Turbomachinery CAD CWM PART 2

RADIAL FLOW WATER TURBINE

Exercise 1 - Design of an Improved Guide Vane Linkage Mechanism

Component Assembly and Kinematic Simulation

This part of the exercise involves the assembly of instances of the components previously modelled, into a hierarchal structure that preserves the precise geometric relationships between the elements of the guide vane steering mechanism, so that kinematic simulation may be carried out with the appropriate choice of inter-element joint properties.
See *Guide Vane Mechanism Assembly* Drg. No. GV000

Create a new assembly with **File, New, Assembly**.

The **Begin Assembly** form opens.
Browse to the *Crank* part (GV001), select open and click on the **Green Tick** to load the part and place it at the origin. This is what we want - alternatively you can drag and drop it into the assembly – in which case it is located where it is 'dropped'.
This part is fixed in place - see its name in the tree with an adjacent (f).

Save the file as *GV Sub Assembly*.
Use **Insert Components** (from the **Assembly** tab) to load the *Guide Vane (GV005)*.
Place it in the graphics window at any convenient position.
If you use the **Green Tick** and the parts are fixed, you will have to right click on the part in the tree and select **Float** to make them float and remove the (f) next to the part name.

Assemble these parts to make a rigid sub assembly -
Use the drawing - *Guide Vane Mechanism Assembly* Drg. No. GV000 to help you understand how the parts go together. Look at the first-year notes in Canvas especially the "Guidelines for Engineering Drawing" document to remind yourself about 3rd angle projection etc if necessary.
Use the **Mate** command to make the *Guide Vane* shaft concentric to the lower hole in the *Crank*.

Make the end of the *Guide Vane* shaft coincident with the face of the *Crank*.

It sometimes helps to rotate parts to a position close to their final position when creating mates. Rotate the *Guide Vane* by clicking and dragging it to approximately the correct angular location. Check the *Guide Vane* drawing (GV005) to see the orientation of the vane relative to the top plane.

Set up an **Angle Mate** between the **Top Plane** of the *Assembly*, and the **Top Plane** of the *Guide Vane*. Select the planes from the feature tree in the graphics window. **Save** the sub assembly. Double check that it is correct, to save further work later.

The sub assembly is now rigid. The crank is fixed, and the guide vane position is fully defined.

> *Note - If you build up the mates, and then find you have to 'flip' one, it may affect the subsequent mates in the list, causing errors. To avoid too many problems, it may help to suppress these mates before 'flipping' the problem mate. Then go through the subsequent mates in turn un-supressing and modifying as required.*

Now create the *GV Mechanism Assembly* (GV000) –
Create a new assembly with **File, New, Assembly**.
The **Begin Assembly** form opens.
Browse to the *Sealing Ring* (GV006) part (make sure that the filter is set to **All Files, Part Files** or **SolidWorks files,** not **Assembly Files**). Click on the **Green Tick** to load the part – it will be fixed at the origin, which is what we want.
Save the assembly as *GV000*.

Insert two of the *GV Sub Assembly* at any position in the graphics window by dragging and dropping (so they are not fixed at the origin). The icon displayed next to each of these parts in the tree should indicate that they are rigid. .
If the alternative 'flexible' icon is shown , then right click on the sub assembly in the tree and select the **Make Subassembly Rigid** icon (This icon is the 'flexible' icon shown above with a dark grey background).

Look at the *Guide Vane Mechanism Assembly* Drg. No. GV000 -We will start by locating the GV Sub Assembly which will have the handle fixed to it. Create a hinge mate between one of the *GV Sub Assemblies* and the correct hole in the *Sealing Ring -*

To do this, firstly right click on the *Guide Vane* and select the 'crossed through eye' icon to **Hide** this part. 
Create **a Hinge Mate** (Start the **Mate** command and look in the **Mechanical Mates** form) between the *Crank* hole and *Sealing Ring* hole. Use the cylindrical surfaces for the concentric selection, and the flat faces for the coincident selection.

Right click on the *Guide Vane* in the tree (for the correct sub assembly) and select the 'eye' icon to **Show** this part again. Note that icons in the tree for hidden parts are displayed in light grey as opposed to yellow for visible parts (making them easier to find).

