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Advanced SolidWorks Workshop

Surface Modelling

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Introduction

This workshop aims to upskill and develop an understanding of '**Surface Modelling'** and Master Modelling techniques using the SolidWorks software for teachers of Design and Communication Graphics.

Learning Intentions

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

- Explore a modelling technique using planar or non-planar geometry with **zero thickness**.
- Understand and apply **Surfacing features** such as Extruded Surface, Revolved Surface, Lofted Surface, Boundary Surface, Filled Surface, Delete Faces, Trim Surface, Knit Surface and Thicken in a SolidWorks design model.
- Understand the use of **spline** sketching tool in the creation of curves objects and manipulation of their properties to design surfaces.
- Explore the use of **Sketch Pictures** in the creation of a design solution when using Surfaces.
- Develop a better understanding of the creation of photorealistic images when using **Photoview 360**.
- Develop an understanding of **master-modelling techniques** in the design of objects.

Key Messages for this workshop:

- 'Surfaces' can be used to develop more complex design-solutions in SolidWorks.
- Advanced sketching techniques are required when modelling with Surfaces.
- A number of different master-modelling techniques can be utilised to develop a design in SolidWorks.

Introduction to Surfaces

What are Surfaces?

Surface modelling is a method of creating planar or non-planar complex geometry which has zero thickness.

Planar Surface

Non-Planar Surface

Advantages of Surfaces:

1. Surfaces give much more flexibility when creating complex shapes that cannot be done using solid features.
2. Surfaces can be used to build a shape face-by-face rather than all at once.
3. Surface can be used as reference geometry.
4. Surfaces can be more efficient than solid features depending on object.

Disadvantages of Surfaces:

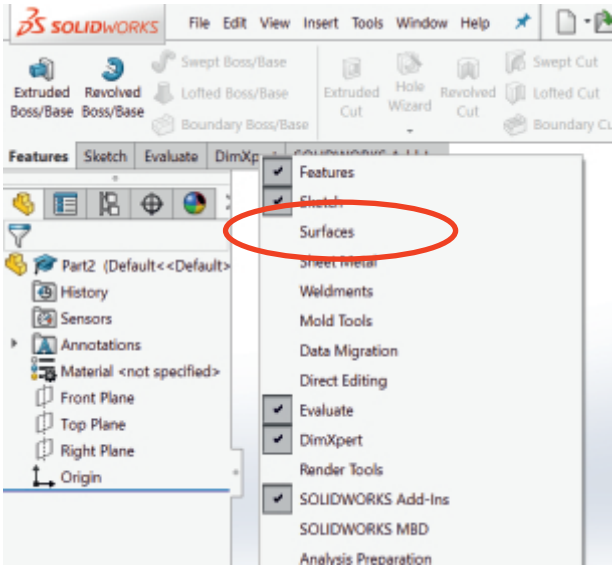
1. Modelling with surfaces is almost always more work than modelling with solids.
2. There is a much more complex workflow when creating surfaces and then converting to solids.

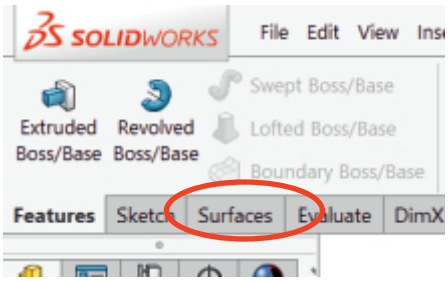
Finding the Surfaces Toolbar

The Surfaces Toolbar is not visible by default.

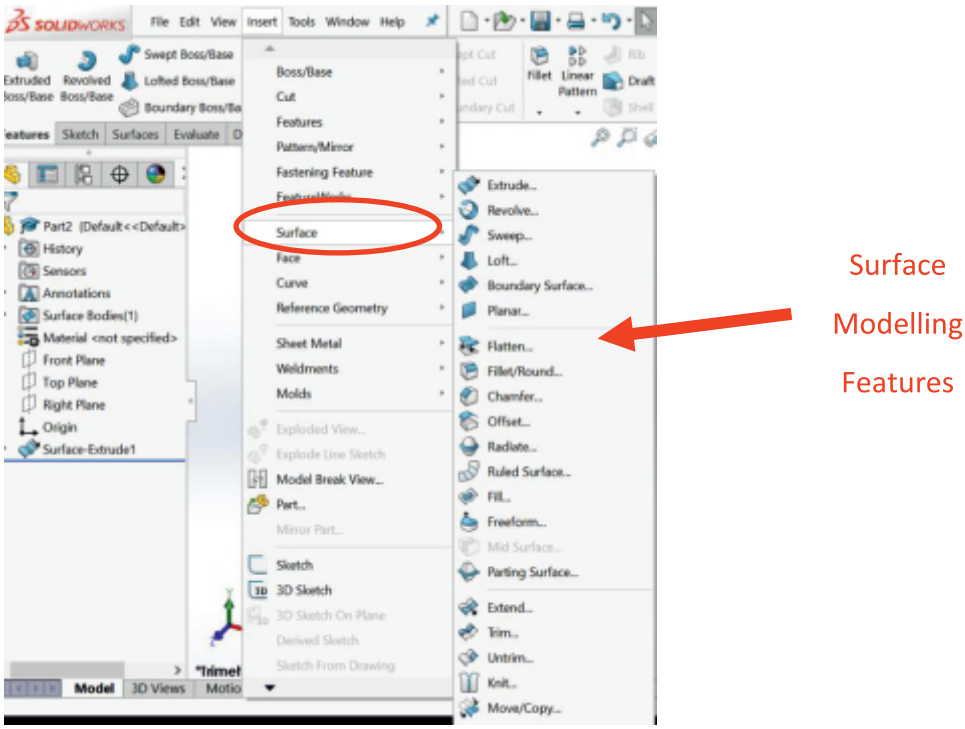
To turn on the toolbar in SolidWorks:

- Open a New part
- **Right-Click** on one of the Ribbon Tabs
- Activate Surfaces by checking the box
- The **Surfaces Tab** will then appear





Surfacing Tool are also available from the main menu in the **Insert – Surface** menu.



Many of the Surfacing features function in the same manner as the solid features but there are a few new terms and features to examine.

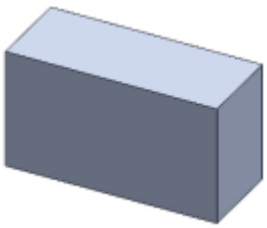
Solid Modelling v's Surface Modelling

The interrelationship between solid and surface modelling is very important to understand before interrogating **Surfaces**. In a solid, every edge is bounded between exactly two faces and solid has a volume.

Solid Model

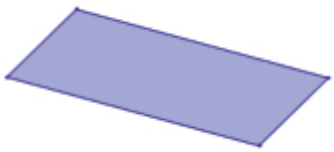
Create a **rectangular** sketch on the **Top Plane** without dimensions.

Extrude Boss/Base the sketch to any height. This is a solid with a mass and volume.

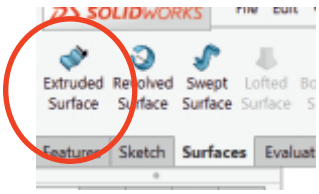


Surface Model

Create another rectangular sketch on the **Top Plane**.

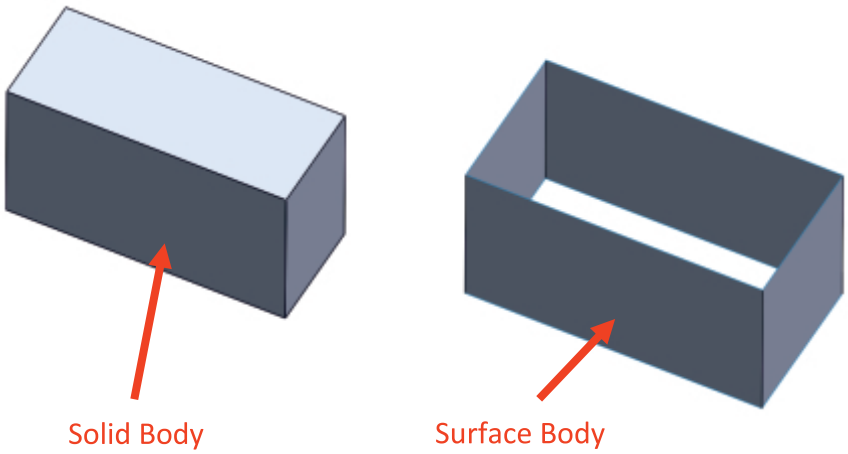


Select **Extruded Surface** from the Surfaces Tab / Menu.

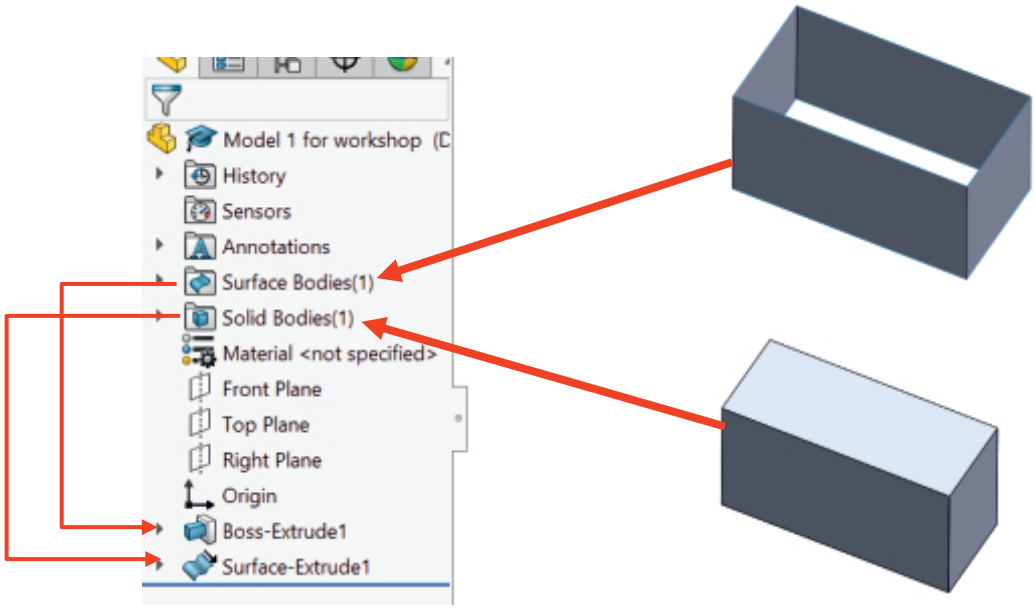


The part now contains a **solid body** and a **surface body**

The **surface body** is a zero-thickness rectangular frame.



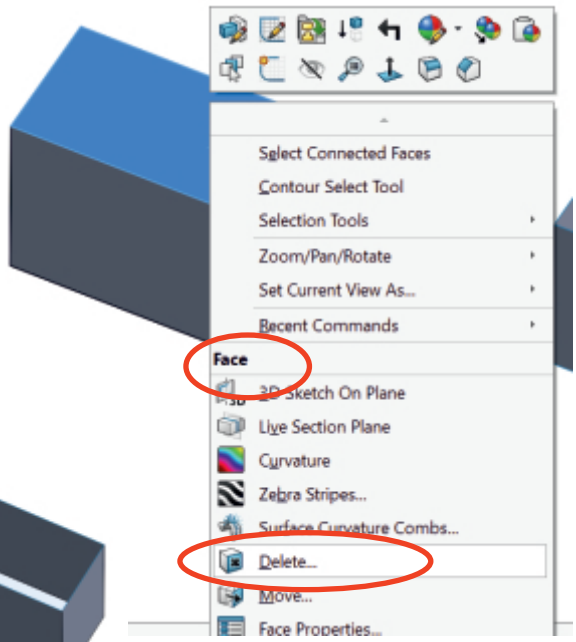
The Solid Body and Surface body also appears in the **Feature Manager** where they can be edited or visibility properties can be changed.



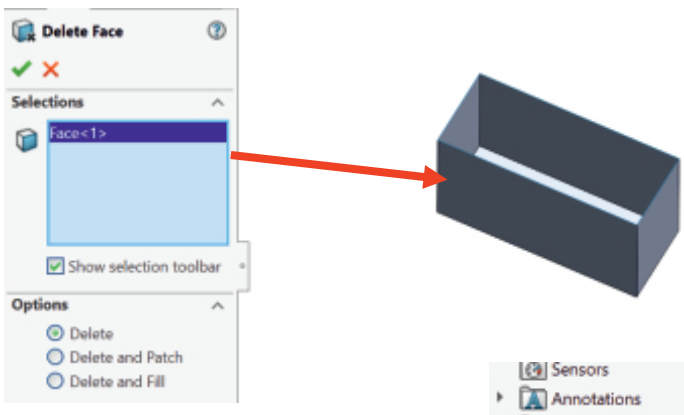
Note: It is very important to understand how and where solid and surface bodies appear in the **Feature Manager** for future editing when dealing with Surfaces.

Turn a Solid into a Surface

Select the top surface of the solid, Right - Click on it.



