

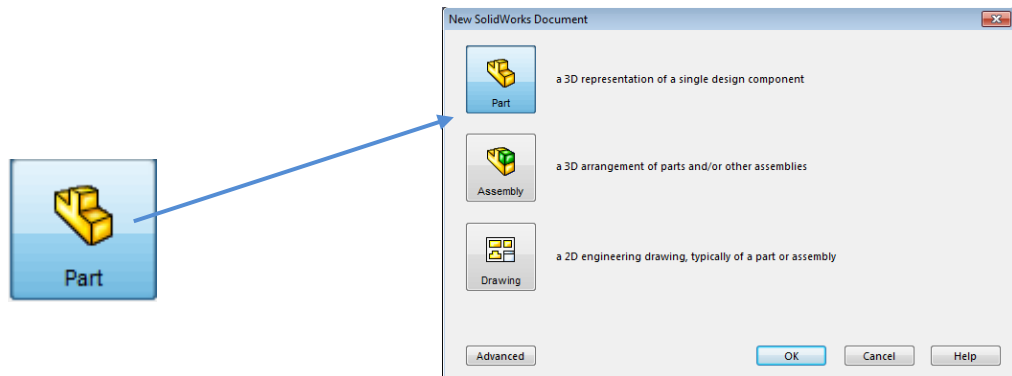
Ladybird Project - Vacuum Mould



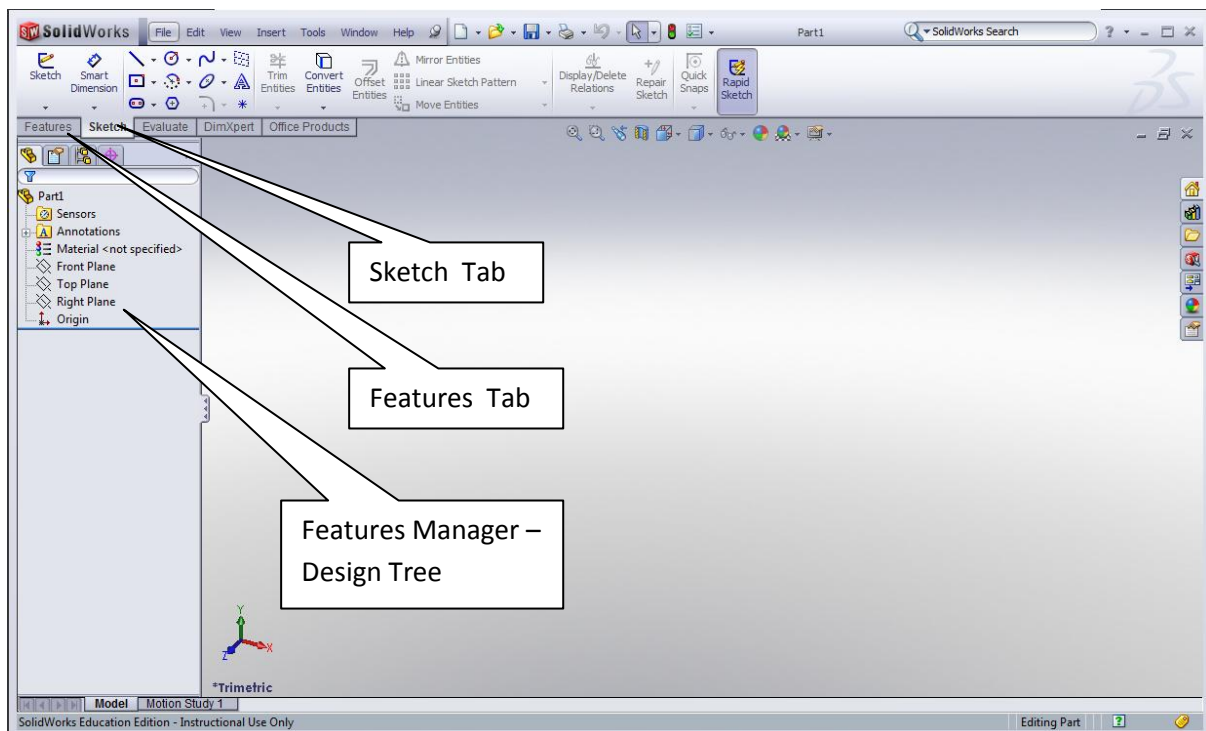
Prerequisite knowledge	Navigating the interface, file management, opening and saving of files & basic sketching.
Focus of the Lesson	<p>On completion of this exercise you will have used:</p> <ul style="list-style-type: none">• Sketch• 3 Point Arc• Relationship• Smart Dimension• Extrude• Apply Appearance• Saving in "STL" file format.
Commands Used	<i>Line, 3 Point Arc, Smart Dimension, Extrude, Edit Sketch, Save As.</i>

Creating the Mould in SolidWorks.

Open Solid works. Select **File, New...** Choose **Part** from the dialog box displayed.



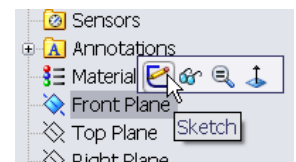
The two tabs used in modelling this part are the **Sketch Tab** and the **Features Tab**.



The **Features Manager design tree** on the left of the screen supplies information on the status of the part and allows access to features and sketches for editing.

Select the **Front Plane** and choose **Sketch** from the pop-up menu.

The front plane will rotate so that your view is perpendicular to it.



With the Sketch Tab active choose the Line option.

Sketching on the plane is now similar to drawing on a sheet of paper.

