Application Examples Manual (Bridge, Civil & Structural)

LUSAS Version 15.0 : Issue 1

LUSAS
Forge House, 66 High Street, Kingston upon Thames,
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999
Fax +44 (0)20 8549 9399
Email: info@lusas.com
http://www.lusas.com

Distributors Worldwide

Table of Contents

Introduction

Where do I start?

Start by reading this introduction in its entirety. It contains useful general information about the Modeller user interface and details of how the examples are formatted.

Software requirements

The examples are written for use with version 14.6 of LUSAS software products.

The LUSAS software product (and version of that product) and any product options that are required in order to run an example will be stated in a usage box like this:

For software product(s):	All (except LT versions)
With product option(s):	Nonlinear

Note that Composite examples can be run in any software product if a Composite product option has been purchased. Similarly, LUSAS Analyst or LUSAS Composite products can run bridge or civil examples if a Bridge or Civil product option was purchased. The LUSAS Academic software product will run any example.

What software do I have installed?

To find out which software product(s), which version of that product, and which software options are installed and licensed for your use run LUSAS and select **Help > About LUSAS Modeller** and press the **Key Information** button to display a dialog that lists the facilities and options supported by your software license.

About the examples

The examples are of varying complexity and cover different modelling and analysis procedures using LUSAS. The first example in this manual contains detailed information to guide you through the procedures involved in building a LUSAS model, running an analysis and viewing the results. This fully worked example details the contents of each dialog used and the necessary text entry and mouse clicks involved.

The remaining examples assume that you have completed the fully worked example and may not necessarily contain the same level of information. It will benefit you to work through as many examples as possible, even if they have no direct bearing on your immediate analysis interests.

Except where mentioned, all examples are written to allow modelling and analysis to be carried out with the Teaching and Training version of LUSAS which has restrictions on problem size. The teaching and training version limits are currently set as follows:

500	**100**	**250**	**1500**	**10**
Nodes	Points	Elements	Degrees of Freedom	Loadcases

Because of the modelling and analysis limits imposed by the Teaching and Training Versions some examples may contain coarse mesh arrangements that do not necessarily constitute good modelling practice. In these situations these examples should only be used to illustrate the LUSAS modelling methods and analysis procedures involved and should not necessarily be used as examples of how to analyse a particular type of structure in detail.

Other worked examples

A separate worked examples manual ***Application Examples Manual (Bridge, Civil & Structural)*** contains application-specific examples for civil, structural and bridge enegineering.

User Manuals for optional software products and options, such as IMD Plus, or Rail Track Analysis contain worked examples to illustrate the use of each.

Format of the examples

Headings

Each example contains some or all of the following main headings:

- ❑ **Description** contains a summary of the example, defining geometry, material properties, analysis requirements and results processing requirements.
    - **Objectives** states the aims of the analysis.
    - **Keywords** contains a list of keywords as an aid to selecting the correct examples to run.
    - **Associated Files** contains a list of files held in the \<**Lusas Installation Folder>\Examples\Modeller** directory that are associated with the example. These files are used to re-build models if you have problems, or

can be used to quickly build a model to skip to a certain part of an example, for instance, if you are only interested in the results processing stage.

- ❑ **Modelling** contains procedures for defining the features and attribute datasets to prepare the LUSAS model file. Multiple model files are created in some of the more complex examples and these therefore contain more than one ‘Modelling’ section.
- ❑ **Running the Analysis** contains details for running the analysis and assistance should the analysis fails for any reason.
- ❑ **Viewing the Results** contains procedures for results processing using various methods.

Menu commands

Menu entries to be selected are shown as follows:

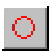
Geometry
Point >
Coordinates...

This implies that the **Geometry** menu should be selected from the menu bar, followed by **Point**, followed by the **Coordinates...** option.

Sometimes when a menu entry is referred to in the body text of an example it is written using a bold text style. For example the menu entry shown above would be written as **Geometry > Point > Coordinates...**

Toolbar buttons

For certain commands a toolbar button will also be shown to show the ‘short-cut’ option to the same command that could be used instead:



The toolbar button for the **Geometry > Point > Coordinates…** command is shown here.

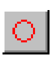
User actions

Actions that you need to carry out are generally bulleted (the exception is when they are immediately to the right of a menu command or a toolbar button) and any text that has to be entered is written in a bold text style as follows:

- Enter coordinates of (**10, 20**).

So the selection of a typical menu command (or the equivalent toolbar button) and the subsequent action to be carried out would appear as follows:

Geometry
Point >
Coordinates...



Enter coordinates of (**10, 20**).

Selecting the menu commands, or the toolbar button shown will cause a dialog box to be displayed in which the coordinates **10, 20** should be entered.

Filling-in dialogs

For filling-in dialogs a bold text style is used to indicate the text that must be entered. Items to be selected from drop-down lists or radio buttons that need to be picked also use a bold text style. For example:

- In the New Model dialog enter the filename as **frame_2d** and click the **OK** button to finish.

Grey-boxed text

Grey-boxed text indicates a procedure that only needs to be performed if problems occur with the modelling or analysis of the example. An example follows:

Rebuilding a model

File
New…

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 **example**

File
Script >
Run Script...

To recreate the model, select the file **example_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Visual Basic Scripts

Each example has an associated set of LUSAS-created VBS files that are supplied on the release kit. These are installed into the \<**Lusas Installation Folder>\Examples\Modeller** directory. If results processing and not the actual modelling of an example is only of interest to you the VBS files provided will allow you to quickly build a model for analysis. These scripts are also for use when it proves impossible for you to correct any errors made prior to running an analysis of an example. They allow you to re-create a model from scratch and run an analysis successfully. For more details refer to Creating a Model From The Supplied VBS files.

Modelling Units

At the beginning of each example the modelling units used will be stated something like this:

"Units used are N, m, kg, s, C throughout"

Model units are specified as part of the creation of a new model and are reported at all times in the status bar at the bottom of the Modeller window. Once set, for all dialogs with grid cells permitting dimensional input, the units expected are displayed as a tooltip when the cursor is hovered over the input cell.

Timescale Units

Timescale units are specified as part of the creation of a new model and can be changed on the Model Properties dialog. Choosing a timescale unit dictates how time-based values are displayed on dialogs during modelling, and how they are output when processing results

Icons Used

Throughout the examples, files, notes, tips and warnings icons may be found. They can be seen in the left margin.

Files. The diskette icon is used to indicate files used or created in an example.

Note. A note is information relevant to the current topic that should be drawn to your attention. Notes may cover useful additional information or bring out points requiring additional care in their execution.

Tip. A tip is a useful point or technique that will help to make the software easier to use.

Caution. A caution is used to alert you to something that could cause an inadvertent error to be made, or a potential corruption of data, or perhaps give you results that you would not otherwise expect. Cautions are rare, so take heed if they appear in the example.

Running LUSAS Modeller

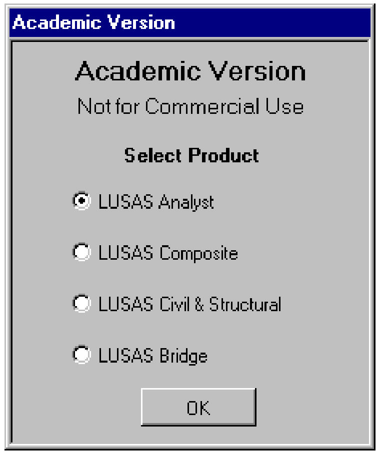
- Start LUSAS Modeller from the start programs menu. This differs according to the operating system in use, but typically is done by selecting:

  Start > All Programs > LUSAS 15.x for Windows > LUSAS Modeller

- The on-line help system will be displayed showing the latest changes to the software.
- Close the on-line Help system window.

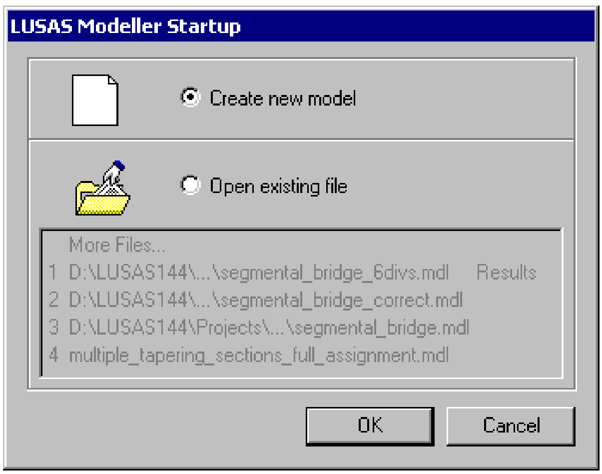
(LUSAS Academic version only)

- Select your chosen LUSAS product and click the **OK** button.



Creating a new model / Opening an existing model

When running LUSAS for the first time the LUSAS Modeller Startup dialog will be displayed.



This dialog allows either a new model to be created, or an existing model to be opened.

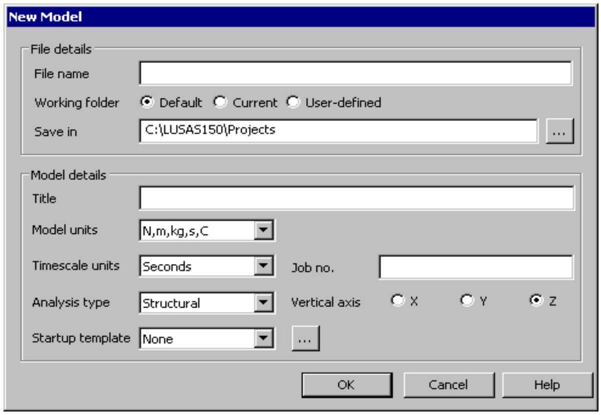
Note. When an existing model is loaded a check is made by LUSAS to see if a results file of the same name exists. If so, you have the option to load the results file on top of the opened model.

Note. When an existing model is loaded, that in a previous session caused a carsh to occur in LUSAS Modeller, options are provided to help recover your model data.

If creating a new model the New Model dialog will be displayed.

- Enter information and make selections for the new model and click the OK button.

Product specific menu entries for the selected software product in use e.g. **Bridge** or **Civil** will be added to the LUSAS Modeller menu bar.



Creating a Model from the Supplied VBS Files

If results processing and not the actual modelling of an example is only of interest, the VBS files provided will allow you to quickly build a model for analysis.

Proceed as follows to create the model from the relevant VBS file supplied:

File
New…

Start a new model file.

- Enter the file name as **example name** and click **OK**
- In general, ensure that the User interface selected is of the same type as the analysis to be carried out.

File
Script >
Run Script...

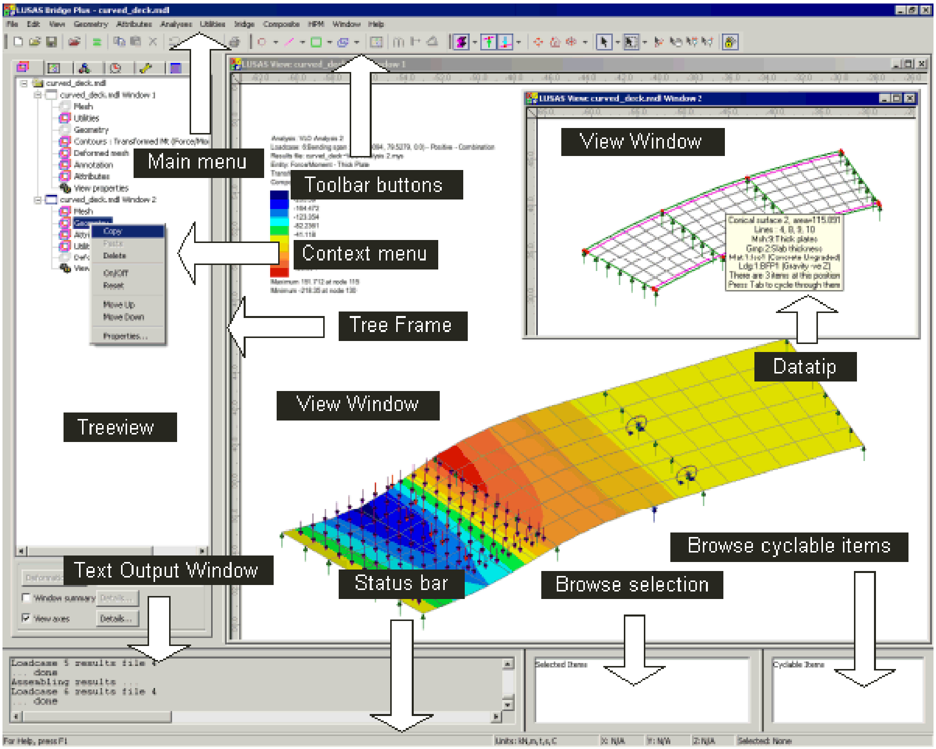
Select the file **example_name_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Note. VBS scripts that create models automatically perform a **File > Save** menu command as the end.

Some additional modelling may need to be carried out prior to an analysis being run. See individual examples for details.

The LUSAS Modeller Interface

The principal regions and components of theModeller user interface are shown on the following image and are described below.



Main Menu and Toolbars

The main menu and associated toolbars which contain toolbar buttons provide the means to define model related geometry and other data. On initial start-up of LUSAS Modeller the Main, Define and View Toolbars are displayed. All toolbars can be shown, hidden, or customised, using the **View> Toolbar** menu item.

View window

A **View Window** is where each model is developed and manipulated. Any number of view windows can be created with each displaying a unique view of a model. Pre-defined view layers inside each window hold model information that can be generally manipulated in the current view window as required.

Modelling in LUSAS

A LUSAS model is graphically represented by geometry features (points, lines, surfaces, volumes) which are assigned attributes (mesh, geometric, material, support, loading etc.). Geometry is defined using a whole range of tools under the **Geometry** menu, or the buttons on the Toolbars. Attributes are defined from the **Attributes** menu. Once defined attributes are listed in the Treeview.

Treeviews

Treeviews are used to organise various aspects of the model in graphical frames. There are a number of Treeviews showing Groups, Attributes, Analyses, Utilities, and Reports.

- The **Layers** Treeview controls the display of selective model data and results data to a view window.
- The **Groups** Treeview is used to store selected user-defined collections of objects (geometry, nodes or elements) under a collective name.
- The **Attributes** Treeview is where defined contains information relating to the model; the element type and discretisation on the geometry; section properties and thicknesses; the materials used; how the model is supported or restrained; and how the model is loaded.
- The **Analyses** Treeview shows all analyses defined; loadcases defined including analysis control loadcases defined during the modelling stage; results loadcases containing solutions for results processing; loadcase combinations and envelopes; and IMD and Fatigue calculations.
- The **Utilities** Treeview contains utility items used in the definition of model geometry or attributes, or to control an analysis, or to provide a particular functionality, such as to define a load combination or produce a report for example.
- The **Reports** Treeview contains a user-defined folder structure of reports, chapter and image entries to allow a report to be generated in a variety of formats.

Treeviews use drag and drop functionality. For example, an attribute in the Attributes Treeview can be assigned to model geometry by dragging the attribute onto an object (or objects) currently selected in the graphics window, or by copying and pasting an attribute onto another valid Treeview item as for instance, a group name, as held in the Groups Treeview.

Text Output Window

The Text Output Window displays messages and warnings during a modelling session. When an error message or warning relating to a particular object is written to the text output window, extra information is usually available by double-clicking on that line of text.

Browse Selection

This window is not displayed by default but can be viewed using the **View> Browse Selection** menu item. Once visible it will contain a list of all currently selected items which may then be individually deselected.

Browse Cyclable Items

This window is not displayed by default but can be viewed using the **View> Browse Cyclable Items** menu item. Once visible it will contain a list of all cyclable items. These may then be individually selected or deselected.

Browse Selection Memory

This window is not displayed by default but can be viewed using the **View> Selection Memory> Browse Memory Selection** menu item. Once visible it will contain a list of all items currently in Selection Memory which may then be individually deselected.

Status Bar

The Status Bar displays progress messages and help text during a modelling session, the model units, the current cursor position in model units (if the model is displayed in an orthogonal plane) and the item or number of items in the current selection. The **View> Status Bar** menu item may be used to hide or show the Status Bar.

Data Tips and Tooltips

Data tips and tool tips provide basic information about whatever is under the cursor. Datatips generally report information relating to the model geometry, attributes and assignments etc. Tool tips report on uses of toolbar buttons or expected input for grid cells etc.

Context Menus

Although commands can be accessed from the main menu, pressing the right-hand mouse button with an object selected usually displays a context menu which provides access to relevant operations. Treeview panels also have a context menu which provides access to additional functionality such as editing of data, control of visibility, visualisation of assignments, and selective control of results plotting on selected attributes.

Properties

General information relating to a model is presented in property dialogs. Properties may relate to the whole model or the current window, or a single geometric feature - in fact most objects have properties. To view an object's properties, select it, press the right mouse button, then choose **Properties** from the context menu

Getting Help

LUSAS contains a comprehensive Help system. The Help consists of the following:

- The **Help** button  on the Main toolbar is used to get context-sensitive help on the LUSAS interface. Click on the **Help** button, then click on any toolbar button or menu entry (even when greyed out).
- Most dialogs include a **Help** button which provides information on that dialog.
- Selecting **Help > Help Topics** from the main menu provides access to all the Help files.

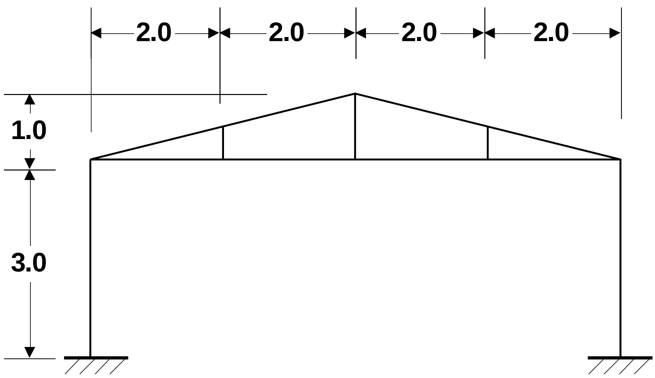
If the Help Contents, the Help Index and the Search facility are not shown when a help page is first displayed pressing the Show button will show these tabs in the HTML Help Window.

Linear Analysis of a 2D Frame

For software product(s):	LUSAS *Civil & Structural* LT and above, or LUSAS *Bridge* LT and above
With product option(s):	None.

Description

A simple 2D frame is to be analysed. The geometry of the frame is as shown.



All members are made of mild steel with a Young's modulus of 210E9 Pa, a Poisson's Ratio of 0.3 and a mass density of 7860 kg/m^3.

The structure is subjected to two loadcases; the self-weight of the structure, and a sway load at the top of the left-hand column.

The units of the analysis are N, m, kg, s, C throughout.

Objectives

The required output from the analysis consists of:

- ❑ A deformed shape plot showing displacements caused by the imposed loading.
- ❑ An axial force diagram showing stresses in the members.

Keywords

2D, Frame, Beam, Standard Sections, Fleshing, Copy, Mirror, Deformed Mesh, Axial Force Diagram, Shear Force Diagram, Bending Moment Diagram, Fibre Results, Report, Printing Results.

Associated Files

- **frame_2d_modelling.vbs** carries out the modelling of the example.

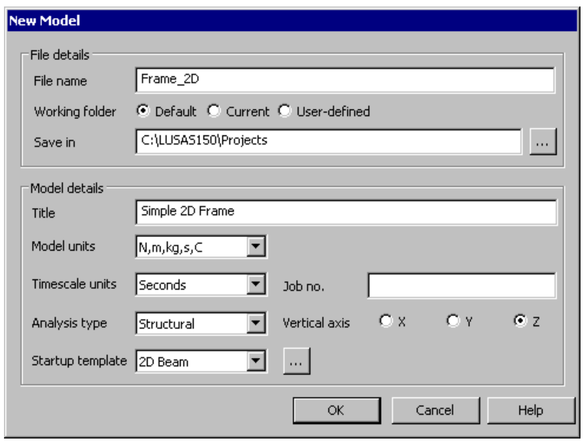
Modelling

Running LUSAS Modeller

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

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

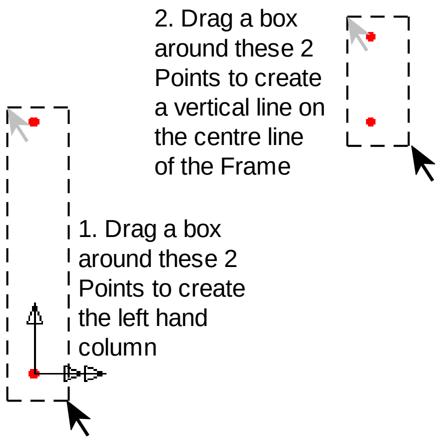


- Enter the file name as **Frame_2D**
- Use the **Default** working folder.
- Enter the title as **Simple 2D Frame**
- Set the model units to be **N,m,kg,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the startup template **2D Beam**. This populates the Attributes treeview with useful basic line mesh and support entries.

- Select the **Vertical Y axis** option.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily.

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



Feature Geometry

Geometry
Point >
Coordinates...

Enter coordinates of **(0, 0)**, **(0, 3)**, **(4, 3)** and **(4, 4)** to define the main setting-out points for one-half of the portal frame. Use the **Tab** key to move to the next entry field on the dialog. With all the coordinates entered click the **OK** button.

Note. Sets of coordinates must be separated by commas or spaces unless the 'Grid Style' method is chosen. The **Tab** key is used to create new entry fields. The Arrow keys are used to move between entries.

- Select the Points on the left-hand side of the model.

Geometry
Line >
Points…

Connect the selected Points with a Line representing the left-hand column of the portal frame.

- Select the pairs of Points on the right hand side of the model.

Geometry
Line >
Points…

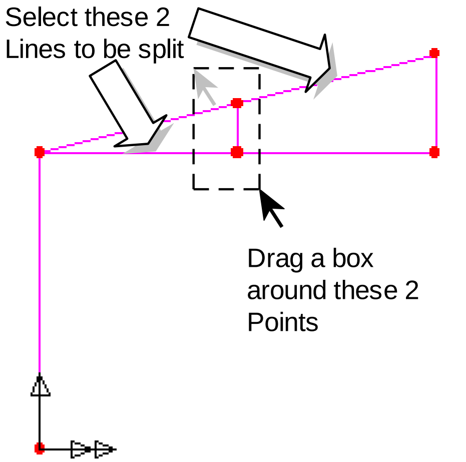
Connect the selected Points with a Line representing the right-hand vertical member on the frame centreline.

- Create the horizontal member by selecting the appropriate Points and define a Line in a similar manner.
- Select the Points at the top of the left hand column and the apex of the roof. (Hold down the **Shift** key to add the second Point to the initial selection).

Geometry
Line >
Points…

Connect the selected Points with a Line representing the sloping member of the frame.

To create the vertical roof member at quarter span the 2 Lines shown are split into equal divisions to create the points required and the new line is then defined.

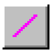


- Select the lower line (first) and the upper line (second) of those shown (Hold down the **Shift** key to add the second Line to the initial selection).

Geometry
Line >
By Splitting >
In Equal Divisions…

- Enter the number of divisions for both Lines as **2**
- Ensure that **Delete original lines after splitting** is set so that the original Lines are deleted and click the **OK** button.
- Drag a box around the 2 Points just created.

Geometry
Line >
Points...

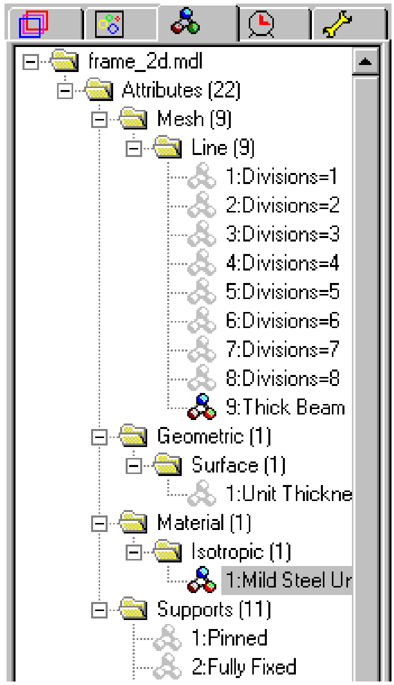
 Create the Line representing the internal member.

This completes the geometry definition for the half-frame.

Model attributes now need to be defined and assigned to the model.

Note. LUSAS Modeller works on a define and assign basis where attributes are first defined, then assigned to features of the model. This can be done either in a step-by-step fashion for each attribute or by defining all attributes first and then assigning all in turn to the model.

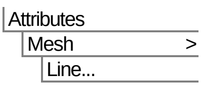
- ❑ Attributes, once defined, are displayed in the Treeview. Unassigned attributes appear 'greyed-out'.
- ❑ Attributes are then assigned to features by dragging an attribute dataset from the Treeview onto previously selected features.



Tip. Useful commands relating to the manipulation of attributes can be accessed by selecting an attribute in the Treeview, then clicking the right-hand mouse button to display a shortcut menu.

Meshing

The Line features are to be meshed using two-dimensional beam elements. A 2D thick beam element is automatically provided in the Treeview but to show the process involved another will be defined:



- Select **Thick beam, 2 dimensional, Linear** elements.
- Set the number of divisions to be **1**
- Enter the attribute name as **Thick Beam** then click **OK**
- Select the whole model. (**Ctrl** and **A** keys together).
- Drag and drop the mesh attribute **Thick Beam** from the Treeview onto the selected model Lines.

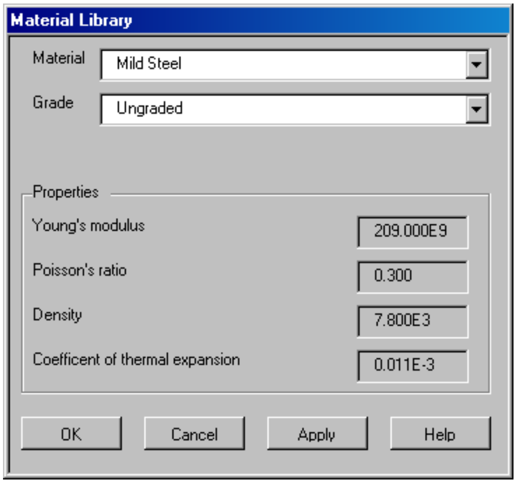
Note. When an assignment is made confirmation of the assignment is written to the Text Output Window.

Material Properties

Attributes
Material >
Material Library…

- Select material **Mild Steel** from the drop-down list, leave the grade set as Ungraded and click **OK** to add the material attribute to the Treeview.

- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material attribute **Iso1 (Mild Steel Ungraded)** from the Treeview onto the selected features.



Geometric Properties

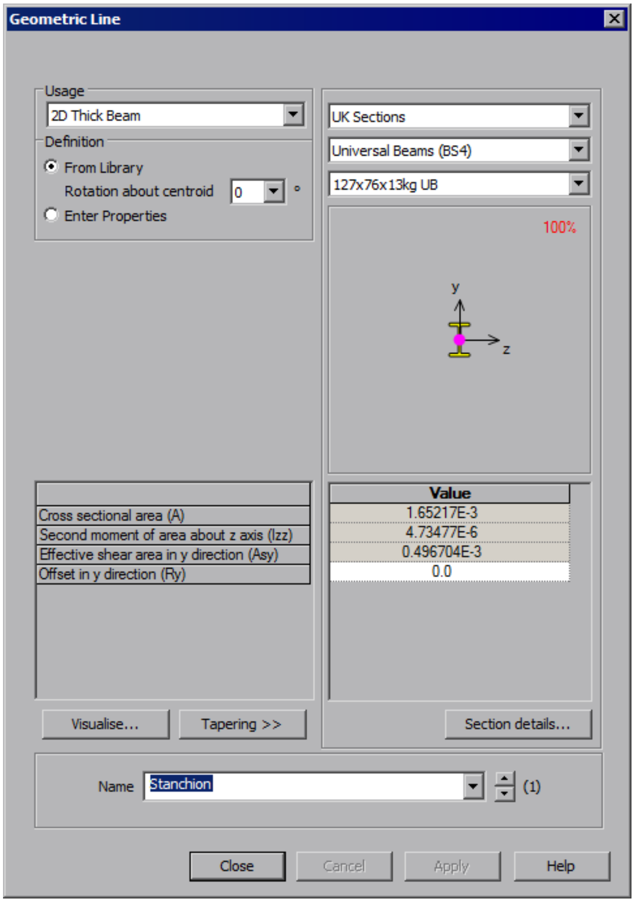
Attributes
Geometric >
Section Library …

The standard sections dialog will appear.

- In the Usage section select **2D Thick Beam**

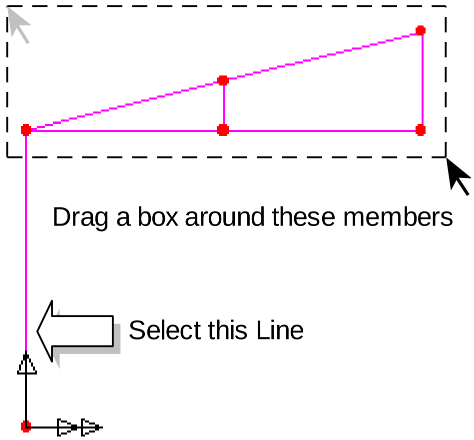
- Select the library of **UK Sections**

- Select section type **Universal Beams (BS4)**
- Select section name **127x76x13kg UB**.
- Enter the Attribute name as **stanchion**.
- Click the **Apply** button to add the Universal Beam attribute to the Treeview.
- Change the library to **EU Sections**
- Change the section type to **Equal** Angles **and** select the **70x70x6 EA** section name.
- Enter the Attribute name as **roof member**
- Click the **OK** button to add the Equal Angle attribute to the Treeview.



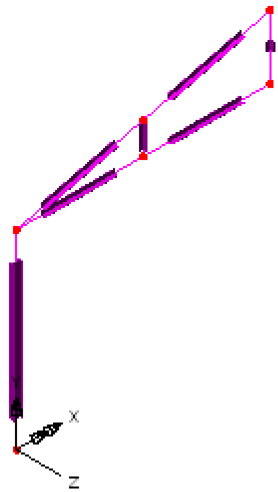
Note. The orientation of the section is appended to the section name which is un-editable. Attribute names (used for identification purposes) can be appended to the section name to make selection of similarly sized members easier.

Assigning Geometric Properties



- Drag a box around the Lines representing the roof members.
- Drag and drop the geometry attribute **70x70x6 EA major y roof member** from the Treeview onto the selected Lines.
- Select the Line representing the left-hand vertical member.
- Drag and drop the geometry attribute **127x76x13kg UB major y stanchion** from the Treeview onto the selected Line.

Geometric assignments are visualised by default.



- Click in a blank part of the Graphics Window to deselect the members.

 Select the isometric button to see the geometric visualisation on the members.

 The Zoom in button can be used to check the orientations.

 Select the Home button to return the model to the default view.

 Select the fleshing on/off button to turn-off the geometric visualisation.

Supports

LUSAS provides the more common types of support by default. These can be seen in the Treeview. The structure will be supported at the end of the stanchion which is in contact with the ground with a fully fixed support condition.

Assigning the Supports

- Select the Point at the bottom of the left-hand vertical member.

- Drag and drop the support attribute **Fully Fixed** from the Treeview onto the selected feature.
- Ensure the **Assign to points** and **All loadcases** options are selected and click **OK**

Mirroring the Model

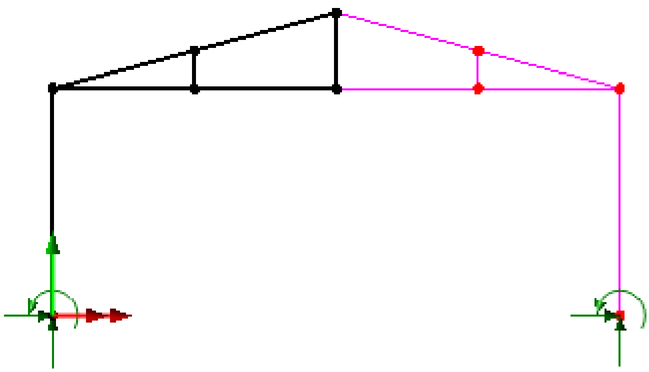
One half of the 2D frame has now been generated. This half can now be copied about a mirror plane to create the complete frame.

- Select the whole model (**Ctrl** and **A** keys together).

Geometry
Line
Copy…

Select **Mirror,** select **Parallel to Y axis** and enter **4** in X, and **0** in Z in the top cells and **4** in X, and **3** in Z in the lower cells to define the mirror plane.

- Click the **OK** button to create the full model.



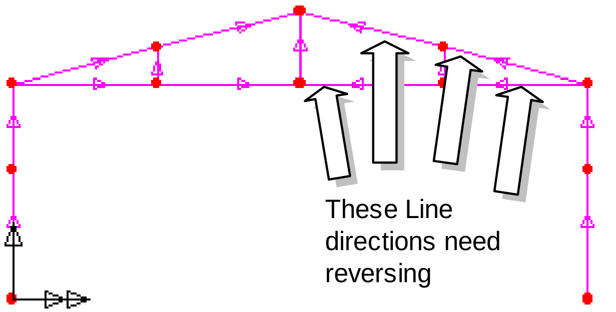
Note. Features that are copied retain their attributes. This means that there is no need to define the mesh, material or geometric properties for the newly created Lines as they will already have the attributes from the original half of the model assigned.

Note. In mirroring the features the Line directions will have been reversed. The orientation of Lines is important as it controls the local beam axes directions which will define the sign convention used to present the results.

- In the Layers Treeview double-click on **Geometry** to display the geometry layer properties. Select the **Line directions** option and click **OK**

The Lines orientations are shown.

- Select the 4 Lines to be reversed. (Hold down the **Shift** key to add lines to the initial selection).



Geometry
Line >
Reverse

The selected Line directions will be reversed.

- Once the Line directions are corrected, de-select the **Line directions** option in the geometry layer properties and click the **OK** button.

Loading

Two loadcases will be considered; self weight, and a concentrated sway load acting at the top of the left-hand vertical member.

Loadcase 1 - Self Weight

Loadcase 1 will represent the self-weight of the structure. This is modelled using a constant body force which is an acceleration loading simulating the force of gravity acting upon the structure.

Attributes
Loading…

- Select the **Body Force** option and click **Next**
- Enter a linear acceleration in the **Y** direction of **-9.81**
- Enter the attribute name as **Self Weight** and click **Finish** to add the attribute to the Treeview.
- Select the whole model (**Ctrl** and **A** keys together) and drag and drop the loading attribute **Self Weight** from the Treeview onto the selected features.
- Ensure that the **Assign to lines** and **Single loadcase** options are selected and the loading is applied to **Analysis 1** and **Loadcase 1**. Click the **OK** button and the loading on the frame will be displayed.

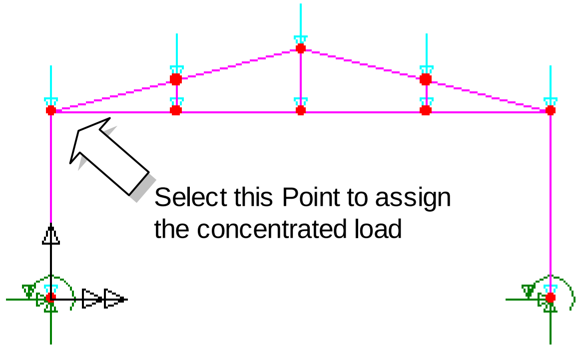
Loadcase 2 - Sway Load

Loadcase 2 is a sway load acting at the top of the left-hand vertical member.

Attributes
Loading…

- Ensure the **Concentrated** option is selected and click **Next**
- Enter a concentrated load in the **X** direction of **500**

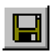
- Enter the attribute name as **Sway Load** and click **Finish**
- Select the Point at the top of the left-hand column.



- Drag and drop the loading attribute **Sway Load** from the Treeview onto the selected Point.
- Ensure the **Assign to points** and **Single loadcase** options are selected, and that the selected analysis is **Analysis 1**. Change the loadcase name to **Loadcase 2**.
- Click the **OK** button to assign the loading as the active loadcase with a factor of **1**

Saving the Model.

File
Save

 Save the model file.

Running the Analysis

With the model loaded:

 Open the Solve Now dialog. Ensure **Analysis 1** is selected and click the **OK** button to perform 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 Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

- ❑ **frame_2d.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

- **frame_2d.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **frame_2d_modelling.vbs** carries out the modelling of the example.

File
New…

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 **frame_2D**

File
Script >
Run Script...

- To recreate the model, select the file **frame_2d_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

Selecting the Results to be Viewed

Analysis loadcase results are present in the Treeview. The loadcase results for the first results loadcase (Loadcase 1) is set active by default. This is signified by the active results bitmap in the Treeview.

Deformed Mesh Plot

A deformed mesh plot is normally displayed initially to highlight any obvious errors with an analysis before progressing to more detailed results processing. The deformed shape will usually show up errors in loading or supports and may also indicate incorrect material property assignments (e.g. where the results show excessive displacements).

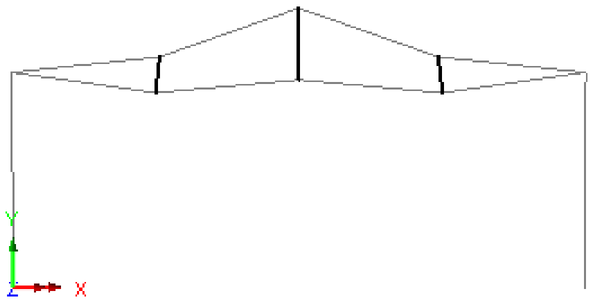
For clarity, the geometry will be removed from the display to leave only the undeformed mesh displayed.

- In the  Treeview turn off the display of the **Geometry** and **Attributes** layers by right-clicking on each layer name and selecting the **On/Off** menu item.
- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select the **Deformed mesh** option. This will add the deformed mesh layer to the  Treeview. Accept the default properties by selecting **OK**

Note. By default the maximum displacement is scaled to the magnitude defined on the deformed mesh dialog so the deformed shape can be easily visualised. The exaggeration factor used to multiply the displacements is displayed on the deformed mesh properties dialog and can be defined if required.

The deformed mesh plot for loadcase 1 (the default active loadcase) will be displayed on top of the mesh layer.

- Turn off the **Mesh** layer from the  Treeview to leave only the deformed mesh displayed as shown.

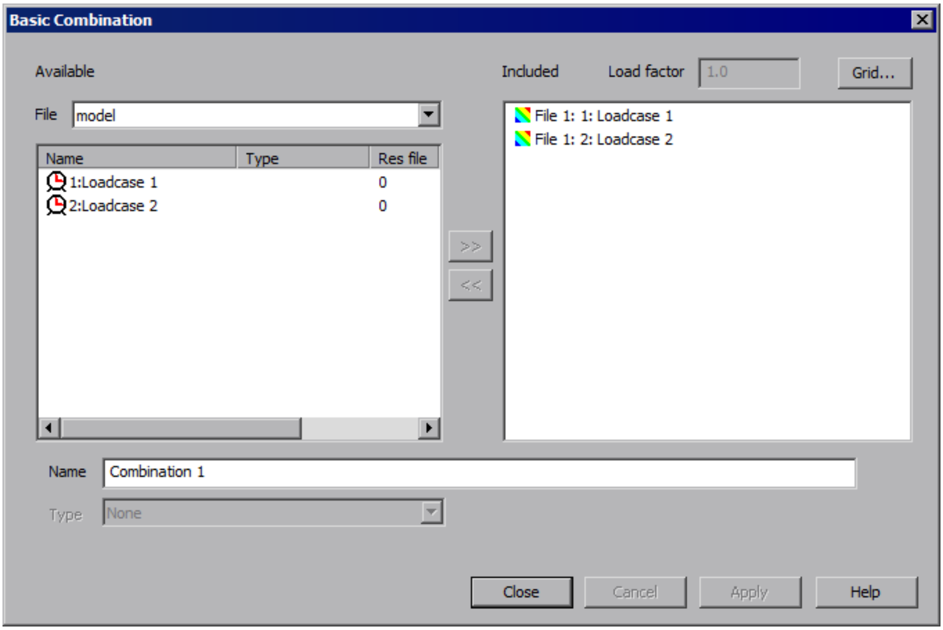


Defining a Combination

Combinations can be created to view the combined effects of multiple loadcases on the structure.

Analyses
Basic Combination

The combination properties dialog will appear.



- Ensure **Analysis 1** is selected from the drop down list.

Both results loadcases should be included in the combination panel.

- Select **Loadcase 1**, hold the shift key down and select **Loadcase 2**. Click the  button to add the loadcases to the load combination.

- Click the **OK** button to finish.

Note. The load factor may be modified by selecting the included loadcase in the right hand list and updating the factor or by updating the factor in the grid, which is displayed by selecting the Grid button.

Note. To obtain the correct effect from the combined loads in this example, the Combination should only include one occurrence of each loadcase.

Viewing a Combination

- In the Treeview right-click on **Post Processing** > **Combination 1** and select the **Set Active** option.

The deformed mesh plot will show the effect of the combined loading on the structure.

Marking Peak Values

- 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.

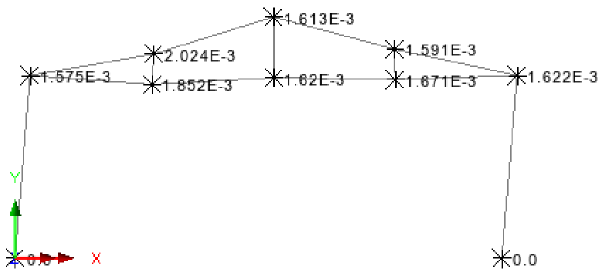
The values layer properties will be displayed.

- Select **Displacement** results from the Entity drop down list and select the resultant displacement **RSLT** from the Component drop down list**.**

- From the **Values Display** tab set **100**% of **Maxima** values to be displayed.

- Set the number of significant figures to **4**

- Click the **OK** button to display the deformed mesh plot showing the displacement at each node.



Note. Because this example is written for LUSAS LT versions that only allow one model or results viewing window to be used the Values layer is best turned off prior to adding further layers. In Standard and Plus versions of LUSAS multiple windows each having different layers can be defined.

- Turn off the **Values** layer in the Treeview.

Axial Force Diagram

- With no features selected click the right-hand mouse button in a blank part of the graphics window and select **Diagrams** to add the diagrams layer to the Treeview.

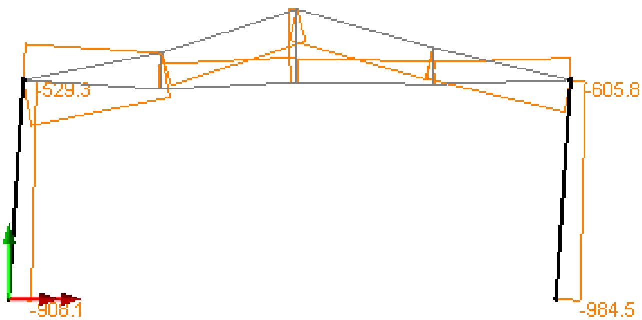
The diagram properties will be displayed.

- Select **Force/Moment – Thick 2D Beam** results of axial force **Fx** in the members

- Select the **Diagram Display** tab and select the **Label values** button.
- Select the **Label only if selected** option.
- Change the number of significant figures to **4**
- Click the **OK** button to finish.

The order of the layers in the Treeview governs the order that the layers are displayed. To see the deformed mesh on top of the diagram plot the deformed mesh layer needs to be moved down the Treeview to a position after the diagram layer.

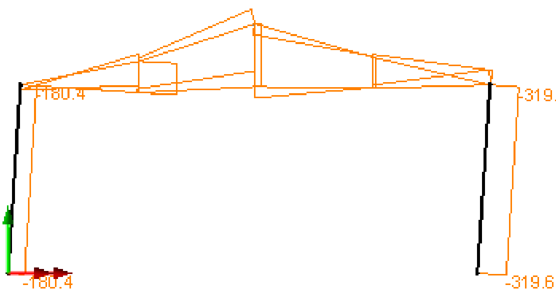
- In the Treeview select the **Deformed Mesh** layer, click the right-hand mouse button and select the **Move Down** option.
- Now add labels to the columns (vertical members). Select the left hand column, hold the **Shift** key and select the right hand column.



An axial force diagram for each member will be displayed with values of Fx displayed on the selected columns.

Shear Force Diagram

- In the Treeview double-click on the **Diagrams** layer. The diagram properties will be displayed.
- Select **Force/Moment – Thick 2D Beam** results of shear force **Fy** in the members.

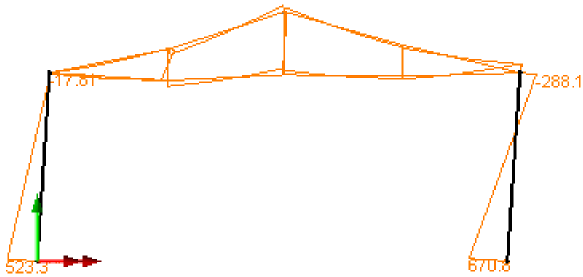


- Click the **OK** button to display a shear force diagram of stresses in each member with labels on the selected vertical members.

Bending Moment Diagram

- In the Treeview double-click on the **Diagrams** layer. The diagram properties will be displayed.

- Select **Force/Moment – Thick 2D Beam** results of bending moments in the members **Mz**



- Click the **OK** button to display a bending moment diagram for each member with labels on the selected vertical members.

Viewing Results on Fleshed Members

As an alternative to plotting diagram results stresses in beams can be plotted on fleshed members.

- Turn off the **Diagrams** layer in the Treeview.

- Click in a blank part of the Graphics Window to deselect the members.

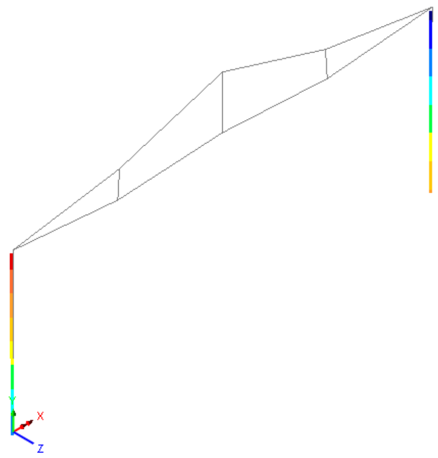
Select the isometric button.

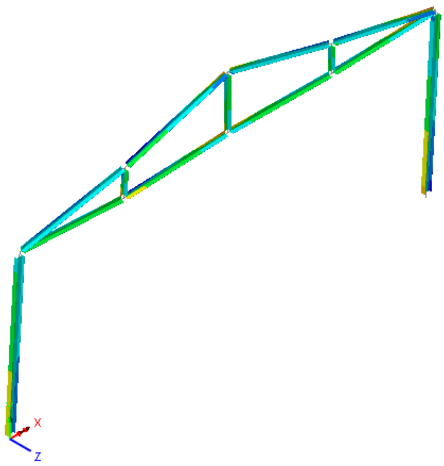
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select **Contours** to add the contours layer to the Treeview.

The contour properties will be displayed.

- Select **Stress – Thick 2D Beam** results of axial and moment results in the members **Sx(Fx Mz)**

Initially contours will be drawn at the current active fibre location on both stanchions as seen on the left-hand image below. Fibre locations will be explained shortly.





Select the fleshing on/off button to visualise the stress results on the fleshed cross-section as seen on the right-hand image above.

Viewing Force/Moment Results at Fibre Locations

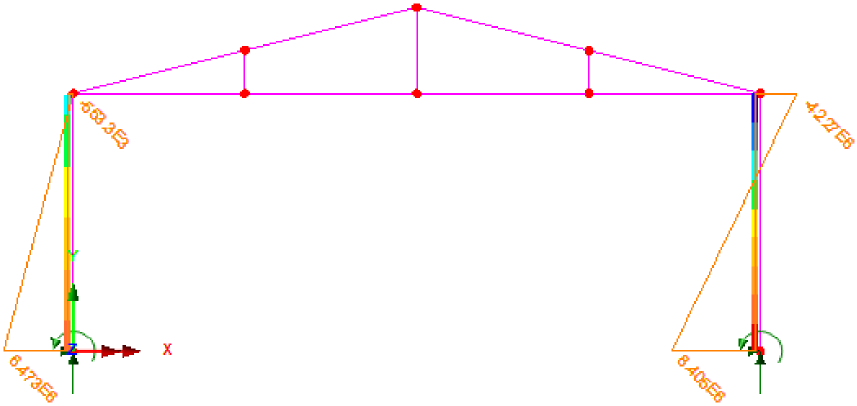
As an alternative to plotting force and moment diagrams along nodal lines they can be also plotted at pre-defined fibre locations on members and be superimposed on other results plots. Standard steel sections have various extreme fibre locations pre-defined. These fibre locations can be seen by expanding the Geometric Line entry in the in the Treeview and can be visualised by double-clicking the Geometric Line attribute name and selecting the **Visualise** button.

Diagram Stress Results at Fibre Locations

- With no features selected right-click in a blank part of the graphics window and select **Diagrams** to add the diagrams layer to the Treeview.

The diagram properties will be displayed.

- Select **Stress - Thick 2D Beam** results of stress **Sx(Fx, Mz)** in the beam.
- Select the **Diagram Display** tab and select the **Label values** button. Change the angle of the text to **-45** and set the number of significant figures to **4**.
- Select the **Scale** tab and set a local scale of magnitude **12**mm.
- Click **OK** to display the stress diagram at Fibre location I1.



Save the Model

File
Save

Save the model file.

Note. When the model file is saved after viewing results, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

Creating a Report

Selective model and results details can be printed to a variety of output formats by using the LUSAS report facility. Reports can include screen captures and imported images or additional text. A report template is used to specify items to be included in a report. This template is saved in the Treeview when a model is saved. A model can have any number of report templates.

Utilities
Report…

- On the Report dialog enter a report title of **2D Frame**
- Leave the modelling units and the number of significant figures set to default values.
- Enter a report name of **Report 1** and click **OK**.

A report entry will be created in the Treeview.

- In the Treeview right-click on **Report 1** and select **Add Chapter**

- With the **Model properties** tab selected select the **All** checkbox.
- Select the **Loadcase/Basic combination results** tab and click the **Add** button
- On the Results Entity dialog select **Displacement** from the entity pull-down menu and click **OK**
- Select the **Add** button again, select **Force/Moment – Thick 2D Beam** from the pull-down menu and click **OK**
- Select the **User content** tab and select the **Capture** button, and save the current screen image as a **JPG** file of the name **frame_2d.jpg**
- Click **OK** to finish

Note. The default order of the report items in the Treeview (and hence the order of any items on any exported report) can be changed by dragging and dropping the report sections as required. Right-clicking on an item gives access to additional options, such as modifying existing data or adding more chapters.

Viewing and Exporting a Report

- In the Treeview right-click on **Report 1** and select **View Report**

After a short pause whilst the report is created, the first page of the report will be displayed. The contents of the report may be browsed using the 'page forward' and 'page backward' buttons at the top of the dialog.

Select the **Export Report** button.

- On the Export dialog and from the **Format:** drop-down menu select **Adobe Acrobat (PDF)**
- From the **Destination:** drop-down menu select **Application** to open the file with Adobe Acrobat and click **OK**
- On the Export Options dialog select a page range of **All** and click **OK**

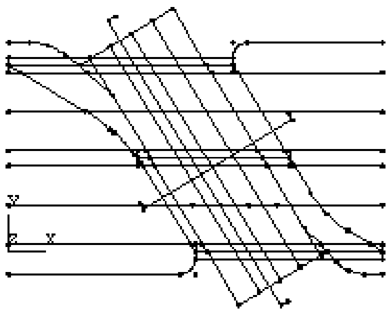
This completes the example.

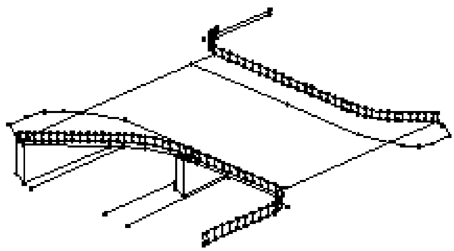
Importing DXF Data

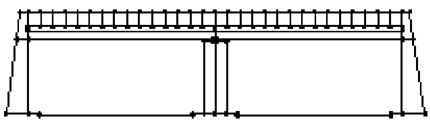
For software product(s):	All (except LT versions)
With product option(s):	None.
Note: This example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example shows the steps involved in importing DXF drawing data for use in LUSAS.







Features imported from DXF file

The initial geometry shown is equivalent to that imported from a typical CAD general arrangement drawing.

Modelling Objectives

The operations in the creation of the model are as follows:

- ❑ Import initial Point and Line feature information from CAD drawing.

- ❑ Rationalise drawing to delete unwanted information.
- ❑ Re-scale imported features to establish analysis units of length (m).

Associated Files

- ❑ **slab.dxf** contains the drawing information available in DXF format.

Modelling

Running LUSAS Modeller

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

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 the file name as **dxf_slab**
- Use the default working folder.
- Enter the title as **Imported DXF file of slab**
- Set model units of **kN,m,t,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Set the Startup template as **None**
- Ensure a vertical **Z** axis is set and click **OK**.

Note. It is useful to save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily by a new user.

Importing DXF Model Data

With DXF data there may be situations where the DXF drawing units need to be converted from scaled ‘paper’ dimensions into full size model dimensions as with this example where modelling units of metres are required. This conversion of units can be done in two ways:

1. By setting a pre-translation scale before importing the data
2. By transforming the data using a scale factor after importing the data

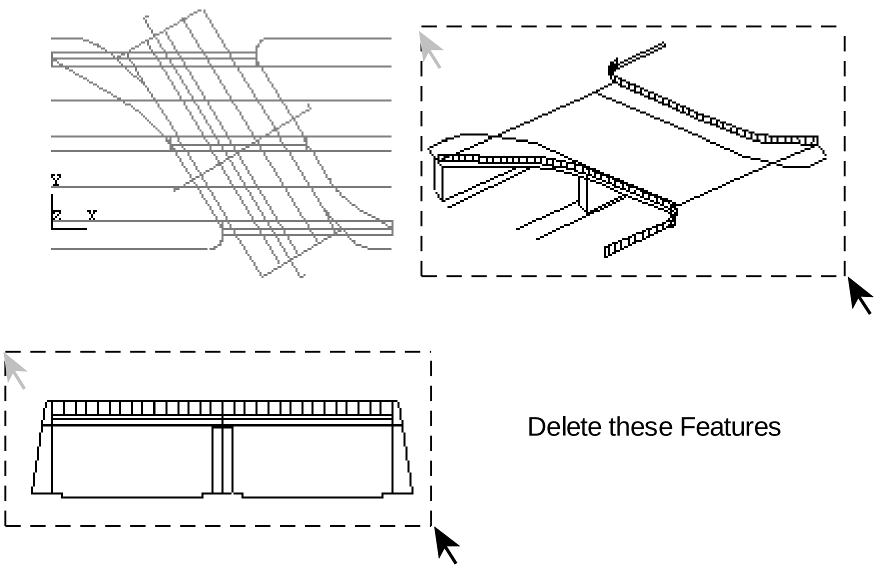
In both cases unless the paper dimensions are already known some measurement of data will be involved in LUSAS prior to either re-importing or scaling the data. By using the Advanced import options entities can be selected for inclusion/exclusion from import and LUSAS modelling-related features can be automatically tidied or merged together. This example covers the use of scaling the data after import. Notes describing the use of a pre-translation scale are at the end of the example.

Importing the DXF Data for Scaling

File
Import…

Select the DXF file **slab.dxf** from the \<**Lusas Installation Folder**>**\Examples\Modeller** directory and click the **Import** button.

Feature Geometry



The DXF features are imported in scaled units defined by the drawing scale. These features will need to be scaled to suit the units of the analysis. Only the features defining the plan view are to be used in creating the analysis model. All other features can be deleted.

- Drag a box around the Point and Line features on the other views of the bridge. Use the **Shift** key to add additional features to the initial selection.

Edit
Delete

Delete the selected features, confirming that the Lines and Points are to be deleted.

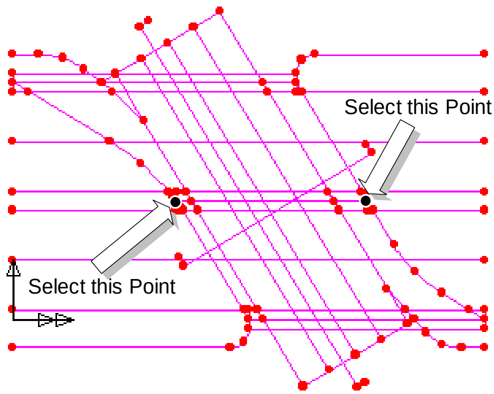
The remaining features will form the basis of the slab definition, but some further manipulation is required before a finite element model can be created.

Converting Units

The imported DXF drawing units are millimetres plotted at 1:100 scale, so these will need to be scaled by 100 to get the actual full-size dimensions of the slab in millimetres. These dimensions in millimetres will then need to be scaled by 0.001 to convert them into metres. To save scaling the data twice, one scale factor of (100x0.001)=0.1 can be used to convert the feature data into metres. Before scaling the model data it is worth checking the dimensions of the slab.

To check the existing model dimensions, 2 sample Points on the centre line of the mid-span support can be selected in turn and their coordinates obtained. If necessary zoom in on the model to make Point selection easier.

Use the Zoom in button to enlarge the view to include both ends of the centreline.



Return to normal cursor selection mode.

- Select the Point on the left-hand edge of the slab deck as shown in the diagram
- With the **Shift** key held down, select the Point shown on the right-hand edge of the deck.

Geometry
Point >
Distance between points

A value between the Points selected of 202.075 will be displayed in the message window.

Resize the model to fit the graphics window.

Scaling the model units

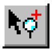
- Drag a box around the slab to select all the features. (Or press the **Ctrl** and **A** keys together).

Geometry
Point >
Move…

 Select the **Scale** option and enter a **Scale** factor of **0.1** in the **X**, **Y** and **Z** fields. Leave the origin set as 0,0,0 and click **OK**

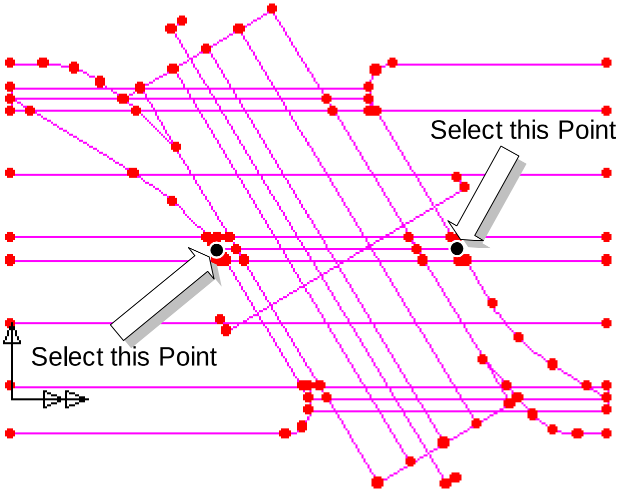
The slab dimensions will be scaled accordingly.

To check the new model dimensions, The same sample Points on the centre line of the mid-span support are to be selected and their new coordinates obtained.

 Use the Zoom in button to enlarge the view to include both ends of the centreline as before.

 Return to normal cursor selection mode.

- Select the Point on the left-hand edge of the slab deck as shown in the diagram



- With the **Shift** key held down, select the Point shown on the right-hand edge of the deck.

Geometry
Point >
Distance between points

A value between the Points selected of 20.2075 will be displayed in the message window.

 Resize the model to fit the graphics window.

Save the model

File
Save

 Save the model.

This completes the conversion and preparation of the DXF data model for use in LUSAS. From the imported data, Surfaces would be defined, a mesh assigned to the model, and geometric properties, supports and loading added.

This completes the example.

Notes on importing the DXF data using a pre-translation scale

As an alternative to scaling data a pre-translation scale could be used. After importing the DXF data and checking the model dimensions in LUSAS as previously described (if required) the DXF file can be re-imported using a pre-translation scale to convert it into the correct dimensions for use.

File
Import…

Select the DXF file **slab.dxf** from the **\<Lusas Installation Folder>\Examples\Modeller** directory and click the **Import** button.

- Click the **Advanced** button
- On the Import Parameters dialog enter a pre-translation scale of **0.1**
- Click the **OK** button
- Select **Import**

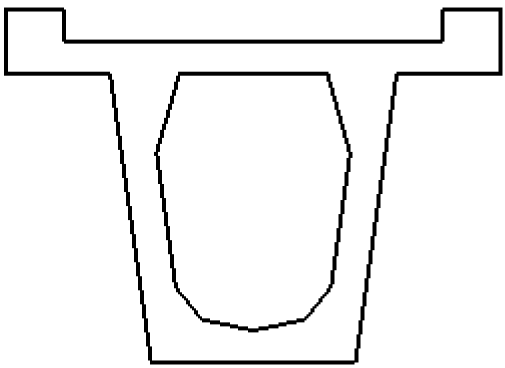
Arbitrary Section Property Calculation and Use

For software product(s):	All (except LT versions)
With product option(s):	None.

Description

The section properties of an arbitrary shaped box section are to be computed and saved in a user library. Section geometry is supplied as a DXF file.

Units of N, mm, t, s, C ar**e** used.



Objectives

The required output from the analysis consists of:

- Section Properties of a box section

Keywords

Section Properties, Arbitrary Section, Holes, Local Library, Server Library

Associated Files

- **box_section.dxf** DXF file containing geometry of section.

Modelling

Running LUSAS Modeller

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

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 the file name as **box_section**
- Use the **Default** working folder.
- Enter the title as **Box Section**
- Set model units of **N,mm,t,s,C**
- Leave the Timescale units as **Seconds**
- Ensure an Analysis type of **Structural** is set.
- Set the Startup template as **None**
- Ensure the vertical axis is set to **Z**
- Click the **OK** button.

Discussion

The arbitrary section property calculator within LUSAS Modeller computes the section properties of any open or closed section. Cross-sections are created either as a single regular or irregular surface, or as a group of surfaces (including holes). Fibre definitions (positions on the cross-section at which stresses or values can be plotted when viewing results) are also created for each drawn section at the time of section property calculation.

Note. 2D cross-sections for section property calculation must be defined in the XY plane.

Feature Geometry

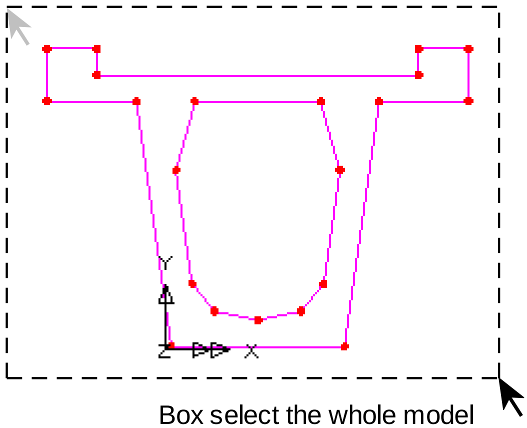
File
Import...

- Set the **Files of type** drop-down menu to **DXF Files (*.dxf)**
- Locate the **box_section.dxf** file in the **\<Lusas Installation Folder>\Examples\Modeller** directory and click the **Import** button to read in the DXF file and create the cross section geometry as shown below.

Defining holes within a section

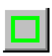
This is done by first defining a surface that represents the total extent of the box-section, then defining a surface that represents the void or hole, and then selecting both the surrounding and inner surface to create the hole in the bounding surface.

- Drag a box around the whole model



Box select the whole model

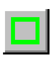
Geometry
Surface >
Lines…



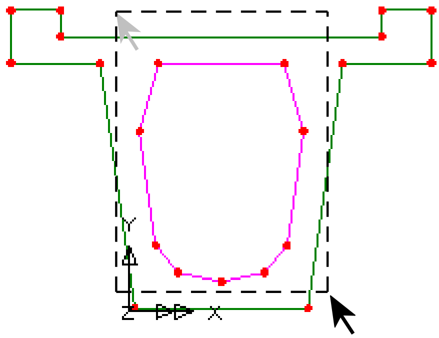
Create an outer Surface from the selected Lines.

- Now, drag a box around the only the lines defining the void.

Geometry
Surface >
Lines…

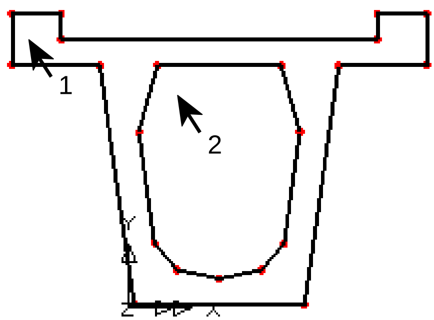


Create an inner Surface from the selected Lines.



Box select the lines defining the void

- Select the outer Surface of the box-section followed by the inner Surface representing the void. (Use the **Shift** key to pick the inner Surface to add it to the initial selection)

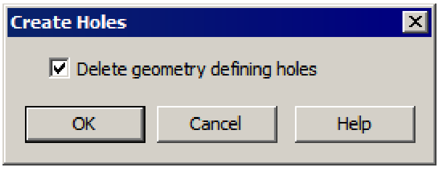


Select these two surfaces

Geometry
Surface >
Holes >
Create

- On the Create Holes dialog, select the option **Delete geometry defining holes**.

A new singular Surface will be created, containing a void



File
Save...

Save the cross-section model for any future modifications or for use with the definition of any additional fibre locations.

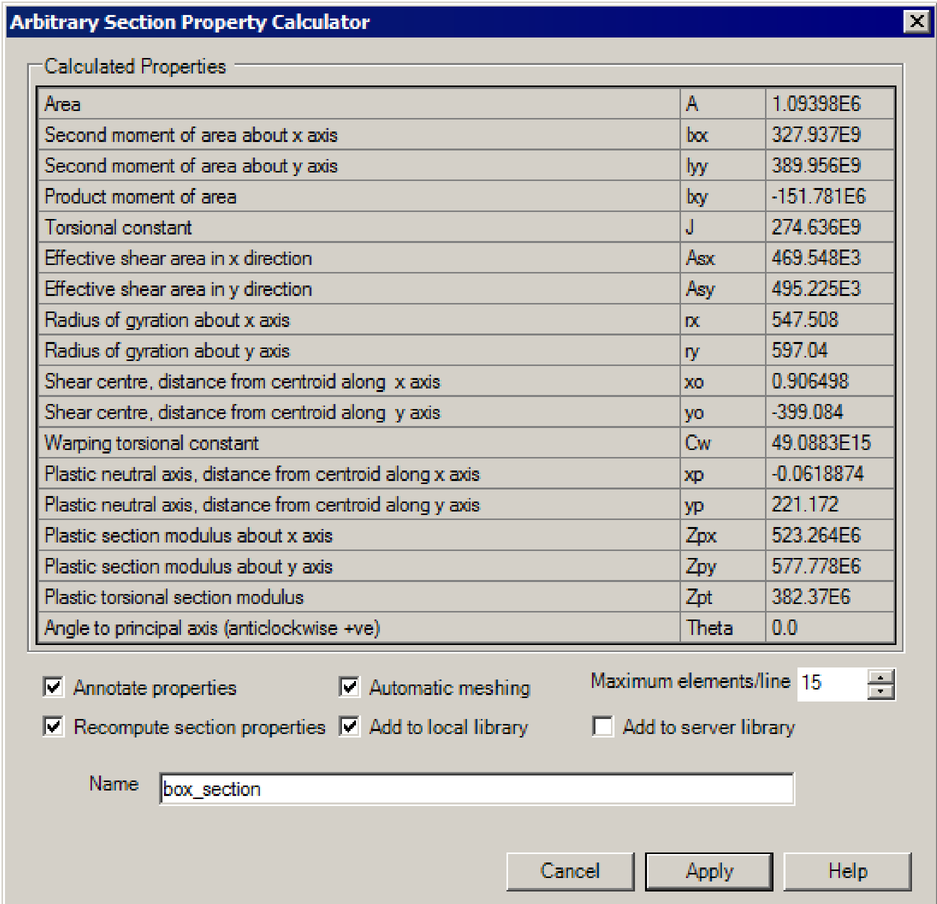
The section properties can now be calculated as follows:

Calculating Section Properties

Utilities
Section Property Calculator >
Arbitrary Section…

Select the **Add to local library** option.

- Click the **Apply** button, and after a short wait the calculated section properties will be displayed in the grid at the top of the dialog.



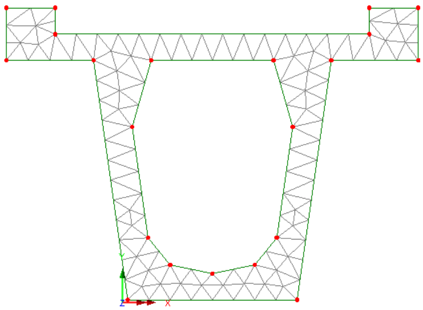
- Click **Cancel** to close the dialog.

Note. There is no need to compute the section properties in the units they are to be used in the analysis model because units conversion is carried out when the section properties are extracted from the section library if it is found to be required.

Note. Section properties may be added to local (user) or server (all users) section property libraries by selecting the appropriate option(s) prior to selecting the **Apply**

button. Alternatively you can deselect the ‘Recompute section properties’ option after selecting the ‘Add local library’ or ‘Add server library’ options. By default the model name is entered as the Section Name. This can be modified if required.

Notes on the automatic meshing used

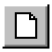


The mesh used to compute the properties of each of the surfaces (and the holes) is displayed in the graphics window.

By default an element size is selected which will assign 15 elements to the longest side and a minimum of 2 elements is applied to the shorter sides. This mesh may be adjusted by deselecting the automatic mesh check box and changing the mesh size in the  Treeview. Alternatively, the maximum element on the longest side may be adjusted by changing the ‘Max elements/line’ option as required prior to selecting the Apply button. As with all finite element models the more elements used the more accurate the results but the slower the calculation. A good compromise of 2 elements across all thin sections has been found to provide reasonable results without using excessive computation time.

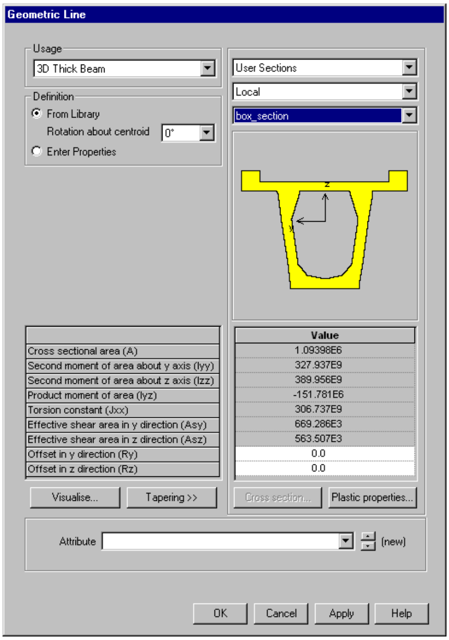
Viewing and using sections in the user library

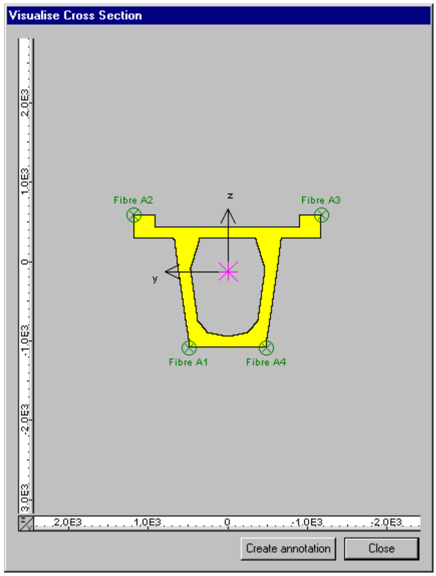
File
New…

 Create a new model called **Test** and select units of **N,mm,t,s,C** from the drop down list provided. Click **OK**.

Attributes
Geometric >
Section library…

- Select **User Sections** from the top-right drop-down on the dialog
- Browse for, and select the **box_section** name. The basic section will be visualised.
- Select the **Visualise** button and the fibre definitions will be seen. Click **Close**

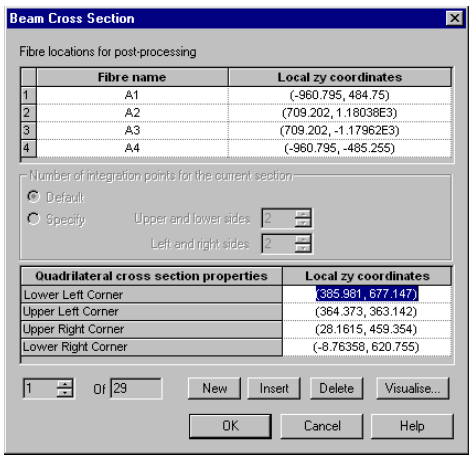




Visualisation of local library item **Visualisation of fibre locations**

Fibre locations can be used to calculate beam stress results for plotting force and moment diagrams. LUSAS automatically creates one for every extreme point on a section.

- On the Geometric Line dialog, select the **Enter Properties** option then press the **Section Details** button.
- Select the **Fibre locations** tab.
- Additional fibre locations can be added manually to the library item by clicking in the Fibre locations area and pressing the **TAB** key until a new line is created, and then entering the coordinate of the fibre location required.
- Press **OK** to save changes and return to the Geometric Line dialog.



Tip. If additional fibre locations are required on the section use Modeller to obtain the properties of any additional points on the saved model and enter the point coordinates in the manner described above.

Finally, to add the local library section to the Attributes Treeview (for subsequent assignment to beams in the model) press the **OK** button.

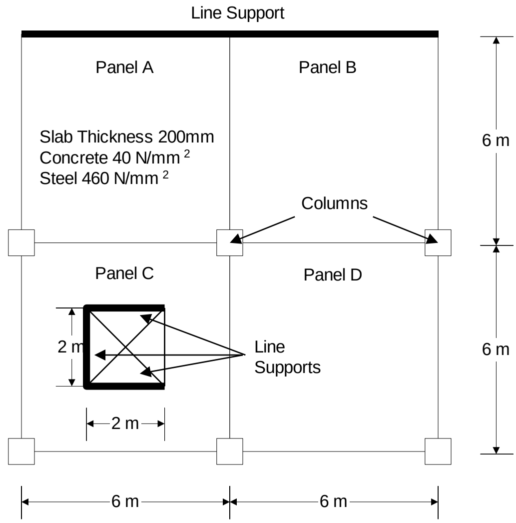
This completes the example.

Simple Building Slab Design

For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	None.
Note: The example as written exceeds the limits of the LUSAS Teaching and Training Version. However, by not increasing the default mesh density to 8 divisions per line (where shown) the analysis can be run using a default of 4 divisions per line.	

Description

Four panels of a concrete slab supported by a wall, columns and a lift shaft are to be analysed, with reinforcement areas computed for bending moment only. Shear and displacement checks need to be carried out separately. The geometry of the slab is as shown.



The slab is subjected to self-weight and a live load.

The units of the analysis are N, m, kg, s, C throughout.

Objective

- To produce areas of reinforcement under ultimate loads and check crack widths under service loads

Keywords

Slab Design, Holes, Reinforcement, Wood-Armer, RC Design, Steel Area, Load Combinations, Smart Combinations, Cracking

Associated Files

- ❑ **slab_design_modelling.vbs** carries out the modelling of this example.

Modelling

Running LUSAS Modeller

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

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

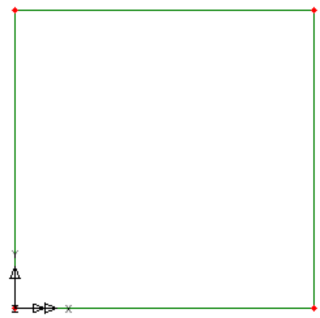
- Enter the file name as **Slab_Design**
- Use the **Default** working folder.
- Enter the title as **Slab Design Example**
- Select model units of **N,m,kg,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the Startup template **Standard** from those available in the drop-down list.
- Ensure the **Z** vertical axis is selected and click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Feature Geometry

Geometry
Surface >
Coordinates...

Enter coordinates of (**0, 0**), (**6, 0**), (**6, 6**) and (**0, 6**) to define the lower left-hand area of slab. Use the Tab key to move to the next entry field. With all the coordinates entered click the **OK** button to create a surface.



- Select the newly created Surface.

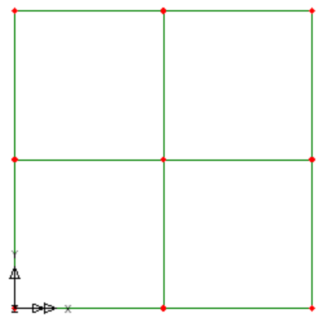
Geometry
Surface >
Copy…

Copy the selected surface with an X translation of **6**. Click the **OK** button to create the new Surface.

- Use **Ctrl + A** keys together to select the whole model.

Geometry
Surface >
Copy…

Copy the selected surfaces with a Y translation of **6**. Click the **OK** button to define two new Surfaces.



Defining a Hole in a Slab

Next, a hole representing the lift shaft needs to be defined. This is done by first creating a surface representing the extent of the lift shaft and then selecting both the surrounding and inner surface to define the hole.

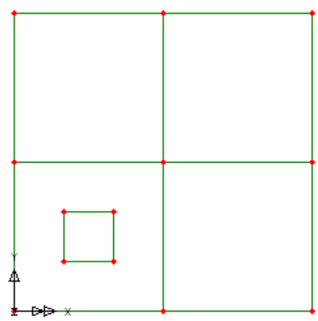
Geometry
Surface >
Coordinates...

Enter coordinates of **(2, 2), (4, 2), (4, 4)** and **(2, 4)** and click **OK** to define the extent of the hole.

- Select the lower left Surface and then the Surface representing the lift shaft. (Use the **Shift** key to pick the second Surface to add to the selection)

Geometry
Surface >
Holes >
Create

- Ensure **Delete geometry defining holes** is selected A new singular Surface will be created.



This new surface containing the hole can be seen / checked by clicking in a blank part of the Graphics Window and then re-selecting the lower-left hand surface.

Note. It is normally good practice to ensure that the orientation of surface axes (and hence mesh element orientation) is consistent throughout the model. However, plate elements, as used in this example, produce results based upon global axes and as such ignore inconsistent element axes.

Meshing

By default lines normally have 4 mesh divisions per line. If you are using the Teaching and Training version this default value should be left unaltered to create a surface element mesh within the limit available. This will give a coarser mesh and correspondingly less accurate results will be obtained. Otherwise, for this example, and to give greater accuracy, 8 divisions per line will be used.

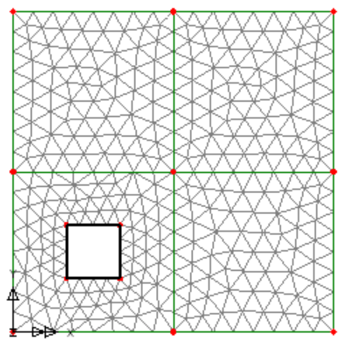
File
Model Properties…

- Select the **Meshing** tab and set the default number of divisions to **8** and click the **OK** button.

Attributes
Mesh >
Surface...

- Select the element type as **Thick Plate**, the element shape as **Triangle** and the interpolation order as **Quadratic**. Select an **Irregular mesh** and ensure the **Element size** is deselected. This forces the number of default line mesh divisions to be used when meshing the surfaces. Enter the attribute name as **Thick Plate** and click the **OK** button.

- With the whole model selected drag and drop the mesh attribute **Thick Plate** from the Treeview onto the selected Surfaces



In the vicinity of the lift core less elements are required.

- Select the 4 lines defining the lift core and drag and drop the line mesh attribute **Divisions=4** from the Treeview onto the selected Lines.

In this manner the mesh density on slabs can be varied according to the levels of detail required.

Geometric Properties

Attributes
Geometric >
Surface…

- Enter a thickness of **0.2**. Enter the attribute name as **Thickness 200mm** and click the **OK** button.

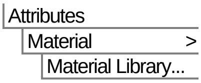
- With the whole model selected drag and drop the geometric attribute **Thickness 200mm** from the Treeview onto the selected Surfaces.

Geometric properties are visualised by default.

- In the Treeview drag and re-order the layers so that the **Attributes** layer is at the top, the **Mesh** layer is in the middle, and the **Geometry** layer is at the bottom.

Select the fleshing on/off button to turn-off the geometric visualisation.

Material Properties

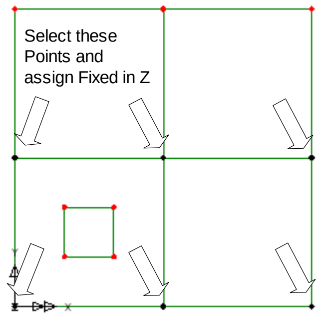


- Select material **Concrete EU** from the from drop-down list, select grade **EN1992-1-1 Table 3.1 fck=40MPa** and click **OK** to add the material attribute to the Treeview.
- Select the whole model and drag and drop the material attribute **Iso1 (EN1992-1-1 Table 3.1 fck=40MPa)** from the Treeview onto the selected Surfaces.

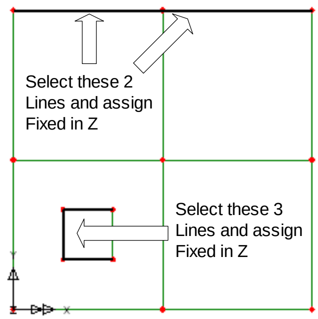
Supports

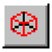
For clarity the mesh layer is not shown on these diagrams.

- Select the 6 points where the columns are located.
- Assign the supports by dragging and dropping the support attribute **Fixed in Z** from the Treeview and assign to **All analysis loadcases** by clicking the **OK** button.

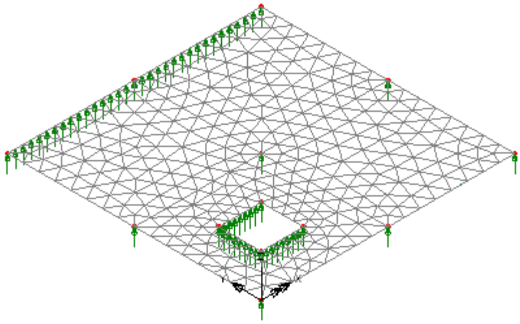


- Select the 2 Lines representing the line support and drag and drop the support attribute **Fixed in Z** from the Treeview and assign to **All analysis loadcases** by clicking the **OK** button.
- Select only the top, bottom, and left-hand lines defining the lift shaft and drag and drop the support attribute **Fixed in Z** from the Treeview and assign to **All loadcases** by clicking the **OK** button.

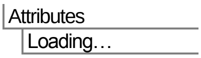


 Select the isometric view button to check that all supports have been assigned correctly.

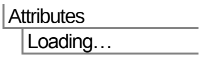
Click this part of the status bar to view the model from the Z direction again.



Loading

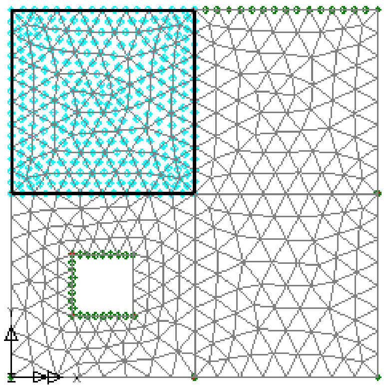


- Select **Body Force** and click **Next**.
- Enter an acceleration of **-9.81** in the Z direction.
- Enter an attribute name of **Self Weight** and click the **Finish** button.



- Select the **Global Distributed** radio button.
- Select the **per unit Area** option.
- Enter **-5000** in the **Z** Direction.
- Enter the attribute name as **Imposed Load 5kN/m2** and click the **Finish** button.

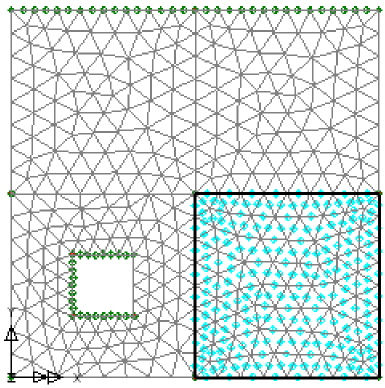
Now the dead and live loading needs to be assigned to each slab panel of the building. This is done using separate load cases so that the loadcases can be combined to determine the most adverse effects.



- Firstly, right click on Analysis 1 > Loadcase 1 in the  Treeview, select the **Rename** option and change the first loadcase name to **Panel A Permanent**
- Select the surface representing the top left-hand panel (Panel A).
- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the  Treeview onto the selection. The loading assignment dialog will be displayed. Select the **Single Loadcase** option, the analysis **Analysis 1** and loadcase **Panel A Permanent**, then click the **OK** button.

Note. The loading assigned to each panel will be visualised as each loadcase is assigned to the model

- Drag and drop the dataset **Imposed Load 5kN/m2** from the Treeview onto the selection and change the loadcase name to **Panel A Variable** and click the **OK** button.
- Select the top right-hand panel (Panel B).
- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the Treeview onto the selection. Change the loadcase name to **Panel B Permanent** Leave the load factor as 1 and click the **OK** button.
- Drag and drop the attribute **Imposed Load 5kN/m2** from the Treeview onto the selection and change the loadcase name to **Panel B Variable** and click the **OK** button.
- Select the bottom-left panel (Panel C).
- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the Treeview onto the selection. Change the loadcase name to **Panel C Permanent** and click the **OK** button.
- Drag and drop the attribute **Imposed Load 5kN/m2** from the Treeview onto the selection. Change the loadcase name to **Panel C Variable** and click the **OK** button.
- Select the bottom right-hand panel (Panel D).
- Assign the dead loading by dragging and dropping the attribute **Self Weight** from the Treeview onto the selection. Change the loadcase name to **Panel D Permanent** and click the **OK** button.
- Drag and drop the attribute **Imposed Load 5kN/m2** from the Treeview onto the selection. Change the loadcase name to **Panel D Variable** and click the **OK** button.



The Treeview should now contain 8 loadcases consisting of a Permanent and Variable loadcase for each panel.

Saving the model

File
Save

Save the model file.

Running the Analysis

With the model loaded:

Select the **Solve Now** button from the toolbar and click **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 Treeview

Addition files will be created in the directory where the model file resides, including:

- **slab_design.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **slab_design.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **slab_design_modelling.vbs** carries out the modelling of the example.

File
New...

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 **slab_design**

File
Script >
Run Script...

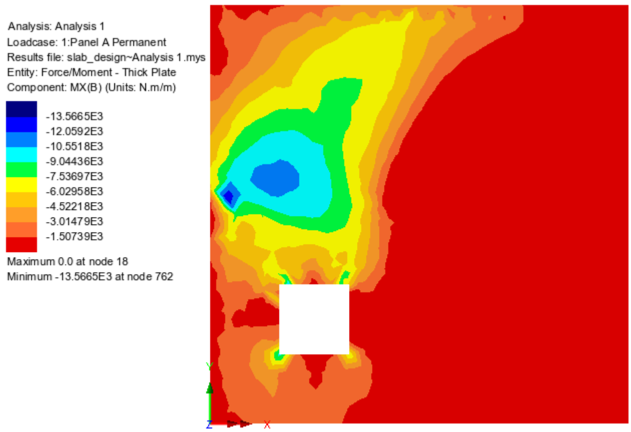
- To recreate the model, select the file **slab_design_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the Treeview, and load case results for Panel A Permanent are set to be active by default.

- Turn off the **Attributes** layer in the Treeview.
- With no features selected, click the right-hand mouse button in a blank part of the active window and select the **Contours** option to add the contours layer to the Treeview.



- Select entity **Force/Moment - Thick Plate** and results component **MX(B)** to see the results for Permanent effects in Panel A.

Defining a Load Combination

The loadcases will be combined to provide the most adverse loading effects. This is achieved using the smart load combination facility within LUSAS Modeller. For Ultimate Limit State (ULS) design EN1990:2002 requires permanent actions to be factored by 1.35 where unfavourable and 1.0 where favourable. Unfavourable variable actions are factored by 1.5 and omitted where favourable. (These factors may be subject to national variation)

For permanent actions this is equivalent to a **Permanent load factor** of 1.0 and a **Variable load factor** of 0.35 since 1.0 will always be applied and 0.35 will only be applied if it creates an unfavourable effect.

For variable actions this is equivalent to a **Permanent load factor** of 0.0 and a **Variable load factor** of 1.5 since the loading will only be applied if it creates an unfavourable effect.

Note. Permanent load factor and **Variable load factor** refer to the LUSAS Smart Combination definitions rather than the EN1990 definitions.

Analyses
Smart Combination...

Creates a smart combination in the Treeview.

- Include all loadcases in the smart combination. To do this select the first load case in the loadcase selector at the bottom-left of the then hold the **Shift** key down and scroll down the list and select the last loadcase.
- Click the >> button to add these loadcases to the included list.
- Select the **Grid** button and set the permanent and variable factors as shown in bold in the table that follows:

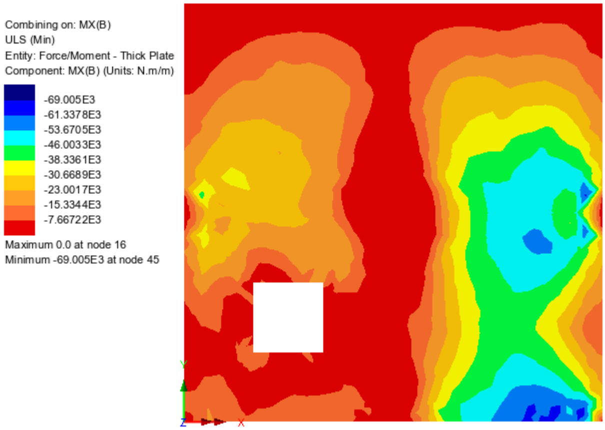
Name	Permanent Factor	Variable Factor
Panel A Permanent	1.0	**0.35**
Panel A Variable	**0**	**1.5**
Panel B Permanent	1.0	**0.35**
Panel B Variable	**0**	**1.5**
Panel C Permanent	1.0	**0.35**
Panel C Variable	**0**	**1.5**
Panel D Permanent	1.0	**0.35**
Panel D Variable	**0**	**1.5**

- Click the **OK** button to return to the combination properties
- Change the smart combination name to **ULS**
- Click the **OK** button to complete the definition of the smart combination

The maximum combination will produce the most adverse hogging moments whilst the minimum combination will produce the most adverse sagging moments.

Note. When the properties of the max combination are modified the corresponding min combination is updated automatically.

- Select **ULS (Min)** from the Treeview with the right-hand mouse button and pick the **Set Active** option.



- Select entity **Force/Moment – Thick Plate** and results component **MX(B)** to combine, applying the variable factors based on the bottom Wood Armer moments in the X direction and click the **OK** button.

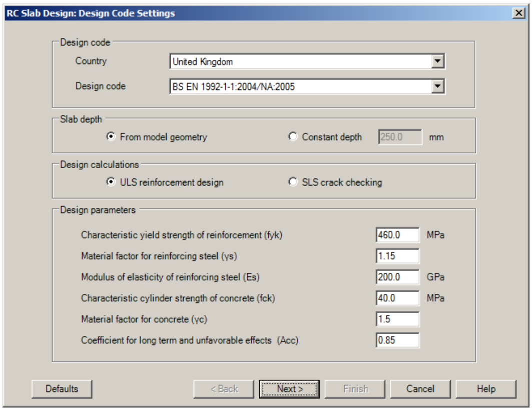
Using the RC Slab Design facility

The RC Slab design facility enables calculation of required steel reinforcement areas for ULS loadcases and calculation of crack widths for SLS loadcases. ULS reinforcement design is based on calculated Wood-Armer moments whereas crack width calculations are based on principal moments. The effective depth is computed from the top and bottom reinforcement bar sizes and covers provided.

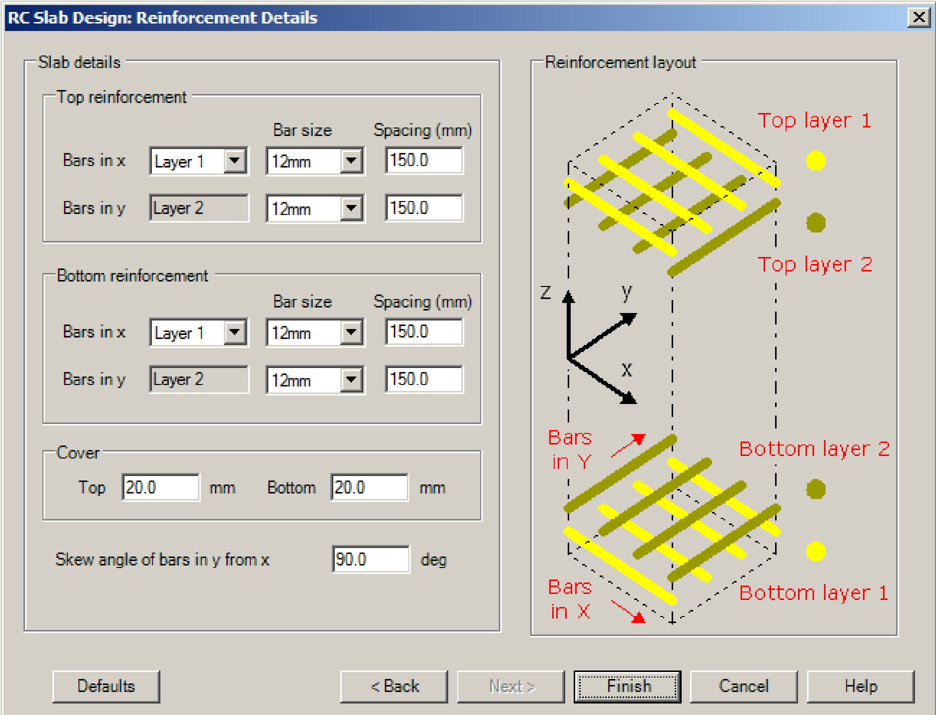
Note. This example uses an irregular mesh of thick plate elements and for these elements the results are output in global directions that, in this case, match the intended reinforcement directions. In other modelling situations ,if the elements in a model are orientated such that their local axes vary from one another, or if an alternative coordinate system is required, the results may will need to be transformed to a consistent direction. This can be achieved by either setting a local coordinate set or choosing an appropriate results transformation option on a results layer property dialog prior to opening the RC Slab Designer.

Bridge
RC Slab Design...

- Ensure that **United Kingdom** and **BS EN 1992-1-1:2000/NA:2005** are selected from the dropdown list of countries and design codes

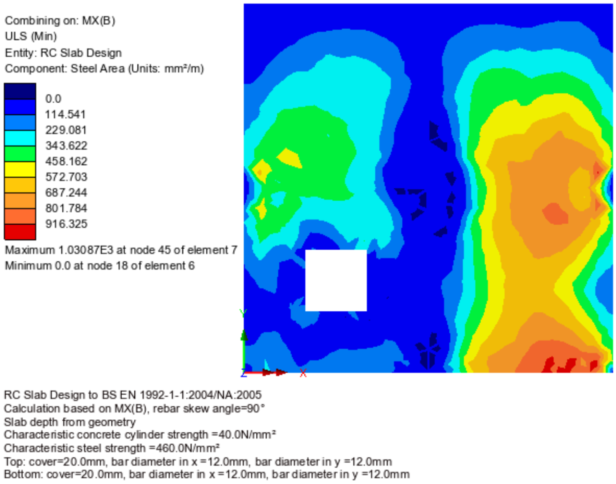


- Ensure slab depth is **from model geometry**
- Ensure that the option for **ULS reinforcement design** is selected.
- Ensure that the characteristic concrete cylinder strength is **40** and the characteristic yield strength of the reinforcement to **460**.
- Click **Next** to proceed to the Slab Properties.



- Enter the slab properties as shown
- Click **Finish** to calculate the reinforcement requirements.

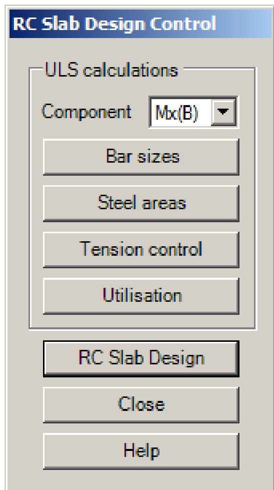
This will close the RC Slab Designer dialog and plot contours of required bottom steel area in the X direction based on the active Loadcase (or combination/envelope).



A floating toolbox is provided to enable results for all four layers of reinforcement to be viewed without having to return to the slab designer dialog.

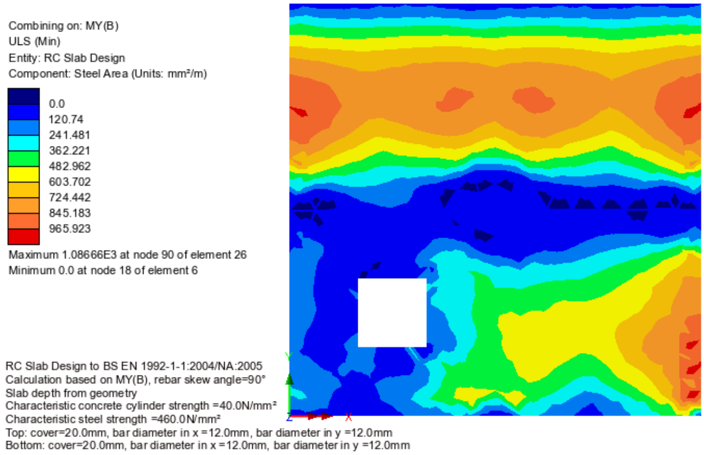
Plotting the contours of Utilisation displays the ratio of the applied moment to the calculated moment capacity and provides a rapid check on the proposed reinforcement arrangement.

To calculate results for the bottom steel in the Y direction:



- Select **ULS (Min)** from the Treeview with the right-hand mouse button and pick the **Set Active** option.
- Select **Force/Moment – Thick Plate** and **MY(B)** from the drop-down lists to combine applying the variable factors based on the Wood-Armer moments in the Y direction and click the **OK** button.
- The Slab Designer will automatically recalculate for the new results however the calculations will be based on MX(B) as shown in the toolbox.
- A message may be issued by the Slab Designer to warn that the current loadcase is optimised for different results than those currently being calculated by the Slab Designer. Change the toolbox dropdown item to be MY(B) and the Slab Designer will recalculate the appropriate steel areas in the Y direction.
- The reinforcement design can be viewed in terms of required area, bar size and utilisation by selecting the appropriate button in the toolbox.

Caution. The ULS design of reinforcement assumes that the slab is "under-reinforced" and therefore the calculations are only valid when this holds true. The contour plot "Tension Control" indicates whether the slab will undergo ductile failure and is hence "under-reinforced." Plots of Tension Control must therefore be checked to ensure a valid design. For more information see the help entry.



- The previous processes can also be repeated to plot contours of required top steel reinforcement using the **ULS (Max)** combination for most adverse hogging moments **MX(T)** and **MY(T)**

Concrete crack checking

The calculation of crack widths in the RC Slab Design facility is based on principal moments rather than Wood-Armer moments used in reinforcement design. Effective steel areas are interpolated based on the orientation of the principal moments with respect to the steel directions. Extensive coverage of the crack calculation procedure is provided in the help manual.

Crack widths will be calculated based on service loads and hence a new combination is required. Click the **Close** button on the Slab Designer toolbox to clear the current reinforcement results.

Analyses
Smart Combination...

Creates a smart combination in the Treeview.

- Include all loadcases (but not the two ULS combinations) in the basic combination. To do this select the first load case in the loadcase selector at the bottom-left of the then hold the **Shift** key down and scroll down the list and select the last loadcase.

- Click the >> button to add these loadcases to the included list.

- The load factors can be entered as follows

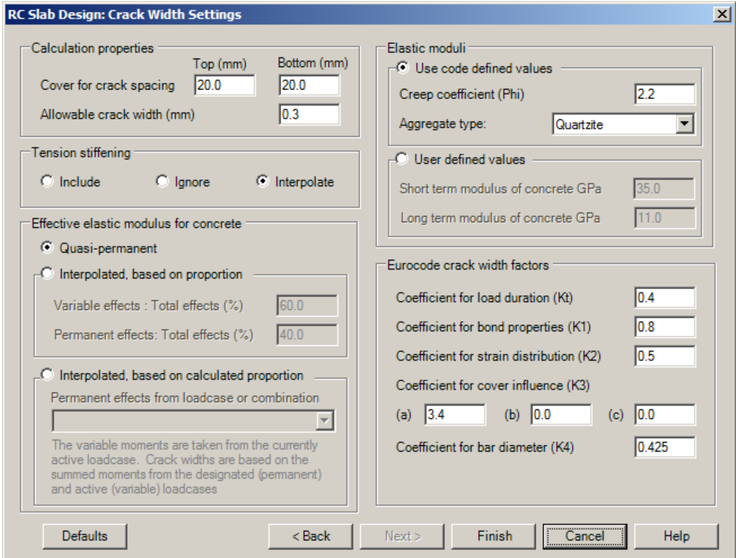
Name	Permanent Factor	Variable Factor
Panel A Permanent	1.0	**0**
Panel A Variable	**0**	**1.0**
Panel B Permanent	1.0	**0**
Panel B Variable	**0**	**1.0**
Panel C Permanent	1.0	**0**
Panel C Variable	**0**	**1.0**
Panel D Permanent	1.0	**0**
Panel D Variable	**0**	**1.0**

- Click the **OK** button to return to the combination properties
- Change the smart combination name to **SLS**
- Click the **OK** button to complete the definition of the smart combination
- Select **SLS (Min)** from the Treeview with the right-hand mouse button pick the **Set Active** option and set the combination for **MMin**.

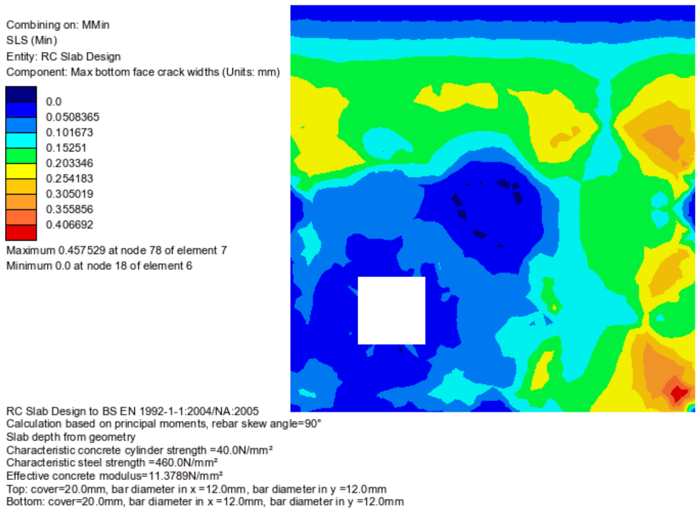
Bridge
RC Slab Design...

Restart the Slab Designer

- On the main page change the design calculation option to **SLS crack checking**
- Select the **Next** button to navigate to the reinforcement details (which remain as set for the ULS design) and again to access the crack width settings.
- Enter the crack width settings as shown on the dialog.

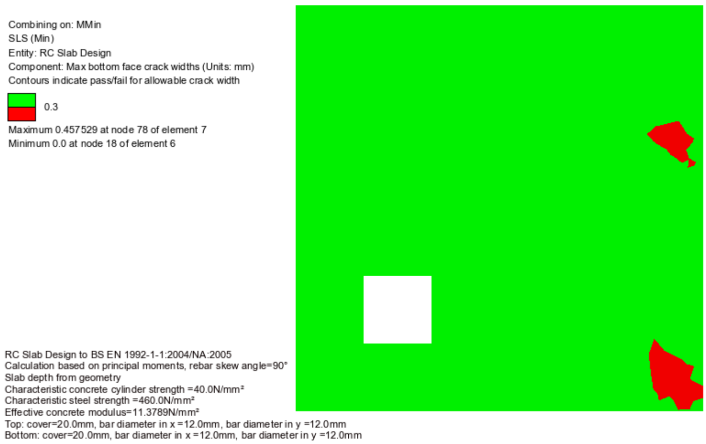


- Click the **Finish** button to plot contours of crack widths throughout the slab. Crack widths to the top and bottom faces can be viewed by selecting either Top of Bottom in the toolbox.



Note. The crack checking calculations are based on strains calculated from an interpolated effective area of steel. However the actual crack width is calculated taking the bar spacing and bar diameter of the layer orientated closest to the principal moment in question. Contour plots for crack widths "controlled" by each reinforcement layer are provided to identify exactly which bars are controlling the cracks. By default the maximum crack width is displayed.

To determine the extent of any excessive cracking press the **Pass/Fail** button on the RC Slab Design Control to show a two-tone contour plot of crack width below and above the acceptable limit.



In this instance excessive cracking is localised and local increases in reinforcement maybe sufficient to prevent the cracks. The reinforcement density will be increased to prevent any excessive cracking. Return to the Slab Design dialog by selecting **RC Slab Design** from the floating toolbox. Click next to display the crack definition page and increase the bottom reinforcement to be **16**mm in both X and Y directions. Click next and then click finish to view the updated results. The extent of the cracking is reduced to a small concentration. Cracking to the top face would be investigated in a similar manner with the combination set for MMax to maximise hogging moments.

Note. The crack checks are based on an interpolated concrete modulus allowing for time-dependant effects. The interpolated value is displayed in the on screen output. For complicated models featuring multiple materials, it may therefore be necessary to reanalyse the model with the interpolated concrete stiffness to more precisely calculate the loading effects.

