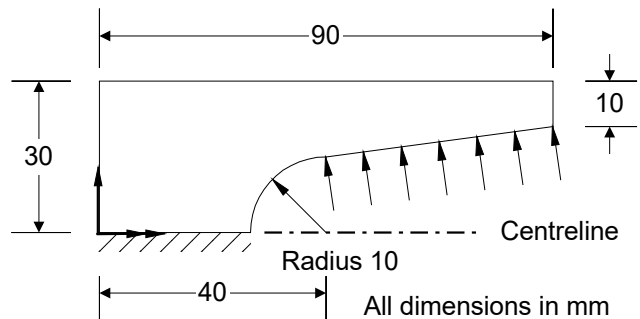


Elasto-Plastic Analysis of a V- Notch

For LUSAS version:	23.0
For software product(s):	All (except LT versions) for the first stage of the example Any Plus version for the second stage of the example.
With product option(s):	Nonlinear (for the second stage of the examples)

Description

A 5mm thick V-shaped, notched specimen is to be subjected to two load types: a pressure load distributed along the inner edge of its opening, and a loading caused by pushing a bolt into the notch of the specimen.



After an initial linear analysis with the specimen subjected to the pressure load, nonlinear material properties are defined, and a nonlinear analysis is carried out using the same pressure loading.

An additional linear and nonlinear analysis is done to investigate the insertion of a bolt into the notch of the specimen with slidelines being used to model the contact behaviour between the two components. Because of symmetry only half of the V-notch need be modelled.

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

Objectives

- ❑ **Initial Yield** Investigate the load which causes initial yielding.
- ❑ **Spread of Yield** Under continued loading investigate the progression of plastic deformation under pressure loading.
- ❑ **Ultimate Capacity (Pressure)** Investigate the ultimate capacity of the specimen under pressure loading.
- ❑ **Transfer of Force from Bolt** Investigate the transfer of forces from the loading bolt from a preliminary linear contact and subsequent nonlinear contact analysis.

Keywords

2D, Inplane, Plane Stress, Contact, Geometrically Nonlinear (GNL), Materially Nonlinear (MNL), Contour Plot, Elasto-Plastic, Yield, Yield Symbol, Plastic Strain, Animation of Results

Associated Files

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



- ❑ **vnotch_linear_modelling.lvb** carries out the modelling for the linear notch analysis.
- ❑ **vnotch_nonlinear_modelling.lvb** carries out the modelling of the nonlinear material model.
- ❑ **vnotch_linear_contact_modelling.lvb** carries out the modelling of the specimen using a linear analysis and a geometrically nonlinear model.
- ❑ **vnotch_nonlinear_contact_modelling.lvb** carries out the modelling of the notch and bolt with nonlinear material properties.

Discussion

The response of this component is dominated by materially nonlinear effects. After an initially elastic response, the material undergoes elastic-plastic yielding. In a simple von Mises model the tensile and compressive stress regions are considered to cause identical plasticity. The post-yield response is governed by the hardening slope. A zero slope denotes elastic-perfectly plastic behaviour.

Stage 1. Linear Material Analysis

An initial linear investigation is performed to verify the model and to find the maximum stress induced by a unit intensity load. This information is used to design the incrementation strategy for the initial coarse nonlinear analysis.

Stage 2. Nonlinear Material Analysis

The material nonlinearity is specified in LUSAS by the addition of plastic material properties and a hardening curve.

The nonlinear strategy is designed such that the first increment (arrived at from the linear analysis) stresses the material to just below yield in a single step. The model is loaded until a specified displacement is reached at a selected Point. The incrementation strategy is designed to develop the yielded region in a gradual and stable manner.

Stage 3. Contact Analysis with Linear Materials

Once the behaviour of the structure is understood under pressure-loaded conditions, the pressure load is removed. A bolt is added to the model and slidelines are defined to model the contact between the notch and the bolt.

