Turbomachinery CAD CWM PART 1

RADIAL FLOW WATER TURBINE

Exercise 1 - Design of an improved Guide Vane Linkage Mechanism

Solid Modelling

This part of the exercise involves the solid modelling of the individual components of the guide vane steering mechanism. Later on these will be assembled into an organised hierarchy for kinematic simulation. The simulation will indicate that the mechanism, presently installed on the water turbine hydroelectric plant, introduces a small but significant error in the relative angles of incidence between the guide vanes. The aim is to modify the linkage mechanism in a simple manner in order to provide equal changes in guide vane angle when varying the fluid throughput, and hence the power output, of the turbine.

The *Linkage Mechanism* (Drg.No. GV000) is assembled from the following components,

Guide Vane Crank	(Drg.No. GV001)
Handle	(Drg.No. GV002)
Guide Vane Link	(Drg.No. GV003)
Indicator	(Drg.No. GV004)
Guide Vane	(Drg.No. GV005)
Guide Vane Sealing Ring	(Drg.No. GV006)

All these components are to be modelled as 3D solids, using SolidWorks. For the drawing numbers refer to the drawing pack

SolidWorks information

Last year you should have used SolidWorks software in the P5 Engineering Drawing & Design Communication Lab. Look at the notes in Canvas, (preferably for the current first year cohort, which is relevant for the latest version of the software) and find the "Introduction to SolidWorks" document. This should act as a good reminder of how to use the software, and as a reference guide. Other documents should also prove useful, especially the "Guidelines for Engineering Drawing" document.

Later on, you will model parts without instruction, but full notes are given to aid construction of the first part, and the initial part of the assembly, as follows…
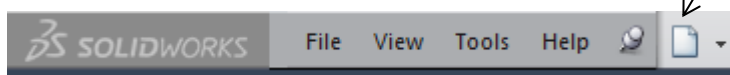
Part generation – Crank GV001

Start by constructing a 3-dimensional model of the *Crank* (Refer to the Drg.No. GV001), working in millimetres, as follows.

Extruding A Simple Shape

Start **Solidworks**.
From the menu select **File** then **New** or select the '**New**' icon in the LH side of the Toolbars.
Choose **Part** and click on **OK**

It is good practice to name and save a file immediately. From the File menu, click on **Save As** and save it as *GV001* in a suitable directory (say TCAD CWM) in your Documents area
(on the local hard drive). It is important to make a backup copy of the folder to your home area (H:) at the end of the day.

On the **Command Manager** make sure that the **Features** tab is selected, then click on the **Extruded Boss/Base** icon to create an extruded feature. A view of three **Sketch Planes** should appear.
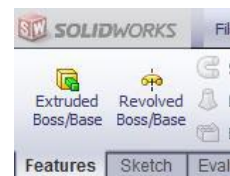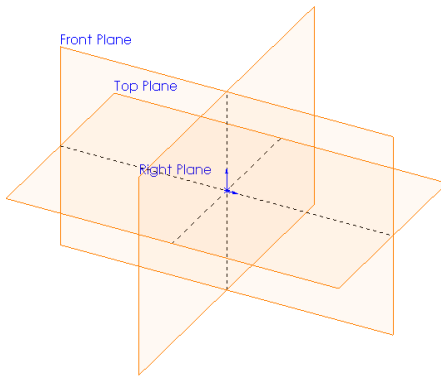
For most objects, it doesn't matter which plane you draw on.
The Front plane is the default view when you create a drawing, but this can be easily changed.

Select the **Front Plane** by clicking on its border and the view rotates and a grid appears, with the origin symbol at the centre. If you cannot see the **origin,** then toggle the **Origins** symbol in the **View** menu.

The **Sketch Menu** should have been automatically selected. If you move the cursor over the icons, a "tool tip" appears to describe the commands.

