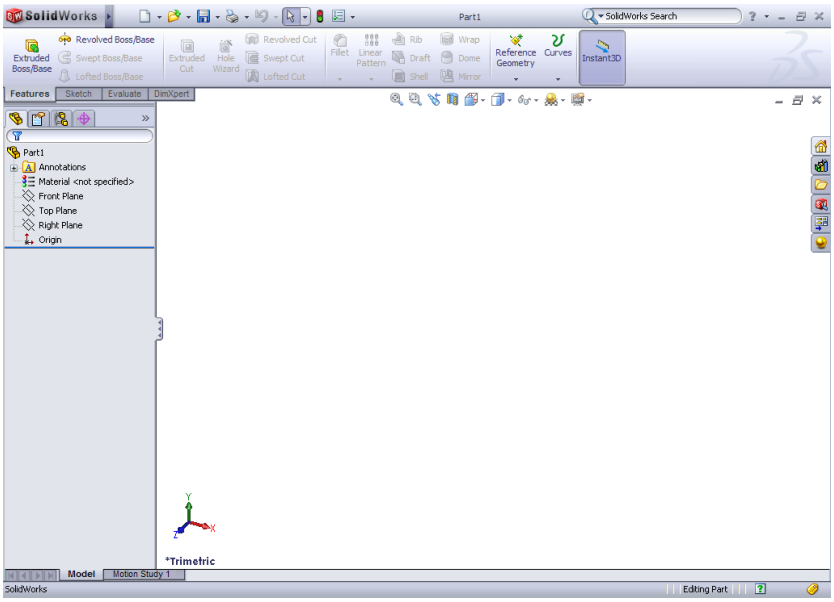
1. Click on **File New** or the **File New** icon  . This will create a new document.



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

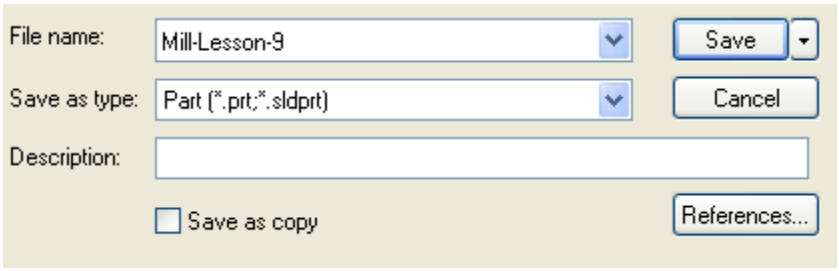


3. Your screen will look like the one below:

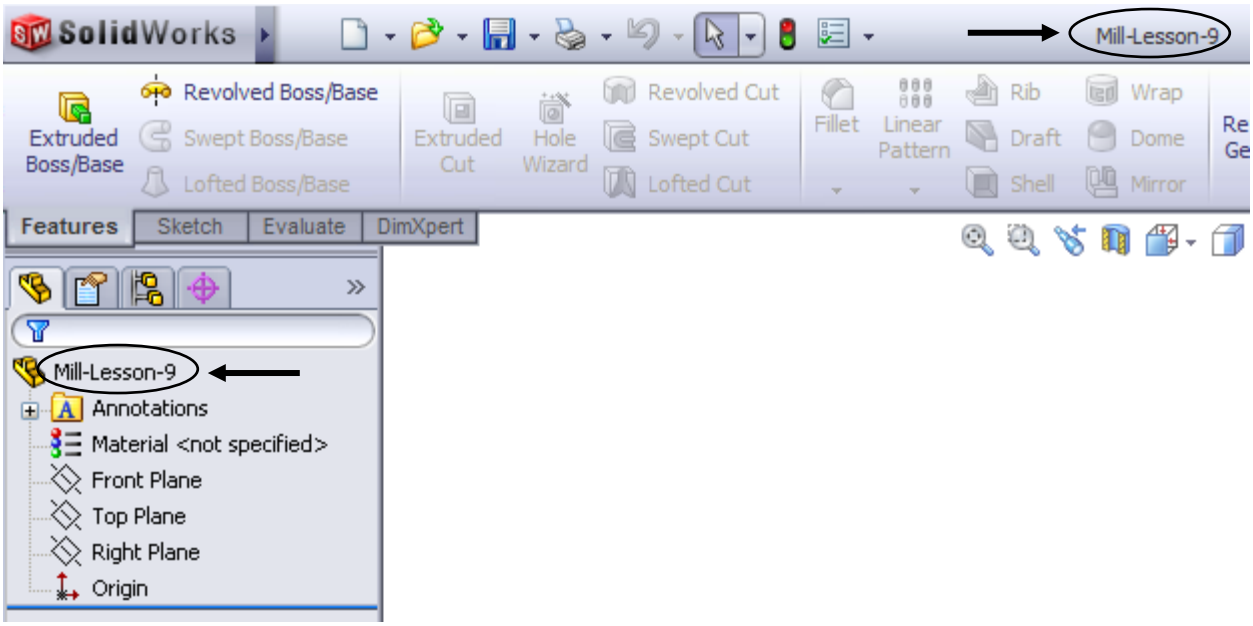


4. Click on the save icon on the menu bar (will perform a save as) or on the arrow beside to expand and click on **Save As**.

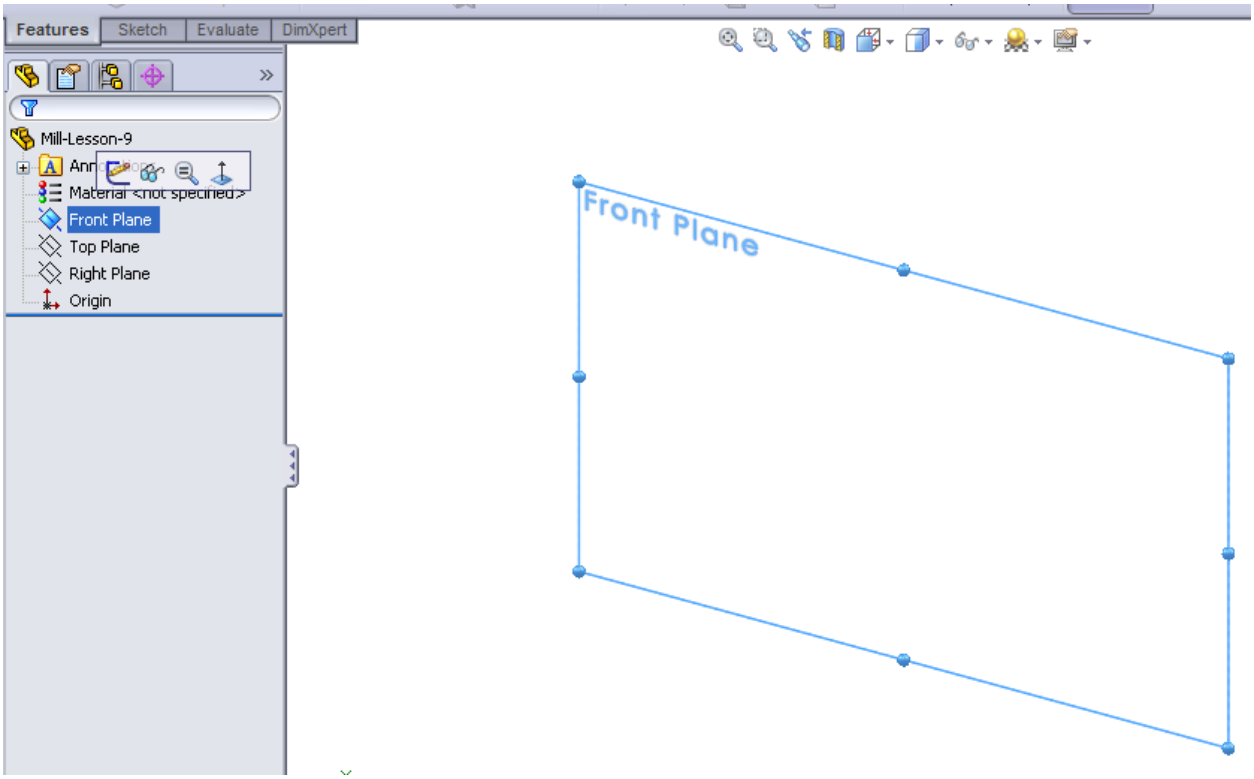
5. Save **Part 1** as **Mill-Lesson-9** in the appropriate directory location you are saving.



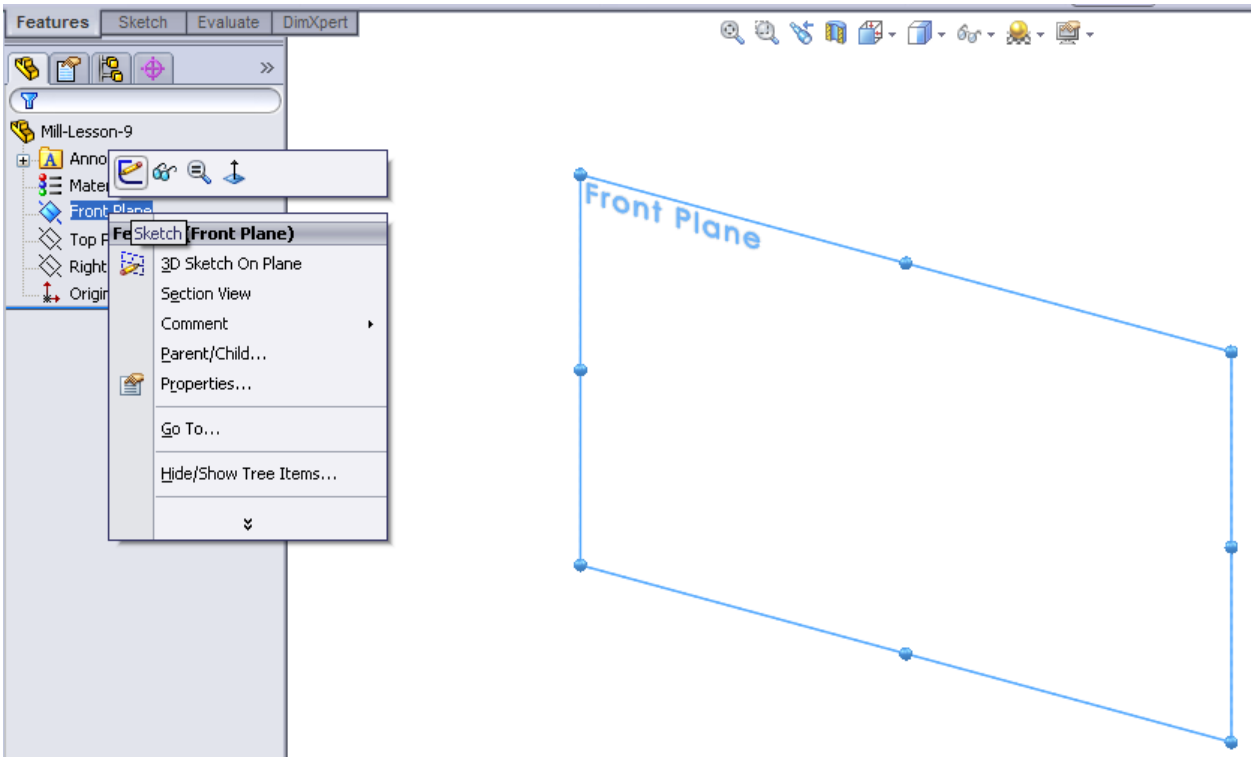
6. Note the part name change in the feature manager design tree from **Part1** to **Mill-Lesson-9** once saved. The part name will also display at the top beside the bar pull-downs.



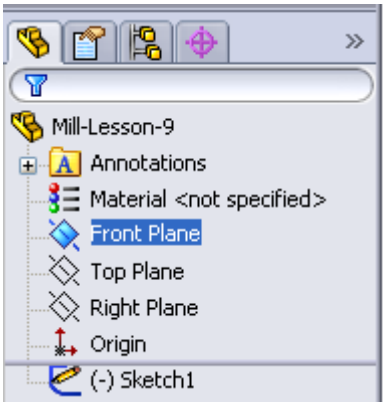
7. Create a sketch using the front plane. **Click on the Front Plane** in the Feature Manager as shown below: Note the rectangular image named Front Plane shown in the drawing area.



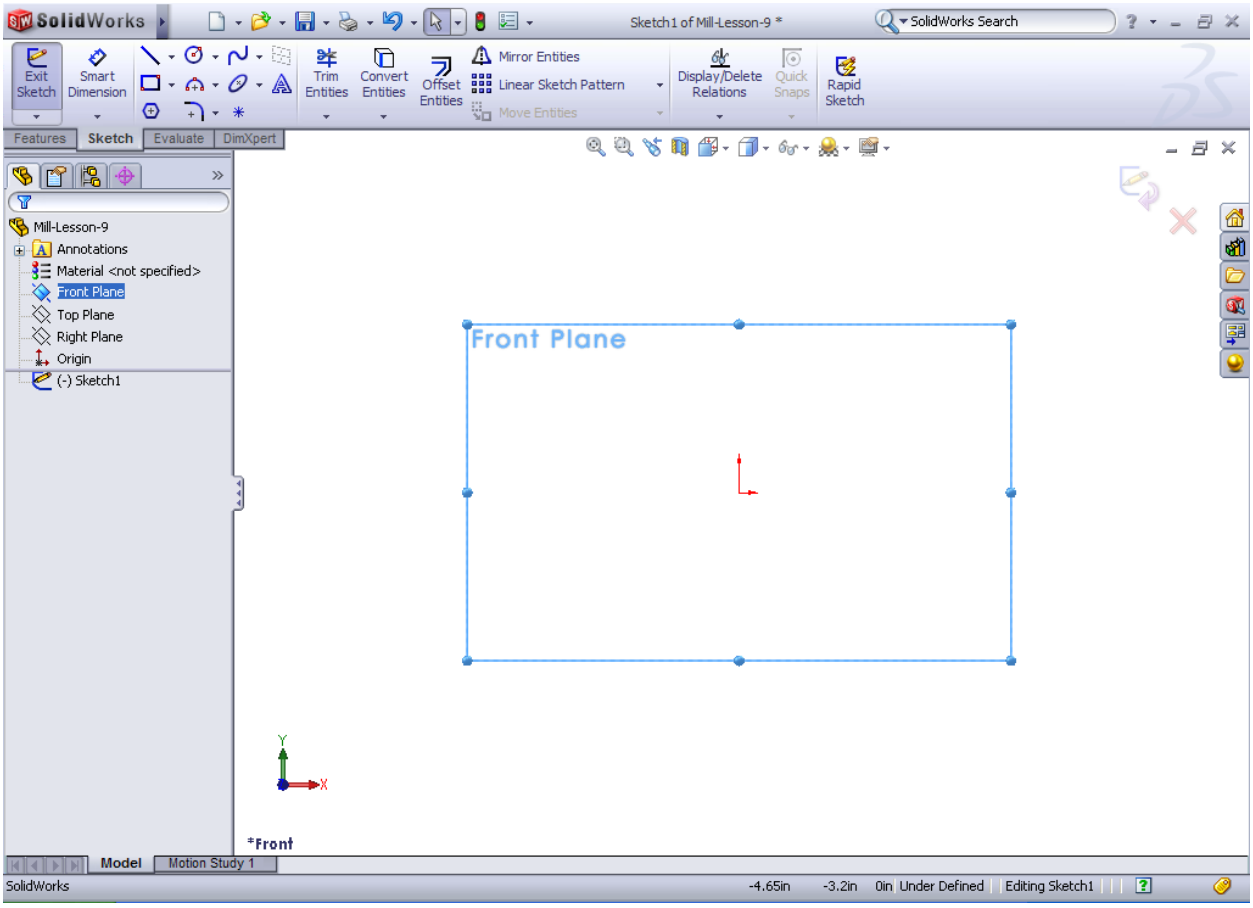
8. Right click on the highlighted plane and select the sketch icon as shown below: 



