**Turbomachinery CAD CWM PART 4**

**RADIAL FLOW WATER TURBINE**

**Exercise 3 - Design of the Water Turbine Casing Wetted Geometry**

**Solid and Surface Modelling**

The task of the *volute* (Drg.Nos.VC001/2) is to provide a uniform radial fluid velocity at the *guide vanes* and the task of the *draft tube reducer* (Drg.No.DT002) is to reduce the exit fluid velocity to provide recoverable pressure energy. In order to do this in an efficient manner, it is important that the design of the internal surface geometry of the turbine casing (Drg.No.TC000) takes account of the relevant energy loss mechanisms. Ether solid or surface modelling can help in this respect and can provide the essential data for computer numerical control machining.

Surface modelling can also be valuable tool to create a simplified shell model for finite element analysis, in order to reduce the number of elements and hence the solve time. This is of particular benefit with large and relatively thin structures and non-linear materials or geometry (i.e. large displacements).

We will use surfacing techniques to create the *draft tube reducer*. Once a closed surface model has been formed, it can be knitted together to form a solid, but first we will build the *volute front* by lofting section sketches around guide curves to directly form a solid model. The *volute rear* can be built by modifying a mirror copy of the front.

**Creating the Volute front (read before attempting).**

Note that the centre of the Volute Front (see Drg. No VC001) needs to be located at the X0, Y0, Z0 position, with the Z axis on the centreline of the Ø122 bore. Start with the 0° to 270° quadrants and build the part in negative Z space.

This method involves creating sketches of four sections at the 0°, 90°, 180° and 270° positions (see the dimension A° in the drawing). Each section comprises two arcs and two lines. One line represents the flat surface 3.5 mm away from the centreline of the volute cavity – see Drg. No VC001). Initially we will build the inner surface of the volute with a constant (5mm) wall thickness, and then build solid material onto that to create the Ø170 face.

The four sections are joined with two guide curves – 'G1' is the outer edge of the volute circular surface (i.e radius D, which starts at the Y= -108 mm position for angle 0°), 'G2' is the inner edge of
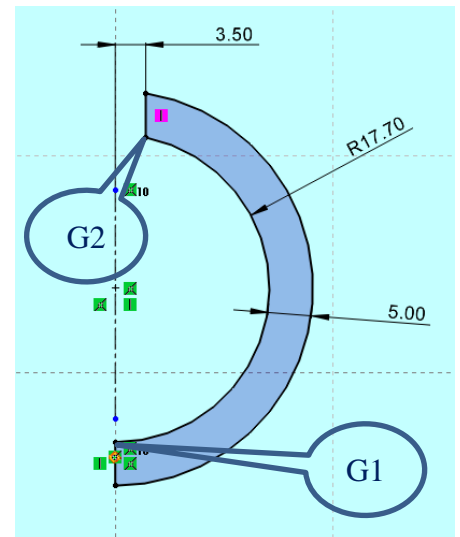


Fig 1. The 90° section (note that X0Y0 is to the top of this). The guide curves pass through G1 & G2.

1

the volute circular surface (i.e. D-ØE offset by -3.5mm from Z0). These curves will pass through the points highlighted in Fig. 1.
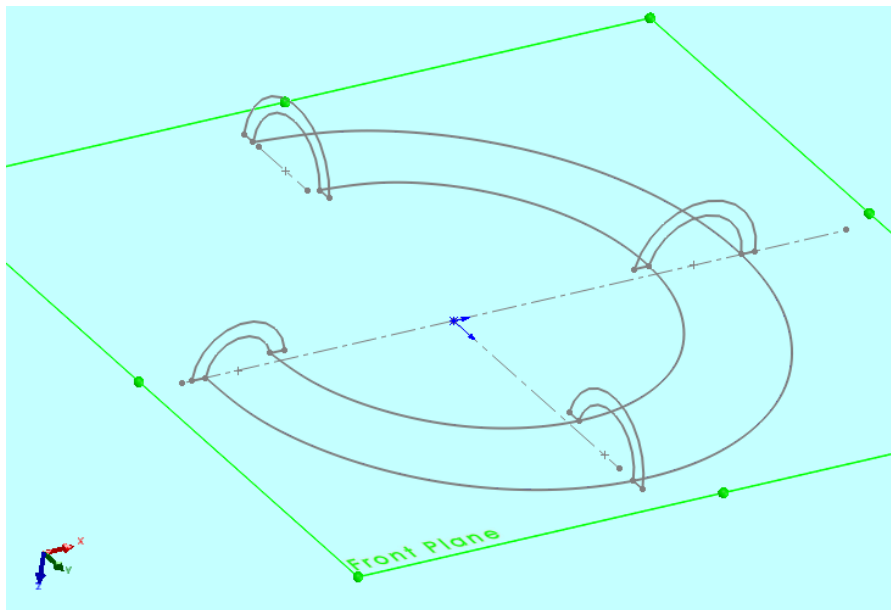


Fig 2. Four cross sections and two guide curves.

Start with the guide curves. Create a spreadsheet to calculate the X,Y positions of the outer guide curve for every 15° from the drawing table (values to two decimal places is sufficient). Copy the calculated values for X and Y into a new sheet and add a Z column with values that should all be zero (but must be included). This new sheet must be formatted with the first 3 columns (A, B, and C) representing the X, Y, and Z data coordinates of the curve, in that order, with no headers. Note that the first part from 0° to 90° is in the +X, -Y quadrant, the second 90° to 180° is in +X,+Y, and the final 180° to 270° is in -X,+Y. The curve should start at (0,-108,0) and end at (-88.7,0,0).

Save this as a **text file** *.txt (tab delimited), making sure that you have the correct active sheet. This is necessary for importing the data to SolidWorks. Read it in using **Insert, Curve, Curve through XYZ points…** Browse to the *.txt file saved earlier (you will have to set the file type to Text Files (*.txt)), wait for the values to load, and select OK. You may have to **close** the Excel file first, or it won't load. Create a sketch on the X,Y plane at Z0, and use **convert entities** to generate the guide curve.

Repeat for the 'G2' curve, but this time the Z values must all be -3.5 mm. Don't forget to allow for the effective increase in curve radius due to the 3.5mm offset. Create a new plane from the **Features** tab using the **Reference Geometry**, **Plane.** Use the **Front Plane** as the **first reference**, and 3.5 as the distance (if necessary, **flip** the offset to position the plane in negative Z)**.** Create a sketch on the plane, and use convert entities to generate the guide curve. Save the part as **VC001**.

