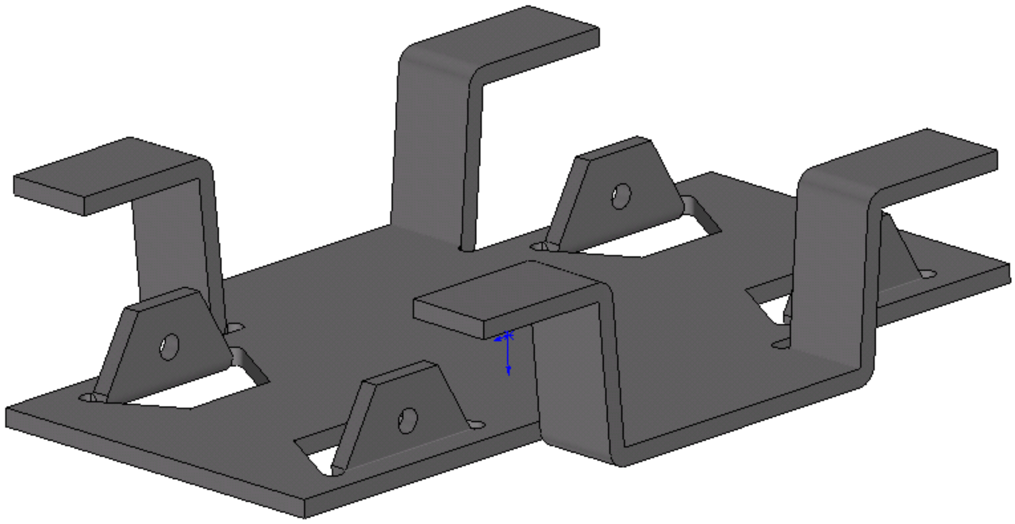
Lesson 5 – Buggy Chassis



Prerequisite Knowledge: To complete this exercise you will need to;

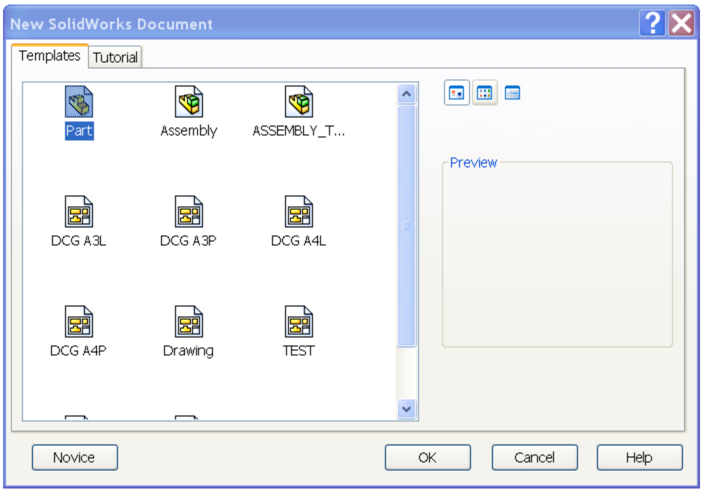
- Be familiar with file navigation and management.
- Have read through the T4 document "Introduction to SolidWorks 2009", available from www.t4.ie
- Worked through lessons 3 and 4 of this series

Focus of the Lesson: On completion of this exercise you will have;

- Used the Sheet Metal features.
- Included design intent into your modeling.
- Familiarised yourself with new features in SolidWorks 2009

Open a new part.

Click **New**  from the Menu toolbar. The **New SolidWorks Document** dialog box is displayed.



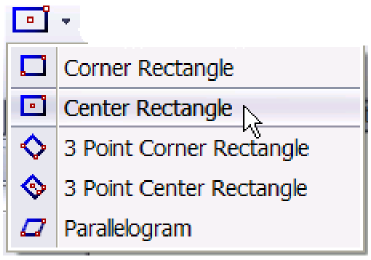
Click the **Part** template

Click **OK** from the New SolidWorks Document dialog box. A new Part document window is displayed.