Repeat the hinge mate creation for the next *GV Sub Assembly* (align to the hole adjacent to the *GV Master Sub Assembly* – anticlockwise looking at the vane side of the assembly).

Insert a *Guide Vane Link* (GV003).

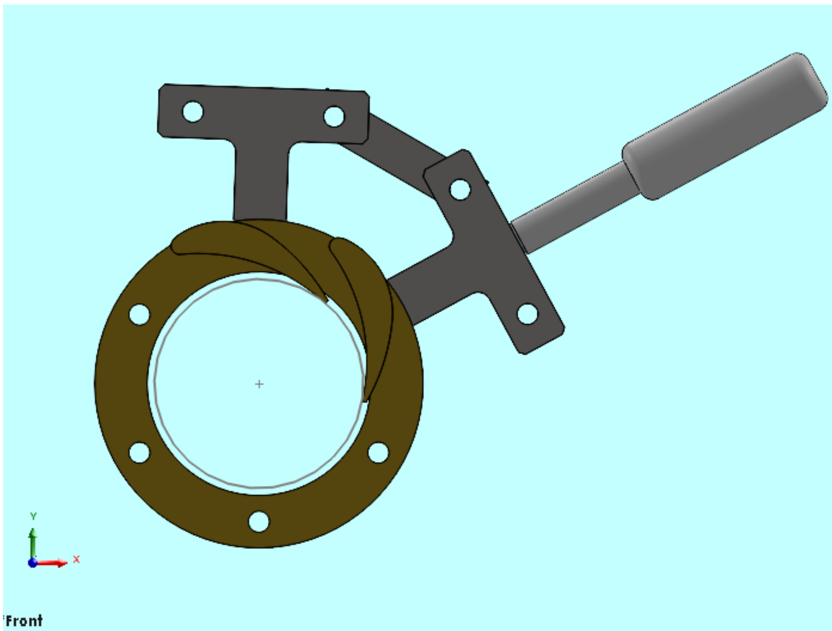
Create **Hinge Mates** between the *cranks* and *Link*. Check the drawing to see which side of the C*rank* the *Link* is located. You will need to reselect the **Hinge Mate** type during this process.
Save the assembly.

Insert the *Handle* (GV002) into the assembly, and mate it to the threaded hole of the correct crank. Use a concentric mate for the cylindrical surfaces of the threads and tick the **lock rotation** box. Use a coincident mate for the abutting flat surfaces. You should be able to mouse click on the handle and drag to move the guide vanes. If the handle is free to rotate it will just spin without adjusting the guide vane angle.

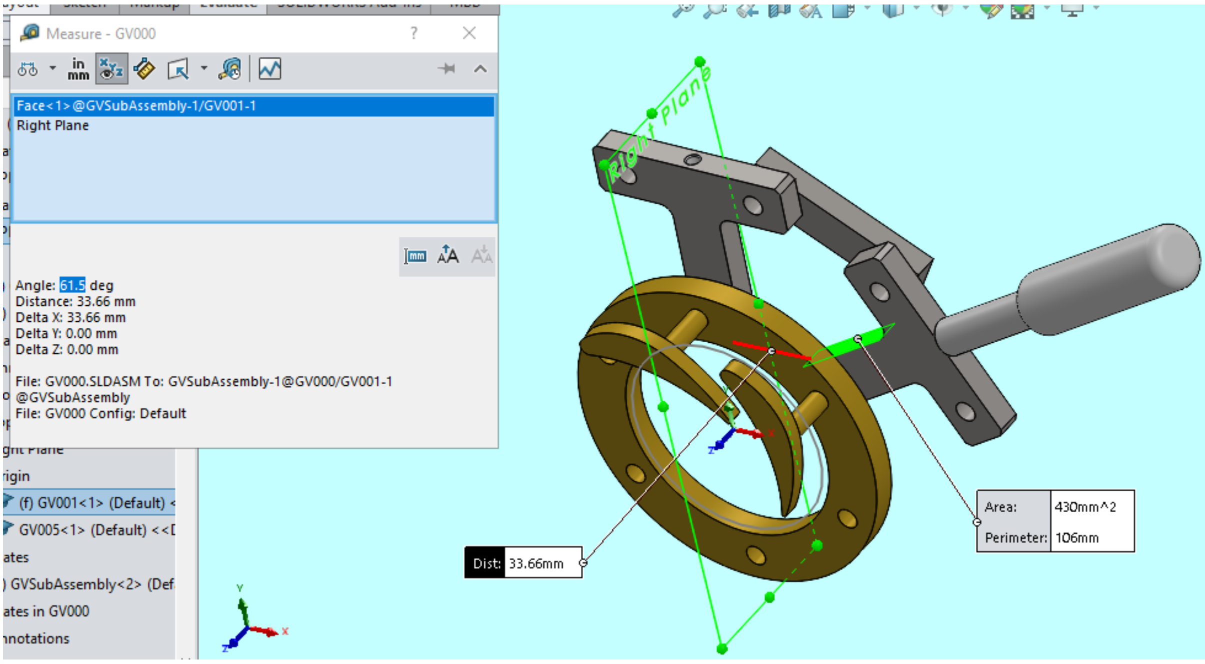
Create a **Sketch** in the assembly on one of the flat faces of the *Sealing Ring*, and draw a circle of Ø79.6mm to represent the outside of the *Runner*.