2

Each of the four cross sections should be on a separate sketch. The outer end of each relevant arc should coincide precisely with the G1 guide curve. To ensure that this happens, sketch the part circle with the correct radius, but allow the centre to 'float' along a centreline from the sketch origin to the outer edge of the volute. Place it so that the arc ends near to the guide curve, then select the point at the outer end of the circular arc and while pressing the control key select the guide curve. Select **Pierce** from the **Add Relations** part of the properties form. The two curves should

coincide. The pierce point symbol…

You should now be able to create a Lofted Solid using the four section sketches as **Profiles** and select the 2 other curves in the **Guide Curve** box. Use the **Features** tab, **Lofted Boss/Base**. Select the same relative position on each of the cross sections as you pick them (e.g. the outer edge of the larger radius) to avoid the loft becoming twisted. The pick points can sometimes be moved after the preview is created to correct any errors.
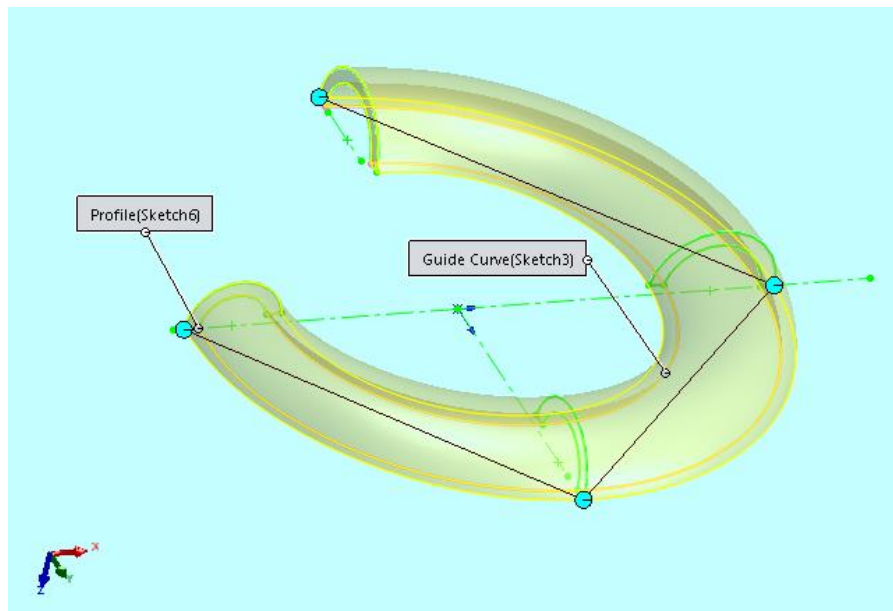


Fig 3. Generating the lofted solid (showing 4 pick point 'dots').

The next step is to form the last quadrant of the involute. This is done in a similar way to the first three quadrants, but this time create section sketches every 15°, with a final 'closing' section at 338°, and just use one guide curve 'G3' which is the outer edge of the volute circular surface (i.e. radius D, which starts at the X= -88.7 mm position for angle 270°). Create a sketch on the **Top Plane** consisting of a **centre line** through X0, Y0 in the negative Z direction as a reference for the new planes. Create new planes from the **Features** tab using the **Reference Geometry**, **Plane**. Use the **Top Plane** as the **first reference**, and 15° as the angle, and the sketched centreline through X0, Y0 as the second reference. **Flip the offset** if necessary. You can create all four 15°planes in one go by typing 4 into the **#** box. Add the 338° plane in a similar way. The sections at 285°, 300°, 315° and 330° follow the same general profile shape as that in Fig. 1. The 270° sketch can be selected by expanding the first lofted feature in the tree. You may find it helpful to hide each

3

section after you create it to make it easier to see the next sketch.

The final section at 338° comprises 3 lines and one arc, as shown in Fig. 5 with the objective of getting the lofted curve to develop into a straight edge. The R5 arc is a sketched 3-point arc dimensioned to R5. The surface generated by this is not critical and will disappear as other parts of the model are added on. Create a dimension to the edge of the curved surface of each sketch (dimension D) to locate the sketch relative to X0Y0, then create the guide curve by drawing a sketch on the Front Plane and creating a **spline** between all of these points (Fig. 6).

Use the **Features** tab, **Lofted Boss/Base** to create the lofted shape from the 6 sketches and single guide curve Fig. 7.
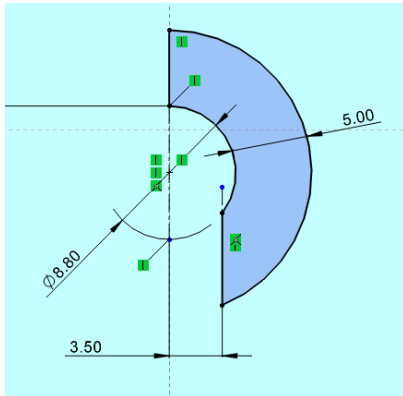