Save the File Select **File** from the **Menu toolbar,** select **Save As,** a dialog box will appear, name the file **Buggy Chassis**.

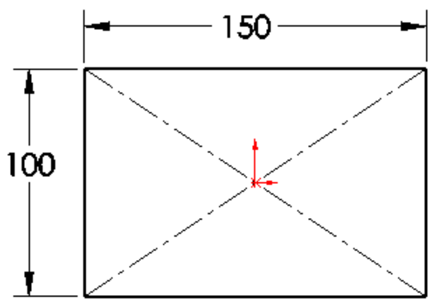
Sketching the Base Select the **Sheet Metal** tab from the **Command Manager.** From the **Sheet Metal** toolbar select **Base Flange/ Tab Command**  . The three planes of reference will appear highlighted. Click on the **Top Plane**. The plane will orientate itself into a **Normal To** position and the **Sketch** toolbar should appear.

From the **Sketch** toolbar select the drop down menu from the **Rectangle Command,** this will display a drop down menu, select the **Centre Rectangle Command** from the list.

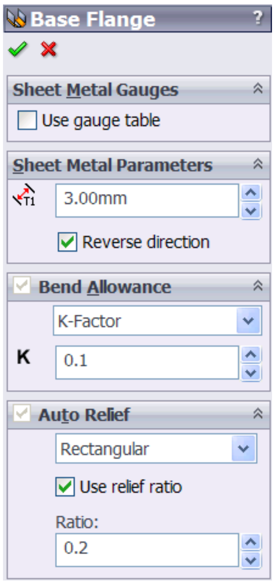


With the origin as centre draw a Rectangle.

Using the **Smart Dimension**  **Command** dimension the rectangle as opposite.



Exit the **Sketch** 



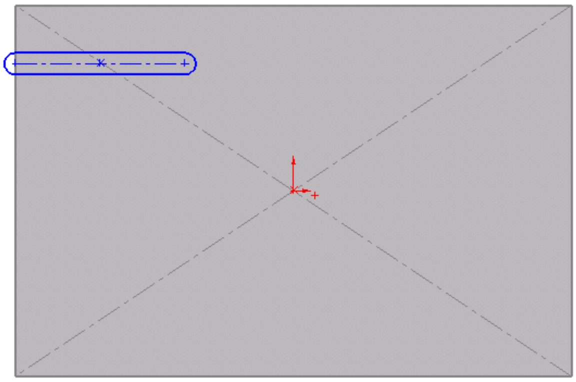
Base Flange/ Tab The Base Flange/ Tab Feature Manager should appear now.

Ensure that all the values are inputted as appear opposite.

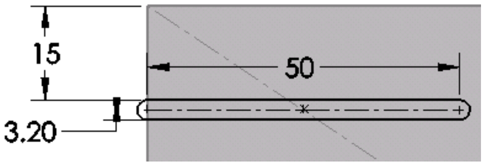
To accept the changes select **OK** 

Sketching Profile Next the slots that facilitate the bending of the wheel arches must be sketched and cut out. Select on the **Sketch Tab** from the **Command Manager**. You are asked to pick a plane or face to sketch on, **Select** the **Top Face** of the part. The next step is to select the **Normal To Command**  from the **Heads UP toolbar.**

Select the **Straight Slot Command** from the **Sketch Toolbar.** This command allows you to draw the centreline of a slot and it will generate the slot from the information inputted. With the start point of the line coincident to the left edge of the draw a horizontal line across the face of the part. A “rubber band” slot should now appear.



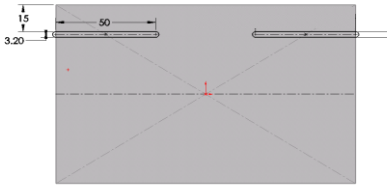
Using the **Smart Dimension Command,** apply dimensions as shown



Mirror this slot across a centreline to produce a copy on the opposite side as shown below.

