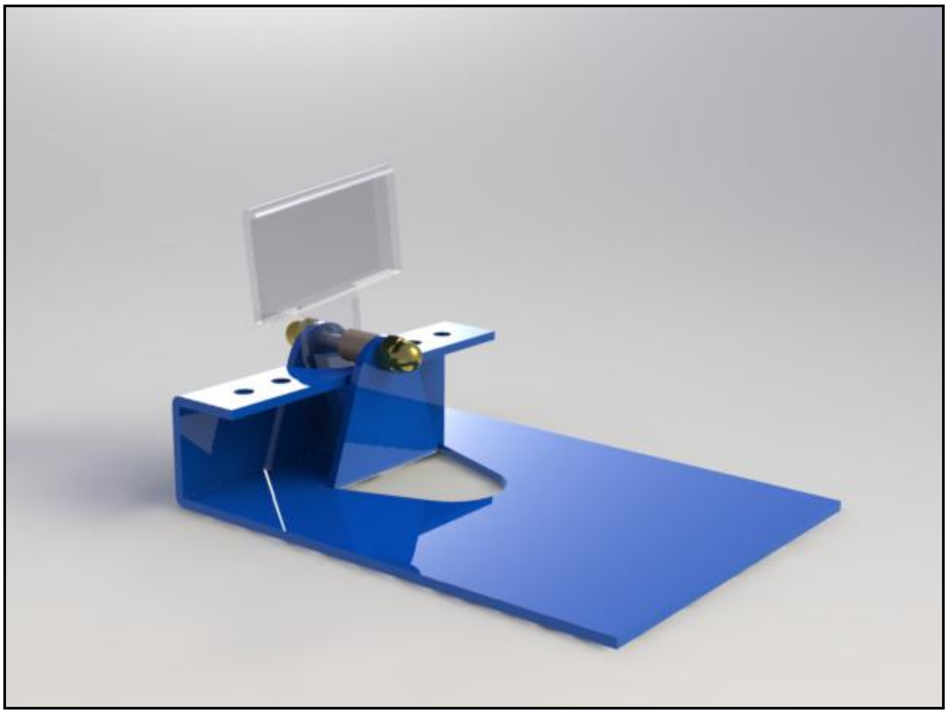
Lesson 4 – Executive Toy



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

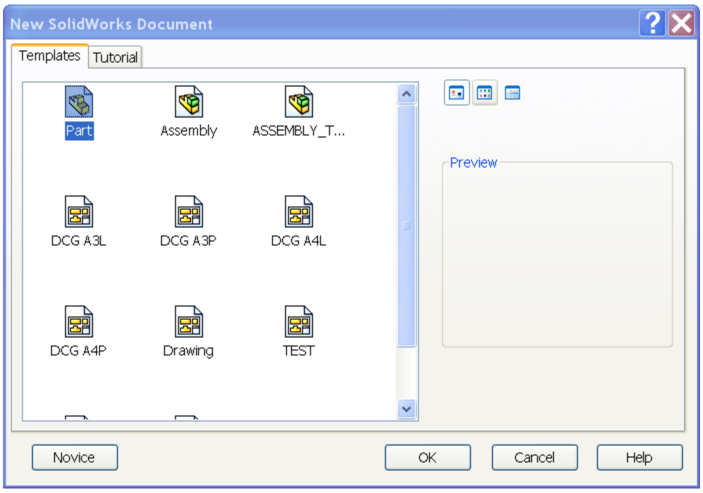
- Be familiar with file navigation and management.
- Read through the T4 document Introduction to SolidWorks. 2009, available from www.t4.ie
- Worked through Lesson 3 of this series.

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.



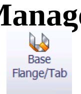
Click the **Part** template

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

Save the File

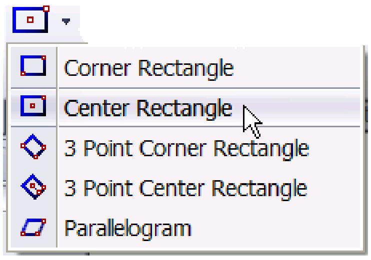
Select **File** from the **Menu toolbar,** select **Save As,** a dialog box will appear. Create a new folder called **Lesson 2 Executive Toy**, name the file **Exec Toy Base** and save it in the folder you just created.

Sketching the Base

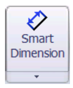
Select the **Sheet Metal** tab from the **Command Manager.** From the **Sheet Metal** toolbar select **Base Flange/ Tab Command** 

. The three planes of reference will appear highlighted. Select the **Top Plane**. The **Sketch** toolbar should appear.

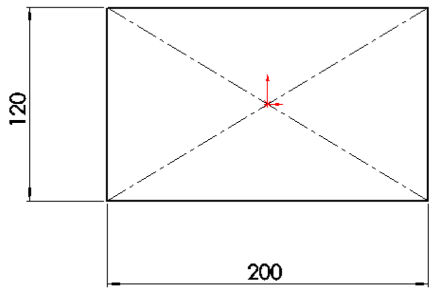
From the **Sketch** toolbar select the drop down menu from the **Rectangle Command,** this will display a drop down menu, select the **Centre Rectangle Command** from the list.



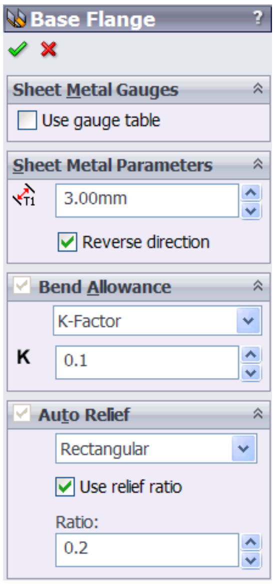
With the origin as centre draw a Rectangle.

Using the **Smart Dimension** 

Command dimension the rectangle as shown opposite.



Exit the **Sketch** 



Base Flange/ Tab

The Base Flange/ Tab Feature Manager should appear now.

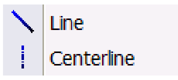
Ensure that all the values are inputted as appear opposite.

To accept the changes select **OK** 

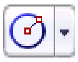
Sketching Profile

The next step is to create the shape that is needed before any bending takes place. Click 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**  m the **Heads UP Toolbar.**

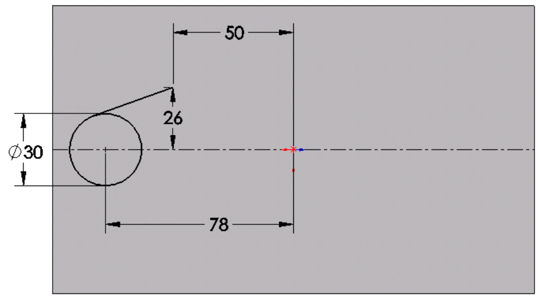
Select the small downward facing arrow to the right of the **Line Command** this displays a drop down menu, select the **Centre Line Command**.



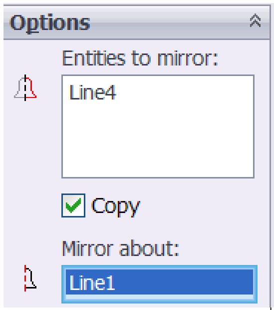
Draw a **Centreline** from the **Left Edge** to the **Right Edge** through the **origin**.

Select the **Circle Command**  **,** draw a circle to the left of the origin with its centre on the centre line. Using the **Smart Dimension Command** from the **Sketch** toolbar dimension the sketch as shown.

Using the **Line Command** draw a tangent line from the circle. **Dimension** the sketch as shown.

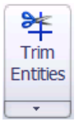


Mirror Command Select the **Mirror Entities Command** from the **Sketch** toolbar. Select the tangential line as the **Entities to Mirror**, and the centreline as the line to **Mirror About.**

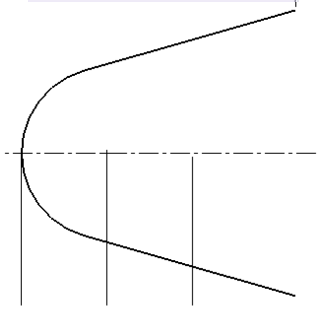


Accept the changes by clicking **OK**

Trim Command Select the **Trim Entities Command**



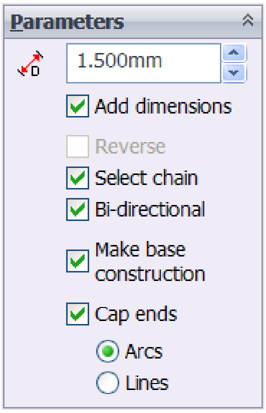
from the **Sketch** toolbar. Click on **Power Trim** from the options list. Holding down the left mouse button drag the cursor over the inside elements circle. This will trim the unwanted entities as shown.



