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

Advanced SolidWorks

Glass Suction Tool

- Surfacing
- Indent
- Deform
- Master Modelling

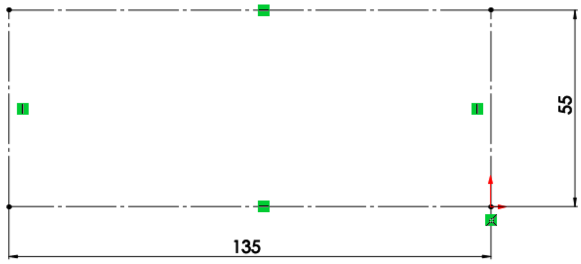
Table of Contents

Glass Suction Tool

Glass Suction Tool

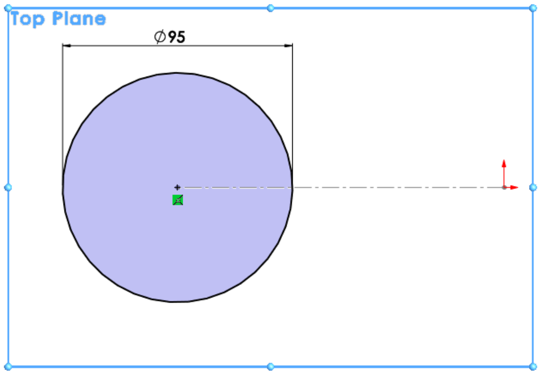
Creating the Multibody Master Model

Create a sketch using **Corner Rectangle** on the **Front Plane**.



Make for Construction & **Dimension** as shown.

Rename as '**Construction Sketch'.**

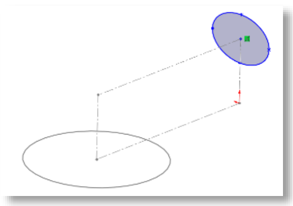


Create a sketch on the **Top Plane**

Sketch a **Circle** with the Centerpoint Coincident with outer edge of construction sketch.

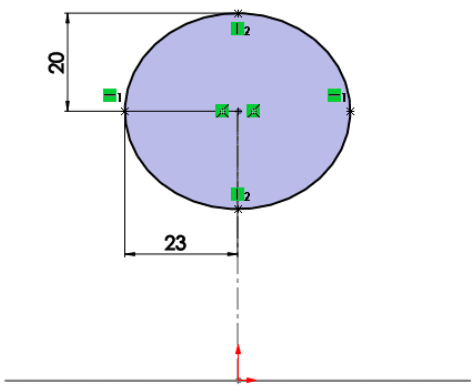
Dimension circle 95mm diameter.

Rename as **'Base Profile'**.



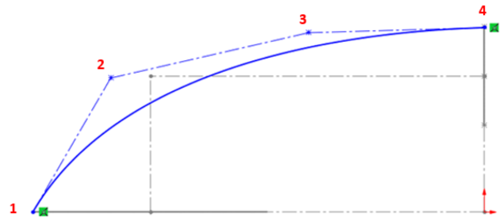
Create a sketch on the **Right Plane**. Sketch an **Ellipse** and add a **Coincident relation** to the top of the Construction Sketch.Add a **Horizontal relation** between the outer points of the ellipse.

Dimension the ellipse as shown.



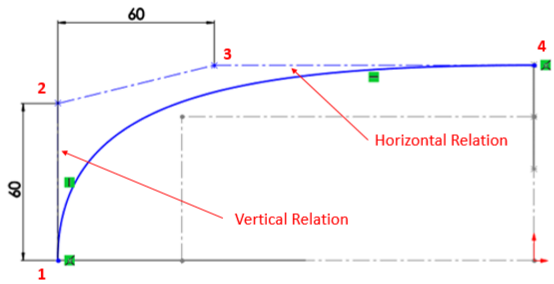
Rename as **'Ellipse Profile'**.

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



Select the **Style Spline** command and create a 4-point style spline as shown.

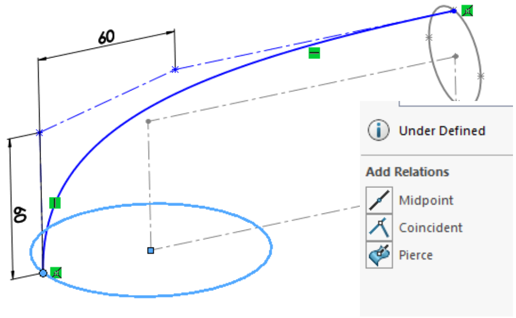
Add a **Vertical Relation** to segment 1-2.



Add a **Horizontal Relation** to segment 3-4.

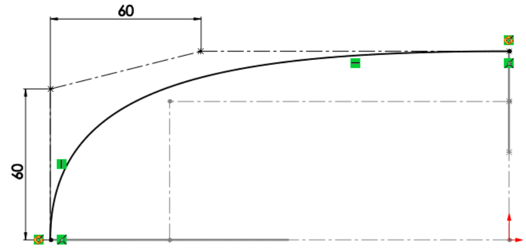
Add **dimensions** as shown.

Add **Pierce Relation** between endpoint of Style Spline (Point 1) and Circle (Base Profile).



Add **Pierce Relation** between endpoint of Style Spline (Point 4) and Ellipse (Ellipse Profile).

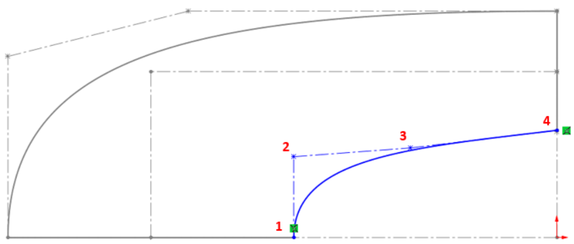
Note: This relation ensures **tangency** when mirrored at a later stage.



The Style Spline will now be fully defined.

Rename as '**Top Curve'**.

Create a **New** sketch on the **Front Plane**.

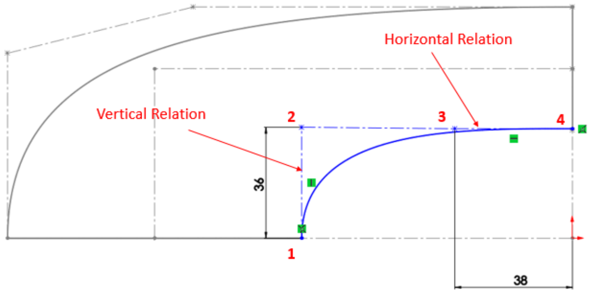


Select the **Style Spline** command and create a 4-point style spline as shown.

Add a **Vertical Relation** to segment 1-2.

Add a **Horizontal Relation** to segment 3-4.

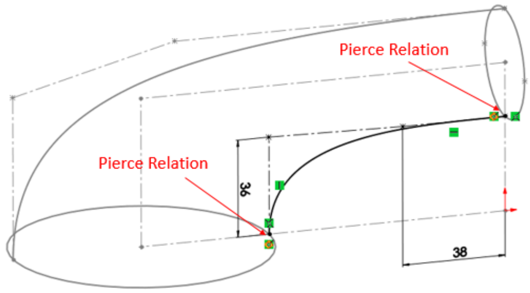
Add **dimensions** as shown.



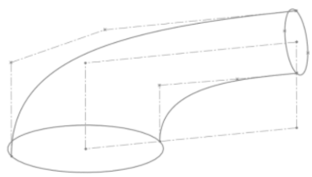
Add **Pierce Relation** between endpoint of Style Spline (Point 1) and Circle (Base Profile).

Add **Pierce Relation** between endpoint of Style Spline (Point 4) and Ellipse (Ellipse Profile).

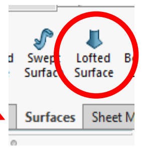
The Style Spline will now be fully defined.



Rename as '**Bottom curve'**.



Surfaces

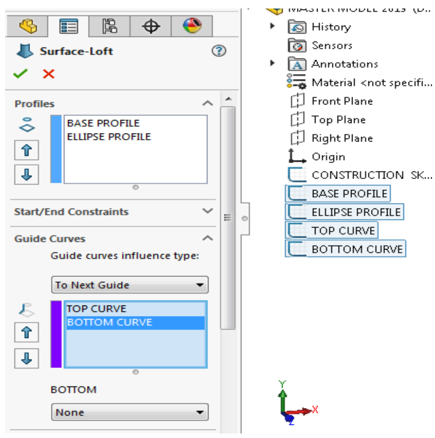


Creating the Lofted Surface

Select the **Lofted Surface** Command from the Surfaces toolbar.

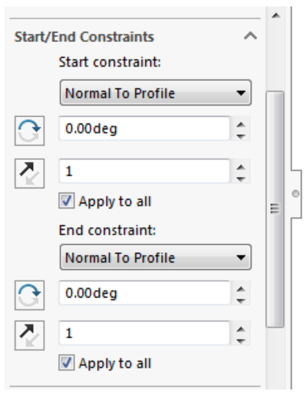
In the Profiles Tab select **Base Profile** & **Ellipse profile** from feature tree.

In the Guide Curves Tab select **Top Curve** & **Bottom Curve**.

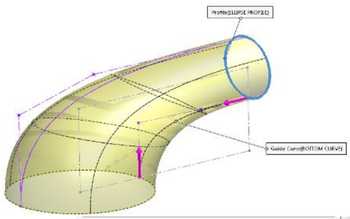


Select the drop down arrow under **Start/End Constraints.**

Select Normal to Profile under both Start and End Constraints.

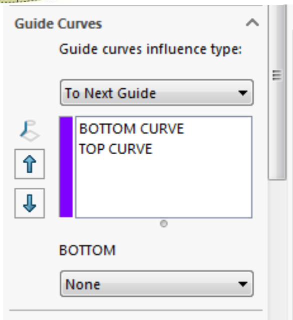


Note; Normal to profile will give a more accurate curve type.

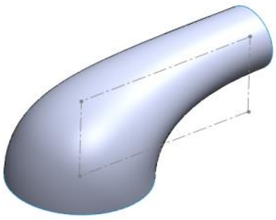


In the Guide Curves Tab, Select **To Next Guide** under guide curves influence type.

Select **None** for Bottom Selection.

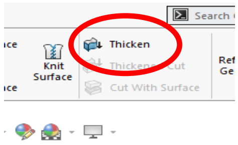


Select **OK** to accept the surface produced.

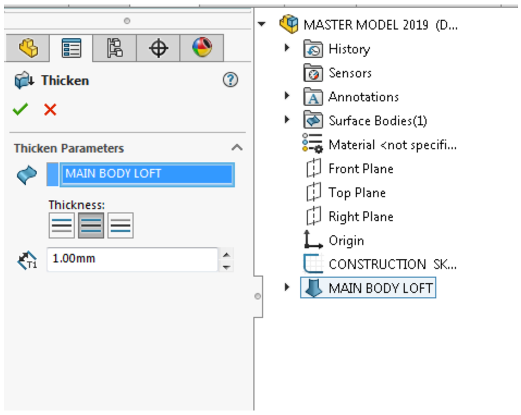


Rename Feature as '**Main Body Loft'**.

Adding a Thickness to the surface



Select **Thicken** from the Surface toobar.



Under the Thicken Parameters Tab, You are required to select a **Surface** to thicken, Select **Main Body Loft** from the feature tree.

Note; Under the Thickness Tab there are three options.

Select **Thicken Both Sides**.

Select **1mm** Thickness.

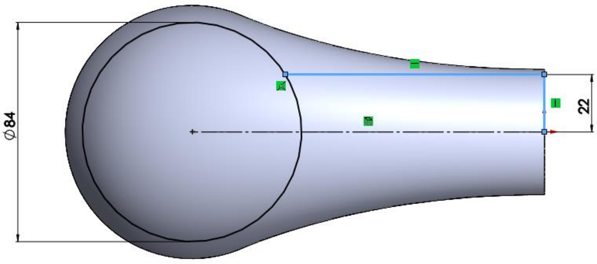
Select **OK** to accept the feature.

Rename as **'Thicken Main Body'**.

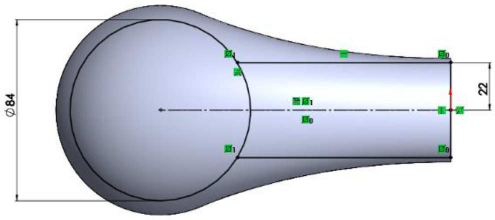
Creating a Split Line Feature

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

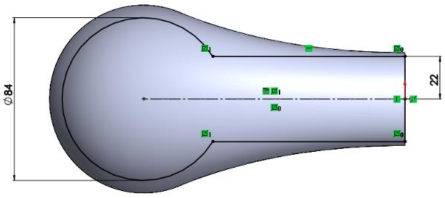
Select **Convert Entities** and select the construction sketch. (Centerline). Under options Tick the for Construction check box, select ok.



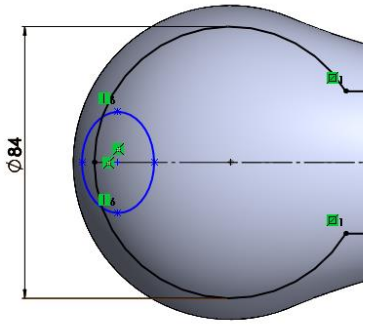
Select **Circle** command and draw circle, add a **Coincident Relation** between the centre of the circle and end centerline.



Select **Line** Command and Draw Horizontal And Vertical Line as shown opposite.
Dimension as shown opposite.
Select **Mirror entities**, Mirror the Horizontal and Vertical lines about the centerline.

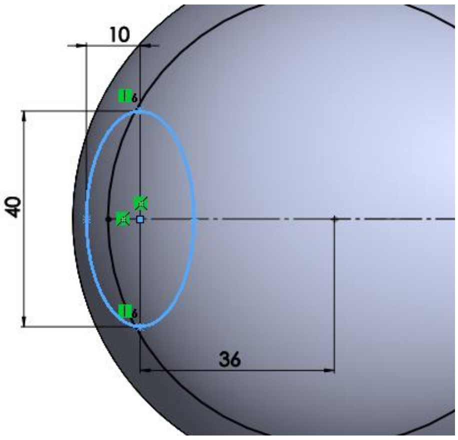


Select **Trim Entities** and use Power trim to remove inner segments of circle.



