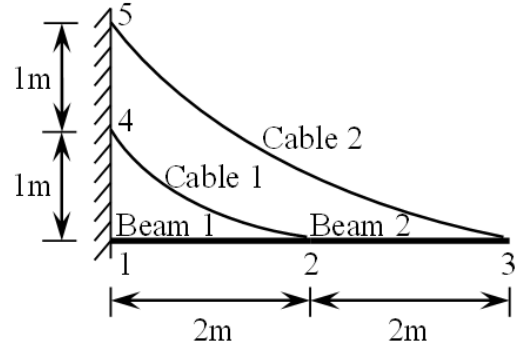
Erection of a cable-stayed cantilever structure

For LUSAS version:	24.0
For software product(s):	LUSAS Bridge / Bridge plus, or LUSAS Civil & Structural / Civil & Structural plus
With product option(s):	Nonlinear.

Description

This example demonstrates the procedure for installing cables during a construction process. Erection of a simplified cable-stayed cantilever structure via several stages is simulated. Cable lengths and cable tensions are specified, and the deformation of the cantilever is determined. Units of N, m, kg, s, C are used throughout the analysis.



The beams have cross-section area $1.18\times10^{-3}m^2$, inertia moment $2.22\times10^{-6}m^4$, weight density $78kN/m^3$, Young's modulus 200GPa; the cables have cross-section area $6.74\times10^{-5}m^2$, self weight per unit undeformed length 0.2kN/m, Young's modulus 170GPa.

The structure is erected in 5 stages that equate to 5 loadcases.

Stage / Loadcase	Description
1	Install beam 1, hang cable 1 with a cable length of 2.24m
2	Tension cable 1 with a tension of 4kN at node 4
3	Install beam 2, hang cable 2 with a length of 4.48m
4	Tension cable 2 with a tension of 4kN at node 5
5	Adjust cable tension of cable 1 to be 6kN at node 4 and cable 2 to be 8kN at node 5

Stages 1 and 3 are shape finding; stages 2, 4 and 5 are cable tensioning.

The deformation states experienced by the cables are shown in the following table.

Stage / Loadcase	Cable 1	Cable 2
1	Tension with given undeformed length	
2	Tension with given cable force	
3	Tension with given undeformed length	Tension with given undeformed length
4	Tension with given undeformed length	Tension with given cable force
5	Tension with given cable force	Tension with given cable force

Objectives

The required output from the analysis consists of:

- Deformed shape plot
- Calculation of cable tension for the stated cable length
- Calculation of cantilever deformation or position

Keywords

3D, Beam, Cable, Shape Calculation, Nonlinear Control, Target values, Activate and Deactivate.

Associated Files

Associated files can be downloaded from the user area of the LUSAS website.

- **cable_stay_cantilever.lvb** this carries out automated modelling of the example with all attributes assigned apart from any cable shape attributes, because these are added to the model as part of this example.

- **cable_stay_cantilever_completed.lvb** this carries out automated modelling of the complete example including all cable shape attribute assignments. It is supplied in case of issues in creating the required model for solving.

Points to note

- A cable is generally modelled as a perfectly flexible 1D continuum, with no rigidity, that undergoes tension-stiffening and large deflection with loading, exhibiting highly nonlinear behaviours.
- When a cable is introduced into a LUSAS model, a shape calculation is undertaken to set up its initial state.
- The analysis of cable structures is strongly nonlinear because the effects of cable sag require large deflection theory (geometric nonlinearity) for accurate estimation, which is intrinsic to the catenary cable elements used.

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller, see the heading ***Running LUSAS Modeller*** in the *Introduction to LUSAS Worked Examples* document.

Creating a New Model

File
New…

- Enter a file name of **cable_stayed_cantilever**
- Use the default **User-defined** working folder
- Ensure an Analysis category of **3D** is set and press **OK**.

Note. There is no need to enter any other new model details when the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script…

To create the model, open the file **cable_stay_cantilever.lvb** that was downloaded and placed in a folder of your choosing.

Saving the model

File
Save

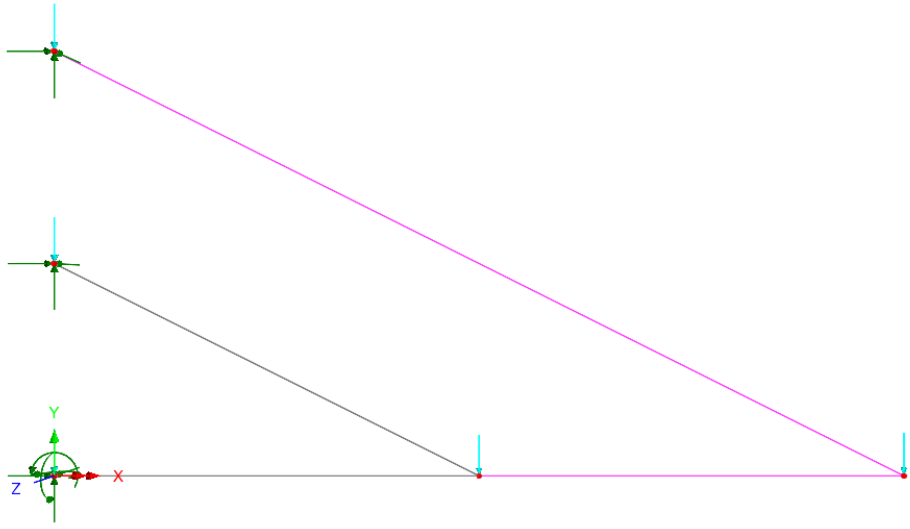
Save the model file.

