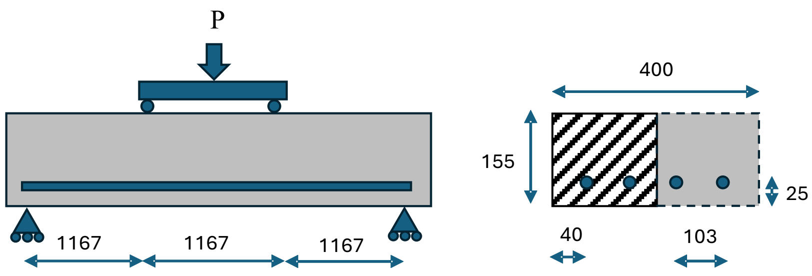
Bond slip modelling using embedded bars

For LUSAS version:	24.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil plus
With product option(s):	Nonlinear

Description

This overview example considers the four-point bending of a reinforced concrete slab including the effects of bond slip. The problem is modelled and analysed in 3D. One quarter of the geometry (as shaded below) is modelled due to symmetry. Steel platens are modelled at loading and support positions.



The modelling units used are N, mm, t, s, C throughout.

Keywords

Overview Example, 3D, Nonlinear, Concrete, Bond Slip, Embedded bars

Associated Files

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

- **bond_slip_complete.mdl** carries out the full modelling of the example.

Running LUSAS Modeller

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

Loading the supplied starting model

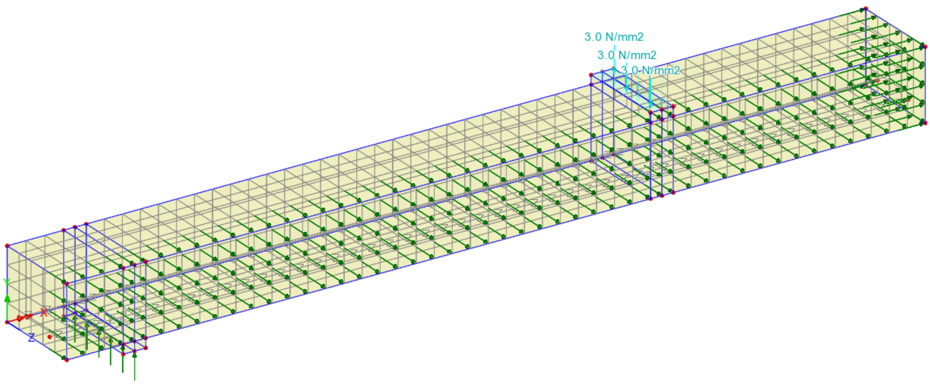
- To load the model, open the file **bond_slip_complete.mdl** that was downloaded and placed in a folder of your choosing.

Objectives

To investigate the load displacement response and evaluate the propagation of crack widths.

Modelling overview

The model is created using an analysis category of **3D Structural** and model units of **N,mm,t,s,C**.

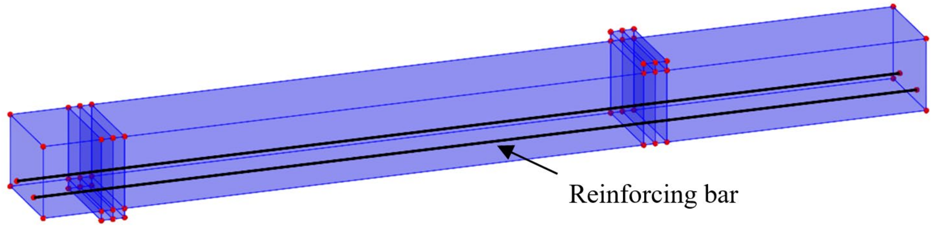


Model created from running the supplied script

Note. This 3D model example uses a Y-axis for the vertical axis instead of the default Z-axis.

Feature Geometry

The model is developed using point, line and surface features which are then used to form the volumes representing the concrete slab and steel platens, whilst lines are used to represent the reinforcing bars.

