PDST

Professional Development Service for Teachers | An tSeirbhís um Fhorbairt Ghairmiúil do Mhúinteoirí

Advanced SolidWorks

Study Lamp

- Surfacing
- Advanced Mates and Patterns
- Advanced Features
- In-Context Modelling

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Introduction & Learning Intentions

Introduction:

This workshop aims to upskill and develop teachers understanding of advanced modelling techniques in SolidWorks 2018. The workshop will further interrogate Surface and Solid Modelling techniques while also examining features such as **Indent**, **Deform**, **Freeform**, **Combine**, **Curve driven Pattern, Fill Pattern, In-Context modelling** and **Advanced Mates.**

Learning Intentions:

At the end of this workshop it is intended you will be able to:

- Explore a modelling technique using planar and non-planar geometry with zero thickness.
- Understand and apply some 'surfacing' features in a SolidWorks design model.
- Explore the use of features including Indent**,** Deform**,** Freeform**,** Combine**,** Curve driven Pattern, Fill Pattern, In-Context modelling and Advanced Mates.

- Surface Modelling can be used to model complex designs within SolidWorks.

- The transition from zero thickness surface geometry to solid objects is a necessary and fluid transition in advanced CAD modelling.

- The appropriate use of advanced commands such as those outlined above, will significantly enhance realistic parametric modelling.

Study Lamp

Base

Open a new part from the SolidWorks Documents dialogue box.

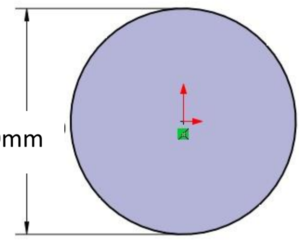
Select **File**. Click **Save as** on the standard toolbar. Save as **Base** in the **Study Lamp folder**.

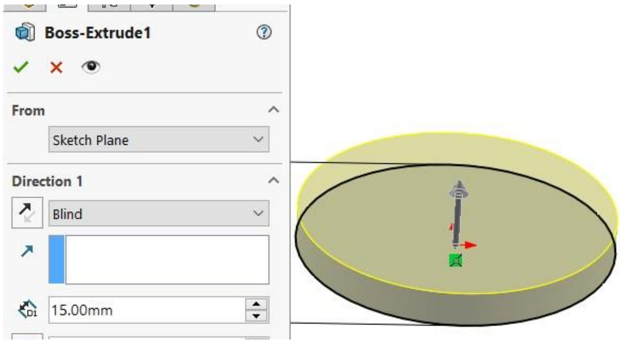
Continue to save periodically throughout the exercise.

Create sketch

Select the **Top Plane** in the design tree and draw a circle having **180mm** diameter.

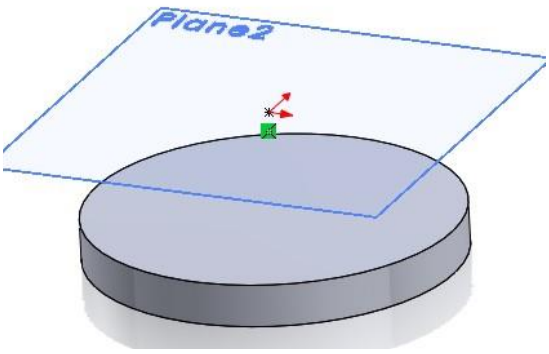
Extrude by **15mm.**



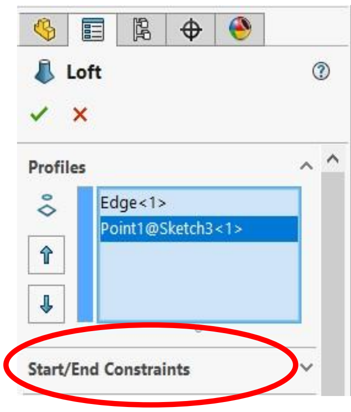


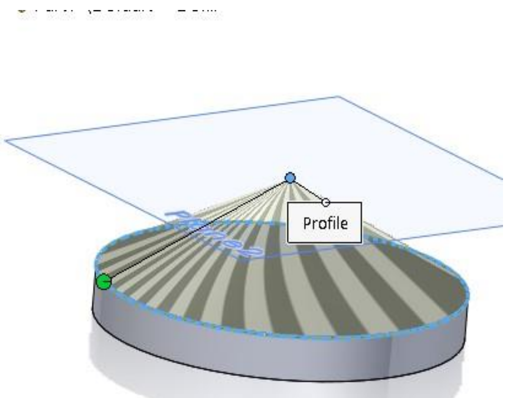
On a **plane 50mm** above the Top Plane, sketch the point as shown.

In the Features commands select **Loft Boss/Base** and select the edge of the circle and the point.



On the left hand side expand the start/end constraints.

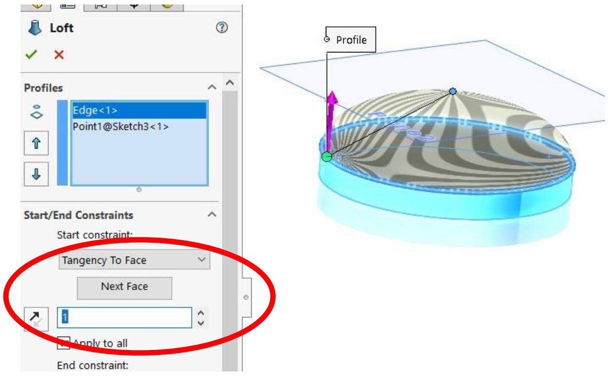


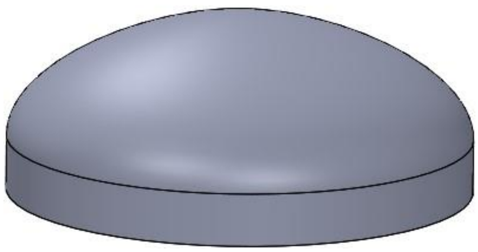


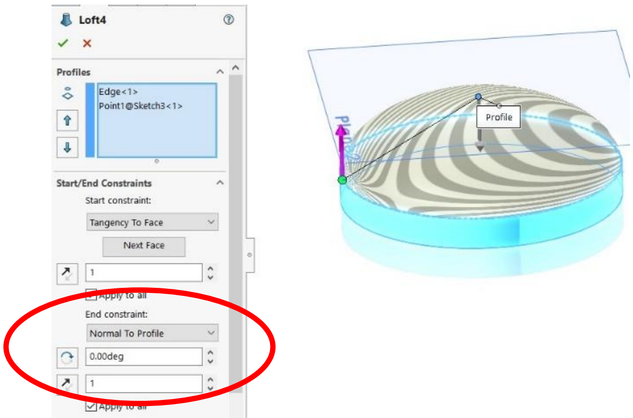
In the window select **edge** as shown and select **Tangency To Face** as the start constraint.

Select the **point** in the window and select the **Normal To Profile** as the end constraint.

Note the arrow direction. Set the height of each arrow to 1.





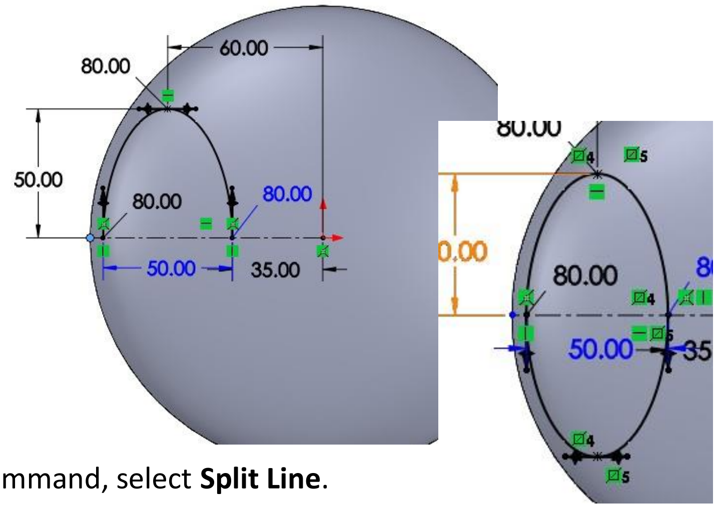


Detailing

On the **Top Plane** draw the sketch shown using the **Spline** command.

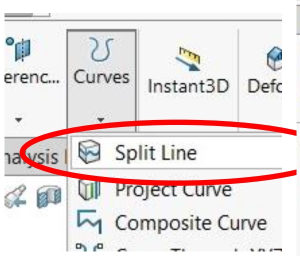
Add the dimensions.

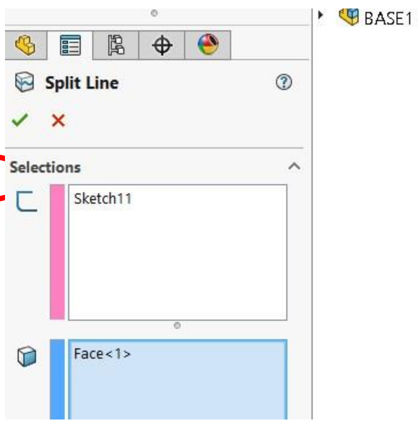
Mirror about the centreline.

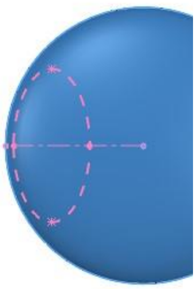


In the **Features** tab, under the Curve command, select **Split Line**.

On the screen select the face to split.

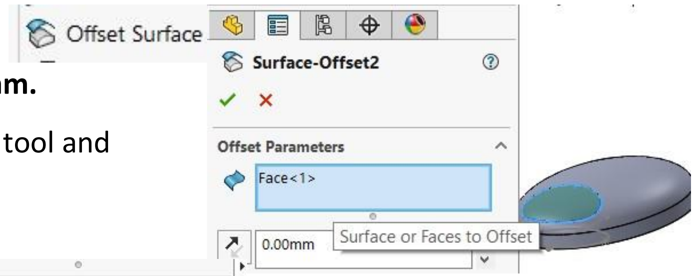






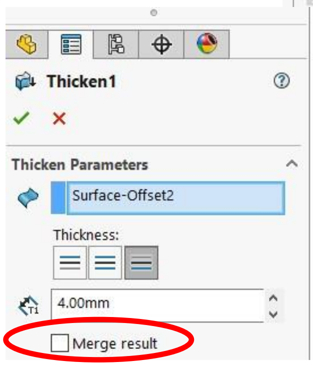
In the surfaces tab select **Surface Offset.**

Select the face as shown and offset by **0mm.**



Still in the surfaces tab select the **Thicken** tool and thicken by **4mm.**

Untick the Merge result box.



Select the **Indent** command.

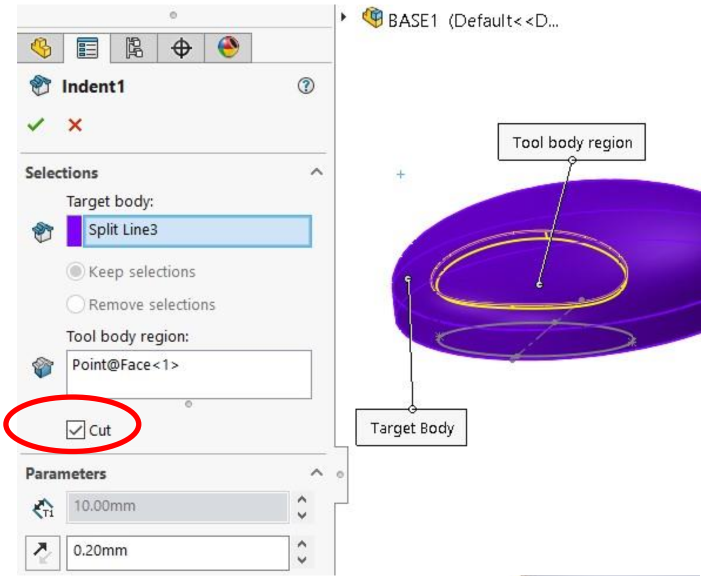
Select the part as the target body.

Select the surfact as the tool body.

Set the offset distance to 0.2mm.

Make sure to tick the **Cut** box.

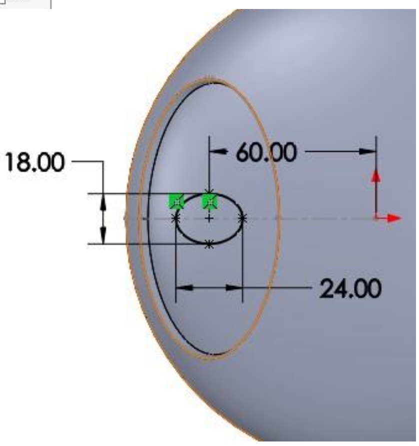
Hide the Body.

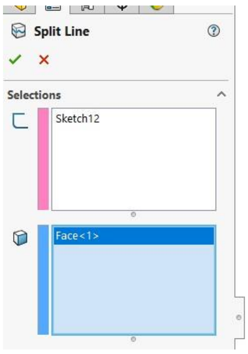


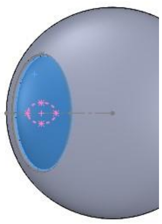
On/Off Button

On the **Top Plane** draw the ellipse to the given dimensions.

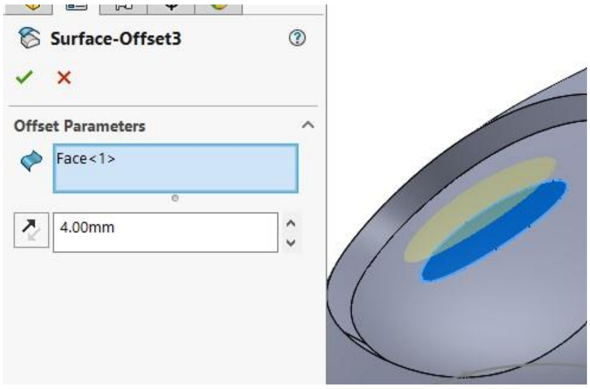
Select **Split Line** and select the sketch and the face to split.