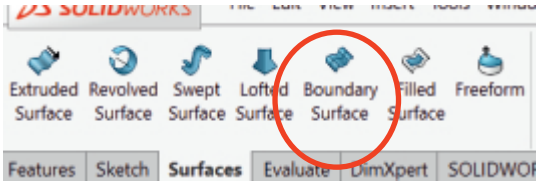
Scroll down to the **Face** section of the menu and select **Delete**.



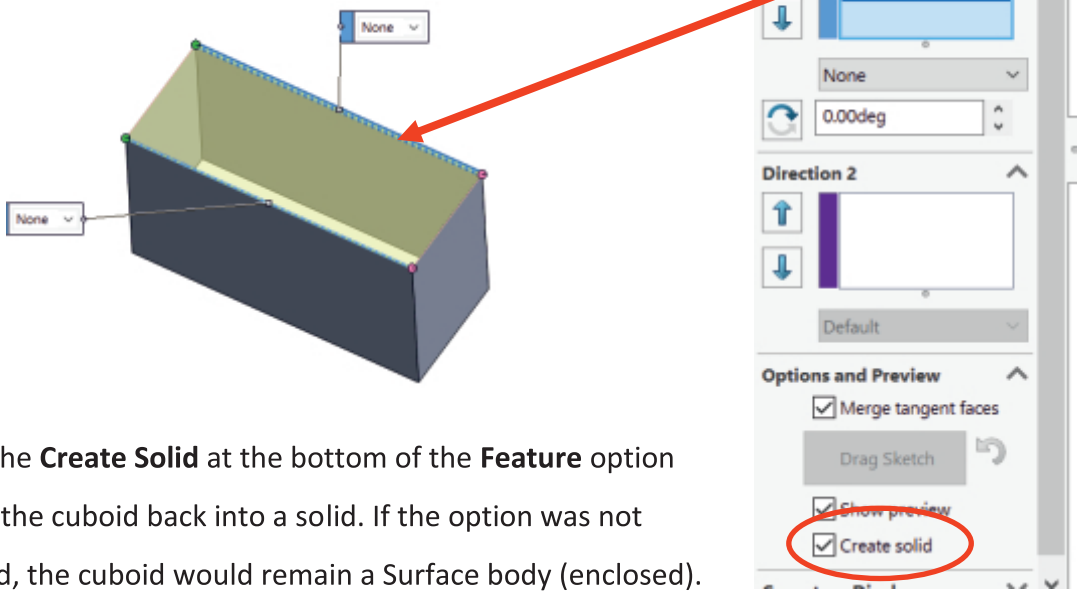
Note: This part now has **2** Surface Bodies.

Turn a Surface into a Solid

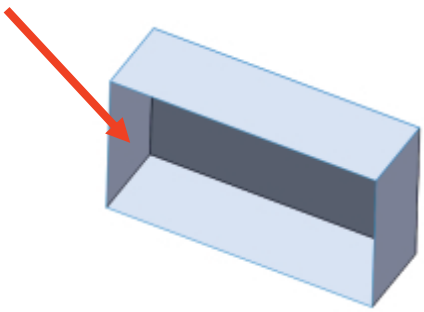
One method of converting this to a Solid would be to select **Boundary Surface** from the Surfaces Toolbar.



Select two of the opposite horizontal edges to fill the ruled boundary of the box.



Select the **Create Solid** at the bottom of the **Feature** option to turn the cuboid back into a solid. If the option was not selected, the cuboid would remain a Surface body (enclosed).



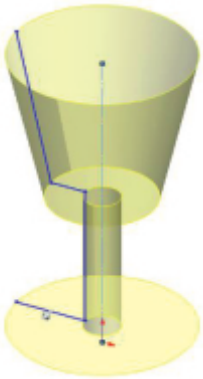
Surfaces are usually an intermediate step to a solid model. SolidWorks enables you to combine the best advantages of solids with the best advantages of surfaces. Solids to Surfaces is a paradigm shift where objects can be created by working between solids and surfaces to achieve the required result.

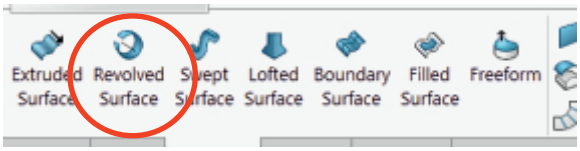
Using Surfaces features

Revolved Surface

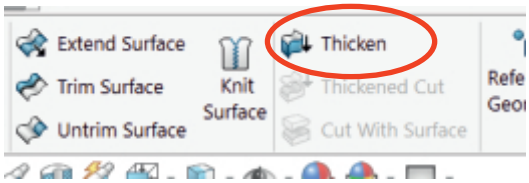
Create a sketch on the Front Plane, to show the outline of a wine glass – no dimension.

Select **Revolved Surface** from the Surfaces Toolbar.

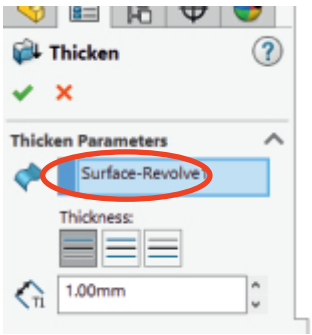


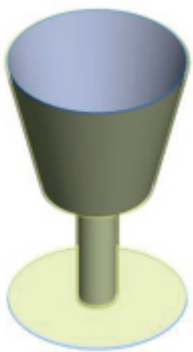


Select **Thicken** from the Surface Toolbar to thicken the glass surface. This will thicken the surface outline of the glass design.



Add a **1mm** thickness to the surface.





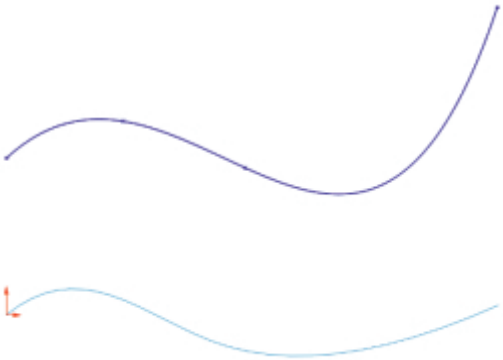
Note: The glass is now a solid.

Boundary Surface, Trim Surface, Knit

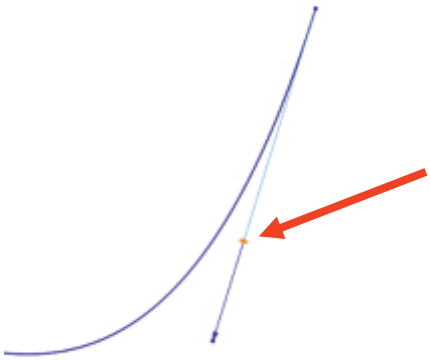
Create a sketch on the **spline** sketch on the Front plane

Extrude Surface this sketch to any dimension.

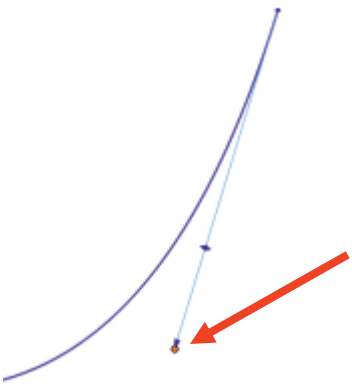
Create another **spline** on the Front plane above the initial sketch.



Select one of the points on the spline to control the curvature of the spline

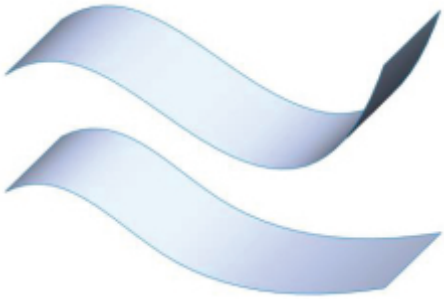


The diamond on the spline handle allows the spline curve to be **steered** or adjusted.

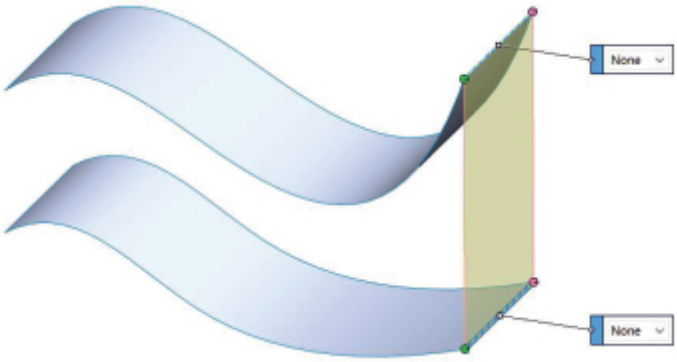


The arrow at the end of the spline handle allows **additional length** to be added to the spline curve.

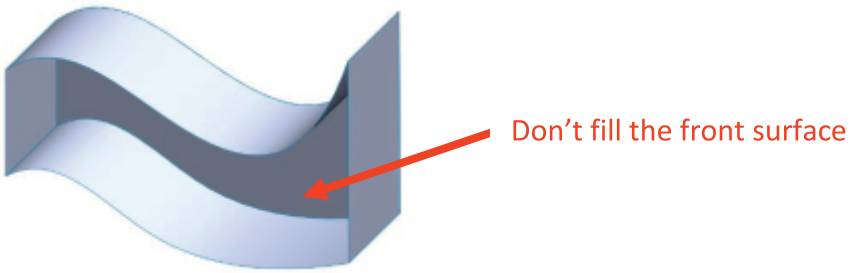
Extrude Surface this spline curve to the same length as the previous



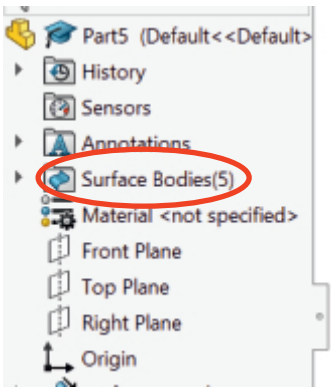
Boundary Surface to complete the shown surface.



Complete the other surface faces with **Boundary Surface**

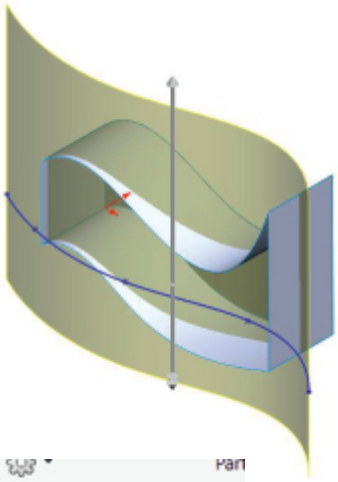


Note: This model is constructed with **5 Surface bodies** as can be seen in the **Feature Manager**

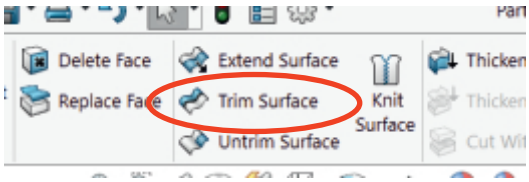


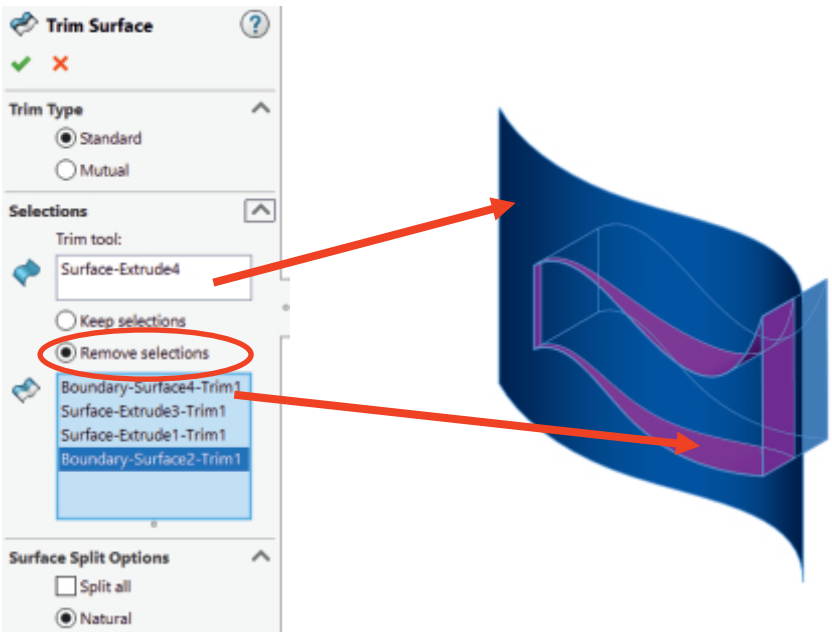
Create a spline sketch on the **Top Plane**

Extrude Surface the sketch in 2 directions to create a surface.