Accept the changes by clicking **OK**

Offset Command Select the **Offset Entities Command**

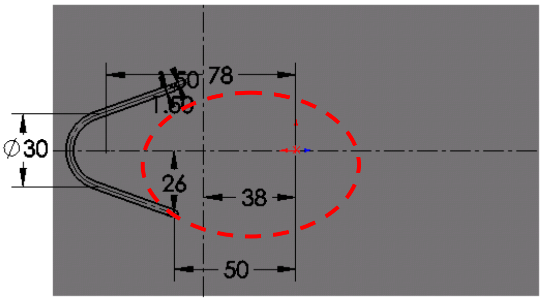
from the **Sketch** toolbar. Set **D1** to **1.5 mm**, ensure the options to the right are selected. By selecting Bi-directional it is possible to add Cap ends which close and fully define the sketch.



Accept the changes by clicking **OK**

Mirror Entities Select the **Centreline Command**, draw a vertical centreline coincident with the two horizontal edges of the part.

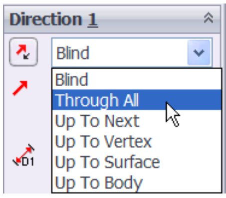
Using **Smart Dimension,** dimension the centreline 44mm from the origin.



Select the **Mirror Entities Command** to mirror the profile about the centreline. Accept the changes, click **OK**

Cut Extrude

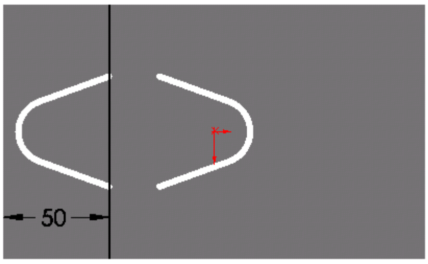
Select the **Features** Tab from the **Command Manager,** and from the **Features** toolbar select the **Extruded Cut Command** Select a ***Through All*** **End Condition.** This will cut through the entire part, regardless of what the dimensions are or if they are changed.



Accept the changes, click **OK**

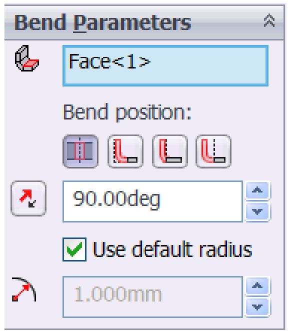
Folding the Part

Select the **Sheet Metal** tab from the **Command Manager,** next select **Sketched Bend** Sketched Bend , select the top face of the part. The Sketch toolbar should appear. Select the **Line Command** from this toolbar. Sketch a line across the part coincident to both horizontal edges.

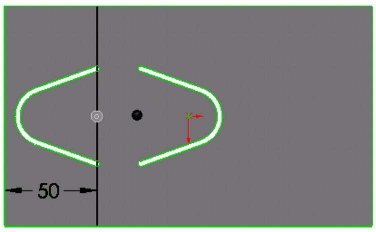


Using **Smart Dimension,** dimension the position of the line as shown. Accept the Sketch by clicking on the **Accept Sketch** button .

The **Sketched Bend Feature Manager** will appear, select the top face of the part to the right of the bend line as the fixed face, as shown by the black dot below. Ensure all the other values are the same as opposite.

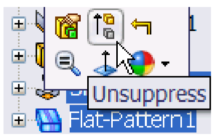


Accept the changes, click **OK**



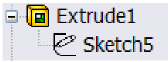
Flattening the Part

Sometimes it is necessary to flatten a part or a specific bend in order to find a measurement, this can be easily achieved by clicking on **Flat-Pattern1** at the bottom of the **Feature**



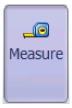
Manager Tree this will display a pop up menu, select **Unsuppress**. This will flatten the entire part.

Viewing Sketches

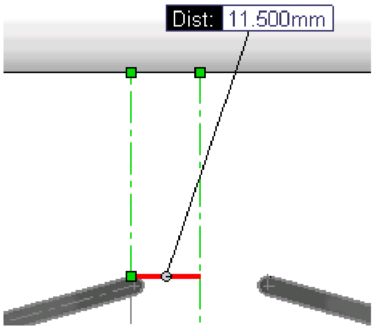
Sometimes in is necessary to view Sketches that are no longer visible in order to find out dimensions of measurements. Here we are looking to view the sketch of the **Extrude Cut**. Select the little **+** to the left of the Extrude Cut feature, this will expand the tree and show the sketch. 

Click on the Sketch, this will display a pop up menu. Select the icon that resembles a pair of spectacles . This will show this sketch.

Measure

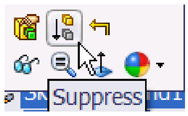
Select the **Evaluate** tab from the **Command Manager**. Select the **Measure Command** 

from the **Evaluate** toolbar. Now select the two vertical lines, this will display the distance between the two lines from the two sketches. Close the **Measure** dialog box.



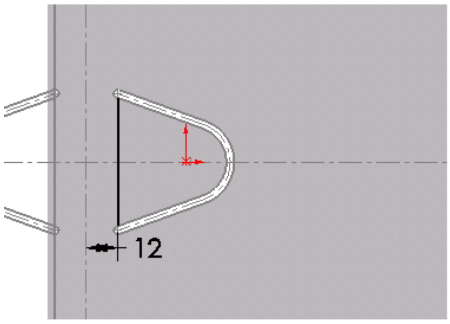
Continue Forming

Next step is to create another sketched bend. To do this you will have to **Suppress** the **Flat-Pattern** command. Click on **Flat-Pattern1** this will display a pop up menu. Select **Suppress.**

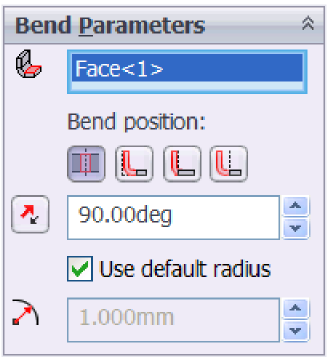


Select **Sketched Bend** 

from the **Sheet Metal** toolbar. Click on the top face of the part, and select the **Normal To**  **Command** from the **Heads Up** toolbar. Using the **Line Command**, sketch a line coincident to the two inside edges of the slot. Dimension using the **Smart Dimension Command**.



Accept the changes to the sketch by selecting

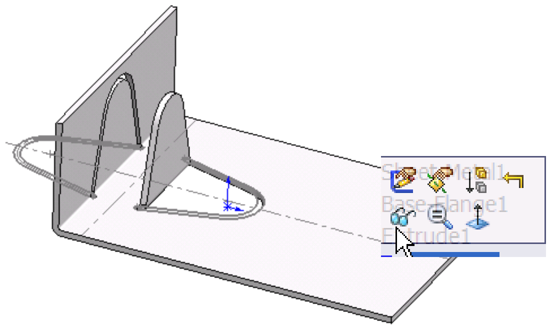


Select the flat face to the left of the bend line as the **Fixed Face**.

Accept the changes by clicking **OK**

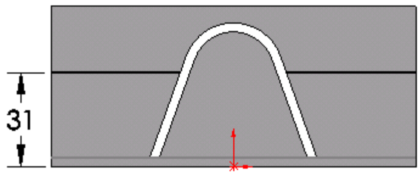
The part is now folded as shown. Now we can hide the sketch.