Stage 4. Contact Analysis with Nonlinear Materials

Once the behaviour of the multiple-bodied structure is understood the nonlinear materials in the specimen are re-introduced and a full geometric, nonlinear contact analysis performed.

Stage 1 : Modelling : Linear Material

This worked example will create a LUSAS model of half of the notched V-specimen. Initially, a linear elastic analysis with an applied unit structural face load will be prepared. Then the material properties will be made nonlinear, and a suitable analysis control defined and assigned.

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 the file name as **vnotch_linear**
- Use the default **User-defined** working folder.
- Ensure the analysis type is **Structural**

Elasto-Plastic Analysis of a V-Notch

- Select an Analysis category of **2D Inplane**
- Select model units of **N,mm,t,s,C**
- Select timescale units as **Seconds**
- Select a Startup template of **2D Plane Stress**
- Ensure the Layout grid is set as **None**
- Enter the title as **V-Notch - Linear Analysis** and click **OK**

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

Feature Geometry



Select the **3 Columns** grid style.

Using the X and Y coordinates shown in the table define Points which mark out half of the model. Use the **Tab** key to move to the next entry field. Use the arrow keys to move around fields.

X	Y
90	20
90	30
0	30
0	0
30	0

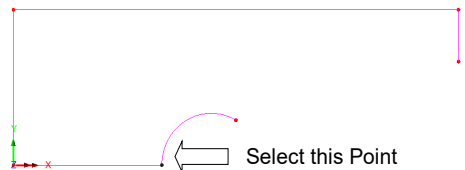
- Click the **OK** button to finish.



Note. The Z coordinate does not have to be entered. A value of zero will be assumed.

The V shaped notch will be created using an arc and a Line drawn tangential to the arc.

- Select the Point shown to generate the arc.

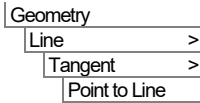
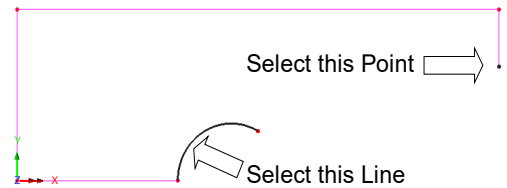


Rotate the Point through an angle of **-120** degrees about an origin of **(40,0,0)**. Click the **OK** button to finish.

Geometry
Line >
By Coords...

Geometry
Line >
Arc/Circle >
By Sweeping Points...

- Select the Point shown and add the Arc to the selection by holding down the **Shift** key.



- With **Multiple straight line tangents** selected, ensure that **Split tangent lines** and **Delete geometry on splitting** are also selected on the Straight Line Tangent dialog.

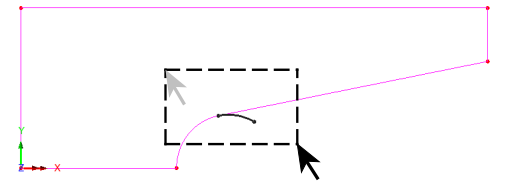
- Click **OK** to draw the Line representing the notch.

The model can now be tidied by deleting the redundant Lines and Points.

- Drag a box to select the features shown.



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



Drag a box to select these features



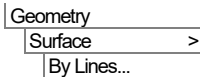
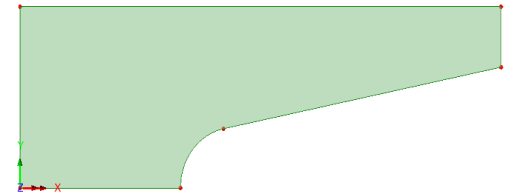
Note. The Points and Lines used to define features extending outside of the area selected will not be deleted.

Finally, a Surface is to be created from the line features.

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



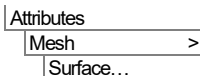
Create a **General Surface** from the selected Lines.



