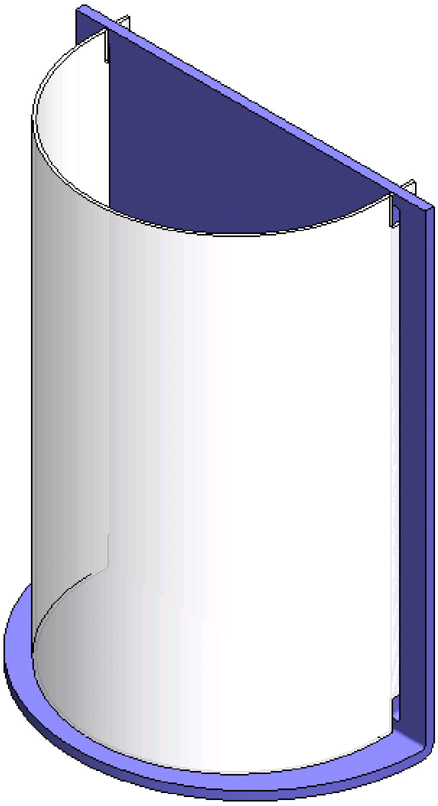
Lesson 3 – Night Light



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

- Be familiar with file navigation and management.
- Read through the T4 document Introduction to SolidWorks 2009, available from www.t4.ie
- Complete units 1 to 5

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

Getting Started

Start the SolidWorks 2009 application.

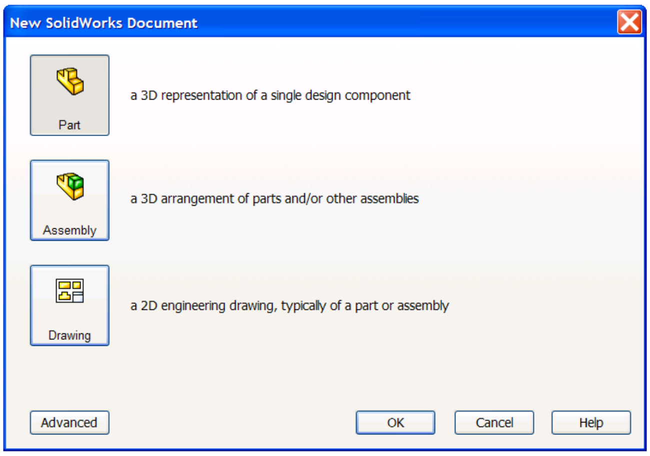
Click the **Start** menu from the Windows interface, located at the bottom left hand corner of the screen.

Select **All Programs**, **SolidWorks 2009**, **SolidWorks 2009 SP 2.1, SolidWorks 2009 SP 2.1.**

You can also quickly start a SolidWorks session by double-clicking the left mouse button on the desktop shortcut, if there is a shortcut icon on the system desktop.

Open a new part.

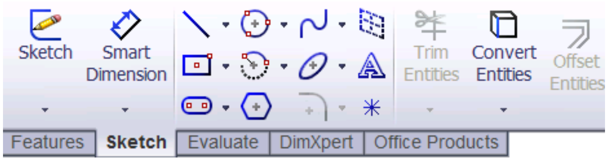
Click **New** from the Menu toolbar. The **New SolidWorks Document** dialog box is displayed.



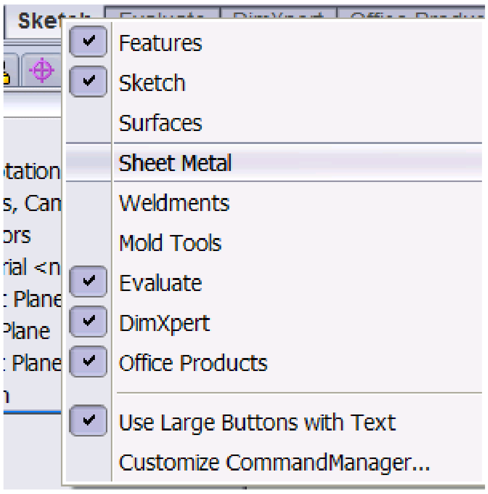
Click the **Part** template

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

Adding Commands We will be using the **Sheet Metal Command** in this lesson. This toolbar is not included as default on the **Command Manager.** The **Command Manager** is the set of gray tabs situated below the toolbar that spans the screen.



To edit the commands that are presented in the **Command Manager**, hover the cursor of the mouse over any of the tabs that already are present, and right click.



A drop down list should appear. Select the **Sheet Metal** option. You should now see the new tab added to the **Command Manager**.

Saving Parts

Select **File** from the **Menu toolbar,** select **Save As.**
Save the file as **Nightlight Back.**

Sketching

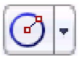
Click on the **Sheet Metal tab** 

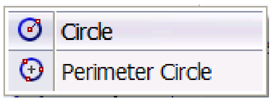
to access the sheet metal tools. From the toolbar click on Base Flange/Tab. 