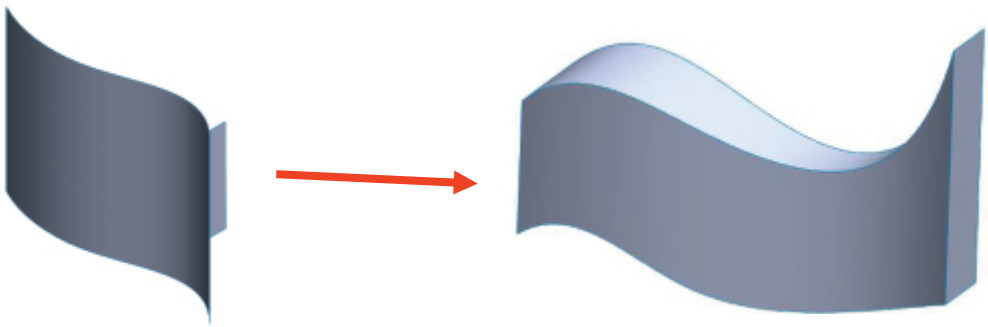


Select **Trim Surface** from the Surface Toolbar



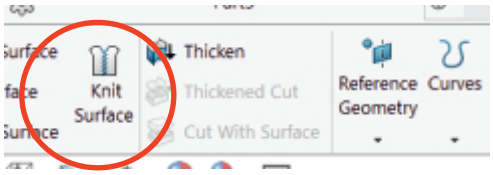


Trim Surface on the front surface



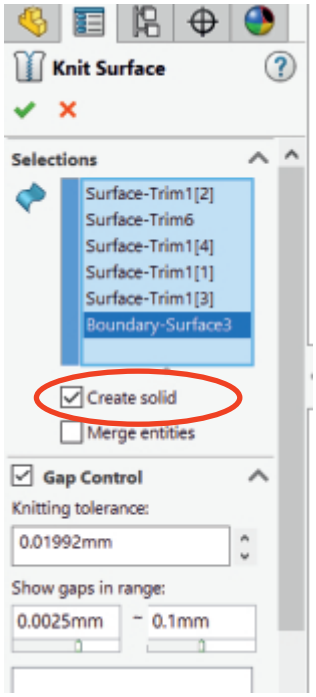
Complete Surface Body

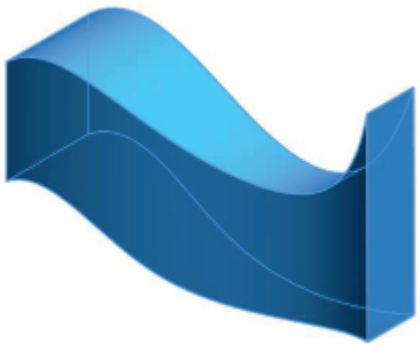
To convert these Surface Bodies into a Solid model, select **Knit Surface**



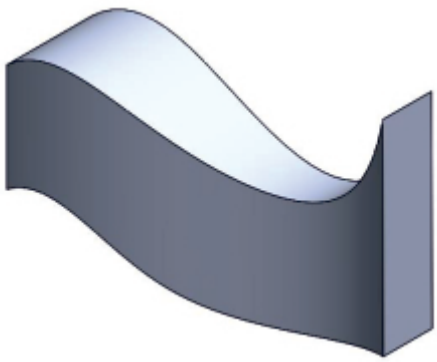
Select the **6 surfaces** of the object in the selections window.

Ensure that the **Create Solid** check box is ticked.





The object is now a **solid**.

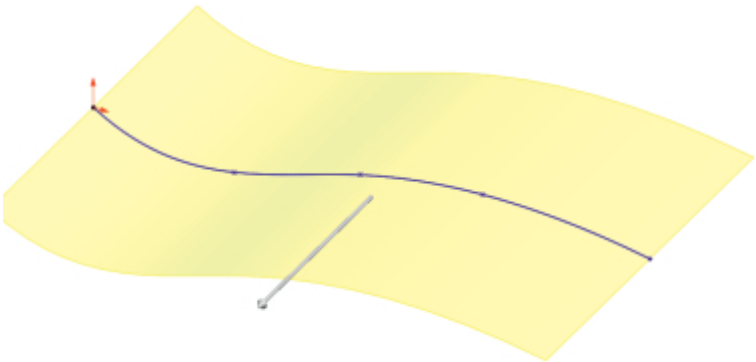


Exercise 1: Shoe Horn

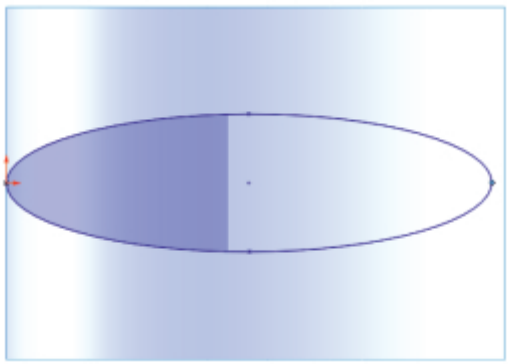
Design attempt at a shoe horn

Create a **spline** sketch on the Front plane of the side profile of the shoe horn.

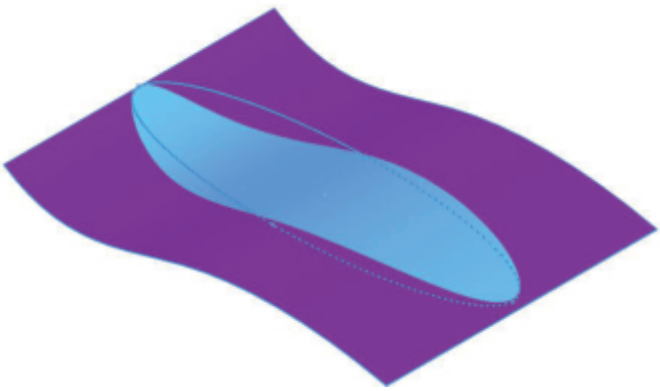
Extrude Surface this sketch Mid Plane



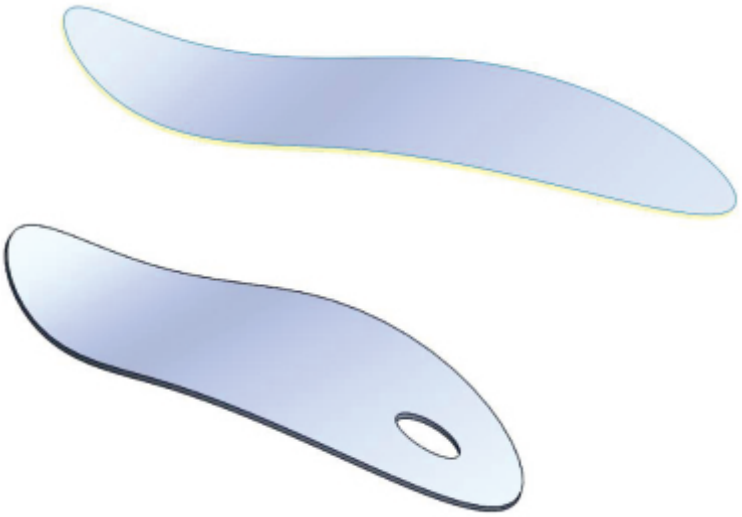
Create an **elliptical** sketch on the Top plane as the outline of the shoe horn.



Trim Surface to create the shoe horn design.



Thicken the surface to 1.5mm



Add an elliptical cut.

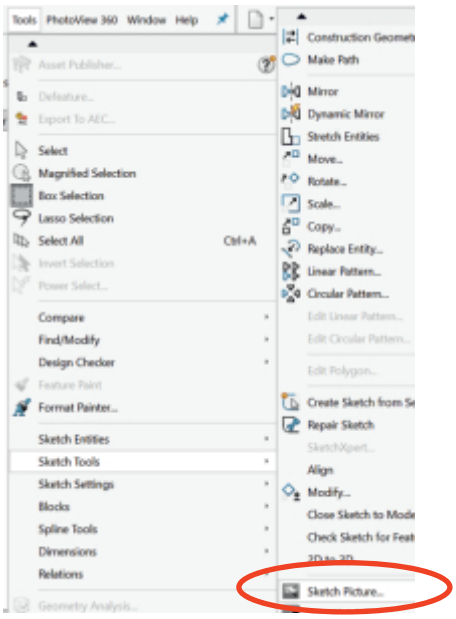
Note: This is **one** method of creating a quick design solution.

Modelling the shoe horn with sketch pictures

Start a **New Part**: This time Sketch Pictures will be used to create an accurate representation of the show horn.

Create a centreline sketch on the Top Plane.

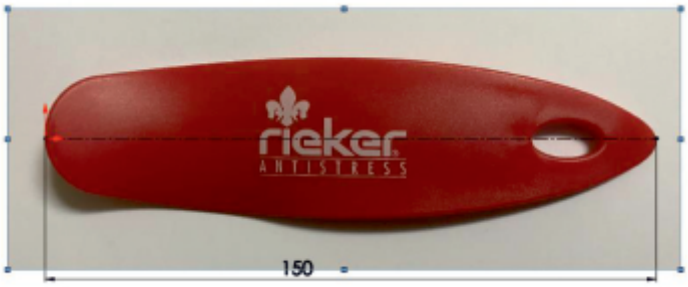
Insert a **Sketch Picture** from Tools menu, **Sketch Tools** and **Sketch Picture**.



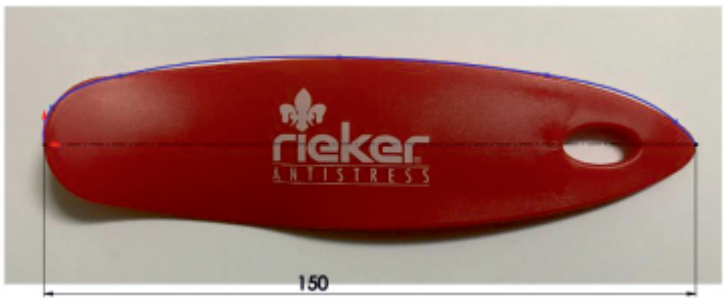
Locate the plan of the shoe horn and insert the image.

Note: Elevation and Plan picture of the object have been taken.

Resize and reposition the image, using the image handles so that the plan image of the shoe horn is position symmetrically on the centerline.

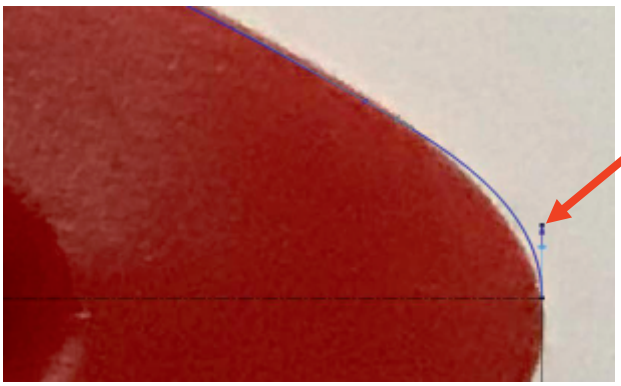


Sketch a **Spline** about half the shoe horn using the end positions of the centerline as start and finishing points.

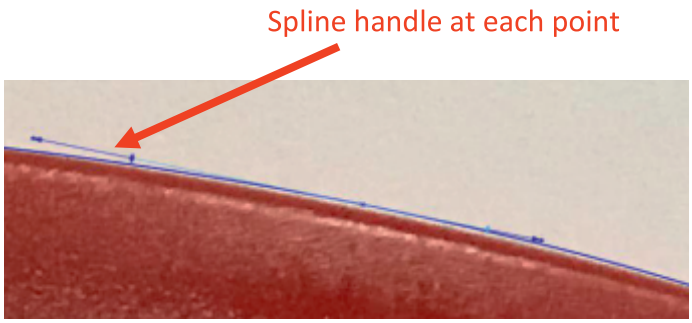


Add a **vertical relation** to the endpoints of the spline.

Note: This vertical relation ensures **tangency** when mirrored.



Make any adjustments to the Spline using the **handles** so that it matches the picture.

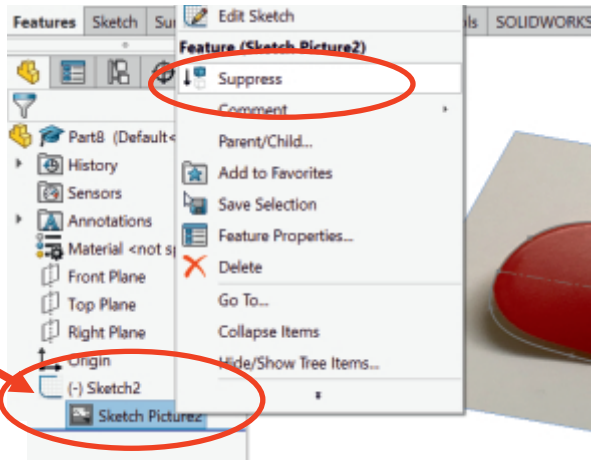


Complete the spline to match the plan picture.

Mirror the sketch across the centerline.

