

3D Nonlinear Static Analysis of a Cable Stayed Mast

For LUSAS version:	22.0
For software product:	Any Plus version.
With product options:	Nonlinear.

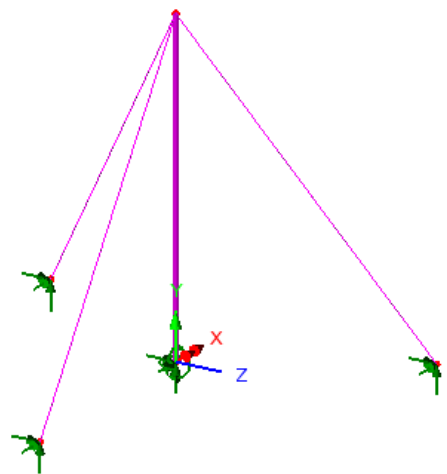
Description

A 10m tall mast with three supporting guy cables is to be analysed. The guy cables are spaced at 120° intervals around the mast and are fixed at ground level at a radius of 7m. Both the mast and cables are made of steel with a Young's Modulus of 210×10^9 Pa, Poisson's Ratio of 0.3, and a mass density of 7800 kg/m^3 .

Units of N, m, kg s, C are used throughout.

Four load types are to be considered:

1. Initial force in the cables of 5000N.
2. Self-weight of the structure.
3. Horizontal wind load of 500N/m distributed along the length of the mast.
4. A horizontal point load at the top of the mast of 5000N (acting in the negative global X direction).



Objectives

The required output from the analysis consists of:

- ☐ **Deformed shape plot**
- ☐ **Force diagram and peak forces in cables**
- ☐ **Bending moment and force diagram of the mast**
- ☐ **Reaction summary at the supports**

Keywords

3D, Thick Nonlinear Beam, Nonlinear Control, Geometric Nonlinear Analysis, Axial Force Diagram, Shear Force, Bending Moment Diagram

Associated Files

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



- ☐ **mast_modelling.lvb** carries out the modelling of the example.

Discussion

Modelling of cables generally requires modelling of tension-stiffening behaviour where the axial stiffness of the cable is dependent upon the sag of the cable, which in turn is dependent upon the tension in the cable. As a result modelling of cables generally requires a geometrically nonlinear analysis. However, in some cases, cables may be adequately represented with linear assumptions.

To model cables when cable sag is present, lines representing cables are modelled using beam elements in conjunction with a geometrically nonlinear analysis. In order to take both large displacements and stress-stiffening into account, a full nonlinear approach should be used.

This example shows the use of thick beam elements with GNL.

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.



Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command

File>New to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter a file name of **Mast**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **3D**
- Set Model units of **N,m,kg,s,C**
- Ensure Timescale units are in **Seconds**
- Select a Startup template of **3D Frame**.
- Ensure the Layout grid is set as **None**
- Enter a Title of **Mast**
- Click the **OK** button.

The Attributes  treeview will be populated with potentially useful basic line mesh, geometric and support entries appropriate to the settings made on the new model dialog.

Set the Vertical Axis

Setting an analysis category of 3D sets the vertical axis to be the Z-axis. For historical reasons this example requires the vertical axis to be the Y-axis.

Set the vertical axis to be the **Y-axis**.

Tools
Vertical axis...

Defining the Geometry

Geometry
Point >
By Coords...



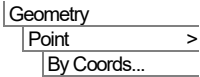
Enter co-ordinates of (0, 0, 0) and click **OK** to define a point at the bottom of the mast.

- Select the Point

Geometry
Line >
By Sweeping...



Enter **10** in the **Y** direction (vertical) and click **OK** to sweep the point.

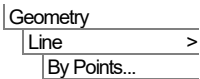


Enter co-ordinates of $(-7, 0, 0)$, $(3.5, 0, 6.0622)$ and $(3.5, 0, -6.0622)$. Click **OK** to define points where the guy cables are fixed to the ground.



