# Nonlinear Analysis of a Concrete Beam

For LUSAS version:	16.0
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.

All dimensions in millimetres   Point Load   C-L

BAR3 Elements   QPM8 Elements

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.
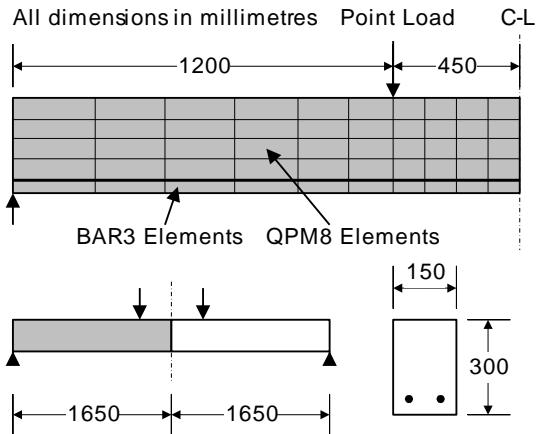
Objectives

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, Inplane, 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 *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter a File name of **beam_nl**

- Use the default User-defined working folder.

- Ensure an Analysis type of **Structural** is set.

- Select an Analysis Category of **2D Inplane**

- Set Model units of **N,mm,t,s,C**

- Leave the Timescale units as **Seconds**

- Select a Startup template of **2D Plane Stress**.

- Ensure the Layout grid is set as **None**

- Enter a Title of **Nonlinear Concrete Beam**

- 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 >
    By Coords...
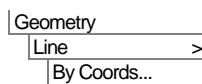
Enter coordinates of (**0, 0**), (**1200, 0**) and (**1650, 0**) to define two Lines representing the bottom of the left hand span of the beam. Click the **OK** button to finish.
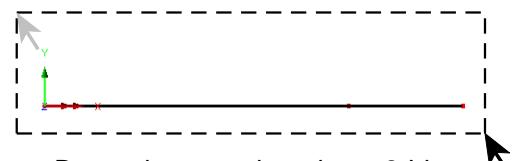
Drag a box to select these 2 Lines

- Select both Lines just drawn by dragging a selection box around them.

Geometry
Surface >
    By Sweeping...
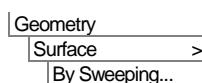
Enter a translation value of **25** in the **Y** direction to create the Surface which represents the concrete cover from the face of the beam to the centreline of the reinforcement.

- Click the **OK** button.

- Select the upper Lines of both of the Surfaces just drawn as shown.

Select these 2 Lines

Geometry
Surface >
    By Sweeping...
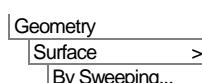
Enter a translation value of **275** in the **Y** direction to create the Surface which represents the extent of the concrete above the centreline of the reinforcement.

3

- 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.

Select these 2 Lines
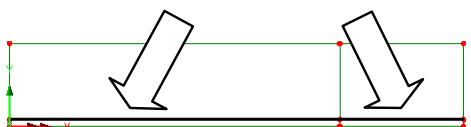
The Surfaces representing the concrete are to be grouped together.

- 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.
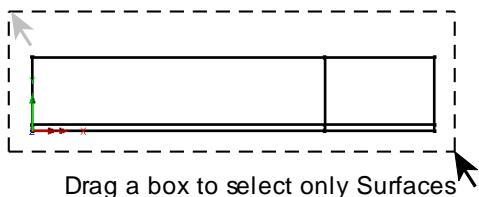

Drag a box to select only Surfaces

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.

- 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, quadratic order, 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, quadratic order, 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..

- Select **Plane stress, Quadrilateral, Quadratic** elements.

- Enter the attribute name as **Plane stress quadratic**

- 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, quadratic order, divs = 6 graded**

The Line mesh properties dialog will appear.

- Change the element type to **None**

- Change the attribute name to **Null, div=6 graded**

- Click the **OK** button to add the attribute to the 🔩 Treeview.

Defining the Geometric Properties

- Select **Bar** 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.

- 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.

- 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.
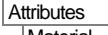
- Enter a Young's modulus of **42000**, a Poisson's ratio of **0.2** and leave the mass density field blank.

- 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**

- Leave the Strain at peak uniaxial compression as the default value.

- Select the **Strain at end of softening curve** radio button, and set the value 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, quadratic order, divs = 6 graded** from the 🔵 Treeview onto the selected Line.