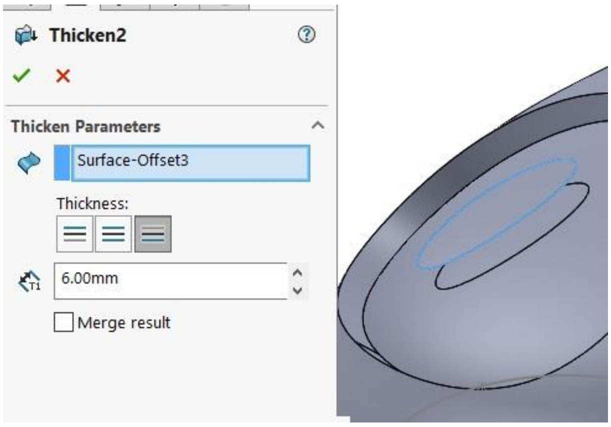


In the surfaces tab select **Surface Offset** and offset the ellipse by **4mm**.



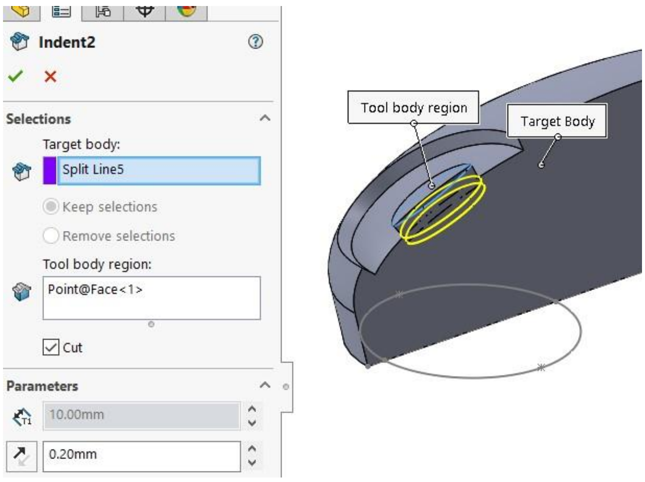
Select the **Thicken** command.

Thicken by **6mm** and **untick Merge results**.



Select the **Indent** command . Select the target body and tool body as shown.

Set the **clearance to 0.2mm**.



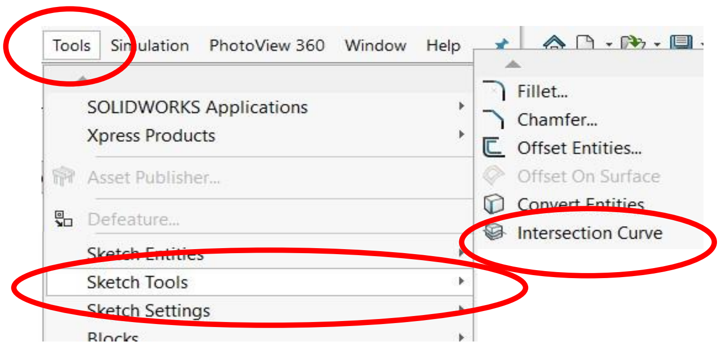
The button now has a clearance of 0.2mm around it and sits into the base part to a depth of 2mm.

On the **Front Plane** create a sketch.

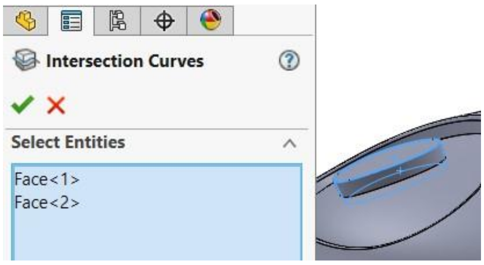
In the tools tab select **Sketct Tools** and then select **Intersection Curve**.

In the drawing area select the top and side of the button.

Accept.

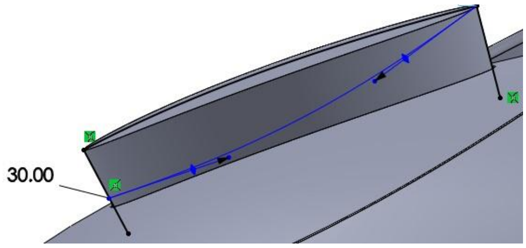


The line of intersection between the plane and the button in now shown.



Select the **Spline** command and draw the spline as shown.

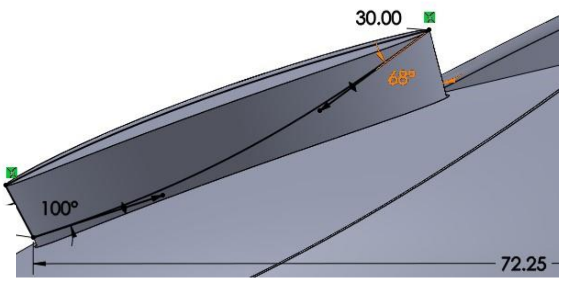
Add the following dimensions to the spline handles.

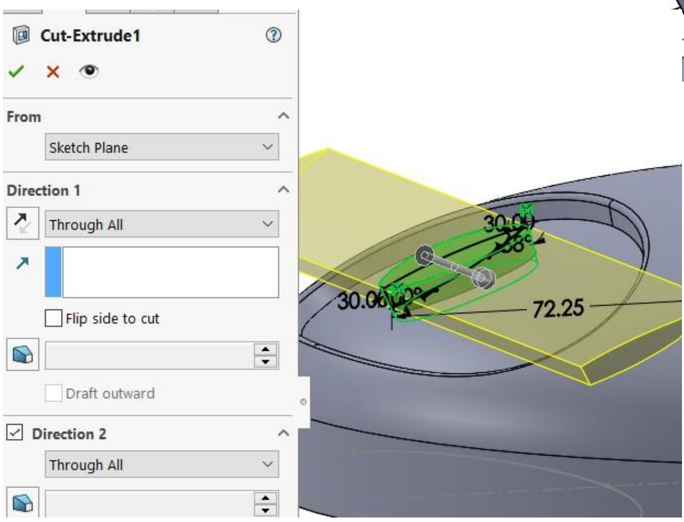


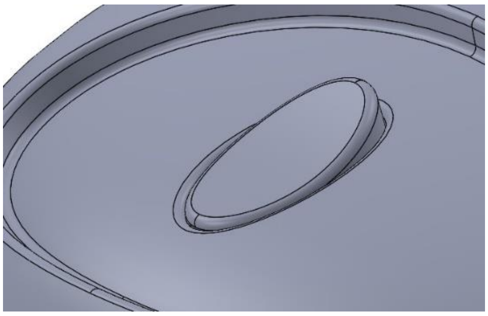
Use the **Trim** command to trim off the excess.

Accept the sketch.

In the features tab select the **Extrude Cut** command to remove the top portion of the button.



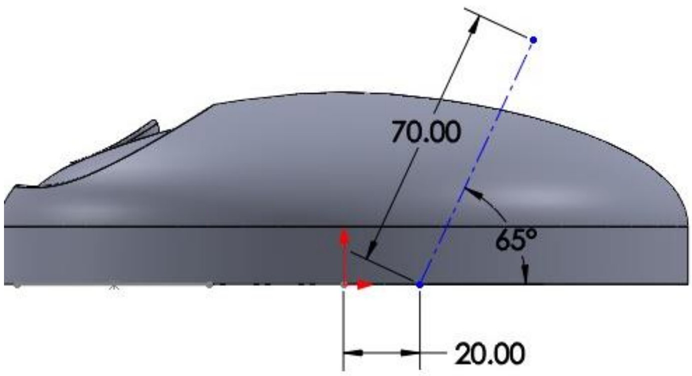




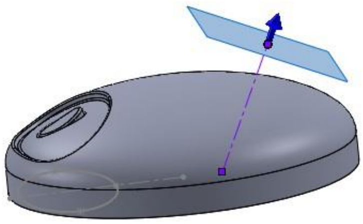
Add a **1mm Fillet** to complete the shape of the button.

Connection to Lamp Arm

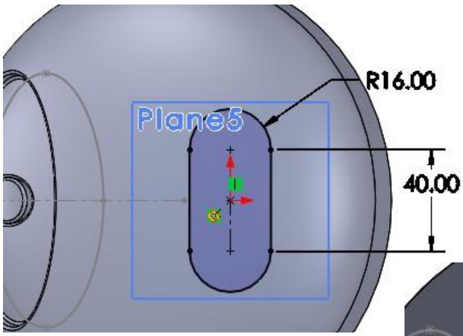
On the **Front plane** draw a Centreline to the following dimensions.

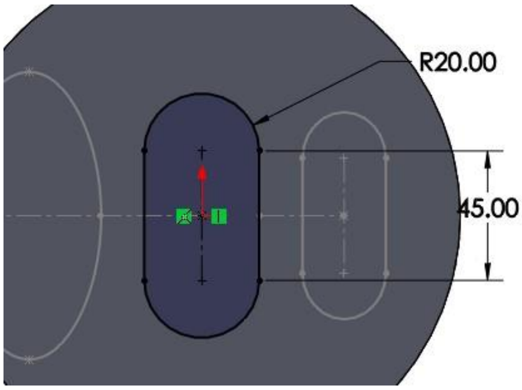


Create a plane perpendicular to this line going through the top point by selecting the line, then the point.



Draw the sketch shown on this plane.

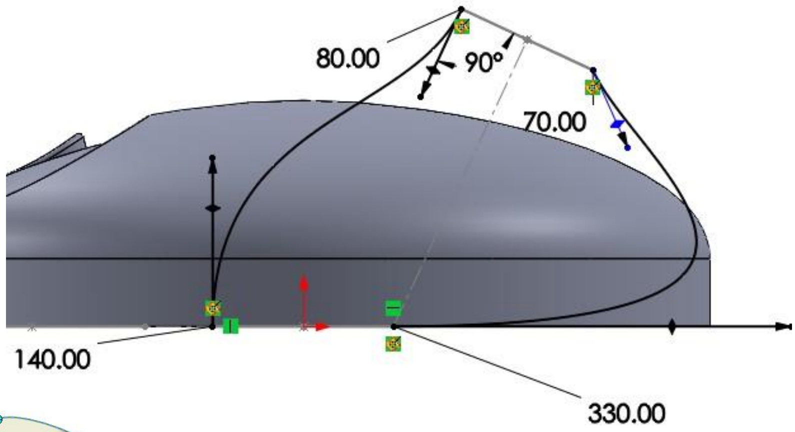


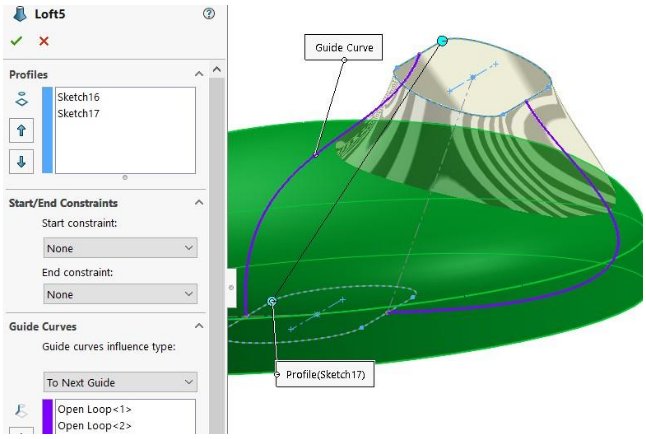


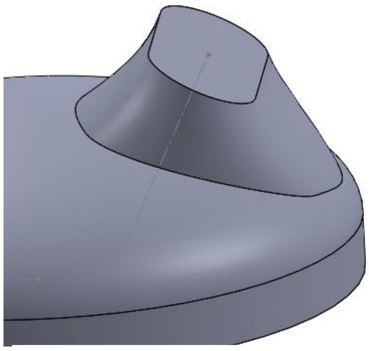
Draw the sketch shown on the **Top Plane**.

On the **Front Plane** draw the **Splines** shown to the given dimensions. Make sure to add the pierce relations.

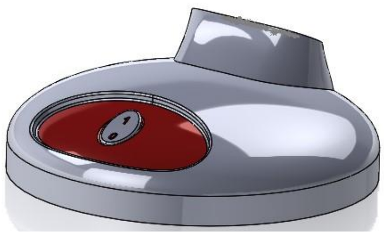
In the features tab select **Loft Boss/Base** to draw the feature.

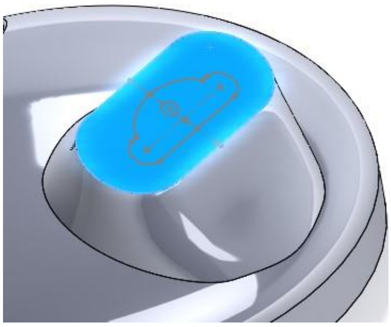


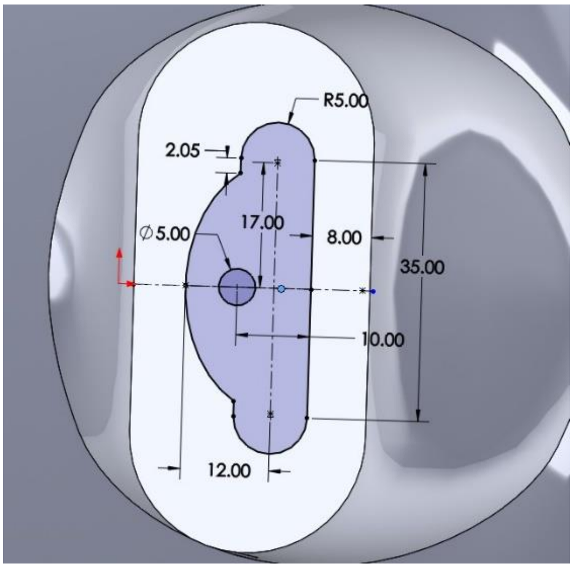




On the surface show, draw a sketch using the **Slot**, **Circle,** and **Ellipse** commands. Use the **Trim** tool and add the following dimensions.