Click on the **Sketch** you wish to hide a pop up menu will appear, select **Hide**

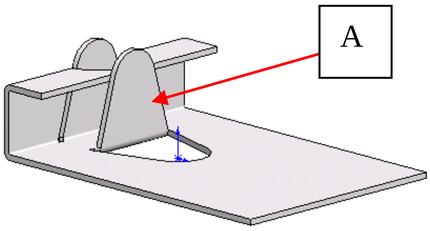


Sketched Bend

To complete our series of bends, another bend is required. Select the **Sheet Metal** tab from the **Command Manager**. Click on the **Sketched Bend Command** Sketched Bend Select the **Upright** face of the part, using the **Line Command**, sketch **two lines** from the outside edges to the outside of the slots. Use **Smart Dimension** to dimension it as shown.



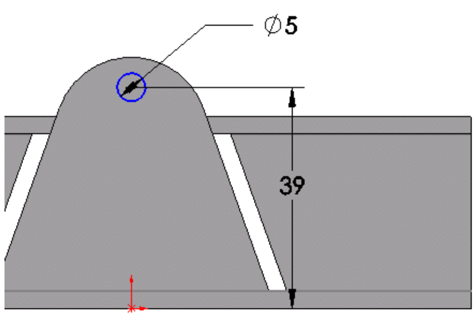
Accept the sketch by clicking on . Pick the face below the bend line as the **Fixed Face**. Accept the changes by clicking on **OK**



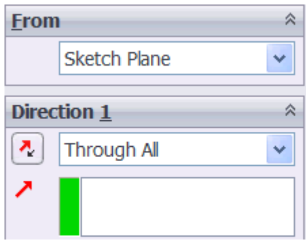
Creating Holes

In SolidWorks you can create holes in two ways, Extrude Cut a circle through the part, or use the Hole Wizard. For this exercise we are going to concentrate on using Extrude Cut to create the holes.

Create a sketch on the Face A as highlighted above sketch a circle on this face, using the **Smart Dimension command** to add the 39mm height and 5mm diameter dimensions as shown opposite.



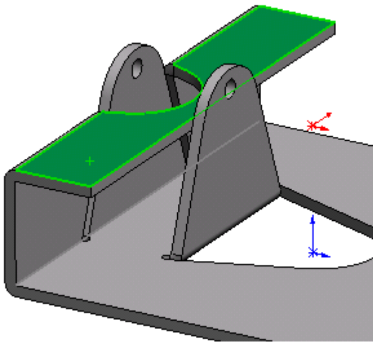
Select the **Feature** tab from the **Command Manager.** Click on the **Extruded Cut** command, set the end condition to **Through All** accept the changes by clicking on **OK**



Pencil Holes

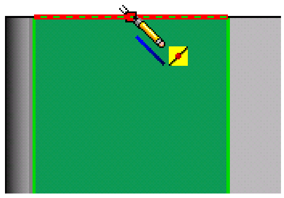
To create the pencil holes, select the **Sketch** tab from the **Command Manager**, click on the **Sketch Command** and select the top face highlighted in green.

Select **Normal To** on the Heads Up toolbar. this will turn the part around to a true shape of the top face.

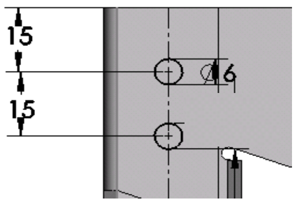


Select the **Centre Line** command from the **Sketch** toolbar.

Draw a vertical centre line from the midpoint of the top edge to the mid point of the bottom edge as shown.



Using the **Circle command,** sketch two circles with their centre on the centre line.

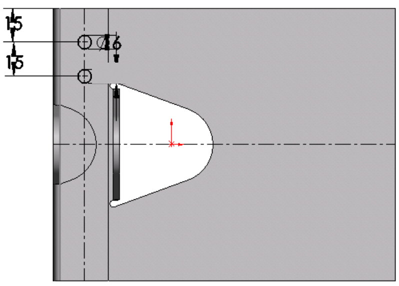


Dimension these as shown using the **Smart Dimension** command. Add an **Equals** relation between the two circles

Mirror Entities

Draw a horizontal centre line down the length of the part. Select the **Mirror Entities**  command from the Sketch toolbar.

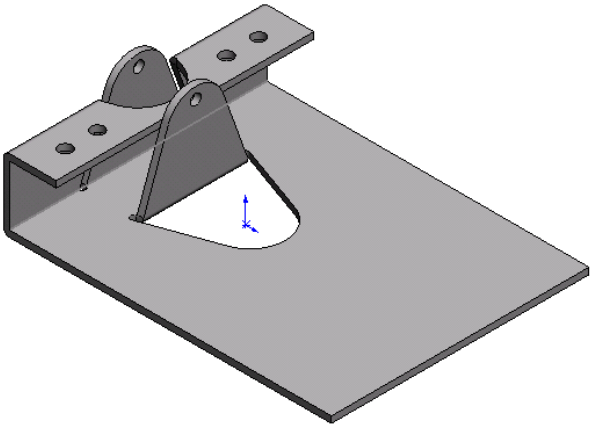
Select the two circles as the *Entities to mirror* and the horizontal centre line as the *Mirror* about entity. Select **OK**



Extrude Cut

Select the **Features** tab from the **Command Manager,** click on the **Extruded Cut** command. Set the **End Condition** to *Up to Next.* Select **OK**

The part should look like this.



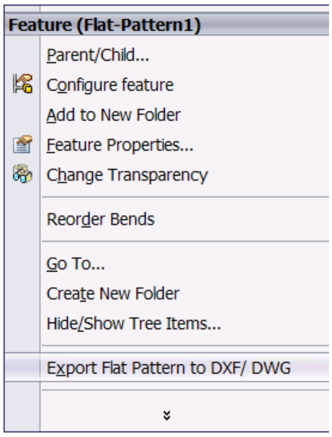
Save

Save the file.

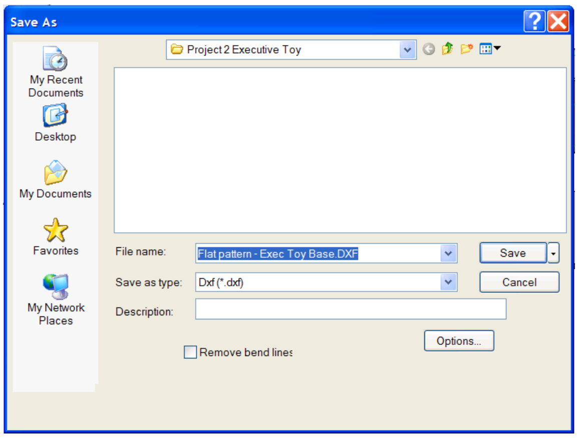
Export the File

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.

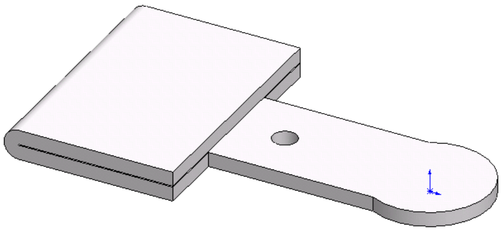
A Feature Dialogue box will appear, scroll down to the last option, **Export Flat Pattern to DXF/DWG**, and click on this.

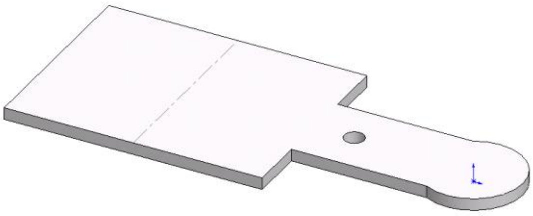


The **Save As** dialogue box will open up. This allows you to save the part in its flat pattern as a ***.dxf*** file. Save this in the **Lesson 2 Executive Toy** folder containing your part file. Click on **Save** button.



Part 2 – Animated Picture Frame



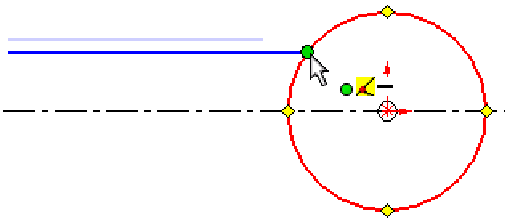


Starting out

Open a New Part, and Save it as **Picture Holder.SLDPRT** in the folder **Lesson 2 Executive Toy**

Creating the Part

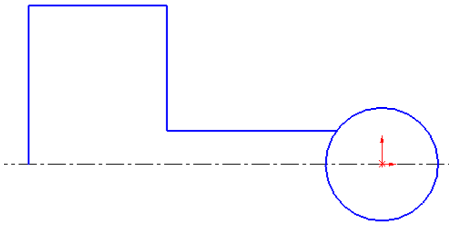
Select the **Sheet Metal** tab from the **Command Manager**. Click on the **Base Flange/ Tab Command**  . Select the **Top Plane**. Select the **Centreline Command** from the **Sketch** toolbar, draw a horizontal centreline through the origin. Next using the **Line Command** draw a line parallel to the centreline to the left of the origin.