Fig 4. The 330° section.
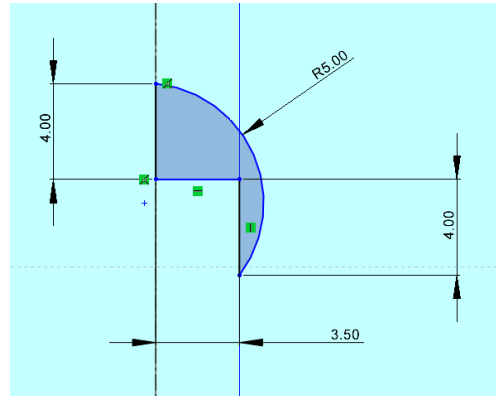


Fig 5. The 338° section.
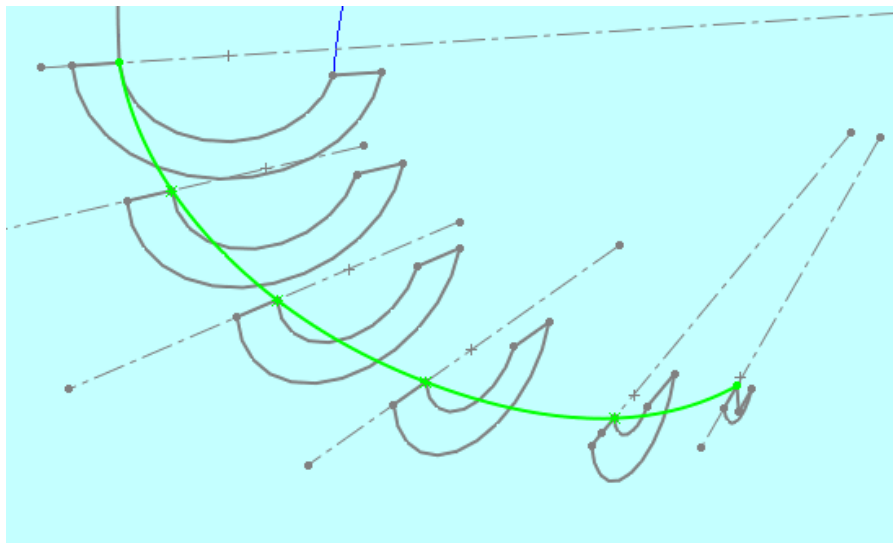


Fig 6. Creating the spline (guide curve) – 270° to 338°.
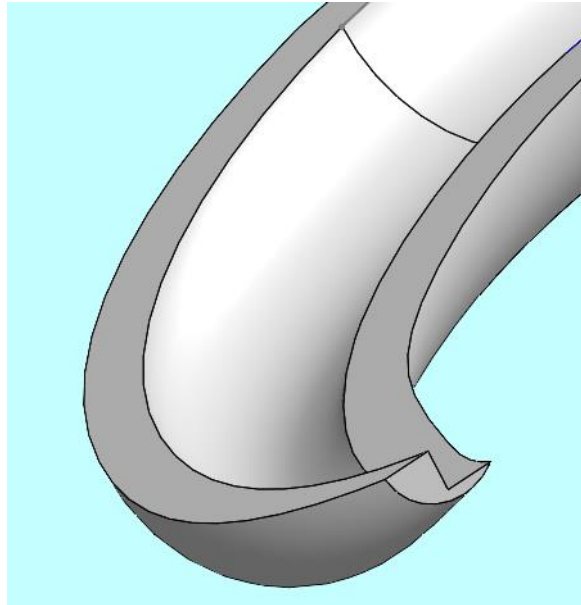
Fig 7. Finished loft shape -4<sup>th</sup> quadrant.

The next step is to extrude the surface at 0° to 140mm length, to build the volute entry pipe. Create a sketch on the surface and use the **Convert Entities** option and pick the surface to create the geometry. Extrude to the correct length, making sure that the Merge Entities box is ticked.

To complete the full diameter of the entry pipe, create a sketch on the end surface, and extrude it by 100mm (The end will get blended in later). To create the centre-point arc shapes in the sketch, hover the mouse over the pipe curve and it will highlight the centre.
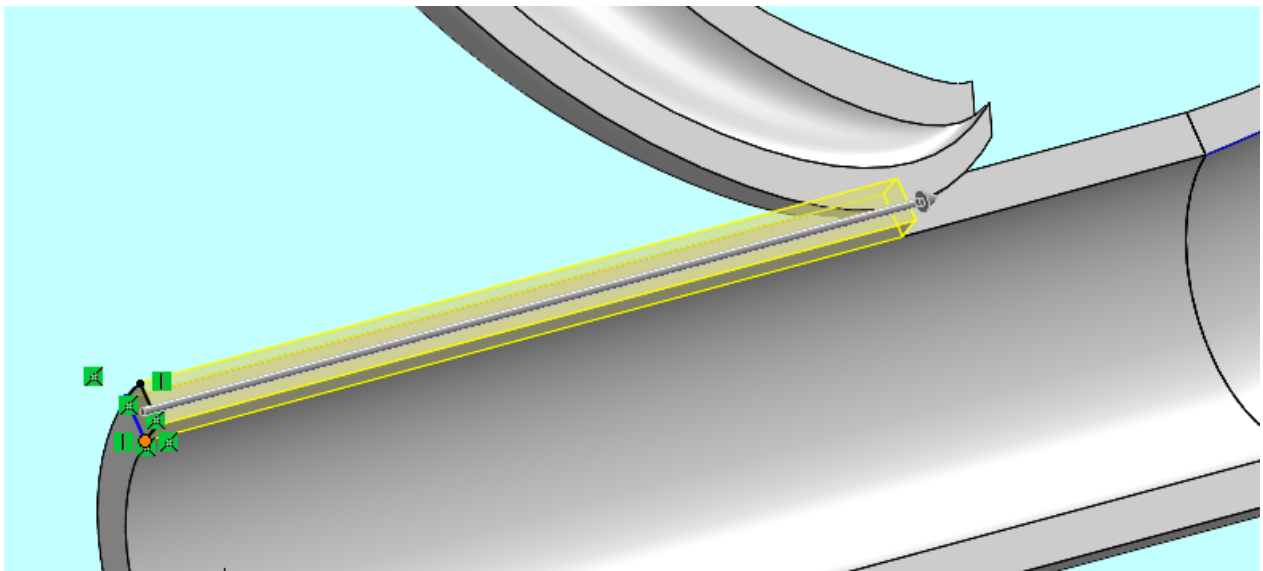


Fig 8. Extrusion closing the entry pipe.

Next, build up the Ø170 boss on the outside surface of the Volute. Create a sketch on a plane parallel to the **Front**, and -24mm distant in Z. Draw two circles on the

5

origin of diameter 170mm and 150mm. Extrude the resulting ring using the **Up to body** option and pick the outer surface of the volute. We want the bore of the boss to be Ø122, but if we had chosen that diameter we wouldn't be able to extrude "up to the body", so we have to create this in stages. Extruding to a fixed length will cause the boss to show on the internal surfaces of the volute, which we don't want.

Create a sketch with a Ø122 circle on a plane at Z=-3.5. In order to fill this shape to create an extrusion, generate three offset curves 2mm inside the volute edge and draw a straight line to join the ends (Fig. 10). Extrude this 'ring' shape by 2mm in negative Z (Fig. 11).
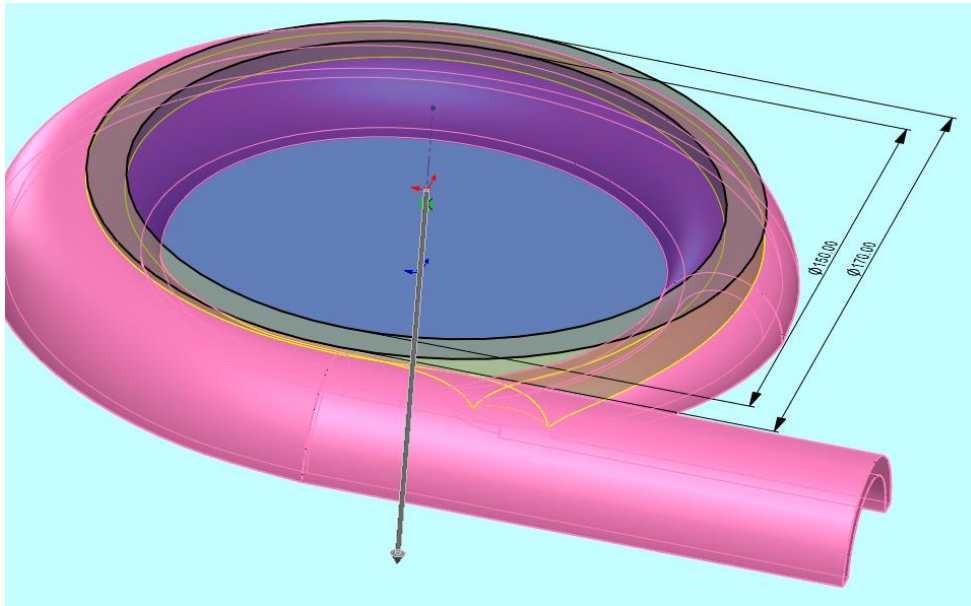
Fig 9. Extruding the Ø170 boss 'Up To Body'.