Close the sketch and deselect everything.

Look at the front view of the assembly and rotate the handle such that the trailing tip of the guide vane is just touching the circle representing the runner.



Carefully rotate the view without adjusting the vane angle and measure the angle between the assembly right plane and the crank (with handle). Use the **Evaluate** tab, and **Measure**. Select the right plane from the assembly and a crank surface that is parallel to the main axis of the handle. Select the down arrow on the pop-up window to see the angle measurement.



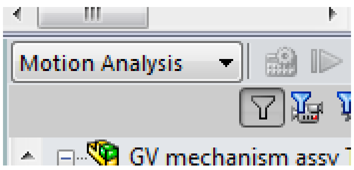
You should get an angle of approximately 61.5° (but make a note of your value).

Re-orient the view to look on the front plane and rotate the handle such that the guide vanes are just touching in the closed position. Carefully rotate the view and measure the angle again (it should be about 49° - but make a note of your value.)

The handle needs to rotate the crank by about 12.5° to move the vanes from fully open to fully closed. You will use this information later.

Load the **SolidWorks Motion** add-in (from the **SolidWorks Add-Ins** tab, or **Tools, Add-Ins…**).

Click on **Motion Study 1** tab – bottom left corner of screen (it may be labelled **Animation1**).
Select **Motion Analysis** from the drop down tab (probably showing **Animation**).

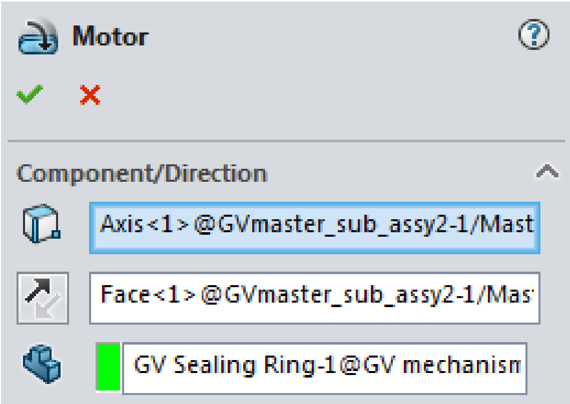


Add a motor to the *Crank* arm as follows...
Right click on the *Guide Vane* in the *GV Sub Assembly* (with the handle) and select the ‘crossed through eye’ icon to **Hide** this part. 

View via **Hide/Show** the **Temporary Axes** (if not already on).

Select the **Motor** icon (in **Motion Analysis**), and choose type **Rotary Motor** 

In the **Component/Direction** tab select the Axis of the clamp hole in the *Crank* for the motor location, the cylindrical surface of the *Crank* hole for the motor direction, and the *Sealing Ring* as the component to move relative to. The rotation direction should tend to close the guide vanes (i.e. anticlockwise). Click on the arrow if you need to change the direction.