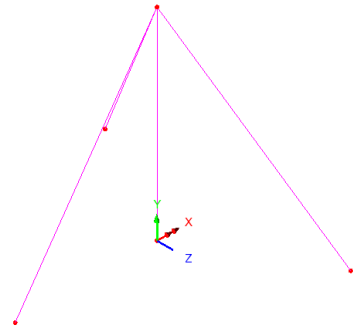
Select the isometric view button to rotate the model so that all points are visible

- Define a cable by selecting a base Point followed by an apex Point. (Use the Shift key to make multiple selections)



The lines defining the cable will be drawn.

- Repeat for the other two pairs of points.



Defining Groups

To simplify the assignment of model attributes and assist with post processing the features will be grouped together.

- Select the Line representing the mast.



- Select the **Group** button.

- Change the group name to **Mast** and click **OK**

- Select the three Lines representing the cables. (Use the Shift key to make multiple selections)



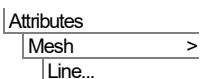
Select the **Group** button.

- Change the group name to **Cables** and click **OK**

Meshing

The Line features are to be meshed using three-dimensional nonlinear beam elements.

- Select **Thick beam, Linear** elements. (This will select beam element BMI21)
- Change the number of divisions per line to **10**
- Enter the attribute name as **Mast** and click the **Apply** button.
- Change the attribute name to **Cables** and click the **OK** button

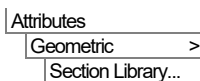


- Select the line representing the mast.
- Drag and drop the mesh attribute **Mast** from the Treeview onto the selected Line features using an element orientation with a beta angle of **0**. Click **OK** to assign.
- Select a line representing a cable and holding down the **Shift** key select the other 2 lines representing the cables.
- Drag and drop the mesh attribute **Cables** from the Treeview onto the selected Line features using an element orientation with a beta angle of **0**. Click **OK** to assign.

Geometric Properties

Mast

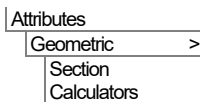
For the purposes of the example the mast will be defined as a Circular Hollow Section (CHS).



- From the **UK Sections** library select the **CHS (EN10210)** section type and select the **168.3x8 CHS** section.
- Enter an attribute name of **Mast**
- Click the **OK** button to add the CHS attribute to the Treeview.

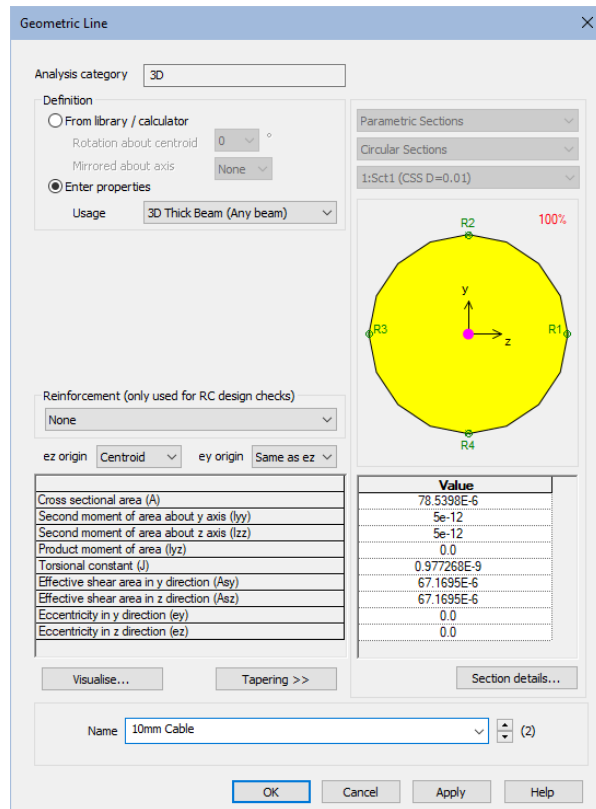
Cables

The 10mm diameter guy cables will be defined using a standard section but with the bending properties subsequently adjusted to a value of $1/100^{\text{th}}$ of the actual bending stiffness in order to not attract any bending moments. The torsional constant and shear area values will be left unaltered.