The planes of reference will now highlight. Select the **Top Plane.**

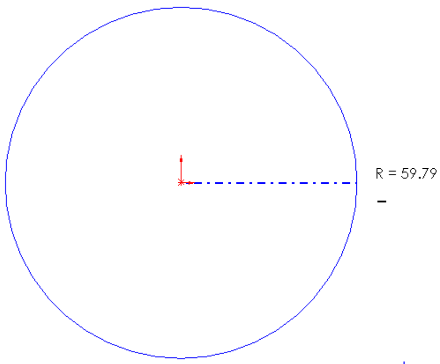
Note how the tool bar at the top switches from the Sheet Metal toolbar to the Sketch toolbar.

Adding the Sketch

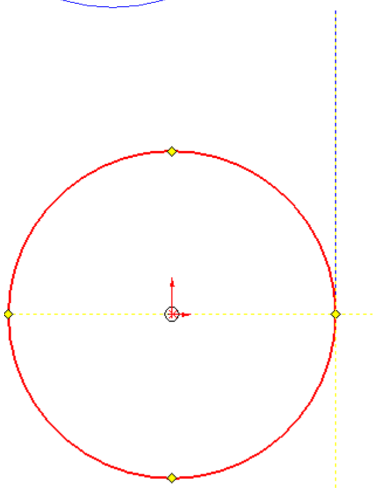
Click on the small downward facing arrow to the right of the **Circle** command  . This will display a dropdown menu, giving choices on the type of circle you wish to draw. The first option, Circle, is the one we require.



With the origin as centre draw a circle of approximately **R60 mm**, we will add relations and dimensions later. Exit the **Circle** command by either clicking on the green  tick at the top of the **Circle Feature Manager** or by pressing the **ESC** button at the top left hand side of the keyboard.



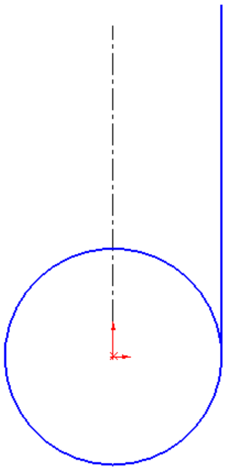
Select the small arrow to the right of the Line command  , this will display a dropdown menu giving the user choices of which line option they wish to choose. Select the first option.



Hover the cursor of the mouse over the extreme right of the circle, this will highlight the four quadrant nodes of the circle. Click on the right node to position the start point of the line and move the mouse vertically up approximately **150mm** and click again to create the end point of the line.

Exit the **Line** command by either clicking on the **green tick**  or by pressing the **ESC** button on the keyboard**.**

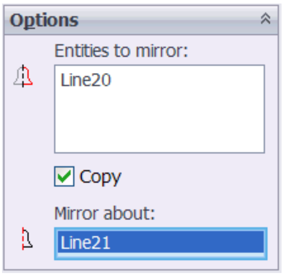
Next we are going to draw a **Centre** line. To access this command click on the arrow to the right of the **Line** command  , this time select the second option **Centreline.**



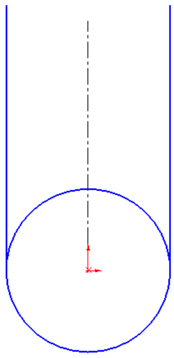
Draw a **Centreline** vertically up from the origin, and exit the command.

Select the **Mirror** command 

button from the **Sketch** toolbar. You are asked for to select **Entities to mirror**, click on line to the right of the sketch. Also asked in the options box is which line you want to mirror these entities about. Click the empty box below **Mirror about** and then click on the centre line in the sketch.

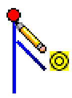


To mirror the selected entities accept the changes to the command by clicking on the Green Tick  at the top of the **Mirror Feature Manager.**



The next step is to draw a line from the end point of one of the vertical lines to the other.

Select the **Line** command , draw a line from the left vertical line to the right, ensuring that the end points have a coincident relation. This can be

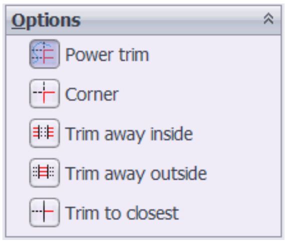


achieved by ensuring this symbol is displayed when drawing the line across. This symbol shows that the start point and end point are concentric. Exit the **Line** command.

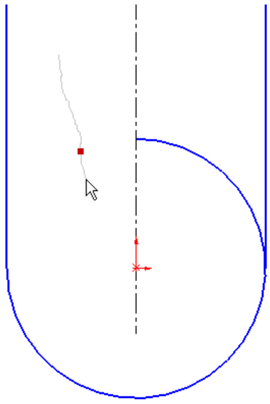
Trimming Entities

Select the **Trim Entities** button 

from the Sketch toolbar.



Four options appear. Ensure that **Power Trim**, the first options is selected.



