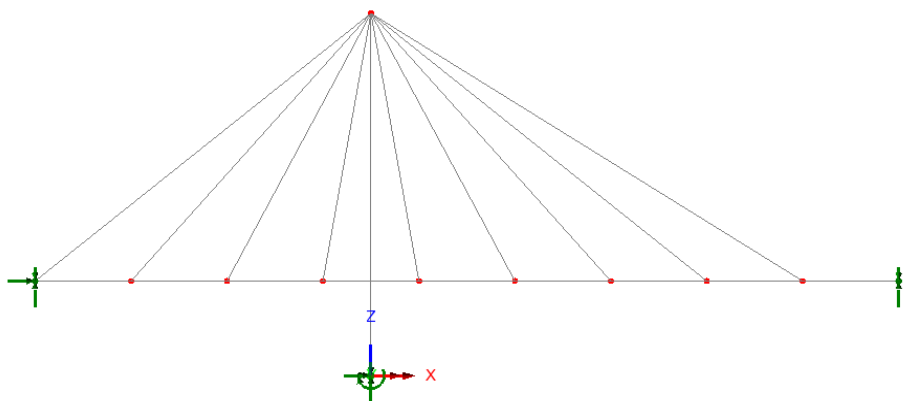


# Linear Cable Tuning Analysis of a Pedestrian Bridge

For LUSAS version:	20.0
For software product(s):	LUSAS Bridge / Bridge plus, or LUSAS Civil & Structural/ Civil & Structural plus
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

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



- cable\_tuning.lvb** creates the model geometry with all of the required attributes assigned. This model is used as a starting point for the example.

## 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 a New Model

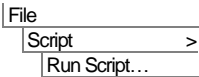
- Enter a file name of **cable\_tuning**

File \_\_\_\_\_  
New... \_\_\_\_\_

- 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 the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.



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



Save the model file.



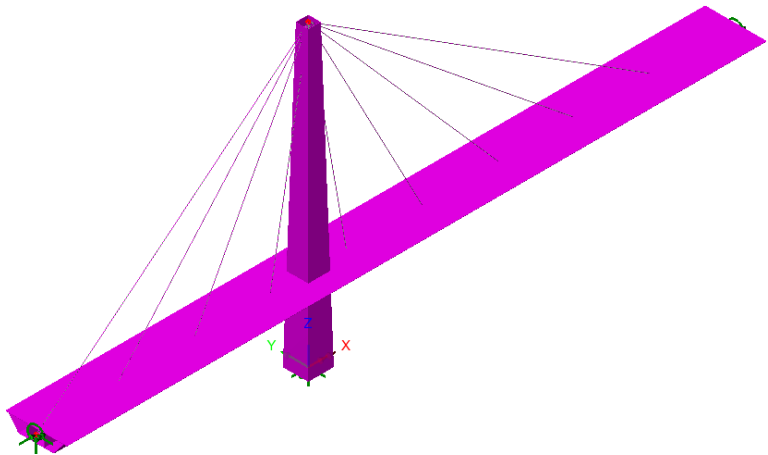
**Note.** The Undo button may be used to correct a mistake. The undo button allows any number of actions since the last save to be undone.

## Model Supplied



Press the Isometric button to see the bridge created

The model geometry and attributes are already defined in the created model.



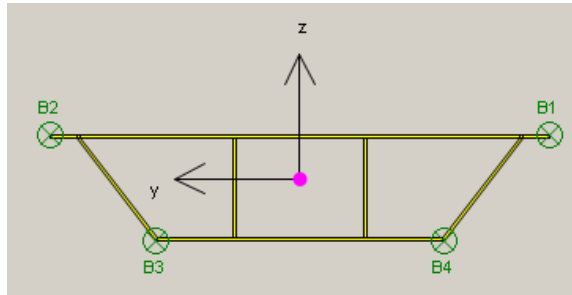
The bridge to be analysed consists of a 46m high tower supporting a deck of total length 108m.

