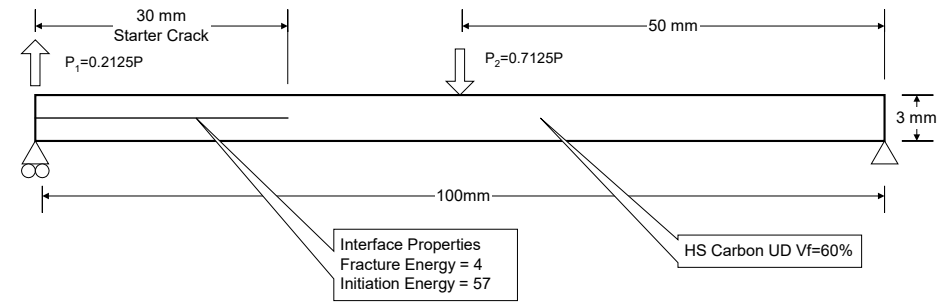


Mixed-Mode Delamination

For software version:	22.0
For software product(s):	Any Plus version.
With product option(s):	Nonlinear.
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example demonstrates the use of delamination elements in a 2D analysis. Both 2D and 3D delamination elements are available within LUSAS. These elements have the ability to model all modes of crack growth under mixed-mode conditions - mode I (open) mode II (shear) and mode III (tear 3D only).



The model is required to predict the growth of a crack within a unidirectional composite laminate subjected to mixed-mode loading. A starter crack 30 mm long is introduced at one end of a double cantilever beam (DCB) specimen. A tensile (crack opening) load is applied to the cracked end of the sample while a compressive (crack closing) load is applied to the centre of the specimen. The resulting delamination crack propagates along the length of the sample under mixed-mode conditions.

The DCB specimen is constrained at either end and prescribed displacements are applied to the centre of the span in a negative Y direction and to the cracked arm, in a

positive Y direction. No axes of symmetry other than the plane strain assumption may be made. The geometry of the strip and support positions are as shown.

Units of N, mm, t, s, C are used throughout.

Objectives

This analysis will:

- ☐ Predict the initiation of delamination
- ☐ Predict the propagation of a delamination crack with increasing displacement
- ☐ Produce an animated Deformed Mesh Plot showing the growth of delamination
- ☐ Produce a Contour Plot showing the redistribution of direct stress caused by delamination on the cracked Surface.
- ☐ Produce an animated Yield Plot showing the behaviour of the interface during incrementation

Keywords

2D Inplane, Plane Strain, Delamination, Composite, Shell, Nonlinear, Yield Symbol, Contour Plot, Animation

Associated Files

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



- ☐ **delamination_modelling.lvb** carries out the complete modelling of the example.

Modelling : Delamination Model

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 **delamination**
- 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**
- Ensure Timescale units are **Seconds**
- Select a Startup template of **2D Plane Strain**
- Ensure the Layout grid is set as **None**
- Enter a Title of **Delamination Model**
- Click the **OK** button.

The Attributes  treeview will be populated with potentially useful basic line mesh, geometric and support entries appropriate to the settings made on the new model dialog.



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

File
Model Properties...

- Select the **Meshing** tab and set the default number of divisions to **2**
- Select the **Options** tab, select **Element Options** and select the option for **Fine integration for stiffness and mass**
- Click the **OK** button to set the element options and the **OK** button to set the model properties.

Defining the Geometry

Geometry
Point >
By Coords...



Enter coordinates of **(0,0)**, **(30,0)**, **(50,0)** and **(100,0)** and click **OK** to define the Points.

- Use the **Ctrl** and **A** keys together to select the Points just defined.

Geometry
Line >
By Points...



This defines a series of straight Lines between the Points.

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

Geometry
Surface >
By Sweeping...



Ensure that the **Translate** option is selected and enter a value of **1.5** in the **Y** direction.



- Click the **OK** button to finish. This generates the first half of the model.