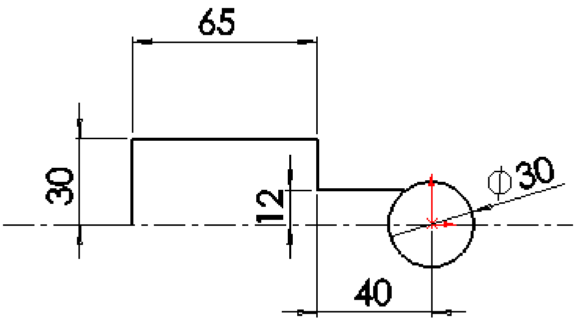
Using the Circle Command sketch a circle that has its centre on the origin.

Create a coincident relation between the line and the circle.

Using the **Line Command** draw the rest of the profile as shown.

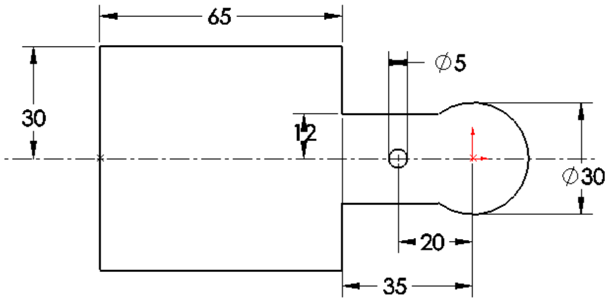


Use the **Smart Dimension Command** to dimension the profile as shown



Mirror Entities Select the **Mirror Entities Command** from the **Sketch** toolbar. Mirror all the lines about the centre line.

Circle Command Use the **Circle Command** to draw a circle of diameter 5mm, ensure that the circles centre lies upon the centre line.

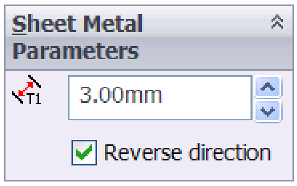


Using the **Trim Entities Command,** trim the inside entities of the circle as shown.

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

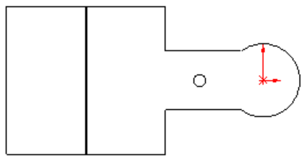
Confirm the Sketch by clicking on 

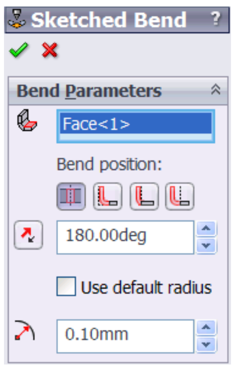
The Base Flange/ Tab Manager should appear. Keep all the default settings and set the thickness **T1** to **3mm.**



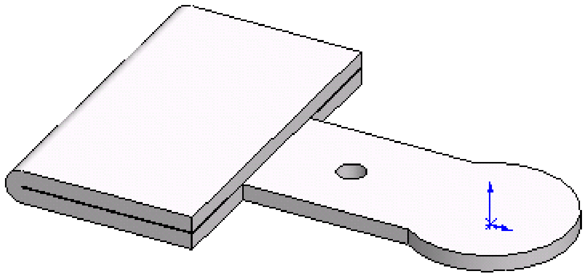
Accept the changes by Clicking **OK** 

Sketched Bend Select the **Sheet Metal** tab from the **Command Manager** click on the **Sketched Bend Command**, select the top face of the part. Select the **Line Command** from the **Sketch** toolbar. Hover your mouse cursor over either of the horizontal lines until a midpoint relation is display, select this as the start point, select the end point as the midpoint of the parallel edge.





Select the **Fixed Face** as any point to the right of the bend line. As the piece is bent back on itself, set the **angle** to **180deg**. **Deselect** the **Use default radius** option and set the radius as **0.1mm.**



Export the File

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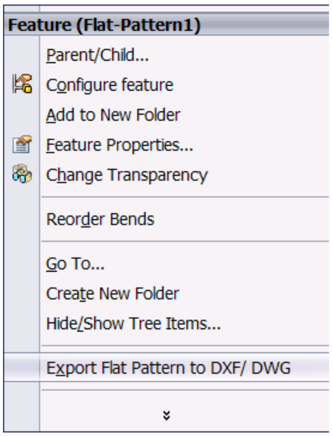
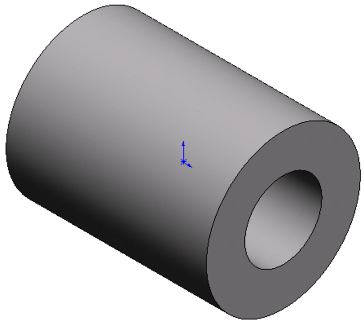
A Feature Dialogue box will appear, scroll down to the last option, **Export Flat Pattern to DXF/DWG**, and click on this.



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 **Lesson 2 Executive Toy** folder containing your part file. Click on **Save** button.

Part 3 – Sleeve



Starting out

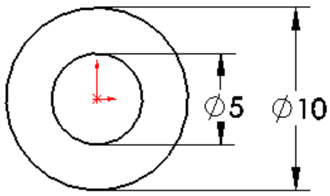
Open a New Part, and Save it as **Sleeve.SLDPRT** in the folder **Lesson 2 Executive Toy**

Sketching the Part

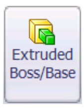
Select the **Sketch** tab from the **Command Manager**. Click on the **Sketch Command** 

.

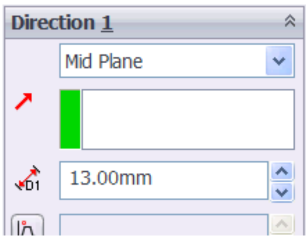
Select the **Right Plane**. Using the **Circle Command** sketch a circle that has its centre on the origin. Create a second circle inside this. Using the **Smart Dimension** command, add dimensions as shown.



Extrude

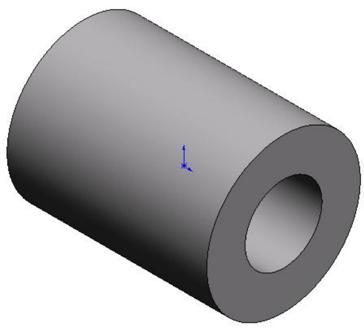
From the **Features** tab on the command manager, select the **Extrude Boss/ Base** command. 

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

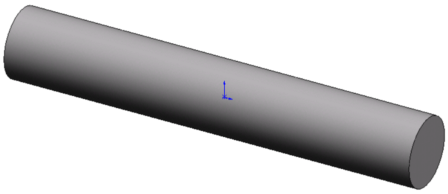


Set the **End Condition** to **Mid Plane**, and D1 to 13mm in the **Feature Manager** as shown opposite.

Choose **OK** 



Part 4 – Threaded Bar



Starting out

Open a New Part, and Save it as **Bar.SLDPRT** in the folder **Lesson 2 Executive Toy**

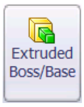
Sketching the Part

Select the **Sketch** tab from the **Command Manager**. Click on the **Sketch Command** 

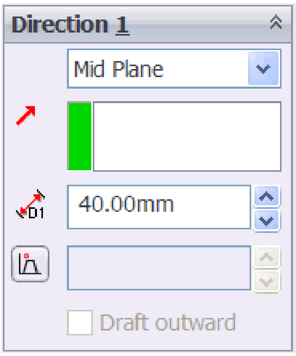
.

Select the **Right Plane**. Using the **Circle Command** sketch a circle that has its centre on the origin. Using the **Smart Dimension** command, dimension the circle diameter 5mm

Extrude

From the **Features** tab on the command manager, select the **Extrude Boss/ Base** command. 