## Linear Cable Tuning Analysis of a Pedestrian Bridge

---

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

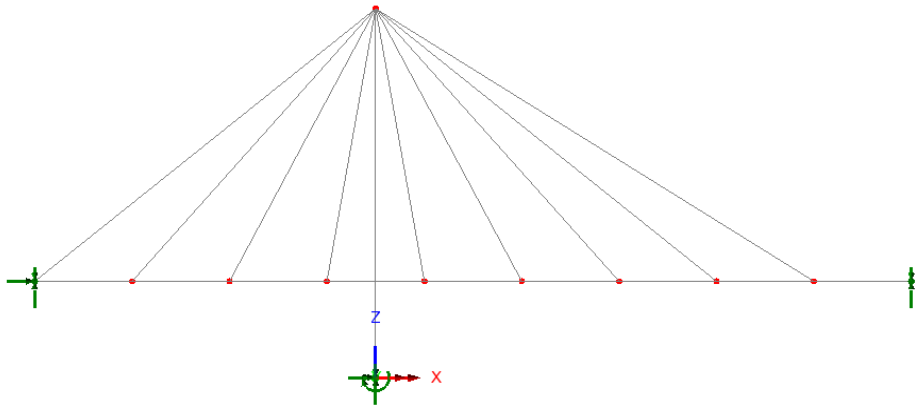
The lines representing the tower and the deck are meshed with thick 3D thick nonlinear beam elements (BMI21). The lines representing the cables are each meshed with a single 3D bar element (BRS2). Each cable is represented by a circular solid section of 50mm diameter.

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.
- Surfacing** of 20kN/m assigned to the deck members only.

### To start

- Turn 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

- 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**.

Utilities

Print Results  
Wizard...

	Component ▲	FX[kN]	FY[kN]	FZ[kN]	MX[kN.m]	MY[kN.m]	MZ[kN.m]	RSLT[kN]
1	Maximum	326.453	2.58431E-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	-94.2089E-27	-15.0168E3	0.0	1.18379E3
4	Node	1	4	4	1	1	1	4
5	Total	0.117382E-9	2.58431E-27	11.9342E3	N/A	N/A	N/A	N/A

Model info | 1:Self Weight(Summary) | 2:Surfacing(Summary) /



- With the Self weight tab selected ensure that the total self-weight reaction (FZ) is **11934kN**.
- With the Surfacing tab selected ensure the total surfacing reaction (FZ) is **2160kN**.

## 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'.

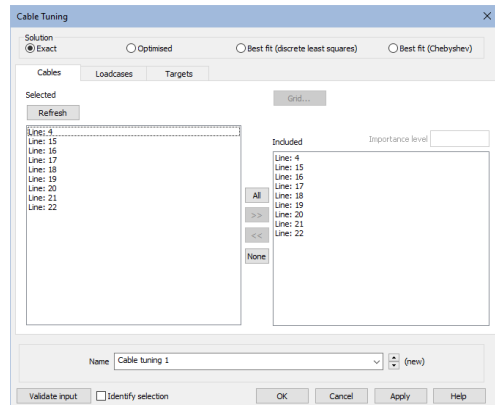
### Select lines representing cables

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


- In the  Treeview turn off the display of the **Deformed mesh** layer.
- In the  Treeview, right-click the geometric attribute **Cable** and click **Select assignments**.

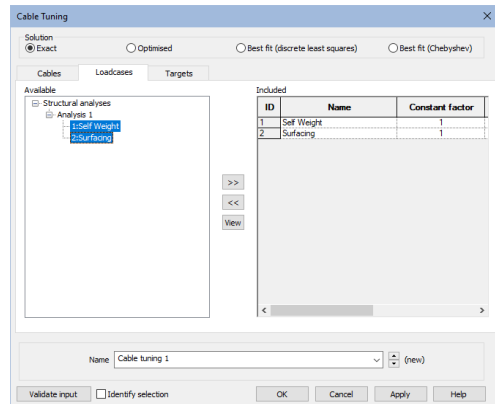
With the Cables tab active, the Selected panel contains a list of the lines which have just been selected to represent the cables. Optimised forces for all of these lines will be required.

