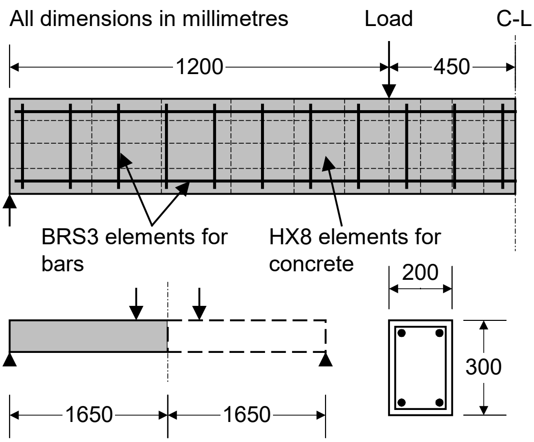
3D Nonlinear Analysis of a Concrete Beam

For LUSAS version:	23.0
For software product:	Any Plus version. Bridge Plus version for the modelling described in the discussion at the end.
With product options:	Nonlinear.

Description

A nonlinear plane stress analysis is to be carried out on a 3D model of a reinforced concrete beam.

Two, 16mm diameter steel bars of reinforcement are provided in the top and lower faces of the beam, with 8mm diameter links provided throughout. 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.



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 of 5000kN is applied to the top of the beam 1200mm from the left-hand end. The concrete section is represented by plane stress (HX8) elements, and the reinforcement bars are represented by parasitic bar (BRS3) 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 steel reinforcement bars.

Note. This example shows the use of lines meshed with parasitic bar elements to model reinforcement. These lines can be defined independently of any other lines, surfaces or volumes that may be used to model concrete in 2D or 3D models.

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.
- **Stress contour plot** showing the stresses in the reinforcement bars.

Keywords

3D, Inplane, Plane Stress, Parasitic Bar Elements, Nonlinear Concrete Model, Element Selection, Concrete Cracking, Steel Reinforcement, Groups, Crack Patterns, Animation, Graphing, Load Displacement Curve

Associated Files

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

- **3d_nl_beam_modelling.lvb** 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 **3d_nl_beam**
- Use the default User-defined working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **3D**
- **Important:** Set Model units of **N,mm,t,s,C**
- Leave the Timescale units as **Seconds**
- Select a Startup template of **3D Beam/Shell**.
- Ensure the Layout grid is set as **None**
- Enter a Title of **3D Nonlinear Concrete Beam**
- Click the **OK** button.

Note. Use the Undo button to correct any mistakes made since the last save was done.

Set the vertical axis

By default, 3D models in LUSAS are set-up with a vertical Z-axis. But this example requires a vertical Y-axis.

Tools
Vertical Axis…

- On the Model Properties dialog press the **Y-axis** radio button.

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.

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

Geometry
Surface >
By Sweeping...

Enter a translation value of **300** in the **Y** direction to create the Surfaces which represent the side of the beam.

- Click the **OK** button.

- Select the two surfaces just drawn

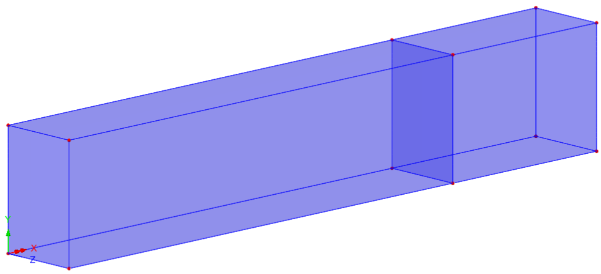
Geometry
Volume >
By Sweeping...

Enter a translation value of **200** in the **Z** direction to create the Volumes which represent the concrete beam.

- Click the **OK** button.

The model should appear as shown.

Rotate the model to see a similar view.

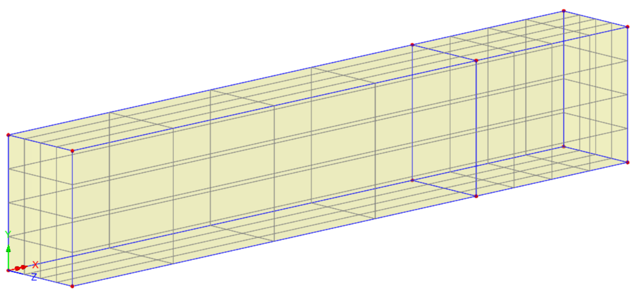


Defining and assigning the mesh for the concrete

The concrete will be modelled using a volume mesh, using line mesh divisions to control the mesh density.

Attributes
Mesh >
Volume…

- Nonlinear concrete is best modelled by HX8 elements. Select **Element name** and enter **HX8**.
- Enter the attribute name as **Stress linear**
- Click the **OK** button to add the attribute to the Treeview.
- Select the whole model and drag and drop the **Stress linear** mesh attribute onto the selection.

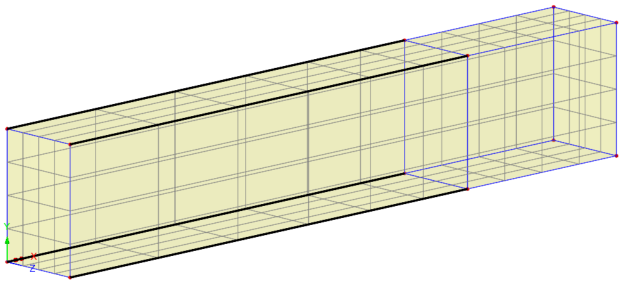


The default mesh density of 4 divisions per line is sufficient for the volumes to the right of where the load will be applied. A graded line mesh will be created for use on the volumes to the left of where the load will be applied.