Suppress the Sketch Picture to hide the picture in the sketch

Right-click on the Sketch Picture in Feature Manager

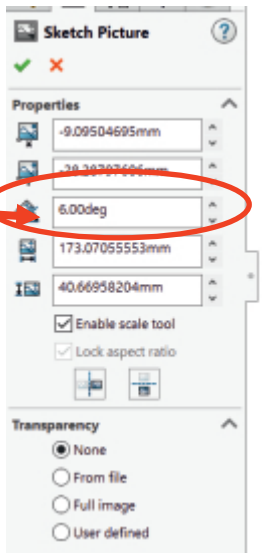


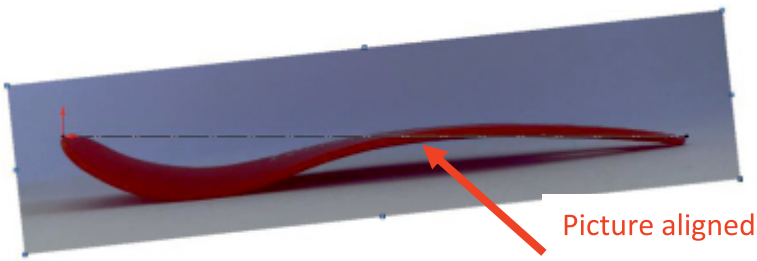
Sketch displayed without picture

Create a sketch on the **Front Plane**.
Sketch the same **centreline** on the Front Plane.

Insert the **Elevation Picture** of the shoe horn.
Adjust the size and angle of the picture so that the end points of the shoe horn align with the centerline

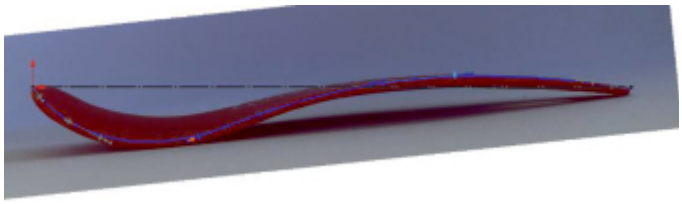
Angle adjustment in Feature Manager





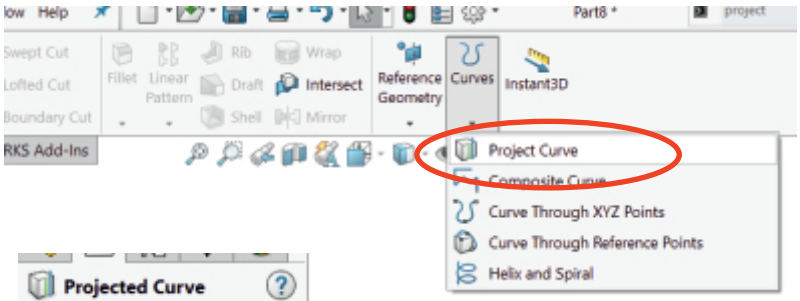
Picture aligned with centerline

Sketch a **spline** along the edge profile of the shoe horn starting and ending on the centerline. Adjust the spline using the spline handles.

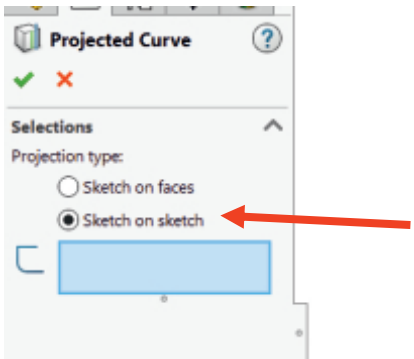


Suppress the Sketch Picture in the Feature Manager to hide the picture.

Select **Project Curve** from the Curves menu.

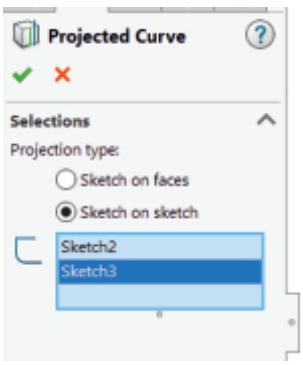


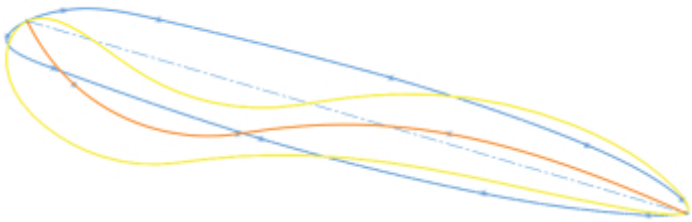
Select **Sketch on sketch** to project the sketches onto each other.



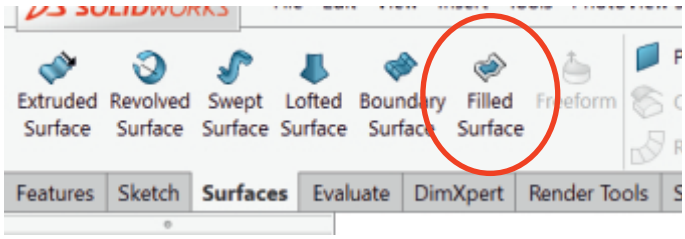
Sketch on sketch to project the sketches together to get outer rim.

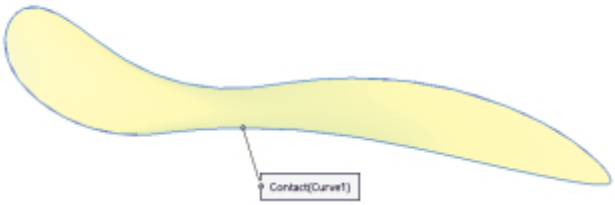
Select the **Elevation** and **Plan** sketches.



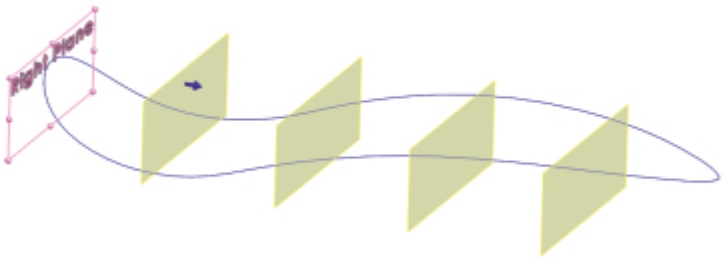


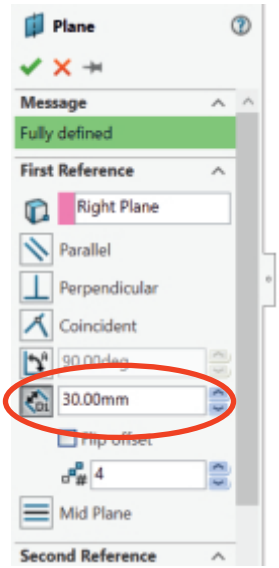
A **Filled surface** could be created using this **Project Curve**. But some **constraint curves** need to be inserted first.



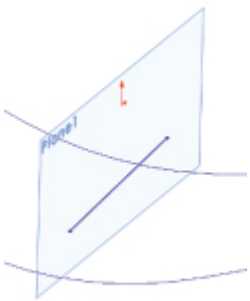


Insert **4** planes parallel to the **Right Plane** at 30mm intervals.

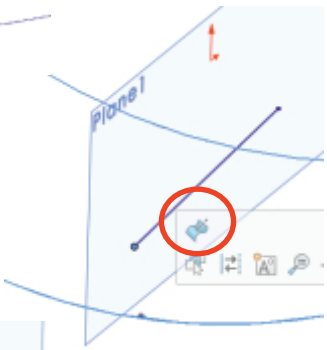




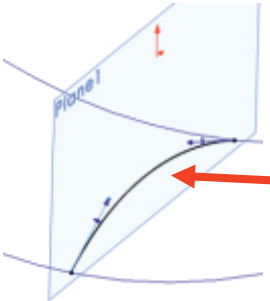
Sketch a two-point **Spline** on Plane 1, not connected to the Project Curve.



Add **Pierce Relation** between endpoint of Spline and Project curve on both sides.

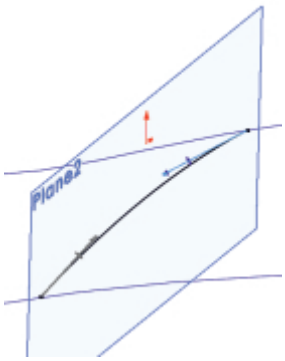


Adjust the curve on the spline using the handles.

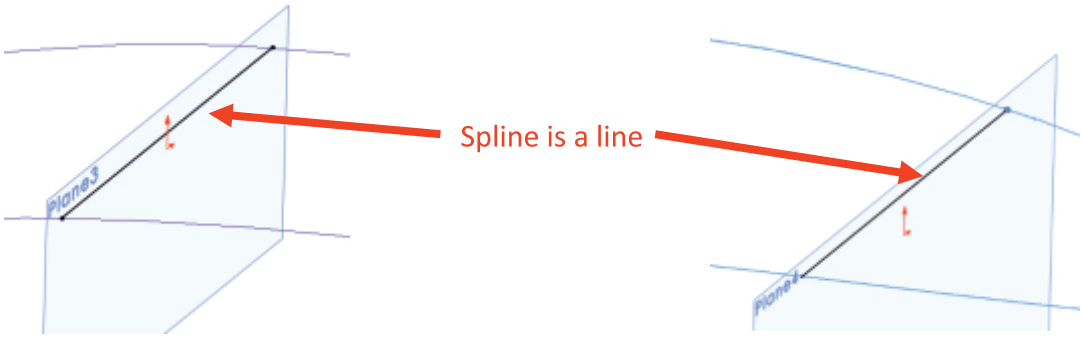


The end profile of the shoe horn is curved.

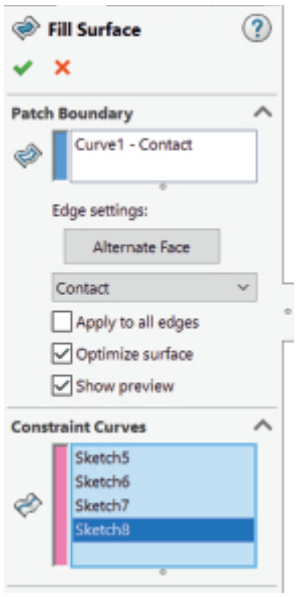
Add **Spline** sketches to the other Planes, with the curved profile graduating to a line on Planes 3 and 4.

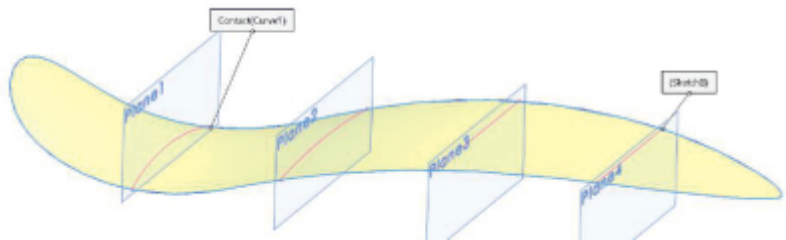


Note: Add a pierce relation to all endpoints and the project curve.

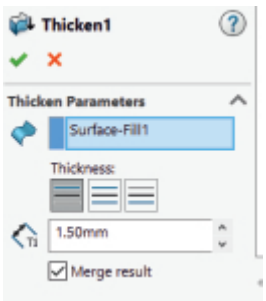


Fill Surface – the project curve is the patch boundary and select the splines on intermediate planes as Constraint Curves.

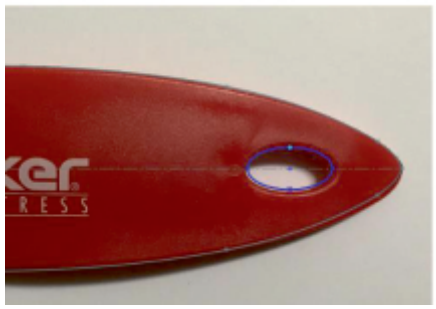




Thicken the surface to **1.5mm**.

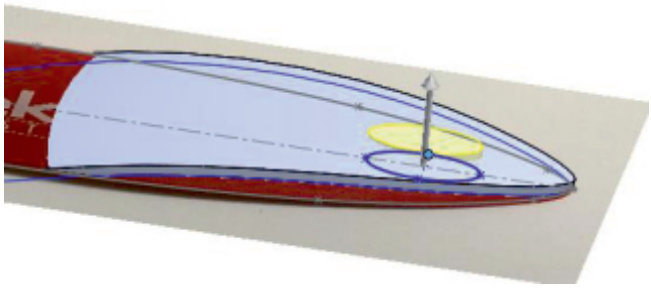


Unhide and **Unsuppress** the Sketch containing the Sketch Picture to see the image.

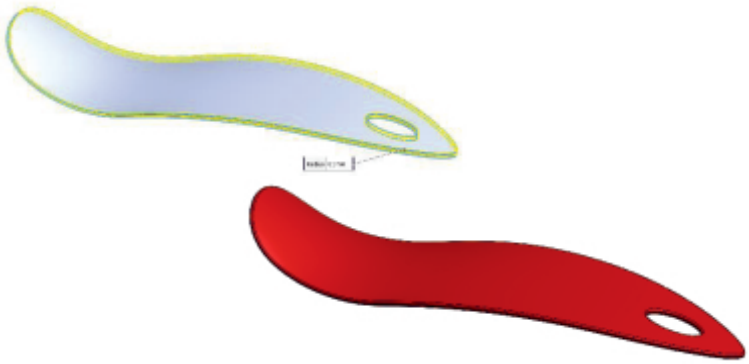


Sketch an **Ellipse** about the hole on the **Top Plane**.