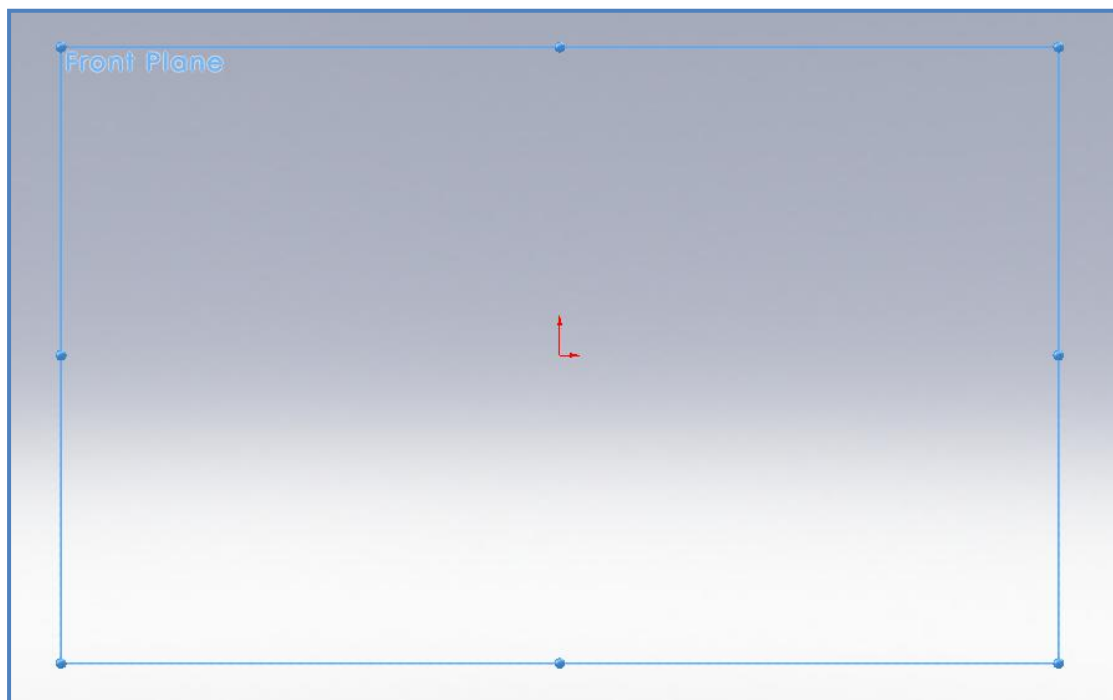



Figure 1

Creating the Sketch

- To begin sketching, left click on the Front Plane to the right of the **origin** as shown (figure 2 - Point1).
- Click on the origin (as shown in figure 2 – point 2) to indicate the line end point.
- Continue to sketch a line coincident with the origin and horizontally across as shown in figure 3 (Length, angle, etc is not important at this stage).

Note: Ensure you capture the automatic horizontal relation identified by 

- To exit the line tool press the “**Esc**” key.

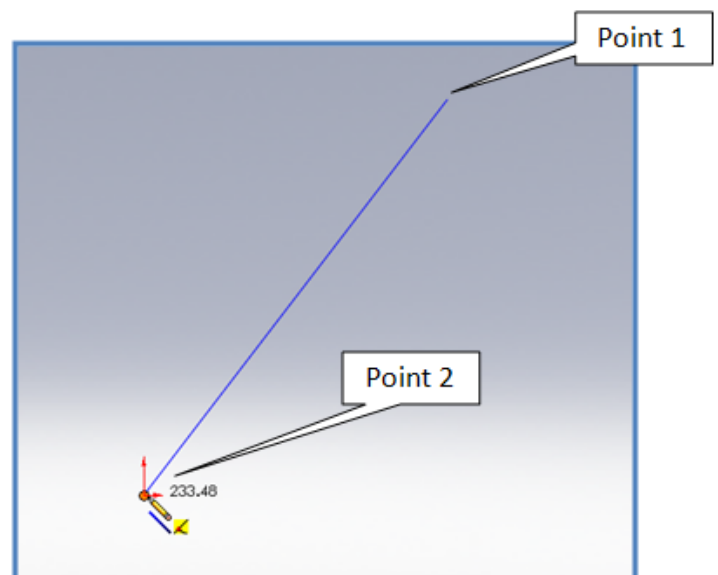


Figure 2

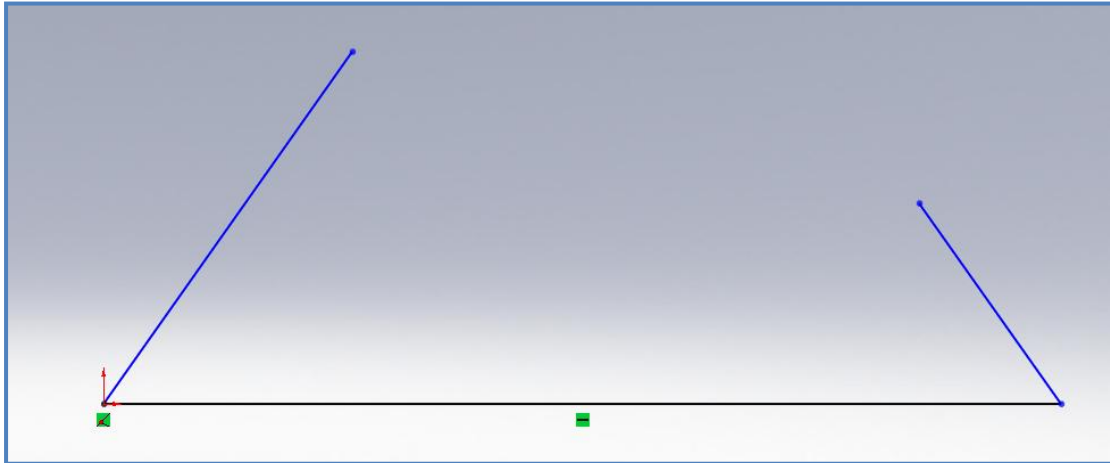


Figure 3

Sketching a 3 Point Arc.