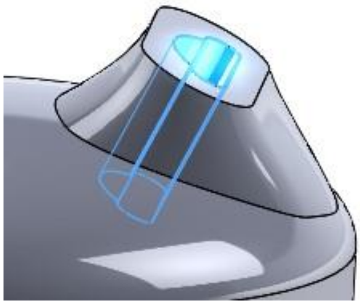






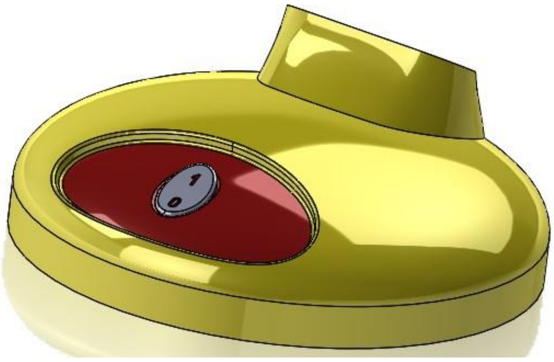
Rename the sketch as **Arm Profile.**

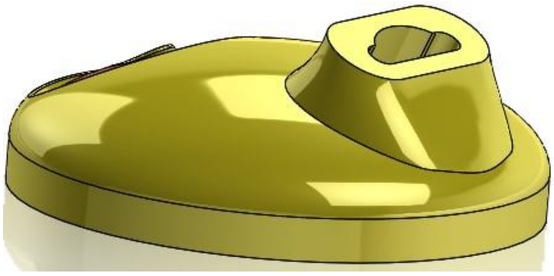
Extrude Cut by **40mm.**



Rename the extrude as **Recess for Arm.**

Apply a **High Gloss Plastic** Appearance to the part.



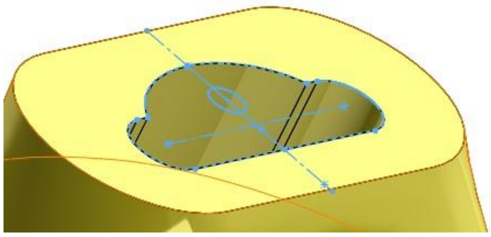


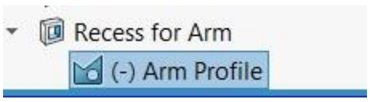
Save the part.

Study Lamp Arm

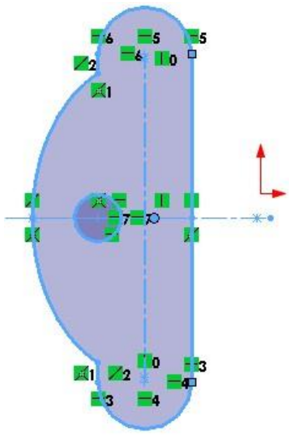
Select the **Top Plane** to draw a sketch. **Open the Base Part also**.

In the design tree for the Base Part, select the Arm Profile sketch. When it is highlighted in blue press the **Control** + **C** on the keyboard.



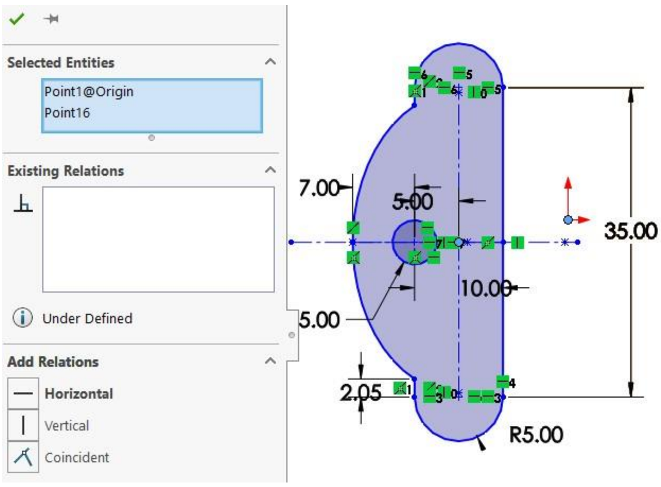


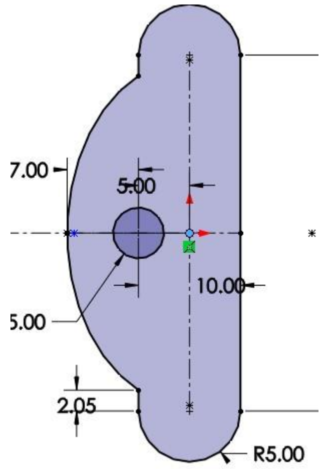
Go into the new Arm part and press **Control + V**



Select the dimensions command, and re-enter the dimensions.

Select the midpoint shown, and the origin, and add a coincident relation between them.



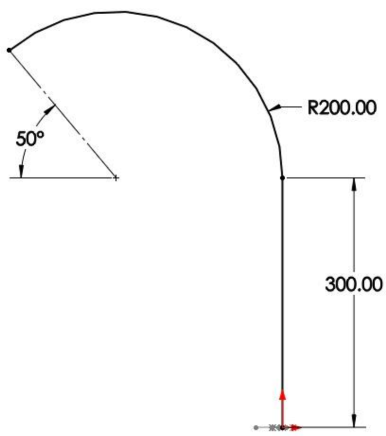


Rename the sketch as "**Arm Profile**"

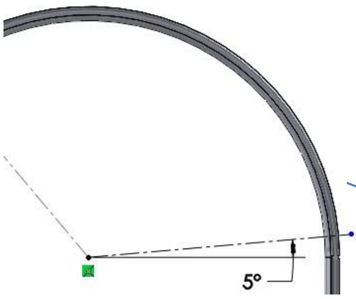
On the **Front Plane** draw the sketch to the following dimensions.

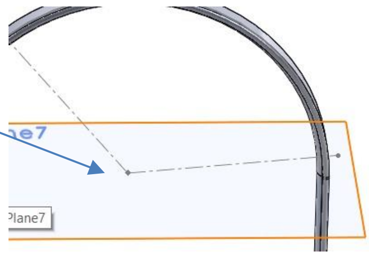
Rename the sketch as "**Arm direction**"

Using **Sweep Boss/Base**, create the feature.



On the **Front Plane** draw a **centreline** from the centre point of the arc as shown.

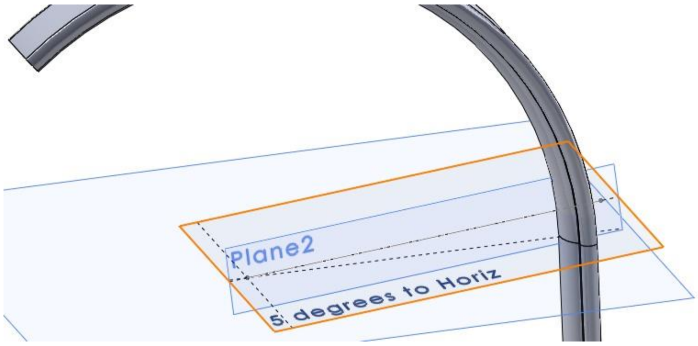




Draw a new plane parallel to the Top Plane at the Centre point.

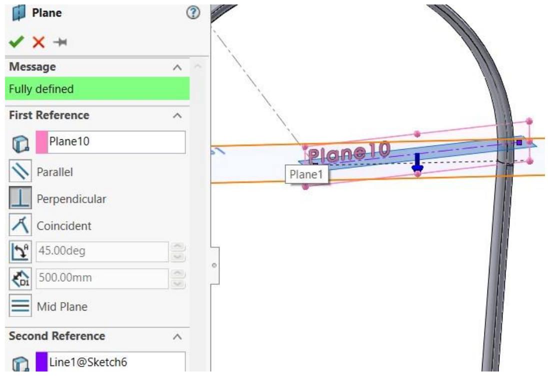
Create a plane that is inclined to this plane and also goes through the centreline.

Rename this plane as “**5 degrees to Horiz”.**



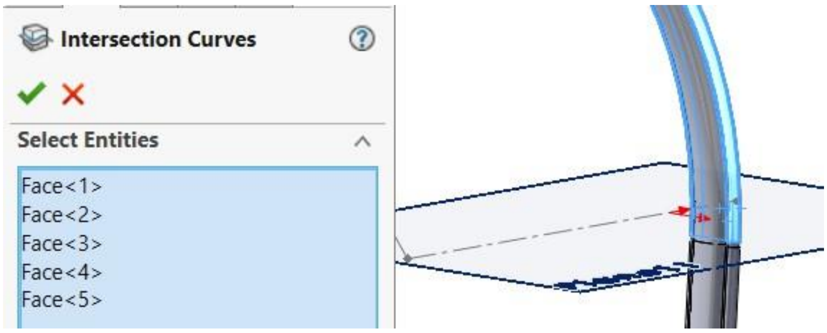
On this inclined plane (“5 degrees to Horiz”) select the sketch command.

In the **Tools** tab select **Sketch Tools** then **Intersection curve.**

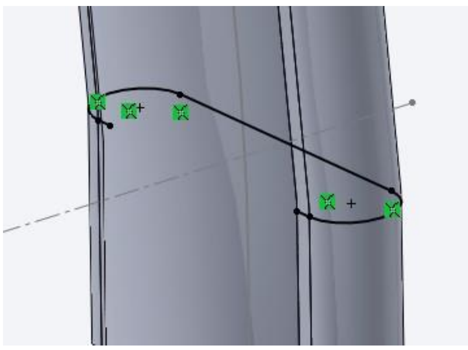


Select all the faces but not the inside curved face.

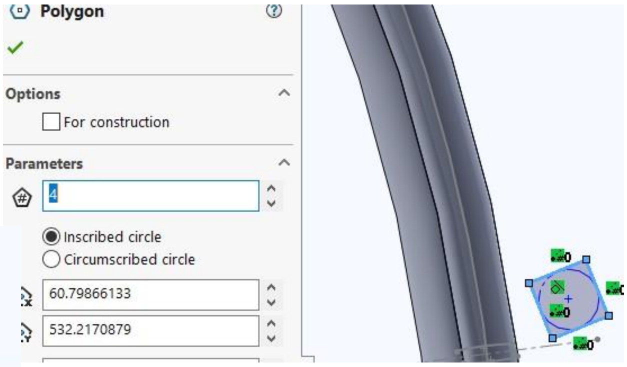
The intersection line between the plane and the feature is now shown.

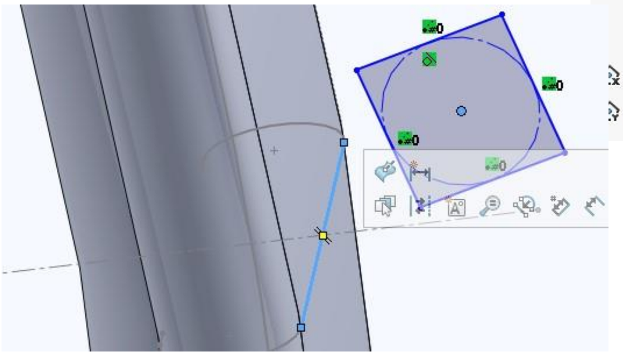


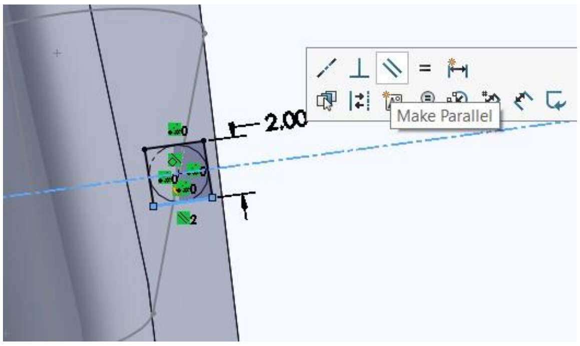
On the **Front Plane**, draw a square using the polygon command.



Add a **Pierce Relation** between the centre point of the square and the line shown.



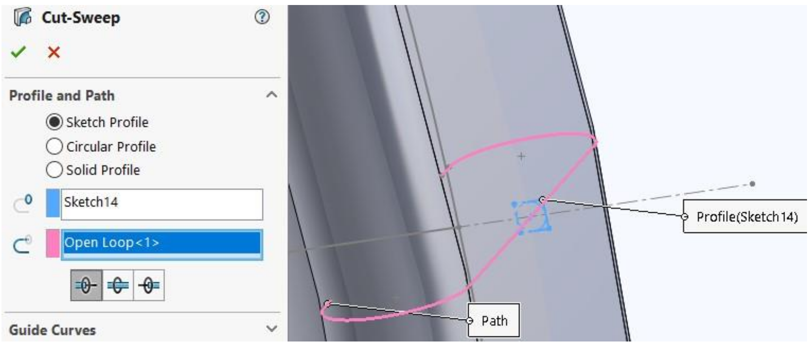


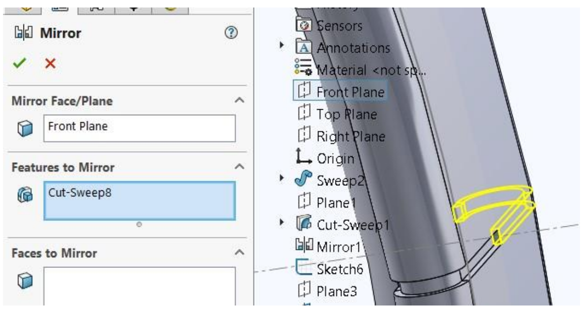


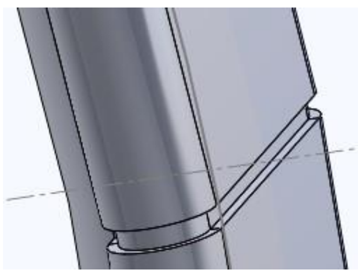
Add a **Parallel Relation** between the bottom of the square and the centreline.

Add a dimension of **2mm** to the side of the square.

Use **Sweep Cut** to remove the material as shown and then **Mirror** about the front plane.







In the Features tab select **Curve Driven Pattern** as shown.

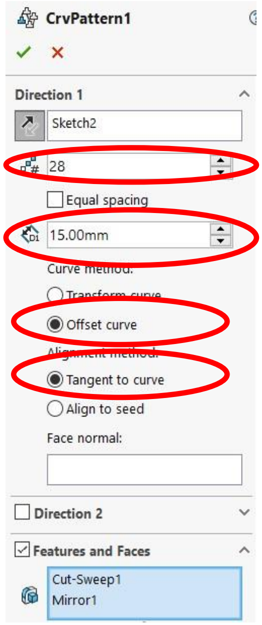
In the direction1 box, select the "**Arc direction**" sketch.

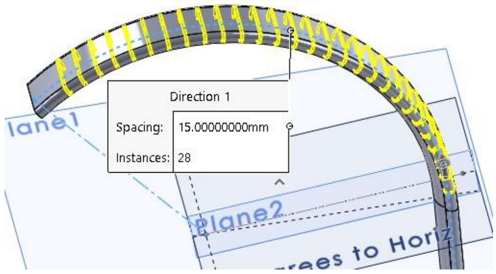
Select the features to pattern from the design tree.