Click **Close** in the Slab Designer toolbox to remove all Slab Designer results and return to modeller. All the Slab Designer inputs are saved to the model and will be saved to disk when the model is saved. Should the Slab Designer need to be run again the saved values will be recovered.

Save the model

File
Save

Save the model file.

Note. When the model file is saved after results processing, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

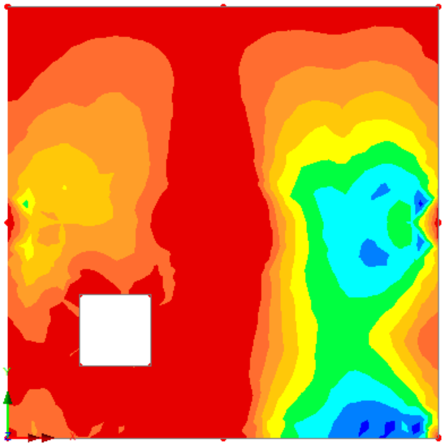
This completes the example.

Discussion

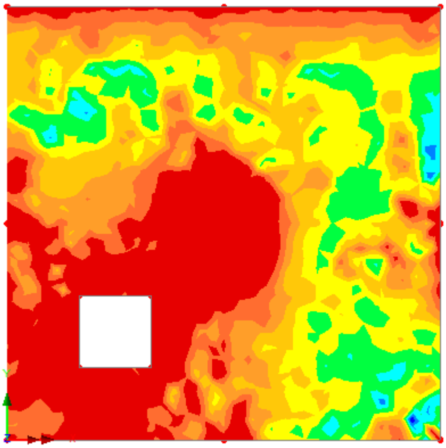
Element types and orientation of results

This example demonstrates the use of the RC Slab designer for a model that uses an irregular mesh of thick plate elements. Within the slab the reinforcement will generally be orthogonal for the layout demonstrated and for thick plate elements the results are output in global directions, so there is no conflict. If, in a real-life example, shell elements were inadvertently used for a model such as this, the use of an irregular mesh would mean that the results used for the Wood-Armer / slab calculations would have different orientations over the whole slab and hence be incorrect. In this circumstance a local coordinate set would need to be used to orientate the shell element results to be consistent with the global axes in order to obtain sensible components for use in the calculations.

The following result contours shows the difference between results from global (left) and local (right) axes for the ULS(Max) MX(B) plot.



Global Axes

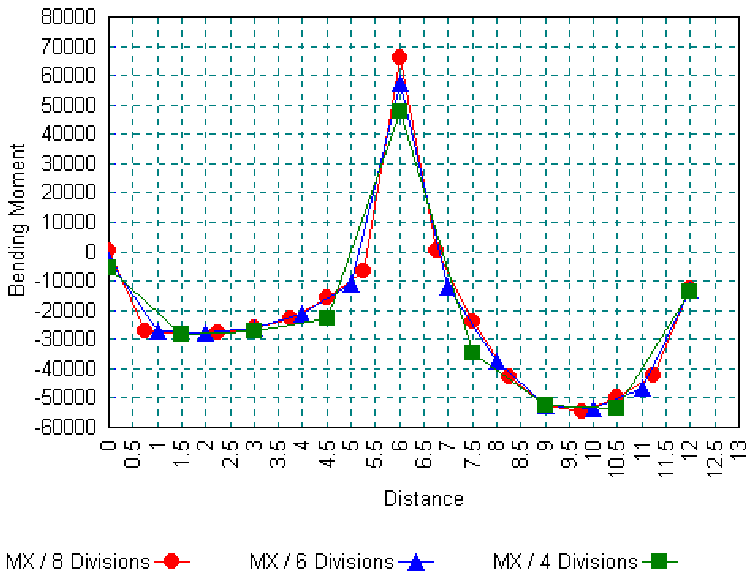


Local Axes

- The mesh orientation can be viewed from the Treeview by selecting **Mesh** > **Properties** and selecting the **Show Element Axis** option. Press **OK** to apply changes.
- Results transformations can be applied using the **Transformed...** button on a results layer's Properties dialog. Selecting **Global axes** will transform local element results into the global axes system. Other options are available..
- Global results are indicated by the uppercase X, Y and Z in the component names, whereas lowercase x, y and z (when seen) are used to denote local results or transformed results.

Element density and results obtained

If the Evaluation Version of LUSAS was used to carry out this example with a reduced default mesh density of 4 divisions per line a reduced accuracy of results will have been obtained. The following graph for ULS(Min) shows how Bending Moment (MX) along a horizontal 2D slice section through the central three columns typically varies with the number of line mesh divisions used.



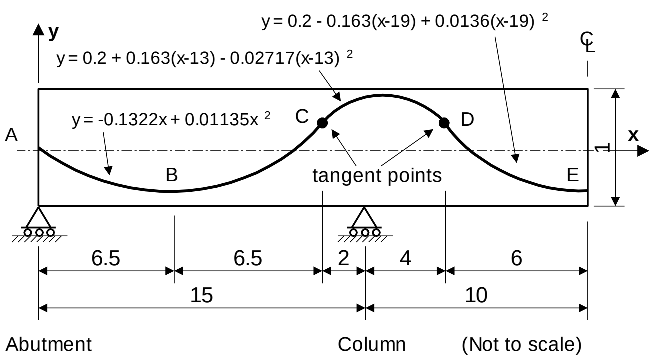
In general, more accurate results are obtained when using more line mesh divisions and hence more elements when modelling slabs of this type. Care should always be taken to use an appropriate number of elements together with a possible refinement of the mesh in areas of interest in order to obtain the best results.

Linear Analysis of a Post Tensioned Bridge

For software product(s):	LUSAS Civil & Structural or LUSAS Bridge
With product option(s):	None.

Description

A 3 span concrete post tensioned bridge is to be analysed. The bridge is idealised as a 1 metre deep beam with a tendon profile as shown in the half model of the bridge.



Half-model of bridge

The initial post tensioning force of 5000 kN is to be applied from both ends. Tendons with a cross section area of 3.5e3mm^2 are located every 2 metres across the section allowing the analysis to model a 2 metres effective width.

Three loadcases are to be considered; self-weight, short term losses, and long term losses.

Units of kN, m, t, s, C are used throughout the analysis. Note that tendon cross-sectional area is specified in mm^2.

Objectives

The required output from the analysis consists of:

- ❑ The maximum and minimum long and short term stress in the concrete due to post tensioning.

Keywords

2D, Beam, Prestress, Post tensioning, Beam Stress recovery.

Associated Files

- ❑ **post_ten_modelling.vbs** carries out the modelling of the example.
- ❑ **post_ten_profile.csv** carries out the definition of the tendon.

Modelling

Running LUSAS Modeller

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

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 the file name as **post_ten**
- Use the **Default** working folder.
- Enter the title as **Post-tensioning of a bridge**
- Select model units of **kN,m,t,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the startup template **Standard**
- Select the **Vertical Y axis** option.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily.

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

Feature Geometry

Geometry
Line >
Coordinates...

Enter coordinates of (**0, 0**), (**15, 0**) and (**25, 0**) to define two Lines representing half the bridge. Use the **Tab** key to move to the next entry field on the dialog.

- With all the coordinates entered click the **OK** button.
- Select all the visible Points and Lines using the **Ctrl** and **A** keys together.

Geometry
Group >
New Group...

To make the manipulation of the model easier create a group of the bridge

- Enter the group name as **Bridge**
- In the Treeview select the group **Bridge** with the right-hand mouse button and select the **Invisible** option to hide these features from the display.

Defining the tendon profile

When using the Prestress utility the tendon geometry is determined from a Line definition which, in practice, is usually a spline curve. The geometry of the tendon may be input into the model directly by manually entering the coordinates, or, as in this example, by copying values from a comma separated file (.csv) which has been opened in a spreadsheet application and pasting these values into the Point coordinates dialog in Modeller. This is done using standard Ctrl + C and Ctrl + V keys.

Note. To prevent the point representing the left-hand end of the bridge and the end point of the tendon profile from becoming stored as just a single point in the model the geometry should be made unmergable.

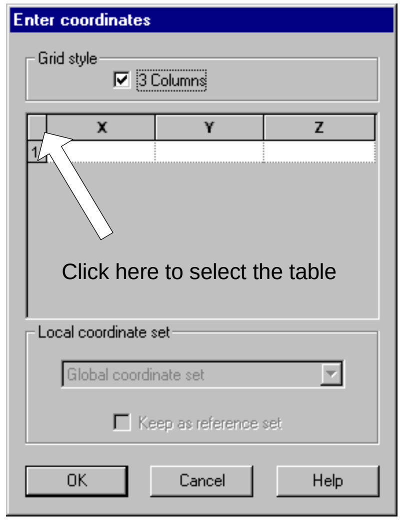
File
Model Properties…

- On the Geometry tab select the **New geometry unmergable** option and click **OK**

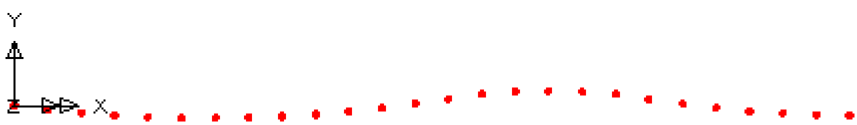
- Read the file **\<LUSAS Installation Folder>\Examples\Modeller\post_ten_profile.csv** into a spreadsheet.

Geometry >
Point >
Coordinates…

- Select the top left-hand corner of the spreadsheet grid to select all the cells and use **Ctrl + C** to copy the data.
- Select the **3 Columns** option to show X, Y and Z columns.
- Select the top left-hand corner of the Enter Coordinates dialog and use **Ctrl + V** to paste the coordinates defining the tendon profile into the table. Note that the use of 'Paste' using a mouse button is not enabled. Click **OK**.



The points defining the tendon profile should appear as shown.



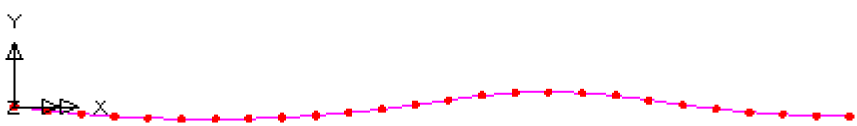
File
Model Properties…

On the Geometry tab de-select the **New geometry unmergable** option and click **OK**

- Use **Ctrl** and **A** keys together to select all the visible Points.

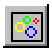
Geometry
Line >
Spline >
By Points…

- Click **OK** to define a spline that is defined in point selection order.



- Use **Ctrl** and **A** keys together to select the spline line and points.

Geometry
Group >
New Group

 Make the model definition easier by placing the prestress tendon into a group.

- Enter the group name as **Tendon** and click **OK**

Note. As well as defining the tendon manually using coordinates the Prestress Wizard allows you to define the profile of the tendon in 2D or 3D space as a series of straight lines, arcs, splines and parabolas, using **Bridge** (or Civil) > **Prestress Wizard** > **Tendon Profile**. The Tendon Profile dialog is used in the Segmental Construction of a Post Tensioned Bridge example.

Defining the Mesh

The concrete bridge is represented by 2D beam elements.

- In the  Treeview select **Bridge** and pick the **Set As Only Visible** option. This turns off the display of the tendon to make bridge assignments easier.

Attributes
Mesh >
Line...

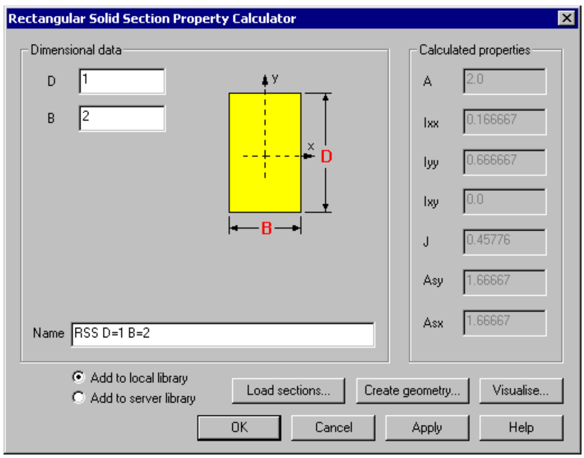
- Select **Thick Beam, 2 dimensional, Linear** elements, enter the dataset name as **2D Beam** and click **OK** to add the mesh dataset to the  Treeview.
- Use **Ctrl + A** to select all the features.
- Drag and drop the mesh dataset **2D Beam** onto the selection.

Defining the Geometric Properties

The beam idealisation represents a two metre width of the bridge which is one metre deep. A section of this size will be created and added to the local library for subsequent assigning to the model.

Utilities
Section Property Calculator >
Rectangular Sections > Solid

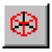
- Select the **Rectangular Solid Section** (RSS)
- Enter a depth (D) of **1** and a breadth (B) of **2** and click **OK** to add the section to the local library.

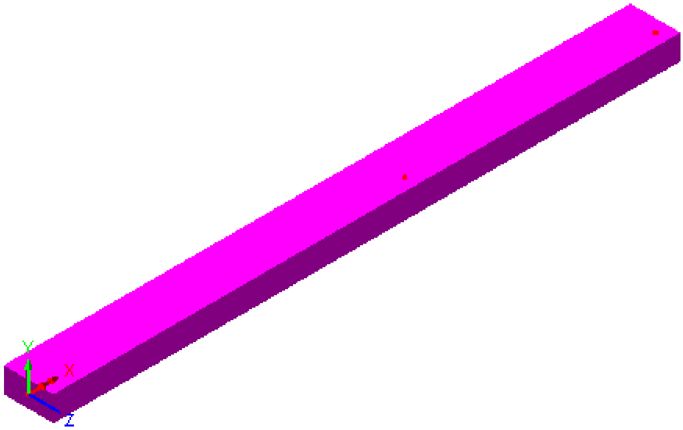


To add this section to the Treeview:

Attributes
Geometric >
Section Library..

- From the Usage section select **3D Thick Beam (Any beam)**
- Select **User Sections** from the top-most drop-down list, select **Local** library, then select **RSS D=1 B=2**. Since the initial section was defined simply in an xy plane, a rotation about the centroid will be required. As a result:
- Select a Rotation about centroid of **90**
- Enter the attribute name as **Beam Properties** and click **OK**
- With the whole model selected drag and drop the geometric dataset **RSS D=1 B=2 Beam Properties (RSS D=1 B=2 major y)** from the Treeview onto the selected features.

 Select the Isometric button to view the visualised beam.



 Select the fleshing on/off button to turn-off the geometric visualisation.

 Select the Home button to return the model to the default view.

Defining the Material

Attributes
Material >
Material Library…

- Select material **Concrete** of grade **Ungraded** from the drop down list and click **OK** to add the material dataset to the Treeview.

- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material dataset **Iso1 (Concrete Ungraded)** from the Treeview onto the selected features and assign to the selected Lines by clicking the **OK** button.

Defining the Supports

LUSAS provides the more common types of support by default. These can be seen in the Treeview.

The bridge is supported with rollers at each of the abutments.

- Select the point at the left-hand abutment and the point at the column. (Hold the **Shift** key to add to the initial selection)

- Drag the support dataset **Fixed in Y** from the Treeview and drop it onto the selected Points in the graphics window. Choose options to **Assign to points** for **All analysis loadcases** and click **OK**

- Select the right-hand point on the centre line of the bridge.

- Drag the support dataset **Symmetry YZ** from the Treeview and drop it onto the selected Point in the graphics window. Choose options to **Assign to points** for **All analysis loadcases** and click **OK**

Loading

Three loadcases are to be applied. The first represents self-weight of the structure. The second represents the prestress with short term losses only. The third represents the prestress with short and long term losses.

Defining the Self-weight

- Gravity can be applied to any loadcase from the Treeview.

- Select **Loadcase 1** from the Treeview with the right-hand mouse button and select **Gravity**. Note that, when applied to a loadcase as an option load arrows representing self-weight are not shown

- Select **Loadcase 1** from the Treeview with the right-hand mouse button and pick the **Rename** option and change the loadcase name to **Self Weight**

Turn-on the display of the tendon

Now that the bridge assignments have been made the tendon can be re-displayed.

- In the Treeview select **Tendon** with the right-hand mouse button and select the **Visible** option.

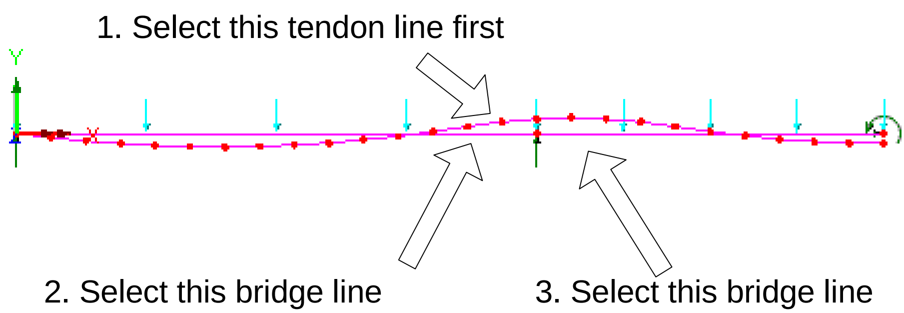
Rebuilding a model after a previous failed analysis

If a previous analysis of this example has failed you need to return to this point to continue after having run the supplied file stated.

Defining Prestress with Short Term Losses

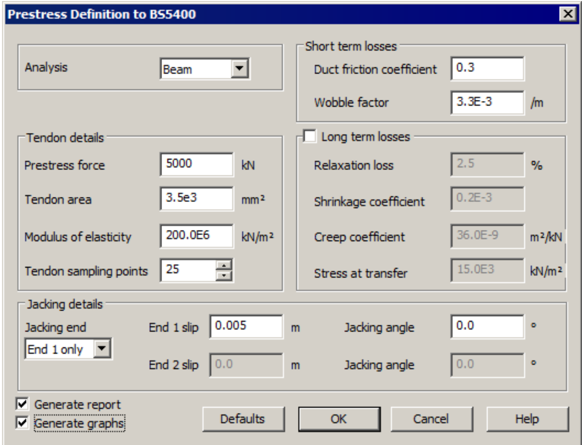
Analyses
Loadcase…

- Change the loadcase name to **Short Term Loss** and click **OK**
- Select **Short Term Loss** from the Treeview with the right-hand mouse button and pick the **Set Active** option. Ensure that the Automatically add gravity to this loadcase option is not selected.
- Select the spline Line representing the tendon in the Treeview with the right-hand mouse button and click the **Select Members** option. Then select the 2 beams representing the concrete bridge. (Hold the **Shift** key to add the other 2 lines to the initial selection)



Caution. Prior to running the Prestress Wizard to calculate short term and long term tendon losses the correct Short Term or Long Term loadcase must be set active to ensure that the values are written into the correct Tendon Load assignment attributes as held in the Treeview. With the Short Term Loss loadcase active:

Bridge
Prestress Wizard >
Single Tendon >
BS5400…



- Select the **Defaults** button.
- Ensure the Analysis type is set to **Beam**
- In the Tendon details section enter the Prestress force as **5000**
- Enter the Tendon area as **3.5E3** (note that this is entered in mm2)
- Leave the Modulus of the elasticity of the tendon as **200E6**
- Set the number of Tendon sampling points to **25**
- In the Short term losses section set the duct friction coefficient to **0.3**
- Ensure the Long term losses option is deselected.
- In the Jacking details section enter an End 1 (anchorage) slippage of **0.005**
- Click the **Generate report** option. This will generate a report in HTML format in the **<Lusas Installation Folder>/Projects** directory summarising the prestress tendon forces.
- Click the **Generate graphs** option. This generates graph datasets in the Treeview for subsequent graphing to be carried out.
- Click **OK**

Equivalent nodal loading is then calculated from the values entered on the Prestress Definition dialog.

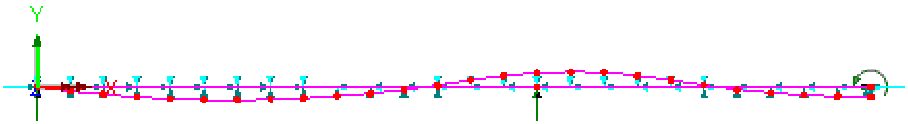
On completion, an HTML report will be opened automatically in the default web browser. This report is also saved into the specified folder. The report is a complete summary of the tendon profile, properties and loading assignments made. On closing the report the tendon loading assignments on the model can be seen for the active loadcase.

Caution. Subsequent modification of the current tendon profile or beam lines will not update the calculated loading until the prestress wizard is run again.

- Click **OK** to dismiss the warning message displayed

Modeller will assign the datasets to the selected Lines representing the concrete beams for the current loadcase and will also add two load datasets to the Treeview.

- In the Treeview double-click the **Utilities** entry and on the Tendon profile tab click the **None** button to stop the display of the tendon profile setting-out details.



Note. Tendon sampling points determine the number of points used when graphing and creating reports on short and long term losses.

Defining Prestress with Short and Long Term Losses

Analysis
Loadcase…

- Change the loadcase name to **Long Term Loss** and click **OK**
- Select **Long Term Loss** from the Treeview with the right-hand mouse button and pick the **Set Active** option.
- Re-select the tendon profile and the two lines representing the bridge, taking care to ensure the tendon profile is selected first.

With the Long Term Loss loadcase active:

Bridge
Prestress Wizard >
Single Tendon >
BS5400…

The values previously entered on the dialog will be retained. Select the **Long term losses** option. Leave all other values as they were set for the short term loss loadcase.

- Click the **Generate report** and the **Generate graphs** options. This generates a report in HTML format and graph datasets in the Treeview for subsequent graphing to be carried out.
- Click **OK** to apply the prestress force to the current loadcase.

Equivalent nodal loading is calculated from the values entered on the Prestress Definition dialog.

- Click **OK** to dismiss the warning message about subsequent modification of the current tendon profile or beam lines not updating the calculated loading until the prestress wizard is run again.

Graphing of Prestress Force in the Tendon

When the Generate graphs option is selected on the Prestress dialog two graph datasets are created. One contains the distance along the beam and the other contains the force in the tendon at each point. To graph these:

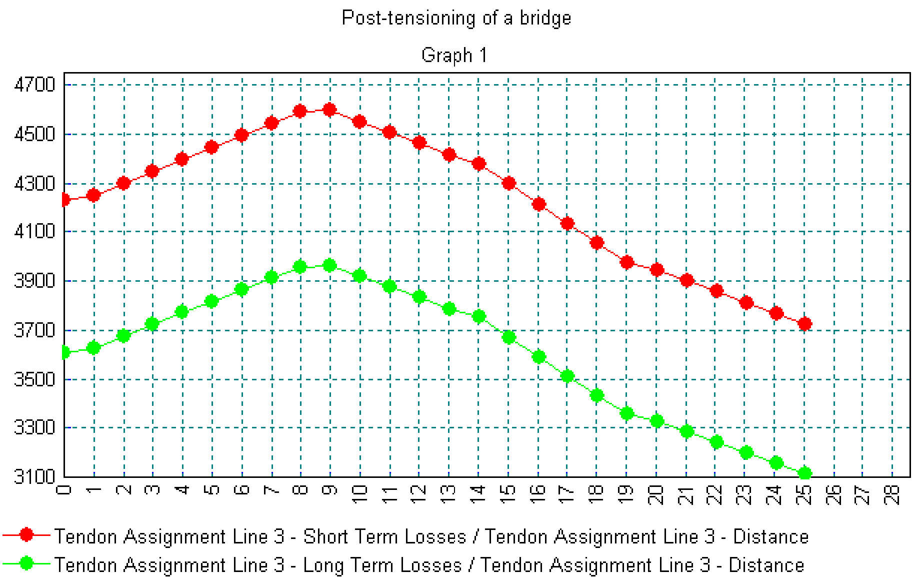
Utilities
Graph Wizard…

- Select the **Specified datasets** option and click **Next**
- From the first drop down list select **Tendon Assignment Line 3 Distance** and from the second select **Tendon Assignment Line 3 - Short Term Losses** and click **Next**
- Click **Finish** to display the graph

Now add a graph of the Force after long term losses:

Utilities
Graph Wizard…

- Select the **Specified datasets** option and click **Next**
- From the first drop down list select **Tendon Assignment Line 3 Distance** and from the second select **Tendon Assignment Line 3 - Long Term Losses** and click **Next**
- Select the **Add to existing graph** option and click **Finish** to update the graph.



 Close the graph window.

Saving the Model

File
Save

Save the model file.

Running the Analysis

Press the **Solve Now** button 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...

LUSAS loadcase results will be added to Treeview.

In addition, 2 files will be created in the Additional Model Data folder, where the model file resides:

- **post_ten.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **post_ten.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **post_ten_modelling.vbs** carries out the modelling of the example up to the point of defining the prestress.

File
New…

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 **post_ten**

File
Script >
Run Script...

- To recreate the model up until the point of defining the prestress, select the file **post_ten_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Now return to the section entitled **Defining Prestess With Short Term Losses** earlier in this example and re-define the tendon properties.

Viewing the Results

Selecting the Results to be Viewed

The results will be loaded on top of the current model and the loadcase results for the first loadcase (the self weight loadcase) will be set to be active by default.

Deformed Mesh Plots

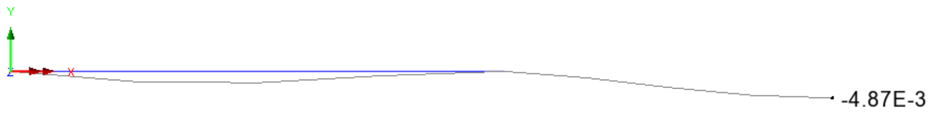
A deformed mesh plot helps highlight any obvious errors with an analysis before progressing to detailed results processing. The deformed shape will usually show up errors in loading or supports and may also indicate incorrect material property assignments (e.g. where the results show excessive displacements).

Deformed Mesh Plot for Self Weight

- If present, turn off the display of the **Geometry, Attributes** and **Mesh** layers from the Treeview.

The **Deformed mesh** layer should be already present in the Treeview, and the View window will be showing the deformed mesh for the self-weight alone.

- With no model features selected, click the right-hand mouse button in the graphics window and select **Values**
- Select entity **Displacement** and component **DY**
- Select the **Values Display** tab.
- Select the **Show values of selection** option.
- Change the number of significant figures to **4** and click **OK**
- Select the node on the centre line to add the value of vertical displacement at mid-span to the display.



Deformed Mesh Plot for Short Term Loss

- In the Treeview right-click on **Short Term Loss** and select the **Set Active** option.

Defining a Basic Combination

Combinations can be created to view the combined effects of the self weight and short term loss loadcases on the structure.

Analyses
Basic Combination…

The combination properties dialog will appear.

- Ensure **post_ten.mys** is selected from the drop down list.
- Select the **Self Weight** and **Short Term Loss** loadcases from those available and click the >> button to add them to the load combination. Change the combination name to **Self Weight and Short Term** and click the **OK** button to finish.

Deformed Mesh Plot for Combination

- In the Treeview right-click on **Self Weight and Short Term** and select the **Set Active** option.

The deformed mesh plot will show the effect of the combined loading on the structure.

Fibre Locations

Force/moment and stress diagrams can be plotted for pre-defined fibre locations on members. All sections created by the Section Property Calculator will have extreme fibre locations pre-defined. These fibre locations can be seen by expanding the Geometric Line entry **Beam properties (RSS D=1 B=2 major y)** in the Treeview, and they can be visualised by double-clicking the Geometric Line attribute name and then selecting the **Visualise** button on the dialog. In doing so, it can be seen that, by default, Fibre S1 is the active fibre and Fibre S1 and Fibre S4 are the upper and lower extreme fibres for which diagram and stress results plots should be created.

Diagram Stress Results at Fibre Locations

- Turn off the display of the **Values** layer from the Treeview.
- In the Treeview right-click on **Short Term Loss** and select the **Set Active** option.
- With no features selected right-click in a blank part of the graphics window and select **Diagrams** to add the diagrams layer to the Treeview.

The diagram properties will be displayed.

- Select **Stress - Thick 2D Beam** results of stress **Sx(Fx, Mz)** in the beam.
- Select the **Diagram Display** tab and deselect the **Peeks only** option.
- Change the angle of the text to **45**, set the number of significant figures to **4** and click **OK** to display the stress diagram at Fibre location S1 for the Short Term Loss loadcase

Diagram Results for Short Term Loss

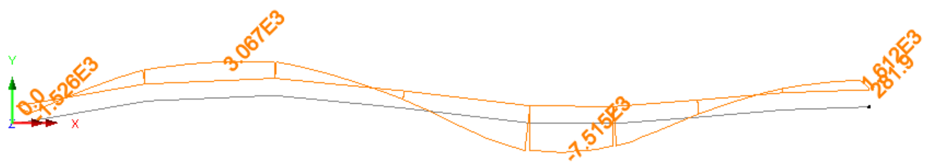


Diagram Results for Combination

- In the Treeview right-click on the combination **Self Weight and Short Term** and select the **Set Active** option to display the stress diagram at Fibre location S1 for the Self Weight and Short Term combination

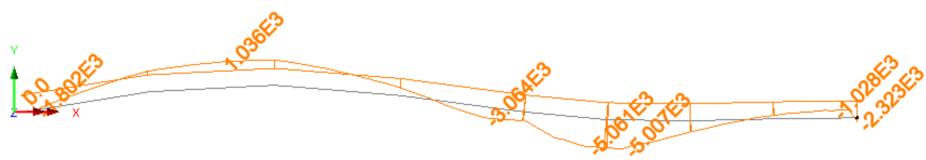


Diagram Results for Combination at Fibre 4

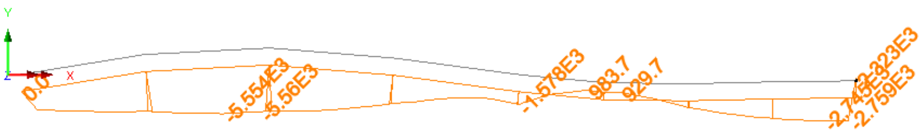
Once a loadcase has been set results can be plotted for any defined fibre location. In this example, because of their location and because of the applied loading, Fibre S1 and

Fibre S2 will give the same results at the top of the beam. Fibre S3 and Fibre S4 will both also give the same results at the bottom of the beam – so only a diagram for Fibre S4 needs to be plotted.

With the combination **Self Weight and Short Term** still set active in the Treeview:

- In the Treeview expand Geometric Line entry **RSS D=1 B=2 major y beam** and right-click on **Fibre S4** and select **Set Fibre Active**.

The diagram display will update to show the stresses at Fibre S4:

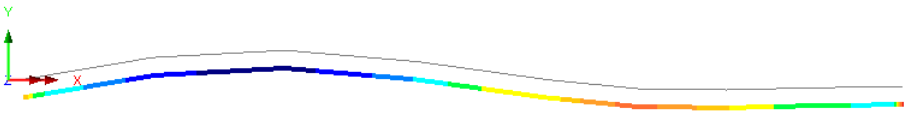


Viewing Results on Fleshed Members

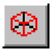
- As an alternative to plotting diagram results, stresses in beams can also be plotted on fleshed members.
- Turn off the display of the **Diagrams** layer from the Treeview.

Contour Results for Short Term Loss

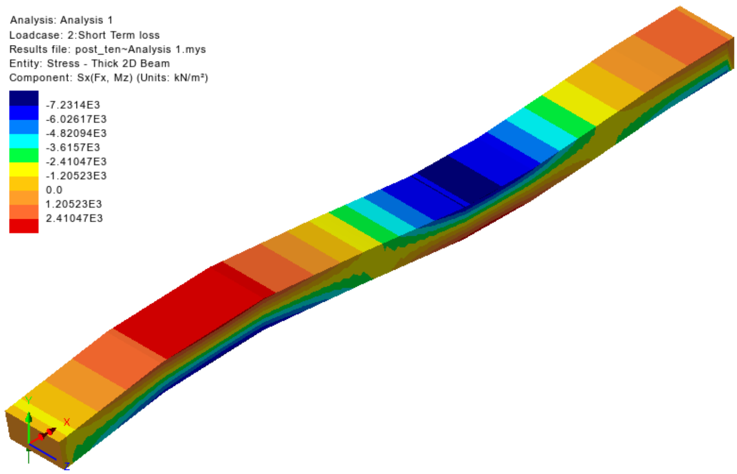
- In the Treeview right-click on **Short Term Loss** and select the **Set Active** option.
- Click in a blank part of the Graphics Window to deselect the members.
- With no features selected right-click in a blank part of the graphics window and select **Contours** to add the contours layer to the Treeview.
- Select **Stress – Thick 2D Beam** results of axial and moment results in the members **Sx(Fx, Mz)** and click **OK**
- Initially the contour properties will be displayed for the fibre location (Fibre S4) that is currently set active.



To see the stresses on the fleshed beam member:

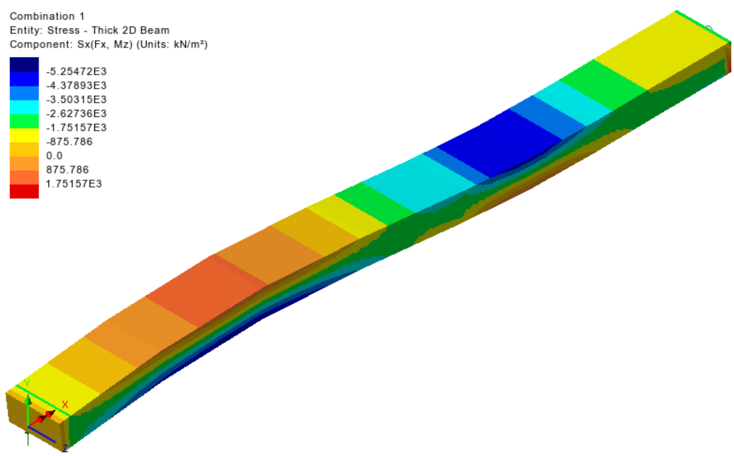
 Select the isometric button.

 Select the fleshing on/off button to visualise the stress results for the fully fleshed cross-section.



Contour Results for Combination

- In the  Treeview right-click on the combination **Self Weight And Short Term** and select the **Set Active** option.



Note. Diagrams and contours may be displayed at the same time to show results for selected fibre locations.

This completes the example.

Discussion

This example illustrates the use of the single tendon prestress wizard, which does not take into account any stressing or unstressing of any other tendons, should they also be present in the model, which in this example they are not.

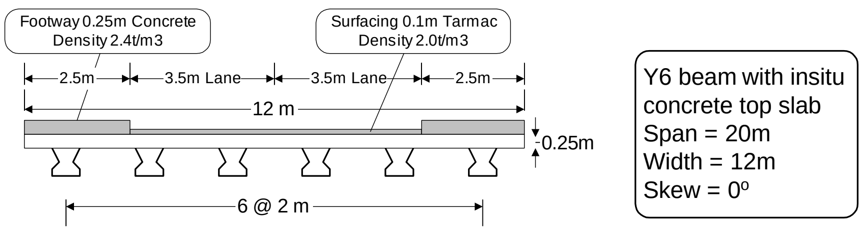
The **Multiple tendon prestress wizard** should generally be used instead of the single tendon wizard since it takes into account elastic shortening due to stressing of other tendons according to the selected design code or user-defined percentage losses, and is more suited to staged construction analysis. If the option to ignore effects due to elastic shortening is chosen the loading computed will be the same as that calculated by the single tendon wizard

Simple Grillage

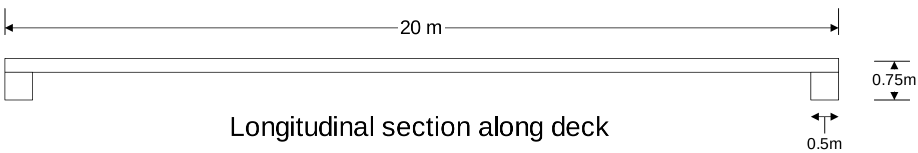
For software product(s):	LUSAS *Bridge*.
With product option(s):	None.

Description

A bridge deck is to be analysed using the grillage method. The geometry is as shown below. All members are made of C50 concrete to BS5400. Section properties of the longitudinal beams and diaphragms are to be calculated using the Section Property Calculator facility.



Cross-section through deck



Longitudinal section along deck

The structure is subjected to four loadcases: Dead load, Superimposed dead load, Lane loads in both lanes (UDL and KEL), and an abnormal load (HB) in the lower notional lane with a lane load (UDL and KEL) in the upper lane.

Units of kN, m, t, s, C are used throughout when modelling the grillage.

Units of N, mm, t, s, C are used in calculating the section properties of selected components.

Objectives

The required output from the analysis consists of:

- ❑ A deformed shape plot showing displacements caused by the imposed loading
- ❑ A diagram showing bending moments in the longitudinal members for the design load combination

Keywords

2D, Y6 Precast Section, Section Property Calculation, Local Library, Grillage, Basic Load Combination, Smart Load Combination, Enveloping, Deformed Mesh, Bending Moment Diagram, Print Results Wizard

Associated Files

- ❑ **grillage_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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.

Before creating the grillage model the section properties of the longitudinal beams and end diaphragms are to be computed using the section property calculator and stored for future use. Calculation of section properties requires section geometry to be drawn/defined in the XY plane.

Creating the Longitudinal Beam Model

- Enter the file name as **Y6**
- Use the **Default** working folder.
- Enter the title as **Y6 Precast Beam**
- Select model units of **N,mm,t,s,C** from the drop down list provided.
- Ensure the timescale units are **Seconds**

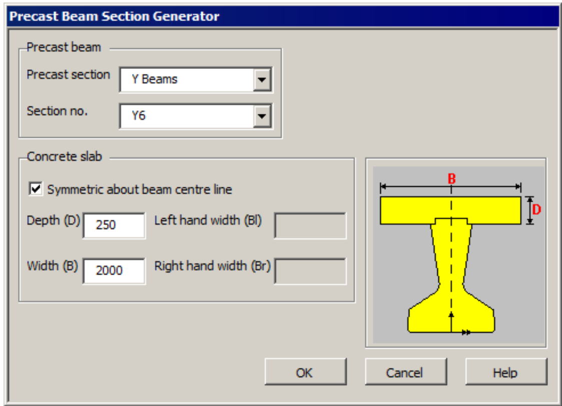
- Ensure the analysis type is **Structural**
- Ensure the startup template is set as **None**
- Ensure the **Vertical Z Axis** option is selected
- Click the **OK** button.

Defining Longitudinal Beam Geometry

Utilities
Section Property Calculator >
Precast Section…

- From the **Y Beams** section series select a **Y6** section.
- Specify a slab depth of **250**
- Enter a slab width of **2000**
- Click the **OK** button.

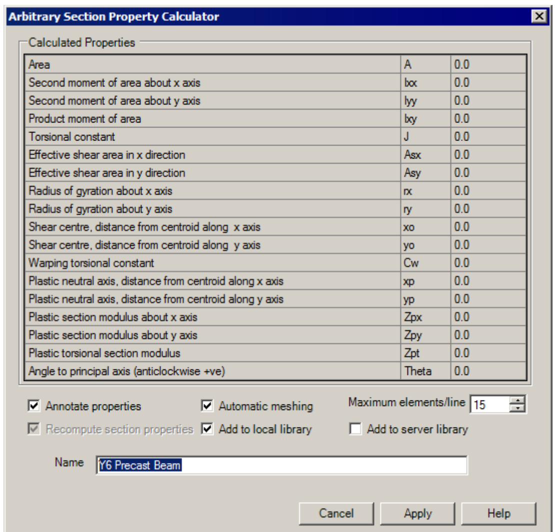
The section will now be drawn.



Calculating Longitudinal Beam Section Properties

Utilities
Section Property Calculator >
Arbitrary Section…

- Press **Ctrl** and **A** to select the two surfaces defining the Y6 section
- Select the option **Add to local library** so the calculated properties will be available from the local library when required.
- Name the section **Y6 Precast Beam**.
- Click the **Apply** button to compute the section properties. These will be displayed in the greyed text boxes on the right hand side of the dialog and written to the local library.



- Click the **Cancel** button to close the dialog.

Creating the End Diaphragm Model

File
New…

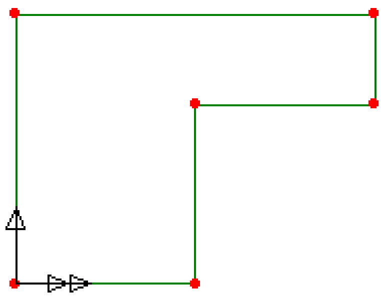
Create a new model and discard the changes to the previous model.

- Enter the file name as **diaphragm**
- Use the default working folder.
- Enter the title as **End diaphragm**
- Ensure units of **N,m,kg, s,C** are selected.
- Ensure the **Structural** user interface is selected
- Ensure the startup template is set as **None**
- Ensure the **Vertical Z Axis** option is selected
- Click the **OK** button.

Defining Diaphragm Geometry

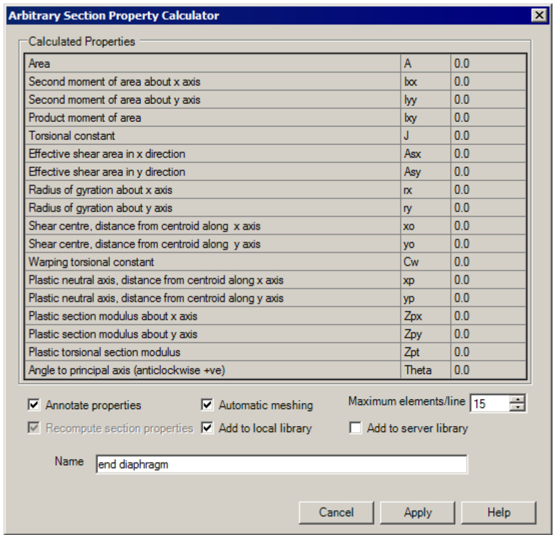
Geometry
Surface >
Coordinates…

Enter coordinates of **(0,0), (0.5,0), (0.5,0.5), (1,0.5), (1,0.75), (0,0.75)** to define a surface representing the end diaphragm and slab (which is to be represented by the end beam on the grillage model) and click **OK**



Calculation of End Diaphragm Section Properties

Utilities
Section Property Calculator >
Arbitrary Section…



- Select the option **Add local library** and name the section **End Diaphragm**.
- Click the **Apply** button to compute the section properties. These will be displayed in the greyed text boxes on the right hand side of the dialog and written to the local library file for future use.
- Click the **Cancel** button to close the dialog.

Creating the Grillage Model

Now that the beam and diaphragm properties have been calculated the grillage model can be created.

File
New…

Create a new model and discard the changes to the previous model.

- Enter the file name as **grillage**
- Enter the title as **Simple grillage analysis**
- Set the units as **kN,m,t,s,C**
- Ensure the **Structural** user interface is selected
- Ensure the startup template is set as **None**
- Ensure the **Vertical Z Axis** option is selected.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the undo button to correct any mistakes made since the last save.

Using the Grillage Wizard

In this example the grillage wizard is used to generate a model of the bridge deck. The grillage wizard defines the grillage geometry, assigns grillage elements to each of the lines, and assigns supports to the end beams. It also creates Groups to ease member identification and the application of section properties.

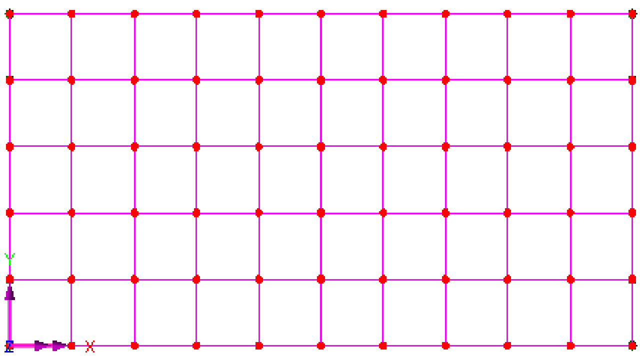
Note. It is difficult to make absolute recommendations as to how individual structures should be modelled using a grillage. A few basic recommendations are however valid for most models:

a) Longitudinal beams within the grillage should be coincident with the actual beams within the structure.

b) Transverse beams should have a spacing which is similar or greater than that of the longitudinal beams and the total number of transverse beams should be odd to ensure a line of nodes occur at mid span.

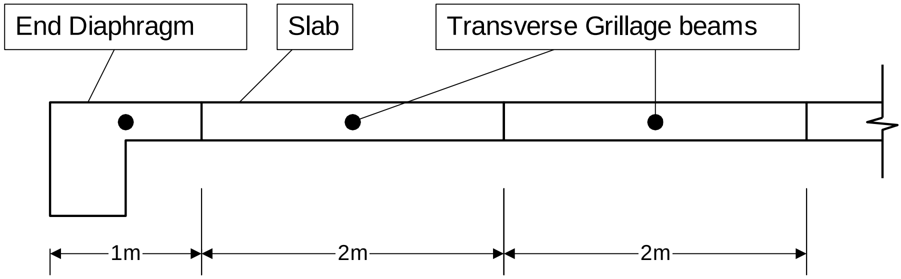
Bridge
Grillage Wizard…

- Select the **Set defaults** button
- Ensure **Slab deck** is selected and click **Next**
- The grillage is **Straight** with **0** degrees skew so click **Next** again
- Enter the grillage width as **10** and the number of internal longitudinal beams as **6 .** Select **evenly spaced** longitudinal beams and click **Next**
- Leave the number of spans set as **1**
- Enter the length of span as **20** and the number of internal transverse beams as **9**
- Click **Finish** to generate the grillage model.



Calculation of Transverse Beam Section Properties

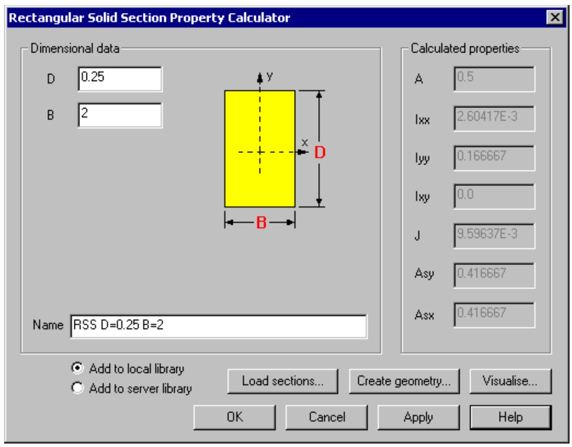
The internal transverse beams each represent 2m of slab so the section properties are computed for an equivalent solid rectangular section.



Longitudinal Section

Utilities
Section Property Calculator >
Rectangular Sections > Solid

- Enter a depth of D = **0.25**
- Enter a width of B = **2**
- The section properties will be displayed in the greyed text boxes on the right hand side of the dialog.
- Note the torsion constant (J) is calculated as (0.0096). This is based on beam theory and is not appropriate to represent a slab in a grillage analysis. It will be adjusted later in the example.



- A section name of RSS D=0.25 B=2 is automatically created from the entered dimensions.
- Ensure the **Add to local library** option is selected and click **OK** to add the properties to the local section library.

Modifying Section Properties for Grillage Analysis

- When representing an isotropic slab using a grillage model, the effective torsion constant (per unit width) can be shown to be $c = d^3/6$ (per unit width). It is therefore common practice to assume 50% of the value calculated using beam theory for a wide slab-like beam.

- In this example the transverse members represent only the slab and therefore their torsion constant can be entered as c = $bd^3/6$ (i.e. 50% of the section library value). The longitudinal members represent the precast beam and associated width of slab, however, and therefore this reduction is only applied to the proportion of the torsion constant exhibited by the slab.
- When the transverse 0.25m deep 2m wide slab and the Y6 precast (longitudinal) beams are selected from the local section library their section properties will be adjusted to reflect this.

Adding Section Library Items to the Treeview

First, add the transverse slab section geometric attribute:

Attributes
Geometric >
Section Library...

- Select **Grillage** as the usage.
- Select the **User Sections** library from the upper-right drop-down list.
- Select the **Local** library type.
- Select the **RSS D=0.25 B=2** entry from the drop down list.
- Select the **Enter Properties** option and reduce the torsion constant to **0.005 (**as discussed: $c = bd^3/6$).
- Enter an Attribute name of **Transverse Slab major z** and click **OK** to add the section properties to the Treeview.
- Next, add the Y6 beam geometric attribute:

Attributes
Geometric >
Section Library...

- Select **Grillage** as the usage.
- Select the **User Sections** library from the upper drop-down list.
- Select the **Local** library type.
- Select **Y6 Precast Beam** from the name in the drop-down list.
- Select the **Enter Properties** option and modify the torsion constant to be **0.020** (The computed value of the precast beam alone, 0.0143, plus the reduced contribution from the slab; c = bd3/12 = 0.005).
- Enter an Attribute name of **Y6 Precast Beam major z** and click **OK** to add the section properties to the Treeview.

Note. Even though the Y6 beam was defined in millimetres the units can be extracted from the library in metres. The units will be set to metres automatically as these were the units selected on the New Model dialog.

- Lastly, define the end diaphragm geometric attribute:

Attributes
Geometric >
Section Library...

- Select **Grillage** as the usage.
- Select the **User Sections** library from the upper drop-down list.
- Select the **Local** library type.
- Select **End diaphragm** from the drop down list.
- Click **OK** to add the section properties to the Treeview.

Note. When a section is used from a library without amending section properties or the section's orientation (as just done for the End diaphragm) the library name is appended automatically to the automatic identifying name given in the dialog.

Assigning Geometric Properties to the Grillage Members

Use the Isometric View button to rotate the model so that the following assignment of the geometric properties can be seen.

Ensure the fleshing button is depressed in the toolbar menu.

Longitudinal members

The Y6 beam section properties are to be assigned to all the longitudinal members.

- In the Treeview select the **Y6 Precast Beam major z** entry and click on the copy button.

- In the Treeview select the **Edge Beams** group and click the paste button to assign the Y6 beam section properties to the edge beams.

Confirmation of the assignment will appear in the text window.

- Select the **Longitudinal Beams** group and click the paste button again to assign the Y6 beam section properties.

Transverse slab members

The slab section properties are assigned to the transverse members in a similar fashion.

- In the Treeview select the **Slab D=0.25 B=2 (m) major z** entry and click on the copy button.

- In the  Treeview select the **Transverse Beams** group and click the  paste button to assign the slab section properties.

To clarify the display prior to assigning the diaphragm members the extent of the fleshing of each grillage member can be modified as follows:

- In the  Treeview, double click on the **Attributes** name, click the **Geometry** tab and press the **Settings** button. Select **Automatic** cross-section end shrinkage and click **OK** to update the display

Diaphragm members

The diaphragm section properties are assigned to the end diaphragms in a similar fashion.

- In the  Treeview select the **End diaphragm (m) major z** entry and click on the  copy button.

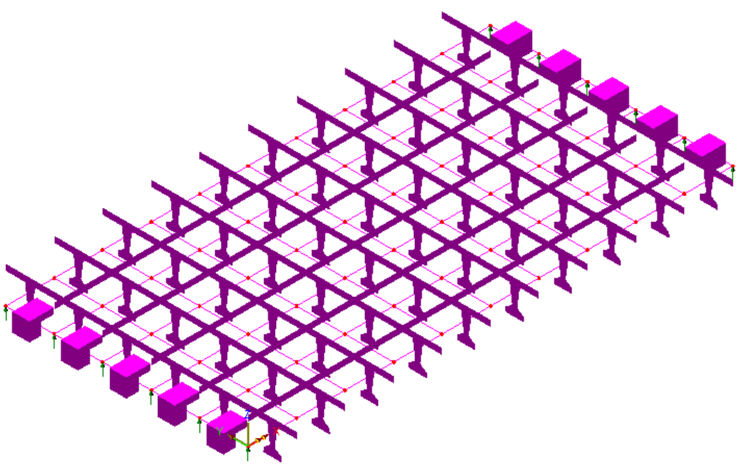
- In the  Treeview select the **End Diaphragms** group and click the  paste button to assign the slab section properties.

From the fleshed image it can be seen that the end diaphragm members for the far end are incorrectly displayed, and as a result the line directions of the lines to which they have been assigned need to be reversed.

- Select the 5 lines at the far end of the grillage

Geometry
Line >
Reverse…

This reverses the line directions of the selected lines to give the following image.



Note. You can also check assignments by right-clicking on a group name in the Treeview and selecting Select Members.

Defining the Material

Note. In this example a single material property will be used. If deflections and rotations are of interest then separate analysis runs with short and long term properties may be appropriate.

Attributes
Material >
Material Library…

- Select material **Concrete BS5400** from the drop down list, and **Short Term C50** from the grade drop down list.
- Click **OK** to add the material dataset to the Treeview.
- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material dataset **Iso1 (Concrete BS5400 Short Term C50)** from the Treeview onto the selected features and assign to the selected Lines by clicking the **OK** button.

Loading

In this example seven loadcases will be applied to the grillage. These will be enveloped and combined together to form the design combination.

Renaming the Loadcases

- In the Treeview expand **Analysis** 1 and right click on **Loadcase 1** and select the **Rename** option.
- Rename the loadcase to **Dead Load** by over-typing the previous name.

Dead Load

Dead load is made up of the self-weight of the structure, which is defined as an acceleration due to gravity.

Note. When a bridge deck is modelled by a grillage the slab area is included by both the longitudinal and transverse members. This means that self-weight should only be applied to the longitudinal members.

Turn off the display of the fleshed members

Bridge
Bridge Loading >
Gravity

- A load dataset named **BFP1 (Gravity -ve Z)** will be added to the Treeview.

- In the Treeview select the **BFP1 Gravity -ve Z** entry and click on the copy button.
- In the Treeview select the **Edge Beams** group and click the paste button.
- With the **Assign to lines** and **Single Loadcase** options selected click the **OK** button to assign the gravity loading to the **Dead Load** loadcase.

The self-weight dead load will be displayed on the edge beams.

- In the Treeview select the **Longitudinal Beams** group and click the paste button to gravity loading.
- With the **Assign to lines** and **Single Loadcase** options selected click the **OK** button to assign the gravity loading to the **Dead Load** loadcase.

The self-weight dead load on the internal longitudinal beams will be added to the display.

Superimposed Dead Load

Superimposed dead load consists of the surfacing loads. These represent the self-weight of the footways and the surfacing on the road.

Bridge
Bridge Loading >
Surfacing…

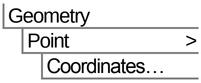
Specify the surfacing loading for the footway:

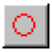
- Leave the density as **2.4**
- Change the thickness to **0.25**
- Set the length to **20** and set the width to **2.5**
- Leave the skew angle as **0** and the origin as **Centre**
- Click the **Apply** button to add the loading to the Treeview.

Now specify the tarmac highway surfacing load:

- Change the density to **2.0**
- Change the thickness to **0.1**
- Leave the length as 20 but change the width to **3.5**
- Click the **OK** button to add the loading to the Treeview.

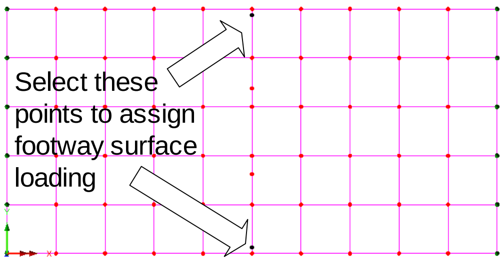
Discrete point and patch loads are positioned by assigning them to points which do not have to form part of the model.



 Enter the coordinates of the mid point of each footway and each notional lane **(10,0.25), (10,3.25), (10,6.75), (10,9.75)** and click **OK**

On the status bar at the bottom of the display, click the Z axis button to return to a global Z direction view.

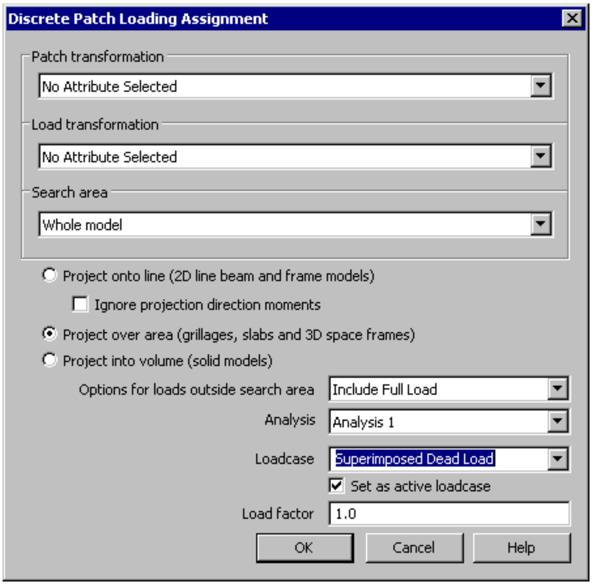
- Select the points at the centre of each footway by holding the **Shift** key down to select points after the initial selection.



- Drag and drop the discrete loading dataset **Pch2 Surfacing 20mx2.5m Skew=0.0 Thickness=0.25m Density=2.4 tm^3** onto the selected points.

- Select **Include Full Load** from the drop down list. This will ensure the portion of the pavement load which is overhanging the edge of the grillage model is applied to the edge beams.

- Enter **Superimposed Dead Load** as the Loadcase and click **OK** to assign the loading.



The loading will be visualised.

Patch load divisions

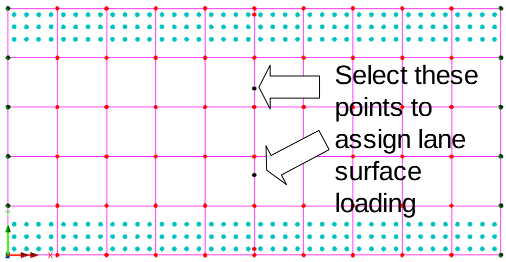
The Patch divisions object seen in the  Treeview controls the number of discrete point loads used to represent a patch load. By default a specified number of 10 divisions is used. However, for this example, and for most real life uses a greater number of divisions is required to accurately reflect the surfacing loading.

- In the  Treeview double-click the **Patch divisions** object

- Select the **Distance between loads** option and specify **0.5**. Click **OK** to update the patch divisions.

Now the road surfacing is to be assigned:

- Select the two points at the centre of each notional lane.
- Drag and drop the discrete load dataset **Pch3 Surfacing 20x3.5 Thickness=0.1 Density=2** from the Treeview onto the selected points.



- Leave the loading option for loads outside the search area set as **Exclude All Load** because for this load type it is irrelevant whether include or exclude is used since the load length, which is positioned centrally, is the same length as the span length
- Select **Superimposed Dead Load** from the Loadcase drop down list.
- Click **OK** to assign the road surfacing load.

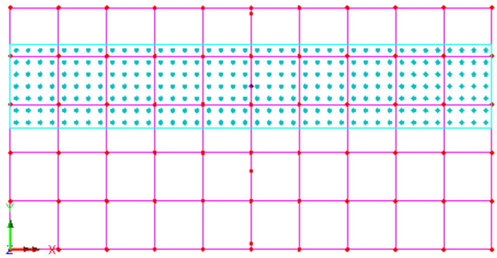
Vehicle Load Definition

Bridge
Bridge Loading >
United Kingdom…

HA loading is to be applied to each notional lane in loadcases 3 and 4. These loads are defined using the UK vehicle loading definitions supplied with LUSAS *Bridge*.

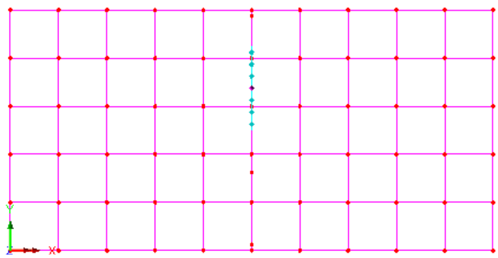
- Select the **Lane load (HA load)** button.
- Select loading code **BD 37/88**, change the length to **20** and select the **OK** button to add the load dataset to the Treeview.
- Select the **Knife edge load (KEL load)** button.
- Leave the notional width as **3.5** and the intensity as **120** and click the **OK** button to add the load dataset to the Treeview.
- Select the **Abnormal load (HB vehicle)** button.
- With the axle spacing set to **6** and **45** units of HB load select the **OK** button to add the load dataset to the Treeview.
- Click the **Close** button to close the UK bridge loading dialog.

Assigning HA Loading



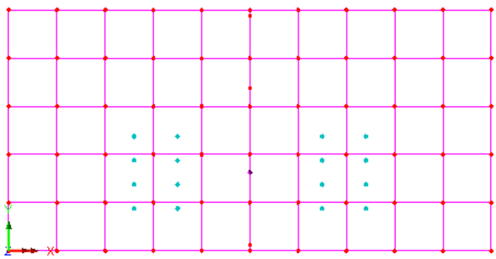
- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pch4 HA BD37/88 20m x 3.5m Skew=0 (Centre)** from the Treeview.
- Enter **HA upper** as the Loadcase, leave other values as their defaults, and click **OK**
- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pch4 HA BD37/88 20m x 3.5m Skew=0 (Centre)** from the Treeview.
- Enter **HA lower** as the Loadcase, leave other values as their defaults, and click **OK**

Assigning KEL Loading



- Select the point defined at the centre of the upper notional lane.
- Drag and drop the dataset **Pch5 KEL 120kN Width=3.5m Offset=0 Skew=0 (Centre)** from the Treeview.
- Enter **KEL upper** as the Loadcase and click **OK**
- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pch5 KEL 120kN Width=3.5m Offset=0 Skew=0 (Centre)** from the Treeview.
- Enter **KEL lower** as the Loadcase, leave other values as their defaults, and click **OK**

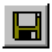
Assigning Abnormal HB Loading



- Select the point defined at the centre of the lower notional lane.
- Drag and drop the dataset **Pnt6 HB 6m spacing 45 units** from the Treeview onto the selected point.
- Enter **HB lower** as the Loadcase, leave other values as their defaults, and click **OK**

Save the model

File
Save

 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 solve.

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 Treeview.

In addition, 2 files will be created in the directory where the model file resides:

- ❑ **grillage.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **grillage.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- ❑ **grillage_modelling.vbs** carries out the modelling of the example.

File
New…

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 **grillage** and click **OK**

File
Script >
Run Script...

To recreate the model, select the file **grillage_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

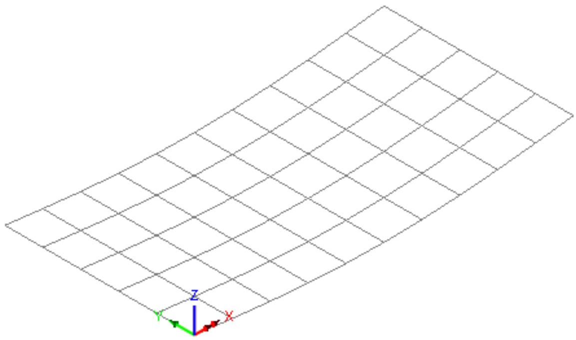
Analysis loadcase results are present in the Treeview, and results for the first load case will be set to be active by default.

Deformed Mesh and Summary Plot

A deformed mesh plot helps highlight any obvious errors with an analysis before progressing to detailed results processing. The deformed shape will usually show up errors in loading or supports and may also indicate incorrect material property assignments (e.g. where the results show excessive displacements).

- If present, turn off the display of the **Geometry, Attributes** and **Mesh** layers in the Treeview.

- The **Deformed mesh** layer should be turned on by default in the Treeview.. Double click its name and select the **Specify factor** option and enter **300** Click the **OK** button to display the deformed mesh for loadcase 1.



If necessary use the Isometric View button to rotate the model.

- Step through each of the loadcases in the Treeview using the **Set Active** option and check each deformed shape looks correct for the supposed loading.

Z: N/A Return to the view from the global Z direction.

Defining Envelopes and Combinations

The design combination will consist of all dead loads and an envelope of all live loads factored by the appropriate adverse or relieving factor.

Loadcase name	Adverse Factor		Relieving Factor
	γ_{f1}	γ_{f3}	
Dead Load	1.15	1.10	**1.0**
Super Dead Load	1.75	1.10	**1.0**
HA alone	1.5 (*)	1.10	**0**
HA with HB	1.3 (*)	1.10	**0**

Table 1

Note. According to BS5400 part 1 two safety factors should be applied to adverse loading. γ_{f1}accounts for the uncertainty in the applied loading and γ_{f3} is a safety factor to allow for modelling inconsistencies / inaccuracies.

Note. (*) When designing to BD 37/88 the HA lane loading factors also include additional lane factors. For a two lane structure these are noted in the tables which follow.

Defining a Basic Load Combination 1

Analyses
Basic Combination…

A basic load combination to investigate HA and Knife Edge loads will be defined.

On the Basic Combination dialog:

Add loadcases (**3) HA upper,** (**4) HA lower, (5) KEL upper, (6) KEL lower**

Note. To add a number of loadcases all together select the first loadcase in the list, hold down the **Shift** key and select the last loadcase in the list (scrolling down the list if necessary) and click the button.

Each loadcase selected then needs a corresponding lane factor to be specified.

- Select the **Grid** button and update the factors as shown in **Factor** column of the following loadcase combination table.

Loadcase name	**Load Factor**		**Lane Factor**	**Lane Factor to be used**
	γ_{f1}	γ_{f3}		
HA upper	1.5	1.10	0.956	**1.6**
HA lower	1.5	1.10	0.956	**1.6**
KEL upper	1.5	1.10	0.956	**1.6**
KEL lower	1.5	1.10	0.956	**1.6**

Table 2

- Click **OK** to return to the combination properties dialog.
- Change the combination name to **HA + KEL both lanes**
- Click **OK** to save the combination definition.

Defining a Basic Load Combination 2

A basic load combination to investigate HA, HB and Knife Edge loads will also be defined.

Analyses
Basic Combination…

On the Basic Combination dialog:

>> Add result loadcases (**3) HA upper, (5) KEL upper, (7) HB lower**

Each loadcase selected needs the factor to be specified.

- Select the **Grid** button and update the factors as shown in **Factor** column of the following loadcase combination table.

Loadcase name	Load Factor		Lane Factor	Lane Factor to be used
	γ_{f1}	γ_{f3}		
HA upper	1.3	1.10	0.956	**1.4**
KEL upper	1.3	1.10	0.956	**1.4**
HB lower	1.3	1.10	0.956	**1.4**

Table 3

- Click **OK** to return to the combination properties dialog.
- Change the combination name to **HB lower, HA + KEL upper**
- Click **OK** to save the combination definition.

Enveloping the Basic Live Load Combinations

Analyses
Envelope

On the Properties dialog:

>> Add combinations (**8)HA+KEL both lanes** and (**9)HB lower, HA+KEL upper**

- Change the envelope name to **Live Load Envelope**
- Click **OK** to save the envelope definition.

Note. When either a Max or Min smart combination or envelope is modified the corresponding Max and Min dataset will be updated automatically.

Defining a Smart Combination

Smart load combinations take account of adverse and relieving effects for the loadcase being considered. The Self-weight, Superimposed Dead Load, and the Live Load Envelope will all be combined using the Smart Load Combination facility to give the design combination.

Analyses
Smart Combination…

On the Smart Combination dialog:

>> Add loadcase **(1)Dead load** and (**2)Superimposed DL** to the Included panel.

>> Add **(10)Live Load Envelope (Max)** and **(11)Live Load Envelope (Min)** to the Included panel.

Each loadcase/envelope selected needs the permanent and variable factors to be specified.

- Select the **Grid** button and update the **Permanent Factor** for the Live Load Envelopes to **0** and ensure the **Variable Factor** for all loadcases are as shown in the table.

Loadcase name	**Variable Factor**		**Permanent (relieving) Factor**	**Variable Factor to be used**
	γ_{f1}	γ_{f3}		
(1)Dead Load	0.15	0.10	**1.0**	**0.265**
(2)Superimposed DL	0.75	0.10	**1.0**	**0.925**
(10)Live Load Envelope (Max)	-	-	**0**	**1.0**
(11)Live Load Envelope (Min)	-	-	**0**	**1.0**

Table 4

Note. In this table the permanent factor is based upon the relieving factor from Table 1. The variable factor for Dead Load and for Superimposed Dead Load is based upon the product of the adverse factors for both from Table 1 minus the permanent (relieving) factor. The live load envelopes have already been factored in previous load combinations (Tables 2 and 3) and, as a result, only a unity factor is applied as a variable factor.

- Click **OK** to return to the combination properties dialog.
- Change the envelope title to **Design Combination**
- Click **OK** to save the smart combination.

Selecting Loadcase results

- In the Treeview right-click on **Design Combination (Max)** and select the **Set Active** option.
- Select entity **Force/Moment – Thick Grillage** results of component **My** from the drop down lists to combine and apply the variable factors based on the moments about the Y axis and click the **OK** button.

Note. When activating a smart combination the selected component is used to decide if the variable factor should be applied. (The variable component is only applied if the resulting effect is more adverse) Viewing results for a component other than the selected component will result in display of the associated values (coincident effects). When the results of an envelope or smart combination are printed the column used to compute the combination or envelope is denoted with an asterisk in the column header.

Selecting Members for Results Processing

Results are to be plotted for selected longitudinal members of the grillage only. The grillage wizard automatically creates groups which are useful in the results processing.

- In the Treeview select the **Longitudinal Beams** with the right hand mouse button and pick **Set as Only Visible**
- In the Treeview select **Edge Beams** with the right hand mouse button and pick **Visible**
- Turn off the display of the **Deformed mesh** layer in the Treeview.
- Add the **Mesh** layer to the Treeview and click **OK** to accept the default properties.

Bending Moment Diagram

A plot showing the bending moment from the design combination is to be displayed for the selected members of the grillage.

- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Diagrams** option to add the diagram layer to the Treeview.

The diagram properties will be displayed.

- Select entity **Force/Moment – Thick Grillage** results of component bending moment **My**
- Select the **Diagram Display** tab

- Select the **Label values** option.
- Select the **Orientate flat to screen/page** option.
- Set the **Label font** to size **12**
- Enter an **Angle** of **45** degrees
- Set the **Number of significant figures** to **4**
- Set **% of element length** to **60**
- Click the **OK** button to display the bending moment diagram.

Note. Results plots which are to be printed are best created in the page layout view. This provides a view that will appear similar to the printed output. Labels however may however be difficult to read in the page layout view since they reflect the size of the labels on the final printout. When this situation arises the zoom facility may be used to examine labels of interest more closely.

View
Page Layout Mode

- Switch to page layout view.

Utilities
Annotation >
Window border

- Add a border to the page which contains the title, date and version of the LUSAS software in use.

File
Page Setup…

- Ensure the orientation is set to **Landscape**. Change the page margins to enable the annotation to be added without obscuring the display. Set the left margin to **50**, the right margin to **15** and the top and bottom margins to **10.** Click **OK**

Utilities
Annotation >
Window Summary

A summary of results will be added to the graphics window showing the loadcase name, diagram component, maximum and minimum diagram values, and element numbers in which the maximum and minimum moments occurs.

- Select the annotation by clicking over any piece of text and then drag it the summary text to an appropriate location on the plot.

Note. The location of any model feature, element or node can be found by using the Advanced Selection facility. This can be used to find the location of the maximum and minimum results values as the element number is output in the window summary text.

As well as creating a results plot, results can be printed for saving or copying to a spreadsheet using standard Windows copy and paste.

Printing results for the active loadcase

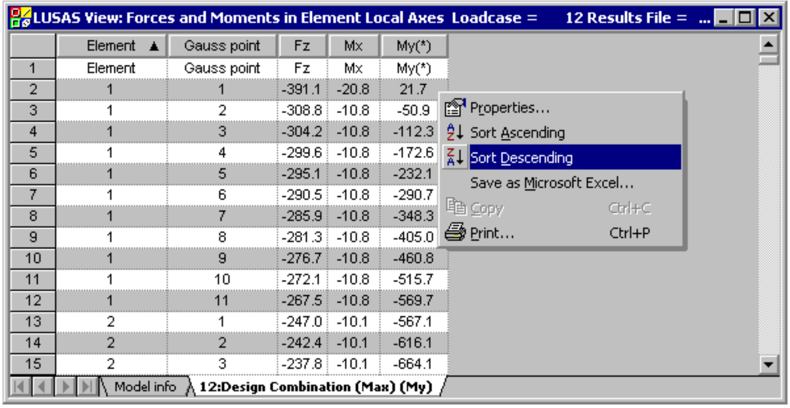
Utilities
Print Results Wizard…

Results values may be output to the screen in a tabular listing format for the active loadcase or for any selected loadcase. With Design Combination (Max) currently set active:

- On the Results Wizard dialog ensure Loadcases **Active** is selected and click **Next**.

- On the Results Entity dialog, select entity **Force/Moment – Thick Grillage,** type **Component** and location **Gauss Point**. Ensure Primary component **My** is selected.

- Set the number of decimal places to **1** and click the **Finish** button to display the results.



	Element ▲	Gauss point	Fz	Mx	My(*)
1	Element	Gauss point	Fz	Mx	My(*)
2	1	1	-391.1	-20.8	21.7
3	1	2	-308.8	-10.8	-50.9
4	1	3	-304.2	-10.8	-112.3
5	1	4	-299.6	-10.8	-172.6
6	1	5	-295.1	-10.8	-232.1
7	1	6	-290.5	-10.8	-290.7
8	1	7	-285.9	-10.8	-348.3
9	1	8	-281.3	-10.8	-405.0
10	1	9	-276.7	-10.8	-460.8
11	1	10	-272.1	-10.8	-515.7
12	1	11	-267.5	-10.8	-569.7
13	2	1	-247.0	-10.1	-567.1
14	2	2	-242.4	-10.1	-616.1
15	2	3	-237.8	-10.1	-664.1

Note. When the active loadcase is an envelope or smart combination the results printed will show the primary component (**My** in this case) marked with an asterisk.

Note. Hovering over the contents of a cell will display a datatip showing location information and an associated value.

Saving printed results to a spreadsheet

When the Printed Results window is shown a context menu can be displayed allowing the printed results to have their number of significant figures or decimal places changed, be sorted in ascending or descending order, be saved to a spreadsheet or copied for pasting elsewhere.

- Right-click inside the Printed Results window and select **Save as Microsoft Excel…**
- Enter a file name of **grillage_results**
- Ensure the save option **All tabs** is selected and click **Save**.

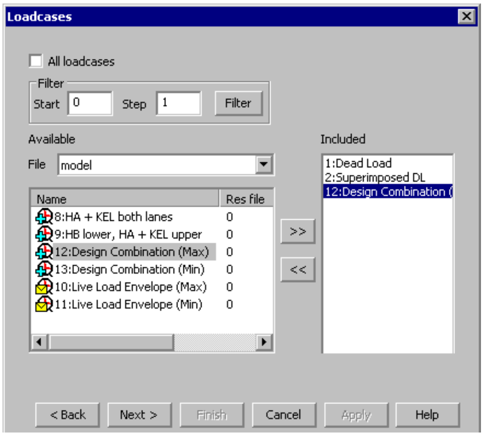
Note that Microsoft Excel may impose limitations on the length of tab name permitted.

Printing results for selected loadcases

Results values may be printed for selected loadcases. To illustrate usage:

Utilities
Print Results Wizard…

- On the Results Wizard dialog ensure Loadcases **Selected** is chosen and click **Next**.
- On the Loadcases dialog select results files or load combinations from the model and results panels and press the Add to button  to add them into the Included panel and click **Next**.



- On the Results Entity dialog, select entity **Force/Moment – Thick Grillage,** type **Component** and location **Gauss Point**. Ensure Primary component **My** is selected.
- Set the number of decimal places to **1** and click the **Finish** button to display the results.

Save the model

File
Save

Save the model file.

Note. If the model file is saved after results processing, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

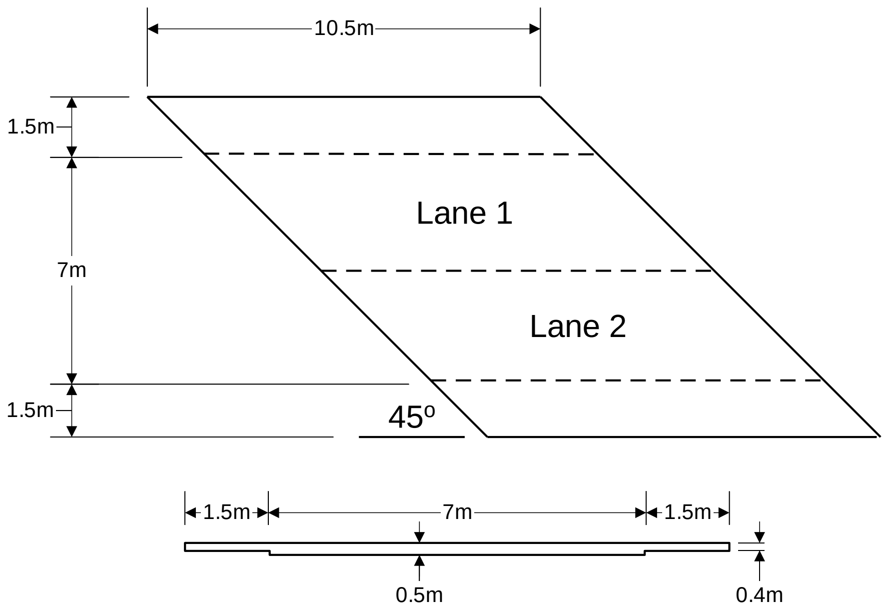
This completes the example.

Simple Slab Deck

For software product(s):	LUSAS *Bridge*.
With product option(s):	None.

Description

A simply supported bridge structure consisting of an in-situ slab deck is to be analysed. The geometry of the deck is as shown. The slab has 0.5m thick carriageways and 0.4m thick footways.



The structure is subjected to loading defined in the Departmental Standard BD 37/88 Loads for Highway Bridges.

The units of the analysis are kN, m, t, s, C throughout.

Objectives

The required output from the analysis consists of:

- ❑ A deformed shape plot showing displacements caused by the imposed loading
- ❑ A diagram showing the bending moments in the slab

Keywords

2D, Shell, Bridge Loading, Moving Load, Eccentricity, Basic Load Combination, Load Envelope, Peak Values, Deformed Mesh, Contours, Graphing, Wood Armer

Associated Files

- ❑ **bridge_slab_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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 the file name as **bridge_slab**
- Use the **Default** working folder.
- Enter the title as **Simple Bridge Slab Analysis**
- Change the model units to **kN,m,t,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the Startup template **Standard**.
- Select the **Vertical Z axis** option and click **OK**

Note. It is useful to save the model regularly as the example progresses. Use the undo button to correct a mistake. The undo button allows any number of actions since the last save to be undone.

Defining the Material

Attributes
Material >
Material Library…

- Select material **Concrete** grade **Ungraded** from the drop down list and click **OK** to add the material attribute to the Treeview.
- Select the **Iso1 (Concrete Ungraded)** attribute from the Treeview with the right-hand mouse button and select **Set Default** to make concrete the default material for all newly created features.

Feature Geometry

Geometry
Line >
Coordinates...

Enter coordinates of (**0, 0**), (**10.5, 0**) to define the lower Line. Use the **Tab** key to move to the next entry field on the dialog. With all the coordinates entered click the **OK** button.

- Select the line.

Geometry
Surface >
By Sweeping...

Sweep the line through a translation of X= **-1.5**, Y= **1.5**, Z = **0** and click **OK** to create a Surface which defines the lower footway.

- Select the Line at the top of the footway.

Geometry
Surface >
By Sweeping...

Sweep the Line through a translation of X= **-7**, Y= **7**, Z= **0** and click **OK** to create a Surface which defines the carriageway.

- Select the Line at the top of the carriageway.

Geometry
Surface >
By Sweeping...

Sweep the Line through a translation of X= **-1.5**, Y= **1.5**, Z= **0** and click **OK** to create a Surface which defines the upper footway.

Meshing

The slab deck will be meshed using a Surface mesh. The number of elements in the mesh can be controlled, by defining Line meshes, on the Lines defining the Surface boundary. Other methods of controlling mesh density are also available.

Defining a Surface Mesh

Attributes
Mesh >
Surface...

- Select a **Thick shell** element type which is **Quadrilateral** in shape with a **Linear** interpolation order.
- Ensure a **Regular Mesh** that will allow **Transition**, **Irregular** and **Automatic meshing** if possible.
- Enter the mesh attribute name as **Thick Shell**
- Click the **OK** button to add the mesh attribute name to the Treeview.

To assign the mesh to the model all features must be selected:

Edit
Select All

- Select all the features of the model, (the shortcut is to press the **Ctrl** and **A** keys at the same time).
- Drag and drop the Surface mesh attribute **Thick Shell** from the Treeview onto the selected features.

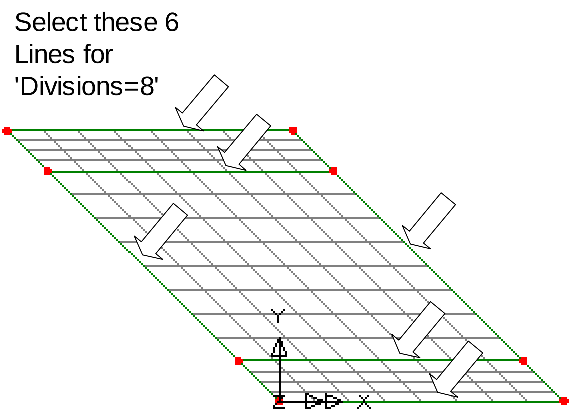
The mesh will be displayed with each Line being split according to a default number of divisions.

Note. One way of controlling the mesh density is by specifying the number of mesh divisions along each Line. Since Modeller is fully associative, this will control Surface mesh density.

Using Line Mesh Divisions

In this example 8 divisions per Line are required for the longitudinal Lines and transverse Line representing the carriageway. A number of Line mesh divisions are defined by default. These can be seen in the Treeview.

- Select the 4 longitudinal Lines as shown. Hold the **Shift** key down after selecting the first Line to add the other Lines to the selection.
- Still holding the **Shift** key select the 2 transverse Lines of the surface representing the carriageways.



- Drag and drop the Line mesh attribute **Divisions=8** from the Treeview onto the selected features.

The mesh will be re-displayed to show 8 mesh divisions along each selected line.

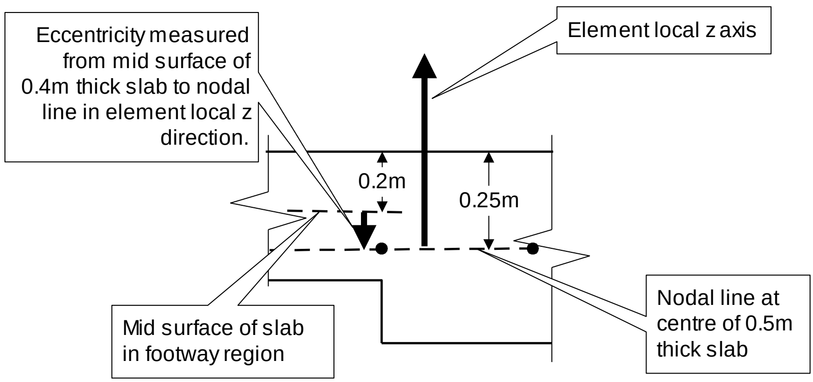
Note. Holding down the **L** key while selecting ensures that only lines will be selected. Similarly the **P** key limits the selection to Points, the **S** key; Surfaces, the **V** key; Volumes and the **G** key; Geometry. Similar shortcuts exist for elements, nodes, etc., which can be found in Modeller Manual.

Defining the Thickness and Eccentricity

The slab is 0.5 m thick in the carriageway region and 0.4 m thick beneath the footways.

Attributes
Geometric >
Surface...

- Enter a thickness of **0.5** and leave the eccentricity field empty.
- Enter the attribute name as **Thickness 0.5**
- Click the **Apply** button to define the first geometry attribute and allow another dataset to be defined.
- Change the thickness to **0.4**



Note. The direction of the eccentricity is measured from the mid-surface of the slab to the nodal line in the local element direction which follows the local surface direction. Since in this example the mid-surface of the slab representing the footway is above the nodal line, and the element local Z axis direction is vertically up, the eccentricity is input as negative.

- Enter the eccentricity as **-0.05**
- Change the attribute name to **Thickness 0.4**
- Click the **OK** button to finish.

The thickness is now assigned to each Surface.

- Select the Surface representing the carriageway.
- Drag and drop the geometry attribute **Thickness 0.5** from the Treeview onto the selected surface.

Geometric assignments are visualised by default.

- Select the two footway Surfaces, (hold down the **Shift** key to add the second Surface to the selection).
- Drag and drop the geometry attribute **Thickness 0.4** from the Treeview onto the selected surfaces.

Select the isometric button to see the geometric visualisation on the members.

The Zoom in button can be used to check the eccentricity has been assigned correctly.

Select the Home button to return the model to the default view.

Select the fleshing on/off button to turn-off the geometric visualisation

Supports

The more common types of support are provided in the Treeview by default. In this example the slab deck is to be restrained in the vertical and lateral direction at the abutment edges, Additionally, a single pinned support will be assigned to one point on the deck to prevent rigid body motion.

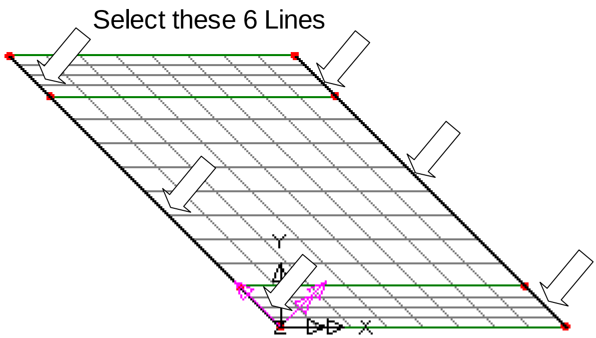
Before the supports can be assigned a local coordinate system needs to be defined so the lateral displacement can be restrained in the skew direction along the length of the abutment edges.

Attributes
Local Coordinate…

- Enter an angle of **45** for the rotation about the **Z-axis**.
- Enter the dataset name as **Skew** and click **OK** to add the Local coordinate attribute to the Treeview.
- By default the new **Skew** local coordinate system, is visualised within the view global axes.

To assign the abutment supports:

- Select the six transverse Lines representing the end diaphragms (ensure the **Shift** key is used to add to the initial selection)
- Drag and drop the local coordinate attribute **Skew** onto the selected Lines.



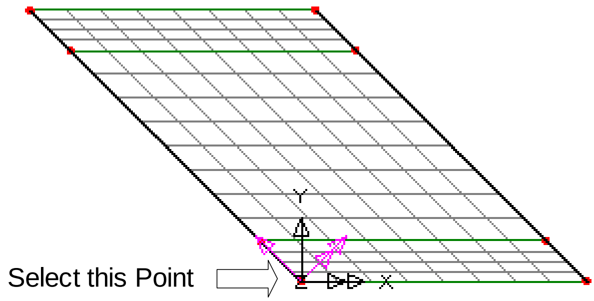
- Drag and drop the support attribute **Fixed in YZ** onto the selected Lines.

The Assign Support dialog will be displayed.

- Ensure that the attribute is assigned to **All analysis loadcases**
- Click the **OK** button to finish.

To assign the pinned support:

- Select the single Point at the origin
- Drag and drop the support attribute **Pinned** onto the selected Point.



The Assign Support dialog will be displayed.

- Ensure that the attribute is assigned to **All analysis loadcases**
- Click the **OK** button to finish.

Note. Since the slab is flat the in-plane rotation degrees of freedom for the thick shell elements can be eliminated from the element formulation by deselecting the following element option. This will remove ill-conditioning of the equations and reduce the problem size. It should however be noted that if moment supports are to be applied with this option deselected, rotation about the Z axis must be restrained to eliminate mechanisms in the solution.

File
Model Properties…

- Select the **Solution** tab and select the **Element Options** button. Deselect the option to **Assign 6 DOF to all thick shell element nodes** and click the **OK** button to return to the Model Properties, Click **OK** to return to the display.

Defining the Loads

Four loadcases will be defined:

- ❑ The self-weight of the slab (ignoring any surfacing loads).
- ❑ A patch load equivalent to a traffic load over the full length and width of the upper traffic lane.
- ❑ A knife edge load acting at mid-span of the upper lane.
- ❑ An HB vehicle in the lower lane.

Self-weight

Bridge
Bridge Loading >
Gravity

The self-weight is defined as a gravity acceleration in the negative Z direction. The correct units are automatically calculated when the gravity loading is selected from the bridge menu.

Lane Load

Bridge
Bridge Loading >
United Kingdom…

The second load will represent a lane load along the upper traffic lane.

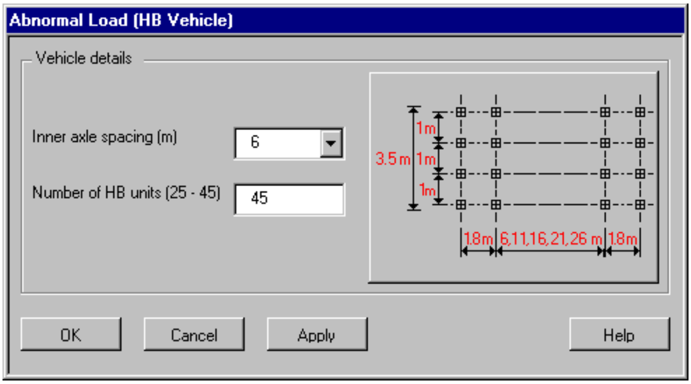
- From the Vehicle Loading dialog select the **Lane load (HA Load)** button.
- Select loading code **BD37/88** from the pull-down list.
- Enter the loaded length as **10.5**, leave the notional width set as **3.5**, set the skew angle as **-45** and the origin as **Centre**, then click **OK** to add the lane load patch loading to the Treeview.

Note. Setting a skew angle on this patch load does not skew the whole load – it only skews the ends of the patch load to match the angle of the skewed deck.

Knife Edge Load

- From the Vehicle Loading dialog select the **Knife edge load (KEL load)** button.
- Leave the skew angle as **0** and click **OK** to add the lane load patch loading to the Treeview.

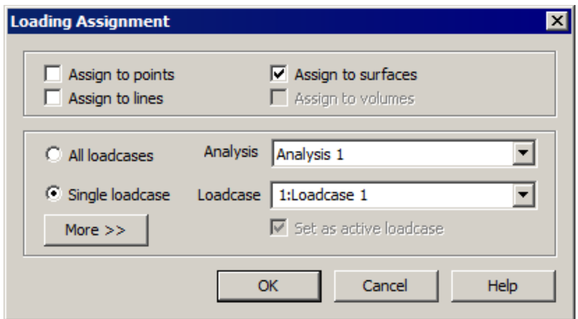
Abnormal Loading



- From the Vehicle Loading dialog select the **Abnormal Load (HB Vehicle)** button.
- Leave the **Axle spacing** as **6** and the **Number of HB units** as **45** and click **OK** to add the HB vehicle point load to the Treeview.
- Select **Close** to close the Vehicle Loading dialog.

Assigning the Self-weight Load

- To assign the gravity loading select all active features using the **Ctrl** + **A** keys together and drag and drop the loading attribute **BFP1** (**Gravity -ve Z)** from the Treeview onto the selected features.



The Loading Assignment dialog will be displayed.

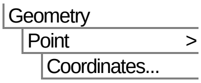
- Ensure that the **Assign to Surfaces** button is selected and click the **OK** button to assign the load to **Analysis 1**, **Loadcase 1**.

The load visualisation for Loadcase 1 will be displayed.

- To rename Loadcase 1 in the Treeview expand **Analysis 1** and select **Loadcase 1** with the right-hand mouse button, then choose the **Rename** option. Overtype the loadcase name with the name **Self Weight**

Assigning the Lane Load and Knife Edge Loads

Vehicle and lane loads are represented by patch loads that are assigned to Points which may form part of the model, or may be defined specifically to locate the bridge loads as in this case:



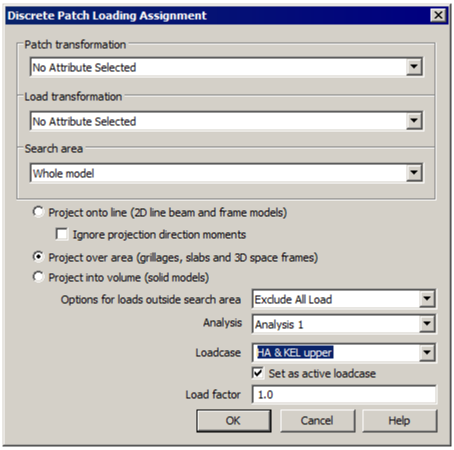
Enter coordinates as (**-1.5, 6.75**) and click the **OK** button to create a new Point in the middle of the upper lane.

- In the Treeview move the Geometry layer name down to sit beneath the Attributes layer so that the point defined can be seen.

The patch and knife edge loads can now be assigned to the Point that has just been defined.

- Select the Point defined at the centre of the upper traffic lane.
- Drag and drop the loading attribute **Pch2 (HA BD37/88 10.5m x 3.5m Skew=-45 (Centre) Patch 1)** from the Treeview onto the selected Point.

The Loading Assignment dialog will appear.



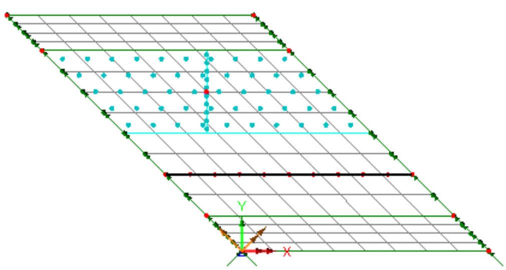
The loading option to **Project over area** will have been automatically selected.

- Overtype the name of the loadcase to be **HA & KEL upper**
- Click on the **OK** to complete the assignment.

The patch load will be visualised on the model.

Note. The number of patch load divisions in the patch is set in the Patch divisions object. By default ten patch divisions are used in the long direction of the patch. For more detailed information on discrete point and patch loads refer to the on-line help.

- Drag and drop the loading attribute **Pch3 (KEL 120kN Width=3.5m Offset=0 Skew=0 (Centre))** from the Treeview onto the selected Point.
- Select the loadcase name **HA & KEL upper** from the loadcase drop down list and click the **OK** button to assign the load.



The assigned patch loading for the upper lane should be as shown.

Note. In this example the patch loading has been shaped to match the skew of the deck but it could have alternatively been defined oversize in length and when assigned to the model any loading outside of the model could have been excluded as necessary.

Rebuilding a model after a previous failed analysis

If a previous analysis of this example has failed you need to return to this point to continue after having run the supplied file stated.

Assigning the Moving HB Vehicle Load

The HB vehicle will be assigned to the points using the moving load generator. To use this facility a line defining the path of the vehicle must be created.

- Click on a blank area of the model window to clear the current selection.

Geometry
Line >
Coordinates...

Enter coordinates of **(-3.25, 3.25)** and **(7.25, 3.25)** to define a Line describing the path of the HB vehicle and click **OK**

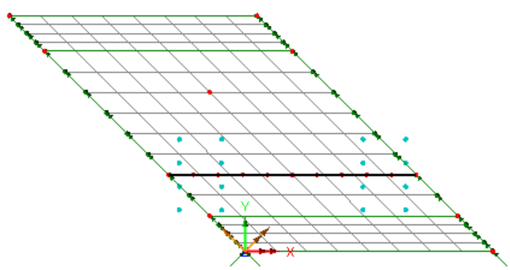
- Select the Line just defined.

Bridge
Moving Load…

- Ensure the **Moving vehicle load** is set to **Pnt4: (HB 6m spacing 45 units)**
- Set the option **Move the vehicle along path in equal number of divisions** to **10** and click **OK** to generate loadcases 3 to 13 representing the HB load moving forward across the bridge.

The isometric view button can be used to rotate the model so the load directions can be visualised.

Checking loading



Loads on the model may be visually checked at any time by changing the active loadcase.

- From the Treeview select **Load ID=4 Line=17 Pos=6**, click the right-hand mouse button and select the **Set Active** option to see the mid-span position of the moving load.

This process may be repeated for any loadcase.

Saving the Model

The model is now complete and the model data should be saved before running analysis.

File
Save

Save the model file.

Running the Analysis

With the model loaded:

Press the **Solve Now** button 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 Treeview.

In addition, 2 files will be created in the Associated Model Data directory:

- **bridge_slab.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **bridge_slab.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **bridge_slab_modelling.vbs** carries out the modelling of the example.

File
New…

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 **bridge_slab** and click **OK**

File
Script >
Run Script...

To recreate the model up to the point of defining the moving load, select the file **bridge_slab_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Now return to the section entitled **Assigning the Moving HB Vehicle Load** earlier in this example and continue from this point.

Viewing the Results

In this section the results produced by the analysis of the slab will be examined. There are a number of ways to do this in LUSAS, allowing users to choose the most appropriate way to present their results. For this problem the following are required:

- ❑ A plot of the deformed mesh.
- ❑ A colour-filled contour plot of bending moments in the X direction due to a load combination.
- ❑ A graph showing the variation of bending moment along a section of the slab.
- ❑ A contour plot of Wood-Armer reinforcement moments in the slab.

Selecting the results to be viewed

Loadcase results can be seen in the Treeview, and the loadcase results for the first loadcase (self weight) are set active by default.

Using Page Layout Mode

The model was created using a Working Mode view which allows a model of any size to be created. Results could be viewed using this mode of operation, but in order to allow additional information to be added without obscuring the model, Page Layout Mode can be used instead.

View
Page Layout Mode

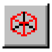
The graphics window will resize to show the model on an A4 size piece of paper in landscape orientation.

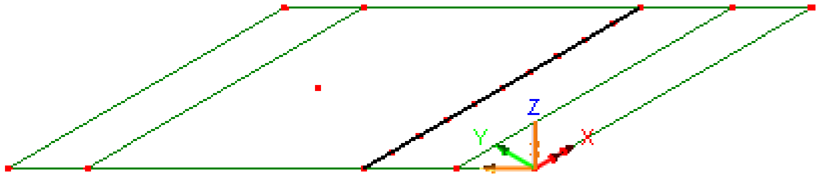
File
Page Setup...

- Ensure that left, right top and bottom margins are set to **60, 10, 10** and **10** respectively and click **OK**

To plot a deformed mesh the Attributes and Mesh layers will be hidden.

- Click the right-hand mouse button on the **Attributes** entry in the Treeview and select the **On/Off** option. The geometry will be hidden from the display.
- Repeat to turn off the Mesh layer.





If necessary, use the Isometric view button to rotate the model to the view shown.

Note. When required, the local coordinate system axes can be turned off by deselecting the Show Definition option for the Skew entry in the Treeview.

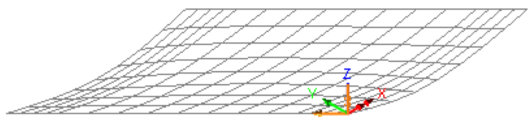
This page layout view can be saved for subsequent re-use with other models.

Window
Save View...

Enter the view name as **Landscape Page Layout** and click **OK**

Displaying the deformed shape

- In the Treeview turn-off the display of the **Geometry** layer.
- With no features selected click the right-hand mouse button in a blank part of the Graphics Window and select the **Deformed mesh** option to add the deformed mesh layer to the Treeview.



- Click the **OK** button to accept the default deformed mesh properties values.

The deformed mesh plot for the first loadcase will be displayed.

Contour Plots

Results can be plotted as colour line or filled contour plots. Here, longitudinal bending moments will be plotted as colour filled contours.

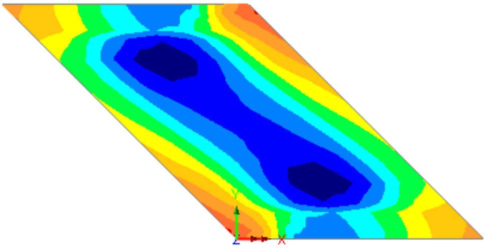
Z: N/A

Return the model to the default starting view by pressing the Z axis field on the status bar at the bottom of the graphics window.

- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Contours** option to add the contours layer to the Treeview.

The contours properties dialog will be displayed.

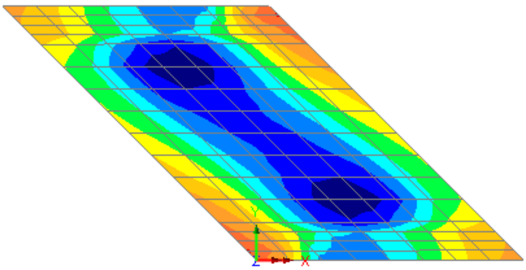
- Select Entity **Force/Moment - Thick Shell** Component moments in the X direction **Mx**
- Click the **OK** button to display contours of moments in the X direction.



Re-ordering the display of the layers

Note. The order of the layers in the Treeview governs the order in which the layers are displayed in the graphics window with layer names at the top of the list drawn to the screen first. To see the mesh layer on top of the contours layer the mesh layer needs to be moved down the Treeview to a position after the contours layer.

- In the Treeview select the **Deformed mesh** layer, click the right-hand mouse button and select the **Move Down** option. (This can also be done by dragging and dropping a layer name on top of another layer name).



Note. Contours of results may be displayed which are either averaged at the nodes, to give a smoothed contour display, or unaveraged to contour on an element by element basis. Unaveraged nodal results are useful for checking for sufficient mesh refinement (since they will display inter-element discontinuities in the solution). They are also used to prevent the averaging of stresses across elements with dissimilar material or geometric properties. By default contours are smoothed.

Moving information on the annotation layer

Text and other objects such as the contour key may be moved after their initial placement.

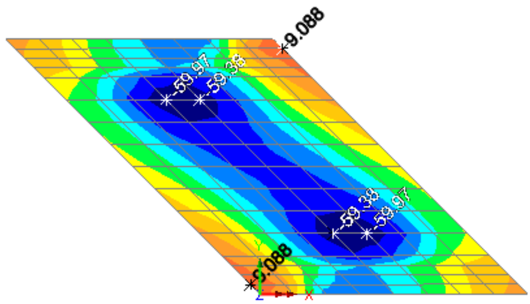
- Click on one of the items of text on the contour key. (This will select the key and enable it to be moved).
- Click and drag the object to a new position on the left-hand margin of the page.

Marking Peak Values

- With nothing 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. 

The values properties dialog will be displayed.

- Select **Force/Moment - Thick Shell** contour results of bending moment **Mx**
- On the same dialog, select the **Values Display** tab and ensure the **Maxima** and **Minima** check boxes are selected to display the top **1%** of results.
- Set the number of significant figures to **4**
- Change the angle to **45** degrees
- Select the **Choose Font** button and set the text size to **16** and click **OK**
- Click the **OK** button to redisplay the contours with peak values marked. Note that on the diagram shown above the mid-span values have been artificially coloured white in order to be visible when printed.



Defining an Envelope

Envelopes are used to automatically assemble the worst effect from a selection of loadcases, combinations, or other envelopes. In this example an envelope is to be assembled to show the worst effect from the moving HB vehicle.

Analyses
Envelope

To include just the moving load loadcases in the envelope:

- With **Analysis 1** selected from the drop-down list select **(3) Load ID=4 Line 17 Pos=1 Dir=Fwd** from the loadcase panel. To select all HB moving loadcases hold down the **Shift** key, scroll down the list and select **(13) Load ID=4 Line 17**

Pos=11 Dir=Fwd and click on the button to add all the selected datasets to the Included panel.

- Change the name to **Moving load lower** and click **OK**
- Two envelope attributes representing the maximum and minimum results will be created in the Treeview.

Note. Changing the properties of the max or min envelope will change the definition of both envelopes.

Defining a Combination

Combinations can be used to view the combined effects of several results loadcases, combinations or envelopes.

Analyses
Basic Combination

The combination properties dialog will appear. Loadcases Self Weight, HA & KEL upper and the moving load envelope are to be included in the combination.

- With the **model** file selected from the drop-down list select both **Self Weight** and **HA & KEL upper** and click the button to add the loadcases to the combination dataset with a factor of **1**
- Select the envelope **Moving load lower (Max)** and click the button to add the loadcase to the combination dataset

Note. By default loadcases are added to the combination with a factor of 1. This factor may be changed by selecting the loadcase in the Included panel and changing the load factor in the text box provided. Alternatively the factors may be changed in a grid by selecting the Grid button.

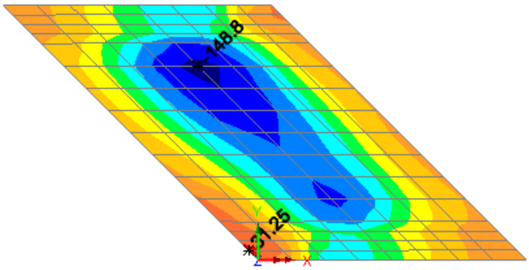
- Change the combination name to **Max Design Combination** and click the **OK** button to finish.

Note. To obtain the correct effect from the combined loads the combination should only include one occurrence of each loadcase. Envelope and Combination definitions are held within the model file.

Selecting and Viewing the Combination Results

- In the Treeview right-click on **Max Design Combination** and select the **Set Active** option.
- Select **Force/Moment - Thick Shell** results of component **Mx** and click **OK**

The contour plot for Mx from the load combination will be displayed.



Note. The values layer is still listed in the Treeview so the maximum and minimum values of Mx for the load combination will also be shown.

Graphing the Results of a Slice

A graph is to be plotted showing the variation in bending moment through a specified section of the slab.

Utilities
Graph Through 2D

If the model is in the XY plane the By Cursor dialog will appear.