Fig 11. Ø122 Extrusion x 2mm deep
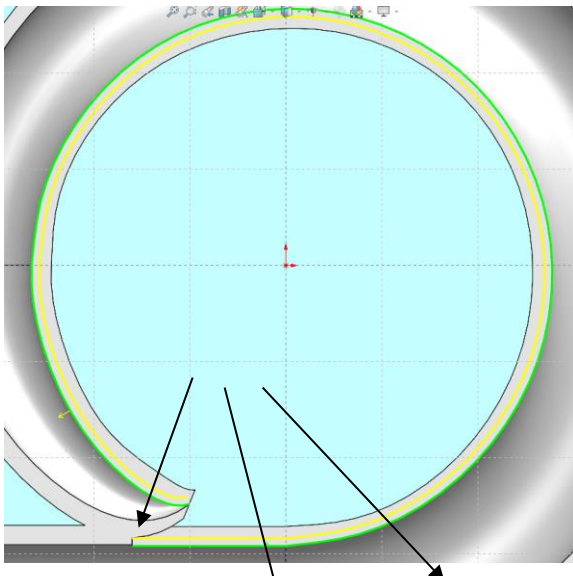
Fig 10. Ø122 Extrusion – the three 2mm Offset curves.
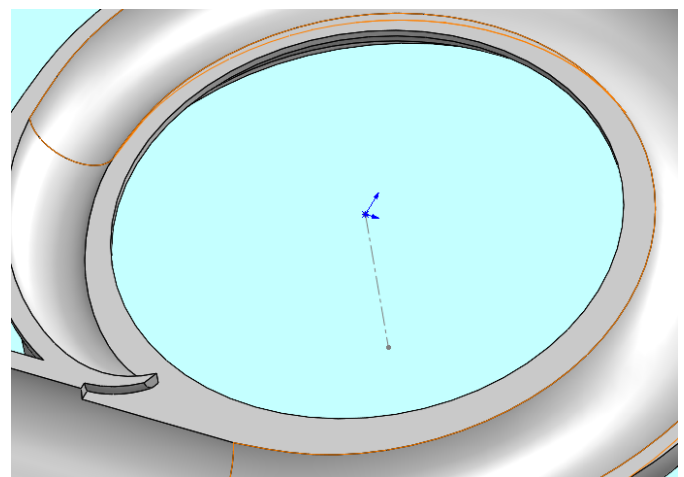
Now create another sketch on the outer surface at Z=-24. This sketch will comprise a ring of ID 122mm, and the OD is created by using **Convert Entities** on the Ø150 edge. Extrude this using the **Up to body** option and pick the outer surface of the volute (Fig. 12).
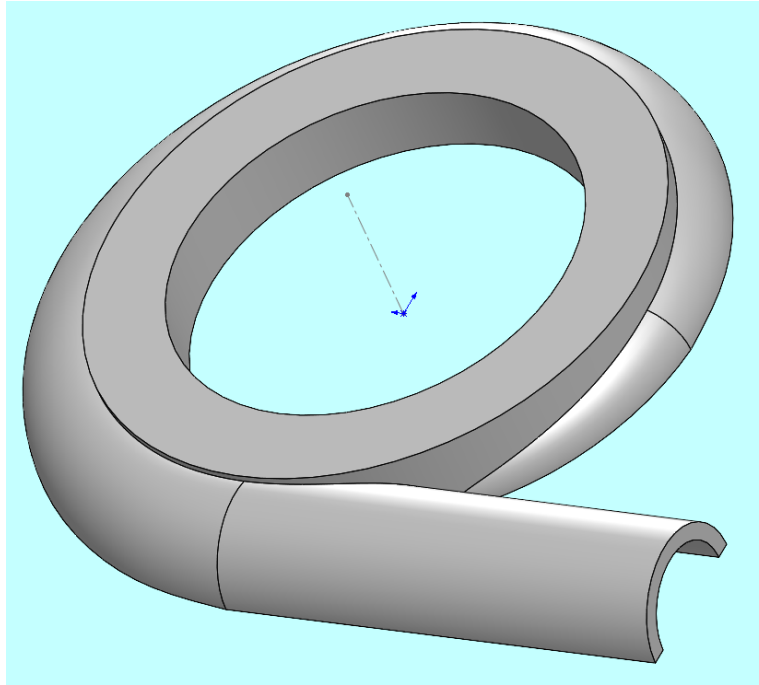


Fig 12. Ø170 boss completed.

Create a 4mm fillet along the edge where the Ø 170 boss meets the rest of the part and include the internal corner where the entry pipe joins the volute (Fig. 13).
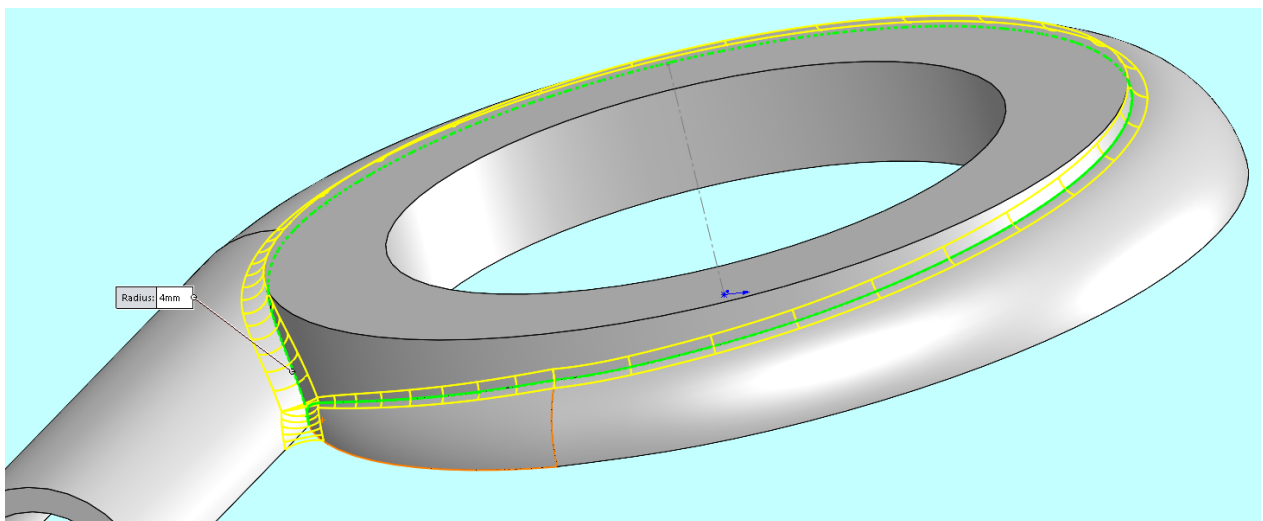


