PDST

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

USB Key

Design & Communication Graphics
Technology
Engineering
Construction Studies

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Introduction & Learning Intentions

Introduction:

This introductory lesson is aimed at participants with little to no prior knowledge of SolidWorks, or those who are looking for a refresher course in SolidWorks. It aims to guide the participants through the basic sketching and features commands of SolidWorks. There will be an emphasis on economy of design when creating each part of the project. In completing the USB key, you will look at parts, assemblies and the application of a decal to a part.

Learning Intentions:

At the end of this workshop you should be able to:

- Use the various sketch commands such as line, rectangle, circle, offset, convert entities and dimensioning.
- Use the features commands such as extrude boss/base, extrude cut, fillet and be able to add additional planes to help in creating various features on a part.
- Apply materials and decals onto a part to make the object look more realistic.
- Assemble the parts to create a realistic object that has all the required movements necessary to enable the object to function.

USB Key

USB Key Body

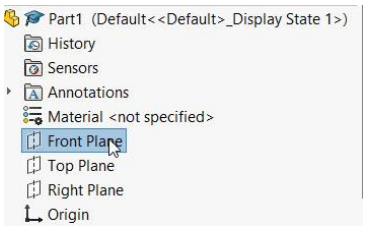
Open a new part from the SolidWorks Documents dialogue box.

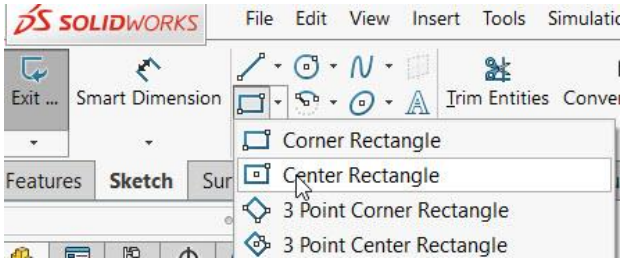
Select File. Click Save, as on the standard toolbar. Save as USB Key Body in the PDST USB folder.

Continue to save periodically throughout the exercise.

Create sketch

Select the **Front Plane** in the design tree and select the sketch icon.

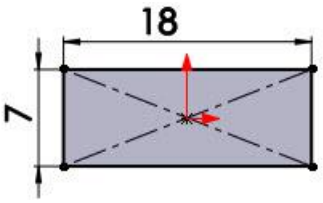




In the sketch commands select **Center Rectangle** as shown.

Select Smart Dimensions.

Draw the rectangle to the given dimensions



When the following dimensions are added the lines of the rectangle turn black indicating that the sketch is fully defined.

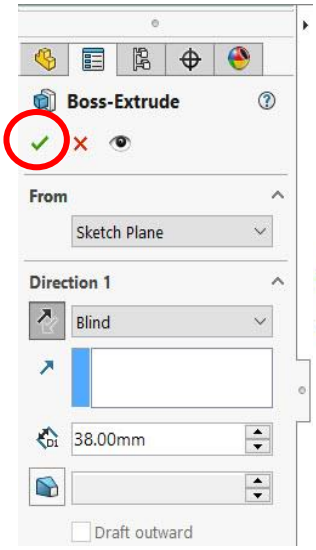
Accept the sketch by selecting the icon on the top right of the drawing area.

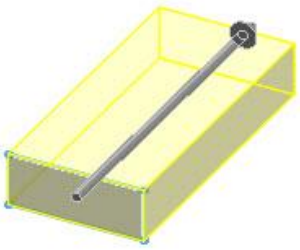
In the features tab

select the extrude

icon

Extrude by 38mm.

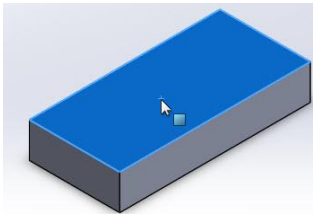


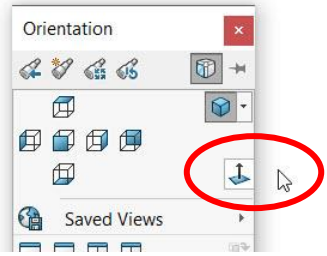


The direction of the extrusion can be reversed by clicking on the direction icon shown.

Accept the extrusion by selecting the green tick.

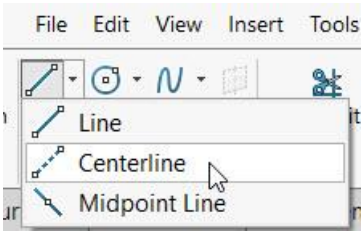
Select the top face of the object and select the sketch icon from the ghost toolbar.

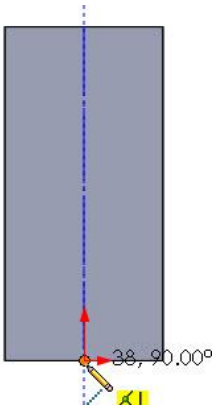




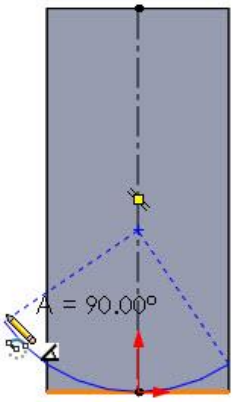
Press the space bar on the key board and select the **Normal To** Icon on the screen.

Select the **Centerline** command as shown. Draw the centerline from the midpoint of the top of the rectangle to the bottom.

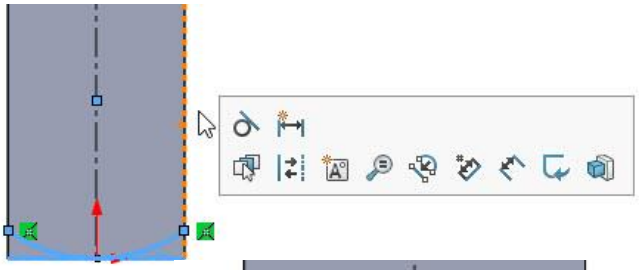




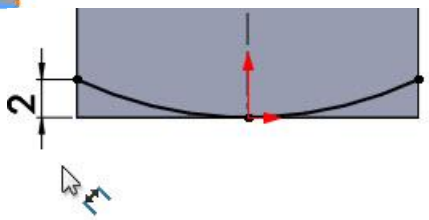
Select the **Centerpoint Arc** command and place the centerpoint anywhere in the centerline and draw an arc as shown.



Select the **arc**. Then press and hold down the control key on the key board to select the bottom line of the rectangle also. Release the control key once the two are selected and select the tangent icon on the ghost toolbar.

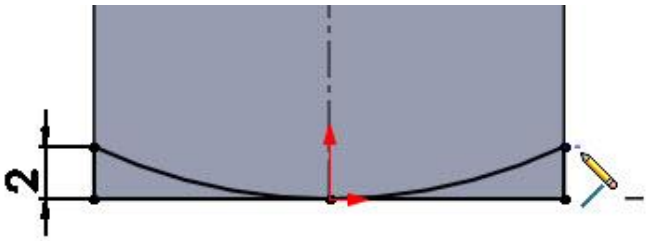


Select **Smart Dimension** and add **2mm** between the end of the arc and the bottom line as shown.



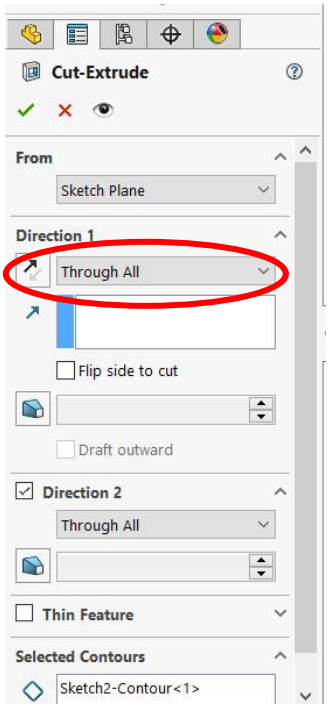
To complete the sketch, select the line command and draw a line from each end of the curve to the bottom of the rectangle The sketch is fully defined.