Select **Extend Entities** to extend centerline to Outer edge of Circle.

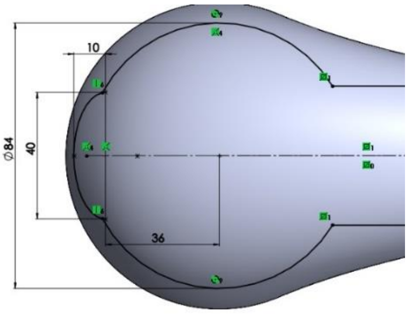
Select **Ellipse** and sketch an ellipse **Coincident** to the centerline as shown opposite.



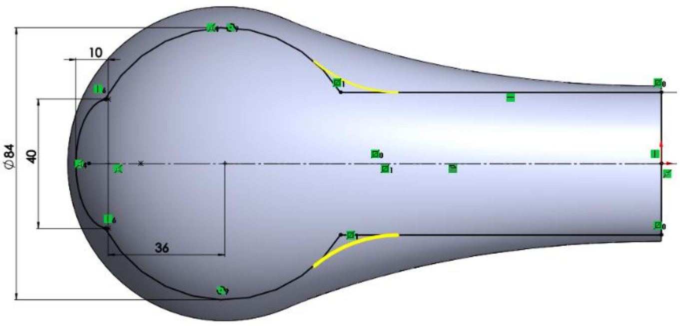
Add a **Vertical Relation** between the outer points of the major axis.

Dimension as shown opposite.

Select **Trim Entities** and remove inner segments of circle and ellipse.



Select **Sketch Fillet** from the sketch tools, select the **intersection point** between the circle and the horizontal lines.

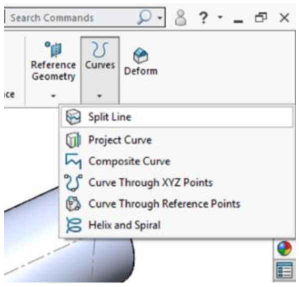


Set the **Fillet Parameters** to **40mm.**

Select **OK** to accept the fillet.

The sketch is now **fully defined**.

Rename as **'Split Sketch'**.

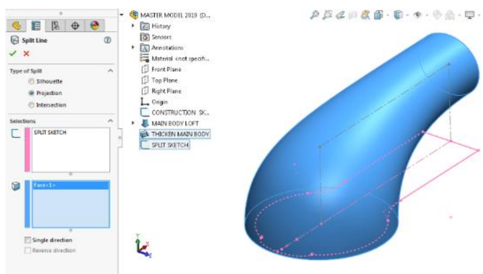


Creating a split on the Body

Select **Surfaces** toolbar and go to **Curves**, Under the Curves drop down select **Split Line**.

Under the **Split Line Tab**, select **Projection**.

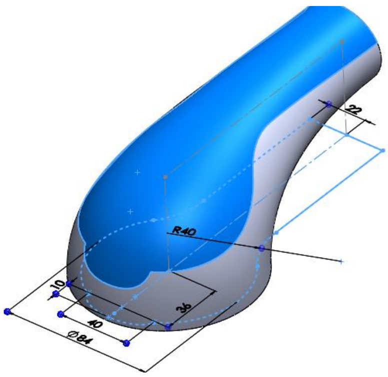
Under the **Sketch tab**, select **Split Sketch** from feature tree.



Under the **Faces tab**, select the **outer face** of the body.

Select **OK** to accept the Split line.

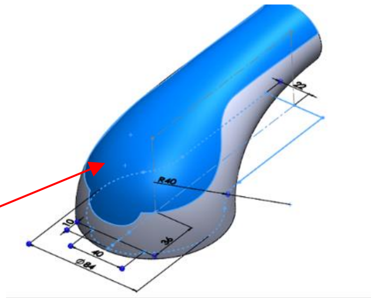
The Face selected has now been **Split** into seperate surfaces.



Click on the face to identify the Surfaces.

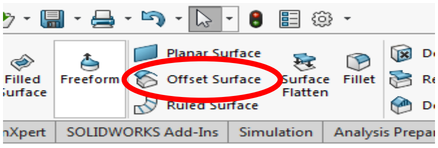
Rename Split Feature as **'Split Top Surface'**.

Offsetting the New Surface



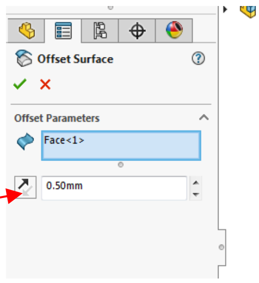
Note; Solidwork will not permit the user to work with the existing surface that has been created as this belongs to the existing body, the user must create a new surface. To do this the Offset Surface Feature is used. This feature acts like a copy button, The offset surface can be controlled by direction and distance.

Select the **Surface** that was Split.



From the surfaces toolbar select **Offset Surface Feature**.

Under the Offset Surface, Set offset distance to **0.5mm**.



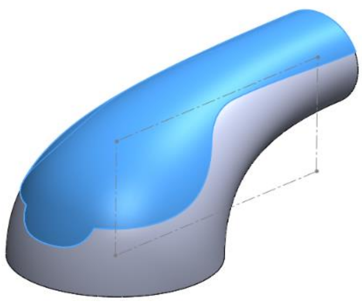
Select **OK** to Accept the Offset Surface.

Rename as **'Top Surface Offset'**.

Note: The Flip Offset Direction is automatically set to offset to the outside of the Surface. Should the user need to change this to the inside, this button can be used to Flip Offset direction.

Note; If the user required the surface to remain flush with the existing surface you could set the offset distance to zero. This will copy the existing surface and allow the user to use the new surface for other features.

Adding a Thickness to the new offset surface.



Select the **'Top Surface Offset'** from the feature tree.

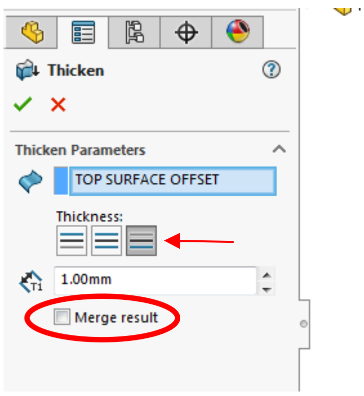
From the Surfaces toolbar select **Thicken**.

Set Thicken Parameters to;

Thicken Side 2

Thickness 1mm.

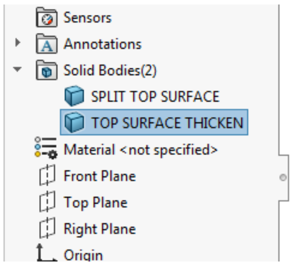
Untick the Merge result



Select **OK** & rename as **'Top Surface Thicken'**.

Note; At this stage by deselecting the merge result, a new solid body will be formed containg the top surface. This style of modelling is known as Multbody Modelling.

All solid bodies created in this manner can be saved as individual parts at any stage.
It is good practice to create the full multibody and save the bodies as new parts when all editing has been finalised.

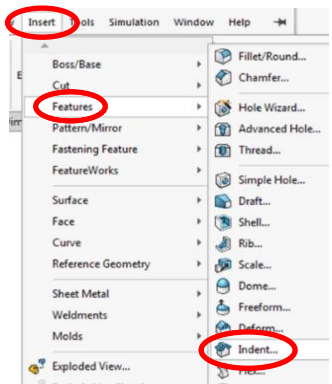


Viewing the Solid Bodies

From the feature tree select **Solid bodies folder**.
Two seperate bodies can now be seen in this folder. While these appear as two seperate bodies they can be saved as seperate parts at a later stage.

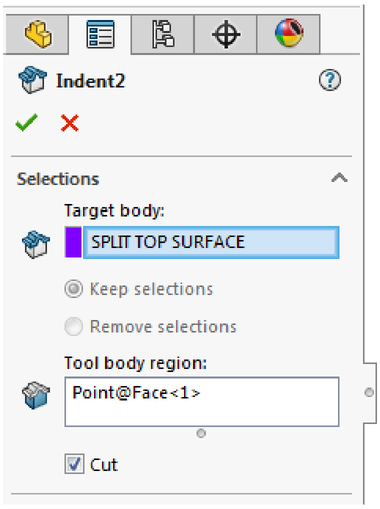
Note; The name of the solid body is derived from the name of the last feature created on this body. The names of the solid bodies can be changed at any stage.

Using the Indent Feature



The indent feature will now be used to take a cut in the shape of the Top Surface directly out of the Main Body resulting in two bodies that will fit perfectly into each other.
Note; The indent Feature can be found under the Insert Tab > Features > Indent.

Select **Indent**



Under the Indent Parameter select;

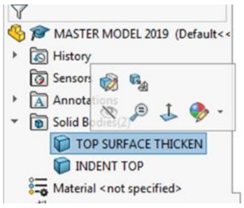
Target Body as Select **face of Main Body**.

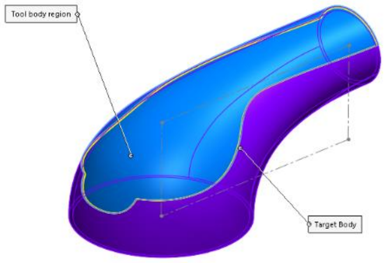
Tool Body Region as Select **face of Top Surface**.

Tick the Cut Checkbox.

Select **Ok** & rename feature as **'Indent top'**.

To view the resulting feature select Top Surface Thicken from the **Solid Bodies** folder.



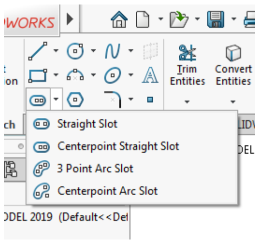


Select **Hide**.

The resulting effect of the indent feature can be seen.

Select **Show** Top Surface Thicken to resume editing.

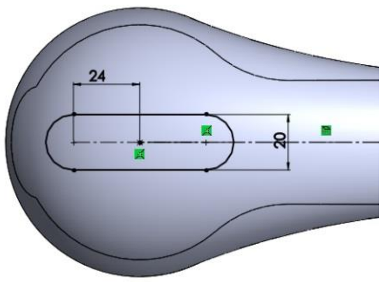
Creating the Button



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

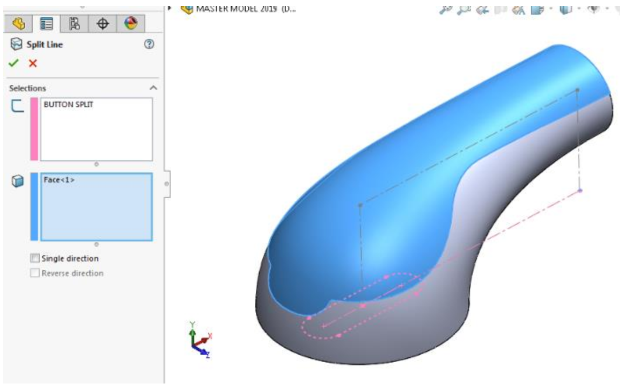
Select **Convert Entities** and select the construction sketch. (Centerline). Under options Tick the **for Construction** check box, select ok.

Sketch a **Centerpoint Straight Slot** with the centerpoint **Coincident** with the Outer edge of the Construction sketch.



Dimension as shown opposite.

Exit Sketch & Rename as **'Button Sketch'**.

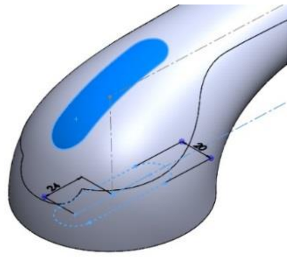


Select **Split line** from the Curves toolbar.

Type **Projection**

Under Sketch select **Button Sketch** sketch.

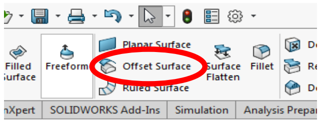
Under face Select the **Top Face** of Top Surface thicken.



Select **OK**.

Rename feature as **'Split Button'**.

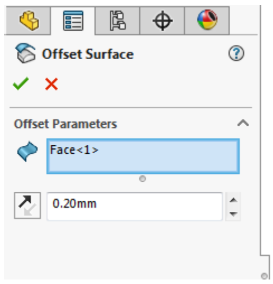
Select the **Surface** that was Split.



From the surfaces toolbar select **Offset Surface Feature**.

Under the Offset Surface, Set offset distance to **0.2mm**.

Select **OK** to Accept the Offset Surface.



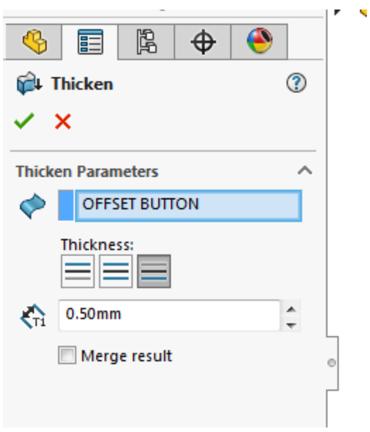
Rename as **'Offset Button'**.

Adding a Thickness to the Button offset.

Select the **'Offset Button'** from the feature tree.

From the Surfaces toolbar select **Thicken**.

Set Thicken Parameters to;

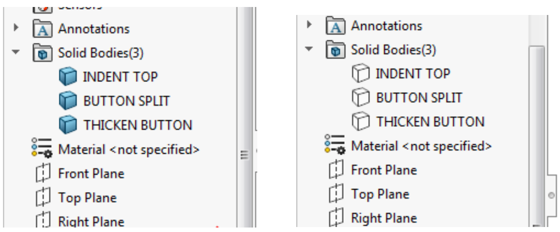


Thicken Side 2 – 0.5mm

Untick the Merge result.