Click on the **Line** command. Create a rough closed profile shape, similar to that shown below.
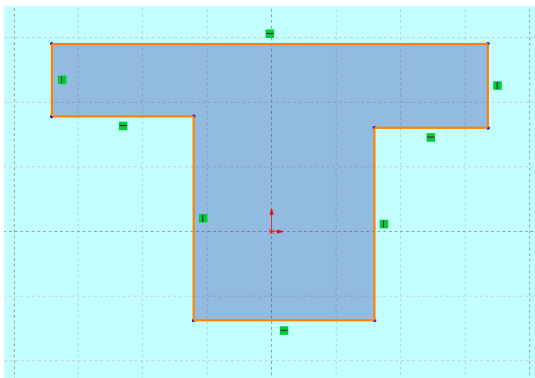
Note the position of the **Origin** – ideally we want to locate it on an appropriate central feature (in this case the centre of the Ø8 mm hole in the narrow section), so that we can make use of the pre-defined **Right Plane**, to mirror features. (This can also be very useful when building assemblies). Note the symbols indicating vertical or horizontal lines. If you create a line that is not aligned in the required way, it is easy to change this. Simply select the line, and select the required relation from the **Line Properties**, **Add Relations** tab.

Use the **Smart Dimension** command to create the dimensions as shown below. For length dimensions of lines (e.g. 20), just pick the line and place the dimension. For dimensions between lines or lines and points (e.g. 10) select the lower horizontal line and then the point, before placing the dimension. Select both horizontal lines for the 70 dimension. It is good practice to lay out the dimensions neatly (this will help when creating the detail drawing).

The lines will change colour from blue to black when their position is fully defined. Note the dimensions used to position the part relative to the origin (10 & 10).

When complete, select the **Exit Sketch** command (or the icon in the upper right hand corner of the sketch window – not the cross).
The view will rotate round and a **Boss-Extrude** dialogue box will open up.

In the **Direction 1** part of the dialogue box, change the **End Condition** from **Blind** to **Mid-Plane.** The **Depth D1** should already be the required 10 mm (if changing you don't need to type the "mm"). Click on the green tick to finish the command.
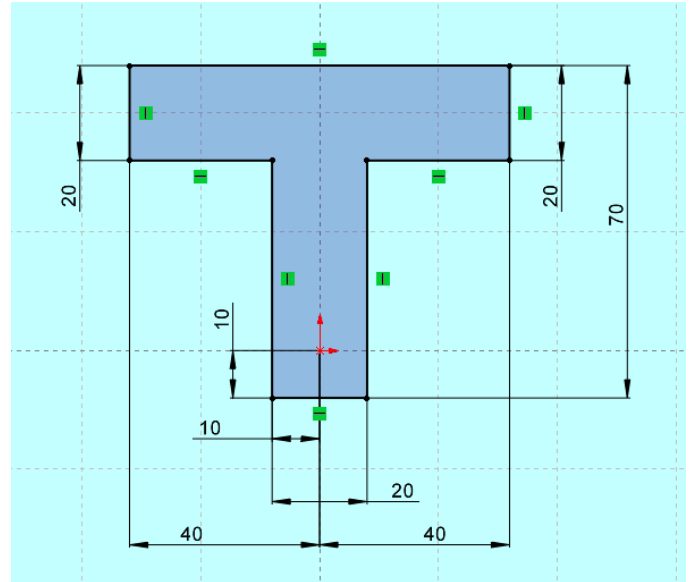
Press the **ESC** key to remove the green image of the sketch. Rename **Boss-Extrude1** to **Profile Shape** (select it in the tree and press F2). Naming a feature like this is helpful for later editing.

View the part by **rotating** it: press the middle mouse button and move the mouse! This is also a good time to practice **zooming** in and out using the mouse wheel. (If it goes off the screen you can use the **Zoom to Fit** icon to get it back or press the **f** key).