- Select the **3 point Arc** command from the Sketch Tab as shown (figure 4).
- Click on the end point of the 2 lines as shown in figure 5 (Points 1 & 2).

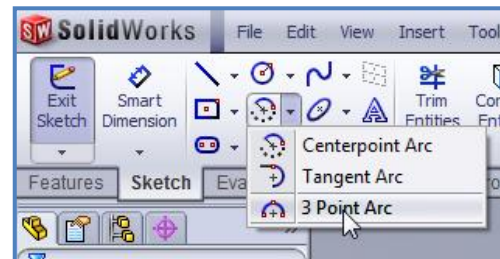


Figure 4

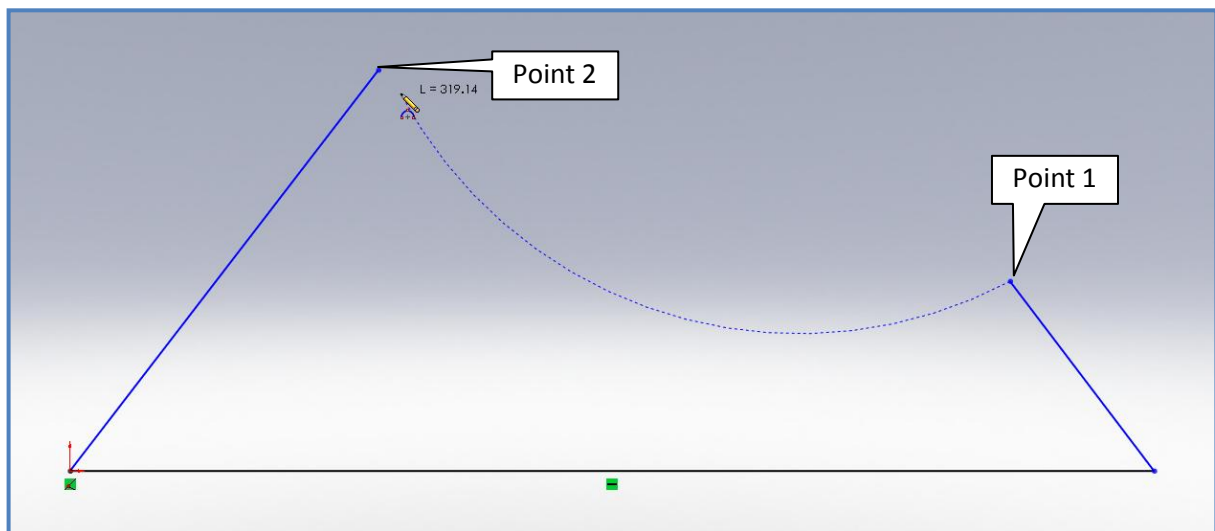


Figure 5

- Choose point 3 in the approximate position indicated in figure 6.
- Press the **Esc** key to exit the command.

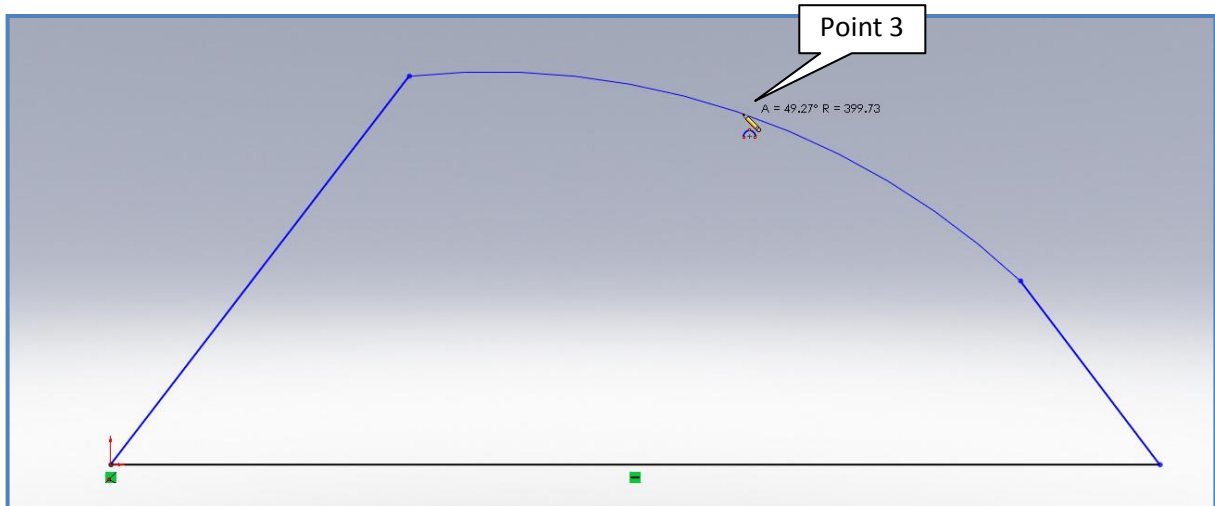



Figure 6

Smart Dimensioning (Line)

- Choose "Smart Dimension" from the "Sketch" toolbar (Figure 7).
- Click on the line to dimension.
- Drag the cursor and click to position.
- Type the required dimension size in the modify box and choose OK 
- Continue to dimension the remainder of the block as shown in figure 8.