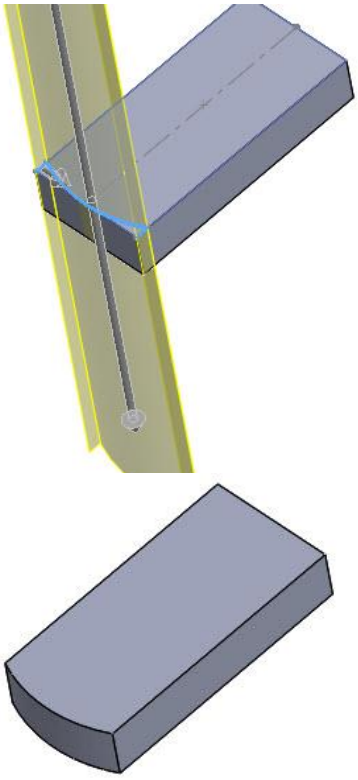
Accept the sketch.



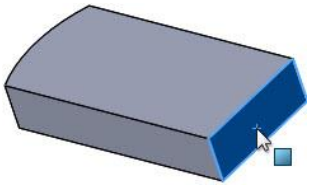
In the Features Tab select **Extrude Cut**

For direction 1, select **Through All**.





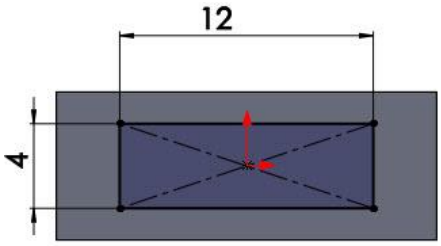
Rotate the part by pressing down on the wheel of the mouse and moving the mouse.



Select the end of the part and select the sketch icon.

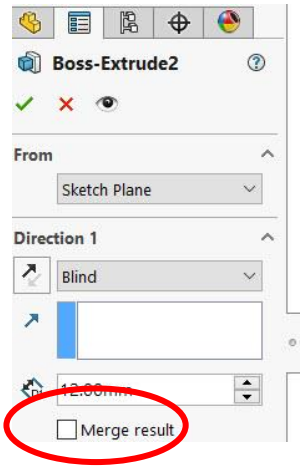
Use the space bar and select **Normal To**. This will make it easier when drawing the sketch.

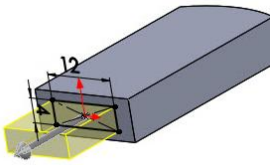
Using the **Centre rectangle** command, draw a rectangle to the following dimensions.



Accept the shetch.

In the features tab select **Extrude Boss/Base** and extrude by **12mm.**



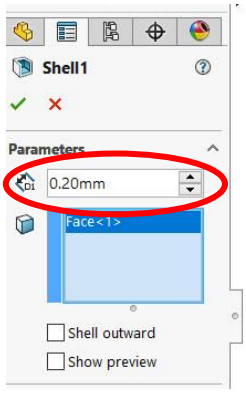


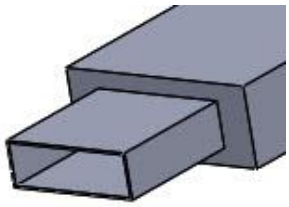
Note: Make sure to untick the merge result box.

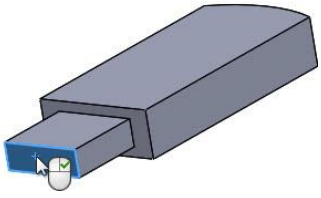
Select the shell command

in the features tab.

Select the face as shown and shell to a thickness of 0.2mm.

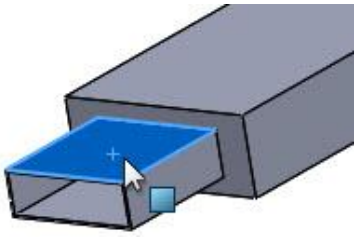






Accept.

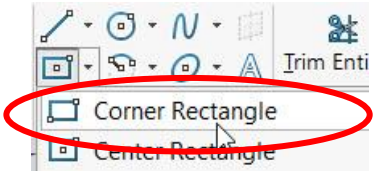
Select the face shown and select the sketch command.

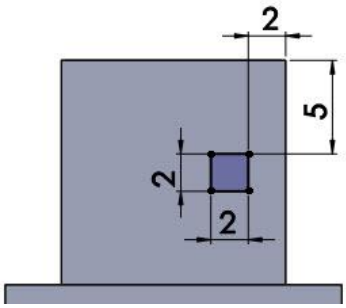


Use the space bar to select **Normal To.**

Draw the following sketch using the **Corner Rectangle** command.

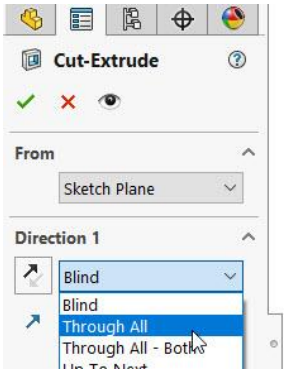
Accept the sketch.



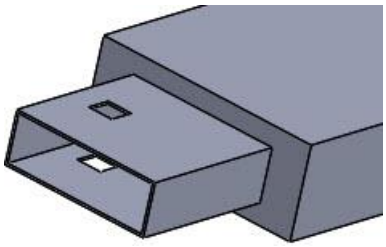


Select **Extrude Cut** in the featues tab.

For direction 1 select **Through All.**



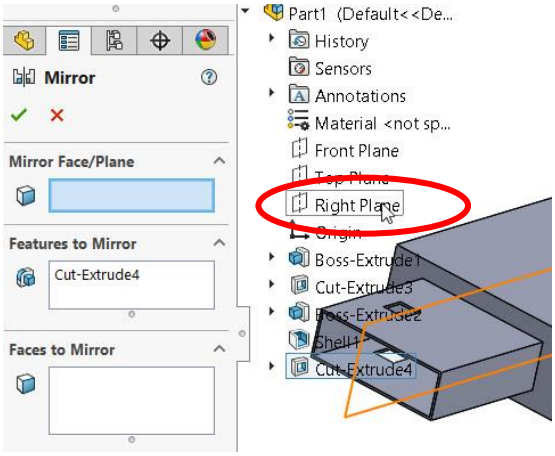
Accept.

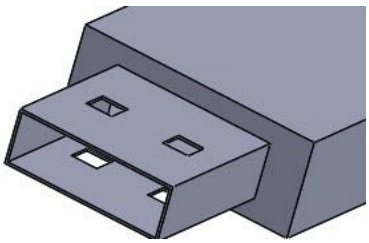


Still in the Features tab, select the

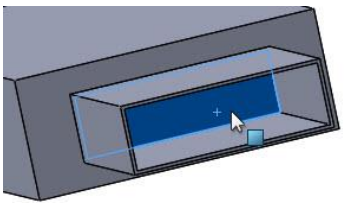
Mirror command.

Expand the design tree and select the **Right Plane** as the mirror plane as shown.



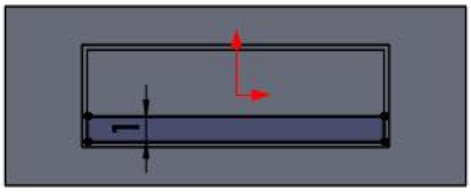


Select the face shown to draw the next sketch.