Select the **Extruded Cut** icon. Select the front face of the part (parallel to the Front plane) to define the sketch plane (this should be the face normal to the Z axis). If the view gets rotated off the plane you can select the **Normal To** button from the **View Orientation** command drop down.

Sketch a circle at the origin using the **Circle** command, and dimension it to Ø8mm with the **Smart Dimension** command. Exit the sketch and select **Through All** from the **Direction 1** tab in the **Cut-Extrude** form. Rotate the view to check that the cut is going in the right direction. If not select the **Reverse Direction** icon (arrows next to **Through all**). Click on the **Green Tick** to finish the command. Rename **Cut-Extrude1** to **Lower Hole** (select it in the tree and press **F2**).

Select the **Extruded Cut** icon, and then the front face of the part to define the sketch plane. Sketch a circle at the top right corner of the part using the **Circle** command, and dimension it to Ø8mm with the **Smart Dimension** command. Locate its position using the **Smart Dimension** command, as shown below.

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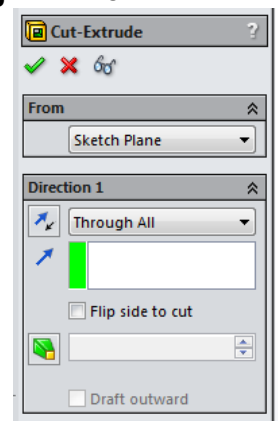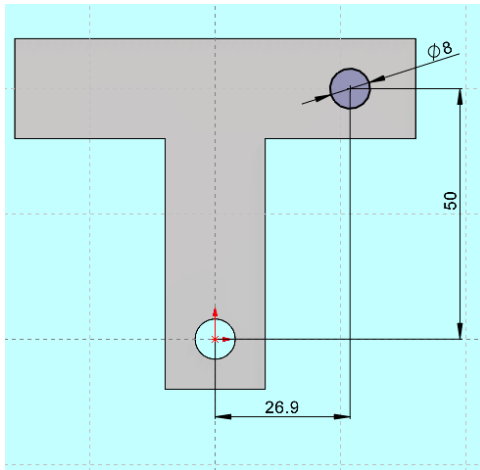
Once again, exit the sketch and select **Through All** from the **Direction 1** tab in the **Cut-Extrude** form. Rotate the view to check that the cut is going in the right direction. Click on the **Green Tick** to finish the command. Rename **Cut-Extrude2** to **Upper Hole Right**.

Make sure that you keep saving the file!

Select the **Mirror** command (you may need to click on the arrow under the **Linear Pattern** command).

Select the predefined **Right Plane** as the mirror plane, from the tree in the graphics window (click on the arrow ► to expand).
Select **Upper Hole Right** as the **feature to mirror** and click on the **Green Tick** to finish.

I recommend that you deselect the **RealView Graphics**, and **Shadows in shaded mode** for clarity.
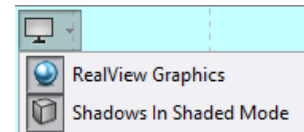
We will use the **Hole Wizard** to create the threaded hole for the handle and indicator screw. Select the **Hole Wizard** icon. The hole type is selected with the icon **Straight Tap.**
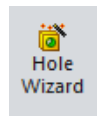
The **Hole Standard** is **ISO**, and the **Hole Type** is **Tapped Hole**. **Size** is **M6**, and **End Condition** - **Blind**. The blind hole depth is set to 15mm, and the thread 12mm.
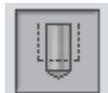
Select the **Cosmetic Thread** icon in the **Options** form. **Nearside countersink** to be Ø6.05 x 90°. Select the **Positions** tab and select the top face of the part. Click on the face origin to locate the hole. Note that you can click anywhere on the face and use **Smart Dimension** to position a hole feature. Click on the **Green Tick** to finish.
The cosmetic thread may not be visible, in which case select the **Options** icon, **Document properties**, **Detailing** and in the **Display Filter** box tick **Cosmetic threads**, and **Shaded Cosmetic Threads**.