Make sure **offset curve** and **tangent to curve** are selected.

Input **28** for the number of instances.

Add a spacing of **15mm**.





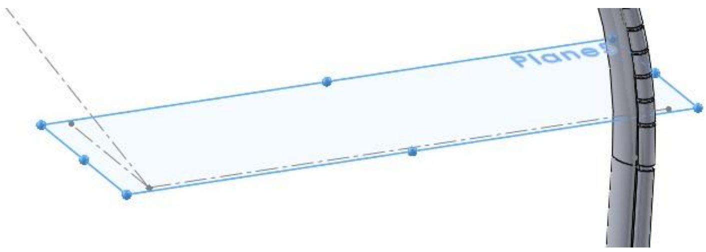
Accept.

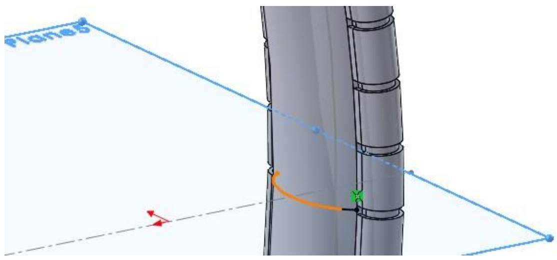
Use of Freeform tool

Select plane "**5 degrees to Horiz**".

Find the line of intersection between the underside of the feature and the plane.

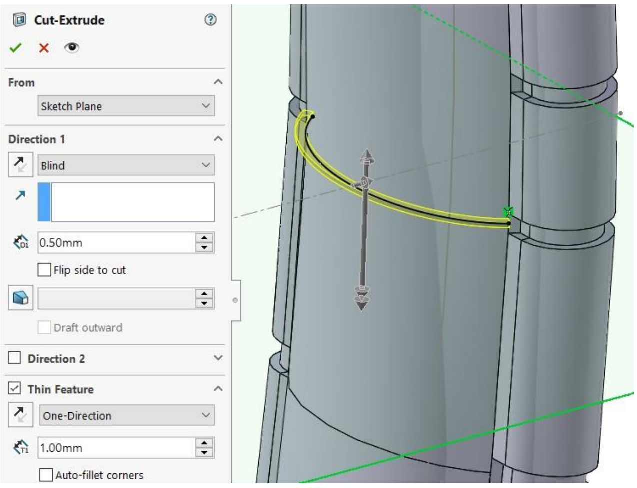
To do this select **Tools**, **Sketch Tools**, **Intersection Curve**.





In the features commands select **Extrude Cut** and cut in direction 1 **by 0.5mm.** De-select direction 2 and tick the thin feature box.

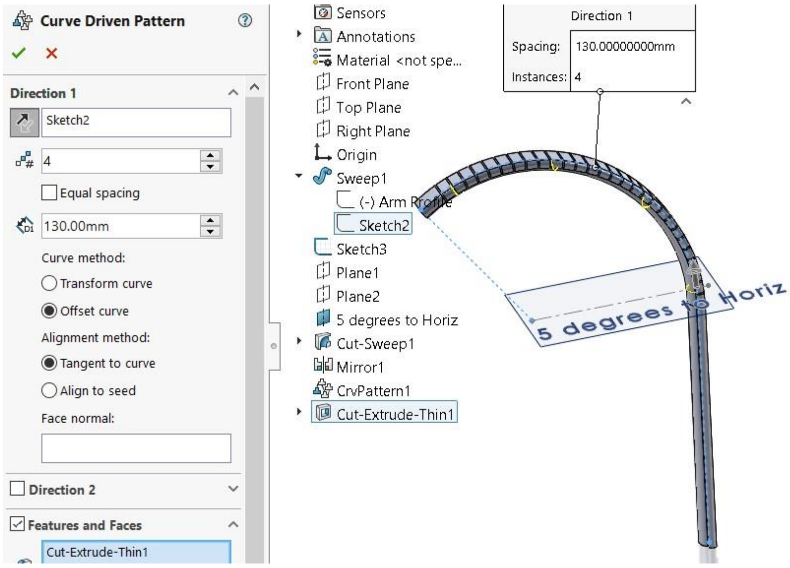
Accept.



Select **Curve Driven Pattern** in the features commands, and add the additional features as shown.

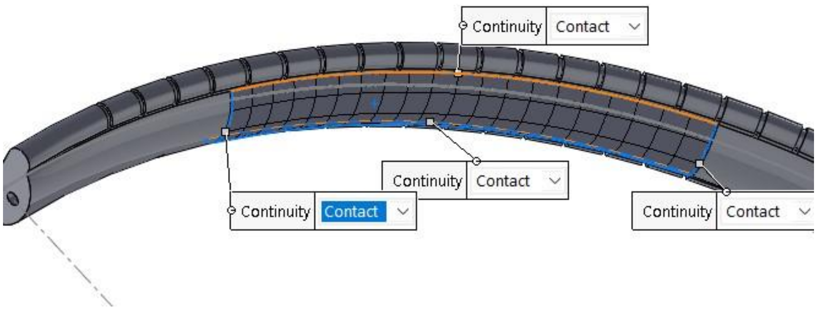
The underside surface is now divided into four sections.

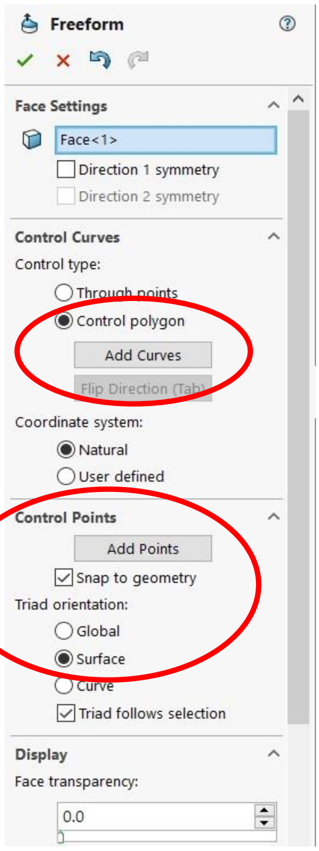
To create the bulge effect, use the freeform feature on each section.



Select **Freeform** from the features commands.

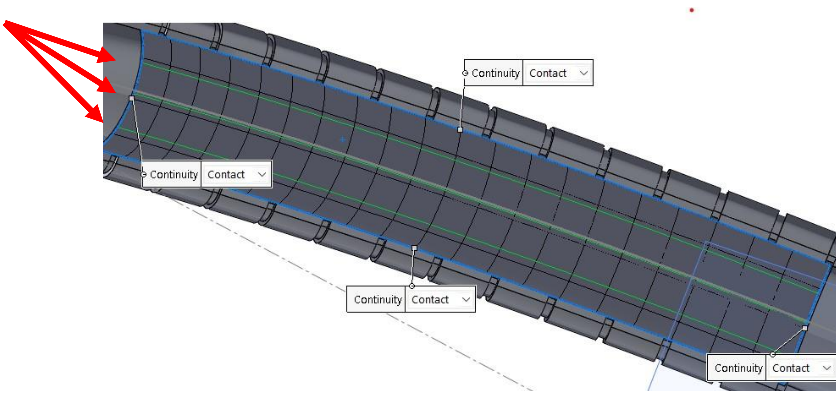
Select the face onto which to apply this feature.





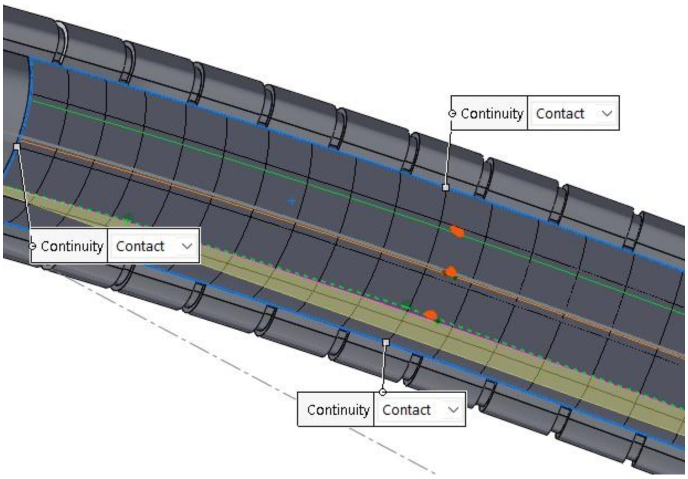
On the left hand side select **Control polygon** and select the **Add Curves** button.

Move the cursor onto the surface, and click on the area to add the curves. They will appear as green.



Activate the **Add Points** tab, and add points roughly in the centre of these curves.

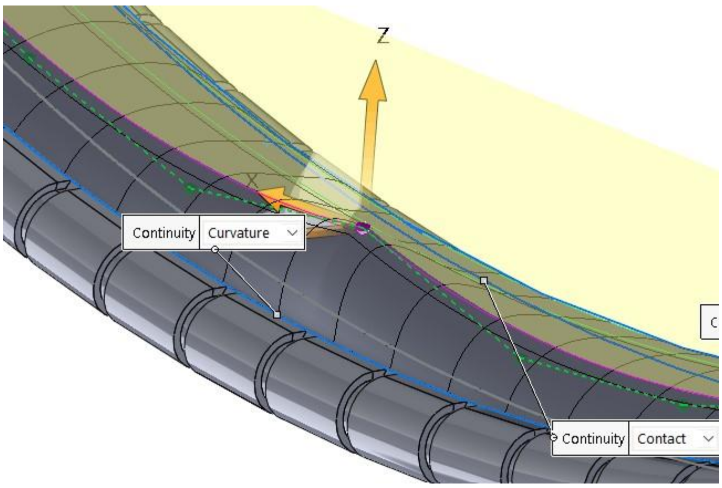
Click on the **Add Points** tab again to deselect this command.

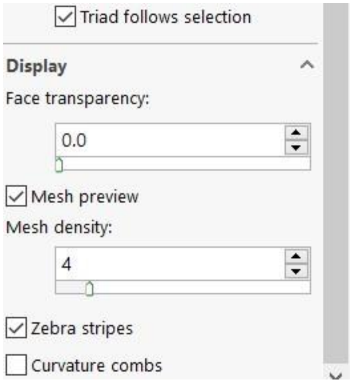


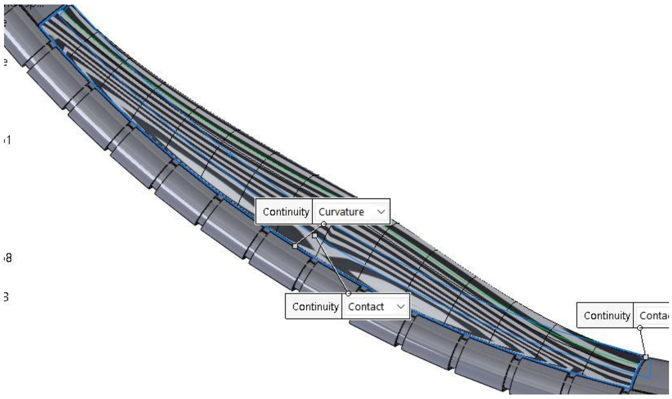
Double click on a point, and the triad will appear. Drag the Z arrow to create the deform.

Drag the points on each curve for the best effect.

Select the Zebra stripes to help to create a smooth curve



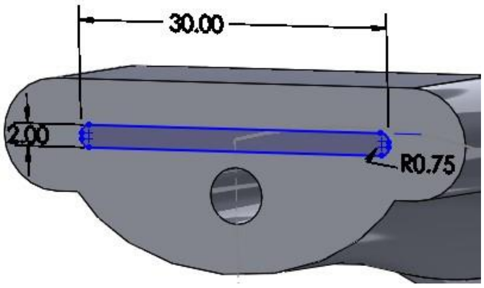




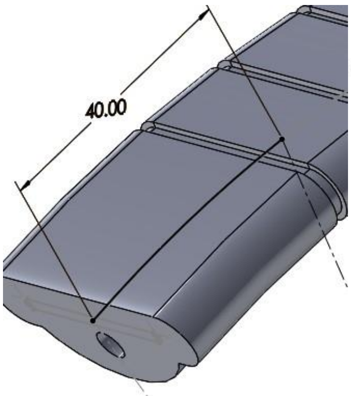
Accept.

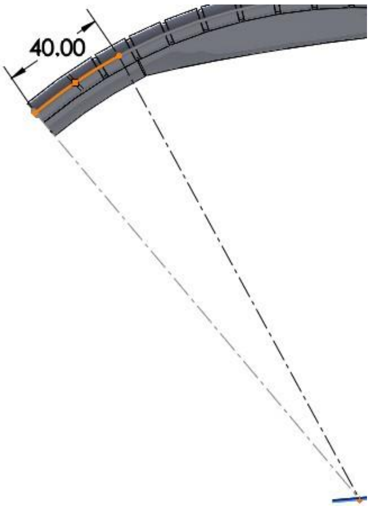
Repeat the process for the other faces on the underside of the Arm part.

On the face shown, draw the sketch to the following dimensions.



On the front plane draw the sketch shown.

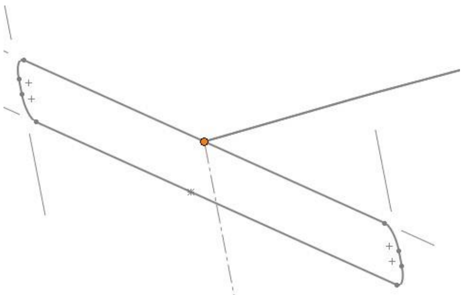




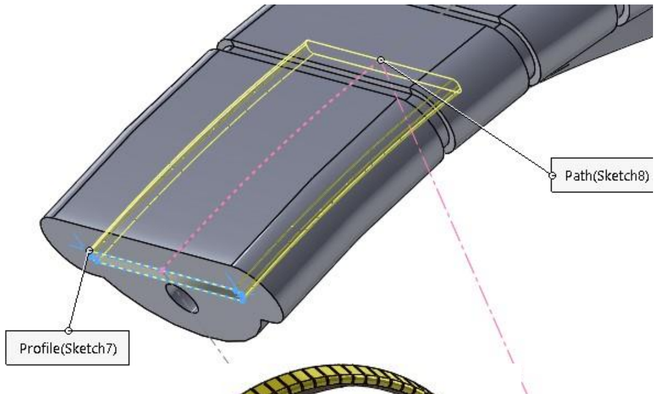
Use **Centerpoint Arc** to draw the segment shown. The CenterPoint being the same centre point as the "*Arm Direction*" sketch.