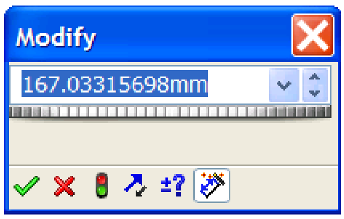
Holding down the left mouse button, drag the left mouse button over the portion of the circle on the inside of the sketch. This will trim the unwanted entities. Repeat for the rest of the circle until you end up with the required shape.

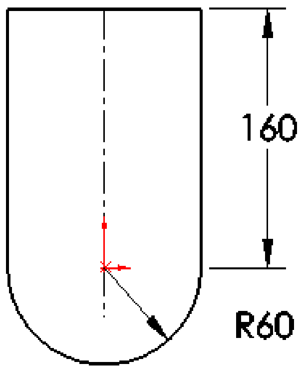
Click on the **Green Tick**  to accept the changes.

Add Dimensions

Select Smart Dimension 

from the Sketch toolbar. You can dimension any line by clicking on it. A rubber band dimension will appear, this will be attached to your mouse, drag this into a clear space in the graphics area and left click. A Modify dimension dialog box will appear. Click in the area highlighted in blue and type in the dimension you want. You can save the changes by clicking on the **Green Tick**  or by pressing the **Return** key on the keyboard.





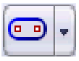
Dimension the sketch as shown

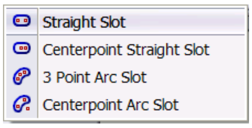
Design Intent

Design intent when working with SolidWorks is basically doing the most amount of drawing in the least amount of steps. It may be asked why this is useful? It is useful for a number of reasons.

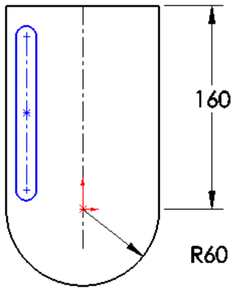
- If you want to change the component, you will have fewer revisions to make.
- The less steps involved the quicker you get the part file created.

Creating the Slots

Now we will include the slots that hold the acetate to the piece. Select the little arrow to the right of the **Slot**  command. This will display a drop down menu. Select the first option, **Straight Slot.**

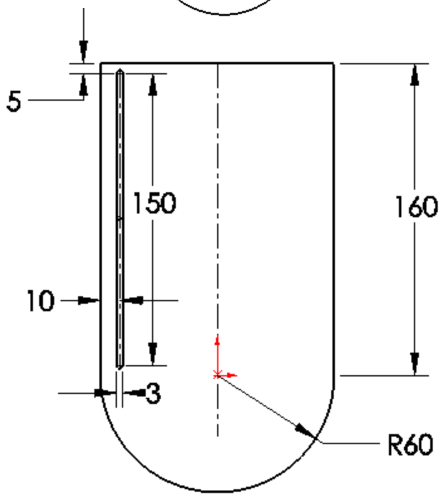


The slot command draws the centre line of the slot first, and then creates the width, so draw a vertical line somewhere to the left of the existing centreline. Then select a width that is within the initial sketch.



Dimensioning Slot

Dimension the slot as shown.

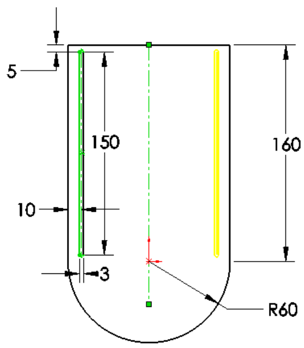


Mirror the Slot

Select **Mirror Entities** 

you are asked which entities you want to mirror. Click on any part of the slot drawn, this will select the whole slot.

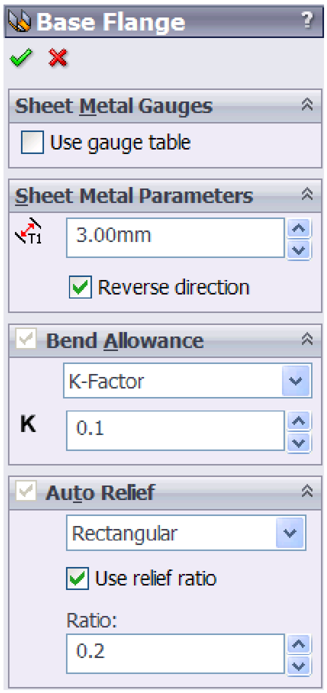
Next you are asked which line you wish to mirror about, select the centreline of the sketch. A preview of the mirrored entity will appear in yellow. To accept click on the **Green Tick** 



Creating the Base Flange/ Tab

Having created the sketch we will now generate the feature.

To accept the sketch click on the **Confirmation Corner** at the top right hand corner of the graphics area.