- From the droplist available select **Parametric Sections**, then **Circular Sections**, then **New...**
- On the Circular Sections dialog enter a diameter (D) of **0.01** and then click in the Name field.
- Leave the section name as the default name of **Sct1** and press the **OK** button to return to the parent dialog.
- On the Geometric line dialog, in the Definition panel, press the **Enter Properties** radio button to allow editing of the section data.
- In the Value table change the Iyy value to be **5e-12** and change the Izz value to be **5e-12** (both 100 times smaller than initially calculated)

3D Nonlinear Static Analysis of a Cable Stayed Mast

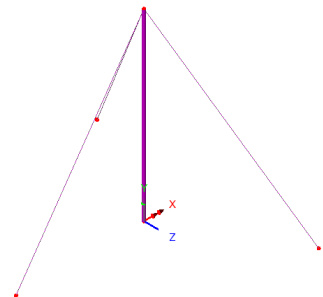


- Enter the attribute name as **10mm Cable** and click the **OK** button.

The cable geometry dataset will be added to the Treeview.

Assigning Geometry Properties

- In the  Treeview, right-click on the group name **Mast**. Choose the **Select Members** option. Click **Yes** to deselect any members already selected.
- Drag and drop the Geometric attribute **Mast (168.3x8 CHS)** from the  Treeview onto the selected line. Click **OK** to **Assign to lines**. The section assigned will be visualised.
- In the  Treeview, right-click on the group name **Cables**. Choose the **Select Members** option. Click **Yes** to deselect members already selected.



- Drag and drop the geometry attribute **10mm Cable** from the  Treeview onto the selected Lines. Click **OK** to **Assign to lines**.
-  Select the fleshing on/off button to turn-off the geometric visualisation.

Material Properties

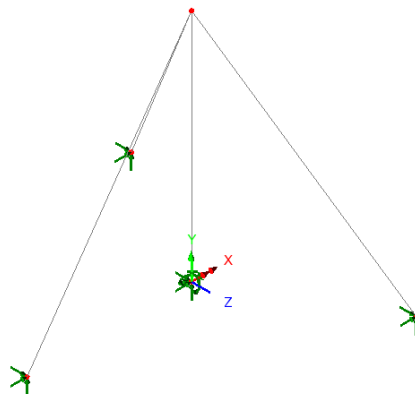
- Select the whole model (**Control + A** keys) and drag and drop the material attribute **Steel (Mild Steel | Steel Structural)** from the  Treeview onto the selected features.

Supports

The Standard template provides the more common types of support by default. These can be seen in the Treeview. The structure will be supported at the base of the mast with a fully fixed support condition. The cables will be supported in translation directions only.

Assigning the Supports

- Select the Point at the base of the mast.
- Drag and drop the support attribute **Fully Fixed** onto the selected Point for all loadcases. Click **OK** to assign.
- Select the three Points at the base of the three cables.
- Drag and drop the support dataset **Pinned** from the  Treeview onto the selected Points for all loadcases. Click **OK** to assign.



Loading

Four load types will be considered:

- ☐ Initial stresses in the cables
- ☐ Self-weight
- ☐ Horizontal wind loading
- ☐ Point load acting at the top of the structure.

The loading will be assigned to four individual loadcases.

Load type 1 - Initial Stress

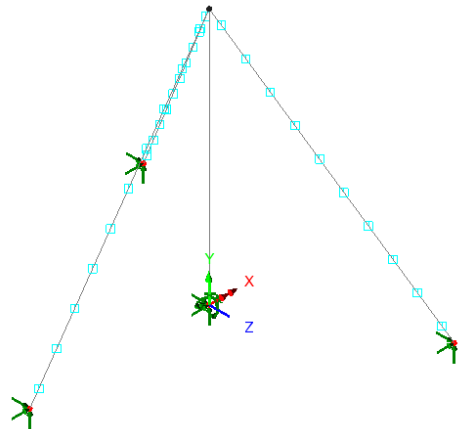
- In the  Treeview expand **Analysis 1** then rename **Loadcase 1** to be **Initial stress in cables** by selecting Loadcase 1 with the right-hand mouse button and using the rename option.