Defining a graded mesh

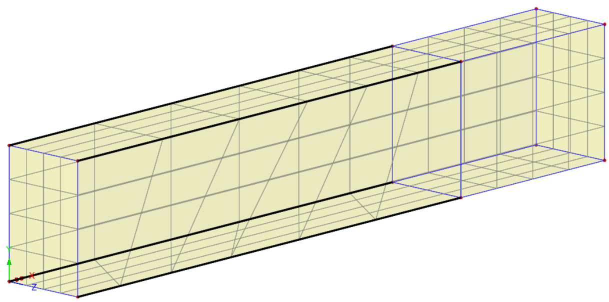
Attributes
Mesh >
Line…

- For an element type of None, 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 **Null, div = 6 graded**
- Click the **OK** button to finish to add the attribute to the Treeview.
- Select the 4 lines on the left-hand side of the model as shown and drag and drop the line mesh attribute Null, div = 6 graded from the Treeview onto the selection.



The mesh arrangement will be updated initially as shown right.

This shows that the line direction of the lower line in the 'front' of the model will need to be reversed in order to obtain the desired mesh arrangement.

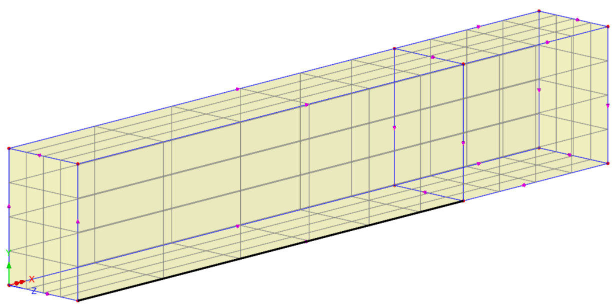


To do this:

- Select only the lower line at the 'front' of the model.

Geometry
Line >
Reverse Direction…

The line direction will be reversed, and the mesh arrangement updated.



Note. Line directions, and surface and volume axes can be viewed by visiting the Treeview and double-clicking on the **Geometry** entry and making appropriate selections.

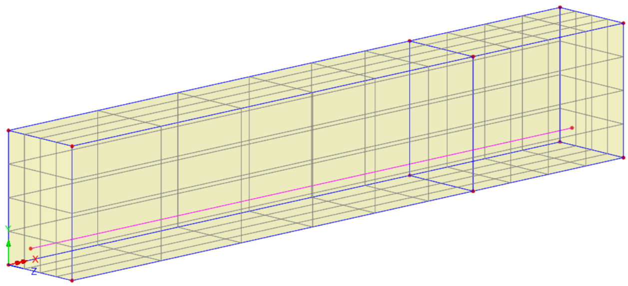
Defining the geometry for the reinforcement bars

Bottom reinforcement bars:

These will have 30mm cover at the left-hand end of the beam and extend to the mid-point of the model on the right-hand side. Note that the position of the centreline of the bar is defined.

Geometry
Line >
By Coords...

Enter coordinates of (**30, 38, 38**) and (**1650, 38, 38**) to define the centreline of a single straight reinforcement bar in the bottom of the beam.

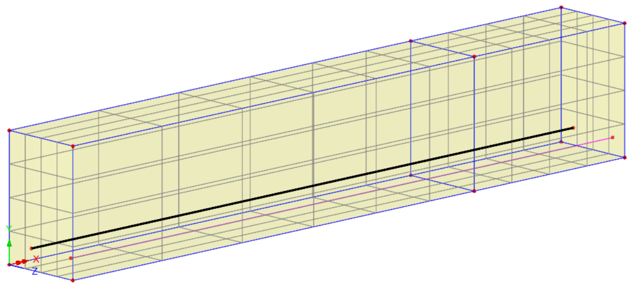


- Select the line just drawn

Geometry
Line >
Copy…

Enter a Z value of **124** and click **OK**.

The other bottom bar will be created.



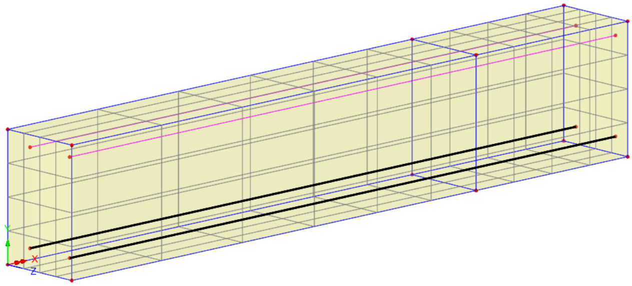
Top reinforcement bars:

- Select both lines representing the bottom bars of reinforcement

Geometry
Line >
Copy

Enter a Y value of **224** and click **OK**.

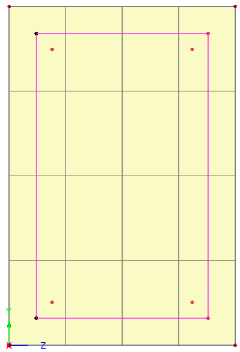
The top bars will be created



Defining links

Links are required to start 75mm from the left-hand end of the beam, then spaced at 150mm centres. One link will be defined initially, before being copied along the beam.

- Press the **Shift** key + X: N/A to see the left-hand end view of the beam. (If the Shift key is not pressed, the other end of the beam would be incorrectly viewed instead.)



Geometry > Line > By Coords...

Enter x, y, z coordinates of (**75, 24, 24**), (**75, 24, 176, 24**), (**75, 276, 176**) and (**75, 276, 24**) to define corner points that will be used to define the links.

- Select the two points on the left-hand side that are yet to define a line.

Geometry > Line > By Points...

This draws a line connecting the two selected points.

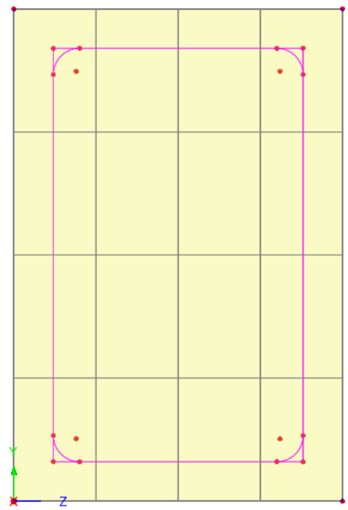
Note. Depending upon the level of detail and accuracy of results, the link could be left without radiused corners, but in this example, radii will be added to represent the true shape of the link.

To add radii to the corners of the link:

- Select two lines of the link that meet at a common point.