The sketch should automatically rotate around and the **Base Flange Feature Manager** will appear on the left hand side of the screen. This allows you to input the various values of the component such as

T1	=	Thickness of Material
Bend Allowance	=	Neutral Axis Definition
Auto Relief	=	Ratio of relief T1

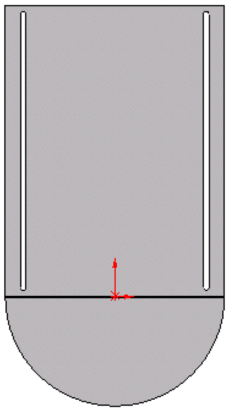
Type in the values as they appear opposite and accept the changes by clicking on the **Green Tick**

Bending a Part

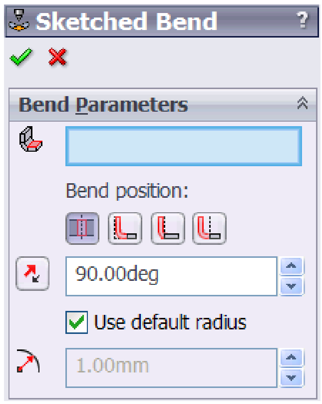
Select the **Sketched Bend** button from the **Sheet Metal** toolbar. You must pick the face where you want the bend to be part. Click on the top face of the part. Select the **Normal To** from the **Heads UP Toolbar.**

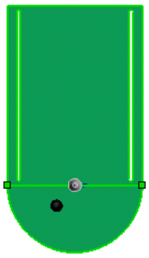
Select the **Line Command** from the **Sketch toolbar.**

Sketch a line through the origin from the left side of the part to the right side as shown. **Accept the Sketch** by clicking on this icon in the top right hand corner of the graphics area.

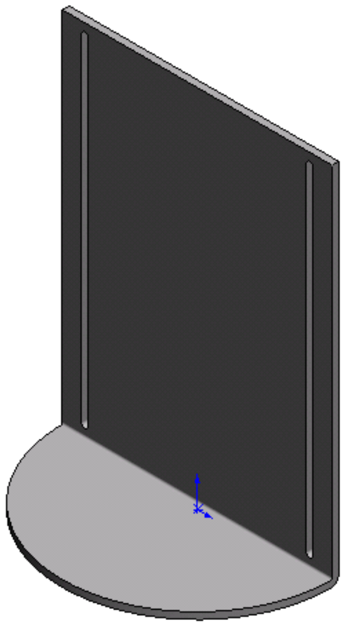


The **Sketched Bend Feature Manager** should appear. Here you need to input the face that is fixed, click on the area between the semi-circle and the sketched line. Ensure all other values are the same as opposite.





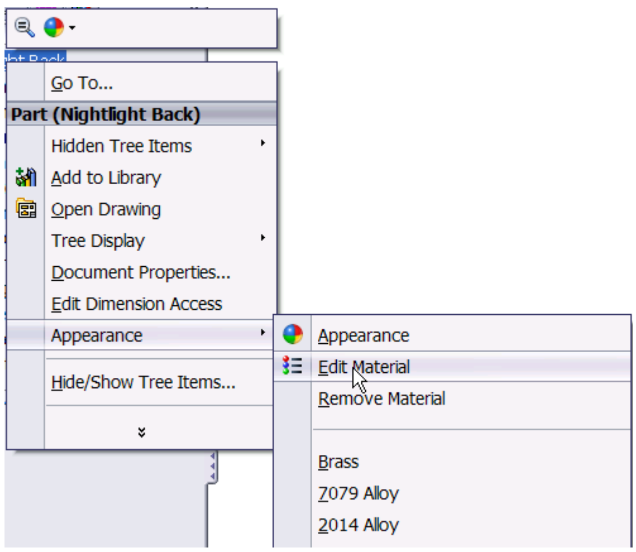
Accept the changes by clicking on the **Green Tick** 

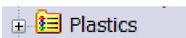


The part should now like this.

Editing Material

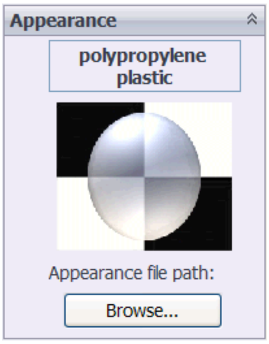
Locate the file name in the **Feature Manager Tree**, it should be at the top of it. Right click on the part name. A pop up menu will appear, scroll down to **Appearance**, another pop up menu will appear, select **Edit Material.**



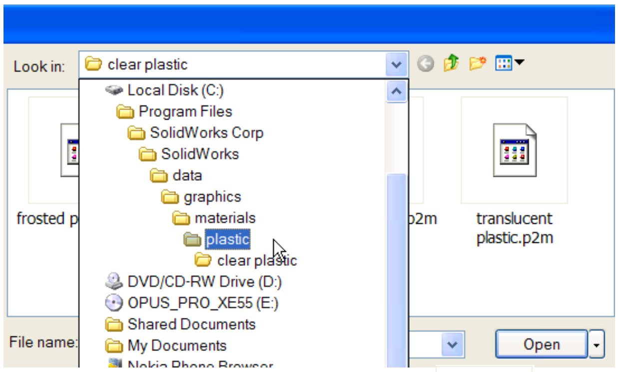
The **Material Manager** should appear, scroll down to a folder called **Plastics**  and double click on it. Scroll down and select a material called **Perspex ™ GS Acrylic Cast Sheet**. Click on **Apply** at the bottom of the Menu.