Accept and rename as **'Thicken Button'**.

In the feature tree under **Solid Bodies folder**, three Solid Bodies can now be seen.



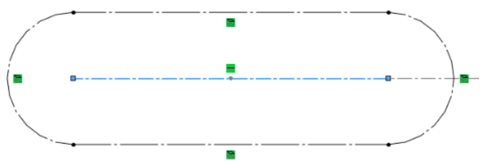
Hide the three Solid Bodies in the Solid Bodies folder.

Adding the Arrors to the Button

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

Select **Convert Entities** and select the Button sketch from the feature tree. Under options Tick the **for Construction** check box, select ok. (To be used as a guide)

Select **Centerline** and sketch from center to center of slot curves.

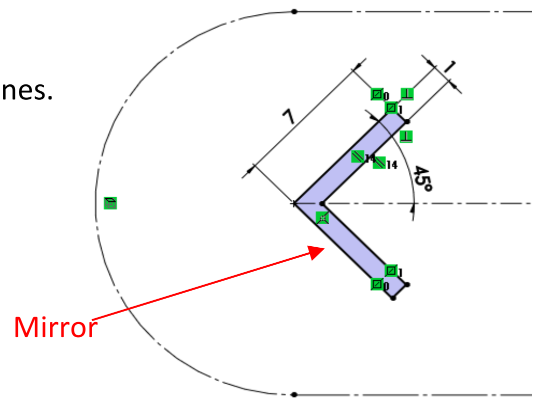


Select **Line** and create sketch starting on the end of the centerline as shown opposite.

Add **Parallel & Perpindicular** Relations between the lines.

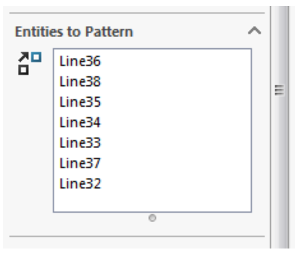
Dimension as shown.

Mirror the Entities using the centerline.



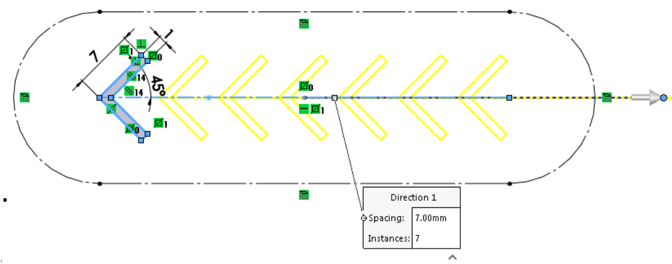
Select **Linear Sketch Pattern** from sketch tools.

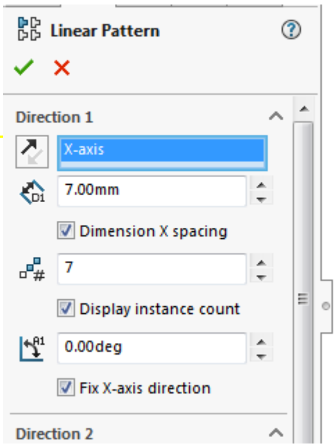
Set Entities to pattern – **Select arrow** (Tip – Select a point inside arrow to automatically select all lines)



Set **Direction 1** to the **centerline**

Set **D1** to 7mm



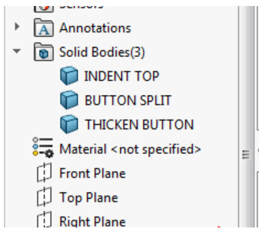


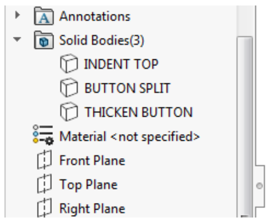
Set **Instances** to 7.

Select **OK** to accept the pattern

Rename as **'Arrows'**.

In the feature tree under **Solid Bodies Folder.**

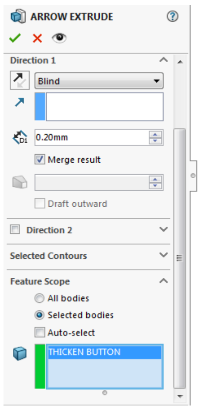


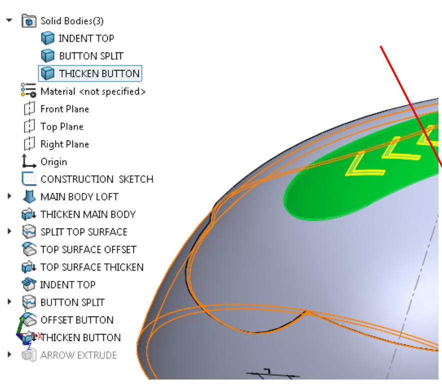


Show the three Solid Bodies in the Solid Bodies folder.

Select the **Arrows sketch** from the feature tree.

Select Top Face of Button





Select **Extruded Boss/Base**

Set The following parameters;

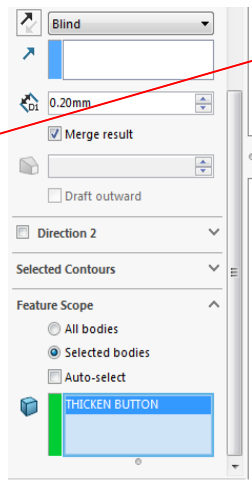
From – **Surface/Face/Plane**

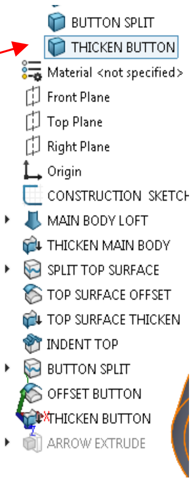
Surface – **Select top face of button**

Select – **Blind** & Distance 1 – **0.2mm**

Under Feature Scope – **Tick selected Bodies**

Untick Auto-select





Select the body to merge with from the feature tree – **Thicken Button**

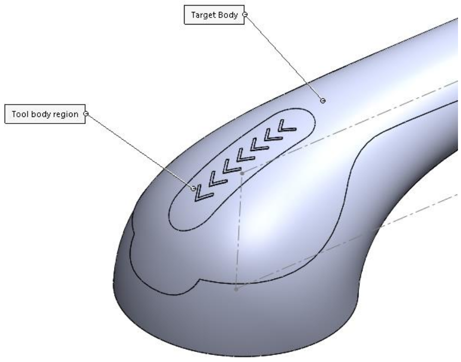
Note; This is an important feature of the extrude boss/base, it allows the user to select which of the bodies to merge with. If all bodies was selected it would merge all bodies in the entire model resulting in a single solid body.

Select **Ok** to accept the feature & rename as **'Arrow Extrude'**.

Indent Feature for Button

Select **Indent Feature** – Insert>Features>Indent

Under the Indent Parameter select;



Target Body as **Top Surface/Split Button.**

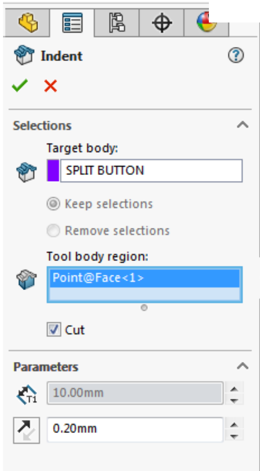
Tool Body Region as **Button**.

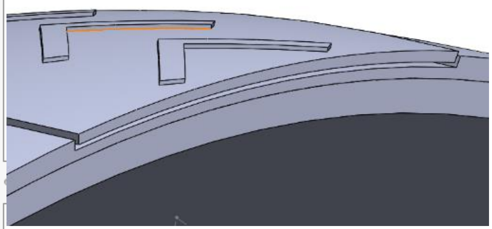
Tick the Cut Checkbox.

Under Parameters;

Set the **thickness to 0.2mm**

Select **Ok** & rename as **'Indent Button'.**



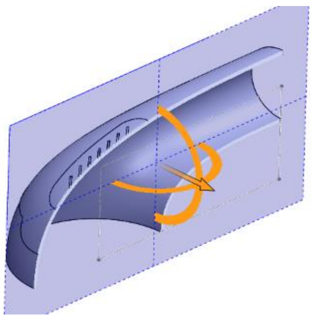


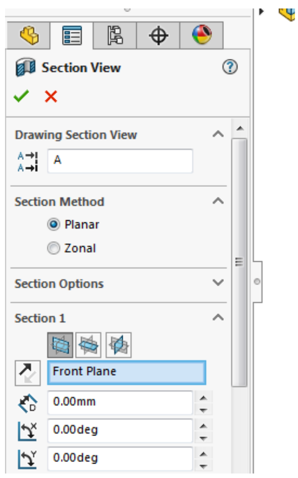
Note; The thickness parameter can be used to create a gap between the two bodies in the indent. The gap is created around the outside of the bodies and between the bodies.

Revolving the Bottom of the Suction Cup

Create a Section View using the **Front Plane**.

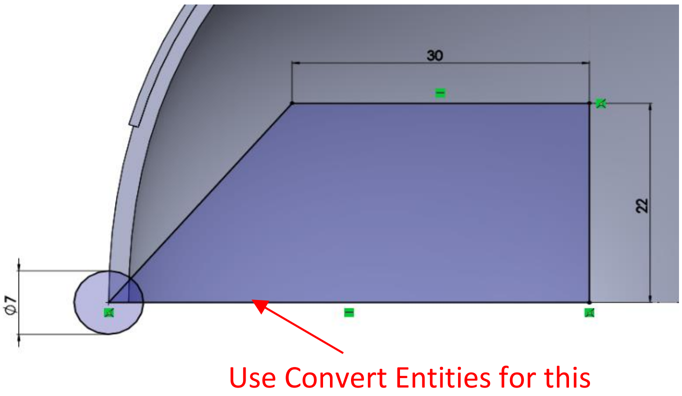
Select **OK**.





Create a sketch on the **Front Plane** and **Convert Entities** for **base line.**

Select **Line** and complete sketch as shown opposite.



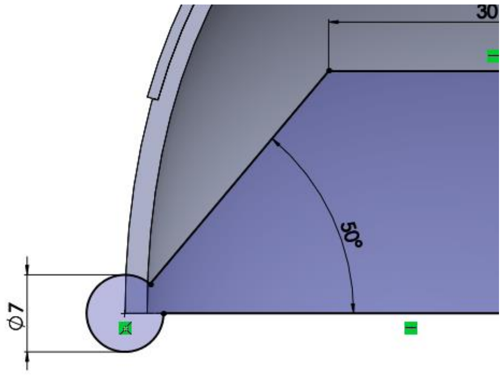
Select **Circle** and sketch a circle with its centerpoint **Coincident** to the outer corner of the model.

Dimension as shown.

Select **Trim Entities** and remove as Shown opposite.

Trim right side of converted entity line.

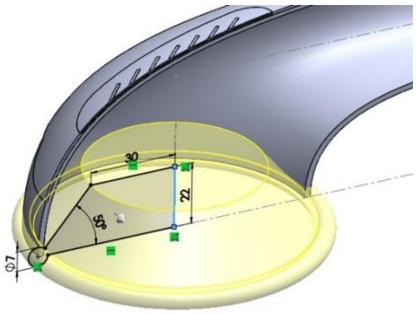
Dimension Angle as shown to fully define sketch.



Rename as **'Inner Section'**.

Select Revolved Boss/Base

Select the 22mm line as axis of reveloution



Select Parameters as shown below;

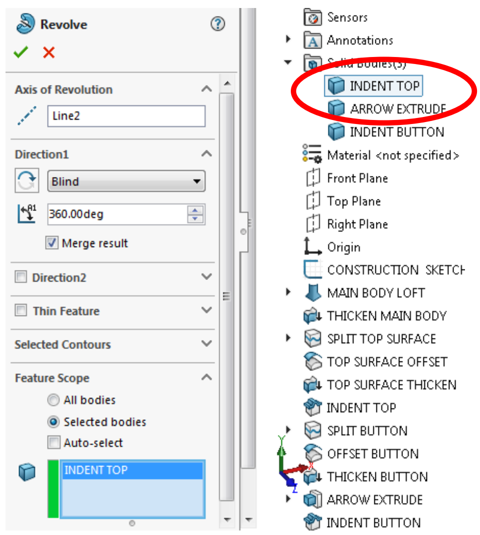
In the **Feature Scope**

Set to **Selected Bodies**

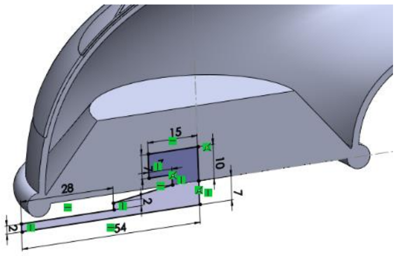
Untick Auto-select

Select **Indent Top** from feature tree as the Selected Body.

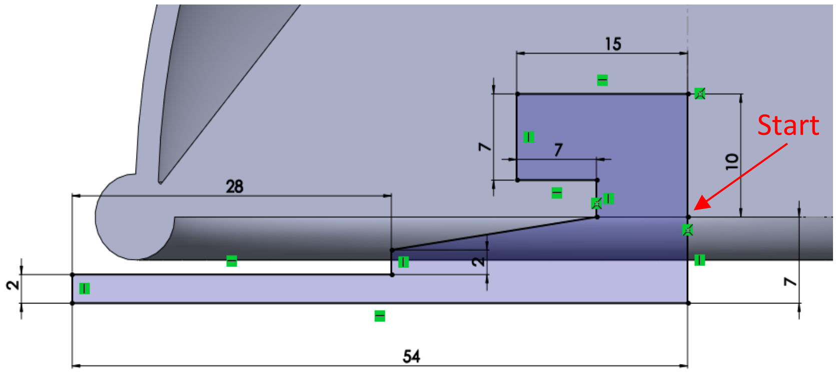
Select **OK** & rename as **'Revolve Base'**.