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

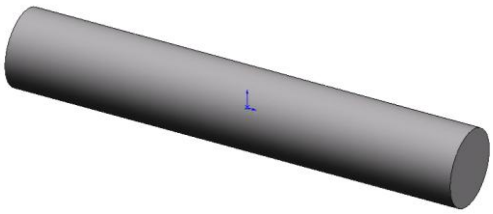


Set the **End Condition** to **Mid Plane**, and D1 to 40mm in the **Feature Manager** as shown opposite.

Accept the changes by selecting **OK** 

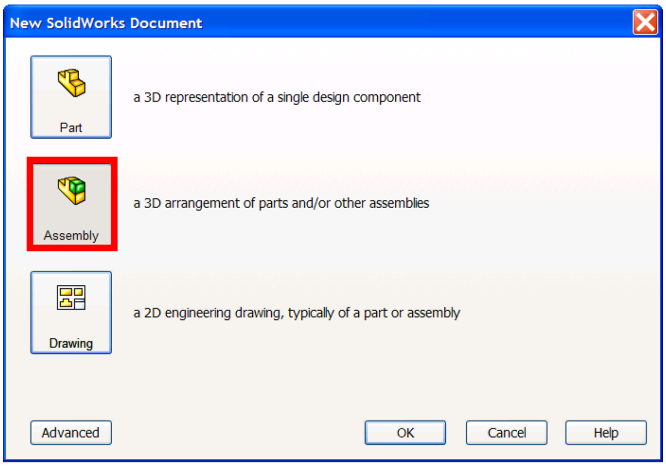
Save

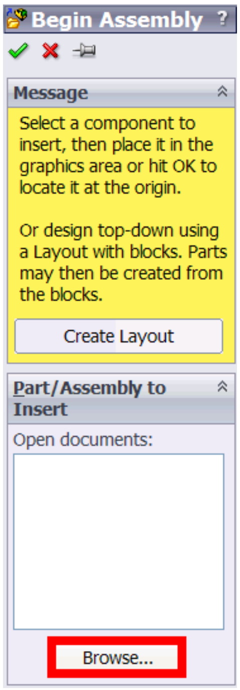
Save the changes.



Assembling Parts

To create an assembly of parts you must file open a new assembly file. This can be done by selecting **File** from the **Menu** toolbar, then select **New.** The **New SolidWorks Document Wizard** should appear, select **Assembly** (highlighted), then click on **OK**





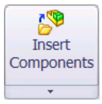
Save

Save the assembly file as **Exect Toy Assembly.SLDASM**

Select the Parts

Any part files already open will be displayed in the **Open documents,** if the part file you need does not appear in this box then select the browse button highlighted, and find the file manually. For this assembly the first file to bring in is **Exect Toy Base.SLDPRT**. Find the file and then select **OK** 

To insert the other components select the **Insert Components** command from the **Assembly** toolbar.

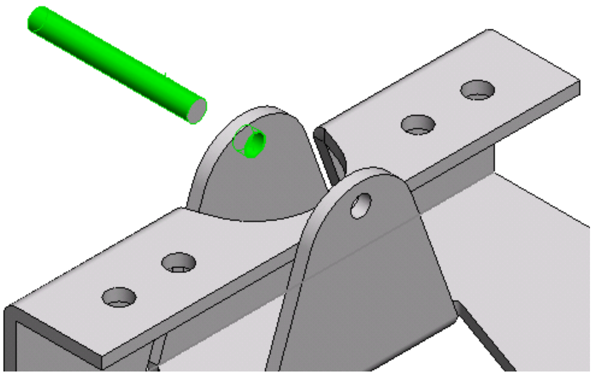


Insert **Bar.SLDPRT** now. Instead of clicking OK this time simply click anywhere in the graphics area to place the part in that area.

Mating

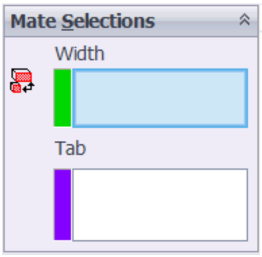
Select the **Mate** command 

from the **Assembly** toolbar. SolidWorks is now looking for the edges or faces you wish to mate. Select the inside of the hole on the Base and the outside perimeter of the cylinder as shown below.



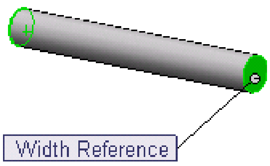
SolidWorks will identify these parts as both being circular in shape and will automatically assign a **Concentric** mate to these two parts. This can been seen in the pop up tool bar that appears, note the button for concentric mate is pressed. Select the green tick to accept this mate.

The next step is to select the **Advanced Mates** options. This can be done by clicking on the little downward facing arrows to the right of the advanced mates heading as shown. Here we want to create a width mate. The width mate will position the bar part so that it is equidistant to the both edges of the base part.

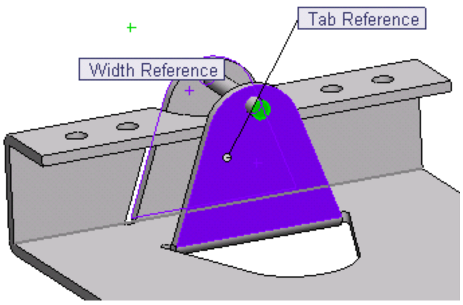


Select the **Width Mate** icon .

The **Width** selection box is automatically highlighted as shown. Click on the two end faces of the cylinder that is the **Bar** part as shown.



Now click into the **Tab** selection box, and select the two vertical outside faces of the Base part as shown below



This will push the bar through the hole.
Accept the changes by clicking on **OK**

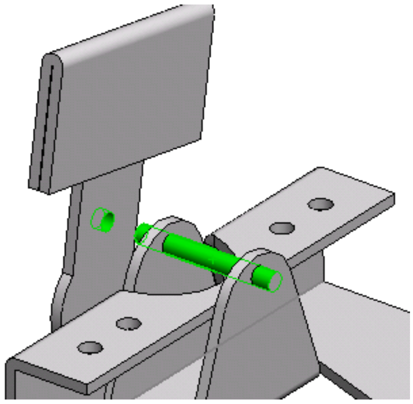
Insert Part

Now insert the part **Picture Holder.SLDPRT** to insert this part select the **Insert Components** command from the **Assembly** toolbar. Browse to where the part is saved and select it. A ghosted image of the part will appear attached to the cursor, click anywhere in the graphics area to place the part in that area.

Mate Parts

Select the **Mate** command 

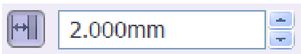
from the **Assembly** toolbar. Select the inside of the hole on the Picture Holder and the outside circular face of the Bar part as shown. Again SolidWorks will identify these parts as both being circular in shape and will automatically assign a **Concentric** mate to these two parts.



Accept the changes by clicking on **OK**

The next step is to select the **Advanced Mates** options. This can be done by clicking on the little downward facing arrows to the right of the advanced mates heading as shown. Here we want to create a distance mate. The distance mate will position the picture holder part so that it is a mated a certain distance from a part.

Select the **Distance Mate** icon. Select the inside vertical face of the Base and the outside face of the Picture Holder as shown. The box beside the distance box allows the user to specify the distance between the two select faces. Type in a distance of **2mm** as shown.



Accept the changes by clicking on **OK** .

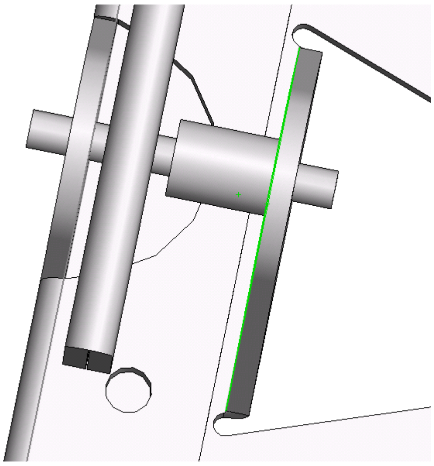
Insert Sleeve

Insert the part file Sleeve.SLDPRT into the assembly drawing.

Accept the changes by clicking on **OK** .

Apply a Concentric Mate between inside hole of the Sleeve and the outside of the Bar.

Apply a **Coincident Mate** between the flat face of the sleeve and the inside vertical face of the base part as shown.