Fig 13. 4mm fillet

The final modelling step is to blend in the spur. Draw a sketch on the Z=-3.5 face of the spur (Fig. 14). Convert the curve and straight line edges and 'close' the profile with a 3 point arc dimensioned to be R15.
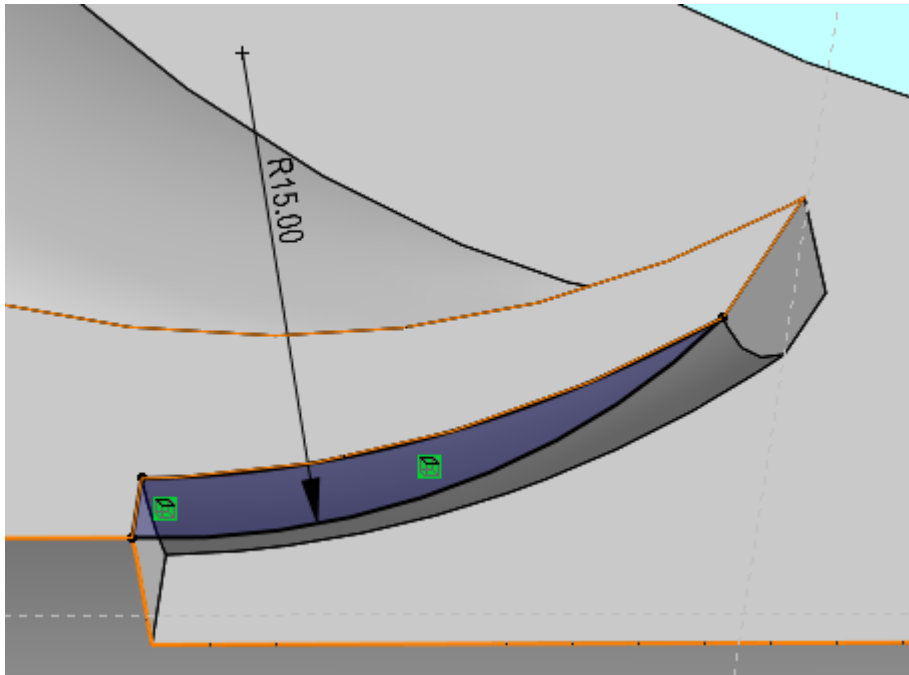
Fig 14. Spur blend sketch with R15 3 point arc.

Create a **Swept Boss/Base feature** of the sketch using the internal corner edge as the **Path**, and the two external corners as **Guide Curves**. Tick the **Merge Smooth Faces** box, and in the **Options** tab select **Keep Normal Constant** for the **Profile Orientation**.
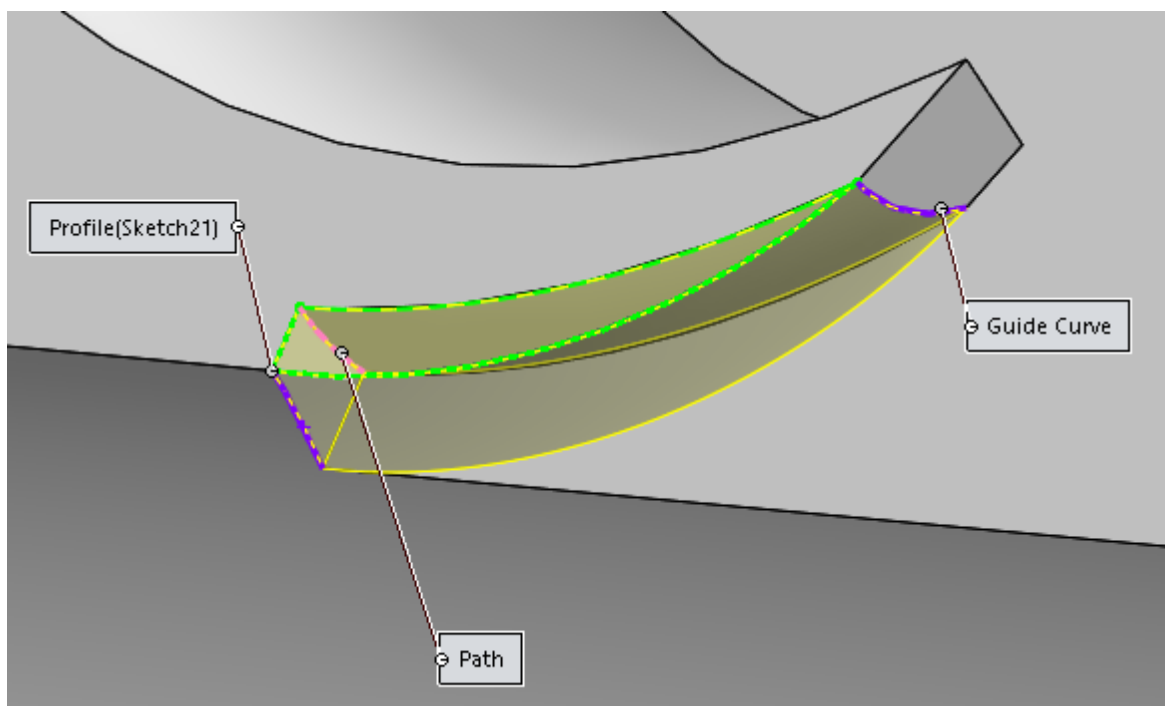


Fig 15. Swept Boss/Base feature for Spur blend.

8

Finally, use Edit Material to select Perspex.

## Creating the Volute rear

Congratulations! You have created the *volute casing front* (Drg VC001). Now you can create the *volute casing rear* (Drg VC002). This will be much quicker and easier!

Create a mirrored part... (Make sure the part is saved first).
**Select** the face to mirror the part (i.e. the flat face on the centreline of the volute chamber).
Then use the menu item **Insert, Mirror Part…**

Tick the **Solid Bodies**, **Surface Bodies, Axes, Planes, Absorbed sketches,** and **Unabsorbed sketches** boxes in the **Transfer** window. Tick the **Break link to original Part** box in the Link window. Then complete the command with the **Green tick**.
If you get a message stating 'The template used for making the derived part has different units…' select **Yes** to change the units of the derived part to those of the base part.