Mirroring the Profile

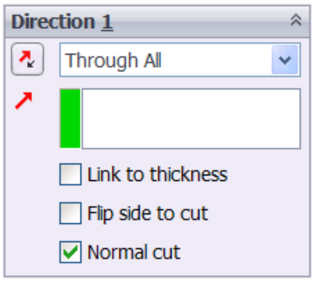
As all slots are the same dimensions and are in specific locations, it is best practice to mirror the entities, rather than redrawing the slot again The first step is to draw the centre line that the entities will be mirrored about.



Select the **Mirror Entities Command** from the **Sketch Toolbar**. Now select the slots as the entities to be mirrored. Click into the box below **Mirror about** and select the horizontal centreline. Accept the changes by selecting **OK**  at the top of the **Mirror Command Manager.**

Adding Features

Select the **Features Tab** from the **Command Manager**, select the **Extruded Cut Command.** Set the end condition of the feature to **Through All.**

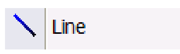


Accept the changes by clicking on **OK** 

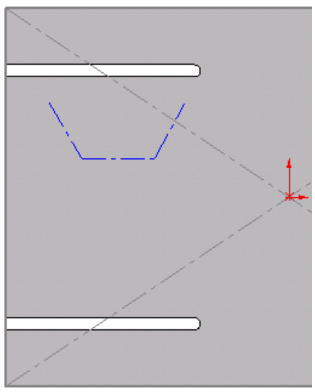
Creating Axle Holders

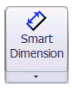
Select the top surface of the part. The part should turn green and a pop up menu will appear. Create a new sketch by selecting the **Sketch** button (highlighted across).

From the **Sketch Toolbar** select the small downward facing arrow to the right of the **Line Command** this displays a drop down menu, select the **Centre Line Command**.



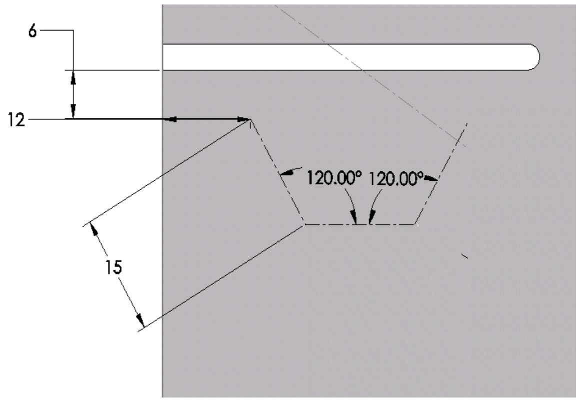
Create a line profile in the top left hand corner of the part, as shown in the blue opposite.





Add Dimensions

Using the **Smart Dimension Command** dimension the rectangle as shown.



Add an equals relation between the three 15mm lines.

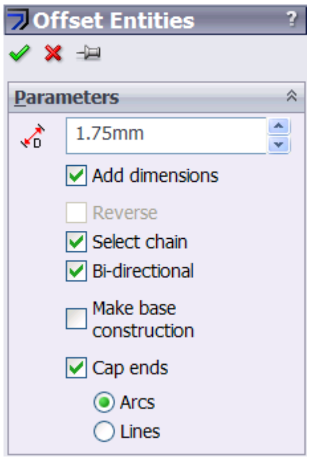
The end points of the lines are 6mm from the slots , and the angle between the inclined lines and the horizontal lines is 120 deg.

Offset Command

Select the **Offset Command** from the **Sketch** toolbar 

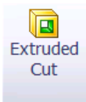
. Using the left mouse button select the centrelines just drawn.