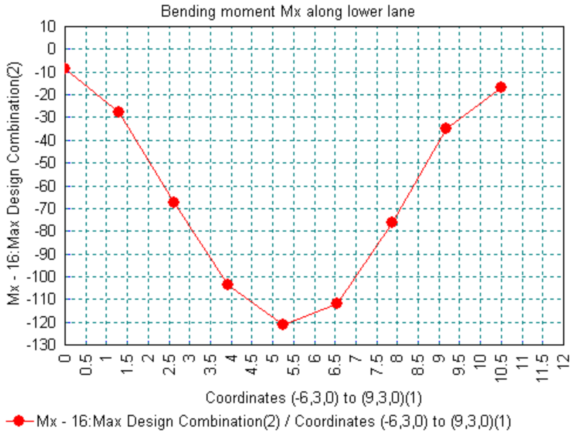
- Ensure that **Snap to grid** is selected with a grid size of **1**
- Click the **OK** button.
- Click and drag the cursor along the centre of the lower lane (Y=3) to define the location of a section slice through the slab.

Note. The Y ordinate of the slice can be seen in the status bar Y:3. The X and Y scales on the sides of the window will also help this selection.

The graph X axis values of distance through the deck are defined by the slice section. The graph Y axis results now need to be specified on the slice data dialog.

- Select **Force/Moment - Thick Shell** results for Bending Moment in the X direction **Mx** and click the **Next** button.

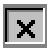
The graph Y axis results have now been defined and title information for the graph can be added.



- Enter the graph title as **Bending Moment Mx along lower lane**
- Leave the graph X and Y axis names blank.
- Click the **Finish** button to display the graph.

Note. If the graph axes labels are left unspecified default names will be used.

Note. The graph properties can be modified by clicking the right hand mouse button in the graph window and selecting the Edit Graph Properties option.

 Close the graph window.

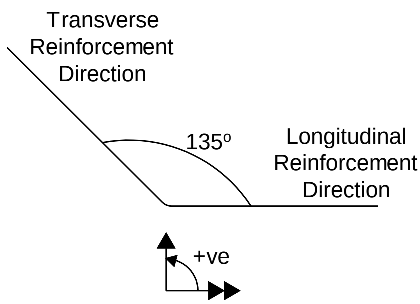
 Click on the selection mode button to return to normal cursor mode.

Wood Armer Reinforcement Plots

- In the  Treeview double-click on the **Contours** layer.

The contour properties dialog will be displayed.

- With the **Force/Moment - Thick Shell** entity selected, select Component **My(B)** to display results of Wood Armer moments in the Y direction in the bottom of the slab.
- Click the **Wood Armer** button.
- On the Wood Armer Options dialog enter a **Reinforcement skew angle** of **135** degrees.
- Click the **OK** button to return to the contour properties dialog.
- Click the **OK** button to display a plot of Wood Armer contours of My(B) for the active combination.



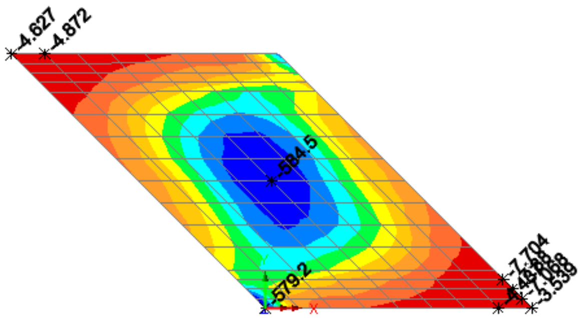
Note. Specifying the distance from top or bottom surface to centre of top or bottom reinforcement is only used when plotting contours of M[UtilB] or M[UtilT]

Caution. The selected results component (My(B)) is different to that being used to compute smart combinations and/or envelopes (Mx) and therefore the results displayed will be coincident effects.

- Click the **OK** button to accept this.

Caution. The contour plot will show bending moment contours My(B) but the Values layer will currently show values of bending moments in the X direction. i.e. Mx since these were specified previously.

- Click **Yes** to change the values layer to show maxima and minima values of My(B).



Save the model

File
Save

Save the model file.

Note. When the model file is saved after results processing, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

This completes the example.

Note. A separate example covers Wood Armer slab assessment in more detail.

Note

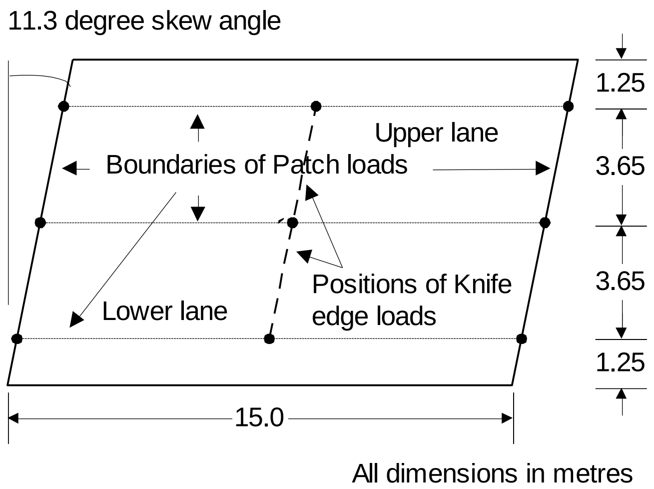
- In this example the geometry and mesh attributes result in a regular mesh aligned to the global axis. The use of an irregular mesh would mean that the results used for the Wood-Armer plots will have different orientations over the whole slab and hence be incorrect. In this circumstance a local coordinate set would need to be used to orientate the shell element results to be consistent with the global axes in order to obtain sensible components for use in the calculations.

- The mesh orientation can be viewed from the Treeview by selecting **Mesh** > **Properties** and selecting the **Show Element Axis** option. Press **OK** to apply changes.
- Results transformations can be applied from the Treeview. Press **Transformed...** and select **Global axes** to transform local element results into the global axes.

Wood-Armer Bridge Slab Assessment

For software product(s):	LUSAS *Bridge*.
With product option(s):	None.

Description



A Wood Armer slab assessment is to be carried out on a 15m long, 9.8m wide single span, reinforced concrete bridge deck. The deck has a skew angle of 11.3 degrees and is 0.9m deep.

A load combination comprising self weight, upper and lower lane loads, and upper and lower knife edge is to be created from the results obtained. Slab reinforcement is to be arranged orthogonally and has capacities of 1700 and 300 kNm in the chosen Mx and My directions.

Units of kN, m, t, s, C are used throughout.

Note. In order to create a generic example, no reference is made to any particular design code or loadcase type.

Keywords

2D, Simple Slab, **Skew Angle**, **Wood-Armer Reinforcement**, **Wood-Armer Assessment**, **Safety Factors**, **Self Weight, General Loading**, **Knife Edge Loading**, **General Patch Loading**, **Combination**, **Contour Plotting**, **Display Peak Values.**

Associated Files

- ❑ **wood_armer_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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 the file name as **wood_armer**
- Use the **Default** working folder.
- Enter the title as **Wood Armer Slab Example**
- Select model units of **kN,m,t,s,C** from the drop-down list provided.
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Ensure the **Standard** startup template is selected
- Select the **Vertical Z axis** option
- Click the **OK** button.

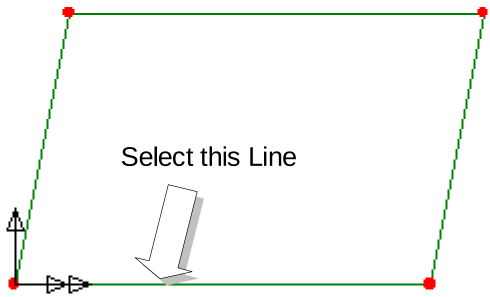
Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Feature Geometry

Geometry
Surface >
Coordinates...

Enter coordinates of (**0, 0), (15, 0**), (**16.96, 9.8**) and **(1.96, 9.8**) and click **OK** to define the slab as a single Surface.

Next, the edge of carriageway and lane widths will be defined by copying the lower Line of the main span.



- Select the lower horizontal Line of the model.

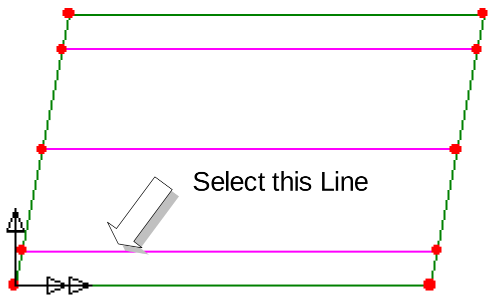
Geometry
Line >
Copy…

Enter the **X** translation as **0.25** [derived from 1.25*tan(11.3^o)]

- Enter the **Y** translation as **1.25** metres, and click the **OK** button.

The edge of the lower carriageway will be drawn.

To define 2 notional lane widths of 3.65 metres.



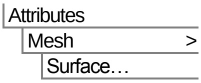
- Select the Line just drawn.

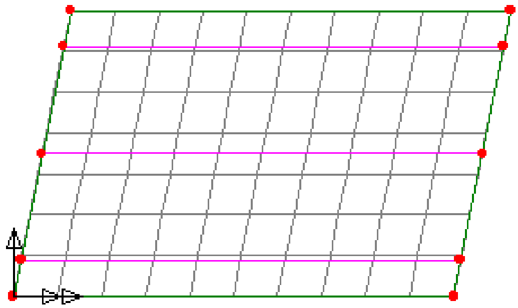
Geometry
Line >
Copy…

Enter the **X** translation as **0.73** [derived from 3.65*tan{11.3^o}]

- Enter the **Y** translation as **3.65** metres.
- Enter the number of copies as **2**
- Click the **OK** button to finish.

Meshing





- Select **Thick plate**, **Quadrilateral**, elements with **Linear** interpolation.
- Deselect the automatic divisions option and enter **10** divisions in the local **X** direction and **7** divisions in the local **Y** direction.
- Enter the attribute name as **Thick Plate** and click the **OK** button to finish.

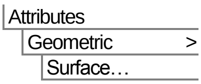
LUSAS will add the Surface mesh attribute to the Treeview.

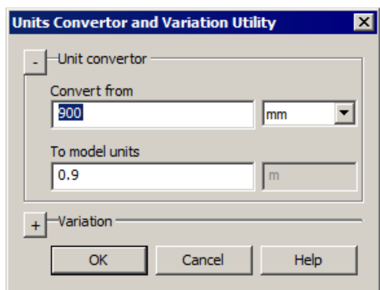
- Select the whole model using the **Ctrl** and **A** keys together**.**
- Drag and drop the Surface mesh attribute **Thick Plate** from the Treeview onto the selected features.

Note. At any time the mesh (and other features) displayed in the Graphics area may be turned on or off by right-clicking on the layer name in the Treeview and selecting the On/Off option.

- Turn off the display of the **Mesh** as just described.

Geometric Properties





- Open the Units Convertor and Variation Utility from the Thickness textbox.
- Convert from **900**, selecting **mm** to convert 900mm to model units. Press **OK** to return.
- Enter the attribute name as **Thickness 0.9**. Click **OK** to add surface geometry attribute to the Treeview.
- Select the whole model.
- Drag and drop the geometry attribute **Thickness 0.9** from the Treeview onto the selected features.

Select the fleshing on/off button to turn-off the geometric visualisation. If at any time during the example you wish to visualise the geometry select this button.

Material Properties

Attributes
Material >
Material Library…

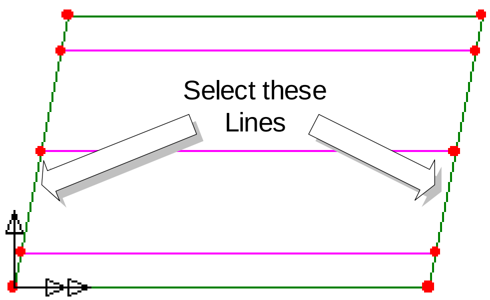
- Select material **Concrete** from the drop down list, leave grade as **Ungraded** and click **OK** to add the material attribute to the Treeview.
- With the whole model selected, drag and drop the material attribute **Iso1 (Concrete Ungraded)** from the Treeview onto the selected features, ensuring that **Assign to surfaces** is selected click **OK**

Supports

LUSAS provides the more common types of support by default. These can be seen in the Treeview. Both inclined edges of the slab are to be simply supported in the Z direction.

Assigning the Supports

- Select the 2 inclined Lines at either end of the slab. (hold the Shift key down to add to the first selection).
- Drag and drop the support attribute **Fixed in Z** from the Treeview onto the selected features.



- Ensure **Assign to lines** and **All analysis loadcases** are selected and click **OK**

The supports will be visualised.

Loading

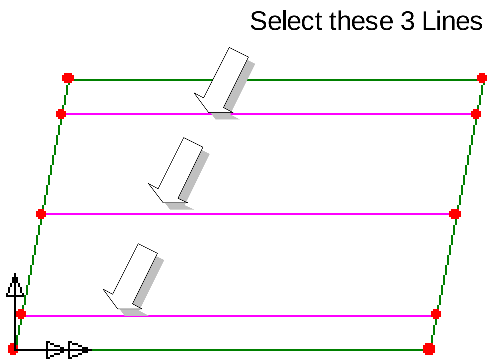
Five loadcases will be considered, with an additional two combinations of these loads created at the results processing stage.

Modifying the Geometry to assign loading

To position knife edge loads additional Points are required at mid-span positions.

Geometry > Line > By Splitting > In Equal Divisions...

- Select the 3 horizontal Lines that define the traffic lane boundaries. (Hold the **Shift** key down to add to the selection)
- Select **Use same divisions for all lines** and enter the number of divisions required for all as **2**
- Ensure the **Delete original lines after splitting** option is selected and click **OK**



Additional Points will be created at mid-span and existing Lines will be broken into 2 new Lines. The new Points will be used later to define the position of the mid-span knife edge loads for each traffic lane.

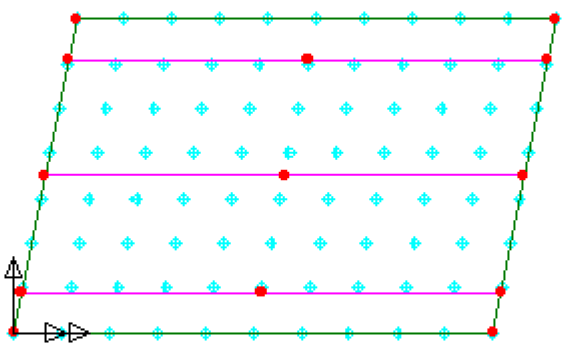
Loadcase 1 - Self weight

Attributes > Loading…

- Select the **Body Force** option and click **Next**
- Define the linear acceleration due to gravity in the **Z** direction as **-9.81**
- Enter the attribute name as **Self Weight** and click **Finish**
- With the whole model selected, drag and drop the loading attribute **Self Weight** from the Treeview onto the selected features ensuring the loading is assigned to Surfaces as **Loadcase 1** and click **OK**

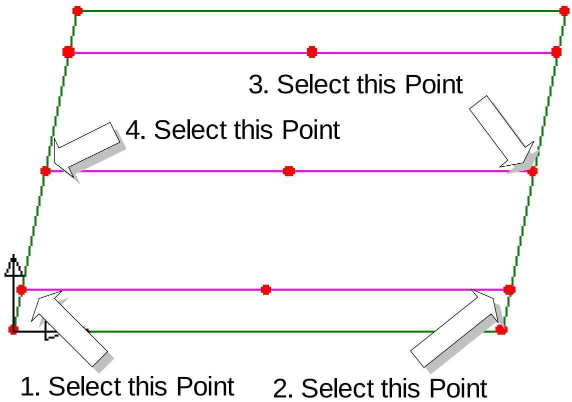
The self weight loading will be displayed.

- In the Loadcases Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select the **Rename** option.
- Rename **Loadcase 1** to **Self Weight**



Loadcase 2 - Lane load (lower lane)

- Select the first point, hold down the **Shift** key and select point 2, 3 and 4 in the order shown to define the patch area.



Attributes
Loading…

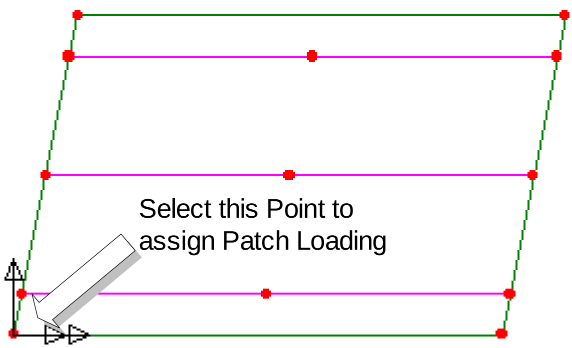
- In the Discrete panel select the **Patch** option and click **Next**

The coordinates of the Points selected will be inserted into the coordinate fields.

- Enter a load value of **-15** (kN/m^2) for each coordinate.
- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction and **0** in the y direction so that the number of loading points in the patch y direction will be computed automatically from the aspect ratio of the patch.
- Enter the attribute name as **Lane load (lower)** and click **Finish**

Note. The order in which the Points are selected determines the local x and y directions of the patch load. The local x direction is from the first Point to the second Point selected. The local y direction is from the second to the third Point.

- Select the Point shown at the origin of the structure.



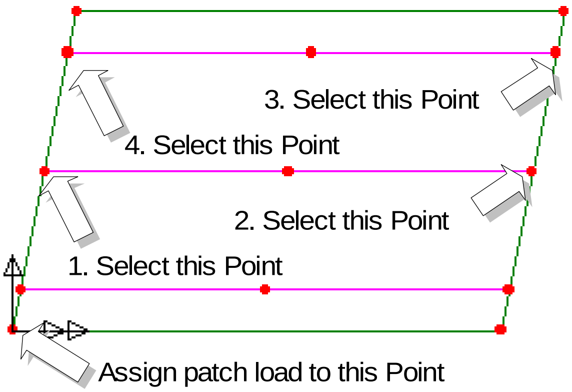
- Drag and drop the loading attribute **Lane load (lower)** from the Treeview onto the selected Point.
- Ensure **Options for loads outside search area** is set to **Exclude all load**
- Enter the loadcase as **Lane load (lower)** with a load factor of **1** and click **OK**

The loading will be displayed.

Note. Because the coordinates of the patch load were taken from the coordinates of the Points, the patch load must be assigned to the Point at the origin of the structure.

Loadcase 3 - Lane load (upper lane)

- Select the Points in the order shown to define the patch area for the upper lane.



Attributes
Loading…

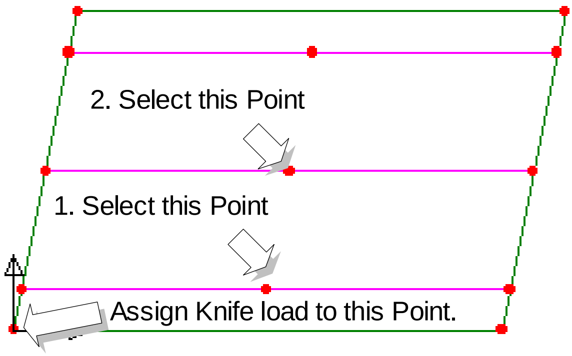
- In the Discrete panel select the **Patch** option and click **Next**

The coordinates of the Points selected will be inserted in the coordinate fields.

- Enter a load value of **-15** (kN/m^2) for each coordinate.
- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction and **0** in the y direction.
- Enter the attribute name as **Lane load (upper)** and click **Finish**
- With the Point at the origin of the structure selected, drag and drop the loading attribute **Lane load (upper)** from the Treeview onto the selected Point.
- Ensure **Options for loads outside search area** is set to **Exclude all load**
- Enter the loadcase as **Lane load (upper)** with a load factor of **1** and click **OK**

Loadcase 4 - Knife edge load (lower lane)

- Select the Points in the order shown to define the knife edge load position for the lower lane.



Attributes
Loading…

- In the Discrete panel select the **Patch** option and click **Next**
- Select the **Straight Line** option.

The coordinates of the Points selected will be inserted in the coordinate fields.

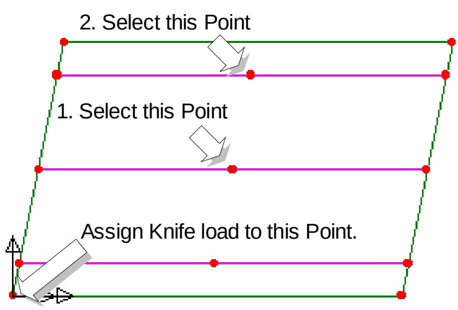
- Enter a load value of **-32.24** (kN/m) for each coordinate.
- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction.
- Enter the attribute name as **Knife edge load (lower)** and click **Finish**

Note. The order in which the Points are selected determines the local X direction of the knife edge load. The Local X direction is from the first Point to the second Point selected.

- With the Point at the origin selected, drag and drop the loading attribute **Knife edge load (lower)** from the Treeview onto the selected Point.
- Enter the loadcase as **Knife edge load (lower)** with a load factor of **1** and click **OK**

Load case 5 - Knife edge load (upper lane)

- Select the Points in the order shown to define the knife edge load position for the upper lane.



Attributes
Loading…

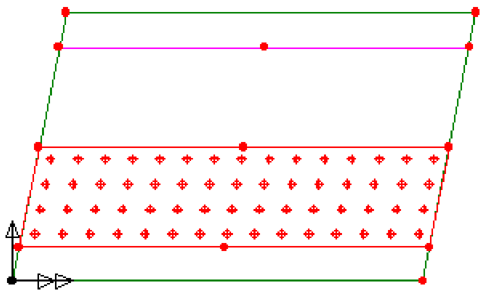
- Select the **Discrete Patch** option and click **Next**
- Select the **Straight Line** option.

The coordinates of the Points selected will be inserted in the coordinate fields.

- Enter a load value of **-32.24** (kN/m) for each coordinate.

- De-select the Use default button in the Patch load divisions panel and enter **15** in the x direction.
- Enter the attribute name as **Knife edge load (upper)** and click **Finish**
- With the Point at the origin selected, drag and drop the loading attribute **Knife edge loading (upper)** from the Treeview onto the selected Point.
- Enter the loadcase as **Knife edge load (upper)** with a load factor of **1** and click **OK**

Visualising Loadcases



Loadcases can be visualised at any time by activating each loadcase in the Treeview.

- From the Treeview select the **Lane load (lower)** loadcase, click the right-hand mouse button and select the **Set Active** option.

Saving the model

File
Save

Save the model file.

Running the Analysis

With the model loaded:

Press the **Solve Now** button 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 Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

- ❑ **wood_armer.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

- ❑ **wood_armer.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- ❑ **wood_armer_modelling.vbs** carries out the modelling of the example.

File
New…

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 **wood_armer** and click **OK**

File
Script >
Run Script...

- To recreate the model, select the file **wood_armer_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

Combinations

A combination will be created to apply the ultimate limit state (ULS) factors and lane factors to loadcases 1 to 5. Typical ULS and lane factors can be found in relevant design codes.

Dead load factor ULS combination

- Dead load factor γ_{fl} = 1.15
- Additional factor γ_{f3} = 1.1

Live load factors for ULS combination

- Lane factors β_1 = 1.0 and β_2 = 1.0
- Live load factor γ_{fl} = 1.5
- Additional factor γ_{f3} = 1.1

Loadcase	Factor Calculation	Factor
Self weight	γ_{fl} x γ_{f3}	1.265
Lane load (lower)	β_1 x γ_{fl} x γ_{f3}	1.65
Lane load (upper)	β_2 x γ_{fl} x γ_{f3}	1.65
Knife edge load (lower)	β_1 x γ_{fl} x γ_{f3}	1.65
Knife edge load (upper)	β_2 x γ_{fl} x γ_{f3}	1.65

Defining a Combination

Combinations can be created to view the combined effects of multiple loadcases on the structure. A combination dataset **Combination 1** is to be created in the Treeview

Analyses
Basic Combination

The combination properties dialog will appear. All 5 loadcases should be included in the load combination panel.

- Select loadcase **Self Weight** hold the **Shift** key down, scroll down, and select **Knife edge load (upper).**
- Click the >> button to add the loadcases to the combination dataset.

Note. To obtain the correct effect from the combined loads in this example the Combination should only include one occurrence of each loadcase.

Assigning load factors

- In the Included panel on the Combination dialog, select the **Self Weight** loadcase and enter load factor of **1.265** in the load factor text box.
- For each of the other 4 loadcases, select each loadcase in turn and enter a factor of **1.65**
- Select the **Grid** button to check all the factors are entered correctly and click **OK** to return to the combination dialog.
- Click **OK** to create the combination dataset.

Selecting Loadcase results

- In the Treeview right-click on **Combination 1** and select the **Set Active** option.
- If present in the Treeview turn off the **Geometry** and **Attributes** layers.
- Turn on the display of the **Mesh** layer in the Treeview.

Contour Plots

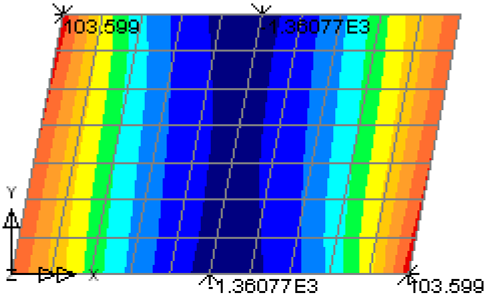
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Contours** option to add the contours layer to the Treeview.

The contour properties dialog will be displayed.

Moments in the X direction

- Select **Force/Moment - Thick Plate** contour results.
- Select the **MX** component.
- Click the **OK** button to display contours of MX for Combination 1.
- To display the mesh on top of the contours, select the **Mesh** entry in the Treeview and drag on drop it on top of the **Contours** entry in the Treeview.

Marking Peak Values

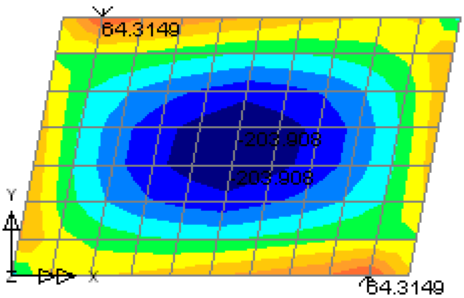


- With nothing selected click the right-hand mouse button in a blank part of the graphics window and select the **Values** option to add the values layer to the  Treeview.

The properties dialog of the values layer will be displayed.

- Select the **Force/Moment – Thick Plate** entity.
- Select the **MX** component.
- Select the **Values Display** tab and set **Maxima** and **Minima** values to display the moments for the top **0.1**% of results.
- Click the **OK** button to redisplay the contours with peak values marked.

Moments in the Y direction



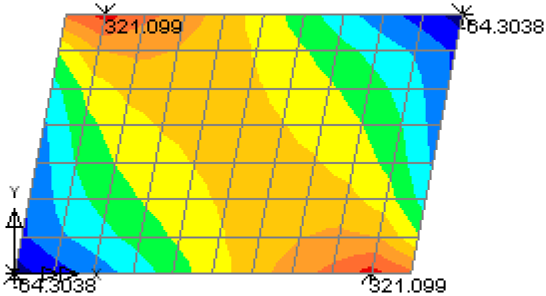
- In the  Treeview double-click on the **Contours** layer.
- Select the **Force/Moment - Thick Plate** entity.
- Select the **MY** component.
- Click the **OK** button to display contours of MY for Combination 1.

Note. The values layer is currently displaying results for MX.

- Select **Yes** to change the values layer results to match those of the contours layer.

Moments in XY direction

- In the Treeview double-click on the **Contours** layer.
- Select the **Force/Moment - Thick Plate** entity.
- Select the **MXY** component. Click the **OK** button.
- Select **Yes** to change the values layer results to match those of the contours layer.



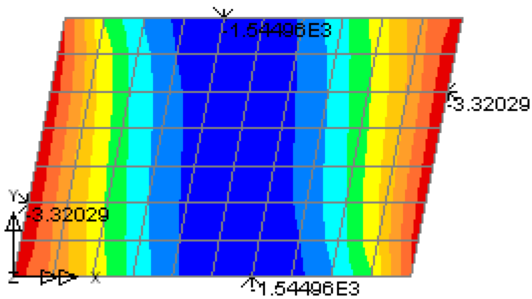
Wood-Armer Reinforcement Moments

Note. For the following section, it is assumed that you are familiar with the theory of Wood Armer as an explanation is beyond the scope of this example. If additional information is required consult the LUSAS Theory Manual. Initially, Wood Armer reinforcement moments Mx(B) in the bottom surface of the slab will be calculated.

Wood Armer Moments in X direction of the bottom of the slab

- In the Treeview double-click on the **Contours** layer.
- Select the **Force/Moment - Thick Plate** entity.
- Select the **MX(B)** component to view contour of Wood Armer moments in the X direction for the bottom of the slab. Click the **OK** button.
- Select **Yes** to change the values layer results to match those of the contours layer.

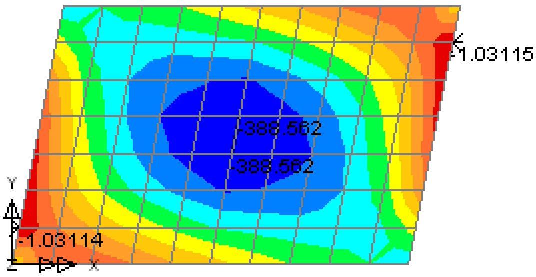
From the resulting Wood Armer plot it can be seen that the maximum value of Mx(B) is 1545 kNm. This value is compared with the maximum slab capacity (1700 kNm) to determine the safe load carrying capacity. In this case the slab passes the assessment.



Wood Armer Moments in Y direction of the bottom of the slab

The Wood Armer moments in the direction of the reinforcement on the bottom Surface My(B), are now to be examined.

- In the Treeview double-click on the **Contours** layer.
- Select the **Force/Moment - Thick Plate** entity.
- Select the **MY(B)** component to view contours of Wood Armer moments in the Y direction for the bottom of the slab.
- Click the **OK** button.
- Select **Yes** to change the values layer results to match those of the contours layer.



The plot indicates a maximum applied sagging Wood Armer moment My(B) of 389 kNm This exceeds the slab capacity of 300 kNm and therefore this slab fails assessment.

Note. Normally the analysis would be continued to calculate the reduction in live load that would be necessary to obtain a situation where the applied moment My(B) equals the capacity of the slab My*(B). This would result in a restriction in the load carrying capacity of the slab.

Wood Armer - Discussion

It is generally accepted that the Wood Armer equations when used in the assessment of the load carrying capacity of slabs provide a safe but conservative estimate of the structural capacity. It is possible to obtain significant improvements using alternative equations based on the fundamental principles of Wood Armer.

Note. A detailed explanation of the modified equations is beyond the scope of this example, however the user may find it beneficial to consult the following references:

- **Concrete Bridge Design to BS5400.** L.A Clark. Published by Construction Press. Appendix A. Equations For Plate Design.
- **The Assessment of Reinforced Concrete Slabs.** S.R Denton, C.J Burgoyne. The Structural Engineer. Vol. 74. No. 9. May 7.

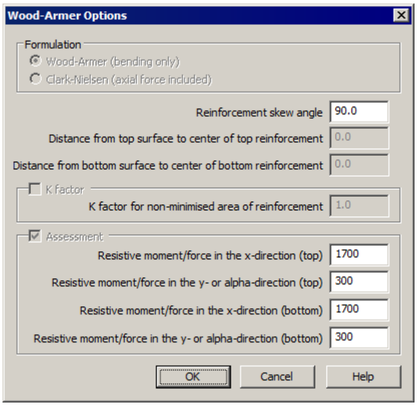
In outline, the method adopts a safety factor approach, where the user inputs the slab capacity of the reinforcement and the reinforcement skew angle. Since the slab can potentially fail in flexure about any axis in the plane of the slab, the method examines the applied moment field (Mx, My, Mxy) against the moment capacity field (Mx*, My*, Mxy*) for all possible reinforcement skew angles.

The method returns minimum safety factors for top (hogging) and bottom (sagging) reinforcement for each nodal position.

Note. Top and bottom safety factors are possible at any single position due to the application of mixed moment fields.

Wood Armer Assessment

- In the Treeview delete the **Values** layer.
- In the Treeview double-click on the **Contours** layer and select the **Force/Moment – Thick Plate** entity.
- Select the **MUtil(B)** component to select contours of Utility factors in the bottom of the slab.
- Click the **Wood Armer** button.
- On the Wood Armer Options dialog ensure the **Reinforcement skew angle** is set to **90** and the Distances to the centre of the reinforcement are set to **0**.



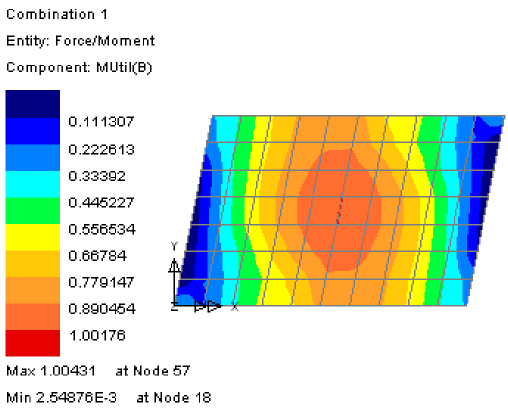
The moment capacity of the slab in the sagging and hogging zones needs to be entered.

- Enter the Resistive moment in the x direction (top) as **1700**
- Enter the Resistive moment in the y direction (top) as **300**
- Enter the Resistive moment in the x direction (bottom) as **1700**
- Enter the Resistive moment in the y direction (bottom) as **300**
- Click the **OK** button to return to the contour dialog.
- Click the **OK** button to display contours of safety for the moment capacities entered.

Note. Specifying the Distance from top or bottom surface to centre of top or bottom reinforcement is not applicable for Thick Plate elements. It is only used with Shell and Ribbed Plate Elements.

Note. Wood Armer skew angles are measured anticlockwise from the x-axis towards the y-axis.

Note. In the Wood Armer dialog, hogging moment capacity values are entered as +ve (positive) values and the sagging moment capacity values are entered as -ve (negative) values.



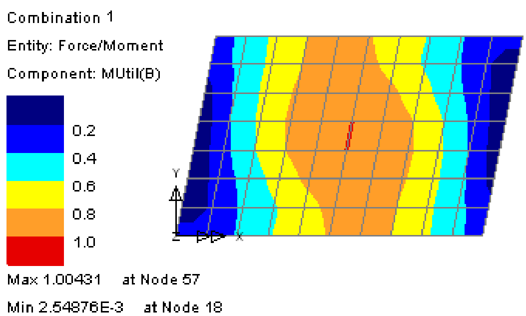
Changing the Contour range

If a utility factor plot shows values of less than 1.0, the slab is deemed to satisfy the reinforcement criteria and no live load reduction is necessary.

To help clarify whether any regions of the slab have failed the assessment the contour range will be modified.

- In the Treeview double-click on the **Contours** layer and select the **Contour Range** tab.
- Set the contour **Interval** as **0.2** and the **Value to pass through** as **1**
- Click the **OK** button to finish.

Contours of utility factors from 0 to 1 will be displayed showing a small region in the centre of the slab that has failed the assessment according to the moment capacities used.



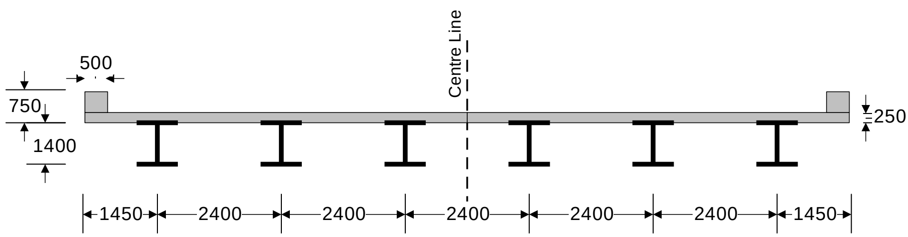
This completes the example.

Grillage Load Optimisation

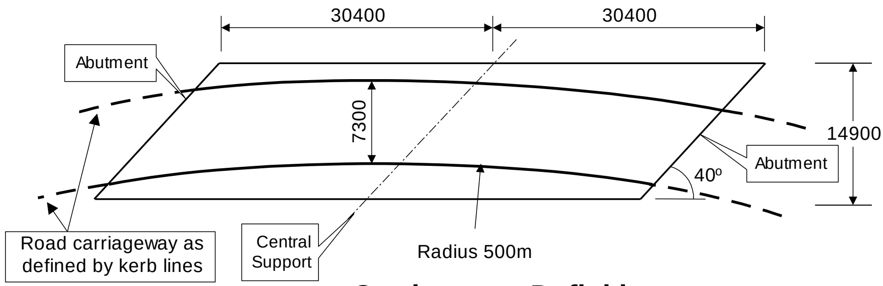
For software product(s):	LUSAS *Bridge* and LUSAS Bridge *Plus*
With product option(s):	Autoloader.
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example illustrates how the vehicle load optimisation facility is applied to grillage models. Autoloader vehicle load optimisation is used using the reciprocal influence method.



Cross Section



Carriageway Definition

The structure to be analysed consists of steel plate girders supporting an in-situ reinforced concrete deck slab. Two spans of 30.4m each (skew span measurement) span between the centreline of the support bearings.

The parapet beams are cast onto the structure after the deck slab has cured. This will make the edge beam properties different from the internal beams however for simplicity this will be ignored in this example. This assumption makes both the internal and edge beam properties the same. The properties are computed assuming 6 transverse beams in each span.

Note. The section properties stated are for the composite sections ignoring shear lag effects, i.e. taking into account the full width of the slab. The composite sections have been transformed using short term concrete modulii and are stated in “steel” units in the table below.

Section Property Table	**Area m^2 (Asz m^2)**	**Iyy m^4 (Izz m^4)**	**J m^4**
Edge Beam/Internal Beams (Uncracked)	0.12359	0.03777	0.697E-3
Edge Beam/Internal Beams (Cracked)	0.04151	0.01387	5.366E-6
Transverse Beams	0.1188	0.5012E-3	0.997E-3
Skew Beam	0.0684	0.2886E-3	0.575E-3
Diaphragms	0.05775	0.029E-3	0.107E-3

The structure will be designed to carry full HA loading and 30 units of HB loading as derived from BD37/88.

Units of kN, m, t, s, C are used throughout.

Objectives

The required output from the analysis consists of:

- ULS and SLS values for selected locations on the structure.

Keywords

2D, Grillage, Autoloader, Load optimisation, Load visualisation, Influence surface, Basic Load Combination, Smart Load Combination, Envelope

Associated Files

- **spaced_beam_geometry.vbs** carries out the definition of the section properties.

- ❑ **spaced_beam_modelling.vbs** carries out the modelling of the example without the loading applied.

Vehicle load optimisation in LUSAS

Vehicle load optimisation (VLO) makes use of influence surfaces and influence analysis to identify the most onerous vehicle loading patterns on bridges for a chosen design code and to apply these loading patterns to LUSAS models. A vehicle load optimisation wizard provides the means of defining parameters, for a particular design code, to generate the most critical traffic loading pattern for each influence shape under consideration. The actual vehicle load optimisation software that is used to generate this loading depends upon the design code chosen.

- For traffic loading assessment to Eurocode EN-1991-2, Australia AS5100-2:2004 and New Zealand (Transit New Zealand Bridge Manual) LUSAS Traffic Load Optimisation (LUSAS TLO) software is used.

- For vehicle loading assessment to other supported design codes, Autoloader Vehicle Load Optimisation software is used.

In this example, the grillage is loaded according to design code BD37/88, so Autoloader is used.

Modelling

Running LUSAS Modeller

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

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 the file name as **spaced_beam**
- Use the **Default** working folder.
- Enter the title as **Two span grillage analysis with load optimisation**
- Select model units of **kN,m,t,s,C**
- Ensure the timescale units are **Seconds**

- Ensure the analysis type is **Structural**
- Ensure the startup model template is set to **None**
- Select the **Vertical Z axis** option.
- Click the **OK** button.

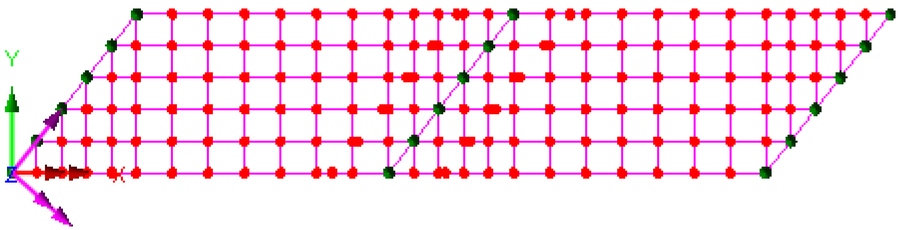
Note. It is useful to save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily by a new user.

Feature Geometry

The grillage geometry will be created using the Grillage Wizard.

Bridge
Grillage Wizard...

- Select the **Set defaults** button
- Select **Spaced beam and slab deck** and ensure the cracked section is **15**%. Click **Next**
- The grillage is **Straight** with **40** degrees skew. Click **Next**
- Select an **Orthogonal** grillage arrangement and click **Next**
- Enter the width of grillage as **12** and the number of longitudinal beams including edge beams as **6**. Ensure **Evenly spaced** beams are used and click **Next**
- Change the number of spans to **2**
- For each span enter the length of span as **30.4** and change the number of internal transverse beams to **6**



- Click **Finish** to generate the grillage model.

Defining the Geometric Properties

The geometric properties for this model are listed in the section property table at the beginning of this example. However, for ease of use they will be read in using a file that is supplied.

File
Script >
Run Script...

Open the file **spaced_beam_geometry.vbs** which is located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Assigning Geometric Properties

This is best done by using copy and paste in the and Treeview panes.

- Select the **Edge Beam/Internal Beams (Uncracked)** properties from the Treeview and click on the button.
- In the Treeview select the **Longitudinal Uncracked Section** group and click on the button.

Confirmation of the assignment will appear in the text window.

- Select the **Edge Beam/Internal Beams (Cracked)** properties from the Treeview and click on the button.
- In the Treeview select the **Longitudinal Cracked Section** group and click on the button.
- Select the **Transverse Beams** properties from the Treeview and click on the button.
- In the Treeview select the **Transverse Intermediate Beams** group and click on the button.
- In the Treeview select the **Transverse Central Beams** group and click on the button.
- Select the **Skew Beams** properties from the Treeview and click on the button.
- In the Treeview select the **Transverse Skew Beams** group and click on the button.
- Select the **Diaphragms** properties from the Treeview and click on the button.

- In the Treeview select the **Internal Diaphragms** group and click on the button.

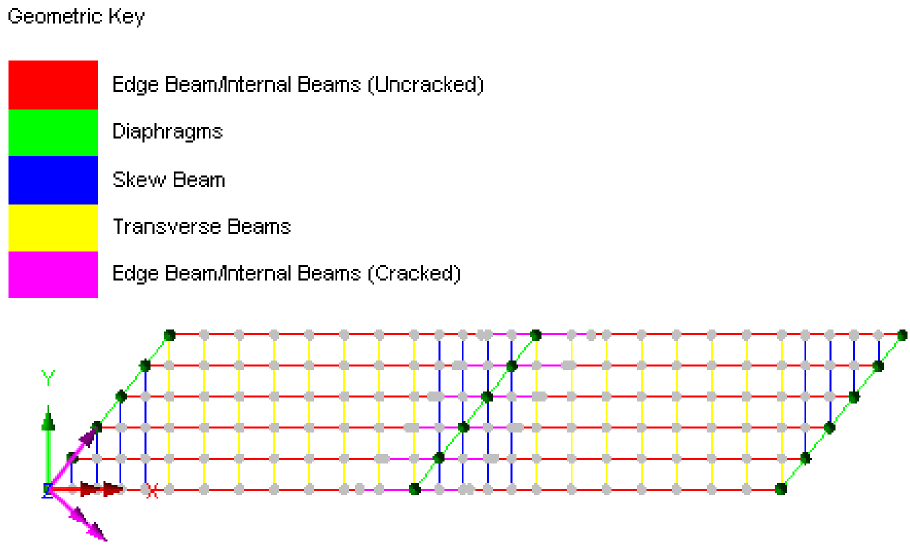
- In the Treeview select the **End Diaphragms** group and click on the button.

Checking for correct geometric assignment

- Turn off the display of the **Mesh** layer from the Treeview.

- Double click on the **Geometry** layer in the Treeview and select **Assignment** from the Colour by drop down list.

- Click the **Set** button and select **Geometric** from the Attribute Type drop down list.



- With the **Generate key** option selected click the **OK** button to return to the Geometry properties dialog and **OK** again to display the geometry coloured by geometric assignment with the key annotated.

- To remove the 'Colour by Attribute' assignment double click on **Geometry** in the Treeview, select **Own Colour** from the Colour by drop down list and click **OK** to update the display.

Defining the Material

Attributes
Material >
Material Library…

- Select material **Mild Steel** of type **Ungraded** from the from the drop-down list and click **OK** to add the material dataset to the Treeview.
- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material dataset **Iso1 (Mild Steel Ungraded)** from the Treeview onto the selected features.

Note. Because the geometric properties provided are transformed section properties (as mentioned in the example's description) Mild Steel properties are assigned to the complete model.

Supports

The deck is supported in the vertical direction at each of the diaphragms. Since there are no in-plane degrees of freedom no in-plane supports are required. The grillage wizard automatically restrains the grillage from vertical displacement at each diaphragm.

Note. If in-plane effects such as braking forces are to be considered the grillage elements should be replaced by 3D beam elements and the appropriate properties and supports should be assigned.

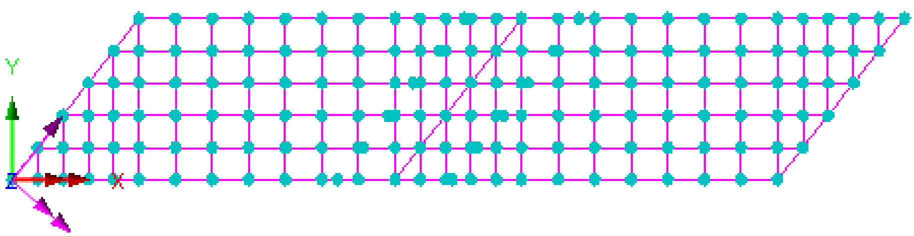
Self-Weight

Bridge
Bridge Loading >
Gravity

This command adds **Gravity -ve Z** to the Treeview.

- With the whole model selected, drag and drop the loading dataset **BFP1 (Gravity -ve Z)** from the Treeview onto the selected features. Ensure the **Assign to lines** option is selected and click **OK** to assign to **Loadcase 1**.

The gravity loading will be visualised on the model.



- From the Treeview expand **Analysis 1** and then, using the right-hand mouse button, select **Loadcase 1** and select the **Rename** option. Change the loadcase name to **Self Weight**

Loading the model using the Vehicle Load Optimisation (Autoloader) facility.

In order to use the vehicle load optimisation facility the locations at which the optimised load combinations are to be calculated and the parameter to be computed must be chosen. To do this, influence attributes are defined and assigned to the model. LUSAS then calculates the influence surfaces for any specified position in the structure using the Muller-Breslau principle whereby the mesh is automatically "broken" at each specified node and automatically constrained to act in the required manner. An influence surface is generated for each of the chosen locations and parameters.

For this example three influence surfaces will be defined.

Defining Influence Attributes

Influence Point 1

The first influence attribute will be used to investigate the reaction at the left hand abutment.

Attributes
Influence >
Reciprocal Theorem...

- Select a **Reaction** influence type for a **Vertical axis** influence direction for a **Positive** displacement direction. Enter the influence attribute name as **Reaction Support 1** and click **OK**.

The influence attribute will be added to the Treeview.

Influence Point 2

The second influence attribute will be used to investigate mid span bending in the first span.

Attributes
Influence >
Reciprocal Theorem...

- Select a **Moment** influence type **About Transverse** axis direction for a **Negative** displacement direction (because we are interested in the maximum sagging moment). Enter the influence attribute name as **Bending Span 1** and click **OK**.

Influence Point 3

The third influence surface attribute will be used to investigate the hogging moments at the central pier.

Attributes
Influence >
Reciprocal Theorem...

- Select a **Moment** influence type **About Transverse** axis direction for a **Positive** displacement direction (because we are interested in the maximum hogging moment). Enter the influence attribute name as **Bending Internal Support** and click **OK**.

Note. Longitudinal and Transverse directions are specified using the **Direction definition** object in the Treeview . In this example the longitudinal direction is defined to be the Global X direction, and the transverse direction is always assumed to be orthogonal to the longitudinal direction.

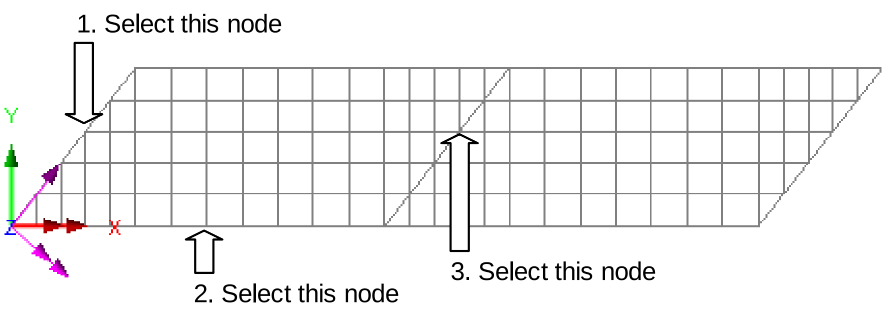
Turning loading visualisation on and off

At any time during this example any applied loading can, for reasons of clarity, be turned on and off by selecting the Loading on /off button.

Assigning Influence Attributes

Influence attributes can be assigned to nodes or points. When modelling grillages created by the grillage wizard there will always be a node at a point on the model, so influence attributes could be assigned to points. But, in general usage, and for slab analysis especially, influence attributes are better assigned to nodes. Influence attributes can be assigned to as many nodes (or points) on the model as necessary. So, for example, the Sagging Span 1 influence attribute could be assigned to all nodes in the first span to specify influences for every node even though all nodes do not necessarily need to be examined for particular influence surface examinations. However, for this example, and for speed and clarity, only one node will be selected for each influence attribute.

- Turn off the display of the **Geometry** layer in the Treeview.
- Turn on the display of the **Mesh** layer in the Treeview
- If necessary turn off any loading visualisation.



The first influence attribute will investigate the reaction near the centre of the left hand abutment.

- Select the node on the end diaphragm as shown.
- Drag and drop the influence attribute **Reaction Support 1** from the Treeview onto the selected feature.

The second influence attribute will investigate edge mid-span bending in span 1.

- Select the mid-span node at the edge of span 1 as shown.
- Drag and drop the influence attribute **Bending Span 1** from the Treeview onto the selected feature

The third influence attribute will investigate hogging near the centre of the internal support.

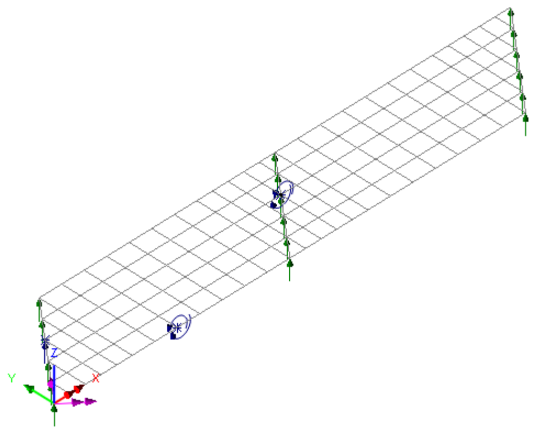
- Select the node at the internal support as shown.
- Drag and drop the influence attribute **Bending Internal Support** from the Treeview onto the selected feature

Visualising the Defined Influence Points

Influence attribute assignments are visualised as they are assigned to the model. To check that the influence attribute orientations are correct (meaning that the correct influence directions have been defined) an isometric view of the model should be used.

 Select the isometric button.

- Ensure that the influence visualisations are as shown



Using the Vehicle Load Optimisation facility to calculate the worst case loading patterns.

The vehicle load optimiser (Autoloader) automates the creation of load datasets in accordance with the chosen loading code for the locations and effects specified.

Before invoking the Vehicle Load Optimisation Wizard the kerb lines need to be defined. This can be done by specifying coordinates but is best done by defining kerb lines on the model.

Select the Home button to return the model to the default view.

- In the Treeview turn on the display of the **Geometry** layer.

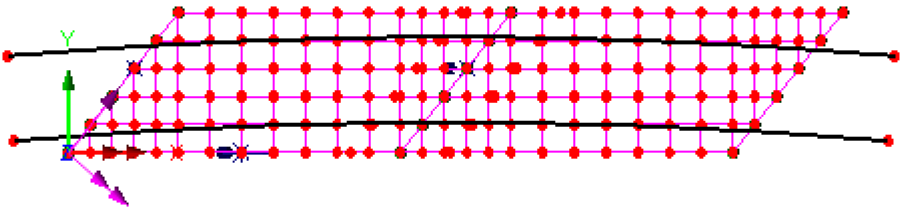
Defining Kerb Lines

Geometry
Line >
Arc/Circle >
From Coords/Points …

- Specify the first point as **(-5, 1)** the second point as **(75, 1)**. Define the third point as **(35, 20)** and select the **Direction** option for the third point.
- Enter the radius as **500** and ensure that **Minor Arc** is selected.
- Click **OK** to define the first kerb line.

Geometry
Line >
Arc/Circle >
From Coords/Points …

- Specify the first point as **(-5.584, 8.2766)** and the second point as **(75.584, 8.2766).** Define the third point as **(35, 20)** and set the **Direction** option for the third point.
- Enter a radius of **507.**3 and ensure that **Minor Arc** is selected
- Click the **OK** button to define the second kerb line.
- **Select the first and second kerb line**. (Use **Shift** to add to the initial selection)



Save the model

File
Save

Save the model file.

Rebuilding a model after a previous failed analysis

If a previous analysis of this example has failed you need to return to this point to continue after having run the supplied file stated.

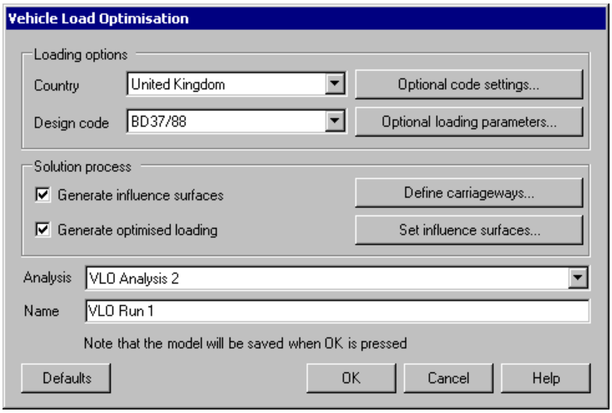
Carrying out Vehicle Load Optimisation

- **Ensure the two kerb lines are selected**.

Bridge
Vehicle Load Optimisation…

With the kerbs selected the vehicle load optimisation can begin.

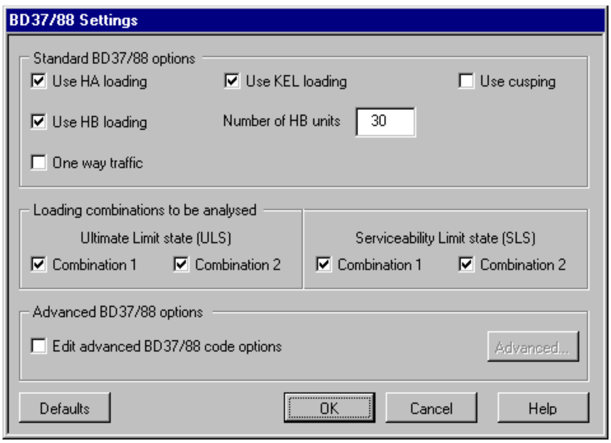
- Select the **Defaults** button to reset any settings from any previous vehicle load optimisation runs.
- Select **United Kingdom** from the Country drop down list.
- Select **BD37/88** from the Design code drop down lists.



- **VLO Analysis 2** will be automatically entered for the Analysis entry in the Treeview. (Note that an alternative name can be entered by selecting the New option from the drop-down list).
- Enter the VLO run Name as **VLO Run 1**
- Select the **Optional code settings** button.

The BD37/88 optional code settings will appear.

- Ensure the options **Use HA Loading**, **Use HB Loading** and **Use KEL Loading** are selected.
- Set the number of HB units to **30**
- Deselect the option to **Use cusping**
- Click **OK** to set the selected options and return to the main vehicle load optimisation dialog.



- Select the **Optional loading parameters** button to display the optional loading parameters dialog.
- Ensure the vehicle library is set as **\<Lusas Installation Folder>\Programs\Scripts\atl\VehicleFiles\autoload.vec**
- Select **HB** from the drop down Vehicles list.
- Set **Longitudinal increment** to **0.25** and **Transverse increment** to **0.25**
- Ensure vehicle direction is set to **Both** and click **OK** to set the options and return to the main vehicle load optimisation dialog.

Note. By selecting 'Edit Advanced Code Options' on the Optional code settings dialog and clicking the Advanced button some of the lesser used loading options may be modified. This is also available for the loading options.

Having specified the loading options the carriageway positions need to be defined. In this example the carriageway positions are defined from the selected kerb Lines.

- Select the **Define carriageways** button.
- Select the **Kerbs from selection** option and click **Apply** to define the carriageways and return to the main dialog.

Finally the influence surfaces to be utilised must be defined.

- Select the **Set influence surfaces** button.
- Ensure **Include all influence surfaces** is selected.
- Ensure that each loaded influence area is set to **Positive** in the grid. This will instruct Autoloader to solve for the worst positive effects on each of the specified influence surfaces.
- Ensure the increment for influence surfaces is set to **0.25**
- Click **OK** to set the influence data and return to the main dialog.

The optimisation parameters are now set-up.

- Lastly, ensure the **Generate influence surfaces** and **Generate optimised loading** are selected and click **OK** to carry out the load optimisation.

LUSAS Modeller will run the Solver to generate each of the three influence surfaces and then run Autoloader to compute the worst position of vehicles for each influence surface, generate the loadcases in Modeller.

Running the analysis

Finally, the model needs to be solved using those critical loadcases:

- Press the **Solve Now** button and on the dialog ensure every analysis is selected and click the **OK** button to perform the analysis.

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 the Model

If errors have been made in defining the model that for some reason you cannot correct, a file is provided to re-create the model information correctly.

❑ **spaced_beam_modelling.vbs** carries out the modelling of the example up to the point of using the vehicle load optimisation facility.

File
New…

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 **spaced_beam** and click **OK**

File
Script >
Run Script...

To recreate the model up to the point of using the vehicle load optimisation facility, select the file **spaced_beam_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Now return to the section entitled **Vehicle Load Optimisation** earlier in this example and continue from that point.

Viewing the Results

Analysis loadcase results for VLO Run 1 are present in the Treeview.

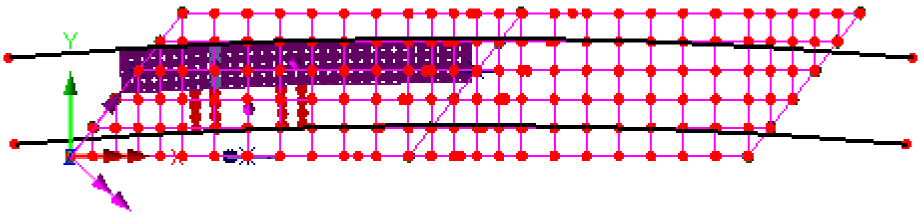
Note. Because each influence attribute may be assigned to many nodes, the coordinates of the node to which an attribute is assigned are also included in the loadcase name in order to make it unique.

Load Visualisation

If it is required to visualise the load pattern for a particular loadcase, this can be done by setting the results loadcase to be active.

If the **Attributes** layer is present and the Loading on/off button is depressed (signifying it is 'on') the loading patterns for the various results loadcases can be displayed:

- In the Treeview expand both the **VLO Analysis** and **Vechile Load Optimisation Run 1** branches, then right-click on the model loadcase **Bending Span 1 - (*Point...*) - ULS1/Positive (4)** and select the **Set Active** option to display the load pattern.



Reactions at Influence Point 1

A plot showing the reactions is to be displayed.

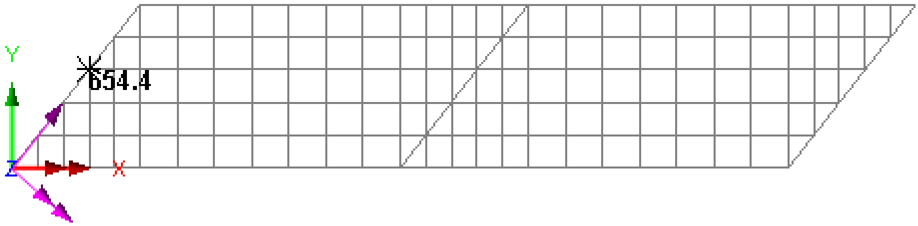
- Turn off the display of the **Geometry** and **Attributes** layers in the Treeview.
- In the Treeview right-click on the results loadcase **Reaction Support 1 – (*Point...*) - ULS1/Positive (2)** and select the **Set Active** option.
- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Values** option to add the values layer to the Treeview.

The values properties will be displayed.

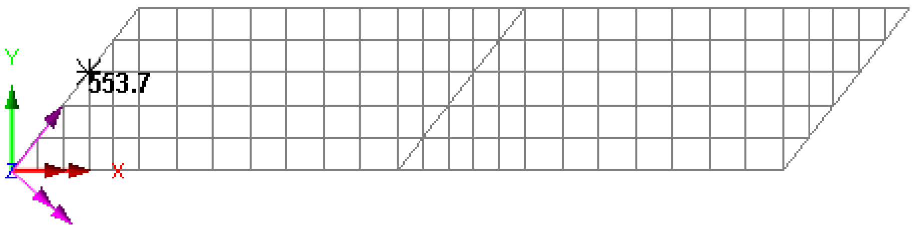
- Select entity **Reaction** results for the component vertical reaction **FZ**
- Select the **Values Display** tab
- Plot the top **10%** of the maxima values

- Set the **Number of significant figures** to **4**
- Select the **Choose Font** button
- Select a font **Arial** style **Bold** size of **12**
- Click **OK** to return to the values properties dialog and **OK** again.

The design reaction at the fixed bearing (previously selected for influence surface analysis) for the current loadcase **Reaction Support 1 – (*Point...*) - ULS1/Positive (2)** will be displayed.



- In the Treeview, right-click on the **Reaction Support 1- (*Point...*) - ULS2/Positive (8)** loadcase and select the **Set Active** option to display the design reaction value.



- Select the remaining Reaction Support 1 results loadcases in turn and choose **Set Active** to display the other corresponding SLS reaction values which are shown in the following table:

Location (Group name)	**Quantity**	**ULS1**	**ULS2**	**SLS1**	**SLS2**
Reaction Support 1 (End diaphragms)	Vertical reaction (FZ)	**654.4**	**553.7**	553.7	503.4

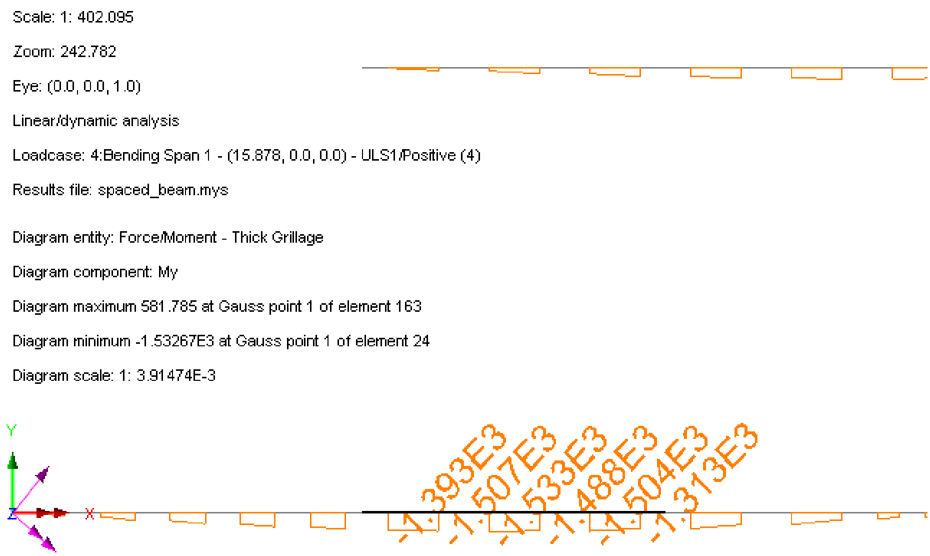
Moments at Influence Point 2

To view the bending moments in an edge beam the groups facility will be used in conjunction with the Diagrams layer.

- In the Treeview right-click on **Edge Beams** group and select the **Set as Only Visible** option.
- Turn off the display of the **Values** layer in the Treeview
- In the Treeview, right-click on the **Bending Span 1 - (*Point...*) - ULS1/Positive (4)** loadcase and select the **Set Active** option
- Add the **Diagrams** layer to the Treeview
- Select entity **Force/Moment – Thick Grillage** results for the bending moment **My**
- Select the **Diagram Display** tab
- Select the **Label values** option, **Label only if selected** option, **Orientate flat to screen/page** option and the **Window summary** option. Deselect the **Peaks Only** option. Set the % of element length to **50**, the label font to be size **20**, the font Angle to **45** and the significant figures to **4**
- Click **OK** to finish.

The window summary shows the maximum and minimum values.

- Use the Zoom in button to enlarge the left-hand end of the lower-beam.
- Drag a box around the beams of interest to view moment values on the beam.



By changing the active loadcase this table of results for the design of the edge beam can be generated.

Location (Group name)	Quantity	ULS1	ULS2	SLS1	SLS2
Lower edge beam (Edge Beams)	Bending moment (My)	**-1533**	-1297	-1297	-1179

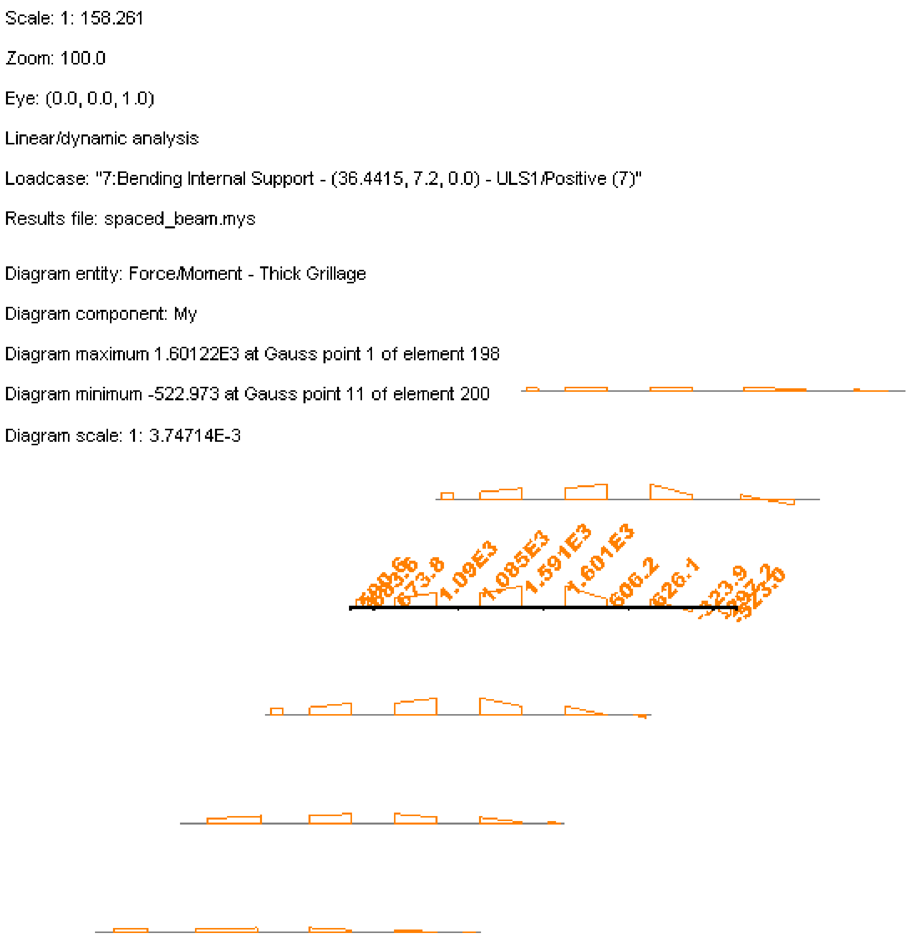
Note. By selecting **Peaks Only** in the **Diagram Display** tab the labels can be limited only to the maximum and minimum in the selection.

Note. If the values layer was used to investigate bending moments at points in the grillage, care must be taken to ensure that unaveraged values are displayed for any number of grillage elements coming together at a node rather than averaged values. If so done, end shrinkage settings will allow the separate values to be seen in isolation.

Moments at Influence Point 3

- In the Treeview right-click on **Longitudinal Cracked Section** group and select the **Set as Only Visible** option.
- In the Treeview, right-click on the **Bending Internal Support - (*x*,*y*) - ULS1/Positive (7)** loadcase and select the **Set Active** option.

- Drag a box around the beams of interest to view hogging moment values on the beam.



By changing the active loadcase the remaining ULS and SLS results for the design of the longitudinal beam over the support can be generated.

Location (Group name)	**Quantity**	**ULS1**	**ULS2**	**SLS1**	**SLS2**
Central pier (Longitudinal Cracked Beams)	Bending moment (My)	**1601**	1355	1355	1232

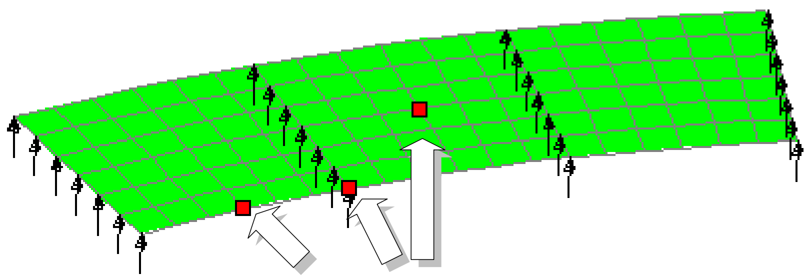
This completes the example.

Bridge Slab Traffic Load Optimisation

For software product(s):	LUSAS *Bridge* and LUSAS *Bridge Plus*
With product option(s):	LUSAS Traffic Load Optimisation (TLO).
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example uses the LUSAS TLO software option to aid in the linear static analysis of a 3-span curved concrete bridge deck subject to Eurocode traffic loading.



Points of interest for influence surface generation

The structure is modelled using thick plate elements, representing a deck of inner radius 75m, outer radius 86m and thickness 0.7m. The deck has a width of 11m consisting of a 10m wide carriageway region and two 0.5m wide verges. The live loading is to be calculated for three defined influence surfaces using the LUSAS TLO software option.

Units used ar**e** kN, m, t, s, C throughout.

Note. LUSAS TLO is a software option for LUSAS *Bridge*. It extends the vehicle loading capabilities in LUSAS *Bridge* and produces worst-case traffic load effects more easily and much faster than by manual methods. LUSAS TLO generates the most onerous traffic load patterns according to a selected code of practice, based upon influence surfaces for specified load effects (moments, shears, reactions, stresses etc) at selected locations in the structure. A number of Eurocode National Annexes and other codes are supported. This example analyses a bridge deck using EN1991-2 Recommended Values. Subsequent analysis for comparison purposes is carried out using the UK and Swedish National Annex options.

Modelling Objectives

The operations in the creation of the model are as follows:

- Sweep a curved deck from a line.
- Mesh the Surfaces with thick plate elements (regular mesh).
- Define and assign geometric properties to describe the slab thickness.
- Define and assign material properties.
- Define and assign bearing supports.
- Define and assign dead loading.
- Define three influence surfaces which will allow calculation of the most onerous traffic load pattern for:
    1. Mid-span bending, inner curve, first span.
    2. Mid-span bending, mid-node, second span.
    3. Reaction at the first internal support.
- Use LUSAS TLO to calculate the most onerous traffic load pattern for the three influence surfaces.

Keywords

2D, Slab, Modelling, Eurocode, Element Axes, Influence Surface, LUSAS TLO, Vehicle Load Optimisation, Load Combination, Bending Moments, Reactions, Transformed Results, Peak Values

Associated Files

- **deck_modelling.vbs** carries out the modelling of the slab deck for subsequent use with the LUSAS TLO software option.

Vehicle load optimisation in LUSAS

Vehicle load optimisation (VLO) makes use of influence surfaces and influence analysis to identify the most onerous vehicle loading patterns on bridges for a chosen design code and to apply these loading patterns to LUSAS models. A vehicle load optimisation wizard provides the means of defining parameters, for a particular design code, to generate the most critical traffic loading pattern for each influence shape under consideration. The actual vehicle load optimisation software that is used to generate this loading depends upon the design code chosen.

- For traffic loading assessment to Eurocode EN-1991-2, Australia AS5100-2:2004 and New Zealand (Transit New Zealand Bridge Manual) LUSAS Traffic Load Optimisation (LUSAS TLO) software is used.

- For vehicle loading assessment to other supported design codes, Autoloader Vehicle Load Optimisation software is used.

In this example, the bridge deck is loaded according to EN1991-2 Recommended Values, so LUSAS Traffic Load Optimisation is used.

Modelling

Running LUSAS Modeller

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

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 the file name as **deck**

Note. When saving a model for use with Autoloader a filename of 8 or less characters must be specified.

- Enter the title as **Deck analysis using traffic load optimisation**
- Use the **Default** working folder.
- Set model units of **kN,m,t,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the model template **Standard**
- Select the **Vertical Z axis** option.
- Click **OK**

Note. Save the model regularly as the example progresses. This allows a previously saved model to be re-loaded if a mistake is made that cannot be corrected easily.

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

Geometry

In this model, one span of the complete bridge deck is modelled in its entirety (including specifying geometric properties, material, supports, etc.) before it is then copied to form the complete 3-span structural model.

Geometry
Line >
Coordinates...

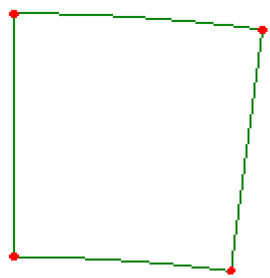
Enter coordinates of (**0,75**) and (**0,86**) to define a Line representing the end of the deck.

- Click the **OK** button to finish.
- Select the Line just drawn.

Geometry
Surface >
By Sweeping…

Select the **Rotate** option and enter an angle of **-7.5** degrees to rotate about the **Z-axis** about the origin **0,0**

- Click **OK** to sweep the Line to create a surface

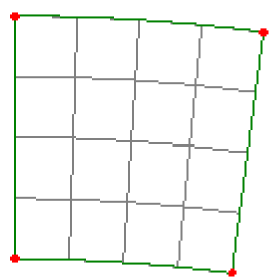


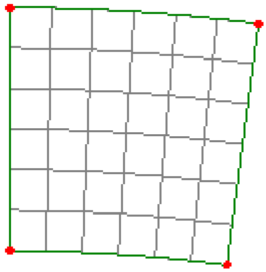
Meshing

This analysis will consider only out-of-plane bending effects, and no in-plane behaviour therefore plate elements will be used. Note that when thick plates are used care must be taken to ensure that results are transformed correctly when plotting contours and values of bending moments.

Attributes
Mesh >
Surface...

- Select **Thick plate, Quadrilateral, Linear** elements. Do not specify any mesh discretisation parameters, this will be controlled by the mesh divisions on the lines.
- Enter the dataset name as **Thick plates** and click **OK**
- With the whole model selected, drag and drop the surface mesh dataset **Thick plates** from the Treeview onto the selected features.





The default number of line mesh divisions of 4 divisions per line will be drawn. This gives a somewhat coarse mesh so the default number of line mesh divisions will now be changed.

File
Model Properties...

- With the **Meshing** tab selected, set the default number of mesh divisions to **6** and click **OK**

The new mesh divisions will be displayed.

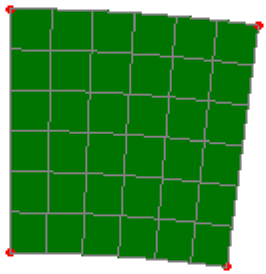
Geometric Properties

Attributes
Geometric >
Surface...

- Enter a thickness of **0.7**

The eccentricity box should, in this case, be left blank. Using an eccentricity would infer some in-plane effects, when, as described above, this analysis is intended to consider only out-of-plane effects. In 3D structures where behaviour is dependent upon both flexural and membrane effects– shell elements may be used.

- Enter the dataset name as **Slab thickness** and click **OK**



- With the whole model selected, drag and drop the geometric dataset **Slab thickness** from the Treeview onto the selected features.

Geometric property assignments are visualised by default.

- In the Treeview re-order the layers so that the **Attributes** layer is at the top, the **Mesh** layer is in the middle, and the **Geometry** layer is at the bottom.

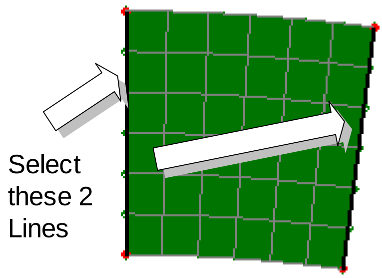
Material Properties

Attributes
Material >
Material Library…

- Select material **Concrete** of type **Ungraded** and click **OK** to add the material dataset to the Treeview.
- With the whole model selected (**Ctrl** and **A** keys together) drag and drop the material dataset **Iso1 (Concrete Ungraded)** from the Treeview onto the selected features and assign to the selected surfaces by clicking the **OK** button.

Supports

LUSAS provides the more common types of support by default. These can be seen in the Treeview. In this example the deck is to be simply supported so the **Fixed in Z** support will be used.

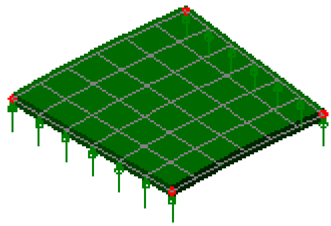


- Select the 2 support Lines shown.
- Drag and drop the support dataset **Fixed in Z** from the Treeview onto the selected Lines.
- Ensure the **Assign to lines** and **All analysis loadcases** options are selected and click **OK**

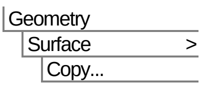
The supports will be visualised on the model.

To check the location and direction of the supports:

Use the isometric view button to rotate the model.



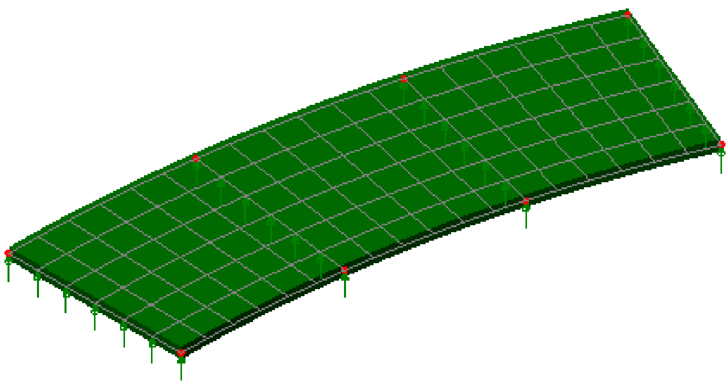
- To complete the full model of the bridge deck select the whole model using the **Ctrl** and **A** keys together.



Select the **Rotate** option and copy the features selected through an angle of **-7.5** degrees about the **Z-axis** and about the origin **0,0**

- Enter the number of copies as **2** and click the **OK** button.

The full 3-span model of the bridge deck will be created.



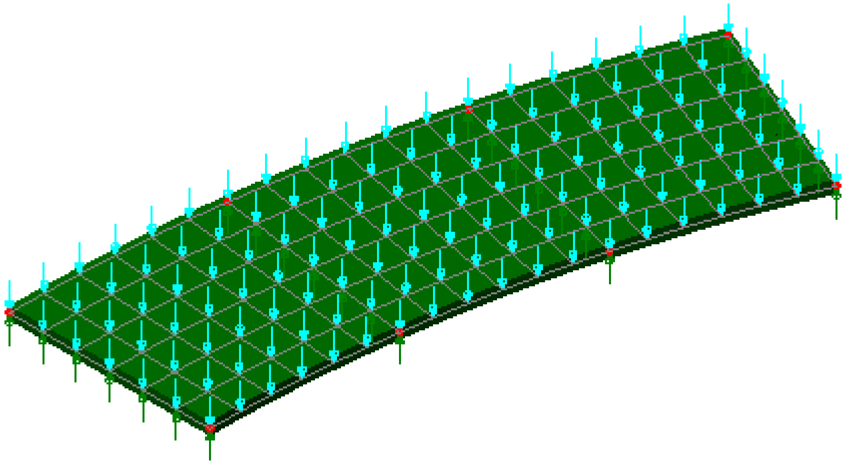
Note. Many of the more common tasks are provided in a context menu for the View window. With the model geometry selected clicking the right-hand mouse button within the view will display a context menu with **Copy**, **Delete**, **Move** and **Sweep**.

Self-Weight

Bridge
Bridge Loading >
Gravity

This command adds **BFP1 (Gravity -ve Z)** to the Treeview.

- With the whole model selected, drag and drop the loading dataset **BFP1 Gravity -ve Z** from the Treeview onto the selected features. Ensure the **Assign to surfaces** option is selected and click **OK** to assign to **Analysis 1**, **Loadcase 1**.



The gravity loading will be visualised on the model.

- In the Treeview, using the right-hand mouse button, select **Loadcase 1** and select the **Rename** option. Change the loadcase name to **Self Weight**

Defining a local coordinate system

Since this model is a curved deck a cylindrical local coordinate system will be used for the direction definition. Specifying a direction definition sets the vertical, longitudinal and transverse axes for a model to assist with model orientation and the calculation of particular effects. A local direction definition is used for aligning influence attributes along a singly curved bridge deck.

Attributes
Local Coordinate…

- Select the **Cylindrical** option and ensure that the Z-axis option is set. Enter a dataset name of **Cylindrical about Z-axis** and click **OK**

To use this local coordinate system:

Utilities
Direction definition…

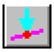
- In the longitudinal section of the dialog set the local axis to be **Cylindrical about Z-axis** for **Theta** and click **OK**.

Influence Attributes

In order to use the vehicle load optimisation facility, it is necessary to identify what quantity is to be optimised and at what location in the structure. This is achieved by specifying appropriate influence surfaces - by defining and assigning influence attributes to the model. LUSAS generates influence surfaces for each of the chosen locations and load effects, and in this case, by using the Muller-Breslau (reciprocal) theorem. Note that Direct Method influence attributes are also available and provide different capabilities.

To make it easier to select the nodes required to define the influence parameters the geometric and load visualisation will be removed from the display.

 Select the fleshing on/off button to turn-off the geometric visualisation.

 Select the loading on/off button to turn-off the loading visualisation

- Turn off the display of the **Geometry** layer in the  Treeview.

For this example three influence surfaces will be defined.

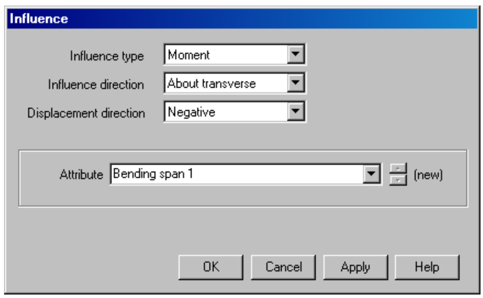
Influence Surface Definition - Edge Mid-span Bending

The first influence attribute to be defined will be used to investigate mid span edge bending in the first span.

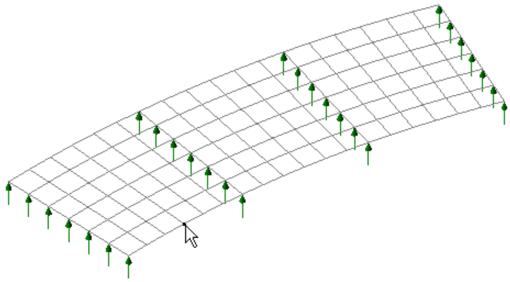
Attributes
Influence >
Reciprocal Theorem...

- Select a **Moment** influence type **About transverse** influence direction for a **Negative** displacement direction (because we are interested in the maximum sagging moment).

- Enter the influence attribute name as **Bending span 1** and click **OK**.



- Select the mid-span node at the inside edge of the first span.
- Drag and drop the influence attribute **Bending span 1** from the Treeview onto the selected node.

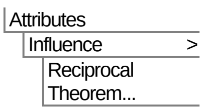


This adds an influence assignment (with a corresponding coordinate) to the Treeview.

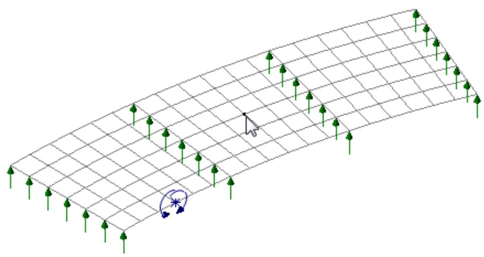
Note. Influence attributes can be assigned to as many nodes or points on the model as necessary. So, for example, the Bending span 1 influence attribute could be assigned to all nodes in the first span to specify influences for every node (even though not all nodes are necessarily required to be examined). However, for this example, and for speed and clarity reasons, only one node will be selected for each influence attribute.

Influence Surface Definition – Mid-span Bending Span 2

The second influence attribute to be defined will be used to investigate mid-span bending in the middle of the slab in the second span.



- Select a **Moment** influence type **About transverse** influence direction for a **Negative** displacement direction (because we are interested in the maximum sagging moment).
- Enter the influence attribute name as **Bending span 2** and click **OK**.
- Select the node at mid-span of the second span.
- Drag and drop the influence attribute **Bending span 2** from the Treeview onto the selected node.



This adds the influence assignment to the Treeview.

Tip. Giving influence attribute names that relate to each span and support (rather than just defining one influence attribute called 'Bending' or 'Reaction' enables easier identification of the influence results in the Treeview.

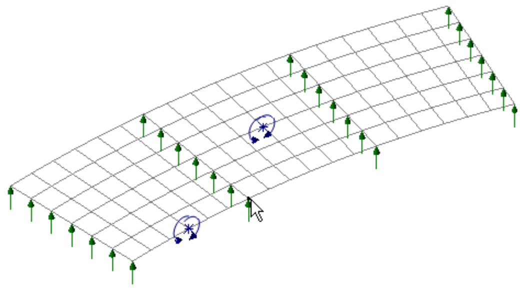
Influence Surface Definition - Maximum Reaction

The third influence attribute to be defined will be used to investigate the maximum reaction at the inner edge of the second line of supports.

Attributes
Influence >
Reciprocal Theorem...

Select a **Reaction** influence type in the **Vertical axis** influence direction for a **Positive** displacement direction.

- Enter the influence attribute name as **Reaction support 2** and click **OK**.
- Select the node at the edge of the first support as shown.
- Drag and drop the influence attribute **Reaction support 2** from the Treeview onto the selected node.

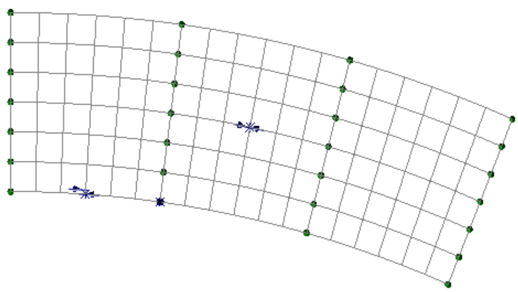


Visualising the Defined Influence Points

Influence attribute assignments are visualised as they are assigned to the model. To check that the influence attribute orientations are correct (meaning that the correct influence directions have been defined) a plan view of the model should be used in this case

Select the Home button to return the model view to a plan view.

- Ensure that the influence visualisations are in the orientations shown.



Define the Kerb Lines

Kerb lines are used to define the extent of a loadable carriageway.

- Turn on the display of the **Geometry** layer in the Treeview.
- Select the Point at the left-bottom corner of the deck

Geometry
Point >
Copy...

Enter a translation of **0.5** in the **Y** direction. Click **OK**.

- Select the Point just created

Geometry
Point >
Copy...

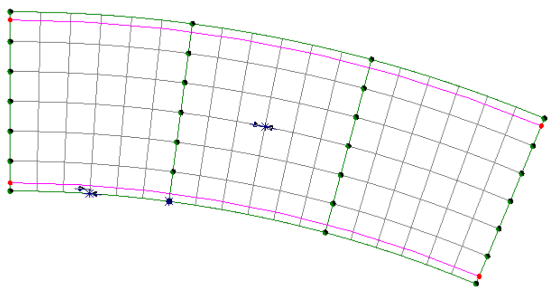
Enter a translation of **10** in the **Y** direction. Click **OK**.

- Select the two Points that have been created

Geometry
Line >
By Sweeping…

Select the **Rotate** option and enter an angle of **-22.5** degrees to rotate about the **Z-axis** about the origin **0,0**

- Click **OK** to sweep the Points to create a two lines representing the kerbs



Save the model

File
Save

Save the model file.

This completes the preparation of the model for influence analysis.

Rebuilding the Model

If errors have been made in the modelling of this example that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully.

❑ **deck_modelling.vbs** carries out the modelling of the example including the loading.

File
New…

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 **deck**

File
Script >
Run Script...

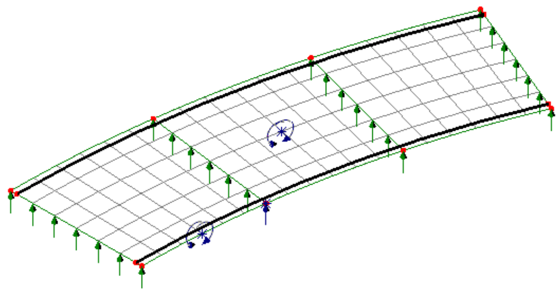
- To recreate the model, select the file **deck_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Using the Vehicle Load Optimisation facility to calculate the most critical loading patterns

The vehicle load optimisation software option automates the creation of traffic load patterns in accordance with a selected design code for locations specified on the model. The example as written uses EN1991-2 Recommended Values, which is supported by the LUSAS TLO software option. Other EN1991-2 National Annexes may be chosen. Results for selected other National Annexes are shown at the end of the example.

Select the kerb lines

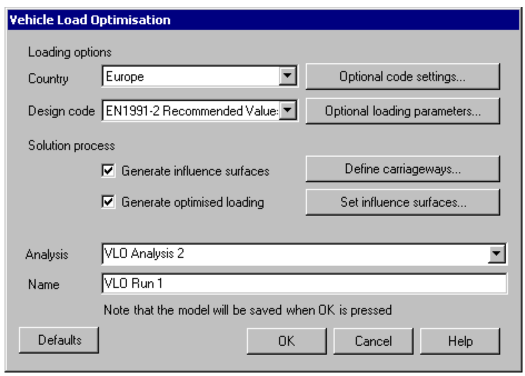
Prior to running the vehicle load optimisation the kerb positions defining the extent of the loadable area of carriageway must be selected. Ensure that the Geometry layer is turned-on, then:



- **Select the two lines** representing the kerb positions / extent of the traffic lane.

Define Vehicle Load Optimisation parameters

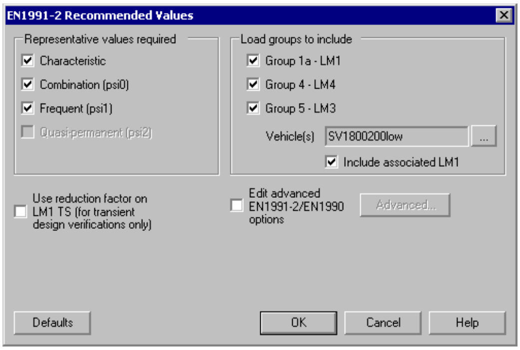
Bridge
Vehicle Load Optimisation…



- First select the **Defaults** button to reset any VLO settings from any previous use.
- Select **Europe** from the Country drop down list.
- Select **EN1991-2 Recommended Values** from the Design code drop down list.
- **VLO Analysis 2** will be automatically entered for the Analysis entry in the Treeview. (Note that an alternative name can be entered by selecting the New option from the drop-down list).
- Enter the VLO run Name as **VLO Run 1**

Browse the optional code settings

- Select the **Optional code settings** button and the optional loading parameters dialog will appear for the design code selected.



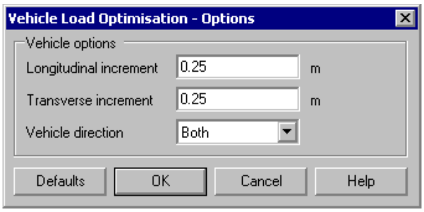
On this dialog any representative values that are required and load groups that are to be included can be specified. Special vehicle types can be specified and advanced settings can be defined. No changes are required for this example.

Note that a single special vehicle (**SV1800200low**) is specified as a default in these recommended values and that a vehicle(s) appropriate to the structure being designed should be specified in practice.

- Click the **OK** button to accept the default values and return to the main VLO dialog

Browse the optional loading parameters

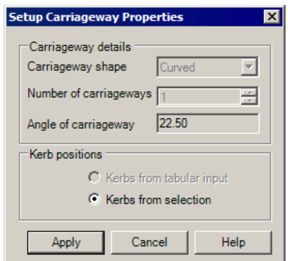
- On the main VLO dialog, select the **Optional loading parameters** button
- On this Options dialog ensure the **Longitudinal increment** is set to **0.25,** the **Transverse increment** is set to **0.25** and the Vehicle direction is set to **Both.**



- Click **OK** to return to the main Vehicle Load Optimisation dialog.

Define the carriageways

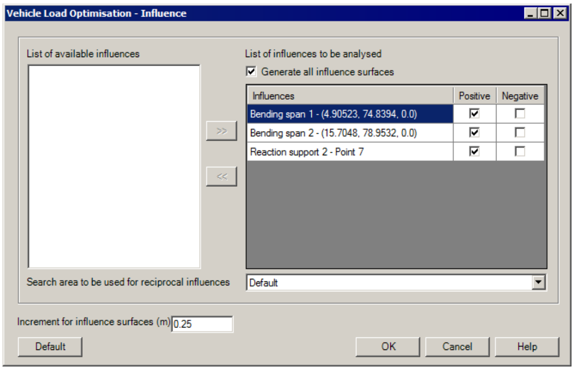
- On the main VLO dialog, select the **Define carriageways…** button.



Because the kerb lines have been selected the carriageway shape and angle of carriageway are known. **Kerbs from selection** will be automatically selected

- Click **Apply** to return to the main Vehicle Load Optimisation dialog.

Define the influence surfaces



- On the main Vehicle Load Optimisation dialog, select the **Set influence surfaces** button.
- Ensure the **Include all influence surfaces** option is selected.
- Ensure the **Increment for influence surfaces** is set to **0.25**
- Ensure the **Positive** direction option is selected for each influence
- Click **OK** to return to the main load optimisation dialog.

Important. The definition of the data for the load optimisation is now complete but **before** pressing the final **OK** button (which would run the vehicle load optimisation analysis) for the purposes of this example the influence surfaces will be visualised:

Solving and Visualising Influence Surfaces

If it is required to visualise influence surfaces and confirm they are correct before they are used to generate optimised loading patterns, this can be optionally done. With the main Vehicle Load Optimisation dialog displayed:

- Deselect the **Generate optimised loading** option and click the **OK** button.

The LUSAS Solver will process each influence surface in turn, but note that it is not until the Solve Now button is pressed (see next step) that loadcase results for the influence surfaces are added to the Treeview.

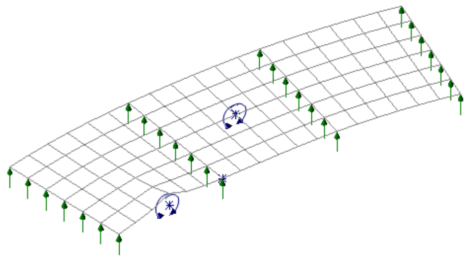
- Press the Solve Now button and deselect the Analysis 1 option to leave just the **Reciprocal Influence Analysis** check box ticked. Click the **OK** button.
- When solved, the Treeview will contain one results file for each influence surface, and an Influence Shape entry will be added to the Treeview.
- If present, turn-off the display of the **Mesh** and **Geometry** layers from the Treeview.
- Add or ensure that the **Influence shape** entry is present in the Treeview, and ensure that a deformation of **6mm** is specified.

- In the  Treeview double-click on the Attributes entry, click the **Supports** tab, and select the **All** button. Select the option **From results file** and click **OK.**

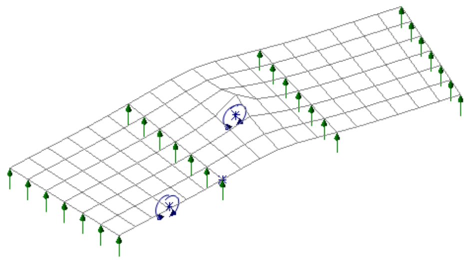
 Rotate the model to an isometric view.

- From the  Treeview right-click on each results loadcase and select the **Set Active** option to view each influence surface.

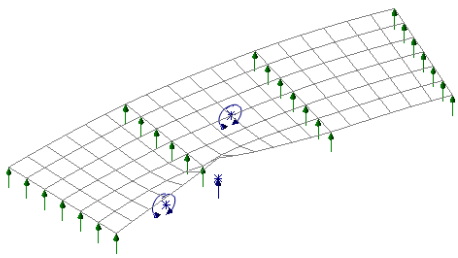
Bending span 1 (moment) influence surface



Bending span 2 (moment) influence surface



Reaction support 2 (reaction) influence surface



Generating Optimised Loading Definitions

- Double-click the **VLO Run 1** entry in the Treeview to re-access the settings made for this analysis.
- Deselect the **Generate Influence Surfaces** option to make use of the previously generated influence surfaces.
- Select the **Generate optimised loading** option.
- Click **OK** to run the vehicle optimisation process.

This loading optimisation will take just a few seconds on modern personal computers, and when successfully completed, Characteristic, Combination and Frequent loadcases will appear for each influence surface in the Treeview.

Visualising Load Definitions

- In the Treeview turn off the display of the **Influence shape** layer.
- In the Treeview turn on the display of the **Mesh** layer.

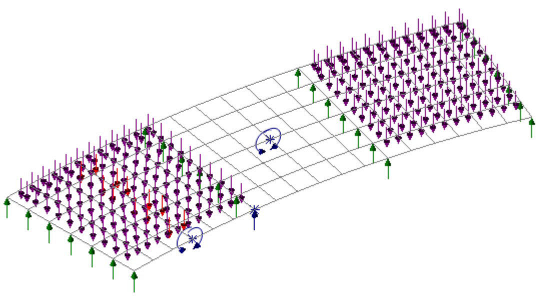
Select the loading on/off button to **turn-on** the loading visualisation

If necessary, select the supports on/off button to **turn-on** the support visualisation

The load definition may be visualised for each loadcase as follows:

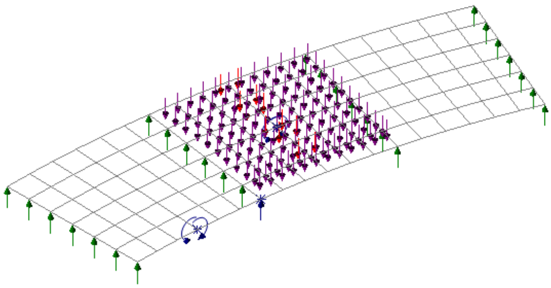
Viewing Characteristic Loading for Edge Mid-span 1

- In the Treeview right-click on the loadcase **Bending span 1 - (*x,y*) – Characteristic** and pick the **Set Active** option.



Here, it can be seen that Group 1a dominates and the load pattern is made up of LM1 tandem systems and LM1 udl patches. The 10m carriageway width accommodates three 3m wide lanes. With reference to EN1991-2:2003 table 4.2 and clause 4.2.4 (4), Lane Number 1, with the heaviest tandem and udl loads, is positioned adjacent to the influence definition on one side of the deck. Moving away from the location of interest, lanes 2 and 3 appear in order. The remaining area (1m wide) appears on the far side of the bridge deck. Spans 1 and 3 only are loaded (span 2 is not part of the adverse area).

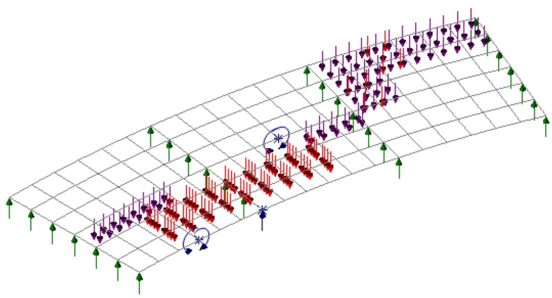
Viewing Characteristic Loading for Mid-span Bending Span 2



- In the Treeview right-click on the loadcase **Bending span 2 - (*x,y*) - Characteristic** and pick the **Set Active** option.

Here, again, Group 1a dominates and the load pattern is made up of LM1 tandem systems and LM1 udl patches. Lane Number 1, with the heaviest tandem and udl loads, is positioned adjacent to the influence definition, near the middle of the deck. Lanes 2 and 3 appear each side of Lane 1 and the 1m wide remaining area appears on the opposite side of the bridge deck compared to that when considering the previous loadcase. This illustrates how the ranking and location of lanes is modified as appropriate (EN1991-2:2003 clause 4.2.4(2)). Span 2 only is loaded (spans 1 & 3 are not part of the adverse area).

Viewing Characteristic Loading for Reaction at Support 2



- In the Treeview right-click on the loadcase **Reaction support 2 - (*x,y*) - Characteristic** and pick the **Set Active** option.

In this last example, giving maximum reaction at the inner support, Group 5 dominates and the load pattern is made up of LM3 with associated LM1 tandems and udl patches. The SV1800/200 is positioned adjacent to the influence definition, near the edge of the deck over spans 1 and 2. This is considered to be Lane Number 1 (EN1991-2:2003 Annex A clause A.3). Lanes 2 and 3 appear on the far side of the deck and are loaded only in the adverse area which happens to be in span 3. The 1m wide remaining area appears between lanes 1 and 3, illustrating yet another arrangement of lane rank and location, used to generate the most onerous traffic load pattern.

Note. The VLO run includes loading arrangements but no results. Results are available only after the VLO analysis has been solved.

Solving for the optimised loading arrangements

Only the VLO analysis has to be solved in order to produce the results for the optimised loading arrangements for each influence assignment.

- Press the **Solve Now** button.

The Solve Now dialog indicates that the VLO analysis results are not up to date and that they need to be solved. The other two analyses are unselected by default, as their results are up to date.

- Press **OK** to solve the VLO run.

Viewing Load Combinations

The LUSAS TLO facility outputs a design value – Characteristic, Combination or Frequent, with the meaning of each given in the TLO Help. In this example the vehicle loading is assumed to be the dominant effect and all other variable actions (wind, snow, temperature etc.) are being ignored. For this case, and in accordance with EN1990 6.4.3.2(3) equation 6.10, the loads can be combined as γ_G*[dead loads] + γ_Q*[traffic loads]. According to clause A2.3.1(4), for the design of structural members not involving geotechnical actions, the γ factors can be found in Table A2.4(B) as $\gamma_G = 1.35$ and $\gamma_q = 1.5$ for traffic. So, to define a basic combination:

Analyses
Basic Combination

On the Combination Properties dialog:

- From the **model** listing add **Self Weight** to the included panel.
- Add **Bending span 1 - (*x*,*y*) - Characteristic** to the included panel.
- Click the Grid button and enter factors of **1.35** for self weight loadcase and **1.5** for the Bending span 1 loadcase
- Change the combination name to **Combination Bending Span 1** and click the **OK** button to update the combination in the Treeview.

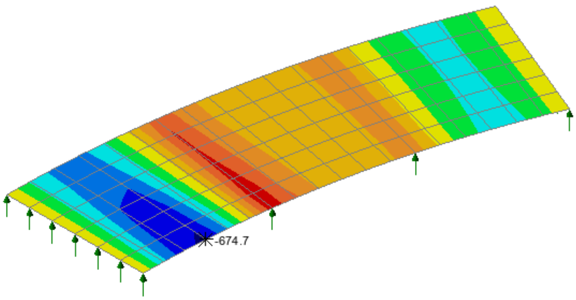
Plotting Contours of Moment and Peak Values

- In the Treeview right-click on the loadcase **Combination Bending span 1** and pick the **Set Active** option.
- The Solve Now dialog is displayed showing the status of the analyses that have been defined. Click **OK** to resolve the analyses indicated as needing to be solved.
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Contours** option to add the contours layer to the Treeview. Note that by default, and in accordance with established theory, thick plate element results are given along a chosen global axis. Because we wish to look at radial results along the deck the cylindrical local coordinate system must be selected in order to make the appropriate results components available for selection.

- Select Entity **Force/Moment - Thick Plate** and then select the **Transformed** button and select the **Local coordinate** option. Ensure **Cylindrical about Z-axis** is selected in the drop down list. Leave the **Shell plane for resultants** as **theta/z** and click **OK.**

- Now, as a result of transforming the results a new component is available in the Component drop down list. Selected moments in the theta direction **Mt**

- Click the **OK** button to display contours of moments in the longitudinal direction.



- To display the mesh on top of the contours, select the **Mesh** entry in the Treeview and drag and drop it onto/below the **Contours** entry in the Treeview.

- 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.

The values properties dialog will be displayed.

- Select Entity **Force/Moment - Thick Plate** and Component moments in the theta direction **Mt**, and select **Averaged at nodes.**

- Select the **Values Display** tab.

- Deselect the **Symbols** display option.