Geometry | Line > | Arc/Circle > Tangent to Lines…

- Enter a tangent radius of **16**, accept all other defaults, and click **OK**.



The unwanted lines that are left behind will be deleted after a radius has been added to the other three corners.

- Repeat the above operation for the other 3 corners of the link to see the image shown right.

To remove unwanted lines:

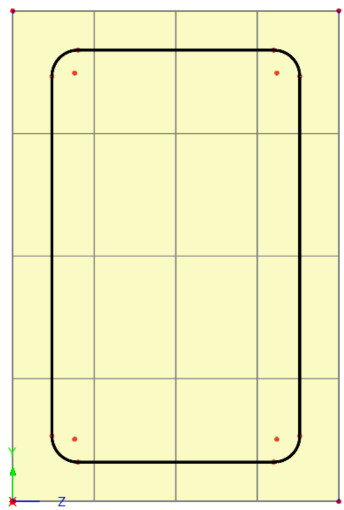
- Select the 8 unwanted remaining lines that are present in each corner and press the **Delete** key. Click **Yes All** to confirm.

To copy the link defined:

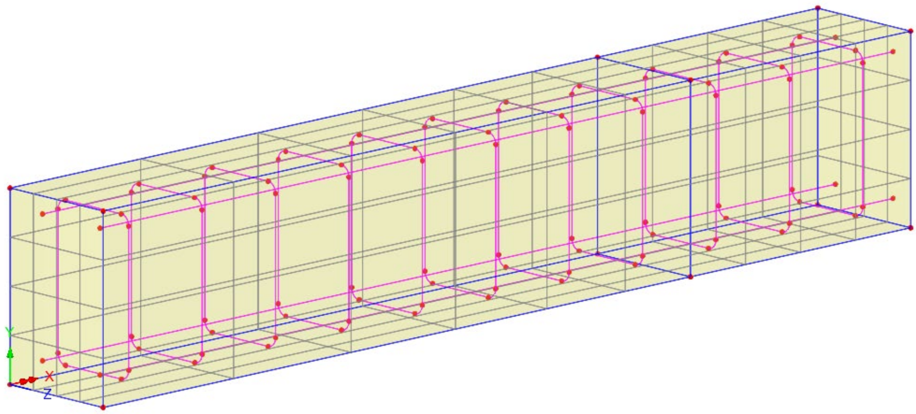
- Select the 8 remaining lines defining the link

Geometry | Line > | Copy…

 Ensure **Translate** is selected, enter an X value of **150** with the number of copies set to be **10** and click **OK**.



Rotate the model to see the generated links.

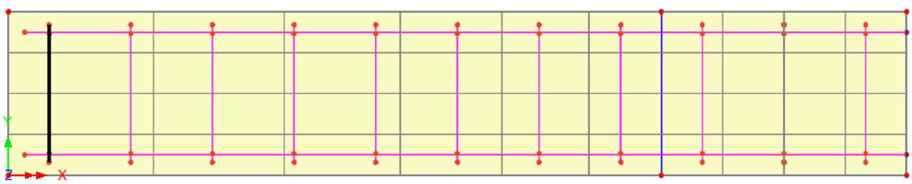


Defining Groups

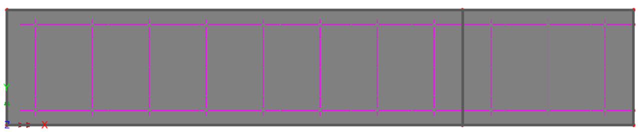
To simplify the assignment of model attributes certain model features will be grouped together to allow selection by name in the  Treeview as opposed to selection by cursor in the view window. This is easiest done by viewing a side view of the model.

- Press the Z-axis 

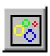
button to see a side view of the beam



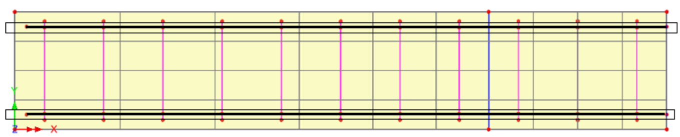
- First, select only the two volumes representing the concrete.



Geometry
Group >
New Group

 Enter **Concrete** for the group name and click **OK**.

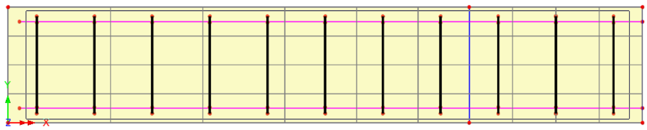
- Now box-select only the two sets of two lines that represent the top and bottom bar reinforcement. Take care with the box-selections to not to include the lines representing the transverse legs of links. Use the **Shift** key to add to the initial selection.



Geometry
Group >
New Group

Enter **Main reinforcement** for the group name and click **OK**.

- Change the selection cursor to select only Lines



- Now box-select only the lines representing the links. See preceding image.

Geometry
Group >
New Group

Enter **Links** for the group name and click **OK**.

Change the selection cursor back to selecting all items.

Note. Model attributes will now be defined but they will not be assigned to the model straight away. They will be assigned to the model later by making use of the Groups facility.

Defining the bar reinforcement mesh

The reinforcement bars will be modelled using a parasitic bar elements. These sit wholly within a host element. The line mesh spacing along the line to which the attribute is assigned is determined by the intersection of the line and the faces of the host elements through which the line or lines pass. The numbers of elements along each line may be adjusted if required.

Attributes
Mesh >
Line…

- Set element type to **Parasitic bar** and Interpolation order to **Quadratic.**
- Press the **Spacing** button and set the **Minimum number** of element divisions to be **2**. Leave other settings as their defaults and click **OK.**
- Enter the attribute name as **Bar, quadratic** and click **OK** to add the attribute to the Treeview.

Defining the top and bottom reinforcement bar size

Attributes
Geometric >
Line…

- Select **From library / calculator** and then from the adjacent droplists select **Parametric Sections** then **Circular Sections** then **New…**
- Enter a diameter, D, of **16** and click **OK** to accept the default utility name
- Back on the Geometric line dialog enter the attribute name as **16 dia bar** and click the **OK** button to add the attribute to the Treeview.