Set the distance D1 as 1.75 mm, and ensure to select **Add dimensions**, **Select chain**, **Bi-directional** and **Cap ends** are selected

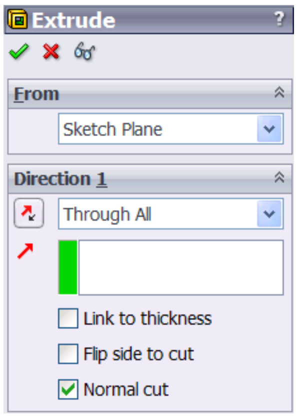


Select **OK** 

Extruded Cut

Select the **Features Tab** from the **Command Manager**. Click on the 

Extruded Cut Command



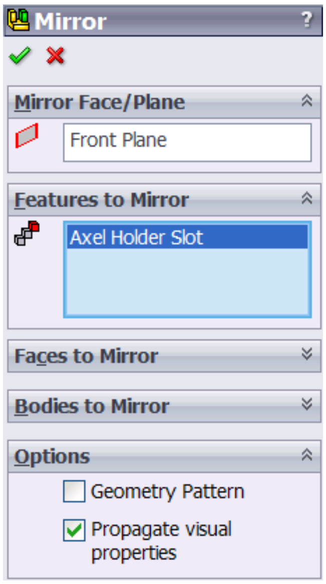
Set the **End Condition** to **Through All,** and ensure that **Normal cut** is selected.

Select **OK** 

Mirror Feature

From the **Features Tab** on the **Command Manager**, select the **Mirror Command** 

From the Feature Manager tree select the **Front Plane** as the **Mirror Face/ Plane,** and select the **Axel Holder Slot** as the **Feature to Mirror**

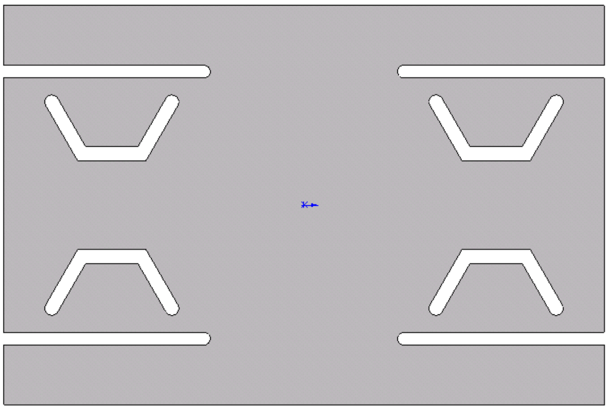


Select **OK** 

Repeat the Mirror Command this time select the **Right Plane** as the **Mirror Face/ Plane,** and select the **Mirror1** as the **Feature to Mirror.**

Select **OK** 

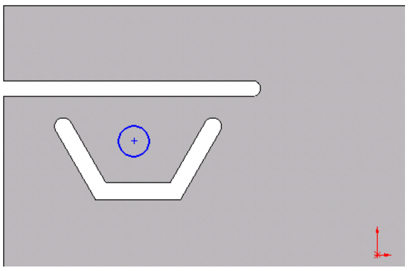
The part file should look as opposite.



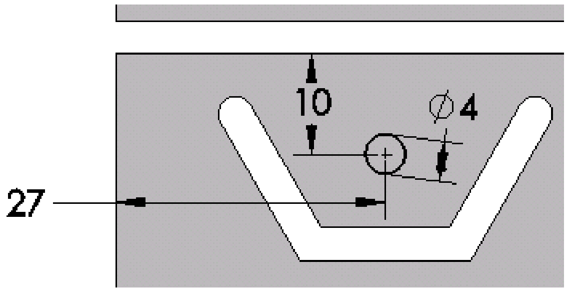
Sketching Axle Holes

To add the holes that the axels will fit into, click on the top surface of the part. A pop up menu will appear select **Sketch.**

From the **Sketch** toolbar select the **Circle Command.** Draw a circle anywhere inside the top left hand axel holder.

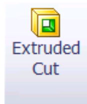


Using **Smart Dimension Command**, dimension the circle as shown opposite.