Toolbox

Now we must add two domed cap nuts to the two end of the threaded bar. We can do this by using the toolbox feature of SolidWorks.

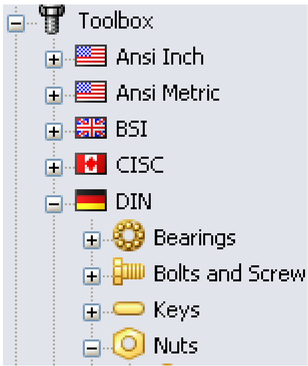
To access the toolbox feature select the **Design Library** icon  from the task pane on the right hand side of the graphics area.

Open the toolbox by clicking on the little + to the left of toolbox

Scroll down to the **DIN** folder, and expand this.

Scroll down to the **Nuts** folder and expand this.

Select **Hex Nuts Cap**



Click on the picture of the dome nut, and holding down the mouse button drag and drop it into the graphic area.

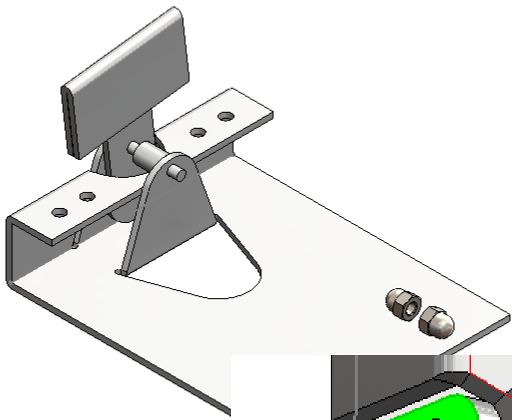
The configuration manager will appear select **M6** and select **OK** 

A ghost out domed nut will appear now, you can click anywhere in the graphics area to add another domed nut. Do this once and click on the red x to exit the command. 

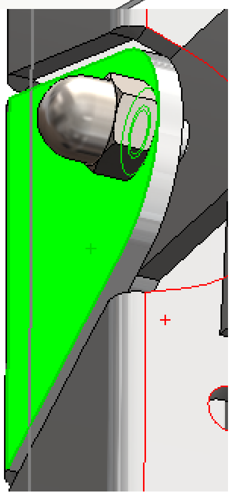
Apply Mates Select the **Mate** command 

from the **Assembly** toolbar. Select the circular edge of Bar part.

Now select the inside cylindrical face of the domed nut, SolidWorks will automatically apply a concentric mate to these two parts. Click **OK** 

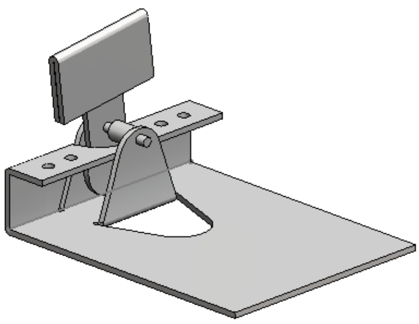


Now select the flat face of the bottom of the dome nut and the outside face of the base part. SolidWorks will automatically assign a coincident mate to these two parts. Click **OK** 



Repeat the process for the other side.

Save Save the changes to the assembly.

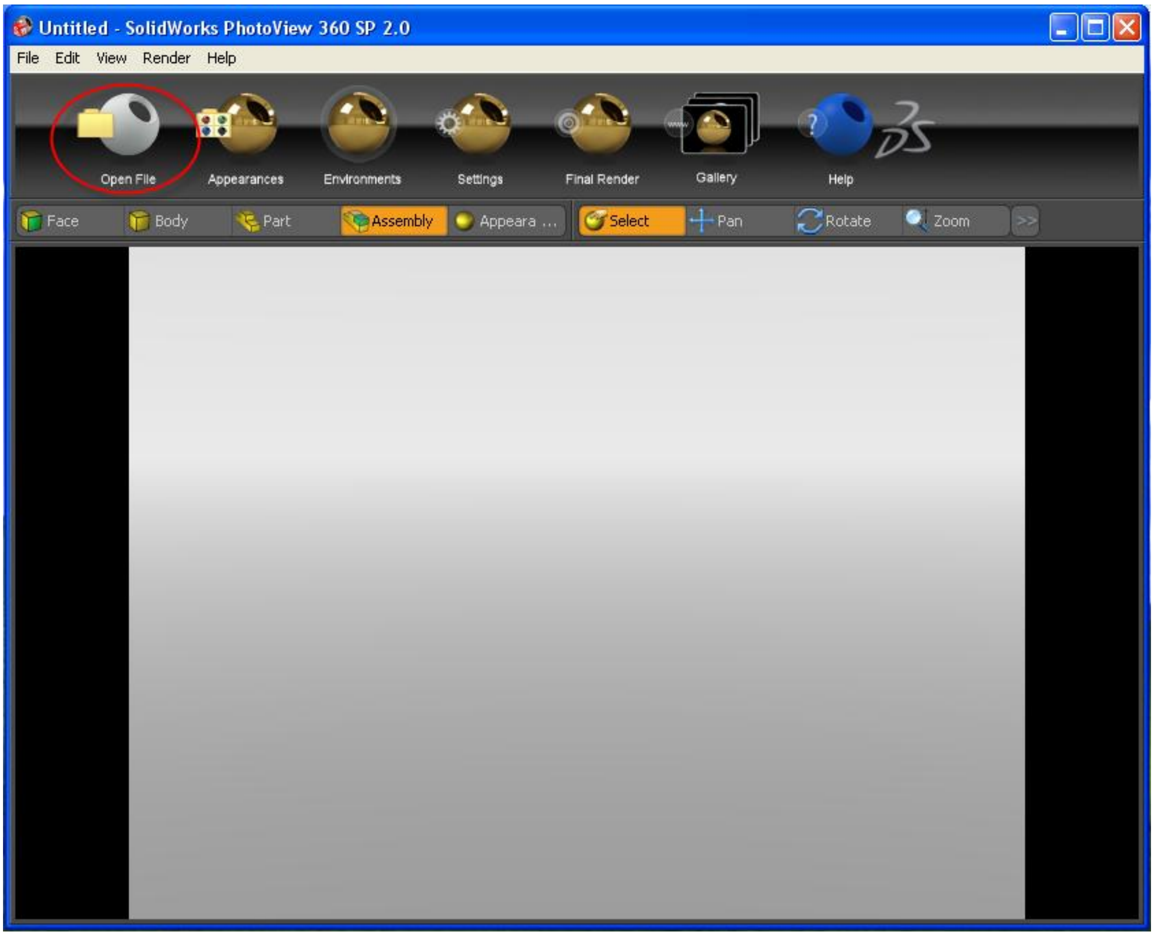


The assembly should look like this when finished.

Rendering in Photoview 360

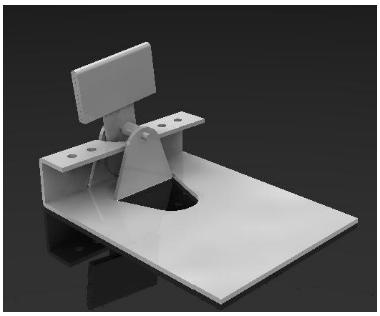
Open 360 Find the Photoview 360 icon 

on your desktop. Double click on it to launch the program.



Click on the Open File button on the top left corner of the graphics area (highlighted). Browse for the folder called **Lesson 2 – Executive To,** open it and open the file called **Exect Toy Assembled.SLDASM.**

The Assembly will be imported into Photoview 360 as shown.



Select Entity

In order to apply a colour, material, or texture you must first select what entities you wish to apply changes to. To do this select the Part button as highlighted, as in this exercise we are going to apply materials to each part.

Edit Appearance

Select the **Appearances** 

button

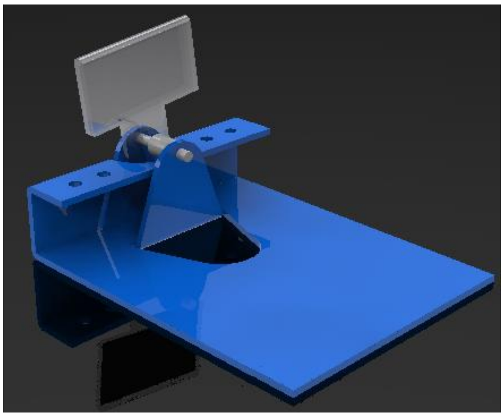
A list of materials will appear. These are headings, sub heading may be accessed by clicking on the small arrow to the left of the heading. This will reveal further choice and sub materials.