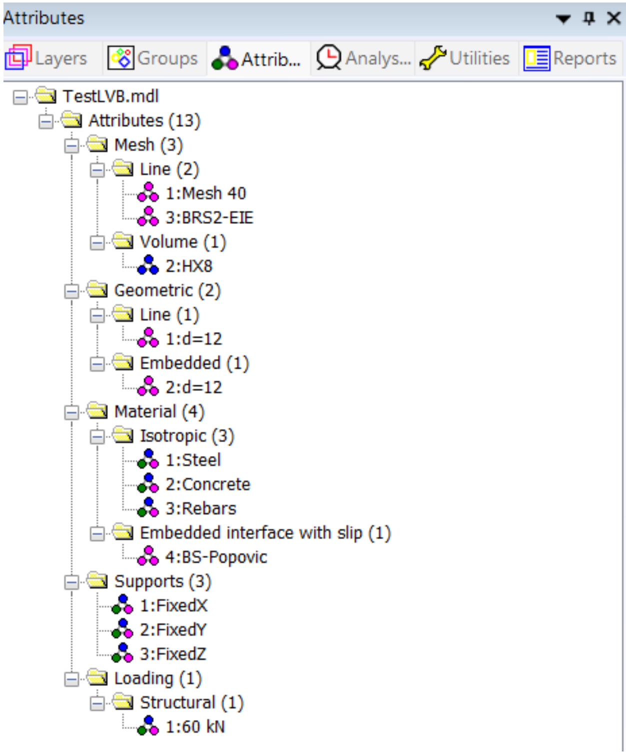


Concrete slab containing longitudinal reinforcing bars

Note. When using embedded bars it is important to note that reinforcing bars that are to be connected need to be represented by lines that join at common points.

Model Attributes

Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model.

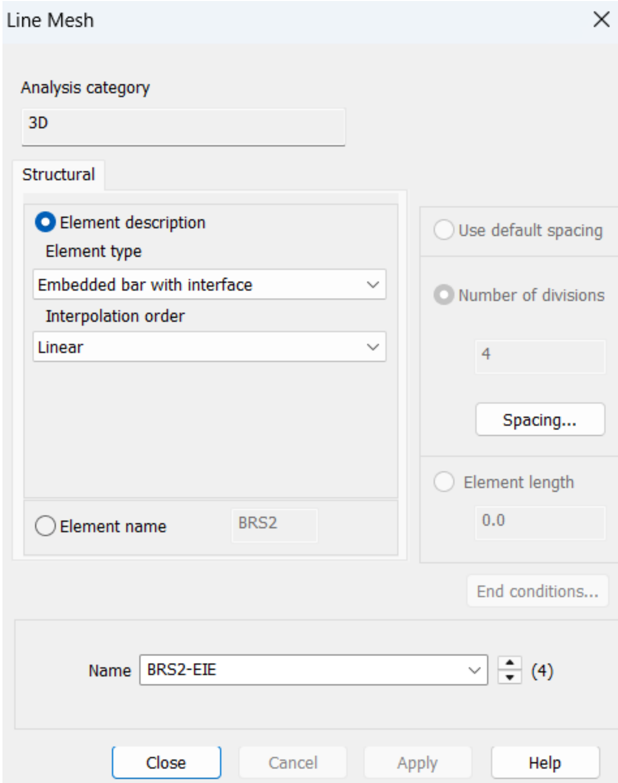


Model Attributes

Mesh

The volume is meshed with linear hexahedral elements the size of which has been set by assigning a null line element of length 40mm to the lines of the model.

The reinforcing bars are meshed using embedded bar elements that are found under the menu item **Attributes** > **Mesh** > **Line** > **Embedded bar with interface.**



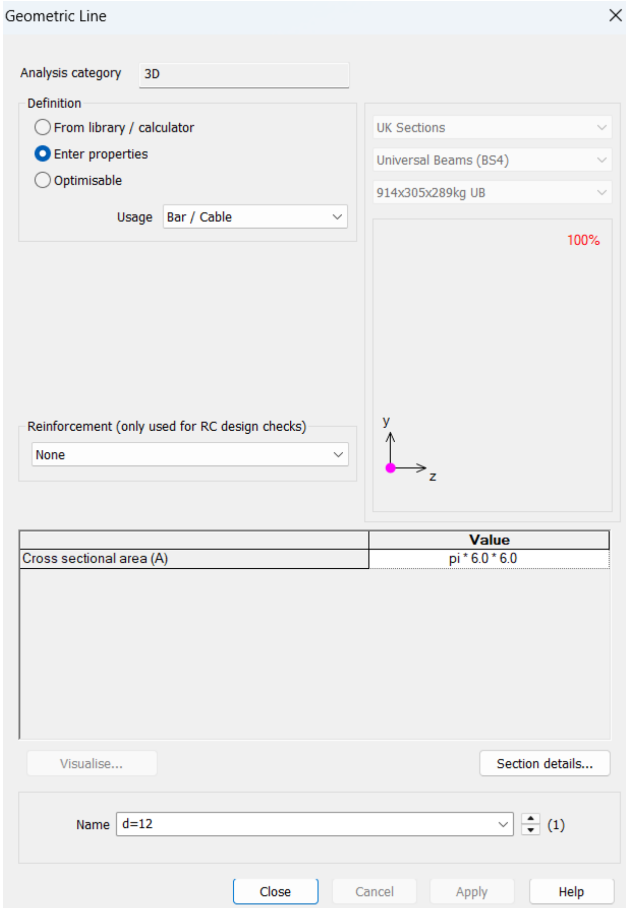
Line mesh dialog

The embedded bar with interface elements are ssigned to the lines that define the reinforcing bars.