Note. The Undo button may be used to correct a mistake. The undo button allows any number of actions since the last save to be undone.

Model Supplied

The model contains a single analysis, with five nonlinear structural loadcases to model the construction sequence.

The model geometry and all attributes excluding the required cable shape attributes are already defined and assigned in the supplied model.



The two lines representing the cantilever are meshed with one 3D thick nonlinear beam element (BMI21). The two lines representing the cables are defined with a line direction from the support to the point on the cantilever and are each meshed with a single 3D catenary element (CTNS2).

Note. A single catenary element is enough to model a whole cable in a static analysis; however, multiple elements are required for a dynamic analysis.

A deactivate attribute specifying 'Custom inactive treatment' with a stiffness set to '1E-6' and 'Inactive node control' set to 'Line mesh control' has already been assigned to the lines representing beam 2 and cable 2 in loadcase 1.

Activate attributes, are already assigned to the same lines in loadcase 3.

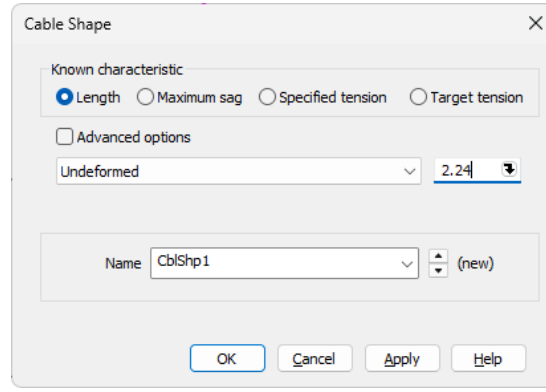
Defining cable properties

Loadcase 1

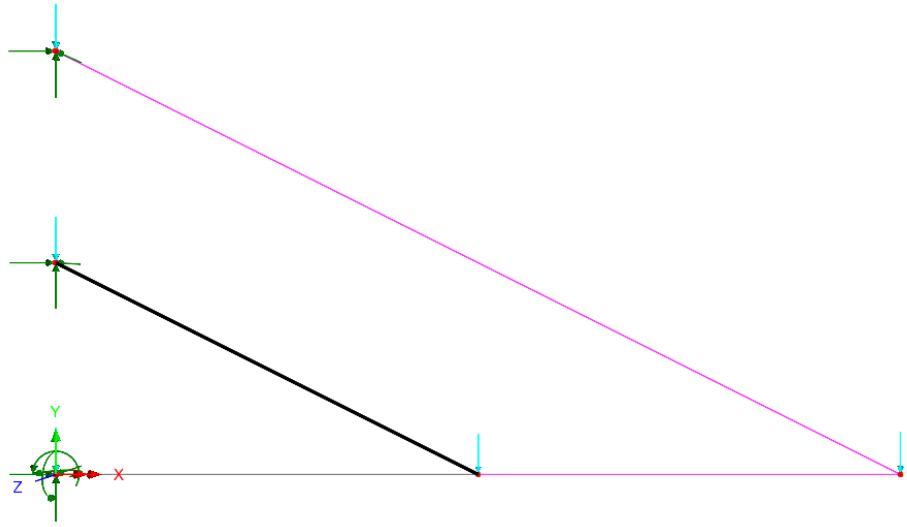
At stage 1 the first cable is installed. (The second beam and second cable have already been deactivated in this loadcase).

Attributes
Cable Shape…

- For stage 1 the cable is defined as being **Undeformed** with a length of **2.24**.
- Accept the default name of **CblShp1** and press **OK**.



- Select the line representing cable 1.



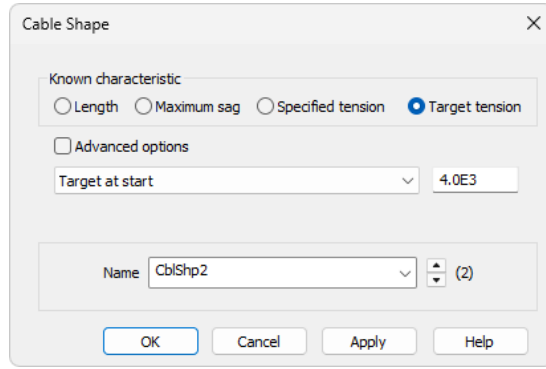
- From the Attributes treeview drag and drop **CblShp1** onto the selected feature. Select the **From loadcase** option to assign it to **loadcase 1.**

Loadcase 2

At stage 2 the installed cable is to be tensioned.

Attributes
Cable Shape…

- Specify a **Target tension** with a **Target at start** of **4.0E3**.
- Accept the default name of **CblShp2** and press **OK**.



Note. The order in which the points are selected when initially defining the lines representing cables determines the direction of the lines and therefore relate to the location of the 'Value at start' as referenced by the cable shape attributes.