Extrude Cut the ellipse through the shoe horn.



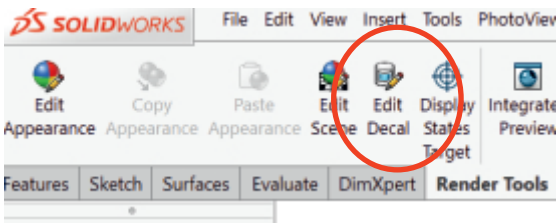
Add a 0.5mm **fillet** about all edges



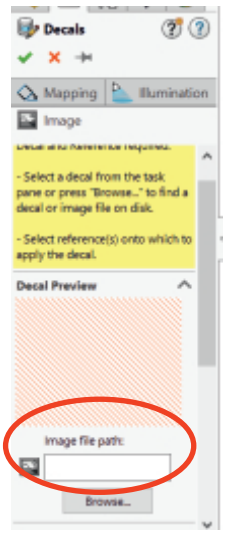
Add a **High Gloss Plastic** to the shoe horn

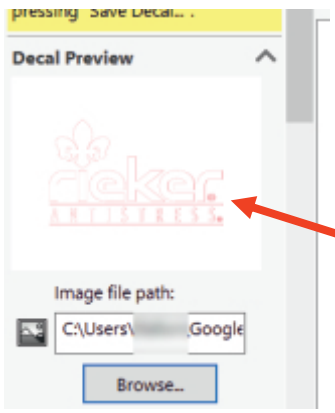
Add a Decal

Select **Edit Decal** from the Render Tools Toolbar



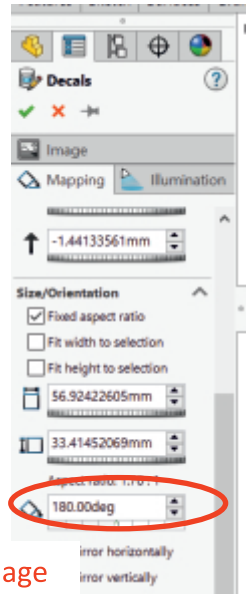
Browse for the required **Decal** image. Locate the Rieker image





Preview of Decal image

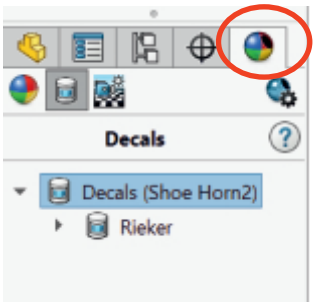
In the **Mapping** options, the image size and orientation can be edited. Edit the image as shown.



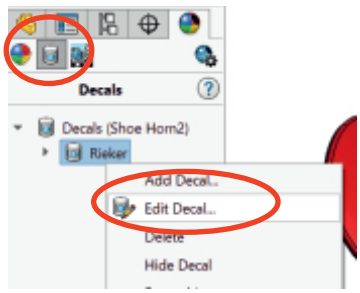
Accept the Decal.

The background of the image needs to be removed

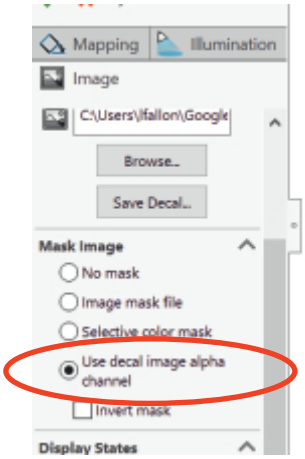
To **Edit** the existing **Decal**, select the Display Manager.



Select **View Decals**, right-click on the Ricker decal and **Edit Decal**.



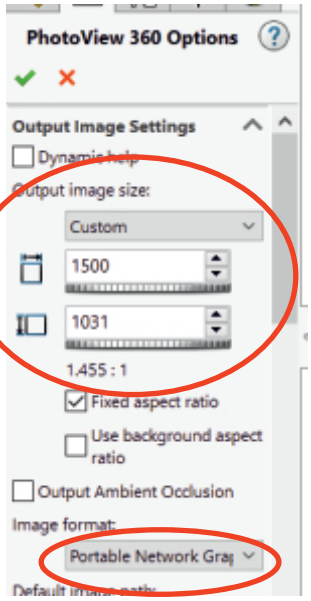
In the Mask image options, **select Use decal image alpha channel** to remove the background.



Background removed on image

Create a Photorealistic image of the shoe horn

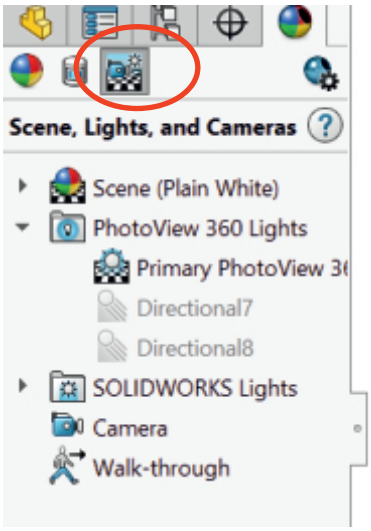
Select **Options**, to increase the **output size** of the image, **image format** (png) and image **Render Quality**.



Edit the Lights

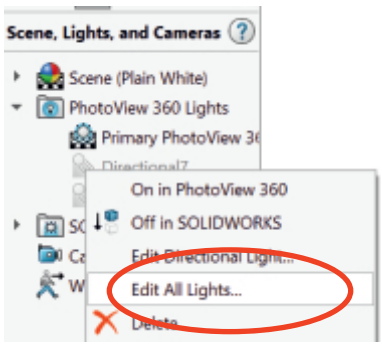
There are times when **scene** and **light** settings may need to be edited to achieve a better quality image.

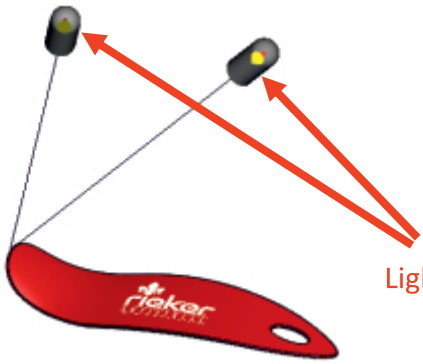
Select the **View scenes, lights and cameras** option in the feature manager.



Lights can be turned on/off, repositioned and illumination adjusted here.

Right-click on one of the light, select **Edit All Lights**.





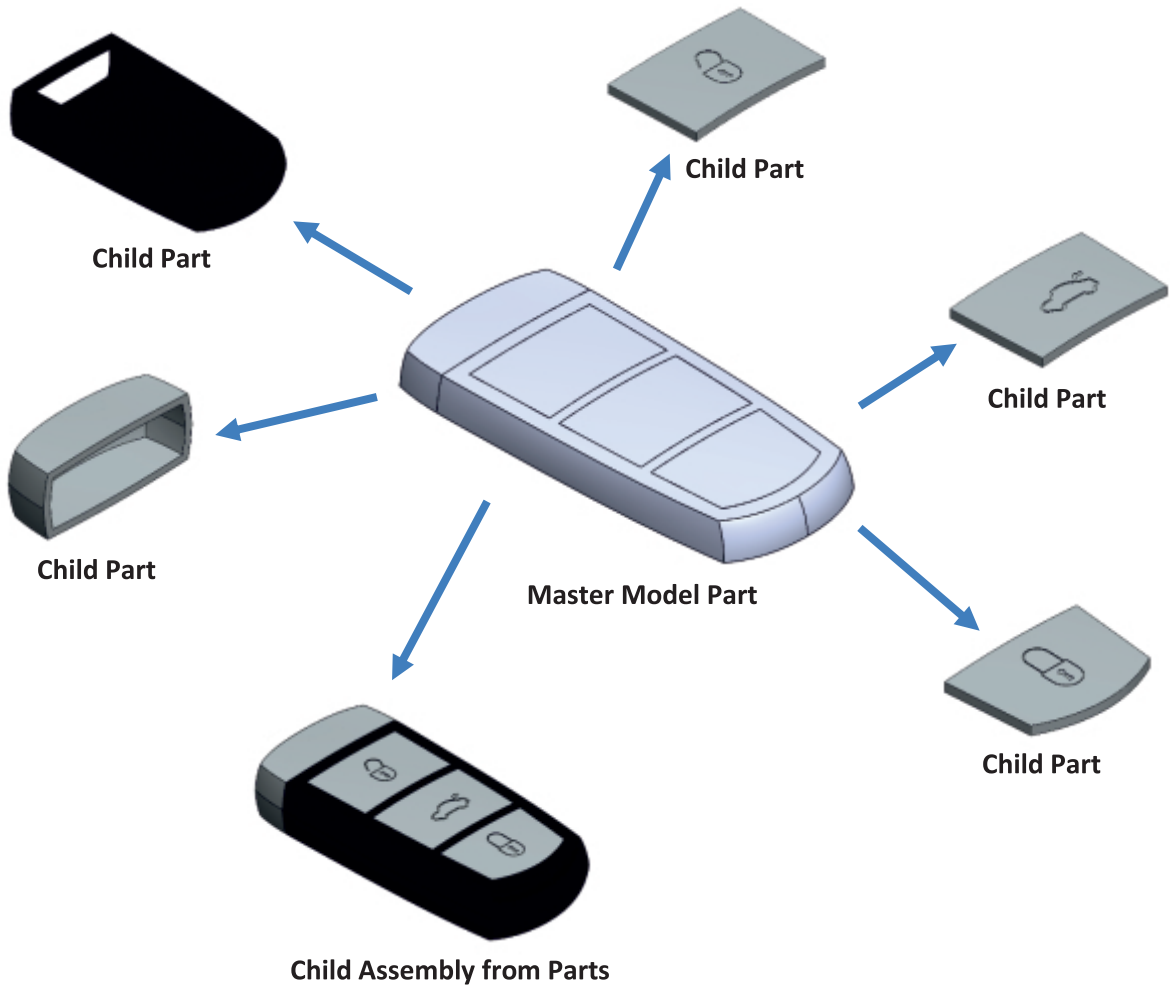
Light positions can be moved

Master Modelling Technique

Master Modelling Technique can be used to design objects that contain multiple interlocking parts/bodies to form the overall shape in one part. This technique allows control over large amounts of geometry from one part, thereby controlling all child components/parts from this one position.

Example of Master Modelling Technique:

The initial **Master Part** controls all the geometry contained in its child parts/assembly. Not all Master Model techniques offer this level of control though. When using the Bottom-up method there is no parametric control following completion.



Possible Methods of Master Modelling

There are numerous methods in which Master Modelling Techniques can be utilised to design an object. These are **four** possible methods:

1. **Bottom – Up**

   This means that a solid master model (parent) is split into solid bodies, which are saved as individual part files (children).

2. **Top – Down**

   The entire master model is brought into each child part and the design for each component proceeds from there.

3. **Save As Copy**

   Use the **Save As Copy** options in the save options to copy the master part into the remaining parts.

Advantages of Master Model Technique

1. Overall control can be achieved from the Master Model in the design and assembly of the final object.
2. The design process is much quicker.
3. The ability to perform rapid model changes, because the key relationships between the parts are maintained.
4. It saves assembly time as the Master Model creates the relative position for all parts.

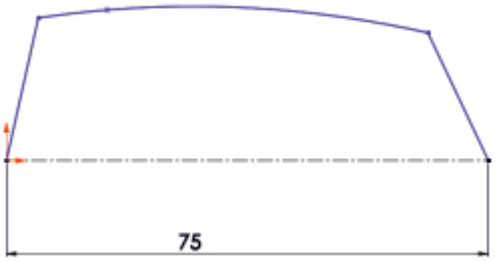
Exercise 2: Car Keys

To examine the use of the Master Model Technique in association with Surfaces, a car key will be used as an example. The key contains multiple parts on a curved surface which must finished co-planar when complete.

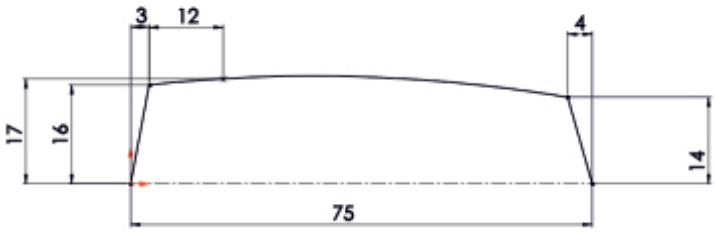
Modelling the Master Part

Create a sketch on the **Top Plane**.

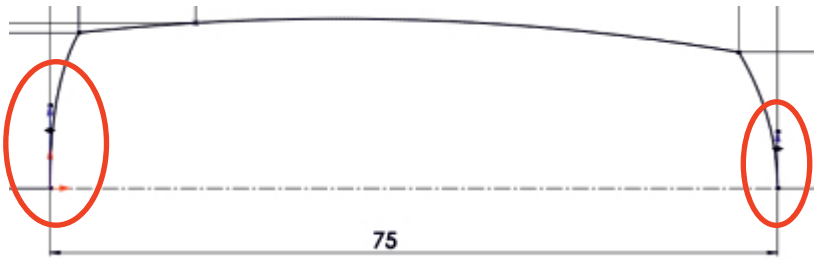
Sketch the shown **centerline** and **Spline** (two and three point) curves.