**Save** the part as **VC002.**

**Right click** on **Material** in the tree and select **Edit Material**. Select **Malleable Cast Iron** from the **Iron** folder. Select **Apply** and **Close**.
**Save** the part.

Now modify the part as required to match the VC002 drawing. Create a sketch on the inner flat face of the part, and draw two circles of diameter 35mm, and 127mm. **Extrude Boss/Base** using **Up To Surface** and pick the outer flat face.

Cut extrude the Ø80 x 3.5mm recess on the internal flat face.

Cut extrude one of the Ø8 through holes on the Ø105 pitch circle. Show the temporary axes and create a circular pattern of the hole to result in 6 equi-spaced instances to finish the part.
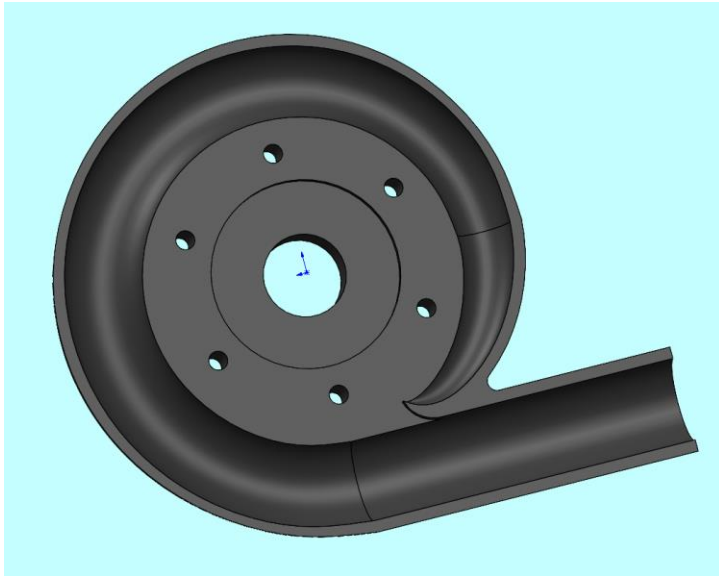
Fig 16. VC002 – bore, recess and the 6 equi-spaced  holes.

**Creating the Draft Tube Reducer using surface modelling methods.**

Create a new part in SolidWorks and refer to the *draft tube reducer* (Drg.No. DT002) The internal shape will be constructed from **revolved** surfaces, and **lofted** surfaces, which are generated between two or more profiles.

Start with the smallest diameter straight and tapered bores. Create a sketch on the **Front Plane** consisting of a centreline through the origin, a parallel line starting at X0,Y18 and ending at X20,Y18. Then create another line from that point and extending in the +ve X and Y direction at 4° to the horizontal and ending at X =95mm.

If the **Surfaces** Command Manager tab is not visible, then right click on the **Features**, or **Sketch** tab, and select '**Surfaces**', or use **Tools**, **Customize**…, then tick the **Surfaces** box in the **Toolbars** tab. The **Surfaces** icons are now more easily accessible (usually on the left-hand edge of the screen).

Create a **Surface-Revolve** from the sketch, using the centreline as the **axis of revolution,** and selecting **blind** & **360°** in the **Direction1** form**.**

The next part will be a **Swept Surface**.
Create a **Plane** by picking the edge of the largest circle as the **First Reference**. Then make a sketch on the plane and use **Convert entities** to create a circle of the large diameter. Make a sketch on the front plane of an arc of radius 45mm, extending from the centre of the circle for 70°. Exit the sketch. Create a **Swept Surface** using the circle sketch as the **profile**, and the arc as the **path** (Fig. 17).
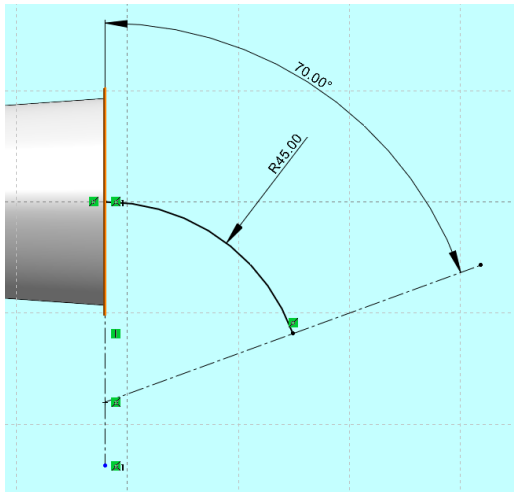
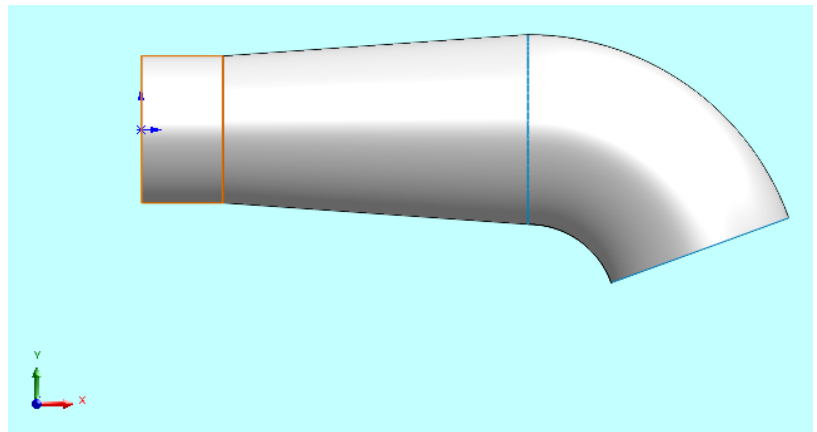Fig 17. Sweep path construction.        Fig 18. Revolved and Swept surfaces.