- Deselect the **Maxima** option and select the **Minima** option since the sagging moment is negative. Change the percentage of values to display to **1**

- Set the number of significant figures to **4**

- Click the **OK** button to display contours of moments in the longitudinal direction with the bending moment value displayed.

Plotting stress contours and peak values for other load combinations and for other influence points of interest may be obtained in a similar manner to that described above.

Save the model

File
Save

Save the model file.

Note. When the model file is saved after results processing, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

This completes the example.

Discussion

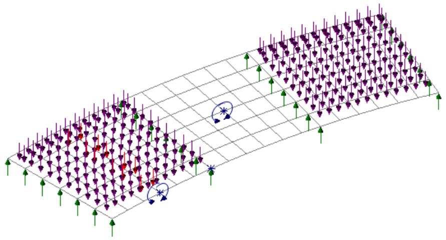
Traffic Load Optimisation for EN1991-2 design codes

To illustrate general usage this example analysed traffic load optimisation to EN1991-2 using the Recommended Values design code option.

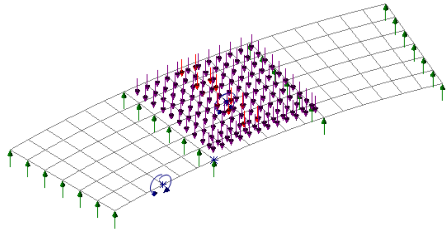
By re-selecting the lines representing the kerbs on the Geometry layer and selecting an appropriate design code on the main Vehicle Load Optimisation dialog and re-running the traffic load optimisation analysis, the results for other supported EN1991-2 Design Code and National Annexes values can be similarly obtained. When re-running the LUSAS TLO facility on the existing model care should be taken to ensure that the correct loadcase is included in the load combination and that the correct load factor has been specified.

Traffic Load Optimisation for EN1991-2 UK

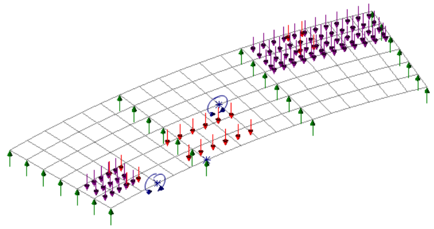
By following a similar procedure to that previously described the optimised load definitions for EN1991-2 UK (using a single SV80 special vehicle) can be obtained along with a contour plot for a basic combination of self weight and combination results for bending in span 1, as shown in the following images.



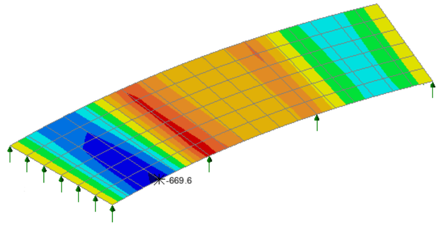
Optimised Characteristic Loading for Influence Point at Edge Mid-span 1



Optimised Characteristic Loading for Influence Point at Mid-span Bending Span 2



Optimised Characteristic Loading for the Influence Point at Reaction at Support 2

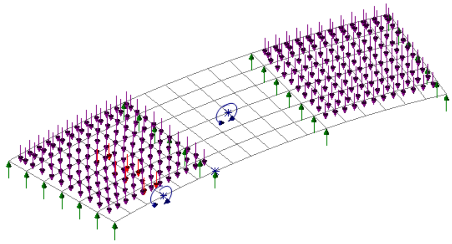


Combination results for Influence Point at Edge Mid-span 1

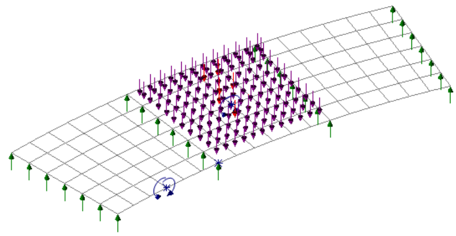
Here, the two bending influences show Group 1A loading to be dominant. The LM1 tandem and LM1 udl are however different in value to those in the earlier example due to the use of the appropriate adjustment factors from NA to BS EN1991-2 Table NA.1. The reaction influence shows the Group 5 to be dominant. The UK SV80 appears in Lane Number 1, with associated LM1 also appearing in Lane Number 1,with a minimum 5m distance from the SV to the first axle of the LM1 TS. Lane rank and location are similar to those described in the earlier example.

Traffic Load Optimisation for EN1991-2 Sweden

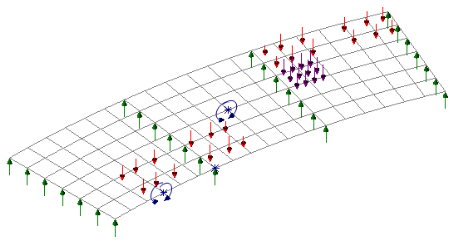
By following a similar procedure to that previously described the optimised load definitions for EN1991-2 Sweden (using all Complementary load model loads from ‘a’ to ‘l’) can be obtained along with a contour plot for a basic combination of self weight and combination results for bending in span 1, as shown in the following images.



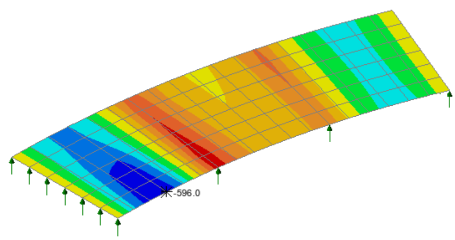
Optimised Characteristic Loading for the Influence Point at Edge Mid-span 1



Optimised Characteristic Loading for the Influence Point at Mid-span Bending Span 2



Optimised Characteristic Loading for the Influence Point at Reaction at Support 2



Combination results for the Influence Point at Edge Mid-span 1

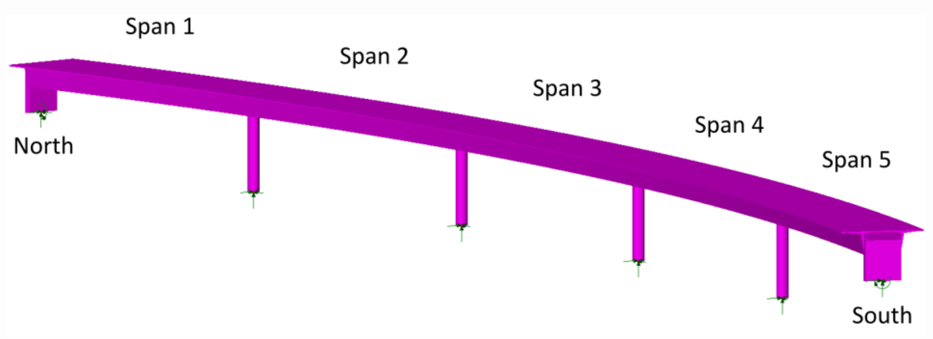
Here, the two bending influences show Group 1A loading to be dominant. In accordance with SIS/PAS NA to EN 1991-2:2003 clause 4.3.2(3) and VVFS 2009:19 Chapter 6, clause 4, Table 7.1, adjustment factors for the LM1 tandem and LM1 udl are applied – hence there is no LM1 tandem in Lane Number 3 (factor α_{q3}=0). Other comments on the EN1991-2 load patterns above also apply here. The reaction influence shows the Swedish Complementary Load Model to be dominant. Type L vehicles appear in Lanes 1 & 2, with udl q appearing in Lane 3 and no loading in the “remaining area”. Lane rank and location are similar to those described in the earlier example.

Vehicle Load Optimisation of a Box Beam Bridge

For software product(s):	Bridge and Bridge Plus
With product option(s):	

Description

A 5-span concrete box curved bridge is modelled with beams to illustrate the use of the Direct Method of Influence (DMI) and the Vehicle Load Optimization (VLO) facilities in LUSAS.



A pre-defined base model with assigned mesh, material and support attributes is used in this example.

Units used are kN, m, kg, s, C throughout.

Objectives

The required steps in the analysis consist of:

- ❑ Run a Direct Influence Method Analysis to calculate the effects of a unit load
- ❑ Define influence attributes and assign them onto the model
- ❑ Create influence surfaces
- ❑ Create optimised vehicle loading results with VLO

Keywords

Bridge, Concrete, Box Girder, Direct Influence Method, Vehicle Load Optimisation

Associated Files

- ❑ **curved_bridge_preliminary.mdl** Basic beam model of the bridge.

Modelling

Running LUSAS Modeller

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

Loading the model

File
Open…

To start this example, open the read-only file **curved_bridge_preliminary.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

The basic bridge geometry will be displayed.

File
Save As…

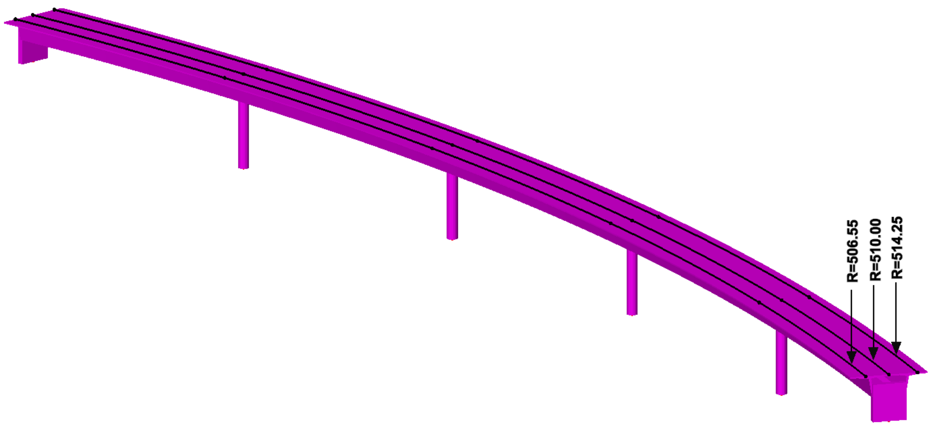
- Save the file as **\<LUSAS Installation Folder>\Projects\curved_bridge**
- Save the model into this new folder as **curved_bridge**

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Model Description

The 5-span, box girder bridge has spans of 45m, 50m, 50m, 50m and 45m, all set-out to a horizontal curvature of 510m radius. For simplicity the cross section of the bridge is considered constant across its length, and the deck is supported on 5.5m high abutments and 12.5m high circular pier columns, which are considered fully integral with it. The box girder of the deck, including cantilever outstands, is 11.5 m wide. The carriageway itself is 7.7m wide, with space for a footway on the inside and hard standing margin on the outside.

The deck is modelled as a series of beams along the centreline of the carriageway at a radius of 510m. The kerb lines defining the extent of the loadable carriageway are defined at the deck level, above the nodal line, using lines of 506.55m radius and 514.25m radius – a difference of 7.7m, which relates to the width of the carriageway.



The geometric lines of this model have been assigned a line mesh of thick beam elements with 10 divisions, and a cylindrical coordinate system has been used to orientate the abutments and piers supports. Concrete EU f_{ck}=50MPa has been assigned to all members. Pin supports restrain the base of the piers and other pin supports, having the additional fixity, THY, are used at the abutments. A local cylindrical coordinate system has been assigned to the end points to orientate the supports as needed.

The Direct Method Influence method is to be used for vehicle load optimisation analysis instead of the Reciprocal Theorem because it allows the structure to be analysed as a simple line-beam model.

Defining a reference path

- A reference path will be created to be used as centreline to define a grid which will be automatically used for loading locations and creating the influence shape in Direct Method Influence Analysis. The use of a reference path is recommended for most vehicle loading and load optimisation situations.

- From the Treeview right-click on **Deck** and click **Select Members** to select the lines representing the bridge centreline.

Utilities
Reference Path…

- Change the name to be **Bridge Centreline** and click **OK**. A corresponding path definition entry will be created in the Treeview.

Defining a search area

A search area can be used to limit the area or features over which loads are applied, so that the effect of the load on certain features may be removed from the analysis. It is strongly recommended to define a search area for VLO analyses.

Attributes
Search Area…

- Change the name to **Deck** and click **OK**.

- From the Treeview, drag and drop the newly defined **Deck** search area on to the model to assign the attribute to the already selected bridge centrelines.

Defining a direct method influence analysis

A Direct Method Influence analysis is defined in order to investigate the most onerous traffic load patterns on the carriageway, to a selected code of practice. The direct method influence enables the construction of influence surfaces for any results component at any node in the structure.

Analyses
Direct Method
Influence Analysis…

- Choose **Deck** as the **Search area** in order to apply a unit load only on the deck beams.

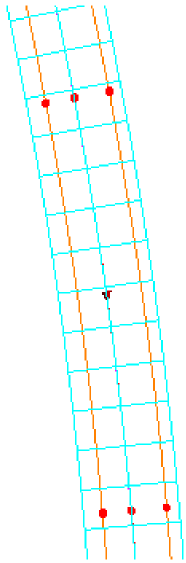
For a line beam model, where the geometric section represents a beam with a loadable top slab, a grid of points must be defined to represent the slab. This virtual grid is equivalent to the nodes or points present in a shell or plate model and is used to create an influence surface for location of interest.

- Ensure the **Grid** option is selected, and that the chosen Centreline is the previously defined **Bridge Centreline** reference path.

- Set a Transverse width of **11.5**. This is the width of the grid to be loaded for influence analysis. Grid settings control the number of load locations within this defined width.

Note. The Grid Settings advanced dialog controls the spacing and number of load locations within the defined width and search area length. For this analysis the default spacing will be used resulting in a minimum of 50 longitudinal points by 2 transverse points. Using a larger number of transverse points across a single beam will not make the analysis more accurate, and is not required because results are only calculated for the two extreme fibres. If a very fine grid was defined the solution stage may take some time, as the solution time is proportional to the mesh and grid density.

- Change the name to **DMI** and press **OK** to exit the dialog.



A defined grid is now shown on the model and will be used to apply the unity load on the grid locations that are defined on the structure.

The grid can be displayed or hidden on the screen.

- From the Treeview right click on **DMI** and select **Show Grid** to un-tick the entry and make the grid invisible.

Running the Analysis: Influence Analysis

- Press the **Solve Now** button. On the Solve Now dialog ensure both **Analysis 1** and **DMI** are checked and press **OK** to run the analysis.

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 is written to the output files 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.

Defining influence attributes

Now that the influence analysis has been solved the construction of influence surfaces for any component and any node in the structure can take place.

Attributes
Influence >
Direct Method…

- Select Entity as **Force/Moment-Thick 3D Beam**
- Ensure the Direction is **Local**
- Select **My** as the Component
- Choose **Automatically choose elements for averaging** to create an individual influence based on the average of all element values at the node of interest.
- Enter **My Influence** as name and press **Apply**
- Repeat to define influence attributes for **Mx** and the **Fx c**omponents, naming them **Mx Influence** and **Fx Influence** respectively. Press **OK** after defining the final attribute to close the dialog.

Once created, an influence attribute is held in the Treeview for assignment to selected nodes or points. Subsequent assignment of the same attribute to other nodes or points of interest will use the same influence type settings.

- From the Treeview turn **Geometry** and **Utilities** off.
- Select the node in the mid-span of span 2 (Node 1490) and assign the **My Influence** from the Treeview onto it.

A new influence loadcase will be added to the **DMI** analysis in the Treeview, indicating the name and position of the influence attribute assigned.

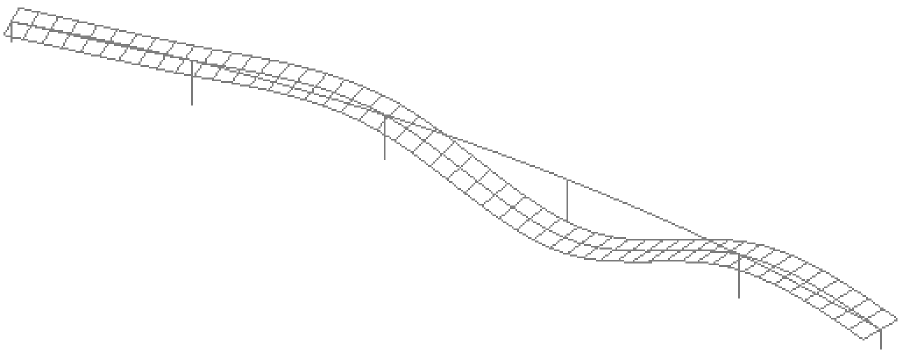
- Assign **Mx Influence** from the Treeview to the centre of span 3 (Node 1071).
- Assign **Mx Influence** from the Treeview to the base of the south abutment (Node 25).

Note that each **Mx Influence** assignment creates a new influence loadcase in the **DMI** analysis Treeview.

- From the Treeview turn **Geometry** on,
- Assign **Fx Influence** from the Treeview to the head of the span 3/4 pier (Point 4).

Three new **Fx Influence** loadcases will be added to the Treeview. Each influence loadcase name identifies the specific higher-order feature for which the effects will be assessed. In this example **Fx Influence (Fx) – Point 4 – (Line 9)** represents the influence shape for the axial load in the top of the pier, whilst **Point 4 – (Line 4)** and **Point 4 – (Line 3)** represents the influence shape for the in-plane load at the connection along spans 3 and 4 respectively.

- From the Treeview right-click **Fx Influence – Point 4 – (Line 9)** and choose **Set Active** to view the influence shape for the axial load at the top of pier 3/4.
- With no features selected right-click in a blank part of the Graphics window and select the **Influence shape** option.
- From the Treeview turn **Geometry** off,



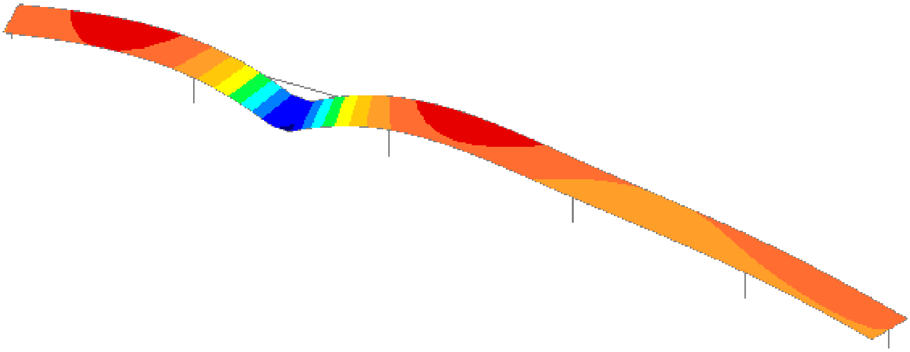
Note. In large models with lots of elements, nodes and attributes you can quickly locate an assigned influence by right clicking on the Direct Method Influence Analysis attribute assignment entry in the Treeview and then choosing **Find**. A temporary indicator will appear, highlighting the target node with the assigned influence attribute.

Contouring the influence results

After the influence attributes have been defined and assigned, contours can be added to the view to indicate just what effect the degree and direction of influence loading applied along the surface will have on the assigned node.

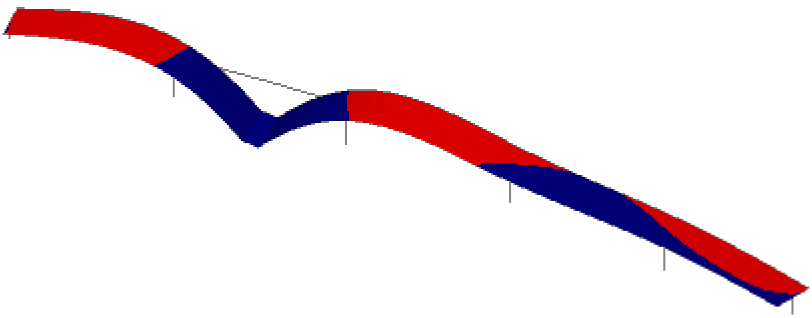
- From the Treeview right-click the **My** Influence and select **Set Active**
- With no features selected right-click in a blank part of the Graphics window and select the **Contours** option to add the Contours layer to the Treeview.
- Select the Entity **Influence result**. Note that **t**he Component **inf** will be automatically selected.

- Select the **Contour Display** tab and tick **Deform**
- Press **OK** to exit the dialog.



The contours can be simplified to indicate which areas of the structure, when loaded, will have a positive, or negative, influence of the selected influence assignment.

- Double-click the **Contours** layer to the Treeview to open the properties dialog
- Select the **Contour Range** tab.
- Ensure **Number** is selected and change the value to **1**
- Press **OK** to exit the dialog.



Positive influence areas are shown in red, negative influence areas are shown in blue..

Vehicle Load Optimisation

Now that influence attributes have been defined and assigned, the VLO facility can be used to define optimised traffic loading for the bridge, based on a chosen code of practice. In this example EN1991-2 UK will be used.

- From the Treeview turn off **Contours** and **Influence shape** and turn on **Geometry**
- In the Groups Treeview, right-click on the **Kerblines** group, and then select **Select Members** to select the lines representing the extent of the carriageway.

Bridge
Vehicle Load Optimisation...

- First select the **Defaults** button to reset any VLO settings from any previous use.
- Select **United Kingdom** from the Country drop down list and choose **EN1991-2 UK 2009** as the Design code.

For simplicity only the effects of a characteristic LM1 load will be considered in this example.

- Press **Optional Code settings** and deselect all options to leave **Characteristic** as the only representative values required, and **Group 1a –LM1** as the only one of the load groups to include. Press **OK** to exit the dialog.
- Press **Define carriageways**. Ensure **Kerbs from selection** is selected and press **Apply** to exit the dialog.
- Press **Set influence surfaces** and use the >> button to add **My Influence** to the list of influences to be analysed.
- Ensure that both **Positive** and **Negative** are selected for the My Influence to be analysed. This means that the VLO will produce loading for positive and negative effects of My. Press **OK** to return to the main VLO dialog.
- To specify a non-default analysis name choose **New** from the drop-down list, and enter the analysis name to be **VLO DMI**
- Change the VLO run Name to be **LM1 Span 2 My**
- Press **OK** to exit the dialog and generate optimised loading.

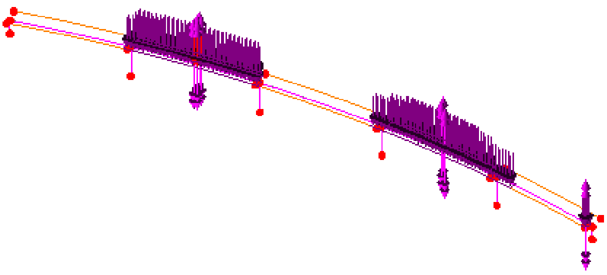
In the Treeview the new **VLO DMI** analysis will be created. This contains the **LM1 Span 2 My** VLO run, which includes a positive and a negative My Influence assignment.

Note. The VLO run includes loading arrangements but no results. Results are available only after the VLO analysis has been solved.

Check loading visualisation settings

- From the Treeview double-click **Attributes**
- Select the **Loading** tab and press **Settings...** button
- Ensure **Show discrete loading by definition** is selected and press **OK** to exit the dialog.
- Press **OK** to return to the model.
- From the Treeview select the **My Influence (My) - Negative** placed under the **VLO DMI** and set it active.
- Press to visualise loading.

The optimised loading for the characteristic case of EN1991-2 UK that gives the maximum My effect in the mid-span of span 2 for negative influence will be displayed.



Solving optimised loading arrangements

Only the VLO analysis has to be solved in order to produce the results for the optimised loading arrangements for each influence assignment.

- Press the Solve Now button.

The Solve Now dialog indicates that the VLO analysis results are not up to date and that they need to be solved. The other two analyses are unselected by default, as their results are up to date.

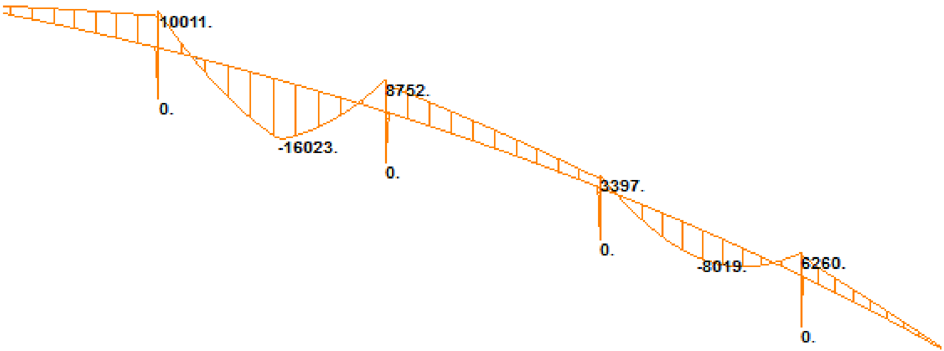
- Press **OK** to solve the VLO run.

The results will be loaded on top of the current model, with the first influence analysis (My Influence (My) – Positive) set active.

Plotting bending moments

- From the Treeview turn off the **Geometry** and **Attributes** layers.

- With no features selected right-click in a blank part of the Graphics window and select the **Diagrams** option to add the **Diagrams** layer to the Treeview
- From the diagram properties drop-down menu, pick **Force/Moment – Thick 3D Beam** from the entity and **My** from the component drop down list.
- From the **Diagram Display** tab untick the **Also use for labels** option, select **Decimal places** and change the value to **0**
- Press the **Label font...** button and change the style to **Bold**
- Press **OK** to return to the properties dialog, and then press **OK** exit.
- From the Treeview, within the **VLO DMI** analysis entry select **My Influence (My) - Negative** and set it active.



The diagram shows the worst-case sagging moment, My, for the centre of the second span, due to characteristic LM1 loading.

This completes the example.

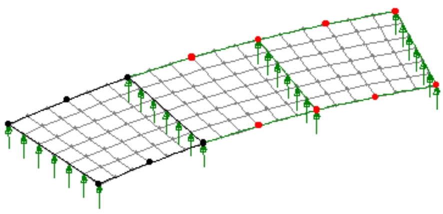
BRO Slab Analysis

For software product(s):	LUSAS *Bridge.*
With product option(s):	None
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example uses a previously created LUSAS bridge model that contains Swedish BRO loadings and loadcases. The BRO load combination wizard is used to investigate worst case positive and negative effects.

The structure is modelled using thick plate elements, representing a deck of inner radius 75m, outer radius 86m and thickness 0.7m. The deck has a width of 11m consisting of a 10m wide carriageway region and two 0.5m wide verges.



Objectives

- ❑ Generation of load combination in accordance with Swedish bridge code BRO

Keywords

2D, Slab, BRO Load Combinations, BRO Wizard, Positive Effects, Negative Effects

Associated Files

- ❑ **deck_comb_loaded.mdl** Model file of the structure.

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

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. Select the **Cancel** button. Select the menu command **File>Open** and then browse to the **\<Lusas Installation Folder> \Examples\Modeller** directory and select the **deck_comb_loaded.mdl** file.

File
Open…

Open the read-only file **deck_comb_loaded.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Loadcases considered

The model in use was created using the Swedish BRO startup template. When using a country-code specific template such as this, a number of blank loadcases are created in the Treeview. These are the characteristic loadcases that are considered possible within the Swedish code of practice BRO. Individual vehicle and lane loadings have also been created in the Treeview and assigned to these loadcases. The BRO load combination wizard will combine them in accordance with the Swedish code.

The following characteristic loadcases have been assigned to the model:

- Dead Load
- Surfacing
- Support yielding 1
- Support yielding 2
- Support yielding 3
- Support yielding 4
- Temperature 3
- Equivalent load 1 to Equivalent load 10

- View the BRO loadings and loadcases in the and Treeviews.

File
Save As…

- In the **\<LUSAS Installation Folder>\Projects** folder create a new directory called **deck_comb_loaded**
- Save the model into this new folder as **deck_comb_loaded** This helps keep all relevant files separate from other analyses and is good practice.

Note. Loadcases would normally be defined using the **Bridge > Bridge Loading > Sweden** menu entry which would display the dialog shown. However, because the purpose of this example is to concentrate on showing the use of the BRO Combination Wizard these loadcases have already been defined and assigned to the model that is in use.

Running the Analysis

With the model loaded:

Open the **Solve Now** dialog. Ensure **Analysis 1** is selected and press **OK** to begin 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 Treeview.

In addition, two files will be created in the Associated Model Data directory where the model file resides:

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

Viewing the Results

BRO Combinations

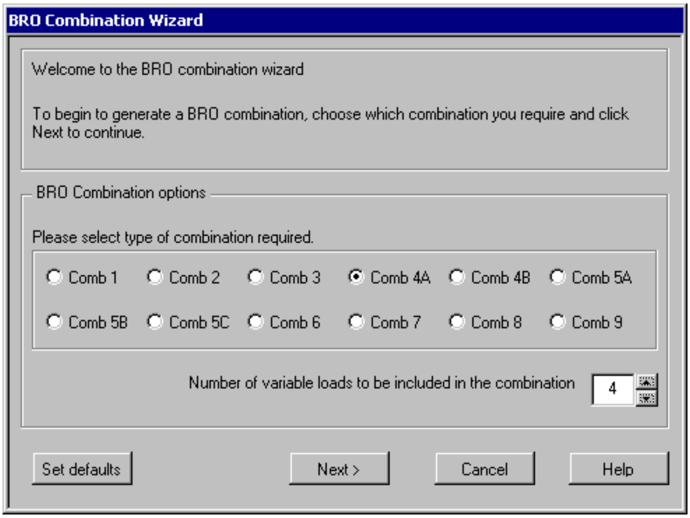
The BRO combination wizard allows you to choose which loads to combine into combinations. By selecting the combination type i.e. Comb4A (Combination 4A), the wizard defaults the load coefficients to the correct values as stated in table 22-1 of BRO. Once all the required permanent, variable and live loads have been selected the resultant combination can be produced. The resultant combination will be formed by combining a combination of permanent loads, support yielding combination or envelope, a combination of variable loads and envelopes of live loading.

Note. The BRO combination wizard can only be used if the BRO loadcase template has been specified to define the characteristic loadcases at the Model Startup stage.

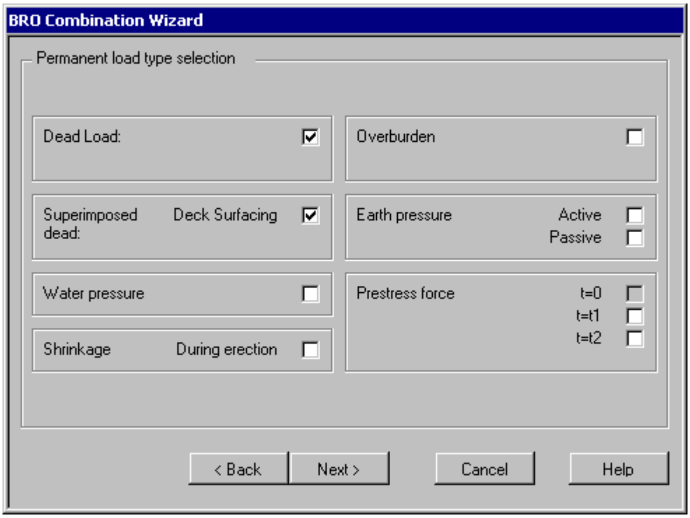
Bridge
Loading
Combinations >
BRO...

The BRO Combination Wizard dialog will appear.

- Select the **Set defaults** button.
- Select combination **Comb 4A**
- Ensure the **Number of variable loadcases** is set to **4**
- Click **Next**

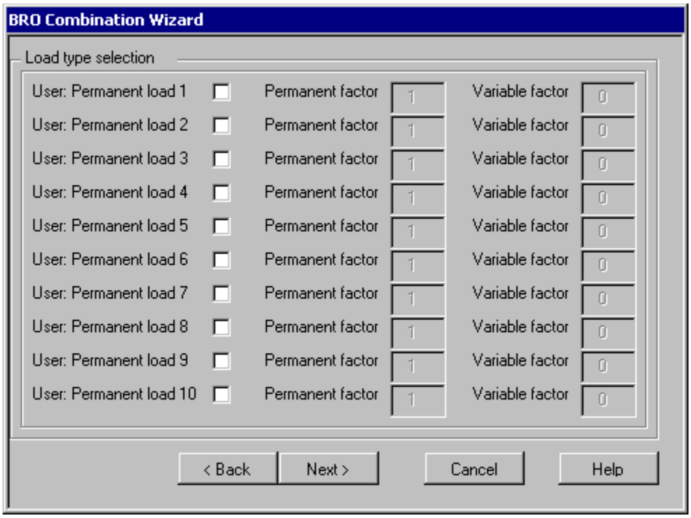


- For the permanent load type selection ensure the **Dead load** and **Superimposed dead: Deck surfacing** options are selected
- Click the **Next** button to continue.

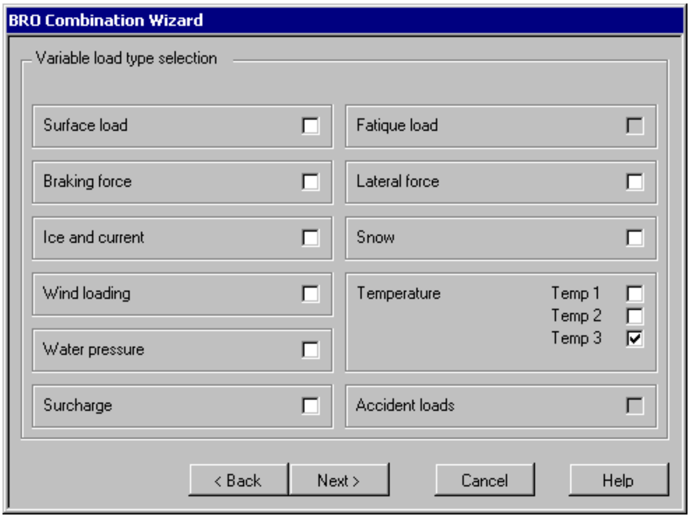


- In the Support yielding section select the **Combine envelope of loadcases** option.
- Click the **Next** button to continue.

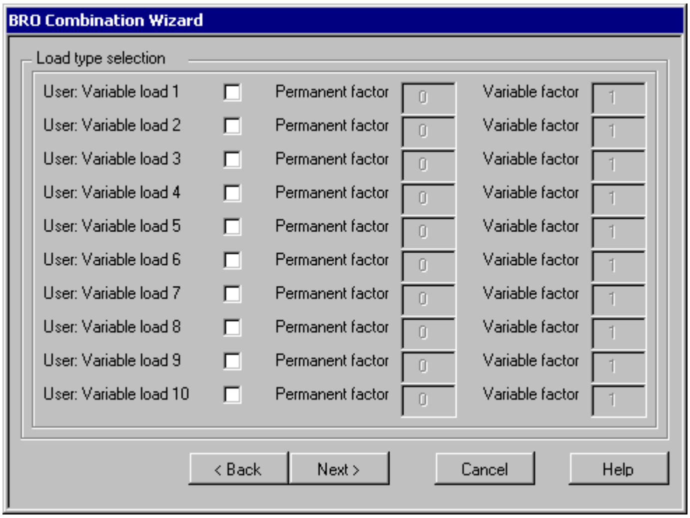
- No user defined permanent loads have been defined for this analysis so ensure the dialog is left blank and click **Next** to continue.



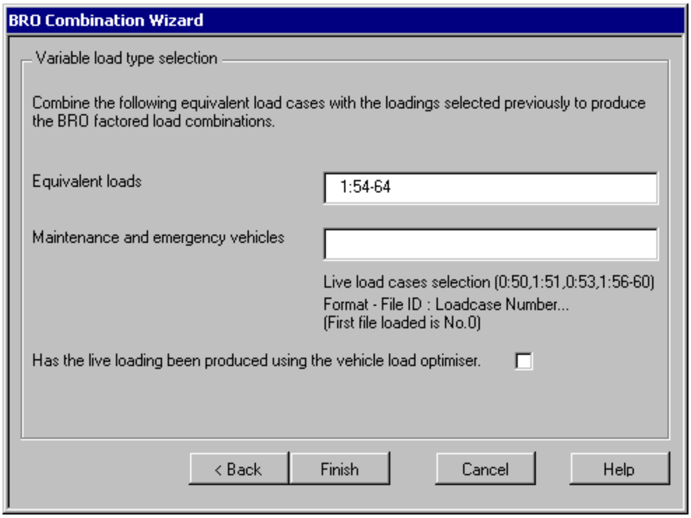
- For the variable load type selection pick **Temperature** type **Temp 3**
- Click **Next** to continue.



- No user defined variable loads have been defined for this analysis so ensure the dialog is blank and click **Next** continue.



- Finally, enter the **Equivalent loads** as attributes **1:54-64** (see the Treeview)
- Leave the **Maintenance and emergency vehicles** blank.
- Select **Finish** to generate the BRO Resultant combinations.



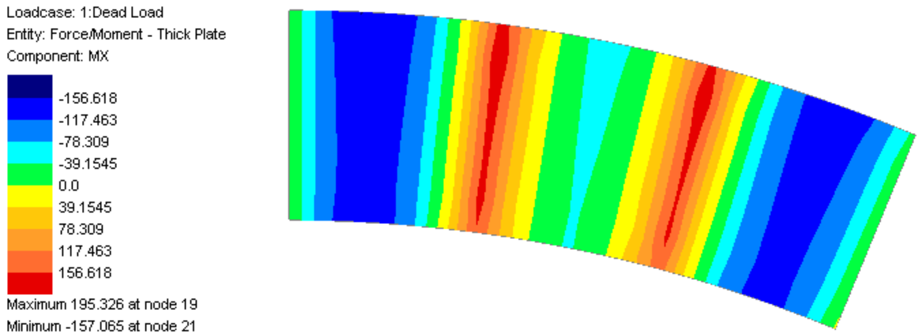
Displaying contours for BRO Max and Min Effects

- If present, turn-off the display of the **Geometry** and **Attributes** layers in the Treeview.
- With no features selected, click the right-hand mouse button in a blank part of the graphics area and select the **Contours** option to add the contours layer to the Treeview.

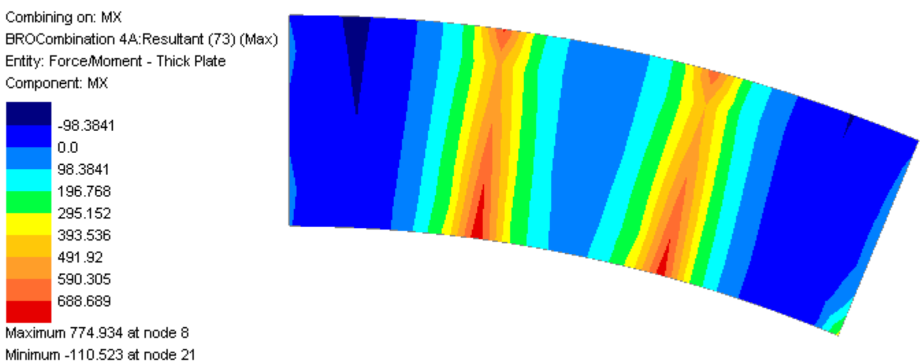
The contours properties dialog will be displayed.

- Select Entity **Force/Moment - Thick Plate** component moments in the X direction **MX** and click **OK**

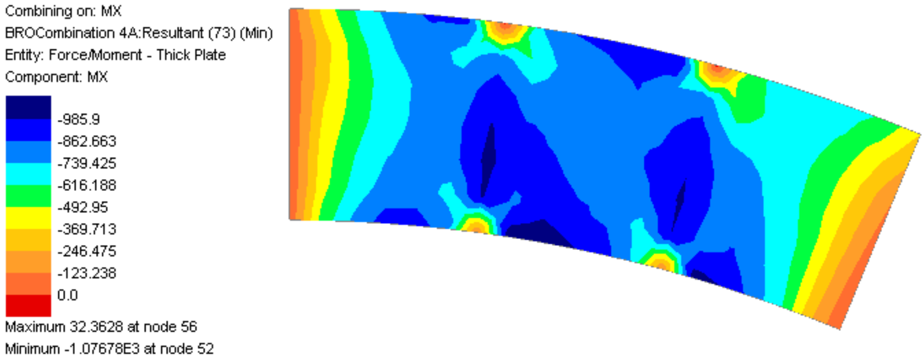
The Contour results for Loadcase 1 - Dead Load will be displayed.



- In the Treeview right-click on **BRO Combination 4A:Resultant(Max)** and select the **Set Active** option to view the worst positive effects (i.e. hogging moments and reactions). Select entity **Force/Moment - Thick Plate** and component **MX** to be used to assess the relieving or adverse affects when processing the combinations and click **OK**. To continue with partial results click **OK**



- Set combination the **BRO Combination 4A:Resultant(Min)** active to view worst negative effects (i.e. sagging moments and vertical displacements). Select entity **Force/Moment - Thick Plate** and component **MX** to be used to assess the relieving or adverse affects when processing the combinations and click **OK**. To continue with partial results click **OK**



Additional results processing can be carried out in a similar manner to that described in the Simple Slab example.

Save the model

File
Save

Save the model file.

Note. When the model file is saved after results processing, all load combinations, envelopes, and graph datasets, if defined, are also saved and therefore do not have to be re-created if the model is amended and a re-analysis is done at a later date.

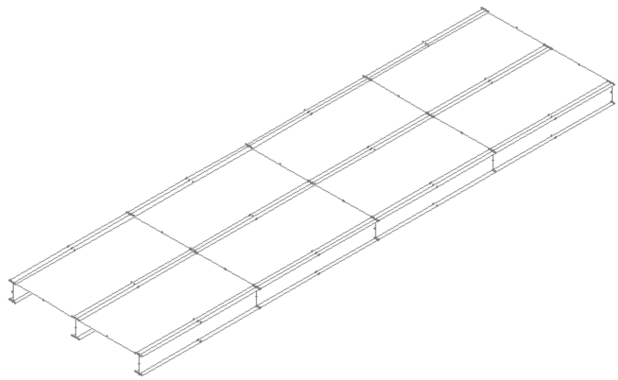
This completes the example.

Section Slicing of a 3D Shell Structure

For software product(s):	Any Plus version.
With product option(s):	None

Description

This example uses the Slice Resultants Beams/Shells facility to investigate the behaviour of a bridge deck modelled using thick shell elements. The results are converted into forces and moments of an equivalent beam using this facility.



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

Objectives

The output requirements of the analysis are:

- ❑ Forces and bending moments along length for whole structure
- ❑ Forces and bending moments associated with central girder effective width

Keywords

Force, Bending moments, Slicing, Shells.

Associated Files

- ❑ **shell_slicing.mdl** Model file of the structure.

Modelling

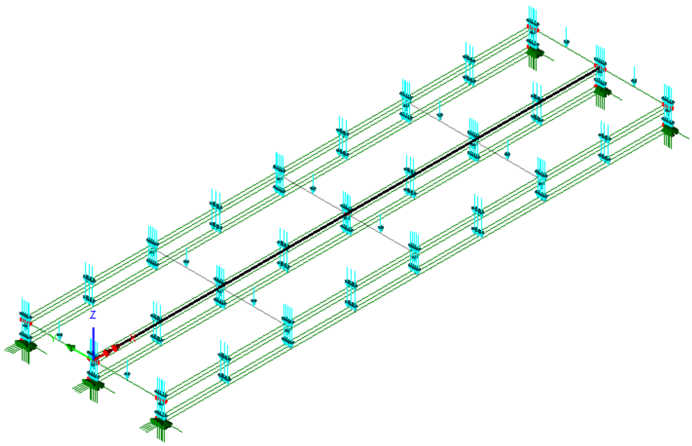
Running LUSAS Modeller

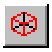
For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

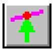
Creating a new Model

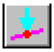
File
Open…

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



 If necessary, click the isometric button to view the model in 3D

 If necessary, turn-on the display of the supports.

 If necessary, turn-on the display of the loading.

File
Save As…

- Save the model to your local projects folder to prevent overwriting the model in the examples directory.

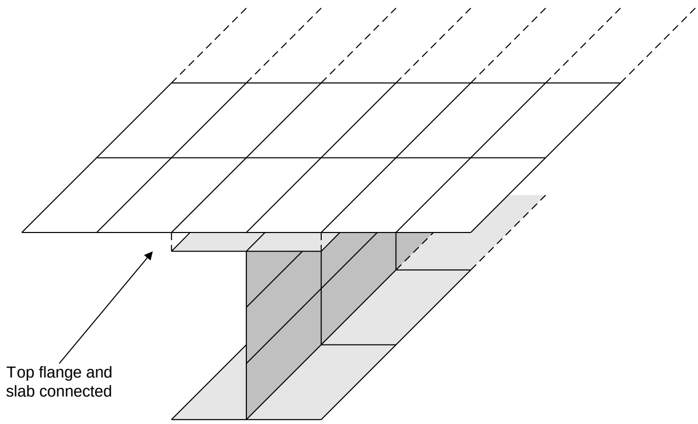
In the Treeview two loadcases can be seen. One corresponds to the self-weight of the structure and the other to an applied load of a United Kingdom HB vehicle. For the post-processing, a combination has also been defined which combines these two load cases.

Turning off all layers other than the mesh allows visualisation of the general layout of the model. The deck is 4.21m wide and consists of a concrete slab and three steel girders as indicated in the following figures. The mesh discretisation of the model is extremely coarse for the purpose of this example.

Note. When carrying out this type of slicing on realistic models it is recommended that a much finer mesh discretisation is used to accurately capture the response of the structure, particularly if 3- and 4-noded shell elements are being used for the modelling.

Idealisation of the Bridge Structure

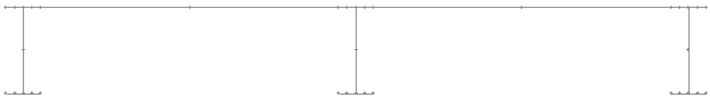
For this example the idealisation of the bridge structure into the analysis model has been done using only shell elements.

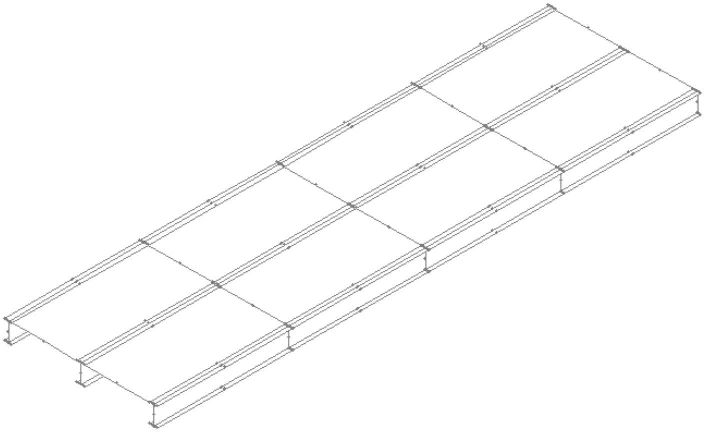


Shell element modelling of the whole structure

Shell elements have been used for the girder components and also the concrete slab as illustrated. For the modelling, discrete features have been defined for the top flange and the slab since their materials are not identical. The slab has an eccentricity to define a bending plane that is not coincident with the top flange and these two components are combined using an equivalence attribute. This is just one of many ways that the idealisation for this type of bridge structure can be undertaken.

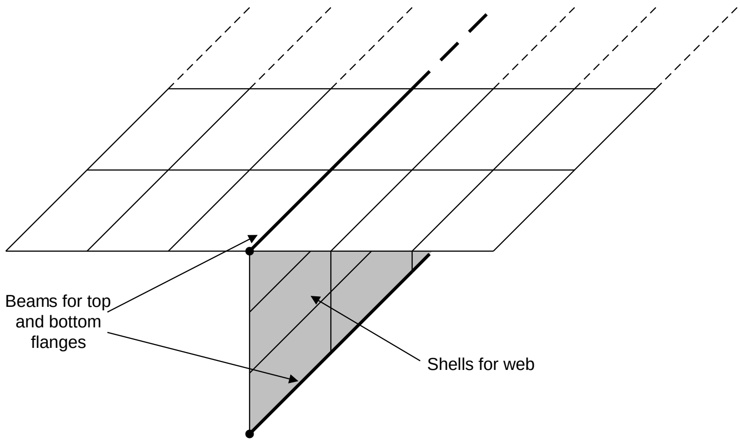
Based upon this idealisation method the whole model showing element discretisation is as shown:





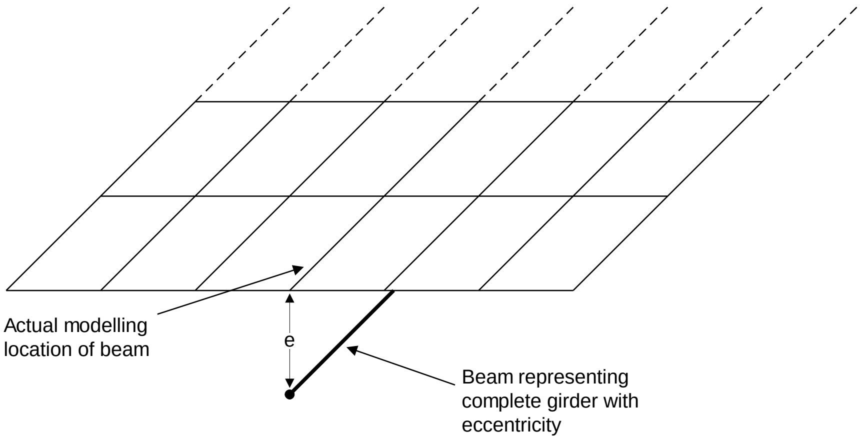
Alternative idealisations

The following figure shows the same structure but this time the shell elements representing the top and bottom flanges of the girder have been replaced by beam elements running along the top and bottom of the girder with the appropriate properties for the flange members.



Beam element modelling of flanges

In the last figure the whole girder has been replaced by beam elements with appropriate geometric properties and an eccentricity to offset the bending plane of the beams.



Beam element modelling of the whole girder

The choice of which analysis idealisation should be used depends upon what is required from the results of the analysis. For instance, if a curved girder analysis is being performed and the lateral forces in the top and bottom flanges are required then Shell element modelling of the whole structure or beam element modelling of the flanges (as shown in the first two figures) are suitable but beam element modelling of the whole girder (as shown in the last figure) is not.

Running the Analysis

With the model loaded:

 Open the **Solve Now** dialog, ensure **Analysis 1** is selected and press **OK**.

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

Viewing the Results

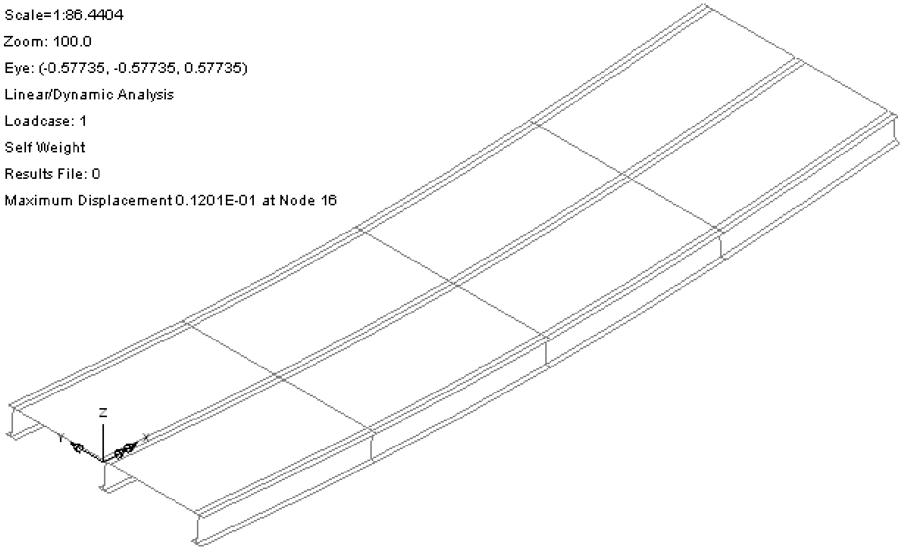
Analysis loadcase results are present in the  Treeview, and the loadcase results for the self-weight loadcase will be set to be active by default.

Plotting Deformed Shapes

- Ensure the **Mesh, Geometry** and **Attributes** layers are present and turned off in the Treeview.
- With no features selected click the right-hand mouse button in a blank part of the Graphics area and select the **Deformed mesh** option to add the deformed mesh layer to the Treeview.

Utilities
Annotation >
Window Summary

- Show the **Window Summary** for an overview of the results.

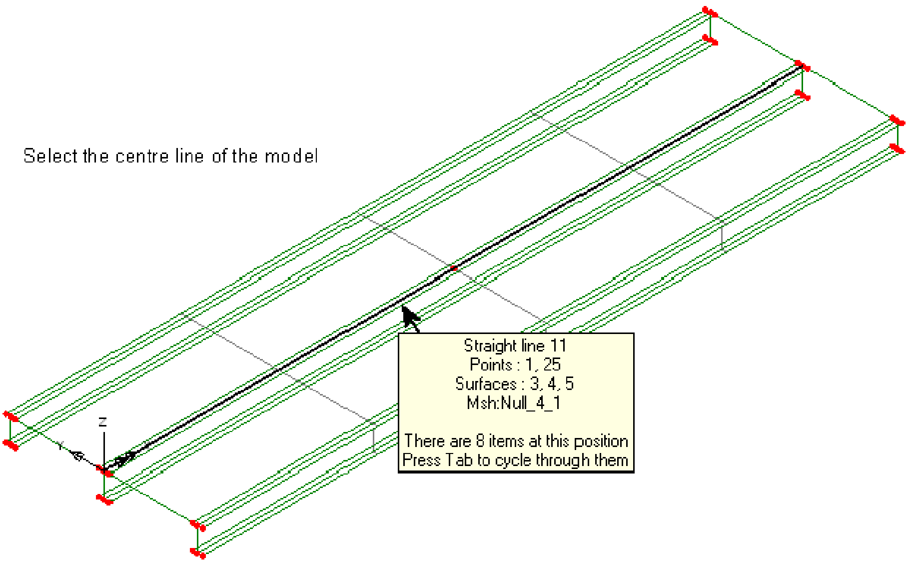


Computing Forces And Bending Moments For Combination

- Turn off the **Deformed Mesh** and **Annotation** layers in the Treeview.
- With no features selected click the right-hand mouse button in a blank part of the Graphics area and select the **Mesh** option to add the mesh layer to the Treeview. In the mesh dialog box click the **OK** button to accept the default values.
- With no features selected click the right-hand mouse button in a blank part of the Graphics area and select the **Geometry** option to add the geometry layer to the Treeview. In the geometry dialog box click the **OK** button to accept the default values.

- In the  Treeview expand **Analysis 1** then right-click on **Combination** and select the **Set Active** option. The results for the combined effects of the factored self weight and HB vehicle load will now be available.

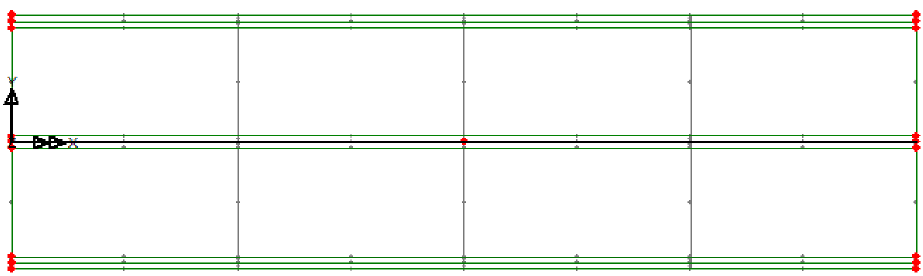
- Select the centre line of the model (Line 11). This line will be used to define the path along which the slicing will take place.



- 

Set the view axis to be along the global Z-axis by pressing the Z-axis button at the bottom of the graphics area.

The current Modeller window should look as follows:

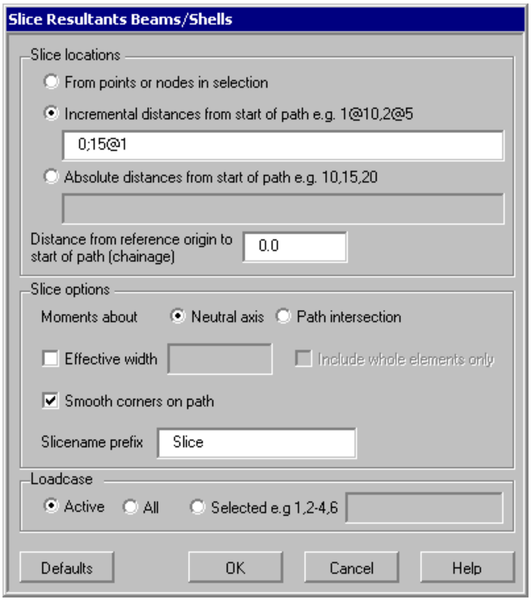


Note. Whilst visualisation of the model can be done using an isometric view the definition of the slice cutting planes must, for this example at least, always be done with the model viewed along the global Z-axis.

With the centre line of the model (Line 11) still selected:

Utilities
Slice Resultants Beams/Shells

The slicing dialog will be displayed.

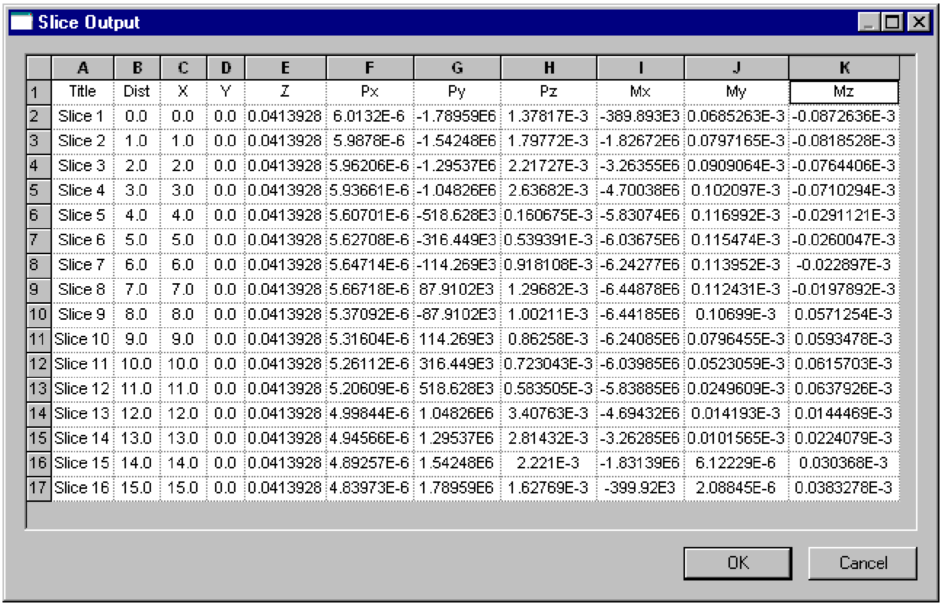


- Select the **Incremental distance from start of path** option.
- Enter **0;15@1** into the associated box which will cut slices at the start of the line (0m) and every 1m up to and including 15m.
- Set the **Distance from reference origin to start of path (chainage)** to **0**
- Select the **Neutral axis** option to calculate the forces and moments about the sliced section neutral axis.
- Ensure that the **Effective width** option is deselected.
- Set the slice **Slicename prefix** to **Slice** . This will be the prefix for the naming of the sliced sections.
- Click the **OK** button to start the automated slicing the model.

Note. Three methods are available for the definition of the slice locations. **From points or nodes in selection** allows the definition of the slicing locations by projection of the points/nodes onto the slicing path. **Incremental distances from start of path** allows the definition of progressive slice distances governed by the supplied increments. These increments can be positive or negative so long as the accumulated distance is within the path length. The **Absolute distances from start of path** allows the definition of known distances along the slicing path for slices to be taken.

Note. **Smooth corners on path** is used to traverse sharp corners in the defined slicing path. If smoothing is selected then the orientation of the slice plane is set to be at an angle of half the path angle change at the connection if this location coincides with a slice. If smoothing is not selected, two slices will be taken at a sharp corner with angles taken from both lines/arcs at the connection.

The forces and moments for the slices are output in a grid as indicated in the following figure. The output is relative to the slice axes and consists of the following information: Slice title, distance associated with slice location, the global coordinates of the sliced section neutral axis, shear forces (Px and Py), axial force (Pz), bending moments (Mx and My) and torsion (Mz). If moments are calculated about the path intersection then the coordinates will be equal to the location on the path associated with the distances selected.

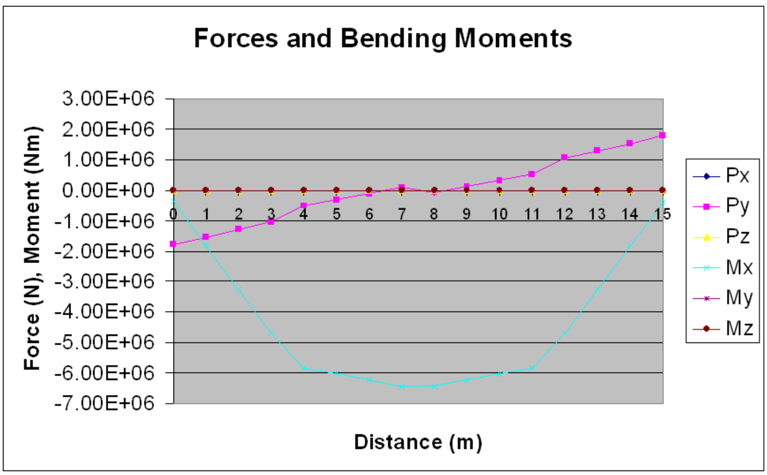


Slice Output

	A	B	C	D	E	F	G	H	I	J	K
1	Title	Dist	X	Y	Z	Px	Py	Pz	Mx	My	Mz
2	Slice 1	0.0	0.0	0.0	0.0413928	6.0132E-6	-1.78959E6	1.37817E-3	-389.893E3	0.0685263E-3	-0.0872636E-3
3	Slice 2	1.0	1.0	0.0	0.0413928	5.9878E-6	-1.54248E6	1.79772E-3	-1.82672E6	0.0797165E-3	-0.0818528E-3
4	Slice 3	2.0	2.0	0.0	0.0413928	5.96206E-6	-1.29537E6	2.21727E-3	-3.26355E6	0.0909064E-3	-0.0764406E-3
5	Slice 4	3.0	3.0	0.0	0.0413928	5.93661E-6	-1.04826E6	2.63682E-3	-4.70038E6	0.102097E-3	-0.0710294E-3
6	Slice 5	4.0	4.0	0.0	0.0413928	5.60701E-6	-518.628E3	0.160675E-3	-5.83074E6	0.116992E-3	-0.0291121E-3
7	Slice 6	5.0	5.0	0.0	0.0413928	5.62708E-6	-316.449E3	0.539391E-3	-6.03675E6	0.115474E-3	-0.0260047E-3
8	Slice 7	6.0	6.0	0.0	0.0413928	5.64714E-6	-114.269E3	0.918108E-3	-6.24277E6	0.113952E-3	-0.022897E-3
9	Slice 8	7.0	7.0	0.0	0.0413928	5.66718E-6	87.9102E3	1.29682E-3	-6.44878E6	0.112431E-3	-0.0197892E-3
10	Slice 9	8.0	8.0	0.0	0.0413928	5.37092E-6	-87.9102E3	1.00211E-3	-6.44185E6	0.10699E-3	0.0571254E-3
11	Slice 10	9.0	9.0	0.0	0.0413928	5.31604E-6	114.269E3	0.86258E-3	-6.24085E6	0.0796455E-3	0.0593478E-3
12	Slice 11	10.0	10.0	0.0	0.0413928	5.26112E-6	316.449E3	0.723043E-3	-6.03985E6	0.0523059E-3	0.0615703E-3
13	Slice 12	11.0	11.0	0.0	0.0413928	5.20609E-6	518.628E3	0.583505E-3	-5.83885E6	0.0249609E-3	0.0637926E-3
14	Slice 13	12.0	12.0	0.0	0.0413928	4.99844E-6	1.04826E6	3.40763E-3	-4.69432E6	0.014193E-3	0.0144469E-3
15	Slice 14	13.0	13.0	0.0	0.0413928	4.94566E-6	1.29537E6	2.81432E-3	-3.26285E6	0.0101565E-3	0.0224079E-3
16	Slice 15	14.0	14.0	0.0	0.0413928	4.89257E-6	1.54248E6	2.221E-3	-1.83139E6	6.12229E-6	0.030368E-3
17	Slice 16	15.0	15.0	0.0	0.0413928	4.83973E-6	1.78959E6	1.62769E-3	-399.92E3	2.08845E-6	0.0383278E-3

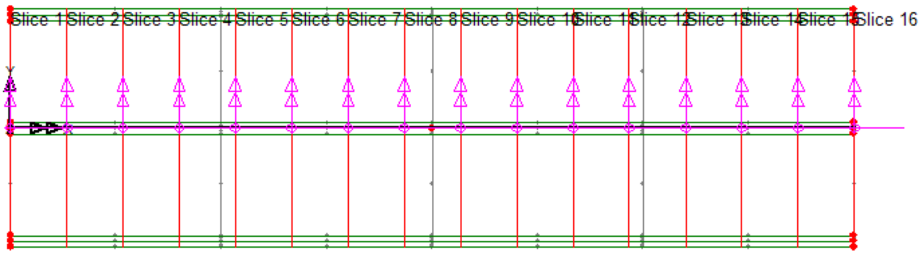
OK Cancel

This grid can be resized and the results copied and pasted using the **Ctrl + C** and **Ctrl + V** keys into an application such as Microsoft Excel to generate force and bending moment diagrams as illustrated.

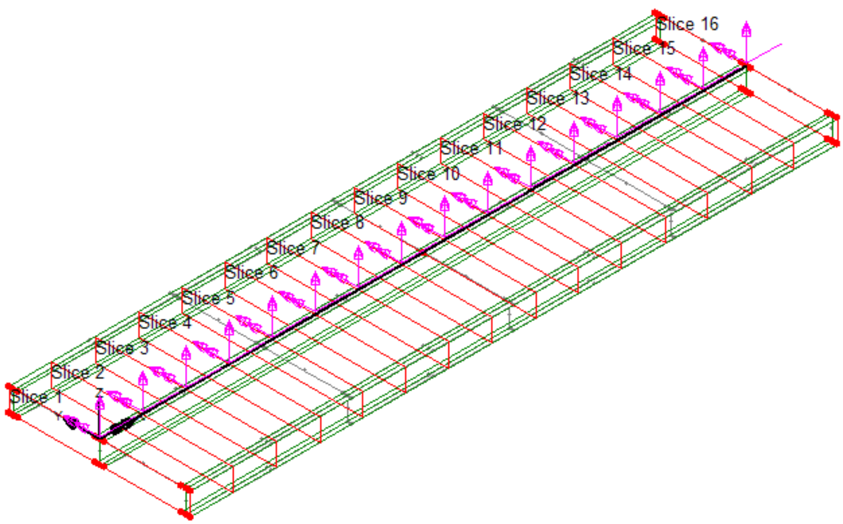


- Click the **Cancel** button to close the slice output grid.
- Turn on the Annotation layer in the Treeview.

The locations of the slices are indicated on the model as illustrated below. These annotations include the physical location of the slice, its title and a local coordinate set indicating the axes of the slice. If the option to calculate the forces and moments about the neutral axis has been selected this local coordinate set will be defined at the neutral axis location. If however the option to calculate the forces and moments about the slice path has been selected, the local coordinate set will be defined at the intersection between the slice path and the slice plane.



- Click on the isometric button to view the model in 3D to view the extents of the slices.



Note. Whilst visualisation of the slices generated is possible in an isometric view the definition of the slice cutting plane must, for this example at least, always be done with the model viewed along the global Z-axis.

Utilities
Remove Beam/Shell Slice Annotations

- Delete the current slice annotations.

Computing Forces and Bending Moments for Girder Effective Width

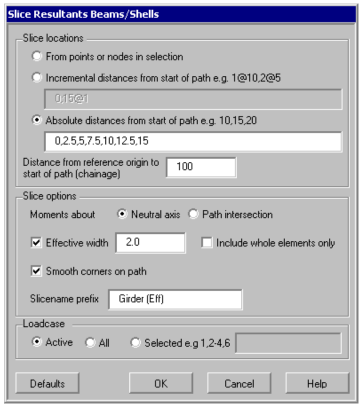
- 

Set the view axis to be along the global Z-axis

With the centre line of the model still selected:

Utilities
Slice Resultants Beams/Shells

The slicing dialog will be displayed.



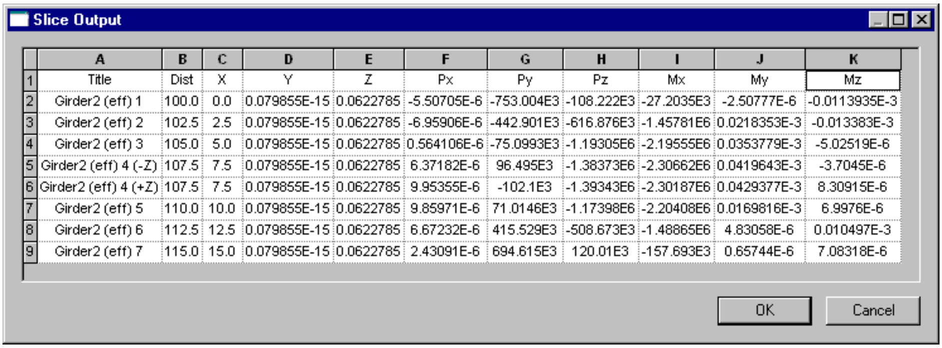
- Select the **Absolute distances from start of path** option.

Note. List separators vary according to your PC's regional locale settings. For example, English (UK) and English (US) settings use a comma as a list separator. French, Swedish and other language settings use a semi-colon.

Please enter whatever is appropriate to your region. In this example an English (UK) separator is shown.

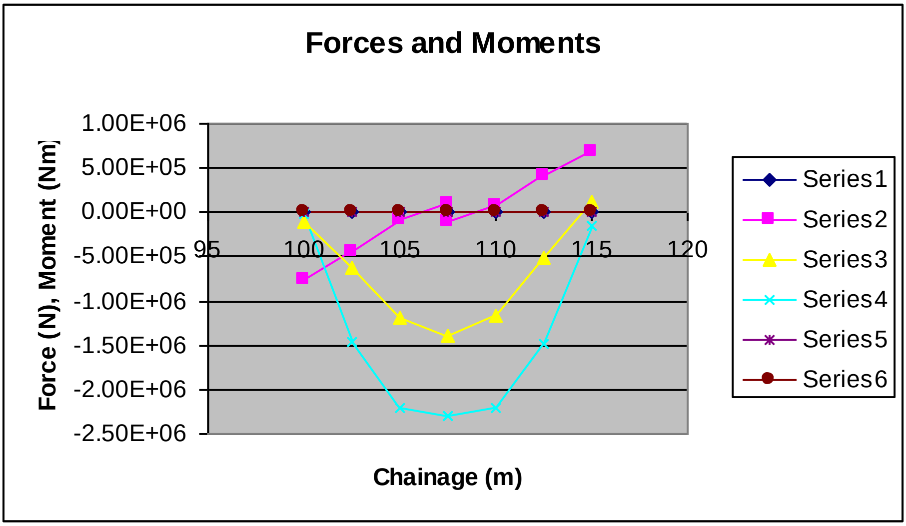
- Enter **0,2.5,5,7.5,10,12.5,15** (using an appropriate list separator for your regional settings) into the associated box. This will cut slices at the start of the line (0m) and every 2.5m up to and including 15m.
- Set the **Distance from reference origin to start of path (chainage)** to **100** . The start of the slicing path will therefore have a distance/chainage of 100m and the end of the slicing path will have a distance/chainage of 115m.
- Select the **Effective width** option and enter **2.0** in the associated box. This will set the effective width to be 2.0m centred on the slicing path (i.e. 1.0m either side)
- Set the slice **Slicename prefix** to **Girder2 (Eff)** . This will be the prefix for the naming of the sliced sections.
- Click the **OK** button to start slicing the model.

The forces and moments for the slices are output in a grid as shown:


Slice Output

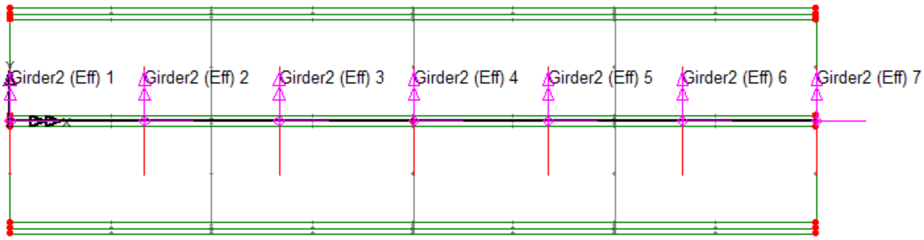
	A	B	C	D	E	F	G	H	I	J	K
1	Title	Dist	X	Y	Z	Px	Py	Pz	Mx	My	Mz
2	Girder2 (eff) 1	100.0	0.0	0.079855E-15	0.0622785	-5.50705E-6	-753.004E3	-108.222E3	-27.2035E3	-2.50777E-6	-0.0113935E-3
3	Girder2 (eff) 2	102.5	2.5	0.079855E-15	0.0622785	-6.95906E-6	-442.901E3	-616.876E3	-1.45781E6	0.0218353E-3	-0.013383E-3
4	Girder2 (eff) 3	105.0	5.0	0.079855E-15	0.0622785	0.564106E-6	-75.0993E3	-1.19305E6	-2.19555E6	0.0353779E-3	-5.02519E-6
5	Girder2 (eff) 4 (-Z)	107.5	7.5	0.079855E-15	0.0622785	6.37182E-6	96.495E3	-1.38373E6	-2.30662E6	0.0419643E-3	-3.7045E-6
6	Girder2 (eff) 4 (+Z)	107.5	7.5	0.079855E-15	0.0622785	9.95355E-6	-102.1E3	-1.39343E6	-2.30187E6	0.0429377E-3	8.30915E-6
7	Girder2 (eff) 5	110.0	10.0	0.079855E-15	0.0622785	9.85971E-6	71.0146E3	-1.17398E6	-2.20408E6	0.0169816E-3	6.9976E-6
8	Girder2 (eff) 6	112.5	12.5	0.079855E-15	0.0622785	6.67232E-6	415.529E3	-508.673E3	-1.48865E6	4.83058E-6	0.010497E-3
9	Girder2 (eff) 7	115.0	15.0	0.079855E-15	0.0622785	2.43091E-6	694.615E3	120.01E3	-157.693E3	0.65744E-6	7.08318E-6

Copying these results into a spreadsheet application and generating the graphs gives the force and bending moment diagrams as shown:

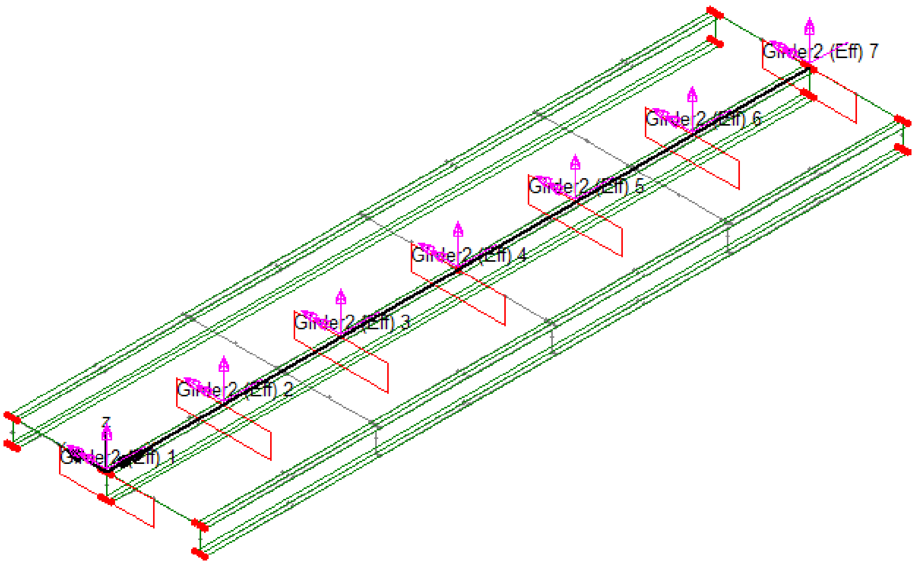


- Click the **Cancel** button to close the slice output grid.

The locations and extents of the slices through the bridge are indicated on the model as illustrated below. These annotations include the physical location, title and local axes as before.



- Click on the  isometric button to view the model in 3D to view the extents of the slices.



Utilities
Remove Beam/Shell Slice Annotations

- Delete the current slice annotations.

This completes the example.

Notes

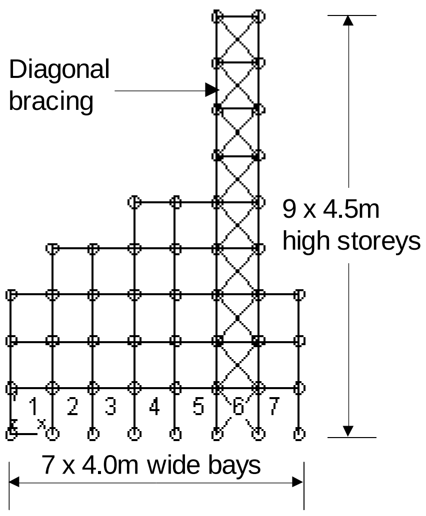
- Curved structures can also be sliced using this facility.

Seismic Response of a 2D Frame (Frequency Domain)

For software product(s):	All (except LT versions)
With product option(s):	None.

Description

This example examines the Spectral Response analysis of a 2D braced tower frame. The geometry is simplified to a wire-frame or stick representation, with each of the structural members being represented by Point and Line features only.



The model is comprised of thick beam elements for the concrete column, beam members, and steel diagonal bracing members which have pinned end connections. The structure is fully restrained against displacement and rotation at ground level.

Since the global response of the structure is required, the model of the tower is further simplified by meshing each Line with a single element. This will effectively avoid the extraction of local panel modes for individual beams and columns. These local modes could be investigated independently in a more detailed analysis.

The spectral response analysis is performed in three distinct stages. Firstly a natural frequency analysis is performed. This is used here to calculate the first 10 natural modes of vibration of the structure. The eigenvalues, frequencies and eigenvectors (mode shapes) are stored and used in the subsequent spectral response analysis. Although the natural frequencies are obtained from an eigenvalue analysis any information regarding the magnitudes of deformations or moments is non-quantitative.

The second phase of the analysis is the spectral response calculation, in which the spectral response computation is performed interactively as a results processing operation using the Interactive Modal Dynamics (IMD) facility. This is an alternative to performing a non-interactive spectral response analysis in LUSAS.

In a spectral response analysis, the structure is subjected to a support condition excitation. In this example this is assumed to be the effect of Seismic/Earthquake motion although any support motion could be envisaged. The excitation is specified as a spectral curve, in terms of displacement, velocity or acceleration and represents an envelope of these effects over a measured frequency range. Damping may also be specified at this stage.

From the spectral analysis participation factors are obtained which indicate, for each mode, the degree of structural response associated with this applied excitation.

The last phase of the analysis considers self weight in conjunction with the spectral response and to this end two load combinations are defined from which worst case force/moment envelopes may be obtained.

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

Keywords

2D, Plane Frame, Beam, End Releases, Fleshing, Natural Frequency, Eigenvalue, Spectral Response, Seismic, Axial Force, Shear Force, Bending Moment Diagrams, Participation Factors, Modal Dynamics, Interactive Modal Dynamics (IMD), Modal Response, Global Response, Graphing, Animation, Load Combination.

Associated Files

- **tower_modelling.vbs** carries out the example modelling.
- **tower_spectrum.vbs** defines a typical spectral curve excitation.

Modelling

Running LUSAS Modeller

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

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 the file name as **tower**
- Use the **Default** working folder.
- Enter the title as **Tower**
- Set model units of **N,m,kg,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the **Standard** startup template.
- Select the **Vertical Y Axis** option.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

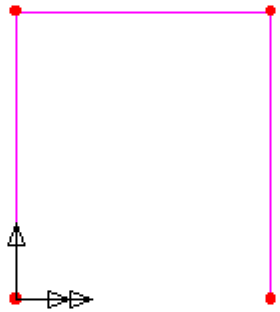
Defining the Column and Beam Members

Geometry >
Line >
Coordinates…

Enter coordinates of (**0, 0**), (**0, 4.5**), (**4, 4.5**) and (**4, 0**) and click **OK** to define the first bay of the ground level storey.

Note. Sets of coordinates must be separated by commas or spaces unless the 'Grid Style' method is chosen. The **Tab** key is used to create new entry fields. The arrow keys are used to move between entries.

Meshing - Columns and Beams



As the global response of the structure is required only one beam element need be used along each Line. This will prevent LUSAS from picking up any panel modes caused by local vibration of the columns and beams.

Attributes > Mesh > Line…

- Select **Thick Beam, 2 dimensional, Linear** elements.
- Enter the number of divisions as **1**
- Enter the attribute name as **Thick Beam** and click **OK**
- With the whole model selected, drag and drop the Line mesh attribute **Thick Beam** from the  Treeview onto the selected features.

Geometric Properties - Columns and Beams

Attributes > Geometric > Line…

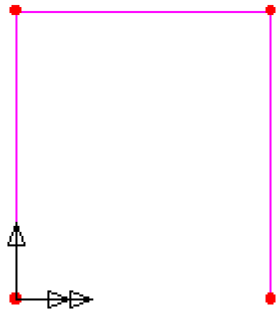
- From the Usage section select **2D Thick Beam** from the drop down list and change the values on the dialog to match those shown in the table for a column member.
- Enter the attribute name as **Concrete Column** and select the **Apply** button to create the attribute and re-use the dialog.
- Change the values on the dialog to match those shown in the table for a beam member.

	Column Member	**Beam Member**
Cross-sectional area	0.1	0.08
Second moment of area about z axis Izz)	0.833e-3	0.533e-3
Effective Shear Area in Y Direction (Asy)	0.1	0.08
Offset in y direction (Ry)	0	0

- Change the attribute name to **Concrete Beam** and click **OK**

Note. Whilst in practice column sizes would vary with height and beam sizes may vary throughout the structure, in this example, for simplicity, all of the concrete columns and beam sections are of the same size and assumed to be of solid square section.

- Select the 2 vertical Lines representing the column members. (Hold the **Shift** key down to add to the initial selection).
- Drag and drop the geometry attribute **Concrete Column** from the Treeview onto the selected features.
- Select the horizontal Line representing the Beam.
- Drag and drop the geometry attribute **Concrete Beam** from the Treeview onto the selected Line.



Geometric property assignments are visualised by default. In general the diagrams in this example will not show geometric visualisations.

Material Properties - Columns and Beams

Attributes
Material >
Material Library...

- Select material **Concrete** from the drop down list, leave the grade **Ungraded** and click **OK** to add the material attribute to the Treeview.

The columns and beams are made of concrete.

- Select the whole model by either using the **Ctrl** and **A** keys to select all features or by dragging a box around the model.
- Drag and drop the material attribute **Iso1 (Concrete Ungraded)** from the Treeview onto the selected features.

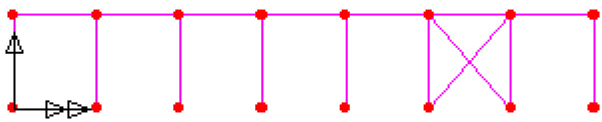
Copying the Column and Beam Members

Now that one bay has been fully defined it will be copied to form the other 6 bays of the ground storey of the tower. With the whole model selected:

Geometry
Line >
Copy…

Enter a translation in the **X** direction of **4** metres.

- Enter the number of copies required as **6**
- Click the **OK** button to create 6 additional bays.



Bay 6 of the structure requires diagonal bracing to be added with the ends of both members being free to rotate about the Z direction in order to model a pinned connection.

Defining the Bracing Members

- Select 2 diagonally opposite Points in bay 6. (Hold the **Shift** key down to add to the selection).

Geometry
Line >
Points…

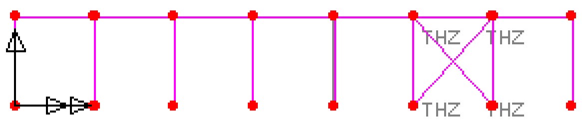
The line will be drawn.

- Repeat for the other pair of diagonally opposite Points.

Meshing - Bracing Members

Only one beam element is to be used along each bracing member. However, the ends of the bracing members must be free to rotate about the Z direction in order to model a pinned connection, therefore the existing mesh dataset (as used for the beam and column members) cannot be used. A new mesh dataset must now be created from the existing one.

- In the Treeview double-click the **Thick Beam** attribute name.
- Click the **End releases** button and set the **First node end releases** and **Last node end releases** to allow **Rotation about Z** by selecting the option.
- Click the **OK** button to return to the Line mesh dialog.
- Change the dataset name to **Thick Beam (Pinned Ends)** and click the **OK** button.
- With the 2 Lines representing the bracing members selected. Drag and drop the Line mesh attribute **Thick Beam (Pinned Ends)** from the Treeview onto the selected features.



Note. The visualisation of the beam end releases is shown by default. If it is required to turn-off their visualisation select the Treeview and double click on **Mesh**, select the Visualise tab and deselect the **Beam end releases** option.

Geometric Properties - Bracing members

For the purposes of this example the bracing members are to be defined as Universal Beams.

Attributes
Geometric >
Section Library…

The standard sections dialog will appear.

- In the Usage section ensure that **2D Thick Beam** is selected

- From the **UK Sections** library select the **Universal Beams (BS4)** section type and select the **254x146x43kg UB** section.
- Enter an attribute name of **Bracing Member** and click the **OK** button to add the Universal Beam attribute to the Treeview.
- Select the 2 diagonal Lines representing the bracing members.
- Drag and drop the geometry attribute **Bracing member (254x146x43kg UB major y)** from the Treeview onto the selected features.

Select the fleshing on/off button to turn-off the geometric visualisation. If at any time during the example you wish to visualise the geometry select this button.

Material Properties - Bracing Members

Attributes
Material >
Material Library...

- Select material **Mild Steel** from the drop down list, leave the grade **Ungraded** and click **OK** to add the material attribute to the Treeview.
- With the 2 bracing members selected, drag and drop the Material attribute **Iso2 (Mild Steel Ungraded)** from the Treeview onto the selected features.

The first bay of the ground storey of the tower is now completely defined. The support conditions are not defined or assigned yet because this bay will be copied upwards to create other storeys where supports will not be required.

Copying the Model Features

The other storeys of the tower can now be generated by copying selected features through specified distances in the Y direction.

- Select the whole model using the **Ctrl** and **A** keys together.

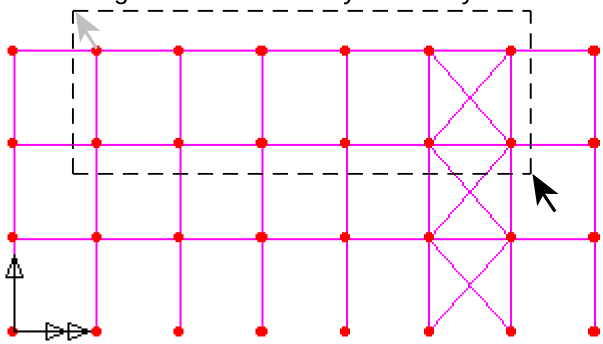
Geometry
Line >
Copy…

Enter a translation in the **Y** direction of **4.5** metres. Enter the attribute name as **Y=4.5**. Click the **Save** button to save the attribute for re-use. Enter the number of copies required as **2** and click the **OK** button to finish to generate two additional storeys.

Note. All attributes assigned to the features on the ground level storey, for example, line mesh, geometric and material properties, will be duplicated on the other storeys automatically when the new features are created by copying.

- Drag a box around the bays shown to create the next level of the structure.

Drag a box to select only these bays

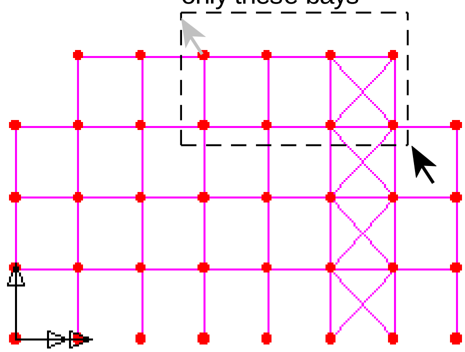


Geometry
Line >
Copy…

Select the attribute **Y=4.5** from the drop down list and click the **OK** button to create **1** copy of the features selected.

- Drag a box around the bays shown to create the next level of the structure.

Drag a box to select only these bays

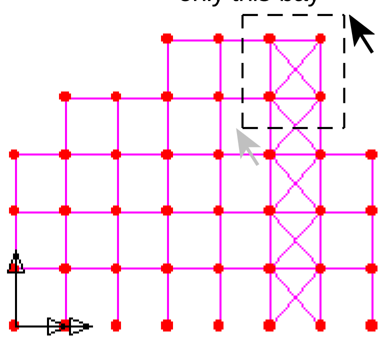


Geometry
Line >
Copy…

Select the attribute **Y=4.5** from the drop down list and click the **OK** button to create **1** copy of the features selected.

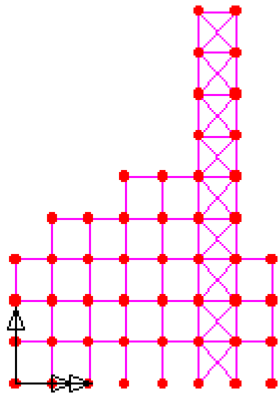
- Drag a box around the bay shown which will be used to create the remaining levels of the structure.

Drag a box to select only this bay



Geometry
Line >
Copy…

Select the attribute **Y=4.5**

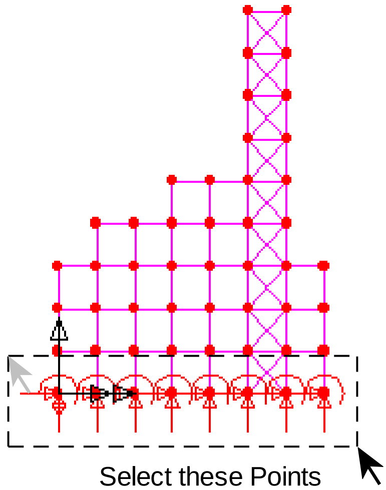


- Change the number of copies to **4**
- Click the **OK** button to create the remaining levels.

Supports

LUSAS provides the more common types of support by default. These can be seen in the Treeview. The tower must be fully restrained at the base, therefore 3 degrees of freedom (U, V and rotation about the Z axis) must be restrained. A fully fixed support type is therefore required.

Assigning the Supports



- Drag a box around the 8 Points along the base of the structure.
- From the Treeview drag and drop the support attribute **Fully Fixed** onto the selected features, ensure the **All analysis loadcases** option is selected and click **OK**

The model geometry is now complete.

Loading

Note. No static structural loading is required for the first part of this analysis because the spectral combination is carried out during results processing using the results from the natural frequency analysis which is independent of applied loading. The modelling will now be completed by defining the controls necessary to extract the natural frequencies.

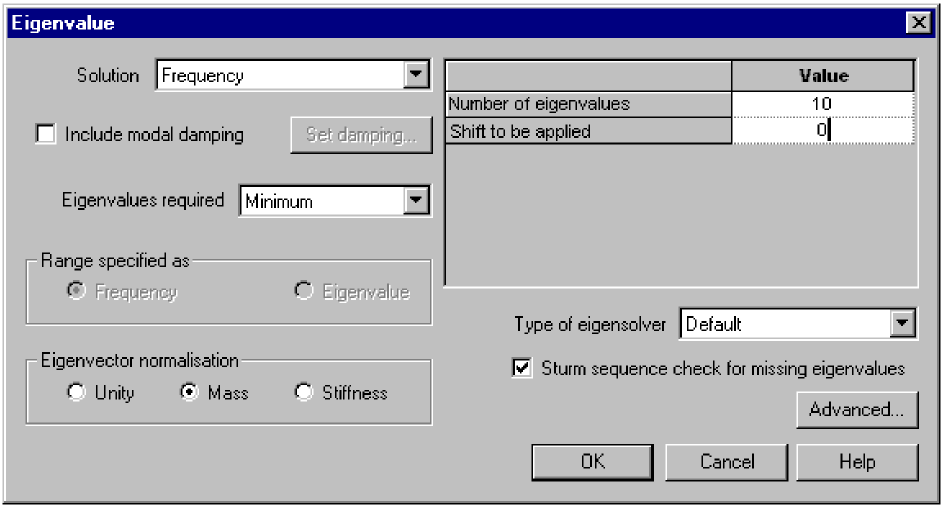
Eigenvalue Analysis Control

Since the first stage of a spectral response analysis involves a computation of the natural modes of vibration an initial eigenvalue extraction analysis must be carried out. The solution parameters for eigenvalue analysis are specified using an eigenvalue control dataset. In this example the first 10 natural modes of vibration of the tower are computed.

Eigenvalue analysis control properties are defined as properties of the loadcase.

- In the  Treeview, expand **Analysis 1** then right-click on **Loadcase 1** and select **Eigenvalue** from the **Controls** menu.

The Eigenvalue dialog will appear.



The following parameters need to be specified:

- Set the **Number of eigenvalues** as **10**
- Ensure the **Shift to be applied** is set as **0**
- Ensure the **Type of eigensolver** is set as **Default**

Note. Eigenvalue normalisation is set to **Mass** by default. This is essential if the eigenvectors are to be used for subsequent IMD analysis when results processing.

- Click the **OK** button to finish.

Saving the model

File
Save

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.

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 Treeview

In addition, 2 files will be created in the new Associated Model Data directory where the model file resides:

- **tower.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **tower.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **tower_modelling.vbs** carries out the modelling of the example.

File
New…

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 **tower**

File
Script >
Run Script...

- To recreate the model, select the file **tower_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

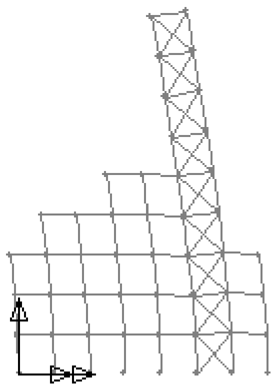
Rerun the analysis to generate the results.

Viewing the Results

Eigenvalue loadcase results are present in the Treeview, with Eigenvalue 1 set to be active by default.

Plotting Mode Shapes

- Turn off the display of the **Mesh, Geometry** and **Attributes** layers in the Treeview.
- By default the **Deformed mesh** layer is visible. To see other mode shapes:
- In the Treeview right-click on **Eigenvalue 2** and select the **Set Active** option. The deformed mesh plot for Eigenvalue 2 will be displayed.
- By setting each Eigenvalue to be active the deformed mesh can be seen for all mode shapes.



Note. The mode shape for eigenvalue 1 may be inverted. This is because the sense is arbitrary since during vibration deformed shape will appear in both directions.

Printing Eigenvalue Results

Eigenvalue results for the whole structure can be displayed in the Text Output window.

Utilities
Print Results Wizard...

Ensue **Active** is selected and press **Next**. Then select **None** from the Entity drop down list. Ensure **Eigenvalues** is displayed in the Type drop down list and click **Finish.** The Eigenvalue results will be printed to the Text Output window. Error norms may vary.

```
Results File  = L:\models\tower.mys ID        0
     MODE     EIGENVALUE      FREQUENCY       ERROR NORM
        1     147.296         1.93160        0.945062E-12
        2     1527.44         6.22017        0.252379E-12
        3     7481.98         13.7667        0.992755E-13
        4     12463.9         17.7683        0.261939E-12
        5     16464.9         20.4221        0.968822E-13
        6     26008.4         25.6671        0.346929E-12
        7     31861.1         28.4087        0.455993E-10
        8     33854.1         29.2837        0.157390E-07
        9     34920.0         29.7411        0.110320E-06
       10     35963.4         30.1822        0.499202E-06
```

Note. The frequency in Hertz can be obtained by dividing the square root of the eigenvalue by 2π. The period of vibration in seconds is obtained using the reciprocal of frequency (1/frequency). Values of error norm may vary from those shown.

Caution. The system eigenvectors have been normalised (in this case with respect to mass) therefore any derived quantities such as displacement and moment are also normalised and are not true design values.

- Close the text window by selecting the close button in the top right hand corner of the window.
- Use the maximise button to increase the size of the graphics window.

Spectral Response Analysis

Spectral response calculations are performed using the IMD (Interactive Modal Dynamics) facility. Setting up the spectral response results involves the following steps:

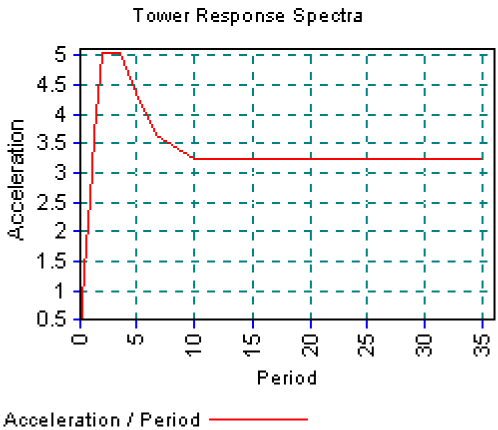
- ❑ Define the Spectral Excitation curve.
- ❑ Define the dynamic excitation type (support motion acceleration in this example), direction, and specify the type of results required using a CQC (Combined Quadratic Combination) for spectral response.
- ❑ Set the IMD loadcase to be active.

The spectral combination results can be displayed using a deformed shape plot or force and moment diagrams.

Note. Additional damping information may also be set. Unlike results from a natural frequency analysis, the output values obtained from a spectral combination are design values.

Stage 1 - Defining a Spectral Excitation Curve

Values for the spectral excitation curve need to be specified. For this analysis Acceleration versus Period values are to be used with a damping value of 5%.



Note. The Utilities > Response Spectra command would normally be used to define response spectra data. However, to simplify entering the required data, the full response spectrum definition for this example is contained in a command file.

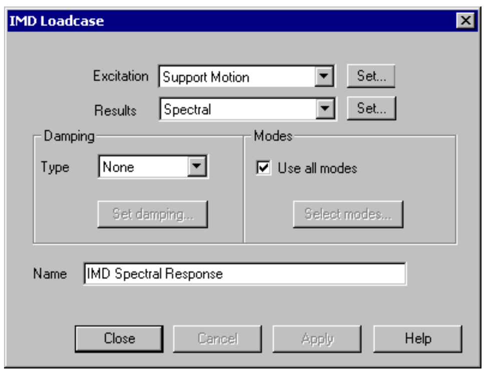
File
Script
Run Script…

- In the \<**LusasInstallationFolder>\Examples\Modeller** directory select the file **tower_spectrum.vbs** and click **OK**
- A response spectra dataset **TOWER** and corresponding graph datasets containing the spectral data for period and acceleration will be created in the Treeview.

Stage 2 - Defining the Dynamic Excitation

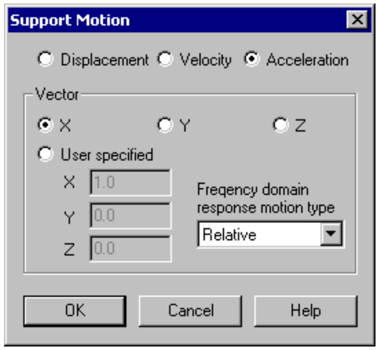
Analyses
IMD Loadcase…

- Select **Support Motion** from the Excitation drop down list and **Spectral** from the Results drop down list. Ensure the **Use all modes** option is selected.
- Change the IMD loadcase name to be **IMD Spectral Response**

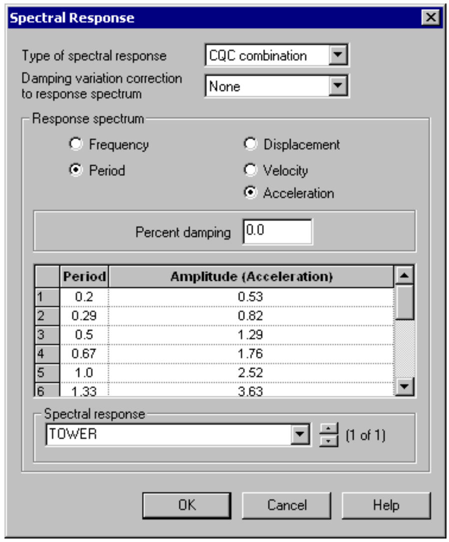


- Click the Excitation **Set** button.

- On the Support Motion dialog select **Acceleration** support motion option in the **X** direction which is **Relative**
- Click the **OK** button to return to the IMD loadcase dialog.



- On the IMD loadcase dialog click the Results **Set** button.
- On the Spectral Response dialog set the type of spectral response to **CQC Combination** with the damping variation correction set as **None**. The response spectrum **TOWER** read in from the command file will already be selected in the drop down list.
- Click the **OK** button to return to the IMD loadcase dialog.
- Click the **OK** button to finish.



Selecting the IMD Results Loadcase

- In the Treeview right-click on the **IMD Spectral Response** loadcase name and select the **Set Active** option.

The spectral combination results are now activated. Any subsequent results plots, such as deformed shapes or moments, will be for the spectral combination results only.

Using Page Layout Mode

The model was created using a Working Mode view which allows a model of any size to be created. Whilst previous results have been viewed using this mode of operation, in order to allow additional information to be added without obscuring the model, Page Layout Mode will be used instead.

View
Page Layout Mode

The graphics window will resize to show the mesh layer on an A4 size piece of paper.

File
Page Setup…

Set the page layout to **Portrait** and click **OK**

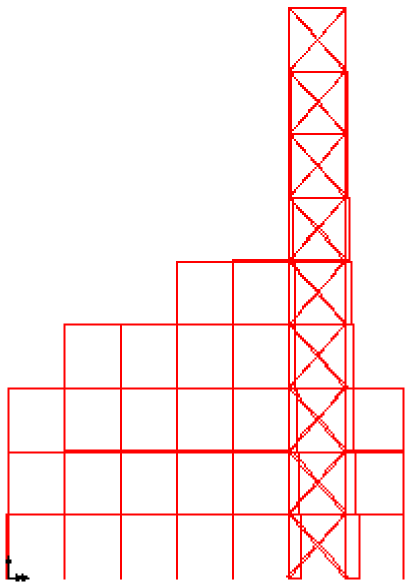
Displaying Results for Spectral Combination

A plot showing the tower deformed shape, bending moment and axial force for the spectral combination is to be displayed.

- Delete the **Deformed Mesh** layer from the  Treeview.
- Add the **Mesh** layer to the  Treeview.

Axial Force Diagram

- With no features selected, click the right-hand mouse button in a blank part of the Graphics area and select the **Diagrams** option to add the Diagrams layer to the  Treeview.
- Select **Force/Moment - Thick 2D Beam** results of axial force **Fx** in the members.
- Select the **Diagram Display** tab, select the **Label values** option and choose the **Label only if selected** option
- Click the **OK** button to finish.



Results will be calculated for the active IMD spectral response loadcase and an axial force diagram of stresses in each member will be displayed.

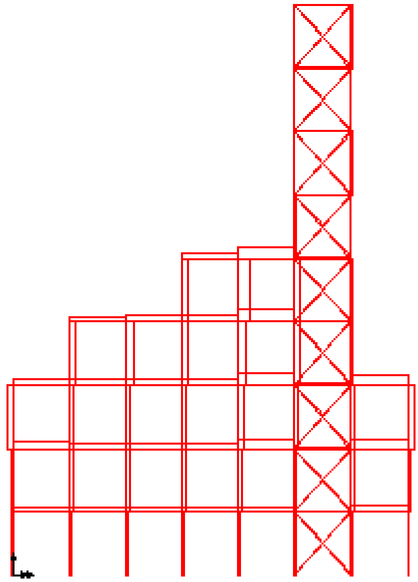
Displaying results on selected members

- Hovering over a member with the cursor will display previously chosen results information for gauss points along that member.
- Selecting a member will display results for that member. Remember more than one member can be selected by holding down the **Shift** key.

If necessary, use the Zoom in button to view results information.

These type of element results can also be plotted in isolation on selected model features. This is covered later in the example.

Shear Force Diagram

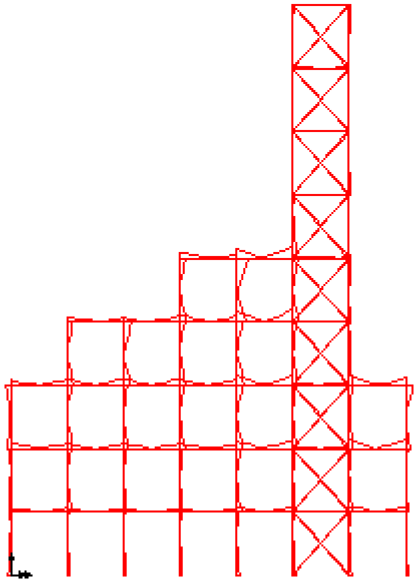


- In the Treeview double-click on the **Diagrams** layer. The diagram properties will be displayed.
- Select **Force/Moment - Thick 2D Beam** results of shear force **Fy** in the members.
- Click the **OK** button to finish.
- A shear force diagram of stresses in each member will be displayed.

Bending Moment Diagram

- In the Treeview double-click on the **Diagrams** layer. The diagram properties will be displayed.
- Select **Force/Moment - Thick 2D Beam** results of bending moment **Mz** in the members.
- Click the **OK** button to finish.

A bending moment diagram of stresses in each member will be displayed.



Printing Results

Note. Text and border annotation may be added to the display from the Utilities> Annotation menu.

Utilities
Annotation >
Window border

To add a border to the display.

File
Print Preview...

Now view how the results would print.

This shows how the bending moment results for the whole model will appear on printed output.

- Click the **Close** button to exit the print preview window.

Utilities
Annotation >
Window border

To remove the border from the display.