Creation the Bottom Suction Cup



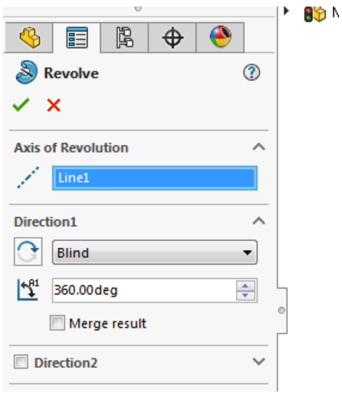
Create a sketch on the **Front Plane** using the **line command**. Make first point of line **Coincicent** with midpoint of previous revolve.

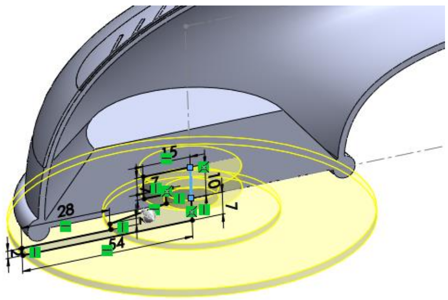


Dimension as Shown.

Select Revolve Boss/Base

Set the **10mm line** as the axis of revolution.



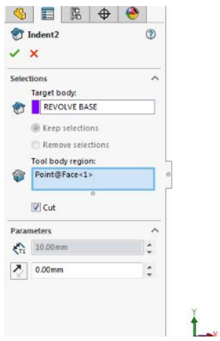


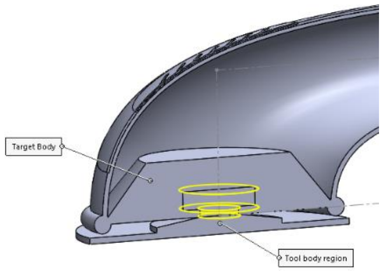
Untick the Merge result tickbox.

This will form a new Solid Body.

Select **OK** to accept.

Rename as **'Base Suction Cup'**.





Select **Indent Feature**

Select the **Revolve Base** as the **Target Body.**

Select the **Base Suction Cup** as the **Tool body region**.

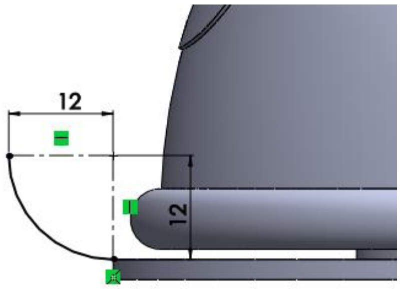
Tick the Cut checkbox.

Note; In this instance space is not required between the two bodies so the gap setting will be set to zero.

Rename as **'Indent Main Body'.**

Turn Off Section View.

Adding a Grip Tab to the Suction Cup



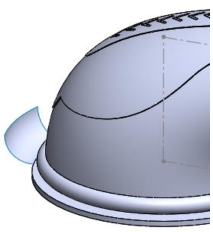
Create a sketch on the **Front Plane** as shown opposite.

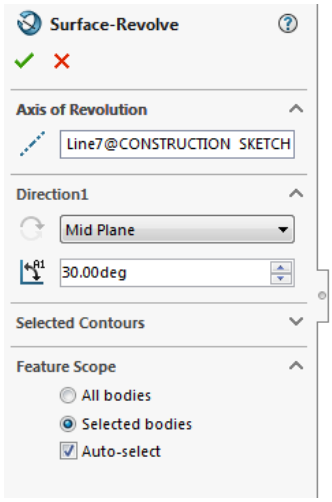
Select **Centerline** and sketch a Vertical centerline with a coincident on the corner of the suction cup and a Horizontal centerline.

Dimension as shown.

Select **Centerpoint arc** to complete the arc.

Select **Revolved Surface.**

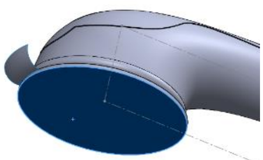




Use Centerline from **Construction sketch** as axis of revolution.

Set Revole **Parameters** as shown opposite.

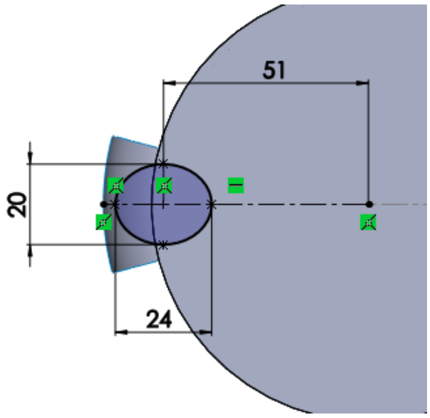
Select **OK** & rename as **'Revolve Tab'**.



Create a sketch on the **Bottom Surface** of the Suction Cup.

Select **Centerline** and sketch from Construction sketch to edge of surface.

Select **Ellipse** and sketch Ellipse with Major axis **Coincident** with Centerline.



Dimension as shown.

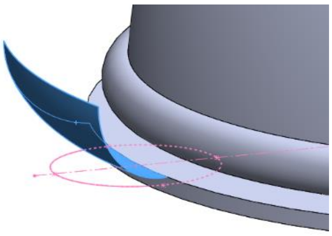
Rename sketch as **'Tab Sketch'**.

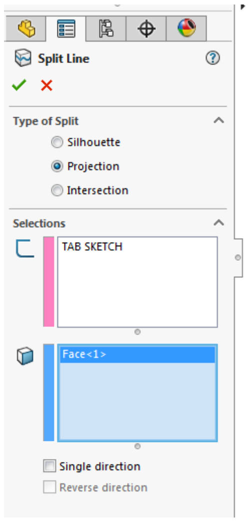
Select **Split line** command from **Curves toolbar**.

Select Tab sketch

Select Face of surface

Select **Ok** to accept.





Rename as **'Tab Surface'**.

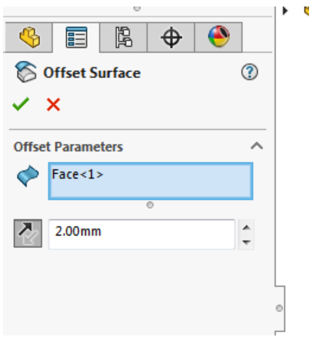
Select the Tab portion of the Surface.

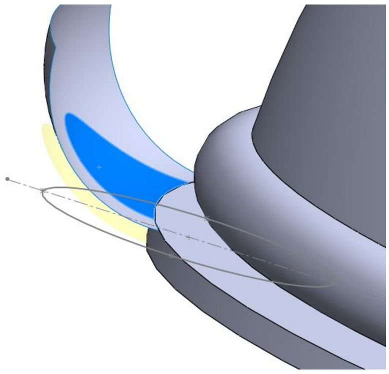
Select Offset Surface.

Create a 2mm offset to the underside of the surface.

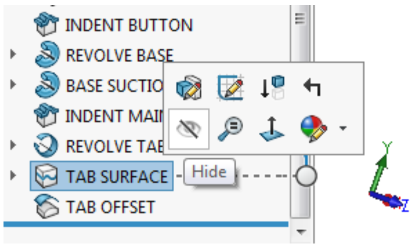
Select Ok.

Rename as **'Tab Offset'**.





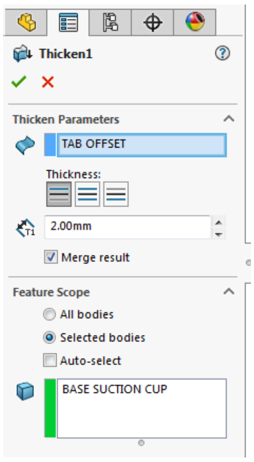
Select **Tab Surface** from Feature tree & **Hide** surface.

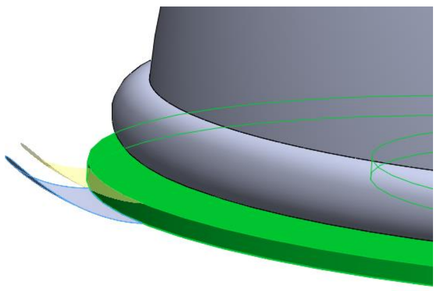


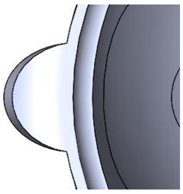
Select **Thicken**

Set **Parameters** as shown opposite.

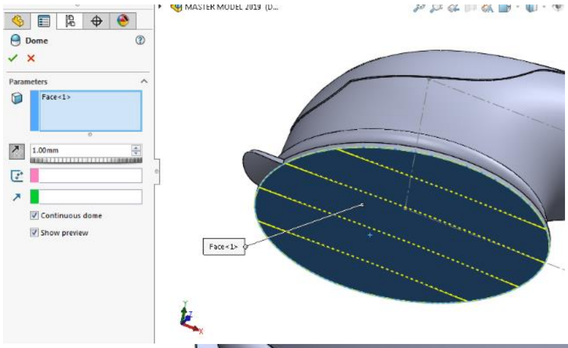
Rename as **'Thicken Tab'.**







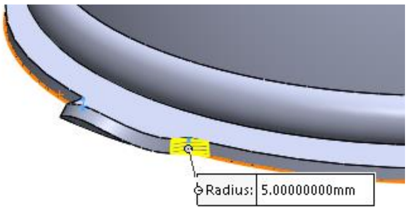
Select the **underside face** of the suction cup.



Select the **Dome** feature.

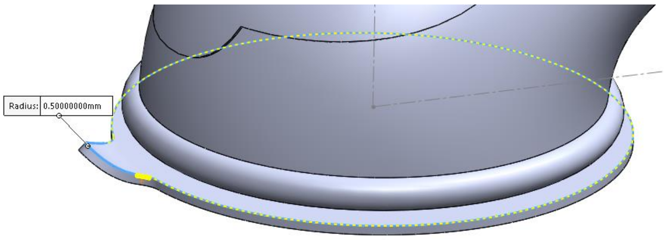
Reverse Direction.

Select **OK** to accept &rename as **'Dome Underside'.**



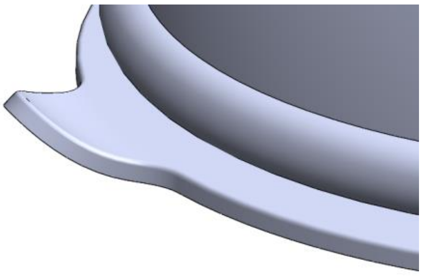
Select **Fillet.**

Add a **5mm Fillet** each side of the Tab.



Rename as Fillet Tab Edge.

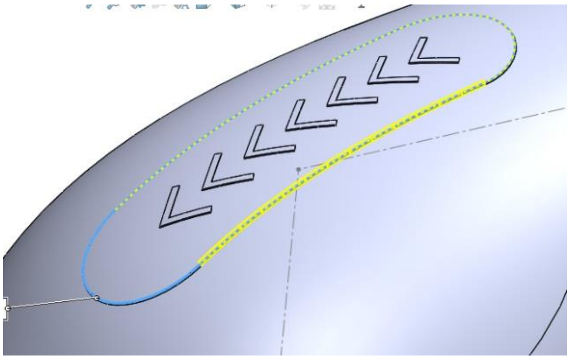
Select **Fillet**



Add a 0.5mm fillet to the top edge of the suction cup.

Rename as Fillet Suction Top.

Select **Top Edge** of Button.



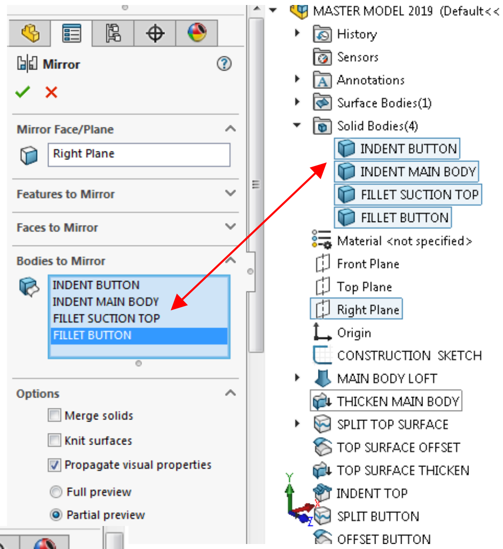
Select **Fillet**.

Add a **0.5 mm** Fillet.

Rename as **'Button Fillet'.**

Mirror Solid Bodies

Select **Mirror** from the features toolbar.



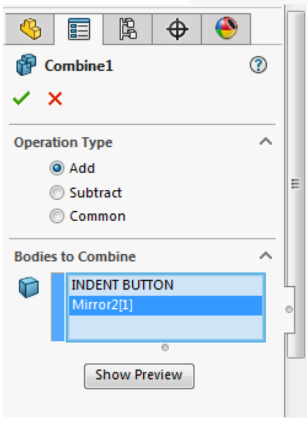
Set Mirror **parameters** as shown opposite.

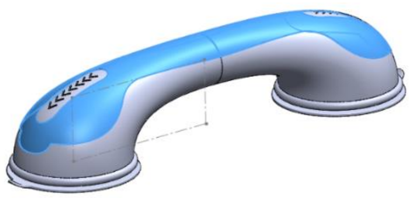
Select **OK**.

Rename as **'Mirror Bodies'**.

Select **Combine** Feature

Select **Add**



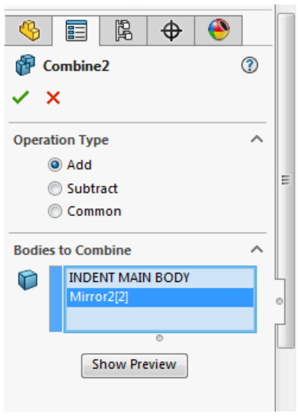


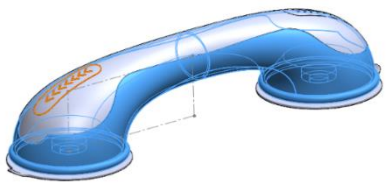
Select the **Top Surfaces**

Select **OK** & rename as **'Top Casing'**.