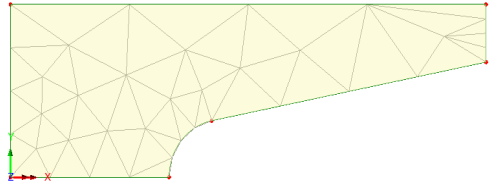
Meshing

- Define **Plane stress**, **Triangle**, **Quadratic** elements with an **Irregular** mesh spacing. Ensure that a specified element size is not selected.
- Enter the mesh attribute name as **Plane Stress (TPM6)** and click **OK**

LUSAS will add the mesh attribute to the  Treeview.



- Select the newly created surface by pressing using the **Ctrl** and **A** keys together.
- Drag and drop the Surface mesh attribute **Plane Stress (TPM6)** from the  Treeview onto the selected features. Click **OK** to complete the mesh assignment.



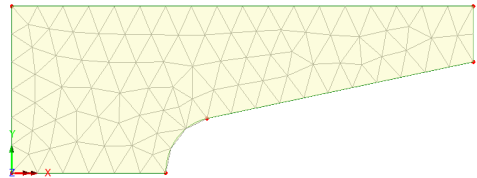
Note. When selecting lines to create a surface, the selection as viewed on the screen was only for the lines and not the new surface. It is for this reason that the whole model (and hence the surface) must be selected again using the Ctrl + A keys as described above.



Note. TPN6 elements are low-order elements and are used in this example to keep within the evaluation version limits. More accurate modelling is obtained when using Plane Stress, Quadrilateral, Quadratic elements (QPM8) which have more nodes per element.

LUSAS will mesh the Surface based upon a default Line division of 4. To improve the shape and arrangement of elements, a Line mesh attribute will be used to control the Surface mesh density.

- With the Structural element type set as **None**, define an average **Element length** of **5 mm**. Enter the Line mesh attribute name as **Element Length 5** and click **OK**
- With the whole model selected, drag and drop the Line mesh attribute **Element Length 5** from the  Treeview onto the selected features.

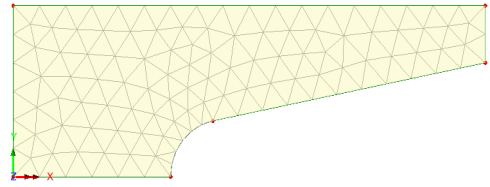


Local mesh refinement will now be applied by giving the Arc a finer Line mesh.

- Double-click on the Line mesh attribute **Element Length 5** from the  Treeview to re-open the Line Mesh dialog and use the attribute as a template.
- Change the average **Element length** to **2.5** (millimetres)
- Enter the Line mesh attribute name as **Element Length 2.5** and click **OK** to create a new attribute.

Attributes	
Mesh	>
Line...	

- Select only the arc
- Drag and drop the Line mesh attribute **Element Length 2.5** from the Treeview onto the selected Line.



The mesh will be refined as shown.

Geometric Properties

Attributes	
Geometric	>
Surface...	

- Define a geometric property attribute with a thickness of **5 mm**.
- Enter the attribute name as **Thickness=5** and click **OK** to add the attribute to the Treeview.
- With the whole model selected, drag and drop the geometry attribute **Thickness=5** from the Treeview onto the selected surface feature.

Material Properties

The only material properties that are essential for a linear elastic static analysis are Young's modulus and Poisson's Ratio. The units of Young's modulus in this particular example are N/mm².

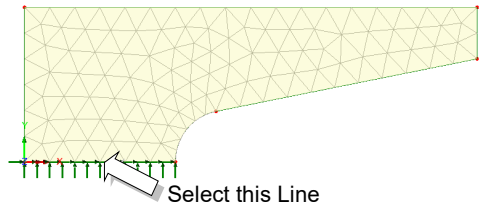
Attributes	
Material	>
Isotropic...	