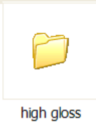
Editing Colour

You can edit the colour by clicking on the **Edit Appearance** button  on the **Heads UP Toolbar**. This will launch the **Appearances Feature Manager**, there are two modes available at the top of the page ensure that **Basic** is selected. The **Selected Geometry** should be should as default be the part name **Nightlight Back.SLDPRT**



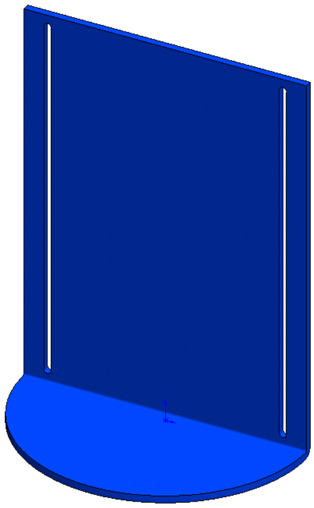
Under the **Appearance** section we will change the appearance from transparent plastic to an opaque plastic. This can be done by clicking the **Browse** button, this will display a dialog box, click the small downward facing arrow beside the name **clear plastic,** this will present a drop down menu, select **plastic** this will present a list of folders, scroll down to find folder **High Gloss** and double click on it.





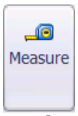
Select the folder called **Blue High Gloss Plastic**, to accept the changes click on the **Choose OK** 

Your part should look like this when finished

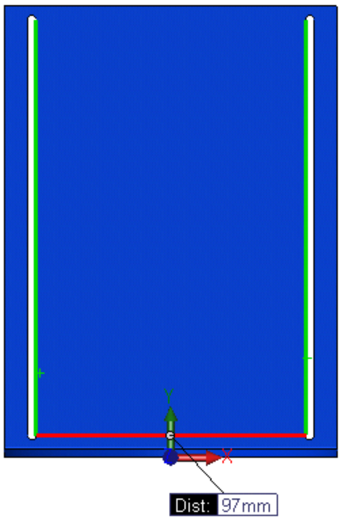


Save the file when done.

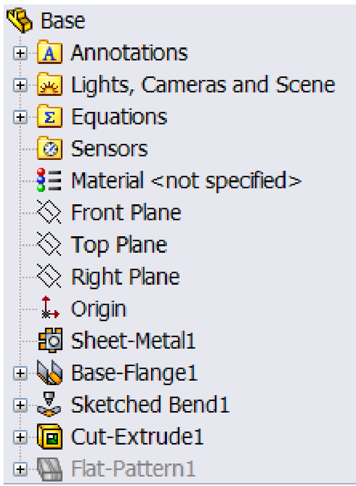
Measure Command Since we have to create a new part which will fit exactly into place with the **Nightlight Back part** it is useful to be able to measure elements of a part. This is possible by using the **Measure Command**. To access it select **Evaluate** from the **Command Manager.** This toolbar contains a variety of tools that allow the user to inspect and analysis the part.

Click on the **Measure Command** 

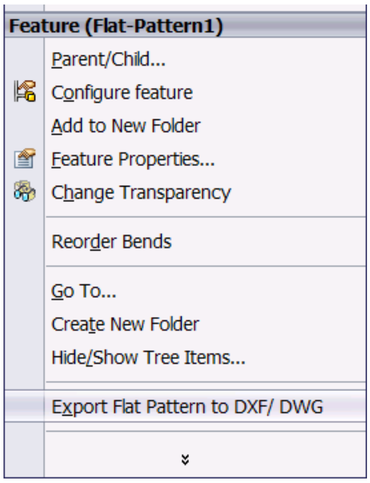
button. Your mouse cursor will change to look like a measuring tape  . Click on the two inside edges of the slots as shown. The two green lines represent the reference lines, the red line indicates the distance line, and **Dist** is the distance of 97mm. we can use this information to create our next part.



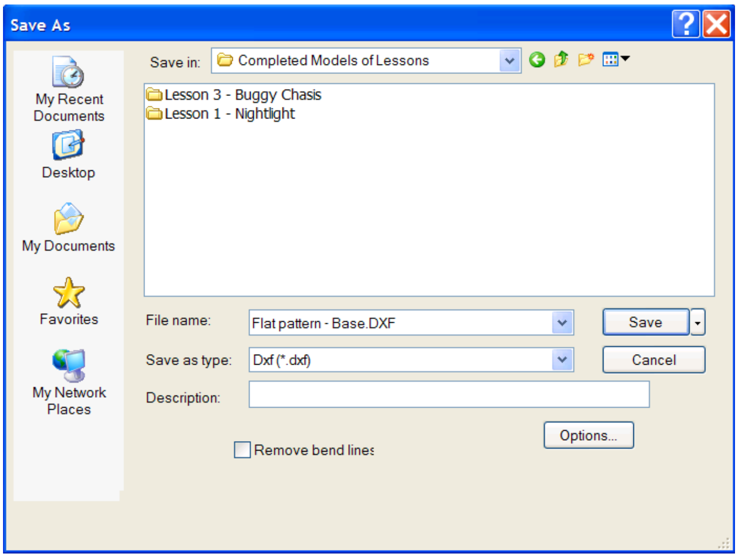
Export the File Later in this tutorial we will be creating our designs on a CNC router, in order to use the file on a CNC Router we must export it in a format the router will understand, this format is ***.dxf***. The simplest way to export a Sheet Metal part is to scroll down the **Feature Manager Tree** 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.