An initial force in the guy cables of 5000 N is to be defined and assigned to the model. Since the elements used to represent the mast and cables are BMI21 elements which are formulated using forces and strains (see the *Element Reference Manual* for details) the force is entered by selecting the Stress and Strain option on the structural loading dialog.

- Select the **Strain and stress** option and then in the lower panel, select **Stress and strain** and click **Next**
- In the Usage panel, ensure that a **Line** element description is specified.
- Select **Thick beam** from the drop-down list.
- Ensure the Stress and Strain Type is set to **Initial**
- Enter a value of **5000** for Fx.
- Enter the attribute name as **Initial Stress**
- Click **Finish** to add attribute to the  Treeview.

The copy and paste facility will be used to assign this loadcase to the cable group.

- In the  Treeview, right-click on the loading attribute **Initial Stress** and select the **Copy** option.
- In the  Treeview, right-click on the group name **Cables** and select the **Paste** option.
- On the Loading Assignment dialog select the loadcase name **Initial Stress in Cables**. Click **OK** to assign loadcase.



Load type 2 - Self Weight

Load type 2 represents the self-weight of the structure. This is modelled automatically by applying gravity to the loadcase. The self-weight will be assigned to both Loadcase 2 and Loadcase 3 for reasons that are explained during the Nonlinear Analysis Control section of this example.

Analyses

Loadcase...

- Select the **Automatically add gravity to this loadcase** option
- Name the loadcase **Self Weight** and press **Apply**.
- Now change the loadcase name to **Self Weight Freeze** and press **OK** to create the second self-weight load type.



Note. With this method of assigning gravity (to the loadcase) no visualisation of the loading will take place in the View window.

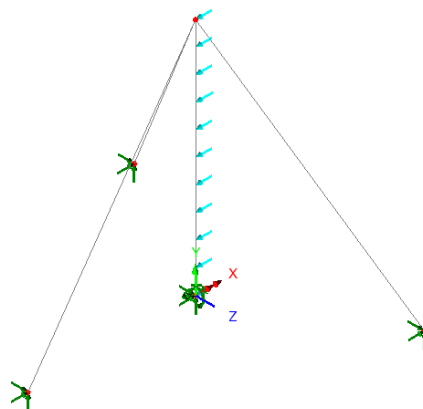
Load type 3 - Horizontal Wind Load

Load type 3 is a horizontal wind load of 500 N/m distributed along the length of the mast.

Attributes

Loading...

- Select the **Distributed loads** option, then in the lower panel select **Global Distributed** option and click **Next**
- Select the **Per Unit Length** option. Enter a value of **-500 N/m** in the **X** direction.
- Enter the attribute name as **Distributed Wind Load**.
- Click **Finish** to add the attribute to the Treeview.
- In the Treeview, right-click on the loading attribute **Distributed Wind Load** and select the **Copy** option.
- In the Treeview, right-click on the group name **Mast** and select the **Paste** option.
- On the Loading Assignment dialog change the loadcase name to **Wind and Point Load** by typing over the loadcase name shown.
- Click **OK** to assign the loading to that specified loadcase.

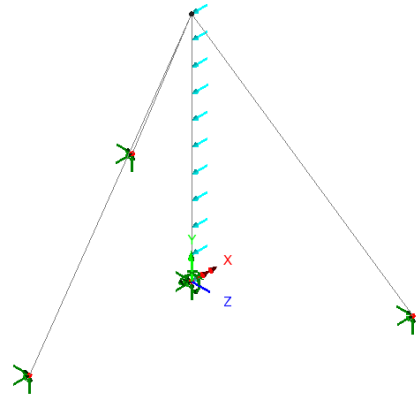


Load type 4 - Concentrated Point Load

A point load of 5000 N is acting at the top of the mast.

Attributes
Loading...