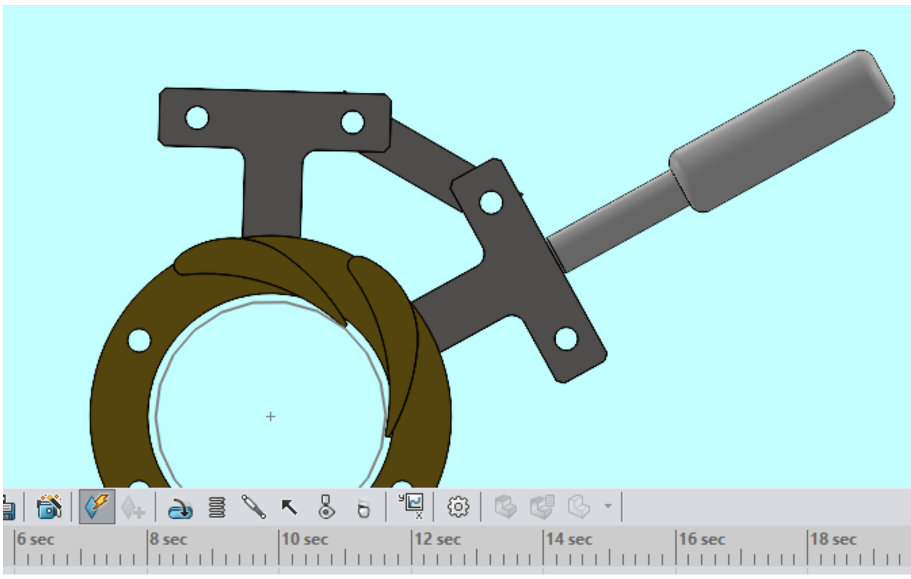
In the **Motion** tab set the function as **Distance** and the displacement as x° - the calculated angle of rotation, from 0 to 5 s.

Right click on the *Guide Vane* in the tree (for the correct sub assembly) and select the 'eye' icon to **Show** this part again.

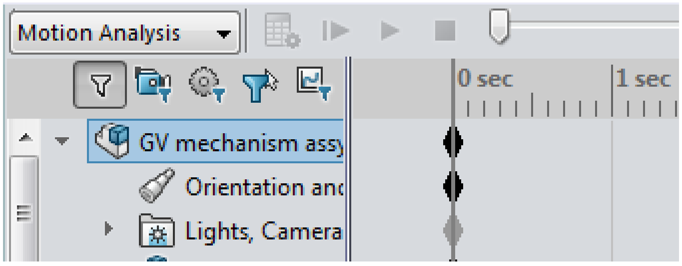
View the assembly on the Front Plane (so that the *Guide Vanes* are not obscured by the *Sealing Ring).*

Make sure that the *Guide Vanes* are fully open (i.e. almost touching the *Runner* circle sketch).

Zoom in to the assembly, so that half of the *Sealing Ring* is visible, and the *Handle* is just in the window, as follows.



In the **Motion Analysis** tab, make sure that the time bar is at the **Zero** seconds position, and if not drag it back to zero (by clicking on it in the grey time axis).



Right mouse button select the **Orientation and Camera Views** icon and select **Disable Playback of view Keys.**

You should have a Time marker (black diamond) at the 5 seconds point for the top-level assembly.

You can zoom into the timescale with the icons in the bottom right corner of the window.

Select the first of these (**Zoom to fit**).

Click on the **Calculate** icon. 

The handle and assembly should move until the *Guide Vanes* are in the fully closed position.

The Play back mode can be set to one of three options -

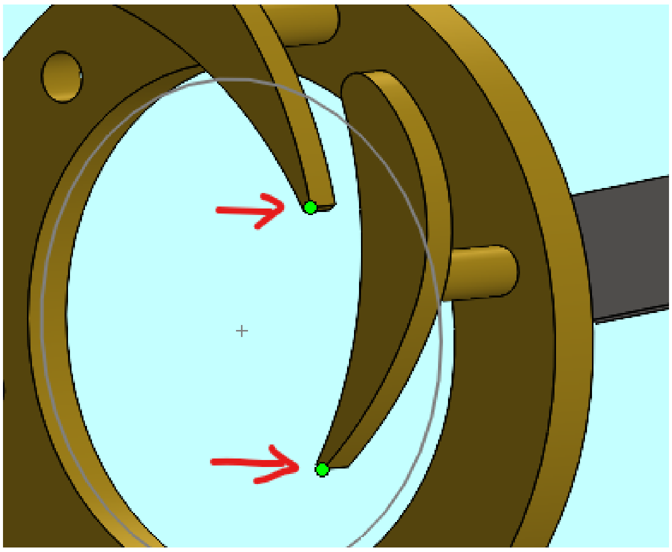
Set it to the **Reciprocate** option.