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⚙️ ▾ The **Options** icon.

To see the tapped hole feature, we will temporarily section the part. Select the **Section View** icon, 📖 and the **Right Plane**. Select the **Section View** icon again, to see the whole part.

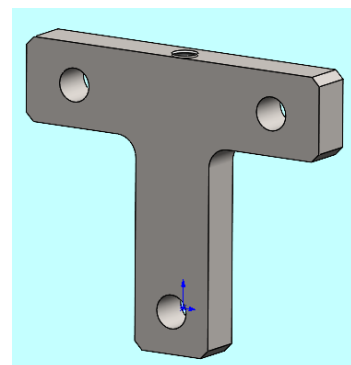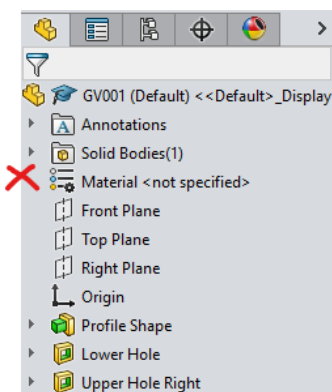The next stage is to add the chamfers in the corners of the part.

Select the **Chamfer** icon (click on the black 'pull-down' arrow ▼ below the **Fillet** icon. Pick all the external corner edges (you may have to rotate the part). Select **Angle Distance**, and set the angle to 45°, and the distance to 2mm (see below – Full Preview selected). **Green Tick** to finish.

Distance: 2mm
Angle: 45deg
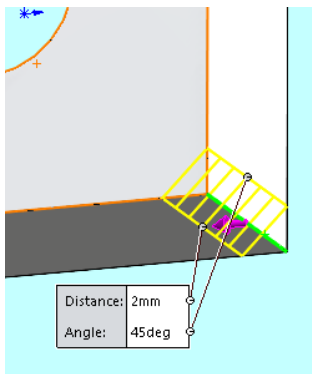
Select the **Fillet** icon. Pick the two internal corner edges (you may have to rotate the part). Select the **radius** to 5mm (see below – Full Preview selected). **Green Tick** to finish.

Radius: 5mm
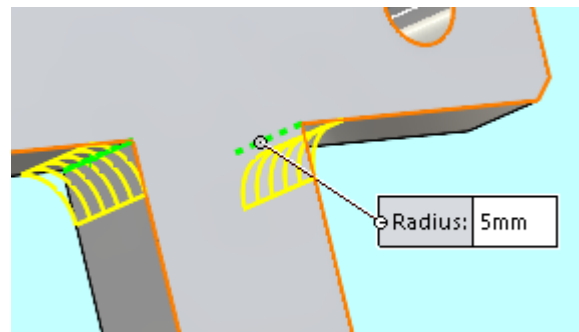
Right click on **Material** in the tree, and select **Edit Material**, then pick **AISI 304** (Stainless Steel) from the list, and **Apply**, then **Close**, and Save the part.

- GV001 (Default) <<Default>_Display
  - ▸ 🅰 Annotations
  - ▸ 🔘 Solid Bodies(1)
  - ❌ Material <not specified>
  - 📄 Front Plane
  - 📄 Top Plane
  - 📄 Right Plane
  - 📐 Origin
  - ▸ 🟢 Profile Shape
  - ▸ 🔲 Lower Hole
  - ▸ 🔲 Upper Hole Right

Finished crank

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You can easily edit sketches and features. Right mouse button click on the feature (in the tree list) and select the Edit Feature or Edit Sketch icon.

Part generation – Handle GV002

To construct the *Handle* (Drg. No. GV002), use the following different techniques to show what is possible - boss extrude and revolve boss.

Create a new part using the menu **File, New...** Save the part as GV002.
Select the **Revolved Boss/Base** icon and the **Front Plane**.