Mixed-Mode Delamination

The second half of the model will now be created by copying the existing data and specifying a suitable gap to enable the interface elements to be embedded in the model. Once the interface elements have been assigned, this gap will be closed.

- Select the whole model using the **Ctrl + A** keys.



Ensure that the **Translate** option is selected and enter a value of **20** in the **Y** direction.

Note. At this stage the two halves of the model are separated to simplify the definition of the delamination interface elements.

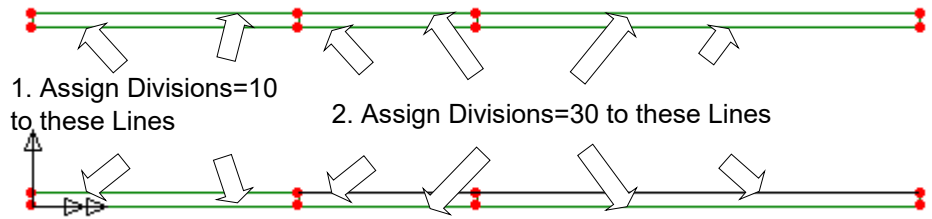
Meshing

Delamination modelling requires a fine mesh density. The mesh density is controlled using Line mesh attributes.

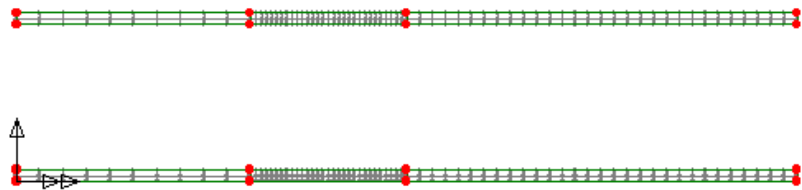
- Select **Plane strain, Quadrilateral, Quadratic** elements. Ensure the **Regular mesh** option is selected with automatic divisions so that Modeller uses the default number of mesh divisions on each line.
- Give the attribute the name **Plane Strain QPN8**
- Click the **OK** button to add the attribute to the  Treeview.
- Select the whole model using the **Ctrl + A** keys.
- Drag and drop the **Plane Strain QPN8** attribute from the  Treeview onto the selected features.

The  Treeview contains some commonly used line meshes. Null line meshes with 10 and 30 divisions need to be added to those present.

- With the Element description set to **None** define a line mesh attribute containing **10** divisions and named **Null, div = 10**
- Click the **Apply** button to add the attribute to the  Treeview.
- Edit the number of divisions to be **30** and change the attribute name to **Null, div = 30**
- Click the **OK** button to add the attribute to the  Treeview.



- Assign the **Divisions=10** attribute to the horizontal lines on the left-hand section of the model.
- Assign the **Divisions = 30** attribute to the remaining horizontal lines of the rest of the model to give a mesh arrangement as shown.

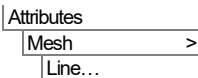


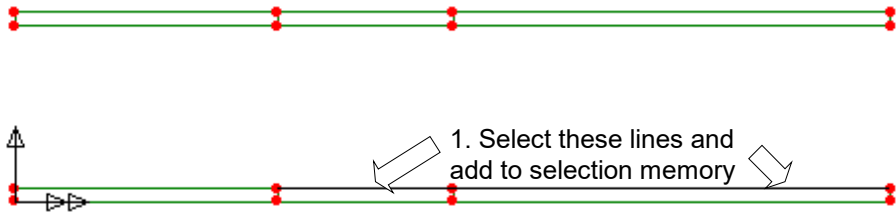
Geometric Properties

Because this model is using plane strain elements there is no need to define or assign a geometric thickness to the model. Plain strain elements assume infinite thickness.

Interface Elements

- On the Line Mesh dialog define a line mesh for an **Interface (plane strain)** element type, of **Quadratic** order for assignment **Between two features** with **30** divisions.
- Name the dataset **Interface mesh**
- Click **OK** to add the dataset to the Treeview.
- Select the two horizontal lines on the right-hand side of the upper face of the bottom set of surfaces as shown in the following diagram.