Defining the link bar size

Attributes
Geometric >
Line…

- Select **From library / calculator** and then from the adjacent droplists select **Parametric Sections** then **Circular Sections** then **New…**
- Enter a diameter, D, of **8** and click **OK** to accept the default utility name
- Back on the Geometric line dialog enter the attribute name as **8 dia bar** and click the **OK** button to add the attribute to the Treeview.

Defining the material properties

Steel

Nonlinear steel properties will be defined for the bar elements representing the reinforcement.

Attributes
Material >
Isotropic…

- Enter Young's modulus as **210e3** and Poisson's ratio as **0.3** and leave the mass density field blank.
- Select the **Plastic** check box, and then select the **Plastic** tab 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.

Concrete

Nonlinear concrete material properties will be defined for the volume elements representing the concrete.

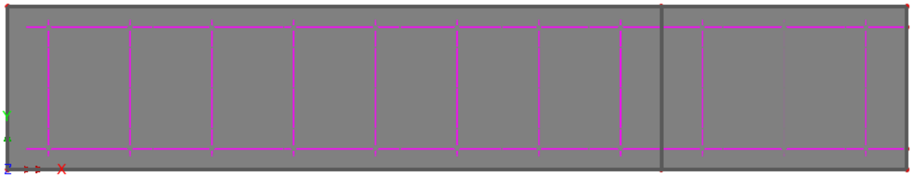
Attributes
Material >
Concrete (Nonlinear)…

- Ensure the concrete model **Smoothed multi-crack (model 109)** is selected.
- Ensure that Creep model code **None** is selected.
- Ensure that Shrinkage model code **None** is selected.
- Select the **Strain at end of softening curve** radio button to add a row containing a default value to the grid and accept all the remaining default values.
- Enter the attribute name as **Nonlinear Concrete**
- Click the **OK** button to add the attribute to the Treeview.

Assigning concrete material to the model

The material and geometric attributes and parasitic line mesh 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 **Concrete**. Select the **Select Members** option. Click **Yes** if necessary to de-select any previously selected items.
- Drag and drop the material attribute **Nonlinear Concrete** from the Treeview onto the selected features. Ensure the **Assign to volumes** option is selected for Analysis 1 and click **OK**

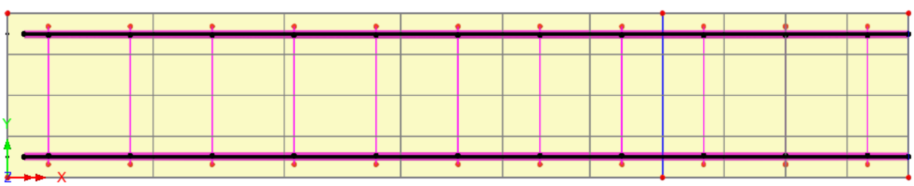


Assigning attributes to the main reinforcement

- In the Treeview right-click the group name **Main reinforcement**. Select the **Select Members** option. The line features in the group will be highlighted.
- Drag and drop the mesh attribute **Bar, quadratic** from the Treeview onto the selected features. Click **OK**.

Note. There will be a short pause whilst the line mesh divisions for the bars are calculated for each intersection with the concrete mesh

- Drag and drop the geometric attribute **16 dia bar** from the Treeview onto the selected features. Click **OK**.
- Drag and drop the material attribute **Nonlinear Steel** from the Treeview onto the selected features. Click **OK**.



Now that the parasitic line mesh and bar geometry have been assigned, the fleshed size of the reinforcement can be seen.

Assigning attributes to the link reinforcement

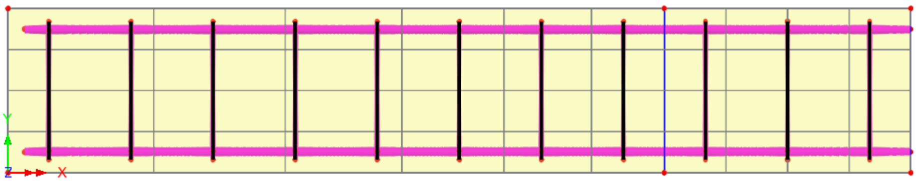
- In the Treeview right-click the group name **Links**. Select the **Select Members** option. The line features in the group will be highlighted.

- Drag and drop the mesh attribute **Bar, quadratic** from the Treeview onto the selected features. Click **OK**.

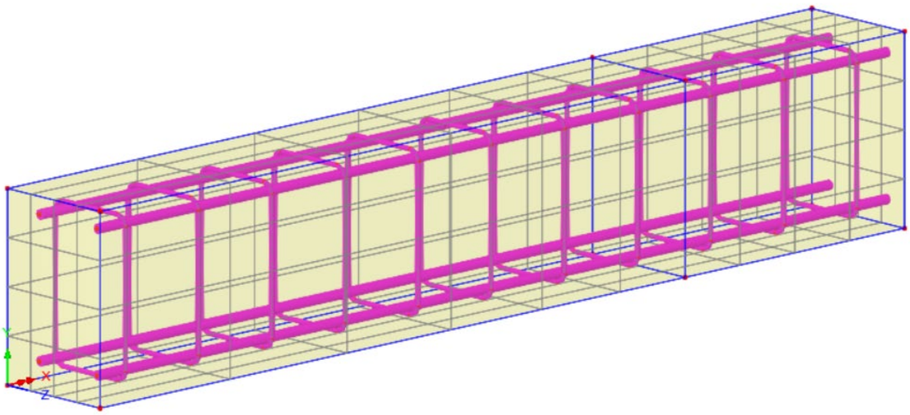
Note. There will be a short pause whilst the line mesh divisions for the links are calculated for each intersection with the concrete mesh

- Drag and drop the geometric attribute **8 dia bar** from the Treeview onto the selected features. Click **OK**.

- Drag and drop the material attribute **Nonlinear Steel** from the Treeview onto the selected features. Click **OK**.



When the model is rotated the reinforcement cage can be seen.