Draw a horizontal centreline through the origin. Draw a 80 x 12mm rectangle (X by Y) on the centreline, with one corner at the origin to represent the largest diameter of the handle. Exit the sketch and revolve the part by 360°.
Create an extrude boss/base feature on the face at the origin and sketch a Ø14 circle centred at Z0Y0. Extrude to 50mm length. Extrude a Ø6 spigot by 12mm long to represent the thread. Add the two external fillets, one internal fillet and the two chamfers.

Type **Cosmetic** in the **Search Commands** box and select **Cosmetic Thread.**
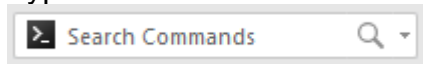
Select the circular edge of the threaded road (outer diameter – not the chamfer) to define the cylinder. Select **Standard - ISO, Type - Machine Threads, size – M6** and **Up to Next (End Condition)**.

The cosmetic thread may not be visible, in which case select the **Options** icon, **Document properties**, **Detailing** and in the **Display Filter** box tick **Cosmetic threads**, and **Shaded Cosmetic Threads**.

It is not really necessary to use cosmetic threads in the model, especially if you produce a fully detailed drawing, however it may be helpful when showing images or animations of parts and assemblies, to aid understanding.

Right click on **Material** in the tree, and select **Edit Material**, then pick **PVC Rigid** from the plastic list, and **Apply**, then **Close**.

Save the part.

Part generation – Sealing Ring GV006

To construct the *sealing ring* (Drg. No. GV006), create a new part, and name it GV006. Sketch two circles on the front plane, centred at the origin, to represent the ring. Use Smart dimension to size them correctly and create a mid-plane extrusion of the appropriate thickness.

Save the part and assign **Brass** as the material.

Make the holes. Create a new sketch on the front surface of the disc (i.e. the flat surface that you see when viewing the part on the front plane – this is not the same as the Front plane) and draw a vertical centreline through the ring. Draw a circle on that centreline, and smart dimension it to 8mm diameter. Also draw a circle of diameter 105mm at the origin to represent the PCD (Pitch Circle Diameter). Tick the '**For Construction**' box to make the circle a centreline. Select the centre of the Ø8 circle, press ctrl key, and select the Ø105 circle. Select **Coincident** in the **Add relations** box. Exit the sketch and create an extrude cut (through all).

To copy the holes around the ring use **Circular Pattern** (the icon is found by clicking the 'pull-down' arrow ▼ below the **Linear Pattern** icon). Go to the **View** menu, **Hide/Show** and select **Temporary Axes** – this will make it easier to select the axis for the pattern. With the **Circular Pattern** form open, select the blue centreline at the origin of the part (this is the primary axis of the ring – you will have to zoom in and rotate the part to see it clearly) as the **Pattern Axis** (the upper box in the **Direction 1** form).
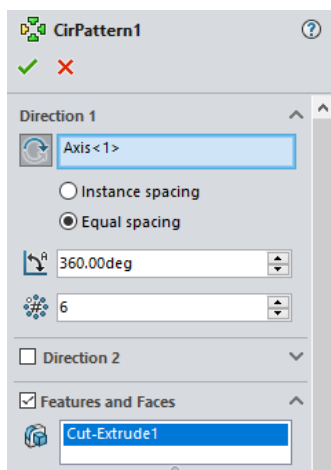
Tick the **Equal Spacing** radio button. The **Angle** should be 360°, and **Number of Instances** is 6. Select the previously created hole from the tree list in the **Features to Pattern** box (e.g. **Cut-Extrude1** - see form). Select the **Full** or **Partial Preview** button and check that it looks OK. Accept with the **Green Tick**.
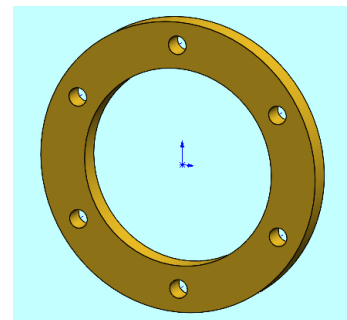
Part generation – Guide Vane GV005

The *Guide Vane* (Drg.No. GV005) is constructed by adding fillet radii to blend to the curves which have known centres. Create a sketch on the front plane. Draw three centre point arcs for the curves of radius 77, 71 and 10mm located on the centres as shown in the drawing. You will have to estimate the start and end points of the curves.

