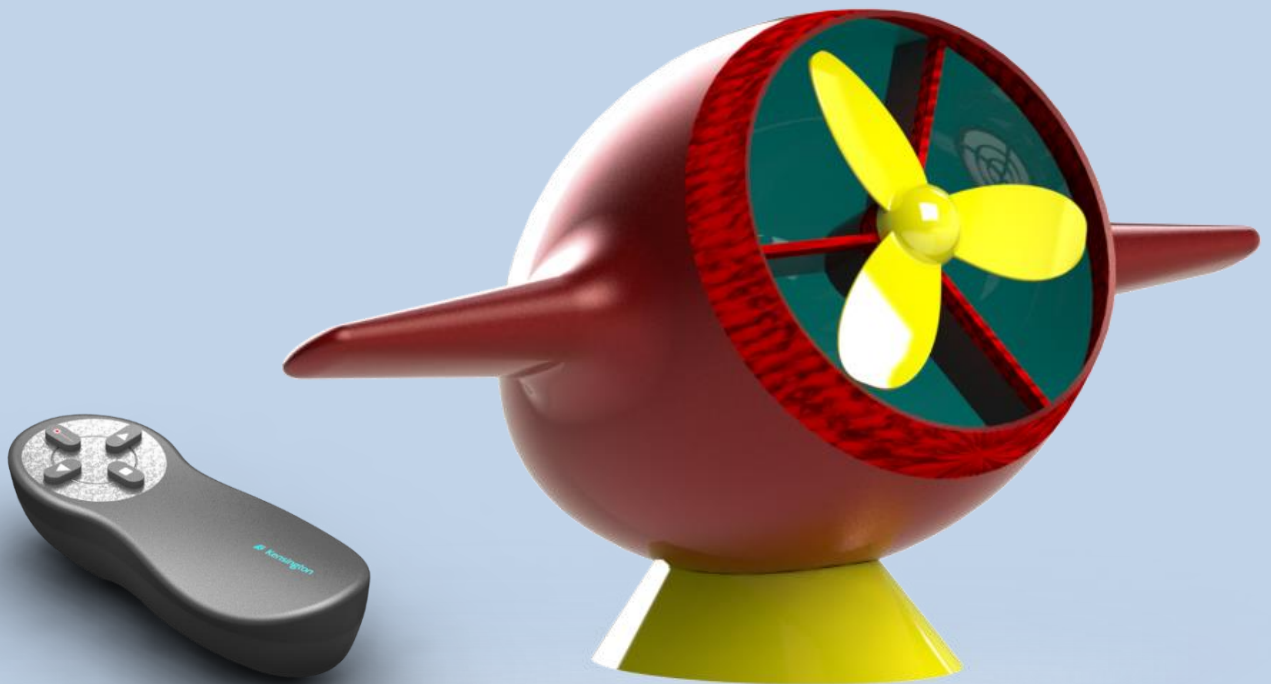




Professional Development  
Service for Teachers

An tSeirbhís um Fhorbairt  
Ghairmiúil do Mhúinteoirí



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## Advanced SolidWorks Workshop 2018/19

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- Surfacing
- Advanced Mates
- Tips & Tricks including Visualize



# Introduction

This workshop aims to upskill and develop teachers understanding of advanced modelling techniques in SolidWorks 2018. The workshop will further interrogate '**Surface Modelling**' techniques for teachers of Design and Communication Graphics while also examining features such as **Flex, Split Line, Rib, Combine, Lip & Groove, Vent**, and advanced **Mechanical Mates**.

## 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 some 'Surfacing' features in a SolidWorks design model.
- Understand and apply the use of non-planar 3D sketching.
- Explore the use of features including Flex, Split Line, Rib, Combine, Lip & Groove, Vent, and advanced Mechanical Mates.
- Explore best practice when using SolidWorks including the use of SolidWorks Rx, Pack & Go and toolbar settings
- Develop and understanding of SolidWorks Visualise, for the creation of enhanced photorealistic images within the software.



## Key Messages for this workshop:

- Surface Modelling can be used to model complex designs within SolidWorks
- The transition from zero thickness surface geometry to solid objects is a necessary and fluid transition in advanced cad modelling.
- The appropriate use of advanced commands such as those outlined above, will significantly enhance realistic parametric modelling.
- 3D sketching (as opposed to sketching on a planar face) is a valuable technique in appropriate applications.

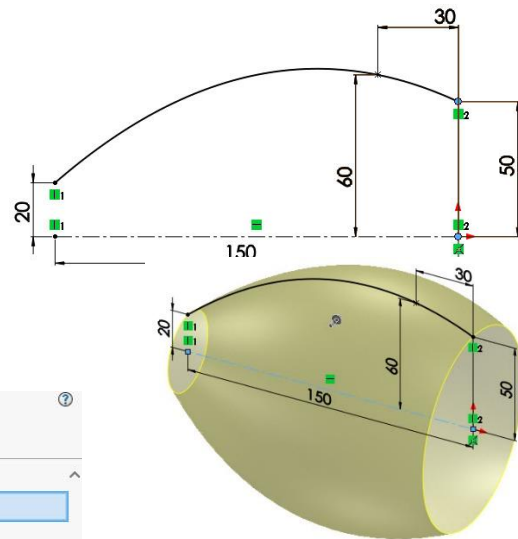
# Desktop Fan

## Desktop Fan Body

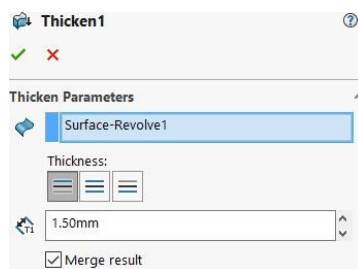
Create a sketch on the **Front Plane**.

Sketch the shown centreline and spline (three point) curve.

Select **Revolved Surface** from the surfaces tab.



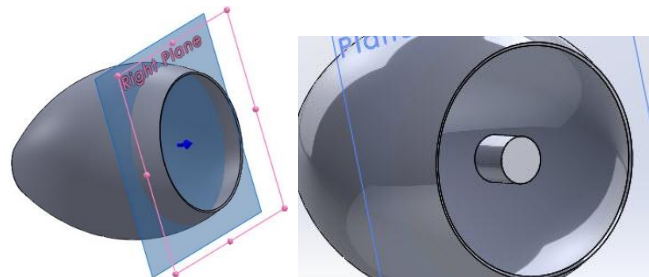
Thicken by **1.5mm** to the inside.



Create a new plane offset by **20mm** from the **Right Plane**.

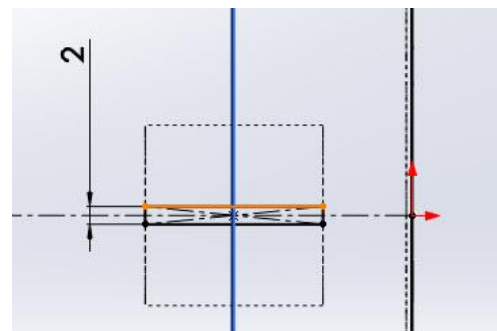
On this new plane draw a circle of diameter **20mm**.