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

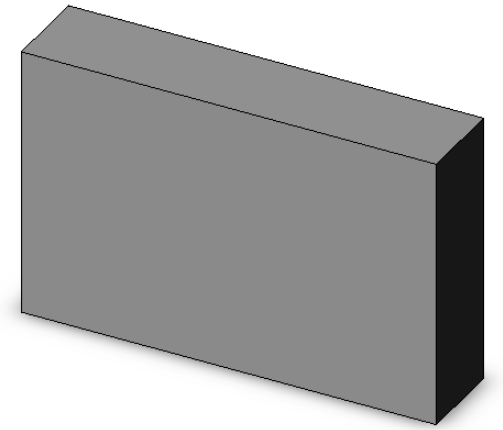


10. You are now ready to start creating geometry.



CAD TASK 2:
CREATE THE BASE SHAPE SKETCH AND EXTRUDE

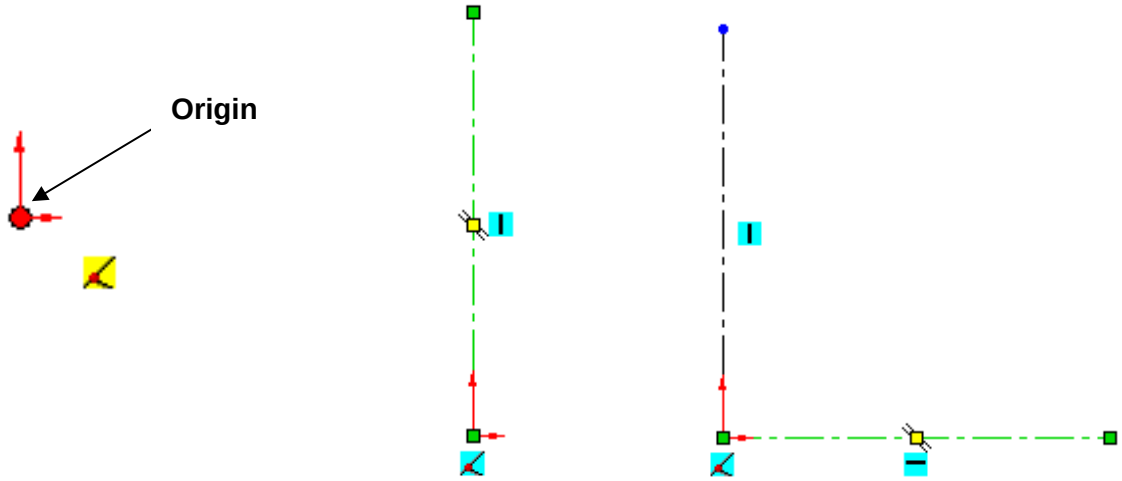
- This task explains how to create the part geometry for SolidWorks Lesson-9.
- Note the location of the origin and centerlines in relation to the geometry.
- You will start by creating the solid part to satisfy design intent.



➲ Create Centerline about the origin

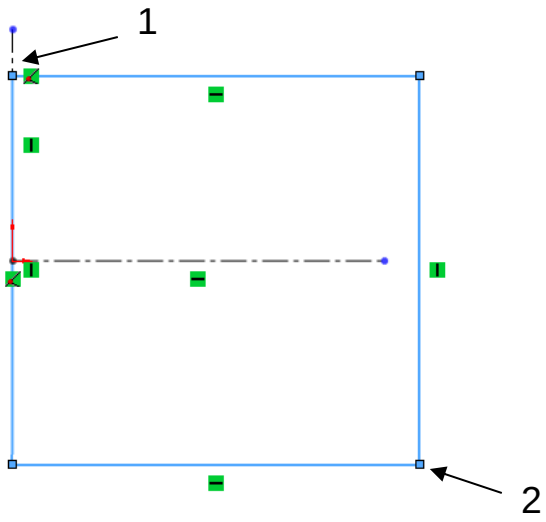
1. Select from sketch toolbar: click on the arrow pull down beside the line icon to click on **Centerline….**

2. 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.



➲ **Create Rectangle starting on the vertical centerline**

3. Click on the **corner rectangle icon** in sketch toolbar. Create a rectangle as shown below by clicking on the vertical centerline shown by number 1 below 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).

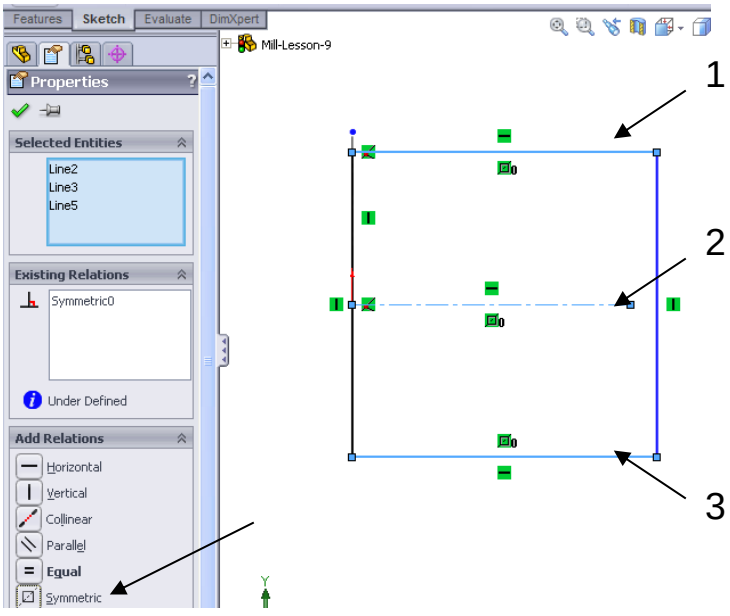


➲ **Add relations**

4. By starting the rectangle on the vertical centerline the edge of the rectangle is automatically coincident with the origin and thus its location is defined.

5. Click on the **horizontal line** (1 shown in diagram below). **Hold the control key down** and select the **horizontal centerline** (2) and the other **horizontal line** (3). Notice the properties box appears and shows the lines in the selected entities area.

6. Select symmetric to see the relation. This will make the **retangle symmetrical about the horizontal centerline**. Click on the **check mark** to accept the relation.



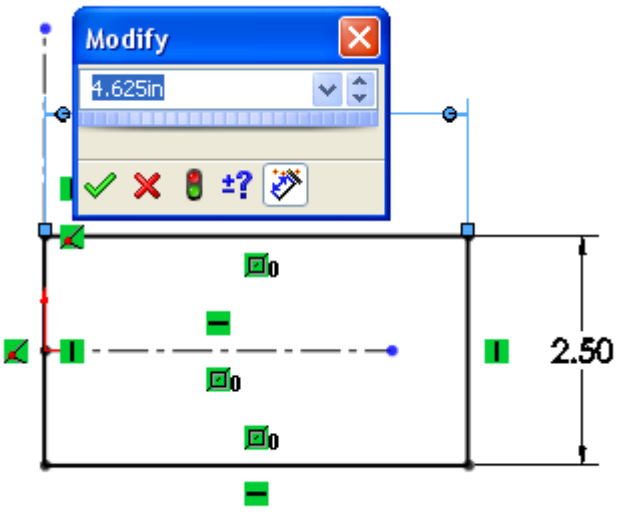
➲ **Add dimensions to fully defined rectangle**

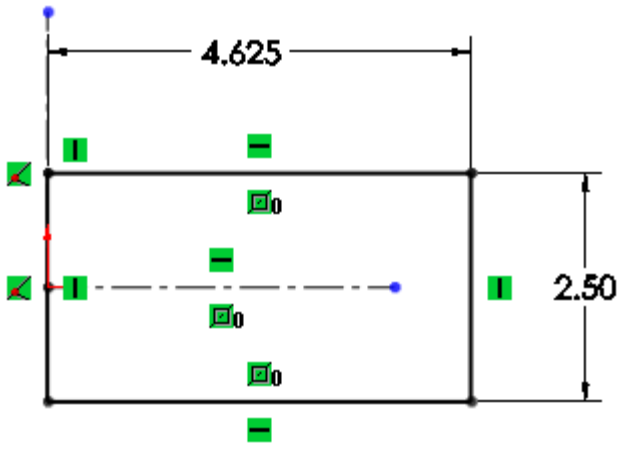
7. Click on the **smart dimension icon** 

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 **2.50** in the **Modify** window and accept. Refer to the Lesson #9 drawing on page 9-2 as a reference for the dimensions.

8. Next click on the **top horizontal line** and drag up to place the dimension. Enter **4.625** in the **Modify** window. Notice how it remains symmetrical about the origin.

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

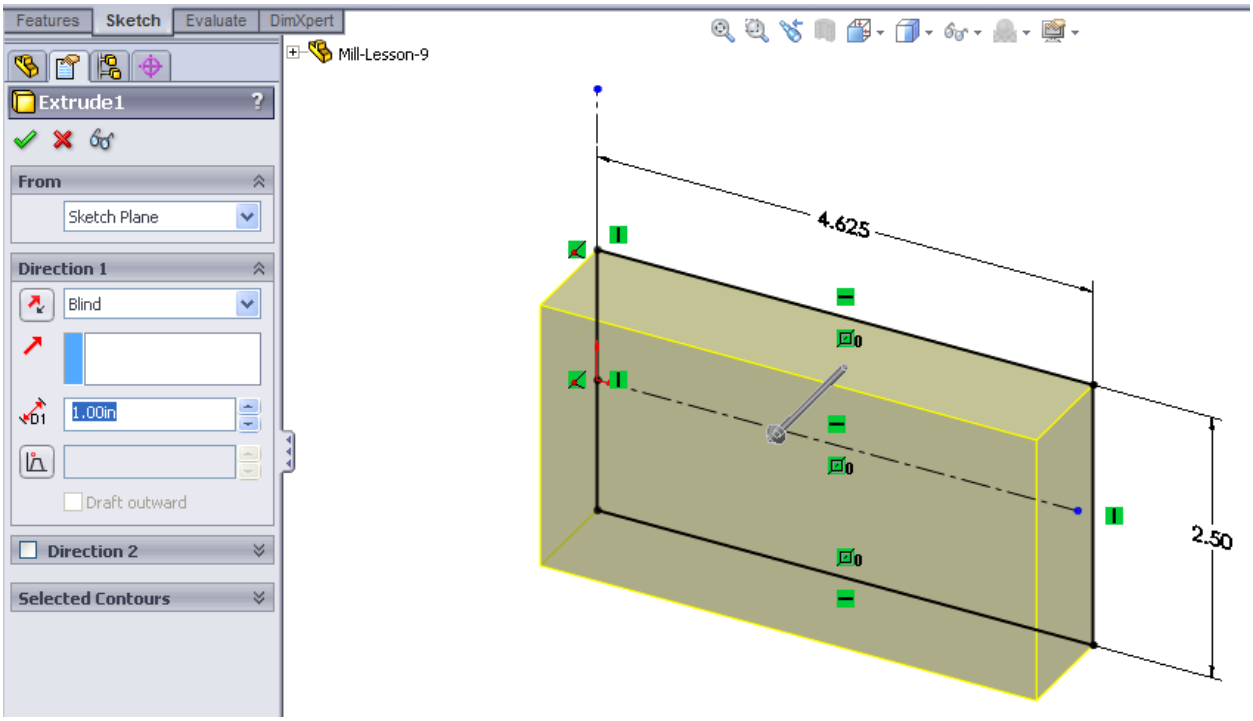


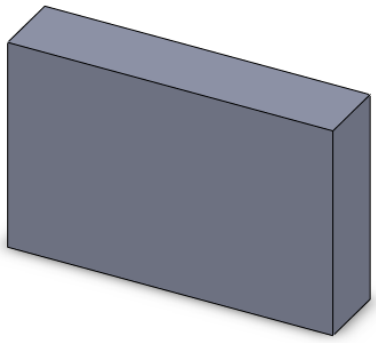


➲ **Extrude the fully defined sketch**

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

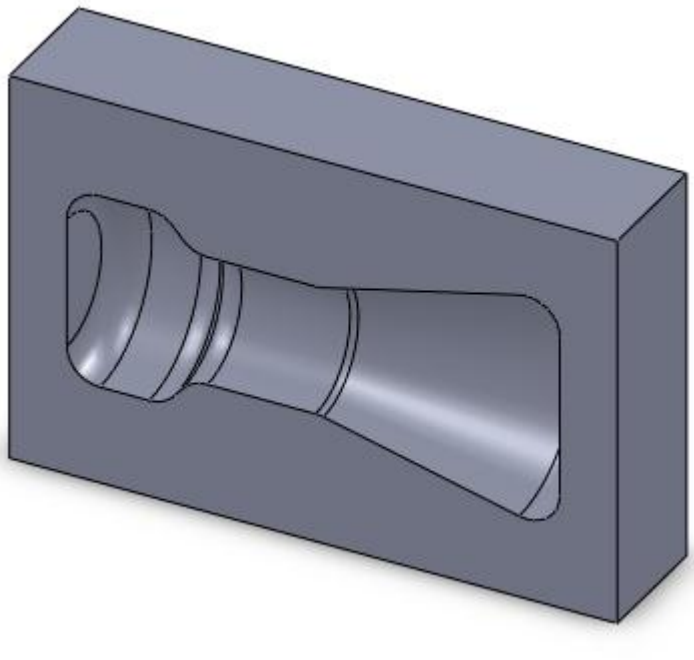
11. Under **Direction 1** in the Extrude Properties, select **Blind** from the drop down list. Then enter **1.00** in the **D1** window as shown below. Click on **OK**.





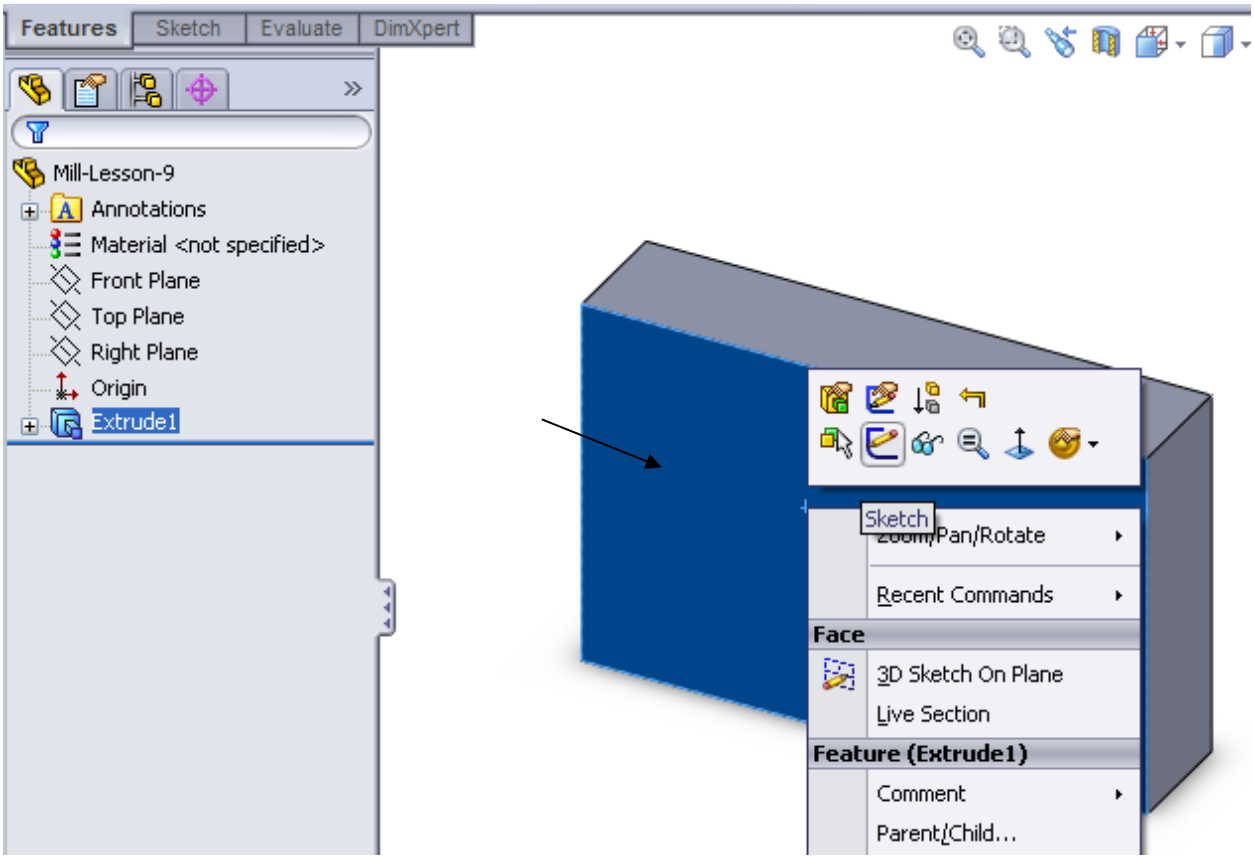
CAD TASK 3:
SKETCH INTERNAL PROFILE AND REVOLVE CUT…

- This task explains how to add a new sketch to create the revolved cut shape for SolidWorks-Lesson-9. The sketch entities will turn black and the sketch will indicate fully defined.
- Once sketch is fully defined perform a revolved cut into the 3D model.