For this exercise select, **plastic**, and **high gloss**.

This will open a window displaying samples of the material. Left click on the material ***blue high gloss plastic***, holding down the left mouse button drag and drop onto the base of the assembled model. The base part should change from the default grey plastic colour to the ***blue high gloss plastic*** colour

Now close the materials palette, by clicking on the  icon.

The model should look like this.



Picture Holder

Select the **Appearances** button

Again a list of material will appear. The plastic sub menu should still be expanded from the previous edit. This time select **clear plastic**.

Select the polycarbonate option and drag and drop this on to the Picture Holder part. The part will still appear to be grey, however if you study it you can seen a greater reflection off the part now.

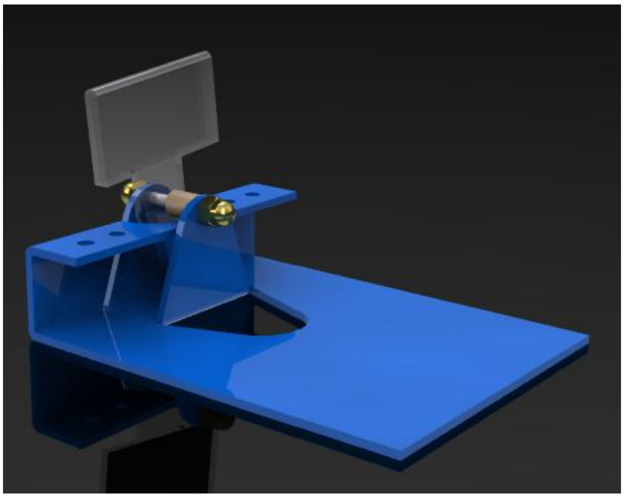
Again close the materials palette, by clicking on the icon.

Sleeve

Now we are going to apply a wooden material to the Sleeve part. Select the **Appearances** command, expand the **organic** menu, scroll down to the **wood** menu, expand this too, now select pine. Drag and drop the **unfinished 3d pine** material onto the sleeve part.

Dome Nuts From the appearances menu expand the **metal** menu. Scroll down to brass. Drag and drop the polished brass material onto the dome nut part. The second part will automatically update.

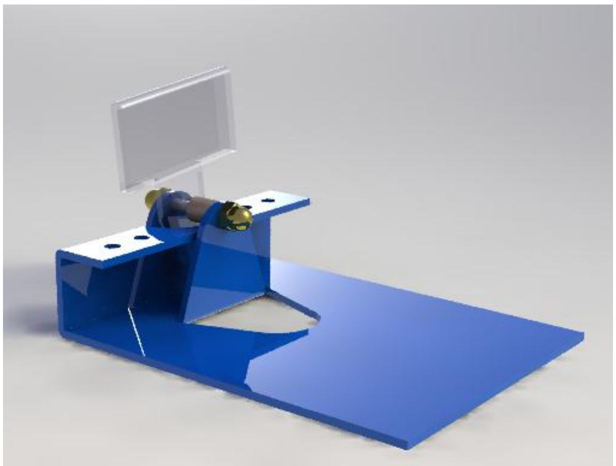
Your assembly should now look like this.



Select Environment It is possible to choose to render your model in front of a variety of different environments in Photoview 360. Select the **Environments** button**.**

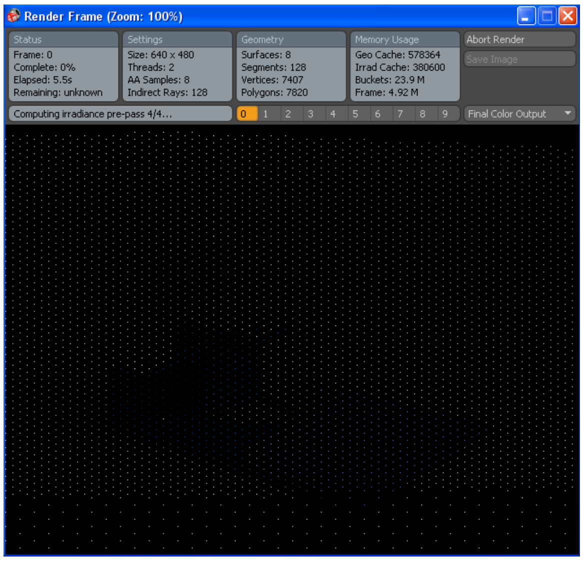
Scroll down and find Abstract Studio. Simply drag and drop it into the graphics area of Photoview 360 to change the environment.

The assembly should now look like this.



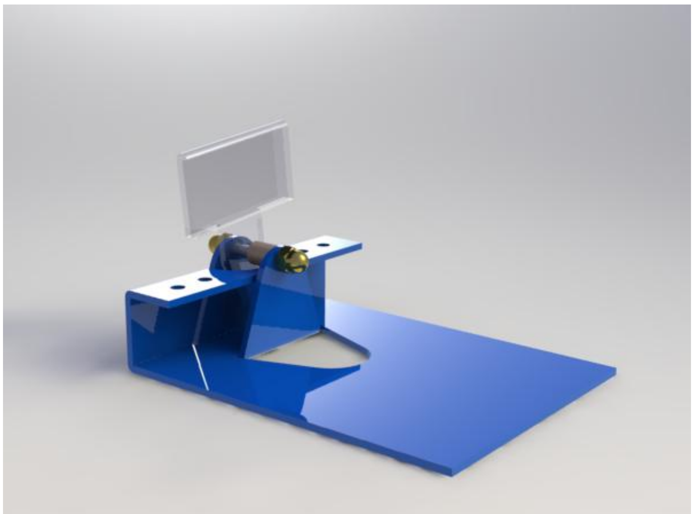
Final Render Select the Final Render button 

Photoview 360 will now begin to render the scene and the assembled project pixel by pixel. This may take some time, be patient!



When the render is finished, click on the Save as button in the top right of the screen 

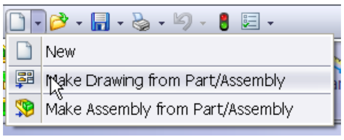
. This allows the rendered image to be saved as a JPEG that can be manipulated later, or can be included in a project report.



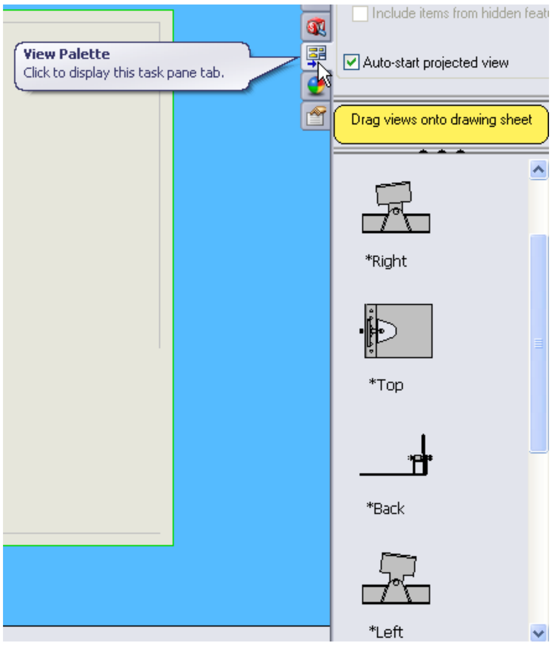
Final Rendered Image!!

Open Assembly Reopen the Assembly file **Exect Toy Assembly.SLDASM.**

Create Drawing Select **New, Make Drawing from Part/Assembly,** from the standard toolbar.



Choose the **DCG A3L** template from the available list. The Drawing document is opened with the DCG A3L template displayed. The view palette, located on the task pane may be used to insert drawings views.