Select this Line for
Bar Elements - Divs=6 graded

Select this Line for
Bar Elements - Divs=4

- Select the right hand Line of the two Lines representing the bars.

- Drag and drop the Line mesh attribute **Bar, quadratic order, 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 made invisible and the lines of the Concrete group will be made visible.

- Select the left-top and left-bottom Lines as shown.

- From the 🔵 Treeview drag and drop the Line mesh attribute **Null, div=6 graded** onto the selected features.

Select this Line

... and this Line

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

- Drag and drop the Surface mesh attribute **Plane stress quadratic** from the 🔵 Treeview onto the selected features.
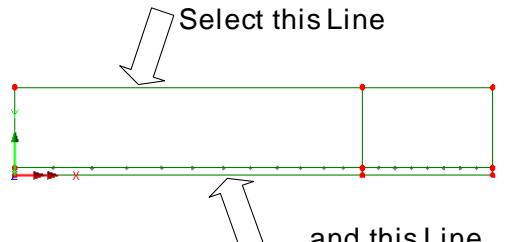
A graded mesh will be drawn on the left-hand Surface and a uniform mesh will be drawn on the right-hand Surface.
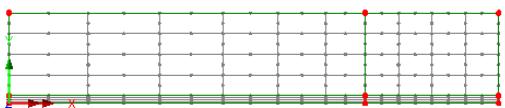
- Drag and drop the geometry attribute **Beam Thickness** from the 🔵 Treeview onto the selected features.

⚡ 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 for Analysis 1 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 **Null, div =1** from the 🔵 Treeview onto the selected Lines.
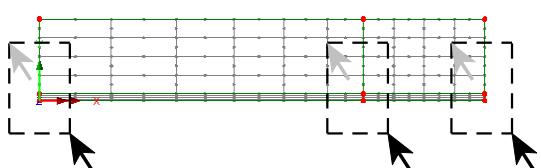
The mesh will be redisplayed with the revised mesh pattern.

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.
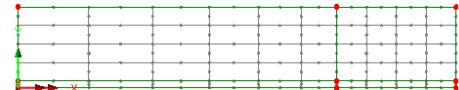
Supports

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.

Select these 2 Lines for support 'Fixed in X'

- 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**

Select lower Point for support 'Fixed in Y'

- 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**
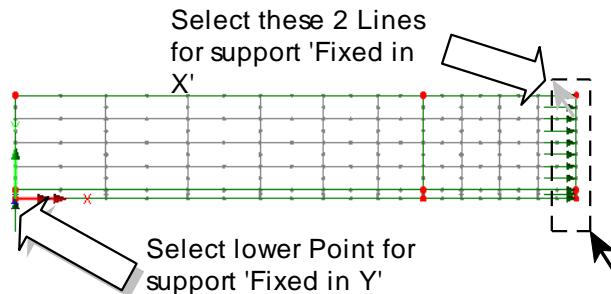
Loading

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 **Concentrated load in Y Dir** loading value of **-1**

- 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.

Select this Point

- Ensure the **Assign to points** option is set and click **OK** to assign the load to **Analysis 1**, **Loadcase 1**.
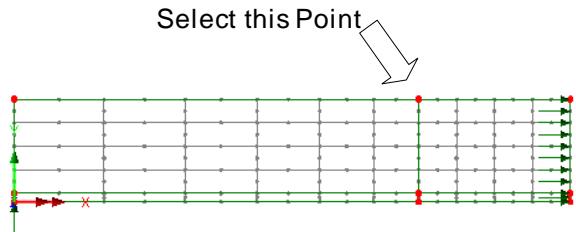
Nonlinear Control

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 from the **Controls** menu select **Nonlinear & Transient**