Select **Combine** Feature

Select **Add**





Select the **Main Body and Mirror**

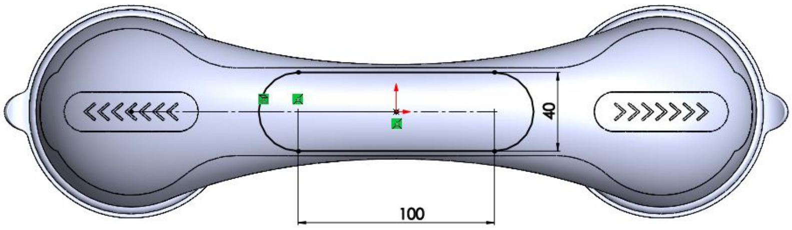
Select **OK** & rename as **'Main Body'**.

Creating a Grip

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

Select **Convert Entities** and select the **Construction sketch** from the feature tree. Under options Tick the **for Construction** check box, select ok.

Select Centerpoint slot **Coincident** with Origin and side point **Coincident** with the centerline converted.



Dimension as shown.

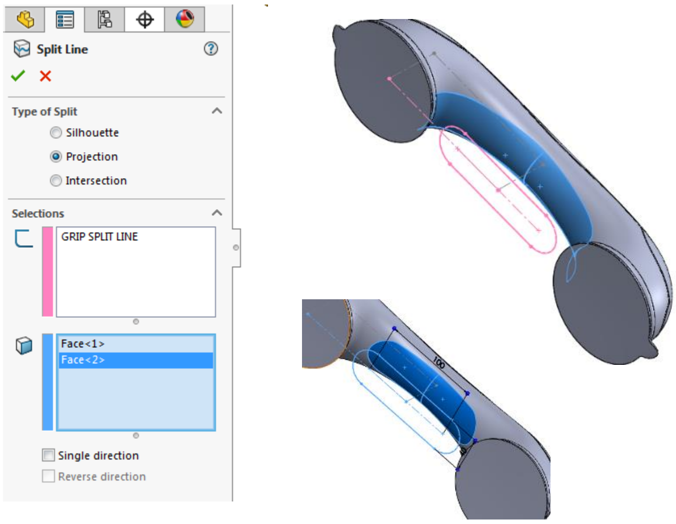
Rename as **'Grip Split Line'.**

Select **Split Line** from Curves toolbar.

Set **Parameters** as shown.

Ensure **both faces** are selected.

Select **OK** & rename as **'Grip Split'**.

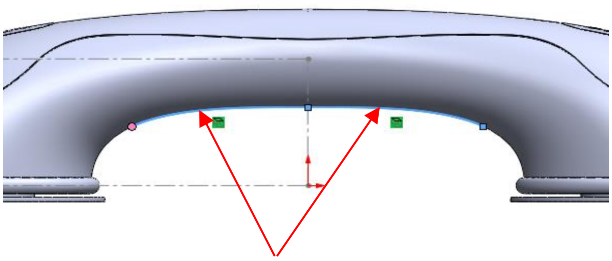


Using the Deform Tool To Create Grips

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

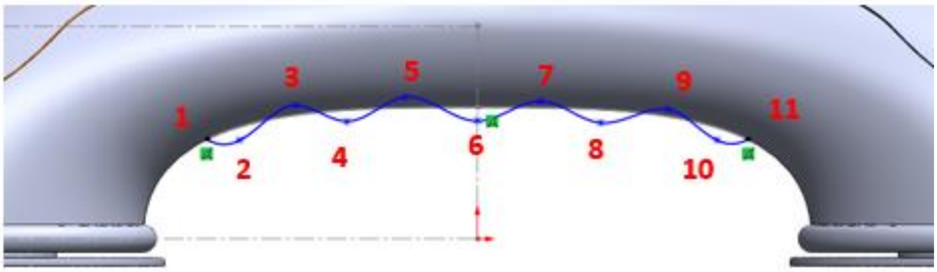
Select **Convert Entities** and select **both edge curves.**

Rename as **'Initial Curve'.**

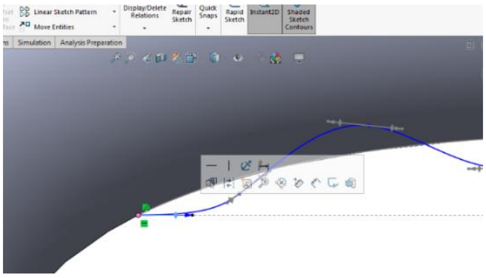


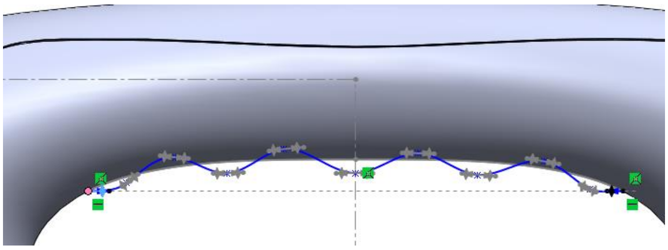
Create a sketch on the **Front Plane.**

Create a **Spline** as shown in the sketch opposite. Ensure **start and end point** of spline are **Coincident** with Initial Sketch.



Add a **Horizontal Relation** to both ends of the **spline handles**.





Rename as **'Target Curve'.**

Note; Try to keep the spline just outside and inside the body for this feature.

Select **Deform Feature**

Insert>Features>Deform

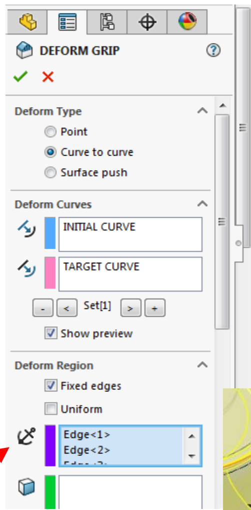
Select **Curve to Curve**

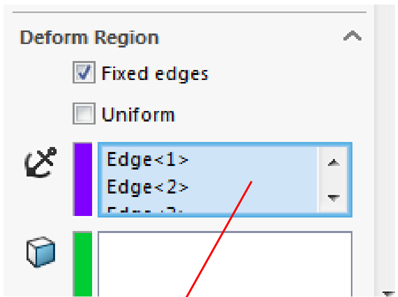
Select **Initial Curve**

Select **Target Curve**

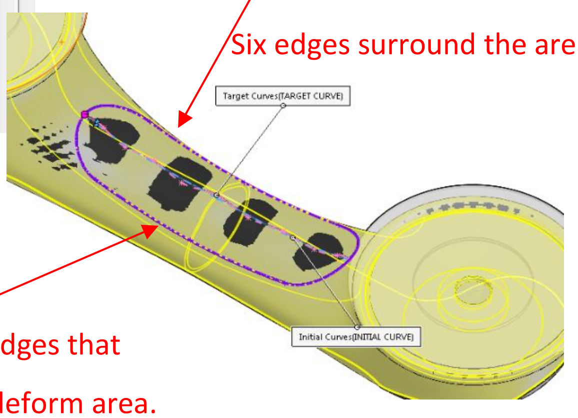
Select **fixed Edges**

Select **OK** to accept.





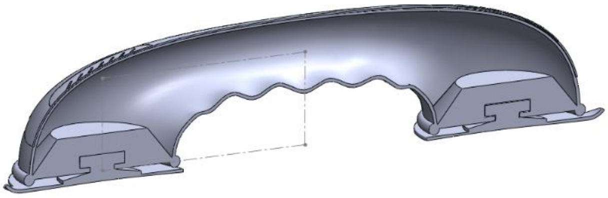
Six edges surround the area.



Select the edges that surround the deform area.

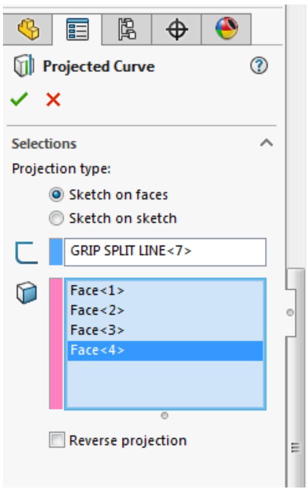
Rename as **'Deform Grip'.**

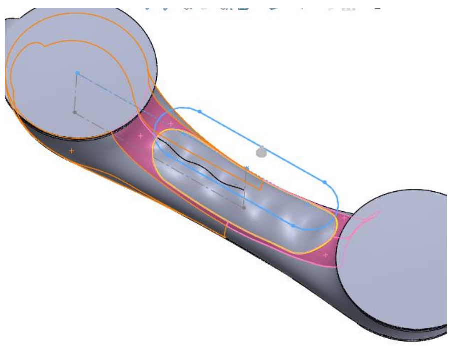
Hide Initial Sketch **and** Target Sketch in feature tree.



Note; Use **Section View** – Front Plane to best view Grip.

Creating a Design around the grip.



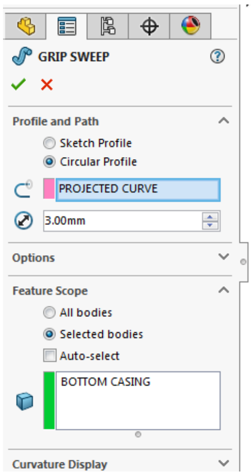


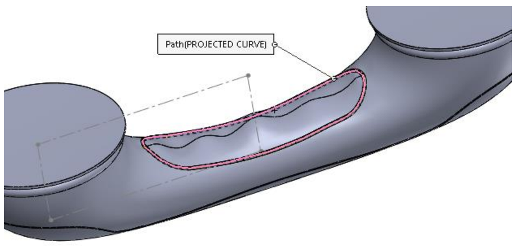
Select **Grip Split Line** sketch.

Select **Projected Curve** from Curves toolbar.

Select **faces** to project onto as shown.

Select **OK** and Rename as **'Projected Curve'.**





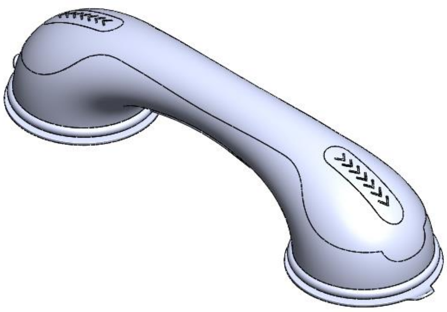
Select Feature **Swept Boss/Base**

Select Projected curve

Select **Circular Profile**

Set Diameter – **3mm**

Select **OK** and Rename as **'Grip Sweep'**.

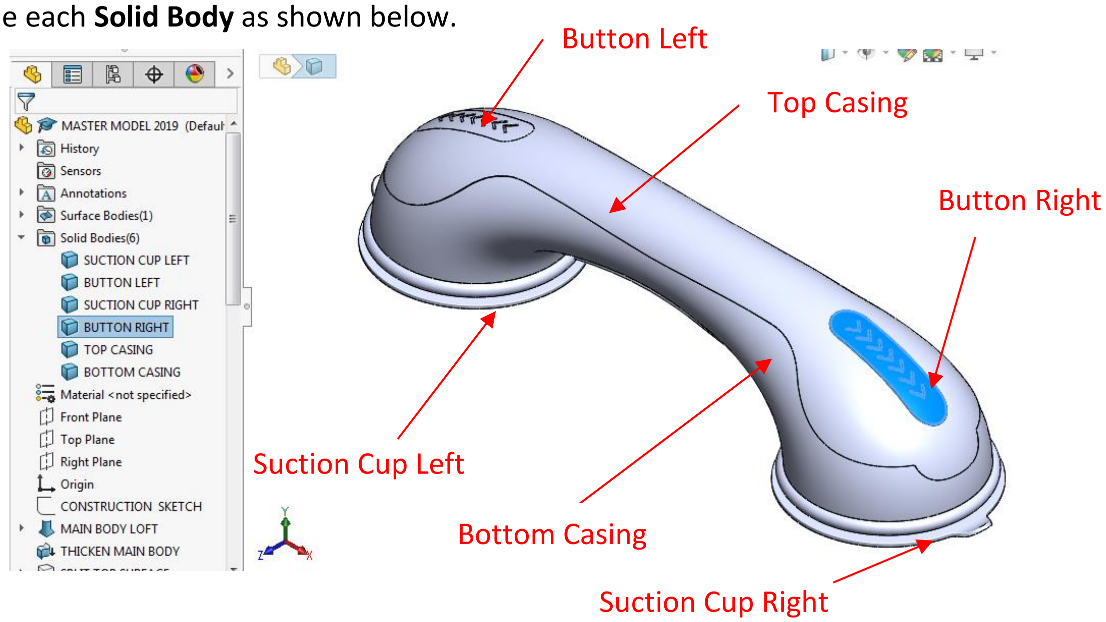


Hide the Projected Curve.

Hide the Construction Sketch.

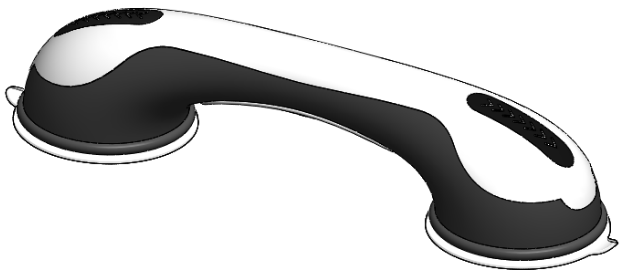
Renaming The Solid Bodies

Rename each **Solid Body** as shown below.



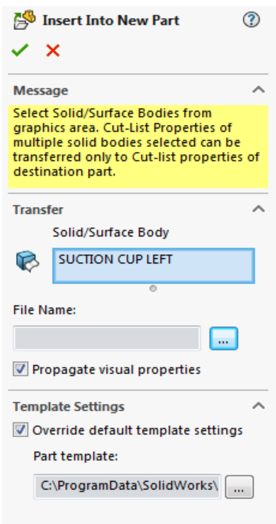
Adding Appearances

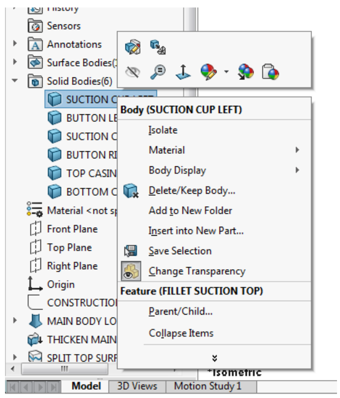
Add suitable appearances to each solid body.