This adds these two lines to Selection Memory. These will form the secondary surfaces of the interface elements.

Edit
Selection Memory >
Set



Note. Items can also be added to selection memory by using the right-hand mouse button in the display area and selecting the **Selection Memory>Set** menu entry from the popup menu, or pressing **Ctrl + M**.

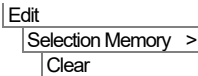
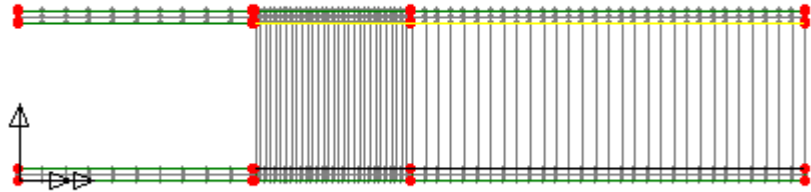


Note. The order in which the lines are selected is important. The interface elements are drawn between sequential pairs of lines in the two selections, i.e. the first set will be drawn between the first line in the selection memory and the first line in the selection.

- Click in a blank area to deselect the initial selection (note that the lines in selection memory will remain highlighted)



- Select the two horizontal lines on the right-hand side of the lower face of the top set of surfaces as shown in the previous diagram. Ensure the lines in the selection match those in selection memory as described in the note above.
- Drag and drop the **Interface mesh** attribute onto the model. Ensure **Mesh from primary to secondary** is selected. Click **OK** to finish the assignment.



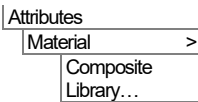
This clears the selection memory.



Note. It is only necessary to make use of Selection Memory if assignment of the interface mesh is to be carried out on more than one line in one operation. By selecting a lower primary line, holding the Shift key down, and selecting the corresponding upper secondary line the assignment could also be carried out on a line by line basis.

Composite Material

LUSAS provides a number of the more common types of composite material in the Composite Material Library. In this example the component is fabricated from a unidirectional High Strength Carbon fibre reinforced polymer matrix composite.



- Select **HS Carbon UD Vf=60%** from the drop down box.
- With the **Composite properties** option selected, choose **Plane Strain** and click the **OK** button to add the properties to the  Treeview.

The composite material is to be assigned to the surfaces of the model. Firstly check the surface axes to show the directions for the orthogonal material.

- In the  Treeview click the right-hand mouse button on the **Geometry** layer and select **Properties**
- On the dialog select the **Surface Axes** option and click the **OK** button. All surfaces should appear with the double arrow (X direction) horizontal.



Note. If required the **Surface > Reverse**, **Cycle** and **Cycle Relative** tools can be used to re-orientate surface axes as desired.

- Deselect the display of Surface axes as described above.

Now assign the material properties:

- Select the whole model using the **Ctrl + A** keys.

Mixed-Mode Delamination

- Select the **Orth3 (HS Carbon UD Vf=60% Plane Strain)** attribute from the  Treeview and drag and drop it onto the model.
- Ensure the **Assign to surfaces** option is selected for **Analysis 1** and click **OK**

Interface Material

Information concerning the fracture energies and the initiation stresses for the relevant failure modes are defined to describe the behaviour of the interface delamination model. For this example it is assumed that the interface characteristics are similar for the two modes, with mode 1 representing opening and mode 2 shear.

- Select the **Delamination** Interface option and click **Next**.
- Enter the value **4** for the fracture energy and **57** for the initiation stress for Mode 1 and Mode 2.
- Name the attribute **Interface Material** and click **Finish** to add the attribute to the  Treeview.