With the line representing cable 1 still selected:

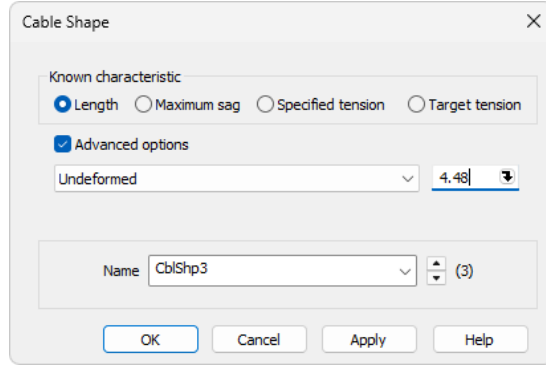
- From the Attributes  treeview drag and drop **CblShp2** onto the selected feature. Select the **From loadcase** option and assign it to **loadcase 2**

Loadcase 3

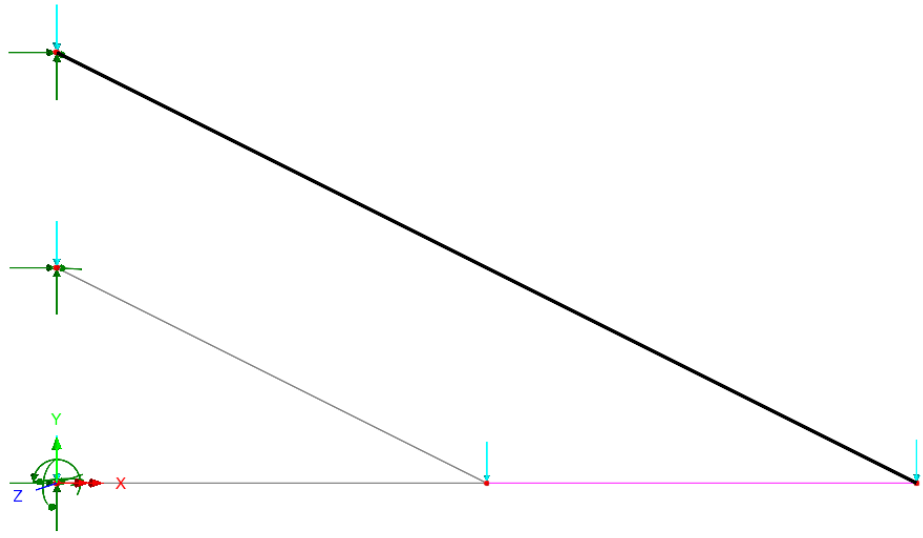
At stage 3 the second cable is installed. (The second beam and second cable have already been activated in this loadcase).

Attributes
Cable Shape…

- For stage 3 the cable is defined as being **Undeformed** with a length of **4.48**.
- Accept the default name of **CblShp3** and press **OK**.



- Select only the line representing cable 2.



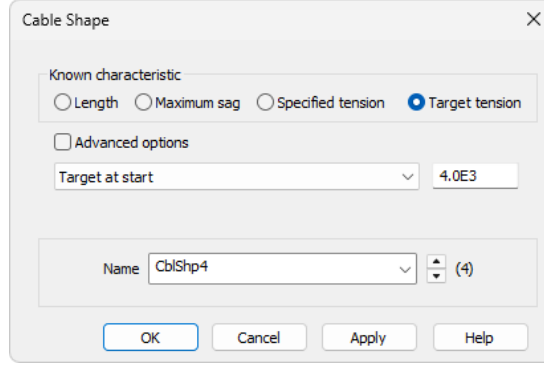
- From the Attributes treeview drag and drop **CblShp3** onto the selected feature, assigning it to **loadcase 3**

Loadcase 4

At stage 4 the second cable is tensioned.

Attributes
Cable Shape…

- Specify a **Target tension** with a **Target at start** of **4.0E3**.
- Accept the default name of **CblShp4** and press **OK**.



With only the line representing cable 2 still selected:

- From the Attributes treeview drag and drop **CblShp4** onto the selected feature, assigning it to **loadcase 4**

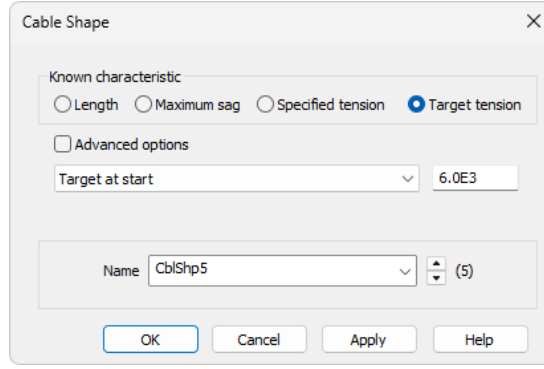
Loadcase 5

At stage 5 the tension in both cables is adjusted.

For cable 1:

Attributes
Cable Shape…

- Specify a **Target tension** with a **Target at start** of **6.0E3**.
- Accept the default name of **CblShp5** and press **OK**.

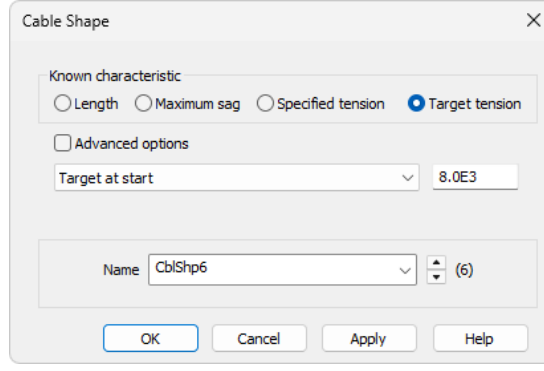


- Select only the line representing cable 1.
- From the Attributes treeview drag and drop **CblShp5** onto the selected feature, assigning it to **loadcase 5.**

For cable 2:

Attributes
Cable Shape…

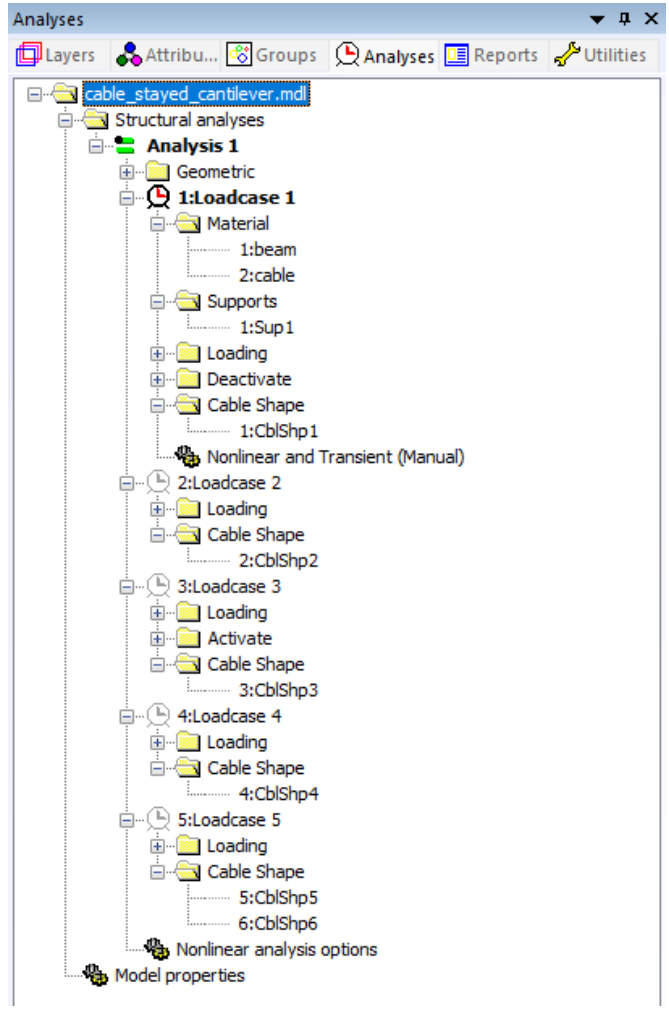
- Specify a **Target tension** with a **Target at start** of **8.0E3**.
- Accept the default name of **CblShp6** and press **OK**.



- Select only the line representing cable 2.
- From the Attributes treeview drag and drop **CblShp6** onto the selected feature, assigning it also to **loadcase 5.**

Analyses treeview

After the creation and assignment of the cable shape attributes to the model the Analyses treeview should look like this:



Running the Analysis

Open the **Solve Now** dialog and press **OK** to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Analyses Treeview.

In addition, these files will be created in the LUSASFiles\<model_name> folder:

- **cable_stay_cantilever.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **cable_stay_cantilever.mys** this is the LUSAS results file which is loaded automatically into the Analyses Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

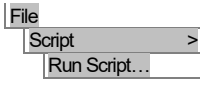
If errors are listed that for some reason cannot be corrected, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully. You may download this file from the user area of the LUSAS website.

- **cable_stay_cantilever_completed.lvb** carries out automated modelling of the example.

Enter a file name of **cable_stayed_cantilever**

- Use the default **User-defined** working folder.
- Ensure an Analysis type of **3D** is set.
- Click the **OK** button.

Note. There is no need to enter any other new model details when the intention is to run a script to build a model, since the contents of the script will overwrite any other settings made.

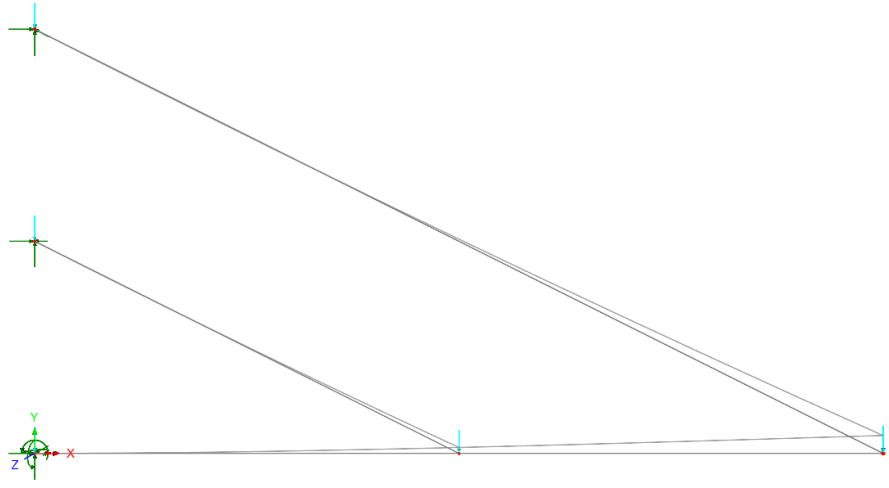


To create the model, open the file **cable_stay_cantilever_completed.lvb** that was downloaded and placed in a folder of your choosing.

Rerun the analysis to generate the results.

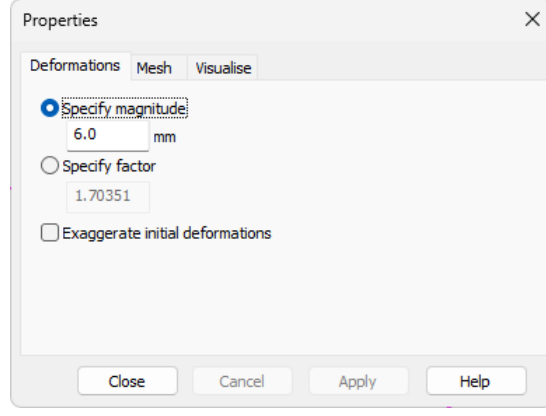
Viewing the Results

A deformed mesh plot will be seen in the model view window. The results for each loadcase can be seen in the Analyses Treeview. The last active loadcase (loadcase 5 in this case) is set to be active by default.