Extrude by **20mm** using **Midplane**.



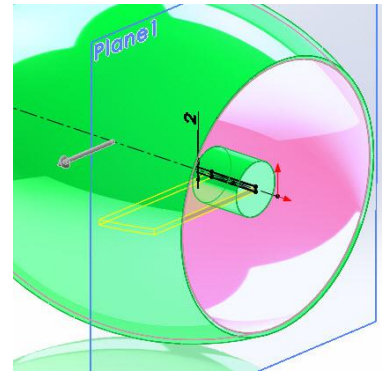
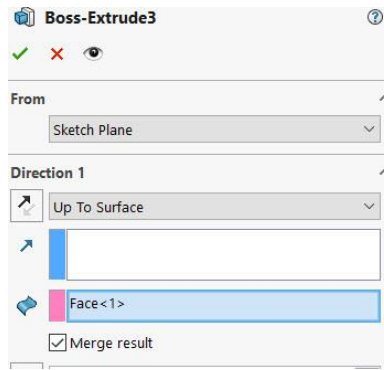
On the **Front Plane** draw the rectangle shown.

It is coincident with the ends of the cylinder with its centre on the centreline.

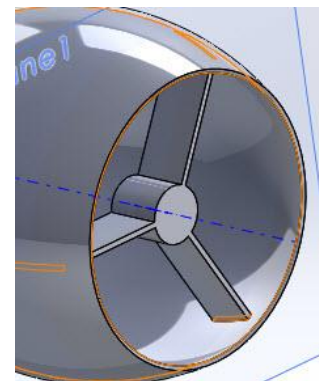
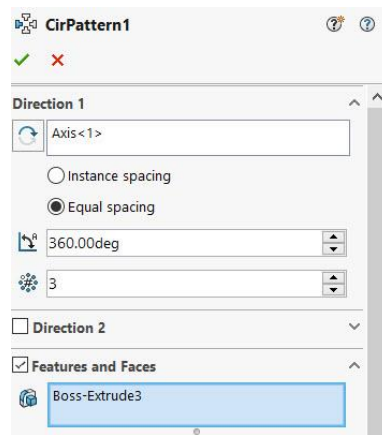




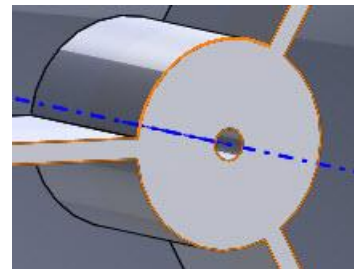
Select **extrude** in the features tab and extrude up to the inside surface of the shell.



Select circular pattern to create the other supports.

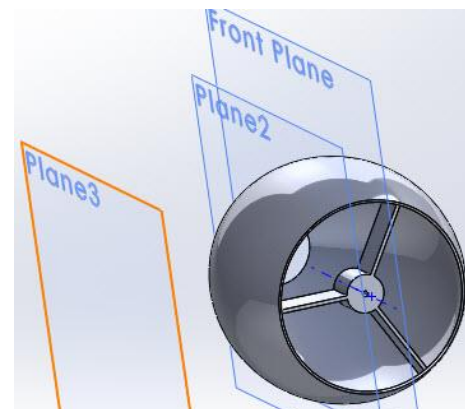


Draw a 3mm diameter circle on the face of the central hub and **Extrude Cut** through all.

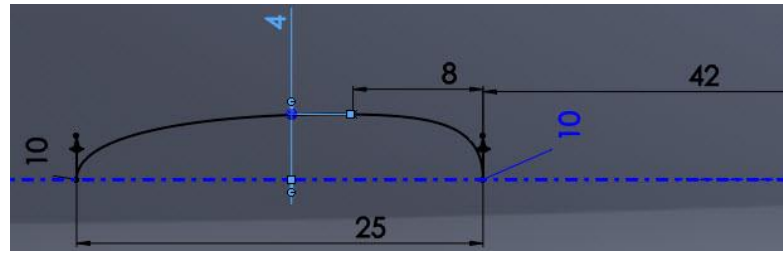


## Creating the Wings

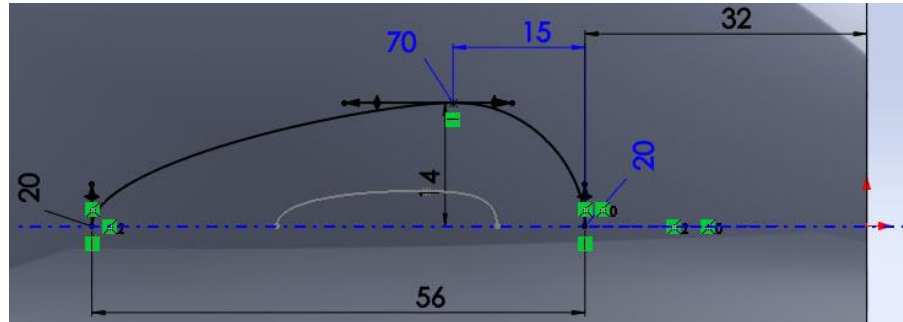
Create two **planes** parallel to the front plane, **30mm** and **120mm** respectively from the front plane as shown.



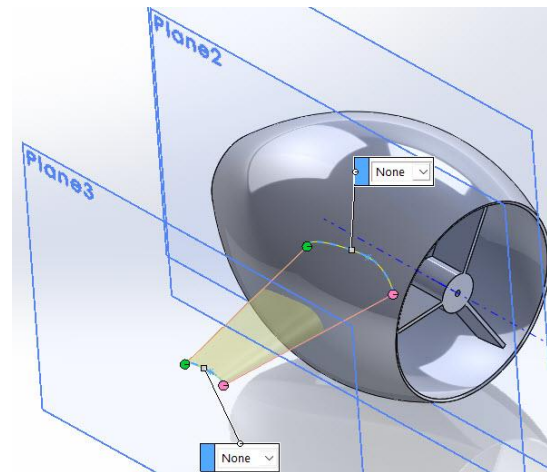
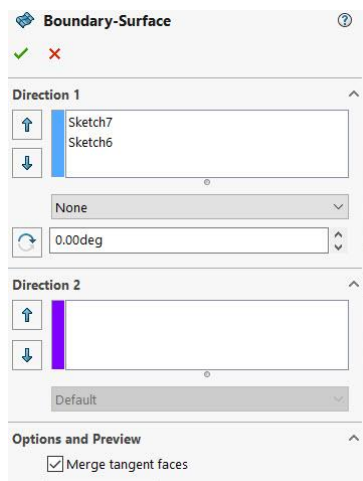
Select plane 3 and draw the 3-point spline on the centreline as shown. Add vertical relations to the two end points of the spline. The spline is 42mm from the origin.