➲ **Create new sketch**

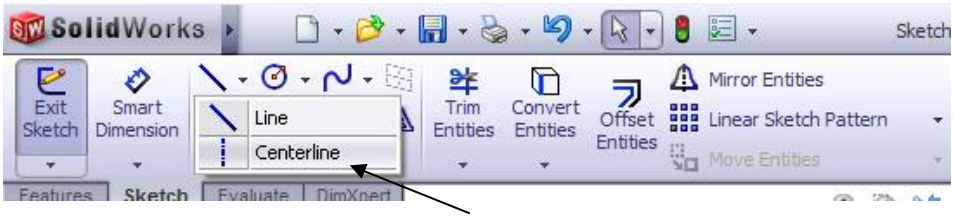
1. **Right click** on the front surface as shown below (indicated by arrow) and select **Sketch Icon**  . This will create a new sketch on this surface.



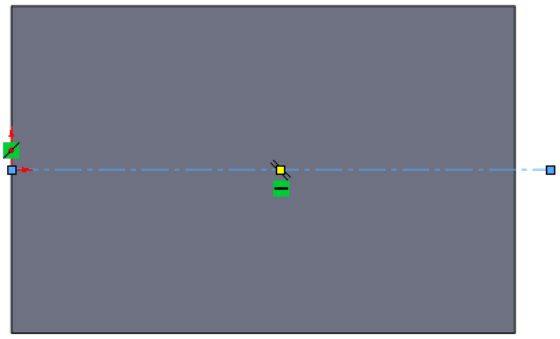
2. Press the space bar and the view orientation will appear. Double click on ***Normal To** to change the orientation to be looking straight on the sketch plane.

➲ **Create horizontal Centerline**

1. Select from sketch toolbar: click on the arrow pull down beside the line icon to click on **Centerline….**



2. Hover over the **origin**  of the sketch area. **Click on the origin, drag horizontally** and release to produce the horizontal centerline. Click on the **check mark** to accept the centerline created followed by the **Esc key** on the keyboard .

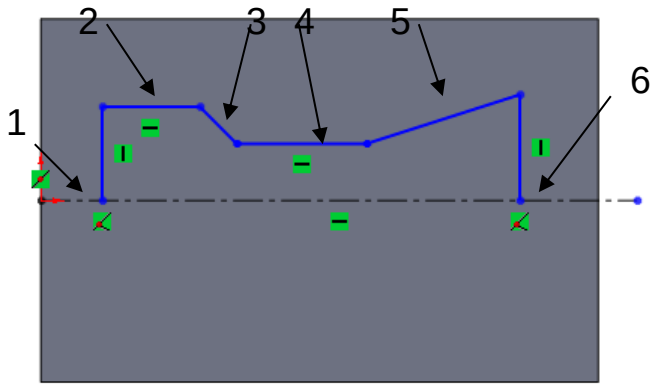


➲ **Sketch half of pocketed profile**

3. Select **line** from the sketch toolbar and sketch half of the profile

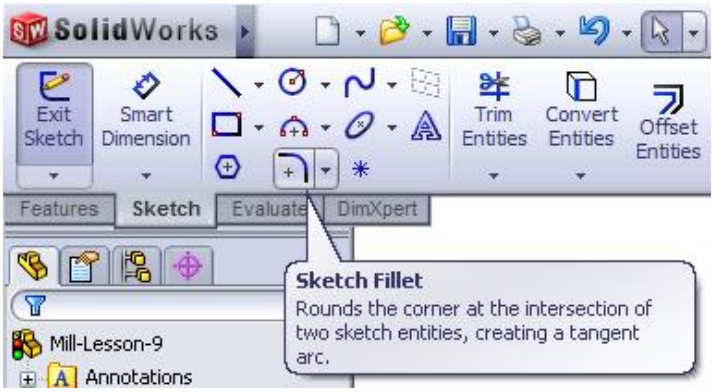
 (see below, do not worry about exact size or corner fillets for now.

4. Start on horizontal centerline (1) below . Follow with lines (2) thru (6) with the last point being on the horizontal centerline .

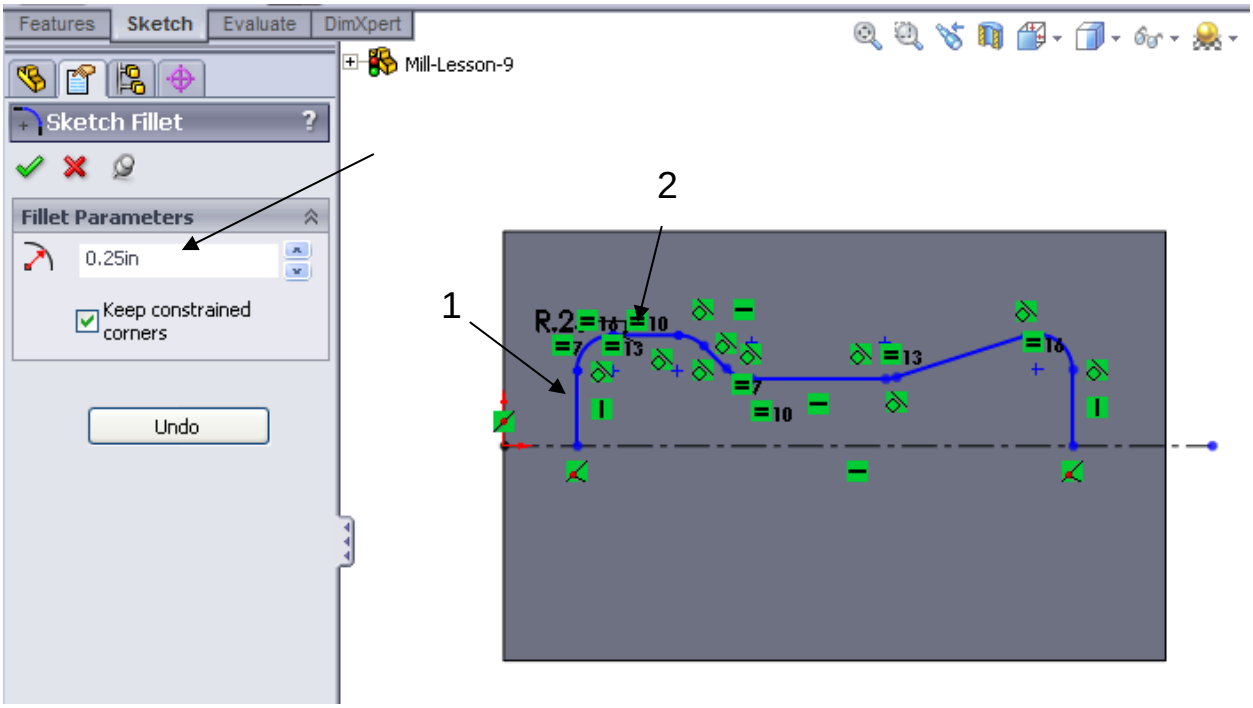


Note: the horizontal  and vertical  relations of the sketched lines.

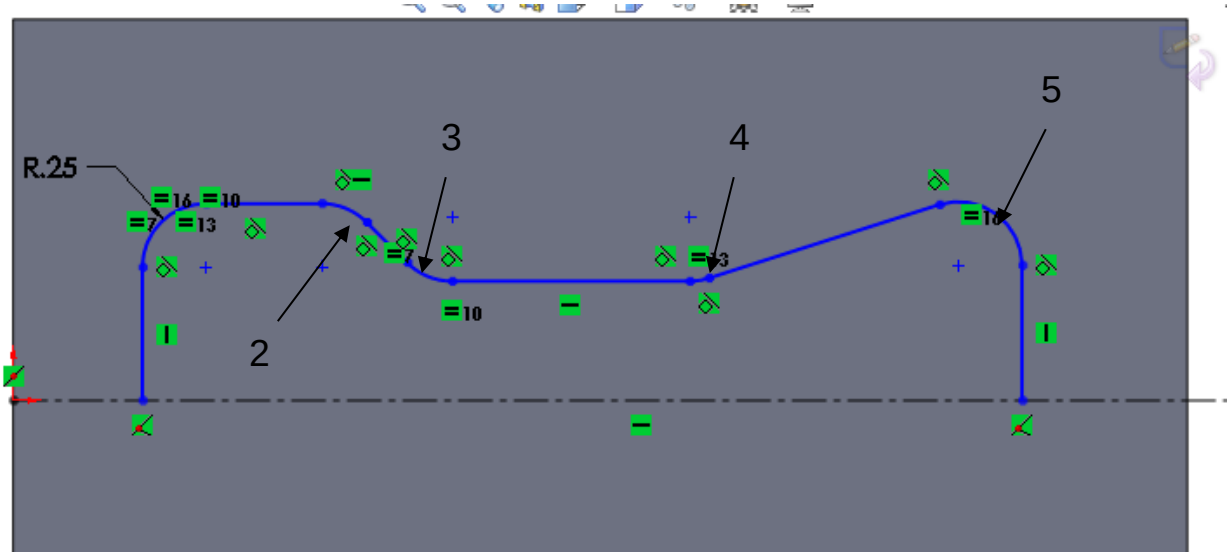
5. Add the **.25 inch fillets** to your sketch in each corner by the **sketch fillet icon**  on the sketch toolbar.



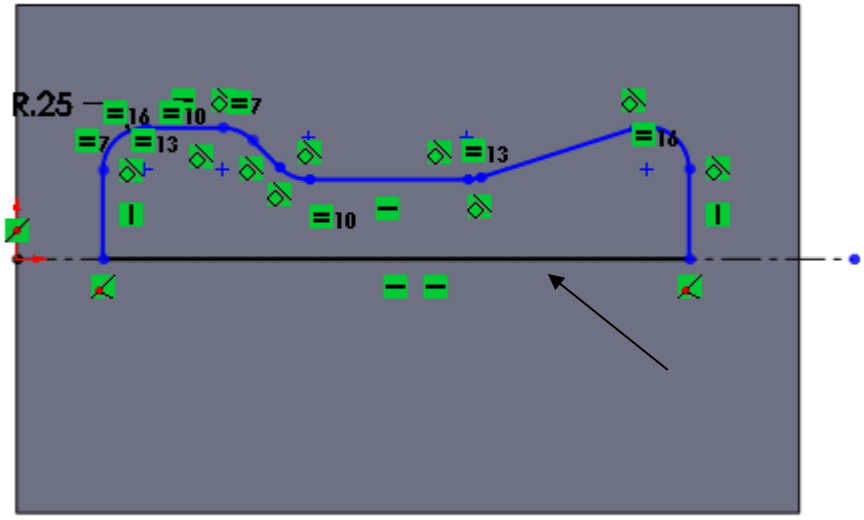
6. **Setting the radius to .25**. Click on a line (1) then the adjacent line (2); the fillet will appear. Continue working your way around each intersection adding the fillets (2), (3), (4), and (5). Click on the **check mark to accept when all corners have fillets**.



Enlarged view of fillets below.



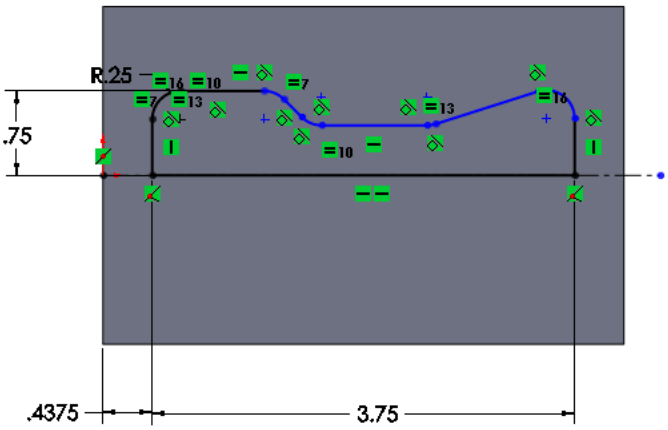
7. Close sketch with one final horizontal line.



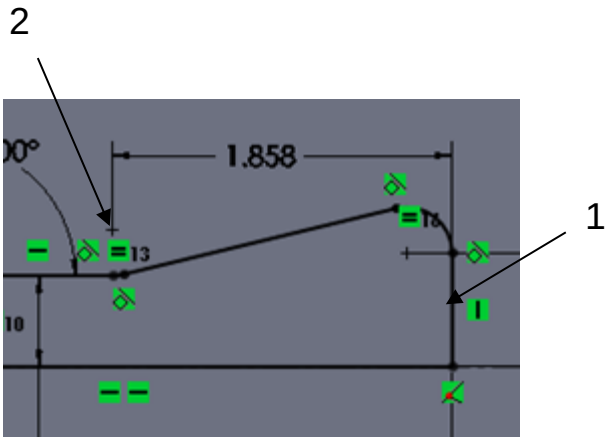
8. From the toolbar click on the **smart dimension icon** 

. Dimension sketch as shown below, **Refer to drawing of SolidWorks-lesson-9**.

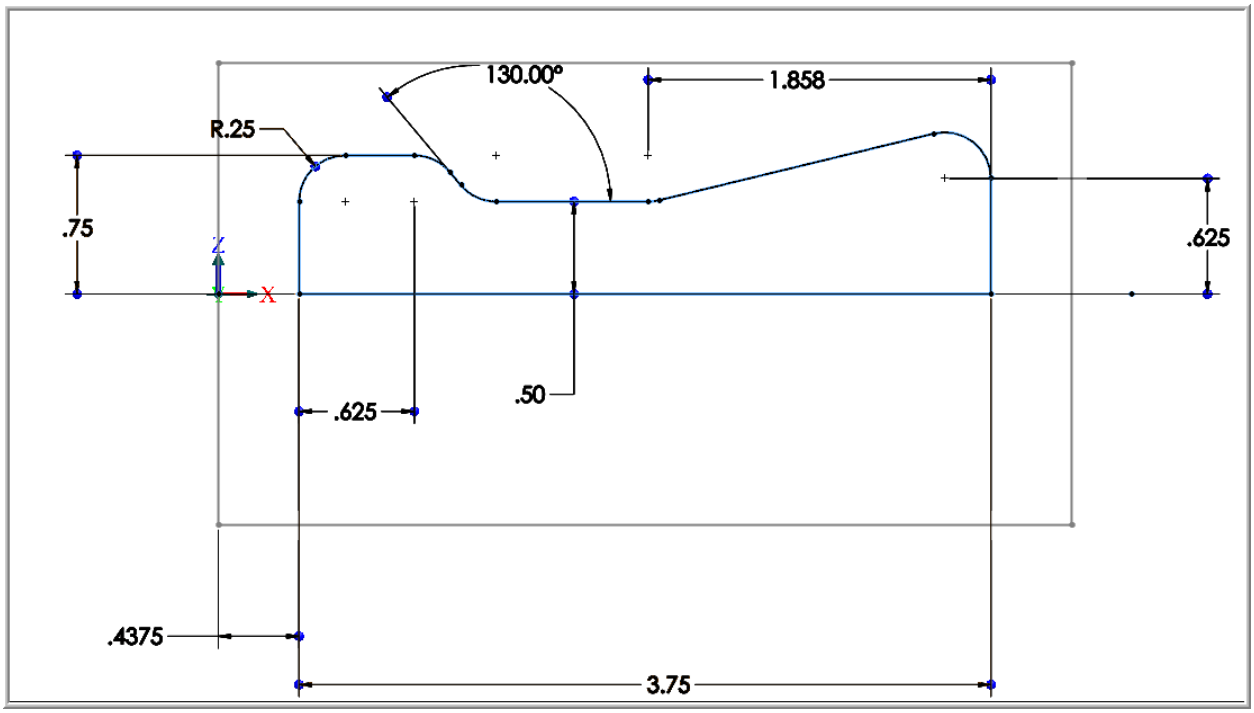
9. Start with the .**4375** from the edge of the block. Follow with the **3.75** dimension clicking on two vertical ends of sketched shape. Then add the .**75** dimension as shown below.