Select this Point
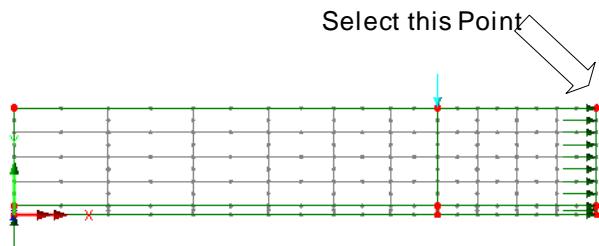
The Nonlinear & Transient dialog will appear:
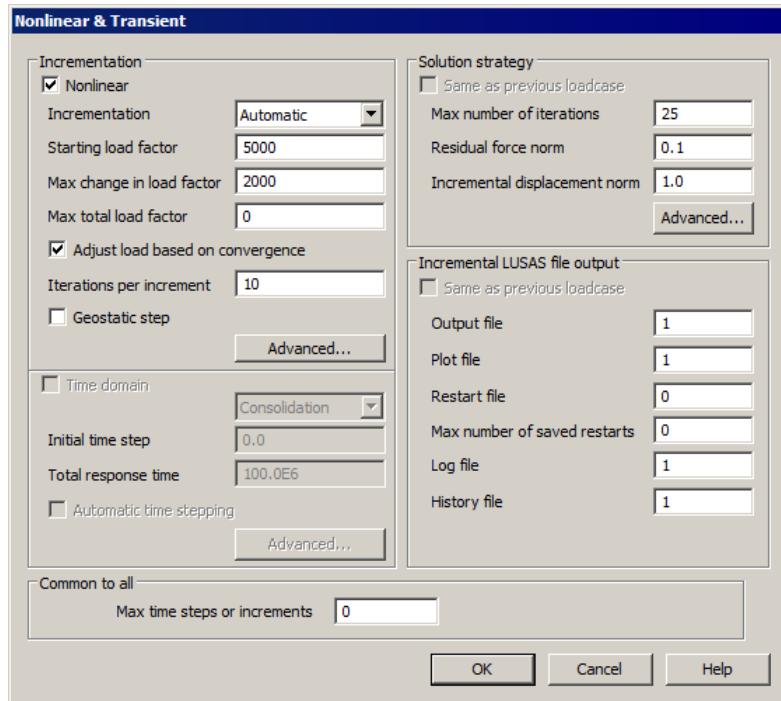
Select the **Nonlinear** option and set Incrementation to **Automatic**

- 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** because 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.

- In the Incrementation section of the dialog, select the **Advanced** button.
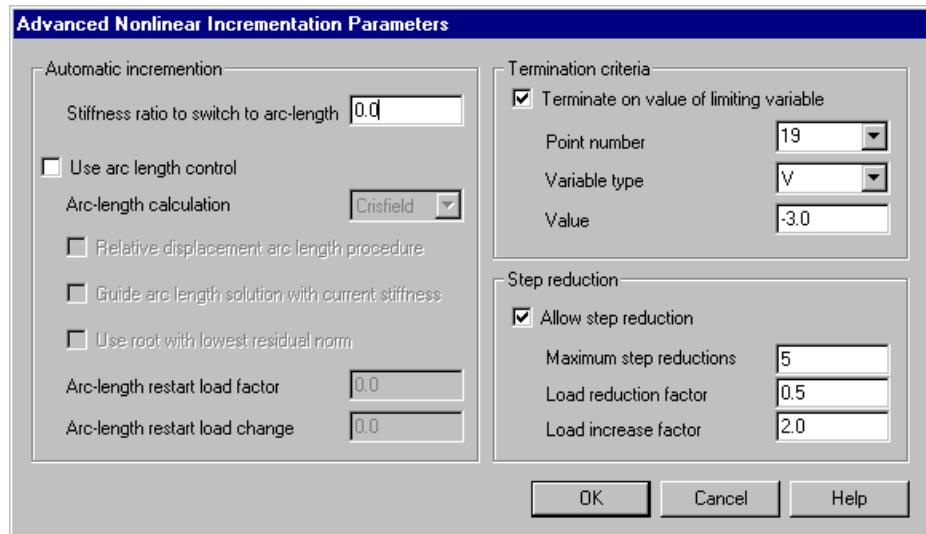
- Ensure that the **Stiffness ratio to switch to arc length** value is set to **0.0**

- Select the **Terminate on value of limiting variable** option.

- The point number of the previously selected point (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 **Options** tab.

- Click on the **Element Options** button and select the **Fine integration for stiffness and mass (structural)** option.