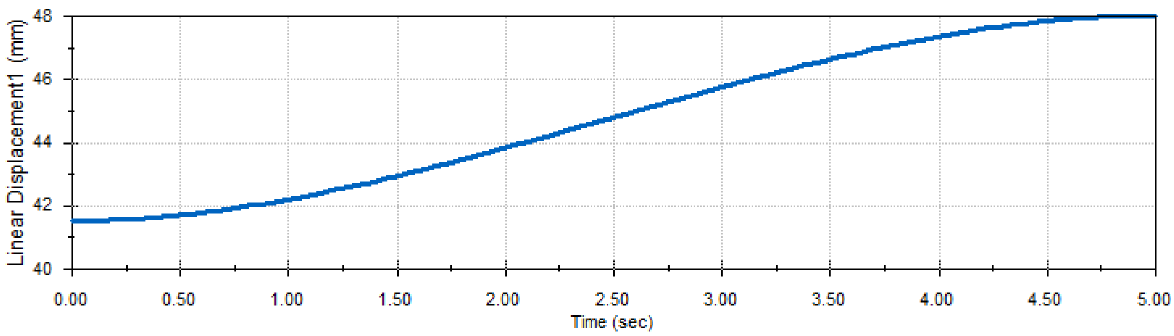
Select the **Play from Start** icon 

Watch the animation for a few cycles. Then select the Stop Playback icon to finish.

In the **Motion Analysis** tab, drag the time bar to the **Zero** seconds position.

Use the Results and Plots icon  to create a results plot. Select **Displacement/Velocity /Acceleration** and **Linear Displacement, and Magnitude.** Pick the end points of the 1st and 2nd *Guide Vane* (see following image). Rename the plot *Vane End Points* (Select the item in the tree under **Results** and press the **F2** key).





Note that you can **Right mouse button click** on the plot in the tree and **Export to Spreadsheet**, so that you can see the displacement values against time.

Alternatively you can **Right mouse button click** on the graph to **Export CSV** (save to a comma separated variable file).

If the results plot is ‘greyed out’ in the tree, you can right mouse button click on it and select ’Show Plot’.

Save the assembly (and I recommend that you save a backup copy).

Go to the **Model** tab (i.e. go out of the **Motion Analysis** study).

Add in the next *GV sub assy*. The easiest way to do this is to click on the *Guide Vane sub assembly* in the tree to highlight it. Then press and hold the **Ctrl** key, and click and drag the item from the tree into the graphics window. Let go of the mouse button to locate the item in the window.

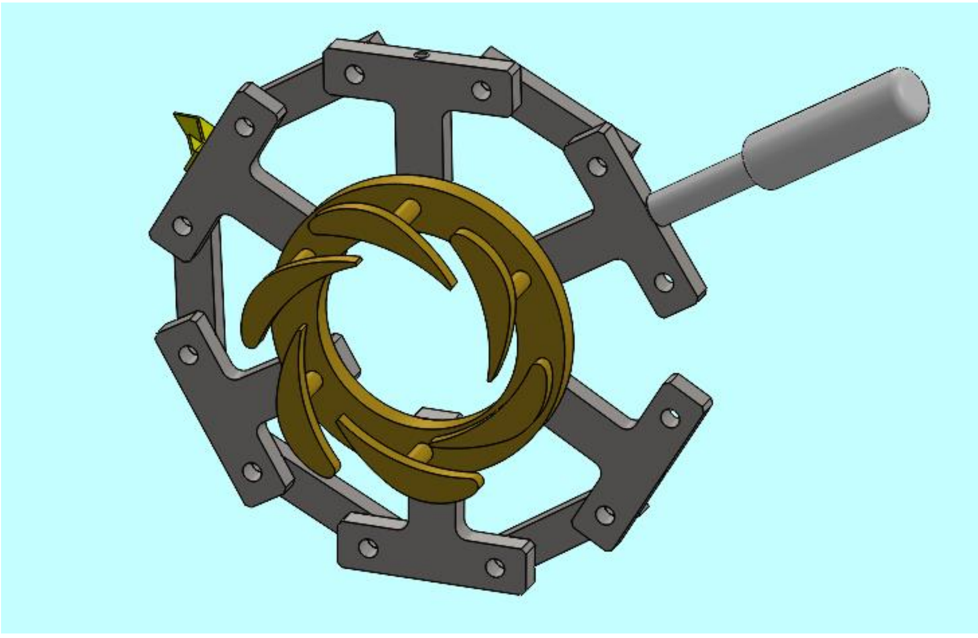
As before, create a hinge mate between the *GV Sub Assembly* and the correct hole in the sealing ring (see assembly drawing). Hide the *Guide Vane* first. Then create a hinge mate between the *Crank* hole and the *Sealing Ring* hole. Use the cylindrical surfaces for the concentric selection, and the flat faces for the coincident selection.