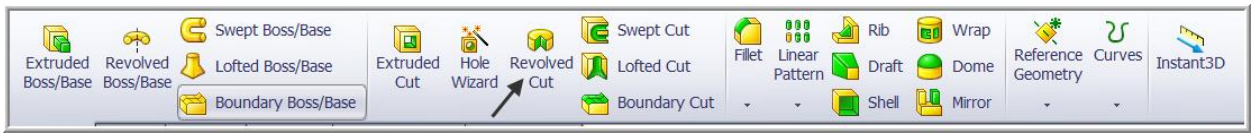
10. When dimensioning the .675 and the 1.858 dimensions pick the end of the shape **(1)** followed by the **center point of the fillet radius (2)**.



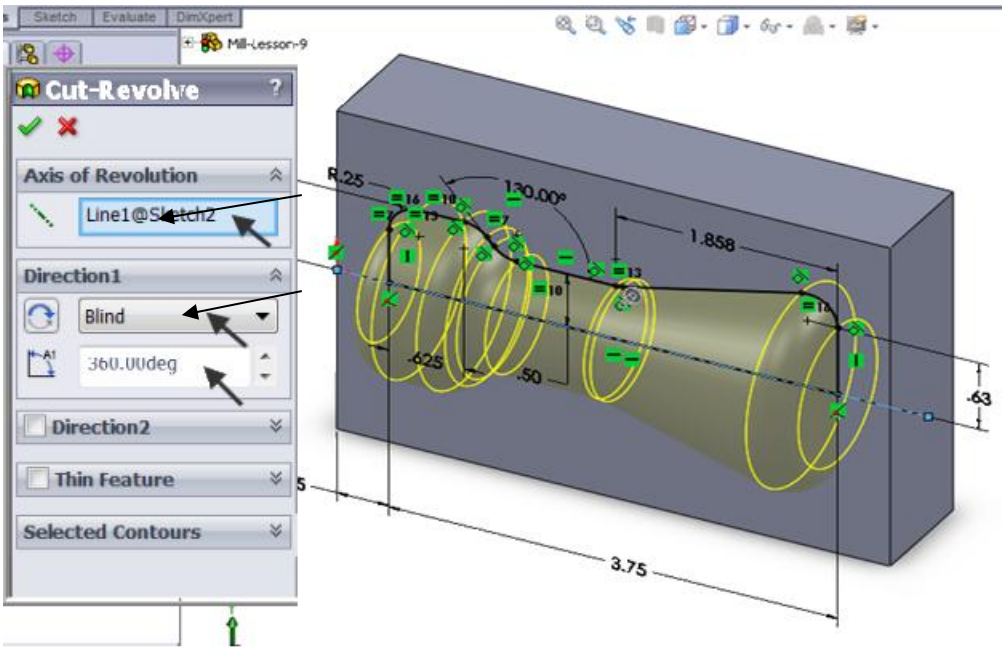
11. Finish the remaining dimensions as shown below. The part will turn black indicating fully defined. Press the **escape key** once fully dimensioned.

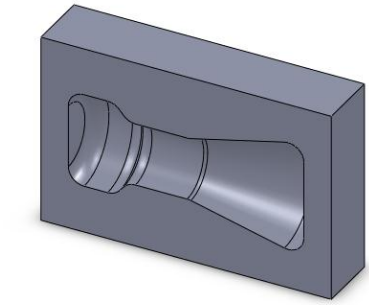


12. From the feature tab click on **Revolved Cut**. Press **space bar** and change part view to ***Trimetric**.

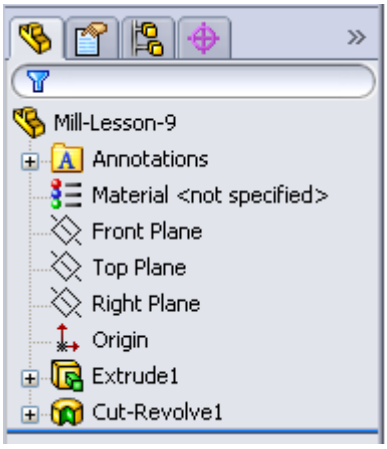


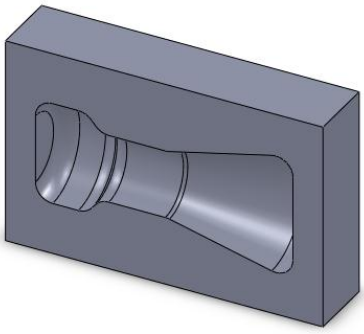
13. **Revolve 360 degrees around the horizontal centerline**.





Your part SolidWorks-lesson-9 is now complete. **Remember to save**.





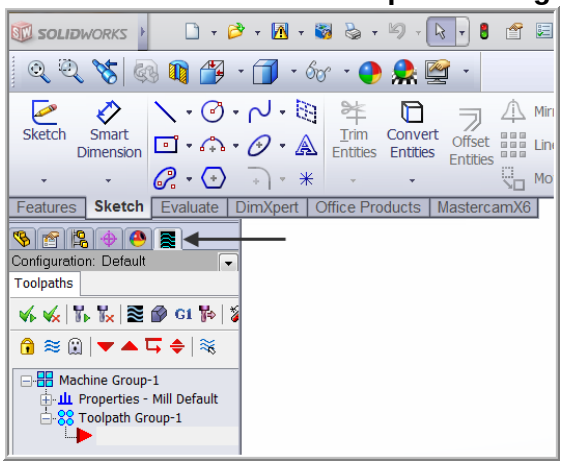
Note: The dimensions and relations established are to define the sketch and do not necessarily represent the placement of the dimensions on the finished drawing. This lesson was dimensioned this way to establish symmetry about the middle of the part (origin). The design intent is to allow the rotated cut to remain centered about the horizontal centerline.

Toolpath Creation

TOOLPATH TASK 1:
DEFINE THE ROUGH STOCK USING STOCK SETUP

➲ Defining the Rough Stock using stock setup.

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

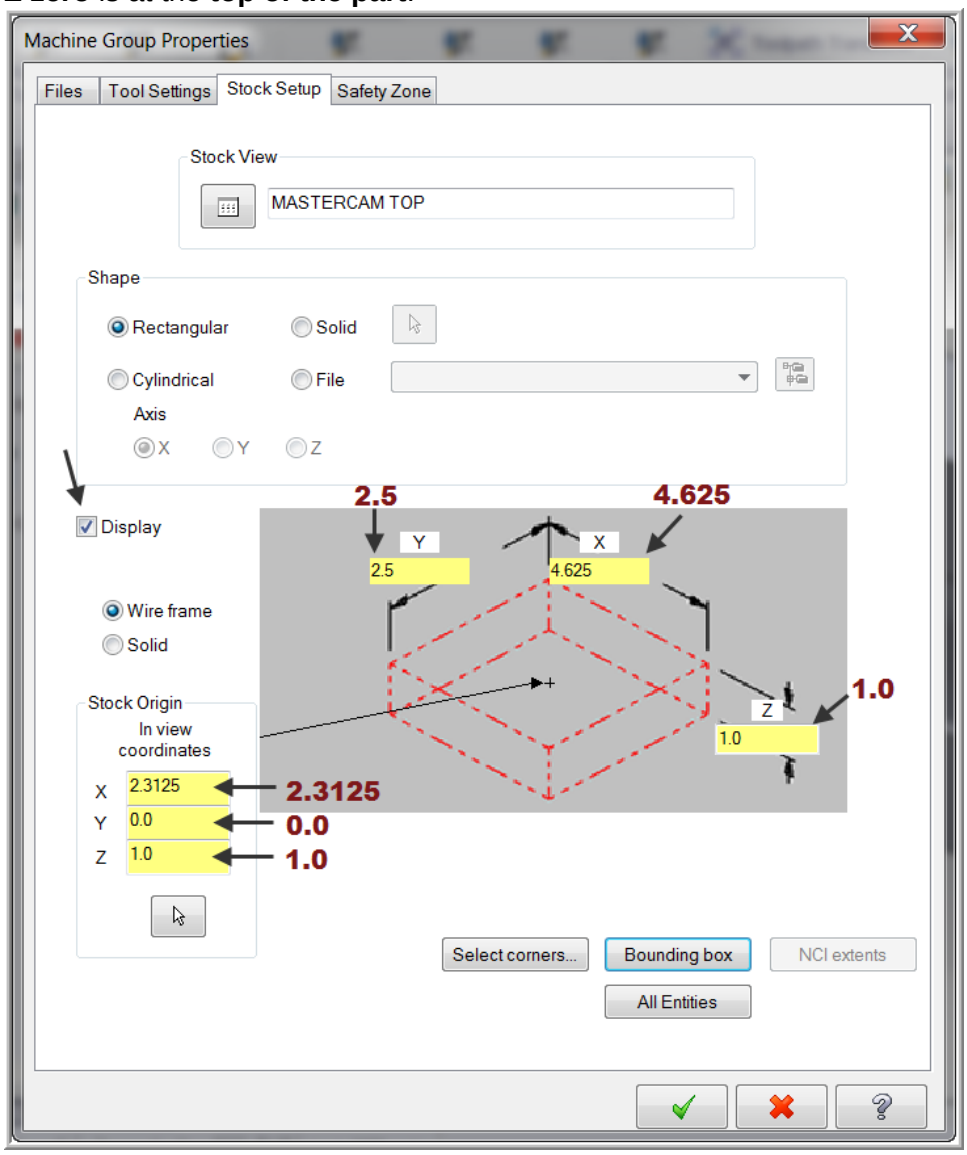


2. Select the plus in front of **Properties** to expand the **Toolpaths Group Properties.**

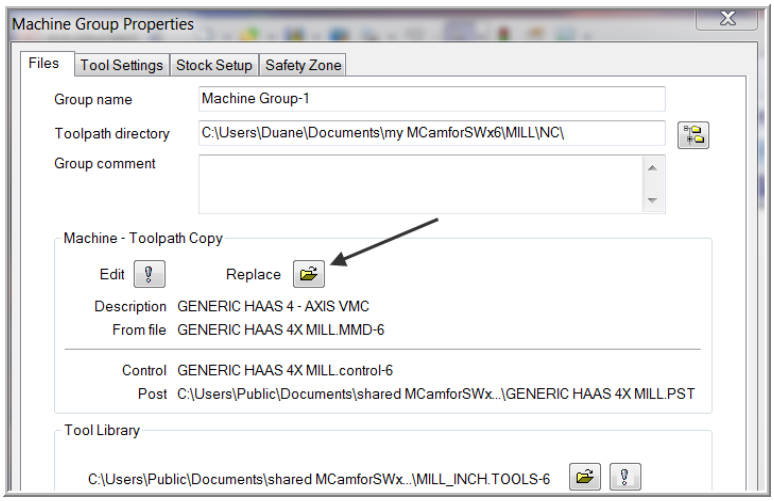
3. Select **Stock setup** in the toolpath manager window.

4. Use Bounding Box to establish the **Stock Setup.** The **coordinates** should be the same as the screenshot below.

➲ **Z zero** is at the **top of the part**.

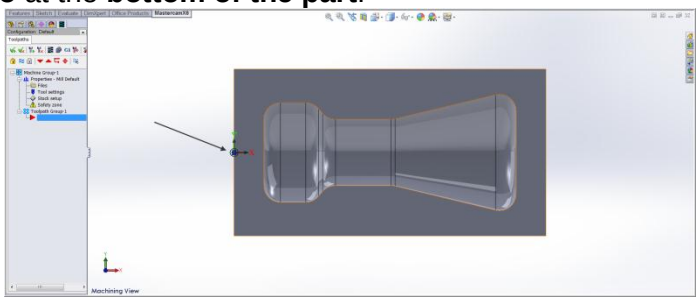


5. ***To select the machine that will be used to machine this part;*** Select the **Files** tab at the top of the **Machine Group Properties** window.
6. Click on the **Replace** icon to open the Machine folder. Select the proper machine from the list and click on **Ok**  . For this lesson we have chosen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



7. Position the part to **Front View** by pressing the space bar and double clicking on ***Front**.

Your part should look similar to the screen shot below. With **X0 Y0** at the middle left side and **Z zero** at the **bottom of the part**.



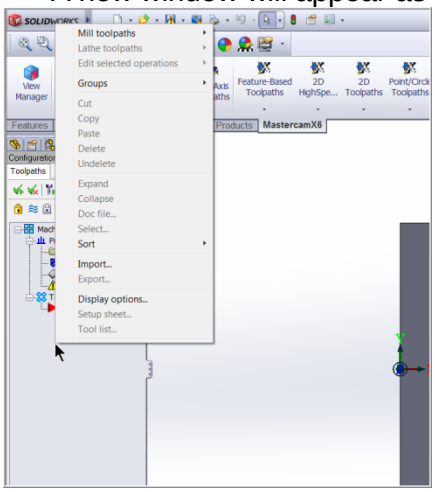
Surface roughing toolpaths typically use larger tools, multiple stepovers, and multiple step downs to quickly remove larger volumes of stock and leave an even amount of stock for finishing.

The roughing toolpaths you choose for your part depend on the shape of the part, shape of the stock, and machining situation. Mastercam provides several roughing strategies.

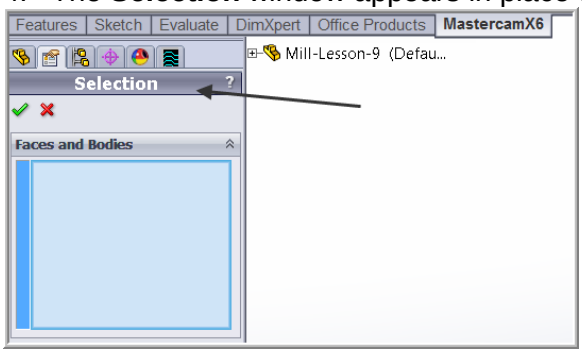
TOOLPATH TASK 2:
ROUGH THE POCKET USING SURFACE POCKET

➲ In this task you will use a 0.5 diameter bull endmill with a 0.125 corner radius to rough out the pocket.