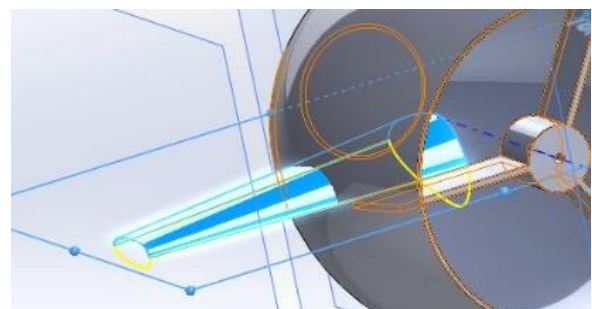
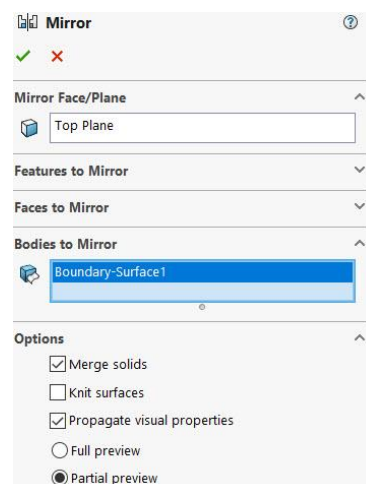
Accept the sketch and draw the sketch shown on Plane 2.



Select boundary surface from the surfaces tab.



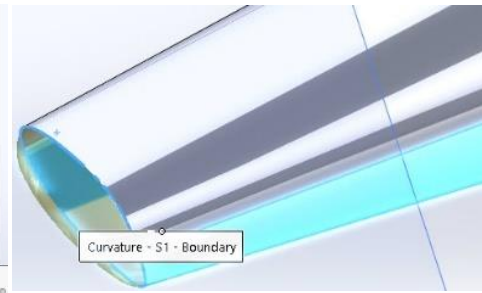
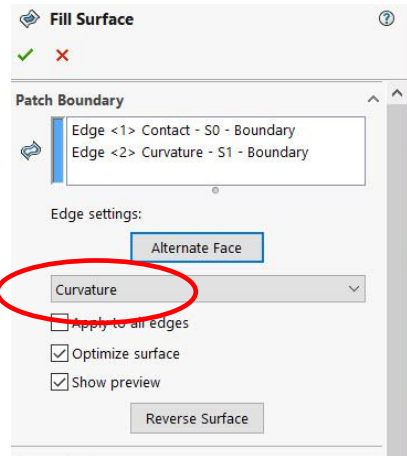
In the features tab select **Mirror** and mirror about the top plane.





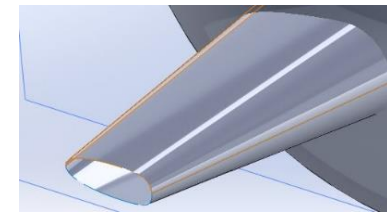
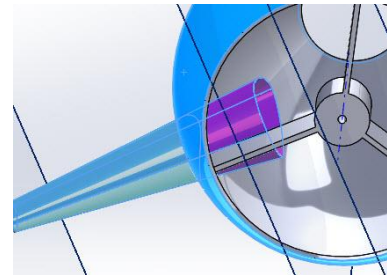
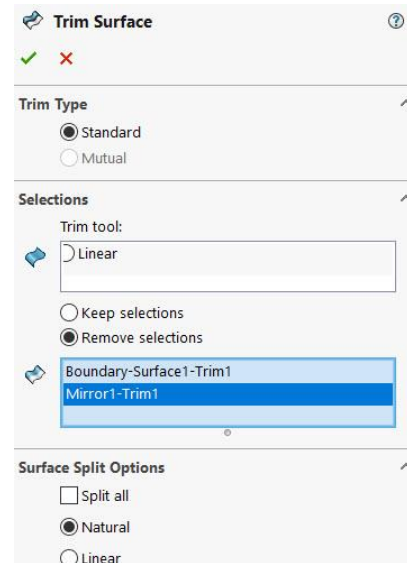


Select the **Fill Surface** command to close the end of the wing.  
Select Curvature as the edge setting.



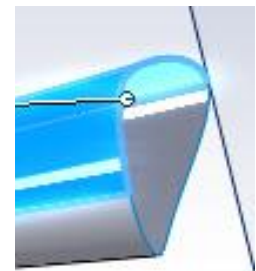
The excess of the wing needs to be trimmed.

In the surface commands select **Trim Surface**. Select the outer shell as the Trimming tool and select the wing bodies to be removed.



Hide the main body to show the other end of the wing.

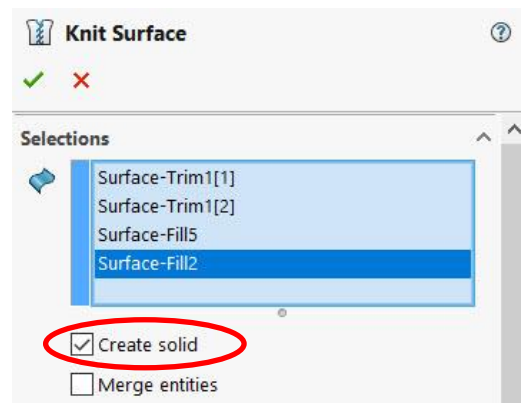
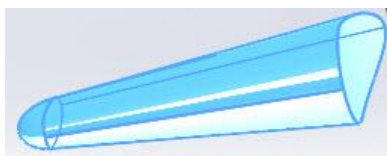
Select **filled surface** and click on the edges to fill in the end.



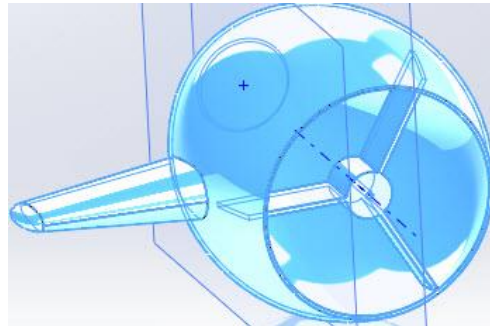
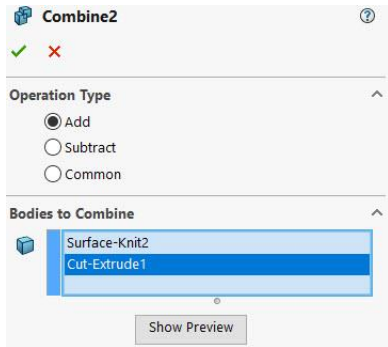
Select knit surface and select the four surfaces of the wing to knit.



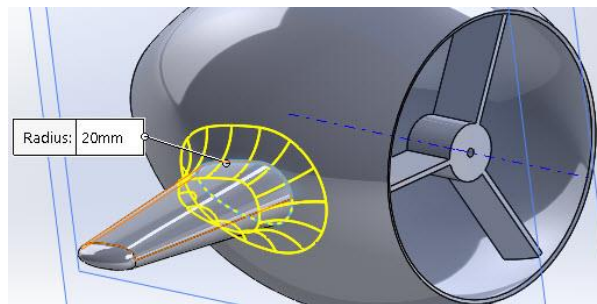
Tick the 'create solid' box.