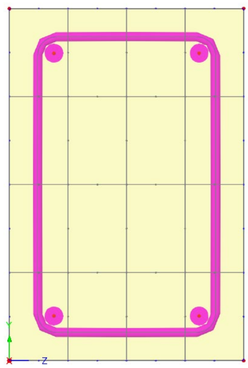
Notes about the parasitic line mesh assigned

The intersection of the parasitic bar meshed lines and the host volume element mesh determines fixed nodal locations on the parasitic line that cannot be modified, however additional elements may be added by using the line mesh spacing options available.

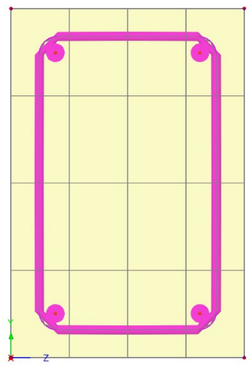
- Press the **Shift** key +  to see the left-hand end view of the beam.

By setting the parasitic spacing parameter earlier to use a minimum number of 2 quadratic elements the link shape will match the left-hand image that follows.

Otherwise, by default, a single quadratic element would be used to represent each radiused corner of the link. When fleshed, these would appear as shown on the right-hand image that follows.



With parametric spacing set to a minimum number of 2



With default values for parametric spacing

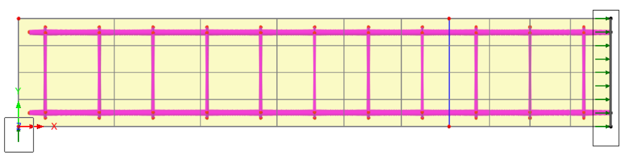
Note. Increasing the number of parasitic line mesh elements will noticeably increase the time required to mesh any changes to the model.

Supports

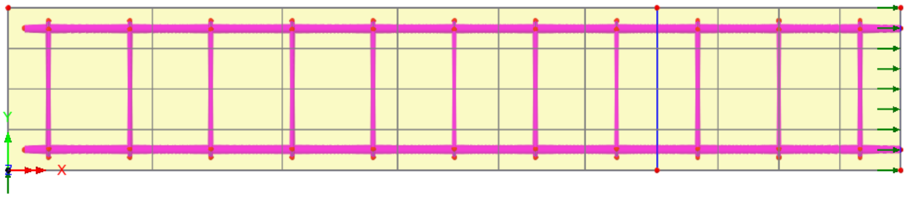
The beam is to be simply supported in the Y direction at its left-hand end and supported by a horizontal restraint in the X direction at its right-hand side to satisfy the symmetry requirements at mid-span.

- Press  to view the model from the side as shown below.

- Box-select the lowest point at the left hand end of the model as shown. (This will ensure that the line to which the support will be assigned is also selected.)



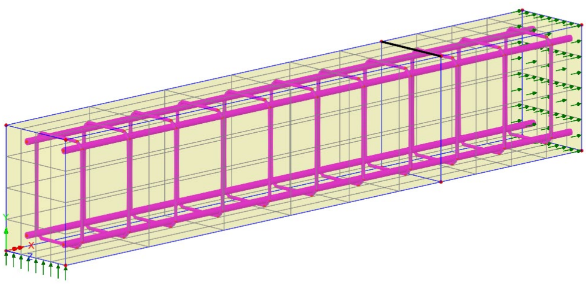
- Drag and drop the support attribute **Fixed in Y** from the Treeview onto the selection. Ensure the **Assign to lines** and **All analysis loadcases** options are selected and click **OK**
- Box-select around the line at the right-hand end of the model (as shown on the previous image). This will ensure that the surface to which the support will be assigned is also selected.
- Drag and drop the support attribute **Fixed in X** from the Treeview onto the selection. Ensure the **Assign to surfaces** and **All analysis loadcases** options are selected and click **OK**



Loading

A load is to be applied to the line 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.

- Rotate the model to a similar view as shown.
- Select the line on the top of the beam as shown.

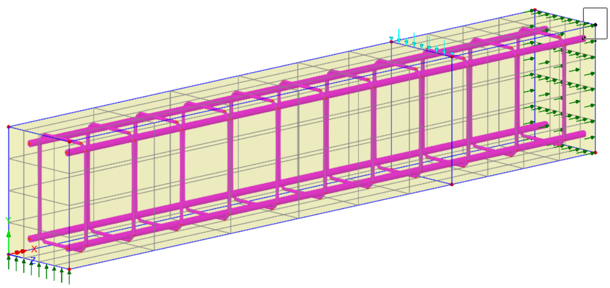


Attributes
Loading...

- Select a load type of **Distributed loads** of type **Global distributed** and click **Next**
- Enter a **Total** load of **-5000** for the **Y** direction
- Enter a name of **Distributed total load** and click **OK**.
- Drag and drop the loading dataset **Distributed total load** from the Treeview onto the selected line and click **OK** to assign the load to **Analysis 1**, **Loadcase 1**.

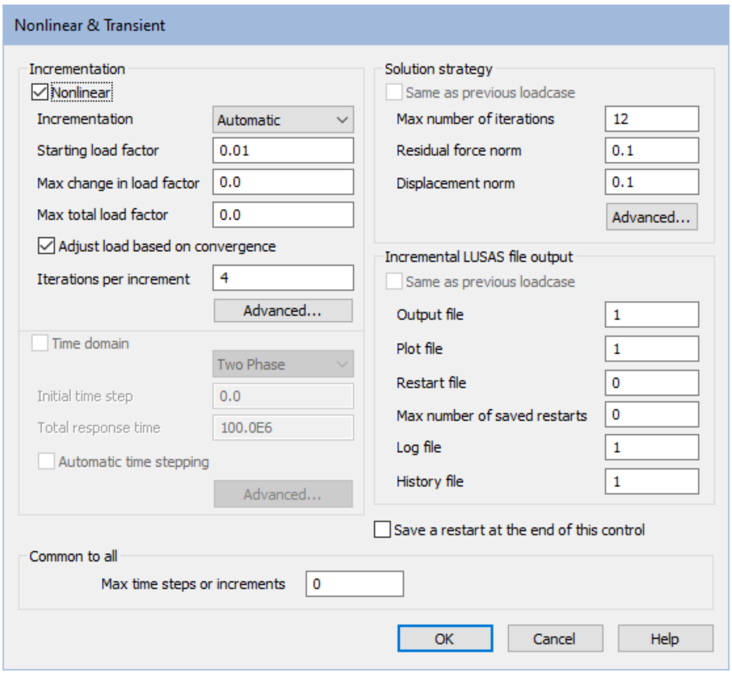
Nonlinear Control

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 (which represents the mid-point of the actual beam).
- In the Treeview ensure the **Analysis 1** entry is expanded, then right-click on **Loadcase 1** and from the **Controls** menu select **Nonlinear & Transient**