- Enter a Young's modulus of **210e3** and a Poisson's Ratio of **0.3**. Leave the other fields blank.
- Enter the attribute name as **Steel (N mm)** and click **OK** to add the attribute to the Treeview.
- With the whole model selected, drag and drop the material attribute **Steel (N mm)** from the Treeview onto the selected features and click **OK** to assign to surfaces.

Support Conditions

When using the Standard template, LUSAS provides the more common types of support by default. These can be seen in the Treeview. The specimen is to be restrained along the horizontal axis of symmetry in the X and Y axes.

- Select the horizontal Line along the axis of symmetry.
- Drag and drop the support attribute **Fixed in XY** from the  Treeview onto the selected Line and click **OK** to assign to lines.



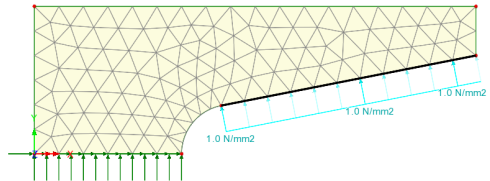
Loading

A unit pressure load is to be defined and applied to the edge of the Surface

- Select load type **Distributed loads** then **Face** and click **Next**
- Define a face load in the y direction of **1**
- Enter the attribute name as **Face Load 1** and click **Finish**

 **Note.** Face loads are pressure loads which can be applied to the edges of Surfaces or the faces of Volumes.

- Select the angled Line of the notch.
- Drag and drop the loading attribute **Face Load 1** from the  Treeview onto the selected Line and click **OK** to assign it to Lines.



 **Note.** Structural face loading uses local element directions. If the loading is incorrectly oriented, it may be due to the local element direction of the Surface. To rectify this, either reverse the element direction of the Surface by selecting the Surface using **Geometry> Surface> Reverse**, or double-click on the loading attribute name in the  Treeview and change the sign of the loading.

Saving the model



Save the model file.

File
Save

Running the Analysis : Linear Material



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

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

If the analysis is successful...

Analysis loadcase results are added to the  Treeview

In addition, these files will be created in the LUSASFiles32\<model_name> folder:



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

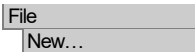
If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

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.



- ☐ **vnotch_linear_modelling.lvb** carries out the modelling of the example.

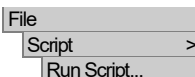


 Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **vnotch_linear_modelling**
- 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 **vnotch_linear_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Rerun the analysis to generate the results.

Viewing the Results : Linear Material

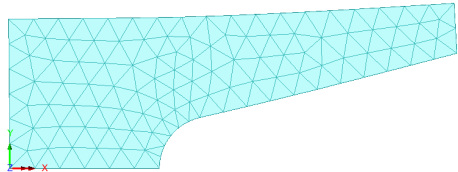
Analysis loadcase results are present in the  Treeview.

Deformed Mesh Plot

A deformed mesh layer is automatically added to the  Treeview.

Once the linear version of the model has been run it is prudent to check the deformed shape for obvious errors such as overlarge displacements in unexpected areas, which could be accounted for by incorrect properties, incorrect positioning of the load or incorrect support conditions. The deformed shape also provides a general check on the overall load direction.

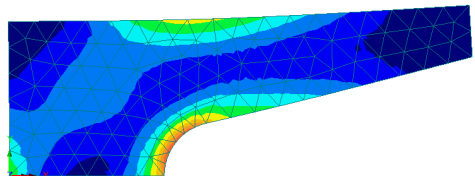
- If present in the  Treeview turn off the **Mesh**, **Geometry** and **Attributes** layers by right-clicking on a layer name and selecting the On/Off menu item.
- The Deformed mesh layer should be visible by default. If not, with no features selected, click the right-hand mouse button in a blank part of the View window and select the **Deformed mesh** option to add the deformed mesh layer to the  Treeview.



Von Mises Stress Contours

The linear elastic results provide an opportunity to establish a faster loading scheme for the nonlinear analysis to come. Checking the maximum von Mises stress values will allow calculation of a factor of load that can be sustained without yielding the material.