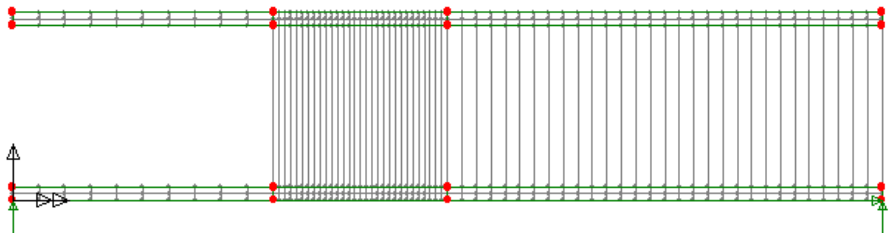


Note. The interface material attribute need only be assigned to the primary features.

- Click the right-hand mouse button on the **Interface Mesh** attribute in the  Treeview and choose the **Select Primary Assignments** option from the drop down menu.
- Drag and drop the **Interface Material** attribute from the  Treeview onto the model to assign to Lines. Click **OK** to assign the material

Supports

LUSAS provides the more common types of support by default. These can be seen in the  Treeview. The model will be supported in the Y direction at the cracked (left-hand) end and in the X and Y directions at the uncracked (right-hand) end.



- Select the point at the bottom left-hand corner of the model, drag and drop the support attribute **Fixed in Y** onto the selection and click **OK** to assign to Points.

- Select the point at the bottom right-hand corner of the model, drag and drop the support attribute **Fixed in XY** onto the selection and click **OK** to assign to Points.

The supports will be visualised as shown.

Loading

The model is subjected to two prescribed displacements. The left-hand (cracked) end of the model has a crack opening load assigned to it. A crack closing load is assigned to the mid-span.

Attributes

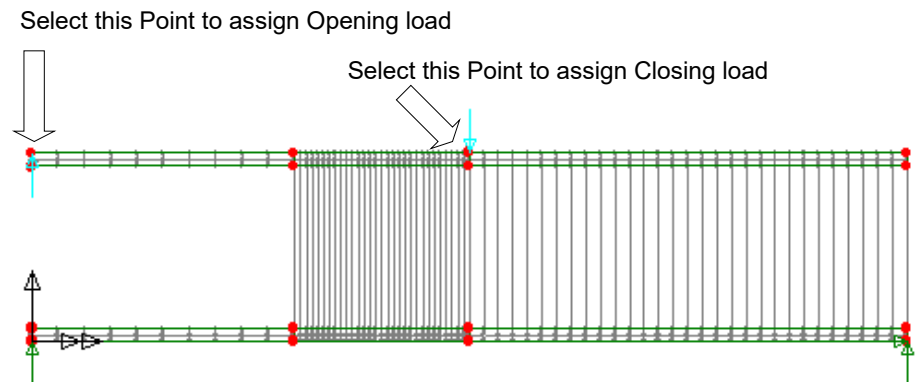
Loading...

The structural Loading attributes dialog will be displayed.

- With the **Concentrated** option selected click **Next**
- On the Concentrated dialog enter a **Concentrated load in Y Dir** of **0.2125**
- Enter the attribute name as **Opening**
- Click the **Apply** button to add the attribute to the Treeview.
- Edit the value in the **Y** direction to be **-0.7125**
- Enter the attribute name as **Closing**
- Click the **Finish** button to add the loading attribute to the Treeview.

Now assign the loads to the model.

- With reference to the following diagram, select the uppermost Point in the top left-hand corner of the model.



- Drag and drop the loading attribute **Opening** from the Treeview onto the selected Point.

- Click the **OK** button to accept the default **Loadcase 1** and assign the loading to the model.
- With reference to the previous diagram, select the uppermost Point in the middle of the top section of the model.
- Drag and drop the loading dataset **Closing** from the  Treeview onto the selected Point.
- Click the **OK** button to accept the default **Loadcase 1** and assign the loading to the model.