View
Working Mode

Plotting Results for a Selected Part of the Model

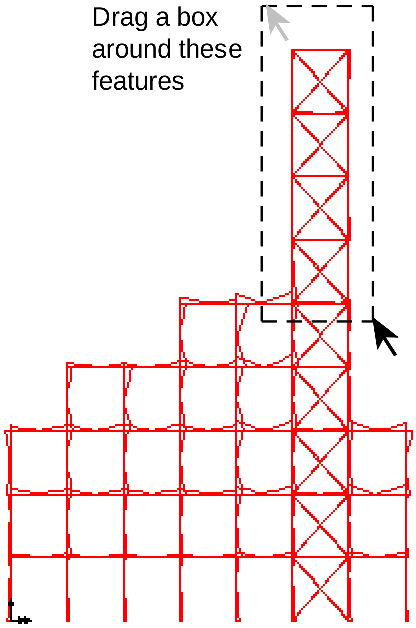
The graphics window will resize to show the model in normal working mode.

Note. Changing from Page Layout to Working Mode will resize and reposition any displayed annotation. Page layout mode should be used when preparing a view for printing. Working Mode should be used when viewing on the screen.

If results are to be plotted for a selected part of the structure only, a Group may be created of the area of interest. For example, to plot results of just the top 4 storeys of the tower:

- In the Treeview double-click the **Mesh** layer, select **Visualise** tab and de-select the option to visualise **Beam end releases**. Click the **OK** button.

- Drag a box around the upper 4 storeys of the tower as shown.

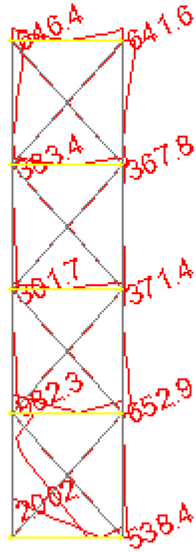


Geometry
Group >
New Group

Enter **Upper 4 storeys** for the group name. Click the **OK** button to complete creation of the group.

- Click the right-hand mouse button on the **Upper 4 storeys** group in the Treeview, and select the **Set as Only Visible** option to turn on the display of this group only.

The results for just the upper 4 storeys will be displayed.



- To add values to selected members double click on the **Diagrams** layer in the Treeview and select the **Diagram Display** tab.
- Ensure that the options to **Label values** and **Label only if selected** are chosen.
- Set the number of significant figures to **4**
- Choose the **Label Font** button and change the size to **11** and click **OK**
- Change the Angle to **25**
- Select the **Scale** tab and set the magnitude to **2**
- Click **OK** and the diagram will be displayed without any values because no elements have been selected.
- Select the horizontal elements and the labels of My (as previously selected) will appear.

Note. Holding down the **Shift** key after the first selection will enable labels on more than one element to be visualised.

Re-display the Whole Model

To re-display the whole model in working mode:

- Select the Treeview and click the right-hand mouse button on **tower.mdl**, and select the **Visible** option to turn on the display of all groups. Select **Yes** to act on subgroups as well.

Frequency Versus Participation Factor Graphs

A graph of Participation factors (for the X and Y directions) versus Frequency is to be plotted. For the current spectral combination, where the applied ground motion is in the X direction, the graph will show to what extent each structural mode is excited in the X and Y directions. A large modal participation factor indicates a significant response in that direction.

Using the Graph Wizard

The graph wizard provides a step-by-step means of selecting which results are to be plotted on the X and Y axes of the graph. The X axis is always defined first.

Utilities
Graph Wizard...

- Ensure the **Time history** option is selected and click the **Next** button.
- Select the **Named** option and click the **Next** button.
- Select **Natural Frequency** from the drop down list and click the **Next** button.

The Y axis results to be graphed are now defined.

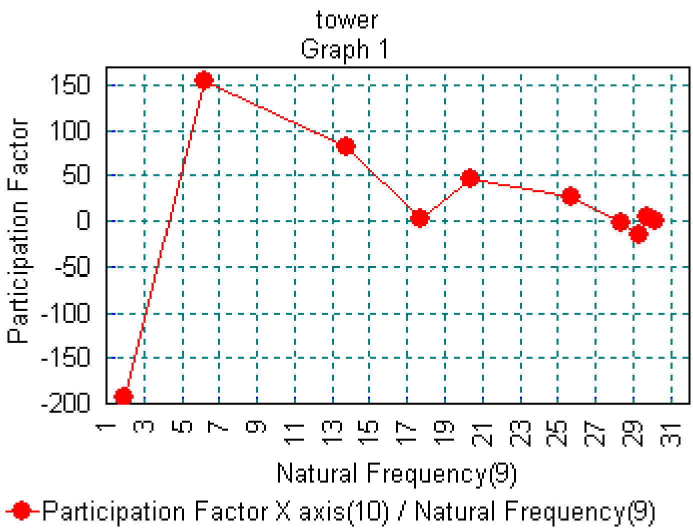
- Select **Named** results and click the **Next** button.
- Select **Participation Factor X** and click the **Next** button.

Additional information for the graph may now to be added.

- Enter the Y axis title as **Participation Factor X axis**
- Ensure that the **Show symbols** button is selected.
- Click the **Finish** button to display the graph.

Note. If no graph or axis titles are entered default names will be used.

Note. The sign of the participation factor displayed on the graph is arbitrary as the mode shape from which it is derived can have a positive or negative amplitude.



LUSAS will create the graph in a new window and display the values used in an adjacent table. To see the graph at the best resolution enlarge the window to a full size view.

From the graph it can be seen that the first 3 modes are dominated by the motion in the X direction.

A graph of Participation factor in the Y direction will now be plotted on top of the existing graph.

Utilities
Graph Wizard...

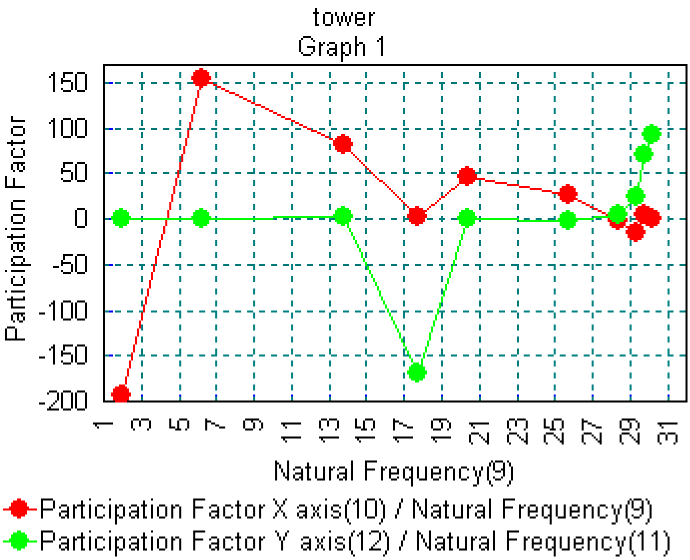
- Ensure the **Time history** option is selected and click the **Next** button.
- Select the **Previously Defined** option. Click the **Next** button.
- Select **Natural Frequency** results. Click the **Next** button.

The Y axis results to be graphed are now defined.

- Select **Named** results and click the **Next** button.
- Select **Participation Factor Y**. Click the **Next** button.

The results are to be plotted on top of the existing graph so no additional title information is to be added.

- Select the **Add to existing graph** option and ensure that the previous graph name **Graph 1** is selected.
- Click the **Finish** button.



The Participation factors for the Y axis are superimposed on the graph.

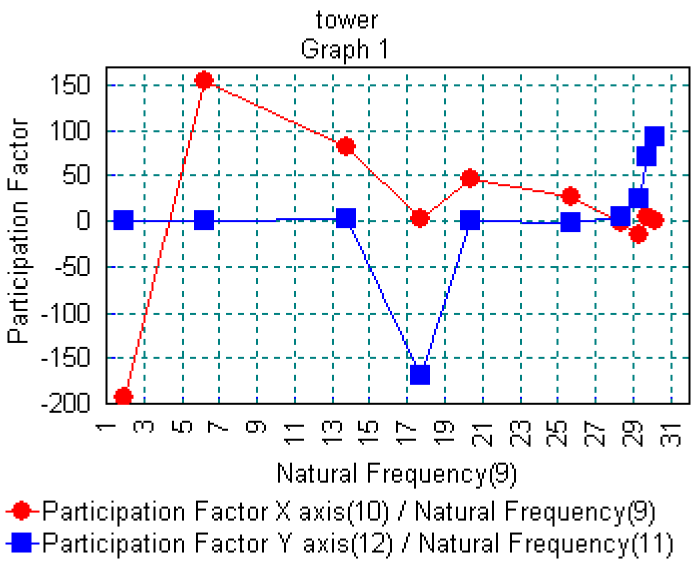
From the graph it can be seen that the fourth mode shape is dominated by motion in the Y direction.

Note. The graph properties may be modified by clicking the right-hand mouse button in the graph display area and selecting the **Edit Graph Properties** option.

To change the symbols on the second graph dataset:

Edit
Edit Graph Properties...

- Select the **Curves** tab and the second dataset **Participation Factor Y / Natural Frequency** from the dropdown list.
- Change the symbol to **Square**
- Select the **Symbol Start Colour** and select **dark blue**
- Select the **Line Colour** and select the same **dark blue**
- Click **OK** to action the change.



Creating Animations of Mode Shapes

The dominant mode shapes (Eigenvalue results 1, 2, 3 and 4) caused by the applied response spectra will now be animated. To see the deformations in the structure the deformed mesh layer must be displayed.

- Re-select the window containing the model data and use the maximise button  to increase the size to fill the graphic area.

- Turn off the display of the **Mesh**, **Diagram** and **Annotation** layers in the Treeview.

- Add the **Deformed mesh** layer and **OK** the dialog properties.

- In the Treeview right-click on Results File **Eigenvalue 1** and select the **Set Active** option.

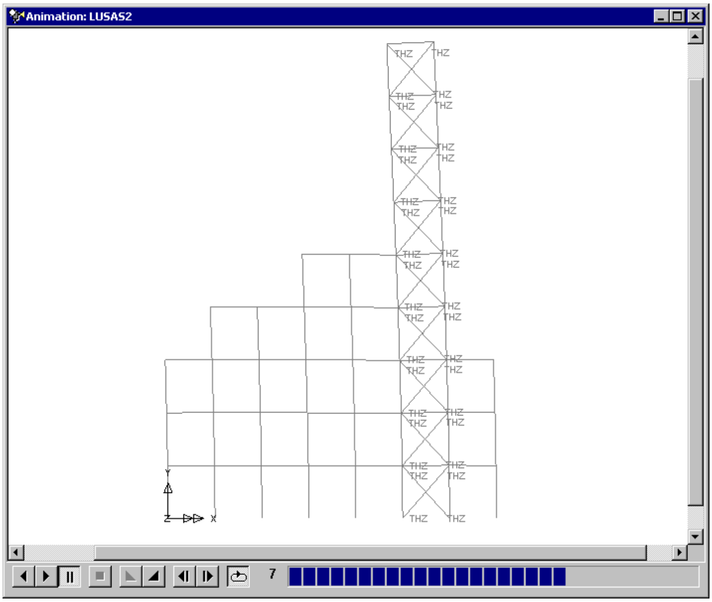
The deformed shape for Eigenvalue 1 will now be displayed.

Utilities
Animation Wizard...

- Ensure the **Active loadcase** button is selected and select the **Next** button.

- Use a **Sine** deformation with **14** frames. Set the deformation magnitude of **6** mm for the range **0 to 1**

- Click **Finish** to build and display the animation.



- To see the animation at the best resolution enlarge the window to a full size view.

The buttons at the bottom of the window may be used to slow-down, speed-up, pause, step through frame by frame, or stop the animation.

Saving Animations

Animations may be saved for replay in LUSAS at any time or saved for display in other windows animation players.

- Ensure the animation window is the active window.

File
Save As AVI...

- Save the animation to your projects folder and enter **tower_1** for the animation file name. An **.avi** file extension is automatically appended to the file name.
- Animations can be compressed to save disk space by altering the **Compress Video** slider.

File
Save

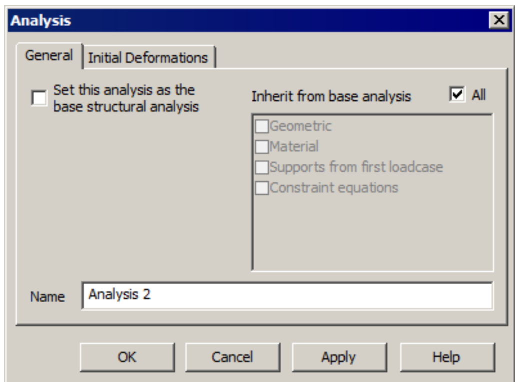
Save the model file.

Viewing the effects of self weight and spectral response

Spectral loadcases can be combined with other loadcases to provide results which include dead and live loads but since spectral loadcases are computed from an eigenvalue analysis the sign of the spectral displacements is always positive. The most adverse results can, however, be obtained using load combinations.

Analyses
Generate Structural Analysis...

- Add a new structural analysis to the model
- Ensure **Set this analysis as the base structural analysis** is not selected, and that **Inherit from base analysis** is selected.
- Name the analysis **Analysis 2** and press **OK**.



Loading from self weight can be automatically added to any loadcase by enabling gravity in the Treeview.

- In the Treeview expand **Analysis 2** and then right-click on **Loadcase 2** and rename it to be **Self Weight**
- In the Treeview right-click on **Self Weight** and select **Gravity**

Running the Analysis

Open the **Solve Now** dialog. Ensure that **Analysis 2** is selected and press **OK**.

Viewing the Results

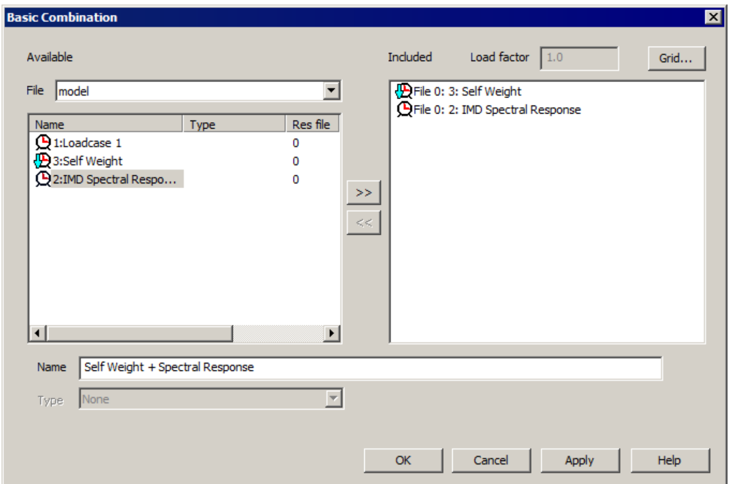
After a successful analysis the Self Weight results file will be added to the Treeview.

Defining Self weight and Spectral load combinations

Two basic load combinations will now be created to view the combined effects of the self weight and spectral results; one for self weight plus the spectral results and one for self weight minus the spectral results.

Analyses
Basic Combination...

The combination properties dialog will appear.



- Ensure results file **model** is selected from the drop down list.
- Select **Self Weight** and click the >> button to add the loadcase to the load combination. Leave the load factor set as **1.0**
- Select **IMD Spectral Results** and click the >> button to add the Spectral results loadcase to the load combination. Leave the load factor set as **1.0**
- Enter **Self Weight + Spectral Response** for the combination name
- Click the **OK** button to finish.

Note. The IMD Spectral Results loadcase references the eigenvalues held in the Analysis 1 results file.

Analyses
Basic Combination...

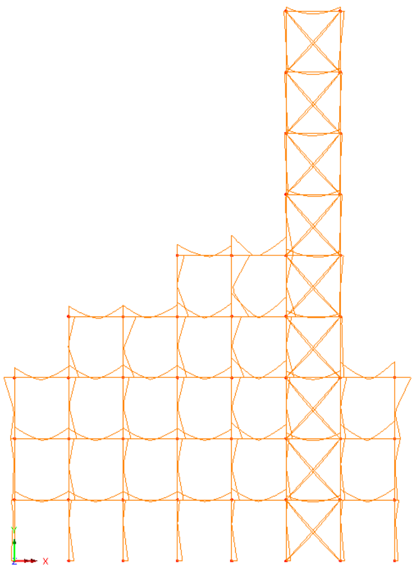
The previous procedure is repeated to create a second basic load combination.

- Ensure the file **model** is selected from the drop down list.
- Include the **Self Weight** loadcase with a load factor set to **1.0**
- From the **model** file include all the IMD Spectral Results and set the load factor for as **-1.0**
- Enter **Self Weight - Spectral Response** for the combination name
- Click the **OK** button to finish.

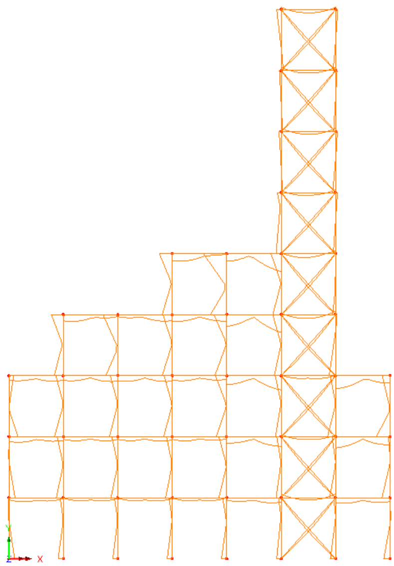
File
Save

Save the model file. This also saves any created load combinations with the model.

By setting a Self Weight and Spectral Response load combination active, shear force and bending moments can be viewed for each combination. Use or modify the Diagrams layer to view values on the frame.



Mz for Self Weight + Spectral Response



Mz for Self Weight - Spectral Response

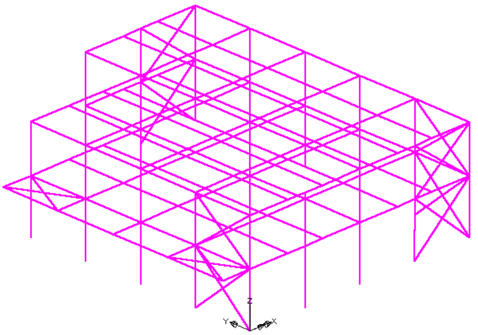
This completes the example.

Seismic Response of a 3D Frame (Frequency Domain)

For software product(s):	All (except LT versions)
With product option(s):	None.
Note: This example exceeds the limits of the LUSAS Teaching and Training version.	

Description

This example examines the spectral response of a 2-storey 3D frame. The geometry of the structure has been simplified to a wire-frame or stick representation with each of the members of the structure being represented by a line feature only.



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

Objectives

The required output from the analysis consists of:

- ❑ Axial force and bending diagram from a CQC combination.

Keywords

Seismic, Spectral, Response, Mass Participation, Interactive Modal Dynamics, Excitation, Eigenvalue, CQC Combination.

Associated Files

- **3D_frame.mdl** Model of the structure.
- **3D_frame_spectrum_EC8.vbs** defines a typical spectral curve to Eurocode 8

Discussion

The mesh definition used on a dynamic analysis is somewhat different from that used on a static stress analysis. In a static analysis, and with experience, it is usually not too difficult to estimate where the high stresses are likely to occur. These estimates can then be used to develop a meshing strategy with a fine mesh in high stress locations and a coarse mesh in less critical locations. For a dynamic analysis the interaction between the stiffness and inertia forces will lead to deflected shapes which can be very different from those expected in a static analysis.

In a dynamic analysis both stiffness and mass distribution has to be considered. Generally, the best strategy for a dynamic analysis is to have a uniform mesh over the entire structure but in stiff regions a more coarse mesh can be used. In regions that are more flexible, or where heavy masses are located, the mesh can be more refined.

In this example the global behaviour of the building is being considered for earthquake response. The lower frequencies will be dominant in this analysis and a relatively coarse mesh will suffice. If the higher frequencies are important, or if a local response due to panel modes for individual beams and columns is to be considered, a revised mesh with more elements would need to be considered.

The spectral response analysis is performed in two distinct stages:

1. A natural frequency analysis is performed. This is used here to calculate the first 30 natural modes of vibration of the structure. The eigenvalues, frequencies and eigenvectors (mode shapes) are stored and used in the subsequent spectral response analysis. In order to carry out a spectral analysis the modes must be normalised with respect to the mass. Although natural frequencies are obtained from an eigenvalue analysis any information regarding the magnitudes of deformations or moments is non-quantitative.

2. A spectral response calculation is performed interactively as a results processing operation using the Interactive Modal Dynamics (IMD) facility. This is an alternative to performing a non-interactive spectral response analysis in LUSAS Solver.

In a spectral response analysis, the structure is subjected to support excitation. In this example this is assumed to be the effect of seismic motion although any support motion could be envisaged. The excitation is specified as a spectral curve, in terms of period versus acceleration. Damping, implicitly included in the spectral curve, may also be specified at this stage.

From the eigen analysis, participation factors indicate, for each mode, the degree of structural response associated with an applied excitation. In the spectral analysis a mode combination is calculated from which a worst-case displacement/moment envelope may be obtained.

Modelling

Running LUSAS Modeller

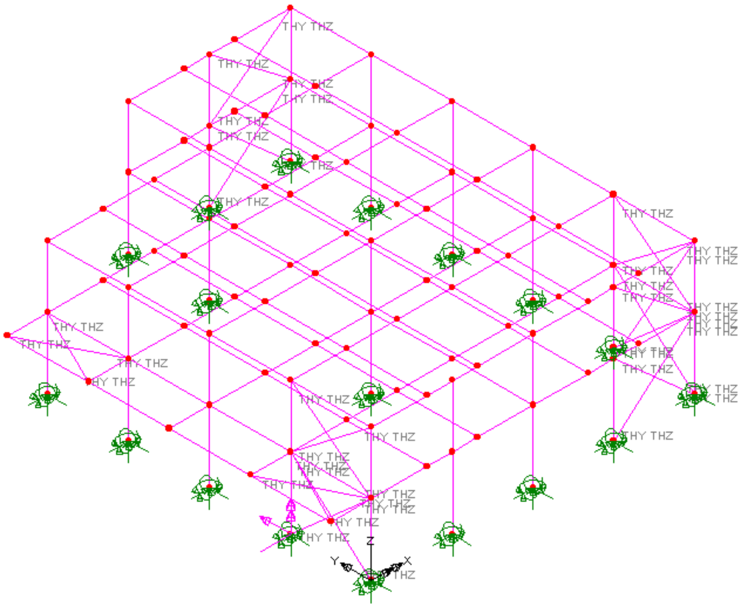
For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

Building and loading the model

For this example a model file is provided:

File
Open…

To create the model, open the read only file **3d_frame.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.



 If necessary, select the isometric button to view the frame in 3D.

Save the model

File
Save As

Save the model file to the **\<Lusas Installation Folder> \Projects** directory.

Note. No static structural loading is required for this analysis because a spectral combination is carried out during results processing using the results from the natural frequency analysis which is independent of applied loading.

The modelling will now be completed by defining the controls necessary to extract the natural frequencies.

Defining Eigenvalue controls

Eigenvalue controls are defined as properties of the loadcase.

- In the Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Eigenvalue** from the **Controls** menu.

The Eigenvalue dialog will appear.

The following parameters need to be specified:

- Set the **Number of eigenvalues** as **30**
- Ensure the **Shift to be applied** is set as **0**
- Ensure the **Type of eigensolver** is set as **Default**

Note. Eigenvector normalisation is set to **Mass** by default. This is essential if the eigenvectors are to be used for subsequent IMD analysis in results processing as they are in this case.

- Click the **OK** button to finish.

File
Save

Save the model file.

Running the Analysis

With the model loaded:

Open the **Solve Now** dialog. Ensure **Analysis 1** is selected and press **OK**.

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

Viewing the Results

Analysis loadcase results for each eigenvalue can be seen in the Treeview. Eigenvalue 1 is set to be active by default.

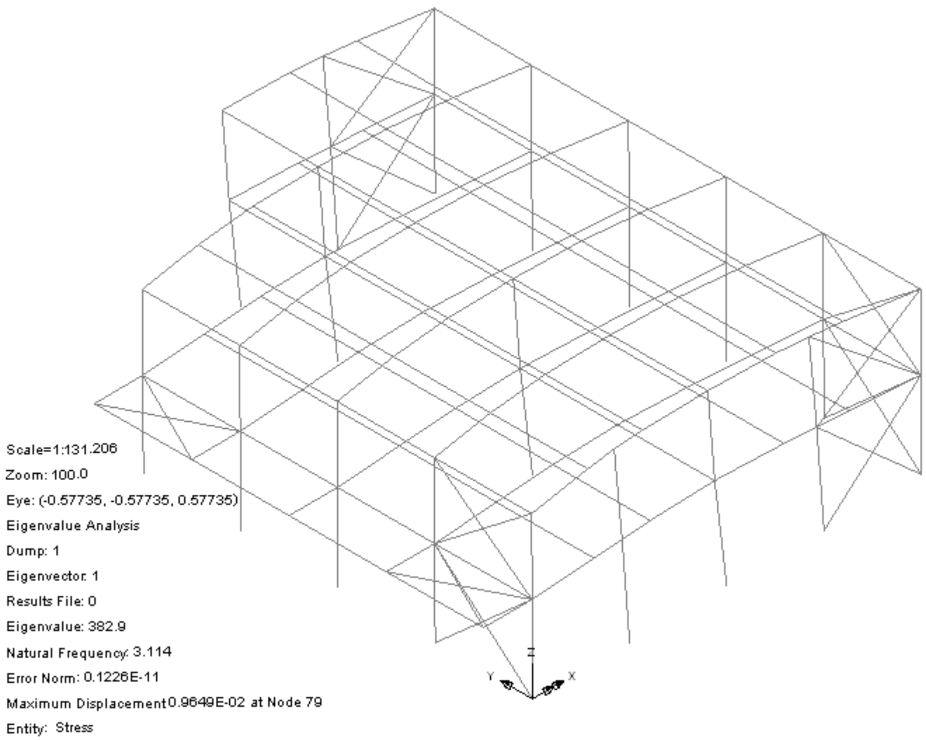
Plotting Mode Shapes

- Turn off the display of the **Mesh, Geometry** and **Attributes** layers in the Treeview.

- With no features selected click the right-hand mouse button in a blank part of the Graphics area and select the **Deformed mesh** option to add the deformed mesh layer to the Treeview. Select the Visualise tab and, if selected, de-select the **Beam end release** option. Click the **OK** button to accept the remaining default values and display the deformed mesh for Eigenvalue 1.

Utilities
Annotation >
Window Summary

- Show the **Window Summary** for an overview of the results.



Note. The window summary displays the values of the eigenvalue and the natural frequency and also a value for displacement at a node. It should be noted that the displacement value is non-quantitative and is related to the amount of mass in a particular mode using the mass normalisation technique. Therefore the only items that can be found using a basic eigenvalue analysis are the frequency and the mode shape.

Note. The mode shape may be inverted from that shown above. This is because the sense is arbitrary since during vibration deformed shape will appear in both directions.

- In the Treeview right-click on **Eigenvalue 2** and select the **Set Active** option. The deformed mesh plot for Eigenvalue 2 will be displayed.

By setting each Eigenvalue to be active the deformed mesh can be seen for all mode shapes.

Printing Eigenvalue Results

Eigenvalue results for the whole structure can be displayed in the Text Output window.

Utilities
Print Results Wizard

- Choose **Active** loadcases and click **Next.**
- Select **None** from the Entity drop down list. Ensure **Eigenvalues** is displayed in the Type drop down list and click **Finish.**

The Eigenvalue results will be printed to the Text Output window. For inspection only the first 10 modes will be printed here.

MODE	EIGENVALUE	FREQUENCY	ERROR NORM
1	382.904	3.11433	0.168140E-11
2	1011.29	5.06126	0.774353E-12
3	1599.54	6.36527	0.431202E-12
4	1891.32	6.92154	0.236809E-12
5	2336.34	7.69287	0.571844E-12
6	2525.21	7.99777	0.384185E-12
7	3924.31	9.97015	0.421926E-12
8	4153.57	10.2572	0.260236E-12
9	4183.07	10.2936	0.248550E-12
10	6658.98	12.9874	0.380041E-12

Note. The frequency in Hertz can be obtained by dividing the square root of the eigenvalue by 2π, and the period of vibration in seconds is obtained using the reciprocal of frequency (1/frequency). Values of error norm may vary from those shown.

Caution. The system eigenvectors have been normalised (in this case with respect to mass) therefore any derived quantities such as displacement and moment are also normalised and are not true design values.

- Close the text window by selecting the close button in the top right hand corner of the window.

Checking the Mass Participation Factor

Note. In order to carry out a successful response analysis you should ensure that a significant proportion of the total mass has been accounted for in the analysis. This requires checking that around 90% of the total mass is in the global x and y directions. Failure to check that a significant proportion of the total mass has been accounted may lead to important modes being missed and subsequent errors in the resulting response analysis results.

Utilities
Print Results Wizard…

- Ensure **Active** loadcases is chosen and click **Next.**
- Select **None** from the Entity drop down list. Ensure **Sum Mass Participation Factors** is displayed in the Type combo box and click **Finish.** The Sum Mass Participation Factors results will be printed to the Text Output window.

For inspection only modes 20 to 25 will be printed here. In can be seen that the 90% value has been achieved in mode 22.

MODE	SUM MASS X	SUM MASS Y	SUM MASS Z
20	0.861201	0.933385	0.122995E-01
21	0.870885	0.937375	0.123688E-01
22	**0.945303**	**0.937380**	**0.123710E-01**
23	0.945422	0.937388	0.130555E-01
24	0.945428	0.937470	0.304988
25	0.945451	0.937589	0.308786

- Close the text window by selecting the close button in the top right hand corner of the window.
- Use the maximise button to increase the size of the graphics window.

Spectral Response Analysis

Spectral response calculations are performed using the IMD (Interactive Modal Dynamics) facility. This involves defining the spectral curve and excitation and specifying the results required in an IMD loadcase. The spectral results can then be interrogated by setting the IMD loadcase active.

With spectral response analysis additional damping information may also be set. Unlike results from a natural frequency analysis, the output values obtained from a spectral combination are design values.

Stage 1 : Defining a Spectral Excitation Curve

Values for the spectral excitation curve need to be specified. For this analysis Acceleration versus Period values are to be used.

3d_frame_spectrum_ec8.vbs defines response spectrum to Eurocode EC8.

File
Script >
Run Script…

- Select the file **3d_frame_spectrum_ec8.vbs,** which is located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory. Click **OK**
- A response spectra dataset **3D_frame_spectrum** containing the spectral data for period and acceleration will be created in the Treeview.

Stage 2 : Defining the Dynamic Excitation

Analyses
IMD Loadcase…

An Interactive Modal Dynamics loadcase **IMD 1** will be added to the Treeview.

- Select **Support Motion** from the Excitation drop down list and **Spectral** from the Results drop down list. Ensure the **Use all modes** option is selected.
- Click the Excitation **Set** button.
- Ensure the **Acceleration** support motion option is set to a **User specified** direction which is **Relative.** In the box next to the X direction enter the value of **1.0** and in the box next to the Y direction enter the value of **0.6**.
- Click the **OK** button to return to the IMD loadcase dialog.

Note. Using the user-specified option a single earthquake response can be applied in multiple directions at the same time. In this case 100% of the earthquake response is being applied in the X direction at the same time as 60% of the earthquake response is being applied in the Y direction. In real-life situations, in the absence of additional data, and depending upon the design code being used, a percentage of the horizontal earthquake response could also be applied in the vertical Z-direction.

- Click the Results **Set** button.
- On the Results dialog ensure the type of spectral response is **CQC Combination** with the damping variation correction set as **None**. The response spectrum **3D_frame_spectrum** read in from the supplied VBS file will already be selected in the drop down list.
- Click the **OK** button to return to the IMD loadcase dialog.
- Click the **OK** button to finish.

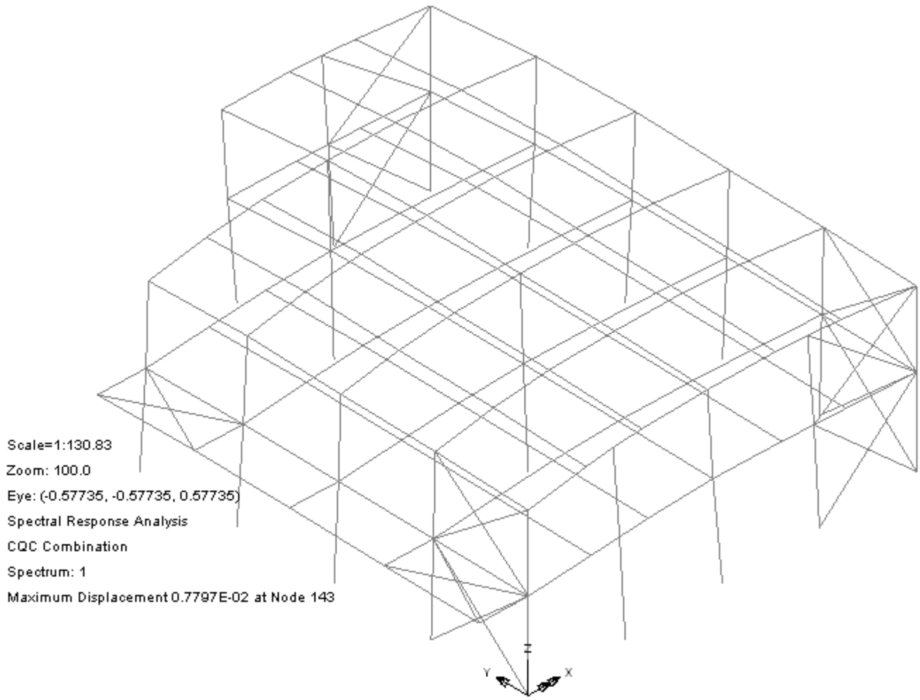
Note. There are a number of different methods of carrying out modal combinations:

1. If the natural frequencies are well separated the square root of the sum of the squares (SRSS) has been shown to produce acceptable results.
2. For a situation where the modes have close separation of natural frequencies then an absolute sum (AbsSum) is taken on the assumption that they can peak in phase with each other.
3. Alternatively a compete quadratic combination (CQC) method can be used (as in this example) which effectively combines both the SRSS and AbsSum methods as it accounts for the separation between modes.

Note. If the defined damping differs from that inherent in the spectral curve a correction may be applied. There are a number of damping variation correction options available for use. For more details on these see the *Theory Manual*.

Stage 3: Selecting the IMD Results Loadcase

In the Treeview right-click on the **IMD Loadcase 1** loadcase name and select the **Set Active** option.

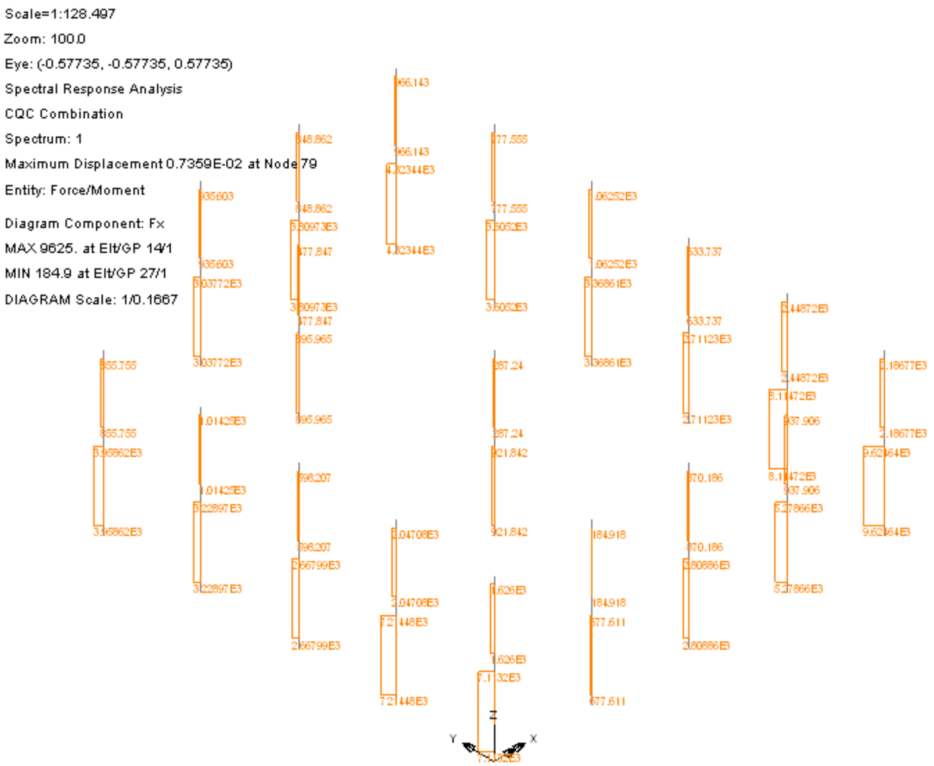


Stage 4 : Displaying the Spectral Combination Results

Plots showing the axial force and bending moments in the members for the spectral combination is to be displayed.

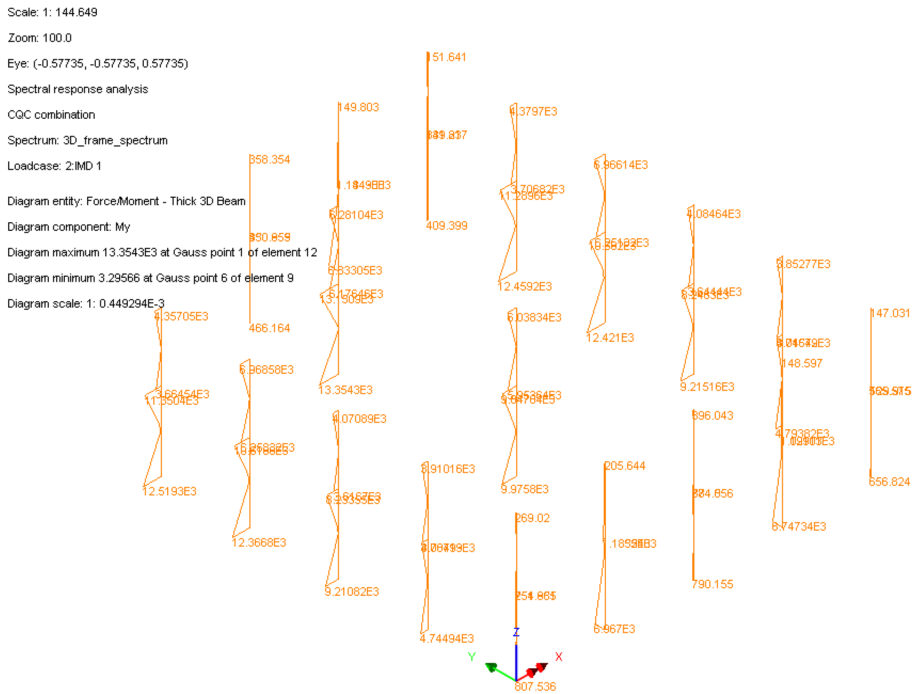
- In the Treeview right-click the group named **Columns** and **Set as Only Visible** to only view the column members.
- Turn off the display of the **Deformed Mesh** layer in the Treeview .
- With no features selected, click the right-hand mouse button in a blank part of the Graphics area and select the **Diagrams** option to add the Diagrams layer to the Treeview.
- Select **Force/Moment - Thick 3D Beam** results of axial force **Fx** in the members. Select the **Diagram display** tab and select the **Label values** option. Plot values on **80%** of the element length. Click the **OK** button to finish.

Note that the Orientate by element axes and Orientate flat to screen/page option are greyed-out because they have no relevance for axial stress diagrams. A 'flat to screen/page' plot is obtained by default.



Results will be calculated for the active IMD load case and an axial force diagram of force in each member will be displayed. If necessary, use the Zoom in button to view results information. These type of results can also be plotted in isolation on selected model features.

- In the Treeview double-click the **Diagram** layer to view its properties.
- Select **Force/Moment - Thick 3D Beam** results of axial force **My** in the members. Click the **OK** button to finish.



Note that Orientate by element axes is the default diagram display option for all components other than Fx. This can be changed by selecting the Diagram display tab and selecting Orientate flat to screen / page as necessary.

File
Save

Save the model file.

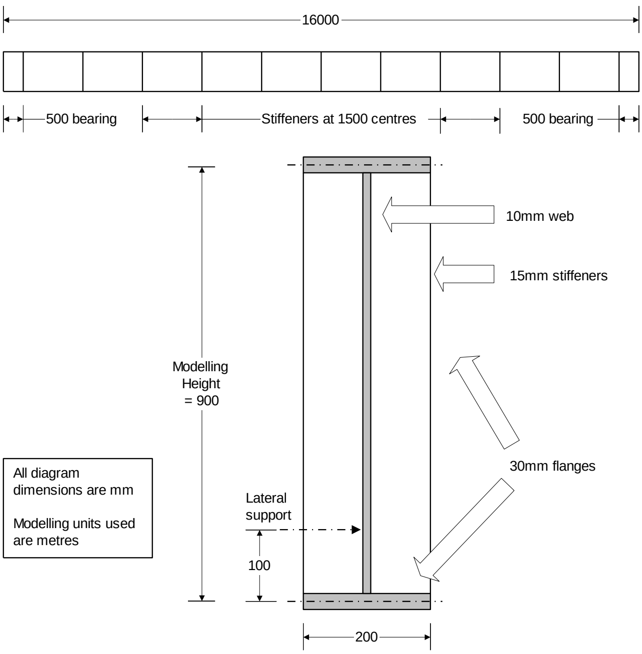
This completes the example.

Buckling Analysis of a Plate Girder

For software product(s):	Any Plus version.
With product option(s):	None.
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

The buckling behaviour of a steel plate girder with web stiffeners is to be examined.



The girder is supported at both ends by fixed and roller bearing supports. A cross-slab connection to the girder provides a partial lateral restraint. A distributed load representing transverse steel decking is applied to a selected part of the web of the girder. The steel plates are modelled using thin shell (QSL8) elements.

Units of kN, m, t, s, C are used throughout the example.

Objectives

The required output from the analysis consists of:

- ❑ A deformed shape plot showing displacements caused by the imposed loading.
- ❑ Animation
- ❑ Buckling load

Keywords

Simple Geometry, Plate Buckling, Linear Buckling, Eigenvalue Buckling, Deformed Mesh Plot, Printing Eigenvalues.

Associated Files

- ❑ **plate_girder_modelling.vbs** carries out the modelling of the example. After creating a new model and running this file, the example can be continued from the section titled ***Running the Analysis*.**

Modelling

Running LUSAS Modeller

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

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 the Longitudinal Beam Model

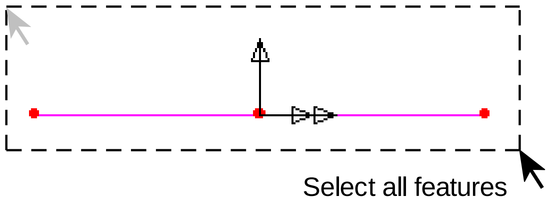
- Enter the file name as **plate_girder**
- Use the **Default** working folder.
- Enter the title as **Buckling analysis of a plate girder**
- Select model units of **kN,m,t,s,C** from the drop down list provided.
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the **Standard** startup template.
- Select the **Vertical Y axis** option and click **OK**.

Feature Geometry

A cross-section of the girder is created initially and swept and copied into a half-model before mirroring to create the full plate girder model.

Geometry
Line >
Coordinates...

Enter coordinates of (**-0.1,0**), (**0,0**) and (**0.1,0**) to define Lines representing the bottom flange of the girder. Click the **OK** button.

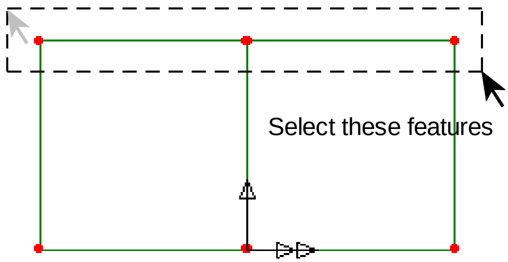


- Drag a box around all the features just drawn.

Geometry
Surface >
By Sweeping...

Enter translation value of **0.1** in the **Y** direction to create surfaces that will define part of the end stiffener. Click the **OK** button to finish.

- Drag a box around the 2 line features shown.

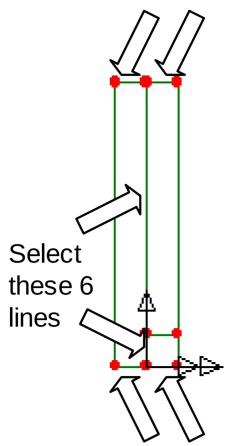


Geometry
Surface >
By Sweeping...

Enter a translation value of **0.8** in the **Y** direction and click **OK** to create surfaces that will define part of the end stiffener.

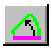
The girder has a 500mm long bearing region at the end which will be created first.

- Select the 6 Lines representing the cross-section of the girder as shown.



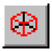
Geometry

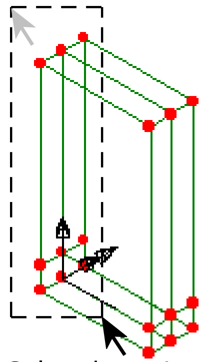
Geometry
Surface >
By Sweeping...

 Enter a translation value of **0.5** in the **Z** direction. Click the **OK** button to finish.

Note. Save the model regularly as the example progresses. The Undo button may be used to correct any mistakes made since the last save.

Note. Holding down the **L** key while selecting ensures that only lines will be selected. Similarly the **P** key limits the selection to Points, the **S** key; Surfaces, the **V** key; Volumes and the **G** key; Geometry. Similar shortcuts exist for elements, nodes, etc., which can be found in Modeller Manual.

 Use the Isometric button to view the model to a similar view to that shown



Select these 4 surfaces

The bearing region has a stiffener either side of it.

- Select the 4 surfaces representing the 2 end stiffeners.

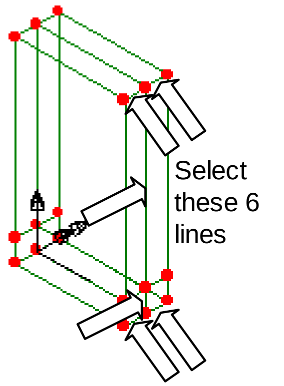
Geometry
Surface
Copy...

 Enter a value of **0.5** in the **Z** direction to copy the surfaces to define the web stiffeners. Click **OK** to finish.

Note. Many of the more common tasks are provided in the context menu. With the model geometry selected right-hand within the view to make the context menu appear. From this menu you can **Copy**, **Delete**, **Move** and **Sweep**.

The girder has 1500mm long internal panel sections along its length.

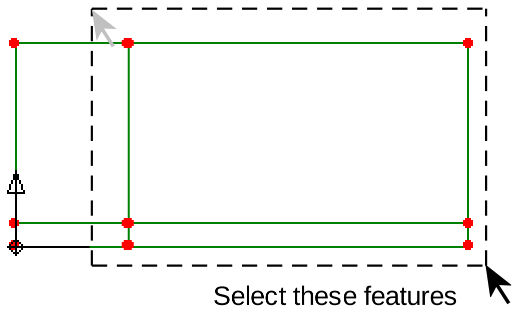
- Select the 6 Lines representing the cross-section of the girder. If necessary use the Zoom facility to simplify the selection.



Geometry
Surface >
By Sweeping...

Enter a translation value of **1.5** in the **Z** direction. Click the **OK** button to finish.

- Right-click on the X: N/A **X coordinates** readout display on the Status bar and select **Rotate +Z +Y**.
- Hold down the **Shift** key on the keyboard and select the **X** coordinate readout panel again to obtain the reverse view looking along the X-axis.
- Drag a box around all the features shown to select the first panel section and web stiffener.



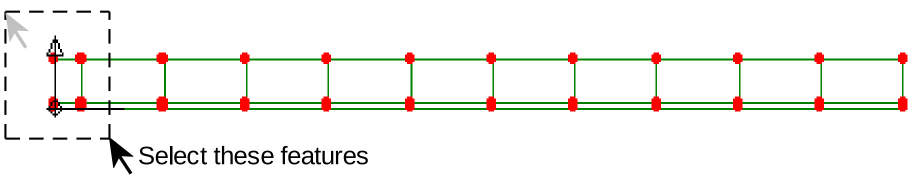
Geometry
Surface >
Copy...

Enter a value of **1.5** in the **Z** direction to copy the surfaces used to define the web and stiffener.

- Enter the number of copies as **9** and click the **OK** button.

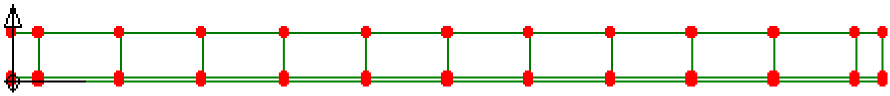
Finally, the other bearing end to the girder has to be created.

- Drag a box around all the features shown to select the bearing end of the girder and web stiffener.



Geometry
Surface >
Copy...

Enter a value of **15.5** in the **Z** direction to copy the surfaces used to define the web and stiffener. Click **OK** to finish.



Meshing the web stiffeners

Line mesh divisions will be used to control the surface mesh density for the plates. All Lines defined so far have 4 Line mesh divisions by default. Certain Lines require different divisions to be specified.

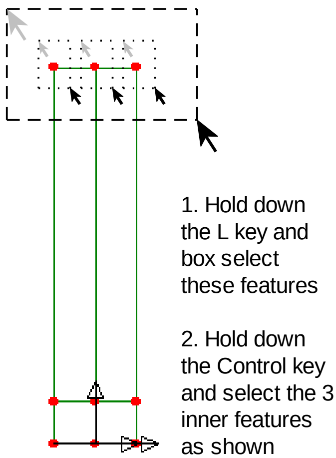
Select the home button to view the girder end on.

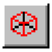
Line mesh

The Lines representing the top flange, lower flange, and part of the web and web stiffeners require 1 mesh division to be assigned to them. These could be selected individually in order to assign the line mesh division to them but LUSAS provides some advanced selection / deselection tools for this type of work.

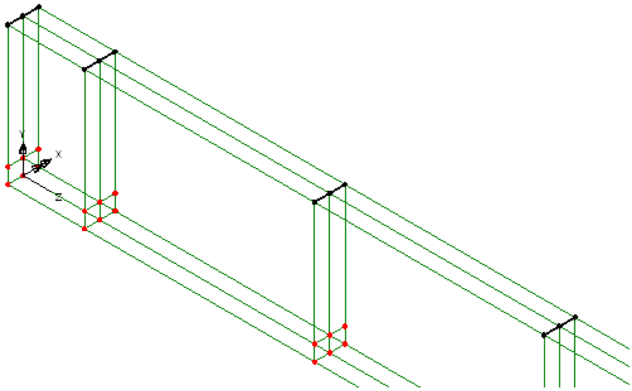
- Hold down the **L key** on the keyboard and drag a box around the area shown to only select the Lines within the box.
- Then, with the lines still selected, hold down the **Control** key and drag separate boxes around each of the 3 points shown.

Note. Using the **Control** key in the manner described deselects the lines going into the screen from the previous box-selection. These lines will therefore remain meshed with 4 line divisions.



 Selection of lines can be checked by using the Isometric button.

Only the lines shown in the diagram should appear highlighted to show that they have been selected.



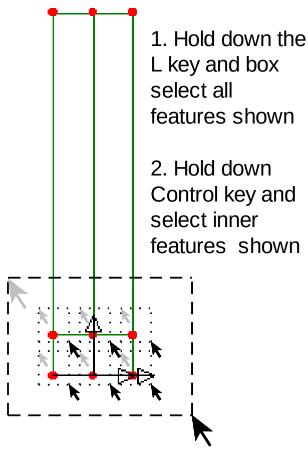
 After checking, select the Home button to return the model to its initial view.

- With the lines selected drag and drop the Line mesh attribute **Divisions=1** from the Treeview onto the selected features.

- In a similar fashion to that done for the top flange, hold down the L key and drag a box around the area shown to select all the Lines within.

- Then, with the lines still selected, hold down the **Control** key, and drag separate boxes around each of the 6 points shown to deselect the lines going into the screen from the initial selection.

- Drag and drop the Line mesh attribute **Divisions=1** from the Treeview onto the selected features.

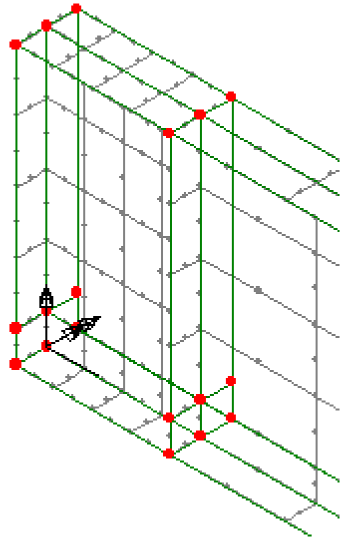


Surface mesh

Attributes
Mesh >
Surface…

- Select **Thin shell, Quadrilateral** elements with **Quadratic** interpolation. LUSAS will select QSL8 elements.

- Enter the attribute name as **Thin Shells (QSL8)** and click **OK** to finish.



- Select the whole model (**Control** + **A** keys)
- Drag and drop the surface mesh attribute **Thin Shells (QSL8)** from the Treeview onto the selected features.

The mesh will be drawn. Note that, by default, the mesh nodes as shown on these accompanying diagrams are not displayed.

Use the Isometric button to view the mesh on the model.

Use the Zoom in button to check the mesh definition.

Use the resize button to show the whole model

Use the Cursor button switch back to the normal cursor tool.

Geometric Properties

Three steel thicknesses are required to model the web, flanges and web stiffeners.

Attributes
Geometric >
Surface…

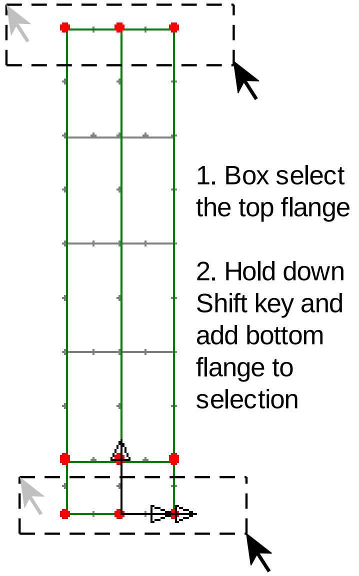
- Enter a value of **0.01** for the thickness. The eccentricity can be left blank. Enter the attribute name as **Thickness=0.01m**. Click the **Apply** button to create the attribute.
- Amend the thickness to **0.015**. Change the attribute name to **Thickness=0.015m**. Click the **Apply** button to create the attribute.
- Amend the thickness to **0.03**. Change the attribute name to **Thickness=0.03m**. Click the **OK** button to finish.

Select the home button to view the beam end on.

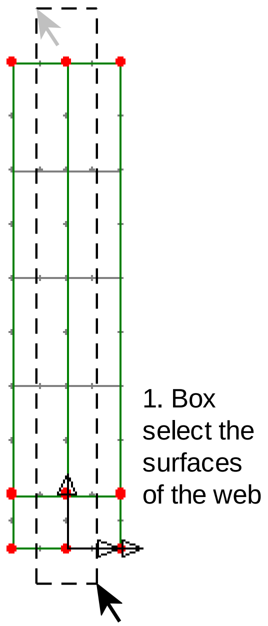
- Select the whole model (**Control + A keys**)
- Drag and drop the geometry attribute **Thickness=0.015m** from the Treeview onto the selected features.

Select the fleshing on/off button to turn-off the geometric visualisation.

- Select the Surfaces representing the top and bottom flanges.
- Drag and drop the geometry attribute **Thickness=0.03m** from the Treeview onto the selected features.

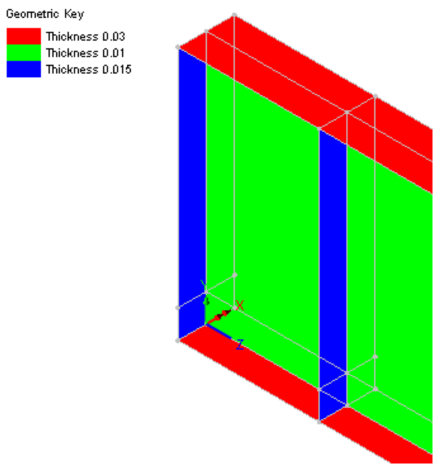


- Select the Surfaces representing the web.
- Drag and drop the geometry attribute **Thickness=0.01m** from the Treeview onto the selected features.
- Once done, click on a blank part of the graphics window to deselect the web surfaces.



Checking attribute assignments

Note. Once assigned to the model, attributes such as geometry may be visualised.



- In the Treeview click the right-hand mouse button on the **Geometry** layer and select **Properties**.
- On the Colour by drop-down menu select **Assignment** then select the **Set** button. Change the attribute type to show **Geometric** attributes and click **OK**. Finally, select **Solid** on the geometry properties box and click **OK**

Use the Isometric button to view the model to a similar view to that shown.

- Check that the geometry assignments have been correctly assigned and then switch off the visualisation. In the Treeview click the right-hand mouse button on the **Geometry** layer and select **Properties**.
- On the Colour by drop-down menu select **Own colour** then Finally, de-select **Solid** on the geometry properties box and click **OK**

Material Properties

Attributes
Material >
Material Library…

Select material **Mild Steel** from the drop-down list, leave the grade as **Ungraded** and click **OK** to add the material attribute to the Treeview.

- With the whole model selected, drag and drop the material attribute **Iso1 (Mild Steel Ungraded)** from the Treeview onto the selected features. Click **OK** ensuring it is assigned to all surfaces.

Supports

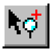
Three support conditions are required. A fixed bearing, a rolling bearing, and a lateral restraint to the flange of the girder.

Attributes
Support…

Set the translation in the **X** and **Z** directions to **Fixed**. Set the spring stiffness in the **Y** direction to **200e8** and specify that the Spring stiffness distribution is **stiffness/unit area**. Enter the attribute name as **Fixed Bearing**. Click the **Apply** button to reuse the dialog for the other support conditions.

- Set the translation in the **Z** direction to **Free** and leave all other translations as specified previously. Enter the attribute name as **Rolling Bearing**. Click the **Apply** button to reuse the dialog for the next support condition.

- Set the spring stiffness in the **X** direction to **100e8.** Set the translation in the **Y** and **Z** directions to **Free**. Ensure that the Spring stiffness distribution is **stiffness/unit length**. Enter the attribute name as **Lateral Support**. Click the **OK** button to finish.

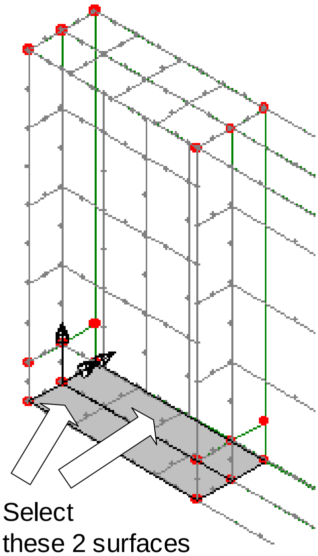
 Use the Zoom in button to enlarge the view of the left-hand end support.

 Use the Cursor button switch back to the normal cursor tool.

- Select the 2 flange Surfaces shown. Take care to not select the lowest web surface by mistake.

Note. For features lying underneath others in the Graphics Window, continually clicking on a feature will cycle through selecting different features in turn.

- Drag and drop the support attribute **Fixed Bearing** from the  Treeview onto the selected features. Click **OK** ensuring that it is assigned to **Surfaces**.



Select
these 2 surfaces

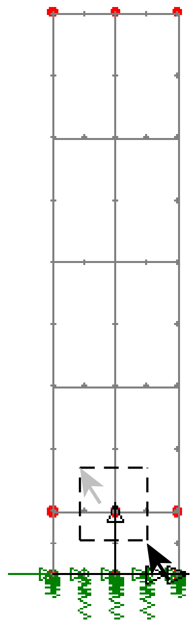
 Use the resize button to show the whole model.

 Use the Zoom in button to enlarge the view of the right-hand end support.

 Use the Cursor button switch back to the normal cursor tool.

- Select the equivalent 2 lower Surfaces for the right-hand support.
- Drag and drop the support attribute **Rolling Bearing** from the  Treeview onto the selected features. Click **OK** ensuring that it is assigned to **Surfaces**.

 To help assign the lateral supports return the model to the default home view.



Select this Point

- Drag a box around the Point shown (All Lines in the same plane will be selected).
- Drag and drop the support attribute **Lateral Support** from the  Treeview onto the selected features, Click **OK** ensuring that the support attribute is assigned to **Lines**.

Loading

Self-weight will be applied to the model as a property of the analysis.

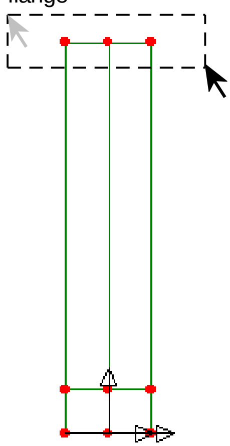
- In the  Treeview, expand **Analysis 1** then right-click on **Loadcase 1** and select **Gravity** to apply self-weight to the model

In addition to the self-weight, a uniformly distributed load is to be applied to the flange of the beam.

Attributes
Loading…

- Select the **Global Distributed** option and click **Next**
- Enter a load of **-10** in the **Y** direction and specify that the load distribution be **Per Unit Area**. Enter the attribute name as **Imposed Loading** and click the **Finish** button.
- Select the Surfaces of the top flange as shown

Box select the top flange



- Drag and drop the loading attribute **Imposed Loading** from the  Treeview onto the selected features. Click **OK** assigning it to **Surfaces** as **Loadcase 1**
- Click in a blank area of the view window to deselect any currently selected features

This completes the modelling of the girder.

Eigenvalue Analysis Control

By default an eigenvalue analysis extracts the natural modes of vibration of a structure. It can also be used to solve buckling load analysis problems. The solution parameters for buckling analysis are specified using an eigenvalue control attribute. In this example only the 1st natural mode of buckling of the girder is to be investigated.

Eigenvalue analysis control is defined as a property of a loadcase.

- In the Treeview right-click on **Loadcase 1** and select **Eigenvalue** from the **Controls** menu.
- Select a **Buckling Load** solution for the **Minimum** number of eigenvalues.
- Change the type of eigensolver to be used to **Subspace Jacobi**
- Enter the **Number of eigenvalues** to be calculated as **1**
- Enter the **Number of starting iteration vectors** as **2**
- Enter the **Shift to be applied** as **-10**
- Click the **OK** button.

Note. The use of a shift and the Subspace Jacobi solver is required when applying uniformly distributed spring supports to quadratic elements in an eigenvalue analysis. This is due to the theoretical negative stiffnesses required by the shape functions to evenly distribute the support stiffness. During the analysis a number of negative pivot warnings will be shown, due to these theoretical values, which can be ignored.

Saving the model

File
Save

To 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.

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 Treeview.

In addition will be created in the Associated Model Data directory where the model file resides, including:

- **plate_girder.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **plate_girder.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **plate_girder_modelling.vbs** carries out the modelling of the example.

File
New…

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 **plate_girder**

File
Script >
Run Script...

To recreate the model, select the file **plate_girder_modelling.vbs** which is located in the **<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

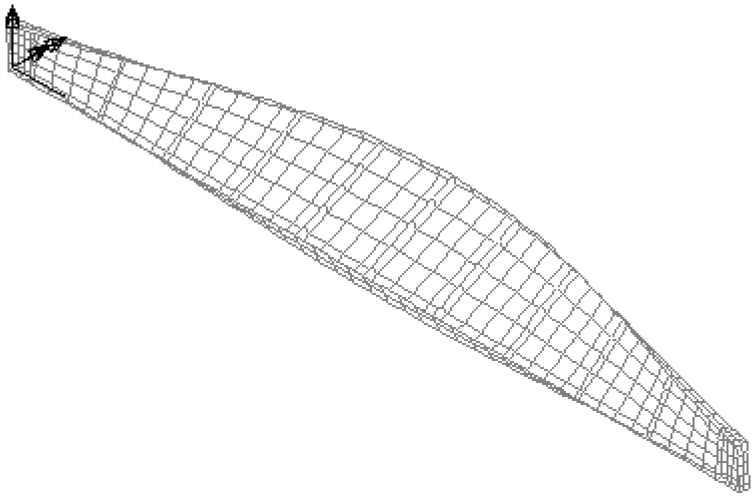
Analysis loadcase results are present in the Treeview.

Select an isometric view of the model.

- Turn off the display of the **Attributes, Geometry** and **Mesh** layers in the Treeview.

Deformed Mesh Plot

- Turn on, and then double-click the **Deformed mesh** layer to the Treeview.
- From the Deformations tab, select **Specify factor** and enter a factor of **2.5**
- Click the **OK** button to display the first eigenmode shape.



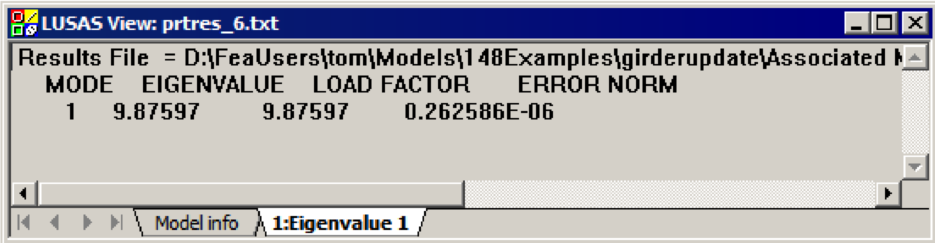
Printing the Buckling Load Factors

In an eigenvalue buckling analysis, the load factors are equivalent to the eigenvalues. Load factors are the values by which the applied load is factored to cause buckling in the respective modes. Eigenvalue results for the whole model can be displayed in the text window.

Utilities
Print Results Wizard...

- Change the entity type to **None**, ensure that results type **Eigenvalues** is selected and click the **Finish** button.

The Eigenvalue results will be printed to the text window with the Load factors being given in the eigenvalue results column. Error norms may vary.



Calculating the Critical Buckling Load

The applied load must be multiplied by the first load factor (9.87597) to give the value of loading which causes buckling in the first mode shape. The initial buckling load is therefore 9.87 x the applied dead and live load.

Note. An applied load of unity could be used in an eigenvalue analysis - in which case the eigenvalues produced would also represent the critical loads at which the structure would buckle. However, to prevent potential convergence problems with the analysis it is more usual to apply actual in-service loading and multiply the applied load by the eigenvalue to give the critical buckling load for each eigenvalue.

This completes the example.

Important Notes Relating to Buckling Analysis

This example has taken you through the process of carrying out a linear buckling analysis on a plate girder. Linear buckling analysis is a technique that can be used to estimate the load that can be supported by a structure prior to structural instability or collapse. However a number of points should be considered when carrying out this type of analysis:

- Buckling analysis is dependent on applied loading. All the loads on the structure must be applied in a single loadcase, this being loadcase 1. In addition, load factors appropriate to the analysis should be included when the loads are assigned.

- Buckling analysis is dependent on the initial model geometry being considered. A perfectly straight (and undeformed) model will provide different answers from a model with imperfect geometry. If required, an imperfection can be built into the initial model either by manually defining the appropriate geometry or by using the deformed mesh from one analysis as the starting point for a further analysis. To do the latter setup a new analysis using the **Analyses > Generate Structural Analysis** menu and select a deformed mesh from the original analysis in the **Initial Deformations** tab.

- An eigenvalue buckling analysis actually requires the use of elements with geometrically nonlinear capabilities in order to create additional stress terms in the stiffness matrix. Some elements do not have a non-linear capability. Refer to the finite element library before such an analysis is commenced.

- The eigenvalue buckling analysis of the structure can only be used to provide the mode shape of the structure and the critical buckling load. The stresses and displacements that are obtained are relative to the unit normalised eigenvector and are generally of no practical use. To obtain member forces for the girder it is necessary to perform a further linear static analysis with the same combination of loading on the structure. The stresses and displacements in the structure when the critical load is applied may be obtained simply by performing a linear static analysis with the loads factored to the buckling load previously derived. These may be compared to other limit state criterion to determine the load carrying capacity of the structure. The critical buckling stress for the mode under consideration will be obtained from the same analysis. If required this can then be used with reference to design codes to calculate the value of the slenderness parameter for lateral torsional buckling λ_{LT} and the limiting compressive stress σ_{lc}.

- This analysis type will provide both local and global buckling modes. However, engineering judgement is necessary to determine which buckling mode is critical in order to select the appropriate buckling load factor.

- If a load factor very close to unity is derived, it would normally be prudent to carry out further investigation, removing some of the assumptions in the initial analysis and perhaps carrying out a nonlinear analysis.

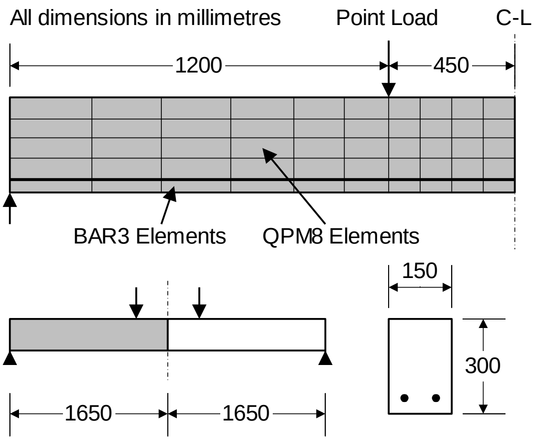
Nonlinear Analysis of a Concrete Beam

For software product:	Any Plus version. Bridge Plus version for the discussion.
With product options:	Nonlinear.

Description

A nonlinear plane stress analysis is to be carried out on a model of a reinforced concrete beam.

The reinforcement is provided in the lower face of the beam and has a total cross-sectional area of 400 mm^2. The superposition of nodal degrees of freedom assumes that the concrete and reinforcement are perfectly bonded. It is assumed that the self-weight of the beam is negligible compared with the applied load and that the effects of any shear reinforcement can be ignored.



Due to the symmetrical nature of the problem, only the left-hand span of the beam is modelled. The beam is simply supported at the left-hand end with a symmetry support at the right-hand axis of symmetry. A concentrated vertical load is applied to the top of the beam 1200mm from the left-hand end. The concrete section is represented by plane stress (QPM8) elements, and the reinforcement bars are represented by bar (BAR3) elements. A nonlinear concrete cracking material model will be applied to the plane stress elements and a von Mises plastic material will be applied to the reinforcement bars.

Units of N, mm, t, s, C are used throughout.

Objectives

The behaviour of the beam under cracking/yielding is to be examined by producing the following:

- **A Deformed Mesh Plot** showing the final deformed shape.
- **A Load Displacement Graph** for the top node on the axis of symmetry of the beam.
- **Stress contour plot** showing the stress distribution in the beam.
- **Crack pattern plot** showing the crack patterns produced.
- **Animation of stresses and crack patterns** for selected load increments.
- **A graph of variation in stress** through selected slice sections through the beam.

Keywords

2D, Plane Stress, Bar Elements, Nonlinear Concrete Model, Element Selection, Concrete Cracking, Steel Reinforcement, Groups, Crack Patterns, Animation, Graphing, Load Displacement Curve, Slice Sections

Associated Files

- **beam_nl_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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

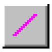
- Enter the file name as **beam_nl**
- Use the **Default** working folder.

- Enter the title as **Nonlinear Concrete Beam**
- Set the units as **N, mm, t, s, C**
- Leave the timescale units as **Seconds**
- Ensure an analysis type of **Structural** is set.
- Select the model startup template **Standard**
- Select the **Vertical Y Axis** option.
- Click the **OK** button.

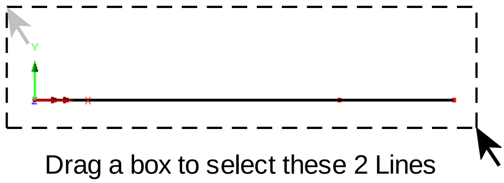
Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Geometry

Geometry
Line >
Coordinates...

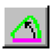


Enter coordinates of (**0, 0**), (**1200, 0**) and (**1650, 0**) to define two Lines representing the bottom of the left hand span of the beam. Click the **OK** button to finish.



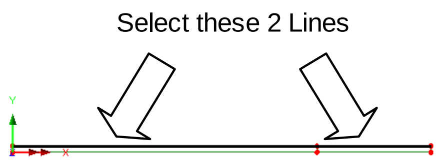
- Select both Lines just drawn by dragging a selection box around them.

Geometry
Surface >
By Sweeping...



Enter a translation value of **25** in the **Y** direction to create the Surface which represents the concrete cover from the face of the beam to the centreline of the reinforcement.

- Click the **OK** button.
- Select the upper Lines of both of the Surfaces just drawn as shown.

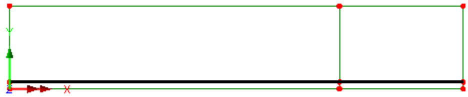


Geometry
Surface >
By Sweeping...

Enter a translation value of **275** in the **Y** direction to create the Surface which represents the extent of the concrete above the centreline of the reinforcement.

- Click the **OK** button.

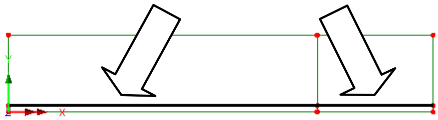
The model should appear as shown.



Defining Groups

To simplify the assignment of model attributes certain model features will be grouped together to allow selection by name in the Treeview as opposed to selection by cursor in the graphics window. The 2 Lines representing the reinforcement bars are to be grouped together:

- Ensure the 2 Lines are still selected as shown.



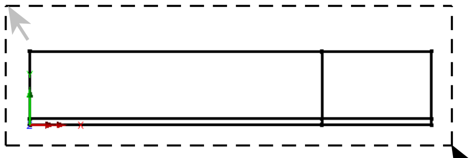
Geometry
Group >
New Group

Enter **Bars** for the group name.

- Click the **OK** button to complete creation of the group.

The Surfaces representing the concrete are to be grouped together.

- Holding-down the **S** key, (and noting that the cursor changes to show the type of feature that will be selected) drag a box around the whole model to select only the Surfaces defining the concrete.


Drag a box to select only Surfaces

Geometry
Group >
New Group

Enter **Concrete** for the group name. Click the **OK** button to complete creation of the group.

Note. In this example, model attributes will be defined but not assigned to the model straight away. They will be assigned to the model later by making use of the Groups facility.

Defining the Mesh Reinforcement Bars

Separate mesh datasets need to be defined for the reinforcement bars and the concrete. For the reinforcement bars a uniform mesh is to be used to the right of the applied load and a graded mesh is to be used on the horizontal lines to the left of the applied load.

The reinforcement bars will be modelled using Line meshes.

Attributes
Mesh >
Line…

- Set Generic element type to **Bar**, Number of dimensions to **2 dimensional** and Interpolation order to **Quadratic**

- Ensure the **Number of divisions** is set to **4**
- Enter the attribute name as **Bar Elements - Divs=4**
- Click the **Apply** button to create the attribute in the Treeview and leave the dialog visible to allow additional datasets to be defined.
- Change the **Number of divisions** to **6** and click the **Spacing** button.
- Select a **Uniform transition** ratio of first to last element of **2** and click **OK**
- Change the attribute name to **Bar Elements - Divs=6 graded**
- Click the **OK** button to finish to add the attribute to the Treeview.

Defining the Mesh Concrete

The concrete will be modelled using a Surface mesh with Line mesh divisions to control the mesh density..

Attributes
Mesh >
Surface…

- Select **Plane stress, Quadrilateral, Quadratic** elements.
- Enter the attribute name as **Plane Stress - Concrete**
- Click the **OK** button to add the attribute to the Treeview.
- The default mesh density of 4 divisions per line is sufficient for the Surface to the right of the applied load. A graded line mesh will be created for use on the Surface to the left of the applied load
- In the Treeview double click the Line mesh attribute name **Bar Elements - Divs=6 graded**

The Line mesh properties dialog will appear.

- Change the attribute type to **None**
- Change the attribute name to **Divisions=6 graded**
- Click the **OK** button to add the attribute to the Treeview.

Defining the Geometric Properties

Attributes
Geometric >
Line…

- Select **Bar/Link** from the drop down list and enter a value of **400** for the total cross sectional area of the reinforcement.
- Enter the attribute name as **Steel Area** and click the **OK** button to add the attribute to the Treeview.

Attributes
Geometric >
Surface…

- Enter a value of **150** for the thickness. Leave the eccentricity blank.
- Enter the attribute name as **Beam Thickness** and click the **OK** button to add the attribute to the Treeview.

Defining the Material Properties

Nonlinear steel properties will be defined for the reinforcing bar elements.

Attributes
Material >
Isotropic…

- Enter Young's modulus as **210e3** and Poisson's ratio as **0.3** and leave the mass density field blank.
- Click the **Plastic** option and enter an **Initial uniaxial yield stress** of **300**
- Select the **Hardening** option, click the **Hardening gradient** button and enter a hardening **Slope** value of **2121** with a **Plastic strain** of **1**
- Enter the attribute name as **Nonlinear Steel**
- Click the **OK** button to add the attribute to the Treeview.

Nonlinear concrete material properties will be defined for the Surface elements representing the concrete.

Attributes
Material >
Isotropic…

- Enter a Young's modulus of **42000**, a Poisson's ratio of **0.2** and leave the mass density field blank.
- Click the **Plastic** option and from the drop-down list select **Concrete**
- For the plastic model Type choose **Smoothed Multi Crack (model 102)**
- Enter a **Uniaxial compressive strength** value of **31.58**
- Enter a **Uniaxial tensile strength** value of **3.158**
- Press the **Advanced...** button
- In the Advanced Concrete Properties dialog change the **Strain at end of softening curve** to be **0.003** and press **OK** to return.
- Enter the attribute name as **Nonlinear Concrete**
- Click the **OK** button to add the attribute to the Treeview.

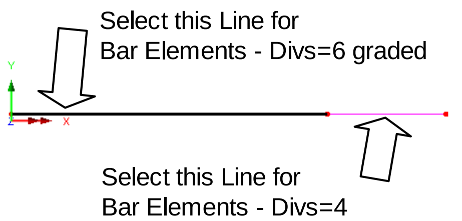
Assigning Attributes to the Bars

The various Line and Surface mesh, geometric and material attributes defined previously will now be assigned to the model using the groups that have been defined.

- In the Treeview right-click the group name **Bars**. Select the **Set as Only Visible** option. The features in the group will be displayed.

- Select the left hand Line of the two Lines representing the bars.

- Drag and drop the Line mesh attribute **Bar Elements - Divs=6 graded** from the Treeview onto the selected Line.



- Select the right hand Line of the two Lines representing the bars.

- Drag and drop the Line mesh attribute **Bar Elements - Divs=4** from the Treeview onto the selected Line.

- In the Treeview double-click on the **Mesh** entry and select **Show nodes**.

The Line mesh divisions will be seen defined with the spacing as shown.

- Select both Lines.

- Drag and drop the geometric attribute **Steel Area** from the Treeview onto the selected features.

- Drag and drop the material attribute **Nonlinear Steel** from the Treeview onto the selected features.

Note. The diagrams in this example show element nodes. To see these at any time you can go to the Treeview and double-click the **Mesh** layer. On the Mesh tab select **Show nodes** and click the **Close** button.

Assigning Attributes to the Concrete

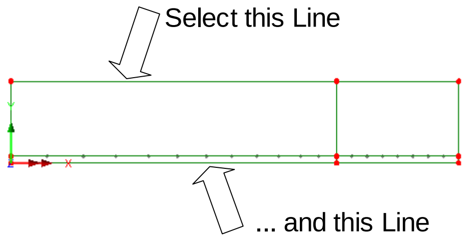
In the Treeview right-click the group name **Concrete**. Select the **Set as Only Visible** option.

The Lines in the Bars group will be removed from the display and the Concrete group will be displayed.

- Select the left-top and left-bottom Lines as shown.

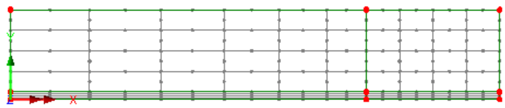
- From the Treeview drag and drop the Line mesh attribute **Divisions=6 graded** onto the selected features.

- Select the whole model using the **Ctrl** and **A** keys together.



- Drag and drop the Surface mesh attribute **Plane Stress - Concrete** from the Treeview onto the selected features.

A graded mesh will be drawn on the left-hand Surface and a uniform mesh will be drawn on the right-hand Surface.



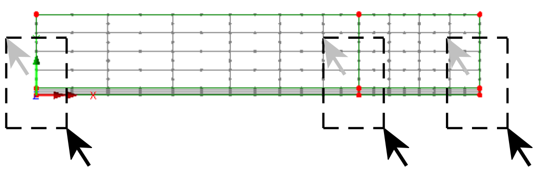
- Drag and drop the geometry attribute **Beam Thickness** from the Treeview onto the selected features.

 Select the fleshing on/off button to turn-off the geometric visualisation. If at any time during the example you wish to visualise the geometry select this button.

- With the whole model still selected, drag and drop the material attribute **Nonlinear Concrete** from the Treeview onto the selected features. Ensure the **Assign to surfaces** option is selected and click **OK**

The mesh on the Lines representing the cover to the centreline of the reinforcement needs to be altered. This is because they currently have a default Line mesh of 4 divisions per line when only 1 division per line is required.

- Drag boxes to select the 3 Lines as shown. (Remember to hold the **Shift** key down after the first line is selected so the other lines are added to the selection)



Drag 3 boxes to select these 3 Lines

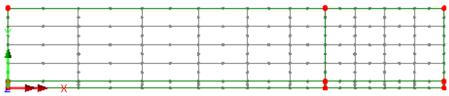
- Drag and drop the Line mesh attribute **Divisions=1** from the Treeview onto the selected Lines.

The mesh will be redisplayed with the revised mesh pattern.

Making all groups visible

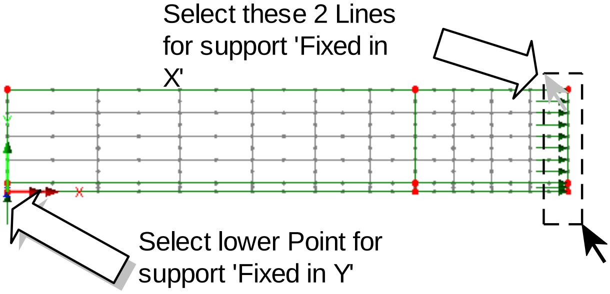
- From the Treeview right-click the group heading name **beam_nl.mdl**. Select the **Set as Only Visible** option. Click **Yes** to act on sub groups as well.

All features in the model will now be displayed as shown.



Supports

When using the Standard template, LUSAS provides the more common types of support by default. These can be seen in the Treeview. The beam is to be simply supported in the Y direction at the left-hand end and a horizontal restraint in the X direction is required to satisfy the symmetry requirements at mid-span.



- Select the lowest Point at the left hand end of the model as shown.
- Drag and drop the support attribute **Fixed in Y** from the Treeview onto the selected Point. Ensure the **Assign to points** and **All analysis loadcases** options are selected and click **OK**
- Drag a box around the 2 Lines at the right hand end of the model as shown.
- Drag and drop the support attribute **Fixed in X** from the Treeview onto the selected Lines. Ensure the **Assign to lines** and **All analysis loadcases** options are selected and click **OK**

Loading

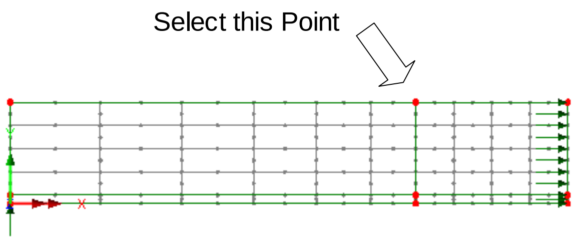
A single concentrated load is to be applied to the Point at the top of the beam. A unit load will be applied and the load factor in the nonlinear control will be used to control the magnitude of loading.

Attributes
Loading...

- With the **Concentrated** option selected click **Next**
- Enter a loading value of **-1** in the component **Concentrated load in Y Dir**
- Enter the attribute name as **Point Load** and click **Finish**

- Select the Point on the top of the beam as shown.

- Drag and drop the loading dataset **Point Load** from the  Treeview onto the selected Point.



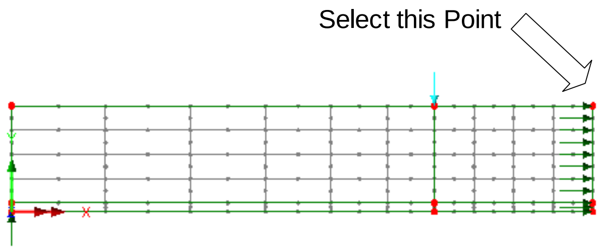
- Ensure the **Assign to points** option is set and click **OK** to assign the load to **Analysis 1**, **Loadcase 1**.

Nonlinear Control

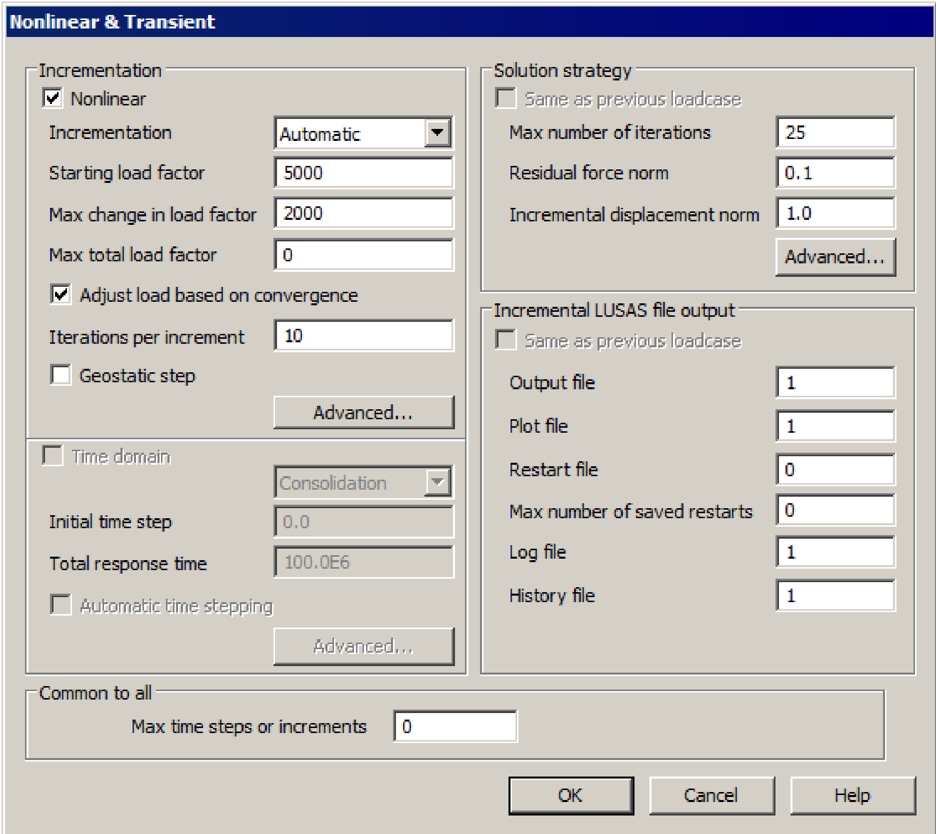
Nonlinear analysis control properties are defined as properties of a loadcase. The nonlinear analysis is to be terminated when the beam deflection at mid span reaches a limiting value.

- Select the point shown.

- In the  Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Nonlinear & Transient** from the **Controls** menu.



The Nonlinear & Transient dialog will appear:

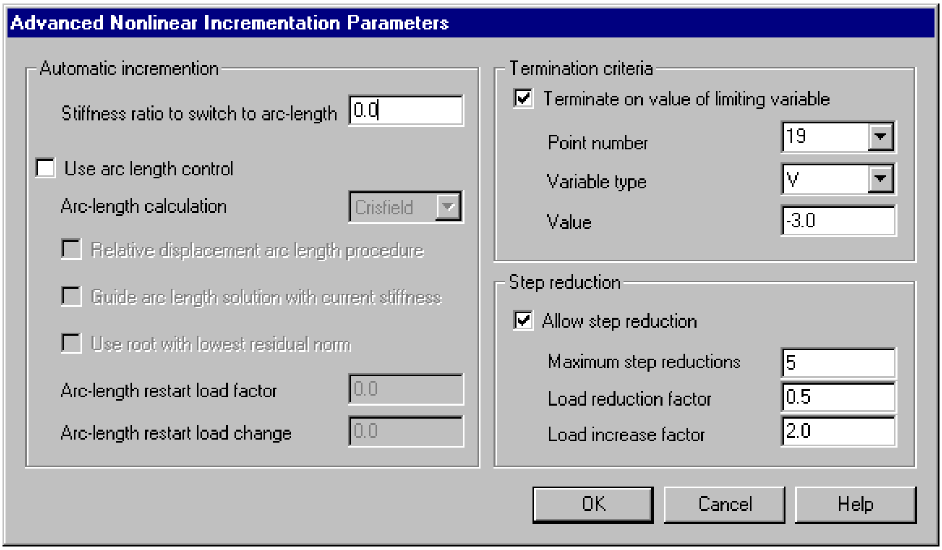


Select the **Nonlinear** option and set Incrementation to **Automatic**

- The initial load to be applied is the actual load applied to the model multiplied by the starting load factor. Set the **Starting load factor** to **5000**
- Enter the **Max change in load factor** as **2000** to restrict the second and subsequent load increment sizes to ensure sufficient points are obtained to observe the load deflection behaviour of the beam.
- Change the **Max total load factor** to **0** as the solution is to be terminated on the limiting displacement at mid span.
- Change the number of desired **Iterations per increment** to **10**

Note. If the number of iterations on the previous increment is less than the desired number the next load increment will be increased (up to the maximum change in load increment) while if the number of iterations is less than the desired number the next load increment will be reduced.

- In the Solution strategy section of the dialog, ensure the **Maximum number of iterations** is set to **25**
- Leave the **Residual force norm** as **0.1** and the **Incremental displacement norm** to **1** so convergence of the solution at each load increment will be achieved when the out of balance forces are as less than 0.1% of the reactions and the iterative change in displacements is less than 1% of the displacements for that load increment.
- Select the **Advanced** button in the Incrementation section of the dialog.



- Ensure that the **Stiffness ratio to switch to arc length** value is set to **0.0**
- Select the **Terminate on value of limiting variable** option.
- The selected point number (this may differ depending on how the model was created) will appear in the Point number drop down list.
- Set the **Variable type** to **V** to monitor the deflection at the selected point in the Y direction.
- Enter a value of **-3** so the analysis is terminated when the central deflection reaches this value.
- In the Step reduction section ensure the **Allow step reduction** option is selected.
- Click **OK** to return to the Nonlinear & Transient dialog.

- Click **OK** again to set the loadcase properties.

One additional setting is required for this analysis to ensure no element mechanisms are induced as the material yields.

File
Model Properties…

- Select the **Solution** tab.
- Click on the **Element Options** button and select the **Fine integration for stiffness and mass** option.
- Click the **OK** button to return the Model Properties dialog.
- Click the **OK** button to finish.

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.

File
Save

Save the model file.

Running the Analysis

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

During the analysis 2 files will be created:

- **beam_nl.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, and so on. Always check the LUSAS output file for error messages.
- **beam_nl.mys** this is the LUSAS results database which will be used for results processing.

Note. Warnings will be written to the text output window during this analysis. Warnings are not to be confused with errors which normally stop an analysis from being run. The warnings are written to advise you of modelling-related issues that are found during the analysis and of the steps LUSAS has taken (if any) to allow the analysis to continue.

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. Select **No** to not view the output file. 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.

- **beam_nl_modelling.vbs** carries out the modelling of the example.

File
New…

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 **beam_nl**

File
Script >
Run Script...

To recreate the model, select the file **beam_nl_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

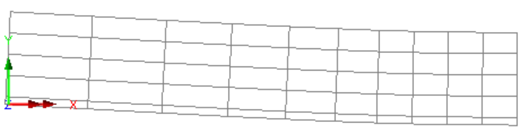
Analysis loadcase results are present in the Treeview, and the loadcase results for load increment 1 are set to be active by default.

Changing the Active Results Loadcase

- In the Treeview right-click on the last load increment **Increment 6 Load Factor = 15000** and select the **Set Active** option.

Deformed Shape

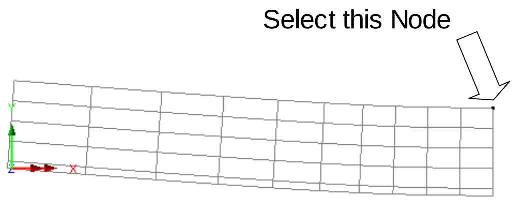
- In the Treeview turn off the **Geometry**, **Attributes** and **Mesh** layers by right clicking on each entry and



selecting the **On/Off** option.

Creating a Load versus Displacement Graph

A graph of displacement at mid-span is to be plotted against the applied load. To do this a node on the line of symmetry is selected:



- With the **Deformed mesh** layer visible, select the top node on the axis of symmetry as shown.

Using the Graph Wizard

The graph wizard provides a step-by-step means of selecting which results are to be plotted on the X and Y axes of the graph. The X axis is always defined first.

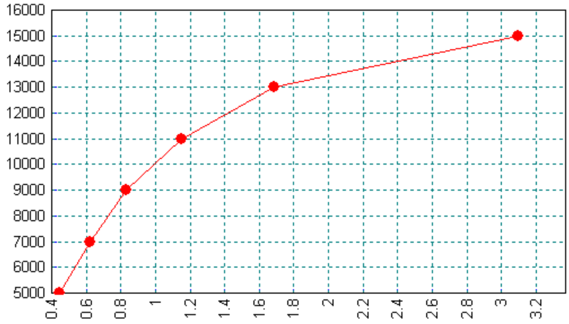
Utilities
Graph Wizard...

- Ensure the **Time history** option is selected and click the **Next** button.
- Ensure the **Nodal** results is selected and click **Next**
- Select entity **Displacement** for component resultant displacement **RSLT**
- The node number selected earlier will be displayed in the drop-down list.
- Click the **Next** button.

The Y axis results to be graphed are now defined.

- Select the **Named** option and click **Next**
- Select **Total Load Factor** from the drop down list.
- Click the **Next** button.

- Leave all title information blank and click the **Finish** button to display the load deformation graph.



Note. Graphs can be modified using the right hand mouse button in the graph window and selecting the **Edit Graph Properties** option.

Close the graph window.

Use the maximise button to increase the size of the graphics window.

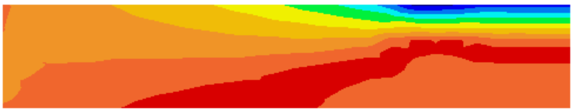
- In the Treeview turn off the **Deformed mesh** layer.

Maximum Principal Stress Contour Plots

- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Contours** option to add the contours layer to the Treeview.

The properties dialog will be displayed.

- Select entity **Stress - Plane Stress** for component of stress in the x direction, **SX**



- In the **Contour Display** tab deselect **Deform**.
- Click the **OK** button to display contours of stresses for the final load increment.

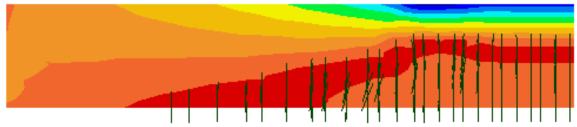
Viewing Crack Patterns

- 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.

The properties dialog will be displayed.

- Select **Stress - Plane Stress** contour results of type **Crack/crush**
- Select **Values Display** tab and press the **Failure details…** button to change the pen colour that is used to plot negative yield to **Black.**

- Click the **OK** button to display the cracking pattern for final load increment superimposed onto the stress contours.



Animating the Results

As an alternative to viewing results individually for each loadcase, the change of stress due to the increasing load increments can be animated instead. To ensure consistent contour values throughout the animation the interval of the range of contours is to be specified.

- In the Treeview double-click on the **Contours** layer.

The contour properties will be displayed.

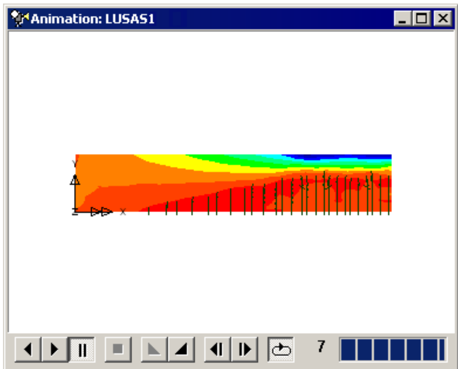
- Select the **Contour Range** tab and click the **Interval** option and set the contour interval as **1**
- Click the **Maximum** button and set the maximum value as **3**
- Click the **Minimum** button and set the minimum value as **-16**
- Click the **Set as global range** and **Use global range** options.
- Click the **OK** button to redisplay the stress contours using the new contour range.

Using the Animation Wizard

Utilities
Animation Wizard...

- Select the **Load History** option and click the **Next** button.
- Select the **All loadcases** option and select the **Finish** button to create the animation sequence.

Note. The buttons at the bottom of the window may be used to slow-down, speed-up, pause, step through frame by frame, or stop the animation.



Saving Animations

Animations may be saved for replay in Windows animation players.

- Ensure the animation window is the active window.

File
Save As AVI...

- Browse to your projects folder and enter **beam_nl** for the animation file name. An **.avi** file extension is automatically appended to the file name. Click **Save**

Note. Animations can be compressed to save disk space by changing the **Compression Quality**. Reducing the quality will, however, result in a lower definition image. It is also possible to reduce file size by setting smaller dimensions on the first page of the **Animation Wizard**.

Close the animation window, choosing not to save changes.

Enlarge the model window to a full size view.

Creating a slice section of results

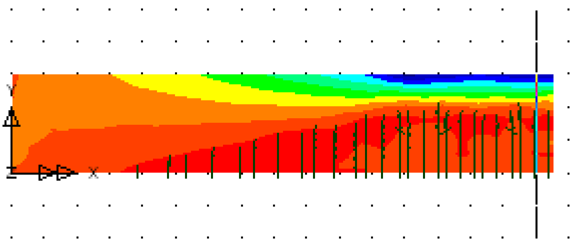
In this example a graph is to be plotted of the variation in stress through the specified section of the beam. The X axis values of distance are defined by the section slice. The Y axis results are specified from the graph wizard dialog.

Utilities
Graph Through 2D

- Ensure the **Snap to grid** option is selected and a grid size of **100** is specified.
- Click the **OK** button.

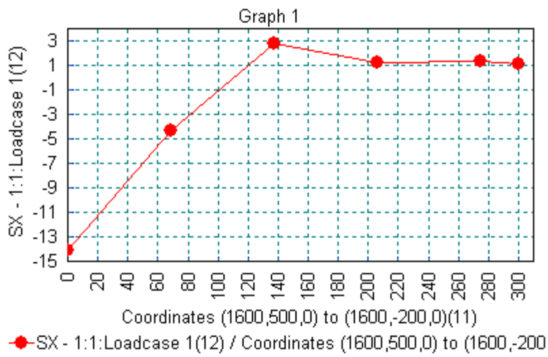
Note. The snap to grid dialog will only appear if the model is viewed in the XY plane. If necessary, return the model to the default starting view by clicking Z: N/A on the status bar at the bottom of the graphics window.

- Using the screen ruler as a guide, click and drag the cursor as shown to define the location of a section slice through the beam at a distance of **1600** from the left-hand end.



- Select **Stress - Plane Stress** results for Stress **SX** and click the **Next** button.
- Leave all title information blank.

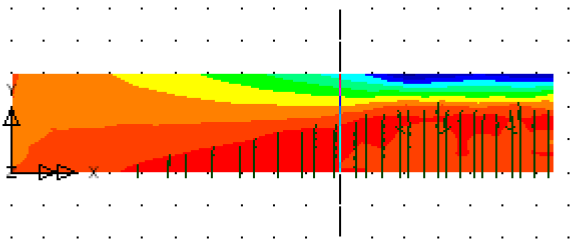
- Click the **Finish** button to create the graph of stress through the section of beam.



Adding Additional Results to a Graph

Window
1 LUSAS View beam_nl.mdl Window 1

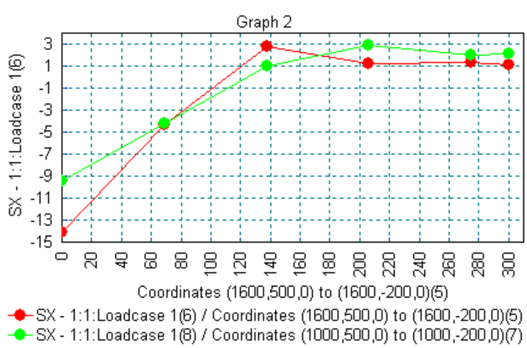
- Re-select the window containing the results contours. The cursor will still be in section slice mode.
- Using the screen rulers as a guide, click and drag the cursor as shown to define the location of a section slice through the beam at a distance of **1000** from the left-hand end.



- Select **Stress - Plane Stress** results for Stress **SX** and click the **Next** button.
- Click the **Add to existing graph** option.
- Click the **Finish** button to add the results for the second slice section to the existing graph.

Window
2 Graph1:Graph 1

- Re-select the existing graph to see the overlaid data



 Close the graph window.

Plotting Crack Width Contours and Values

Crack width values as calculated by the use of the Smoothed Multi Crack Concrete Model can be plotted in contour and value form:

- Double-click the **Contours** layer name in the Treeview.

The properties dialog will be displayed.

- Select entity **Plastic strain - Plane Stress** for component of stress in the x direction, **CWMax**
- Select the **Contour Range** tab and **deselect Use global range**
- Deselect the **Maximum** button
- Deselect the **Minimum** button
- Set the contour **Interval** to be **0.005**
- Click the **OK** button to redisplay the stress contours using the new contour range.
- Press **Yes** to confirm the Values layer will be adjusted to match.

Note. Contouring of crack width values effectively shows the elements affected by cracking and identifies those elements where the maximum crack widths occur. This is done using unaveraged, or unsmoothed values at nodes. To see the actual values of crack width within elements (at the locations where the Crack/crush patterns were plotted earlier in the example), values for gauss points need to be plotted.

To see the crack width values:

- Double click the **Values** layer name in the Treeview.
- With the **Values Results** tab active, select **At Gauss Points**.
- Select the **Values Display** tab and change the **Font angle** to **45** degrees.
- Click **Close** to update the display and show the actual crack width value calculated at gauss points within the element.

This completes the example

Discussion

Plotting Crack Widths to EN1992-1-1

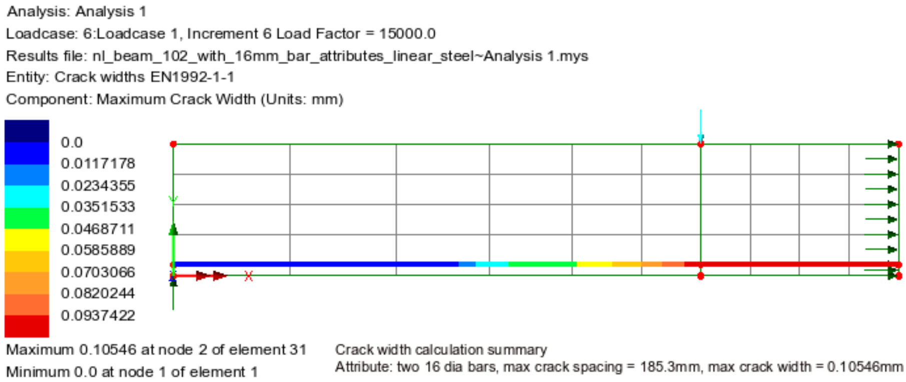
In this example nonlinear steel properties were defined for the reinforcing bar elements. Reinforcement bar attributes (accessed from the **Attributes > Geometric > Bar reinforcement** menu item) may alternatively be used to model the steel reinforcement in reinforced concrete, but only if a linear steel material model is used to represent the steel reinforcement and the Crack Widths calculation utility (accessed from the **Bridge > Crack Widths to EN1992-1-1** menu item) is employed.

When this is done, contours and values of crack widths can be plotted in accordance with EN 1992-1-1:2004 Eurocode 2 by selecting the Entity **Crack Widths EN 1992-1-1** and the Component **Maximum Crack Width** on the Contour Properties dialog. Note that a linear steel material model (rather than a nonlinear one) is required for the steel reinforcement because of the steel strains used in the calculations.

Reinforcement bar attributes are similar to geometric line attributes but need to be assigned to lines meshed with bar elements for a chosen analysis to enable a crack width calculation to be carried out. Bar attributes may be defined for a single reinforcing bar (a Discrete bar), as would be used in a 3D model, or be defined with suitable properties to represent a bundle or simplified arrangement of bars (Equivalent bar) as would be used typically in a 2D plane strain analysis.

When modelled in the way described for two 16mm diameter steel bars, the crack width contours can be plotted along the bar elements (actually on the surface of the fleshed bar section) corresponding to the steel strains used in the calculation, and not on any concrete face or surface in the model. Contours and values are plotted for an active loadcase (or combination) and are re-computed if a different loadcase is set active. This visualisation method is used because the approach used in EN 1991-1-1:2009 to calculate crack widths is generally unclear as to where the crack width calculation applies.

A typical crack width contour plot for a bar reinforcement attribute defined for the steel in this example is shown below. Unaveraged or gauss point values of crack widths could be additionally added using a Values layer.