- 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.

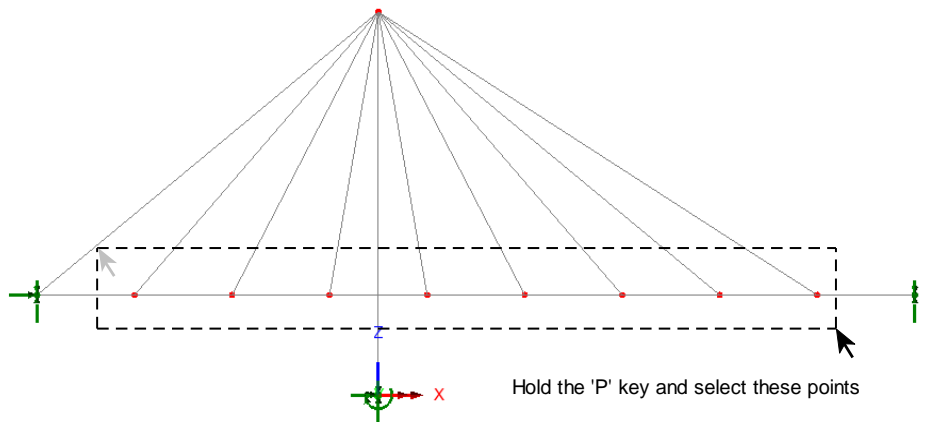
- Select both **Self-Weight** and **Surfacing** loadcases and click  in the centre of the dialog to add both loadcases to the **Included** list.
- Switch to the **Targets** tab of the dialog.



Analyses  
Linear Cable Tuning  
Analysis...

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.



- 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 only on the top point of the tower to select this in isolation from the previous selection, 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.

## Linear Cable Tuning Analysis of a Pedestrian Bridge

Cable Tuning

Solution  
 Exact     Optimised     Best fit (discrete least squares)     Best fit (Chebyshev)

Cables    Loadcases    **Targets**

	Type	Name	Entity	Component	Condition	Value
1	Point	13	Displacement	DZ	=	0.0
2	Point	15	Displacement	DZ	=	0.0
3	Point	17	Displacement	DZ	=	0.0
4	Point	19	Displacement	DZ	=	0.0
5	Point	21	Displacement	DZ	=	0.0
6	Point	23	Displacement	DZ	=	0.0
7	Point	25	Displacement	DZ	=	0.0
8	Point	27	Displacement	DZ	=	0.0
9	Point	5	Displacement	DX	=	0.0

Add    **Add Selected**    Delete    Delete All

Name: Cable tuning 1 (new)

Validate input     Identify selection    OK    Cancel    Apply    Help



**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 were 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  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.

## Solving the cable tuning analysis



- Solve the model again by clicking the Solve  button.
- Click **OK** to solve the cable tuning 1 analysis.



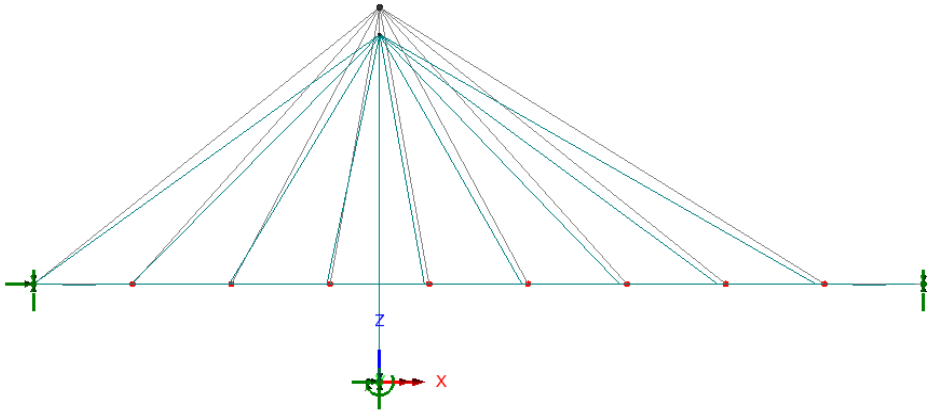
**Note.** In solving the cable tuning analysis, a unit tensile loadcase is automatically generated and assigned to any line selected to represent a cable stay. These loadcases are then factored by the cable tuning optimisation algorithm to try to achieve a solution.