Geometry

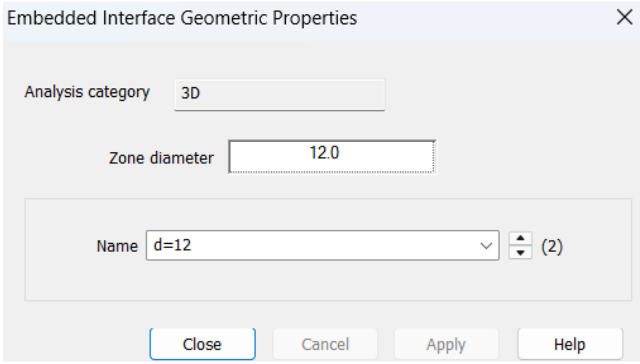
Geometric attributes are required for the reinforcing bars and the embedded interfaces.

The menu item **Attributes > Geometric > Line** is used to define the reinforcing bar properties.



Section properties for reinforcing bar

The menu item **Attributes > Geometric > Embedded interface…** is used to define the embedded interface properties.



Geometric properties for embedded interface

Both the geometric line and embedded interface geometric property attributes are assigned to the lines that define the reinforcing bars.

Materials

The material properties employed in each of the constitutive models can be found in the following Tables:

Concrete material properties

Young's modulus (MPa)	Poisson's ratio (-)	Compressive strength (MPa)	Tensile strength (MPa)	Strain at peak uniaxial compression (-)	Fracture energy (N/mm)
28,920	0.2	40	2.5	0.0022	0.13

Advanced concrete material properties

β_r	Z_0	ψ	m_g	m_{hi}	m_{ful}	r_σ	μ	m_{cf}	f_{pz}	it_{cf}	it_{mx}	R_s	A_p	A_n	C_{z1}	C_{z2}	C_{LM}
1.15	0.6	-0.1	0.425	0.5	5	1.25	0.8	0.05	60	2	2	0.03	0.6	0.75	8	8	3

Steel platen properties

Young's modulus (MPa)	Poisson's ratio (-)
210,000	0.2

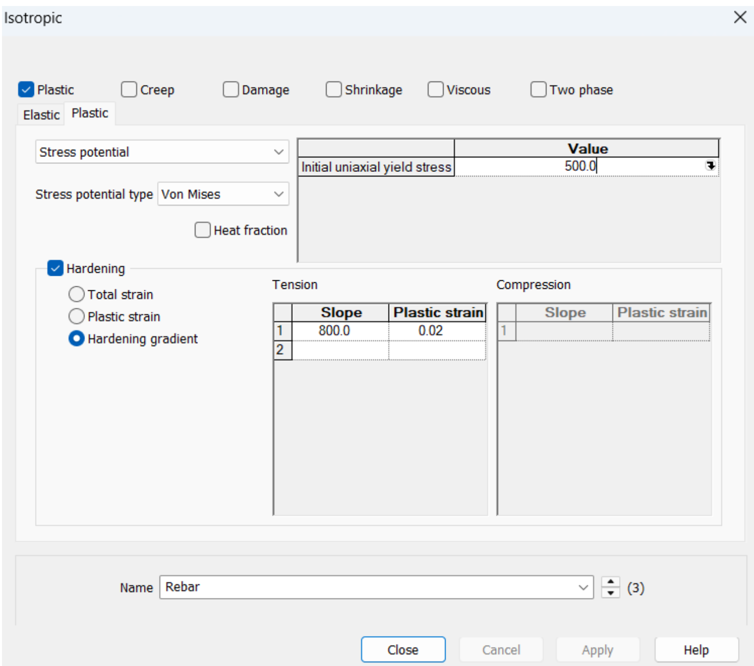
Reinforcement material properties

Diameter (mm)	Young's modulus (MPa)	Poisson's ratio (-)	Yield stress (MPa)	Hardening modulus (MPa)	Plastic strain (-)
12	210,000	0.2	500	800	0.02

Bond slip properties (Popovic)