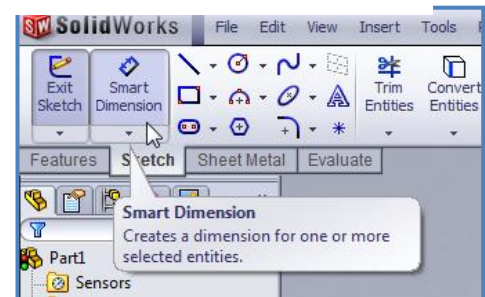


Figure 7

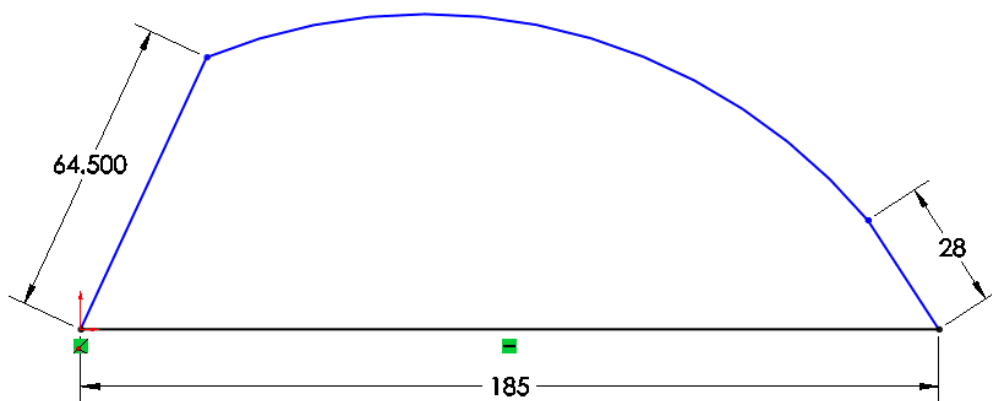


Figure 8

Smart Dimensioning (Angle)

(Smart Dimension still selected)

- Click on one of the lines to select.
- Click on the 2nd line forming the angle.

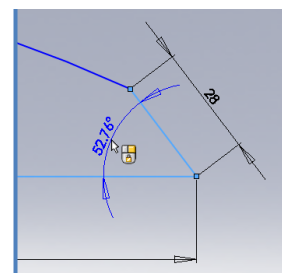


Figure 9

- Insert the required 60° angle in the modify box.
- Accept the new size.
- Repeat procedure for the 2nd angle.

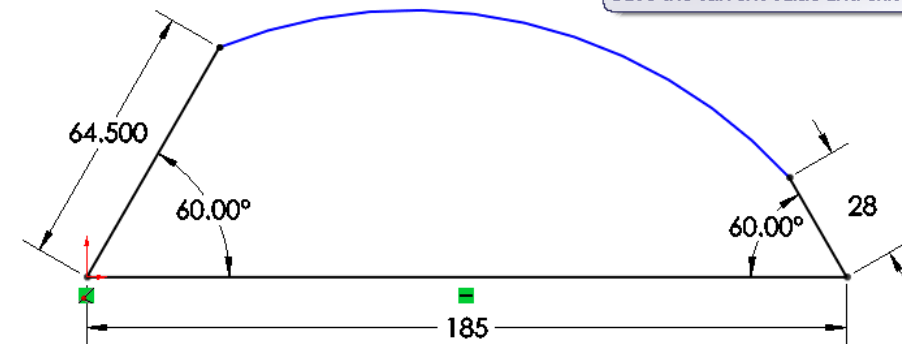
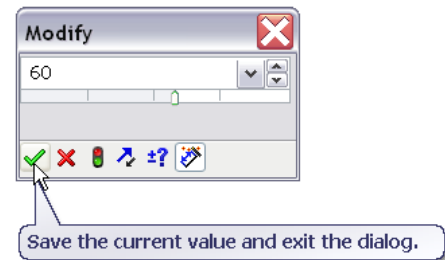


Figure 10

- To complete the sketch “**Smart Dimension**” the radius for the arc as 173mm, by clicking on the curved line

The sketch is now fully defined (figure 11). (lines all black colour)

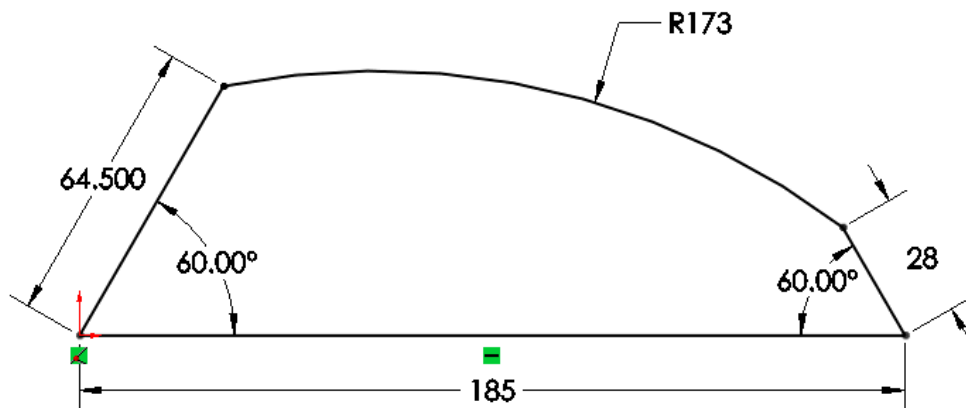



Figure 11

Exit the sketch by choosing  from the confirmation corner located at the top right hand corner of the screen.

Extrude

The “**Sketch**” is completed at this stage. To create the feature from the sketch it must be extruded.