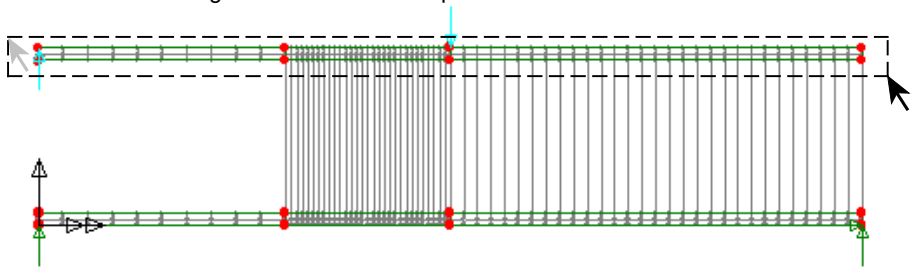
The loading will be visualised.

Final Geometry Manipulation

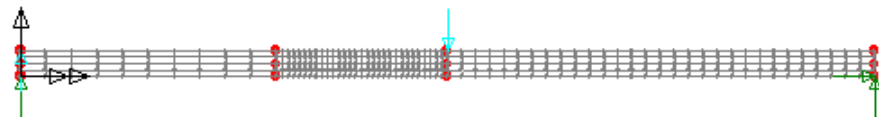
It is significantly easier to assign the appropriate mesh and the material properties of the interface region to the model if the interface surfaces are apart. Once these manipulations are complete it only remains to close the interface to complete the construction of the model.

- Select the top section of the model.

Drag a box to select the top section of the model



This makes the points of the upper section unmergeable, ensuring that the nodes either side of the embedded crack in the model are not merged together. Note that making points unmergeable will also ensure that the lines are also unmergeable.



 Move the selection to join the bottom half of the model by selecting the **Translate** option, enter a value of **-18.5** in the **Y** direction and click **OK** to confirm.

Analysis Control

Since this is a nonlinear problem the load incrementation strategy needs to be defined. The analysis is to be terminated when the vertical displacement at the left-hand tip of the specimen reaches 6mm.

- Select the point at the top-left of the model where the opening load is applied.
- From the  Treeview expand **Analysis 1**, click the right hand mouse button on **Loadcase 1**, then select **Nonlinear & Transient** from the **Controls** menu.

The Nonlinear & Transient dialog will be displayed.

- Select the **Nonlinear** option.

- Set Incrementation to **Automatic**

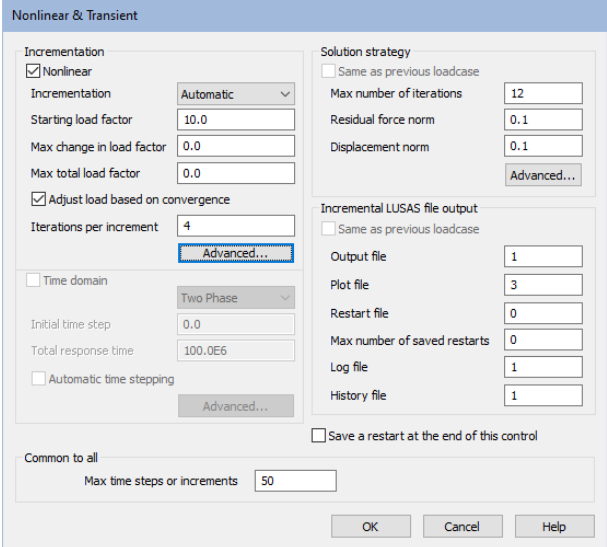
- Set the Starting load factor to **10**. This will multiply the applied loading by a factor of 10 on the first load increment.

- To enable the load to increase as required set the Max total load factor to **0**

- To prevent the analysis carrying on too long if an error has been made set the maximum number of time steps or increments to **50**

- In the Incremental LUSAS file output section set the Plot file value to **3**. This will ensure results are output to the Modeller plot file every third load increment.