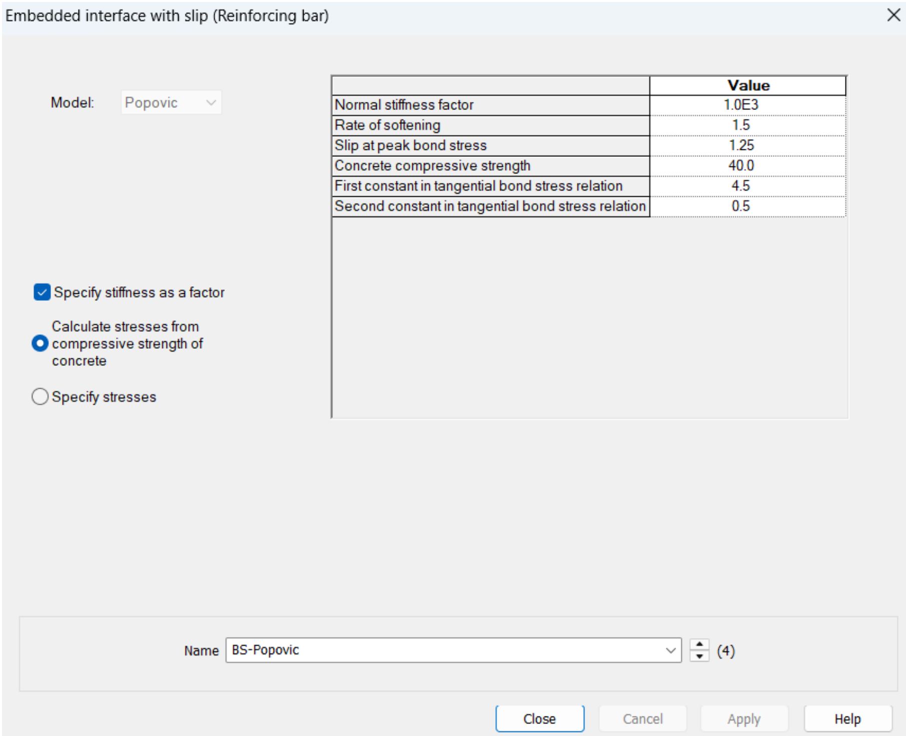
Normal stiffness (MPa/mm)	Curve parameter (-)	Slip at peak bond stress (mm)	Constant c1 (MPa^{c2})	Constant c2 (-)
28,920,000	1.5	1.25	4.5	0.5

The reinforcement properties are defined using the menu item **Attributes > Material > Isotropic**, with the plastic properties defined by checking the **Plastic** checkbox, then under the **Plastic** tab, checking the **Hardening** checkbox and selecting the **Hardening gradient** radio button.



Material properties for reinforcing bar

The bond slip properties are defined using the menu item **Attributes > Material > Specialised > Interface…** and selecting the **Embedded interface with slip (Reinforcing bar)** radio button.

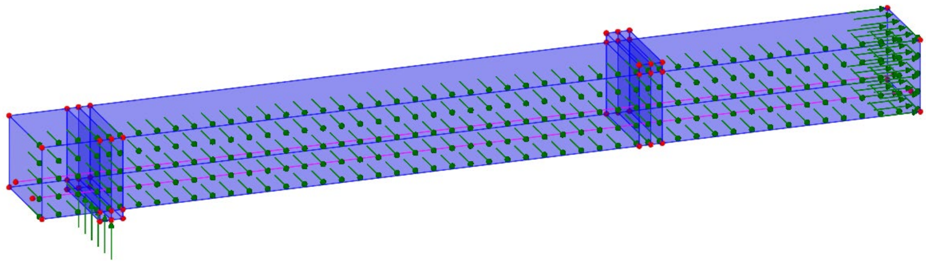


Material properties for bond slip

The material attributes for the reinforcing bar and for bond slip are both assigned to the lines that define the reinforcing bars.

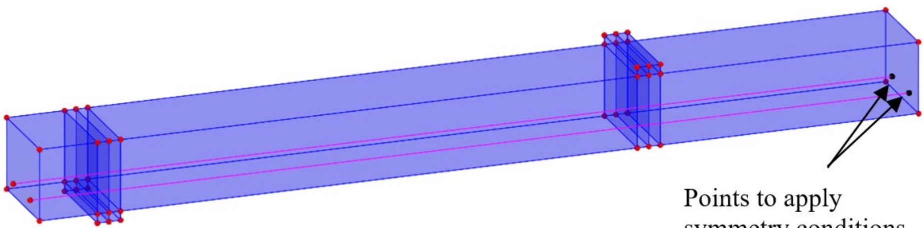
Supports

A vertical support is applied to the central line of the support platen, and symmetry conditions are applied to the central faces of the slab (i.e. the face at centre span and the face at the centre of the width of the slab).



Model geometry showing supports

Symmetry conditions are also applied to the ends of the reinforcing bars at the centre of the span. This can be achieved by selecting the relevant points and assigning the **FixedX** support.