Insert a second *Guide Vane Link*. Click on the item in the graphics window, then press and hold the **Ctrl** key, and click and drag the highlighted item in the graphics window to create a copy. Let go of the mouse button to locate the item in the window.

Create **Hinge Mates** between the appropriate *crank* holes and *Link*. Save the assembly.

Add in all of the GV sub assemblies and links, and setup the appropriate mates. **Show** all the hidden parts. Add the *Indicator* (GV004) as shown in the assembly drawing.

<u>Regularly **Save** the assembly.</u>



Go to the **Motion Study** tab.

You may get a warning message to update parts – say No.

Be aware! Motion Analysis calculations can take a long time due to the number of parts and joints. Though in this case, it shouldn't take too long.

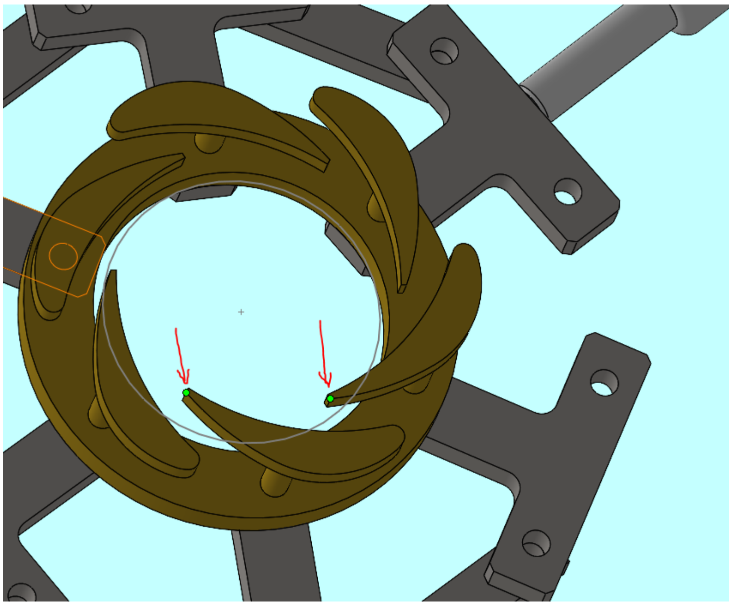
Drag the time bar to zero seconds.

Select the **Calculate** icon.

Select **Play from the start** in the **Reciprocate** mode to check that all is in order.

Save the assembly.

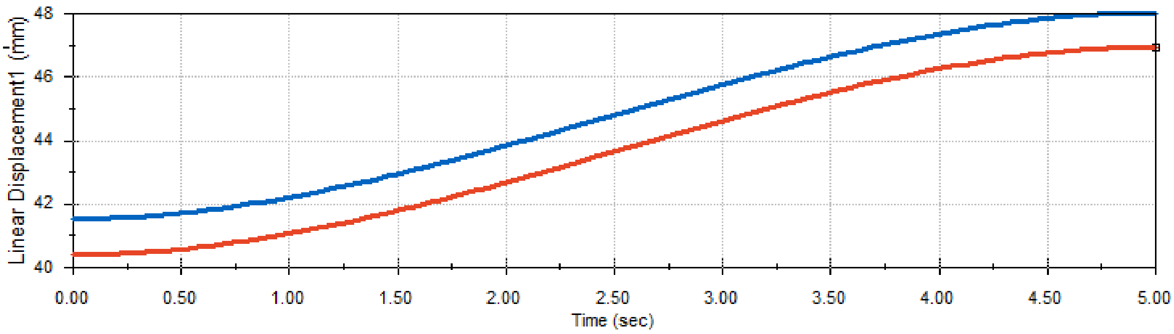
Create a **Results Plot** for the **Linear Displacement (Magnitude)** between the end points of the penultimate and final *Guide Vanes* (see next image). Select **add to existing plot** and pick the plot *Vane End Points*.