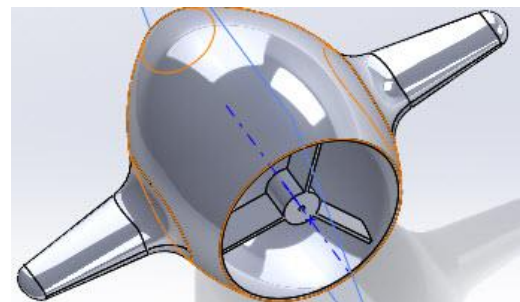
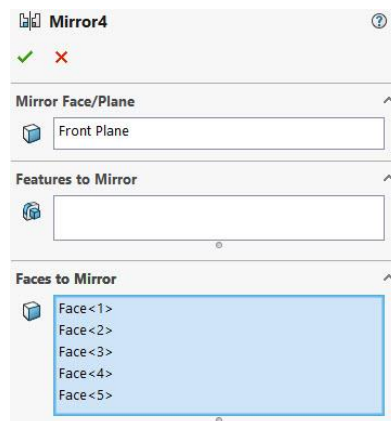
Select the **Combine** command and add the wing to the main part.



Add a **20mm fillet** as shown.



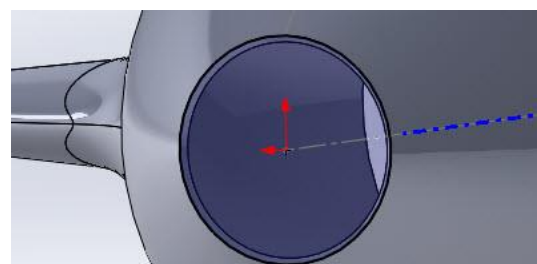
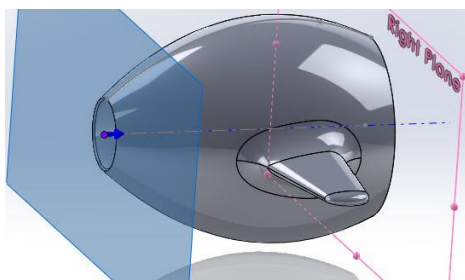
Select the mirror command to complete the wings



## Creating a vent.

Create a plane at the back parallel to the right plane.

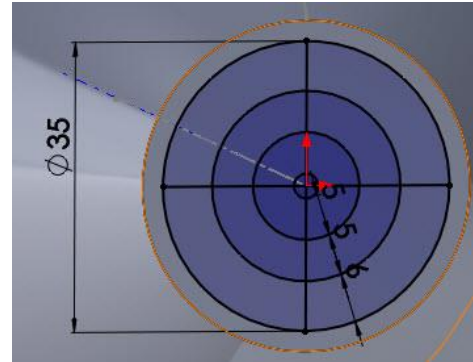
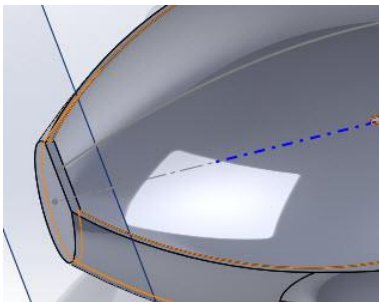
On this plane create the circle by using convert entities.




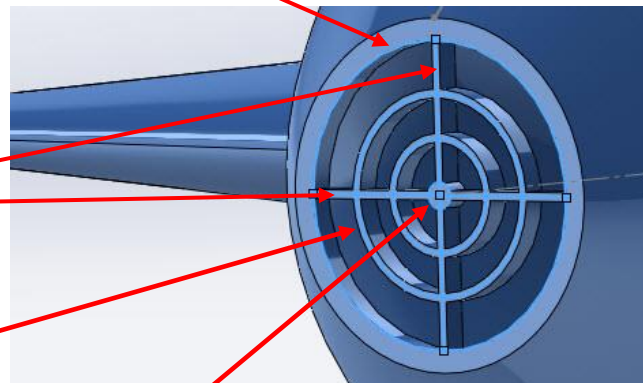
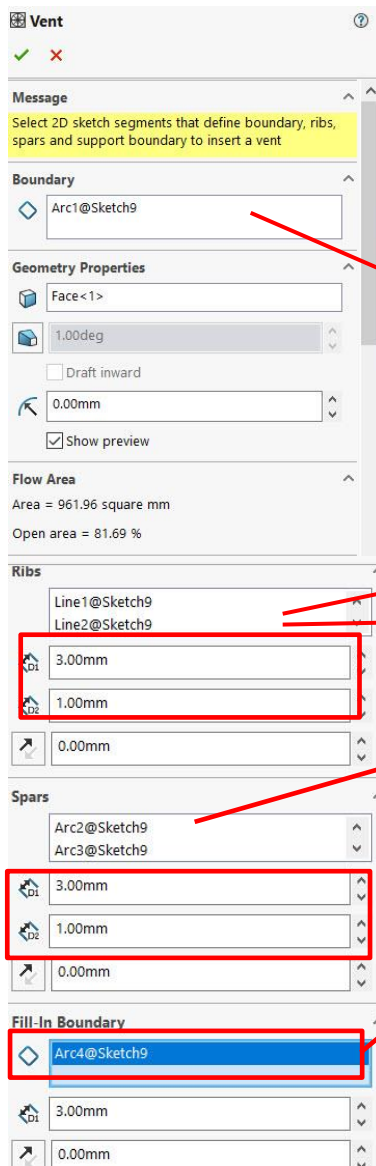


Extrude the sketch inwards by 3mm.

On the back face draw the following concentric circles and vertical and horizontal line.

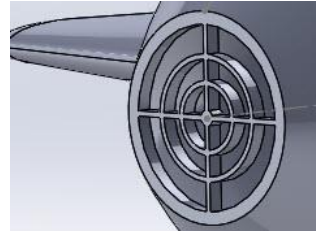


Select the vent  command in the sheet metal tab



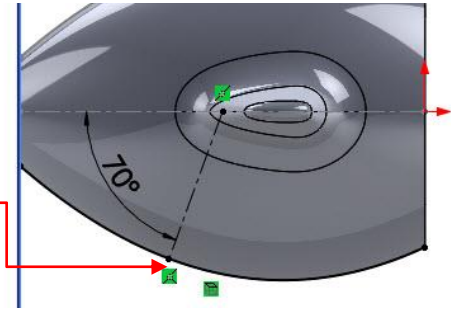
Add the following thicknesses and depths in the relevant columns.

Accept to create the vent shown.

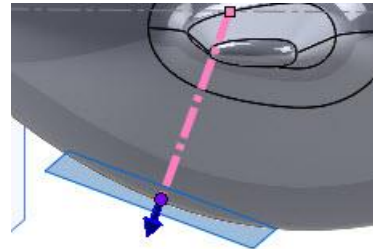


On the front plane draw a centreline at an angle of 70 degrees as shown. The line starts midway along the centreline containing the origin.

Use convert entities to select the bottom arc.