Model geometry showing points to apply symmetry conditions

Loads

The loads considered on this model are the slab weight due to gravity and a quarter of the total applied load of 120kN (i.e. 30kN).

Note. The applied loading is defined as a 'Total' load of -15kN that is assigned to each of the upper two surfaces of the loading platen. Because 'Total' loading is applied per feature, this results in a load of 2 x -15kN = -30kN being applied to the model.

Analyses considered

Initial Phase

In this phase the gravity load is applied.

Main loading regime

In this phase the slab is loaded in four-point bending

Running the Analysis

 Ensure all analyses are selected and press **OK** to run the analysis.

Viewing the Results

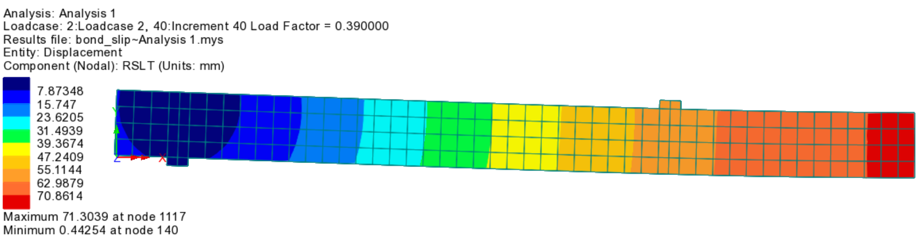
Analysis loadcase results are present in the Analyses treeview.

Main Loading Regime Results

The following images show the deformed shape and effective strain of the concrete and reinforcing bars after the main loading regime, in addition to the predicted crack widths.

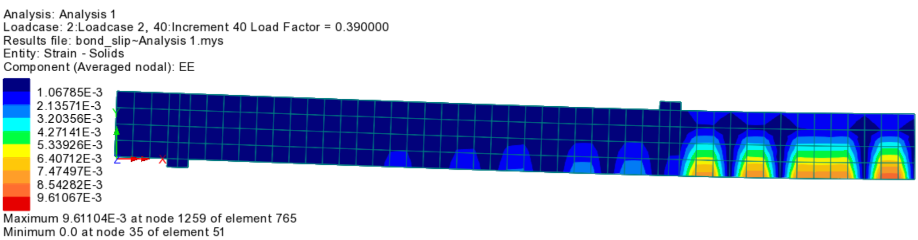
To see these plots:

- Turn off the display of the **Geometry** and **Mesh** layers and **Attributes**.
- Set active **Loadcase 2**
- Add the contours layer to plot contours of entity **Displacement** and component **RSLT**



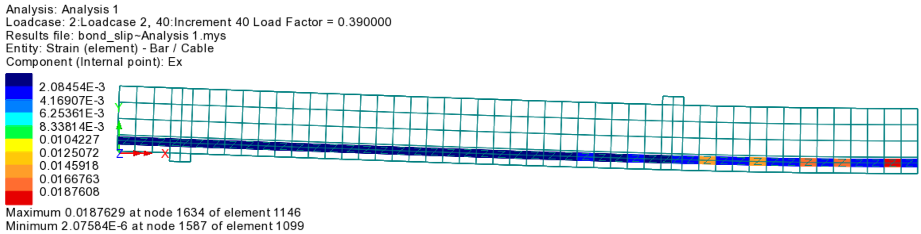
Resultant displacements

- Change the contours layer to plot contours of entity **Strain - Solids** and component **EE**



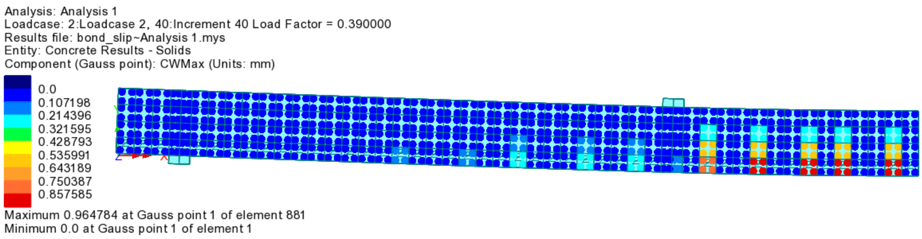
Effective strain

- Change the contours layer to plot contours of entity **Strain (element) – Bar / Cable** and component **Ex**



Effective strain in the bars

- Change the contours layer to plot contours of entity **Concrete Results - Solids** and component **CWMax**

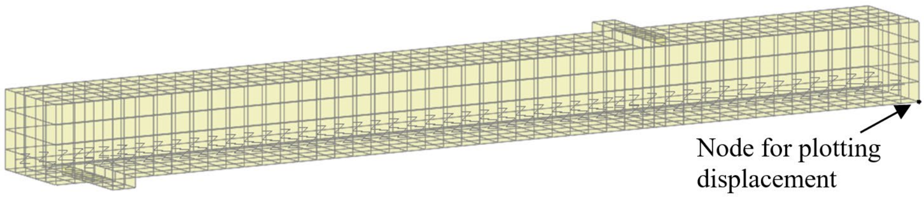


Predicted crack widths

Graph of load-deflection response

To demonstrate the load-deflection response of the slab, as well as the evolution of crack widths with increasing loads, the graph wizard can be used.