Analysis: Analysis 1
Loadcase: 6:Loadcase 1, Increment 6 Load Factor = 15000.0
Results file: nl_beam_102_with_16mm_bar_attributes_linear_steel~Analysis 1.mys
Entity: Crack widths EN1992-1-1
Component: Maximum Crack Width (Units: mm)
0.0
0.0117178
0.0234355
0.0351533
0.0468711
0.0585889
0.0703066
0.0820244
0.0937422
Maximum 0.10546 at node 2 of element 31
Minimum 0.0 at node 1 of element 1
Crack width calculation summary
Attribute: two 16 dia bars, max crack spacing = 185.3mm, max crack width = 0.10546mm

Staged Construction of a Concrete Tower with Creep

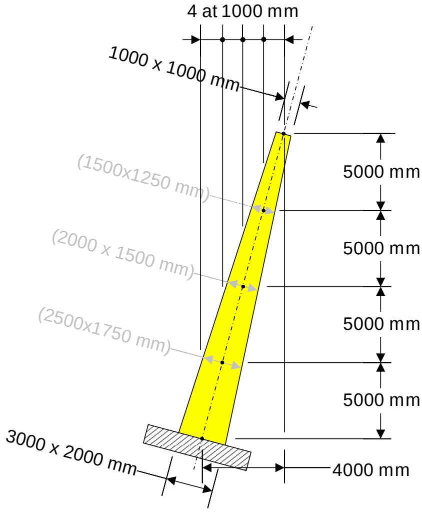
For software product(s):	Any Plus version.
With product option(s):	Nonlinear and Dynamic.

Description

A 20m high concrete tower of varying cross-section is to be constructed in four stages, each 5 metres high. The formwork is struck 14 days after each stage is cast and the casting sequence between each stage is 60 days.

Simplified geometry has been used to allow the example to concentrate on the creation of staged construction loadcases, the definition of the concrete creep properties and running a creep analysis.

Concrete creep is carried out according to CEB-FIP 1990 code. The modelling units used are N, mm, t, s, C throughout.



Tower geometry and construction stages

Objectives

The required output from the analysis consists of:

- A comparison of the moments at the base of the tower after each construction stage.
- Maximum displacement of the tower after each construction stage and the change in the maximum displacement value due to long-term creep.

Keywords

Birth, Death, Staged Construction, Activate, Deactivate, Long-Term, Creep, CEB-FIP Concrete Model, Age, Casting, Reference Path

Associated Files

- **concrete_tower_modelling.vbs** carries out the modelling of the example.

Discussion

Structural concrete is unique among the major construction materials in that it undergoes time-dependent changes in its properties. Generally, the concrete's strength will increase over time, but this increase in strength is accompanied by a certain degree of shrinkage and creep. Particularly in large structures where several concrete pours may take place over a period of months or years, the time-dependent nature of concrete's behaviour can have a significant effect on the behaviour of the structure and these effects must be considered in analysis.

Modelling

Running LUSAS Modeller

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

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 the file name as **concrete_tower**
- Use the **Default** working folder.
- Enter the title as **Concrete creep example to model code CEB-FIP 1990**
- Select units of **N,mm,t,s,C**
- Select timescale units of **Days**
- Ensure the **Structural** analysis type is selected.
- Leave the startup template as **None**
- Select the **Vertical Y Axis** option.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Geometry

Geometry >
Line >
Coordinates…

Enter coordinates of (**0, 0**), and (**1000, 5000**), to define the first section of the tower.

- Select the line just drawn.

Geometry
Line >
Copy…

Enter a translation in the **X** direction of **1000** and in the **Y** direction of **5000**

- Enter the number of copies required as **3** and click **OK** to create the next three sections of the tower.

Defining and Assigning Mesh Attributes

Attributes
Mesh >
Line…

- The tower is to be modelled with **Thick Beam 3D Quadratic** elements (BMI31 elements) with **6** divisions. These elements can correctly capture the effects of the variation in section shape.
- Enter the attribute name as **BMI31 6Div** and click **OK**.
- Select all lines on the model and drag and drop the **BMI31 6Div** mesh from the Treeview onto the selected features.
- Click **OK** to assign a default element orientation using a beta angle of 0.

Caution. If constant cross-section beam elements were used to model the variation in section shape instead of the quadratic elements used, care should be taken to ensure that the line mesh was sufficiently fine to correctly capture the effects of the variation in section shape.

Geometric Properties

There are two ways that the tapering shape of the tower could be modelled. The first option would be to create a tapering beam geometric property for each individual line. This would require different geometric attributes to be created for each of the four lines.

A more efficient method is to use a **multi-varying section** geometric attribute which allows tapering along a **reference path** rather than along individual lines. By using this type of geometric attribute, only a single attribute is required to describe the taper of the tower from its base to tip.

To define the tower cross-section at the base

Utilities
Section Property Calculator >
Rectangular Sections > Solid

- On the Rectangular Solid Section Property dialog enter D=**3000** and B=**2000**.

A name of RSS D=3000 B=2000 will be automatically created.

- Ensure **Add to local library** is selected and click **Apply** to calculate the section properties, add the section to the local library and define another cross-section.

- Change the values to read D=**1000** and B=**1000**.

A name of RSS D=1000 B=1000 will be automatically created.

- Ensure **Add to local library** is selected and click **OK** to calculate the section properties and add the section to the local library.

All required section properties have now been defined and saved in the local user library. They will be used to create multiple varying section line attributes which will be automatically added to the treeview when the attribute is created.

Creating a reference path

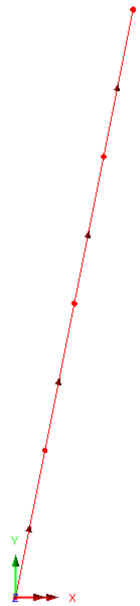
For this example a reference path will be defined along the lines representing the tower. This will be used when defining and assigning the varying section properties of the tower.

- Box-select all lines representing the tower

Utilities
Reference Path

- Leave the name set as **Path 1** and click **OK.**

The direction of the path is shown by arrows now overlaid on each line.



- **Note.** Model geometry can be used to generate a reference path but no subsequent connection between the initial parent geometry and reference path data exists.

- The visualisation of a reference path can be controlled via the Utilities layer. If desired reference path visualisation can be turned off using a context menu option for its entry in the Utilities layer of the Treeview.

Defining the multiple varying section line property

A multiple varying geometric line attribute needs to be defined. This will make use of the previously defined rectangular solid sections.

Define sections

Attributes
Geometric >
Multiple Varying Section…

- On the Multiple Varying Section dialog click in the Section cell that currently reads 914x305x289kg UB and the launch dialog button will appear to allow a different pre-defined section to be chosen from a different section library.

- On the Enter Section dialog, click on the drop-down list button to change the selection from UK Sections to **User Sections**

- Ensure a **Local** library is being accessed.

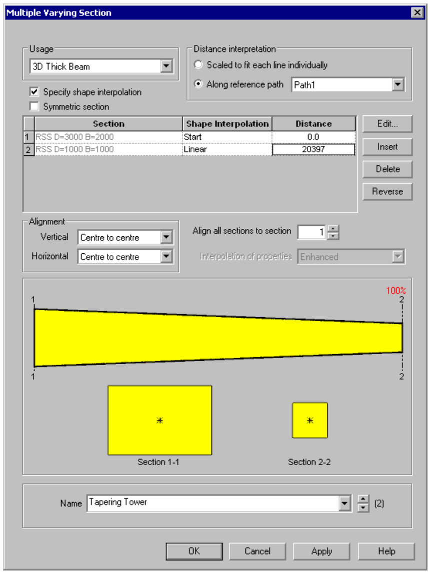
- Select **RSS D=3000 B=2000** from the list of local library items and then click **OK** to add the section to the first row of the table on the Multiple Varying Section dialog.

- On the Multiple Varying Section dialog press the **TAB** key to move between cells and create a new row beneath the existing row.

- In the new Section cell (row 2) click on the launch dialog button again. The drop-down list should still be showing **User Sections** for a **Local** library. Browse and select **RSS D=1000 B=1000** from the drop-down list and click **OK** to add the section to the table.

Note. Before a visualisation of the longitudinal and horizontal alignment of the cross-section shapes can be displayed on the dialog, the shape interpolation method and distance values need to be defined.

Define shape interpolation method



To specify a shape interpolation method for each defined section:

- The **RSS D=3000 B=2000** section as the first in the table is set by default to **Start**
- For the **RSS D=1000 B=1000** section entry click on the drop-down list button  in the Shape Interpolation cell and select **Linear**

Define distance values

- Change the distance interpretation to be **Along reference path** and ensure **Path1** is selected.

Lastly a distance value at which the section will apply should be specified.

- For the **RSS D=1000 B=1000** section click in the Distance cell and enter **20397** (the true length of the tower).

Note. When specifying a distance value at which a defined section will apply it is important to round-up to the next whole number (20397 in this case). If 20396 has been specified the final section would not be applied to the topmost line on assignment, and therefore not shown when fleshed.

Check alignment settings

The vertical and horizontal alignment of a pair or series of sections is controlled by the Alignment options. All sections are aligned with respect to a chosen section.

- Select **Centre to centre** for both alignment options and set the section to align to as **1**.

Name the attribute

- Enter a name of **Tapering Tower** and click **OK** to add the multiple varying section line attribute to the  Treeview. Note that “(Varying – 2 sections)” is automatically appended to the stated name.

Assigning the varying geometric line property

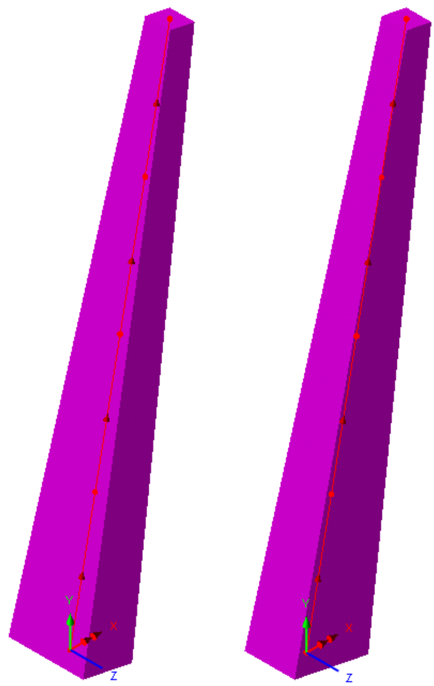
- Box-select all lines representing the tower and drag and drop the **Tapering Tower (Varying – 2 sections)** geometric line attribute from the Treeview onto it.

Check and correct the orientation of sections used

Select the isometric view.

In this example, the 3m x 2m section at the base of the tower has not been oriented correctly for the line axes that are in use. The cross sections at the bottom and the top of the tower both need rotating around their centroids. This is done as follows:

- In the Treeview double-click on the **Tapering Tower (Varying – 2 sections)** attribute name
- On the **Multiple Varying Sections** dialog click in the Section cell for row 1 and click on the launch dialog button to access the detail for **RSS D=3000 B=2000** and change the Rotation about centroid to be **90** degrees and click **OK**.
- Click in the Section cell for row 2 and click on the launch dialog button to access the detail for **RSS D=1000 B=1000** and change the Rotation about centroid to be **90** degrees and click **OK**.



Incorrect Correct

Material Properties

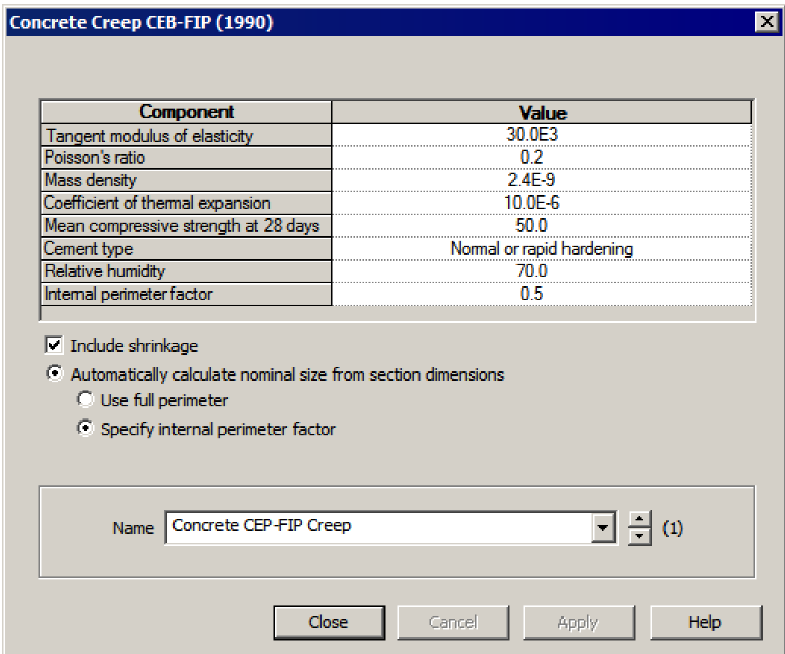
Attributes
Material >
Specialised >
Concrete
Creep
CEB/FIP...

In addition to the common linear isotropic material properties (**Young's modulus at 28 days**, **Poisson's ratio, mass density**, and **coefficient of thermal expansion**), the **mean compressive strength at 28 days**, **relative humidity** and **cement type** need to be specified.

In this example, leave the **Include shrinkage** box ticked to include the effects of shrinkage as well as creep in the material behaviour.

The CEB-FIP creep calculations use a parameter called ‘nominal size’. This is calculated as 2A/U where A is the area of the cross-section and U is the length of the cross-section perimeter. In LUSAS you have the option of manually inputting the nominal size by activating the **nominal size** radio button, or allowing LUSAS to automatically calculate the nominal size by activating the **specify the beam section internal perimeter factor** radio button. The internal perimeter factor is a factor applied to the perimeter length of any internal voids when calculating the section perimeter U. In this case the tower cross-section has no voids so the value has no effect.

- Ensure the values shown on the accompanying dialog are entered, enter a material name of **Concrete CEB-FIP Creep** and click OK to create the material attribute.



- Select the whole model (**CTRL + A)**. From the  treeview drag and drop the **Concrete CEP-FIP Creep** attribute onto the model to assign the new creep material.

Supports

Attributes
Support…

- A fully fixed support is required therefore all degrees of **translation** and **rotation** in the **X**, **Y** and **Z** axes must be set as **Fixed**. Enter an attribute name of **Fully Fixed** and click **OK**
- Select the point at the base of the tower and from the treeview drag and drop the support attribute **Fully Fixed** onto the selected feature, ensure the **All analysis loadcases** option is selected and click **OK**

Loading

Only self weight will be applied in this analysis. The LUSAS auto-gravity facility can be used to quickly apply gravity loading in all loadcases. This will be done later on after creation of all the required loadcases.

Age

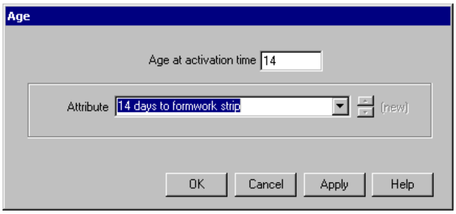
The age of a section within a creep analysis is important because the material properties change with time. The age of a section is specified in days between the casting of the section and when it is first activated in the analysis.

Note. The timescale units of days for the model were set on the New Model dialog at the beginning of this example. Timescale units are distinct from the fundamental model units (i.e. N, mm, t, s, C) and are provided as a convenience for long-duration analyses like creep and consolidation.

Creating and Assigning an Age Attribute

Attributes
Age…

It is assumed that all sections in the tower will be cast and the formwork struck **14** days after the component was first activated. (If this was not the case separate age attributes would be created and assigned to each section as required)



- Name the attribute as **14 days to formwork strip** and click **OK**
- Assign the age attribute to all lines in the model.

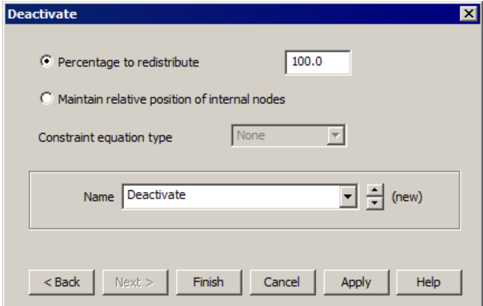
Note. If a construction includes precast sections, the age of a component might be 80 days, while the component may be activated at day 28 in the analysis for example.

Creating Activation and Deactivation datasets

In order to carry out a creep analysis on the model the construction stages are modelled using the birth and death facility within LUSAS.

Attributes
Birth & Death…

- Select the **Activate** option and click **Next**
- Enter the attribute name as **Activate** and click **Apply**. Then click **Back** so the dialog can be reused to define the deactivate attribute.



- Select the **Deactivate** option and click **Next**
- Select the **Percentage to redistribute** option and leave the value as **100%**. Enter the attribute name as **Deactivate**, then press **Finish**

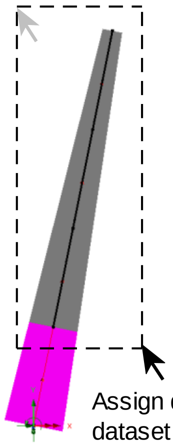
Modelling of Construction Stage 1

 Select the Home button to return the model to the default view.

- In the Treeview expand **Analysis 1** and rename **Loadcase 1** to **Stage 1: 0 to 60 Days** by right-clicking its title and selecting **Rename**

All elements in the model which are not required for the first stage analysis need to be de-activated.

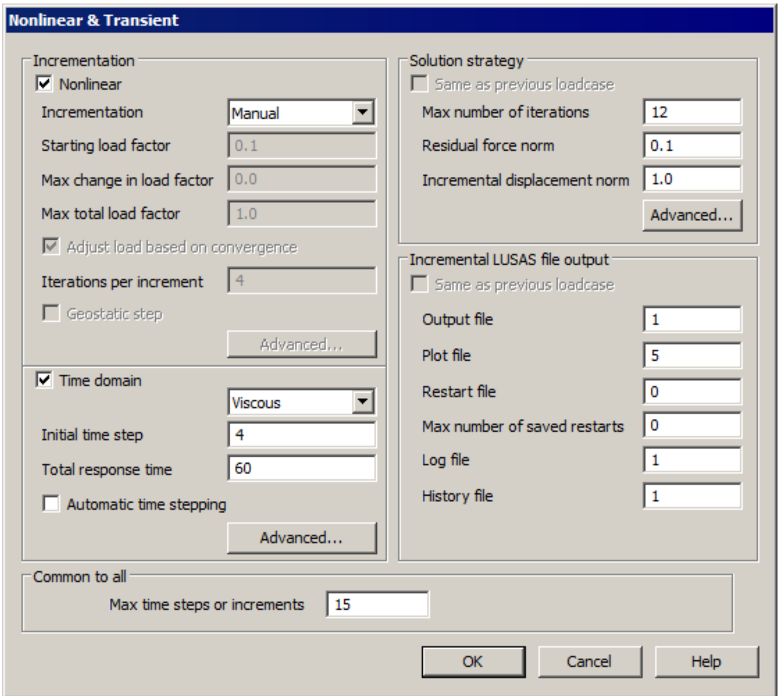
- In the graphics window select the top three lines of the tower.
- From the Treeview drag and drop the deactivation attribute **Deactivate** to the three selected lines ensuring that it is assigned to loadcase **Stage 1: 0 to 60 Days**



Assign deactivation dataset to top 3 stages of construction

Defining loadcase properties

- In the Treeview right-click **Stage 1: 0 to 60 Days** and, from the **Controls** menu item, select the **Nonlinear & Transient** menu option.
- On the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner and leave the incrementation type as **Manual.**
- Enable the **Time domain** option. Select a **Viscous** time domain from the drop-down options. Enter the **initial time step** as **4**, the **total response time** as **60**, ensure the automatic time stepping option is <u>not</u> selected and set the **maximum number of time steps or increments** as **15**
- In the incremental LUSAS file output section enter a **Plot file** value of **5** to only write out the results every 5 time steps. This will simplify the viewing of the results by only creating a results file for the final time step of each loadcase.
- Click **OK** to return to the Modeller working window.



Note. The total response time of 60 represents the 60-day time interval until the next stage of the analysis is activated.

- Having set all the Nonlinear & Transient options select **OK** to return to the Modeller working window.

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.

Construction Stages 2 to 4

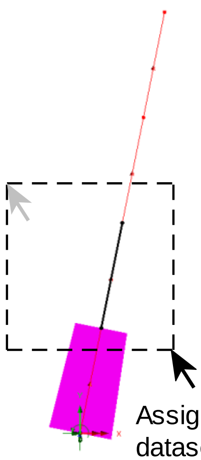
Stages 2 to 4 require loadcases to be generated to specify the duration of the construction process. Lines on the model must be selected according to the construction stage being considered and activation datasets must be assigned to these lines as instructed below.

Stage 2

The elements in the second construction stage must be activated

Analyses
Loadcase…

- Enter the Loadcase name as **Stage 2: 60 to 120 Days** and select **OK**
- In the Treeview select **Stage 2: 60 to 120 Days** using the right-hand mouse button select **Set Active**
- In the graphics window select the second line of the tower.
- With the second stage selected, assign the activation attribute **Activate** from the Treeview ensuring that it is assigned to loadcase **Stage 2: 60 to 120 Days**



Defining loadcase properties

- In the Treeview right-click **Stage 2: 60 to 120 Days** and, from the **Controls** menu item, select the **Nonlinear & Transient** menu option.
- On the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner and leave the incrementation type as **Manual**
- Select the **Time domain** option and select a **Viscous** time domain. Enter the **initial time step** as **4**, the **total response time** as **120** and the **maximum number of time steps** as **15**.

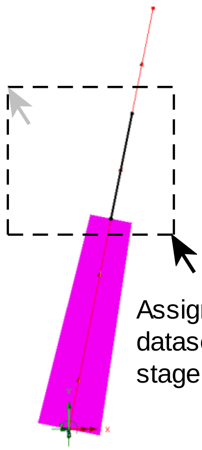
- Ensure **Same as previous loadcase** is selected for the **Incremental LUSAS file output**.
- Select **OK** to return to the Modeller working window.

Stage 3

The elements in the third construction stage now need to be activated.

Analyses
Loadcase…

- Enter the Loadcase name as **Stage 3: 120 to 180 Days** and select **OK**
- In the Treeview right-click **Stage 3: 120 to 180 Days** and select **Set Active**
- In the graphics window select the third stage of the tower.
- With the third stage selected, assign the activation attribute **Activate** from the Treeview, ensuring that it is assigned to loadcase **Stage 3: 120 to 180 Days**



Assign activation dataset to the 3rd stage of construction

Defining loadcase properties

- In the Treeview right-click **Stage 3: 120 to 180 Days** and, from the **Controls** menu item, select the **Nonlinear & Transient** menu option.
- On the Nonl**inear & Transient dia**log select the **Nonlinear** option in the top-left hand corner and leave the incr**ementatio**n type as **Manual**
- Select the **Time domain** option and select a **Viscous** time domain. Enter the **initial time step** as **4**, the **total response time** as **180** and the **maximum number of time steps** as **15**.
- Ensure **Same as previous loadcase** is selected for the **Incremental LUSAS file output**.
- Press **OK** to return to the Modeller working window.

Verifying self weight and activation assignments

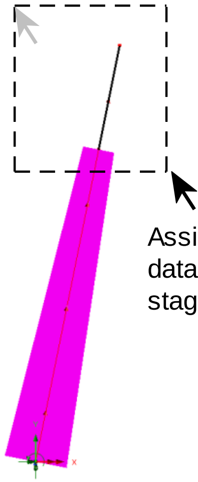
If you need to verify loading assignments such as self weight or to check when particular lines (and hence elements) become active in an analysis select a line on the model using the right-hand mouse button and select **Properties**. Select the **Activate Elements** tab. Note that the 'greyed-out' loadcase is simply the current loadcase and may not be the actual loadcase that the properties are assigned to. Selecting the **Activate** entry in the table of Assigned properties will show the loadcase to which the activation is assigned.

Stage 4

The elements in the fourth construction stage need to be activated.

Analyses
Loadcase…

- Enter the Loadcase name as **Stage 4: 180 to 240 Days** and select **OK**
- In the Treeview right-click **Stage 4: 180 to 240 Days** and select **Set Active**
- In the graphics window select the top line of the tower.
- Drag and drop the activation attribute **Activate** from the Treeview ensuring that it is assigned to loadcase **Stage 4: 180 to 240 Days**



Assign activation dataset to the 4th stage of construction

Defining loadcase properties

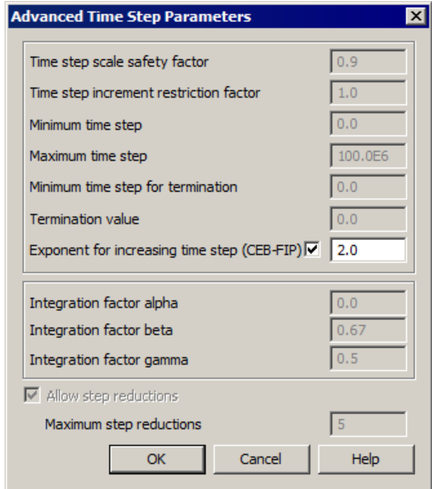
- In the Treeview right-click **Stage 4: 180 to 240 Days** and, from the **Controls** menu item, select the **Nonlinear & Transient** menu option.
- On the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner and leave the incrementation type as **Manual**
- Select the **Time domain** option and select a **Viscous** time domain. Enter the initial time step as **4**, the total response time as **240** and the maximum number of time steps as **15**. Select **OK** to return to the Modeller working window.
- Click **OK** to return to the Modeller working window.

Long-term loadcase

This investigates long-term effects on the tower over time.

Analyses
Loadcase…

- Enter the Loadcase name as **Long-term: 240 to 1000 Days**
- In the Treeview right-click loadcase **Long-term: 240 to 1000 Days** and select the **Nonlinear & Transient** option from the **Controls** menu.



- On the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner and leave the incrementation type as **Manual**

- Select the **Time domain** option and select a **Viscous** time domain. Enter the initial time step as **1**, the total response time as **1000** and the maximum number of time steps as **20**.

- Select the **Automatic time stepping** option and then select the **Advanced** button

- On the Advanced time step parameters dialog select the **Exponent for increasing time step** option and set the value to **2**. Press **OK** to return to the Nonlinear & Transient dialog.

- Having set all the Nonlinear & Transient options select **OK** to return to the Modeller working window.

Note. Specifying an **exponent for increasing time step** will cause the time step increments to increase in duration throughout a loadcase. This is particularly useful with creep analyses where it is often desirable to start a loadcase with short time steps but progress to larger time steps as the age of the concrete increases to investigate long-term behaviour without using an excessive number of time steps. The larger the value specified in the **exponent for increasing time step** box, the greater the ratio of the final to first time step size will be. The analysis will always use the number of time steps specified in the **max time steps or increments** box.

Note. All lines in the model are now activated so no activation dataset needs to be assigned to the model for this loadcase.

Applying Gravity as a property of the analysis

Now that all five loadcases have been created, in the treeview right-click the heading **Analysis 1** and select **Add gravity**. This will automatically assign gravity loading to all loadcases.

File
Save

Save the model file.

Running the Analysis

Click the **solve now** button and press **OK** to run the analysis using LUSAS Solver and load the results file

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, 2 files will be created in the Associated Model Data folder in the directory where the model file resides:

- **concrete_tower.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **concrete_tower.mys** this is the LUSAS results file which is loaded automatically into the 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 are written to the .out 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.

- ❑ **concrete_tower_modelling.vbs** carries out the modelling of the tower.

File
New…

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 **concrete_tower**

File
Script >
Run Script...

- To recreate the model, select the file **concrete_tower_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

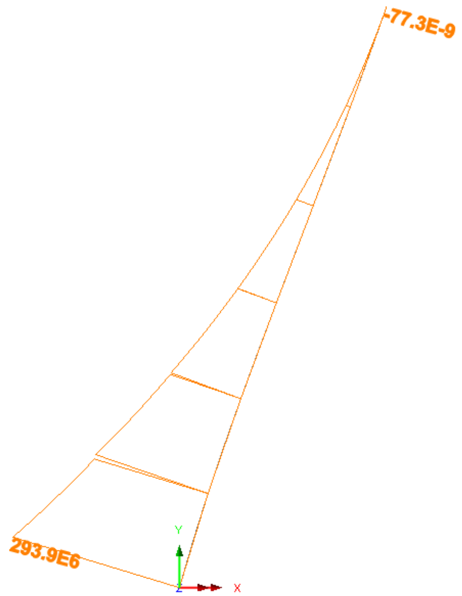
Viewing the Results

Analysis loadcase results for each time step are added to the Treeview. The time step results for the first loadcase are set to be active by default.

Bending moments of Mz at the base of the tower are to be investigated for each stage of the construction process, that is, after 60 days, 120 days, 180 days, 240 days and for a long-term loadcase. A summary of results on each results plot also allows a comparison of maximum displacements for each stage.

Stage 1 results

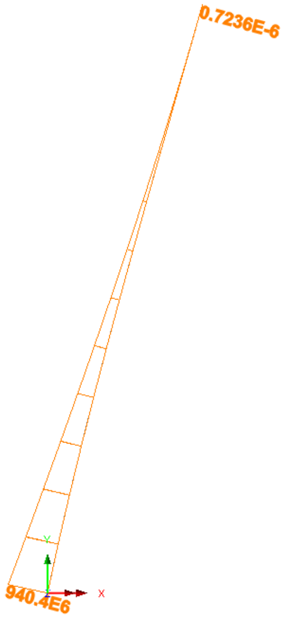
- Turn off the Geometry, Mesh, Utilities and Attributes layers in Treeview.
- Double-click **View Properties** in the Treeview. Select the **View** tab and ensure the check box **Show only activated elements** is ticked. Press **OK** to apply changes and return to the model.
- In the Treeview right-click on **Time Step 15 Time = 60.000** (days) and select the **Set Active** option.
- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select the **Diagrams** option. On the dialog, select **Force/Moment - Thick 3D Beam** contour results from the entity drop down list and component **Mz.**
- Select the **Diagram Display** tab and ensure **Label values** and **Peaks only** are ticked. Press the **Label font...** button to set the diagram font as **Arial**, **Bold**, size **14**. Press **OK** to return and change the angle to **-15°**.



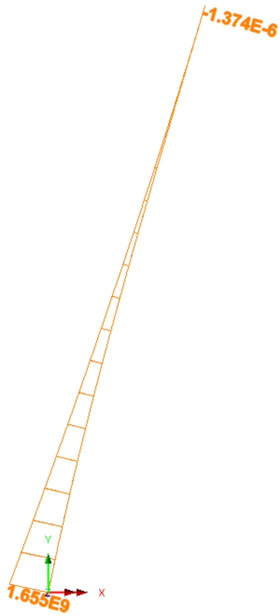
- Select the **Scale** tab select **Use a local scale** and **Specify magnitude** with a value of **10**mm. Click **OK** to display the diagrams layer.

Stage 2 to 4 results

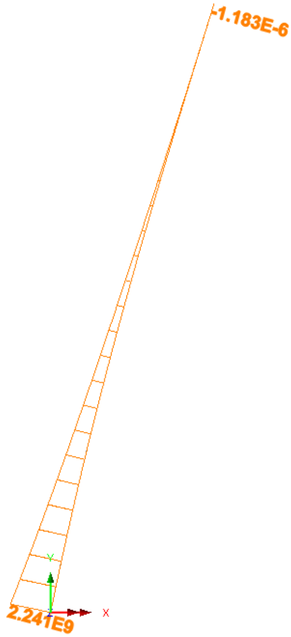
- In the Treeview right-click on the loadcases noted in the following summary table, and select the **Set Active** option to update the diagram results.



Time Step 30
Time = 120.000



Time Step 45
Time = 180.000

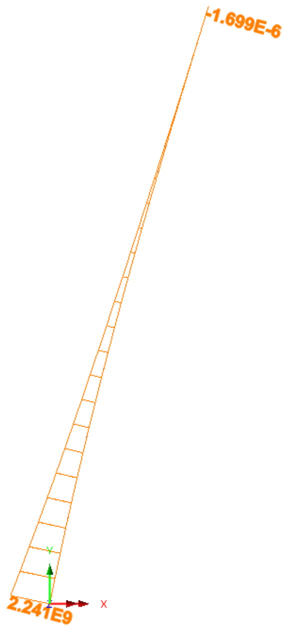


Time Step 60
Time = 240.000

Long-term loadcase

- In the Treeview right-click on **Time Step 79 Time = 1000.00** (days) and select the **Set Active** option.

For the long-term loadcase the bending moment has not changed from the stage 4 analysis since no additional loading has been applied to the model. However, a comparison of maximum displacement between time steps would show an increase due to the effect of the creep deformations.

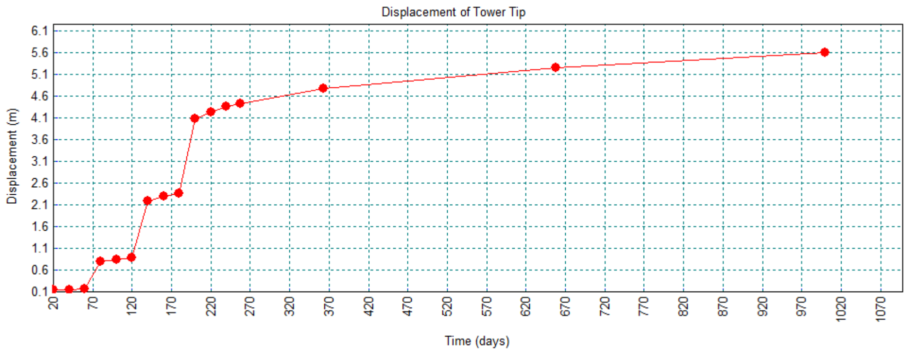


Plotting a Graph of Deflection

- The time-dependent creep effects can be plotted using the LUSAS Graph Wizard.
- Ensure that the mesh and/or deformed mesh layers are visible,
- Select the node at the top of the tower by holding the **N** key and clicking the node.

Utilities
Graph Wizard…

- On the first page of the Graph Wizard, select **Time History** and click **Next**.
- For the X-axis entity, select **Named** and ensure **Whole Analysis** is seleted. Click **Next**.
- For the X attribute named data, select **Response Time** and click **Next**.
- For the Y-axis entity, select **Nodal** and click **Next**.
- Select entity **Displacement** and component **RSLT**. The number of the selected node should automatically be entered in the ‘specify node’ input box. Click **Next**.
- Enter title **Displacement of Tower Tip**, X-axis label **Time (days)** and Y-axis label **Displacement (mm)**. Click **Finish** to plot the graph.



The graph clearly shows the four construction stages, and that the deflection increases with time due to creep and shrinkage effects. Note that the tower tip is not actually activated in the first three construction stages, but deflections of the deactivated mesh can be plotted.

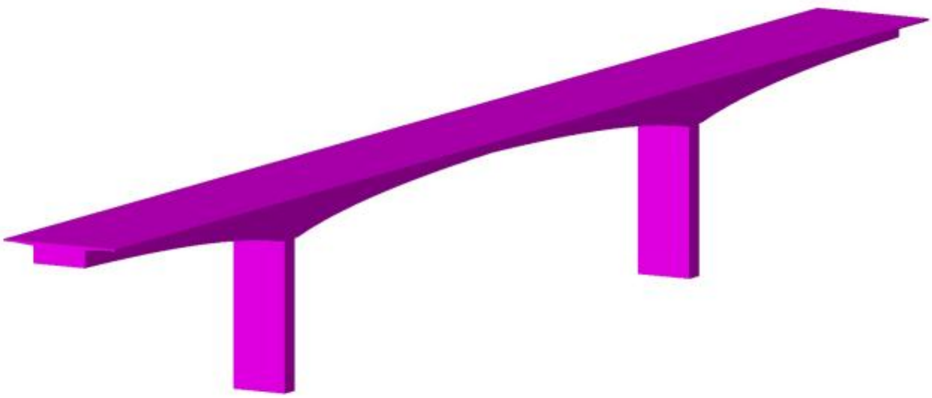
This completes the example.

3-Span Concrete Box Beam Bridge of Varying Section

For software product(s):	Bridge and Bridge Plus
With product option(s):	None

Description

A 3-span concrete box beam bridge of varying cross-section is to be modelled to illustrate the use of the box section property calculator and the use of the multiple varying sections facilities in LUSAS.



Two models are to be created; a preliminary model suitable for prototype / assessment work and a more detailed model suitable for developing further into a staged construction model. Simplified geometry is used for both to allow the example to concentrate on the use of the box section property calculator and the multiple varying section facilities that are used in modelling the structure.

Units used are kN, m, kg, s, C throughout

Objectives

The required output from the analysis consists of:

- A comparison of results from the preliminary model and the more detailed model.

Keywords

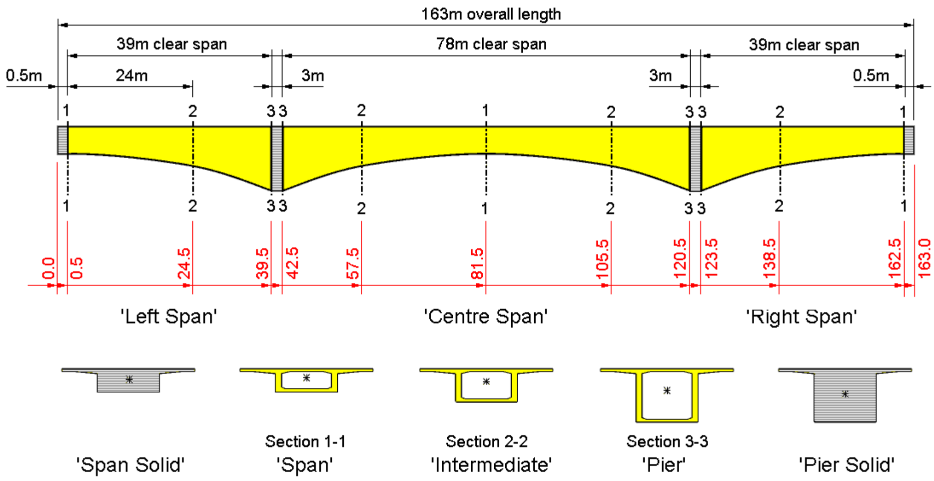
Bridge, Concrete, Multiple Varying Sections, Box

Associated Files

- **concrete_box_bridge.mdl** Basic beam model.

Discussion

The 3-span structure is comprised of varying hollow cross-sections with solid diaphragm sections at the four supports. Cross-section properties for three void locations on the structure (as shown in the image below) will be defined and used in the creation of multiple varying section geometric line attributes which will then be assigned to selected lines on the model.



Schematic showing the section locations at which the varying section definitions are based.

Other section properties, for solid diaphragm sections at the supports and at the ends of the structure (shown on the image above), will also be defined and assigned as required.

When building a model like this the bridge can either be represented as a ‘flat model’ using lines at a common level with offsets being assigned to the elements to represent to eccentricity of the section properties about the line, or geometry lines could follow the centroid of the sections to create a ‘curved’ model. If post tensioning is to be carried out the model should be based upon a ‘curved’ model. Both methods are valid but for this example the ‘flat model’ method will be used.

‘Flat’ modelling method
(where lines following the centroid of one section with other sections offset to align to match)

Indicative diagram showing ‘Curved’ modelling method
(where all lines follow the centroid of sections)

Modelling details

The supplied basic beam model is a ‘Flat’ model with clear spans of 39m, 78m and 39m formed of single straight lines. During the initial part of the example multiple varying section geometric line attributes are assigned to the single lines along with other section properties and a simple linear analysis is carried out.

After creating the simple model a more detailed model is formed by splitting the lines representing each span into shorter 3m long lengths in readiness for a future staged construction analysis. In doing so 3 mesh divisions will be present on each 3m long geometric line.

Caution. When modelling varying cross sections with constant section beam elements care should be taken to ensure that sufficient elements have been assigned to the line(s) representing the tapering beam to achieve accuracy of results. If varying section beam elements are used, only one element per line may be required.

Modelling : Preliminary Model

Running LUSAS Modeller

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

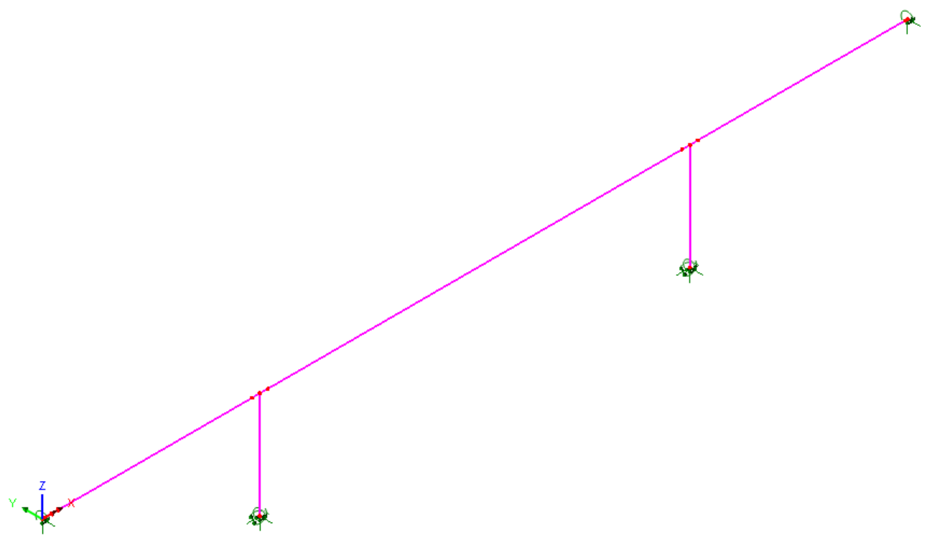
Loading the model

File
Open…

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

The basic bridge geometry will be displayed.

The geometric lines of this model have been assigned a line mesh of thick beam elements with an element length of 1m. Concrete (ungraded) material properties, and a structural gravity load have also been assigned. Fixed supports restrain the piers and roller supports restrain the ends of the deck.



File
Save As…

- In the **\<LUSAS Installation Folder>\Projects** folder create a new directory called **ConcreteBoxBridge**
- Save the model into this new folder as **concrete_box_bridge_preliminary**

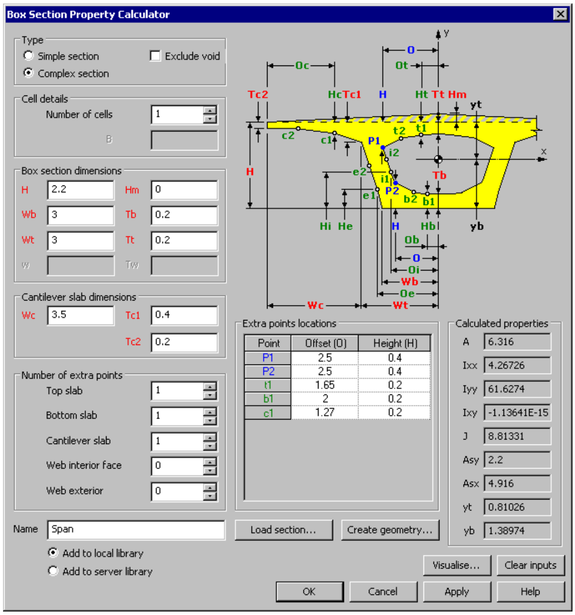
Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining Cross-sectional properties

To start, cross-section properties for three voided box sections that are used in all three spans will be defined.

To define the voided span section

Utilities
Section Property Calculator >
Box Section



- Select the **Complex section** option and enter values as shown on the dialog opposite.

Box section dimensions are:

- H = **2.2**
- Wb = **3**
- Wt = **3**
- Hm = **0** (or leave blank)
- Tb = **0.2**
- Tt = **0.2**

Cantilever slab dimensions are:

- Wc = **3.5**
- Tc1 = **0.4**
- Tc2 = **0.2**

To allow additional points to be entered in addition to P1 and P2 to define the cross-sectional shape the following need to be set in the Number of extra points panel:

- Top slab = **1**
- Bottom slab = **1**
- Cantilever slab = **1**

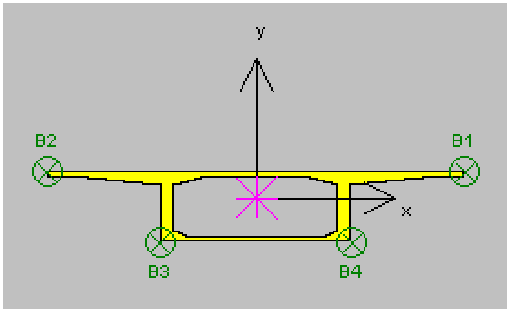
Web interior face and web exterior face points are not required so are left set to zero.

- Enter the name as **Span**

The complete set of values to be entered in the Extra points locations table are:

Point	Offset (O)	Height (H)
P1	**2.5**	**0.4**
P2	**2.5**	**0.4**
t1	**1.65**	**0.2**
b1	**2.0**	**0.2**
c1	**1.27**	**0.2**

- To see the section that will be created by the values entered click the **Visualise** button. Click **Close** to return to the main dialog.
- Ensure **Add to local library** is selected and click **Apply** to calculate the section properties, add the section to the local library, and allow you to continue to define another section.



Tip. Of the section properties calculated one, yt, (**0.81026** for this section) should be noted because it used in subsequent calculations of eccentricity / offset values for other sections.

To define the voided intermediate section

- In the Box section dimensions panel change the value of H to be **3.2**
- Change the name to be **Intermediate** and click **Apply** to calculate the section properties, add the section to the local library, and allow you to continue to define another section.

To define the voided section adjacent to the pier

- In the Box section dimensions panel change the value of H to be **5.2**
- Change the name to be **Pier** and click **Apply** to calculate the section properties, add the section to the local library, and allow you to continue to define another section.

To define the solid pier section

- In the Type panel select **Exclude void**
- Change the name to be **Pier Solid** and click **Apply** to calculate the section properties, add the section to the local library, and allow you to continue to define another section.

Tip. The difference between the yt value of the Span section and the Pier Solid section should be noted because this will be required later in the example to specify an eccentricity / offset value for other sections. In this case the difference in values of yt is 2.4626 - 0.81026 = **1.65234**

To define the solid span section

Existing defined sections such as that of the voided span section can be loaded into the dialog for subsequent modification to create new or similar sections

- Click the **Load section…** button. Browse the sections in the local library to find the **Span** entry and click **OK** to load its section details into the dialog.
- In the Type panel select **Exclude void**
- Change the name to be **Span Solid** ensure **Add to local library** is selected and click **OK** to calculate the section properties, add the section to the local library, and finish defining the deck sections.

Tip. The difference between the yt value of the Span section and the Span Solid section should be noted because this will be required later in the example to specify an eccentricity / offset value of one from the other. In this case the difference in values of yt is 0.98225 - 0.81026 = **0.17199**

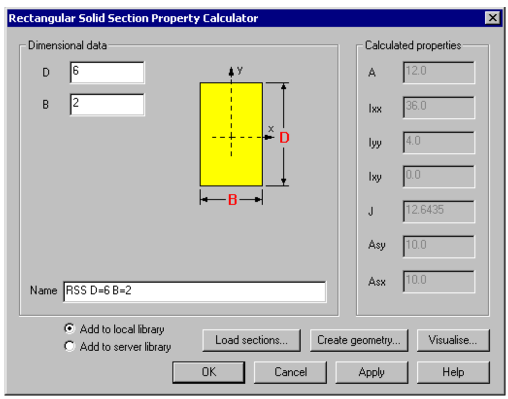
To define the column section

Utilities
Section Property Calculator >
Rectangular Sections > Solid

- On the Rectangular Solid Section Property dialog enter D=**6** and B=**2**.

A name of RSS D=6 B=2 will be automatically created.

- Ensure **Add to local library** is selected and click **OK** to calculate the section properties, add the section to the local library, and finish defining section properties.

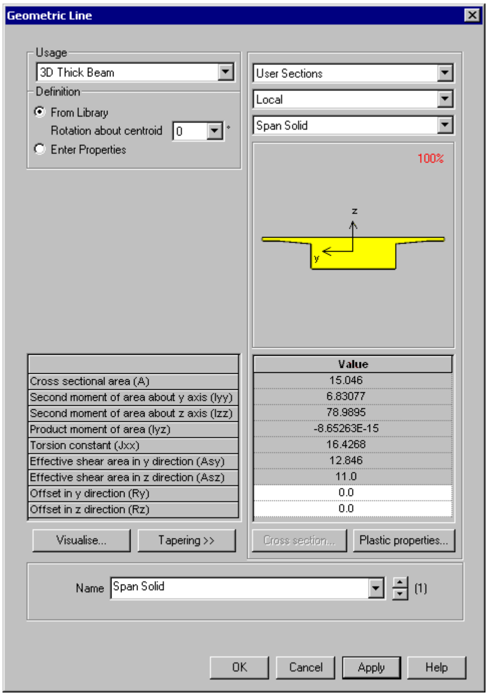


All required section properties have now been defined and saved in the local user library. The solid sections must now be added to the Treeview in order to allow for them to be assigned to the model. The voided span sections will be used to create multiple varying section line attributes which will be automatically added to the Treeview when the attribute is created.

Adding the solid section geometric properties to the Treeview

Attributes
Geometric >
Section Library…

The standard sections dialog will appear.



- Click on the drop-down list button to change the selection from UK Sections to **User Sections**
- Ensure a **Local** library is being accessed.
- Click on the drop-down list button and select **Span Solid** from the list of local library items.
- Change the name to be **Span Solid**
- Click the **Apply** button to add the Span Solid line attribute to the Treeview and allow you to choose another section.
- Click on the drop-down list button and select **Pier solid** from the list of local library items.
- Change the name to be **Pier Solid**
- Click the **Apply** button to add the Pier Solid line attribute to the Treeview and allow you to choose another section.

- Click on the drop-down list button and select **RSS D=6 B=2** from the list of local library items
- Change the name to be **Column**
- Click the **OK** button to add the Column line attribute to the Treeview.

Defining the multiple varying section line properties for the Left Span

A multiple varying geometric line attribute needs to be defined for the left-hand span. This will make use of the previously defined voided sections named Span, Intermediate and Pier.

Define sections

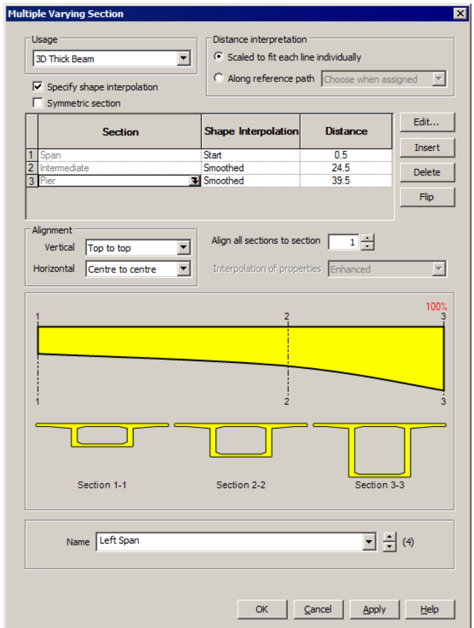
Attributes
Geometric >
Multiple Varying Section…

- On the Multiple Varying Section dialog click in the Section cell that currently reads 914x305x289kg UB and the launch dialog button will appear to allow a different pre-defined section to be chosen from a different section library.
- On the Enter Section dialog, click on the drop-down list button to change the selection from UK Sections to **User Sections**
- Ensure a **Local** library is being accessed.
- Click on the drop-down list button and select **Span** from the list of local library items and then click **OK** to add the section to the first row of the table on the Multiple Varying Section dialog.
- On the Multiple Varying Section dialog press the **TAB** key to move between cells and create a new row beneath the existing row. (Shape interpolation and distance values will be entered after all sections have been added to the table)
- In the new Section cell (row 2) click on the launch dialog button again. The drop-down list should still be showing **User Sections** for a **Local** library. Browse and select **Intermediate** from the drop-down list and click **OK** to add the section to the table.
- Press the **TAB** key to move between cells and create a new row beneath the existing row.
- In the new Section cell (row 3) click on the launch dialog button again. The drop-down list should still be showing **User Sections** for a **Local** library. Browse and select **Pier** from the drop-down list and click **OK** to add the section to the table.

Note. As the multiple varying section is built-up in the table, a visualisation of the longitudinal and vertical alignment and of the cross-section shapes used is displayed on the dialog. Longitudinal section visualisation only takes place once all required data has been entered and only for sections that are compatible.

Define shape interpolation method

To specify a shape interpolation method for each defined section:



- The **Span** section as the first in the table is set by default to **Start**
- For the **Intermediate** section entry click on the drop-down list button in the Shape Interpolation cell and select **Smoothed**
- Repeat for the **Pier** section in the cell below selecting **Smoothed**

Define distance values

Lastly we need to specify a distance value at which each section will apply.

- For the **Span** section click in the Distance cell and enter **0.5**
- For the **Intermediate** section enter **24.5**
- For the **Pier** section enter **39.5** and click somewhere else on the dialog to cause the visualisation to be updated for the last entered distance.

Note. Values entered are treated as proportional distances along a line (for example entering 0, 10 and 20 would specify a section at either end and at a mid-point of any line that was selected and assigned the geometric line attribute). Distances are also mapped to the actual line length so entering 0, 0.5, and 1, in three separate cells would produce the same result. But note that a section does not necessarily have to be defined to start at a distance of 0, (as in this example). In this case the use of a non-zero starting point of 0.5 (a chainage value representing the actual setting-out point of the section on the model) as well using a corresponding intermediate and an end point appropriate to the line beam length will help to ensure an easy conversion to a staged construction model in the second part of this example. This will use the same chainage values in conjunction with the reference path facility.

Check alignment settings

The vertical and horizontal alignment of a series of sections is controlled by the Alignment options. All sections are aligned with respect to a chosen section.

- Ensure the alignment is **Top to top**, **Centre to centre** and with respect to section **1**

Name the attribute

- Enter a name of **Left Span** and click **Apply** to add the multiple varying section line attribute to the Treeview and allow you to continue to define another one.

Defining the multiple varying section line properties for the Right Span

This can be done by reversing the order of the sections used for the left span and then changing the distance values. This time, to prevent inadvertent overwriting of previously defined line attributes we will name the attribute first.

Name the attribute

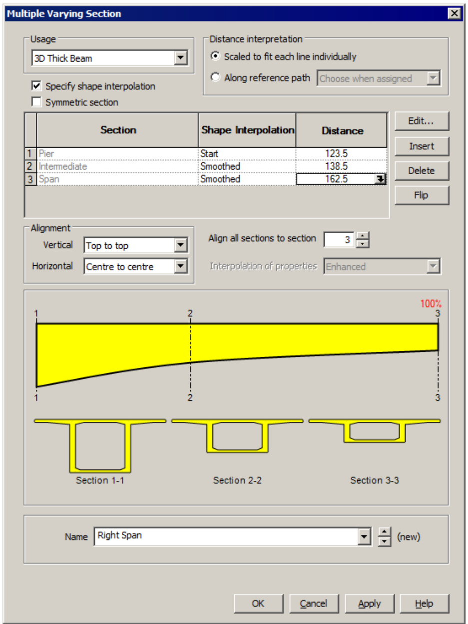
- With the Multiple Varying Section dialog for the Left Span currently displayed overtype the Left Span name with **Right Span**

Define sections

- Click the **Flip** button to change the order of the sections listed.

Check Shape Interpolation type

- The **Pier** section should be set to **Start**
- The **Intermediate** section should be set to **Smoothed**
- The **Span** section should be set to **Smoothed**



Define Distance values

- For the **Pier** section click in the Distance cell and enter **123.5**
- For the **Intermediate** section enter **138.5**
- Repeat for the **Pier** section enter **162.5**

Check alignment

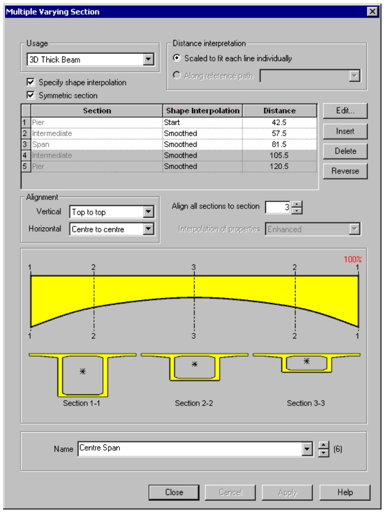
- Ensure that all sections are aligned **Top to top**, **Centre to centre** and with respect to section **3** (because this is now the number identifier for the Span section)

Save the changes

- Click **Apply** to add the multiple varying section line attribute to the Treeview, and allow you to continue to define another one.

Defining the multiple varying section line properties for the Centre Span

This can be done by using the symmetric option to mirror the sections used for the Right Span and then changing the distance values.



Name the attribute

- With the Multiple Varying Section dialog displayed for the Right Span overtype the Right Span name with **Centre Span**

Define sections

- Click the **Symmetric** option to mirror the sections listed.

Check Shape Interpolation type

- The **Pier** section should be set to **Start**

- The **Intermediate** section should be set to **Smoothed**

- The **Span** section should be set to **Smoothed**

Define Distance values

Specify a distance value at which each section will apply:

- For the **Pier** section click in the Distance cell and enter **42.5**

- For the **Intermediate** section enter **57.5**

- Repeat for the **Pier** section enter **81.5**

- Click somewhere else on the dialog to cause the visualisation to be updated for the last entered distance.

Check alignment

- Ensure that all sections are aligned **Top to top**, **Centre to centre** and with respect to section **3** (the number identifier for the main setting-out section)

Save the changes

- Click **OK** to add the multiple varying section line attribute to the Treeview and stop defining any more.

Note. Where all sections have been defined using either a LUSAS supplied standard library item or one of the LUSAS standard section generators an 'exact' calculation is made to arrive at intermediate section properties based upon the defined shape interpolation method.

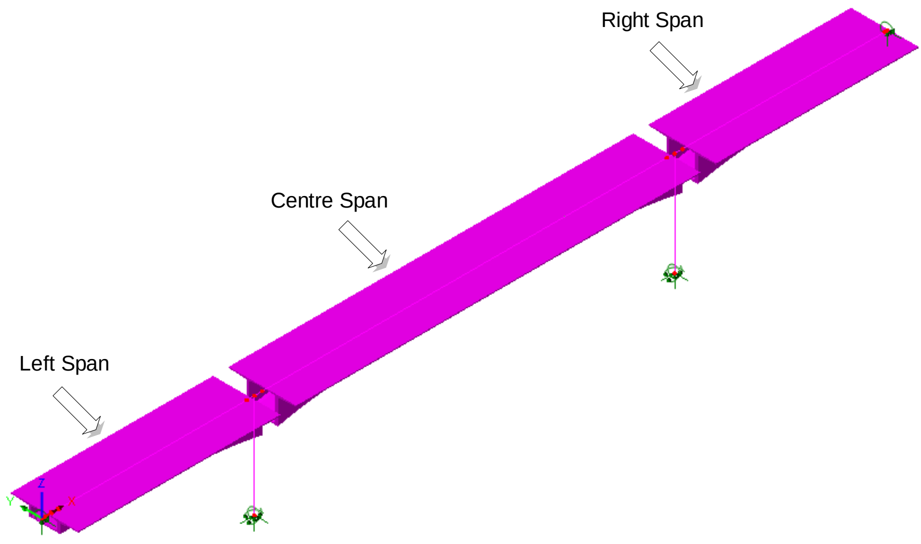
Assigning the varying geometric line properties

- Select the 39m long line representing the left-hand clear span of the bridge and drag and drop the **Left Span (Varying – 3 Sections)** geometric line attribute from the Treeview onto it.

- Select the 39m long line representing the right-hand clear span of the bridge and drag and drop the **Right Span (Varying – 3 Sections)** geometric line attribute onto it.

- Select the 78m long line representing the centre clear span of the bridge and drag and drop the **Centre Span (Varying – 5 Sections)** geometric line attribute onto it.

After assignment (and with fleshing turned on) the model will look like this:

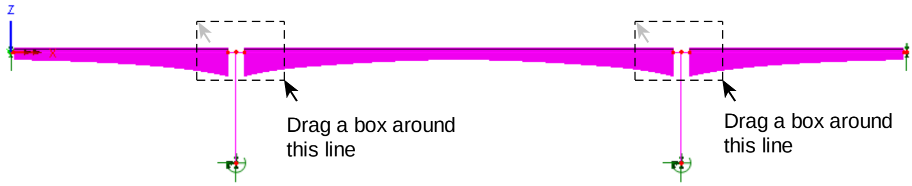


Assigning the solid geometric deck properties

The solid sections at the ends of the deck and at column locations now need to be assigned to the model.

Select the **Y axis** button to view the side elevation of the model.

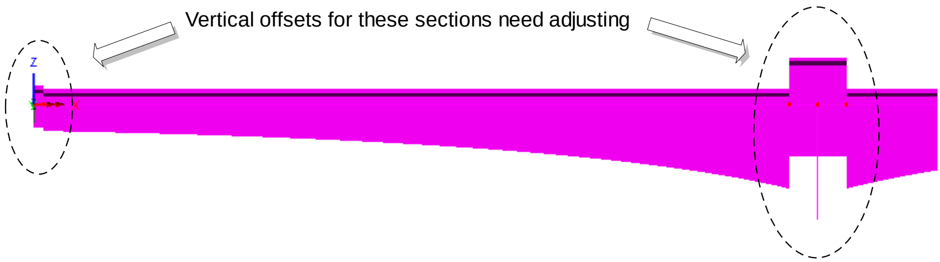
- Box-select the lines at either end of the bridge defining the solid section and drag and drop the **Span Solid (Span Solid major Z)** geometric line attribute onto the selection.



- Box-select the 2 lines at both internal supports defining the solid section of the deck and drag and drop the **Pier Solid (Pier Solid major Z)** geometric line attribute onto the selection.

Adjusting the centroid offsets for solid sections

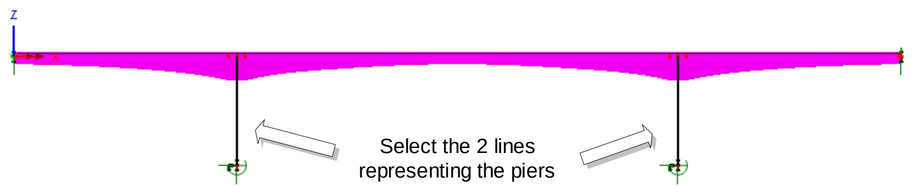
In assigning the geometric line attributes for the solid sections of the deck it can be seen that differences in centroid values in the y direction cause the solid sections to sit too high with respect to the varying sections (which were all set-out with a vertical alignment with respect to the centroid of the section named Span that sits at the left-hand end of the structure). As a result the offset values for these solid sections need amending.



- In the Treeview, double-click the **Span Solid (Span solid major Z)** entry. Enter a value of **0.17199** for the Offset in z direction (ez). This was calculated and noted with a Tip marker earlier in the example.

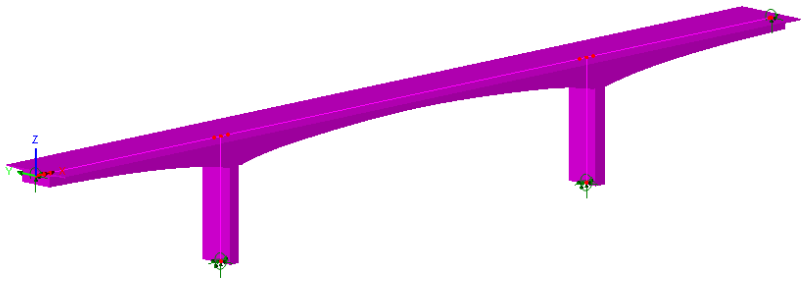
- In the Treeview, double-click the **Pier Solid (Pier Solid major Z)** entry. Enter a value of **1.65234** for the Offset in z direction (ez). This value was calculated and noted with a Tip marker earlier in the example.

Assigning the main column properties



- Select the lines representing the columns (piers) and drag and drop the **Column (RSS D=6 B=2 major z)** geometric line attribute onto the selection.

- Rotate the model to a suitable 3D view



File
Save

Save the model file.

The preliminary model is now complete and ready to be solved.

Running the Analysis : Preliminary Model

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

Note. Normally a general warning relating to the use of sufficient constant beam elements will be displayed (however in this example it has been turned off) whenever a model containing multiple varying geometric line attributes is solved.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

- **concrete_box_bridge_preliminary.out** this output file contains details of model data, assigned attributes and selected analysis statistics.
- **concrete_box_bridge_preliminary.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results to be viewed.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to the output files 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.

Viewing the Results : Preliminary Model

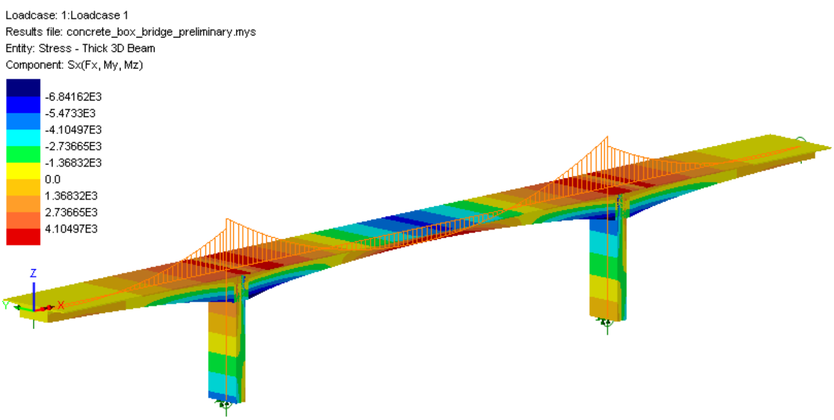
Analysis loadcase results are present in the Treeview.

Add a contour layer

- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select the **Contours** option to add the contour layer to the Treeview.
- From the drop down menu in the dialog, pick **Stress – Thick 3D Beam** from the entity drop down list and pick **Sx(Fx,My,Mz)** from the component drop down list.
- Click **OK** to display stress contours on the fleshed shape.

Add a diagrams layer

- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select the **Diagrams** option to add the diagrams layer to the Treeview.
- From the drop down menu in the dialog, pick **Force/Moment – Thick 3D Beam** from the entity drop down list and **My** from the component drop down list.
- Select the **Diagram Display** tab and deselect **Label values**. Select the **Scale** tab and enter a magnitude of **18**. Click **OK** to display a bending moment diagram on the fleshed shape.



This completes the preliminary part of the example.

Modelling : Detailed Model

The preliminary model has clear spans of 39m, 78m and 39m formed of single lines. A more detailed model is to be formed by splitting the lines representing each span into shorter 3m long lengths in readiness for a future staged construction analysis. To allow this to happen a reference path is created and the multiple varying section geometric line attributes are updated to make use of the reference path that is defined prior to the lines representing the spans being split into the smaller number of line divisions. No re-assignment of geometric properties need be made to the lines created by splitting the existing lines. The multiple varying section geometric line attributes will automatically be applied to the lines using the reference path facility.

File
Save As...

- Enter the model file name as **concrete_box_bridge_detailed** and click the **Save** button.

File
Close All Results Files

- Close the previous result files, for clarity.

Changing the model description

File
Model Properties…

- Change the model title to **Concrete Box Beam Bridge Detailed Model** and click **OK**

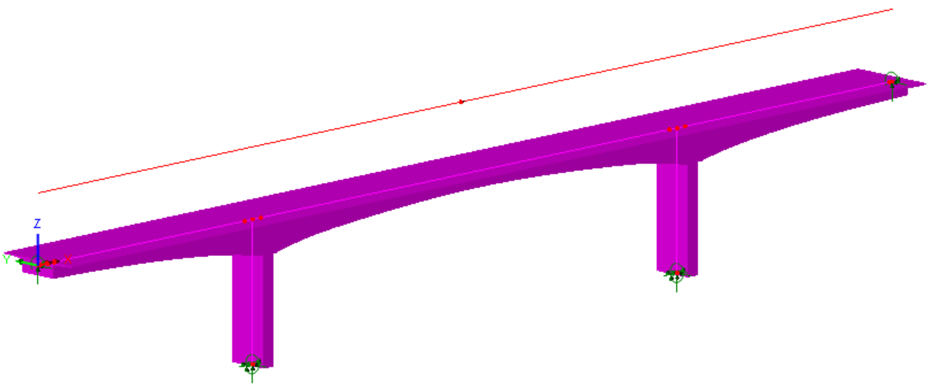
Creating a reference path

A reference path defines a route through a model that provides a concept of distance to each point in the model. Those distances can be used in the definition of a varying section, such that when the section is assigned to lines, the path is used to interpret which part of the section is appropriate to each line. Bridge engineers refer to this reference path concept as chainage.

For this example and for clarity a reference path will be defined above the model using the path definition dialog.

Utilities
Reference Path

- Enter coordinates of (**0, 0, 10**) in the first row of the table. Press the **TAB** key and enter coordinates of (**163,0,10**) in the second row, ensuring the **Type** remains **Straight**, as default.
- Accept all the default values and click **OK** to define a reference path above and of the same overall length as lines defining the deck in the model. A corresponding path definition entry will be created in the Treeview.

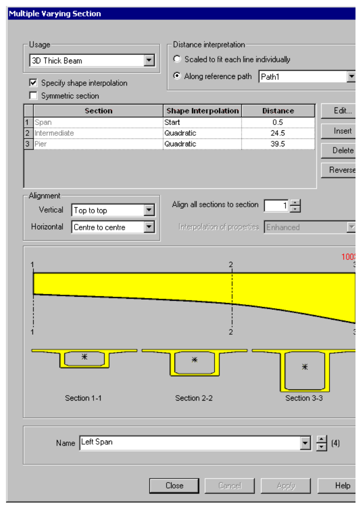


Note. In its simplest form a reference path can be defined as a line between two points (quite separate from the model data, as in this case) if a straight path is to be considered, or be created from the model geometry itself and contain as many defining points as the lines from which it has been created (by using the Utilities > Reference Path menu item) in which case the coordinates of all lines selected would be listed in the Path Definition table. If the latter is done it is important to remember that the model geometry has been used to arrive at the points required to generate a reference path but no connection between the model geometry and reference path data exists.

Editing multiple varying section geometric properties

To make use of the reference path all multiple varying section geometric line attributes need to be edited to refer to it.

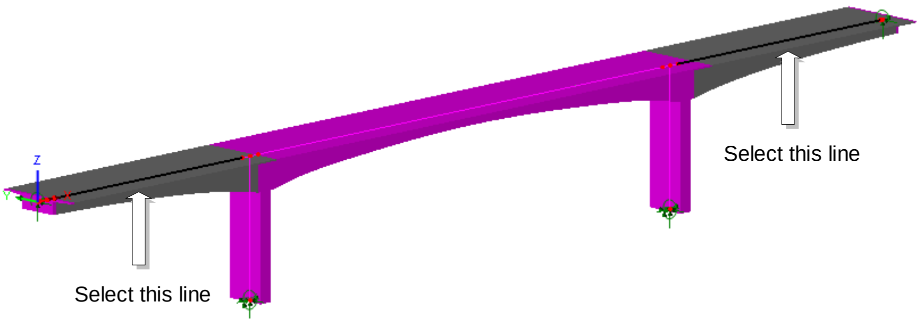
- In the Treeview, double-click the **Left Span (Varying - 3 Sections)** entry.
- On the Multiple Varying Section dialog change the Distance Interpretation setting to **Along reference path** and ensure that **Path1** is selected. Click **OK** to update.
- Repeat the above procedure for the **Right Span (Varying - 3 Sections)** entry in the Treeview.
- Repeat the above procedure for the **Centre Span (Varying - 5 Sections)** entry in the Treeview.



Split the lines representing the clear spans

The single lines representing each clear span are to be split into 3m lengths to represent each proposed construction stage.

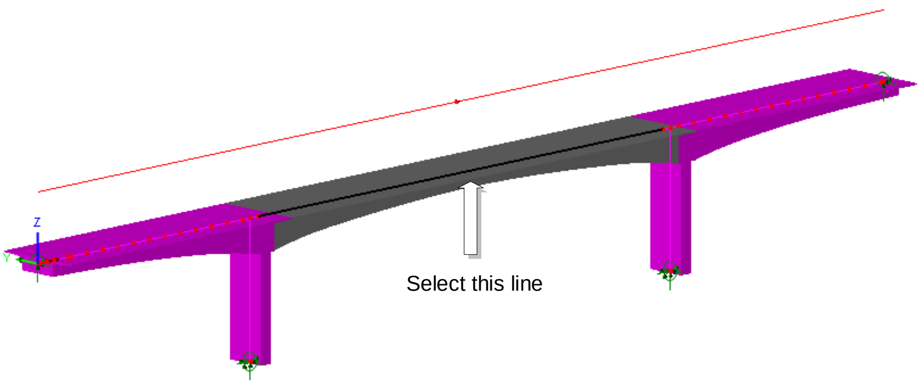
- Select the two 39m long lines representing the left and right clear spans of the bridge



Geometry
Line >
By Splitting >
In Equal Divisions

- Select **Use same divisions for all lines** and Enter **13** for the number of divisions. Ensure that **Delete original lines after splitting** is selected and press **OK**.

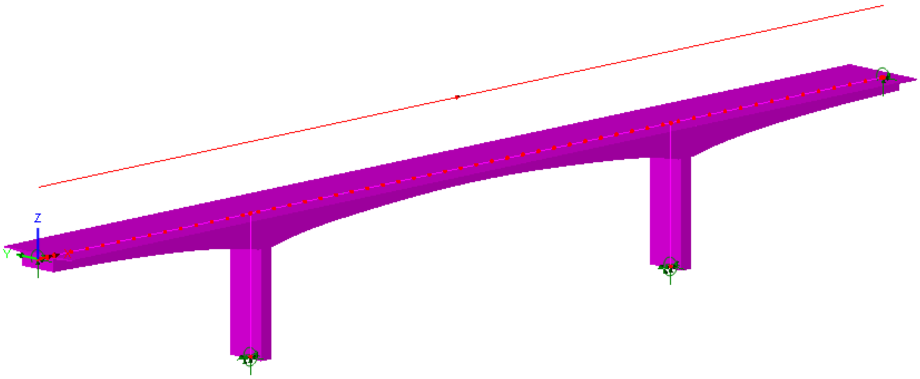
- Lastly, select the 78m long line representing the centre span of the bridge



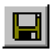
Geometry
Line >
By Splitting >
In Equal Divisions

- Enter **26** for the number of divisions and ensure that **Delete original lines after splitting** is selected.

Splitting the lines in this way produces the following model which is now ready for further manipulation (not covered in this example) to allow a staged construction analysis to be carried out.



File
Save

 Save the model file.

The detailed model is now complete and ready to be solved.

Running the Analysis : Detailed Model

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

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

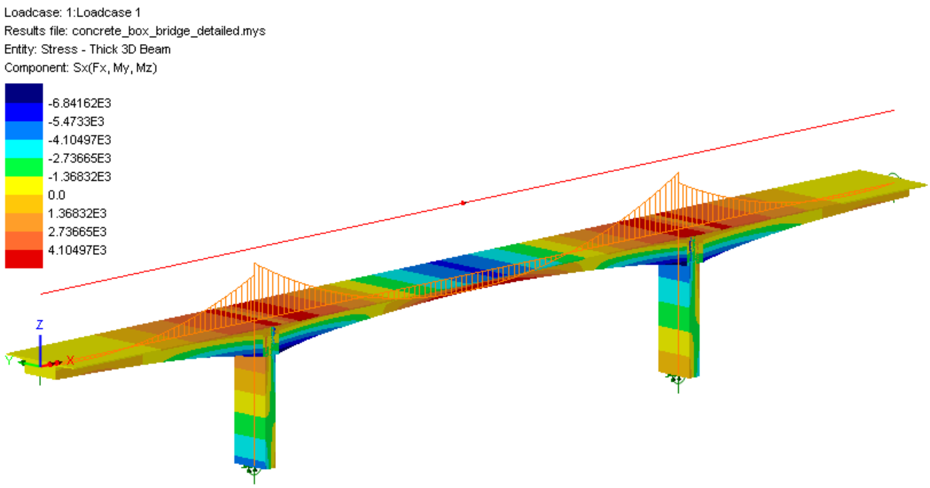
- **concrete_box_bridge_detailed.out** this output file contains details of model data, assigned attributes and selected analysis statistics.
- **concrete_box_bridge_detailed.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results to be viewed.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to a output files 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.

Viewing the Results : Detailed Model

When the detailed model has been built immediately after creating and viewing results for the preliminary model the Contour and Diagrams layers will already be present in the Treeview and the stress contours and diagram plot will be as shown below.



The results obtained can be seen to be identical to those obtained from the simple preliminary model.

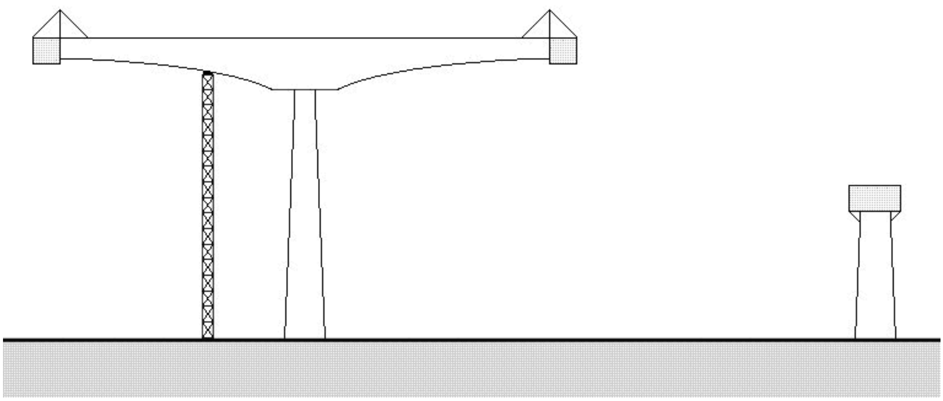
This completes the detailed modelling part of the example.

Segmental Construction of a Post Tensioned Bridge

For software product(s):	LUSAS *Civil & Structural Plus* and LUSAS *Bridge Plus*
With product option(s):	Nonlinear

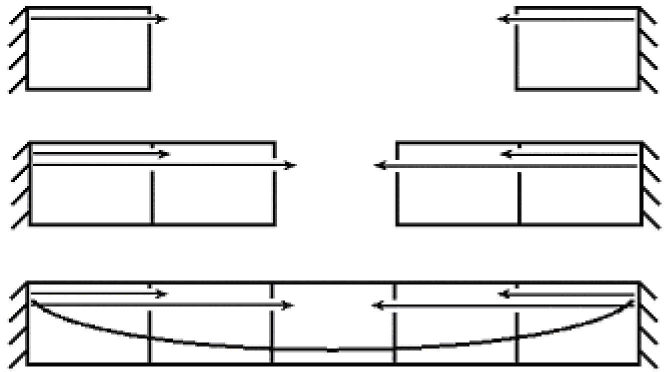
Description

The construction of a balanced cantilever segmentally constructed bridge is to be modelled using a beam analysis.



A complete analysis of a balanced cantilever segmentally constructed bridge is a complex and large analysis to undertake. In this example the geometry has been simplified to concentrate on the definition of the staged construction process and to illustrate the definition and assignment of tendon properties.

An internal span of the bridge is modelled with segments being placed from adjacent piers with a final closing segment to join the two constructions together as shown on the next diagram.



Each stage of the construction analysis considers a 6m long section being added to the construction. These are: Stage 1 (top-left), Stage 2 (top-right), Stage 3 and Stage 4 (middle) and Stage 5 (bottom)

Units used are kN, m, t, s, C throughout.

The example incorporates staged construction and the assignment of multiple tendon prestress loading.

Objectives

The output requirements of the analysis are:

- Maximum moments during construction

Keywords

Staged Construction, Multiple Tendon Prestress Wizard

Associated Files

- **segmental_bridge_modelling.vbs** carries out the modelling of the structure.

Modelling

Running LUSAS Modeller

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

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 the file name as **Segmental Bridge**
- Use the **Default** working folder.
- Enter the title as **Segmental bridge including prestress and creep**
- Select model units of **kN, m, t, s, C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Ensure the startup template **None** is selected.
- Ensure the **Vertical Y Axis** option is selected and click **OK**.

Defining the Geometry

Geometry >
Line >
Coordinates…

Enter coordinates of (**0, 0, 0**), and (**6, 0, 0**), to define the first segment of the bridge and click **OK**

- Select the line just drawn.

Geometry
Line >
Copy…

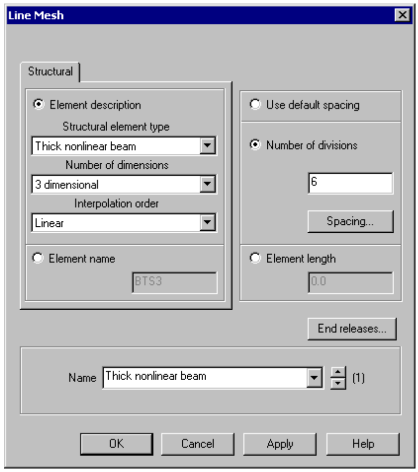
Enter a translation in the **X** direction of **6**

- Enter the number of copies required as **4** and click **OK**

Defining and Assigning Mesh Attributes

Attributes
Mesh >
Line…

- The bridge is to be modelled with **Thick nonlinear beam** elements (BTS3 elements)
- Ensure that **3 dimensional** elements of **Linear** interpolation order are selected.
- Enter **6** for the number of mesh divisions.
- Enter the attribute name as **Thick Nonlinear Beam** and click **OK**
- Select all lines on the model and drag and drop the **Thick Nonlinear Beam** mesh from the Treeview onto the selected features.



- Click **OK** to accept default element orientation.

Note. Forces and moments for a thick nonlinear beam element are constant along each element's length. As a result, sufficient elements should be used to get the desired accuracy. In this example 6 mesh divisions have been chosen for simplicity, more may be needed in real-life modelling situations.

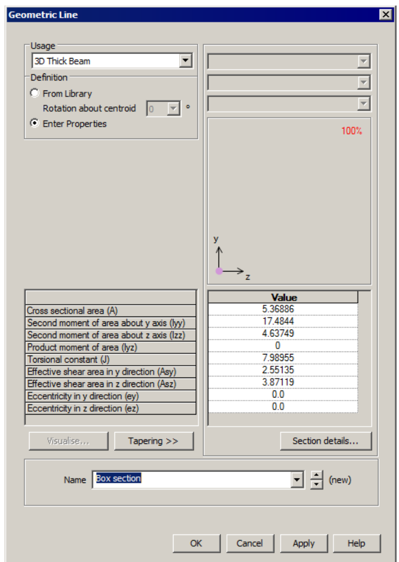
Defining and Assigning Geometric Properties

The bridge is formed from box sections of a constant width and depth. Five construction stages are to be modelled. Each of the stages uses the same section properties.

In this example the geometric properties have already been calculated and will be defined manually and then assigned to the model.

Attributes
Geometric >
Line…

- With the **3D Thick Beam** option selected enter the box section properties in the Vales column as follows:



A = 5.36886,
Iyy = 17.4844, **Izz = 4.63749**, **Iyz = 0.0,**
Jxx = 7.98955,
Asy = 2.55135, Asz = 3.87119,
ey = 0.0 and **ez = 0.0**

- Enter the attribute name as Box Section and click OK.
- Select all the lines on the model (hold down Ctrl + A keys) and drag and drop the Box Section geometry material from the Treeview onto the selected features

Defining the Material

Attributes
Material >
Material Library...

- Select material **Concrete** of grade **Ungraded** from the drop down lists, and click **OK** to add the material attribute to the Treeview.
- Select all lines on the model and drag and drop the **Iso1 (Concrete Ungraded)** material from the Treeview onto the selected features.

Note. For this example an allowable tensile stress of 4 N/mm^2 and an allowable compressive stress of 20 N/mm^2 will be considered for the concrete section.

Supports

Attributes
Support…

- A fully fixed support is required so all translations in the **X, Y** and **Z** and rotation about the **X, Y** and **Z** axes must be **Fixed**. Enter a attribute name of **Fully Fixed** and click **OK**
- Select the points at both ends of the bridge model and, from the Treeview, drag and drop the support attribute **Fully Fixed** onto the selected features, ensure the **All analysis loadcases** option is selected and click **OK**

Loading

As well as the self-weight of the structure the effects of the prestress force will also be considered in this analysis. Firstly, the self-weight will be defined.

Bridge
Bridge Loading >
Gravity

A loading attribute named **BFP1 (Gravity –ve Y)** will be created in the Treeview.

Note. If using the Civil version a gravity load may be defined from the **Attributes> Loading> Structural> Body Force** menu by specifying **–9.81** as Linear acceleration in the Y direction.

Self-weight loading is not to be applied yet. It will be applied during the definition of the staged construction process.

Creating Activation and Deactivation Datasets

In order to carry out a staged construction analysis the birth and death facility is used.

Attributes
Birth and Death…

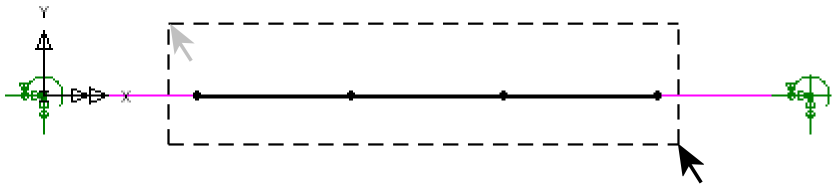
- Select the **Activate** option and click **Next**
- Enter the attribute name as **Activate** and click **Apply**
- Click **Back** so the deactivate attribute can be defined.
- Select the **Deactivate** option and click Next
- Select **Percentage to redistribute** and ensure the value is **100%**. Enter the attribute name as **Deactivate** and click **Finish**

Modelling of Construction Stage 1

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

All elements in the model not required for the first stage analysis need to be de-activated.

- In the graphics window select the three lines defining the segments that are not required in stage 1.



- From the Treeview and with the three segments selected, assign the deactivation attribute **Deactivate** ensuring that it is assigned to loadcase **Stage 1** and click **OK**

Assigning Loading

- Now select the two lines representing the active segments at either end. Assign the **BFP1 (Gravity –ve Y)** loading to **Stage 1** and click **OK**

Defining Loadcase Properties

- In the Treeview select **Stage 1** using the right-hand mouse button and select **Nonlinear and Transient** from the **Controls** menu.

- On the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner, leave the incrementation type as **Manual** and click **OK** to accept all default entries.

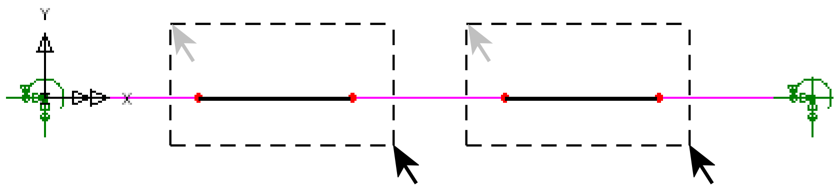
Construction Stages 2 to 3

Stages 2 and 3 require loadcases to be generated to specify the duration of the construction process. Lines on the model must be selected according to the construction stage being considered and activation attributes must be assigned to these lines.

Stage 2

The elements in the second construction stage must now be activated

- In the graphics window select the segments to be added in Stage 2 of the construction sequence.



- Assign the activation attribute **Activate** from the  Treeview. Enter **Stage 2** in the loadcase combo box and ensure that **Set as the active loadcase** is checked. Click **OK** to finish activation of the selected sections.

Assigning Loading

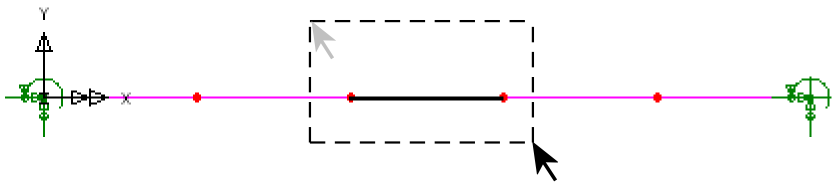
- Now select all the activated segments (i.e. the lines representing the segments from Stage 1 as well as the lines representing the segments from Stage 2) and assign the **BFP1 (Gravity –ve Y)** loading to these lines ensuring the loadcase is set to **Stage 2**. Click **OK** to complete the loading assignment.

Note. As the **Stage 1** loadcase has been set as nonlinear with manual control the subsequent loadcase controls do not need to be defined as they will take the properties defined in loadcase 1.

Note. If you need to verify loading assignments such as self-weight or to check when particular lines (and hence elements) become active in an analysis select a line on the model and then, using the right-hand mouse button, select Properties. Select the **Activate Elements** tab. Selecting the **Activate** entry in the right-hand table of Assigned properties will show the loadcase to which the activation is assigned.

Stage 3

The elements in the third construction stage now need to be activated.



- In the graphics window select the segment to be added in Stage 3 of the construction sequence.
- With the middle segment selected, assign the activation attribute **Activate** from the  Treeview. Enter **Stage 3** in the loadcase combo box and ensure the **Set as the active loadcase** is checked. Click **OK** to finish the activation of the selected section.

Assigning Loading

- Now select all of the activated segments (i.e. all lines in the model) and assign the **BFP1 (Gravity –ve Y)** loading ensuring the loadcase is set to **Stage 3**. Click **OK** to complete the loading assignment.

Note. The active elements at each stage can be visualised by removing the Geometry layer, selecting the **Show activated only** option on the mesh properties dialog and then activating each loadcase in turn.

This completes the definition of the model geometry and the staged construction process.

Rebuilding a model after a previous failed analysis

If a previous analysis of this example has failed you need to return here to continue after having run the supplied file stated.

Prestress Loading

The calculation and assignment of the equivalent prestress force to the loadcases already created is carried out by using the multiple tendon prestress wizard. It requires the following to be defined:

1. **Design code and elastic shortening criteria**
2. **Tendon profiles**

3. **Tendon properties**
4. **Tendon loadings**
5. **Tendon loading assignments**

Once entered, and once the multiple tendon prestress wizard has been run, values for each of these entries are retained and can be viewed and subsequently modified by re-running the wizard. Previously calculated equivalent loadings are automatically re-calculated.

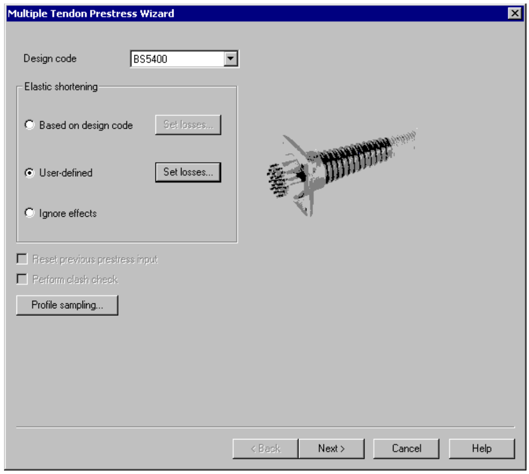
Using the Multiple Tendon Prestress Wizard

Lines on the model will be selected during the use of the wizard, so:

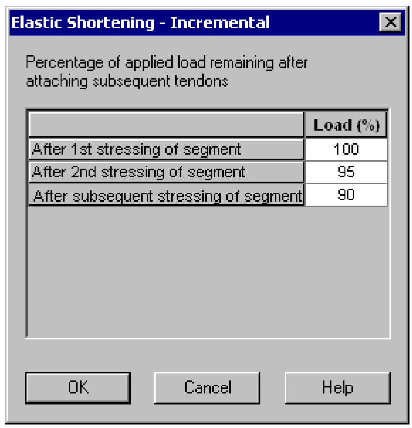
- Ensure that the **Geometry** layer is present in the Treeview.

Bridge
Prestress Wizard >
Multiple Tendon...

- Ensure that the **BS5400** design code is selected from the drop down list.
- Ensure that the **User-defined** option for elastic shortening is selected and then click **Set losses…** to proceed to the Elastic Shortening – Incremental dialog.

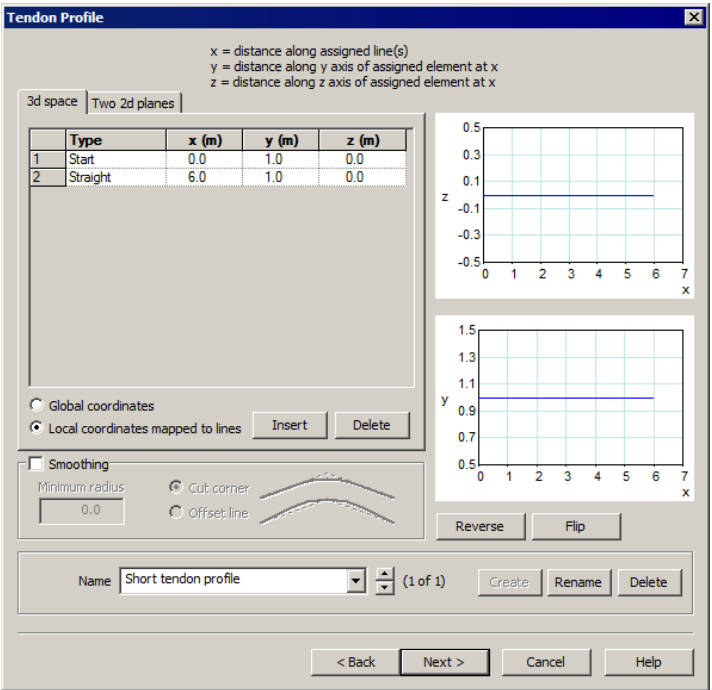


- Enter the losses as **100, 95** and **90** percent. (Use the **Tab** key to create each new line.)
- Click **OK** to return to the first dialog.
- Click **Next** to proceed to the Tendon Profile dialog.



Defining the Tendon Profiles

Any number of tendon profiles can be defined on the tendon profile page prior to moving on to the next page of the Multiple Tendon Wizard. By entering a name the Create button becomes active and tendon details can be defined. Entering another name stores the previously defined profile and enables another set of data to be entered, and so on.



- Enter the attribute name as **Short Tendon Profile** and click the **Create** button.
- Ensure the option **Local coordinates mapped to lines** is selected
- Enter the tendon profile into the grid as

Type	X (m)	Y (m)	Z (m)
Start	0	1	0
Straight	6	1	0

- Now, to start entering another tendon profile, enter the name as **Long Tendon Profile** and click the **Create** button. In doing so, this stores the previous short tendon profile values and clears the grid for the long tendon profile values to be entered.

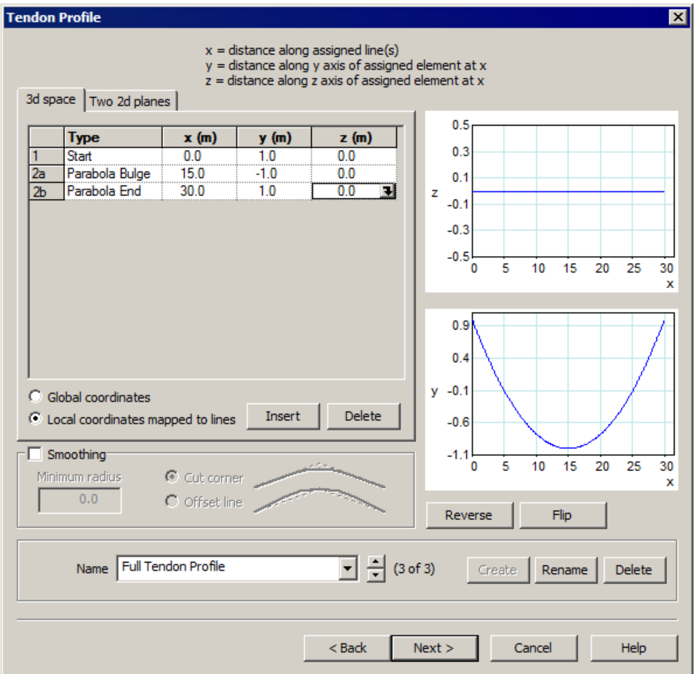
- Enter the tendon profile into the grid as follows:

Type	X (m)	Y (m)	Z (m)
Start	0	1	0
Straight	12	1	0

- Lastly, enter the attribute name as **Full Tendon Profile** and click the **Create** button.
- Enter the tendon profile into the X, Y, Z part of the grid as follows:

Type	X (m)	Y (m)	Z (m)
Start	0	1	0
Parabola Bulge	15	-1	0
Parabola End	30	1	0

- Change the Type settings from Straight to be as shown by clicking inside the cell and selecting **Parabola Bulge** for the second row and **Parabola End** for the last row.



Note. Tendon profile definitions can be reviewed before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

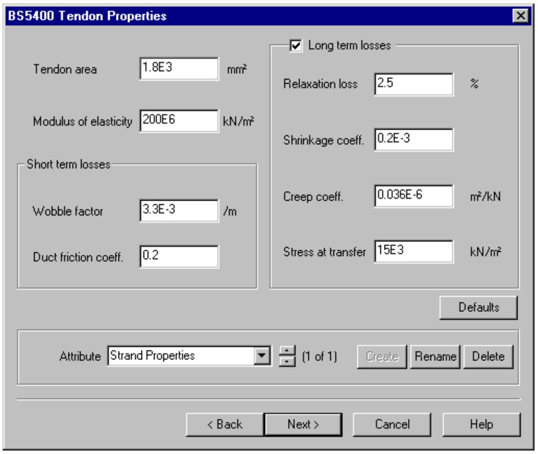
- Click **Next** to proceed to the Tendon Properties dialog.

Defining the Tendon Properties

The Multi-strand prestressing system being considered here consists of 12, 15mm diameter strands that give a nominal area of $1800mm^2$

- Enter the attribute name as **Strand Properties** and click the **Create** button
- Enter the tendon area as **1800**
- Enter duct friction coeff. as **0.2**
- Select **Long term losses**

Any number of tendon properties could be defined on this page prior to moving on to the next page of the Multiple Tendon Wizard.



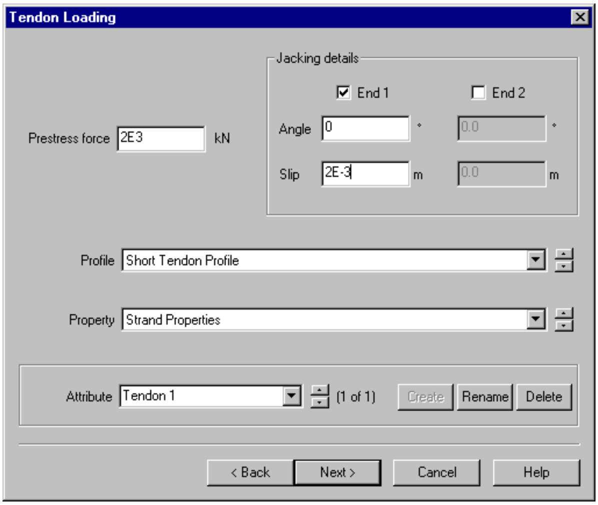
- Accept the defaults for the remaining properties by clicking the **Next** button.

Defining the Tendon Loading

This page allows a tendon load to be associated with previously entered profiles and properties.

For tendon 1:

- Enter the attribute name as **Tendon 1** and click the **Create** button.
- Enter the prestress force as **2E3**
- Ensure that jacking from **End 1** only is selected and enter the slip as **2E-3**
- Ensure the profile combo box is set to **Short tendon profile** and the property combo box is set to **Strand Properties**



For tendon 2:

- Enter the attribute name as **Tendon 2** and click the **Create** button.
- Enter the prestress force as **2E3**
- Ensure that jacking from **End 2** only is selected and enter the slip as **2E-3**
- Ensure the profile combo box is set to **Short tendon profile** and the property combo box is set to **Strand Properties**

For tendon 3:

- Enter the attribute name as **Tendon 3** and click the **Create** button.
- Enter the prestress force as **2E3**
- Ensure that the jacking from **End 1** only is selected and enter the slip as **2E-3**
- Change the profile combo box so that **Long tendon profile** is displayed and the property combo box so that **Strand Properties** is displayed.

For tendon 4:

- Enter the attribute name as **Tendon 4** and click the **Create** button.
- Enter the prestress force as **2E3**
- Ensure that the jacking from **End 2** only is selected and enter the slip as **2E-3**

- Change the profile combo box so that **Long tendon profile** is displayed and ensure the property combo box is set to **Strand Properties**

For tendon 5:

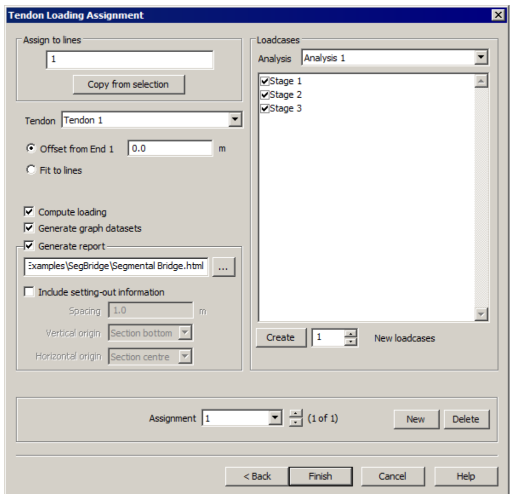
- Enter the attribute name as **Tendon 5** and click the **Create** button.
- Enter the prestress force as **4E3**
- Ensure that the jacking from **End 1** and **End 2** is selected and enter the slip as **4E-3** at each end.
- Change the profile combo box so that **Full tendon profile** is displayed and ensure the property combo box is set to **Strand Properties**

Note. Tendon loading assignments can be reviewed and corrected if necessary before proceeding further by clicking the up and down buttons adjacent to the tendon profile name.

- Click **Next** to proceed to the Tendon Loading Assignment dialog.

Defining the Tendon Loading Assignment(s)

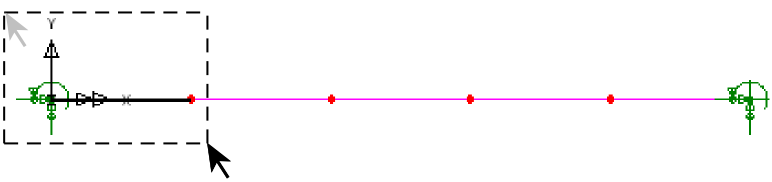
The page of the wizard allows the assignment of the tendon loads that have been defined in the previous dialog to selected lines on the model.



Click the **New** button and the Assignment will change to 1

- Drag the dialog so that the analysis model can be clearly seen in the graphics window

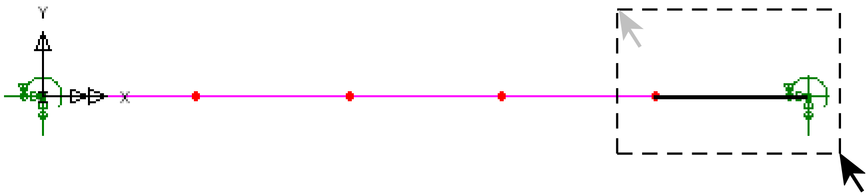
- Select the line tendon 1 acts on.



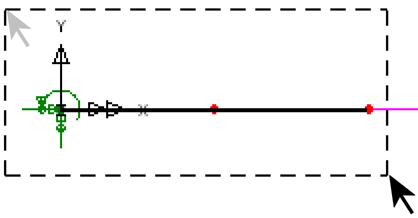
- On the Tendon Loading Assignment dialog click the **Copy from selection** button and the selected line number will be entered in the text box.
- Ensure the tendon combo box has **Tendon 1** displayed.
- In the loadcases section of the dialog, ensure **Stage 1, Stage 2** and **Stage 3** are selected.
- Ensure **Generate graph datasets** is selected.

Note. Clicking a higher loadcase will select or deselect loadcases beneath it.

- Click the **New** button and the Assignment will change to 2.
- Select the line tendon 2 acts on.

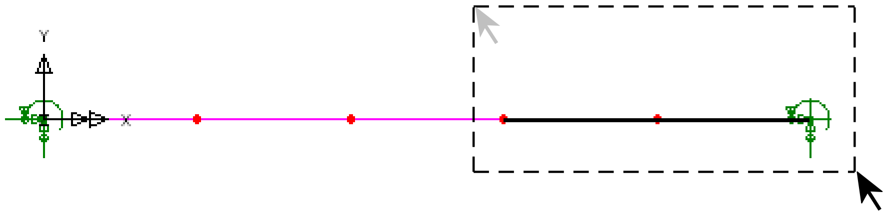


- Click the **Copy from selection** button and the selected line number will be entered in the text box.
- Change the tendon combo box so that **Tendon 2** is displayed.
- In the loadcases section of the dialog, ensure **Stage 1, Stage 2** and **Stage 3** are selected
- Click the **New** button and the Assignment will change to 3.
- Select the two lines tendon 3 acts on.



- Click the **Copy from selection** button and the selected line numbers will be entered in the text box.
- Change the tendon combo box so that **Tendon 3** is displayed.

- In the loadcases section of the dialog, ensure **Stage 2** and **Stage 3** only are selected
- Click **New** and the Assignment will change to 4.
- Select the two lines tendon 4 acts on.