- Select the **Concentrated** option and click **Next**
- Enter a value of **-5000** in the **X** direction.
- Enter the attribute name as **Point Load**
- Click **Finish** to add the attribute to the  Treeview.
- Select the Point at the top of the mast then drag and drop the loading attribute **Point Load** from the  Treeview onto the selected Point.
- Ensure that the loading is assigned to the **Wind and Point Load** loadcase and press **OK**.



Nonlinear Analysis Control

Nonlinear control is defined as a property of a loadcase and must therefore be set for all four loadcases required for this analysis.

Loadcase 1

- In the  Treeview, right-click on **Initial Stress in Cables** and from the **Controls** menu select **Nonlinear & Transient**.
- Select **Nonlinear** incrementation with **Manual** control.
- Use default settings for this initial manual loadcase. Do not modify the dialog.
- Click **OK** to finish.



Note. A  Nonlinear and Transient object will be added to the Treeview. Double-clicking on this object will allow any changes to be made to the control properties for this loadcase.

Loadcase 2

Loadcase 2 (the self weight loadcase) uses automatic load incrementation. In doing so, the load is applied as a very small initial step to overcome any potential convergence problems. The specification of the number of iterations per increment determines how the load step is varied from one increment to the next. The analysis generally converges in 2 or 3 iterations. By setting this value to 12 the load step will rapidly increase.



Note. In LUSAS, the use of automatic nonlinear incrementation immediately following a manual nonlinear incrementation (as in this case) causes the results from

the previous analysis to be retained and used as the starting point for the subsequent analysis.

- In the  Treeview, right-click on **Self Weight** and from the **Controls** menu select **Nonlinear & Transient**.
- Select **Nonlinear** incrementation with **Automatic** control.
- Enter the Starting load factor as **0.01**
- Leave the Max change in load factor set to **0**
- Leave the Max total load factor as **1**
- Set the Iterations per increment to **12**
- In the **Incrementation** section of the dialog select the **Advanced** button.
- Ensure that Incrementation by arc-length control is set to **Off** and click the **OK** button.
- Ensure that the 'Solution strategy' and 'Incremental LUSAS file output' are set to **Same as previous loadcase**.
- Click the **OK** button to finish.

Loadcase 3

Loadcase 3 reapplies the self-weight in a single **Manual** load step. This effectively freezes the self-weight load allowing the wind loadcase 4 to be incrementally increased. If an Automatic (loadcase 4) nonlinear control were to follow a previous Automatic (loadcase 2) nonlinear control, then the loading would be overwritten.



Note. In LUSAS, using manual nonlinear incrementation immediately following an automatic nonlinear incrementation (as in this case) allows a subsequent automatic nonlinear incrementation to be applied without overwriting any previous loading.

- In the  Treeview, right-click on **Self Weight Freeze** and from the **Controls** menu select **Nonlinear & Transient**.
- Select **Nonlinear** incrementation with **Manual** control.
- Ensure that the 'Solution strategy' and 'Incremental LUSAS file output' are set to **Same as previous loadcase**.
- Click the **OK** button to finish.

Loadcase 4

Loadcase 4 applies the wind and point load using **Automatic** load incrementation. In a similar manner to that used for loadcase 2.

- In the  Treeview, right-click on **Wind and Point Load** and select **Nonlinear & Transient** from the **Controls** menu.
- Select **Nonlinear** incrementation with **Automatic** control.
- Enter the Starting load factor as **0.01**
- Leave the Max change in load factor set to **0**
- Leave the Max total load factor as **1**
- Set the Iterations per increment to **12**
- In the **Incrementation** section of the dialog select the **Advanced** button.
- Ensure that Incrementation by arc-length control is set to **Off** and click the **OK** button.
- In the **Solution strategy** section of the dialog deselect the **Same as previous loadcase** option and then select the **Advanced** button.
- In the Nonlinear iterative solution panel, ensure the Strategy is set to **Auto** and click **OK**
- Click the **OK** button to finish.