Add a **Pierce Relation** between the top midpoint of the sketch and the arc.

Select **Sweep Cut** to remove the material.

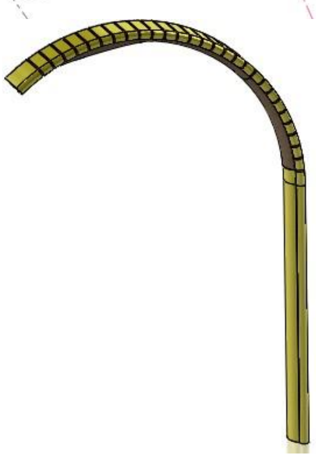


Accept.

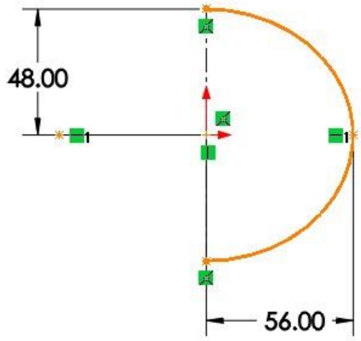


Apply a **High Gloss** Material to the Part.

Save.



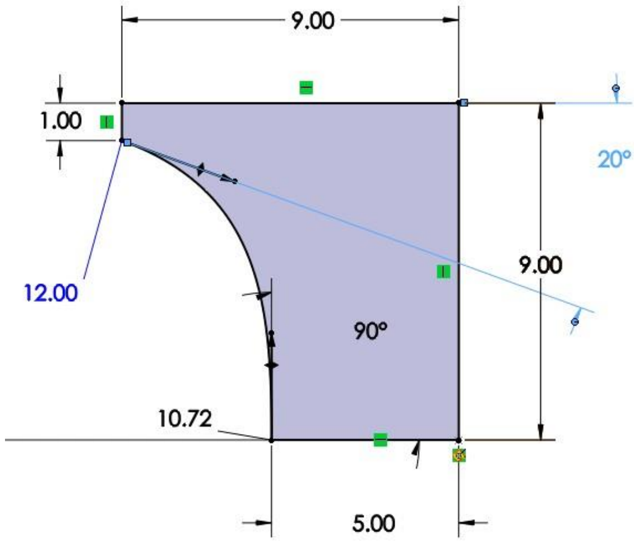
Hood Support



Open a new part from the SolidWorks Documents dialogue box.

On the **Top plane** draw the following sketch using the **Ellipse** command. **Trim** the excess.

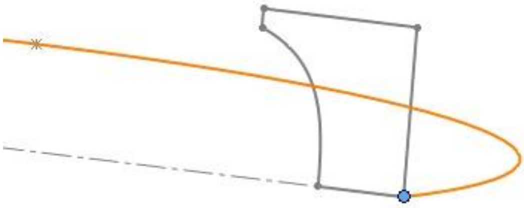
On the **Right Plane** draw the following sketch using the **Line** and **Spline** commands.

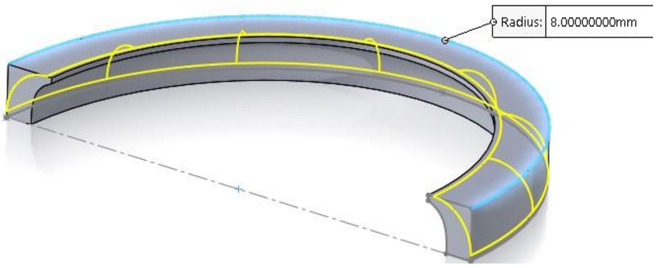


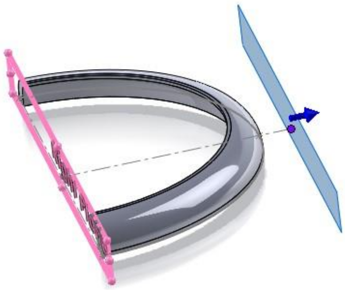
Add a **Pierce Relation** between the arc and the outside corner of the profile as shown.

In the features commands use the **Sweep** command to create the profile.

Add an **8mm Fillet** to the outside edge.

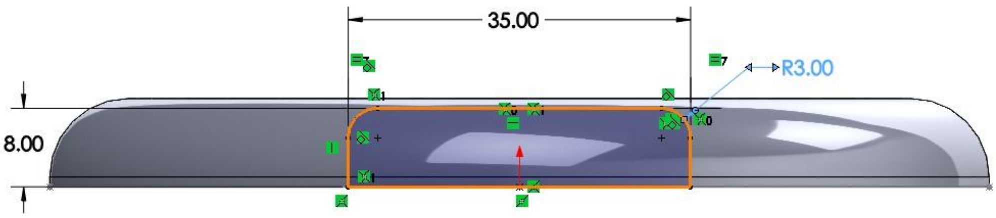




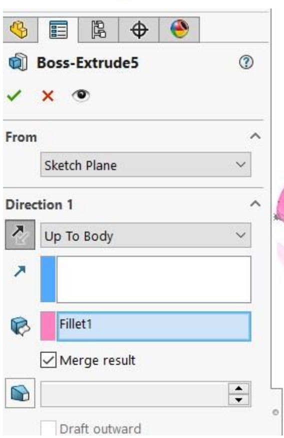


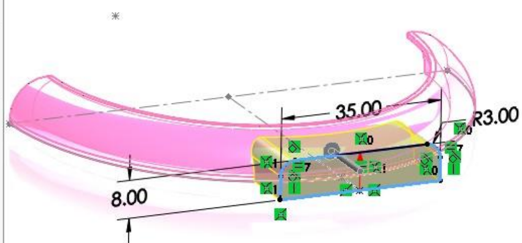
Create a new plane parallel to the right plane at a distance of **65mm** as shown.

Draw the sketch shown.

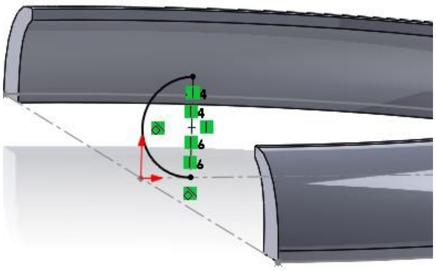


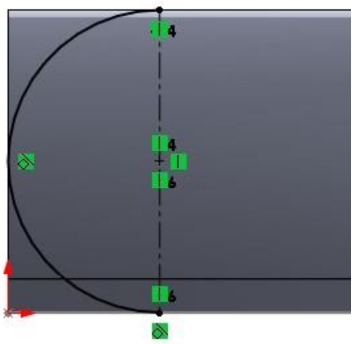
Extrude up to Body.



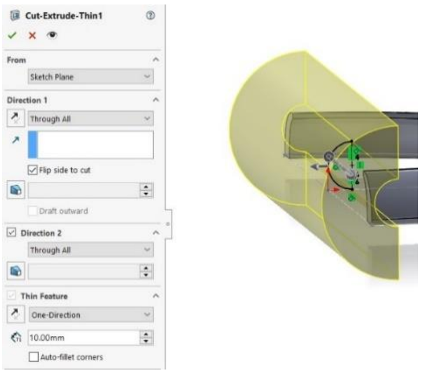


On the **Front Plane** draw a semi-circle tangential to the end, top and bottom edges of the part as shown.

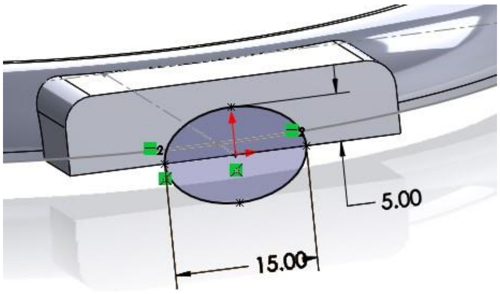




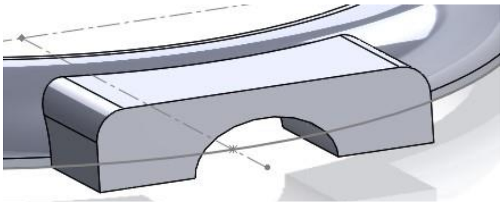
Select **Extrude Cut** and **Through All**



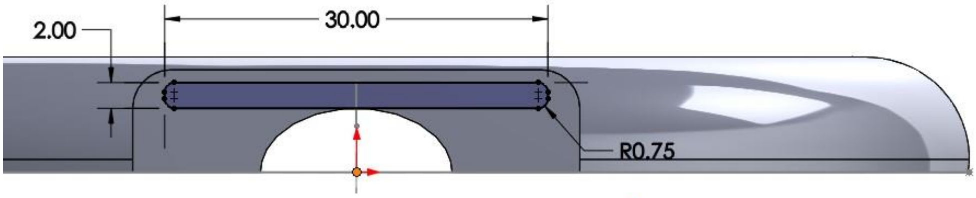
On the face shown draw the ellipse to the given dimensions.

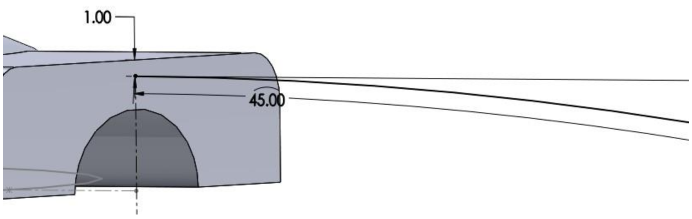


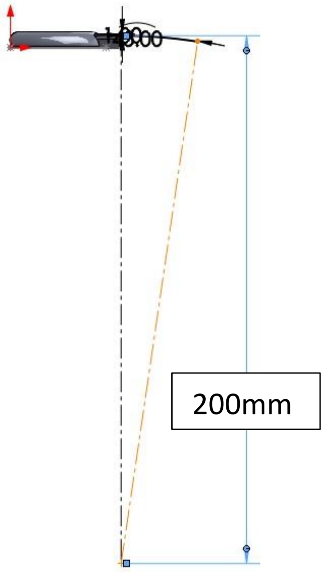
Extrude Cut through all.



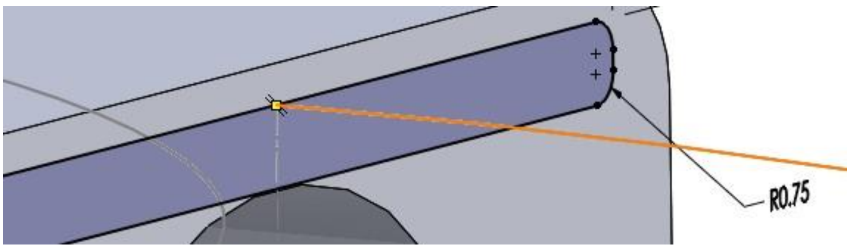
Draw the rectangle on the same face and add a fillet of 0.75 to the corners.





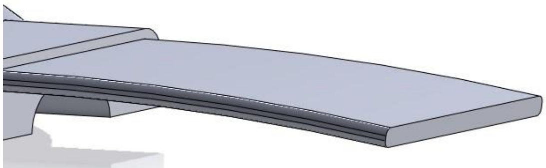


On the **Front plane** draw a **Centre Point Arc** having a radius of **200 mm** with the centre point vertically below the face of the part. The length of the arc is **45mm,** and the arc starts **1mm** below the top face of the Part as shown.

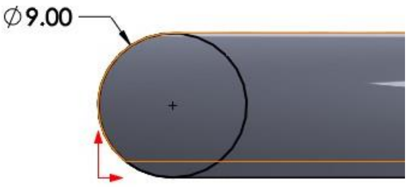


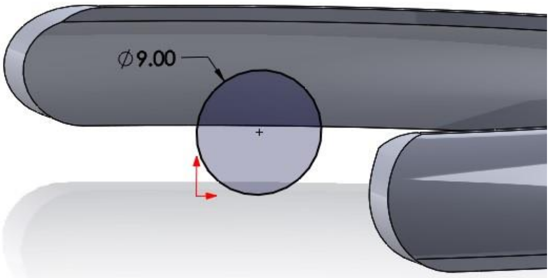
Add a **Pierce relation** between the arc and the Midpoint of the profile.

Use **Sweep Boss/Base** to complete the feature.



On the front face draw the **Circle** shown.

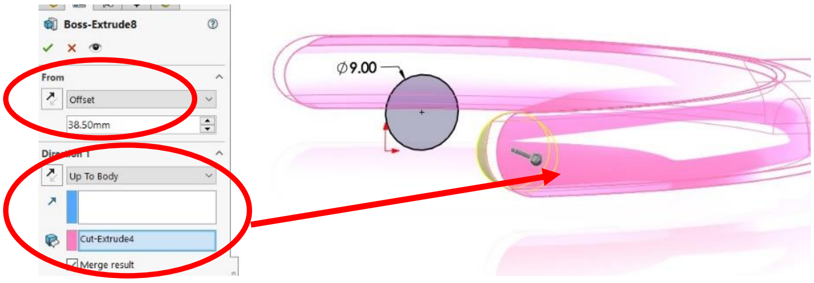




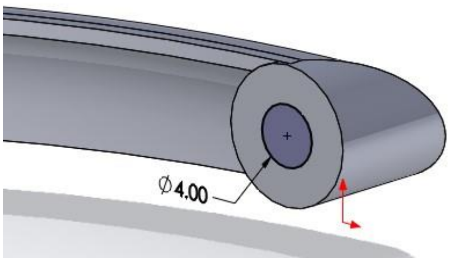
Select the **Extrude** command.

Input the following information.

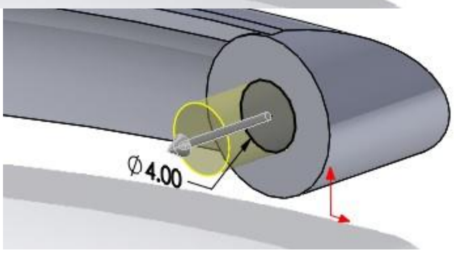
The extrusion begins **38.5mm** from the sketch. Select the body as shown, to indicate where the extrusion ends.