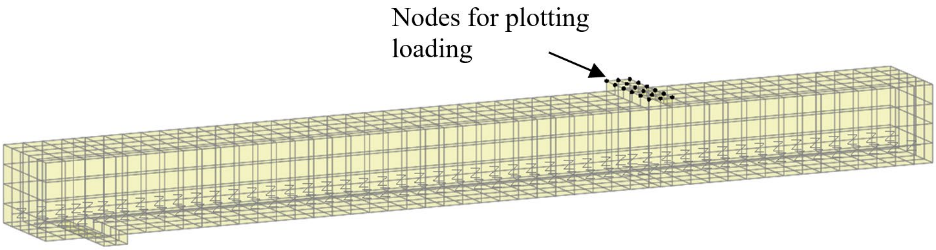
- Turn off the display of the **Contours** layer and **Deformed mesh** layer and turn on the display of the **Mesh** layer.
- Obtain a view of the model like that below.
- Select and make a note of the node number at the bottom of the centre of the beam.



Finite element mesh showing position of node for plotting displacement

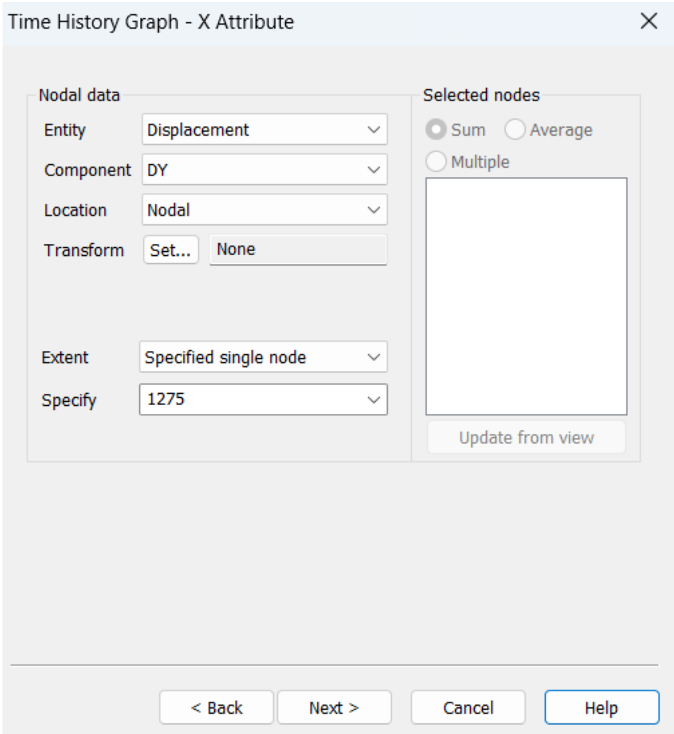
- Following this, the nodes defining the upper surface of the loading platen should be added to the selection.

Tip. View the model along the Z-axis, then holding the Shift + 'N' keys down to add to the current selection and select only nodes, box-select the top line of the platen.



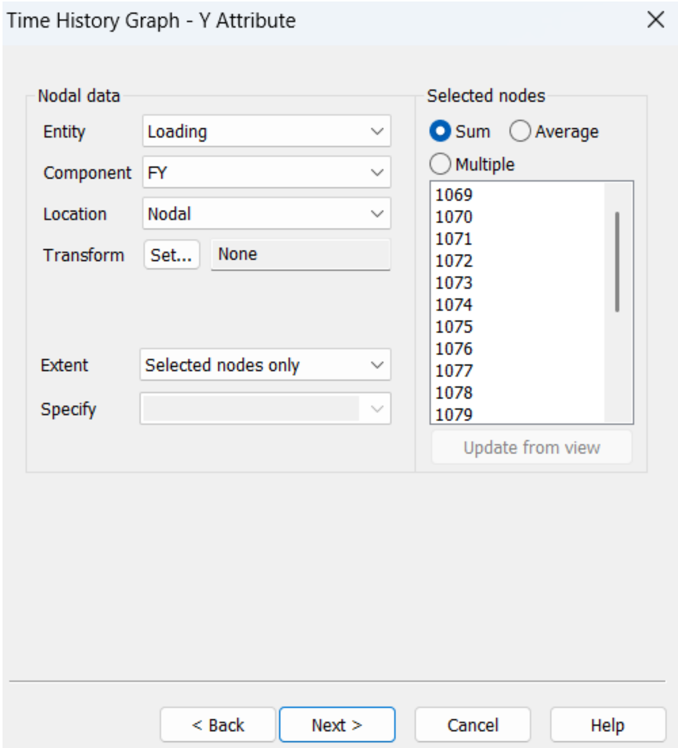
Finite element mesh showing selection of nodes for plotting loading