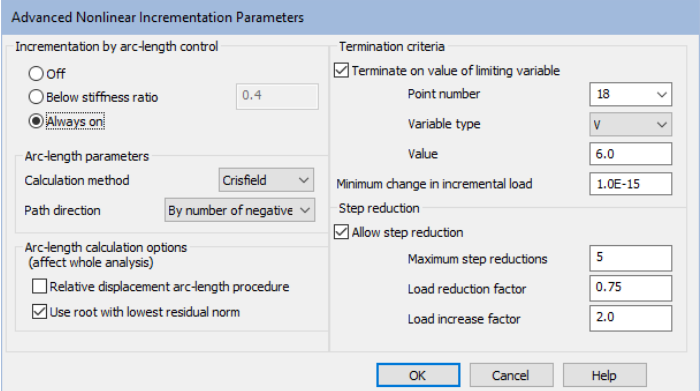
- In the Incrementation section on the top-left of the dialog select the **Advanced...** button.



The image shows the 'Nonlinear & Transient' dialog box with the following settings:

- Incrementation**
 - ☒ Nonlinear
 - Incrementation: Automatic
 - Starting load factor: 10.0
 - Max change in load factor: 0.0
 - Max total load factor: 0.0
 - ☒ Adjust load based on convergence
 - Iterations per increment: 4
 - Advanced...** (button)
- Solution strategy**
 - ☐ Same as previous loadcase
 - Max number of iterations: 12
 - Residual force norm: 0.1
 - Displacement norm: 0.1
 - Advanced...** (button)
- Incremental LUSAS file output**
 - ☐ Same as previous loadcase
 - Output file: 1
 - Plot file: 3
 - Restart file: 0
 - Max number of saved restarts: 0
 - Log file: 1
 - History file: 1
- ☐ Save a restart at the end of this control
- Common to all**
 - Max time steps or increments: 50

Buttons: OK, Cancel, Help



The dialog box is titled "Advanced Nonlinear Incrementation Parameters". It is divided into two main sections: "Incrementation by arc-length control" and "Termination criteria".

Incrementation by arc-length control:

- Three radio buttons: "Off", "Below stiffness ratio" (with a text box containing "0.4"), and "Always on" (which is selected).
- Arc-length parameters:**
 - "Calculation method": A dropdown menu with "Crisfield" selected.
 - "Path direction": A dropdown menu with "By number of negative" selected.
- Arc-length calculation options (affect whole analysis):**
 - "Relative displacement arc-length procedure": An unchecked checkbox.
 - "Use root with lowest residual norm": A checked checkbox.

Termination criteria:

- "Terminate on value of limiting variable": A checked checkbox.
- "Point number": A dropdown menu with "18" selected.
- "Variable type": A dropdown menu with "V" selected.
- "Value": A text box containing "6.0".
- "Minimum change in incremental load": A text box containing "1.0E-15".
- Step reduction:**
 - "Allow step reduction": A checked checkbox.
 - "Maximum step reductions": A text box containing "5".
 - "Load reduction factor": A text box containing "0.75".
 - "Load increase factor": A text box containing "2.0".

At the bottom are three buttons: "OK", "Cancel", and "Help".

- Ensure that incrementation by arc-length control is **Always on**
- Select the calculation option **Use root with lowest residual norm**
- In the Termination criteria section select the **Terminate on value of limiting variable** option. The point at the top left-hand corner of the model should be entered in the drop down list. Note that this may not be the same point number as shown in this dialog.
- To stop the analysis once a displacement of 6mm has been reached at that point, select the Variable type as **V** and the value as **6**
- In the Step reduction section of the form ensure the **Allow step reduction** option is selected and set the Load reduction factor to **0.75**
- Click the **OK** button to return to the control dialog.
- Click the **OK** button to exit the Nonlinear & Transient dialog.

A  Nonlinear and Transient entry will be added to the  Treeview . A separate  Nonlinear analysis options entry will also be added to the loadcase.