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

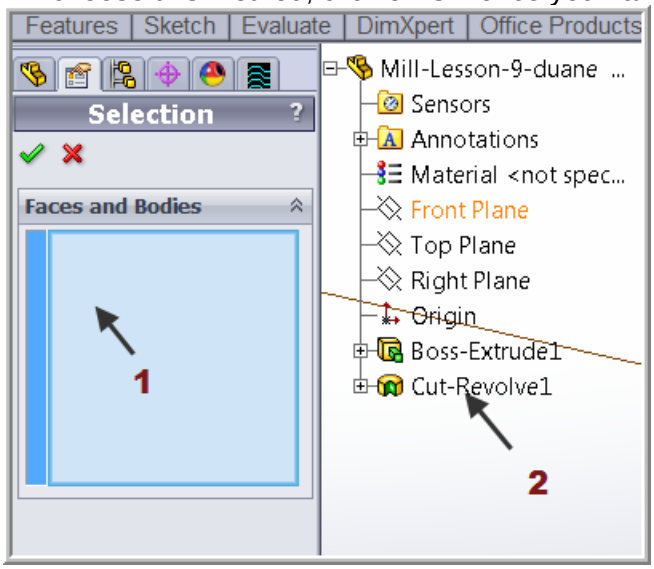


2. Click on **Mill toolpaths>Surface Rough>Pocket…**
3. When prompted to **Enter new NC name** ensure **Mill-LESSON-9** is entered and then select **OK**  .
4. The **Selection** window appears in place of the **Toolpath Manager** window:

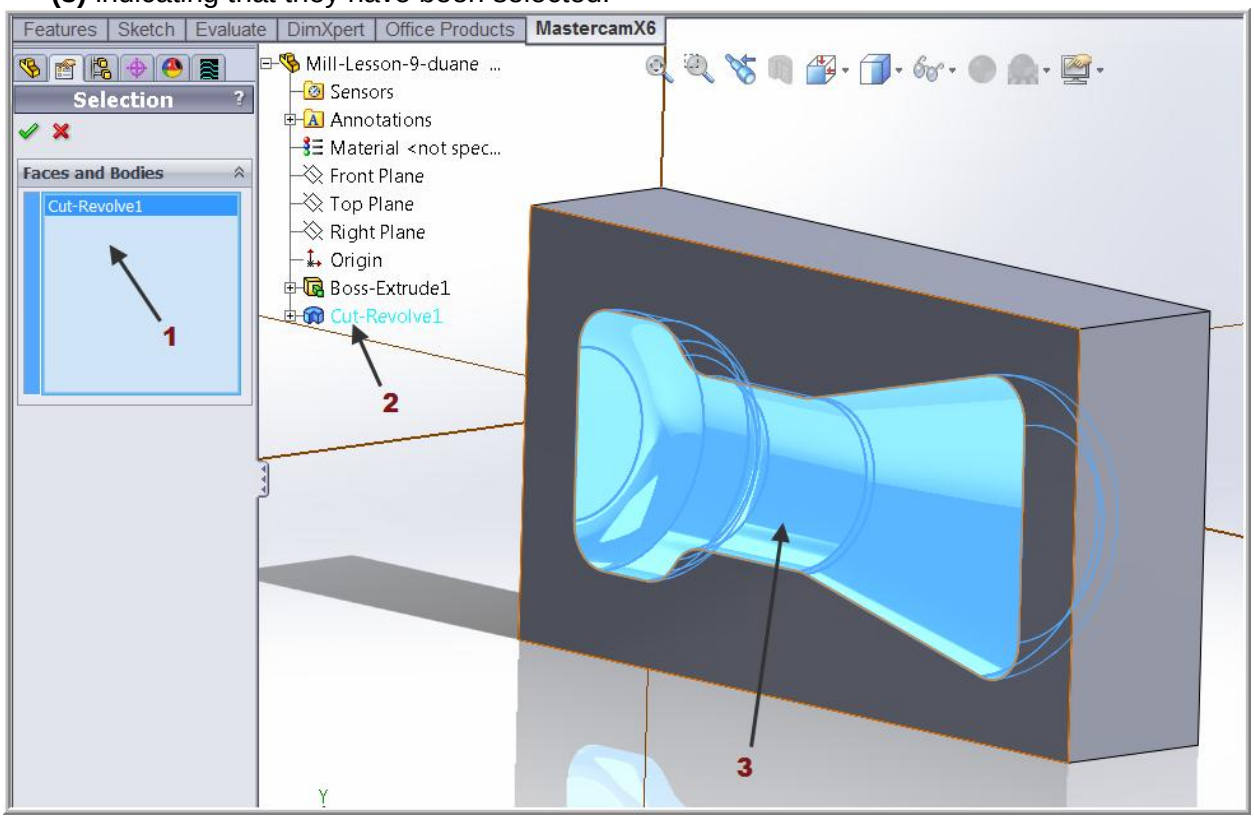


5. The **Faces** now need to be selected. You have a few option for selecting the Faces.

6. You can select the **Faces** by clicking on the **Cut-Revolve1 (2)** as shown below. If you choose this method, click on OK once you have selected Cut-Revolve1 and go to step 16.

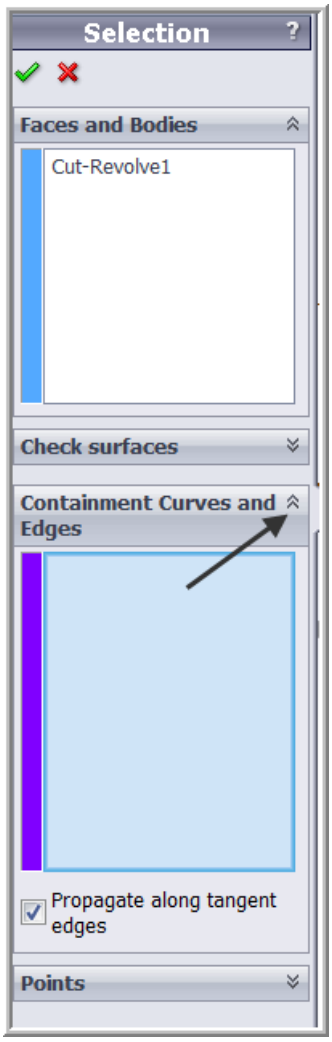


7. In the **Faces and Bodies** window **Cut-Revolve1** is listed **(1)**, the **Cut-Revolve1** item in the Mill-Lesson-9 tree has changed colors **(2)** and the **Faces** on the model have changed colors **(3)** indicating that they have been selected.



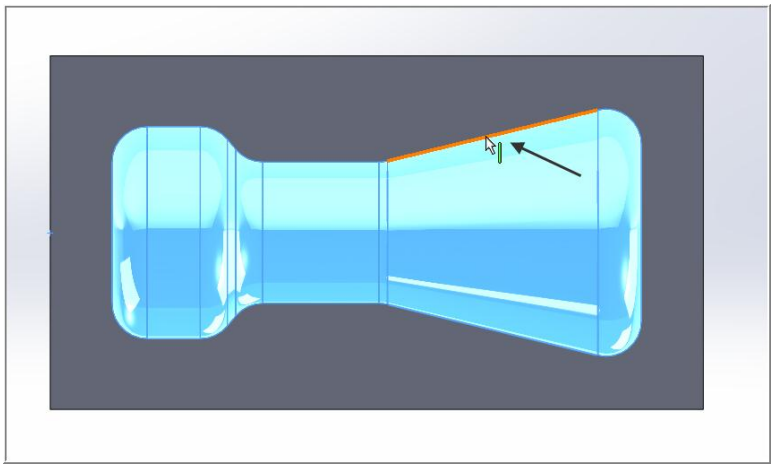
8. Click on **OK** to accept.

9. The **Selection Window** will reappear with a few more options available, one of them being **Containment Curves and Edges**. Click on the arrow to open this window.



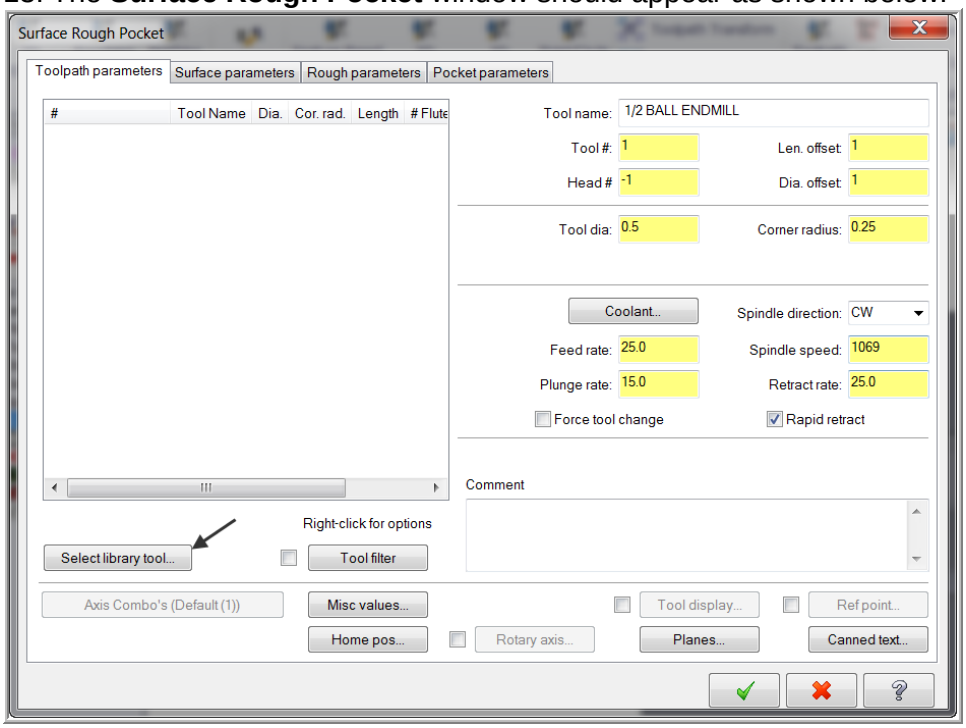
10. Click in the **Containment Curves and Edges window** and make sure **Propagate along tangent edges** is checked. (turned on).

11. To select the **Containment Curve(s)**, select one of the **Curves** as shown below.

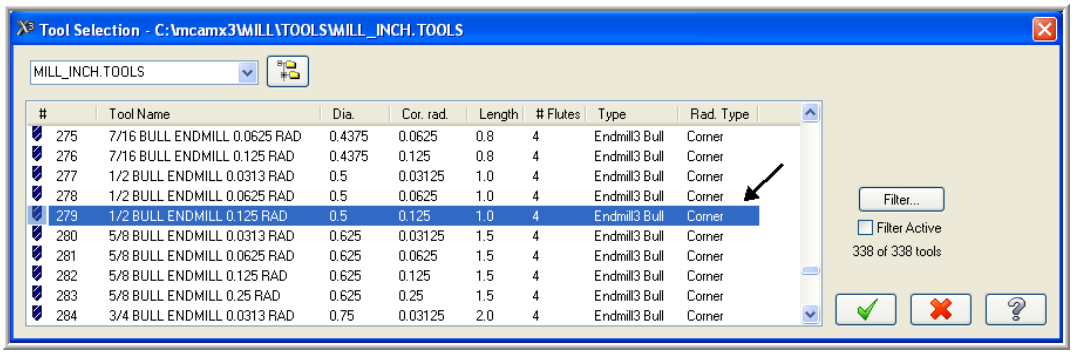


12. Select **OK**  in the **Selection** window.

13. The **Surface Rough Pocket** window should appear as shown below:

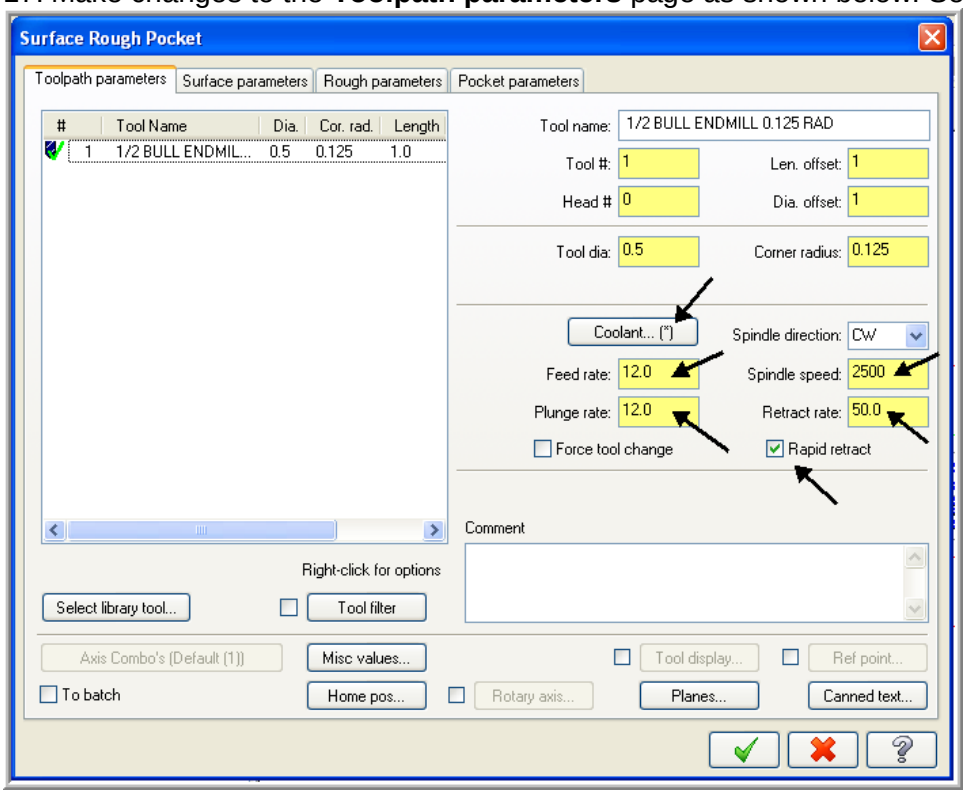


14. In the lower left corner of the **Toolpath parameters** page select the **Select library tool…** button. **Disable Filter active.**
15. Use the slider bar on the right of this dialog box to scroll down and locate a **0.5 diameter bull endmill with a 0.125 corner radius**. Select the endmill by picking anywhere along its row.

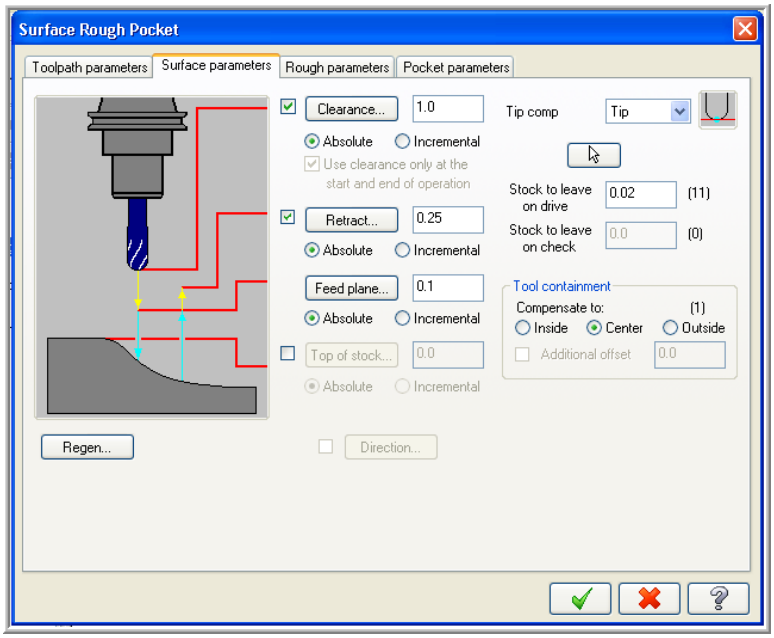


16. Select the OK button  to complete the selection of this tool.

17. Make changes to the **Toolpath parameters** page as shown below. Set coolant on.



18. Select the **Surface parameters** page and make changes to this page as shown below: **Stock to leave on drive** is set to 0.020.