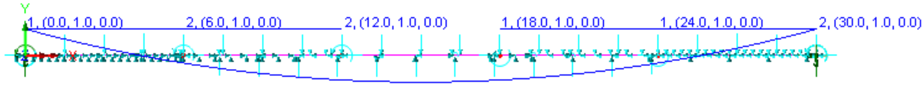


- Click the **Copy from selection** button and the selected line numbers will be displayed in the text box.
- Change the tendon combo box so that **Tendon 4** is displayed.
- In the loadcases section of the dialog, ensure **Stage 2** and **Stage 3** only are selected
- Click **New** and the Assignment will change to 5.
- Select the lines tendon 5 acts on. (All the lines in the model.)
- Click the **Copy from selection** button and the selected line numbers will be displayed in the text box.
- Change the tendon combo box so that **Tendon 5** is displayed.
- In the loadcases section of the dialog, ensure that the check box for **Stage 3** only is selected.
- Ensure that the **compute loading** and **generate graph datasets** and **generate report** options are selected. Graph datasets will be created in the Treeview for graphing if required. A report will be generated in the directory that you are currently working in but this can be changed if required.
- Select the **Finish** button to calculate and generate the equivalent prestress loading.

On completion, an HTML report will be opened automatically in the default web browser. This report is also saved into the specified folder. The report is a complete summary of the tendon profile, properties and loading assignments made.

On closing the report the tendon loading assignments on the model can be seen for Stage 1. By changing the active loadcase the assignments can be seen for the other construction stages also.

Note. Tendon profile visualisations and the display of start and finish coordinates can be controlled by changing the **Current style** setting to **With coordinates** on the **Utilities** layer Properties dialog. Visualisations will then be identified by their start and finish identifiers (1 or 2) and by the coordinates of these points. Because all tendons are active in loadcase 3 the visualisations will overlap.



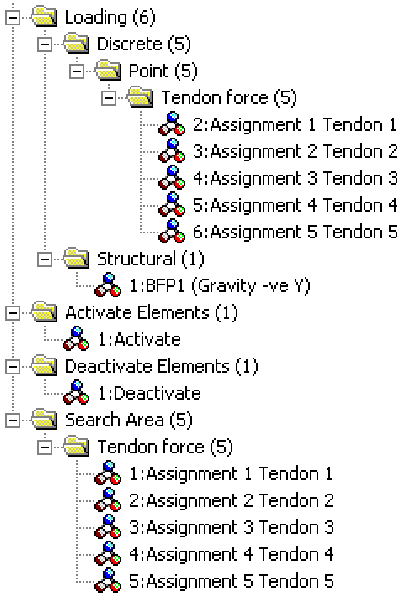
Notes on Prestress Wizard-created data

In using the Prestress Wizard, prestress loading is added to the analysis model as equivalent discrete point loading. Attributes containing this data can be seen in the Treeview in the Tendon force folder.

Search areas are created and used automatically by the prestress wizard to define the target geometry (in this case, the lines) to be loaded. These can be seen in the Treeview.

Load factors are used to represent elastic shortening and these can be seen for each stage in the Treeview.

Graphing datasets (which allow of graphing of prestress losses etc) are added to the Treeview.



Notes on editing Prestress Wizard-created data

If any tendon assignments are incorrect, the Multiple tendon prestress wizard may be re-run (previously entered values will be retained) and the Next button can be clicked to get to the tendon loading assignment page. Incorrect assignments can be deleted or simply corrected as required before clicking the Finish button to recalculate the equivalent tendon loading on the model.

Tendon profile data defined within the Prestress Wizard is also stored in the Treeview. Editing of tendon profile data in the Treeview (that is, outside of the wizard) is permitted but all attributes that require re-calculating by re-running the prestress wizard will be marked with the symbol. Re-running the prestress wizard will update all values accordingly.

Editing of any attribute data calculated and generated by the prestress wizard is not permitted.

Save the model

File
Save

Saving the model file also saves all the multiple tendon prestress input. This allows the prestress definition and assignment data to be modified by re-entering the multiple tendon prestress wizard.

Running the Analysis

With the model loaded:

Open the **Solve now** dialog. Ensure **Analysis 1** is selected 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 Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

- **segmental bridge.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **segmental bridge.mys** this is the LUSAS results file which is loaded automatically into the 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. Select **Yes** to view the output file. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis. Note that a common error is to forget to assign attribute data (such as geometry, mesh, supports, loading etc.) to the model. If the errors cannot be identified the model may be rebuilt and the prestress loading redefined.

Rebuilding the Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model up to the point of defining the Prestress Loading.

- **segmental_bridge_modelling.vbs** carries out the geometric modelling of this example.

File
New…

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 **Segmental Bridge**

File
Script >
Run Script...

- To recreate the model, select the file **segmental_bridge_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

The geometry of the model has been created. Now return to the section entitled **Prestress Loading** earlier in this example and re-define the tendon properties.

Viewing the Results

Analysis loadcase results are present in the Treeview for each stage.

Bending moments of Mz over the segments are to be investigated for each stage of the construction process. A summary of results on each results plot also allows a comparison of maximum displacements for each stage.

In the following images the tendon profile visualisations are not shown. They can be turned off by editing the tendon profile properties of the Utilities layer in the Layers Treeview.

Stage 1 results

- In the Treeview right-click on **Stage 1** and select the **Set Active** Option.
- Turn off the display of the **Mesh, Geometry**, **Utilities** and **Attributes** layers from the Treeview.
- If not there already add the **Deformed mesh** layer to the Treeview.
- Using the **Deformations...** button at the bottom of the Treeview and specify factor as **1E3**. Click the **Window summary** option.

- Add the **Diagrams** layer to the Treeview. On the dialog, select **Force/Moment - Thick 3D Beam** results from the entity drop-down list and component **Mz.** Select the **Diagram Display** tab.
- Select the **Label values** options and deselect the **Peaks only** option.
- Set the % of element length to **60**. Set the Angle to **90** degrees and the number of significant figures to **4**.
- Click **OK** to display the bending moment diagram for the active elements for Stage 1.

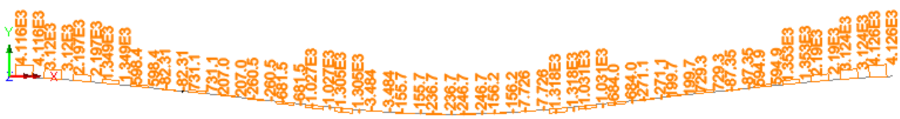
Note. The maximum deflection and maximum and minimum bending moments are shown in the window summary.

Stage 2 results

- In the Treeview right-click on **Stage 2** and select the **Set Active** option.

Stage 3 results

- In the Treeview right-click on **Stage 3** and select the **Set Active** option.



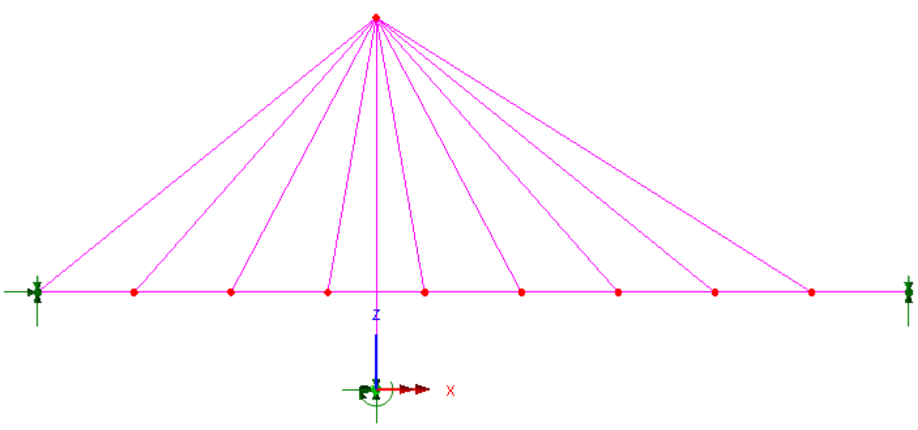
This completes the example.

Cable Tuning Analysis of a Pedestrian Bridge

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

Cable tuning analysis, targets, exact solution, linear

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.
- ❑ **cable_tuning_complete.mdl** contains the complete model ready for analysis.

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:

File
Open…

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

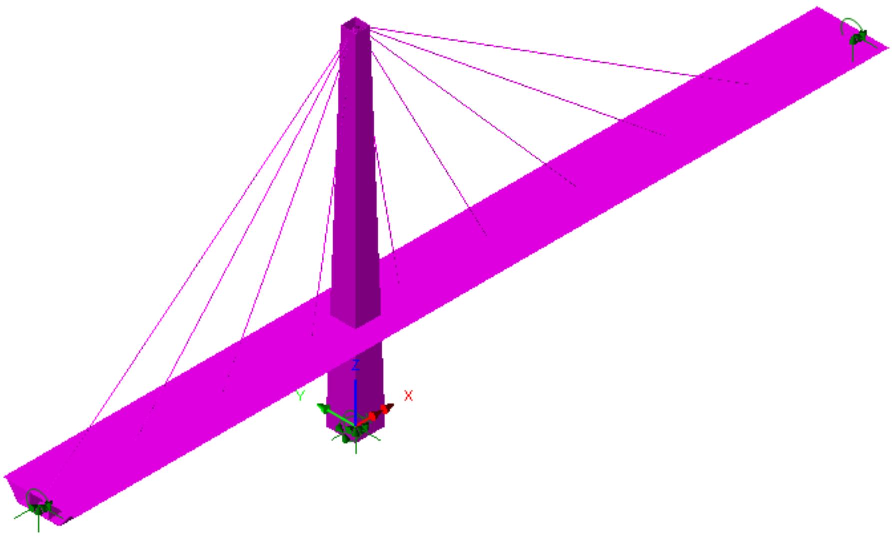
File
Save As

Save the model file to the **\<Lusas Installation Folder> \Projects** directory to avoid over-writing the provided file.

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

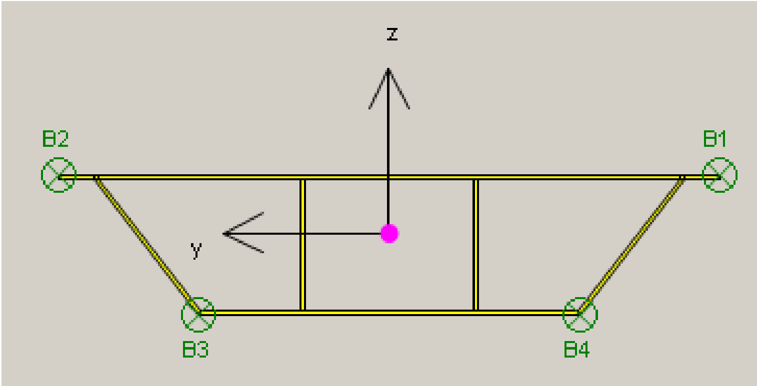
The model geometry and attributes are already defined in the supplied model file.



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 9no. 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

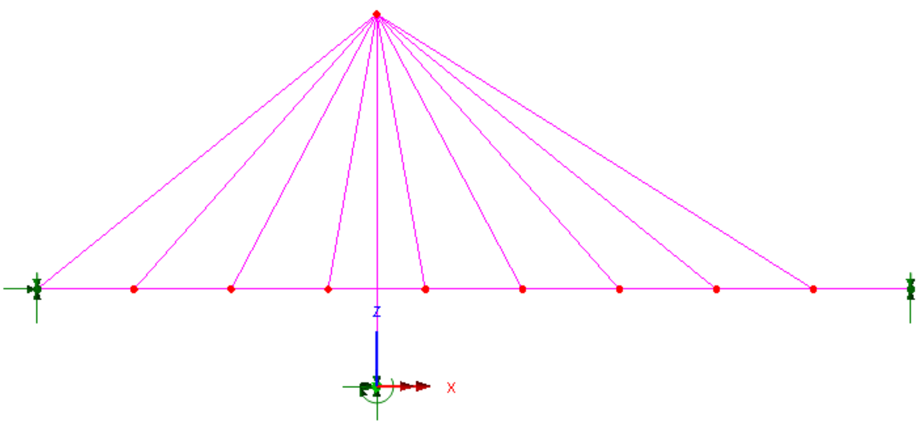
The lines representing the tower and the deck are meshed with thick 3D thick nonlinear beam elements (BMI21). The lines representing the cables are each meshed with a single 3D bar element (BRS2). Each cable is represented by a circular solid section of 50mm diameter.

Checking the Modelling so Far

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).

- 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.

Utilities
Print Results Wizard…

- Check the reactions using **Utilities> Print Results Wizard,** then ensure that **Selected, All loadcases,** for entity **Reaction** and for type **Summary** are selected and click **Finish**.
- With the Self weight tab selected ensure that the total self-weight reaction is **11960**kN. With the Surfacing tab selected ensure the total surfacing reaction is **2160**kN (both in the Z-direction).

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 we will calculate forces 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, right-click the geometric attribute **Cable** and click **Select assignments**.

Analyses
Cable Tuning Analysis…

In the first tab of the Cable Tuning Analysis dialog you should see a list of the lines which represent the cables under the **Selected** heading. Optimised forces for all of these lines will be required.

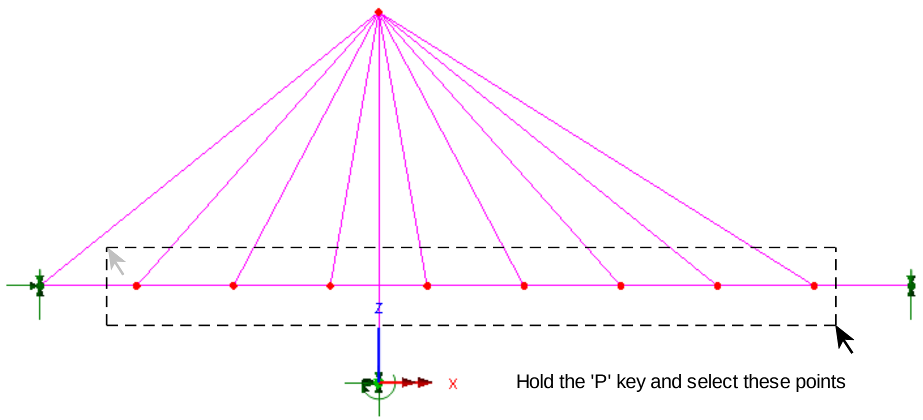
- Click the **All** button in the middle of the dialog to copy them all to the **Included** list on the right-hand side of the dialog.
- Switch to the **Loadcases** tab of the dialog.

This tab enables 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.

Under the Targets tab 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 bottom-left-hand side of the screen and pan the model view so that a clear image of the bridge is shown.
- In the Treeview switch the Deformed mesh layer off.
  - 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.
- 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 cables is equal to the number of targets so an **Exact** solution can be calculated.

Note. If the number of 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 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.

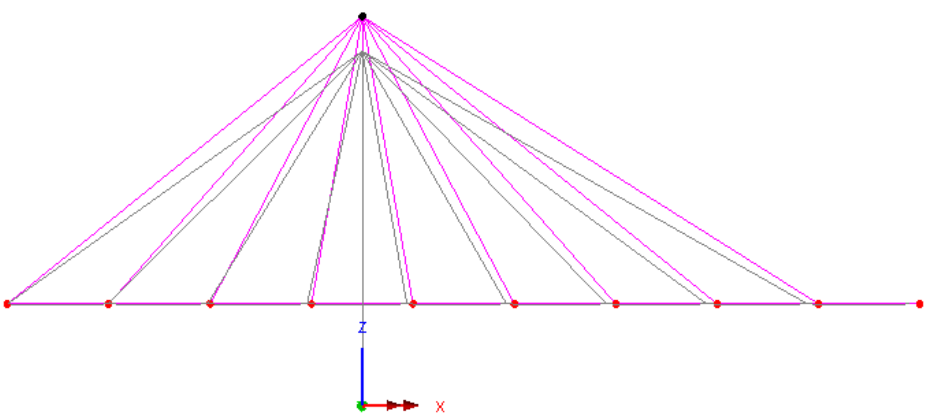
- Assuming that your inputs were successfully validated, click **OK** to conclude the Cable Tuning setup. A Cable tuning analysis entry **Cable tuning 1** will be added to the Treeview along with a loadcase entry for each of the cables selected for tuning, and a cable tuning results entry shown by a 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.

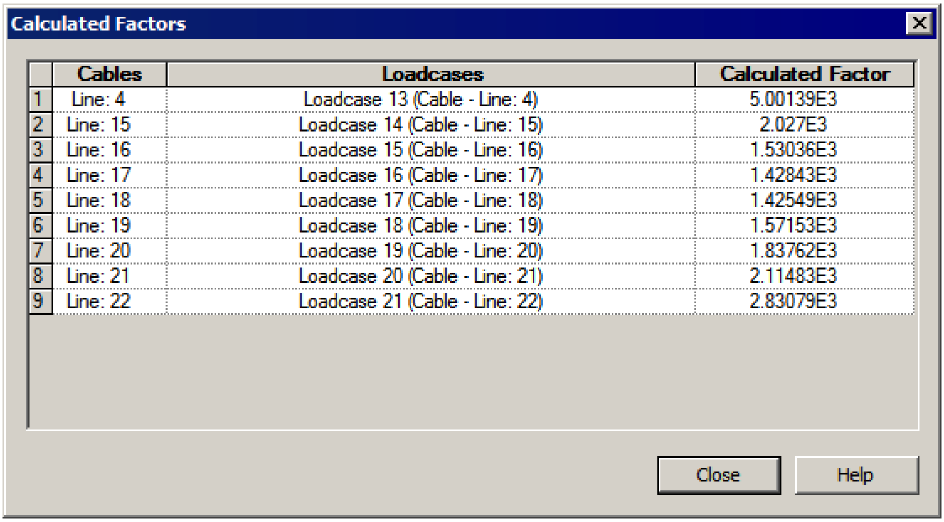
Viewing the results from the Cable Tuning Analysis

- 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 switch the **Deformed mesh** layer on. 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 below.



- To see the calculated load factors for individual cables, in the Treeview right-click the loadcase **Cable Tuning 1 Results** and select **Calculated Factors**.



	Cables	Loadcases	Calculated Factor
1	Line: 4	Loadcase 13 (Cable - Line: 4)	5.00139E3
2	Line: 15	Loadcase 14 (Cable - Line: 15)	2.027E3
3	Line: 16	Loadcase 15 (Cable - Line: 16)	1.53036E3
4	Line: 17	Loadcase 16 (Cable - Line: 17)	1.42843E3
5	Line: 18	Loadcase 17 (Cable - Line: 18)	1.42549E3
6	Line: 19	Loadcase 18 (Cable - Line: 19)	1.57153E3
7	Line: 20	Loadcase 19 (Cable - Line: 20)	1.83762E3
8	Line: 21	Loadcase 20 (Cable - Line: 21)	2.11483E3
9	Line: 22	Loadcase 21 (Cable - Line: 22)	2.83079E3

Note. Although the values shown in the Calculated Factors dialog cannot be edited, they can be copied for pasting into spreadsheets and reports by selecting the data and pressing **Ctrl+C**

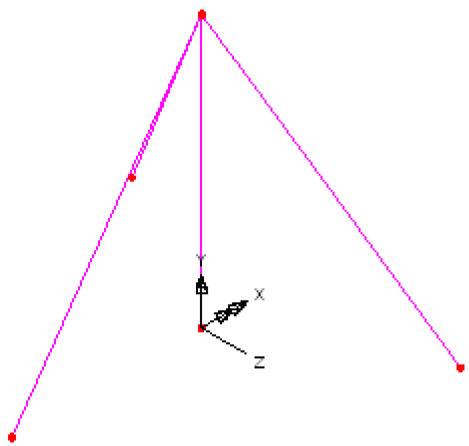
This concludes the example.

3D Nonlinear Static Analysis of a Cable Stayed Mast

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 210e9 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 500 N/m distributed along the length of the mast.
4. A horizontal point load acting at the top of the mast of 5000 N.

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 base of the mast**

Keywords

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

Associated Files

- **mast_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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 the file name and model title as **Mast**
- Use the **Default** working folder.
- Set model units of **N,m,kg,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the model startup template **Standard**
- Select the **Vertical Y axis** option and click **OK**.

Defining the Geometry

Geometry
Point >
Coordinates...

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

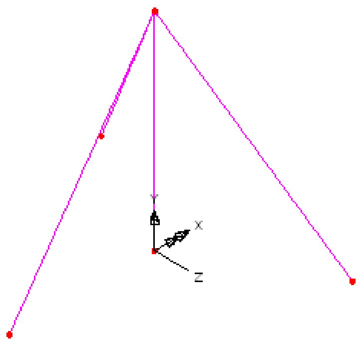
- Select the Point

Geometry
Line >
By Sweeping...

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

Geometry
Point >
Coordinates...

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)

Geometry
Line >
Points...

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.

Attributes
Mesh >
Line...

Select **Thick nonlinear beam**, **3 dimensional**, **Linear** elements.

- Enter the attribute name as **Mast** and click the **Apply** button.
- Change the number of divisions per line to **10**
- Change the attribute name to **Cables** and click the **OK** button
- Select 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

The mast will be defined as a Circular Hollow Section (CHS).

Attributes
Geometric >
Section Library...

- From the Usage section select a **3D Thick Beam** with no rotation specified.
- 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.

The 10mm diameter guy cables will be defined with very low bending properties in order to not attract any bending moments. A value of 1/100th of the actual bending stiffness for the 10mm diameter cable will be used. The torsional constant and shear area values are left unaltered.

Attributes
Geometric
Line…

- From the Usage section ensure a **3D Thick Beam** is selected
- Enter the properties shown below:

  A=7.85e-5

  Iyy=5e-12

  Izz=5e-12

  Iyz=0

  J=1e-9

  Asy=7.85e-5

  Asz=7.85e-5

  ey=0, ez=0

Note that these properties could have been derived by using the standard section property calculator.

- 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 members already selected.
- Drag and drop the geometry attribute **Mast (168.3x8 CHS major z)** from the Treeview onto the selected Line. 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. Note that for manually entered section property data the section properties cannot be visualised unless cross-section data is also defined (which will not be done in this case)

Select the fleshing on/off button to turn-off the geometric visualisation.

Material Properties

A linear approximation to steel properties will be used for the mast and cables.

Attributes
Material
Material Library...

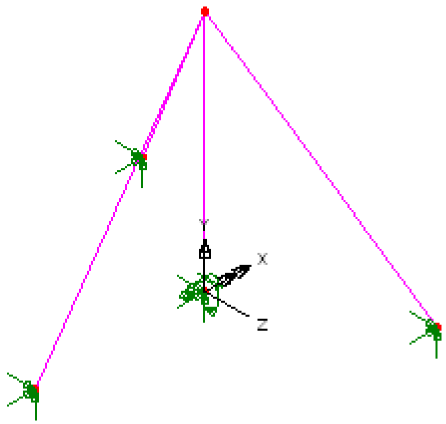
- Select material **Mild Steel**, **Ungraded**, and click **OK**.
- Select the whole model (**Control + A** keys) and drag and drop the material attribute **Iso1 (Mild Steel Ungraded)** from the Treeview onto the selected features.

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 3 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 Load, and a point load acting at the top of the structure. The loading will be assigned to four 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.

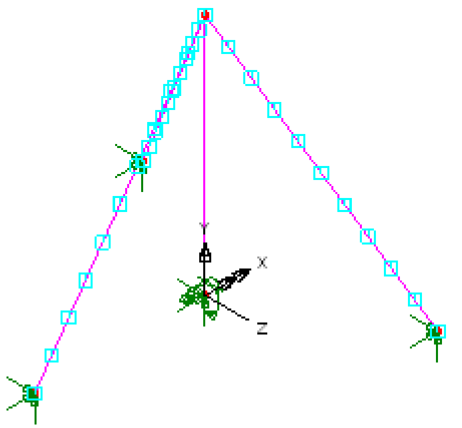
An initial force in the guy cables of 5000 N is to be defined and assigned to the model. Since the elements used to represent the mast and cables are BTS3 elements which are formulated using forces and strains (see the *Element Reference Manual* for details) the force is entered by selecting the Stress and Strain option on the structural loading dialog.

Attributes
Loading…

- Select the **Stress and Strain** option and click **Next**
- Select a Line element description
- Select **Thick nonlinear 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…

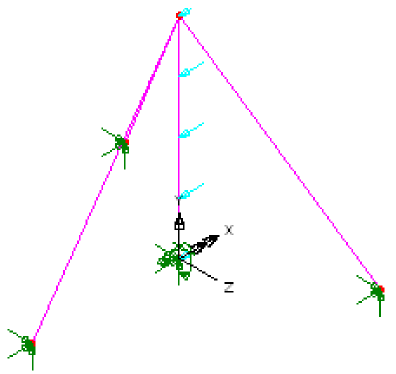
- Name the loadcase **Self Weight**
- Select the **Automatically add gravity to this loadcase** option and press **Apply**.
- Change the name to **Self Weight Freeze** and press **OK** to create the second self-weight load type.

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 **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.



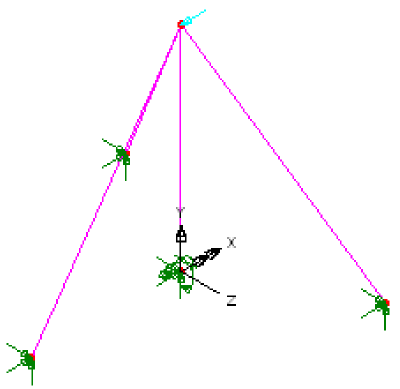
- In the Treeview, right-click on the group name **Mast** and select the **Paste** option.
- On the Loading Assignment dialog change the loadcase name to **Wind and point Load**. Click **OK** to assign loadcase.
- Click **OK** to assign the loadcase.

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 as **Wind and Point Load** 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 select **Nonlinear & Transient** from the **Controls** menu.
- 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 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.
- Enter the **Stiffness ratio to switch to arc length** as **0**. 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 select **Nonlinear & Transient** from the **Controls** menu.
- 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.

- Enter the **Stiffness ratio to switch to arc length** as **0**. 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.
- Set the **Maximum number of line searches** to **0** and click **OK**
- Click the **OK** button to finish.

Defining a Geometric Nonlinear analysis

One additional setting is 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.
- 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.

File
Save

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.

During the analysis 2 files will be created:

- **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 and the results for the first loadcase will be set active by default.

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.

- ❑ **mast_modelling.vbs** carries out the modelling of the example.

File
New…

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**

File
Script >
Run Script...

To recreate the model, select the file **mast_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

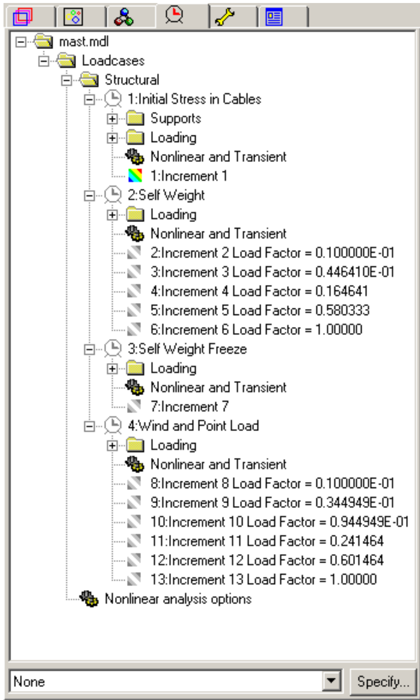
Rerun the analysis to generate the results

Viewing the Results

Selecting the results to be viewed

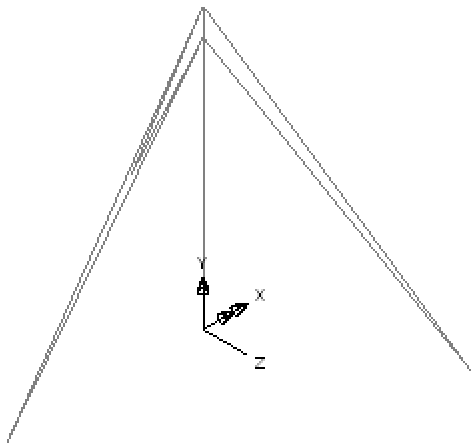
The loadcase results for the first loadcase (in this case the initial stress in cables) are set to be active by default.

Note. With reference to the 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 on the pre-stressed structure.

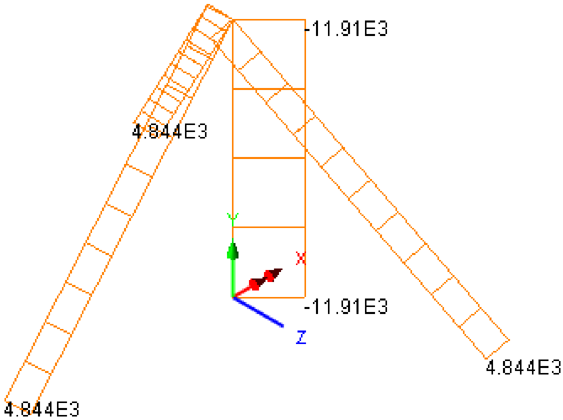


Deformed Mesh Plot

- If present in the Treeview turn off the **Geometry**, and **Attribute** layers.
- Click **Close** to accept the default properties and view the deformed mesh for the first results loadcase **Initial Stress in Cables**
- Turn off the **Mesh** layer



Axial Force Diagram for Initial Stresses



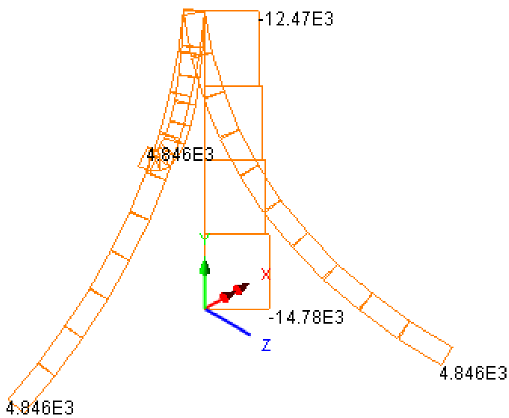
- With no features selected, click the right-hand mouse button in a blank part of the graphics window and select the **Diagrams** option to add the diagrams layer to the Treeview.

- Select the entity **Force /Moment - Thick 3D Beam** of component **Fx** (axial force)

- Select the **Diagram Display** tab and select the **Label values** and **Peak values** option. Untick the **Also use for labels** option and set the number of significant figures to **4**.

- Click **OK** to add the Diagrams layer to the display.

Combined Initial Stress and Self weight



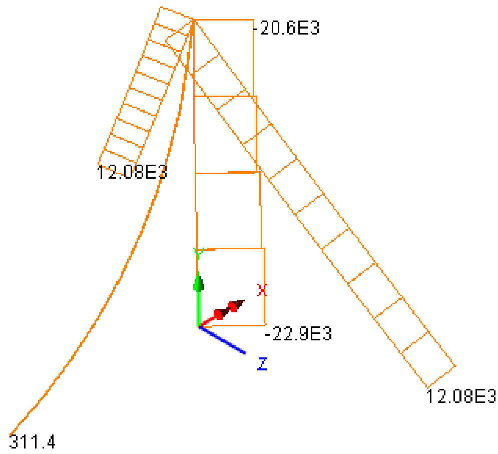
- In the Treeview right-click on the on the last results increment inside the **Self Weight** section for **Load Factor = 1.00000** and select the **Set Active** option.

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

Combined Initial Stress, Self Weight and Wind Load

- 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.

This shows axial force for the combined effect of the initial stress, the self-weight and the wind loading on the mast.



Axial Force Diagram for the mast only

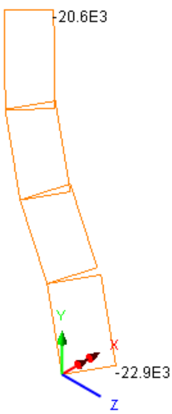
- In the Treeview, right-click on the group name **Mast** and select the **Set as Only Visible** option.

Shear Force Diagram for the mast

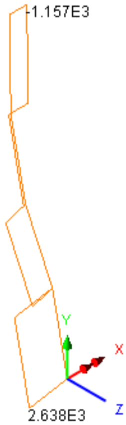
- In the Treeview double-click on the **Diagrams** entry and select **Force/Moment – Thick 3D Beam** results for axial force **Fy**. Click **OK** to update the graphics window.

Bending Moment Diagram

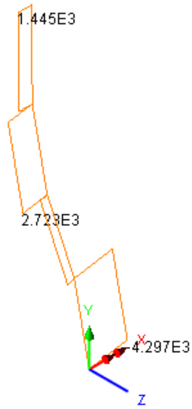
- In the Treeview double-click on the **Diagrams** entry and select **Force/Moment – Thick 3D Beam** results for bending moment **Mz**. Click **OK** to update the graphics window.



Axial force Fx



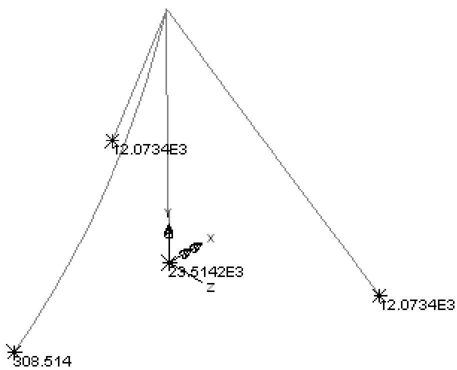
Shear force Fy



Bending moment Mz

Plotting Reactions at the Mast and Cable Supports

- In the Treeview turn off 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.

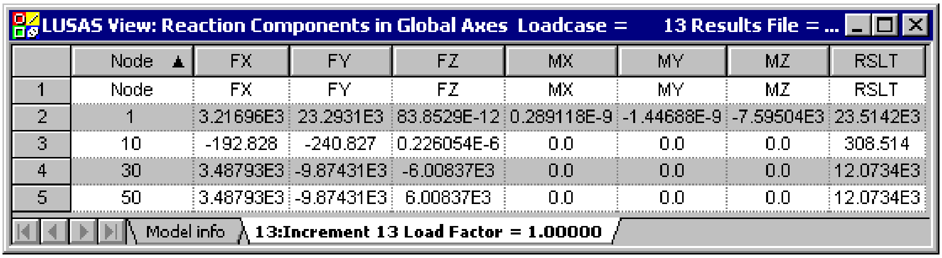
Printing a Results Summary of Reactions

With the last results increment for a load factor of 1.0 still selected in the Treeview:

Utilities
Print results wizard

- Select **Active**
- Select entity **Reaction** for type **Component**.

- Click **Finish** to view the tabulated results summary


LUSAS View: Reaction Components in Global Axes Loadcase = 13 Results File = ...

	Node	FX	FY	FZ	MX	MY	MZ	RSLT
1	Node	FX	FY	FZ	MX	MY	MZ	RSLT
2	1	3.21696E3	23.2931E3	83.8529E-12	0.289118E-9	-1.44688E-9	-7.59504E3	23.5142E3
3	10	-192.828	-240.827	0.226054E-6	0.0	0.0	0.0	308.514
4	30	3.48793E3	-9.87431E3	-6.00837E3	0.0	0.0	0.0	12.0734E3
5	50	3.48793E3	-9.87431E3	6.00837E3	0.0	0.0	0.0	12.0734E3

Model info | 13:Increment 13 Load Factor = 1.00000

Note. The values of FZ, MX and MY for node 1 vary slightly from those shown from machine to machine. These values are effectively zero.

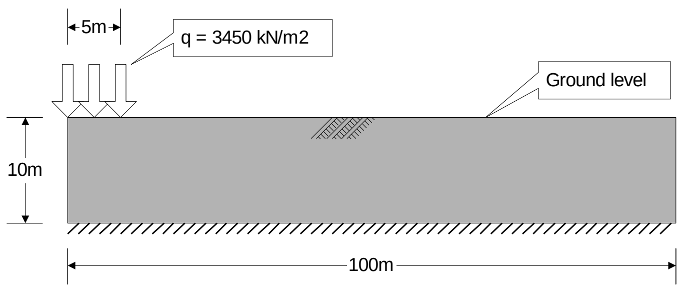
This completes the example.

2D Consolidation under a Strip Footing

For software product(s):	Any Plus version.
With product option(s):	Nonlinear.
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

The pore water pressure dissipation and settlement in a soil following the application of a distributed load is to be investigated.



Young's modulus of the soil	$E = 34.7 \times 10^3$ kN/m^2
Poisson's ratio	$\upsilon = 0.35$
Density	$\rho = 1.9$ t/m^3
Bulk modulus of water	$K_w = 2.2 \times 10^6$ kN/m^2
Void ratio	$e = 1.174$
Unit weight of water	$\gamma_w = 9.812$ kN/m^3
Soil permeability	$k_x = k_y = kz = 1.0 \times 10^{-8}$ m/s

Horizontal displacement at the left edge is restrained to model the symmetrical boundary condition. Vertical displacement at the base of the soil is restrained as the soil rests on solid rock. The pore pressure at ground level is prescribed to zero.

Units of kN, m, t, s, C are used throughout.

Objectives

The required output from the analysis consists of:

- ❑ Settlement at the centre of the footing with time.
- ❑ Pore pressure distribution immediately after application of the load (undrained response)
- ❑ Pore water pressure dissipation with time at the centre of the footing

Keywords

Consolidation, Pore Water Pressure, Time Stepping, Nonlinear, Transient, Settlement, Two Phase Materials, Default Assignments, Graphing

Associated Files

- ❑ **pwp_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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 the File name as **pwp**
- Use the **Default** working folder.
- Enter the title as **Consolidation under a Strip Footing**
- Select model units of **kN,m,t,s,C** from the drop down list provided.
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Select the startup template **Standard**
- Select the **Vertical Y axis** option and click **OK**.

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.

Prior to defining the geometry of the model default surface mesh and material properties will be set-up.

Mesh Definition

Since this analysis requires the modelling of pore water pressure, plane-strain two phase elements will be used.

Attributes
Mesh >
Surface...

- Select **Plane strain two phase, Quadrilateral, Quadratic** elements. Ensure the **Regular mesh** option is selected with **Automatic divisions** so that Modeller uses the mesh divisions assigned to each line.
- Give the attribute the name **Plane Strain Two Phase** and click the **OK** button to add the mesh attribute to the Treeview.
- In the Treeview click on the mesh attribute **Plane Strain Two Phase** with the right-hand mouse button and select the **Set Default** option**.** This will ensure all newly created surfaces will be assigned the elements defined in this mesh attribute.

Material Properties

For consolidation analysis both the elastic and two-phase soil properties need to be defined. The overall 'equivalent' bulk modulus of the soil is related to the bulk modulus of the pore fluid and the bulk modulus of the solid soil particles by the formula:

$$\frac{1}{Ke} = \frac{n}{Kf} + \frac{(1-n)}{Ks} \approx \frac{n}{Kf}$$

Where

Ke is the equivalent bulk modulus of the soil

Kf is the bulk modulus of the pore fluid

Ks is the bulk modulus of the solid soil particles

n is the porosity of the soil

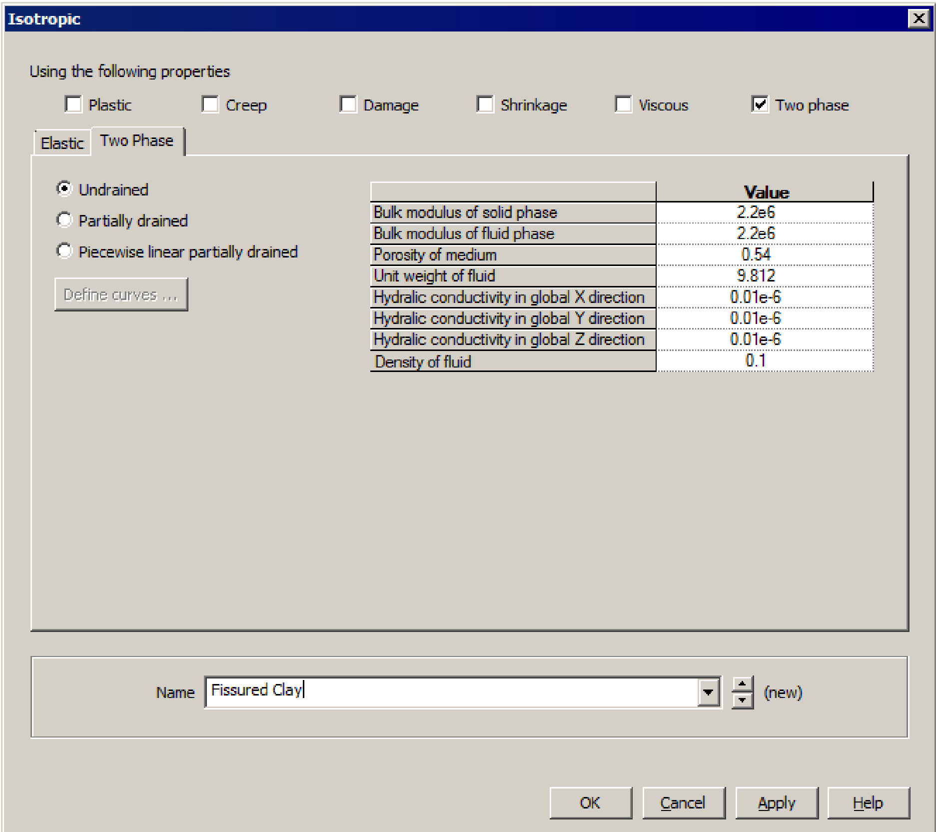
The porosity of the soil is related to the void ratio by the formula:

Porosity $n = \frac{e}{1+e}$ = 0.54

Attributes
Material >
Isotropic...

- Enter the isotropic material properties for the soil as Young's Modulus **34.7E3**, Poisson's Ratio **0.35** and Mass density **1.9**

- Click on the **Two phase** check box at the top right-hand corner of the dialog and select the **Undrained** option.



- Enter the properties as shown in the dialog above..

Note. For the purposes of this example and to comply with a corresponding test case the bulk modulus the undrained bulk modulus of the soil is set to be the same as that of the pore fluid.

- Enter the attribute name as **Fissured Clay** and click the **OK** button to add the attribute to the Treeview.

- In the Treeview click on the attribute **Fissured Clay** with the right-hand mouse button and select the **Set Default** option**.** This will ensure all newly created features will be assigned the properties defined in this material attribute.

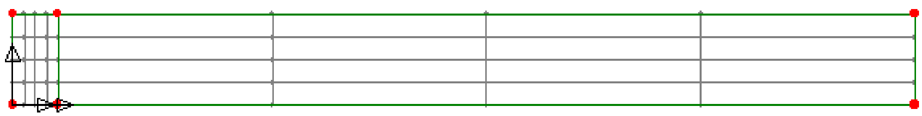
Feature Geometry

Geometry
Surface >
Coordinates...

Enter coordinates of (**0, 0**), (**5, 0**), (**5, 10**) and (**0, 10**) to define the soil under the load. Use the Tab key to move to the next entry field on the dialog. When all coordinates have been entered click the **OK** button.

Geometry
Surface >
Coordinates...

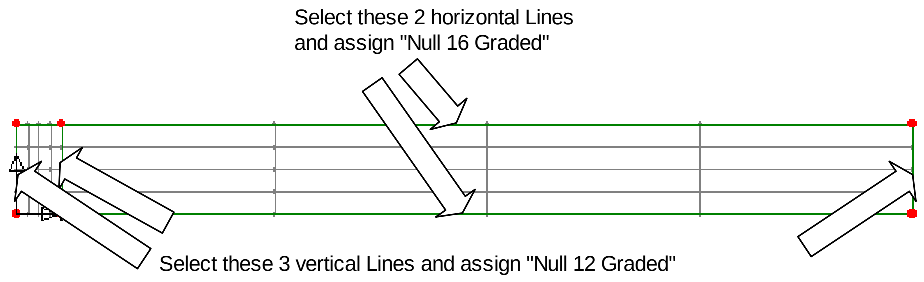
Enter the coordinates of **(5, 0), (100, 0), (100, 10), (5, 10)** to define the remainder of the soil.



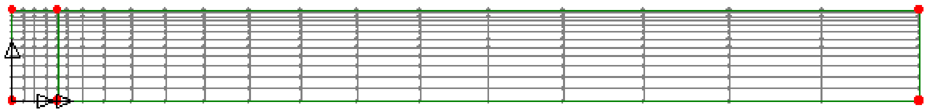
Mesh Grading

Attributes
Mesh >
Line...

- Define a Line mesh with element type **None** and **16** divisions.
- Select the **Spacing** button.
- Choose the **Uniform transition** option and set the **Ratio of first to last element** to **0.1**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 16 Graded** and click the **Apply** button to add the attribute to the Treeview and keep the dialog active.
- Define a Line mesh with **12** divisions.
- Select the **Spacing** button.
- Enter the **Uniform transition** option and set the **Ratio of first to last element** to **5**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 12 Graded** and click **OK** to add the attribute to the Treeview.



Select the horizontal lines of the larger surface as shown and assign the mesh attribute **Null 16 Graded**



- Select all the vertical lines as shown (Use the **Shift** key to add to the initial selection) and assign the mesh attribute **Null 12 Graded**

Note. If the mesh is graded with the smaller elements at the wrong end of a line reverse the line by selecting the line, and using the **Geometry>Line>Reverse** menu.

Supports

Define a support to prescribe zero pore pressure.

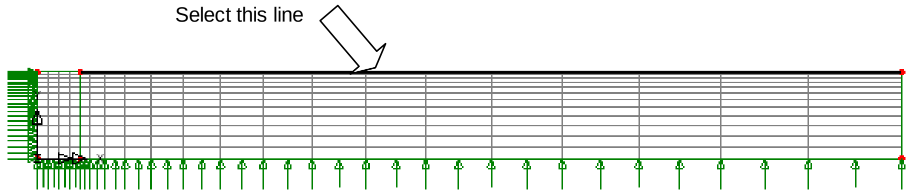
Attributes
Support…

- Click on the option to make the Pore pressure **Fixed**
- Enter the attribute name as **Fixed PWP** and click the **OK** button.

Assign the supports to the model.

- Select the vertical Line on the left-hand side of the model and drag and drop the support attribute **Fixed in X** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**
- Select the 2 horizontal Lines representing the base of the soil and drag and drop the support attribute **Fixed in Y** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**

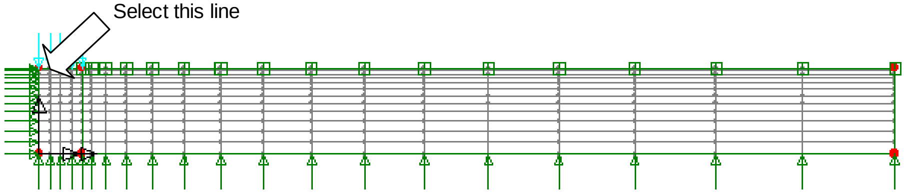
- Select the Line at ground level on the right-hand side of the model (see image) and drag and drop the support attribute **Fixed PWP** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**



Loading

Attributes
Loading...

- Select the **Face** option and click **Next**
- Enter loading of **3450** in the **y Direction**
- Enter the attribute name of **Distributed Load** and click the **Finish** button.
- Select the top line on the left-hand side of the model and drag and drop the **Distributed Load** attribute onto the selection. With the **Assign to lines** option selected click **OK** to assign the loadcase attribute to **Loadcase 1**.



Analysis Control

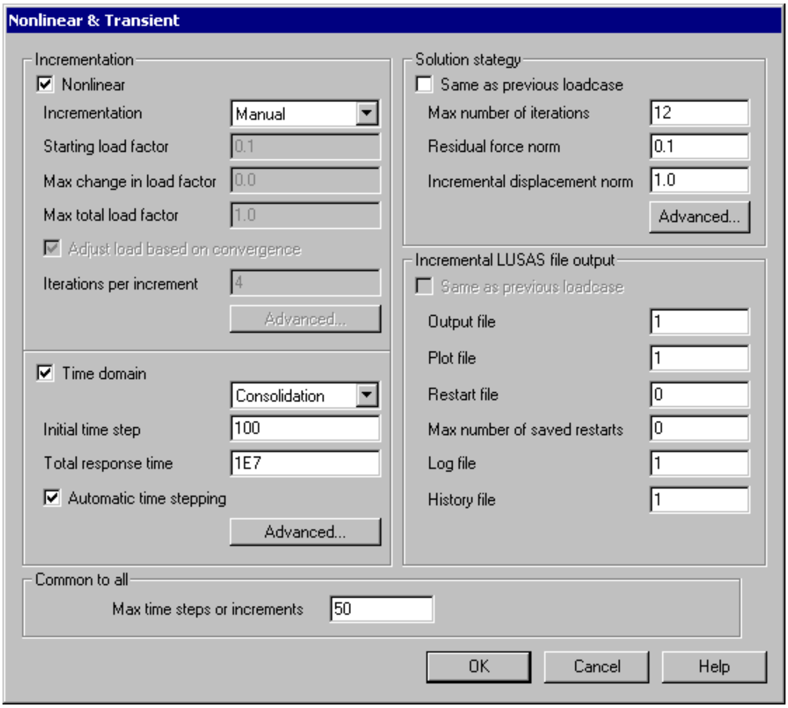
With this consolidation problem an automatic time stepping procedure is adopted. This is because consolidation is a typical diffusion process in which the field changes rapidly at the start of the process before settling down to a steady state condition a considerable time after the initial load is applied. The automatic time stepping procedure enables the time step to be modified so that the small time steps required at the start of the problem can be increased as the analysis progresses. In some cases the overall response time can be orders of magnitude larger than the initial time step.

The initial time step is important since the early variations in pore pressure must be accurately accounted for. Vermeer and Verruijt suggest the following criteria for determining the initial time step where Δh is the minimum distance between nodes.

$$\Delta t \geq \frac{\gamma_{\omega}}{6Ek}(\Delta h)^2 = \frac{9.812}{6 \text{ x } 34.7E3 \text{ x } 1E-8} \text{ x } 0.14^2 \approx 100 \text{ secs}$$

Note. The distance between two nodes can be determined by selecting the two nodes and then picking the **Utilities>Mesh>Distance between Nodes** entry.

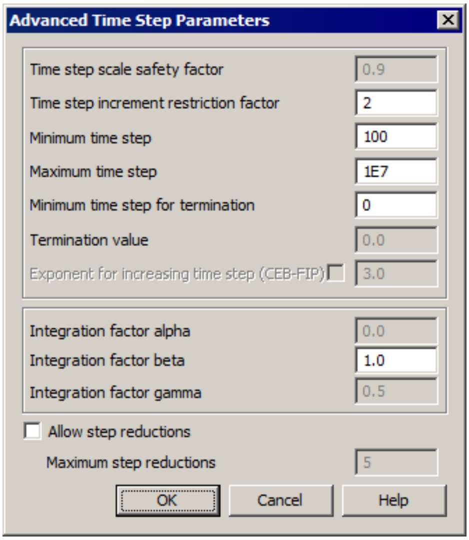
- In the Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Nonlinear & Transient** from the **Controls** menu.



In the Incrementation section:

- Select the **Nonlinear** option and choose **Manual** incrementation.
- Select the **Time domain** option and chose **Consolidation** from the drop down list.
- Enter an **Initial time step** of **100**
- Select the **Automatic time stepping** option.
- Select the **Advanced** button in the Time domain section of the dialog.

- On the Advance time step parameters dialog set the **Time step increment restriction factor** to **2**
- Set the **Minimum time step** to **100**
- Set the **Maximum time step** to **1E7**
- Set the **Minimum time step for termination** to **0**
- Click **OK** to return to the Nonlinear and Transient control dialog.
- On the Nonlinear and Transient control dialog set **Max time steps or increments** to **50**
- Click **OK** to set the loadcase control.



Saving the model

File
Save

Save the model file.

Running the Analysis

Open the **Solve Now** dialog. Ensure A**nalysis 1** is selected 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 Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

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

If the analysis fails...

If the analysis fails, the output file will provide information relating to the nature of the error encountered. Any errors listed in the output file should be fixed 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.

- ❑ **pwp_modelling.vbs** carries out the modelling of the example.

File
New…

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 **pwp**

File
Script >
Run Script...

- To recreate the model, select the file **pwp_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

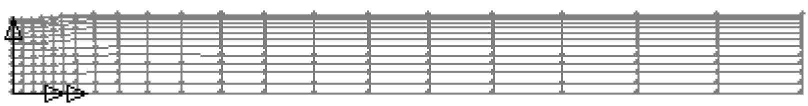
Viewing the Results

Loadcase results can be seen in the Treeview and results for the first time step are set active by default.

- If present, turn off the **Geometry**, **Mesh** and **Attributes** layers in the Treeview.

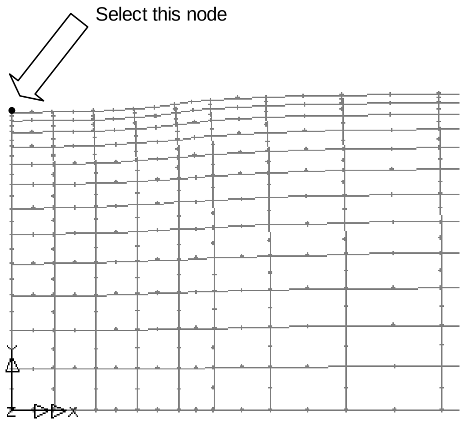
Settlement

- Click the **Deformations** button in the Treeview and select the **Specify factor** option. Specify a factor of **1** and click **OK** to visualise the deformed mesh for the first time step.



A graph of the deformation over time will be created using the graph wizard.

- Zoom into the left-hand side of the model and select the node on the centre line at the centre of the footing.



Utilities
Graph Wizard...

- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis.

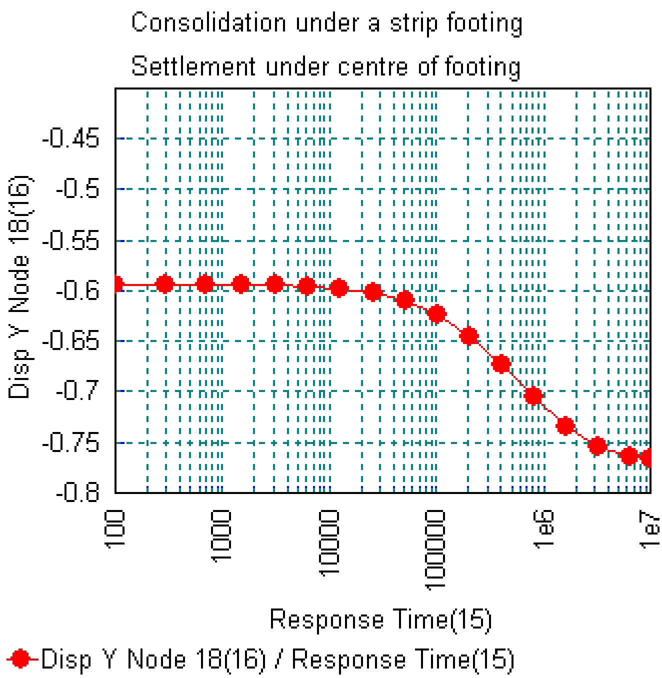
- Selected the **Named** option and click **Next**
- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis.

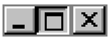
- With the **Nodal** option selected and click **Next**
- Select **Displacement** from the Entity drop down list and **DY** from the Component drop down list.
- The selected node number will appear in the **Specified single node** drop down list. Click **Next**

For the X Scale enter **Manual** values of minimum **100** and maximum **10e6** respectively.

- Select the **Use logarithmic scale** option.
- Title the graph as **Settlement under centre of footing**
- Click **Finish** to display the graph of settlement over time under the centre of the strip footing.

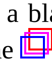


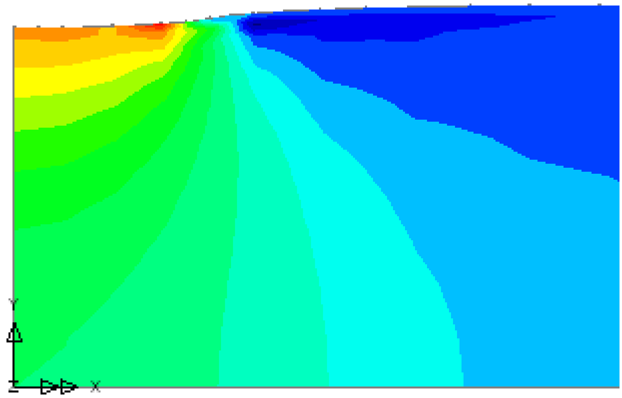
 Close the graph window

 Maximise the graphics window.

Pore water pressure

The distribution of pore pressure is to be shown using contours.

- With no features selected click the right-hand mouse button in a blank part of the graphics area and select **Contours** to add the contours layer to the  Treeview.
- Select **Displacement** from the Entity drop down list and **PRES** from the component drop down list.
- Select the **Contour Range** tab and set the **Interval** to **250**
- Click the **OK** button to display contours of the undrained pore pressure distribution (Time step 0) immediately after the loading is applied.



Note. The dissipation of pore water pressure over time may be observed by creating an animation of the contour display.

- To observe the distribution of pore water pressure at a particular time after the application of loading activate the appropriate time step from the Treeview by selecting the time step with the right-hand mouse button and choosing the **Set Active** option.

The dissipation of pore water pressure under the footing is to be presented on a graph.

- Select the node on the centreline under the footing as before.

Utilities
Graph Wizard...

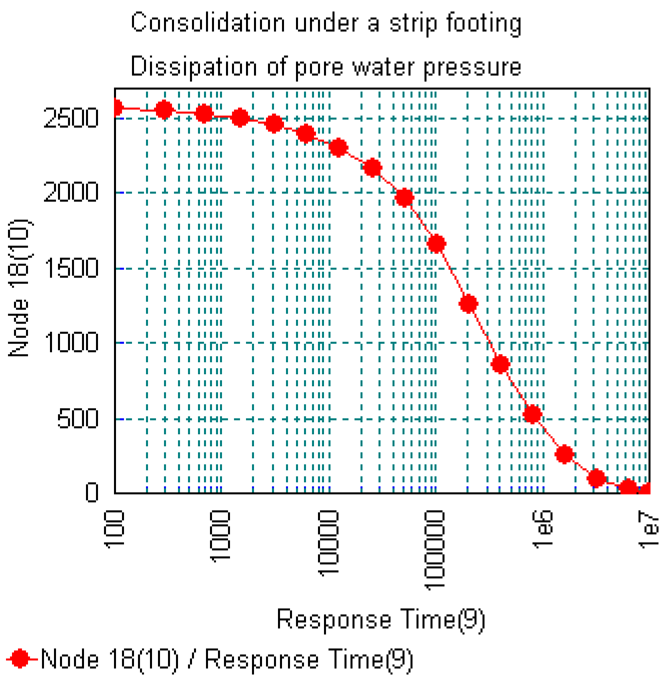
- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis

- Selected the **Named** option and click **Next**
- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis

- With the **Nodal** option selected and click **Next**
- Select **Displacement** from the Entity drop down list and **PRES** from the Component drop down list.
- The extent to be graphed will be set to **Specified single node** and the node selected will be seen in the Specify node field. Click **Next**



- For the X Scale ensure that **Manual** values of minimum **100** and maximum **10e6** respectively are used.
- Select the **Use Logarithmic scale** option
- For the Y Scale leave the values as the defaults.
- Title the graph as **Dissipation of pore water pressure**
- Click **Finish** to display the dissipation of pore water pressure over time under the centre of the strip footing.

Close the graph window.

Maximise the graphics window.

As the pore water dissipates the load is carried by the soil. The increase in effective stress in soil can be observed on a graph of effective stress against response time.

- Ensure the node on the centreline under the footing still selected.

Utilities
Graph Wizard...

- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis

- Select the **Named** option and click **Next**
- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis

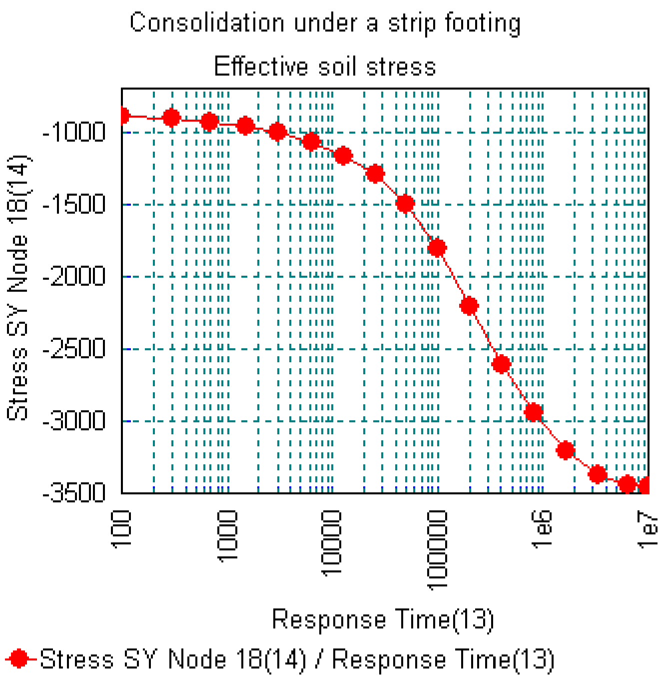
- With the **Nodal** option selected and click **Next**
- Select **Stress - Plane Strain** from the Entity drop down list and **SY** from the Component drop down list.

- The extent to be graphed will be set to **Specified single node** and the node selected will be seen in the Specify node field. Click **Next**

- For the X Scale ensure that **Manual** values of minimum **100** and maximum **10e6** respectively are used.

- Select the **Use Logarithmic scale** option.

- For the Y Scale leave the values as the defaults.



- Title the graph as **Effective soil stress**

- Click **Finish** to display the graph of the effective soil stress over time under the centre of the strip footing.

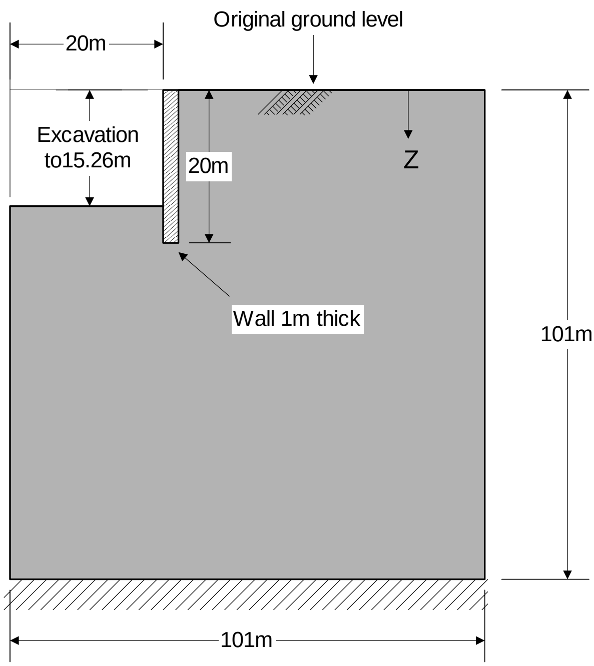
This completes the example.

Drained Nonlinear Analysis of a Retaining Wall

For software product(s):	Any Plus version.
With product option(s):	Nonlinear
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

The behaviour of a single propped retaining wall is to be investigated in the long term. Drained soil parameters are used throughout.



Horizontal displacement is restrained at the left and right hand edge and both horizontal and vertical displacement is restrained at the base of the soil as the soil rests on solid rock.

The wall is 'wished-in-place' in that construction of the wall is not modelled.

Units of kN, m, t, s, C are used throughout.

Objectives

The required output from the analysis consists of:

- ❑ Displacement of the wall toe towards the excavation
- ❑ Surface heave immediately behind the wall
- ❑ Contours of plastic strain around the excavation
- ❑ Prop force per m length of wall
- ❑ Bending moment in the wall at 10m depth

Keywords

Retaining wall, Drained, Nonlinear, Excavation, Surface heave, Plane strain, Prop force, Bending Moment

Associated Files

- ❑ **drained_wall_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

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

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.

Creating a new model

- Enter the File name as **drained_wall**.
- Use the **Default** working folder.
- Enter the title as **Drained analysis of a propped retaining wall**.
- Select model units of **kN,m,t,s,C** from the drop down list provided.
- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Set the startup template as **None**
- Select the **Vertical Y axis** option and click **OK**.

Note. Save the model regularly as the example progresses. Use the undo command to undo mistakes as far back as the last save.

Mesh Definition

The surfaces are to be meshed using plane-strain quadrilaterals with a quadratic interpolation order (QPN8).

Attributes
Mesh >
Surface...

- Select the **Plane Strain, Quadrilateral, Quadratic** elements. Ensure the **Regular mesh** option is selected with **Automatic divisions** so that Modeller uses the mesh divisions assigned to each line.
- Give the attribute the name **Plane Strain quads** and click the **OK** button to add the mesh attribute to the Treeview.
- In the Treeview click on the mesh attribute **Plane Strain quads** with the right-hand mouse button and select the **Set Default** option**.** This will ensure all newly created surfaces will be assigned the elements defined in this mesh attribute.

General Field Variations

Utilities
Variation...

The Young's Modulus of the soil and the in situ stresses vary with depth and therefore general field variations are required.

- Select **General Field Variation** and press **Next**.
- Enter the function as **-6000*Y** and the attribute name as **y_mod** for the definition of Young's Modulus with depth and click **Apply** to add the attribute to the Treeview and keep the dialog active.
- Enter a new function as **20*Y** and the attribute name as **sig_v** for the definition of vertical stress and click **Apply.**
- Enter a new function as **2*20*Y** and the attribute name as **sig_h** for the definition of horizontal stress and click **Finish.**

Note. The LUSAS convention assumes that negative stresses are compressive. The Y coordinate datum is at ground level and so increasing depth will lead to increasingly negative Y coordinates. The variations as entered will ensure that stresses increase negatively with depth while the Young's Modulus increases positively with depth.

Feature Geometry

- Turn off the display of the **Mesh** layer in Treeview.

Geometry
Line >
Coordinates...

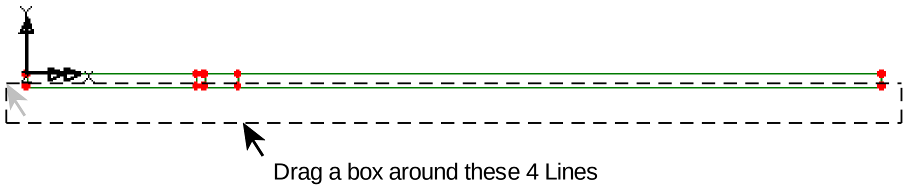
Enter coordinates of (**0, 0**), (**20, 0**), (**21, 0**), (**25, 0**) and (**101, 0**) to define four lines representing the original ground level (all Z coordinates should be zero or left blank). Use the Tab key to move to the next entry field on the dialog. When all coordinates have been entered click the **OK** button.

- Press **CTRL-A** to select all the Lines just drawn.

Geometry
Surface >
By Sweeping...

Ensure the **Translate** option is selected and enter a value of **–1.5** in **Y.**

- Click **OK** to sweep the Lines to create the Surfaces.



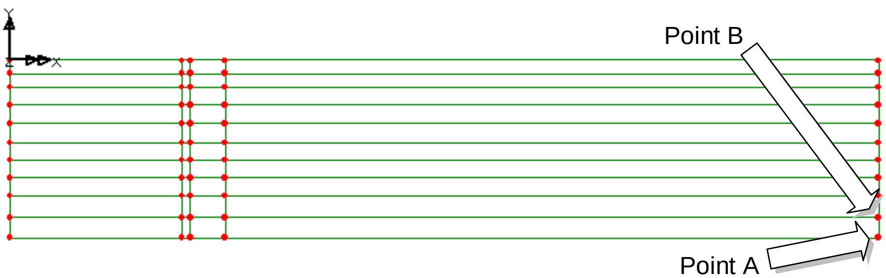
- Select the bottom four horizontal Lines by dragging a box around them.

Geometry
Surface >
By Sweeping...

Ensure the **Translate** option is selected and enter a value of **–1.5** in **Y.**

- Click **OK** to sweep the Lines to create the Surfaces.

Repeat the preceding commands eight times, selecting the bottom four lines of the model each time and sweep the lines by **-2.0, -2.13, -2.13, -2.0, -2.0, -2.0, -2.49** and **-2.25** metres in Y as shown above.



- Select Point A shown.

Geometry
Point >
Move...

Ensure the **Translate** option is selected and enter a value of **–81** in **Y.**

- Click **OK** to move the Point.

- Select Point B shown.

Geometry
Point >
Move...

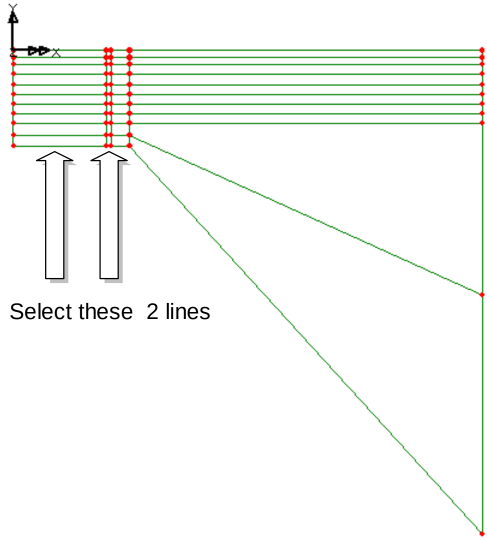
Ensure the **Translate** option is selected and enter a value of **–33.25** in **Y.**

- Click **OK** to move the Point.
- Select the two Lines shown.

Geometry
Surface >
By Sweeping...

Ensure the **Translate** option is selected and enter a value of **–81** in **Y.**

Click **OK** to sweep the Lines to create the Surfaces.

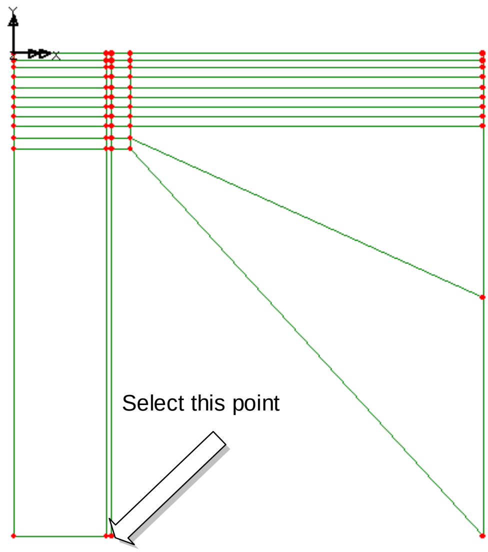


- Select the Point shown.

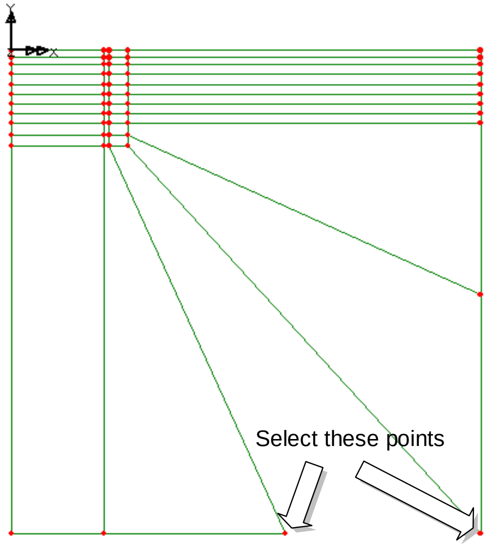
Geometry
Point >
Move...

Ensure the **Translate** option is selected and enter a value of **38** in **X.**

- Click **OK** to move the Point.



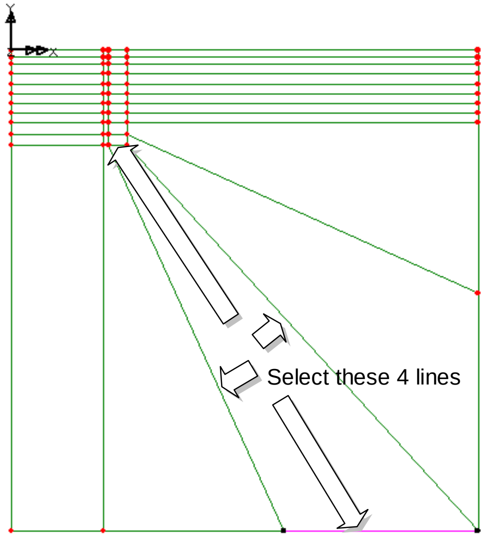
- Select the two Points shown.



Geometry
Line >
Points...

Create a Line between the two selected Points.

- Select the Line just drawn along with the three Lines as shown.



Geometry
Surface >
Lines...

Create a Surface from the four selected Lines.

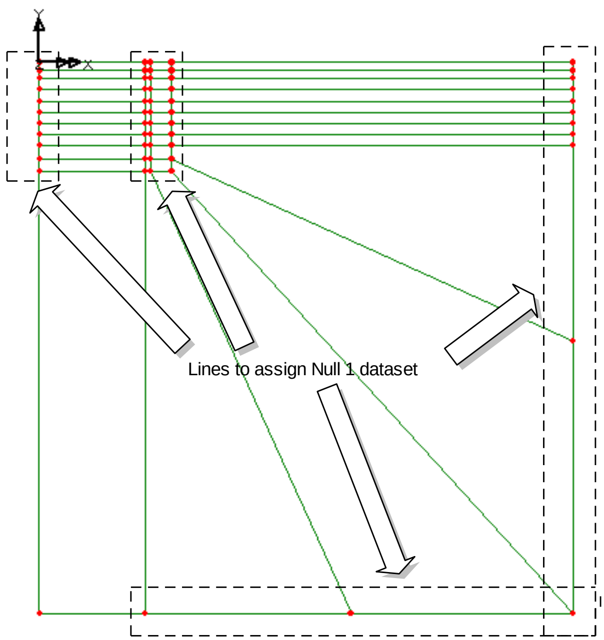
Mesh Grading

Attributes
Mesh >
Line...

- Define a Line mesh with element type **None** and **1** division.
- Enter the attribute name as **Null 1** and click the **Apply** button to add the attribute to the Treeview and keep the dialog active.

- Define a Line mesh with **5** divisions.
- Select the **Spacing** button.
- Select the **Uniform transition** option and set the **Ratio of first to last element** to **4**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 5 Graded** and click **Apply** to add the attribute to the Treeview and keep the dialog active.
- Define a Line mesh with **10** divisions.
- Select the **Spacing** button.
- Select the **Uniform transition** option and set the **Ratio of first to last element** to **0.2**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 10 Graded** and click **OK** to add the attribute to the Treeview.

Assigning the mesh attributes



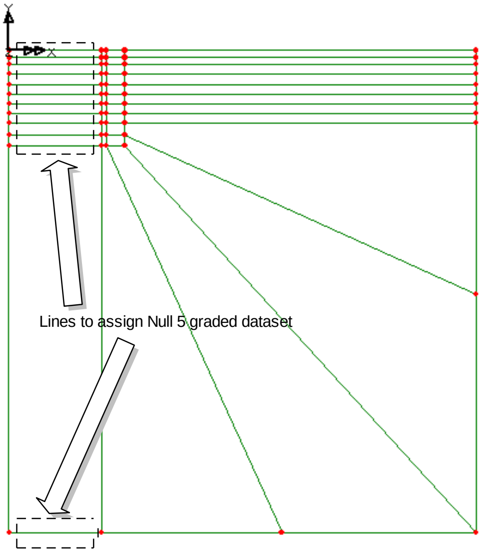
- Box-select the lines as shown and assign the mesh attribute **Null 1.**

Note. The selection of some of the sets of lines as shown in the next two images is best done by changing the selection cursor and additionally using a special keyboard shortcut which causes any lines passing through a box-selection to be selected.

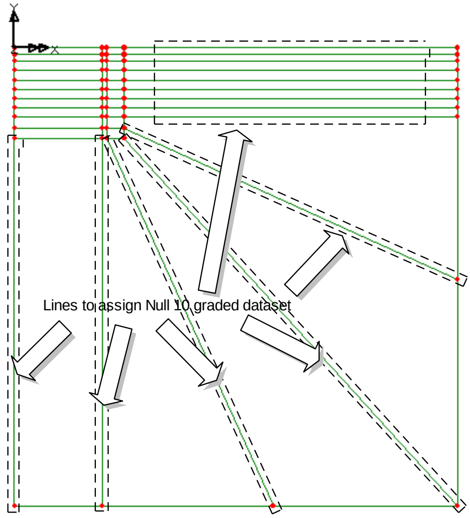
To prepare for this:

 Change the cursor to the Select Lines cursor.

- Hold down the **ALT** key and box-select each set of horizontal lines as shown in the preceding image and assign the mesh attribute **Null 5 Graded.**

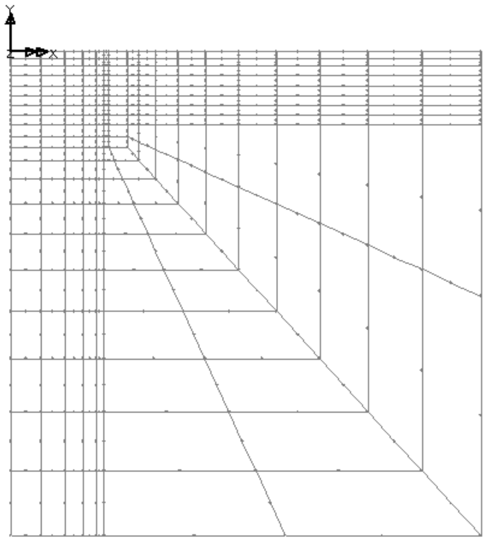


- Select the remaining lines as shown and assign the mesh attribute **Null 10 Graded.**



Note. See the *Keyboard Shortcut Guide* for more details on the use of keyboard shortcuts.

- Turn on the **Mesh** layer in the Treeview to see the mesh.



Note. If the mesh is graded with the smaller elements at the wrong end of a line reverse the line by selecting the line using the **Geometry>Line>Reverse** menu selection. If errors have been made in mesh assignments simply re-select any lines with an incorrect mesh assignment and re-assign the correct line mesh.

Material Properties

An isotropic elastic material will be used for the retaining wall while an isotropic nonlinear material utilising the Mohr-Coulomb failure surface will be used for the soil.

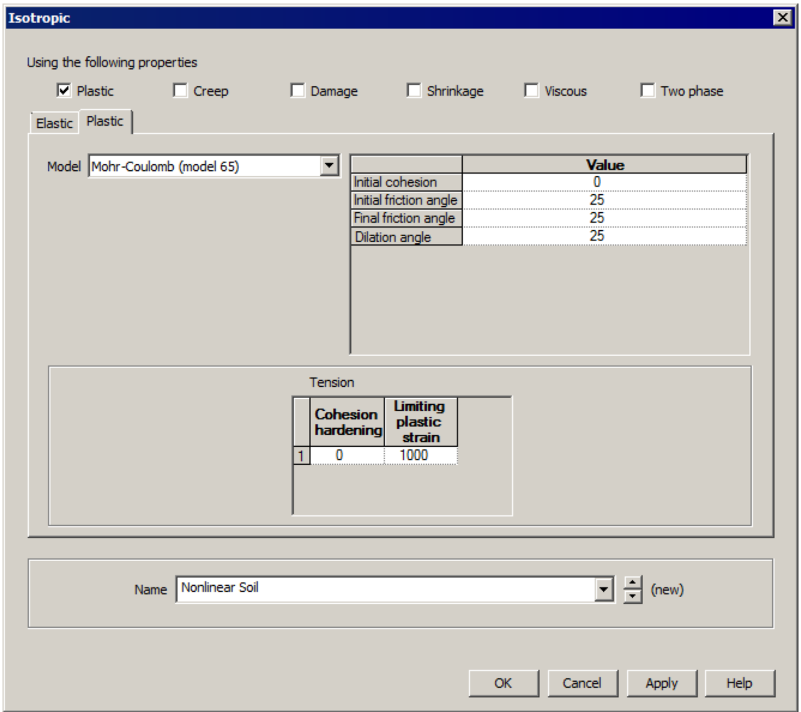
Attributes
Material >
Isotropic...

With the **Elastic** tab selected enter the isotropic material properties for the wall as Young's Modulus **28E6** kN/m^2, Poisson's Ratio **0.15** and Mass Density **2.03874** tonne/m^3.

- Enter the attribute name as **Concrete Wall.**
- Click the **Apply** button to add the attribute to the Treeview and keep the dialog active.
- With the **Elastic** tab selected enter the isotropic material properties for the soil as Young's Modulus **1*y_mod**, Poisson's Ratio **0.2** and Mass Density **2.03874** tonne/m^3.

Note. The **1*y_mod** indicates that the Young's Modulus is defined by a variation scaled by a factor of **1**. As an alternative to typing in the formula, enter **1** in the grid location and click on the button. Select the **y_mod** variation from the drop down list.

- Click the **Plastic** option and from the Model drop-down list select the **Mohr-Coulomb (model 65)** entry.
- Enter an **Initial cohesion** value of **0**
- Enter an **Initial friction angle** value of **25**
- Enter an **Final friction angle** value of **25**
- Enter a **Dilation angle** value of **25**
- In the tension section enter a **Cohesion hardening** value of **0**
- In the tension section enter a **Limiting plastic strain value** of **1000**
- Enter the attribute name as **Nonlinear Soil**
- Click the **OK** button to add the attribute to the Treeview.

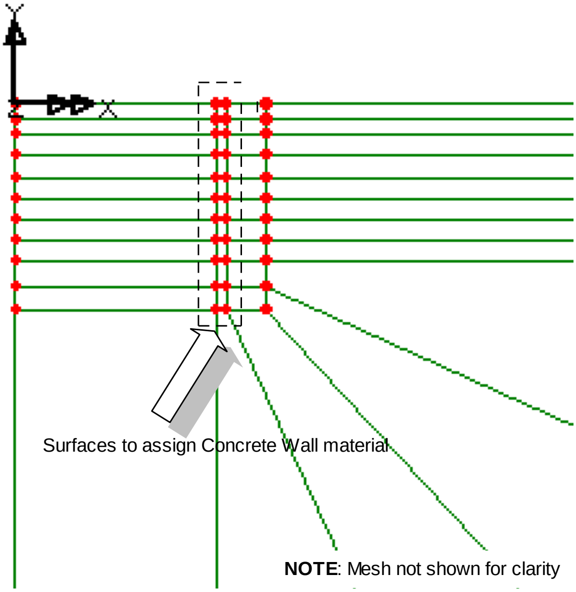


Assigning material properties

To assign the material properties groups will be used. Groups are very useful in allowing features of the model to be turned-on and off to aid the assignment of attributes or the viewing of results on selected features of the model.

 Change the cursor back to the normal cursor.

- Select the ten surfaces that form the retaining wall as shown.



Geometry
Group >
New Group...

Create a group containing the wall elements only and rename it **Wall**. Click **OK**.

- With the ten surfaces of the wall still selected drag and drop the material attribute **Concrete Wall** from the Treeview onto the selection. With the **Assign to surfaces** option selected click **OK** to assign the material attribute.
- In the Treeview right-click on the **Wall** group and select **Invisible**.
- Select the whole visible model by pressing the **Ctrl-A** keys.

Geometry
Group >
New Group...

Create a group containing the soil elements only and rename it **Soil**. Click **OK**.

- Drag and drop the material attribute **Nonlinear Soil** from the Treeview onto the selection. With the **Assign to surfaces** option selected click **OK** to assign the material attribute.
- In the Treeview right-click on the **Wall** group and select **Visible**.

Supports

Define supports to prescribe zero horizontal displacement as well as zero horizontal and vertical displacement.

Attributes
Support…

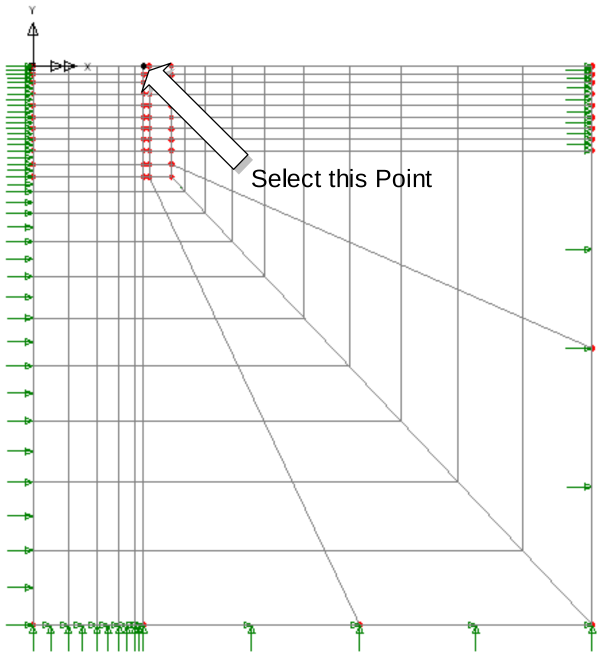
- Click on the option to fix the **Translation in X.**

- Enter the attribute name as **Fixed in X** and click the **Apply** button to add the attribute to the Treeview and keep the dialog active.
- Click on the option to fix the **Translation in Y**.
- Enter the attribute name as **Fixed in X and Y** and click the **OK** button to add the attribute to the Treeview.

Assign the supports to the model.

Select the vertical Lines on the left-hand side and right-hand side of the model and drag and drop the support attribute **Fixed in X** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All loadcases.**

- Select the 3 horizontal Lines representing the bottom limit of the soil and drag and drop the support attribute **Fixed in X and Y** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases.**
- Select the Point which represents the top-left corner of the concrete wall as shown and drag and drop the support attribute **Fixed in X** from the Treeview onto the selection. With the **Assign to Points** option selected click **OK** to assign the support attribute to **All analysis loadcases.** This support represents the prop.

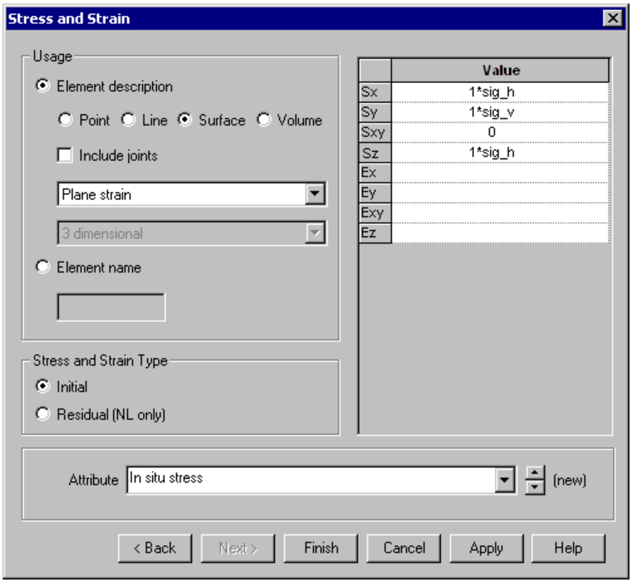


Loading

The initial in situ ground conditions must be established in the first loadcase of the analysis. This is carried out using a stress and strain type load and a gravity (body force) load.

Attributes
Loading…

- Select the **Stress and Strain** option and click **Next**



- On the Stress and Strain dialog ensure the Stress And Strain Type is set to **Initial**
- Select a **Surface** element description
- Select **Plane strain** from the drop down list
- Enter **1*sig_h** for Sx. (This can be done most easily by clicking in the right-hand side of the Sx field and selecting the variation attribute from the drop-down list)
- Enter **1*sig_v** for Sy.
- Enter **0** for Sxy
- Enter **1*sig_h** for Sz. The remaining fields may be left blank (to signify zero)
- Enter the attribute name as **In situ stress** and click **Finish.**

Attributes
Loading…

- Select the **Body force** tab from the Structural Loading dialog and click **Next**
- Enter **–9.81** in **Linear acceleration in Y Dir**.
- Enter the attribute name as **Gravity** and click **Finish**.

Assign the loads to the model.

The in-situ stress is to be assigned to the whole model.

- Select the whole model by pressing **Ctrl-A** and drag and drop the loading attribute **In situ stress** from the Treeview onto the selection. With the **Assign to Surfaces** option selected rename the loading attribute to **Loadcase 1** and click OK.

Gravity is assigned to the whole model

- With the whole model selected drag and drop the loading attribute **Gravity** from the Treeview onto the selection. With the **Assign to Surfaces** option selected click **OK** to assign the loading attribute to **Loadcase 1**

- In the Treeview drag and drop the **Geometry** layer name beneath the Attributes layer name.

Birth and Death

The **Birth and Death** feature must be used to simulate the removal of soil during excavation in front of the wall. Excavation will take place over eight loadcases.

Attributes
Birth and Death...

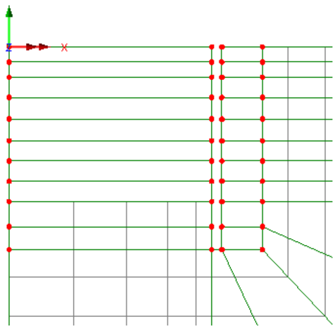
- Select the **Deactivate** tab and click **Next**. Select **Percentage to Redistribute** and ensure the value is set to **100%**. Enter the attribute name as **Excavation**.

- Click **Finish** to add the attribute to the Treeview.

Assign the deactivation to the model.

- Use the zoom-in button to enlarge the view of the top-left corner of the model.

- Select the top surface representing the first layer of soil to be removed in front of the retaining wall and drag and drop the deactivation attribute **Excavation** from the Treeview onto the selection. With the **Assign to surfaces** option selected enter the Loadcase name as **Excavation to –1.50m** and click **OK** to assign the attribute.



- Select the next surface down in front of the wall and drag and drop the deactivation attribute **Excavation** from the Treeview onto the selection. With the **Assign to surfaces** option selected enter the Loadcase name as **Excavation to –3.00m** and click **OK** to assign the attribute.

- Repeat this procedure, selecting the next surface down each time and renaming the loadcases **Excavation to –5.00m**, **Excavation to –7.13m**, **Excavation to –9.26m**, **Excavation to –11.26m**, **Excavation to –13.26m** and **Excavation to –15.26m** respectively. The adjacent image shows the final excavation assignment made.

Note. Deactivated surfaces (and hence elements) can be visualised and checked by setting each loadcase active in the Treeview.

Analysis Control

The use of a Mohr-Coulomb nonlinear material in this example dictates that a nonlinear analysis control must be specified.

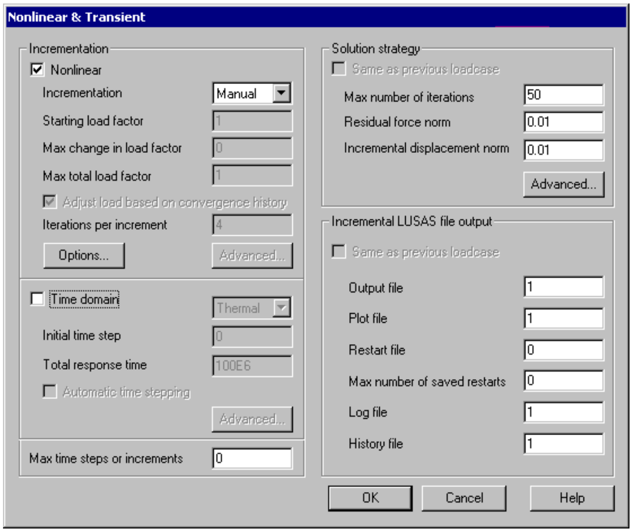
- In the Treeview right-click on **Loadcase 1** and rename it to **In situ**
- In the Treeview right-click on **In situ** and select **Nonlinear & Transient** from the **Controls** menu.

In the **Incrementation** section:

Select the **Nonlinear** option and choose **Manual** incrementation.

In the **Solution strategy** section:

- Set the **Max number of iterations** to **50**.
- Set the **Residual force norm** to **0.01**.
- Set the **Incremental displacement norm** to **0.01**.



- Click **OK** to set the loadcase control.

Saving the model

File
Save

Save the model file.

Running the Analysis

Open the **Solve Now** dialog. 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 Treeview

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

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

If the analysis fails...

If the analysis fails, the output file will provide information relating to the nature of the error encountered. A common mistake made when using LUSAS Modeller for the first time is to forget to assign particular attribute data (geometry, mesh, supports, loading etc.) to the model. Any errors listed in the output file should be fixed 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.

- ❑ **drained_wall_modelling.vbs** carries out the modelling of the example.

File
New…

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 **drained_wall**

File
Script >
Run Script...

To recreate the model, select the file **drained_wall_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results

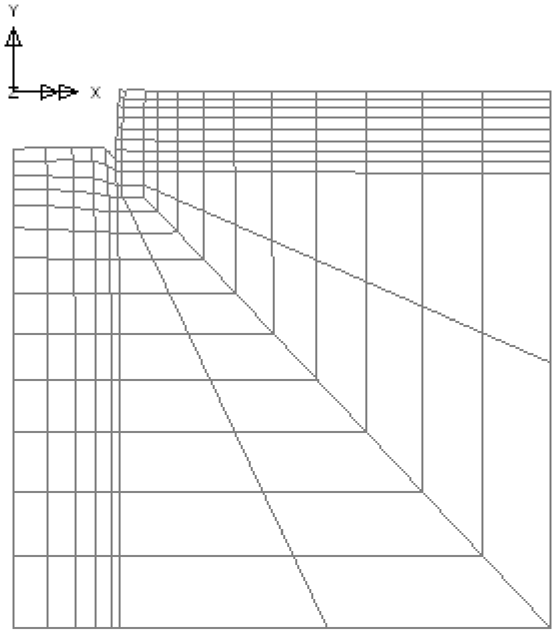
Viewing the Results

Loadcase results can be seen in the Treeview, and the load case results for the first time step (increment 1) are set active by default.

- If present, turn off the **Geometry**, **Mesh**, **Utilities** and **Attributes** in the Treeview.

Wall Displacement

- In the Treeview right-click on the results for **Excavation to –13.26m** and **Set Active**.
- If not already on, add the **Deformed mesh** to add the deformed mesh layer to the Treeview.
- Press the **Deformations...** button in the Treeview, select the **Specify factor** option and specify a factor of **5**. Press **OK** to return.
- Double click the **View properties** control in the Treeview. On the **Mesh** tab ensure the option to **Show activated only** is selected. Click **OK** to visualise the deformed mesh for the active elements at increment 8, magnified by a factor of 5.



A graph of the deformation of the wall toe will be created using the graph wizard.

- Zoom into the top left-hand side of the model and select the node at the wall toe.

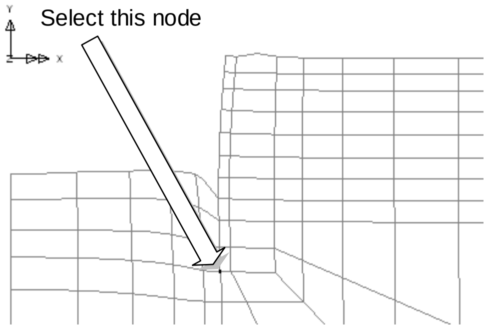
Utilities
Graph Wizard...

- With the **Time history** option selected click **Next.**

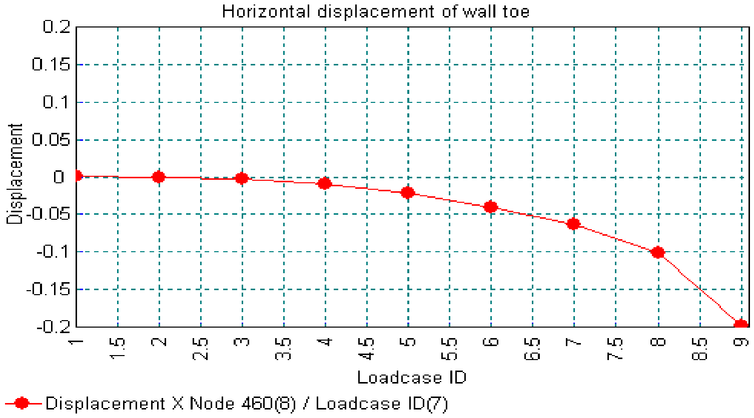
Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next.**
- Select **Loadcase ID** from the drop down list and click **Next.**

Secondly define the data for the Y axis.



- With the **Nodal** option selected and click **Next.**
- Select **Displacement** from the Entity drop down list and **DX** from the Component drop down list.
- The selected node number will appear in the **Specify node** drop down list. Click **Next.**
- Enter suitable text for the graph titles and click **Finish** to display the graph of displacement of the wall toe towards the excavation.

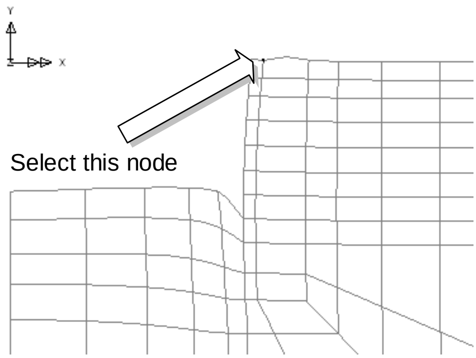


-  Close the graph window.
-  Maximise the graphics window.

Surface Heave

- Select the node at the top of the back of the wall.

- With the **Time history** option selected click **Next.**

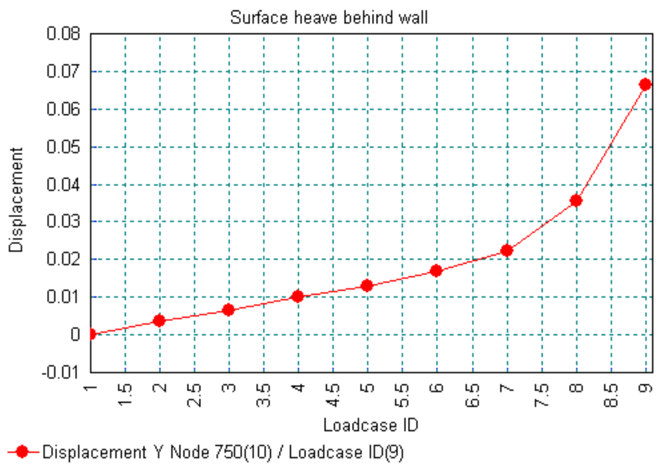


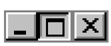
Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next.**
- Select **Loadcase ID** from the drop down list and click **Next.**

Secondly define the data for the Y axis.

- With the **Nodal** option selected and click **Next.**
- Select **Displacement** from the Entity drop down list and **DY** from the Component drop down list.
- The selected node number will appear in the **Specify node** drop down list. Click **Next.**
- Enter suitable text for the graph titles and click **Finish** to display the graph of heave behind the wall.

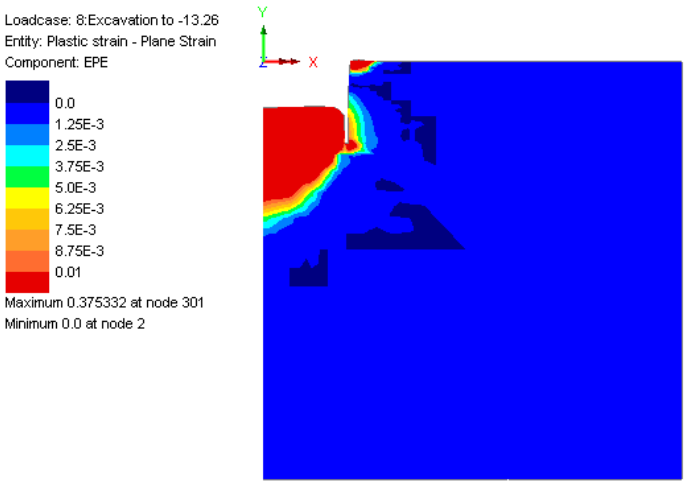


-  Close the graph window then  maximise the graphics window.

Plotting contours of plastic strain in the soil

Because the model contains elements with two different material properties the active set on which results are to be plotted must be selected.

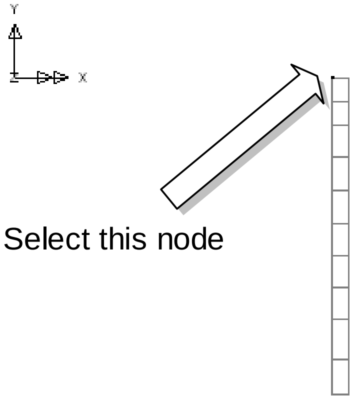
- In the Treeview right-click on the group name **Soil** and select **Show Results > Show Results Only On This Group**
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select **Contours** to add the contours layer to the Treeview.
- Select **Plastic strain – Plane strain** from the Entity drop down list and **EPE** from the component drop down list.
- Select the **Contour Range** tab and set the **Maximum** to **0.01** (1% strain).
- Click the **OK** button to display contours of the plastic strain distribution around the wall.



Prop Force

- Turn off the display of the **Deformed Mesh** and **Contours** layers in the Treeview.
- Turn on the display of the **Mesh** layer in the Treeview.
- In the Treeview right-click on the group name **Wall** and select **Show Results > Show Results Only On This Group**
- Right-click on the group **Wall** in the Treeview and select **Set as Only Visible**.

- Select the node at the top-left of the wall that is supported by the prop in the X direction.



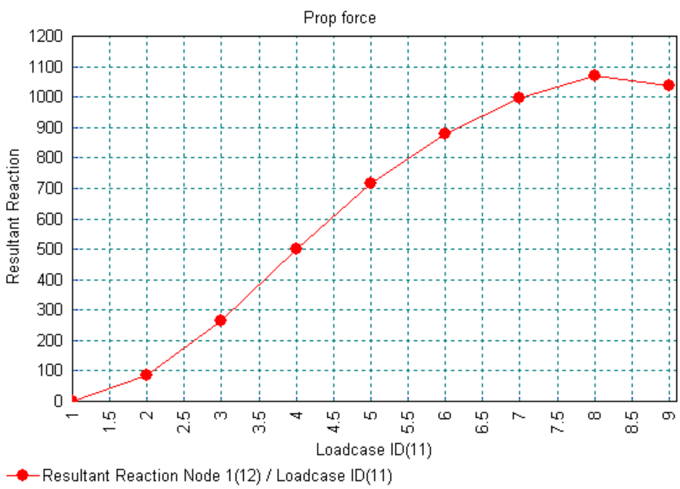
- With the **Time history** option selected click **Next.**

Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next.**
- Select **Loadcase ID** from the drop down list and click **Next.**

Secondly define the data for the Y axis.

- With the **Nodal** option selected and click **Next.**
- Select **Reaction** from the Entity drop down list and **RSLT** from the Component drop down list.
- The selected node number will appear in the **Specify node** drop down list. Click **Next.**
- Enter suitable text for the graph titles and click **Finish** to display the graph of heave behind the wall.



Note. The resultant reaction has been graphed in the above figure. Since the wall is propped only in the horizontal (X) direction at this location (see Supports section) there is no reaction in the vertical direction and therefore the resultant reaction is identical to the horizontal reaction.

- Close the graph window.
- Maximise the graphics window.

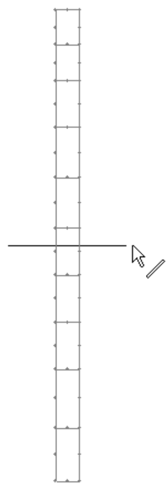
Bending moment in the wall at 10m Depth

Because the model contains elements with two different material properties the active set on which results are to be plotted must be chosen.

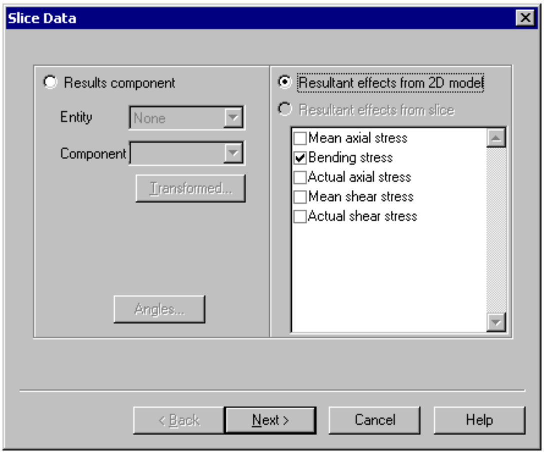
- Turn off the **Deformed mesh** layers in the Treeview, and restore the **Geometry** layer by right-hand clicking on a blank area of the model view and selecting **Geometry**.

Utilities
Graph Through 2D...

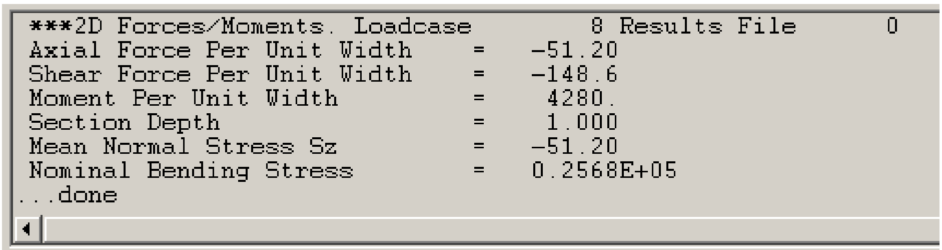
- Ensure the **Snap to grid** option is selected and enter a value of **1** for **Grid size**.
- Click on **OK**.
- Draw a line with the mouse through the wall at 10m depth as shown.



- Select **Resultant effects from 2D model** and select **Bending stress** in the list and click **Next**.



- Enter suitable text for the graph titles and click **Finish**. A graph is drawn and the bending moment (and other information) is displayed in the Text output pane.



```
***2D Forces/Moments. Loadcase        8 Results File      0
Axial Force Per Unit Width      =   -51.20
Shear Force Per Unit Width      =   -148.6
Moment Per Unit Width           =    4280.
Section Depth                   =    1.000
Mean Normal Stress Sz           =   -51.20
Nominal Bending Stress          =   0.2568E+05
...done
```

The bending moment reported is 4280 kN/m^2 per metre run of the wall.

