# Linear Cable Tuning Analysis of a Pedestrian Bridge

For LUSAS version:	18.0
For software product(s):	LUSAS Bridge / Bridge plus, LUSAS Civil & Structural/ Civil & Structural plus only
With product option(s):	None.

Description

This example demonstrates the procedure for setting up a cable tuning analysis on a simplified 3D beam model of a cable-stayed pedestrian bridge. Optimum cable forces to maintain a flat deck and vertical tower under permanent loads are to be determined.
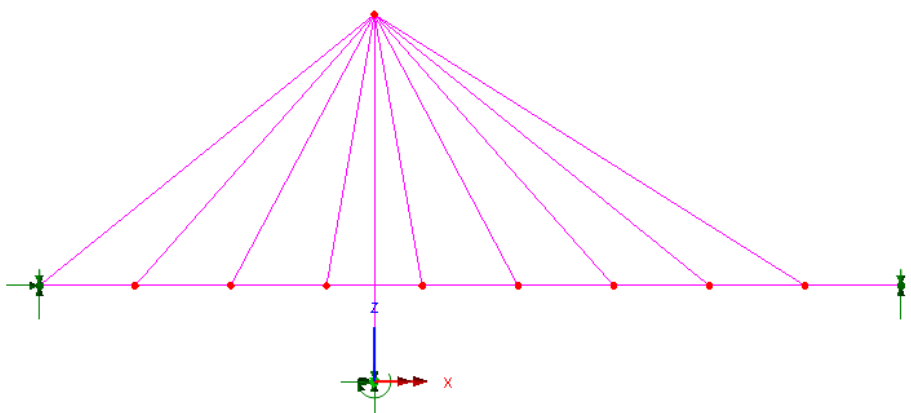
Units of **kN, m, t, s, C** are used throughout the analysis.

Objectives

The required output from the analysis consists of:

❑ Calculation of cable forces required to maintain a horizontal deck and vertical tower under permanent loading.

Keywords

3D, Beam, Bar, Linear Cable Tuning, Target values, Targets, Exact solution.

Associated Files

❑ **cable_tuning.mdl** contains the model geometry with all of the attributes assigned. The example will use this model file as a starting point.

Discussion

Cable tuning analysis enables automatic calculation of cable forces to optimise certain specified criteria, for example, to limit displacements in a bridge deck and/or tower during construction.

The analysis of cable structures is strictly a nonlinear problem because the effects of cable sag require large deflection theory (geometric nonlinearity) for accurate estimation. However, because bridge cables are typically stressed to such a high force that the amount of sag is small, it is often acceptable to idealise such analyses to equivalent linear problems by neglecting the sag effects.

A common method of doing this is to represent the cables with a single linear bar element. Bar elements only carry axial force (no moment or shear) and by using a single element no lateral deflection (i.e. sag) is permitted, so a reasonable approximation to true cable behaviour is achieved. It is important to note that this method would potentially permit the cable elements to carry compressive forces, so the predicted forces must be checked to ensure this is not the case under any in-service combination of loading.

Modelling

Running LUSAS Modeller

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

Creating the Model

For this example a model file is provided:

To create the model, open the read only file **cable_tuning.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Depending upon your access rights it may be necessary to copy this file to another folder before it can be opened, and remove the read-only protection before it can be saved to the new folder. Once opened:
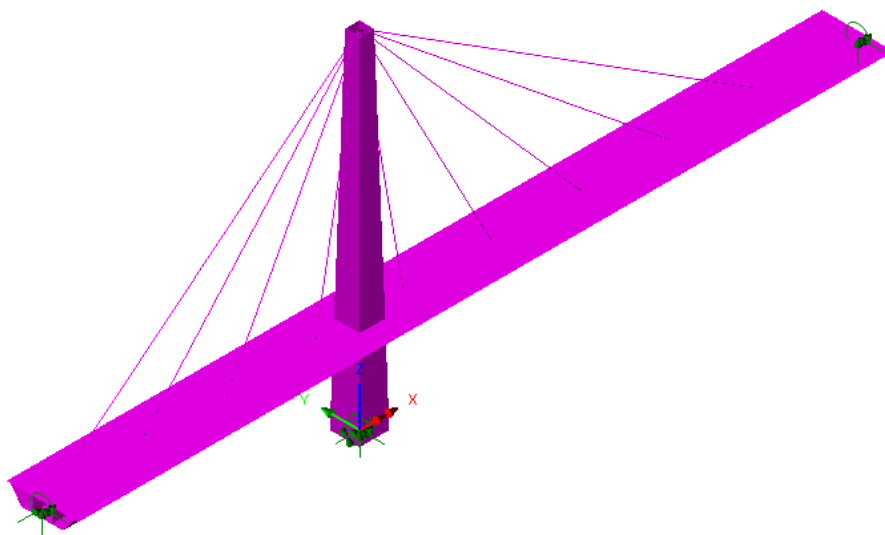
🖫 Save the model file to the **\<Lusas Installation Folder>\Projects** directory.