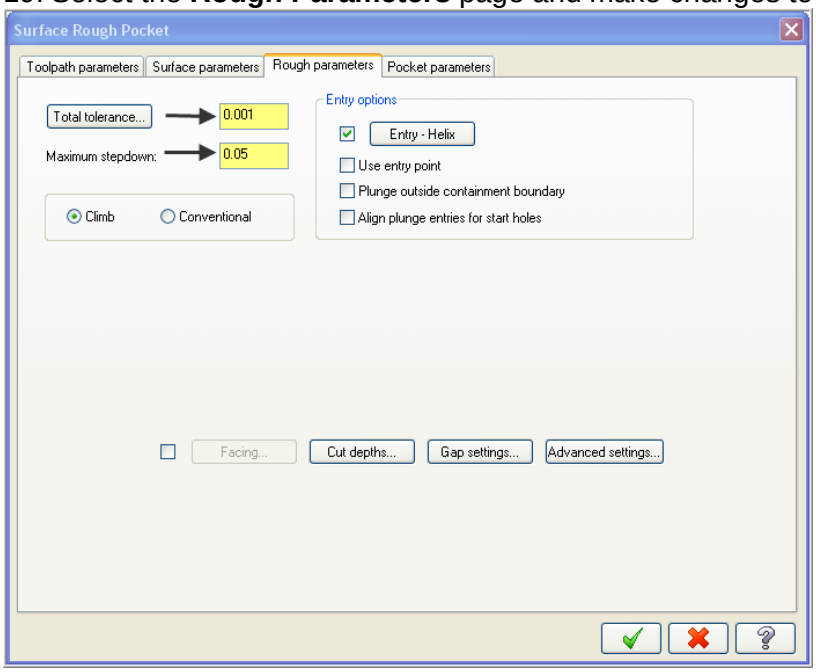
Tool Containment
Specify the behaviour of the containment boundary.

Compensate to Inside
Keep the tool inside this boundary.

Compensate to Center
Keep the tool centerline inside the boundary (allow up to half of the tool to exit the boundary).

Compensate to Outside
Allow the entire tool to exit the boundary but keep the tool edge in contact with the boundary.

19. Select the **Rough Parameters** page and make changes to this page as shown below:

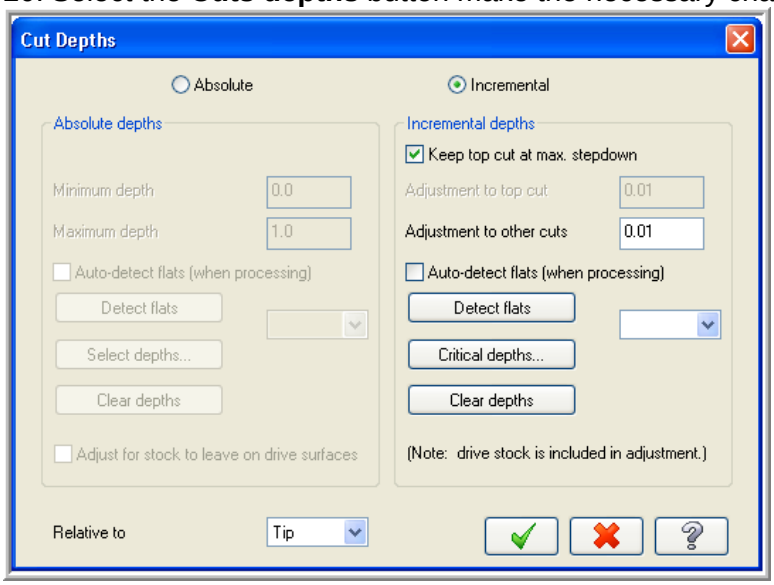


Tolerance selection will vary based on the toolpath type (roughing, semi-finish, or finishing), tolerances of final part, surface finish required, etc.

For the purposes of this tutorial, we will use .001 for both roughing and finishing to find a balance between accuracy and calculation time.

Typical tolerance ranges are from .0001 to .001.

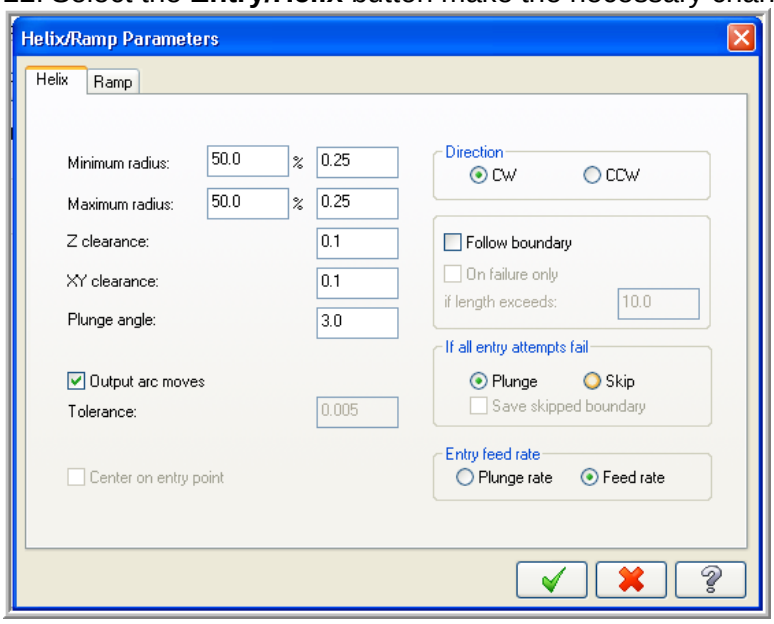
20. Select the **Cuts depths** button make the necessary changes.



Cut depths specify the placement of Z-axis cuts for all rough surface toolpaths and for finish contour toolpaths.
Incremental cut depths
Incremental cut depths are measured from the top and bottom of the part for most rough surface toolpaths and for finish contour toolpaths.
Keep top cut at max stepdown
Available only for rough pocket toolpaths. Forces the tool to make the first cutting pass at the max cut depth instead of the top of the part.

21. Select the **OK** button  to complete this feature.

22. Select the **Entry/Helix** button make the necessary changes.

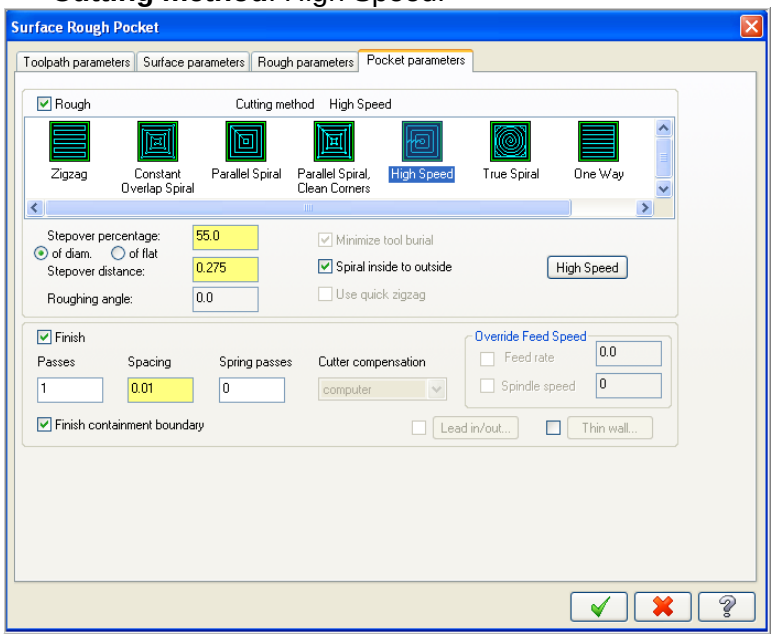


Helix/Ramp
Use this to add a ramp entry move to the pocket roughing operation.

You can add either a helix or a ramp entry, but not both. To switch between ramp or helical entry, just select the other tab and enter the desired entry dimensions.

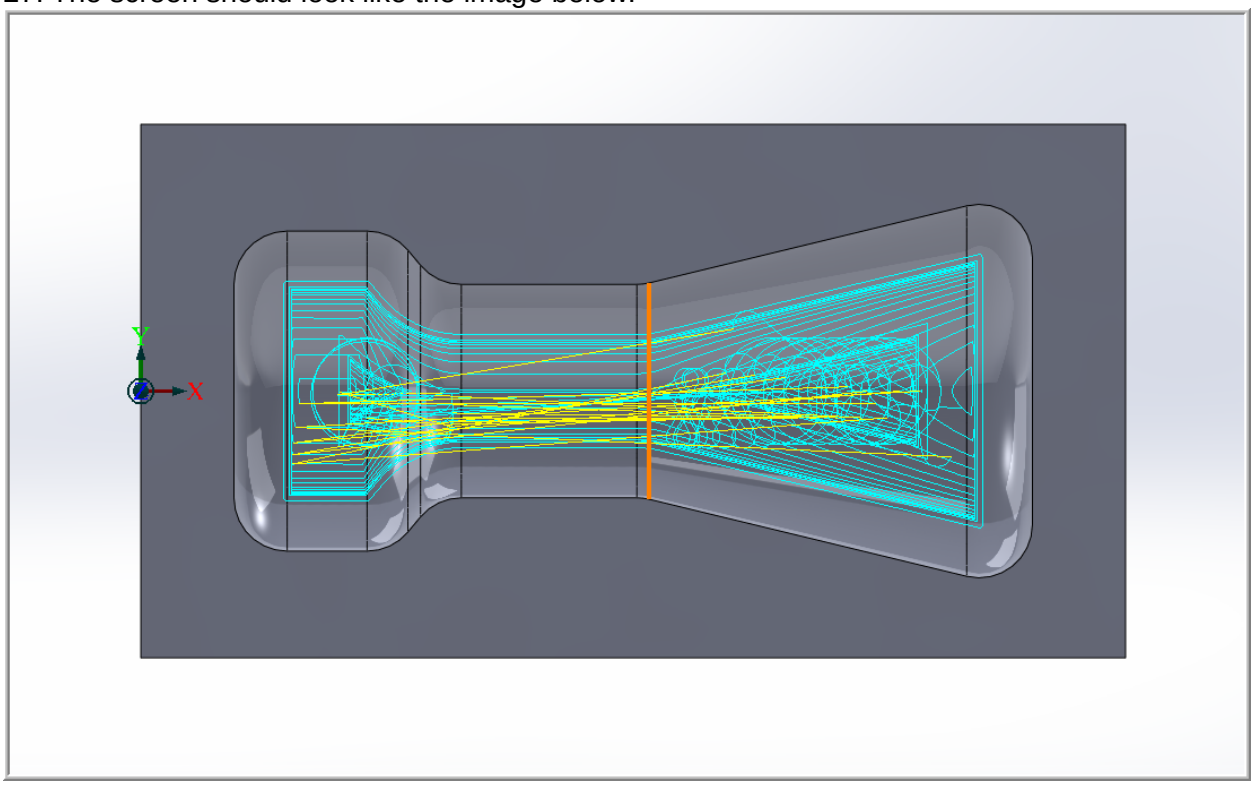
The helix/ramp options are off by default. If no helix/ramp options are set, Mastercam plunges the tool to the pocket depth at the start of the toolpath.

23. Select the OK button  to complete this feature.
24. Select the **Pocket parameters** page and make changes to this page as shown below.
    Cutting method: High Speed.



25. Select the **OK** button  to exit Pocket parameters.
26. It may take a while for Mastercam to create the toolpath.

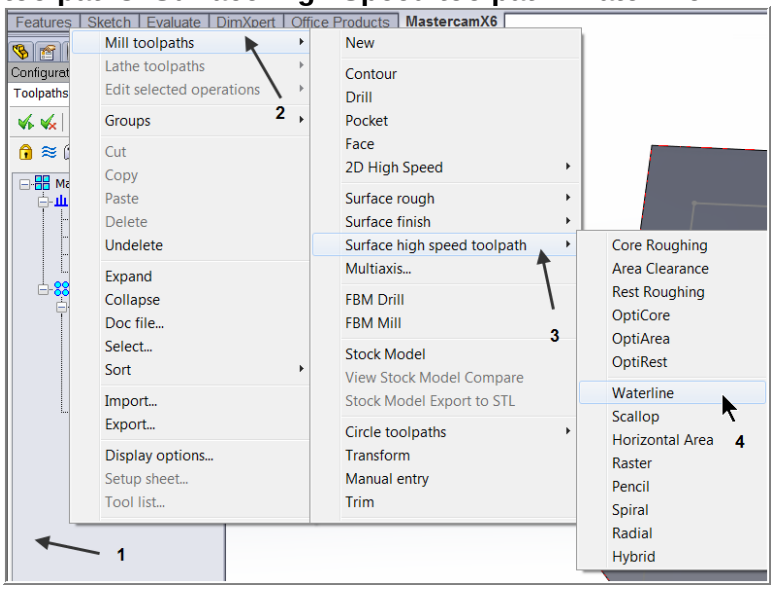
27. The screen should look like the image below:



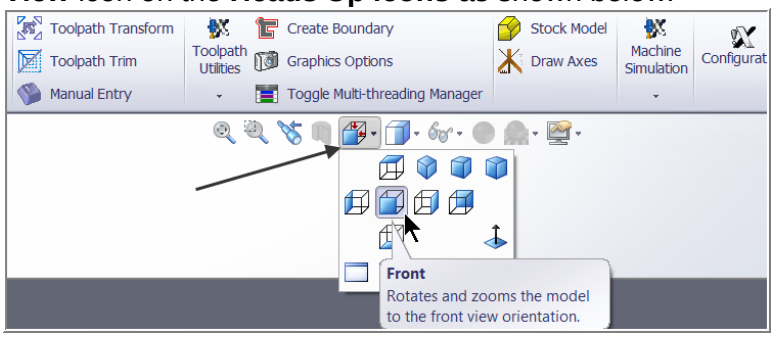
TOOLPATH TASK 3:
FINISH THE POCKET USING SURFACE HIGH SPEED WATERLINE

- In this task you will use a 0.5 diameter Ball endmill to finish the pocket.
- The rough parallel toolpath removes stock quickly using multiple constant Z depth. Surface High Speed Waterline also works at constant Z depth and steps down with cuts directly on the surface.

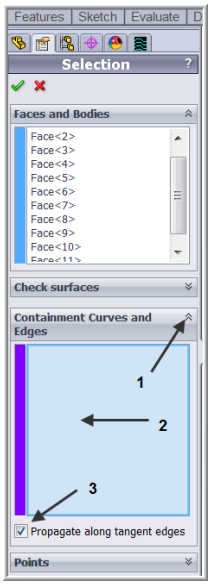
1. Right click in the **Mastercam Toopath Manager** window **(1)** and click on **Mill toolpaths>Surface High Speed toolpath>Waterline...**



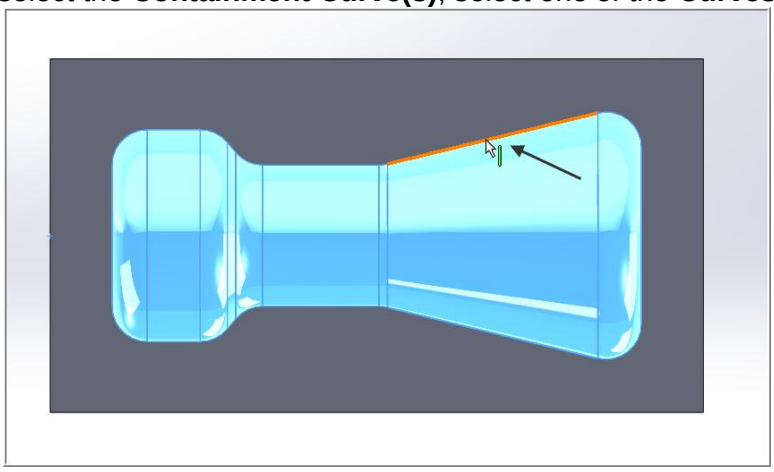
2. The **Selection** screen replaces the Mastercam Toolpath window. First change the view of the part to **Front** by either **right clicking** the mouse on the graphics screen or by using the **View** icon on the **Heads Up icons** as shown below.