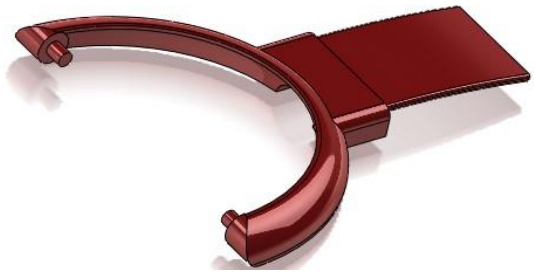
Draw a **Circle** diameter **4mm** on this face, and **Extrude** it by **5mm**.



Mirror these features about the front plane.

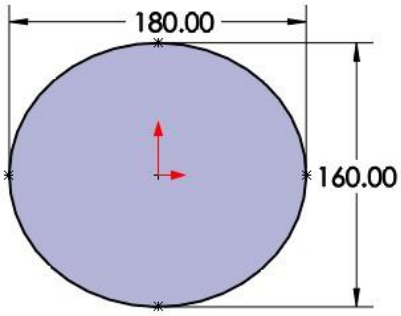


Apply an Appearance to the Part.

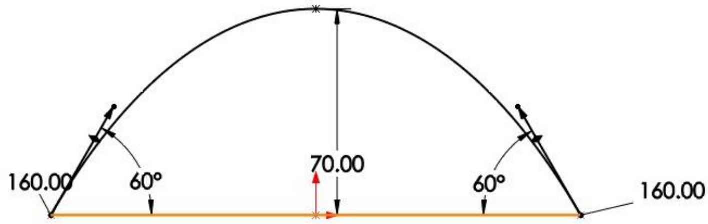


Save.

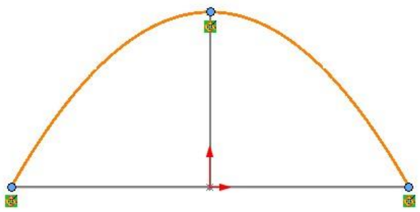
Lamp Hood



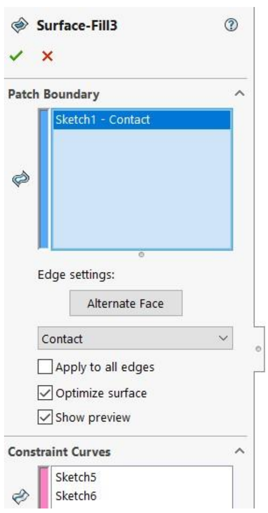
On the **Top plane** draw an **Ellipse** to the following dimensions. On the **Front Plane** select the **Spline** command to draw the following sketch. Add **Pierce Relations** to connect the spline to the sketch on the top plane

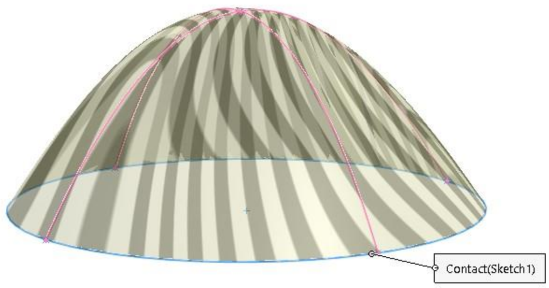


Select the **Right Plane** to draw the three point **Spline** shown. Add the **Pierce Relation** to connect the sketch to the two other sketches.

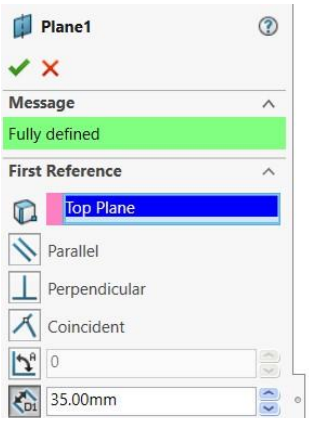


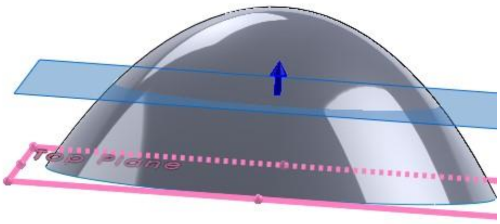
Select **Filled Surface** in the Surfaces Tab.



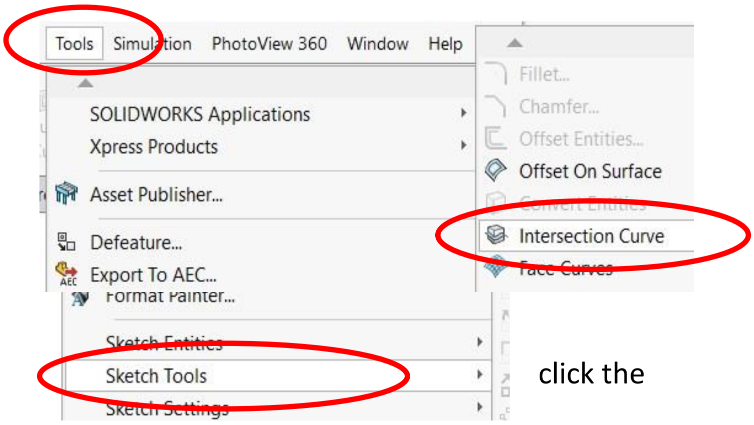


Create a plane parallel to the Top Plane a distance of **35mm** above.

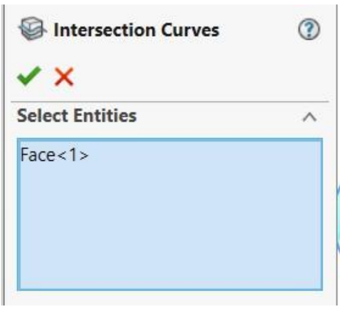


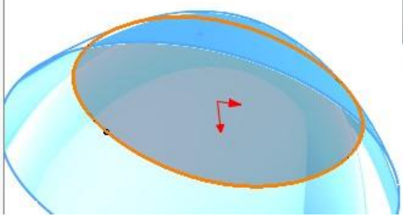


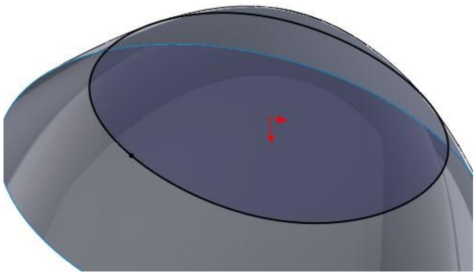
To find the line of intersection between the plane and the hood shell, select **Tools**, **Sketch Tools** and **Intersection Curve**.



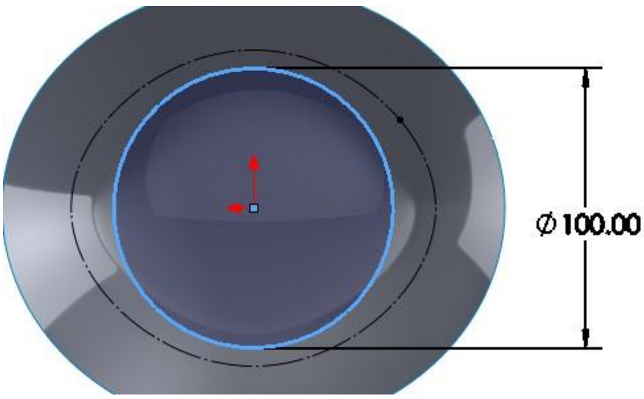
Click on the face of the hood and click the accept.



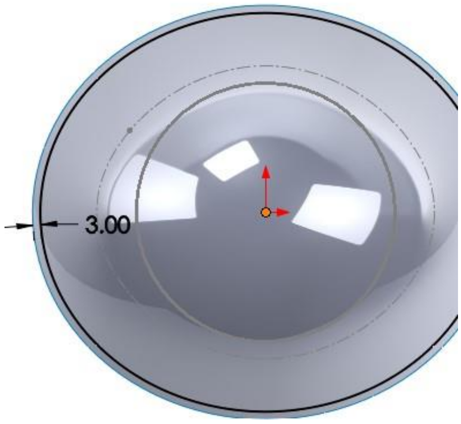




Still in this sketch, draw a **Circle** diameter **100mm** and change the intersection line into a centreline.



Accept the sketch.



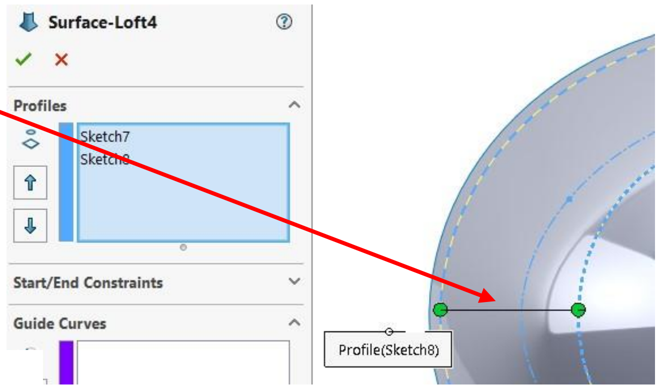
Create a new sketch on the **Top Plane**. Select the **offset** command to draw an ellipse **3mm** inside "***sketch 1"***.

Accept the sketch.

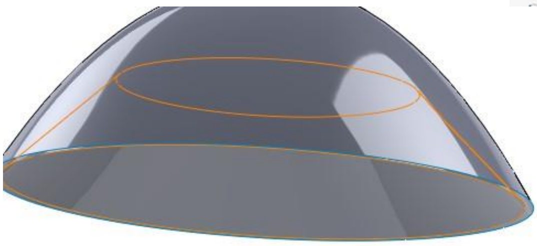
In the surfaces tab select **Lofted Surface.**

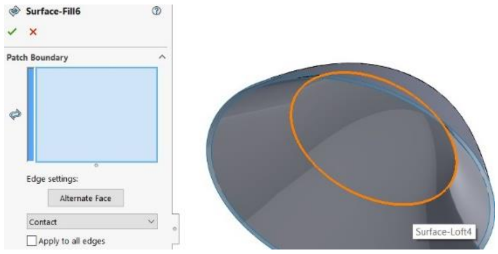
Be careful when selecting each profile.

Try to keep the green dots horizontal,in order to achieve the best shape.



Accept.

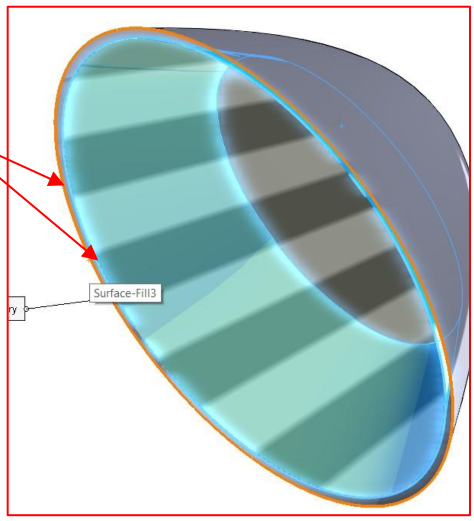




Select the **Surface Fill** command.

Select the edge highlighted.

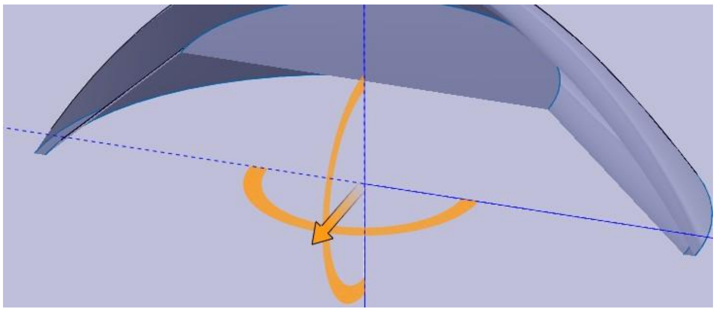
Accept.



Select **Surface Fill** again and select the edges on the top plane as shown.

Accept.

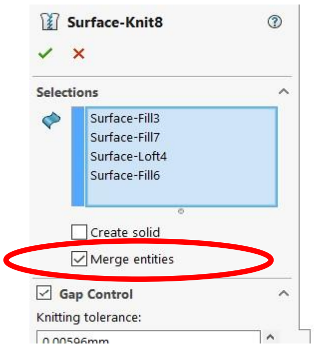
Two new surfaces have been applied.

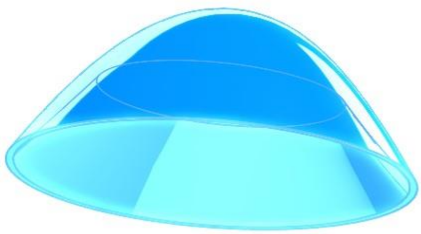


Select the **Surface Knit** command, and click on all four surfaces in the drawing area to knit.

Tick the merge entities box.

Accept.



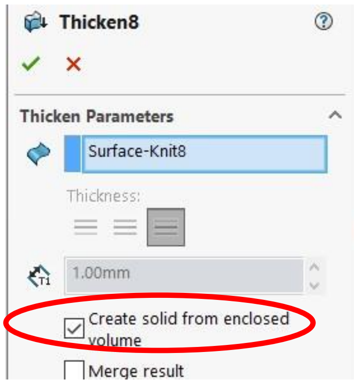


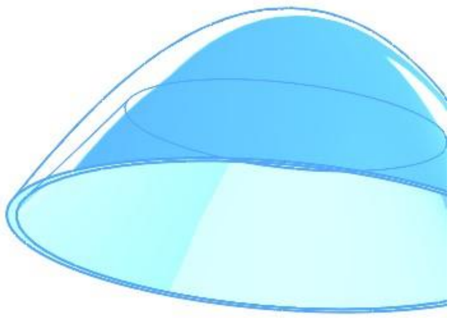
Select the **Thicken** command.

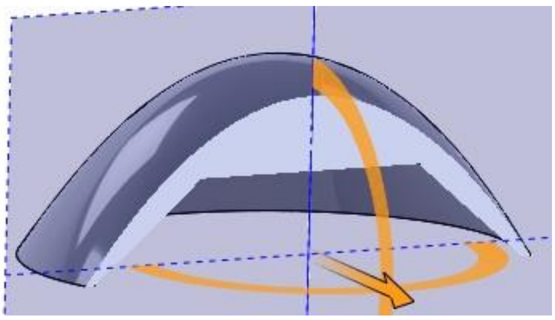
Add a thickness of **1mm** to the inside.