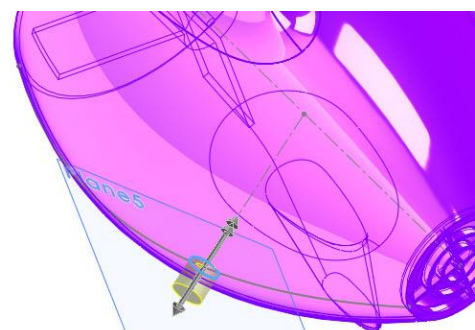
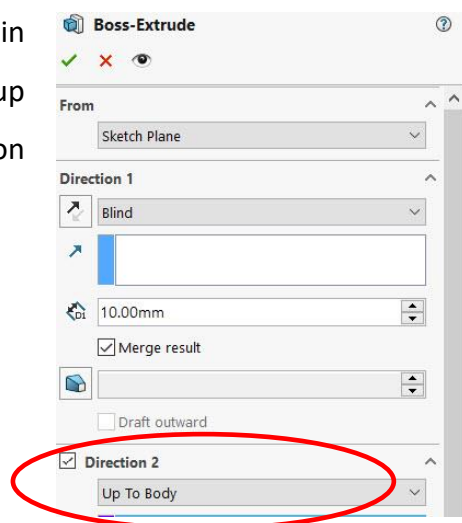
Create a plane perpendicular to the line and containing the end point of the line as shown.



On this plane draw a circle diameter 10mm.



Extrude by 10mm in direction 1 and up to body in direction 2

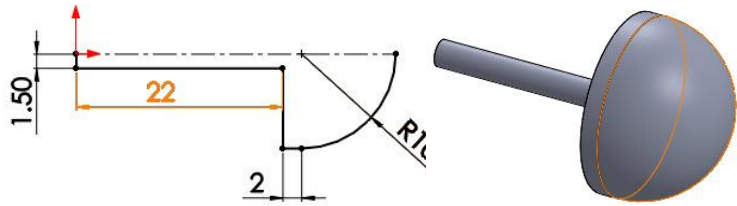


**This cylinder will aid the assembly of the body to the base later.**

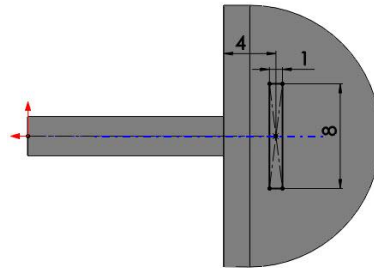
## Fan Blades

Open a new part.

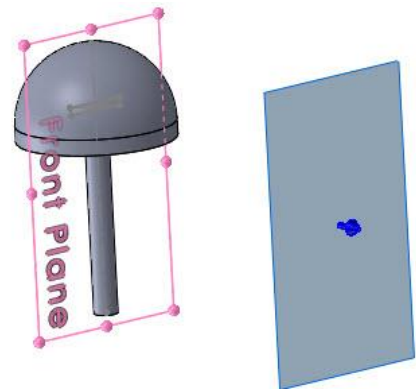
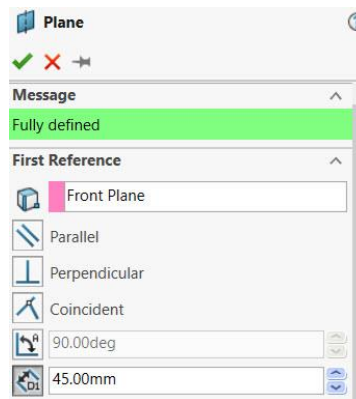
On the **Front plane** draw the sketch shown.



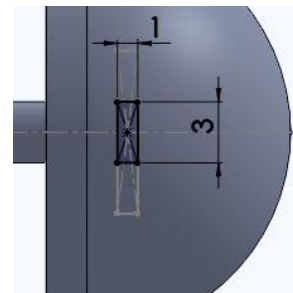
On the front plane draw the rectangle using **centre rectangle**.



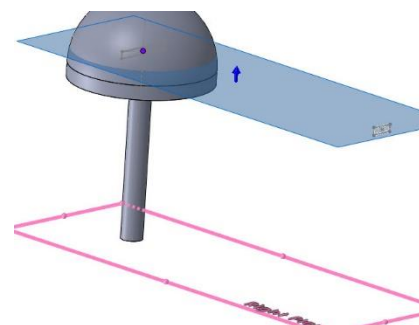
Create a new plane parallel to the front plane a distance of 45mm from the front plane.



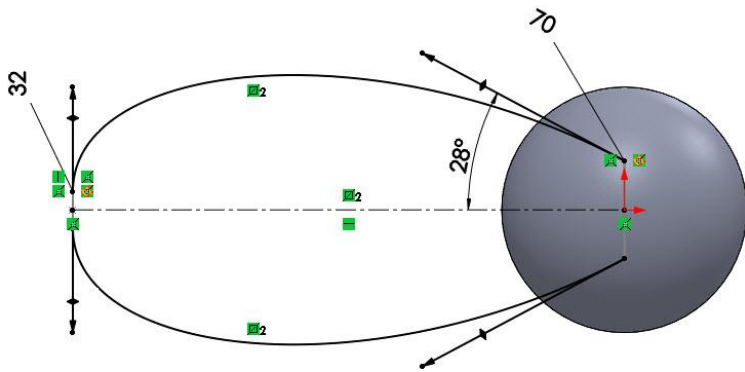
Draw the rectangle with its centre coincident with that of the rectangle on the front plane.



Insert a plane parallel to the right plane that intersects the CentrePoint of the rectangle.

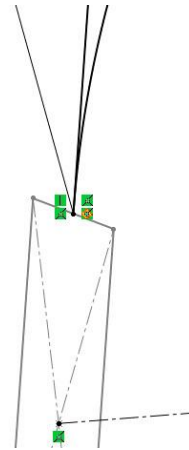


Use two-point **spline** to draw the shape shown.

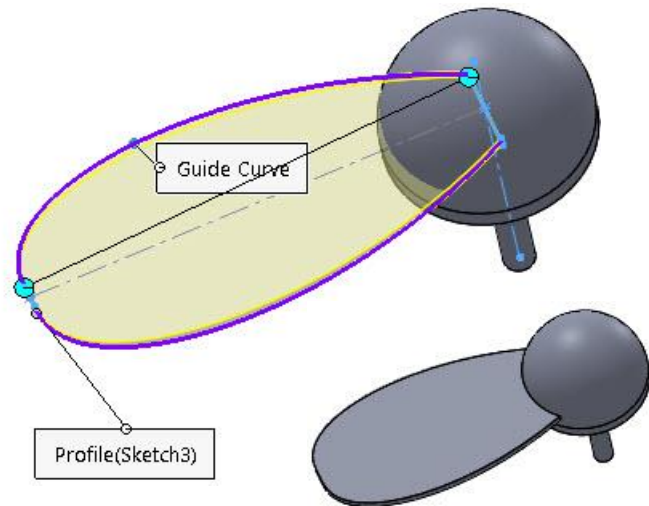
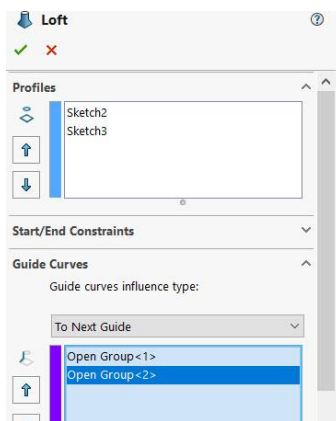


Make a **pierce relation** between the end of the spline and the short side of the rectangles.