- Click on the Features Tab to display the features tools (figure 12).
- Select Extruded Boss/Base.
- Choose Mid Plane End Condition (Figure 13).
- Set the depth of the extrude to 120mm.
- Choose OK

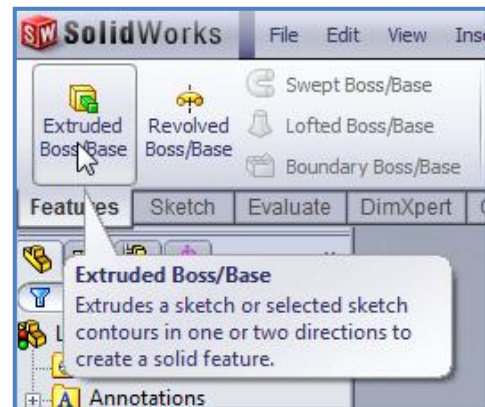


Figure 12

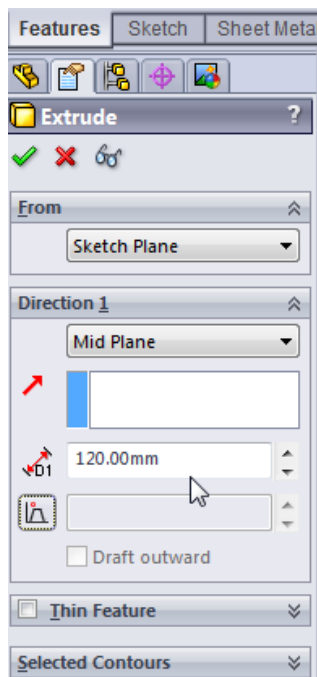
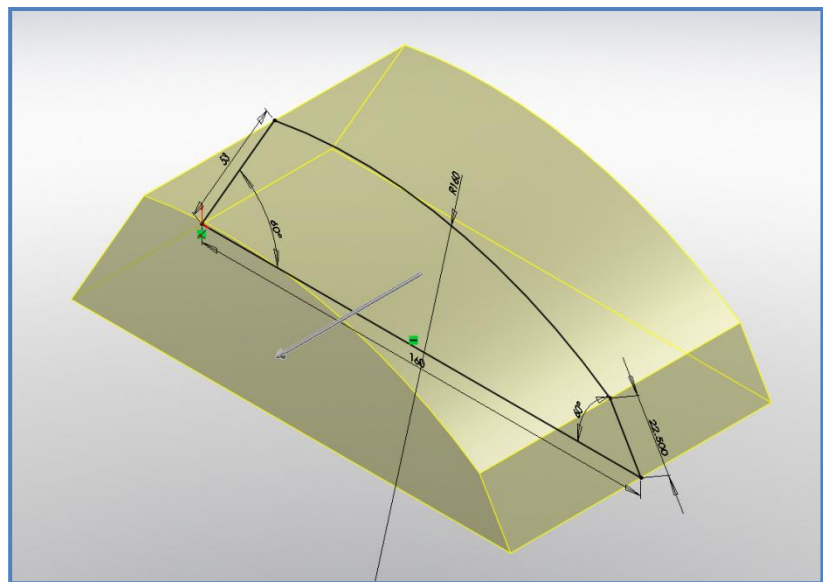



Figure 13



Adding Fillets

- Select Fillet from the Features Tab
- Using the manual option:
Input a fillet size of 10mm.
- Click on the 2 edges to fillet to select them (Figure 15).
- Choose OK  to create the fillet.

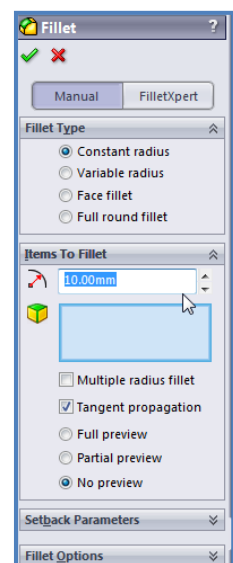
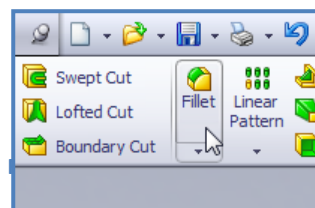