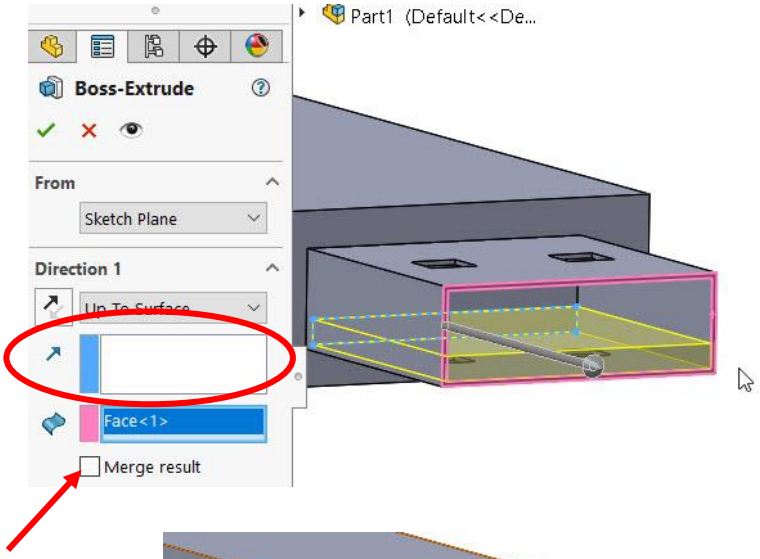
Select the **Corner Rectangle** command and draw a rectangle inside the shelled part to a height of 1mm.

Accept the sketch.



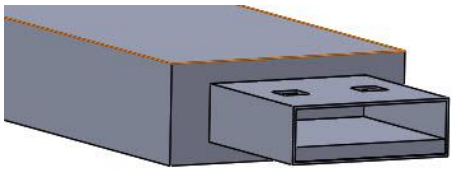
Select Extrude Boss/Base and extrude up to surface.

Select the surface shown in pink, on the drawing, as the end of the extrusion.



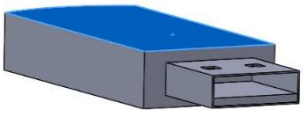
Note: Untick the Merge result box

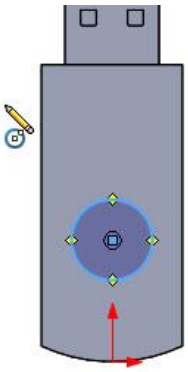
Accept.



Select the face as shown and select the sketch icon.

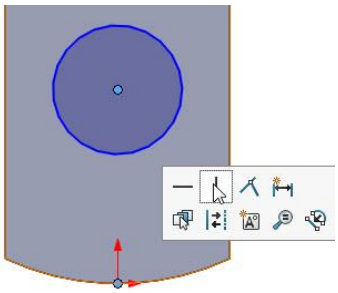
Press the space bar on the keyboard and select **Normal To**.



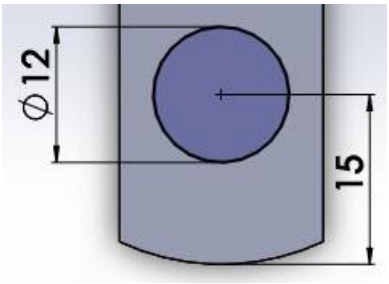


Select centre point circle and draw a circle anywhere on the face.

The centre of the circle must be vertically above the origin. To achieve this, select the centre of the circle and holding down the cotrol key on the keyboard select the origin. When the control key is released the ghost toolbar appears. Select the vertical icon.

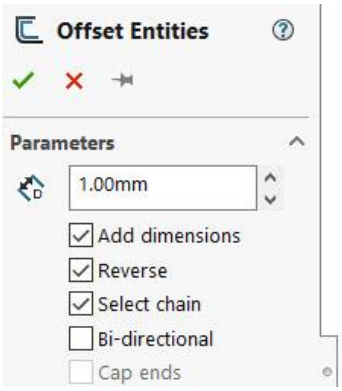


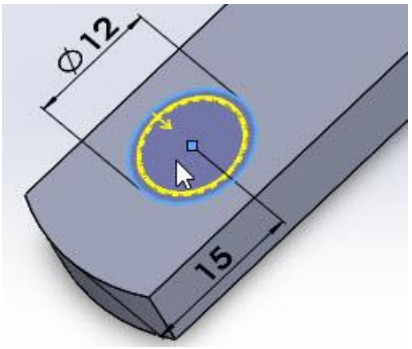
To fully define the sketch add the following dimesions.



Still in the sketch commands, select Offset Entities

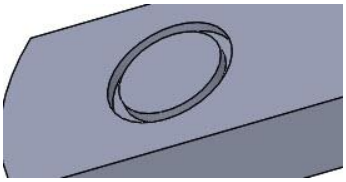
Select the circle and give an offset of 1mm and select reverse if necessary.



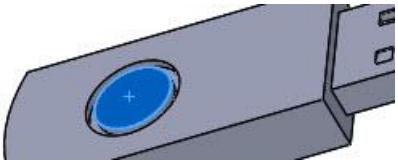


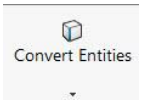
Accept the sketch.

Extrude cut by **1mm.**

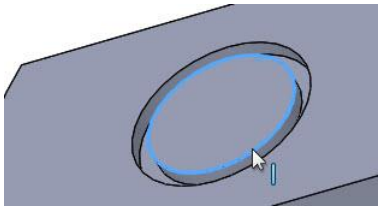


Select the face shown select the sketch command.

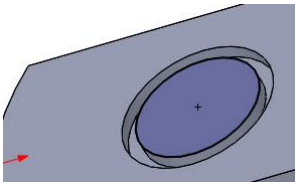


Select convert entities 

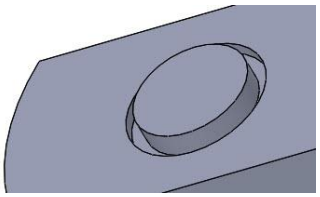
and select the circle shown.



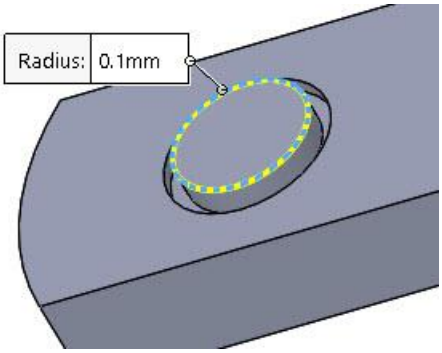
This creates a new circle on this new sketch plane.

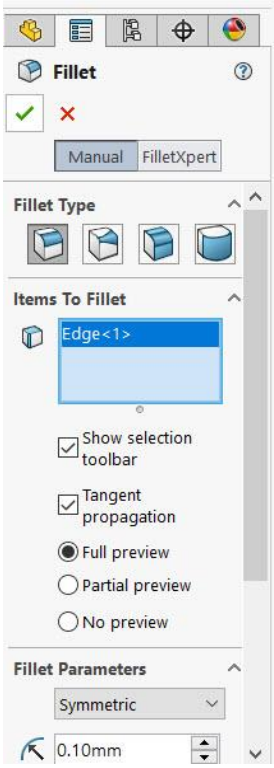


Accept the sketch and **Extrude** the sketch by **1mm**.



Select the **Fillet** command and add a **0.1mm** fillet to the edge shown.