- Click the **OK** button to return to 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.
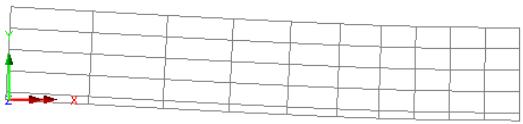
Creating a Load versus Displacement Graph

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:

Select this Node

- With the **Deformed mesh** layer visible, select the top node on the axis of symmetry as shown.
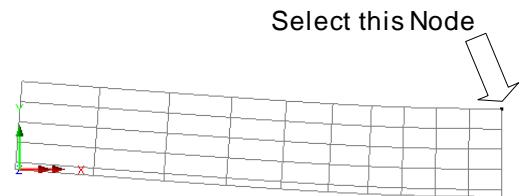
Using the Graph Wizard

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 Specify 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.

Graph1

Total load factor(2) / Resultant Displacement Node 181(1)

Note. Graphs can be modified using the right hand mouse button in the graph window and selecting the **Edit Graph Properties** option.

[X] Close the graph window.

Use the maximise button [- □ X] to increase the size of the graphics window.

15
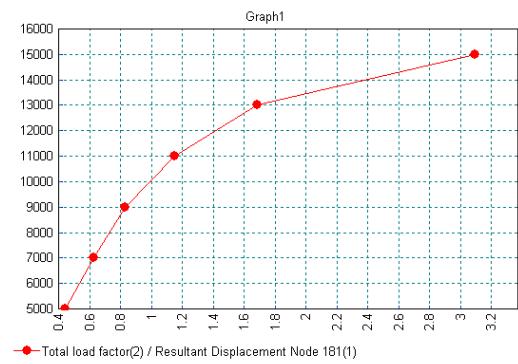
- In the Treeview turn off the **Deformed mesh** layer and turn on the Mesh layer.

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, **SMax**
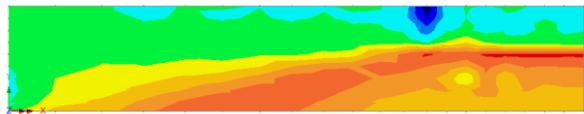
- In the **Contour Display** tab ensure **Deform** is <u>not</u> selected.

- 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.
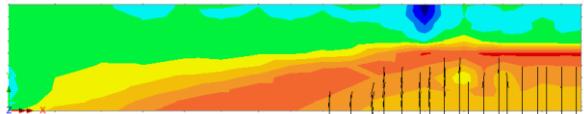
Animating the Results

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.
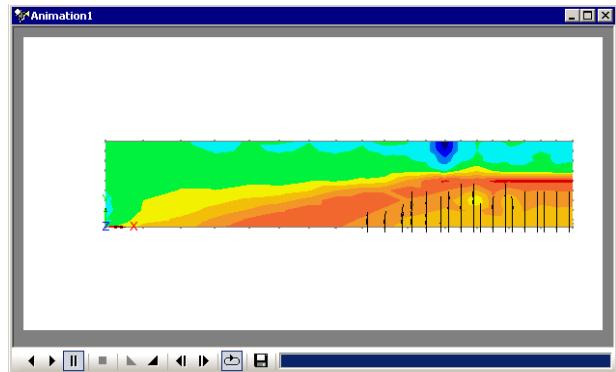
Saving Animations

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**.

X Close the animation window, choosing not to save changes.

_□X 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.
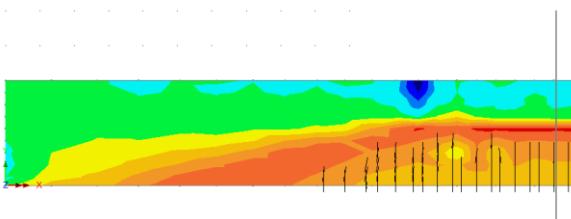
- Select **Stress - Plane Stress** results for Stress **SX** and click the **Next** button.

- Leave all title information blank.

- Click the **Finish** button to create the graph of stress through the section of beam.
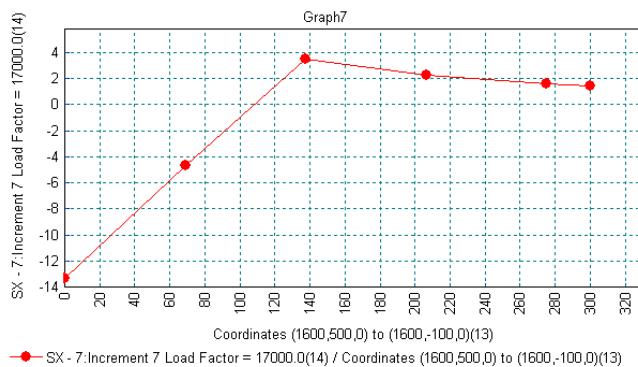
Adding Additional Results to a Graph

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.