Now we can continue to create the rest of the model.

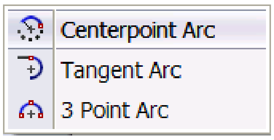
Creating the Front

Create a **New Part File** by clicking on the **New** button which is on the **Menu** toolbar. Select **Part** and click on **OK**

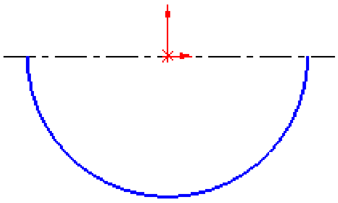
Save this part as **Nightlight Front.SLDPRT**

Create a new sketch by selecting the **Sketch** tab from the **Command Manager**, click on the **Sketch** button. Select the **Top Plane**.

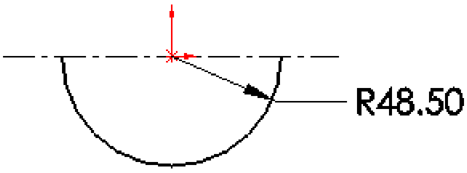
Select the **Centreline Command** from the **Sketch** toolbar. Draw a **Centreline** through the origin.



Click on the small dropdown arrow to the right of the **Arc Command** select the first option the **Centrepoint Arc**. With the origin as centre sketch an arc as shown.



Using **Smart Dimension** add dimensions to the arc as shown.



Helpful Hint

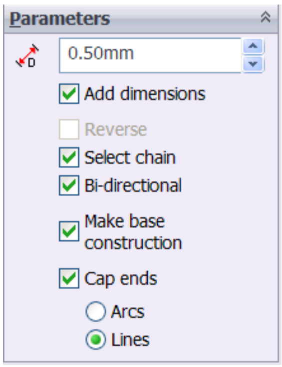
Instead of dividing 98mm by 2 to get the radius, type **98/2** into the **Modify** dialog box, then click on the **Green Tick** or press **Enter** on the keyboard. SolidWorks will do the calculations for you.

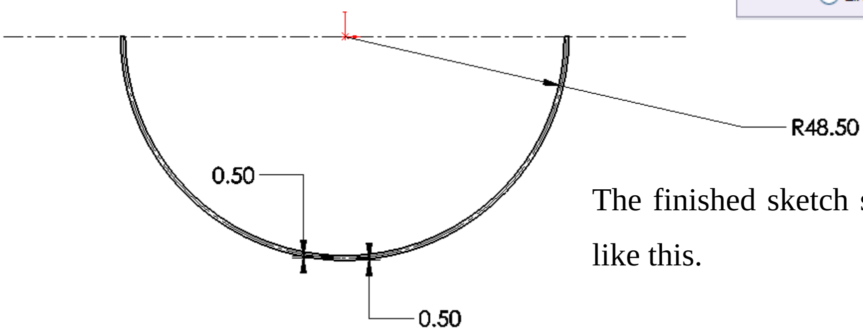
Offsetting Entities

Select the **Offset Entities Command** 

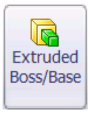
from the **Sketch toolbar.**

Click on the Arc drawn in the graphics area. In the **Offset Entities Command Manager** input the information opposite. **Ensure Make Base Construction** and **End Caps, Lines** are selected.

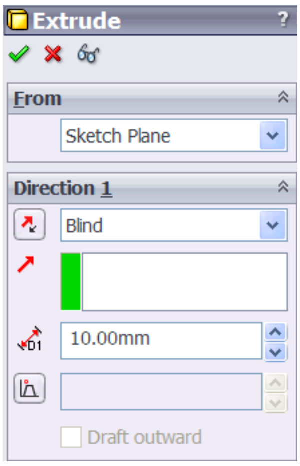




The finished sketch should look like this.

Extruded Boss/Base To extrude this sketch select the **Features** tab from the **Command Manager,** and click on the **Extruded Boss/ Base Command** 

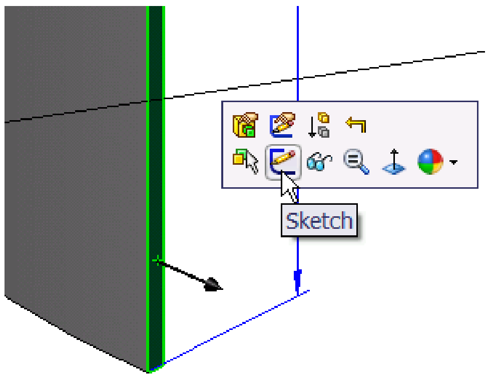
. The **Extrude Feature Manager** will appear on the left.