Figure 14

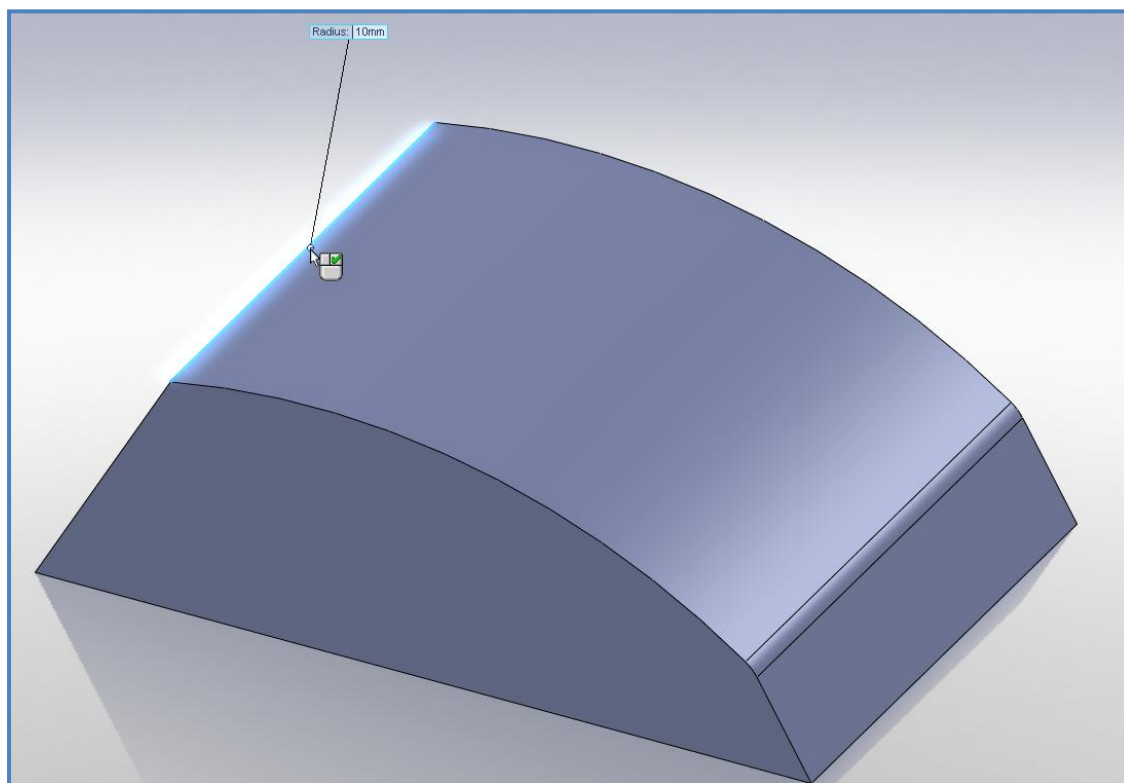
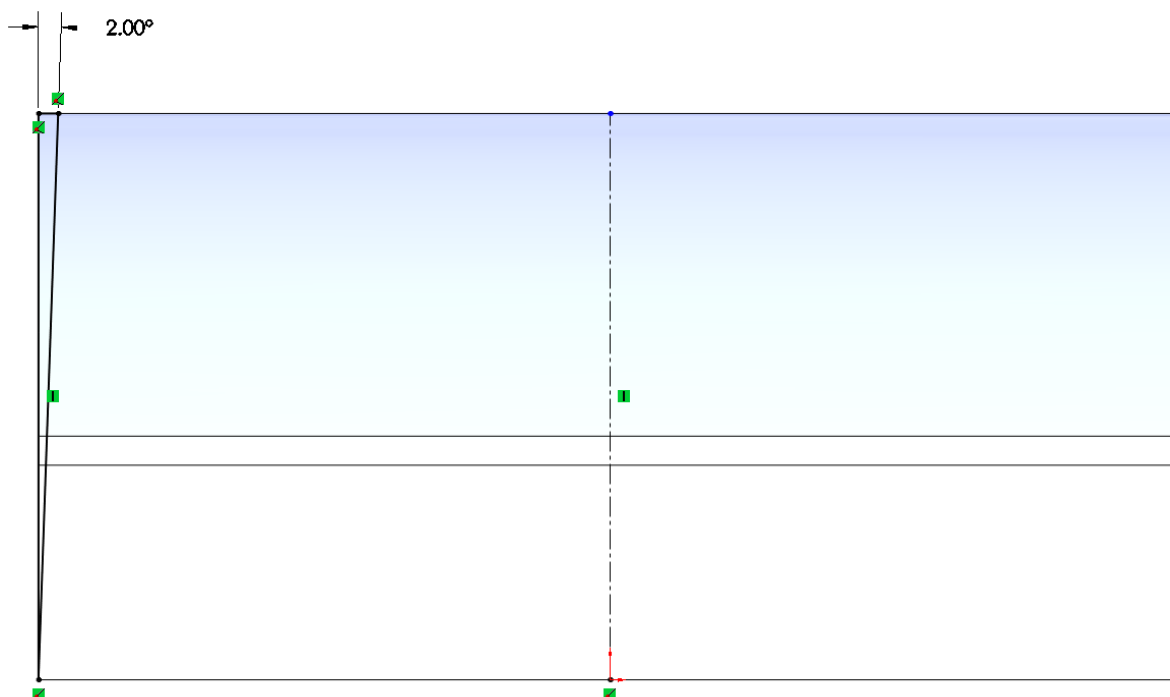


Figure 15

You may wish to add a draft angle to the sides of the mould for ease of removal afterwards. It should be noted however that this will considerably increase machining time for the part.


Adding a draft angle to the to the sides of the mould

Create the sketch shown on the **Right Plane**.



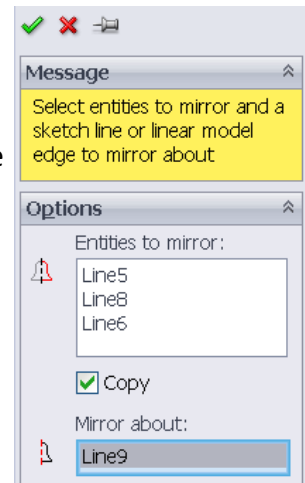
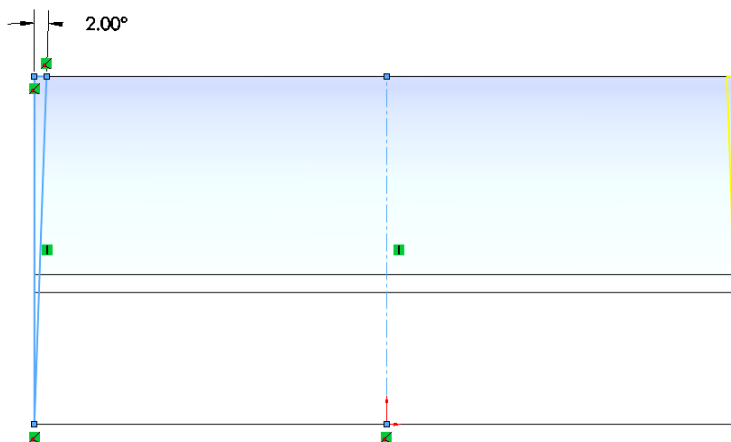
Ladybird Project - Vacuum Mould

We will now mirror the triangle on the left across the centreline.