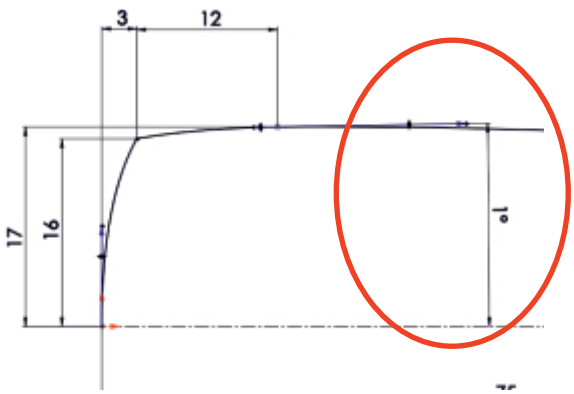
Add dimensions to the sketch

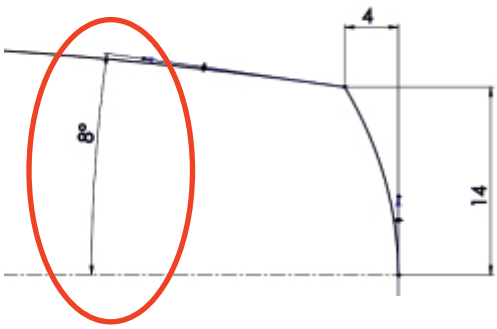


Add a **Vertical** relation to the shown Spline handle



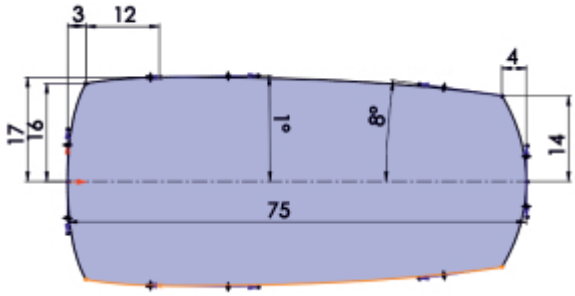
Add the shown dimensions to the **spline handles**.



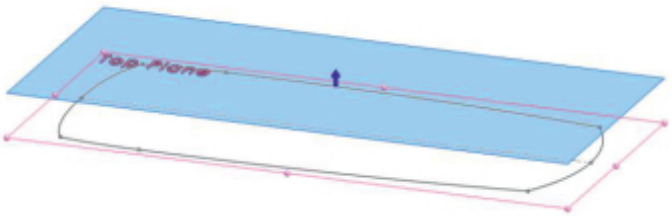


Mirror the sketch.

Rename as '**Base Sketch'**.

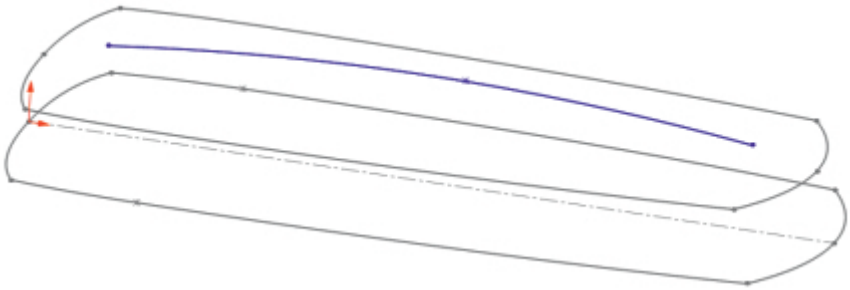


Insert a **Plane 6.5mm** above the **Top Plane**.



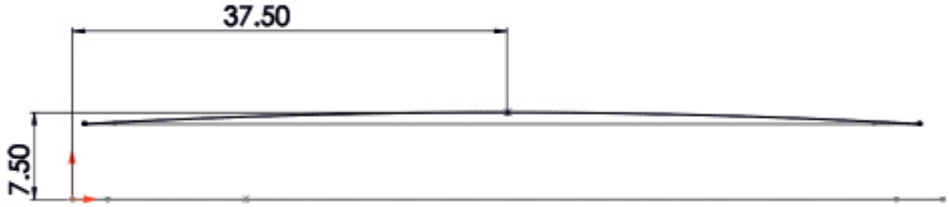
Create a sketch on this **Plane**, **offset** the curves as shown. Rename as '**Top Sketch'**

New sketch on **Front Plane**. Sketch a 3 point **Spline**.

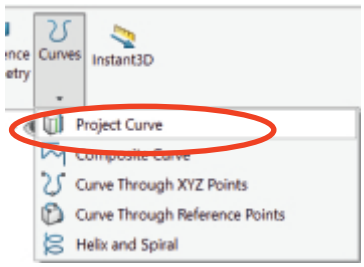


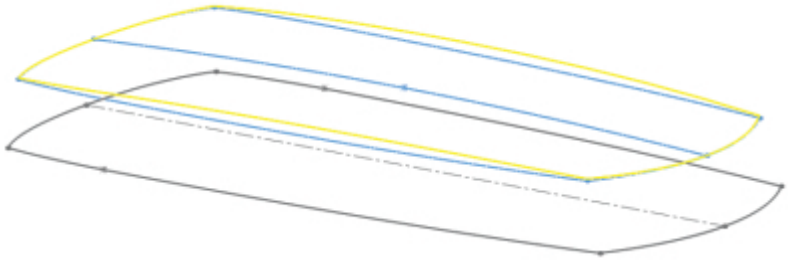
Add a **Pierce** relation between the endpoints of the spline and project curve – to connect.

Add dimension to third point. Rename sketch as '**Centre profile'**.

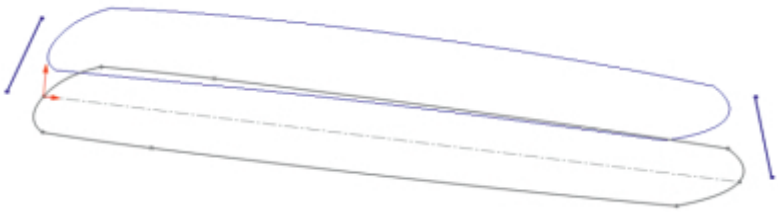


Select the **Project Curve** feature. Add the **Top Sketch** and **Centre Profile** to the selection (sketch to sketch option).

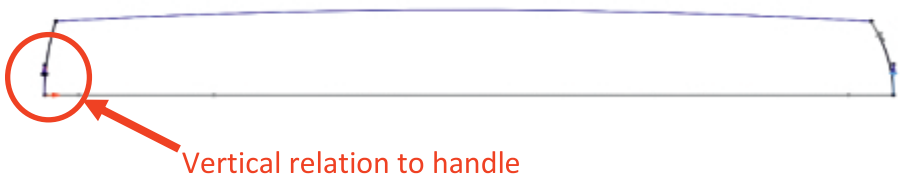




Sketch two point splines on **Front Plane**.



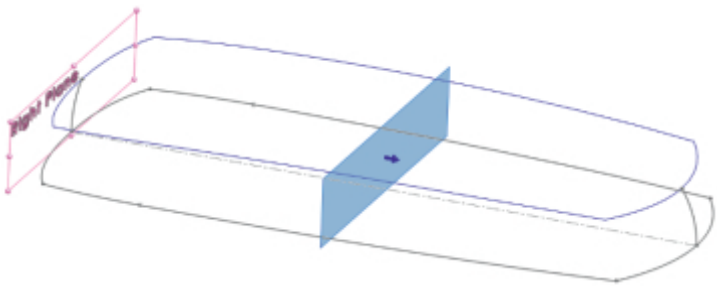
Pierce the endpoints of the Splines to the connecting curves.



Add **Vertical** relation to the base handle of each spline (to ensure tangency when mirrored).

Rename sketch as '**End Profile'**.

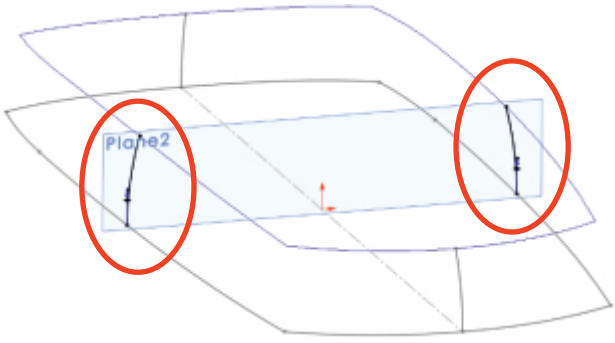
Insert **Plane** which is Parallel to the **Right Plane** and distance of **37.5mm**.



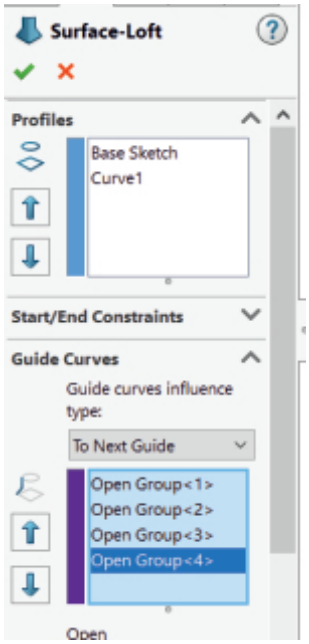
Sketch **2 point spline** on each side.

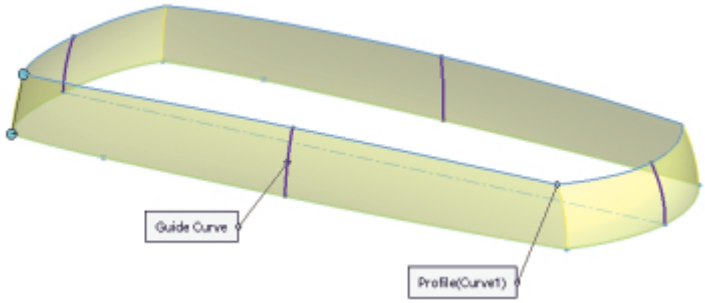
Add **Pierce** relation to the base and project sketches.

Add **Vertical** relation to the base handle of each.

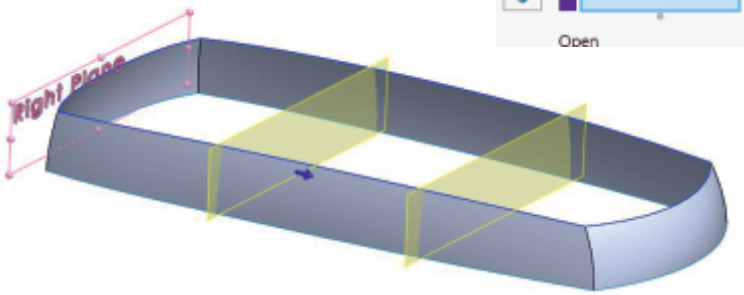


Lofted Surface: select base and project sketch as the **Profiles**. The end and side profiles must be selected individually as **Guide Curves**.





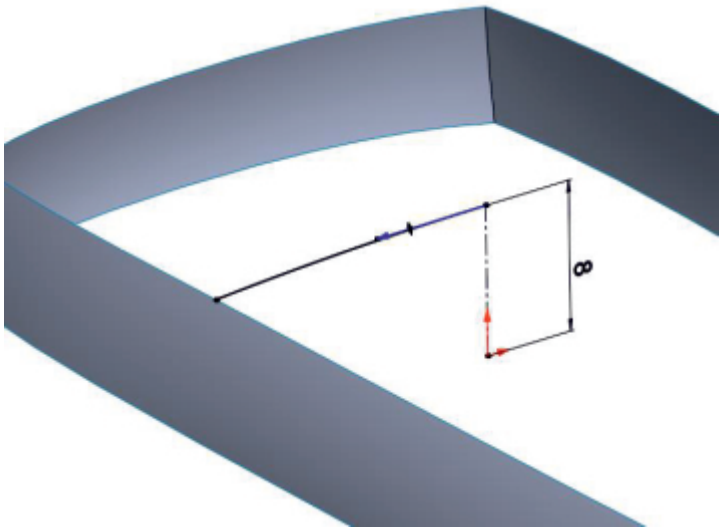
Insert two planes parallel to the **Right Plane** at distances of **25mm**.



Sketch 2 point **spline** and **centerline** on plane 3.

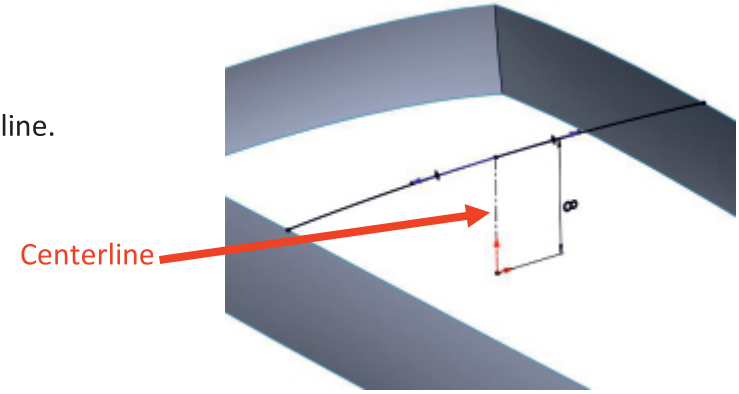
Pierce spline to the **Project Curve** and centerline.