Saving the model

The model is now complete, and the model data must be saved.

File
Save



Save the model file.

Running the Analysis

With the model loaded:



Open the **Solve Now** dialog and press **OK** to run the analysis.



Note. In running this nonlinear analysis a number of load increments are evaluated. On modern personal computers this will take just a minute or so. An indication of the time remaining can be obtained by observing the number of the increment being evaluated.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results will be added to the Treeview.

In addition, these files will be created in the LUSASFiles\<model_name> folder:



- ☐ **delamination.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ☐ **delamination.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

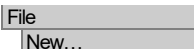
If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully. You may download this file from the user area of the LUSAS website.



- ☐ **delamination_modelling.lvb** carries out the complete modelling of the example.



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 **delamination**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **2D Inplane** is set.

- Click the **OK** button.

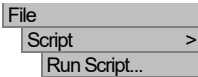


Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

To recreate the model, select the file **delamination_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 will be present in the  Treeview. For a nonlinear analysis the last solved loadcase increment is set active by default.

The first stage in any results processing is to examine the deformed mesh.

Plotting the deformed mesh

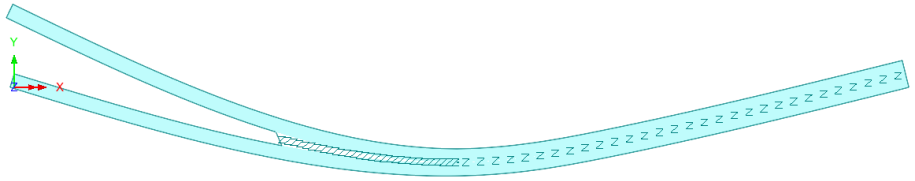
The outline of the crack which has been initiated and grown during the solution with load incrementation may be observed using a solid representation of the model. For this the deformed mesh will be examined alone. All of the model information currently displayed must be removed.

With the last (lowest) available loadcase from the  Treeview set active:

- If the deformed mesh is not already visible, with no features selected click the right-hand mouse button in a blank part of the view area and select **Deformed Mesh** and press **OK**
- Turn off each of the layer entries under the  Treeview except **Deformed Mesh**
- Press the **Deformations** button at the bottom of the  Treeview, and then select the **Specify Factor** option and enter a value of **1**. Press **OK**
- Double click on the **Deformed Mesh** entry in the  Treeview and select the **Mesh** tab.
- Ensure the **Solid** option and **Outline only** options are selected.
- Click the **OK** button to add the deformed mesh layer to the  Treeview.



Note. It is useful to use a magnification factor for displaying the deformed mesh for nonlinear analyses so that change in shape through the analyses can be observed.



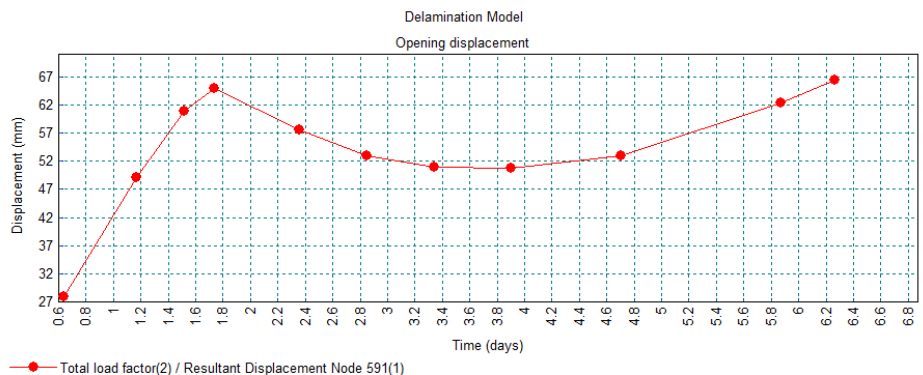
The deformed mesh plot for the final loadcase increment will be displayed.