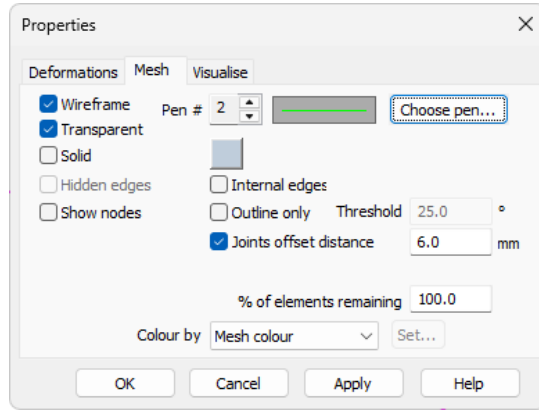


Deformed Mesh Plot

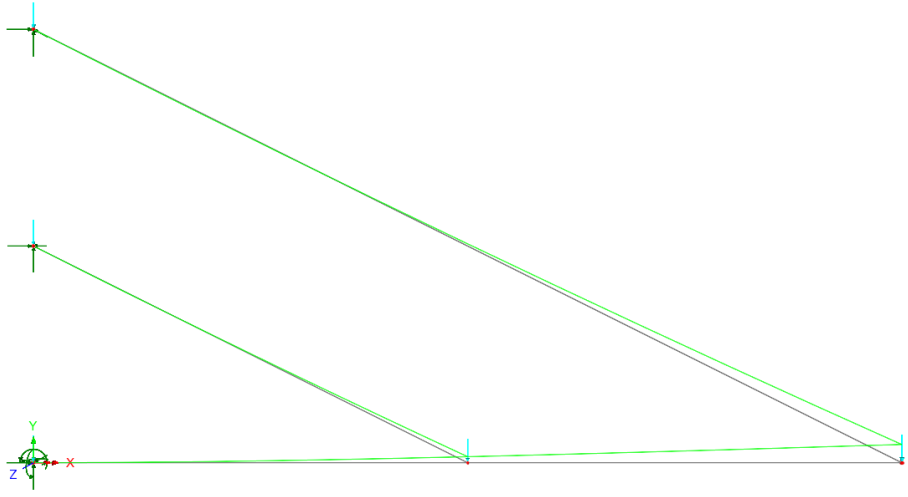
- In the Layers treeview, double click on the **Deformed mesh** entry and on the resulting properties dialog ensure that **Specify magnitude** is selected with a value of **6**.



- Now select the **Mesh** tab and using the **Choose pen…** button specify the deformed mesh colour to be **Green** and press **OK.**

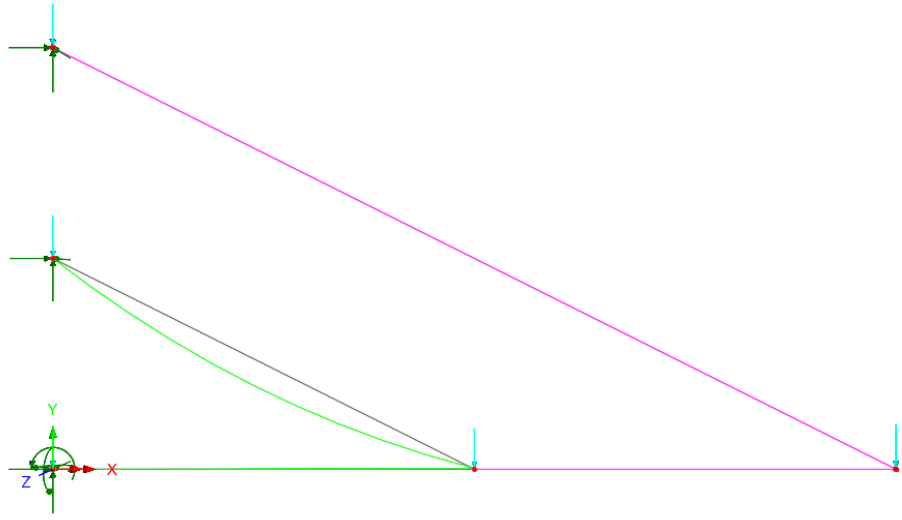


The deformed mesh for loadcase 5 will be shown.



View Displacement Values

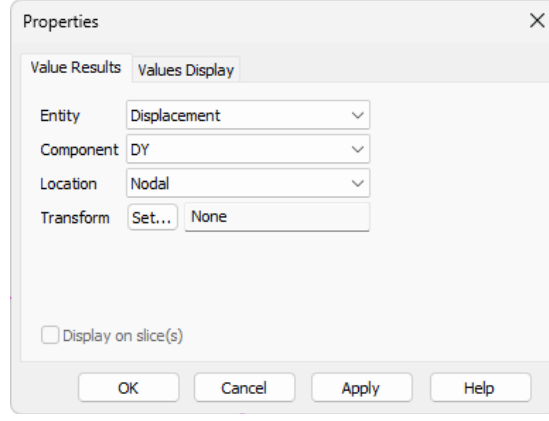
- Set active loadcase 1



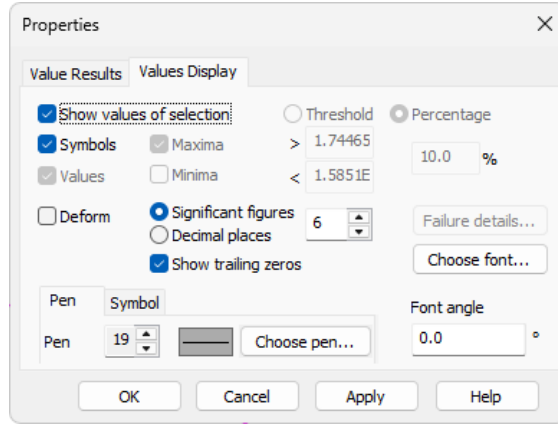
- Click the right-hand mouse button in a blank part of the view window and select the **Values** option to add the Values layer to the Treeview.