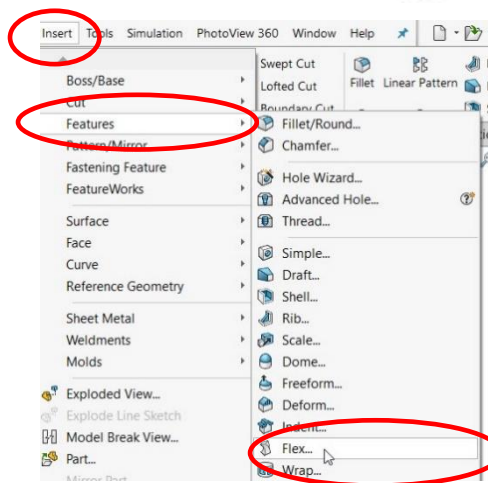
**Mirror** the spline about the centreline.



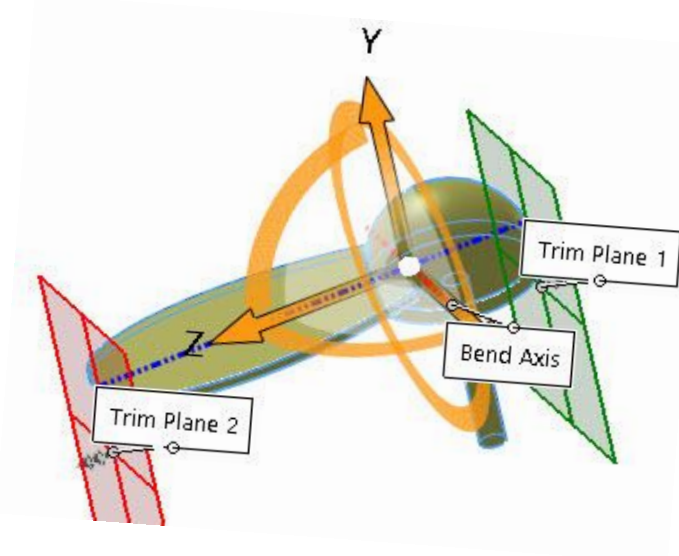
Use the **loft** command to create the blade.



Select the **flex** command as shown.



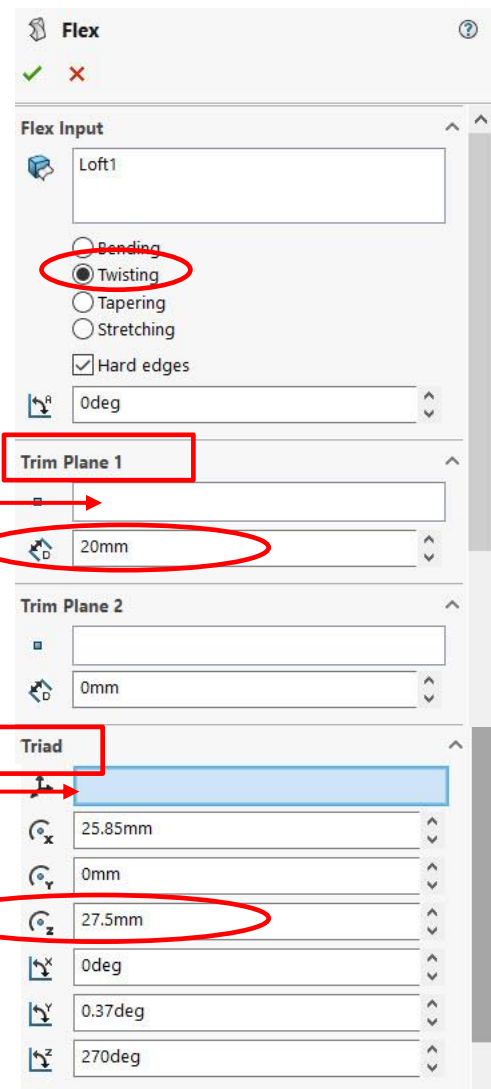
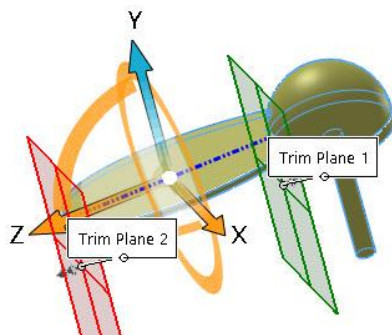
Select the blade in the drawing area to see the following display.



Ensure the twisting button is selected.

To move trim plane1 to its correct position either drag it, using its arrows on the drawing area or activate the trim plane 1 box on the left-hand side.

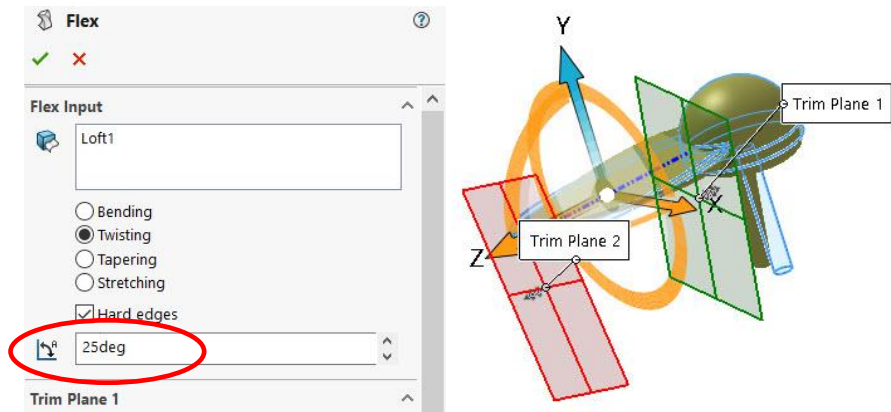
Change the distance to 20mm.



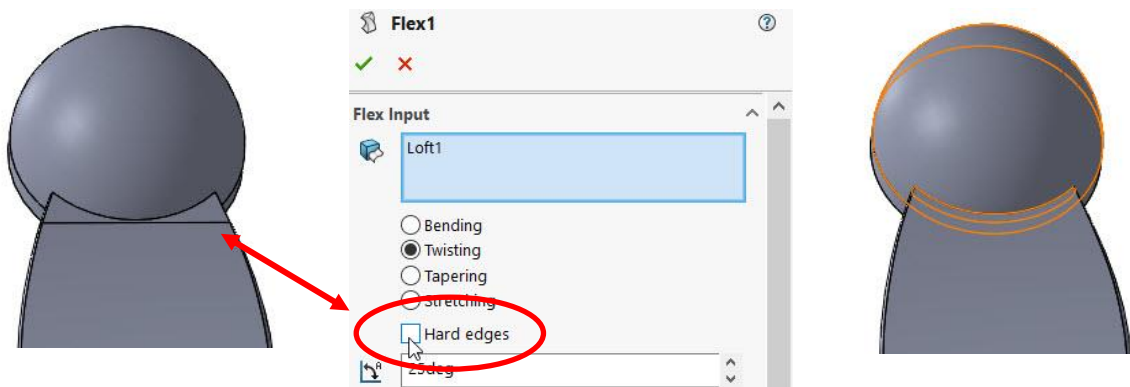
To move the triad, select the box on the left-hand side and change the Z vector to 27.5mm.