Horizontal relation to the top spline handle and dimension.



Mirror the spline about the centerline.

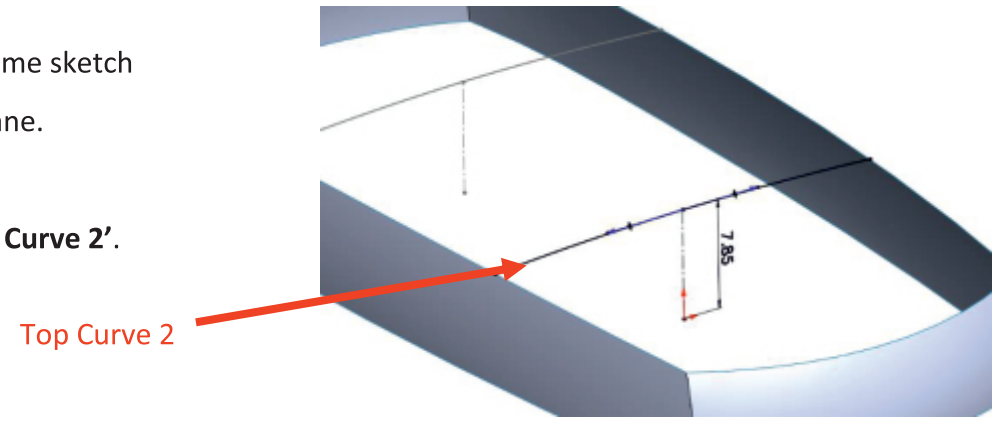
Rename as '**Top Curve 1'**.



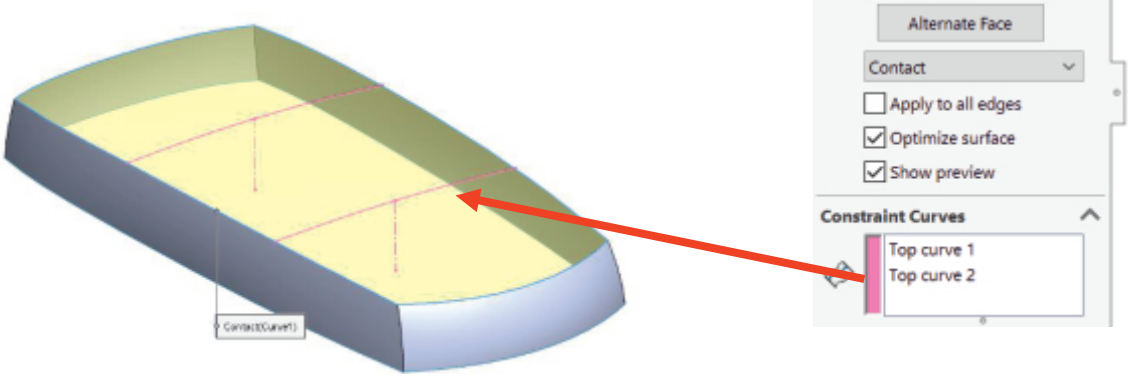
Complete the same sketch

On the other plane.

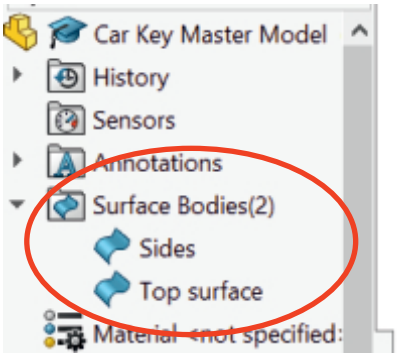
Rename as '**Top Curve 2'**.



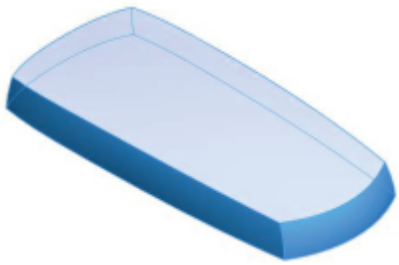
Fill Surface on the top surface, selecting the top edges of the Project curve as the **Patch Boundary** and Top curves as **Constraint curves**.

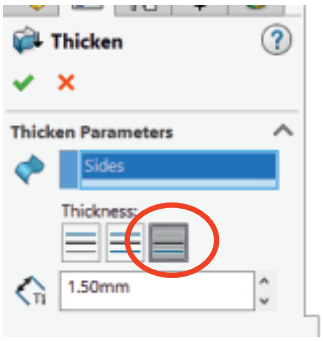


Note: There are now two Surface Bodies in this part.

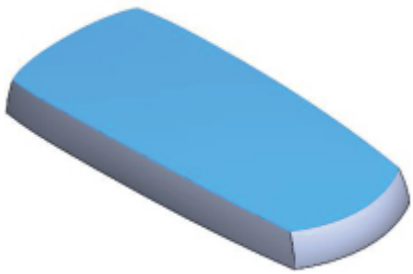


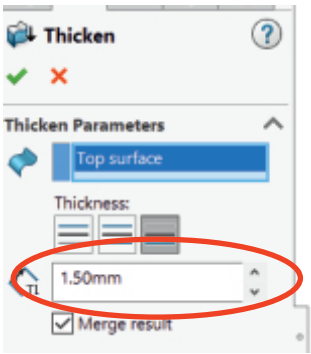
Thicken the side surface inwards at a distance of **1.5mm**.





Thicken the top surface using the same settings.

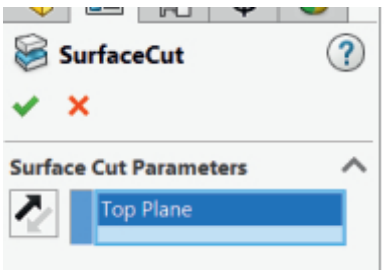


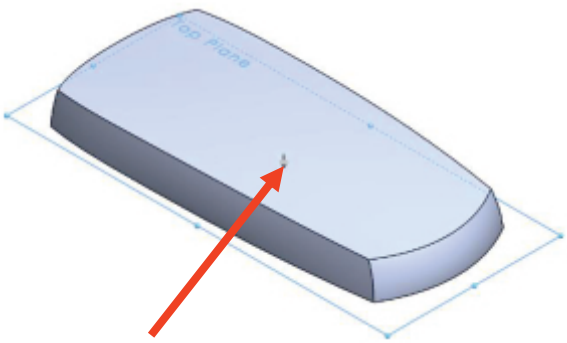


Note: The Surface Bodies have now disappeared into the Solid.

Cut with Surface, using the **Top Plane** as the **Surface Cut**.

This will remove the bottom edge rim created from the thicken surface.

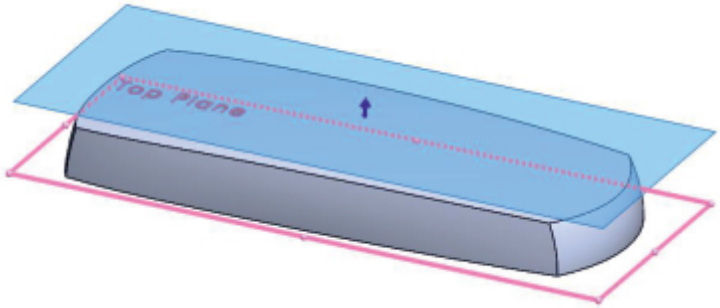




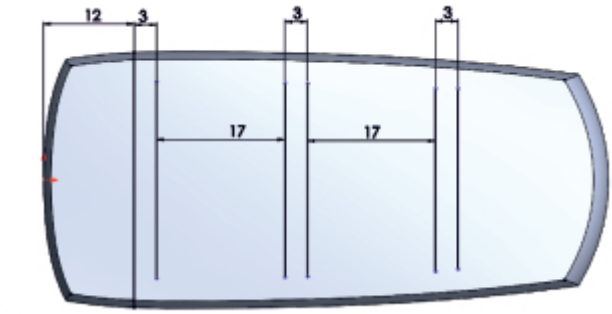
Insure that the correct direction is selected for the cut

Split the Part into Bodies

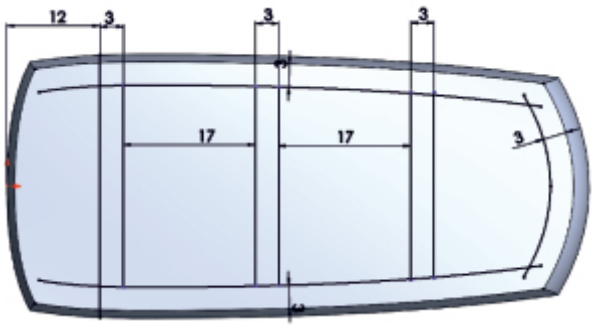
Insert a plane 10mm above the **Top Plane**.



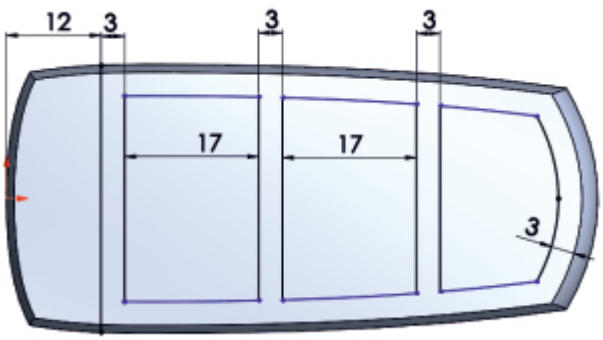
Sketch the shown lines and dimensions on the new Plane.



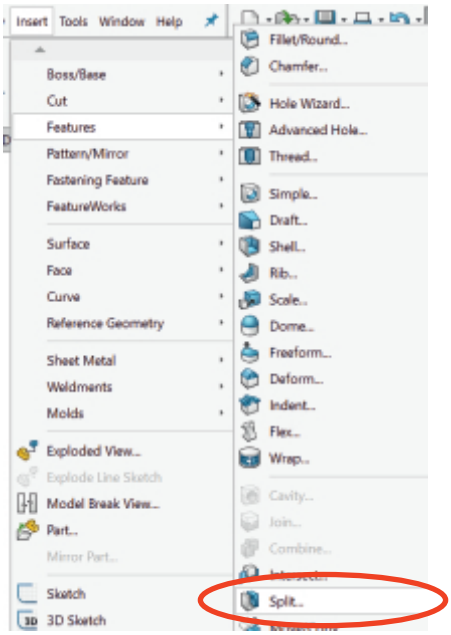
Offset Entities 3mm from the top edges.



Trim Entities on the unwanted lines.

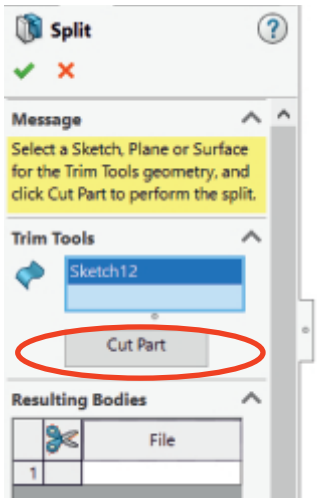


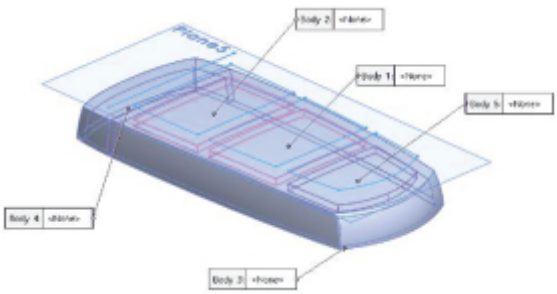
Select **Split** from the Insert, Feature Toolbar



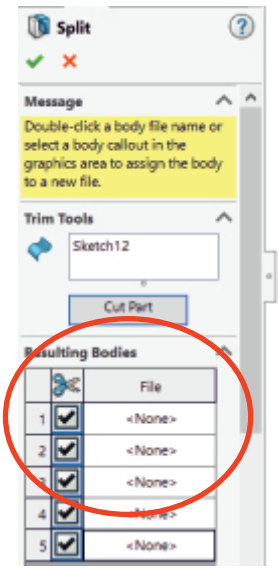
Select the Cut Sketch as the **Trim Tool**.

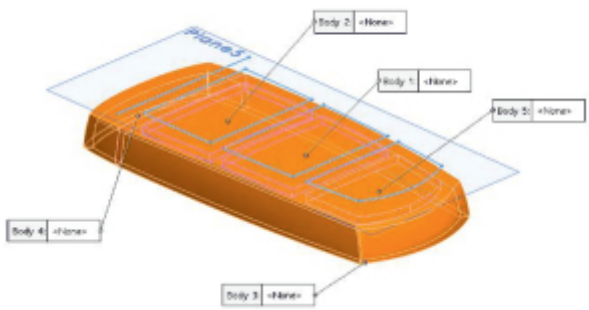
Note: Nothing will happen until **Cut Part** is clicked.