The Values properties dialog will be displayed.

- Select entity **Displacement** and component **DY** and then choose the **Values Display** tab.



- On the **Values Display** tab select **Show values of selection** and press **OK**.

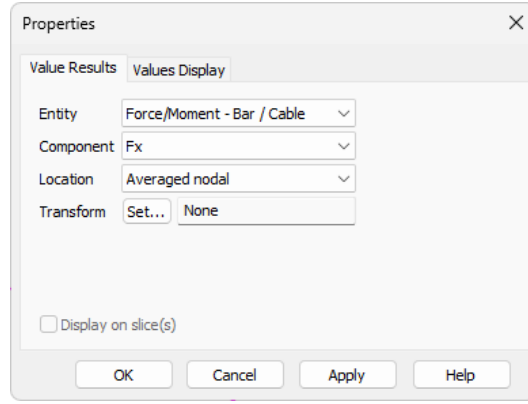


- Select the green line representing the deformed shape of cable 1. The nodal DY values will be shown.

Values of displacement DX can be shown by re-visiting the Values properties dialog and selecting DX.

View Cable Force Values

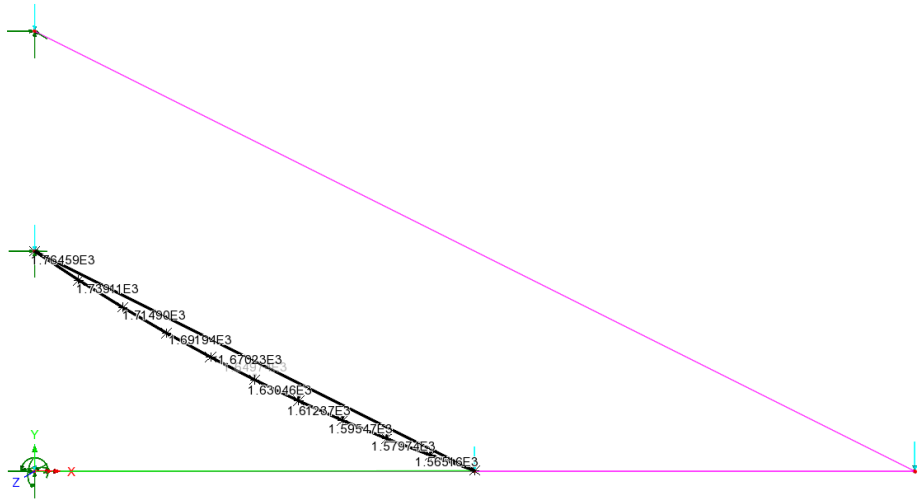
- In the Layers treeview, double-click on the **Values** entry and select entity **Force/Moment – Bar/Cable** and component **Fx**. Ensure that **Averaged nodal** values are selected and press **OK**.



Values of Fx will be shown at the end points (nodes) of the cable.

To view the change in force Fx along the cable:

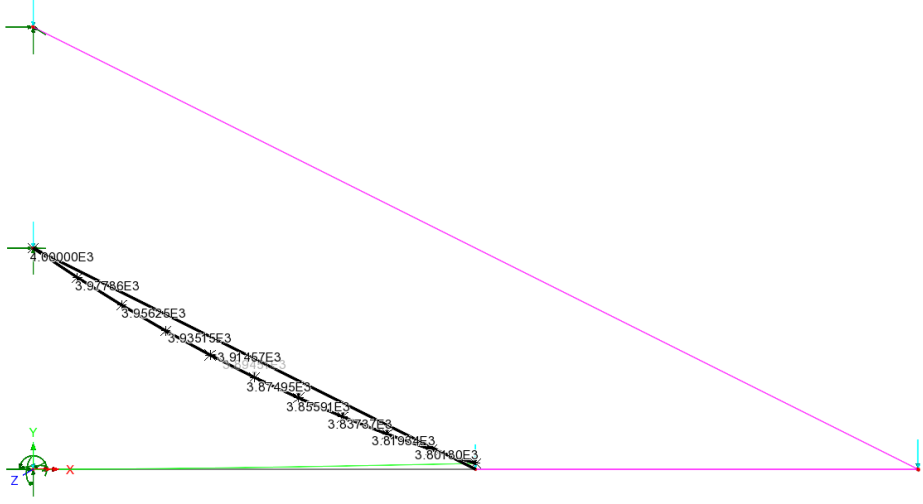
- In the Layers treeview, double-click on the **Values** entry and choose **Internal point** from **Location**.



Viewing results for other loadcases

- In the Analyses Treeview right-click on **Loadcase 2** and select **Set Active**.

The deformed mesh and associated results for Fx for Loadcase 2 will be displayed.



Results for the other loadcases can be viewed in a similar manner.