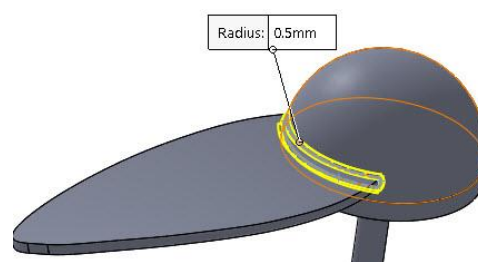
When the planes and arrows icon are in position as shown the blade can be twisted by activating the angle icon and changing the angle to 25 degrees.



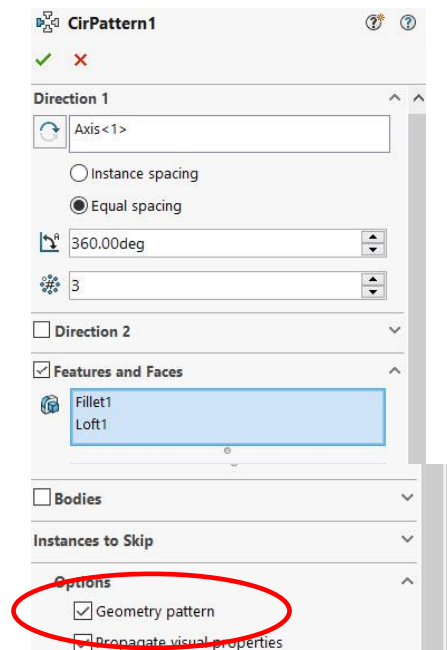
Untick the hard edges box to remove the line on the blade as shown.



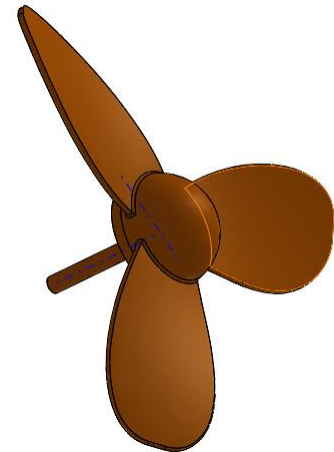
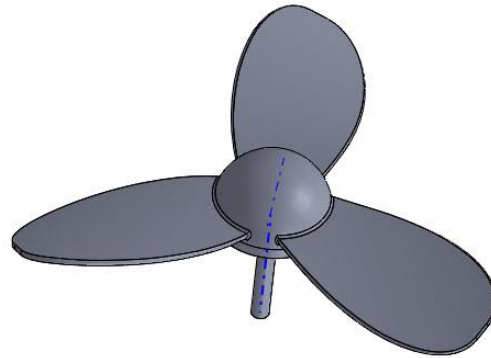
Add a **0.5mm Fillet** to the blade as shown.



Select **Circular pattern** to create the other blades.



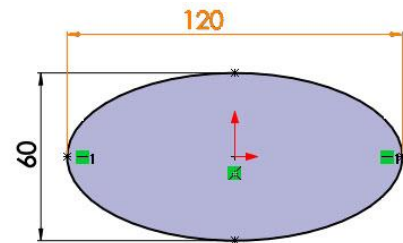
Make sure to tick the geometry pattern box.



Add a **High Gloss Brown** plastic finish to the part.

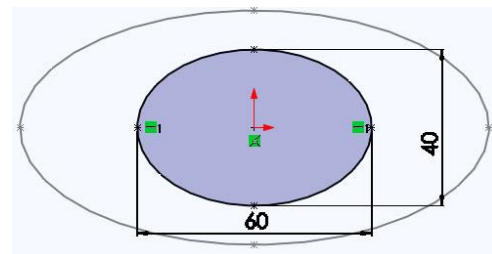
## Desktop Fan Base

On the **Top Plane** draw the ellipse shown.



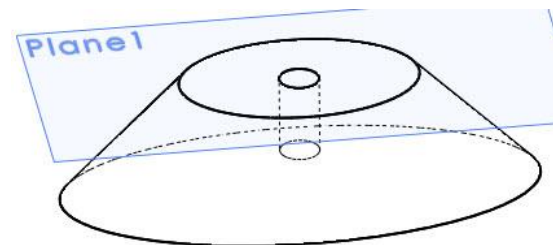
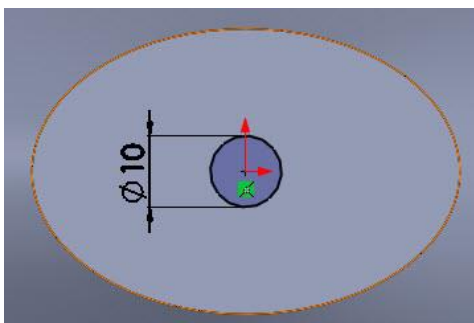
Insert a plane offset **30mm** from the top plane.

Draw another ellipse to the given dimensions.



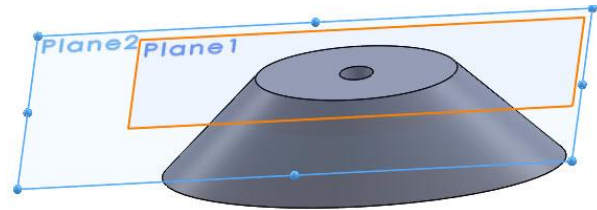
Select **loft** to complete the solid.

Draw a circle on plane 1 as shown and extrude cut by 20mm.

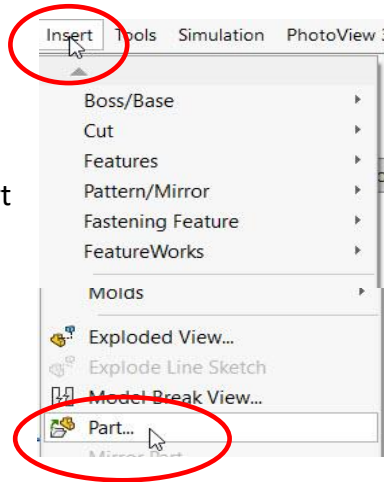


The top of the base needs to be curved to allow the body fit snugly into it. This is achieved by using the Combine feature. The body is brought into the drawing and placed into its final position and the excess of the base is removed.