Set the distance **D1** as **160mm**, ensure all the other properties are the same as opposite.

Sketching the Tabs Rotate the part around by holding down the middle mouse button and moving your mouse around. The part should be orientated so that you can see the concave element of the Extrusion.

Zoom in on the part by rolling your middle mouse button back towards the user. Zoom in until you can see the 1 mm face at the back of the part.



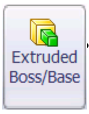
Click on the face. The face will turn green as shown. A pop up menu will appear, select the **Sketch Command** from this pop up menu as shown.

The left face should still be selected. Note if a face is selected it will turn green. Click on the **Convert Entities Command** 

on the **Sketch** toolbar. This will convert the edges to sketch lines.

Click on the right face, and again select the **Convert Entities Command.** This will create a fully defined sketch around the perimeter of the face.

Extruding the Tabs

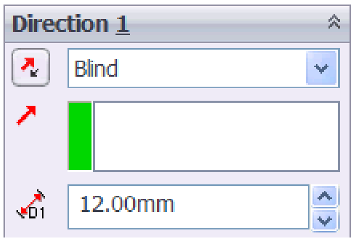
To extrude this sketch select the **Features** tab from the **Command Manager,** and click on the **Extruded Boss/ Base Command** 

. The **Extrude Feature Manager** will appear on the left.

Set the

End Condition = **Blind**

D1 = **12mm**



Accept the changes by clicking on the **Green Tick.**

Extruded Cut

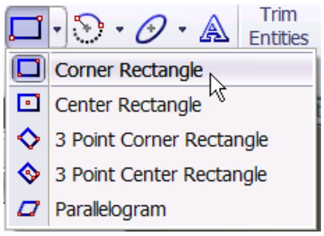
Next step is to create an extruded cut to allow the front part of the nightlight to be retained to the back part. Click on the symbol for **Right Plane** 

in the **Feature Manager Tree**

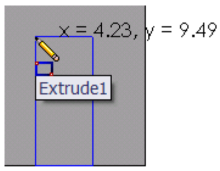
A drop down menu will appear select the **Sketch Command** from the drop down menu. Select the **Normal To Command**  from the **HUD** toolbar.

Select the **Centreline Command** and move the cursor to approximately the half way point of the edge furthest to the right. Sketch a horizontal centreline through the midpoint as shown.

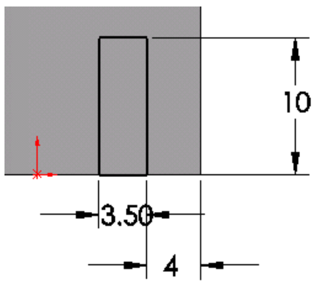
Select the small arrow to the right of the **Rectangle Command** from the **Sketch** toolbar. This will display a drop down menu. Click on the first option the **Corner Rectangle Command.**



To specify the first corner click anywhere on the bottom edge of the part. This will create a rubber band rectangle, click again to define the second corner of the rectangle.



Using the **Smart Dimension Command** from the **Sketch** toolbar, dimension the sketch.

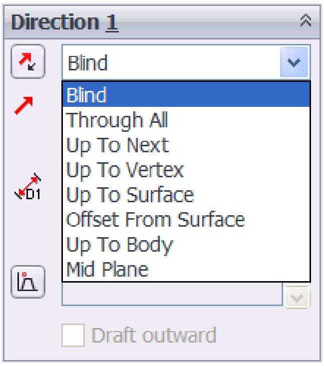


Mirror the sketch about the centreline.

Select **Mirror Entities** from the **Sketch** toolbar. Select the corner rectangle as the entities to mirror and the centreline as the line to mirror about. Choose **OK**

Choose the **Features** tab from the **Command Manager**, then select **Extruded Cut** from the **Features toolbar.**

Up until this point we have only used the **Blind End Condition** this means that the **Extruded Cut** will go solely in one direction. There are other End Conditions, you can choose between them by clicking in the arrow to the right of the current end condition.



Scroll down to and select **Mid Plane**. Set **D1** to a size larger than the diameter of the semi-circle. **125mm** works well. Choose OK to confirm the changes.

Save

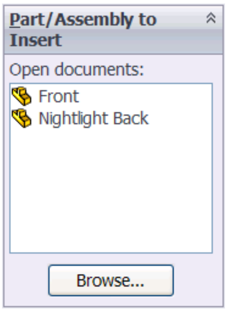
Save the file when done by clicking on the **Save** icon on the **Menu toolbar.**

Assemblies

An assembly file is a file where you group parts together as they would be in real life. This type of file is especially useful in project design as it allows the user to see how parts interact with each other.

Creating an Assembly

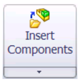
Create a New Assembly file the same as creating a new part file. Select **File** from the **Menu** toolbar, select **New.** The **New SolidWorks Document Wizard** should appear, select **Assembly** 



The **Begin Assembly Manager** should appear on the left, you are asked pick the **Part Files** or **Assemblies** you wish to assemble. Any documents that are open are shown in the first box, However you may need to browse to find the files in the location you saved them.