Saving the Bodies

There are **two methods** of saving the bodies.





Method 1

Right Click on the Solid Body in the Solid Bodies Folder.

Select **Insert into New Part.**

Tick Propagate visual properties.

Tick Override default template settings.

File Name – **Leave blank.**

Select OK.

The Solid body will now appear as a new part file.

The Save as dialog box will appear – It will **automatically** create a name for the part linked to the name of the Solid Body.

Select **Save.**

Note; The new part will automatically be saved into the same folder/location as the master model. Only tick propagate visual properties if you want to carry appearances/decals into the new part.

Repeat this Process for all solid bodies.

Create an **Assembly & Insert Parts** into the Assembly.

Parts can be **mated** using planes**. (front to front)**

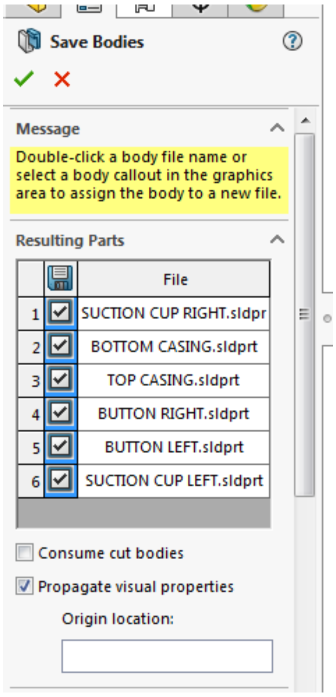
This will create an **identical** setup to the Master Model.

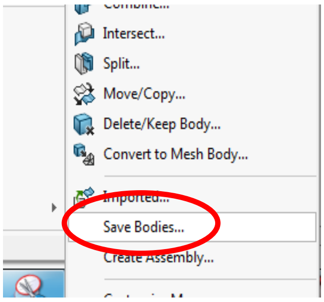
Note; All mating can be achieved as if mating any set of parts.

Note; When Solid Bodies are inserted into a new part, they retain the parametric link to the Master. This is important to note as any changes carried out in the Master Model will automatically update in the new part or assembly.
Any changes made in the part files will not be linked to the master model, these changes will be seen in the assembly. This can be used to an advantage for making different versions of a part.

Method 2

Save Bodies Feature





In the Master Model

Select **Insert> Features > Save Bodies**.

Tick the Bodies to Save.

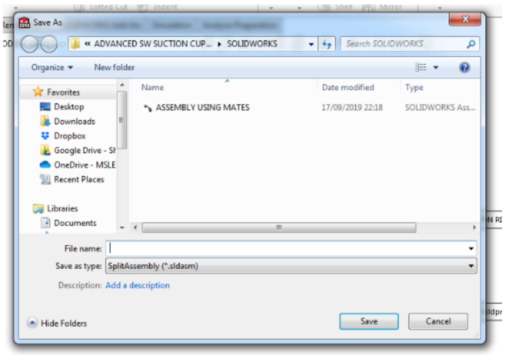
In this case all 6 bodies should be ticked.

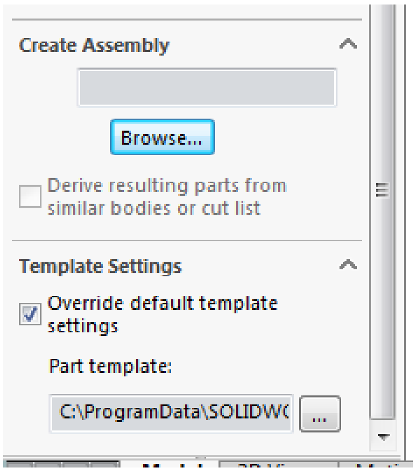
Untick Consume Cut Bodies.

Tick propagate Visual Properties.

Under Create Assembly

Select **Browse**





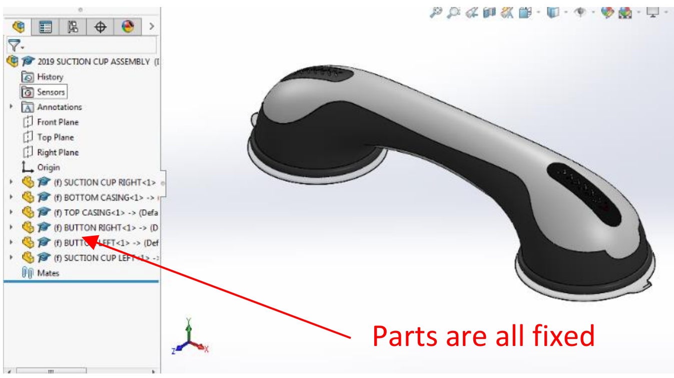
Specify the **file name** and **location** for the assembly.

Select **Save**.

Select OK on the save bodies feature.

Solidworks will **automatically** save each solid body as a new part file, it will also create an assembly of all the part files.

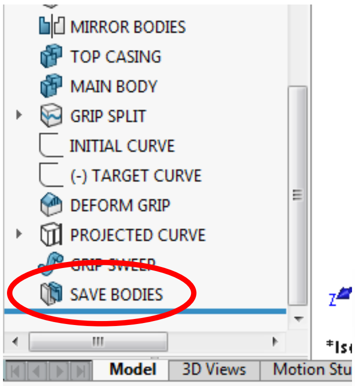
Assembly Environment



Notes;

All parts are fixed in the exact position from the master model.

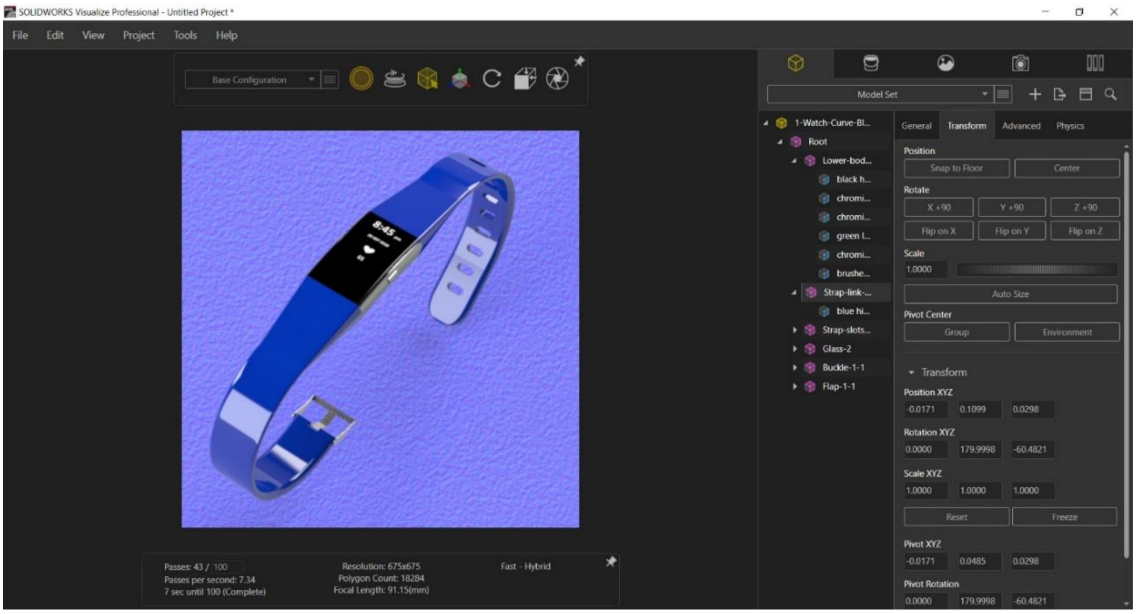
If changes are required in the **Master Model** it is important to look at the feature tree.



A new feature was created when **Save Bodies** was used.

In order to retain a **parametric link** to the assembly, any further changes must be created before the Save Bodies Feature.

SolidWorks Visualize Professional

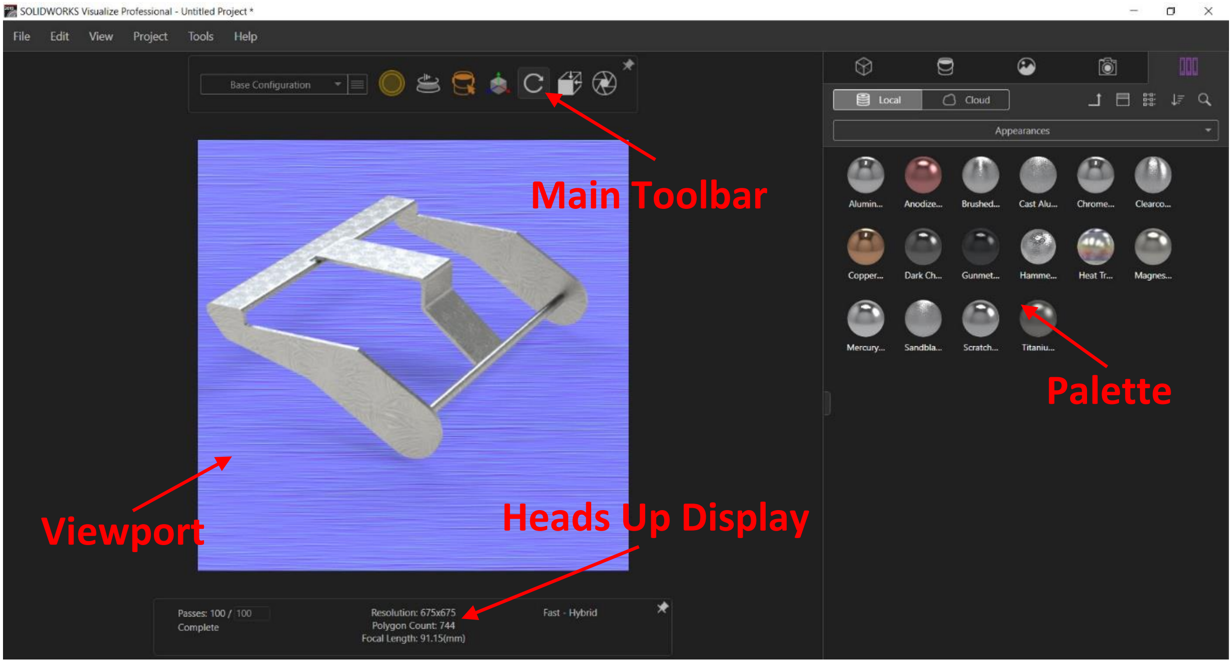


It is essential to understand the importing process in SolidWorks Visualize to control the import settings and the organisation of the original CAD files

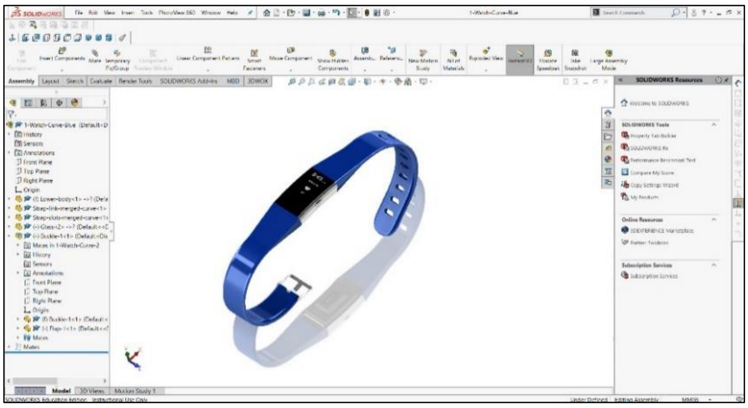
There are two common import settings to setup a Visualize project for different workflows one to make a model quick and easy to work the other to allow live updates and more flexibility.

This presentation will focus in on the workflow that allows the appearances applied to Parts/Assemblies in SolidWorks to Live Update in Visualize. Note reference will be made to the quick and easy workflow (www.mysolidworks.com) at a later stage

General User Interface Of Visualize



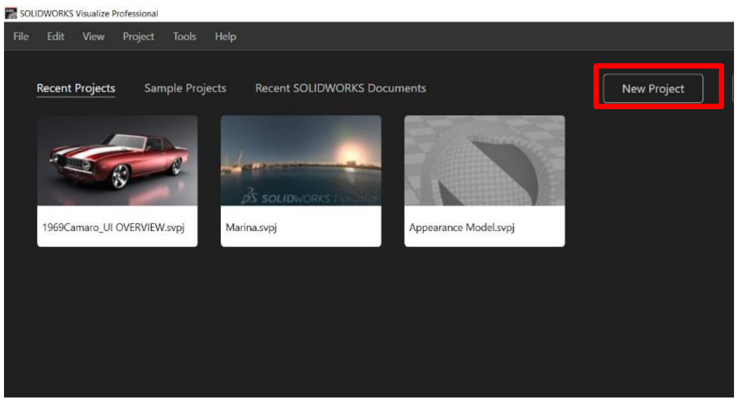
Importing CAD files into Visualize



Open the CAD files Watch-curve-blue and apply appearances save the file as normal.

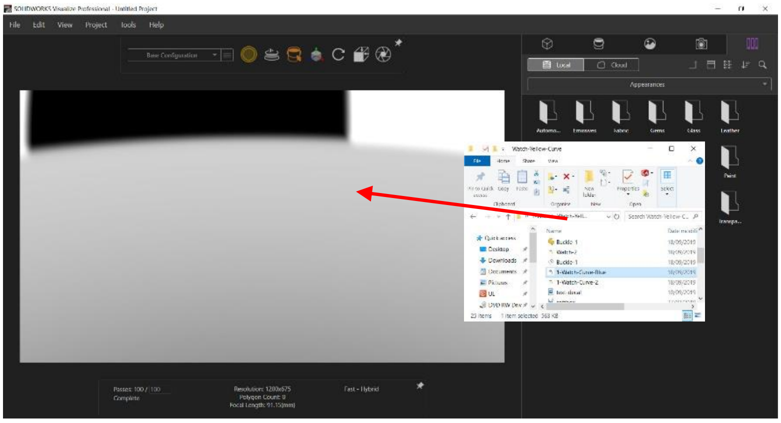
Select the **Visualize Icon** (Desktop)

Select the icon create **New Project** and save.