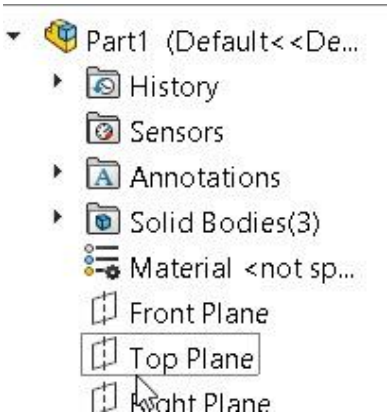


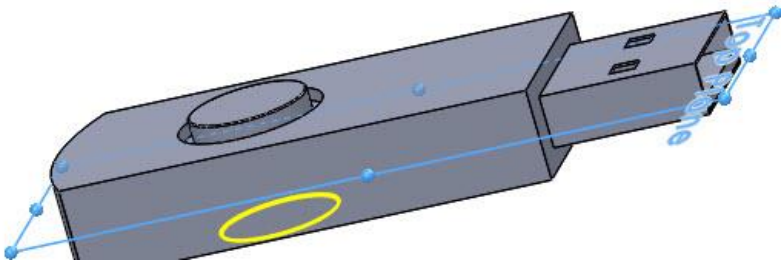


The features created must now be replicated on the opposite face. To do this the mirror command will be used.

Select the **Mirror** command Mirror in the features tab.

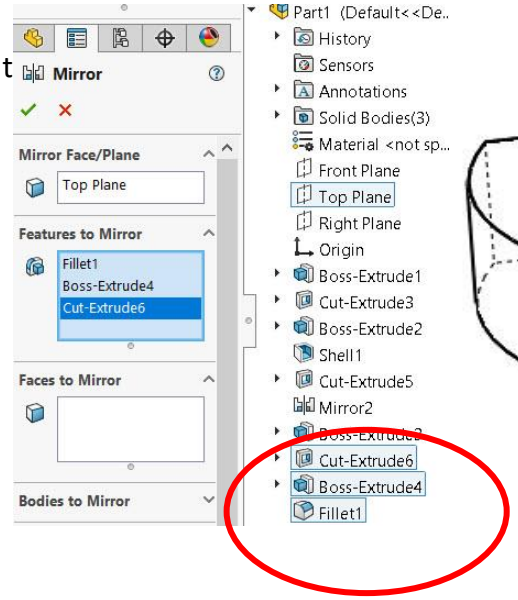
Expand the design tree and select the Top Plane as the mirror plane.

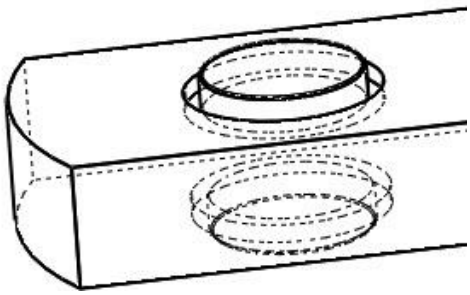




In the design tree select t

the features to mirror.



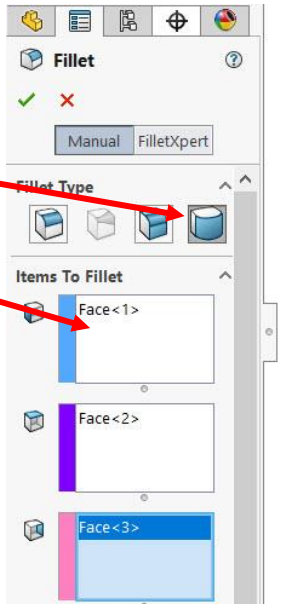


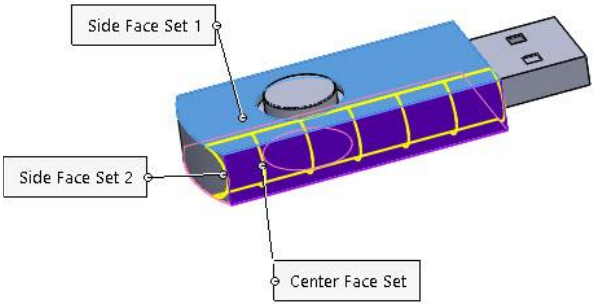
Accept.

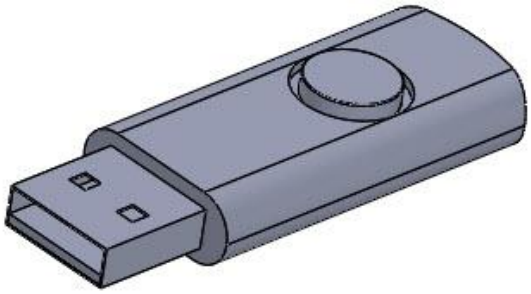
Select the **Fillet** command and select the full round fillet icon.

Click in the first box and then select the top face of the object.

Cilck in the second box and select the side face as shown. Select the third box and select the bottom face of the object.

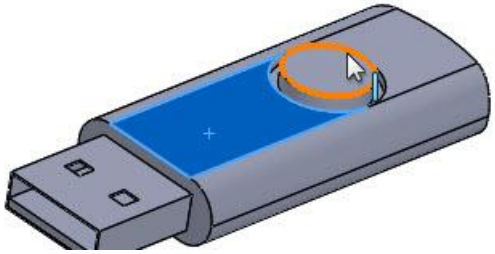


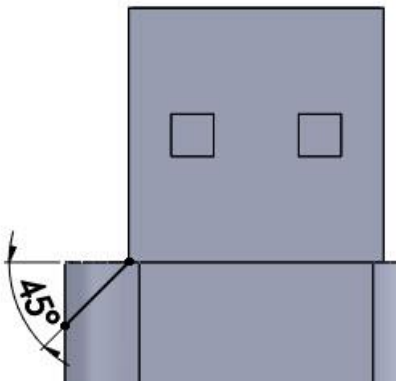




Accept.

On the top surface draw a line at 45 degrees as shown.

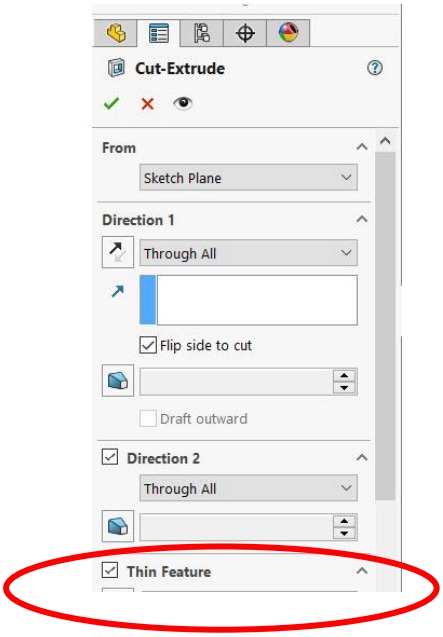




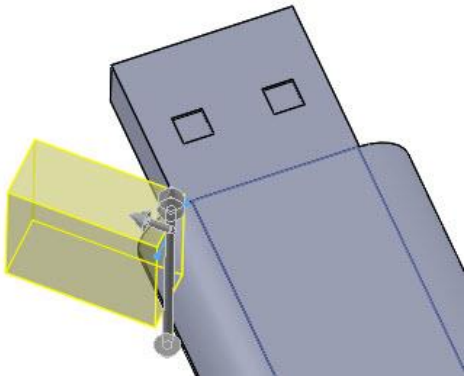
Accept the sketch.

Select **Extrude Cut** and tick the **Thin Feature** box.

Click in the flip side to cut if necessary.

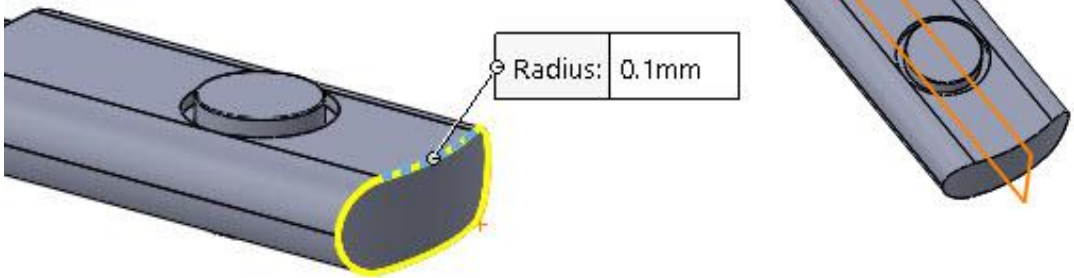


Accept.