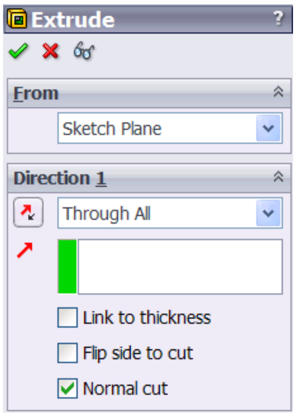


Extruded Cut

Select the **Features Tab** from the **Command Manager**. Click on the **Extruded Cut Command**



Set the End Condition to **Through All**



Select **OK** 

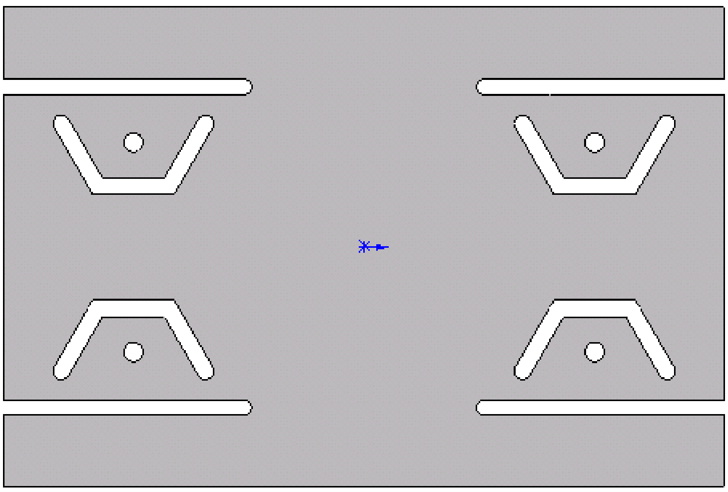
Feature Mirror

From the **Features Tab** on the **Command Manager**, select the **Mirror Command** 

From the Feature Manager tree select the **Front Plane** as the **Mirror Face/ Plane,** and select the **Axel Hole** as the **Feature to Mirror**

Select **OK** 

Repeat the procedure to add the next two axel holes on the right hand side of the part. From the Feature Manager



tree select the **Right Plane** as the **Mirror Face/ Plane,** and select the **Axel Hole** as the **Feature to Mirror**

Select **OK** 

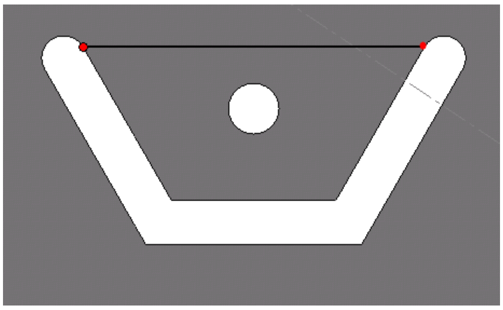
Adding the Bends

Select the **Sheet Metal** tab from the **Command Manager** Click on the **Sketched Bend** 

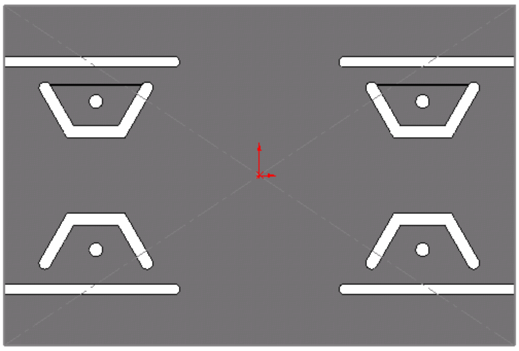
Command you will be asked to pick a plane or face, click on the top surface of the part.

You will notice that the **Sheet Metal** toobar disappears and the **Sketch** toolbar appears, this is for to define a bend line.

Select the Line command and draw a Line horizontally from left to right, from the node at the end of the arc of the slot, to the opposite node. The nodes are highlighted here in red.