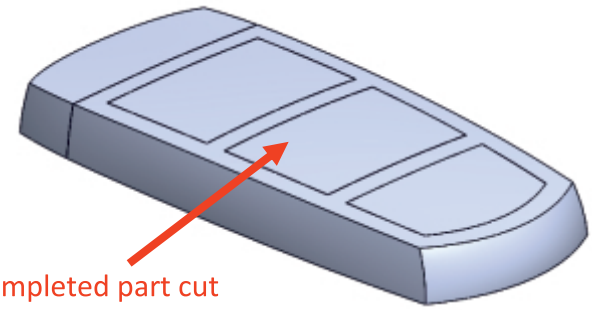


Once **Cut Part** is selected, select the bodies which need to be cut. In this case – select all the bodies in either the **graphics** window or **feature manager**.

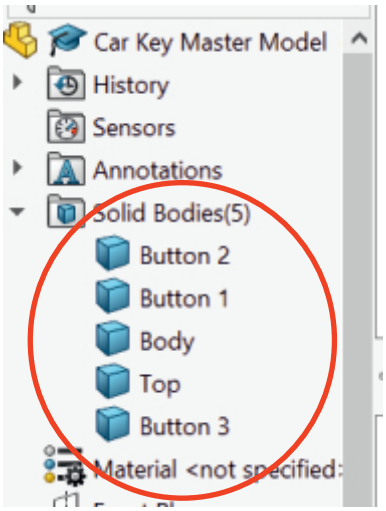




Note: The part now contains **5 bodies** as seen in the Feature Manager. Rename these bodies to an appropriate name.



Completed part cut into 5 bodies (parts).



Ensure the Part is save as '**Car Key Master Part**'.

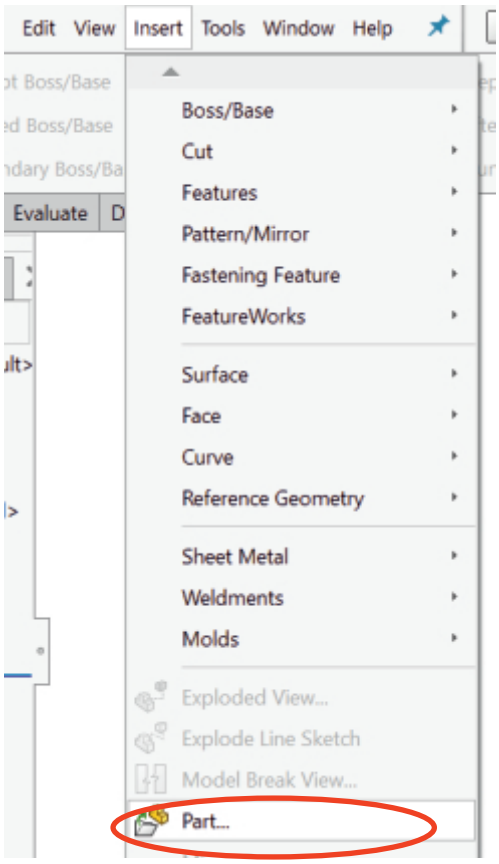
This Master Part will now be used to create all the individual part of the car key. The method used in this tutorial will be the **'Top – down method'**. Where this part will be inserted into a new part and the body extracted from the master part. The advantages of this method is that the master model controls the geometry in all the parts and any editing will be pushed out to the child parts.

Car Key Body Part

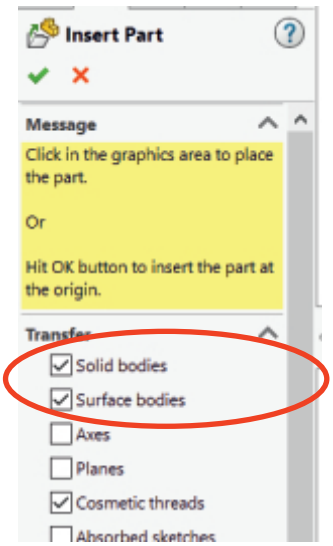
Create a **New Part** and save as Car Key Body

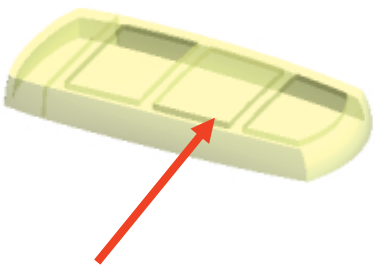
Select Insert and **Part**

Navigate to the **Car Key Master Part** and select.



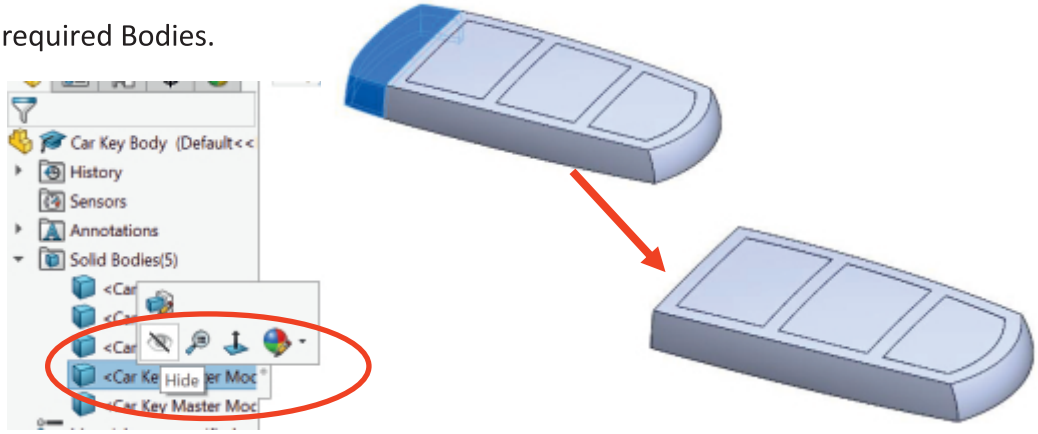
Ensure that the **Solid Bodies** and **Surface Bodies** are checked when inserting the part.



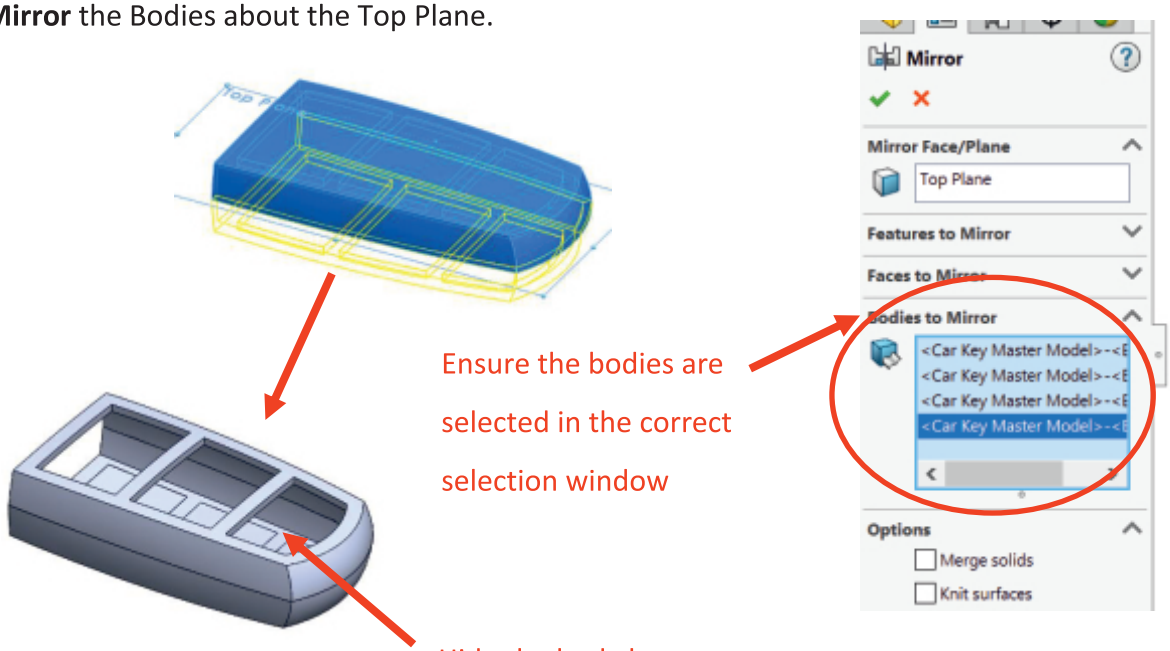


Preview of the Master Part

Hide the unrequired Bodies.



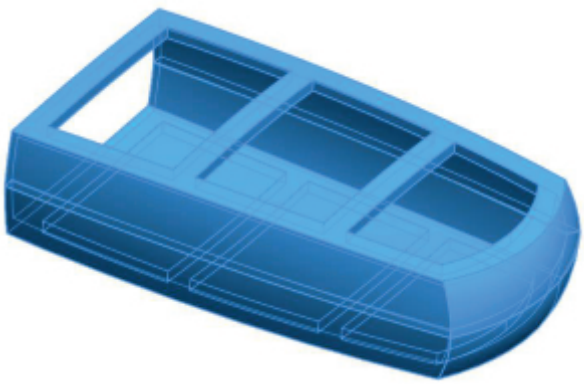
Mirror the Bodies about the Top Plane.

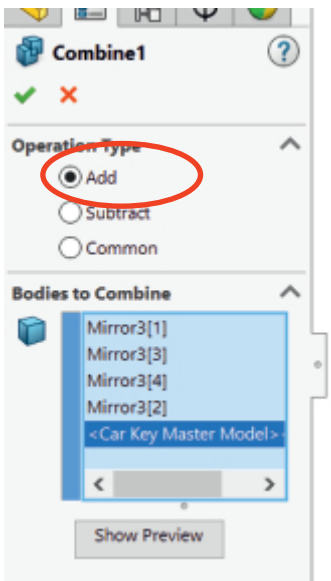


Ensure the bodies are selected in the correct selection window

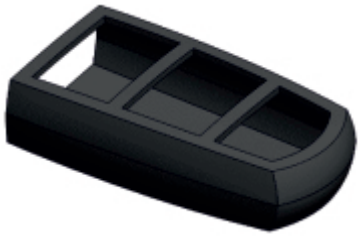
Hide the body buttons on the key

Combine the bodies into one part by using the **Combine** feature. The Operation type is **add** and select all bodies.





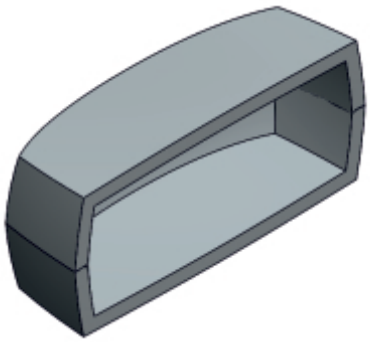
Add a **Medium Gloss Black** plastic to the the completed part. Save.



Car Key Top

New Part and save.

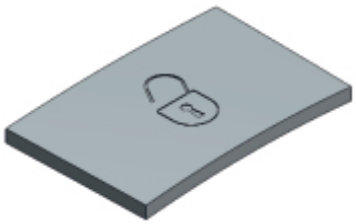
Insert the **Car Key master model** into the Part and hide the unrequired bodies.

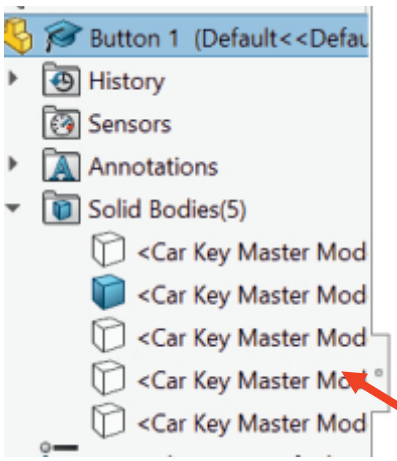


Mirror the body about the Top Plane and add a **Chromium Plate** appearance.

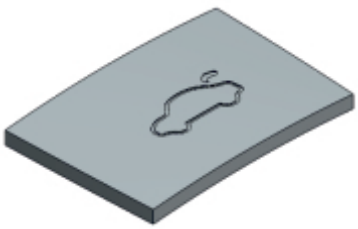
Car Key Buttons

Complete the 3 Car Key buttons using the same procedure – **Insert Part**, **Hide Bodies** and continue editing the part.

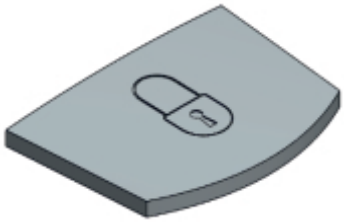




Hide the unrequired bodies in the feature manager



Complete Button parts



Complete Assembly of the Car Key

Insert each of the parts into a New Assembly.

Note: The master part creates the relative position for each of these parts, making it very quick to assemble the key.

Note: When submitting the student assignment, ensure that the master part is in the folder with the SolidWorks model. The **Pack and Go** options in SolidWorks will copy the Master Part into the folder with the other parts.

Notes

Notes

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