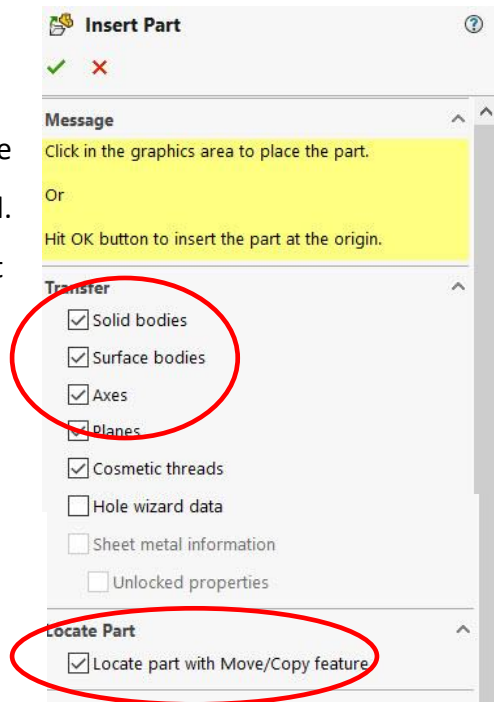
Insert a plane 10mm below plane 1



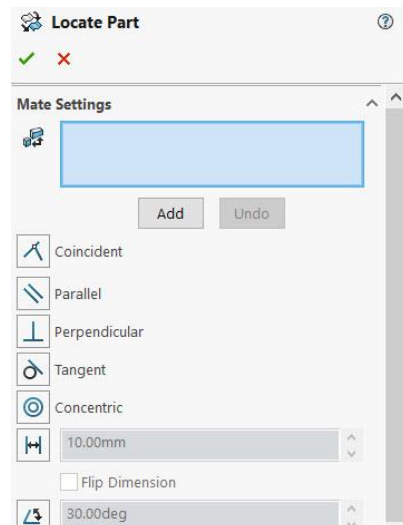
Select Insert, Part



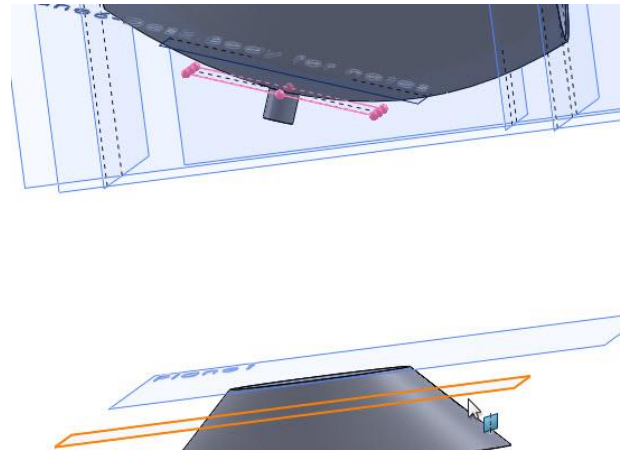
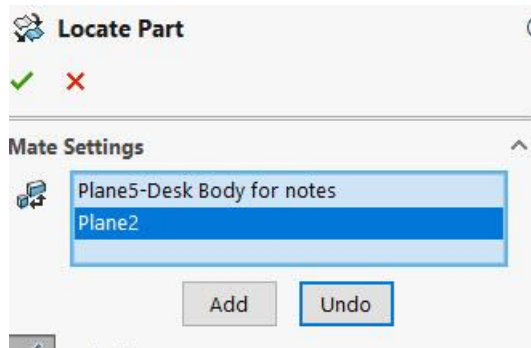
Make sure that the locate part box is ticked. Drop the body part above the base.



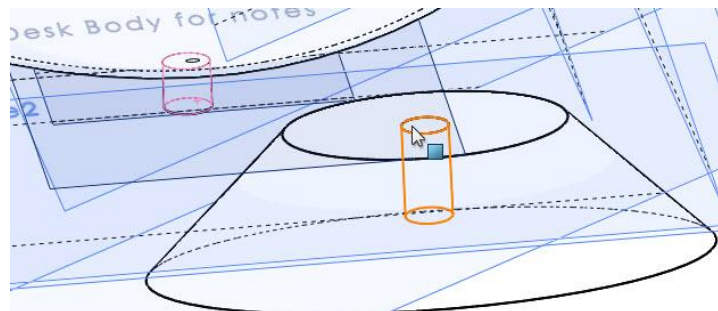
The following mates window appears.



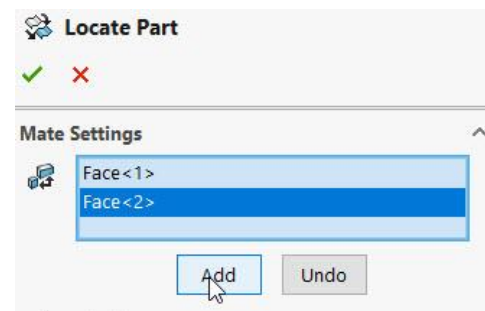
Mate the plane shown on the fan body with the plane highlighted on the base and select add.



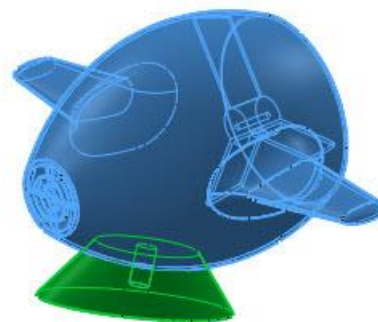
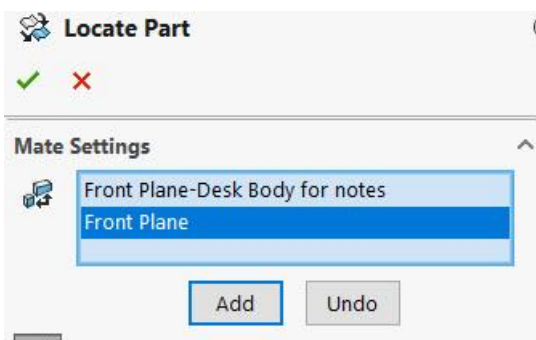
Mate the cylinder at the base of the body with the hole in the base and select add.



Finally mate the front plane of the fan body with the front plane of the base. Select add then accept by selecting the green tick.



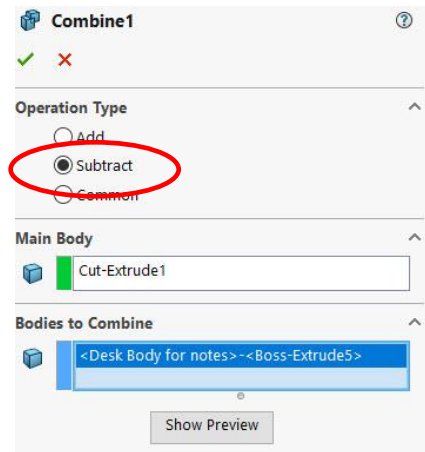
The fan body is now in the correct position.



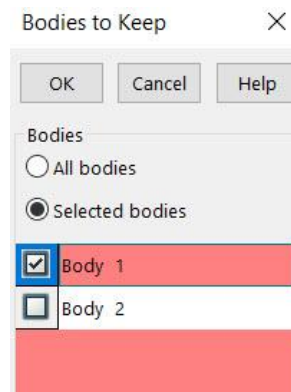
Select the **Combine** command.

Select the subtract button.

Select the base part as the main body and the fan body part as the body to combine.



A window appears. Tick the base part as the body to keep



The base part is now created.

Add a High Gloss brown plastic to the part.





**Assemble** the parts to create the Desk Top Fan.