Draw a vertical centreline 29mm from the origin. Drag the endpoint of the R77 curve to coincide with it (or trim it back to the centreline*). Draw a line from that point at 31.5° to the vertical. Drag the R71 curve endpoint to coincide with that (or trim it back). Trim the line back to where it meets the curve.
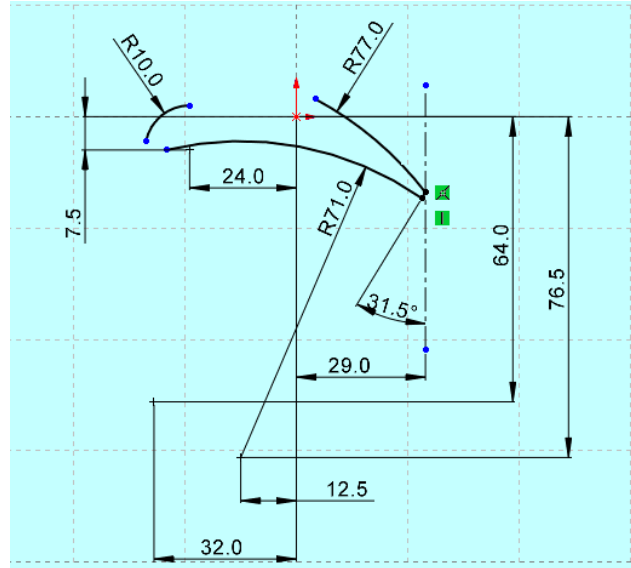
*Use the **Trim Entities** icon and the **Power Trim** option. To trim the line (or arc) –
- position the cursor to one side of the line
- press the left mouse button and drag the cursor through the line
- then release the mouse button.
- Make sure all unwanted parts of the line are removed.

You can also use the **Corner** trim option – select the parts of the lines (or arcs) that you want to keep.

In theory you can draw a fillet radius of specified size to blend the curves. In practice you will have to draw the curves approximately in place and create tangency conditions at each end.
Draw a 3-point arc near the left hand side of the vane, and dimension it to R3.3
Select the R3.3 arc, press the control key and select the R10 arc. Select the Tangent relation (note the Tangency relation markers in the central image below). Repeat the process for the blend to the R71 arc.
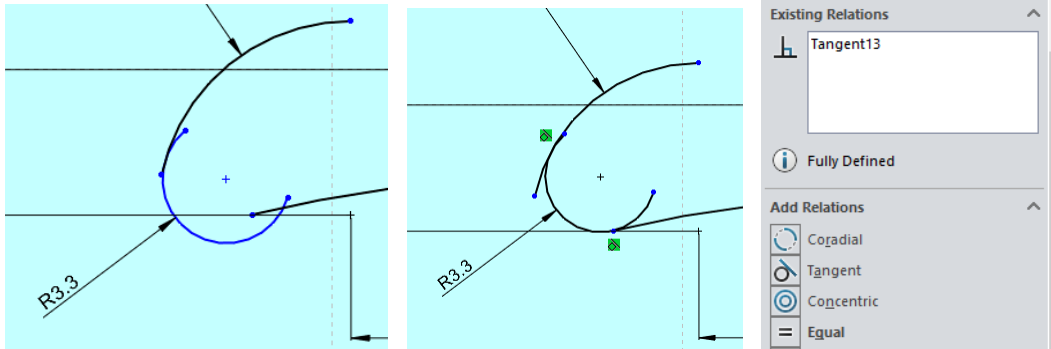
Use **Trim Entities, Power Trim** to trim the arcs back to the tangency point, making sure that any unwanted parts of the arcs are removed.
Repeat the process to blend the R77, & R10 radii with an R31 fillet.