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 geometry and attributes are already defined in the supplied model file.

The bridge to be analysed consists of a 46m high tower supporting a deck of total length 108m.

The tower is a steel square hollow section, varying from 4m square at the base to 2m square at the top, with 50mm thick walls throughout. It is fully fixed at its base and is not connected directly to the deck where the two features cross.

The deck is a steel box section of total width 7m, having a cross-section as shown on the following image. The deck consists of nine 12m spans between supporting cables. The deck is pin-supported at the left-hand end and roller-supported at the other.
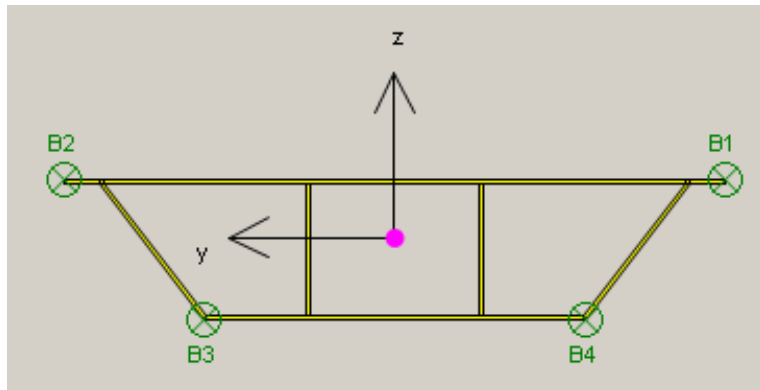
Steel box section for the deck

The lines representing the tower and the deck are meshed with thick 3D thick nonlinear beam elements (BMI21). The lines representing the cables are each meshed with a single 3D bar element (BRS2). Each cable is represented by a circular solid section of 50mm diameter.

The model contains a single analysis, with two linear structural loadcases – **Self-Weight** (where the self weight is assigned to all members using 'auto gravity' as a property of the analysis) and **Surfacing** (20kN/m assigned to the deck members only).

To start

- Switch off section fleshing by clicking and rotate the model into the X-Z plane by pressing in the status bar.
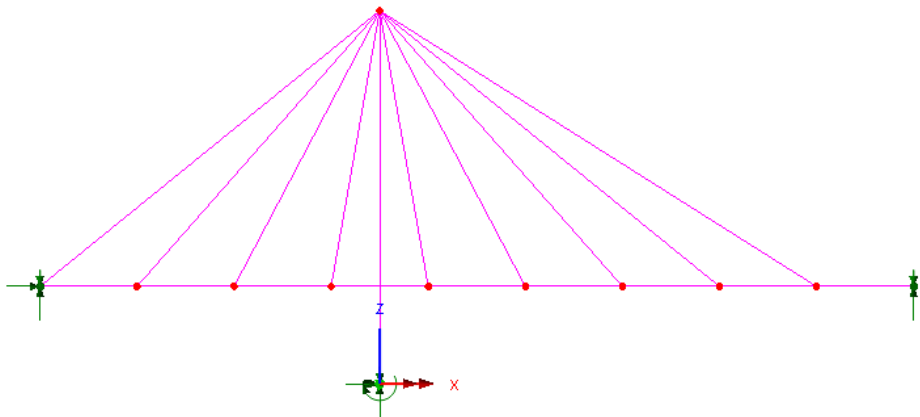
Solving the Supplied model

Press the **Solve Now** button to check the assignments made so far.

The **Deformed Mesh** layer will now be visible, and can be used to check that the deformed shape for each of the two loadcases is reasonable.

Check the reactions

Utilities
Print Results
Wizard…

- Make selections to print results for **All** loadcases, of entity **Reaction**, and output type **Summary**.

- In the Reported components panel, ensure that **All** is selected**.**

- Ensure **Display now** is selected and click **OK**.