Tick the box to create solid from enclosed volume.

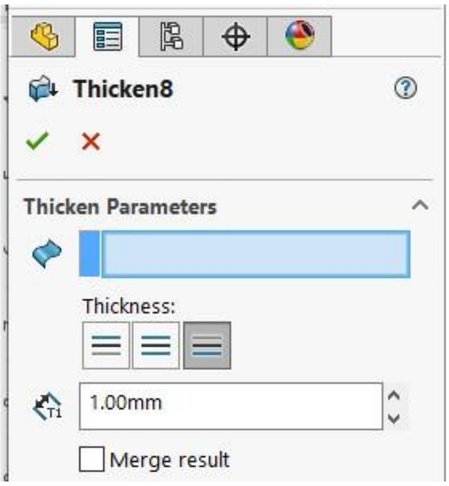
Accept.

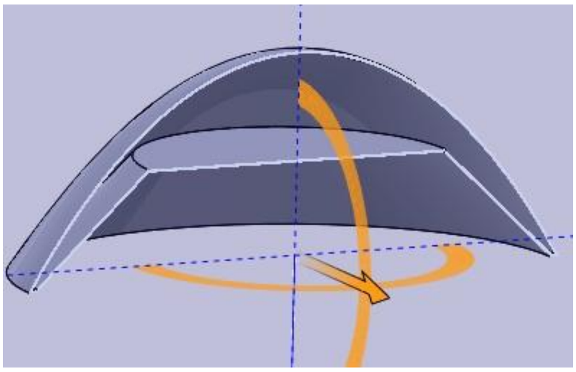






If the box to "create solid" is not ticked, and 1mm thickness is applied, the enclosed space is not filled as shown.



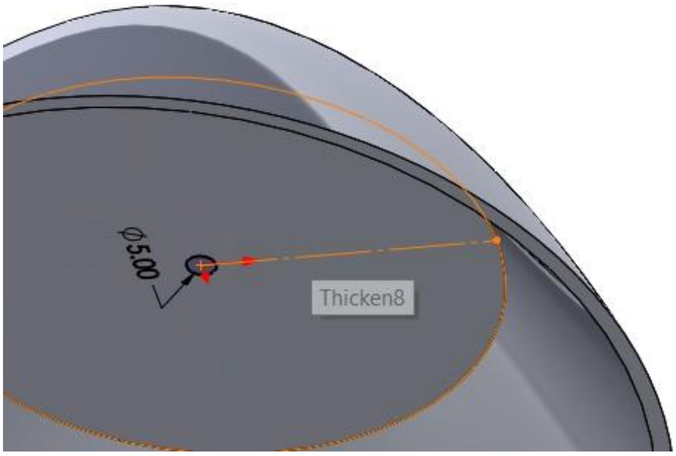


Recess for lights/ Diodes

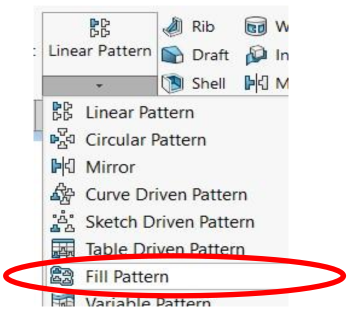
On the face shown draw the **Circle diameter 5mm**.

Draw a **Centreline** from the centre to the perimeter.

Select **Extrude Cut,** and extrude cut the circle by **5mm**.



Select **Fill Pattern** in the feature commands.



Activate the Fill Boundary box, and on the drawing area, select the face.

Select the Pattern layout as shown.

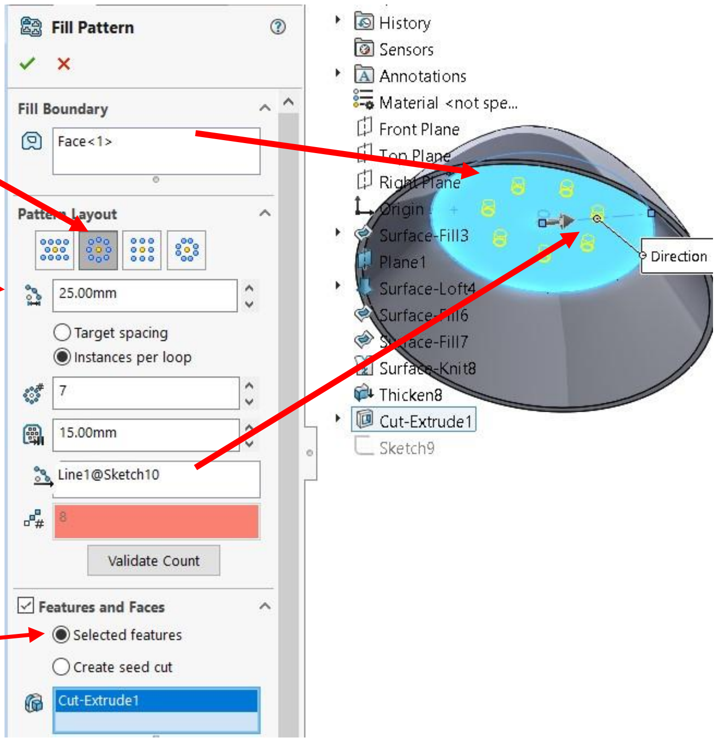
Set the spacing between holes to **25mm.**

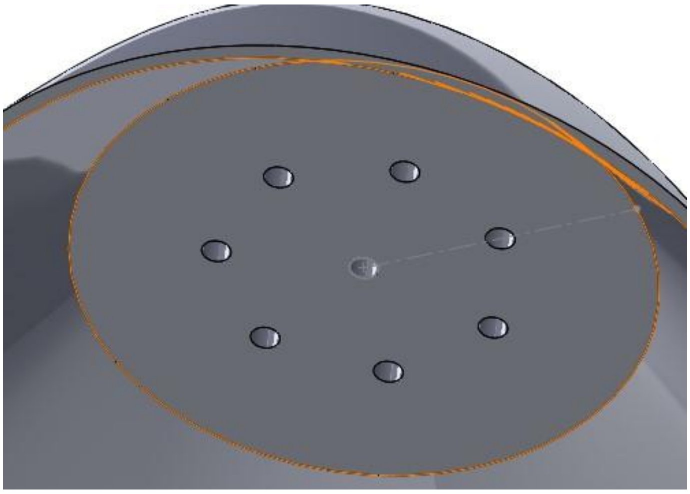
Set the number of **7.**

Activate the direction box and select the centreline.

Press the selected features button and select the Cut Extrude from the design tree.

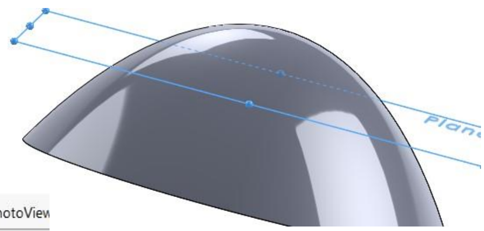
Accept.



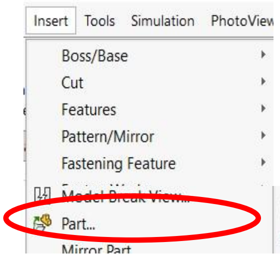


Creating the recess to receive the Hood Support

Create a Plane **45mm** above the **Top Plane**.

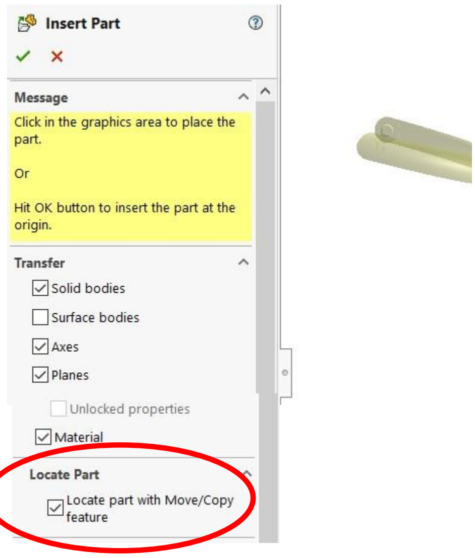


Under the **Inser**t tab select **Part**.



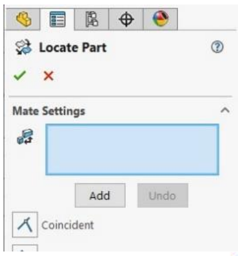
Select the **Hood Support**.

In the display on the Left hand side make sure Solid bodies, Planes and Locate Part boxes are ticked.

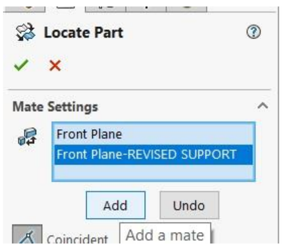


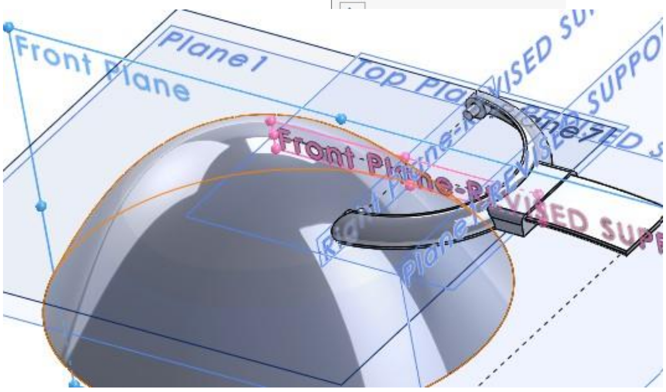
Drop the Hood support part near the Hood part.

The following mates window appears.



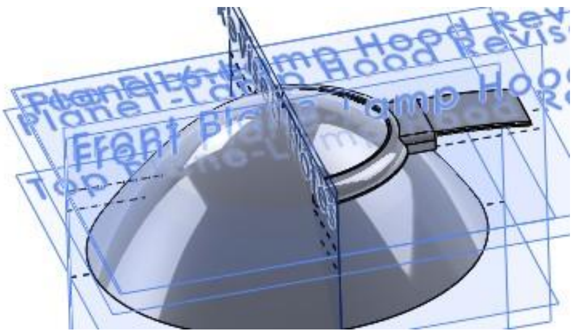
Mate the **Front Planes** as shown, and select **Add.**

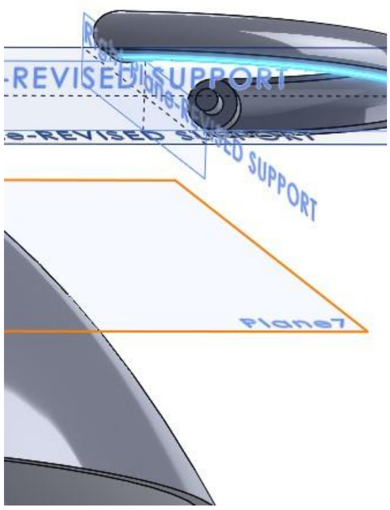




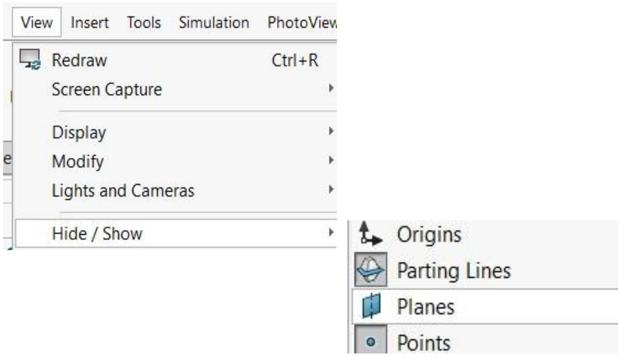
Mate the **Right Planes** and select **Add.**

Mate the plane that was created 45mm above the top plane with the underside of the hood support, as shown.





To hide the planes, click on the **View** tab. Select **Hide/Show,** and then **Planes.**



Select the **Indent** Command.

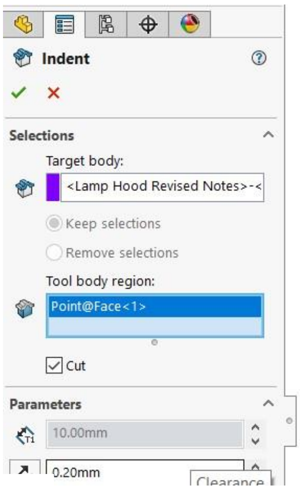
Select the lamp hood as the target body.

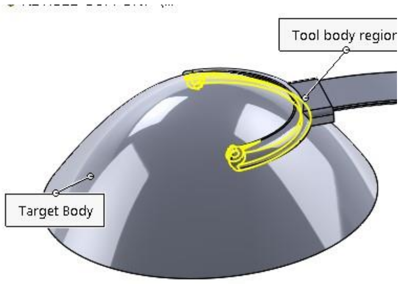
Select the support as the tool body.

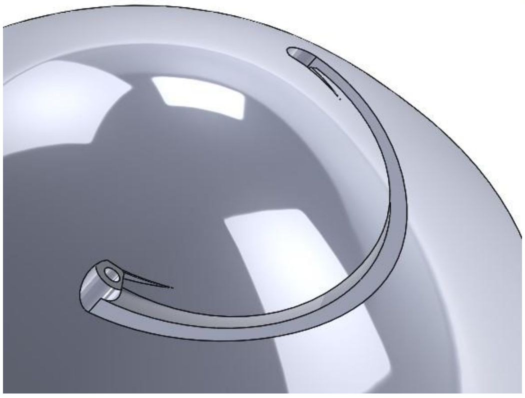
Add a **0.2mm** clearance.

Tick the cut box.

Accept.