The blade profile should be closed, and the internal area should be a shaded colour (if the Shaded Sketch Contours Icon is active – see image below). If this doesn't work, check that the curves do meet and do not overlap, even by a tiny amount. You will have to zoom in very closely to the join to check, and trim if necessary.
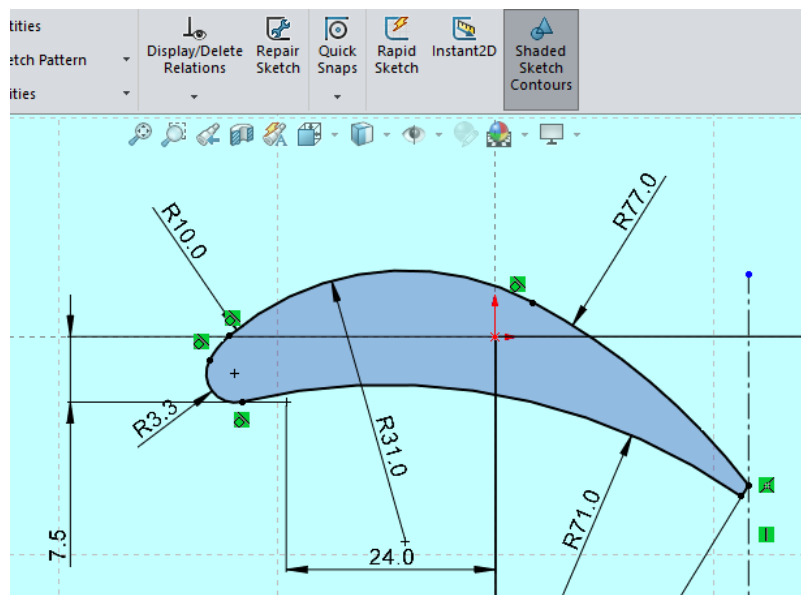
Extrude the sketch to 6.5mm thickness (use the 'blind' option).

Create a new sketch on the 'rear' face so that you can draw and dimension the Ø8 circle and extrude it to 40.8mm length (see drawing GV005). Create the 1 x 45° chamfer, set the material to Brass and save the part.

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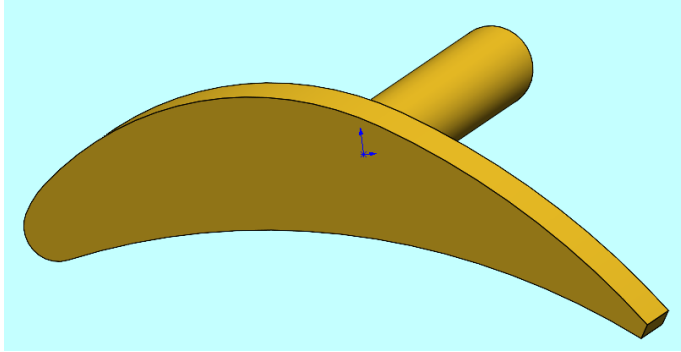
Finished Guide Vane

Have a go at constructing the remaining components, namely:

Guide Vane Link (Drg.No. GV003)
Indicator (Drg. No. GV004)

Edit the Indicator Material to be ABS, as stated in the drawing.
Then set the colour by selecting the **Appearance** tab -

Right click on the label "White PW-MT11150"
Select **Add Appearance**
Highlight the part in the **Selected Geometry** box
Pick the yellow colour from the standard option. Green Tick.

This completes the solid modelling of the guide vane linkage mechanism.