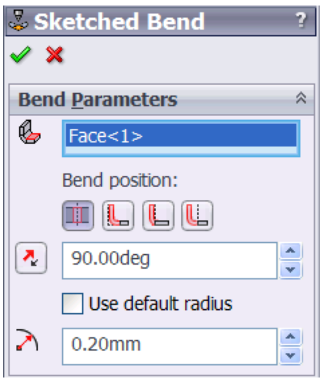


Exit the **Sketch** 



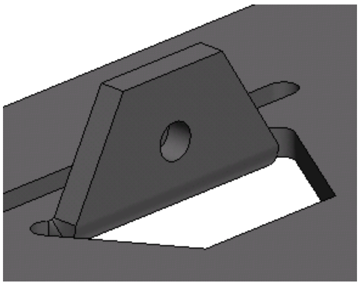
Bend Parameters

In order for the program to determine bend, the fixed face must be selected. Select the part anywhere outside the slot. Set the bend position to centreline as shown. Set the bend angle to **90 degrees**. Deselect the Use default radius box, this will allow you to input a custom bend radius, set this to **0.2mm**.



Select **OK** 

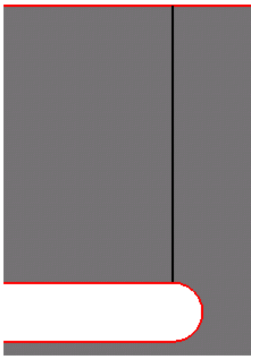
The first two axel holders should now be bent into shape as shown.



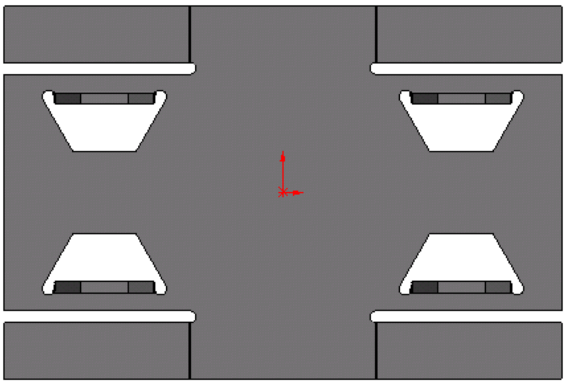
Repeat the procedure above for to bend up the final two axel holders.

Bend 2: Wheel Arch

Select the **Line** command and draw a line vertically up from the end point of the line of the slot, to the edge of the part, as shown.



Repeat for the rest of the initial wheel arch bends as shown below.



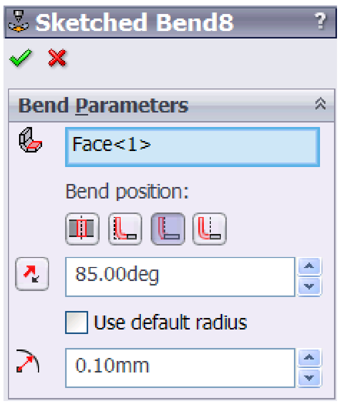
To return to the **Sketch Bend Command** exit the sketch

Set Parameters

Click on the top face of the part as the fixed face.

Set the Bend position to **Material Outside** as shown by selecting the third icon from the left.

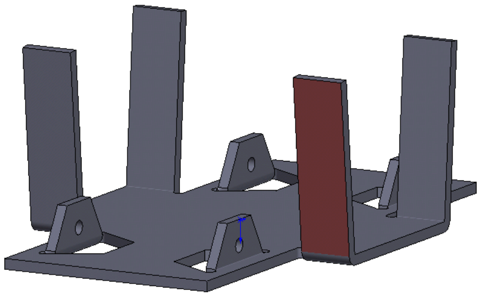
Set the angle to **85 degrees,** and the bend radius to **0.10mm.**



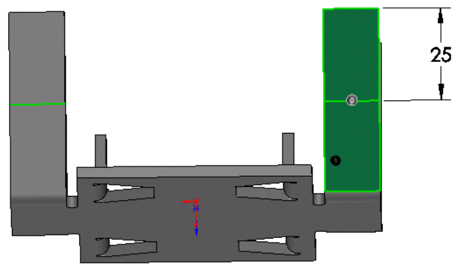
Select **OK** 

Finishing Wheel Arches

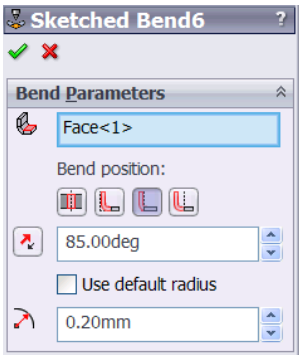
To finish the wheel arches we must perform four more bends, however we can do this in two steps. Select the **Sheet Metal** tab from the **Command Manager,** click on the **Sketched Bend Command** you will be asked to pick a plane or face, select the inside of the wheel arch as shown in red.