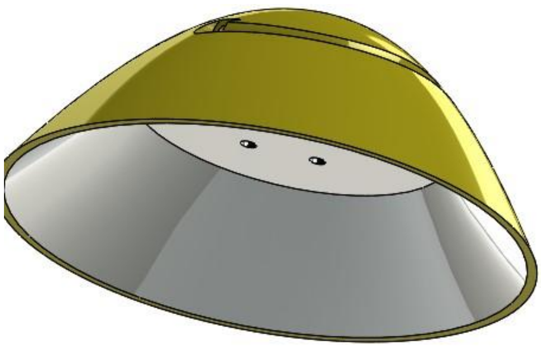


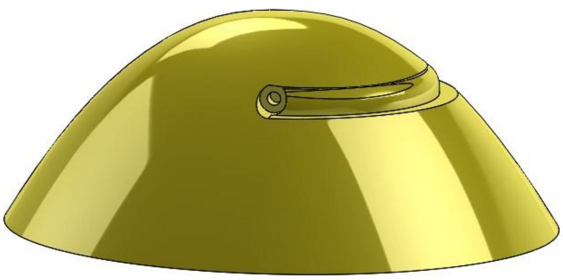




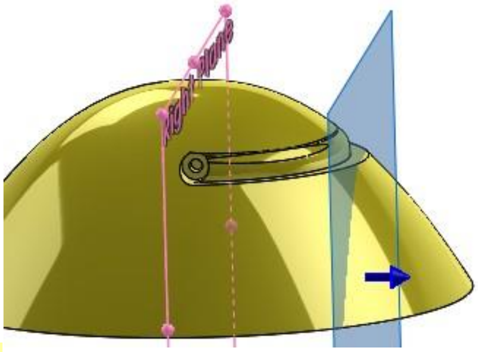
Hide the Hood Support part.

Apply a **High Gloss Plastic** to the part, and a **Chrome** finish to the inside faces.

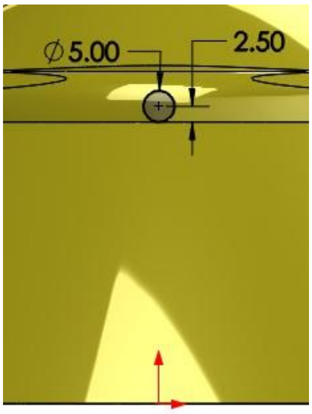




Create a Plane parallel and at a distance of **55mm** from the Right Plane.



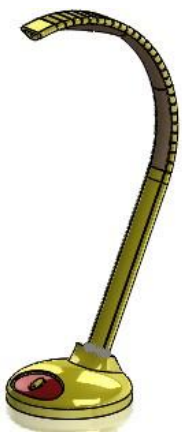
Draw the circle shown.



Extrude cut by 10mm.

Save the Part.

Study Lamp Assembly



The parts for the assembly must be saved in the same folder.

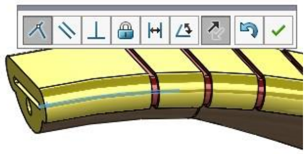
On the top of the screen select **File, New**.

Select **Assembly** in the SolidWorks Document dialog box.

Click OK.

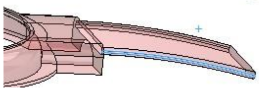
Bring in the Base Part, and select the green tick to fix it to the origin.

Bring in the Arm and apply the required mates.

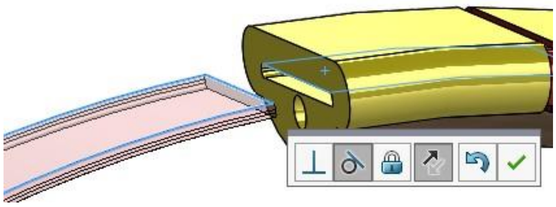


Bring in the Hood Support.

Mate the faces as shown.



Add a tangent mate to the Top of the Support, and to the top of the recess in the Arm, as shown.

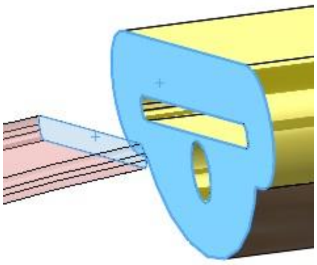


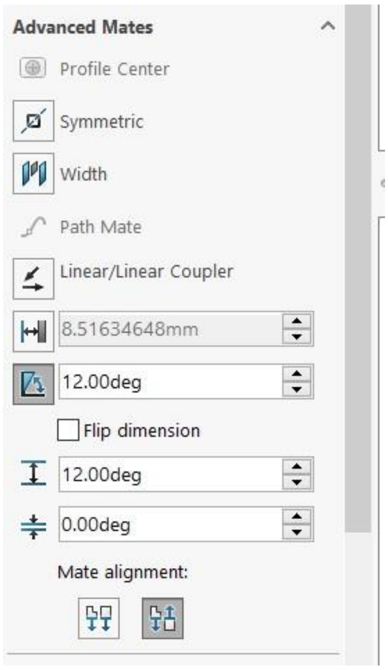
Finally select both faces as shown, and in Advanced Mates select angle mate.

Set the **minimum** angle to **0 degrees.**

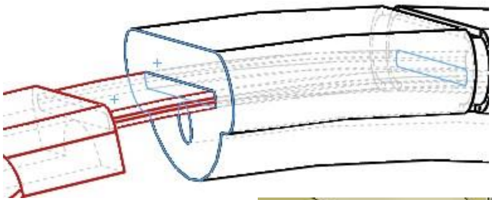
and the **maximum** angle to **12 degrees.**

Accept.



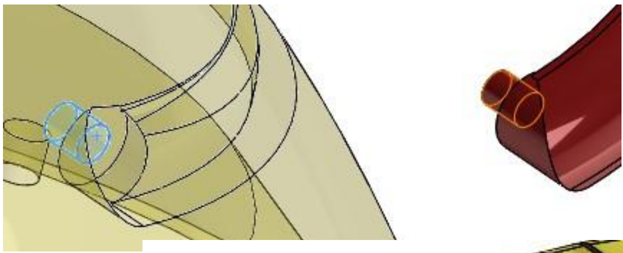


The support is now free to move in the arm to these values.

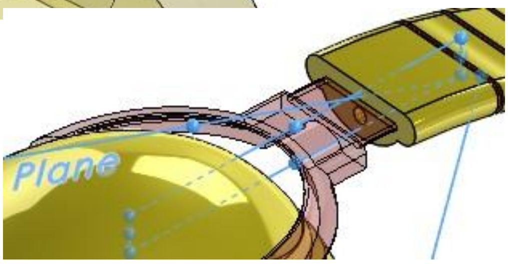


Bring in the Hood.

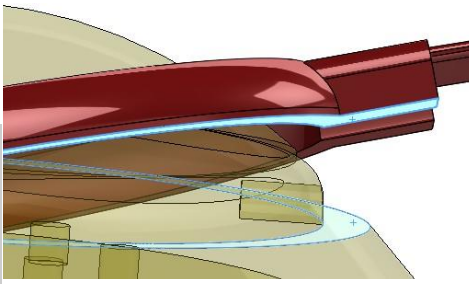
Apply the following concentric mates.

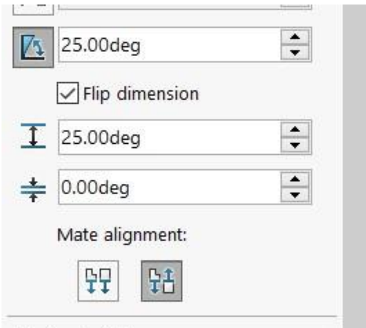


Mate the front planes of each part



Finally in **Advanced mates,** select Angle mate and select the two faces shown. Set the minimum angle to 0 degrees and the maximum angle to 25 degrees.

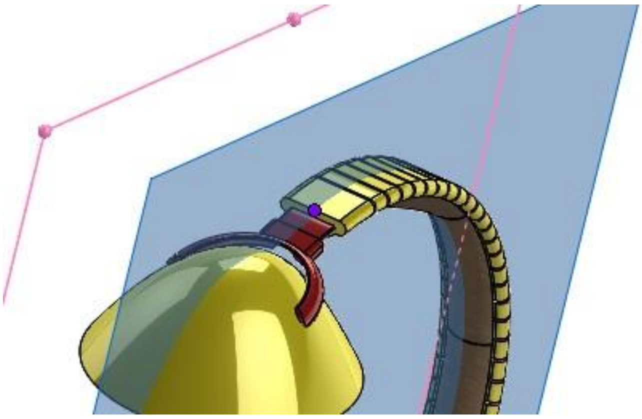




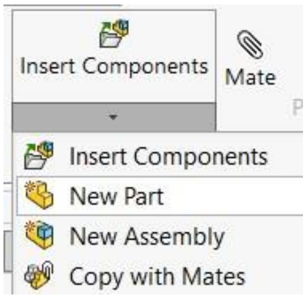
Accept.

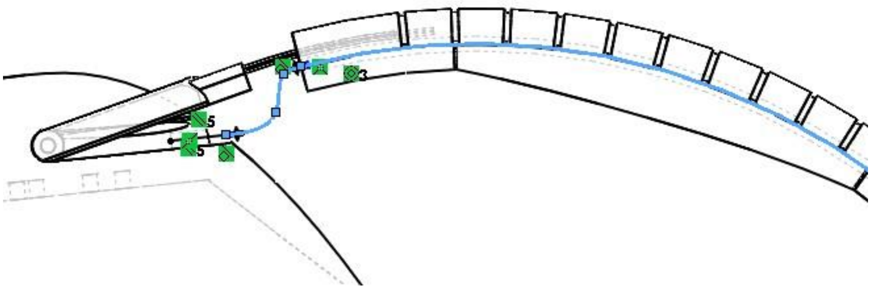
To Add the Cable using In-Context Modelling

In the Assembly, create a plane parallel to the front plane through the centre of the arm, as shown.

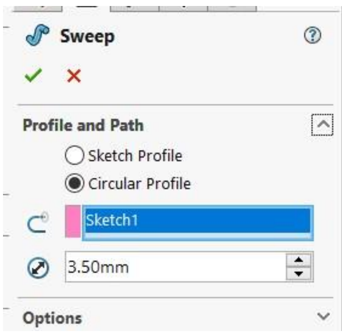


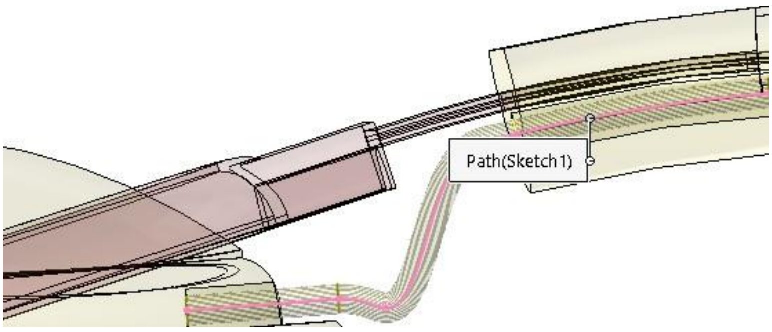
Select the down arrow under **Insert Component,** and select **New Part**. Select this new Plane to draw a sketch. Create the sketch shown using the **Line**, **Spline,** and **CentrePoint Arc** commands.



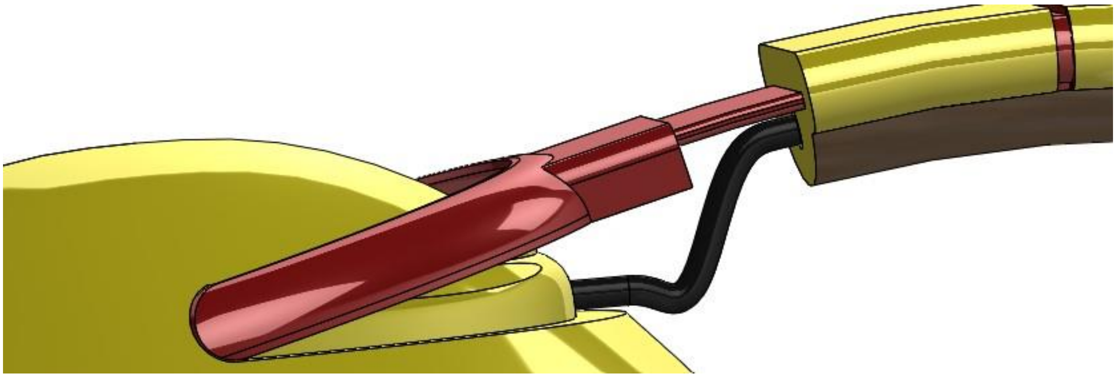


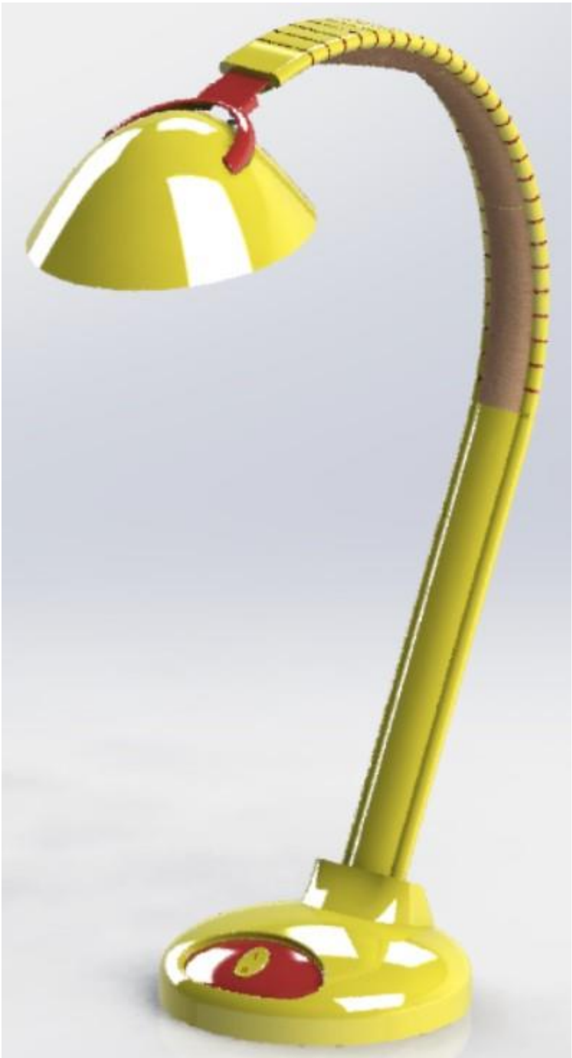
In the features commands, select **Swept Boss/Base.** Select circular profile and a diameter of 3.5mm.





On the top right hand side of the display area, select the icon shown to exit and return to the assembly.





The Task is Complete