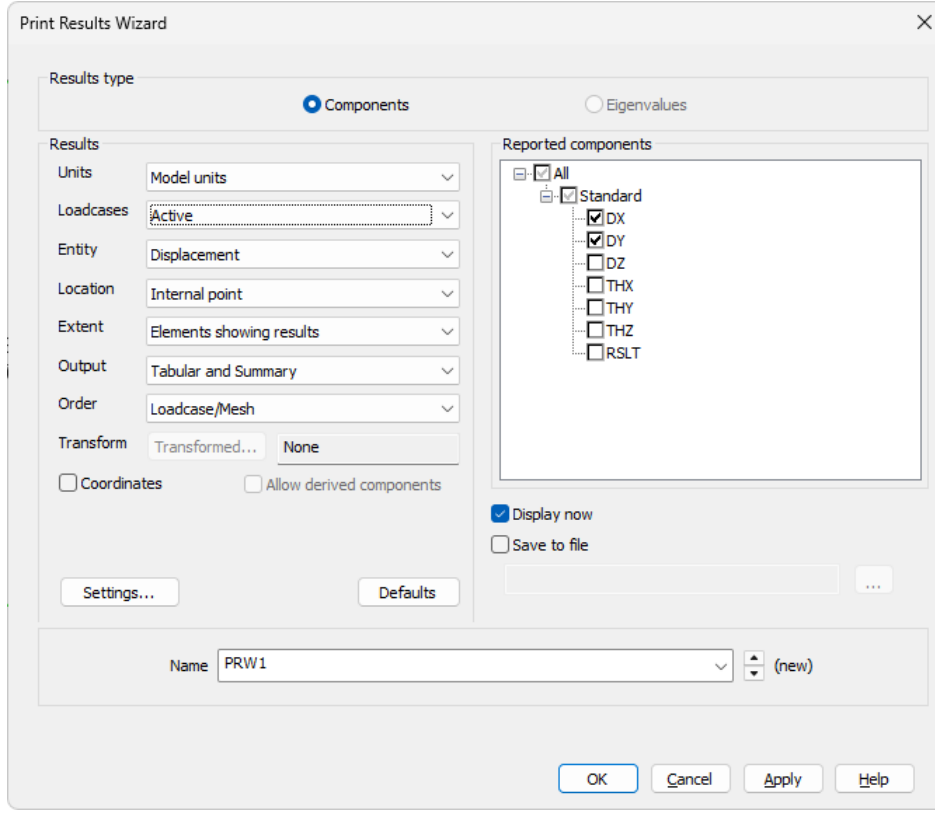
Printing the Results

- In the Analyses Treeview, right-click and set **Loadcase 5** active.

Displacement in cables

Utilities
Print Results Wizard...

- Press the **Components** radio button.
- From the Results loadcases drop-down menu select **Active**, select entity **Displacement** and location **Internal point**.
- Ensure that **DX** and **DY** are the only selected reported components and press **OK.**



The displacements at the internal points of each element for this (final) active loadcase will be output to the print results window.

LUSAS View: cable_stayed_cantilever.mdl Window 1 | Displacement in Global Axes (Elements showing results)

	Element	GP	DX[m]	DY[m]
1	1	1	0.0	0.0
2	1	2	-10.5322E-6	0.680007E-3
3	1	3	-21.0644E-6	2.64666E-3
4	1	4	-31.5967E-6	5.81383E-3
5	1	5	-42.1289E-6	0.010095
6	1	6	-52.6611E-6	0.0154035
7	1	7	-63.1933E-6	0.0216521
8	1	8	-73.7255E-6	0.0287534
9	1	9	-84.2578E-6	0.0366195
10	1	10	-94.79E-6	0.0451624
11	1	11	-0.105322E-3	0.0542936
12	2	1	-0.105322E-3	0.0542936
13	2	2	-0.165608E-3	0.0639272
14	2	3	-0.228834E-3	0.0740119
15	2	4	-0.294681E-3	0.0844988
16	2	5	-0.36283E-3	0.0953389
17	2	6	-0.432958E-3	0.106483
18	2	7	-0.504741E-3	0.11788
19	2	8	-0.577851E-3	0.129482
20	2	9	-0.651961E-3	0.141237
21	2	10	-0.72674E-3	0.153094
22	2	11	-0.801853E-3	0.165003
23	3	1	0.0	0.0
24	3	2	-12.7221E-6	-2.06169E-3
25	3	3	-23.2491E-6	-2.44494E-3
26	3	4	-33.7419E-6	-1.15426E-3
27	3	5	-43.963E-6	1.80467E-3
28	3	6	-54.5667E-6	6.42674E-3
29	3	7	-65.0178E-6	0.0127063
30	3	8	-75.8483E-6	0.0206385
31	3	9	-86.4082E-6	0.0302176
32	3	10	-96.9343E-6	0.0414388
33	3	11	-0.105322E-3	0.0542963
34	4	1	0.0	0.0
35	4	2	-83.8232E-6	-5.95434E-3
36	4	3	-0.163999E-3	-6.85507E-3
37	4	4	-0.244118E-3	-2.72582E-3
38	4	5	-0.324797E-3	6.40887E-3
39	4	6	-0.405137E-3	0.0205243
40	4	7	-0.485141E-3	0.039597
41	4	8	-0.564814E-3	0.063604
42	4	9	-0.645042E-3	0.0925233
43	4	10	-0.725215E-3	0.126333

Model info | 5:Loadcase 5 - 5:Increment 5 | 5:Loadcase 5 - 5:Increment 5(Summary)