Using the **Line** command draw two horizontal lines across the inside surface of the two parallel wheel arches as shown



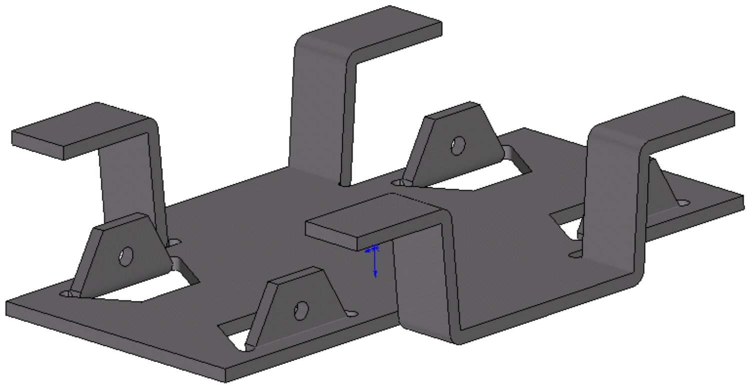
Using **Smart Dimension** add a dimension of 25mm from the top edge of the unfolded wheel arch as shown.



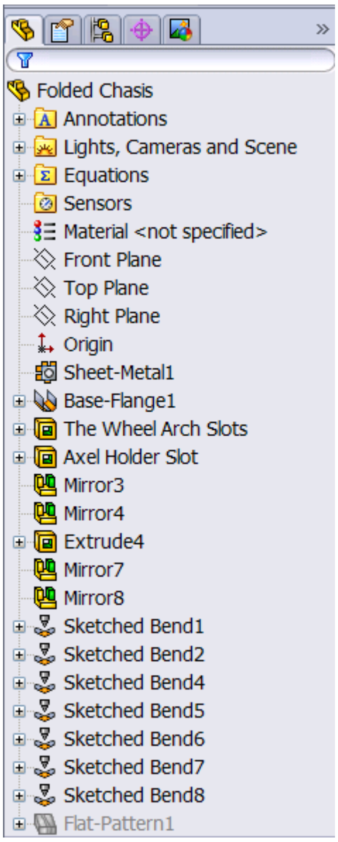
Accept the Sketch and the Sketched Bend dialogue box will appear. Select the Face below the sketch line as the fixed face. Input all the other data as shown.

Select **OK** 

Repeat the process for the rear pair of wheel arches.



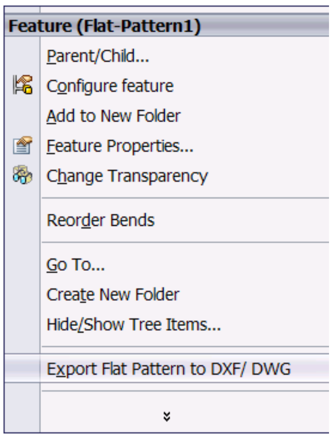
The final product should look like this



Save Save the file by clicking on the **Save** icon 

Export the File In order to use the file on a CNC Router we must export it in a format the router will understand, this format is ***.dxf***. The simplest way to export a Sheet Metal part is to scroll down the **Feature Manager Tree** to the very end of the list. This reads **Flat-Pattern1**. Right click over this greyed out line of text.

A Feature Dialogue box will appear, scroll down to the last option, **Export Flat Pattern to DXF/DWG**, and click on this.



The **Save As** dialogue box will open up. This allows you to save the part in its flat pattern as a

.dxf file. Save this in the folder containing your part file. Click on **Save** button.