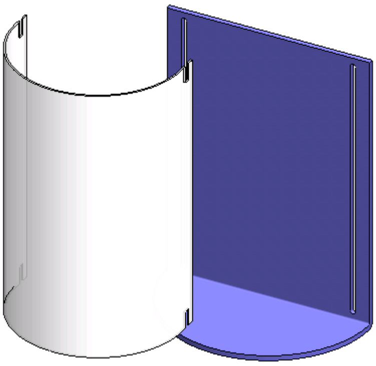
Select the part **Nightlight Back.SLDPRT** and choose **OK**  **.** This is the first component of the assembly, and as we clicked **OK**, the part files **Origin** and the **Origin** of the **Assembly Document** will be aligned and the part **Fixed** in position

Inserting Parts into an Assembly

Click on the **Assembly** tab of the **Command Manager,** select the **Insert Components Command** 

, when the **Insert Component Manager** appears, click on the Browse button and find the file **Nightlight Front.SLDPRT**. The part should now appear in a ghosted preview in the graphics area. Click anywhere in the graphics area to place the part in the assembly file.

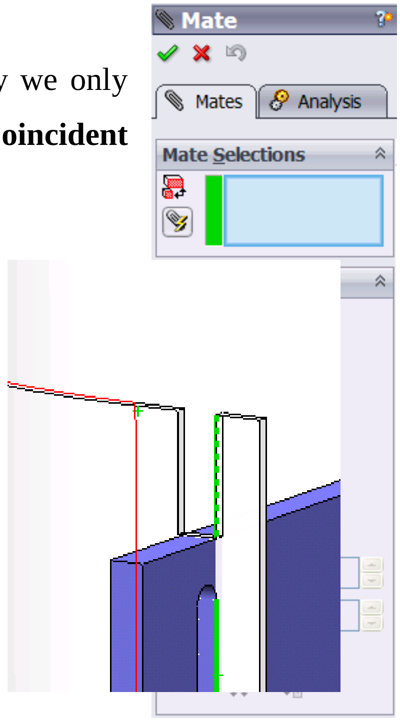
Both parts should now be in the assembly in space.



Adding Mates

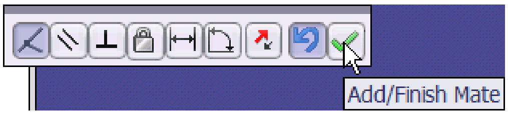
Mates in a SolidWorks Assembly define where a part is placed and also determine how parts interact with each other. Select the **Mate Command** from the **Assembly** toolbar.

There are a number of **Standard Mates** generally we only use a few. For this exercise we will only use **Coincident** from the **Standard Mate** list.

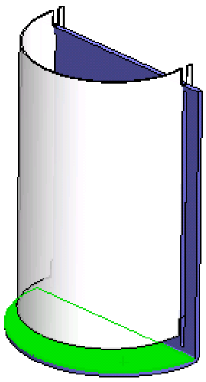


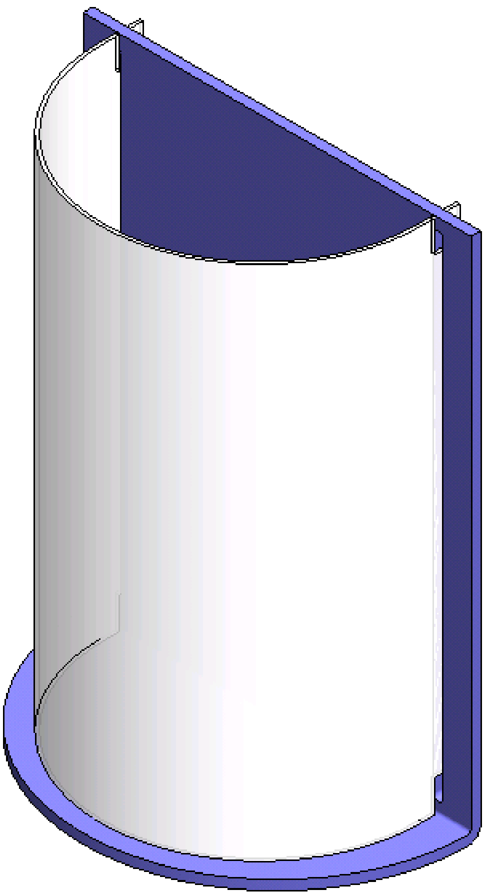
Select the **Outside Edge of the Left Slot** of the **Front part,** and select the Back Face of the Back part. SolidWorks will as default create a **Coincident Mate**.

Accept any changes made by clicking on **Add/ Finish Mate** (the Green Tick)



The next step is to click on the upper face of the base of the **Back Part,** then click on the bottom face of the **Front Part.** A coincident mate should automatically occur. Accept the changes by clicking on **Add/ Finish Mate.**





Exercise Finished !!!