The Nonlinear & Transient dialog will appear:

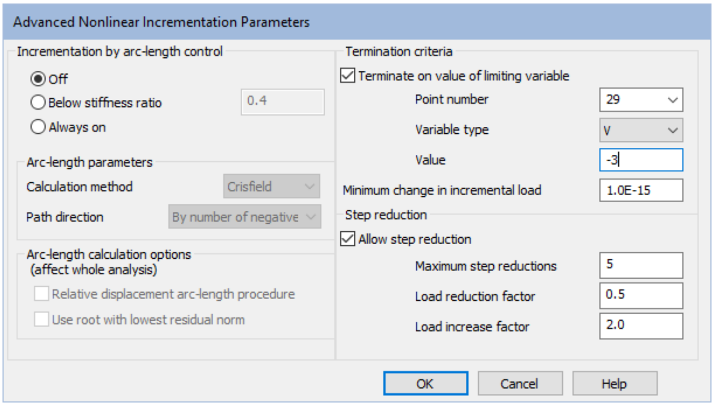


- Select the **Nonlinear** option and set Incrementation to **Automatic**
- The load to be applied to the model is the concentrated load previously defined and assigned to the model multiplied by the starting load factor. Set the **Starting load factor** to **0.01** to apply a minute amount of load initially.

- Ensure the **Max change in load factor** is set as **0** to signify that no limit and let Solver uses load factors according to how the solution is progressing.
- Change the **Max total load factor** to **0** because the solution is to be terminated once the limiting displacement at mid span has been reached for a particular loading increment.
- Ensure the number of **Iterations per increment** is set as **4**

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). If the number of iterations is less than the desired number, the next load increment will be reduced.

- Leave the other settings as shown on the previous dialog image. The residual force norm value of 0.1 means that the 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.
- In the Incrementation section of the dialog, select the **Advanced** button.



- Ensure that Incrementation by arc-length control is **Off**.
- For termination criteria select the **Terminate on value of limiting variable** option. The point number of the previously selected point will appear in the Point number drop down list. (This may differ from the number shown above depending on how the model was created.)
- Set the Variable type to **V** to monitor the deflection at the selected point in the Y direction.
- Enter a value of **-3** (minus three) so the analysis is terminated when the loading increment that causes this central deflection exceeds this value.

- In the Step reduction section ensure that 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:

- **3d_nl_beam.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.
- **3d_nl_beam.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. You may download a zip file containing this file from the user area of the LUSAS website.

3d_nl_beam_modelling.lvb 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 **3d_nl_beam**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **3D** is set.
- Click the **OK** button.

Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script...

To recreate the model, select the file **3d_nl_beam_modelling.lvb** that was downloaded and placed in a folder of your choosing.

Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the Treeview.

Changing the Active Results Loadcase

- In the Treeview the last load increment **Increment 10 Load Factor = 5.3500** should be already **Set Active**.

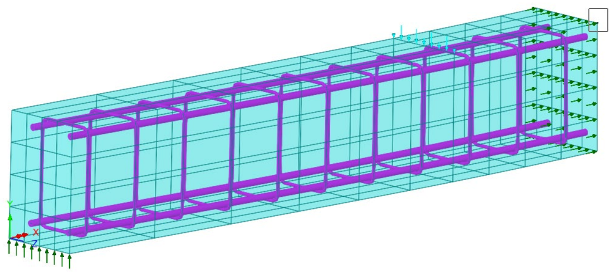
Deformed Shape

- In the Treeview turn off the **Geometry** and **Mesh** layers by right clicking on each entry and selecting the **On/Off** option.

To see the supports and loading arrows in the correct location on a deformed mesh plot:

- In the Treeview double-click on the **Attributes** entry, click the **Supports** tab, and select **From results file**. Click the **Loading** tab and click **From results file**. Click **OK**.

Creating a Load versus Displacement Graph



A graph of displacement at mid-span is to be plotted against the applied load.

- Ensure the node on the line of symmetry is still selected:
- With the **Deformed mesh** layer visible, select the top edge 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.

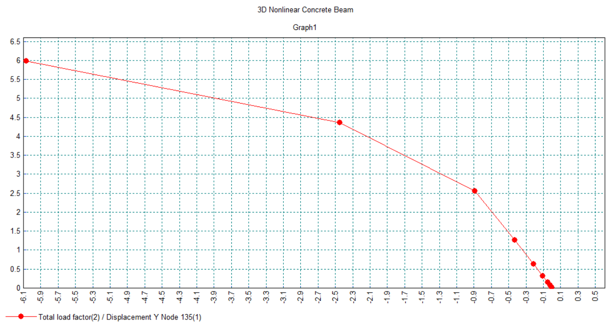
Utilities
Graph Wizard...

- 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 **DY**
- 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.



From the graph and its adjacent data table it can be seen that for the final load increment that needed to be considered, a displacement of -3mm the corresponding load factor is around 4.25

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.