When solved, cable tuning results are held in a separate cable tuning analysis entry in the  Treeview. This contains a folder with the individual solved and generated loadcases for each cable, and a results loadcase (which is effectively a combination) representing the achieved target profile. Note that the loadcase naming convention used is automated and cannot be edited.


## Viewing the results from the Cable Tuning Analysis

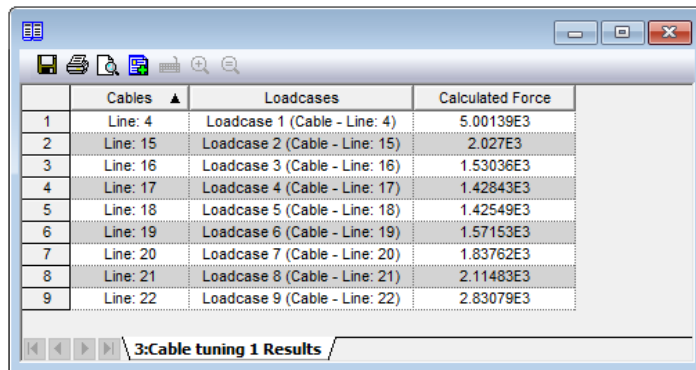
- Set active the **Cable Tuning 1 Results** entry / combination by right-clicking the Cable tuning entry  and selecting **Set active** to check that the targets have been met.
- In the  Treeview, if necessary, turn on the **Deformed mesh** layer. This should show that the deck points are all vertically 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.

## Linear Cable Tuning Analysis of a Pedestrian Bridge



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



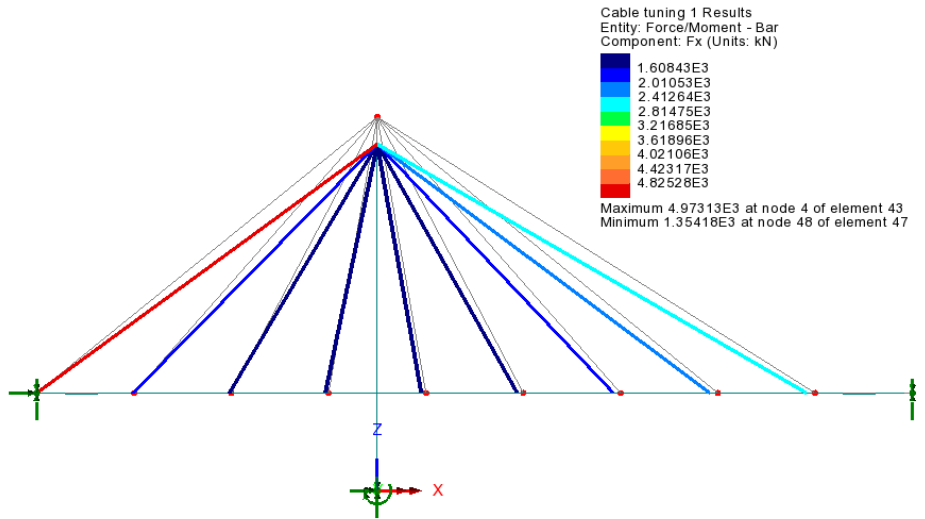
**Note.** Although the values shown in the Calculated Factors dialog cannot be edited, they can be saved for use in spreadsheet (by right-clicking within the grid) or added to a model report by clicking the 'Add to report' icon.

- 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** results of stress **Fx**.

- Select the **Appearance** tab and check the **Deform** option to plot the contours on the deformed shape.
- Click **OK**.



This concludes the example.