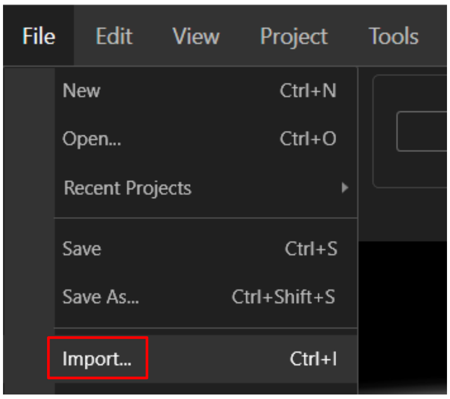
Importing CAD File Option 1

In the **New Project** window open the folder with the contains the file Watch-curve-blue drag the file into the new project window.

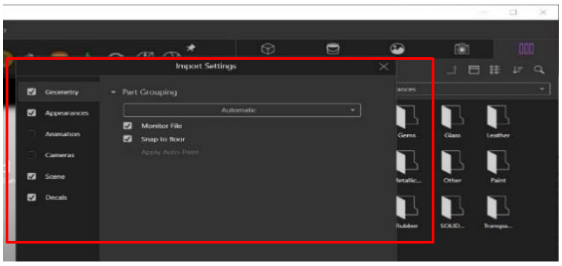


Importing CAD File Option 2

Select File top left and scroll down to import.

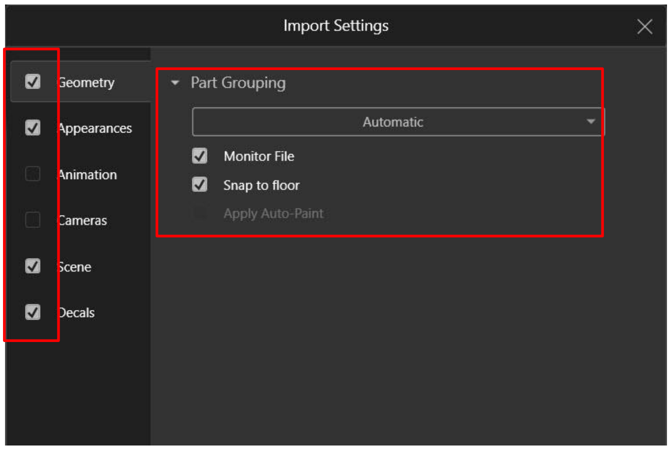


Import window

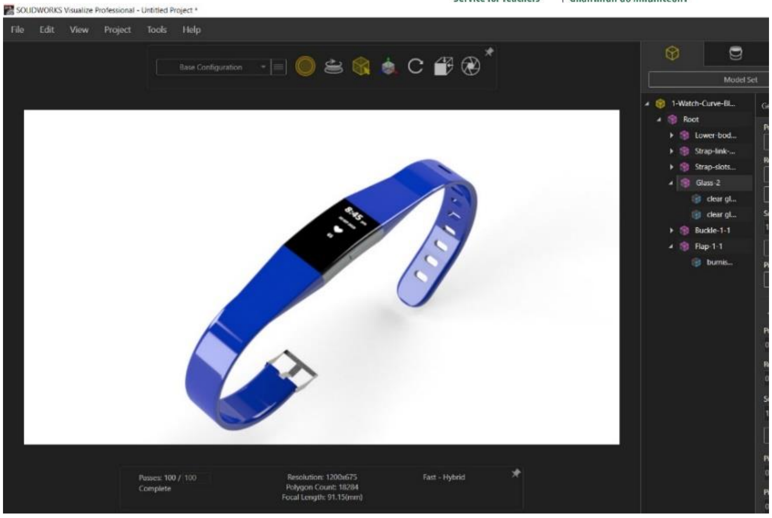


Import Settings Checked

- ✓ **Geometry**
- ✓ **Appearances**
- ✓ **Scene**
- ✓ **Decals**
- o Animations
- o Camera
- ✓ **Automatic**
- ✓ **Monitor File**
- ✓ **Snap to Floor**

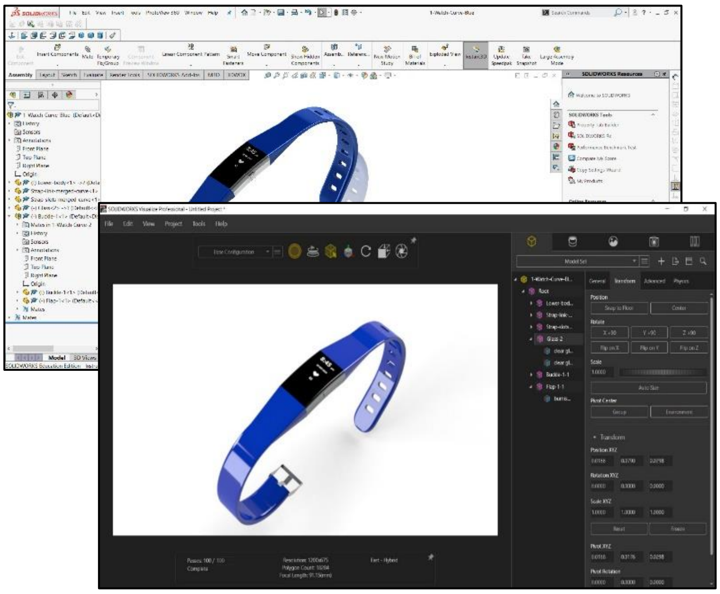


Imported SolidWorks files with settings applied



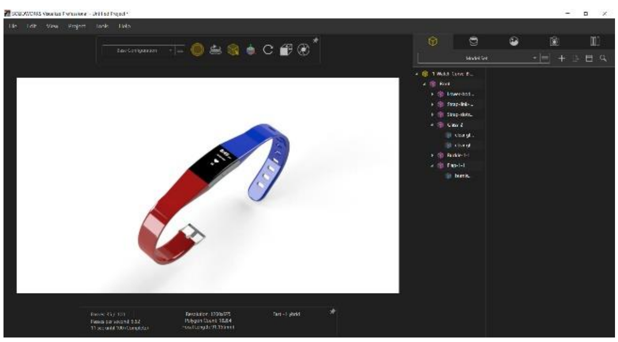
Toggle between

SolidWorks & **Visualize (Alt Tab)**



Edit a part in SolidWorks, Visualize will prompt to **Update Live**

SolidWorks Edit



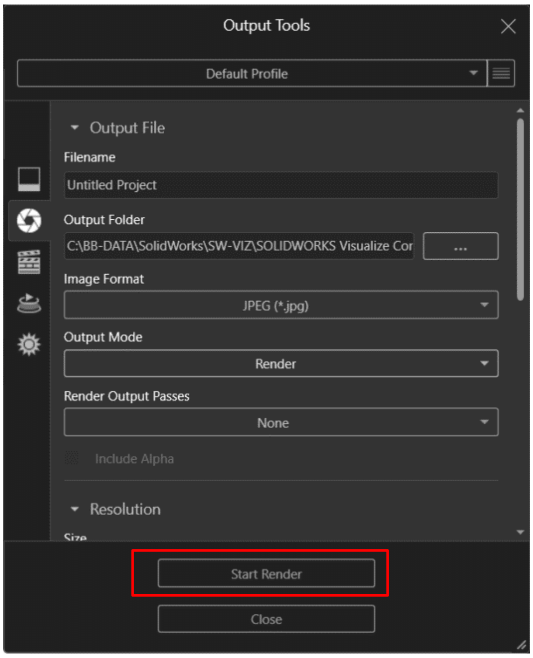
Visualize Live Update

Saving an Image from Visualize

Select the **Output Icon** from the Main Toolbar this will bring up the **Output Tools**

Output Tools

- **File Name**: Appropriate Name for Image
- **Output Folder**: Location to save file
- **Image Format**: JPEG
- **Output Mode**: Render
- **Render Output Passes:** None
- **Start Render**



Render Completed

On completion of the **render** you can select a link to find the location of the file with you have pointed to in the **Output Tools Settings**

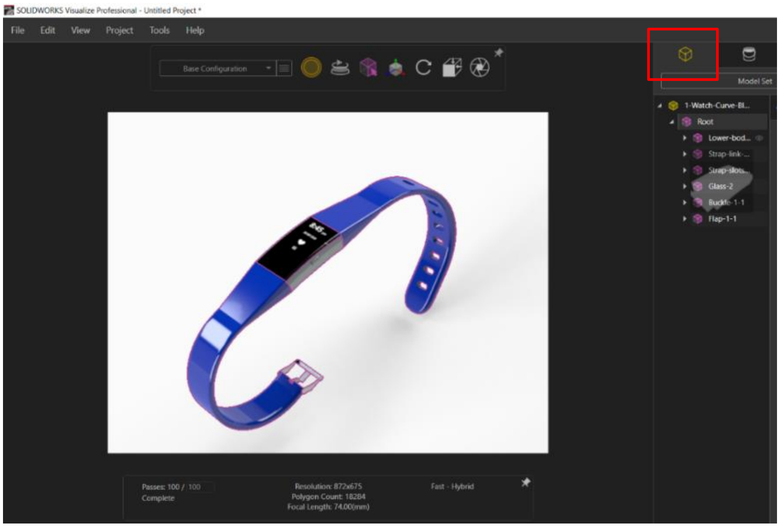
Editing the SolidWorks Model

Main Toolbar

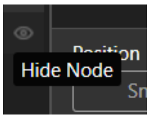
- Renderer Selection
- Turntable
- Selection Tool
- Object Manipulation Tool
- Camera Tool
- View Presets
- Output Tool

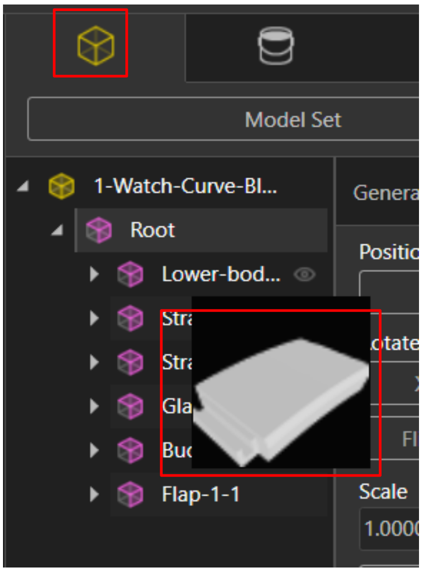
Showing and **Hide Parts**

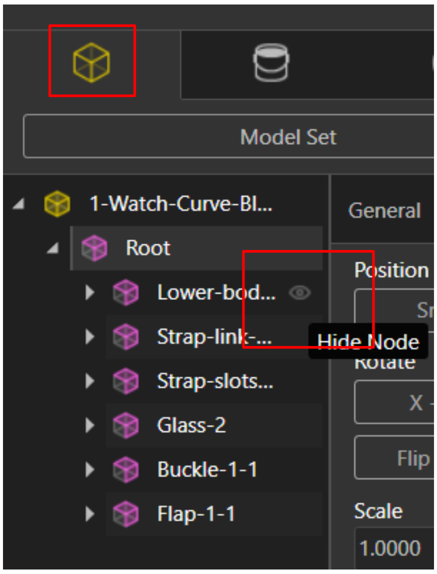
Select the Models Icon



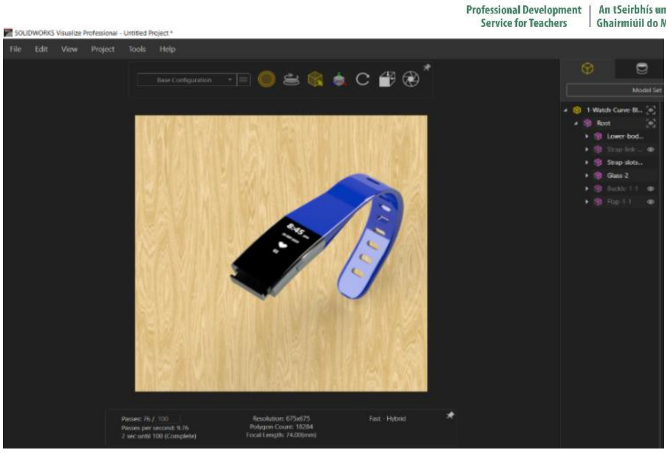
Select a Part and on the right select the **Show/Hide**