- In the 'Include existing graph' panel select the check box for **Graph 2**.

- Click the **Finish** button to see the results for the second slice section alongside the results for Graph 2.

Window
2 Graph2:Graph 3

- Re-select the existing graph to see the overlaid data
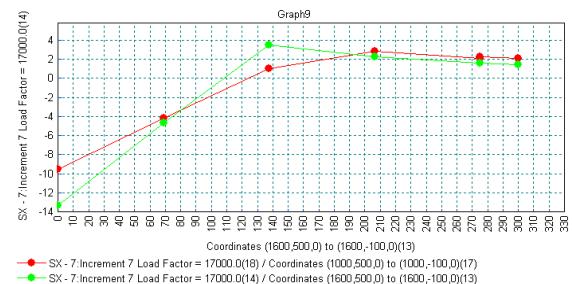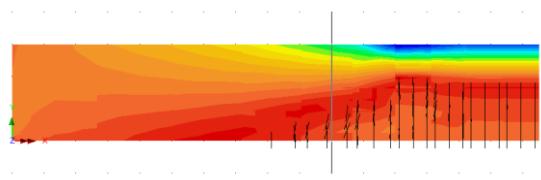
 Close the graph window.

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 <u>deselect</u> **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, ensure the following are selected: **Plastic strain – Plane Stress** of Component **CWMax** at Location **Gauss Point**.

- Select the **Values Display** tab and ensure the **Symbols** and **Values** are checked, and change the **Font angle** to **45** degrees.
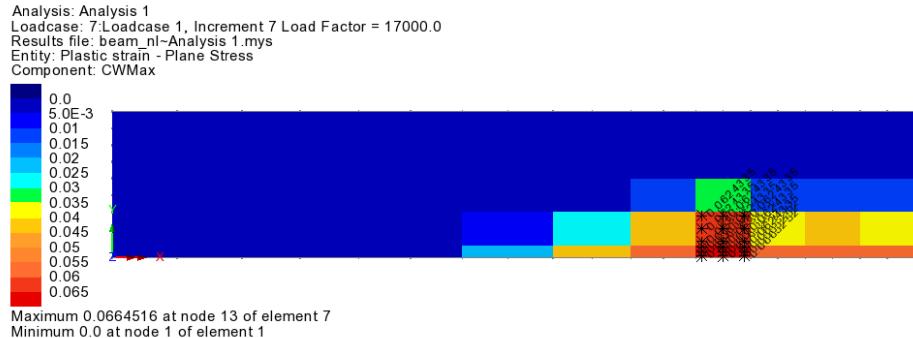
- Click **OK** to update the display and show the actual crack width value calculated at gauss points within the element.

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 in place of the nonlinear steel material 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. 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, such as this example. Typical settings are shown on the adjacent image.
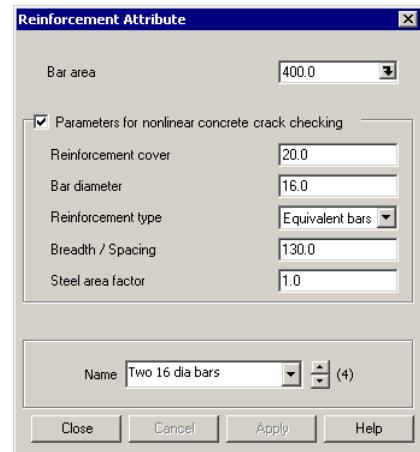
When linear steel material and reinforcement bar attributes have been defined to the lines representing the reinforcement, and the model solved, contours and values of crack widths can be plotted in accordance with EN1992-1-1:2004 Eurocode 2 by selecting the Entity **Crack Widths EN 1992-1-1** and the Component **Maximum Crack Width** on the Contour Properties dialog.

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 1992-1-1:2004 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.

Analysis: Analysis 1
Loadcase: 7:Loadcase 1, Increment 7 Load Factor = 17000.0
Results file: beam_nl_with_16mm_bar_attributes_linear_steel~Analysis 1.mys
Entity: Crack widths EN1992-1-1
Component: Maximum Crack Width (Units: mm)

Maximum 0.0991234 at Gauss point 2 of element 17
Minimum 0.0 at Gauss point 1 of element 11