	Component ▲	FX[kN]	FY[kN]	FZ[kN]	MX[kN.m]	MY[kN.m]	MZ[kN.m]	RSLT[kN]
1	Maximum	326.453	6.77027E-27	8.44981E3	0.0	-15.0168E3	0.0	8.45611E3
2	Node	4	1	1	4	1	1	1
3	Minimum	-326.453	0.0	1.13789E3	-0.227504E-24	-15.0168E3	0.0	1.18379E3
4	Node	1	4	4	1	1	1	4
5	Total	0.106525E-9	6.77027E-27	11.9342E3	N/A	N/A	N/A	N/A

Reaction in Global Axes (Elements showing results)

Model info \ 1:Self Weight(Summary) \ 2:Surfacing(Summary)

- With the Self weight tab selected ensure that the total self-weight reaction (FZ) is **11934**kN.

- With the Surfacing tab selected ensure the total surfacing reaction (FZ) is **2160**kN.

Creating a Cable Tuning Analysis

During the design of cable structures it is usually desirable to determine a set of cable forces that will result in a certain deck profile. In this case the required forces will be calculated to maintain a horizontal deck and vertical tower under permanent loading (gravity plus surfacing). In LUSAS this can be done by setting up a 'Cable Tuning Analysis'.

The first stage of this process is to select the lines representing the cables.

- In the Treeview turn off the **Deformed mesh** layer.

- In the Treeview, right-click the geometric attribute **Cable** and click **Select assignments**.
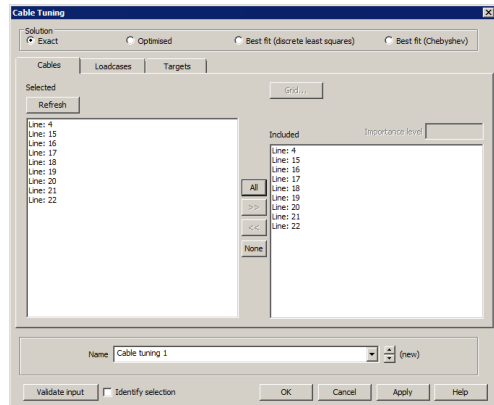
With the Cables tab active, the Selected panel contains a list of the lines which have just been selected to represent the cables. Optimised forces for all of these lines will be required.

- Click the **All** button in the middle of the dialog to copy these lines to the Included list on the right-hand side of the dialog.

- Switch to the **Loadcases** tab of the dialog.

The Loadcases tab allows for a combination of loads to be specified for optimising the cable forces. In this instance we are interested in the bridge profile under permanent loads.

- Select both **Self-Weight** and **Surfacing** loadcases and click ⟫ in the centre of the dialog to add both loadcases to the **Included** list.

- Switch to the **Targets** tab of the dialog.
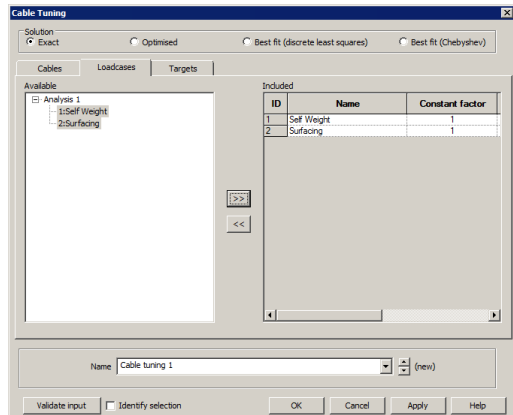
With the Targets tab active, the desired deflections and/or load effects from the cable tuning analysis can be specified. In this instance we want to keep the deck flat and the tower vertical, so we will specify that each of the points on the deck should have vertical deflection of zero and the point at the top of the tower will have a horizontal deflection of zero.

- Move the dialog to the side of the user interface and adjust the model view if necessary so that a clear image of the bridge is shown.

- In the view window, and holding the **P** key (to select only points), box select the whole deck with the exception of the two end points as shown below. The two end points are supported vertically so there is no need to fix their displacement.