Mirror this featuee about the right plane.

Apply a **0.1mm** fillet to the edges shown.

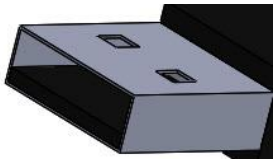


Appearance.

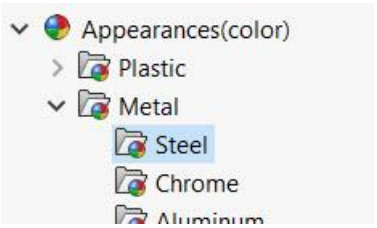
To apply appearances to the part, select the icon on the right hand side of the screen. Expand the appearance folder by selecting the arrow symbol. Scroll down to select the **Plastic** folder and select **Low Gloss**. Apply a **black low gloss plastic** to the body by holding down the left button on the mouse and dragging the black low gloss plastic ball onto the body in the drawing area. Select the body icon as shown.

black low gloss plastic

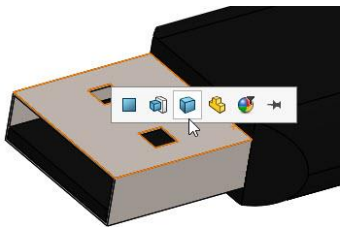
Apply a black low gloss plastic to the insert also as shown.



Apply a brushed steel appearance to the body as shown.

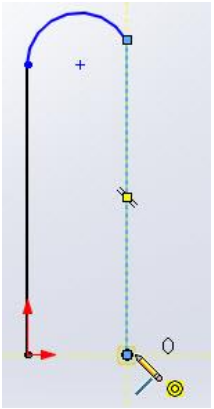


brushed steel



Save the USB Key Body.

USB Key Cover

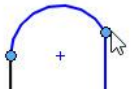


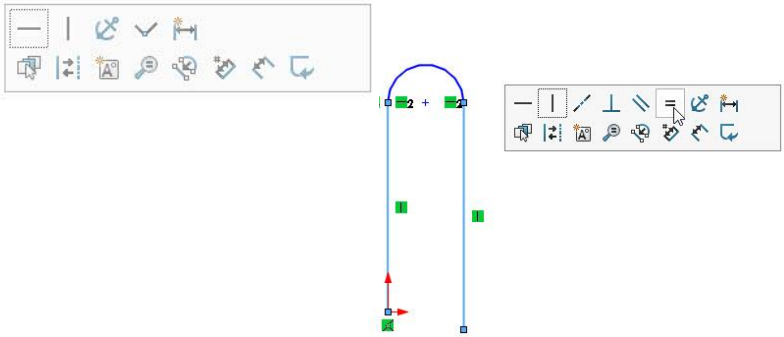
Open a new part and save as **USB Cover**

Select the **front plane** and using the line command draw the sketch shown.

Add a horizontal relation between the two nodes shown.

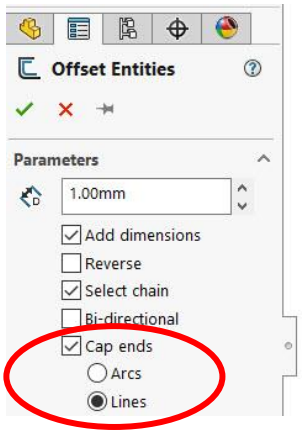
Add an equal relation between the two vertical lines as shown.



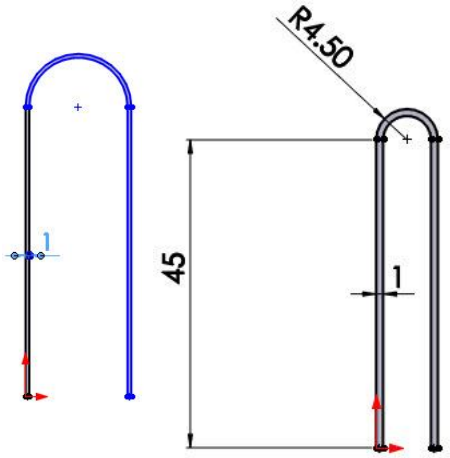


Select the offset entities command

Make sure that the offset is inwards by 1mm. Tick the **Cap ends** box and the **Lines** button to close the sketch.



Select Smart Dimensions and add the dimensions shown.

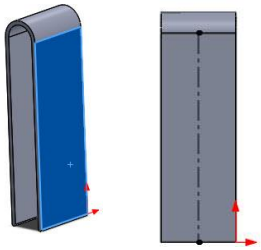


Accept the sketch.

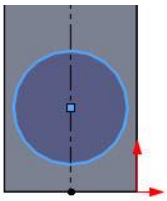
In the features commands select **Extrude Boss/Base** and extrude by **16mm.**

Select the face shown and select the sketch command.

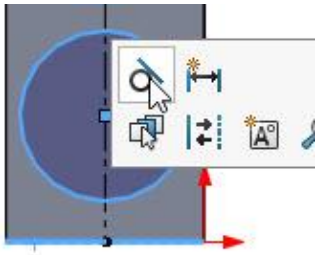
Draw a centreline between the two midpoints.

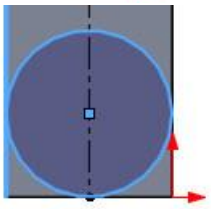


Select the circle command and draw a circle with its centrepoint on the centreline.



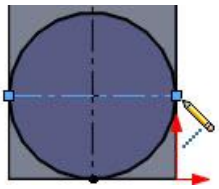
Select the circle and holding down the shift key select the bottomline shown and add a tangent relation.





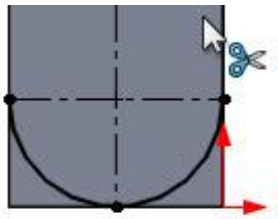
Repeat the process to add a tangent relation between the circle and the vertical line shown.

Select the centreline and draw the following diameter line.

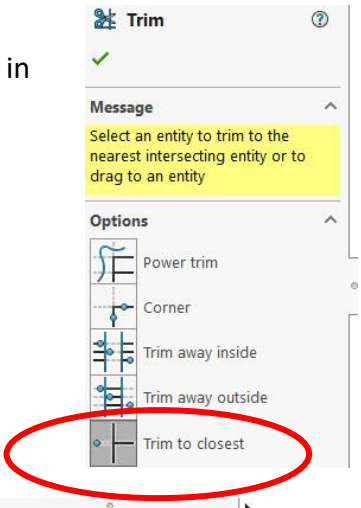


Use the **Trim Command**

to remove the excess from the circle.



Select the **Trim to closest** button then in the drawing area click on the lines to remove.

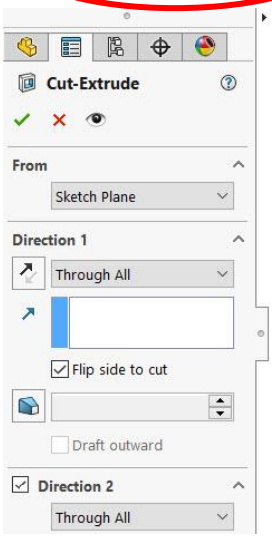


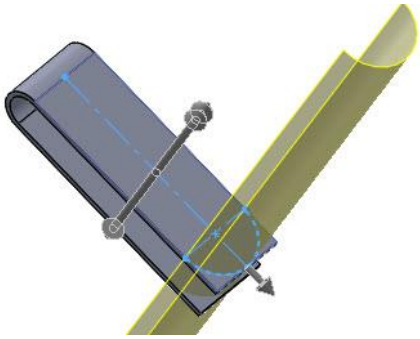
Accept the sketch.