Create a sketch on the front plane to represent the internal surface of the lower part of the tube, but make it longer than shown in the drawing, say 260mm below the origin (Fig. 19), and generate the surface with **Revolved Surface**.
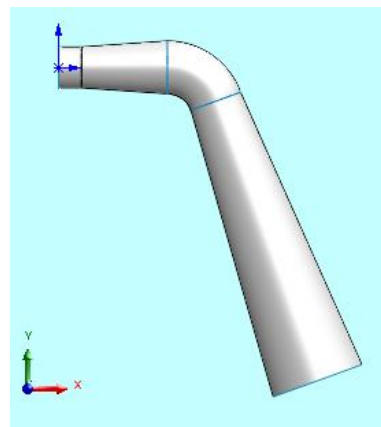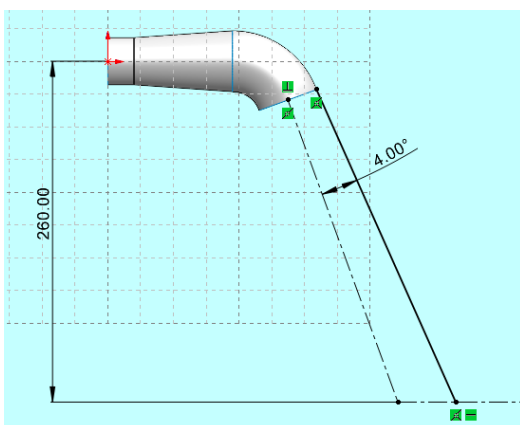


Fig 19 & 20. Line drawing for lower part of tube & Revolved Surface.

Repeat the revolve & sweep processes for the outer surfaces of the tube -
Use four solid lines to create the first revolve section as shown in Fig. 21. The Ø42 surface will be created to join the inner to outer. Note the centreline to be used as the revolve axis.

11
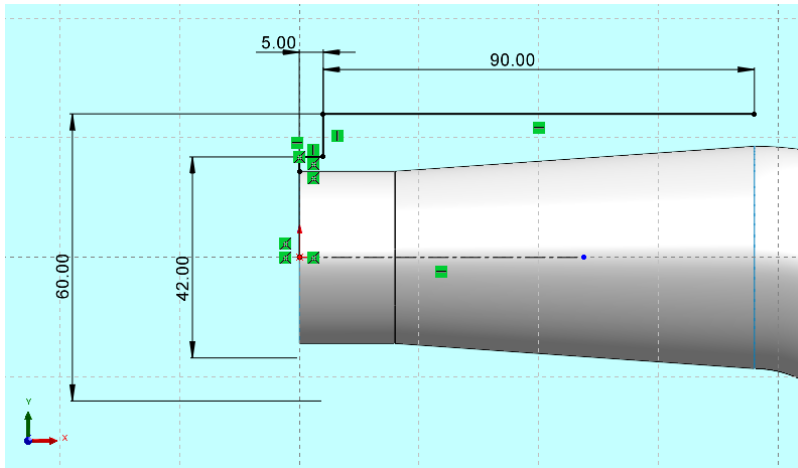
Fig 21

The swept surface can be generated by using the path sketch created for the inner surface, and the profile edge of the newly created outer surface (Fig. 22)
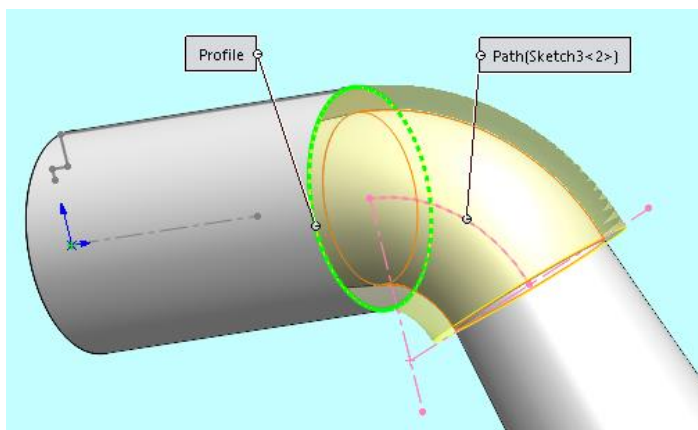


Fig 22

Use a similar sketch to Fig. 19 for the lower outer profile, with one solid line and no attempt to link the inner and outer surfaces at this end (yet).
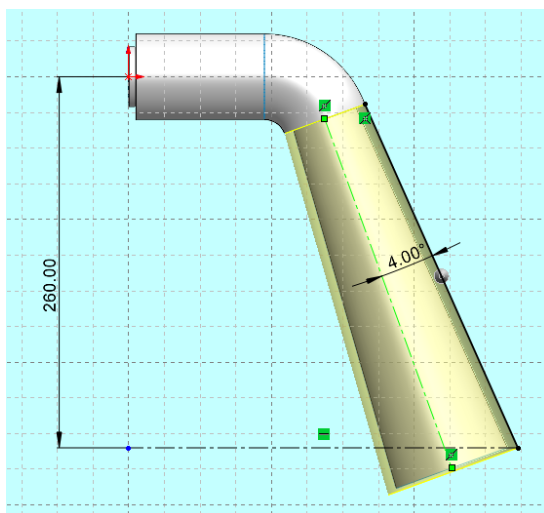


We will need to trim the surfaces and then add a planar surface to close the volume.

Create a plane parallel to the **Top Plane** at Y = -250.

Then select the **Trim Surface** icon and pick the plane as the Trim tool and select the appropriate regions of the model to remove (or keep, but remove is a bit easier), having activated the relevant radio button – i.e. **Remove Selections**.

Fig 23 Lower outer revolve & sketch

12

Create a **Planar Surface**  by selecting the two ellipses to close the volume (Fig. 24).
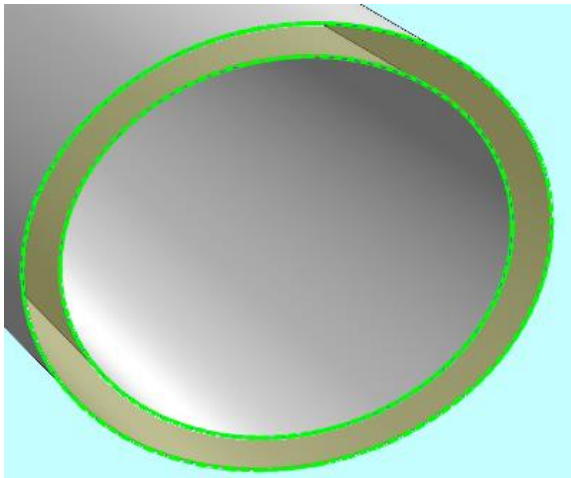


You should now have 7 **Surface Bodies** listed in the construction tree.

To finish the part, select the **Knit Surface** icon  and drag a box around the part to select all the relevant surfaces. Tick the **create solid** box. It shouldn't matter if **Gap Control** is selected, as that is unlikely to be an issue. If the solid is successfully created, you should now see **Solid Bodies** (1) in place of the Surface Bodies listing before.

Fig 24 Closing the volume with a Planar Surface.

Finally select Perspex as the material

## Complete the Turbine Model

Use the drawings to create the remaining parts –

- Mounting Plate *MP001*
- Sealing Plate *SP001*
- Drive Shaft *DS001*

Build an assembly of the whole model, with your redesigned guide vane mechanism.