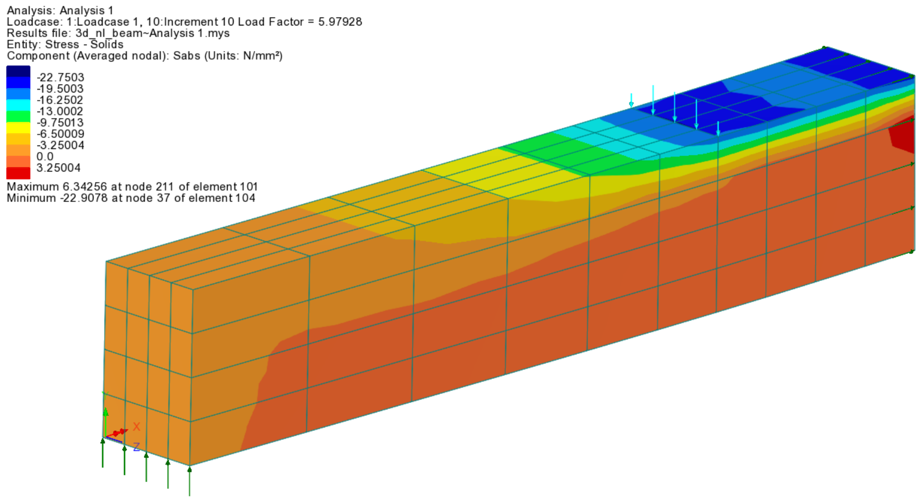
- In the Treeview turn off the **Deformed mesh** layer and turn on the **Mesh** layer.

Maximum Principal Stress Contour Plots

- With no features selected, click the right-hand mouse button in an empty part of the view window and select the **Contours** option to add the contours layer to the Treeview.

The properties dialog will be displayed.

- Select entity **Stress - Solids,** Component of stress in the x direction, **Sabs**, and **Averaged nodal** contours.
- Click the **OK** button to display contours of stresses for the final load increment.

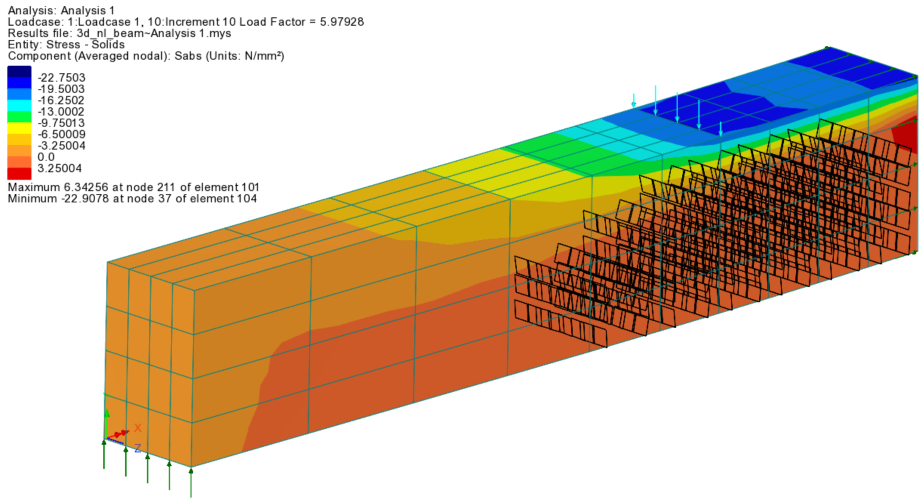


Viewing Crack Patterns

- With no features selected, click the right-hand mouse button in an empty part of the view window and select the **Values** option to add the values layer to the Treeview.

The properties dialog will be displayed.

- Select **Stress - Solids** contour results of type **Crack/Crush**
- Click the **OK** button to display the cracking pattern (no crushing is present) 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, and the maximum and minimum values for the range of contours is to be specified.

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

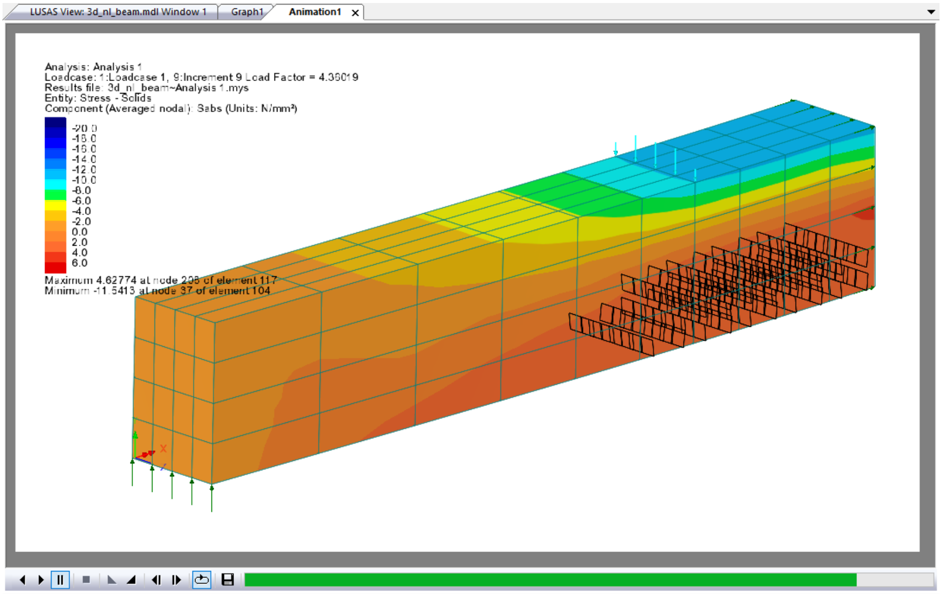
The contour properties will be displayed.

- Select the **Appearance** tab and for the **Classic** contour range press the **Set** button
- Click the **Interval** option and set the contour interval as **2**
- Click the **Maximum** button and set the maximum value as **6**
- Click the **Minimum** button and set the minimum value as **-20**
- Click the **OK** button to return to the parent dialog, and **OK** to redisplay the stress contours using the new contour range.

Using the Animation Wizard

Tools
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.

Note. To see the animation window in isolation use the **Window** > **Layout** > **Multiple documents** menu option.

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 **3d_nl_beam** for the animation file name. An **.avi** file extension is automatically appended to the file name. Click **OK**.

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.