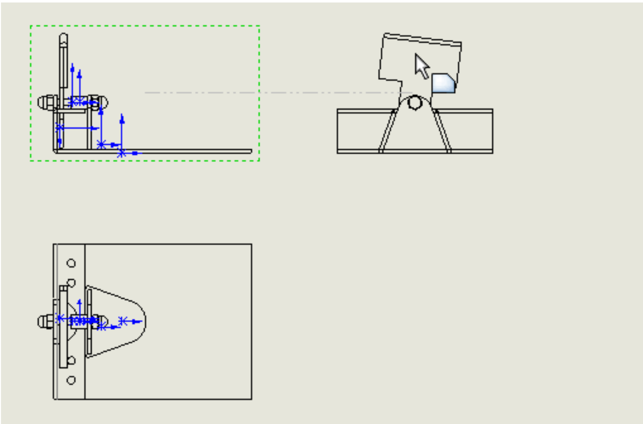


NB: The **View Palette** tab provides the ability to drag and drop drawing views into an active sheet. The View Palette contains images of standard views, annotation views, section views.

Importing drawing views.

Note: Ensure **Auto-start projected view** is ticked. This will allow us to project other views from the parent view.

Click and drag the **front view** from the view palette onto the drawing sheet. By moving the cursor to the right an **end view** is projected from the parent view. Click to add the view.



Project a plan view from the parent front view also.

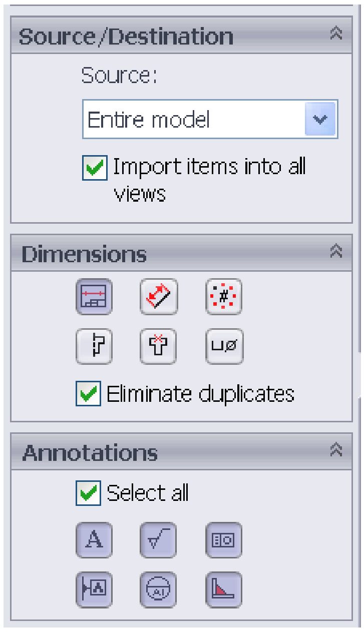
Press Esc to stop **Auto-start projected view**

Drag an **isometric view** from the view palette.

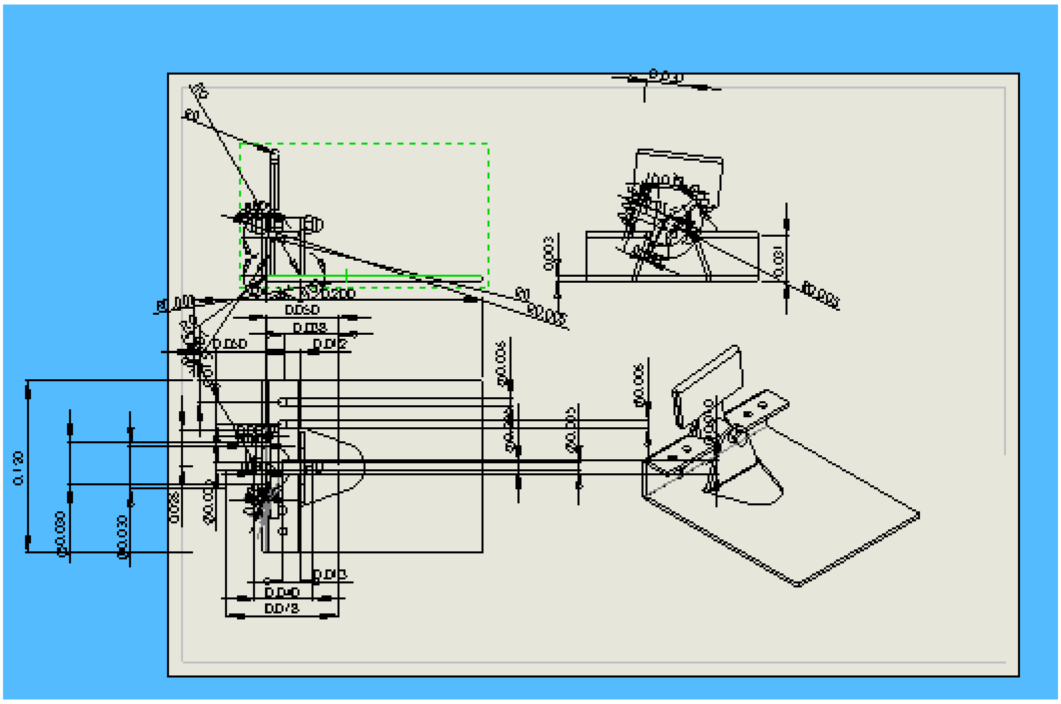
Dimension

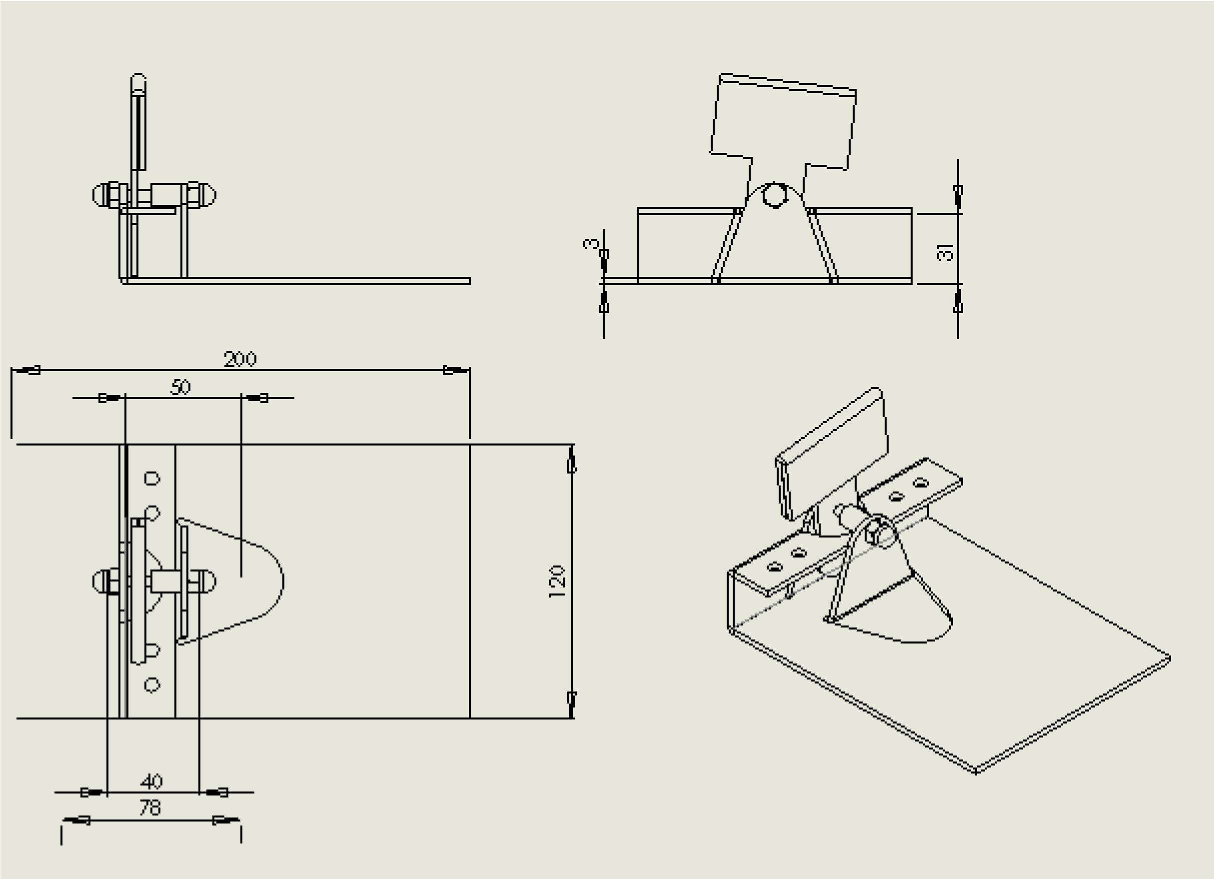
To add dimensions select **Model Items** from the **Annotations** tab of the command manager

Choose the settings as shown across. Select **OK** 



The resulting dimensions can be very messy as below. To delete any items just click on the item and press delete on the keyboard.





Finished Drawing