Choose **Mirror Entities** from the **Sketch** toolbar.  Mirror Entities


Choose the 3 lines of the triangle as 'Entities to mirror' and the centreline as the line to 'Mirror about'

A preview is displayed. Choose **OK** 

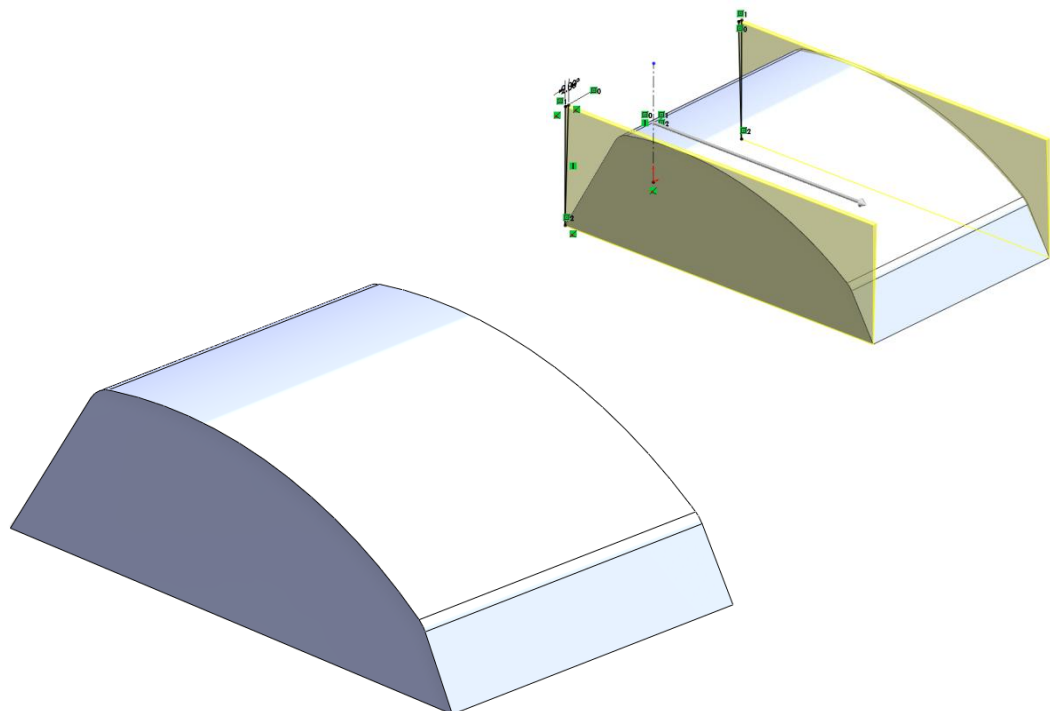


Extruded Cut

We will now cut this profile from the shape of the block.

Choose **Extruded Cut**  from the **Features** toolbar.

Choose **Through All** End Condition and **Reverse Direction** if necessary. Choose **OK**



Adding further fillets.

As the mould will be vacuum formed it may be advisable to add fillets to the remaining corners.

- Choose the Fillet command from Features Tab.
- Input a fillet radius of 3mm.
- Choose the edges to be filleted (Figure 16)
- Choose OK

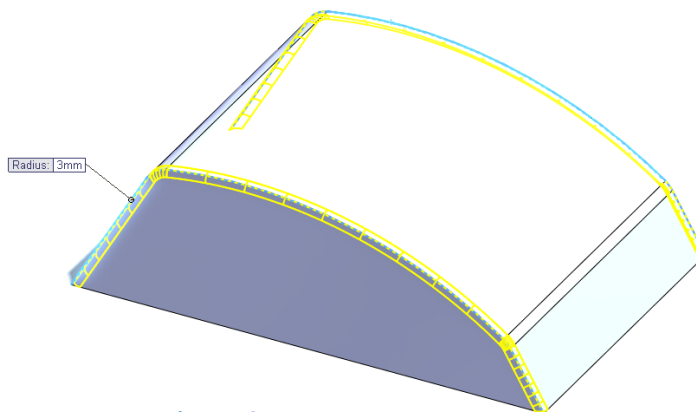
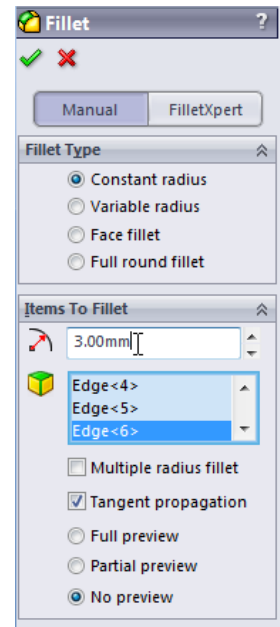


Figure 16



Applying an Appearance

Applying an Appearance to the finished mould can enhance the design.

- Click on the beach ball icon (Appearances and Scenes) on the right of the screen.
- Select **Appearance**.
- Under the **Organic** tab select **Wood**.

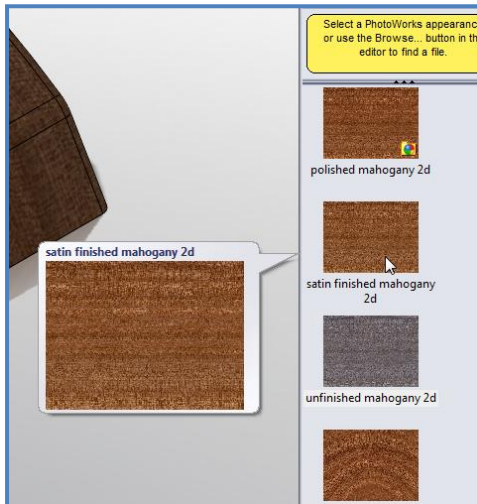


Figure 17

- From the different types of wood select **mahogany** (Figure 17).
- Select the type of finish of your choice by double clicking on it (Figure 18).