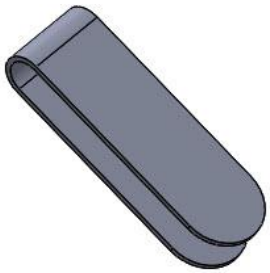
Select **Extrude Cut** in the Features tab.

Make sure the arrow is pointing in the correct direction on the drawing area.

Select through all for Direction 1 and Direction 2.



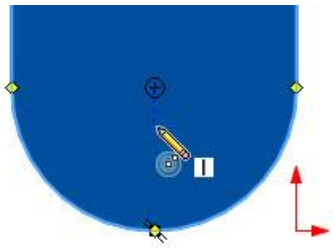


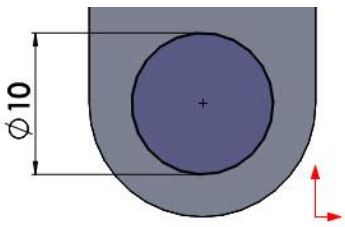


Accept.

Click on the surface shown and using the circle command draw a circle using the same cetre point as before.

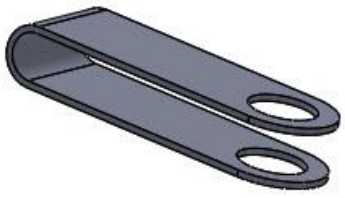
Add the following dimension.





Accept.

Extrude Cut and select **Throug All** for direction 1.

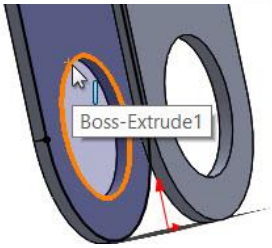


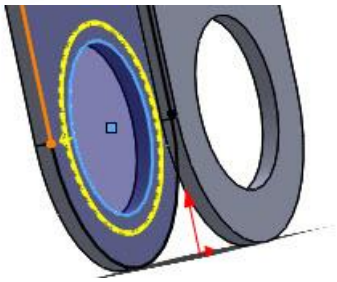
Accept.

Select the inside surface to draw a new sketch.

Select **Convert Entities**

and click on the circle shown to transfer it onto the new sketch.

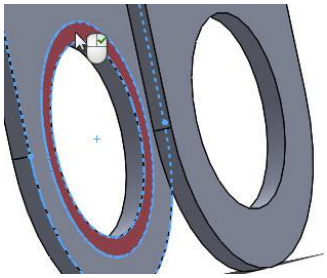




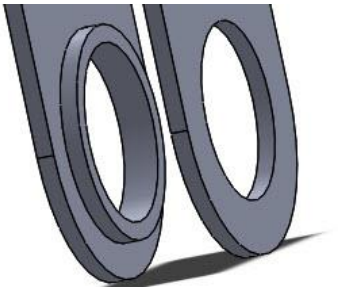
Select offset entities and offset the circle outwards by 1mm.

Accept the sketch.

Select Extrude Boss/Base and click on the area shown to extrude.



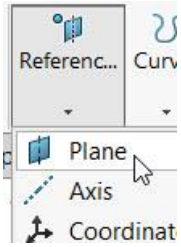
Extrude by **1mm** and **accept.**



To Morror this feature a new plane is needed.

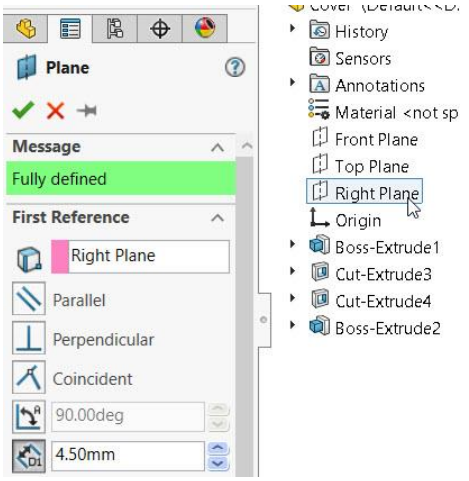
Select the arrow under the reference geometry icon in the Features tab.

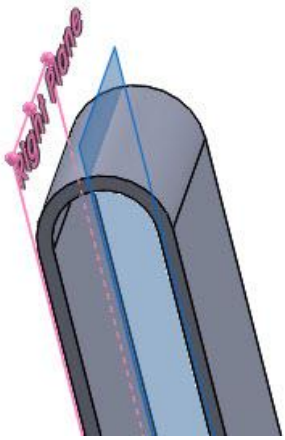
Select the **Plane** icon.



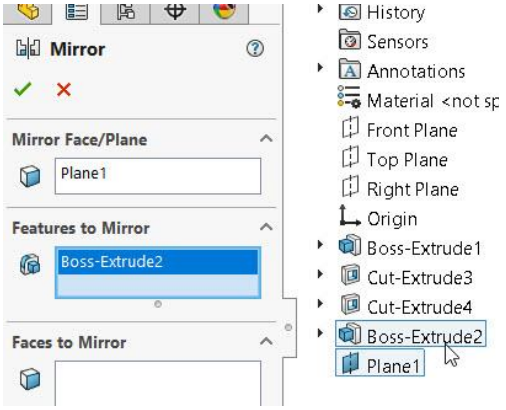
In the design tree, select the **Right Plane** as the first reference and add a distance of **4.5mm** as the distance the new plane is away from this reference plane.

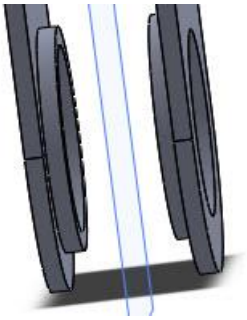
Accept.



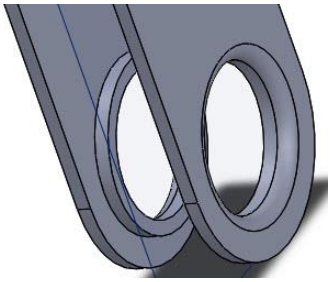


Select the Mirror command as shown to add the circular feature to the other side.



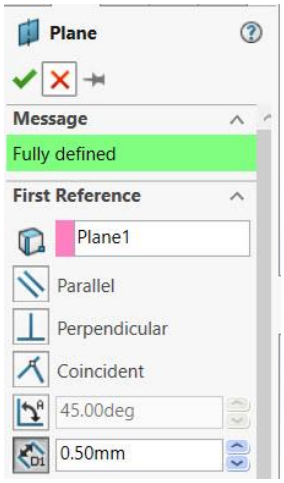


Add a **1mm fillet** to complete the detailing.

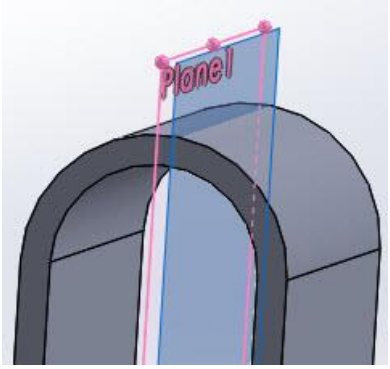


<u>To add the tab on top of the cover.</u>

Under reference geometry select plane. Create another plane parallel to plane 1 a distance of **0.5mm** away.

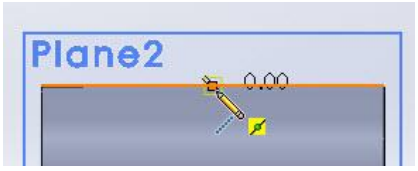


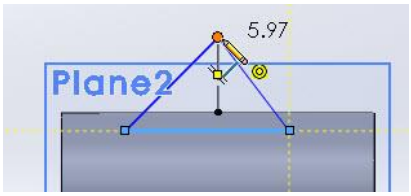
Select this new plane to draw a new sketch.