Defining a Geometric Nonlinear analysis

Two additional settings are required to allow a geometric nonlinear analysis to take place.

- Double-click on the  **Nonlinear analysis options** object in the  Treeview.
- Select the **Co-rotational Formulation** option.
- Select the **Continue solution if more than one negative pivot occurs** option.
- Click the **OK** button to finish.



Note. The  Nonlinear analysis options object sets parameters for the whole analysis and the  Nonlinear and Transient object sets control properties for each loadcase.

Save the Model

The model is now complete and the model data is to be saved before an analysis is run using the LUSAS Solver.



Save the model file.

Running the Analysis

With the model loaded:



Open the **Solve Now** dialog. Ensure **Analysis 1** is selected and press **OK** to run the analysis from Modeller.

These files will be created in the LUSASFiles\<model_name> folder:

- ☐ **mast.out** this contains the statistics of the analysis, for example how much disk space was used, how much CPU time was used, and any errors or warning messages from LUSAS. Always check the output file for error messages.
- ☐ **mast.mys** This is the LUSAS results database which will be used for results processing.

If the analysis is successful...

Analysis loadcase results are added to the  Treeview.

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 it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully. You may download this file from the user area of the LUSAS website.



- ☐ **mast_modelling.lvb** this carries out the modelling of the example.

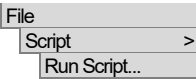


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **mast**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **3D** is set.
- Click the **OK** button.



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



- To recreate the model, select the file **mast_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Re-run the analysis to generate the results

Viewing the Results

Selecting the results to be viewed

Loadcase results can be seen in the  Treeview. A deformed mesh layer is automatically displayed for the active loadcase. A default deformed mesh magnitude of 6mm on screen size is used.

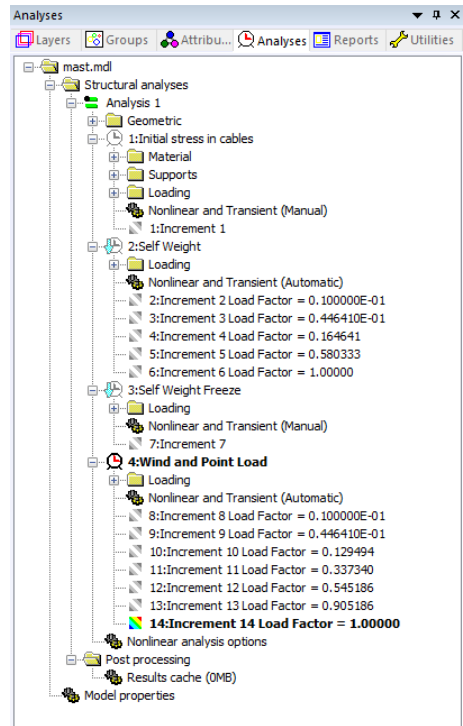


Note. With reference to the Analyses Treeview and loadcase results opposite, the first increment relates to the initial stress loadcase.

The first set of load factor increments relate to the self-weight loadcase.

Increment 7 relates to the ‘freezing’ of the self-weight loadcase prior to loading the structure with the wind and point load.

The remaining load factor increments relate to the application of the wind and point loading on the pre-stressed structure.



Deformed Mesh Plot for Initial Stress in Cables

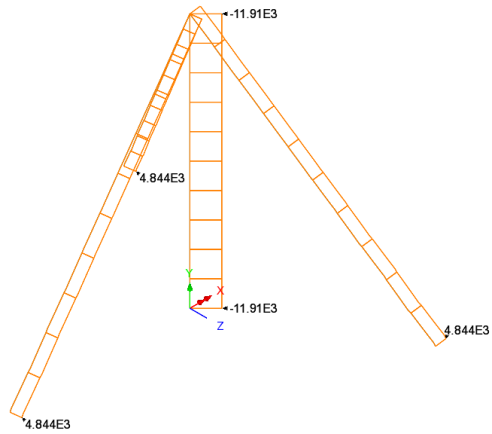
- In the Treeview, right-click on the first load increment **Increment 1** and select the **Set Active** option.
- In the Treeview turn off the **Mesh**, **Geometry**, and **Attribute** layers to view only the deformed mesh for the first results loadcase **Initial Stress in Cables**



Note. With the deformed mesh factor set to 6mm, irregularities in the deformed mesh will be seen for this and some other loadcases. This is due to very small displacements in the elements modelling the cables being magnified by a very large deformation factor. Specifying a screen magnitude of 1mm or a smaller deformation factor would improve the display and is used to create the images that follow.