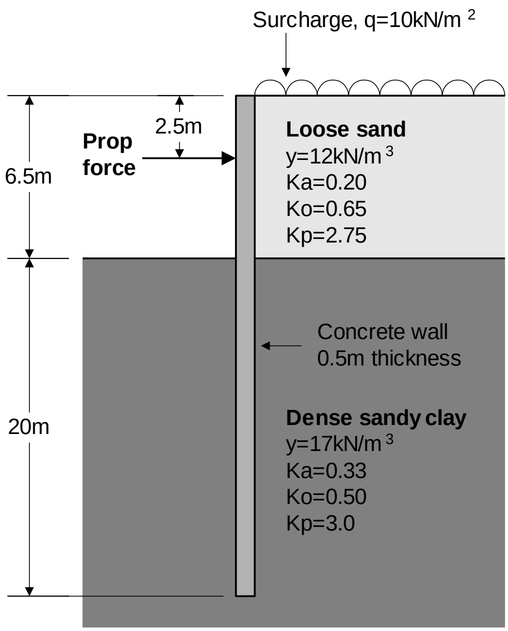
This completes the example.

Embedded Retaining Wall

For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge*.
With product option(s):	Nonlinear.

Description

An embedded retaining wall is required to retain 6.5m of loose sand along with an additional surcharge of 10kN/m^2. A 2D analysis is to be carried out with the soil represented by Tri-linear earth pressure joint material for both an unpropped and a propped wall condition.



The units of the analysis are kN, m, t, s, C throughout.

Objectives

- To calculate bending moment and shear forces in the embedded wall and ensure that deflections are within serviceable limits.
- To investigate the effects of installing a prop and reducing the pile length.

Keywords

Embedded, Retaining, Wall, Trilinear Earth Pressures, Modulus of Subgrade Reaction, Soil-Structure Interaction, Geotechnics

Associated Files

- **embedded_wall_modelling.vbs** carries out the modelling of this example.

Modelling

Running LUSAS Modeller

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

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 the file name as **Embedded Wall**
- Use the **Default** working folder.
- Enter the title as **Embedded Retaining Wall Example**
- Select units of **kN,m,t,s,C**
- Ensure that the Timescale units are **Seconds**, the Analysis type is **Structural** and the Startup template is set to **None**
- Choose **Y** as the vertical axis and click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Wall Geometry

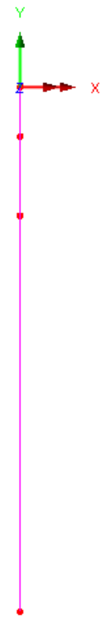
Geometry
Point >
Coordinates...

Enter coordinate (**0, 0, 0**) to define ground level and the top point of the wall. Click the **OK** button to create a single point at the origin.

- Select the newly created Point.

Geometry
Surface >
Sweep…

Sweep the selected point with a Y translation of **-2.5** and click the **OK** button to create the new Line.



- Select the newly created lower point

Geometry
Surface >
Sweep…

Sweep the selected point with a Y translation of **-4.0** and click the **OK** button to define the new Line.

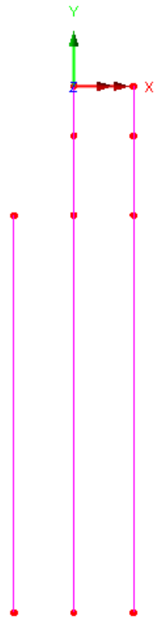
- Select the newly created bottom point

Geometry
Surface >
Sweep…

Sweep the selected point with a Y translation of **-20.0** then click the **OK** button to define the new Line.

Note. Separate lines must be used to represent each change of soil type.

Defining the Earth Geometry



The earth pressure acting on the wall is modelled using joint elements connected between the wall itself and fixed supports. Additional lines are required to assign the fixed sides of the joint elements to.

Note. Joint elements should normally have a zero length, however to reduced confusion from overlapping features during definition they are usually initially located with a gap that is later closed once the model has been setup.

- Select all lines and points (Ctrl + A)

Geometry
Line >
Copy...

Copy all the geometry by **3.0** in the **X** direction and click **OK** to define the retained soil.

- With the wall structure still selected:

Geometry
Line >
Copy...

Copy all the geometry **by -3.0** in the **X** direction and click **OK** to define the retained soil.

Geometry
Line >
Delete...

Select the upper two lines of the excavated side and delete them.

The geometry is now complete. The central series of lines model the wall. The lines to either side are used to model the soil. A load to model the propping force applied to the upper part of the wall will be added to the model later in the example.

File
Save

Save the model file.

Meshing

Note. Due to the underlying implementation of the Tri-linear earth pressure material, the joint elements that the soil material will be assigned to must be orientated in a specific way. For both 2D and 3D joint elements the joint element **local x** axis must align with the horizontal and connect the support to the structure. For 2D joint elements **local y** axis must be aligned to the model vertical axis and for 3D joint elements the **local z** must be aligned to the model vertical axis. The model vertical axis is defined in the new model dialog and can be accessed subsequently by the Utilities > Vertical Axis menu item.

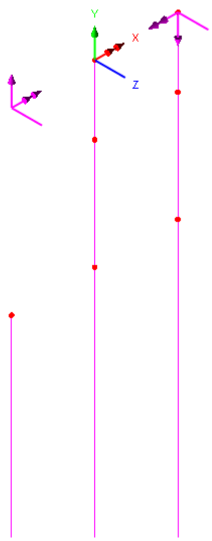
Defining local coordinate systems

To ensure the joint elements are orientated correctly two local coordinate systems are needed: one in the direction of the main model X axis, and one in an opposing direction.

Attributes
Local Coordinates...

- Create a local **Cartesian** coordinate system rotated **0** degrees about the **Z-axis** with an Origin of **(-3,0,0)**, enter the dataset name as **Excavated** and click **Apply**

- Create a second local **Cartesian** coordinate system rotated **180** degrees about the **Z-axis** with an Origin of **(3,0,0)**, enter the dataset name as **Retained** and click **OK**



Defining element types

The joint elements are required to have displacements in the u and v directions to represent horizontal and vertical pressures. For this 2D problem the JNT3 element with only translational freedoms is sufficient.

Attributes
Mesh >
Line...

- Select the element type as **Joint no rotational stiffness**, the number of dimensions as **2 dimensional** and the interpolation order as **Linear**. Select an **Element length** of **1.0**. Enter the dataset name as **Soil Mesh** and click the **OK** button.

The wall requires thick beam elements:

Attributes
Mesh >
Line...

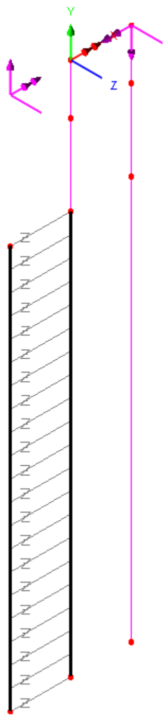
- Select the element type as **Thick beam**, the number of dimensions as **2 dimensional** and the interpolation order as **Linear**. Select an **Element length** of **1.0**. Enter the dataset name as **Thick Beam** and click the **OK** button.

Assigning the elements

The wall is meshed first, and the joint elements are then assigned between the lines representing the soil and the wall.

- Select the three central lines representing the wall and drag the dataset **Thick Beam** the Treeview onto the model.

- Select the left-hand line representing the excavated earth and then, at the same time, also select the corresponding adjacent line of the wall. Drag the dataset **Soil Mesh** from the Treeview onto the model. In the Line Mesh Assignment dialog, select **Excavated** from the specified local coordinates list and ensure **Mesh from master to slave** is selected. Click **OK** to complete the assignment



To assign the mesh for the retained side:

- Select the three central lines representing the wall.

Note. To make joint mesh assignments to sets of multiple lines the selection memory must be used:

Edit
Selection Memory >
Set

Add the lines to the selection memory.

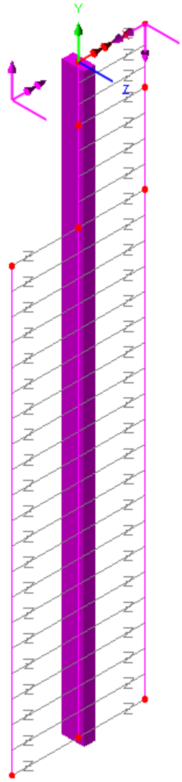
- Select the three lines on the far right representing the retained soil. Drag the dataset **Soil Mesh** from the Treeview onto the model. In the Line Mesh Assignment dialog, select **Retained** from the specified local coordinates list and ensure **Mesh from master to slave** is selected. Click **OK** to complete the assignment

Edit
Selection Memory >
Clear

This clears the selection memory.

Note. If necessary the joint element axes can be visualised and checked by editing the Mesh layer properties (double-click the Mesh layer name in the Treeview) and selecting Show element axes.

Geometric Properties



Utilities
Section Property Calculator >
Rectangular Sections >
Solid...

- Enter a depth **D** of **0.5** and a breadth **B** of **1.0**. Enter a dataset name of **Wall**, ensure **Add to local library** is selected and click the **OK** button.

Attributes
Geometric >
Section Library...

- In the Geometric Line dialog select **2D Thick Beam** for the usage, select **User Sections** in the top right list and dataset **Wall** from the available **Local** sections. Enter the dataset **Wall 1m run** and click **OK** to create the geometric attribute.
- Select the three lines representing the wall and drag the geometric attribute **Wall 1m run** from the Treeview onto the model.

Geometric properties are visualised by default.

- In the Treeview re-order the layers so that the **Attributes** layer is at the top of the Treeview, followed by the **Mesh** layer, followed by **Geometry** layer.

Select the fleshing on/off button to turn-off the geometric visualisation.

Material Properties

Attributes
Material >
Material Library...

- Select material **Concrete EU** from the from drop-down list, select grade **EN1992-1-1 Table 3.1 fck=40MPa** and click **OK** to add the material dataset to the Treeview.
- Select the three lines representing the wall and drag and drop the geometry attribute **Iso1 (EN1992-1-1 Table 3.1 fck=40MPa)** from the Treeview onto the selected lines.

Attributes
Material >
Joint...

- In the Joint Material dialog select **Trilinear Earth Pressure** and select **Next**. Three earth pressure material attributes are required: one for the Loose Sand, and two for the Dense Sandy Clay (where one is for the retained side and one is for the excavated side). All materials are to be defined for use with **Two dimensional joint elements** with the defining geometry being **Lines**.

To create the Loose Sand material, enter the following properties:

Datum used for the calculation of effective overburden pressure	**Zo**	0.0
Surcharge (vertical load per unit area)	**q**	10.0
Unit weight of soil	**γ**	12.0
Angle of shearing resistance between soil and structure	**δ**	0.0

Coefficient of passive earth pressure	**Kp**	2.75
Coefficient of passive earth pressure due to cohesion	**Kpc**	0.0
Cohesion	**c'**	0.0
Coefficient of at-rest earth pressure	**Ko**	0.65
Coefficient of passive earth pressure due to cohesion	**Ka**	0.20
Coefficient of active earth pressure due to cohesion	**Kac**	0.0
Empirical constant used in calculation of kh	**A**	1E3
Empirical constant used in calculation of kh	**B**	250.0
Empirical constant used in calculation of kh	**κ**	1.0
Soil width	**Weff**	1.0

- Observing the local coordinate system established for the retained side, select **Active pressures** to define just what the positive displacement of the local x axis mobilises (i.e towards the wall) and **Matches gravity (points down)** to define just what the positive direction of the local y axis represents.
- Deselect **Consider angle of structure**, enter a dataset name of **Loose Sand Retained** and click **Apply** to create the attribute.

Then, change the parameters to the following:

Datum used for the calculation of effective overburden pressure	**Zo**	-6.5
Surcharge (vertical load per unit area)	**q**	88.0
Unit weight of soil	**γ**	17.0
Angle of shearing resistance between soil and structure	**δ**	0.0

Coefficient of passive earth pressure	**Kp**	3.0
Coefficient of passive earth pressure due to cohesion	**Kpc**	0.0
Cohesion	**c'**	0.0
Coefficient of at-rest earth pressure	**Ko**	0.5
Coefficient of passive earth pressure due to cohesion	**Ka**	0.33
Coefficient of active earth pressure due to cohesion	**Kac**	0.0
Empirical constant used in calculation of kh	**A**	1E3
Empirical constant used in calculation of kh	**B**	250.0
Empirical constant used in calculation of kh	**κ**	1.0
Soil width	**Weff**	1.0

- Enter a dataset name of **Dense Sand Retained** and click **Apply** to create the attribute.

Finally, change the Surcharge value to **0.0** and, observing the local coordinate system established for the excavated side, ensure **Active pressures** is set to define just what the positive displacement of the local x axis mobilises and select **Opposes gravity (points up)** to define just what the positive direction of the local y axis represents. Leave any other settings unchanged. Change the dataset name to **Dense Sand Excavated** and click **Finish** to create the final soil material attribute.

Note. When creating soil layers, if a different soil weight is used for each layer the overburden pressure from any soil layers above must be calculated manually and entered as a surcharge. Where all layers have the same unit weight the datum for each layer can be the same (i.e ground level) remembering that any additional surcharge (q) at ground level needs to be included in the definition of all lower layers. In this example the dense sandy clay on the retained side was given a surcharge of $88kN/m^2$. This was calculated from the depth of sand above plus the applied surcharge such that $q = 6.5x12 + 10 = 88$

- Select the upper two lines representing the loose sand on the retained side of the wall (not the wall itself) and drag the material attribute **Loose Sand Retained** from the Treeview onto the model.
- Select the bottom line representing the dense sand on the retained side of the wall and drag the material attribute **Dense Sand Retained** from the Treeview onto the model.
- Select the line representing the dense sand on the excavated side of the wall and drag the material attribute **Dense Sand Excavated** from the Treeview onto the model.

Note. Earth pressure joint material attributes are held in the Treeview. Context menu entries named Edit Definition... and Edit Attribute... can also be seen by right clicking on an attribute.

Note. Selecting the **Edit Definition...** menu entry or double clicking the attribute name displays the original definition dialog with all the original input data for viewing or editing.

Note. Selecting the **Edit Attribute...** menu entry displays the joint type and stored piecewise linear joint material values that are used to define the earth pressure joint material. These values may be changed but the link to the original definition dialog will be broken.

Supports

For the wall it is assumed that a supported end condition exists.

Attributes
Support...

- Select a **Fixed** translation for the **Y** axis, enter a dataset name of **Fixed in Y** and click **OK** to create the attribute.
- Select the point representing the bottom of the wall then drag the support attribute **Fixed Y** from the the Treeview onto the model.

The Trilinear earth pressure material generates forces in the joint elements appropriate to the position (depth) of the joint element in the model. These forces must be resisted by supporting the free end of the joint elements.

Attributes
Support...

- Select **Fixed** translations for the **X**, **Y** and **Z** axes, enter a dataset name of **Fixed** and click **OK** to create the attribute.
- Select the lines representing the excavated and retained soils and drag the support attribute **Fixed** from the Treeview onto the model. Accept the defaults in the assignment dialog.

Setting the Analysis

The Trilinear earth pressure material is a nonlinear material and therefore a nonlinear analysis is required.

- In the Analyses Treeview, right click on **Loadcase 1**. From the Context menu select **Controls** > **Nonlinear and Transient**
- In the Nonlinear & Transient dialog check the **Nonlinear** check box and select the **OK** button to return to the model.

Saving the model

File
Save

Save the model file.

Caution. For ease of definition the joint elements used have been inserted into the model a set distance away from where they will be acting and, as a result, they have a non-zero length. Prior to an analysis taking place, the lines <u>and</u> points representing the soil should normally be set to be unmergeable prior to being moved to overlay the line representing the wall. For some modelling situations the use of joint elements with a non-zero length will produce unreliable results, and in general joint elements should always be modelled with a zero-length. For clarity in this example merging has been omitted, as there is no significant difference in the results obtained.

To model joints with zero-length

To correctly model the soil joints the joint elements would normally be modelled with zero-length. For this example this would be done as follows:

- Select the lines on either side of the wall that represent the soil.
- Select **Geometry > Line> Make unmergable**
- Select **Geometry > Point> Make unmergable** (because making a line unmergable does not make the points defining it unmergable too.)
- Select the left-hand line representing the soil and select **Geometry > Point> Make unmergable** and move the line **3** metres in the **X** direction
- Select the right-hand lines representing the soil and select **Geometry > Point> Make unmergable** and move the line **-3** metres in the **X** direction
- Save the model with an appropriate filename.

Running the Analysis

With the model loaded:

Select the **Solve Now** button from the toolbar and click **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.

Note. For this analysis the use of non-coincident joints (i.e. omitting the merge step) and linear beam elements within a nonlinear analysis will cause warnings to be written to the text output window. These warnings can be ignored.

If the analysis is successful...

Analysis loadcase results are added to the Treeview

In addition, 2 files will be created in the Associated Model Data where the model file resides:

- **Retaining Wall~Analysis 1.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **Retaining Wall~Analysis 1.mys** this is the LUSAS results file which is loaded automatically into the 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 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.

- **embedded_wall _modelling.vbs** carries out the modelling of the example.

File
New…

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 **Embedded Wall**

File
Script >
Run Script...

- To recreate the model, select the file **embedded_wall_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

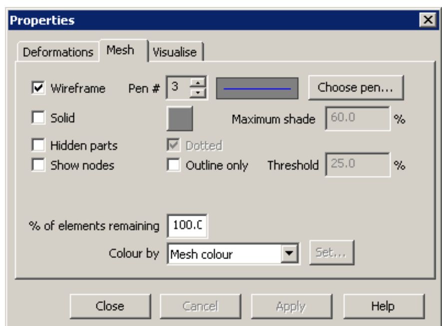
Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the Treeview.

A Deformed mesh layer will be added to the Layers Treeview.

- Double-click on the the **Deformed Mesh** layer in the Treeview and in the **Mesh** tab set the pen to be **blue**



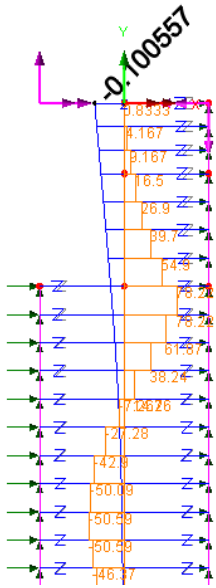
- Turn on the **Diagrams** layer in the Layers Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment – Thick 2D Beam** and the Component to **Mz**

- In the Layers Treeview, tick **Window Summary**.

A maximum Bending Moment of 342kNm is seen to occur in the retaining wall.

Double click the **Diagrams** item in the Layers Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment – Thick 2D Beam** and the Component to **Fy**

A maximum Shear Force of 78kN is seen to occur in the retaining wall.



- Turn on the **Values** layer in the Layers Treeview. In the Values Properties dialog set the Entity to **Displacement** and the Component to **DX**
- On the **Values Display** tab check **Show values of Selection,** check the **Deform** option, set a font **Angle of 45º**, click the choose font button and select a **font size of 16**, click the **OK** button to return to the Properties dialog and click the **OK** button again to return to the model.
- Select the top-most node of the deformed mesh to display the maximum deflection. A maximum deflection of 100mm is seen to occur.

File
Save

Save the model file.

- In the Layers Treeview turn-off the display of the **Diagrams**, **Values** and **Deformed mesh** layers.

Loadcase 2: Adding a propping force

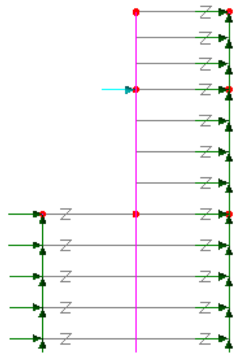
For this example the previously calculated deflection of the wall has been considered to be too large and the wall is to be propped at a depth of 2.5m from the top of the wall. The propping force will be modelled using a concentrated load that will be increased incrementally to calculate the propping force required to reduce the deflection to 30mm. The calculated propping force can them be used to design the anchoring system.

Attributes
Loading...

- Select **Concentrated** and click **Next**
- Enter a value of **10** in the **X direction,** enter a dataset name of **Prop Force** and click the **Finish** button to create the attribute

Analyses
Loadcase...

- Click **OK** to create a second loadcase with the default name Loadcase 2



- Select the prop point on the wall at (0,-2.5). Drag and drop the loading dataset **Prop Force** from the Treeview onto the model

- In the Loading Assignment dialog select **Single loadcase**, select **Loadcase 2** from the Loadcase list and ensure that **Set as active loadcase** is checked.

- Click the **OK** button to make the assignment

- In the Analyses Treeview, right click on loadcase 2. From the Context menu select **Controls** > **Nonlinear and Transient**

- In the Nonlinear & Transient dialog check the **Nonlinear** check box and select **Automatic** Incrementation.

- Set the **Starting load factor** to **1.0**, set the **Max change in load factor** to **1.0** and the **Max total load factor** to **10.0**

- Click the **OK** Button to make the changes and return to the model.

Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.

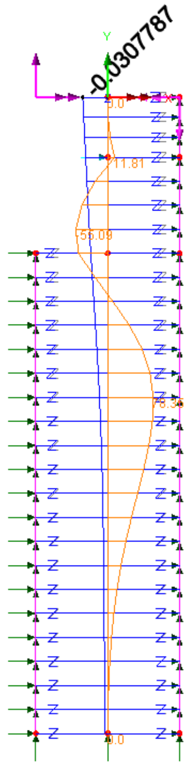
Designing the wall

- In the Layers Treeview turn-on the display of the **Diagrams**, **Values** and **Deformed mesh** layers.

- With the top node of the wall selected (and the values still set to display the horizontal displacement), set each of the load increments (load factors) of **Loadcase 2** active in turn to investigate the effect of applying different prop forces to the top of the wall. From this examination it can be seen that a 30mm top of wall displacement is achieved at **Increment 6**. This increment has a load factor of 5.0 leading to a required propping force of 5x10 = 50kN.

- Double click the **Diagrams** layer in the Layers Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment – Thick 2D Beam** and the Component to **Mz**

The moments in the wall can now be seen to be significantly reduced, with the maximum being 78kNm



Note. This example shows a prop force applied externally to the wall. If a soil anchoring system were to be used to provide the restraint, a different modelling approach would be necessary. In addition, the propping force required to reduce the wall deflections may not be achievable in practice due to the soil type. In this case the design may require some iteration and re-running of the analysis to achieve acceptable forces, moments and deflections.

Once suitable deflections and design forces have been established the required length of pile can be determined by reducing the embedded length and ensuring the effect on the required deflection is acceptable.

An initial estimate of the reduction can be sought by considering where wall forces become negligible. To do this a diagram of Shear force in the wall is to be plotted.

- In the Diagrams Properties dialog set the Entity to **Force/Moment – Thick 2D Beam** and the Component to **Fy**

By inspection the shear within the wall for the bottom 5m length is negligible and can be removed.

Geometry
Point >
Move...

Select the bottom three points of the model and enter **5** in **Y translation**. Click **OK** to reduce the wall depth.

File
Save

Save the model file with a different name if desired.

Select the **Solve Now** button from the toolbar and click **OK** to re-run the analysis.

With the top node of the wall selected and the values set to display the displacement, set each of the load increments (load factors) of Loadcase 2 active in turn to investigate the effect of applying different prop forces to the top of the wall. From this examination it can be seen that a 30mm top of wall displacement is still achieved at **Increment 6** so the reduction in length has not altered the top of wall displacement. Maximum bending moment in the wall is similarly unaltered. Further reduction in embedded length maybe possible by repeating the previous steps and ensuring deflections remain suitable.

This concludes the example.

Trapezoidal earth dam with drainage toe

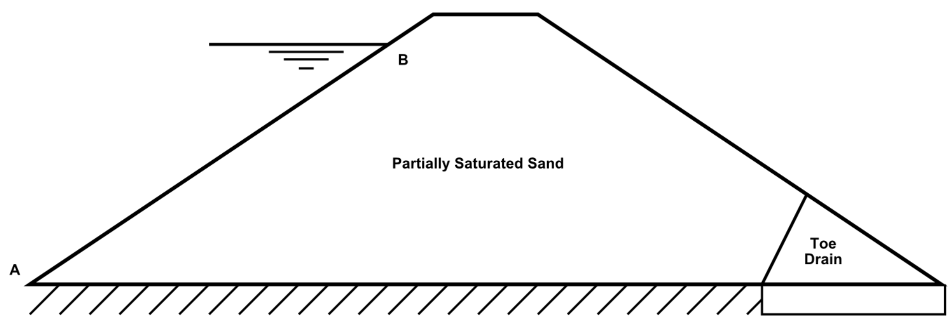
For software product(s):	Any Plus version
With product option(s):	Nonlinear, Dynamic, Thermal, Heat of hydration

Description

This example illustrates water seepage through a trapezoidal earth dam to drain in the granular filter in the downstream toe. The flow unconfined, with the dam built on an impermeable foundation.

On the upstream side, the water is at the reservoir level of 12m which results in equipotential line along AB with maximum head of 12m. The filter drain on the downstream side is considered fully effective.

The modelling units used are N, m, kg, s, C throughout.



Objectives

The required output from the analysis consists of:

- The calculation of steady-state seepage flow through the body of the dam.
- Contours of pore water pressures.

Keywords

Pore Pressure, Two Phase Material, Consolidation, Geostatic step, Transient, Seepage

Associated Files

- **earth_dam_modelling.vbs** carries out the modelling of the example.

Discussion

Not all the flow networks of an earth dam are known from the beginning. The free flow surface, or top flow line (where the pressure is atmospheric), can only be approximated when using hand methods. The upper most flow line inside an earth dam follows a parabolic path, which is corrected to be tangential to the upstream slope and discharge surface, as well as at different material boundaries. Earth dams almost always include drainage filters (having a large value of permeability) positioned in the downstream slope to keep seepage entirely within the dam, and prevent erosion of the downstream surface.

There are two basic assumptions in such analyses:

a) Between two consecutive points of equipotential lines in contact with the free surface there is a constant piezometric difference

b) At the point that the flow from the upstream side to the downstream side begins, the upper most flow line is positioned perpendicular to the upstream slope of the dam

Modelling

Running LUSAS Modeller

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

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 the file name as **Earth_dam**
- Use the **Default** working folder
- Enter the title as **Trapezoidal earth dam with drainage toe**
- Select units of **N, m, kg, s, C**
- Select a timescale unit of **Days**
- Ensure the **Structural** analysis type is selected.
- Select a startup template of **None**
- Select the **Vertical Y Axis** option.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Geometry

Geometry
Surface >
Coordinates…

Enter the following coordinates to define the partially saturated sand part of the dam.

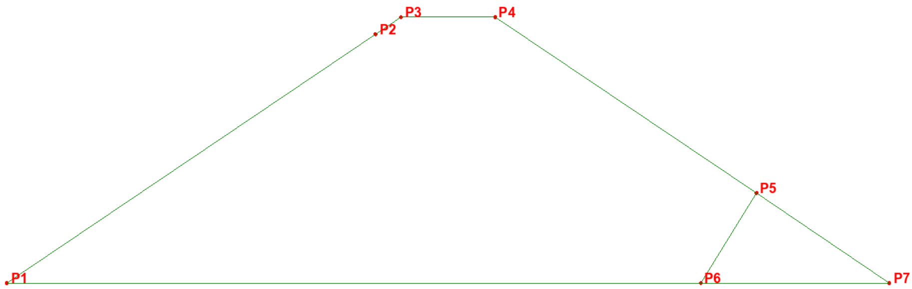
X	Y
0.0	0.0
17.8064	12.0
19.0345	12.8276
23.5862	12.8276
36.2038	4.34484
33.5172	0.0

Geometry
Point >
Coordinates…

Enter the following coordinates (**42.6207, 0.0**) to define the downstream toe point.

Geometry
Surface >
Points…

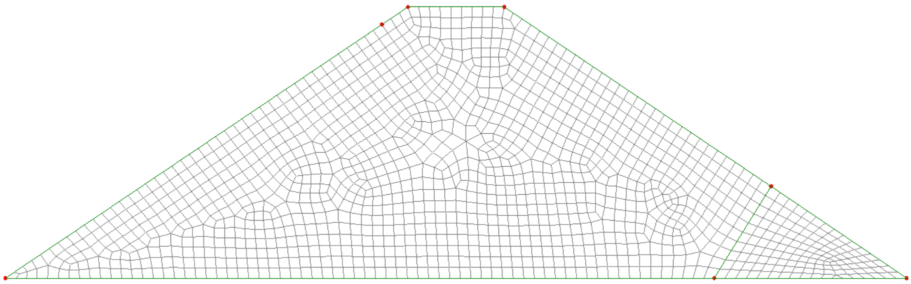
Holding down the **P** key to ensure only points are selected, box-select the three points that form the bottom-right corner (Points 7, 6 and 5) in order to create the filter discharge surface.



Defining and Assigning Mesh Attributes

Attributes
Mesh >
Surface…

- Select **Plane strain two phase** elements types with **Quadrilateral** shape and **Quadratic** interpolation order (QPN8P elements).
- Unselect **Automatic** mesh, tick **Element size** and enter **0.5**
- Enter the attribute name as **QPN8P 0.5m** and click **OK**
- Select the two surfaces on the model and drag and drop the **QPN8P 0.5m** mesh from the Treeview onto both surfaces.



Material Properties

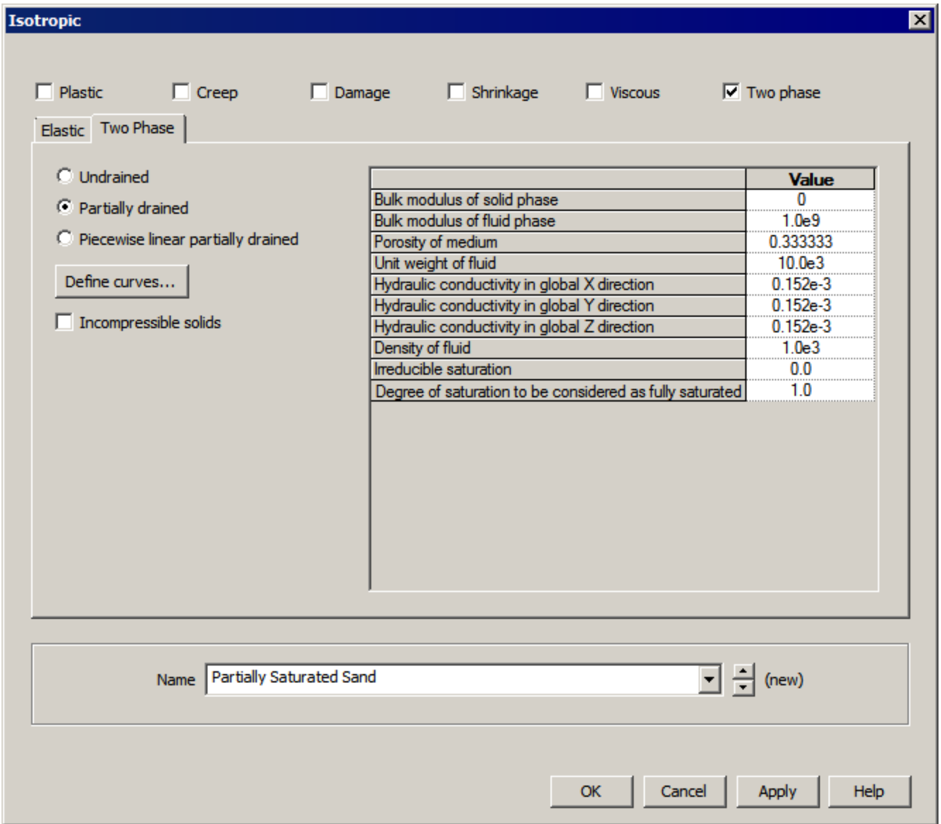
Attributes
Material >
Isotropic...

Two different materials need to be defined; one for the partially saturated sand and one for the filter. For this example both materials need two phase properties defined.

Two-phase material properties are required when performing an analysis in which two-phase elements are used to define a drained and undrained state for soil.

- With the **Elastic** tab selected enter a Young's modulus **1.0E9**, Poisson's ratio **0.2** and Mass density **2.0E3**.

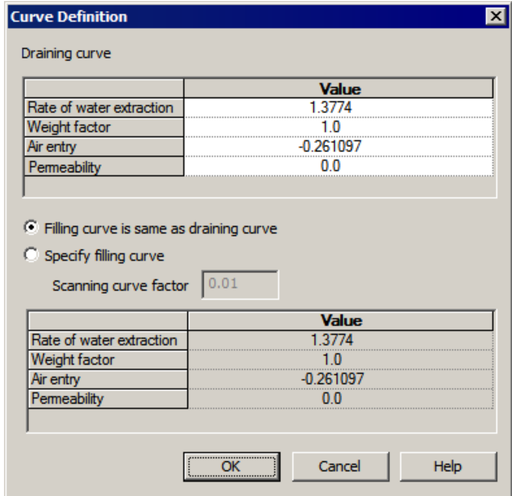
- Tick **Two phase** option and select the **Two phase** tab. Ensure the **Partially drained** option is selected, and enter the Bulk modulus of fluid phase as **1.0e9**, the Porosity of medium as **0.333333**, the Unit weight of fluid as **10.0e3** the Hydraulic conductivity in every global direction as **0.152e-3**, the Density of the fluid as **1.0e3** and the Degree of saturation to be considered as fully saturated as **1.0**.



Note: Usually the value of Bulk modulus during the solid phase is substantially larger than the fluid phase, and therefore does not typically form part of the geotechnical testing. If the bulk modulus of solid phase is left as **0**, LUSAS assumes an incompressible solid phase.

Note: All time dependent material units (e.g. Hydraulic conductivity) are entered as model units (i.e. seconds). The timescale units are only used when defining analysis properties, such as nonlinear timesteps. The units expected by each input are shown as a tooltip.

- Press **Define Curves** to define a new pressure-effective saturation curve.
- Enter the Rate of water extraction as **1.3774**, the Weight factor as **1.0** and the Air entry to **-0.261097**.
- Press **OK** to leave the curve definition dialog.
- Enter the **Name** of the newly defined material as **Partially Saturated Sand** and press **Apply**



- To define the granular filter material; in the Two Phase tab, change the Hydraulic conductivity to **1.0** in all three global (X, Y and Z) directions.
- Change the attribute name to **Filter** and press **OK**
- Assign the two attributes to their corresponding surfaces by dragging and dropping them onto the relevant features.

Loadcases

The geometry specified for the dam represents the as-constructed state, prior to any settlement and the filling of the reservoir. A geostatic loadcase will be used to model the initial stresses in the earth structure (due to self-weight) without any deformation. A second loadcase will then be used to apply the hydrostatic water pressure to the 'as-constructed' dam.

- In the Treeview right-click on **Loadcase 1** and click rename. Change the name to **Initial Condition**

Analyses
Loadcase...

- Add a second loadcase, changing the name to **Seepage** and press **OK**.

Supports

Using two consecutive loadcases requires support attributes to be defined to both add and remove the restraint to the model for each stage.

During the Initial Condition (geostatic) stage pore-water will be allowed to flow freely through the dam to prevent the build-up of pore pressure during the construction stage.

Attributes
Supports…

- Leaving all other degrees of freedom **Free**, set Pore pressure to **Fixed**. Enter the name of the support as **Free Flow** then press **Apply** to define a constant (zero) pressure attribute.
- Switch the X and Y translation freedoms to **Fixed**. Change the attribute name to **Fixed XY (Free Flow)** and press **OK** to define a fixed structure support.

Once the support attributes have been defined, they must be assigned to define the initial condition.

- In the Treeview right-click on **Initial Condition** and press **Set active**
- Select both surfaces and, from the Treeview, drag and drop the support attribute **Free Flow** onto the model. Press **OK** to assign the support.
- Select the two bottom lines of the dam, and assign the **Fixed XY (Free Flow)** support attribute. Press **OK** to assign the support.

During the seepage stage the pore pressure will be allowed to build-up within the structure, with flow restricted to the drainage filter material in the downstream toe.

Attributes
Supports…

- Leaving all degrees of freedom **Free** (including Pore pressure). Enter the name of the support as **Seepage Flow** and press **Apply** to define a 'freeing' support.
- Change the X and Y translation freedoms to **Fixed** and enter the name as **Fixed XY (Seepage Flow)**. Press **OK** to define a structural support that restricts flow.

Non-linear analyses inherit support conditions from the previous stage. These new support attributes are therefore used to free the Pore pressure fixity where required.

- In the Treeview right-click on **Seepage** and press **Set active**
- Select both surfaces and, from the Treeview drag and drop the support attribute **Seepage Flow** onto the model.
- In the Assign support dialog choose the **From loadcase** option, and select the **Seepage** loadcase. Press **OK** to assign.

- Select the left-hand bottom line of the dam (i.e. not within the filter), and assign the **Fixed XY (Seepage Flow)** support attribute, the ensuring **From loadcase** option and **Seepage** loadcase are selected in the Assign support dialog.

There is no need to assign a new support attribute to the line at the bottom of the filter material, as the required support, assigned in the previous loadcase (structurally fixed with free flow), will carry forward.

Structural Loading

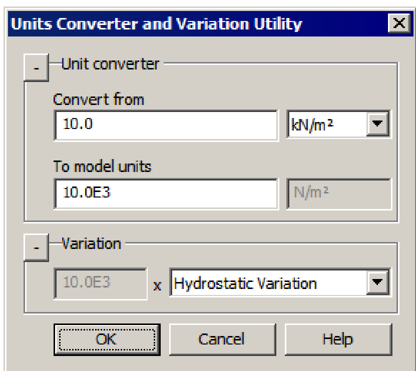
The effects of the self-weight of the dam will be modelled throughout this analysis and can be applied as directly to the analysis.

- In the Treeview right-click on **Analysis 1** and select **Gravity**

For this analysis it will be assumed that the reservoir retained by the dam does not drain significantly due to seepage, or emptying, and therefore the total pressure head can remain fixed at 12m.

Attributes
Loading…

- In the Prescribed section select **Displacement**, click **Next** and in the **Pore pressure** select the **Fixed** radio button and click the button in the **Displacement** field.
- In the Unit Converter and Variation Utility dialog, select **New...** from the Variation drop-down menu to open the Variation dialog.
- Select **General Field Variation**, click **Next** and enter **12-y** in the function field.
- Click **Function limits**, then tick **Max. x coordinate** and enter a value of **17.8065**
- Click **OK**, ensure **Global Coordinates** is selected, enter **Hydrostatic Variation** as the name and click **Finish** to exit the dialog.
- To convert the variation, defined in m-head, to hydrostatic pressure, select **kN/m^2** from the Unit converter options. Then enter a value of **10** in the Convert from box. and click **OK**
- Name the attribute **Hydrostatic Pressure** and click **Finish**



- Select the bottom line of the upstream slope of the dam (line 1) and from the Treeview and drag and drop the loading attribute **Hydrostatic Pressure** onto the selected line, ensure the loadcase **Seepage** is selected, then click **OK**

Non-Linear Control

Use of Geostatic steps and two-phase consolidation materials are inherently non-linear.

- In the Treeview right-click **Initial Condition** and select the **Nonlinear & Transient** option from the **Controls** menu.

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

- In the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner and leave the incrementation type as **Manual**

- Select the **Geostatic step** option.

- Change the Incremental displacement norm to **0**. This leads to convergence being assessed based on the (total) Displacement norm. Click **OK** to exit the dialog.

Note: Most geotechnical problems begin from a geostatic state where the undisturbed soil or rock body remains in equilibrium with the prescribed boundary conditions and geostatic loads, including gravity. The geostatic step produces zero deformations, but establishes the initial stress field that can be used in a subsequent static or coupled field diffusion/stress analysis.

- In the Treeview right-click on **Seepage** and select the **Nonlinear & Transient** option from the **Controls** menu.

- In the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option and select a **Consolidation** time domain. Enter the initial time step as **0.01**, the total response time as **30** and the maximum number of time steps as **30**. Note that analysis time is measured in days, which was the timescale option set when creating the model.

- Tick the **Automatic time stepping** option, press **Advanced** and change the Time step increment restriction factor to **5.0** and the Maximum time step to **5.0.** As consolidation/seepage effects decay exponentially this will allow the time interval to grow accordingly. Press **OK** to exit the dialog.

- Ensure the Solution strategy is set to **Same as previous loadcase**

- Having set all the Nonlinear & Transient options select **OK** to return to the Modeller graphics window.

Running the Analysis

Open the Solve Now dialog, ensure **Analysis 1** is selected and press **OK** to solve.

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 Treeview.

In addition, solution files will be created in the **Associated Model Data\Earth dam** folder.

- ❑ **Earth_dam.dat** this data file contains details of model data and assigned attributes.
- ❑ **Earth_dam.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **Earth_dam.mys** this is the LUSAS results file from the analysis which is loaded automatically into the Treeview to allow results to be viewed.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to the output files 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 the Model

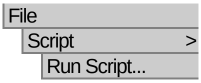
If errors have been made in the modelling of this example that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully.

- ❑ **earth_dam_modelling.vbs** carries out the modelling of the example.

File
New…

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 **earth_dam**



- To recreate the model, select the file **earth_dam_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Run the analysis to generate the results.

Viewing the Results

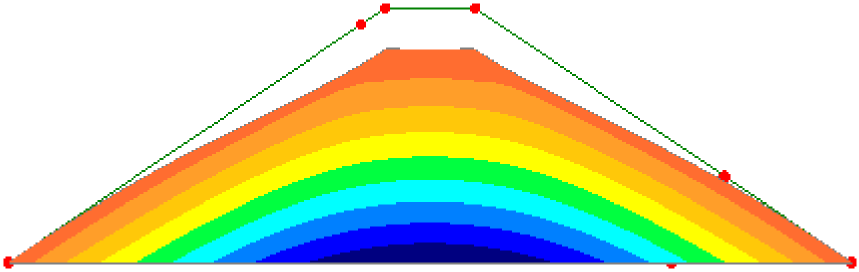
Analysis loadcase results for each time step are present in the Treeview.

Viewing the Initial Stress Distribution

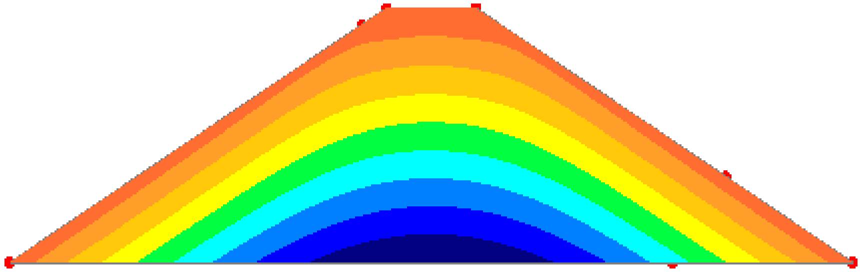
Two steps take place in **Initial Condition**; a first increment and then the Geostatic step.

- In the Treeview right-click on the Initial Condition result-set for **Increment 1** and **Set Active**.
- In the Treeview turn-off the **Mesh** and **Attributes** layers. Press the **Deformations...** button, select **Specify factor** enter a factor of **2.0e3** and press **OK**.
- With no features selected right click in a blank part of the Graphics window and select the **Contours** option to add the **Contours** layer to the Treeview.
- On the dialog set the results entity to **Stress – Plain Strain** and component **SY**, then press **OK**.

The results from Increment 1 show how the structure would behave if gravity was applied to the structure immediately after construction. This is used to get the 'in-situ' stresses then used for the Geostatic step.



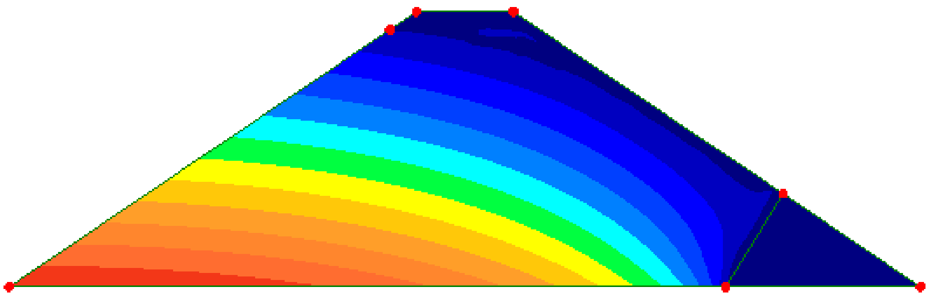
- In the Treeview right-click on the Initial Condition result-set for **Geostatic Step** and **Set Active**.



In the **Geostatic Step**, the deformations are set to zero (to match the as-built geometry) with in-situ stresses maintained

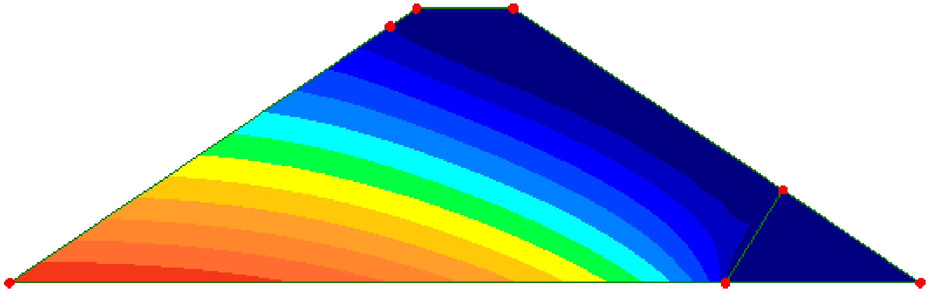
Viewing The Phreatic Line Development

- In the Treeview right-click on the Seepage result-set for **Time Step 1**, and **Set Active**.
- In the Treeview switch off the **Deformed Mesh** layer.
- Double-click the **Contours** layer in the Treeview. On the dialog, set the results entity to **Displacement** and component to **PRES**.
- Select the **Contour Display** tab and deselect **Deform**.
- Select the **Contour Range** tab and set an **Automatic** contour range using an Interval of **10e3**. Enter a Maximum value of **120e3** (12m head) a Minimum value of **0**. Click **OK.**



By setting each time-step active (right-click the time-step in the Treeview and **Set Active**) in turn the gradual development of a semi-steady state seepage flow can be seen. During the first three time-steps (~7.5 hours) the effects of primary consolidation can be observed. From the fourth time-step onwards, however, there is very little change between results.

- In the Treeview right-click on the Seepage result-set for **Time Step 10**, and **Set Active**.



From the basic parabola, the top flow (zero pressure) line curves correctly upward on the upstream side, and downward reaching the filter toe.

Dam Discharge

To calculate the seepage flowrate into the filter (i.e. the rate of water draining through it the body of the dam), the flow reaction of the filter bed is going to be used (as water is being drained only from this location).

- Select the bottom line of the toe-drain filter and press to define a new Group. Enter the name as **Filter** and press **OK** to create the new group.

Ensure that the last time-step increment of the Seepage results (**Time Step 10**) is set active in the Treeview:

Utilities
Print Results Wizard...

- Select **Active**.
- Select the entity **Reaction** and use the results type **Component**.
- Press the **Extent...** button and select new created group **Filter**.
- Press **Finish** to view the tabulated results summary.

LUSAS View: Reaction Components in Global Axes Loadca...

	Node	FX	FY	VFlo	RSLT
1	Node	FX	FY	VFlo	RSLT
2	172	-6.01233E3	17.166E3	0.0	18.1884E3
3	3704	-6.82473	16.5582	-4.87784E-6	17.9095
4	3705	-11.1695E3	33.7625E3	0.0	35.5622E3
5	3706	-5.35537E3	16.5152E3	125.022	17.3618E3
6	3707	-10.0486E3	32.2052E3	0.0	33.7365E3
7	3708	-4.82496E3	15.7607E3	42.3815	16.4828E3
8	3709	-9.26649E3	30.79E3	0.0	32.1542E3
9	3710	-4.46435E3	15.045E3	15.8784	15.6934E3
10	3711	-8.61125E3	29.3719E3	0.0	30.6083E3
11	3712	-4.15867E3	14.3347E3	5.77938	14.9258E3
12	3713	-8.0397E3	27.944E3	0.0	29.0776E3
13	3714	-3.88883E3	13.6144E3	2.33988	14.1589E3
14	3715	-7.52903E3	26.4823E3	0.0	27.5317E3
15	3716	-3.64544E3	12.871E3	1.05219	13.3773E3
16	3717	-7.06183E3	24.9613E3	0.0	25.941E3
17	3718	-3.41982E3	12.092E3	0.514106	12.5663E3
18	3719	-6.62131E3	23.3587E3	0.0	24.2791E3
19	3720	-3.20349E3	11.2674E3	0.26854	11.714E3
20	3721	-6.19059E3	21.6578E3	0.0	22.5252E3
21	3722	-2.98804E3	10.3904E3	0.14719	10.8115E3
22	3723	-5.75343E3	19.8484E3	0.0	20.6655E3

Model info | 12:Seepage, Time Step 10 Time = 0.259200E-

The value of **VFlo** is the volume of flow during the selected time-step. Considering the semi-steady flow achieved by 30 days, the average flow-rate into the filter drain can be calculated from the total discharge shown in these results over the duration of the time-step between the last and second-to-last increments.

- Save the results in a spreadsheet by right clicking on the Print Results Wizard output and selecting Save as Microsoft Excel.
- Sum the **VFlo** column to get the total discharge over the time-step (193m^3).
- Identify the time of the last and second-to-last timesteps from the Treeview (4.2 days)
- Divide the total discharge over the final interval to calculate the average flow rate into the filter dam (45.9m^3/day/m)

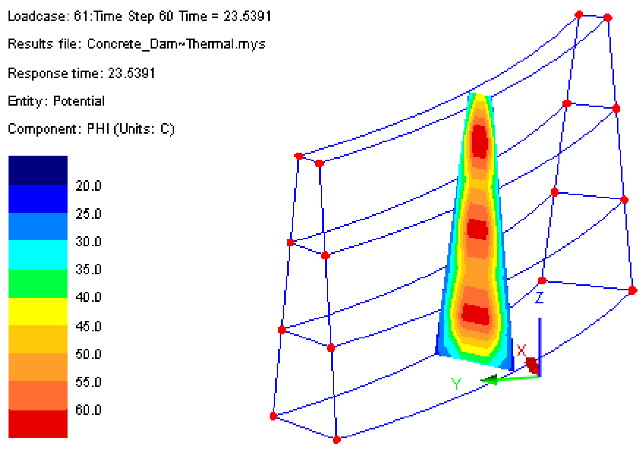
This completes the example.

Staged Construction of a Concrete Dam

For software product(s):	Any Plus version
With product option(s):	Nonlinear, Dynamic, Thermal, Heat of hydration

Description

A 30m high concrete dam, tapering from 10m thick at the base to 3.1735m thick at the crest is to be constructed in three stages, each 10 metres high. Each stage is added 10 days after the previous stage, and the analysis runs for a total of 30 days.



Animations are created showing the variation of temperature and stress during its construction.

Simplified geometry is used to allow the example to concentrate on the definition of the concrete heat of hydration loading and staged construction techniques required.

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

Objectives

The required output from the analysis consists of:

- ❑ A time history of the temperature throughout the dam during construction.
- ❑ A time history of the distribution of maximum principal stress (S1) throughout the dam.

Keywords

Birth, Death, Staged Construction, Activate, Deactivate, Heat of hydration, Semi-coupled analysis, Concrete

Associated Files

- ❑ **concrete_dam_geometric_modelling.vbs** carries out the geometric modelling of the example.
- ❑ **concrete_dam_modelling.vbs** carries out the complete modelling of the example to a ready to run stage.

Discussion

Concrete is generally unique among structural materials in that it interacts with its environment undergoing unavoidable physical and chemical volume changes. Concrete exhibits certain characteristics such as ageing, creep and shrinkage, which are collectively known as time dependent deformations; these are dealt with in other examples in this manual. See the Concrete Tower example if age and creep is of interest to you.

The process of hydration is an exothermic chemical reaction, which leads to large amounts of heat generation. This leads to thermal stresses, which may potentially cause the concrete to crack. A semi-coupled analysis can be used to investigate the connection between the heat generated due to concrete hydration and the thermal stresses induced as a result.

Modelling

Running LUSAS Modeller

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

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.

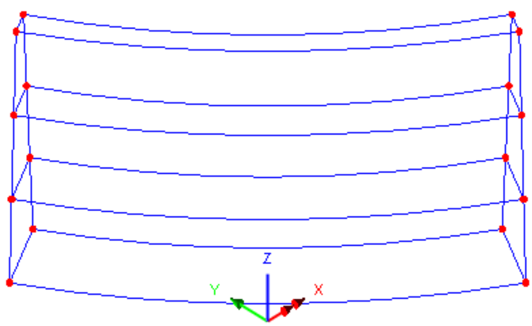
Creating a new model

- Enter the file name as **Concrete_Dam**
- Use the **Default** working folder
- Enter the title as **Concrete dam example to model heat of hydration**
- Set the Timescale units to **Days** and the Model units to **N,m,kg,s,C**
- Ensure the **Coupled** user interface is selected.
- Select a startup template of **None**
- Select the **Vertical Z Axis** option.
- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Geometry

The base of the dam is 10m thick, 54.97m along its centreline, and subtends an angle of 30 degrees. The thickness of the dam tapers with increasing height in a nonlinear manner (such that the thickness at the abutments is very slightly larger than at the centre of the dam).



Since the geometry of the dam is not the focus of this example a script file has been provided which will automatically create the model.

File
Script >
Run Script…

- Select the file **concrete_dam_geometric_modelling.vbs** which is located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory and click **OK**

Select the isometric button which, because of the way the geometry has been defined, will give a front view of the dam. The three volumes represent the three stages of construction.

Defining Groups

It is useful in staged construction analysis to make use of the group facility in LUSAS. This allows parts of the model to be displayed in isolation according to the construction stage being modelled and simplifies the viewing of results.

- Select the lowest volume representing the concrete in construction stage 1.

Geometry
Group >
New Group

Enter the group name as **Stage 1** and click **OK** to finish defining the group.

- With the previous volume still selected, hold the Shift key down and add the middle volume representing construction stage 2 to the selection.

Geometry
Group >
New Group

Enter the group name as **Stage 2** and click **OK** to finish defining the group.

- With the previous two volumes still selected, hold the Shift key down and add the upper volume representing construction stage 3 to the selection.

Geometry
Group >
New Group

Enter the group name as **Stage 3** and click **OK** to finish defining the group.

Defining and Assigning Mesh Attributes

To ensure a well-proportioned mesh, line mesh attributes will be assigned to define a mesh with three elements through the thickness of the dam, and eight elements across the width of the dam. Vertically, lines defining each volume will be split into two divisions. To create this mesh arrangement both default and individual line mesh divisions will be used.

File
Model properties…

- On the Model Properties dialog select the **Meshing** tab. Change the default line divisions to **3** and click **OK.**

This specifies the default number of mesh divisions to be used along a line unless a line mesh attribute is subsequently assigned.

Attributes
Mesh >
Line…

- On the Line Mesh dialog ensure that the Structural element type is set to **None**. Set the number of divisions to **2**, name the attribute **Divisions = 2** and click **Apply** to create the attribute.

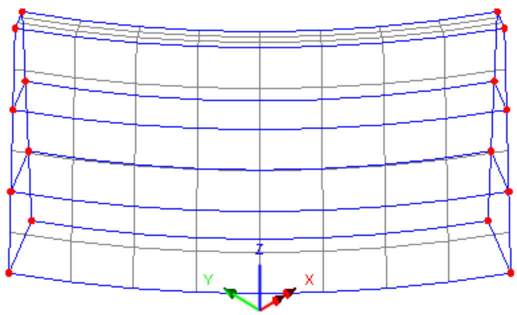
- Change the number of divisions to **8**, change the name to **Divisions = 8** and click **OK**.

Tip. To assign these mesh divisions to the model the selection of lines could be done individually, but lines can also be selected by holding-down the **L** key while box-selecting with the mouse. When this is done any part of a line that is within the area dragged is selected.

- Select the twelve near-vertical lines that define four edges of the dam. Assign the line mesh attribute **Divisions = 2** from the Treeview onto the selected features.

- Select the eight horizontal curved lines from the model. Assign the line mesh attribute **Divisions = 8** from the Treeview onto the selected features.

Attributes
Mesh >
Volume…



- The dam is to be modelled with **Stress, Hexahedral, Linear** elements (HX8M elements) in the structural analysis. By clicking on the Thermal tab it can be seen that these elements are automatically coupled to **Field, Hexahedral, Linear** elements (HF8 elements) in the thermal analysis. Ensure that a **Regular** mesh is used. (The mesh spacing will be governed by the assigned line mesh attributes). Enter the dataset name as **Coupled Brick Elements** and click **OK**

- Select all volumes in the model (or use the **Ctrl + A** keys to select all features) and drag and drop the **Coupled Brick Elements** mesh from the Treeview onto the selected features.

Material Properties

Attributes
Material >
Isotropic...

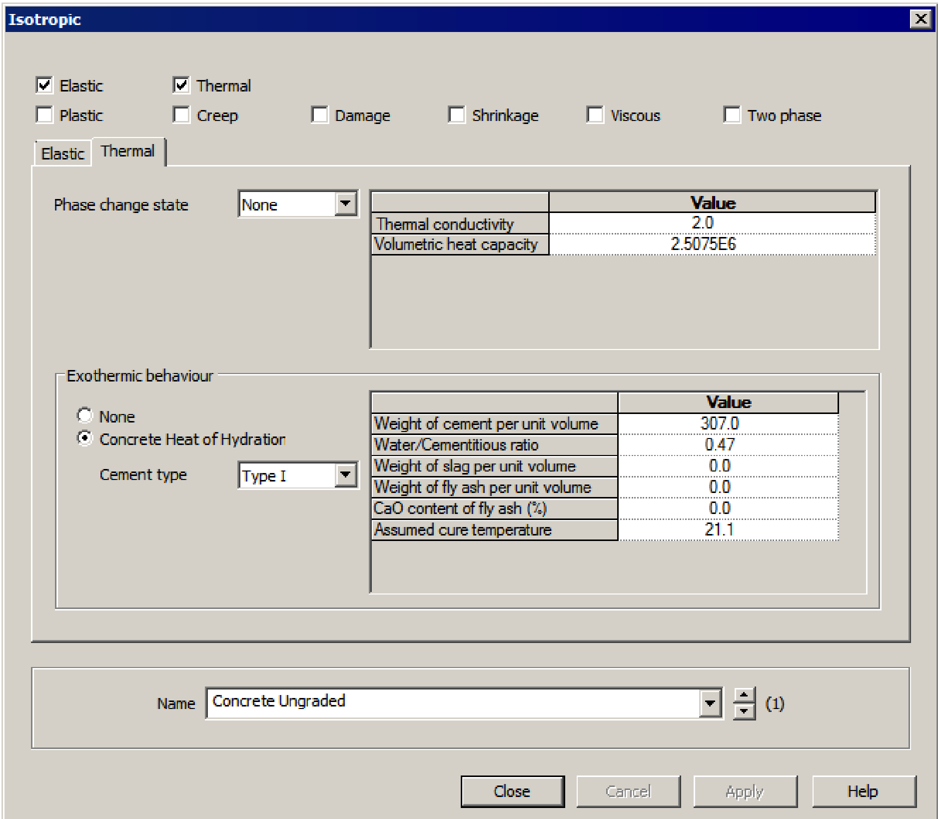
- Enter a Young's modulus of **30e9**, a Poisson's ratio of **0.2**, and a mass density of **2.4e3**. Click the **Thermal expansion** check box and enter a Coefficient of thermal expansion of **10e-6**.

- Click on the **Thermal** tab. Leave the Phase change state set to **None** and enter a Thermal conductivity of **2.0** (J/m^2·s·°C) and a Specific heat coefficient of **2.5075E6** (J/m^3·°C).

The heat of concrete hydration thermal loading will now be defined.

- Select **Concrete Heat of Hydration** from the Exothermic behaviour panel. Then set the Exotherm option to **Concrete** for Cement type **Type I.** Note that the timescale units have already been set when the model was first created and will be used in any heat of hydration analysis.

- Set the remaining values required in the dialog as follows: Weight of cement **307**, Water/Cementitious ratio **0.47**, Weight of slag **0**, Weight of fly ash **0**, CaO content of fly ash **0**, Assumed cure temperature **21.1**
- Change the attribute name to **Concrete Ungraded** and click **OK**. See the following dialog for confirmation of the thermal values required:



Note. The Specific heat coefficient is the specific heat capacity multiplied by the density.

Note. When computing the heat due to the rate of hydration of concrete, hours or days are convenient units to use for the time step and elapsed time. Due to the way the solution algorithms are formulated this is independent of the fundamental units of seconds and Newtons that are defined for the model.

Note. The concrete heat of hydration loading is a type of internal heat generation. From the input parameters LUSAS automatically calculates the amount and rate of this internal heat generation based on formulae presented in published research by Schindler and Folliard. Reference [S18] *LUSAS Theory Manual*.

Assign the Material Properties

- With all three volumes in the model selected, drag and drop **Concrete Ungraded** from the Treeview onto the selected volumes. Press **OK** to assign to **Analysis 1. (**This analysis name will be renamed later to be 'Structural')

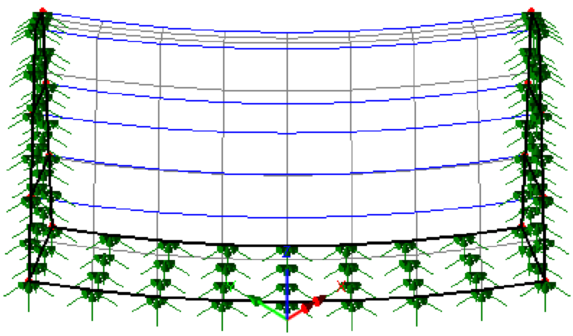
Note. In a coupled analysis, a material with Elastic <u>and</u> Thermal properties defined should always be assigned to the Structural analysis.

Structural Supports

Attributes
Support >
Structural...

A fully fixed support is required for the base and abutments of the dam. All degrees of translation in the **X**, **Y** and **Z**-axes must therefore be set as **Fixed**. Enter the attribute name as **Fixed** and click **OK**

- Select the surfaces that define the base and abutments of the dam and from the Treeview drag and drop the support attribute **Fixed** onto the selected features, ensure the **Analysis 1** and the **All analysis loadcases** option is selected and click **OK**



Tip. Hold-down the **Shift +S** keys and box-select the surfaces required.

Thermal Supports

A thermal support needs to be defined for use later in the example.

Attributes
Support >
Thermal...

- Ensure the temperature support is set to **Free** and enter the attribute name as **Free** and click **OK.**

Do not assign this support to the model yet**.** It is to be used during the activation and deactivation of the thermal loadcases to free-up elements that have had their temperature restrained at the concrete placement temperature.

Thermal Loading

Several thermal loads need to be defined for applying to the model later. Environmental temperature loads for the air and ground temperatures will be defined first, followed by the initial concrete placement temperature.

Air temperature

Attributes
Loading >
Thermal...

- Select **Environmental Temperature** and click **Next**
- Enter a value of **21.1** (ºC) for the environmental temperature, and **8.333** (J/m^2·s·ºC) for the convection heat transfer coefficient. Leave the other two inputs blank. Enter the attribute name as **Air temperature**. Click **Apply**

Ground temperature

- In the first row of the dialog, keep the value of **21.1** (ºC) for the environmental temperature, but over-type a value of **2.777** (J/m^2·s·ºC) for the convection heat transfer coefficient. Change the attribute name to **Ground temperature** and click **Finish**

Initial concrete temperature

The initial concrete temperature will be defined:

Attributes
Loading >
Thermal...

- Select **Prescribed Temperature** and click **Next**
- Select **Fixed** and enter a value of **21.1** (ºC). Ensure the **Total** option is selected. Enter the attribute name as **Initial concrete temperature**. Click **Finish**

Modelling Staged Construction

In order to correctly model the staged construction of the dam, volumes (and hence, elements) in the model must only be included in the analysis after they have been constructed. Similarly, loading attributes (such as surface heat transfer) will need to be applied only for a certain portion of the analysis to reflect the construction stage being considered. In general, model attributes are assigned and changed using Loadcases, whilst loading attributes that apply during particular times during an analysis are applied to selected features of the model using Load curves.

Creating Activation and Deactivation datasets

Modelling of staged construction processes that require activation and deactivation of elements is carried out in LUSAS using the birth and death facility.

Attributes
Birth & Death…

- Choose the **Activate** option and click **Next**

- Enter the attribute name as **Activate** and click **Apply**. Then click **Back** so the dialog can be reused to define the deactivate attribute.

- Choose the **Deactivate** option and click **Next**

- Enter the attribute name as **Deactivate**, select **Percentage to redistribute** and leave the value as **100**%, then select **Finish**

When building staged construction models it is often very helpful to see the elements that are actually activated in the loadcase being viewed.

- In the Treeview, double-click the **View properties** control, and under the **View** tab ensure the **Show only activated elements** option is selected, then click **OK**.

Defining Loadcase properties

- In the Treeview two analysis entries can be seen – one structural and one thermal. These are created automatically as a result of selecting a Coupled user interface option at the beginning of the example There is also a **Coupled analysis options** object. Loadcases need to be created for each construction stage to be modelled – for both the structural loadings and for the thermal loadings.

Defining a structural initialisation loadcase

To make the model easier to manipulate, the default analysis names will be changed to something more descriptive.

- In the Treeview right-click on **Analysis 1** and rename it to **Structural**

- In the Treeview right-click on **Analysis 1 (Thermal)** and rename it to **Thermal**

Defining a structural initialisation loadcase

- Right-click on **Structural** > **Loadcase 1** and rename it to **Structural Initialisation**

- Select **Structural Initialisation** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.

- On the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option. Choose a **Viscous** time domain from the drop-down list. Enter the Initial time step as **1e-6**, leave the Total response time as its default value, and set the Max time steps or increments to **1**

- Click **OK** to return to the Modeller window.

Note. The small initial time step of 1e-6 has been used to minimise the curing effects during this loadcase.

Note. In this instance the large default total response time has no significance because only one time step is processed. This load case will finish after the first small time step.

Defining a structural loadcase for construction stage 1

Analysis
Loadcase

- Ensure the **Structural** analysis is selected, then enter a loadcase name of **Structural stage 1** and click **OK**
- Select **Structural stage 1** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.
- On the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**
- Select the **Time domain** option. Choose a **Viscous** time domain from the drop-down list. Enter the Initial time step as **0.05**, the Total response time as **10**, and set the Max time steps or increments to **100**
- Select the **Automatic time stepping** option and then click the **Advanced** button.
- On the Advanced Time Step Parameters dialog set the Time step increment restriction factor to **1.5**, the minimum time step to **0.05** and the Maximum time step to **0.5** This will allow the time step to increase from its initial value of 0.05 up to the maximum value of 0.5 and thus reduce the number of increments required.
- Click **OK** to close the Advanced Time Step Parameters dialog and return to the Nonlinear & Transient dialog. On this dialog pick the **Coupling** button and change the **Interval between coupled reads** and the **Interval between coupled writes** to **0.05**, and click **OK**. Click **OK** to return to the Modeller window.

Defining a structural loadcase for construction stage 2

Loadcases can be copied and pasted in the Treeview. This saves having to re-enter similar details for each loadcase. Then, just the different values can be defined for each newly copied loadcase.

- Click on loadcase **Structural stage 1**. Press the copy toolbar button, followed by the paste toolbar button. This will create a new loadcase that is an exact copy named **Copy of Structural stage 1**. Rename this new structural loadcase to **Structural stage 2**

- Double-click the **Nonlinear and Transient** object for **Structural stage 2** and set the total response time as **20**

Defining a structural loadcase for construction stage 3

- Repeat the last two procedures to create another copy of the first loadcase called **Structural stage 3**, but set the total response time as **30**

Defining a thermal initialisation loadcase

- Right-click on the thermal Loadcase 2 and rename it to **Thermal Initialisation**
- Select **Thermal Initialisation** using the right-hand mouse button and from the **Controls** menu select the **Nonlinear and Transient** option.
- On the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**
- Select the **Time domain** option. Choose a **Thermal** time domain from the drop-down list. Enter the Initial time step as **1e-6**, leave the Total response time as its default value, and set the Max time steps or increments to **1**
- In the Solution Strategy section set the Max number of iterations to **15**
- Click **OK** to return to the Modeller window.

Note. The small initial time step of 1e-6 has been used to minimise the curing effects during this loadcase.

Note. In this instance again, the large default total response time has no significance because only one time step is processed. This load case will finish after the first small time step.

Defining a thermal loadcase for construction stage 1

Analyses
Loadcase...

- Select the **Thermal** analysis
- Enter a loadcase name of **Thermal stage 1**
- Click on loadcase **Thermal stage 1** using the right-hand mouse button and from the **Controls** menu choose the **Nonlinear and Transient** option.
- On the Nonlinear & Transient dialog tick the **Nonlinear** option and leave the incrementation type as **Manual**
- Select the **Time domain** option and note that the **Thermal** time domain option is the only one available for selection. Enter the Initial time step as **0.05**, the Total response time as **10**, and set the Max time steps or increments to **100**

- Ensure the **Automatic time stepping** option is selected and then click the **Advanced** button.
- On the Advanced Time Step Parameters dialog set the Time step increment restriction factor to **1.5**, the minimum time step to **0.05** and the Maximum time step to **0.5**
- Click **OK** to close the Advanced Time Step Parameters dialog and return to the Nonlinear & Transient dialog. On this dialog, pick the **Coupling** button and change the **Interval between coupled reads** and the **Interval between coupled writes** to **0.05**, and click **OK**. Click **OK** to return to the Modeller window.

Now create a copy of this loadcase:

Defining a thermal loadcase for construction stage 2

- Click on loadcase **Thermal stage 1**. Press the copy toolbar button, followed by the paste toolbar button. This will create a new loadcase that is an exact copy named **Copy of Thermal stage 1**. Rename this new structural loadcase to **Thermal stage 2**
- Double-click the Nonlinear and Transient object for **Thermal stage 2**, set the total response time as **20**

Defining a thermal loadcase for construction stage 3

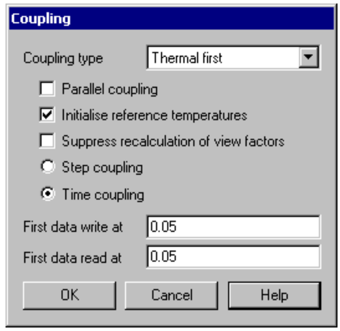
- Repeat the last procedure to create another copy of the first loadcase called **Thermal stage 3**, but set the total response time as **30**

All loadcases have now been defined. Element activations and model attributes will be assigned to these loadcases later.

Note. The total response time in each loadcase represents the total time at which that loadcase in the analysis ends, and the next one begins.

Set coupled analysis options

In this analysis, the thermal results affect the structural behaviour (that is, temperatures cause thermal stresses), but the structural results do not affect the thermal behaviour. A semi-coupled analysis is therefore required, and the appropriate coupled analysis options must be chosen.



- Double click on the **Coupled analysis options** object in the Treeview.

Set the coupling type as **Thermal first** from the drop down list.

- Select the **Initialise reference temperatures** option.
- Ensure the **Time coupling** radio button is selected.
- Set the **First data write** to **0.05** and the **First data read** to **0.05** (i.e. every step)
- Click **OK**

Load Curves

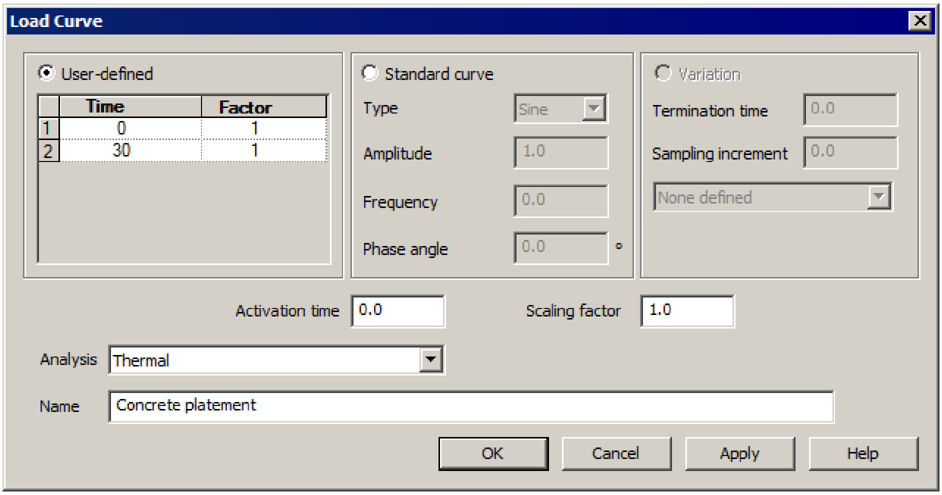
Load curves are used to describe the variation of the loading in nonlinear, transient and Fourier analyses. In a transient thermal analysis, such as in this example, the environmental temperature loading on specific surfaces of the dam will change with time. As the construction progresses some surfaces that were initially external surfaces (that could dissipate heat to the outside environment) become internal ones that can only dissipate heat to other parts of the structure.

Defining Load Curves

- Right-click on the **Thermal** analysis icon, and choose **New** and **Load Curve** to open the Load Curve dialog box.

Defining a concrete placement load curve:

- In the User-defined section of the dialog enter a Time of **0** and a Factor **1**, press the **Tab** key to create a new row. On this new row enter a Time of **30** and a Factor of **1**. Leave the default values of Activation time (0.0) and Scaling factor (1.0) unchanged. Name the load curve **Concrete placement** and click **Apply**



- With reference to data that follows, repeat this procedure four more times to add four more load curves called **External surface stage 1**, **External surface stage 2**, **External surface stage 3** and **Base and abutments**. The User-defined area of each of the dialog boxes should be filled-in according to the tables that follow.

Time	Factor
0	1
9.999	1

Table 1. External surface stage 1

Time	Factor
10	1
19.999	1

Table 2. External surface stage 2

Time	Factor
20	1
30	1

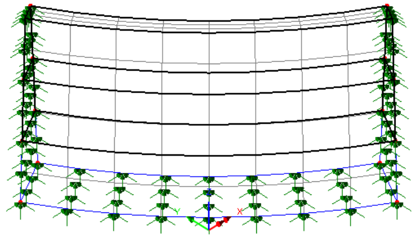
Table 3. External surface stage 3

Time	Factor
0	1
30	1

Table 4. Base and abutments

Assigning Activation and Deactivation attributes

Construction Stage 1



The elements not required for the first construction stage must be deactivated.

- In the Graphics window select the top two volumes of the dam.

Structural activation/deactivation

- Drag and drop the deactivation attribute **Deactivate** from the Treeview, ensuring that it is assigned to analysis **Structural** and loadcase **Structural Initialisation** and then clicking **OK**

Because the option to show the activated mesh only was set earlier in the example the view window will update to show only the mesh elements for the lowest volume.

Thermal activation/deactivation

- With the top two volumes still selected, assign the deactivation attribute **Deactivate** from the Treeview onto the selected features ensuring that it is assigned to analysis **Thermal** and loadcase **Thermal Initialisation**

Verifying self weight and activation assignments

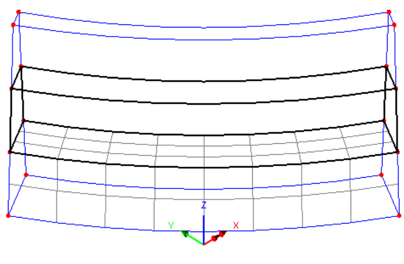
- If, during the course of this example, you need to check when particular elements become active in an analysis, select a feature and then, using the right-hand mouse button, choose **Properties**. Go to the **Activate Elements** tab. Highlighting an entry in the right-hand panel (that shows the assigned attributes) will show the loadcase in which the activation is assigned in the Loadcase box.

Construction Stage 2

The elements in the second construction stage now need to be activated.

- In the graphics window select only the second stage of the dam (i.e. the middle volume).

Structural activation/deactivation



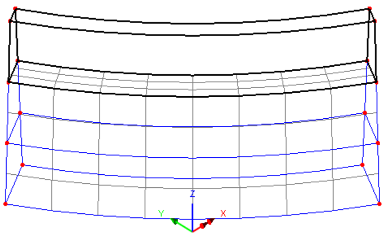
- Assign the activation attribute **Activate** from the Treeview, ensuring that it is assigned to analysis **Structure** and loadcase **Structural Stage 2**, then and clicking **OK**

Thermal activation/deactivation

- With the middle volume still selected, assign the activation attribute **Activate** from the Treeview ensuring that it is assigned to analysis **Thermal** and loadcase **Thermal Stage 2**

Construction Stage 3

The elements in the third construction stage now need to be activated.



- In the graphics window select only the top volume of the dam.

Structural activation/deactivation

- Assign the activation dataset **Activate** from the Treeview, ensuring that it is assigned to analysis **Structural** and loadcase **Structural stage 3**, and then clicking **OK**

Thermal activation/deactivation

- With the top volume still selected, assign the activation attribute **Activate** from the Treeview ensuring that it is assigned to analysis **Thermal** loadcase **Thermal stage 3**

Adding self-weight

In the Treeview right-click on the **Structural** analysis icon and select **Add gravity** to apply self-weight loading to all the structural loadcases in the analysis.

Note. Loadcases with automatic self-weight loading can easily be identified by the gravity loadcase icon.

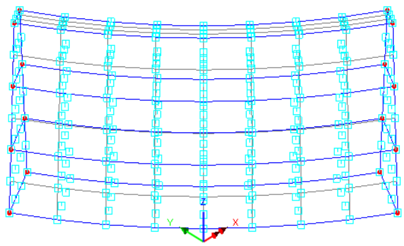
Assigning thermal loading (load curves)

Load curves were defined to model the thermal loading and these need to be assigned to appropriate features of the model.

Assigning the Initial Concrete Temperature

The Initial Concrete Temperature attribute will apply to all the stages of the model as they are activated.

- With all three volumes selected, assign the thermal loading attribute **Initial Concrete Temperature** from the Treeview, ensuring that it is assigned to Load curve **Concrete Placement** in the drop-down list before clicking **OK.**



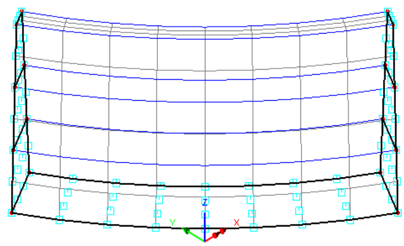
Assigning ground temperature

Ground temperature loading only applies to selected surfaces. These could be selected one-by-one but, as an alternative, they can also be selected by through their support assignment. First, a structural loadcase must be set active to allow selection of an assigned structural attribute:

- In the Treeview right-click on the **Structural Initialisation** loadcase and select **Set active.**

- In the Treeview right-click on the Supports entry **Fixed** and choose **Select Assignments**

- With the bottom and side surfaces only selected, assign the thermal loading attribute **Ground temperature** from the Treeview, ensuring that it is assigned to Load curve **Base and abutments** in the drop-down list before clicking **OK**.



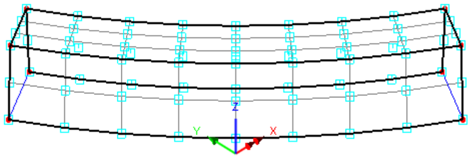
Assigning air temperatures and thermal supports

The surfaces in contact with the air change as the construction progresses. To help ensure the correct surfaces are selected prior to assigning air temperature loading the pre-defined Groups (that were set-up at the start of the example to mimic the construction process) will be used. Thermal supports also need to be assigned to the model at each construction stage to free-up nodes in the model that are restrained at a prescribed temperature by the LUSAS Modeller.

Construction Stage 1

- In the Treeview right-click on the group name **Stage 1** and select the **Set as Only Visible** option.

- Click in the graphics area to allow a keyboard short-cut to be used.

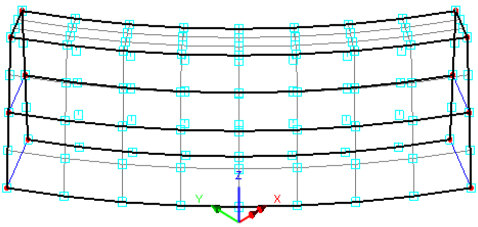


- Holding-down the **Shift + S** keys select the front, back and top Surfaces of the concrete that forms stage 1

- Assign the thermal loading attribute **Air temperature** from the Treeview, ensuring that it is assigned to Load curve **External surface stage 1** in the drop down menu before clicking **OK**

- Select the Volume and assign the thermal support **Free** from the Treeview, ensuring that it is assigned to volumes and selecting **Thermal stage 1** from the loadcase drop down list before clicking **OK**

Construction Stage 2



- In the Treeview right-click on the group name **Stage 2** and select the **Set as Only Visible** option.

- Click in the graphics area to allow a keyboard short-cut to be used.

- Holding-down the **Shift + S** keys select the front, back and top surfaces of the concrete that forms stage 2 (5 surfaces in total)

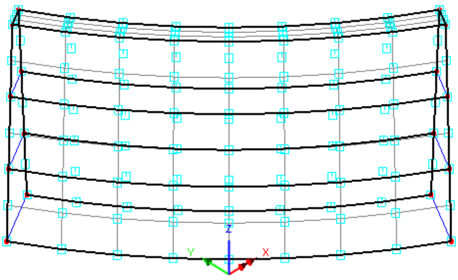
- Assign the thermal loading attribute **Air temperature** from the Treeview, ensuring that it is assigned to Load curve **External surface stage 2** in the drop down menu before clicking **OK**

- Select the two Volumes representing this construction stage and assign the thermal support **Free** from the Treeview, ensuring that it is assigned to volumes, and the **Thermal** analysis. Enable **From loadcase (nonlinear and transient analysis)** and then select **Thermal stage 2** from the loadcase drop down list before clicking **OK**

Construction Stage 3

- In the Treeview right-click on the group name **Stage 3** and select the **Set as Only Visible** option.

- Click in the graphics area to allow a keyboard short-cut to be used.



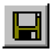
- Holding-down the **Shift + S** keys select the front, back and top surfaces of the concrete that forms stage 3 (7 surfaces in total)

- Assign the thermal loading attribute **Air temperature** from the  Treeview, ensuring that it is assigned to Load curve **External surface stage 3** in the drop down menu before clicking **OK**

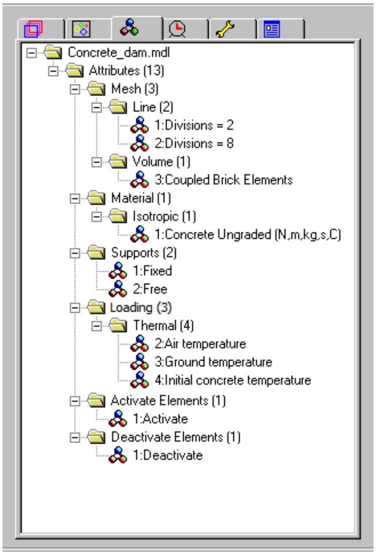
- Select the three Volumes representing this construction stage and assign the thermal support **Free** from the  Treeview, ensuring that it is assigned to volumes and selecting **Thermal stage 3** from the loadcase drop down list before clicking **OK**

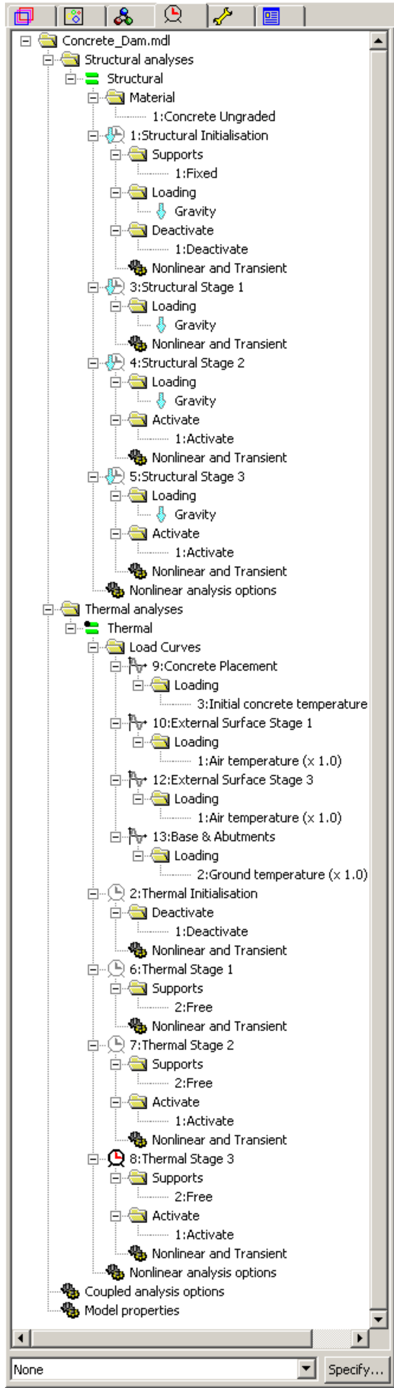
The model is now complete.

File
Save

 Save the model file.

Checking the Attributes and Loadcase Treeviews





In defining and modelling the staged construction of the dam the Attributes and the Loadcase Treeviews now contain all the information for the LUSAS Solver to carry out an analysis.

For a successful analysis your model should contain Treeviews similar to those shown.. Right-clicking on attribute and loadcase data allows any defined values to be checked. If your model has potential errors in your Attribute data or in your Loadcase Treeviews, a file is provided to enable you to re-create the model from scratch and run the analysis successfully. See the next page for details.

Note. The structural and thermal loadcases are always tabulated and solved in Treeview order from top to bottom. The thermal loadcases can be solved before the structural ones by stating the coupling type in the Coupled analysis object.

Running the Analysis

With the model loaded:

Open the **Solve Now** dialog. Ensure that the **Thermal** and **Structural** analyses are selected, 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 Treeview.

In addition, 4 files will be created in the Associated Model Data directory where the model file resides:

- **concrete_dam_therm.out** this output file contains details of model data, assigned attributes and selected statistics of the thermal analysis.
- **concrete_dam_struct.out** this output file contains details of model data, assigned attributes and selected statistics of the structural analysis.
- **concrete_dam_therm.mys** this is the LUSAS results file from the thermal analysis which is loaded automatically into the Treeview to allow results to be viewed.
- **concrete_dam_struct.mys** this is the LUSAS results file from the structural analysis which is loaded automatically into the Treeview to allow results to be viewed.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to a output files 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 the following file is provided to enable you to re-create the model from scratch and run an analysis successfully:

- **concrete_dam_modelling.vbs** carries out the modelling of the example.

File
New…

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 **concrete_dam**
- Select units of **N,m,kg,s,C**
- Ensure the **Coupled** user interface is selected and click the **OK** button

File
Script >
Run Script...

- To recreate the model, select the file **concrete_dam_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Run the analysis to generate the results.

Viewing the Results

Analysis loadcase results for each time step are present in the Treeview. The time step result for the first thermal loadcase is set to be active by default.

Temperature contours throughout the dam are to be investigated for each stage of the construction process. Animations of the change in temperature and of the stress in the dam will be created.

Thermal Results

To illustrate the variation of temperature throughout the model with time, an animation will be created showing contours at different time steps. Firstly, contours of temperature for the current time step will be created.

- Turn off the display of the **Mesh, Deformed Mesh** and **Attributes** layers in the Treeview.
- With no features selected click the right-hand mouse button in a blank part of the Graphics window and select the **Contour** option to add the contour layer to the Treeview. Select **Potential** from the entity drop down list and ensure **PHI** is selected in the component drop down list. Click **OK**.

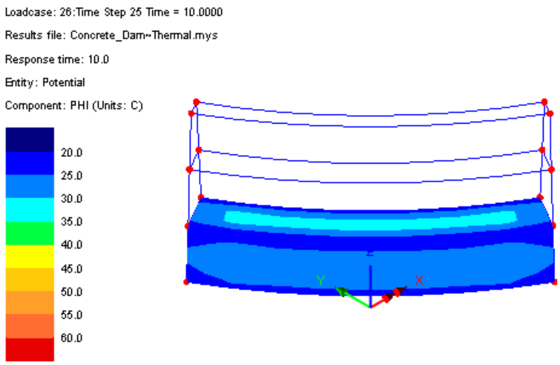
A plot showing initial contours at ambient temperature will be displayed.

When creating animations of contours it is preferable to have a static contour scale. By setting a Time Step active for the end of the construction stage under consideration an estimate of the maximum and minimum values required for the contour key can be obtained.

- In the Treeview right-click on the Thermal loadcase results for **Time Step 25** and **Set Active**

- In the Treeview double-click on the **Contours** layer.

- Select the **Contour Range** tab and ensure that the contour **Interval** contour is set to **5.0**. Set the **Maximum** and **Minimum** values to **60.0** and **20.0** respectively. Click **OK**



Animating Thermal Results

Note. The viewing parameters (e.g. the view angle and contour options) used in the animation are those that are currently specified in the window when the animation is loaded. It is therefore important to have an appropriate view visible in the Modeller window when running the animation wizard.

To create and save an animation of the temperature variation on the external surfaces of the dam throughout the whole analysis, complete the following procedure:

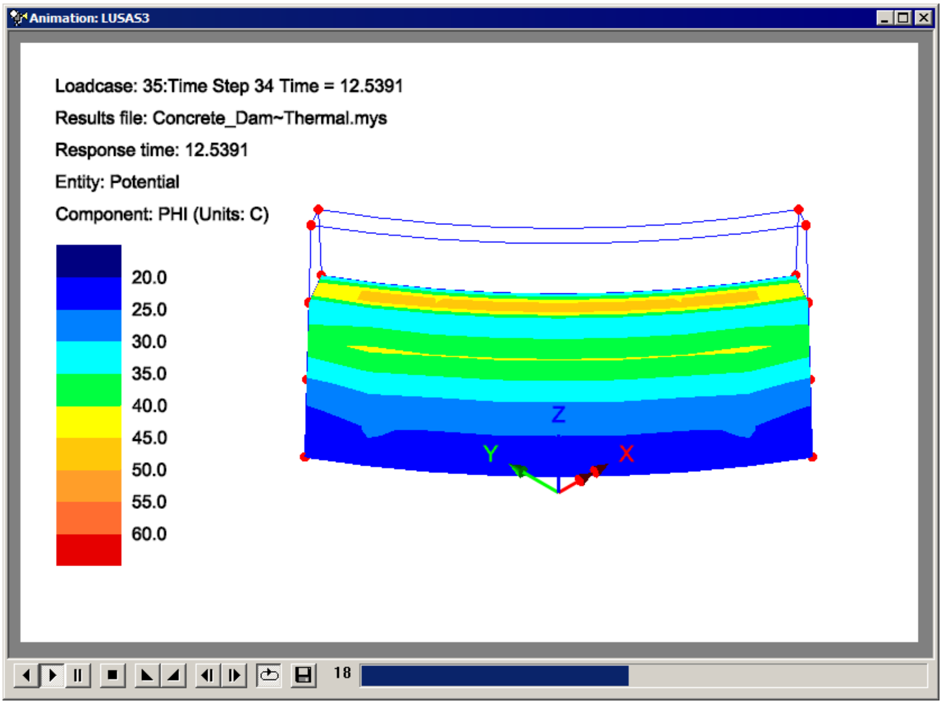
Utilities
Animation Wizard…

- Select the **Load history** radio button and click **Next**

- From the File drop-down menu on the Animation Load History dialog choose the **Thermal** results file

Note. In some analyses fine time increments are required for solution convergence but are not required for animation purposes. It is possible to reduce the size of (and the time required to produce) an animation by using the filter area of the dialog box.

- For the purposes of this example animate every 2nd Time Step by entering a Start value of **1**, a Step value of **2** and click **Filter**. Only every 2nd Time Step will appear in the Available area of the dialog. Select all of these filtered thermal results time steps (by clicking on the first time step, holding the **Shift** key, scrolling all the way to the bottom of the list and clicking on the last time step). Include these time steps by clicking on the >> button and then clicking **Finish**



LUSAS will load-in the selected time step results and open a window showing an animation of the variation of temperature with time. Animations may be saved in compressed AVI format for playback in other applications.

File
Save As AVI…

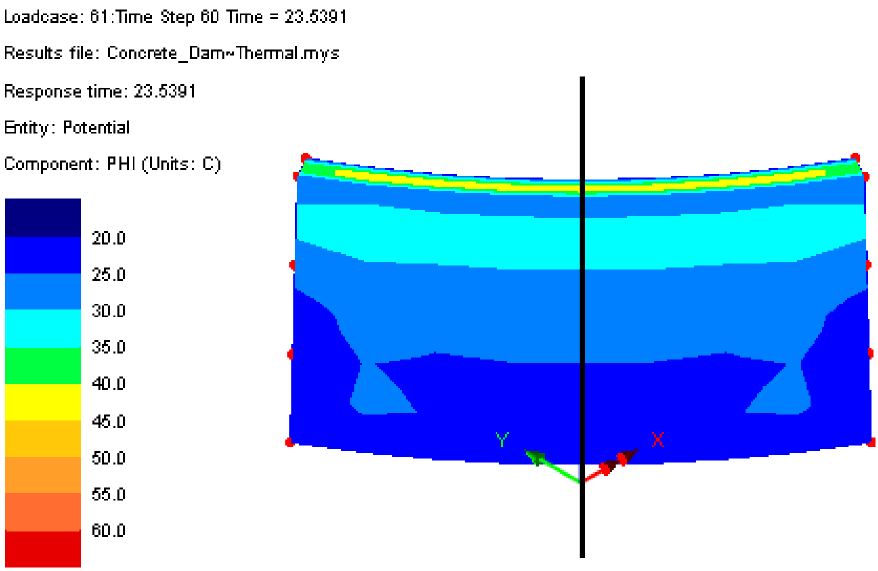
- Save the animation to your projects folder and enter **concrete_dam_thermal.avi** for the filename and press **Save**
- Close the Animation window.

Animating Thermal Results on a Slice Section

With heat of hydration analysis the maximum temperatures reached will occur within the concrete. To plot the change of temperature over time inside the dam slice sections are used:

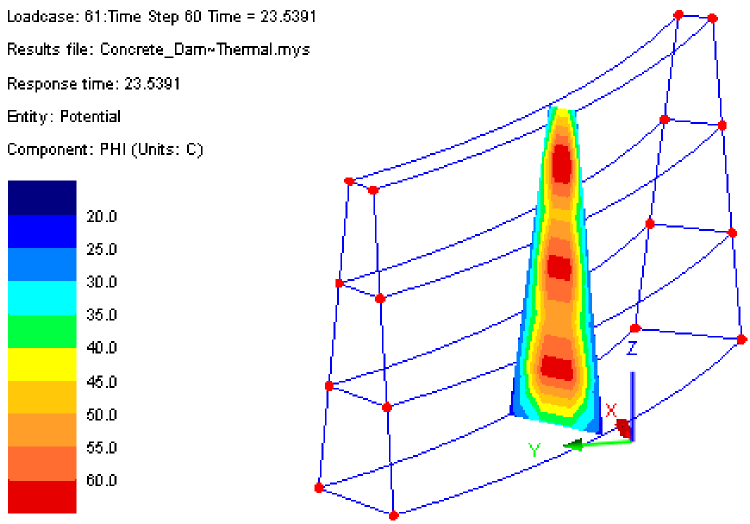
Note. When creating animations using slice sections the animation results are only created for the elements that are sliced. This means that for this example one of the Thermal loadcase Time Step results for Stage 3 must be set active so that the complete mesh for the dam can be seen.

- In the Treeview right-click on the Thermal Stage 3 loadcase results for **Time Step 60** and select **Set Active**



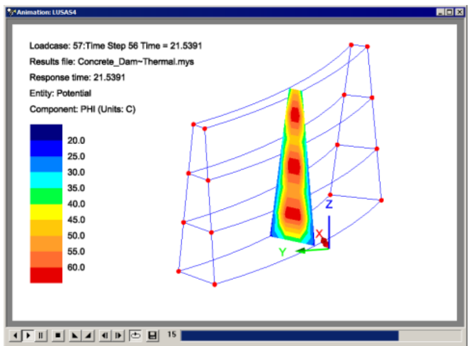
Utilities
Section Through 3D…

- Click **OK** to accept a grid size of 1
- Position the cursor above the dam in-line with the Z-axis and click and drag vertically downwards to define a slice section right through the centre of the dam. A group named Slice 1 will be created in the Treeview.
- In the Treeview select double-click the **Contour** option, ensure the **Display on slice(s)** option is selected and click **OK**
- Use the Dynamic rotate button to rotate the model around to the view shown.



To create and save an animation of the temperature variation on this slice section through the dam, complete the following procedure:

Utilities
Animation Wizard…



- Select the **Load history** radio button and click **Next**
- From the File drop-down menu on the Animation Load History dialog choose the thermal results file **Concrete_Dam_therm.mys**
- For the purposes of this example animate every 4th Time Step by entering a Start value of **1**, a Step value of **4** and click **Filter**. Only every 4th Time Step will appear in the Available area of the dialog. Select all of these filtered thermal results time steps (by clicking on the first time step, holding the **Shift** key, scrolling all the way to the bottom of the list and clicking on the last time step). Include these time steps by clicking on the >> button and then clicking **Finish**

File
Save As AVI…

Save the animation to your projects folder and enter **concrete_dam_thermal_slice.avi** for the filename. Click **OK**

- Close the Animation window.

Plotting contours on the whole model after slicing

- In the Treeview select double-click the **Contour** option and deselect the **Display on slice(s)** option and click **OK**.

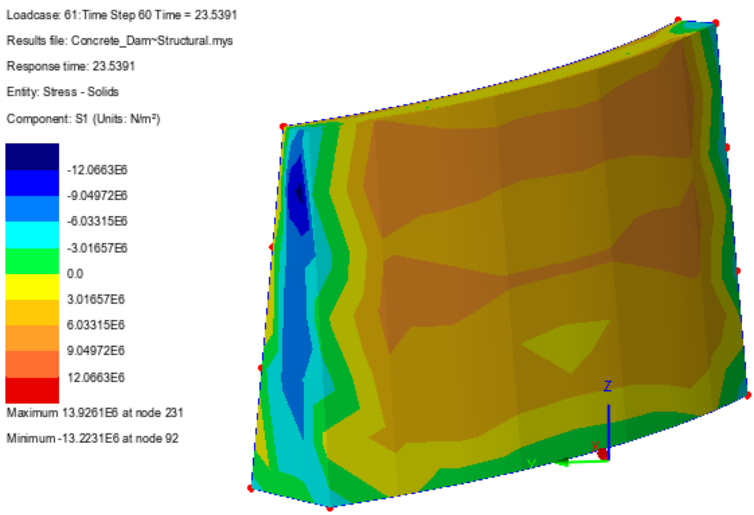
The display will revert to show contours of Potential of entity PHI on the whole model.

- Remove any slices generated by picking the **Slices** group from the Treeview, clicking right-hand mouse button and choosing the **Delete** menu item from the context menu.

Structural Results

Viewing and animating structural results is done in a similar manner to that described for thermal results, ensuring that only Structural loadcase results Time Steps are set active or used in an animation.

- In the Treeview right-click on the structural results for **Time Step 60** and **Set Active**
- Double-click the contours layer in the Treeview. On the dialog, set the results entity to **Stress – Solids** and component **S1**.
- Select the **Contour Range** tab and ensure that an **Automatic** contour range using **9** contours is being used. Deselect the **Maximum** and **Minimum** values and click **OK**



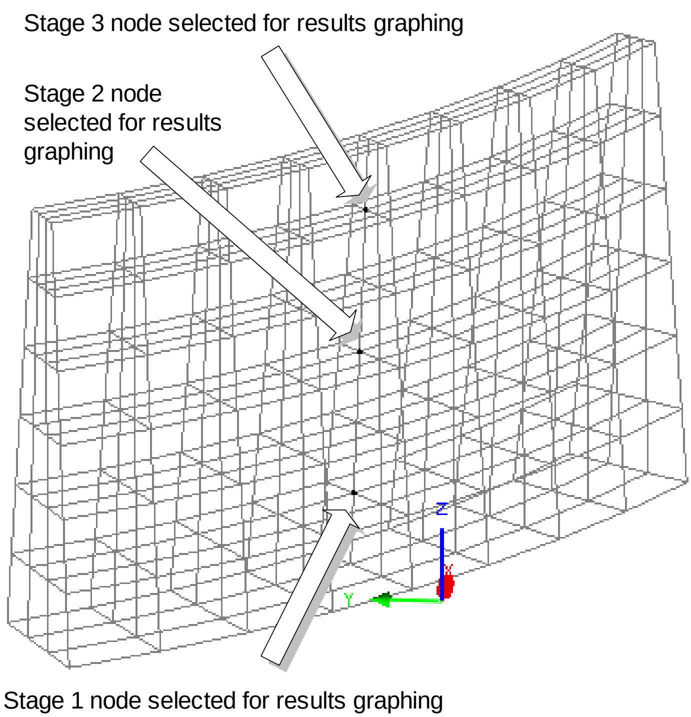
This will show contours of the maximum principal stress (S1) at the current time step.

Graphing of Results at a Selected Node

In analyses of this type it may also be useful to graph the changing temperature at a selected node (or nodes) as the construction progresses. This can be done by selecting a node of interest and using the Graph Wizard - making sure that only thermal results and thermal time steps are selected for plotting.

Continuing on from the last animation created:

- Delete the animation window and maximise the Model window
- In the Treeview, turn off the **Contour** and **Geometry** layers and turn on the **Mesh** layer.
- In the Treeview right-click on **Time Step 60** in the **Thermal Stage 3** loadcase results and **Set Active**
- Double-click the **Mesh** layer and select **Hidden parts** and deselect **Dotted** to view all the mesh.
- Select the stage 1 node for results graphing.



Utilities
Graph wizard…

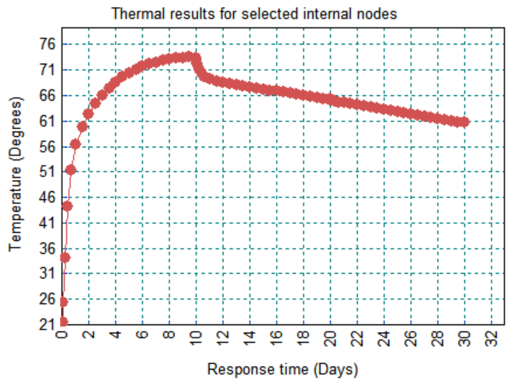
- Select **Time history** and click **Next.**

To define the graph X axis details:

- Select **Named** entity data and in the Entity data panel.
- In the Sample loadcases panel ensure **Whole analysis** is selected and choose the **Thermal** analysis from the drop-down box. Click **Next** to continue.
- Select **Response time** for the data to be plotted and click **Next.**

To define the graph Y axis details:

- Select **Nodal** entity data and click **Next**.
- Ensure entity **Potential** and results component **PHI** are selected for the node shown on the dialog and click **Next**.
- Enter the title as **Thermal results for selected internal nodes**
- Enter the X axis title as **Response time (Days)**



- Enter the Y axis as **Temperature (Degrees)** and click **Finish** to create the graph.
- Without deleting the Graph Window return to the Model Window

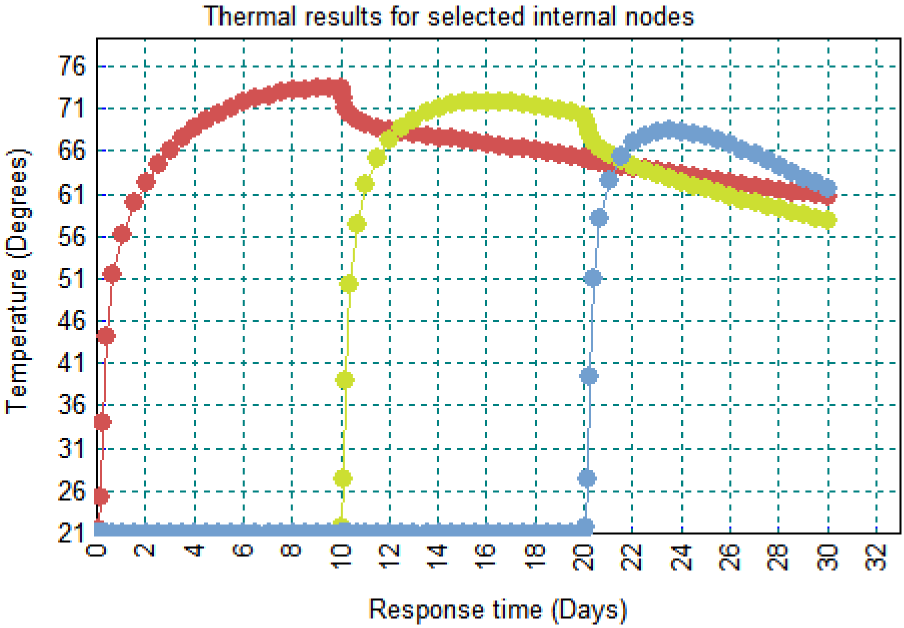
Graphing Results for Other Nodes

When a node in the mid-section of the dam for the other two construction stages is selected the following graph can be created, which clearly shows the initial rise and subsequent reduction of temperature after each stage of concrete is constructed.

Utilities
Graph wizard…

For each additional node identified, select it, and run the Graph Wizard. Repeat the above steps, instead selecting **Previously defined** for the X-Attribute Entity Data and re-using the **Response time** dataset.

Make sure that results for these additional modes are plotted onto the initial graph created by selecting the **Add to existing graph option** at the final stage of the wizard. Graph properties (such as editing axis or curve titles etc) can be edited by right-clicking on the Graph and choosing **Edit Graph Properties**



This completes the example.