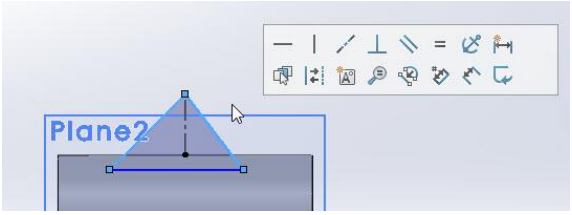
Draw a centreline from the midpoint shown.

Select the line command and draw the triangle shown.

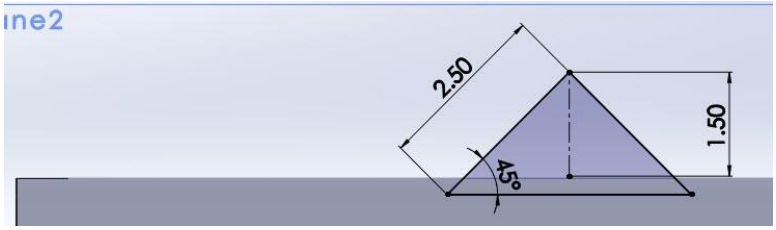




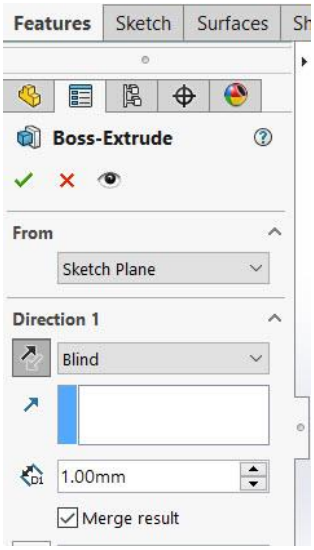
Add an equal relation between the two lines shown.

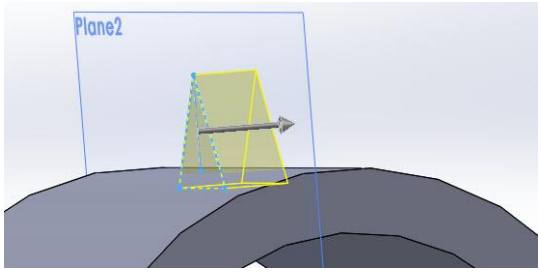


Add the following dimensions to fuly define the sketch.



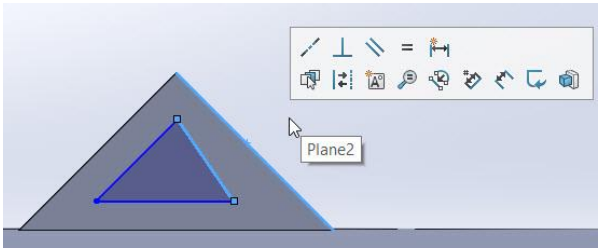
Extrude to a distance of 1mm.





On the face of the triangle draw another sketch using the line command.

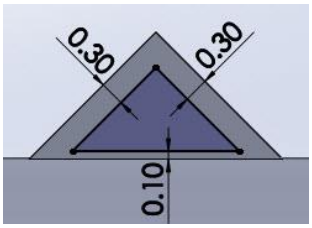
Add a parallel relation between the sides of the inner and outer triangle as shown.

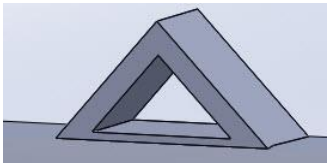


Add the following dimensions.

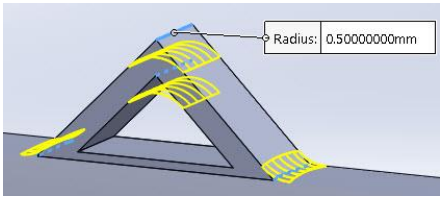
In the features tab select **Extrude Cut**.

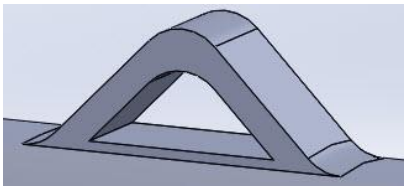
Cut through all.



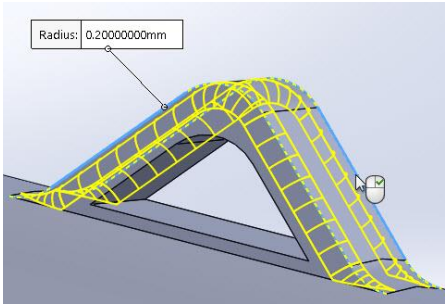


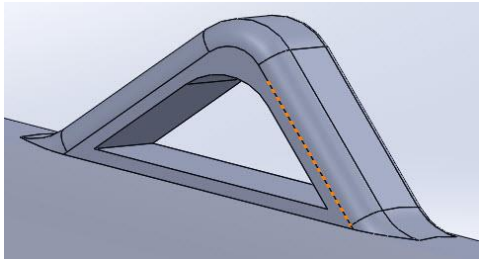
Select the fillet comand and apply a 0.5mm fillet to the edges as shown.





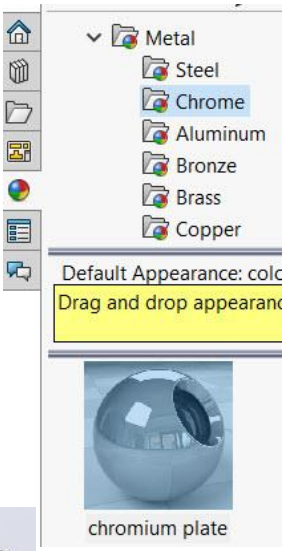
Add a 0.2mm fillet to the edges as shown.





Appearance

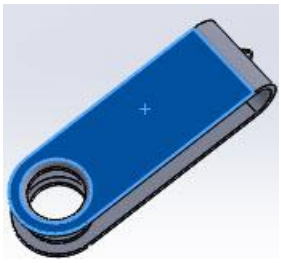
Add a **Chrome Plate** appearance to the part.



Save the part as **USB Cover**.

To add a decal onto the cover

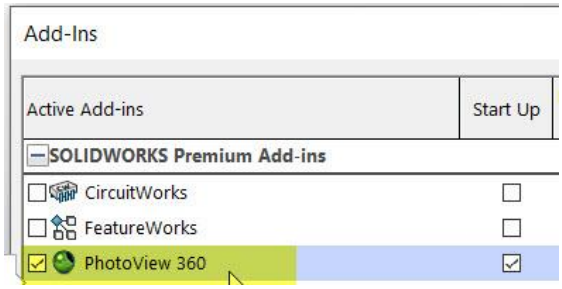
Select the face that is to receive the decal.



The decal is added using **Photoview 360.**

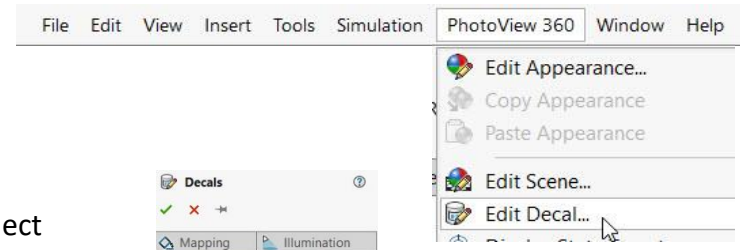
This icon is retrieved by selecting **options** on the top of the screen and selecting **Add-ins..**

Make sure the two boxes each side of Photoview 360 are ticked.

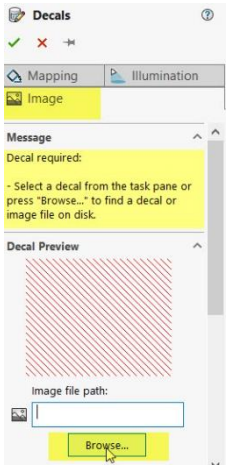


The Photoview 360 tab now appears on the tabs list at the top of the screen.