Axial Force Diagram for Initial Stresses

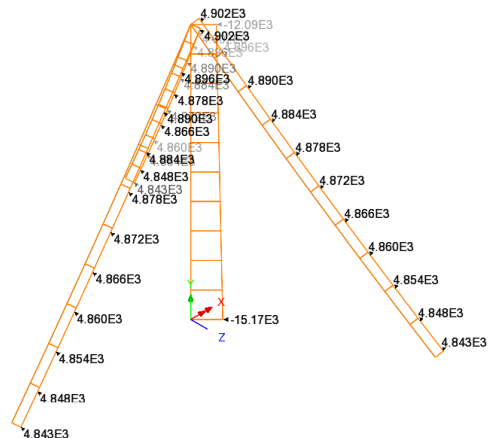
- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Diagrams** option.
- Select the entity **Force /Moment** of component **Fx** (axial force)
- Select the **Diagram Display** tab and ensure the **Label values** and **Peak values** options are chosen. Deselect the **Use for labels too** option (this will display values in the default black colour) and set the number of significant figures to **4**.
- Click **OK** to add the diagrams layer to the  Treeview and view the results



Combined Initial Stress and Self weight

- In the  Treeview right-click on the last results increment inside the **Self Weight** section for **Load Factor = 1.00000** and select the **Set Active** option.

This shows axial force from the combined effect of the initial stress and the self-weight on the mast.



- In the Treeview right-click on the last results increment in the **Wind and Point load** section for **Load Factor = 1.00000** and select the **Set Active** option.