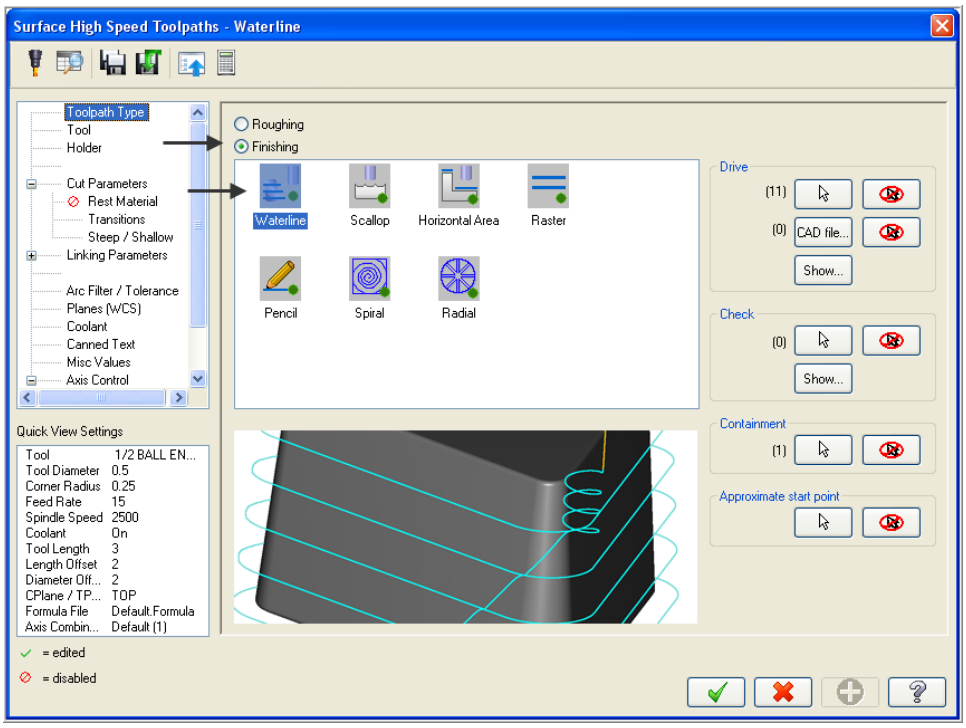
3. Use the same method that was used before to select the faces of the part to be machined. Be sure to select the 2 **inside end faces** that are not visible.
4. There should be a total of **11 faces** selected.
5. Click on **OK** in the **Selection** window.
6. Click on the top of the **Containment Curves and Edges** window to expand it **(1)**.
7. Select the **Propogate along tangent edges** box **(3)**.
8. Click in the newly expanded window **(2)**.



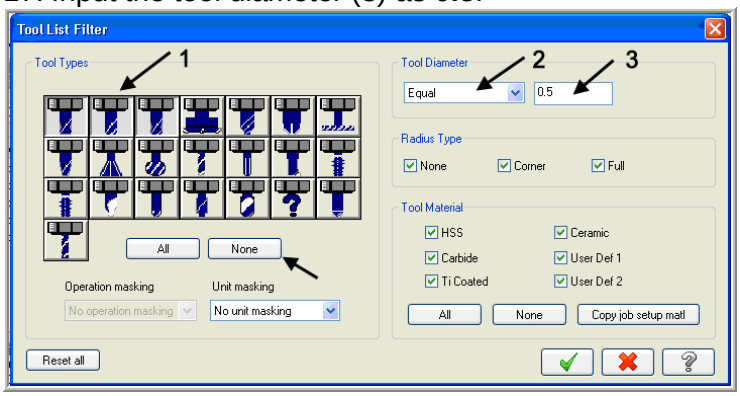
9. To select the **Containment Curve(s)**, select one of the **Curves** as shown below:



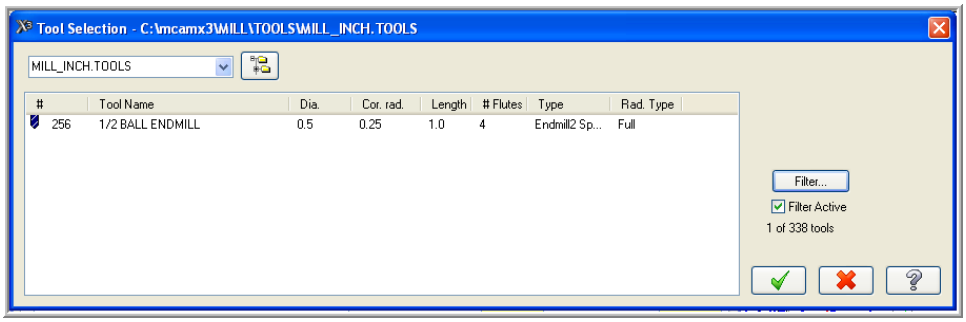
10. Select OK  in the **Selection** window.
11. For the **Toolpath Type**, select the **Finishing** radio button, and the **Waterline** toolpath.



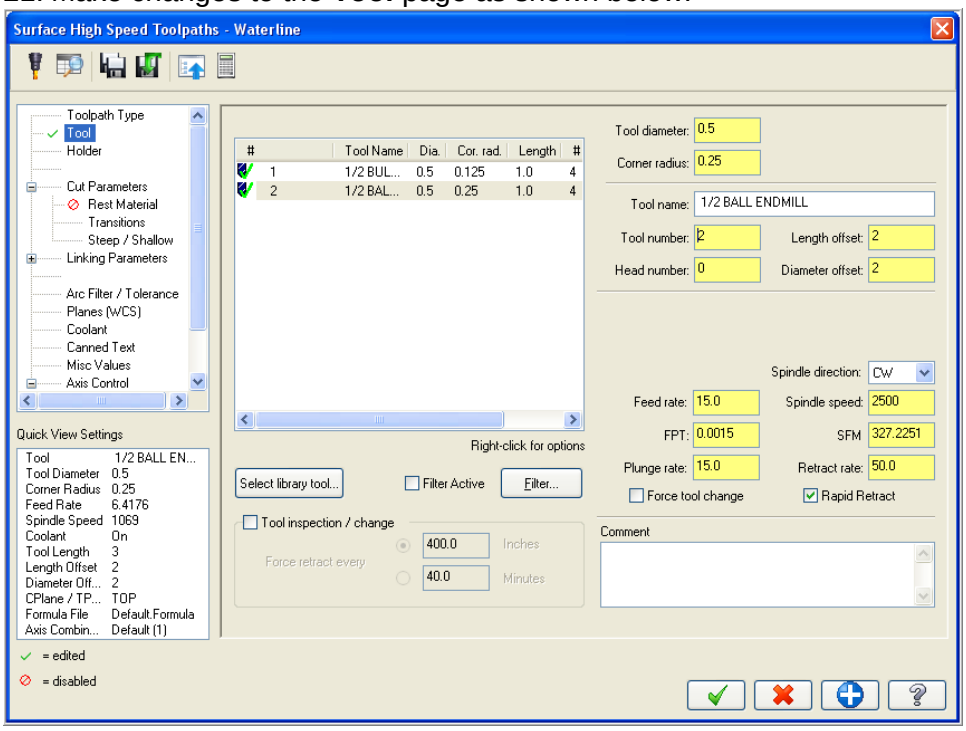
12. Select the **Tool** selection page, then **Select library tool…** button.
13. Select the **Filter** button on the right side of the **Tool selection** dialog box.
14. Select the **None** button in the Tool Types section.
15. Click on the **Endmill2 Sphere** type icon as shown in the picture (1) below:
16. Select the drop down arrow in the **Tool diameter** (2) field and set it to **Equal**.
17. Input the tool diameter (3) as **0.5**.



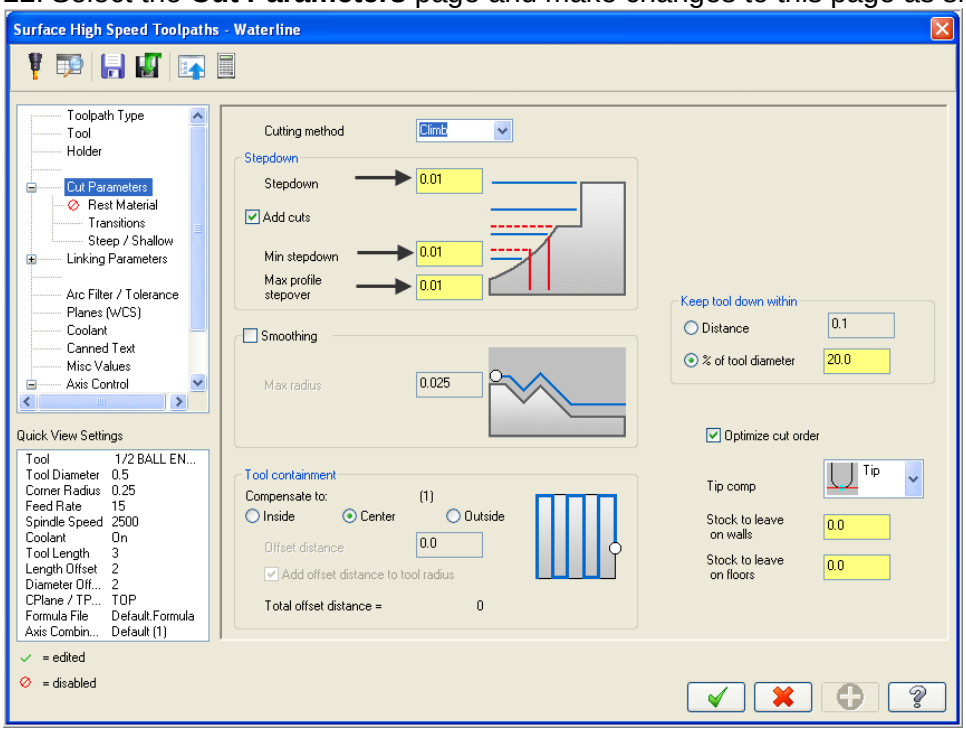
18. Select the **OK** button  to exit.
19. Select the **0.5 ball endmill**.



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



22. Select the **Cut Parameters** page and make changes to this page as shown below:

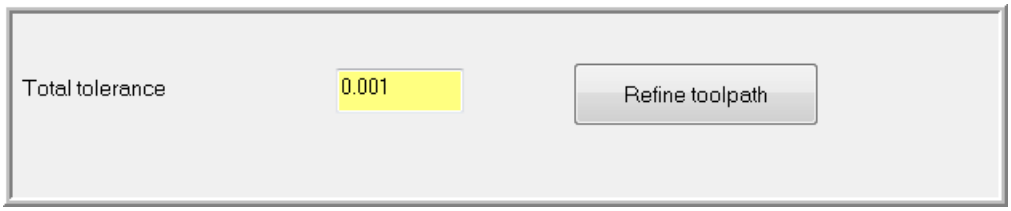


The Add cuts selection allows the user to add cuts in shallow areas to decrease the size of surface scallops.

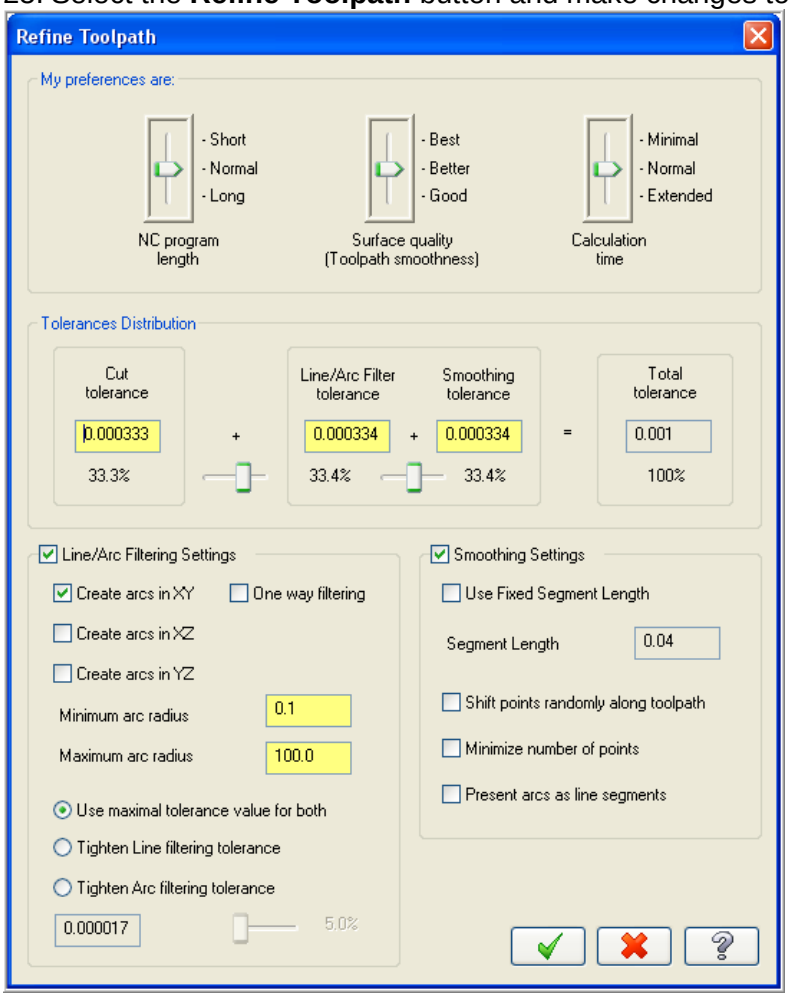
The programmer should consider the tool corner radius, surface shape, and operation type (rough, semi-finish, finish) when making selections here.

23. Select the **Arc Filter / Tolerance** page,

24. Set the **Total** tolerances.

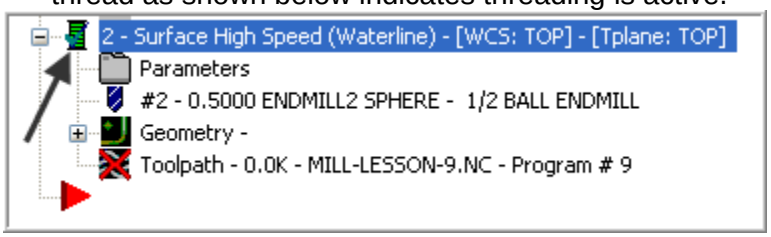


25. Select the **Refine Toolpath** button and make changes to this page as shown below:

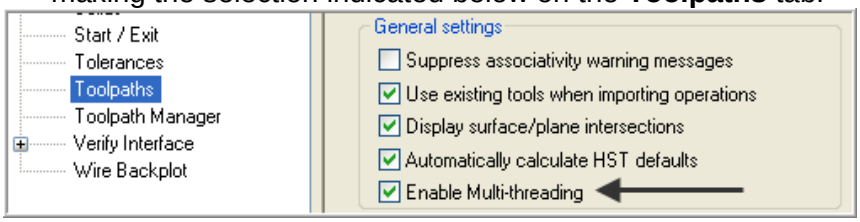


Refine Toolpath
Refine toolpath allows the user to make selections for toolpath calculation tolerance, arc filtering, and smoothness. The beginner programmer can simply adjust the sliders in the preferences section at the top. Advanced users can make selections manually in the windows below.
Tolerances Distribution allows the programmer to manipulate the extent to which the Filtering and Smoothing settings are applied in the toolpath calculation.
Line/Arc Filtering Settings
Settings for creating arcs and lines from sections of linear moves.
Smoothing Settings will process the code in varied ways in order to present it to the CNC controller in the preferred way.