Hold the 'P' key and select these points
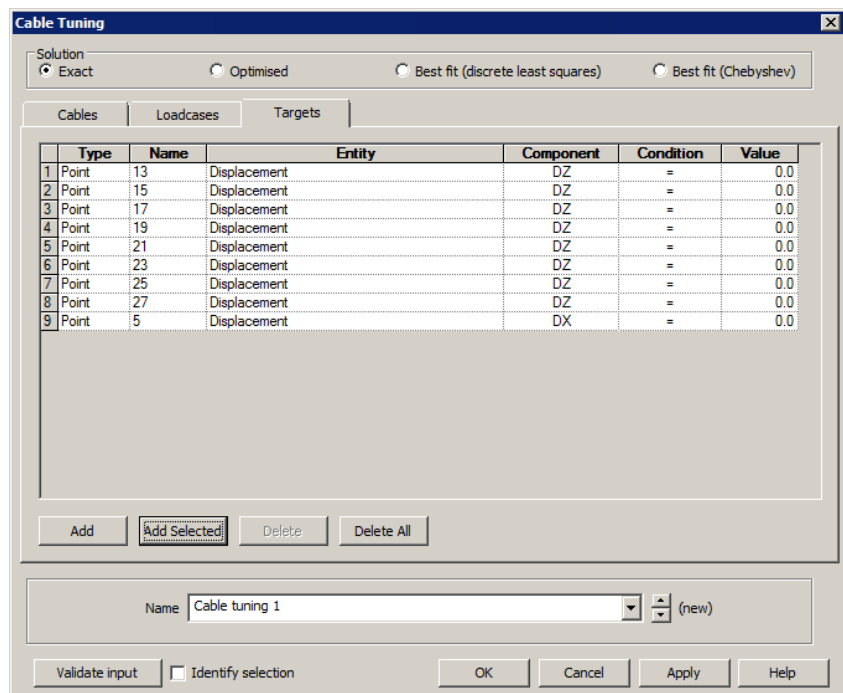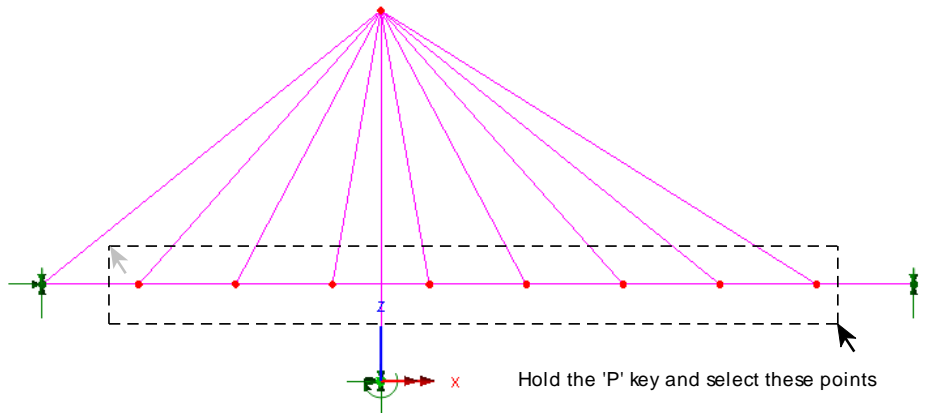
- In the cable tuning dialog, click **Add Selected**. In the window that pops up, select entity **Displacement** and set component **DZ = 0**. Click **OK** to add the selection of points to the targets list.

- In the view window, click on the top point of the tower, then click **Add Selected** on the cable tuning dialog again and specify entity **Displacement** and this time set component **DX = 0**

- For this analysis the number of selected lines representing cables is equal to the number of targets so an **Exact** solution can be calculated.

Note. If the number of selected lines representing cables exceeded the number of targets, there would be more than one possible solution to the cable tuning problem. It would therefore be possible to optimise the cable forces so that the targets were all met exactly, while some additional criteria was also optimised; for example the lowest total force in the cables. This would be set up by ticking the **Optimised** button at the top of the dialog and entering optimisation data in the additional tab which appeared.

Note. If the number of targets exceeded the number of selected lines representing cables, an exact solution would not be possible. In this case the **Best Fit** button should be ticked, and best fit data entered in the additional tab. It is usually possible to achieve a good match to the desired targets using the Best Fit method.

To check that the correct solution method has been selected:

- Click the **Validate Input** button in the bottom-left corner of the dialog. If the above instructions have been followed correctly a pop-up stating Input is valid will be shown. If not, check the three cable tuning tabs to determine the problem – there is probably a mismatch between the number of cables and the number of targets.

- Assuming that the inputs were successfully validated, click **OK** to conclude the Cable Tuning setup.

A Cable tuning analysis entry **Cable tuning 1** will be added to the �e Treeview along with a **Cables** folder containing a loadcase entry for each of the cables selected for tuning. A **Cable tuning 1 Results entry** is also added, as shown by a ⟁ symbol, representing a loadcase that will contain the combination of optimised cable forces and the loadcases specified during the cable tuning setup.