- Select the menu item **Utilities > Graph Wizard** to define graphing data, then select the radio button **Time history**.
- The X Attribute data is defined by choosing **Nodal**, followed by **Displacement** component **DY** specified at the single selected node noted earlier, and click **Next**.



X Attribute Definition

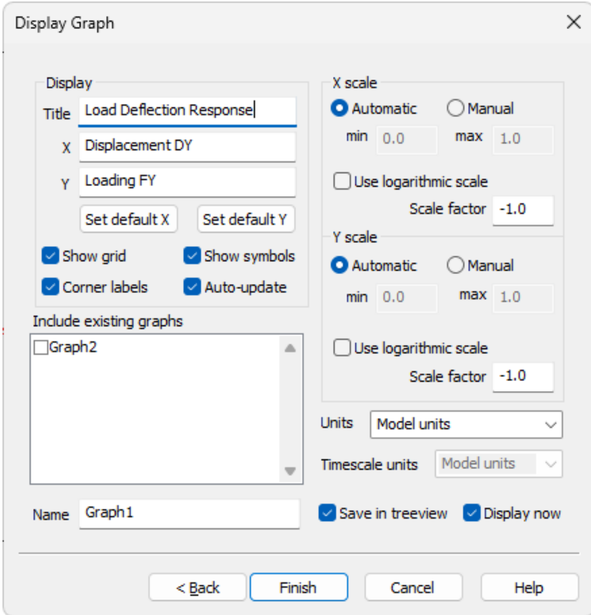
- The Y Attribute data is defined by choosing **Loading**, followed by component **FY** specifying **Selected nodes only**, with the radio button for **Sum** selected in the 'Selected nodes' panel.



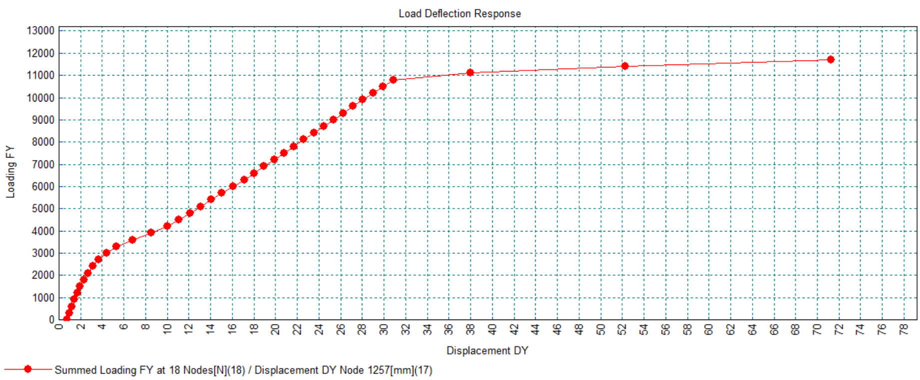
Y Attribute Definition

- Finally, add a graph title and set X and Y scale factors of **-1** as shown in the following dialog.

Note. Setting scale factors of -1 ensures that the displacement and loading values are plotted increasing in value along the X and Y axes of the graph.



Graph Display Options

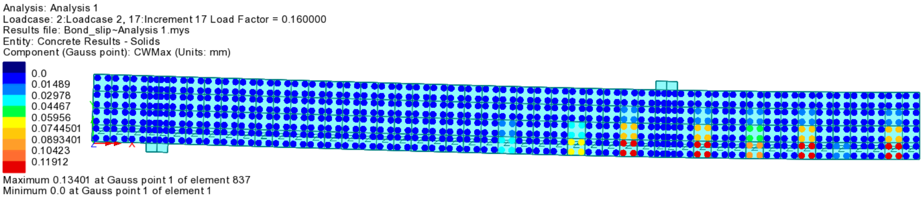


Load Deflection Response

Graph of crack width evolution

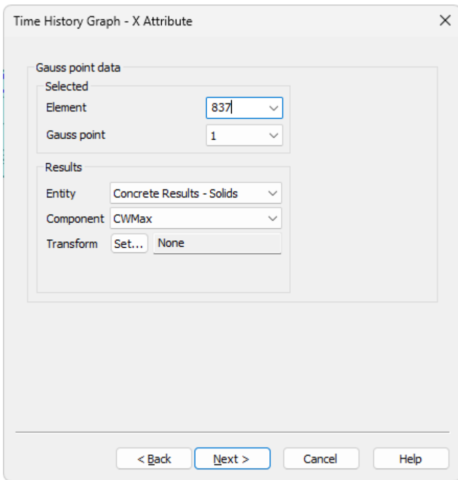
A similar process may be followed for plotting the crack width evolution. A key difference in this case, is that the crack width is a Gauss point value.