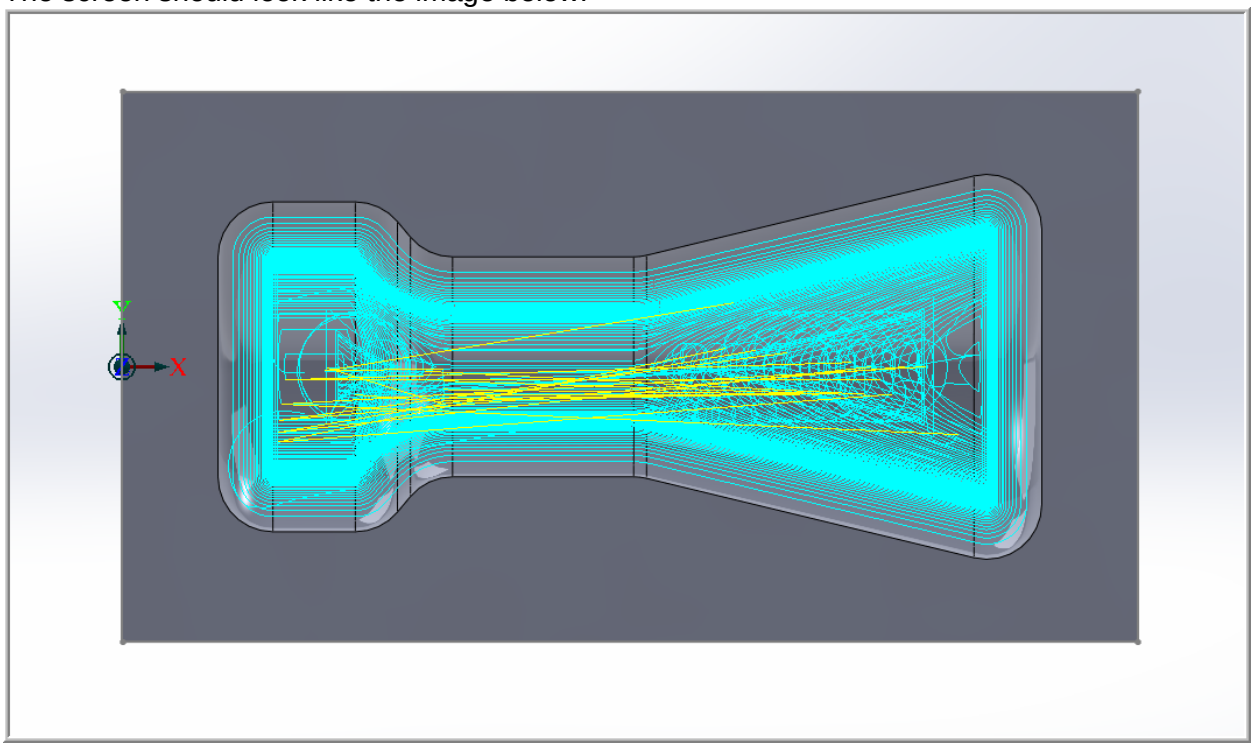
26. Select the **OK** button  to exit **Refine Toolpath**.
27. Select the **OK** button  to exit **Surface High Speed Toolpaths**.
28. It may take a while for **Mastercam** to create the toolpath. Mastercam's multi-threading functionality will calculate the toolpath while you continue to work. The green spool of thread as shown below indicates threading is active.



29. If threading is not active, you can turn it on by selecting **Settings>Configuration...** then making the selection indicated below on the **Toolpaths** tab.

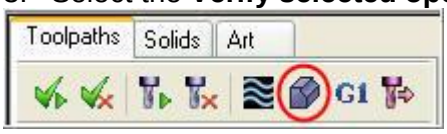


The screen should look like the image below:

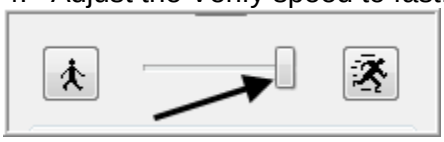


TOOLPATH TASK 4:
VERIFY THE TOOLPATH

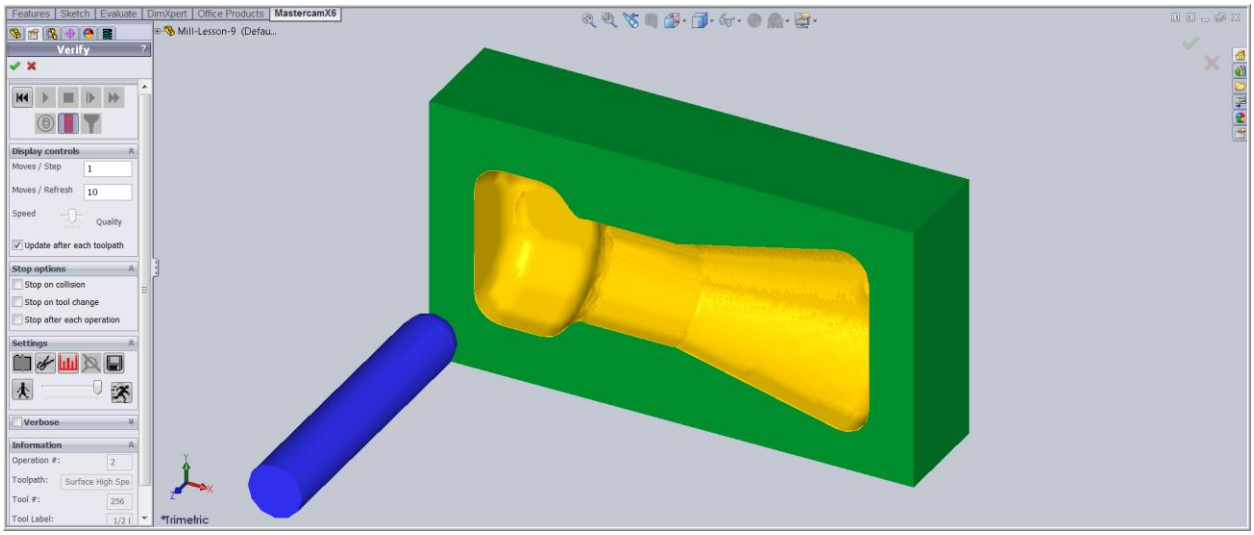
1. Change the view of the part to ***Trimetric**.
2. In the Toolpath Manager pick all the operations to verify by picking the **Select All** icon 
3. Select the **Verify selected operations** button circled below:



4. Adjust the Verify speed to fast.



5. Select the **Play** button to verify the toolpath. If it is taking too long to verify the toolpath click on the **Stop** button and go to step 5.
6. Select the **fast forward** button to speed up the verification of the toolpaths.



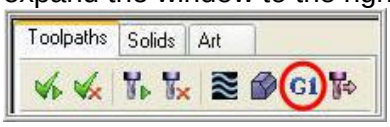
7. Select the OK button  to exit **Verify**.
8. Save the file.

TOOLPATH TASK 5:
POST AND CREATE THE CNC CODE FILE

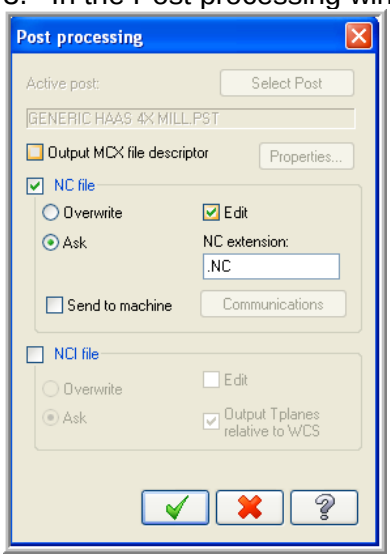
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.

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

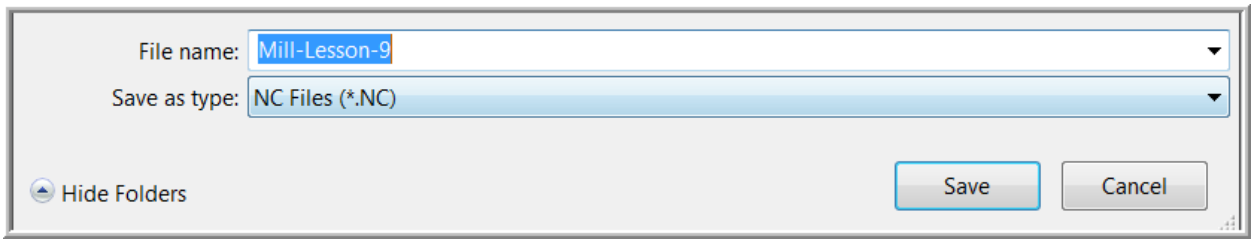
Please Note: If you cannot see **G1** click on the right pane of the Toolpath manager window and expand the window to the right.



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



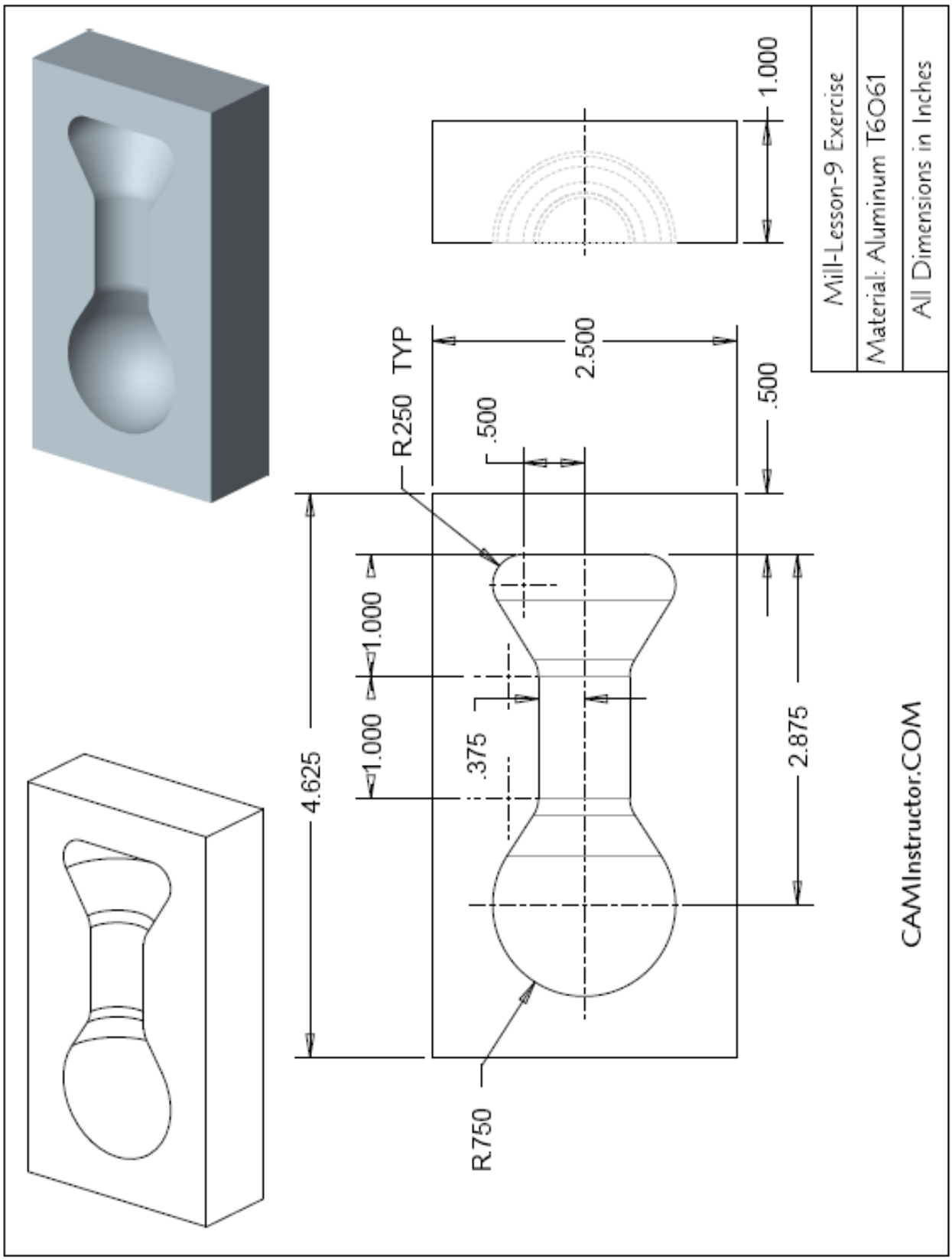
4. Select the OK button  to continue.
5. Enter the same name as your Mastercam part file name in the NC File name field **Mill-Lesson-9**.

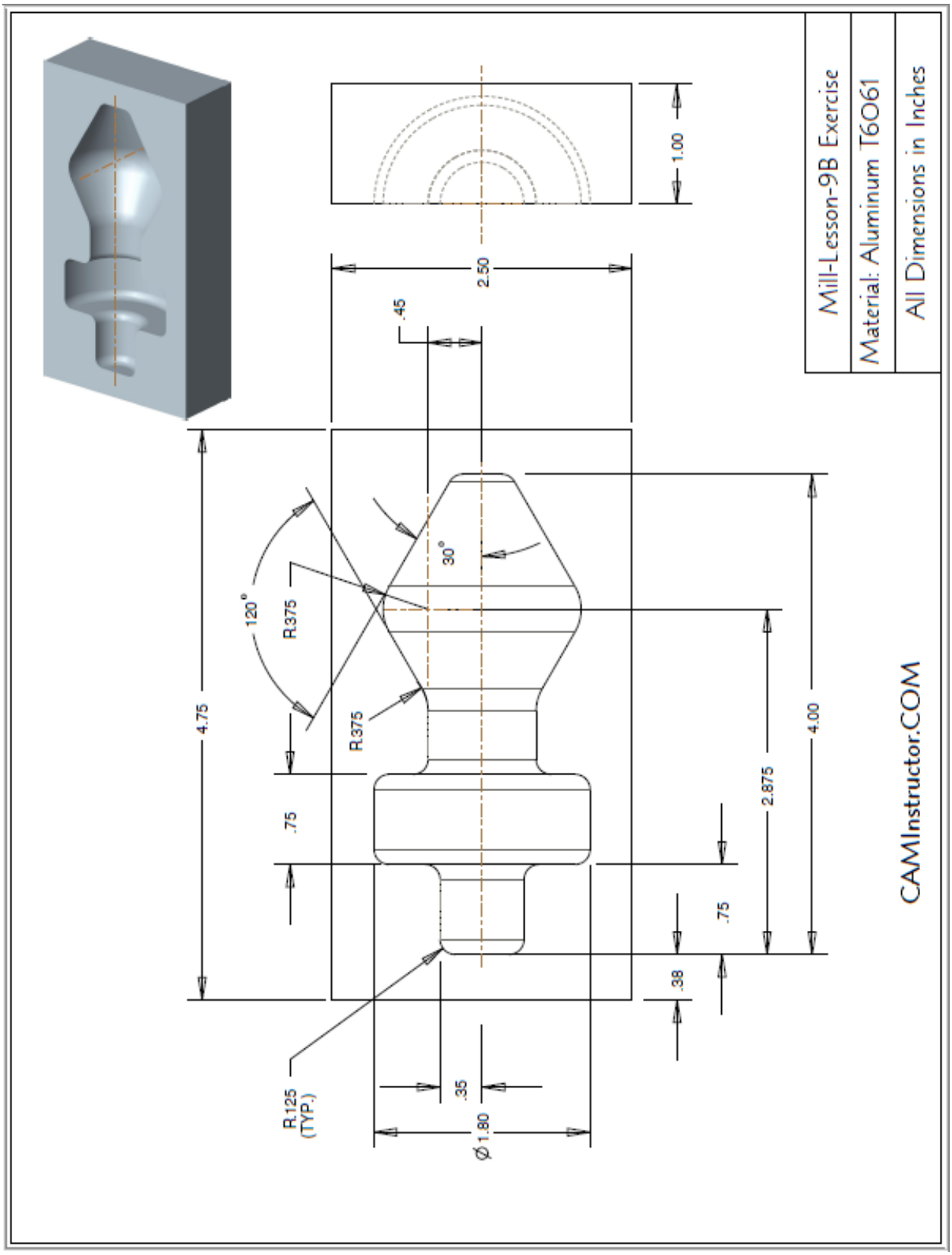


6. Select the **Save** button.
9. The CNC code file opens up in the default editor.
10. Select the  in the top right corner to exit the CNC editor.

This completes Mill-Lesson-9.

MILL-LESSON-9 EXERCISE




Mill-Lesson-9B Exercise
Material: Aluminum T6061
All Dimensions in Inches
CAMInstructor.COM
1.00
2.50
.45
4.75
120°
R.375
30°
R.375
.75
2.875
4.00
.75
.38
R.125
(TYP.)
.35
Ø 1.80