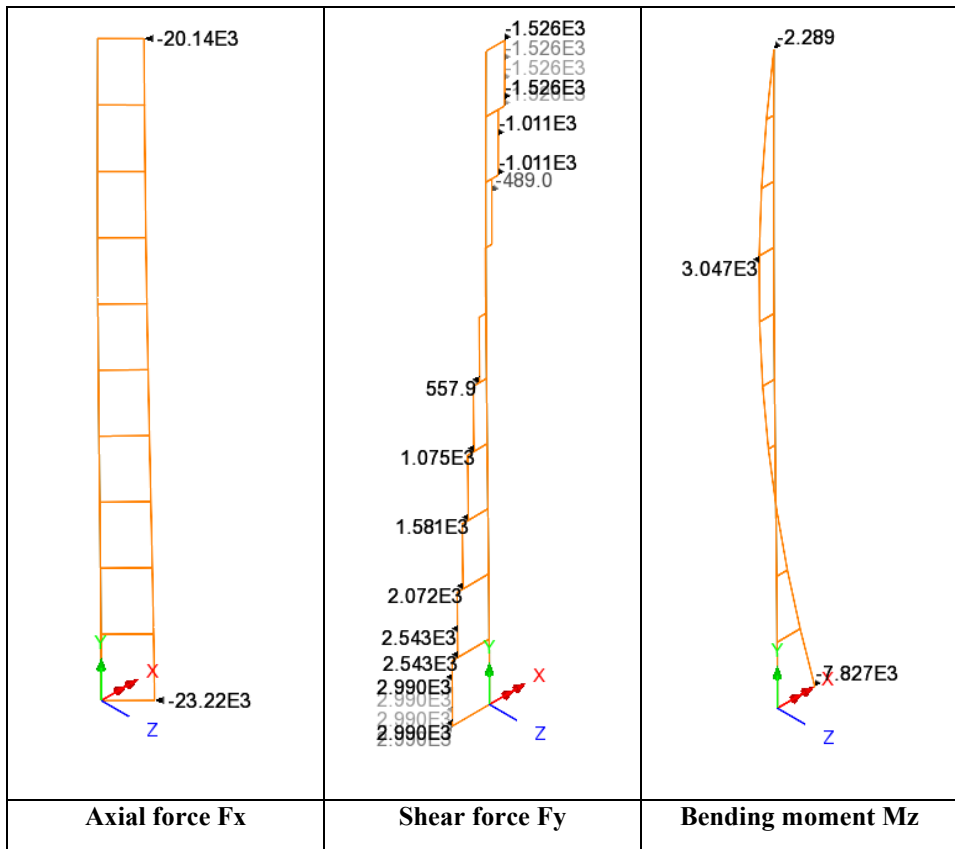
-

Axial Force Diagram for the mast only

- ### Shear Force Diagram for the mast

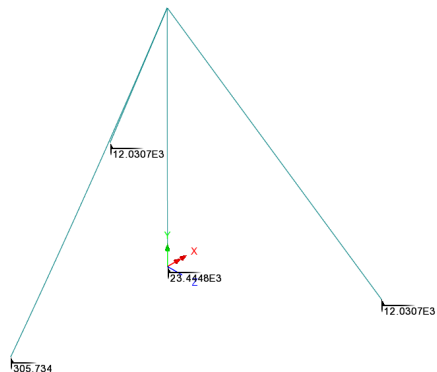
- ### Bending Moment Diagram

- Diagram results for Wind and Point load loading (Load factor 1)



Plotting Reactions at the Mast and Cable Supports

- In the  Treeview turn off the display of the **Diagrams** layer.
- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **All Visible** option.
- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Values** option to add the values layer to the  Treeview. Select results of entity **Reaction** of component



RSLT.

- Select the **Values Display** tab and plot **100%** of **Maxima** values
- Click the **OK** button.

Printing a Summary of Reactions

With the last results increment for Wind and Point load loading (Load factor 1) still selected in the Treeview:

Utilities

Print results wizard

- Select Entity **Reaction**.
- Select output for **Tabular and Summary**
- Leave other settings as their defaults.
- Click **OK** to view the tabulated results and summary of reactions at each support node.

Tabular results tab:

	Node ▲	FX[N]	FY[N]	FZ[N]	MX[N.m]	MY[N.m]	MZ[N.m]	RSLT[N]
1	1	3.24008E3	23.2198E3	-26.5173E-12	-0.365758E-9	-0.362621E-12	-7.82725E3	23.4448E3
2	22	-191.133	-238.625	-0.129082E-12	N/A	N/A	N/A	305.734
3	42	3.47552E3	-9.8393E3	5.98717E3	N/A	N/A	N/A	12.0307E3
4	62	3.47552E3	-9.8393E3	-5.98717E3	N/A	N/A	N/A	12.0307E3
5	Total	10.0E3	3.30259E3	-0.938599E-9	N/A	N/A	N/A	N/A

4:Wind and Point Load - 14:Increment 14 Load Factor = 1.00000 4:Wind and Point Load - 14:Increment 14 Load Factor = 1.00000(Summary)

Summary results tab:

	Component ▲	FX[N]	FY[N]	FZ[N]	MX[N.m]	MY[N.m]	MZ[N.m]	RSLT[N]
1	Maximum	3.47552E3	23.2198E3	5.98717E3	-0.365758E-9	-0.362621E-12	-7.82725E3	23.4448E3
2	Node	42	1	42	1	1	1	1
3	Minimum	-191.133	-9.8393E3	-5.98717E3	-0.365758E-9	-0.362621E-12	-7.82725E3	305.734
4	Node	22	42	62	1	1	1	22
5	Total	10.0E3	3.30259E3	-0.938599E-9	N/A	N/A	N/A	N/A

4:Wind and Point Load - 14:Increment 14 Load Factor = 1.00000 4:Wind and Point Load - 14:Increment 14 Load Factor = 1.00000(Summary)

This completes the example.