Plotting the load deflection graph

- Select the **node** at the upper top left of the model where the opening load is applied.
- With **Time history** selected click **Next**
- With the **Nodal** option selected click **Next**
- Select Entity **Displacement** with Component **RSLT**
- The selected node will be visible in the drop down box. Click **Next**
- Select the **Named** option and click **Next**
- From the drop down list pick **Total load factor** and click **Next**
- Enter the desired graph titles and click **Finish** to display the load deflection graph.

Utilities

Graph Wizard...



- Close the graph window.

Stress contours

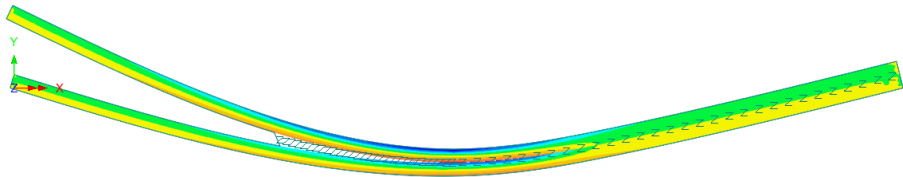
The growth of the delamination seen in the deformed mesh affects the stress distribution within the structure. This stress redistribution is easily visualised using stress contour plots.

With the last (lowest) available loadcase from the  Treeview still set active:

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

The Contour Properties dialog will be displayed.

- Select **Stress - Plane Strain** contour results in the direction **SX**
- Click the **OK** button to finish.



The contour plot and the associated contour key will be displayed.

Animating the effect of delamination growth

By setting other loadcase attributes active the effect of the delamination growth on the stress distribution may be observed. The most effective way to view this effect is to animate the deformed mesh with the stress contours layer showing.

Before animating each loadcase the contour range must be set to avoid the contour key changing between animation frames.

- Double click on the **Contours** layer in the  Treeview.
- Select the **Appearance** tab.
- For the **Classic** contour type, press the **Set** button and set the **Maximum** and **Minimum** values to be **1000** and **-1000** respectively.
- Select the **Set as global range** option
- Click the **OK** button to finish.

To create the animation:

- Select **Load History**
- Select **Next**
- Select the **All Loadcases** option.
- Select **Finish** to display the animated sequence.



Note. Animations may be saved for replay using other Windows animation players using the **File>Save As AVI** menu option.

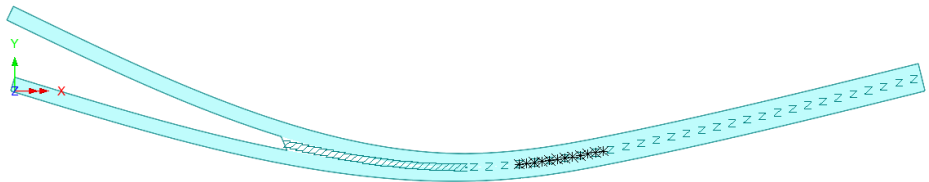


After viewing the animation close the animation window.

Yield Plot

The interface material formulation is an energy based model. The model allows the material to display three zones of behaviour. These are the elastic, softening and failed regions. Yield flags may be used to indicate which nodes within the model (if any) lie within a particular region. This technique allows a precise demonstration of the delamination crack extent within the model.

- Turn off the **Contours** layer in the  Treeview.
- With no features selected, click the right hand mouse button in the view window and select the **Values** layer from the menu.
- In the **Properties** dialogue box select the entity as **Stress - 2D Interface** and the component **Yield**
- Click **OK** to display the softening zone.



The extent of the delamination for the current loadcase can be clearly seen. The delamination growth can be visualised by animating this plot through successive loadcases.

Animating delamination growth with yield flags

To create an animation of the delamination growth with yield flags:

- Select **Load History**
- Select **Next**
- Select the **All Loadcases** option.
- Select **Finish**

This completes the example.

Tools

Animation Wizard...