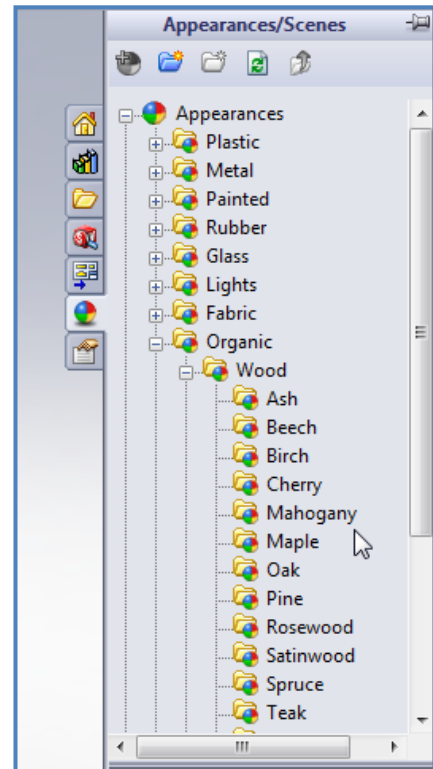


Figure 9



Figure 19

Saving the File

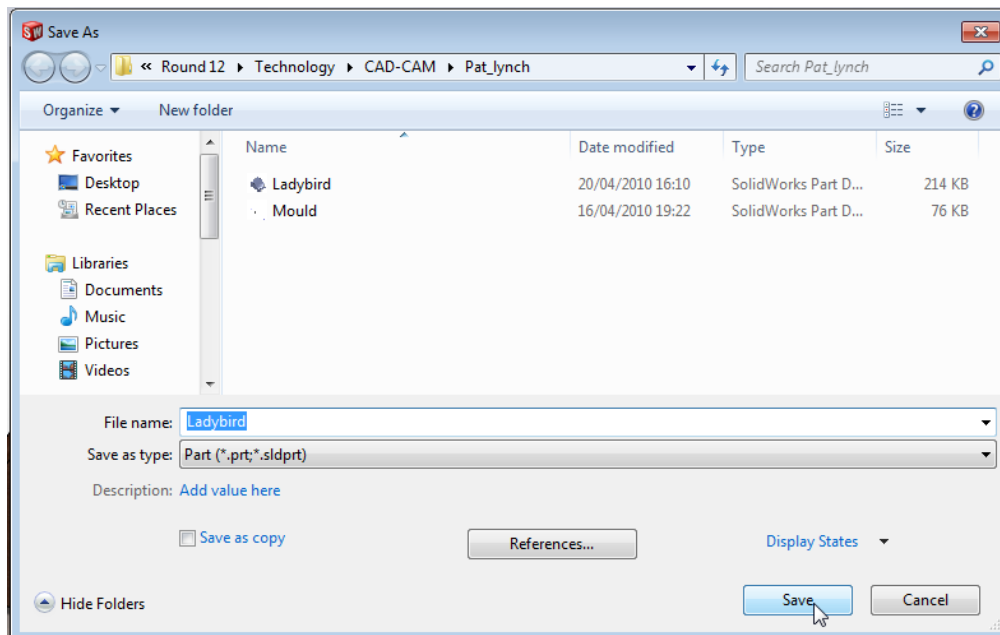


Figure 20

SolidWorks part files are normally saved as .prt files. To machine the component on the CNC Router the file needs to be saved in “STL” format (stereolithography tessellation language) which the Router software will recognise. Choose the STL option from the “Save as type” field as shown below.

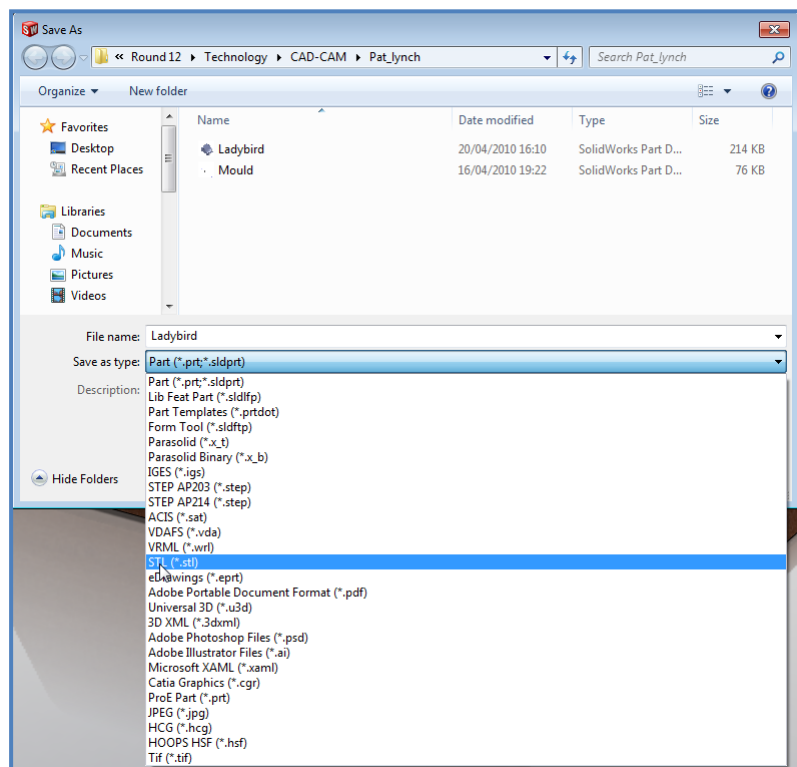


Figure 10