When solved, right mouse click on *Vane End Points* in the Motion Analysis tree and select **Show** to display the graph (if not already displayed). Right mouse click on *Vane End Points* in the tree and select **Export to Spreadsheet.**

With the SolidWorks plot, be aware that there may be a different scale for each plot (possibly a different unit too? – if so **Hide** and **Show** again, then **right mouse click** on the plot and select **Chart Properties…** and **Single Y axis**). If necessary, select **Export to Spreadsheet** again.

The distance between successive *GV* tips and their movement is not uniform; it changes incrementally around the sealing ring by about 0.3mm. Some *Guide Vanes* may clash with the runner.



Go to the Model tab. View on the Front Plane. With the GV's set at either fully open or closed. Zoom in and take a close look at the *Guide Vane* closure, or *Guide Vane* tip distance to the runner circle. You will notice an increasing difference in position appearing as the linkage progressively rotates each *Guide Vane* unit. This is because the linkage is rotating about different centres from that of the axis of symmetry of the turbine.

You can also use the **Evaluate** command manager with the **Measure** icon to compare *Guide Vane* distances.

Your next task is to modify the Guide Vane Mechanism so that uniform *Guide Vane* rotations are produced. You are free to explore your own design, however, the recommended approach is to replace the five *links* with something.

Produce clear hand sketches of possible solutions to the problem. Ideally these sketches will be self-explanatory but use annotation if necessary. Set up a **Motion Study** to prove that your design works. Each student should independently create a model of the modified design.

IMPORTANT!
Create new and/or modified existing parts for your solution. If you modify any parts <u>you must make a copy first – do not overwrite your original part/assembly files – we may wish to see them later. Instructions for easy copying follow</u>.

A simple way to copy the full assembly structure is to use 'Pack and Go'. Select **File, Pack and Go…**
All assemblies and parts in the list must be selected. Tick the 'add a prefix' box and type in Modified_ (or Backup_). Tick to flatten to a single folder and select **Save**.

You are required to create formal drawings to enable complete manufacture of all the parts that you have designed or modified to overcome the mechanism deficiencies. Modified parts can refer to the original drawing as a starting point for the modification work (If you just need to add an extra hole, then the drawing can be the original un-dimensioned part with a fully dimensioned new hole, and the material is specified as the original drawing number).

Create a formal assembly drawing of the mechanism redesign, including a Bill of Materials table and balloons. Drawings should be on A3 format and saved as a **pdf** files to submit to the Canvas assignment, with your report.

How to create a drawing

Think about which views best allow you to properly dimension all features and their positioning on the final drawing.

Remember to allow sufficient space on your drawing for dimensioning all the features properly. Lay out your drawing neatly and evenly.

The key to good dimensioning is clarity. Place dimensions in the view that best shows the feature being dimensioned and keep dimensions of the same feature together in the same view.

A feature should only be dimensioned ONCE and should not be dimensioned in each view.

Spread dimensions out so that they are not cluttered.

Dimensions written at an angle are to read from the bottom or right-hand side of the page.

It helps to give the overall dimensions of an object.

Any circular item, or hole drawn in section through its centre, should have its centreline included.

<u>For practical SolidWorks advice see the presentation –</u>

Producing an Engineering Drawing - Tips & Techniques

W:\TCAD\Drawing Tips.pptx (or pdf version)

This includes information on Views, sheets, scaling, sections, cross hatching, details, assemblies, centrelines, dimensioning, tangent edges, and moving drawing views.

Also see GENERATING A CAD DRAWING on page 28 onwards of the **Introduction to SolidWorks** handout (**see Canvas, P5 Engineering Coursework, Drawing & Design**) and see the **Oil Pump Exercise** page 38 onwards - especially page 50 Adding Balloons and the Bill of Materials (BOM).