Insert the volute casings, and draft tube reducer etc. Mate the parts with the correct alignment. Note the location of the driveshaft in drawing TC000 (distance to Mounting Plate).
Insert your modified version of the GV mechanism assembly and set up the mates. Right click on the guide vane mechanism assembly and select the Make

Subassembly Flexible icon  to allow rotation of the parts.

Create a vertical cross section view of the assembly (on the centreline) using the

**Section View** icon.  for your report.
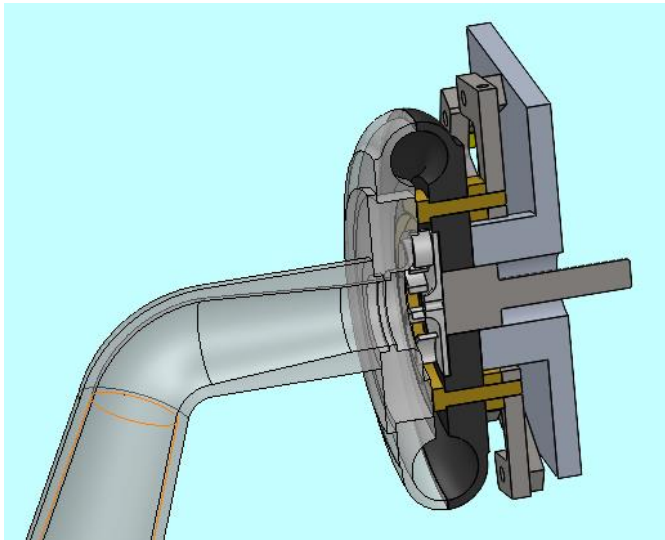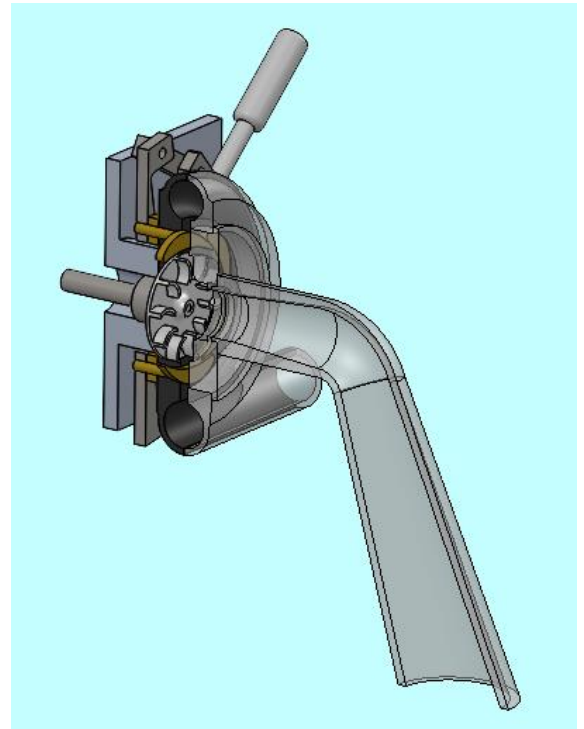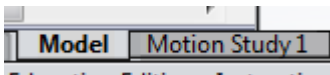
See Fig. 25 & 26 and **Save** the assembly.



Fig 25 & 26 The complete sectioned assembly.
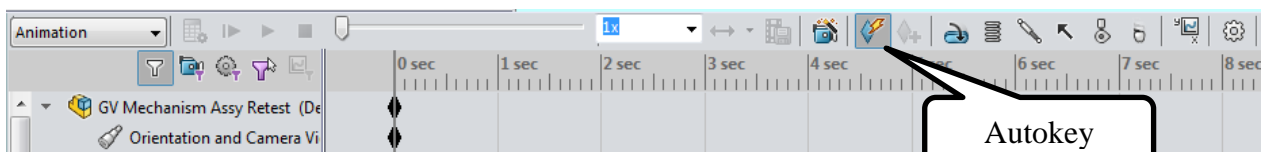(Runner closing plate hidden)

## Animate the model

How to create a 'spin around' or 'fly by' animation…

Click on the motion study tab at the bottom left of the Graphics window (it may be labelled **Animation1**).



The **Type of Study** should be set to **Animation**.



needs to be enabled (with black rather than grey text). If required, right click on **Orientation and Camera views** in the tree, and deselect '**Disable Playback of View Keys**'

Rotate and zoom the part into the position that you want to start the animation (or select a standard view).  Right click on the black diamond in the **Orientation and Camera views** row at the timebar (0 secs). Select the **Replace Key** icon.
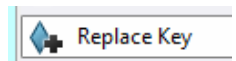
Drag the Orientation and Camera view diamond to the end time point for the animation (say 8 secs).



Rotate and zoom the part into the position that you want to end the animation. Right click on the diamond in the **Orientation and Camera views** row at the timebar (8 secs). Select the **Replace Key** icon.

Press the **Calculate** icon.  The assembly zooms and rotates.

You may have to select the Stop Playback icon (depending on the playback mode).

The Play back mode can be one of three options -



The Normal mode of animation will stop at the end time point.

Drag the timebar to a new end time point for the animation (16 secs).

Right mouse click on the marker key for the **Orientation and Camera views** at 0 secs and select **Copy**. Right mouse click on the timebar at 16 secs and select **Paste**.
Press the **Calculate** icon. The assembly should zoom and rotate from the original start point, to the original end point and then back to the beginning (without a sudden jump).

By manipulating the keypoints for individual parts of the assembly, it is possible to animate their position (but see note below), as well as their colour and display mode. For example – setup a key point at say 5 seconds for the Runner and rotate it before selecting the **Calculate** icon. Also edit the Appearance colour for the runner at a specific time step.
For sub-assemblies and component parts you can select them in the Motion Study tree and select the **Add/Update Key** icon to set a positior appearance.

Experiment with these options.

> **Note** that for some sub-assemblies you may need to right click on the item in
> the tree, and select the **Make Sub-assembly Flexible**  icon to move their
> components. A rigid sub-assemblyll have a different icon -
>
> By default, when you create a subassembly, it is rigid. Within the parent
> assembly, the subassembly acts as a single unit and its components do not
> move relative to each other. However, you can make subassemblies flexible.

15

This allows movement of the individual components of a subassembly within the parent assembly.

Other useful features are –

The **Animation Wizard** icon  - which inserts a view rotation or expand/collapse at the current timebar location.

The **Save Animation** icon  - which saves animation as a 'stand-alone' AVI file, *.mp4 or other type.

Create a short video showing the how the guide vanes or runner moves in a sectioned assembly, or dismantle some parts to better show the internals, and save it as an AVI or *.mp4 file.

You have now completed the Part 4 notes.