- Solve the model again by clicking the Solve ▤ button.
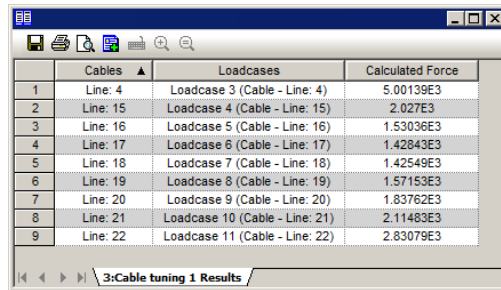
Viewing the results from the Cable Tuning Analysis

- Set active the **Cable Tuning 1 Results** entry / combination by right-clicking the Cable tuning entry ⟁ and selecting **Set active** to check that the targets have been met.

- In the ▣ Treeview, if necessary, turn on the **Deformed mesh** layer. This should show that the deck points are all vertically aligned with the original flat deck profile, and that the tower top point is horizontally aligned with the original vertical tower profile as shown on the image below.
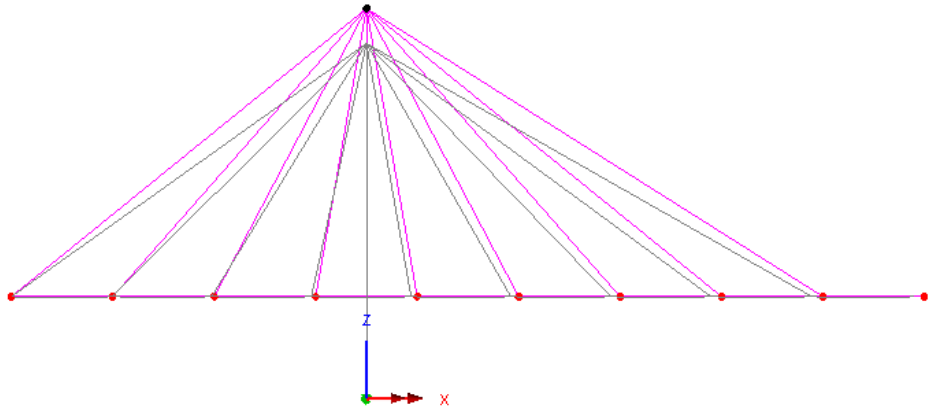
Note. By default the deformed mesh layer shows the largest displacement in the model as a specified on-screen distance of (typically) 6mm. As a result the very small change in tower height following the cable tuning is exaggerated. Specifying an exaggeration factor of 1.0 for the deformed mesh would show the exact position.

- In the ⌕ Treeview right-click the loadcase **Cable Tuning 1 Results** and select **Calculated Forces** to see the calculated load factors for individual cables.

	Cables ▲	Loadcases	Calculated Force
1	Line: 4	Loadcase 3 (Cable - Line: 4)	5.00139E3
2	Line: 15	Loadcase 4 (Cable - Line: 15)	2.027E3
3	Line: 16	Loadcase 5 (Cable - Line: 16)	1.53036E3
4	Line: 17	Loadcase 6 (Cable - Line: 17)	1.42843E3
5	Line: 18	Loadcase 7 (Cable - Line: 18)	1.42549E3
6	Line: 19	Loadcase 8 (Cable - Line: 19)	1.57153E3
7	Line: 20	Loadcase 9 (Cable - Line: 20)	1.83762E3
8	Line: 21	Loadcase 10 (Cable - Line: 21)	2.11483E3
9	Line: 22	Loadcase 11 (Cable - Line: 22)	2.83079E3

3:Cable tuning 1 Results

Note. Although the values shown in the Calculated Factors dialog cannot be edited, they can be saved for use in spreadsheets or added to a model report.

- With no features selected right-click in a blank part of the view window and select **Contours** to add the contours layer to the 🗗 Treeview.

The contour layer properties will be displayed.

- Select **Force/Moment – Bar / Cable** results of stress **Fx**.

- Select the **Contour display** tab and check the **Deform** option to plot the contours on the deformed shape.

- Click **OK**.

Scale: 1: 292.406
Zoom: 100.0
Eye: (0.0, -1.0, 0.0)
Cable tuning 1 Results
Maximum displacement 5.88431E-3 at node 48
Deformation exaggeration: 298.155

Contour entity: Force/Moment - Bar / Cable
Contour component: Fx (Units: kN)
Number of contours: 9
Contour interval: 402.106
Contour maximum 4.97313E3 at node 4 of element 43
Contour minimum 1.35418E3 at node 48 of element 47

Cable tuning 1 Results
Entity: Force/Moment - Bar / Cable
Component: Fx (Units: kN)

1.60843E3
2.01053E3
2.41264E3
2.81475E3
3.21685E3
3.61896E3
4.02106E3
4.42317E3
4.82528E3

Maximum 4.97313E3 at node 4 of element 43
Minimum 1.35418E3 at node 48 of element 47

z

x

This concludes the example.