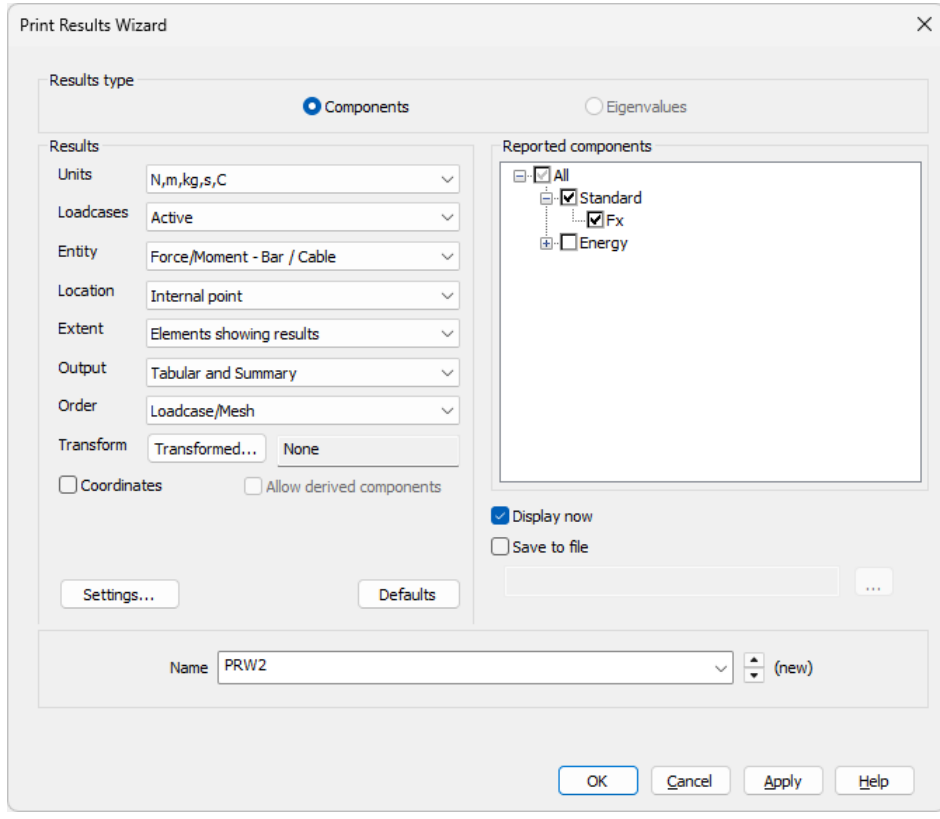
Forces in cables

To print results of force in the cables:

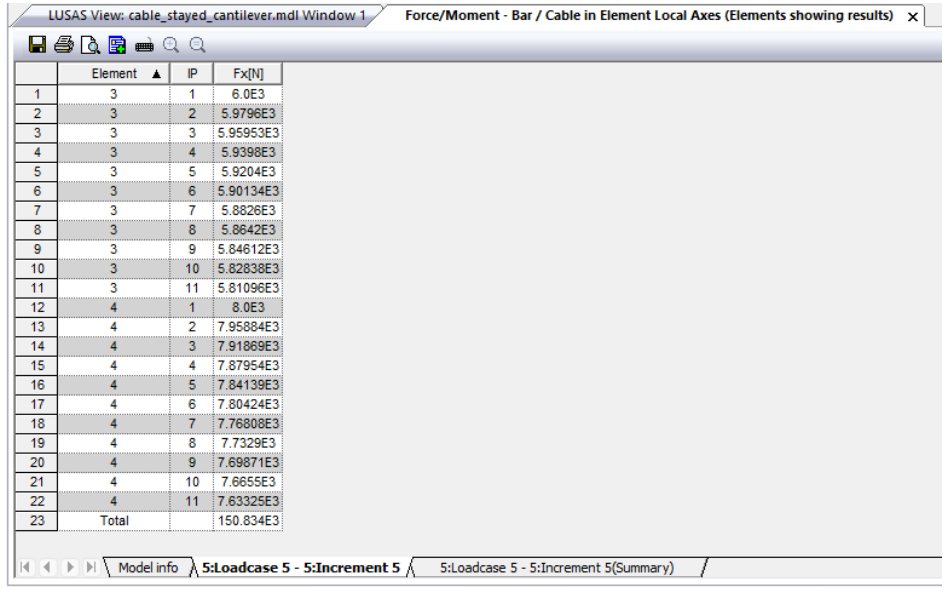
- Re-select the model view window.

Utilities
Print Results Wizard...

- Select **Force/Moment – Bar / Cable** and press the **OK** button to report on component **Fx**.



The forces at the internal points of each cable will be output to the print results window.


LUSAS View: cable_stayed_cantilever.mdl Window 1 | Force/Moment - Bar / Cable in Element Local Axes (Elements showing results)

	Element	IP	Fx[N]
1	3	1	6.0E3
2	3	2	5.9796E3
3	3	3	5.95953E3
4	3	4	5.9398E3
5	3	5	5.9204E3
6	3	6	5.90134E3
7	3	7	5.8826E3
8	3	8	5.8642E3
9	3	9	5.84612E3
10	3	10	5.82838E3
11	3	11	5.81096E3
12	4	1	8.0E3
13	4	2	7.95884E3
14	4	3	7.91869E3
15	4	4	7.87954E3
16	4	5	7.84139E3
17	4	6	7.80424E3
18	4	7	7.76808E3
19	4	8	7.7329E3
20	4	9	7.69871E3
21	4	10	7.6655E3
22	4	11	7.63325E3
23	Total		150.834E3

Model info | 5:Loadcase 5 - 5:Increment 5 | 5:Loadcase 5 - 5:Increment 5(Summary)

This concludes the example.