- Select the model view tab and turn on the display of the **Contours** layer, and set entity **Concrete Results - Solids** and component **CWMax**,
- In the Analysis treeview, set active the load increment corresponding to a load factor of **0.16**
- On the contour key, make a note of the element number and Gauss point number of the location of the maximum crack width – in this case element **837** and gauss point **1**.



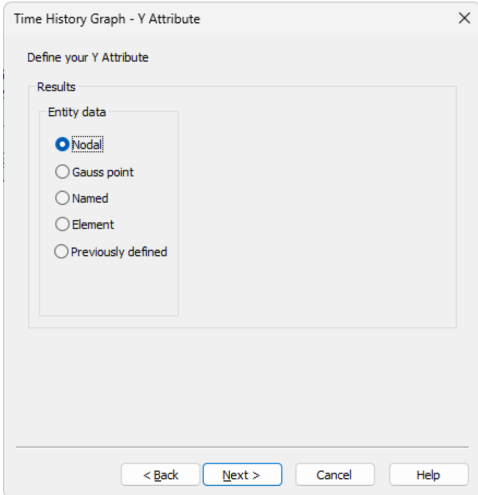
Crack width contours at a load factor of 0.16

- If not still selected, hold down the '**N**' key, and box-select the nodes in the top of the platen.
- Use the menu item **Utilities > Graph Wizard** to define graph data and select the radio button **Time history**.
- The X Attribute is defined as the **Gauss point**, click Next, and select **Concrete Results – Solids** component **CWMax** specified at the Gauss point number and element number noted earlier.



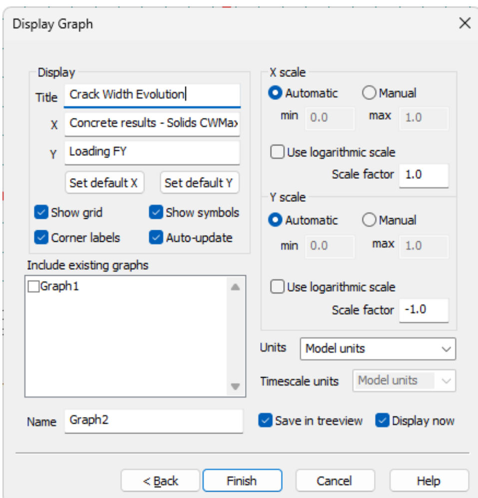
X Attribute Definition

- The Y Attribute data is defined by choosing **Nodal**, clicking **Next** and selecting the **Loading** component **FY** specifying **Selected nodes only** (for the nodes that define the upper surface of the loading platen), with the radio button for **Sum** selected in the 'Selected nodes' panel.

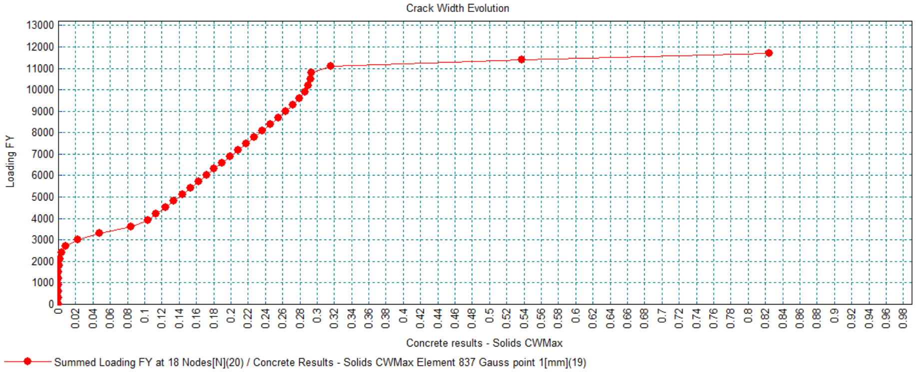


Y Attribute Definition

- Finally, add a graph title and set the X and Y scale factors of **1.0** in X and **-1.0** in Y as shown in the following dialog.



Graph Display Options



Crack Width Evolution

This completes the example.