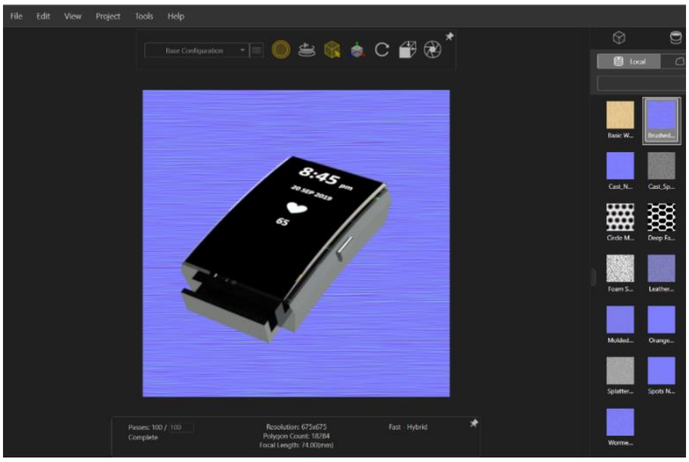






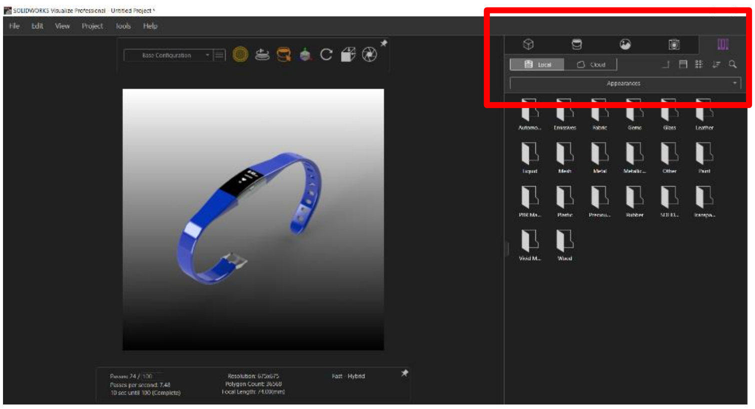
SolidWorks Parts Hidden

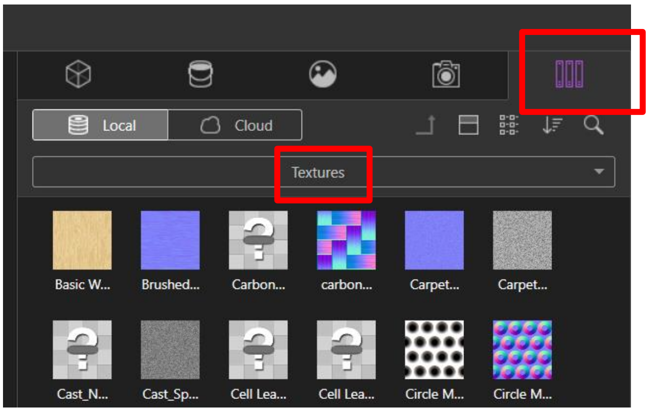




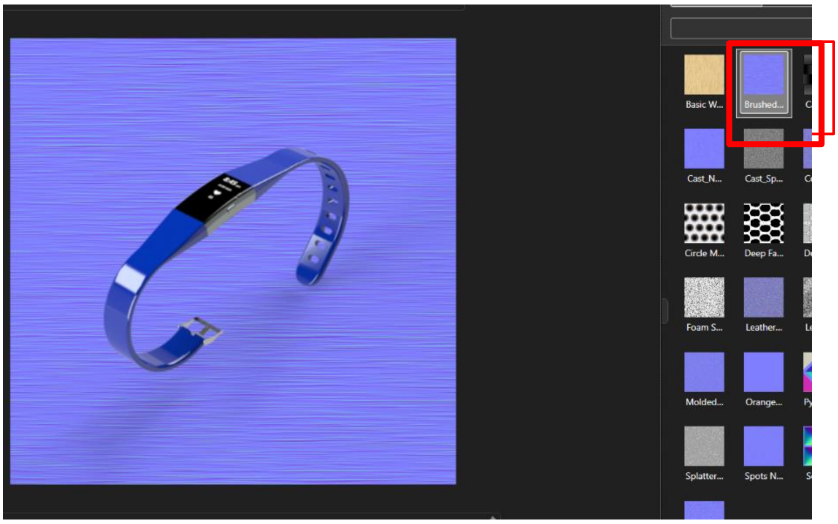
Inserting Textures

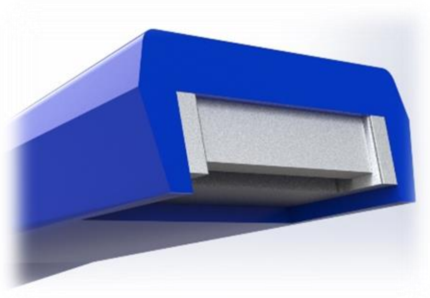
Select **File Library** and **Textures**

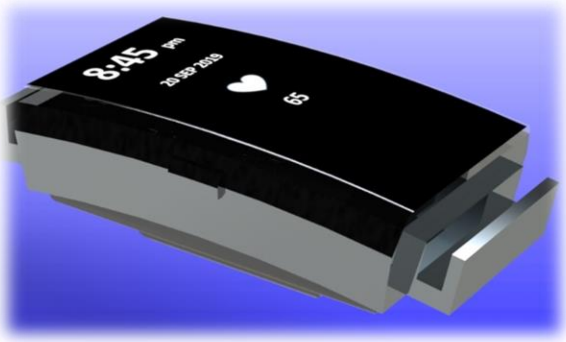




Drag **Brushed Normal** into the Project Window



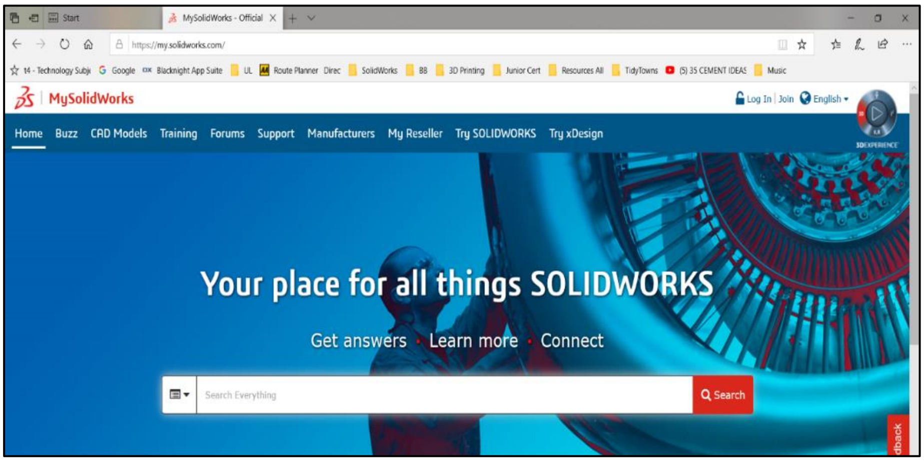




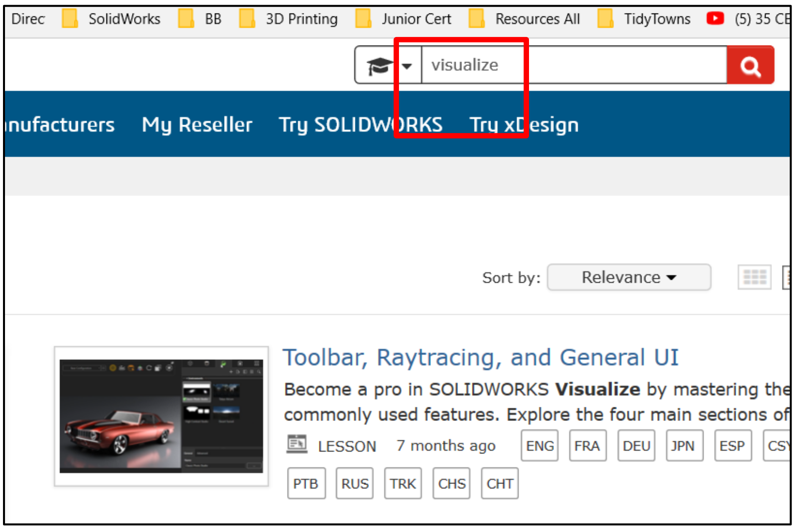
www.mysolidworks.com

Login using **Existing SolidWorks Account**

Create a **New Account** using SolidWorks Serial Number



Search for Visualize



Training Materials

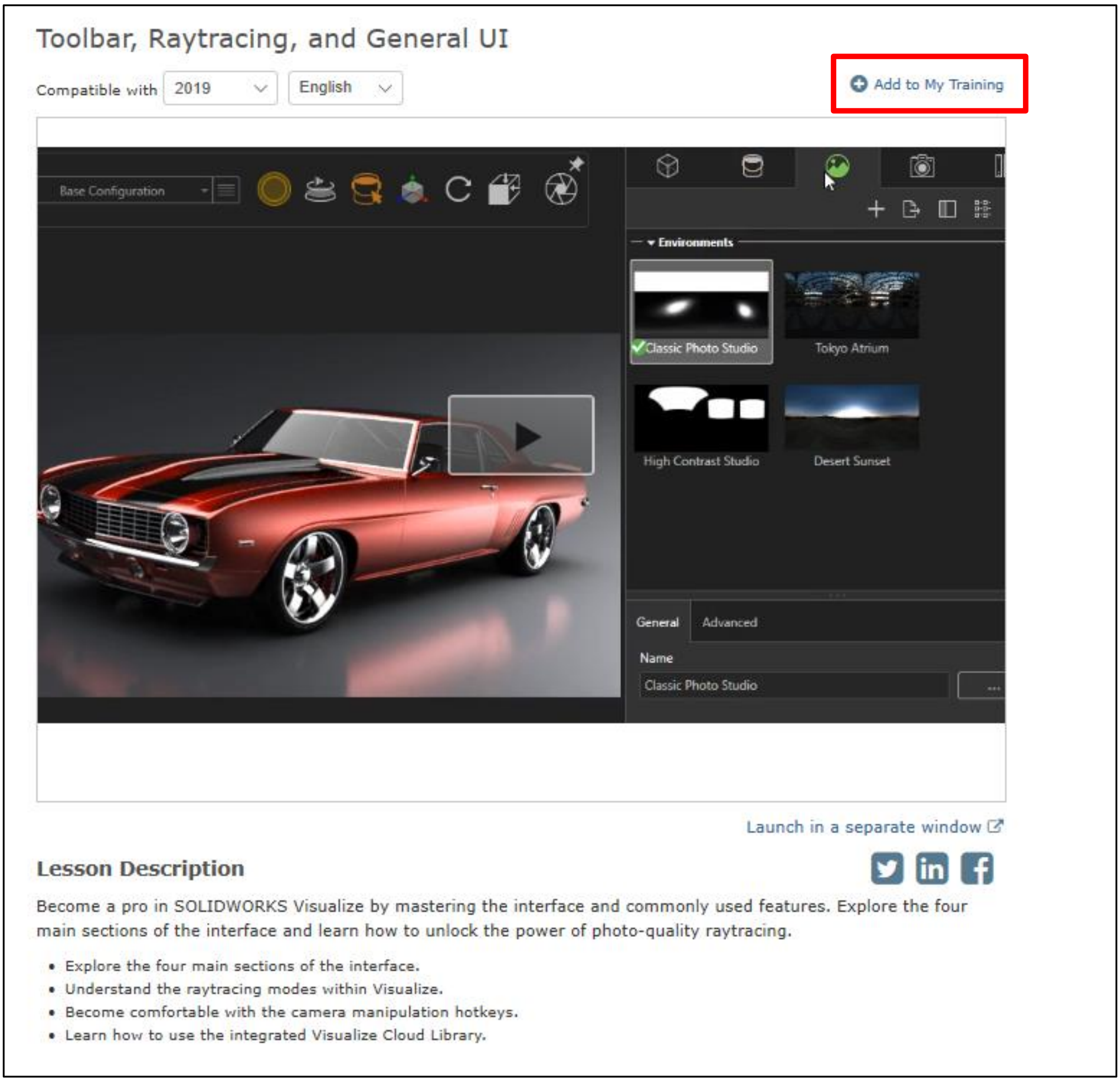
https://my.solidworks.com/training/path/61

SOLIDWORKS Visualize

Add to My Training

Learn about SOLIDWORKS Visualize, which enables anyone to create professional, photo-quality im
textures, and decals to your model, animate your assemblies, and much more.

Lessons	Status	Langua
Import & CAD Live-Update	Completed	ENG
Toolbar, Raytracing, and General UI	In Progress	ENG
Easy Mode Overview	In Progress	ENG
Working With Your Scene Tree	In Progress	ENG
Using Appearances in SOLIDWORKS Visualize	In Progress	ENG
Texture Mapping	In Progress	ENG
Decals	In Progress	ENG
Setting Up Your Scene	Not Started	ENG
Adjust Camera Settings	Not Started	ENG
Image and Video Output Options	In Progress	ENG
Configurations	Not Started	ENG



https://my.solidworks.com/training/lessonviewer/19-ENG-VIS-0001/

https://my.solidworks.com/training/master/333/1/0/toolbar%2C+raytracing%2C+and+general+ui

Create Text & Image Decals using PowerPoint

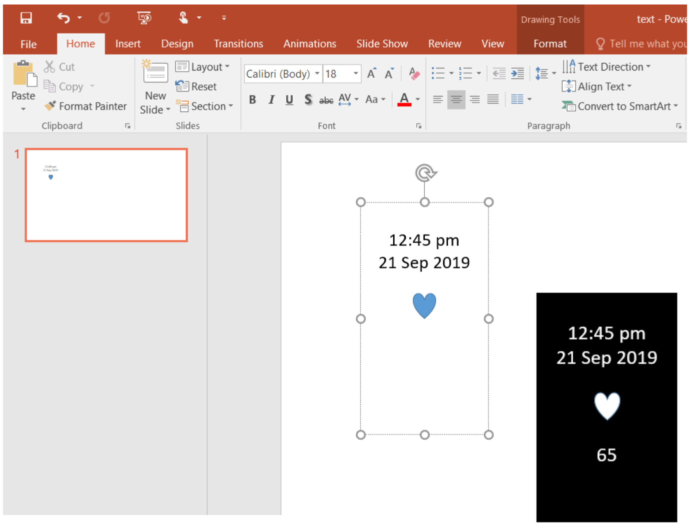
Create a text box insert Text and Images

Format the text box.

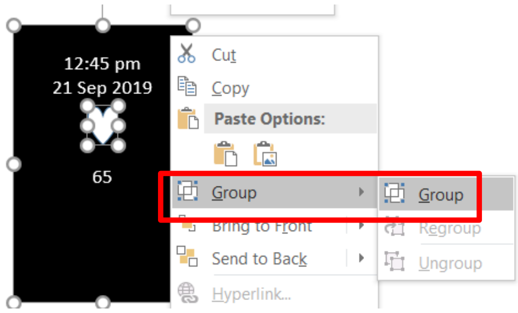
Background Black

Text White

Image White



Group Text and Image



Right click Save as Picture