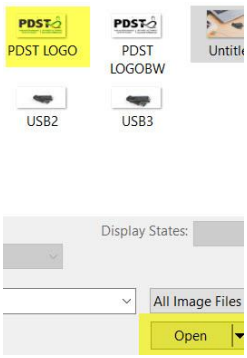
Select the tab and select **Edit Decal.**



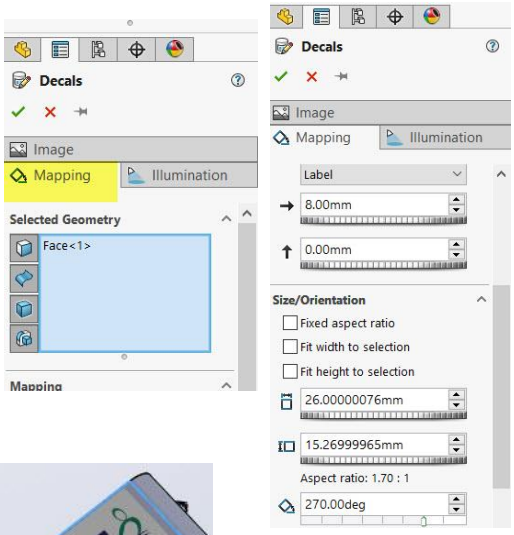
On the left had side of the screen select **image**, and then select **browse.**



Select the **PDST logo,** and press open.



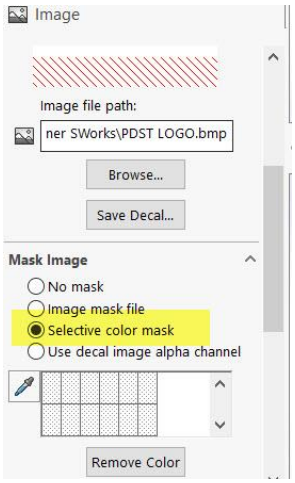
Select **mapping** on the left hand side.

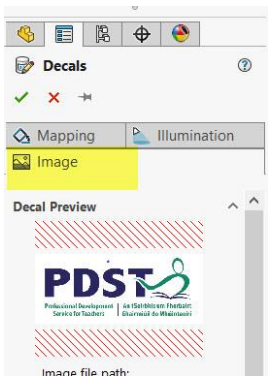


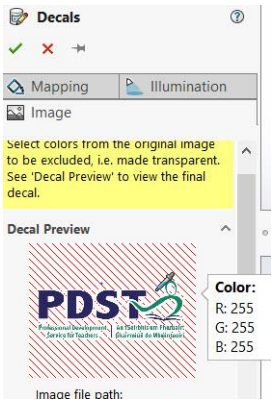
The decal can be rotated, moved, and size changed to fit correctly on the face using the wheels.

To remove the white background from the decal, select the image tab.

Under **Mask Image** select **Selective colour mask**. Select the dropper icon and go to **Decal Preview,** and click on the white background in this preview window.



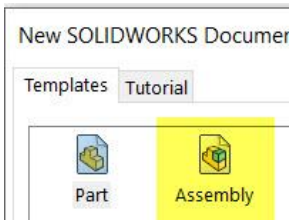




The white background is now removed from the decal on the cover.

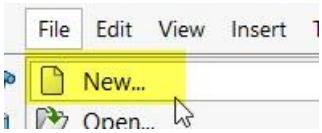
Note: It is important that the decal is saved in the same folder as the Parts and Assembly.

USB Key Assembly



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

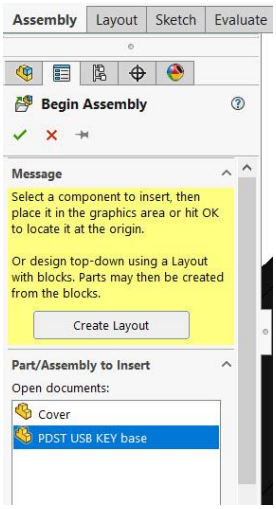
On the top of the screen select **File, New**.

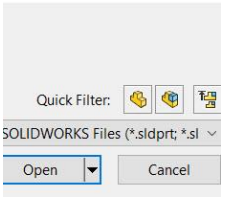


Select **Assembly** in the SolidWorks Document dialog box.

Click OK.

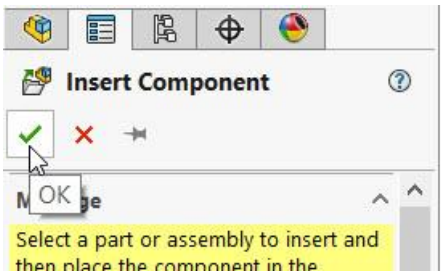
In the following window select browse and select **USB Key Base**. Select Open.





Click **OK** as shown.

The part origin will snap to the origin of the drawing area.



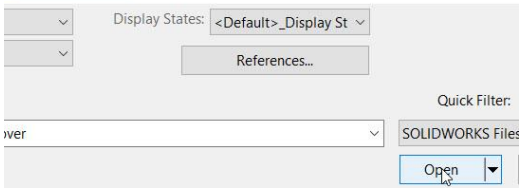
Select File, Save as on the standard toolbar. Save the assembly as USB KEY in the same folder as its parts.

Select Insert component

from the assembly tool bar. Choose browse from the Insert Component dialog box.

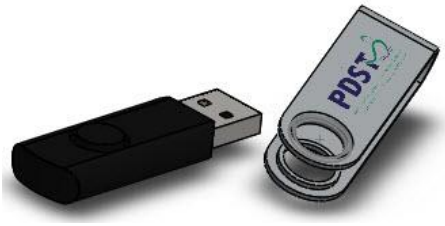
Select **USB Key Cover** and click **Open.**

The part comes in as shown.



If it is not in the correct orientation, click on the x,y or z icon on the screen.

Click anywhere on the graphics area to place the part as shown.

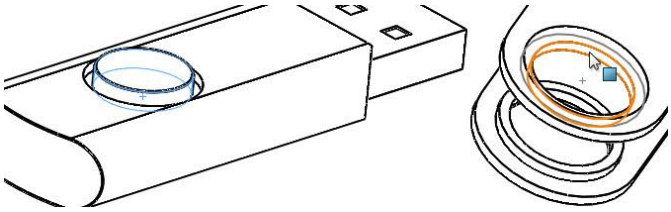


Select **Mate**

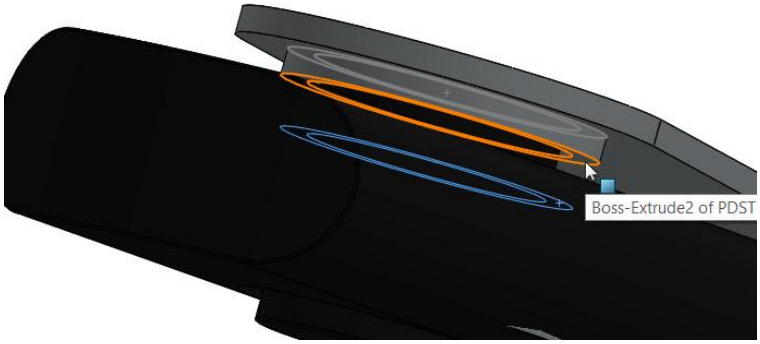
from the assembly toolbar.

Select the inside of the hole on the cover shown and the cylider on the USB key base.

Accept the concentric mate by clicking on the green tick on the ghost toolbar.



Select the bottom of the hole on the **USB base** and the end face of the cylinder on the **USB cover** as shown.



Accept the **Coincident** mate.

Save as USB key assembly.

Note: The assembly must be saved in the same folder as the parts.

PDST