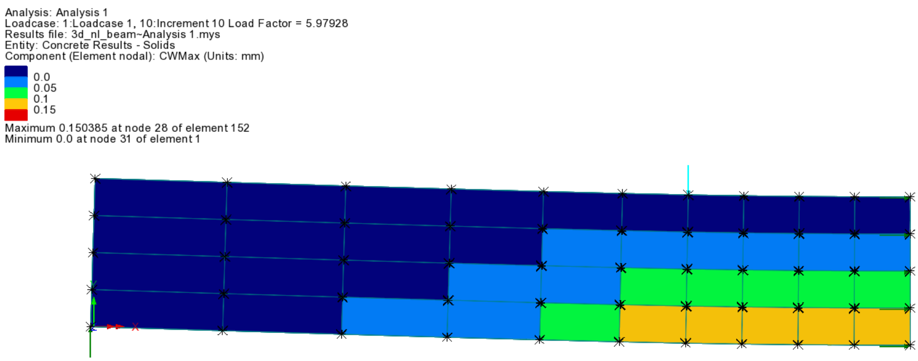
Plotting Crack Width Contours

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 **Concrete Results - Solids** for component of stress in the x direction, **CWMax**, and of **Unaveraged nodal** results.
- Select the **Appearance** tab and for the **Classic** contour range press the **Set** button
- Deselect the **Maximum** button.
- Deselect the **Minimum** button.
- Set the contour **Interval** to be **0.05**
- Ensure the Value to pass through is set to be **0**
- Click the **OK** button on both dialogs to redisplay the stress contours using the new contour range.
- Press **Yes** to confirm the Values layer will be adjusted to match.
- Press the Z-axis button Z: N/A at the bottom of the view window to view the model from along the z-axis.

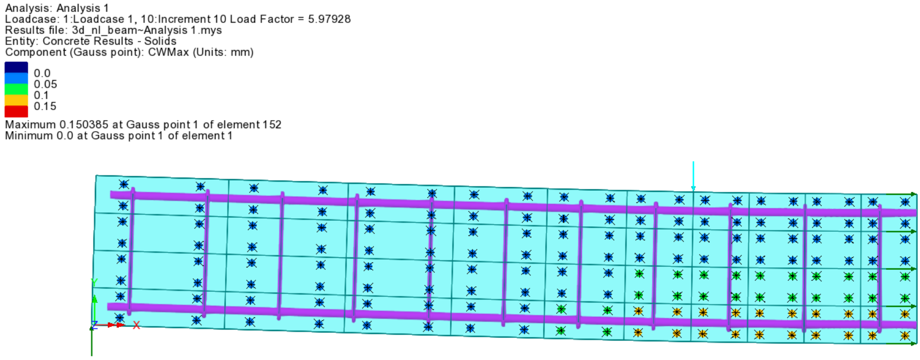


Note. Contouring of unaveraged nodal results effectively shows the elements affected by cracking and identifies those elements where the maximum crack widths occur.

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.

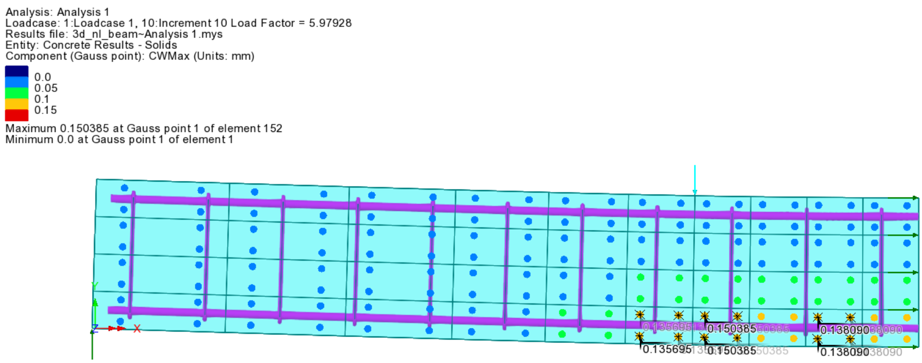
- Double-click the **Contours** layer name in the Treeview and on the dialog select entity **Concrete Results - Solids** for component of stress in the x direction, **CWMax**, and of **Gauss point** results. And click **OK**.

- Press **Yes** to confirm the Values layer will be adjusted to match.



Plotting crack width values

- Double click the **Values** layer name in the Treeview.
- Select the **Values Display** tab, ensure that both the **Symbols** and **Values** options are checked, ensure only **Maxima** is selected, to show the top **10%** of results. Leave the font angle set to 0 degrees.
- Click **OK** to update the display and show the actual crack width values calculated at gauss points within the element.

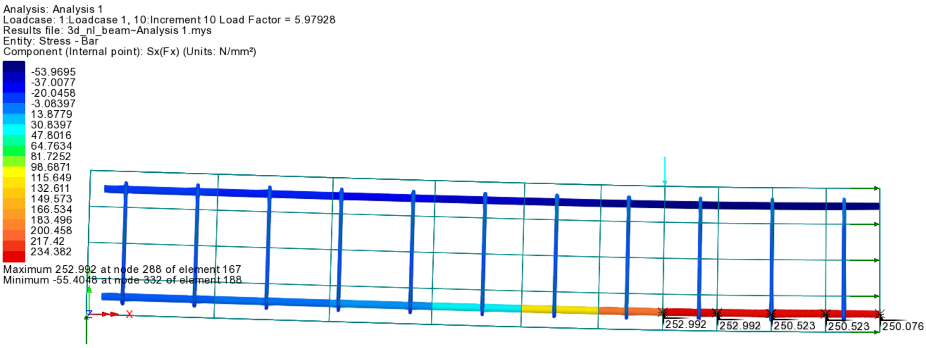


- Zoom-in if you wish to see more detail.

Plotting stress in the reinforcement

- Double click the **Contours** layer name in the Treeview.
- Select entity **Stress – Bar** and component **Sx(Fx)**

- Click **Yes** to update the Values layer to match.
-  Ensure that fleshing is turned on to see the contours drawn on the true size of the reinforcement.



This completes the example.