- With no features selected click the right-hand mouse button in a blank part of the View window and select the **Contours** option to add the contour layer to the  Treeview.
- Select entity **Stress** and component equivalent stress **SE**
- Click the **OK** button to display the contours. A contour key (not shown here) will also be displayed.





Note. The material chosen for the analysis is assumed to yield at 300 N/mm^2 . From the contour key results the maximum stress induced from a unit face load results in a stress of just over 30 N/mm^2 . Therefore, a factored load value of 9 would be suitable for use as the first load increment level in the nonlinear analysis as this would result in stresses just below the yield stress.

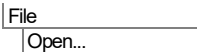
This completes the linear analysis stage of the example.

Stage 2 : Modelling : Nonlinear Material

For software product(s):	Any Plus version.
With product option(s):	Nonlinear.

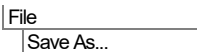
Files. The geometry of the notch is the same as that defined for the linear model. It may be recovered by one of two methods; either by loading the previously created model file, or by creating the initial model from a supplied file as described below.

If the linear analysis was successful:



If the previous linear analysis was performed successfully open the model file **vnotch_linear.mdl** which was saved after completing the first part of this example and select **No** to not load a results file of the same name on top of this model.

- Ensure the display of the **Geometry** and **Mesh** layers is turned on.
- Enter the model file name as **vnotch_nonlinear** and click the **Save** button.



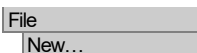
Creating a suitable model from a supplied file

You may download this file from the user area of the LUSAS website.



- ☐ **lug_linear_modelling.lvb** This carries out the modelling of the example.

Creating the initial model from a supplied file



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 **vnotch_linear**
- 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 **vnotch_linear_modelling.lvb** that was downloaded and placed in a folder of your choosing.

Once created:

Change the model description

Change the model description to **V-Notch - Nonlinear Analysis** and click **OK**

Modifying the Material

The linear material dataset needs to be modified to include plastic properties.

- In the  Treeview double-click on the material attribute **Steel (N mm)**
- Select the **Plastic** check box and on the Plastic tab for a **Stress potential** model enter an **Initial uniaxial yield stress** of **300**
- Select the **Hardening** check box, select the **Hardening gradient** option and enter a **Slope** of **0** and a **Plastic strain** of **1000** in the first row of the table.
- Click the **OK** button to finish and overwrite the existing material dataset.



Note. The von Mises material model used assumes identical behaviour in compression and tension.

Nonlinear Analysis Control

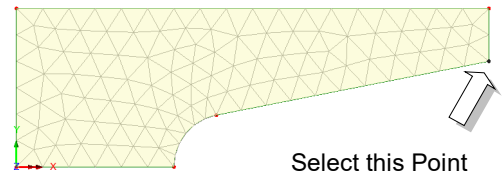
From the results of the Linear analysis, initial yielding is expected at a load factor of nearly 10 (Yield Stress/Max Stress). Therefore the nonlinear loading strategy will be to apply an initial load factor of 9 and use unit factor increments as the yielding progresses. The analysis will be set to stop once the Point at the inside end of the arm of the notch has displaced 10mm from its original position.

- Ensure that the geometry layer is present in the  Treeview and turned On.

- Select the Point at the end of the notch as shown.



Note. The number of the Point selected will be displayed in the status bar at the bottom of the view window.



Nonlinear analysis control options are defined as properties of a loadcase.

- In the  Treeview expand **Analysis 1** then right-click on **Loadcase 1** and from the **Controls** menu select **Nonlinear and Transient**.

The Nonlinear & Transient dialog will appear.

- Select the **Nonlinear** option.
- From the **Incrementation** drop down list select **Automatic** control.
- Set the **Starting load factor** as **9**
- Set the **Max change in load factor** as **1**
- Set the **Max total load factor** as **0** to enable the load to increase without limit.

To terminate the load on a limiting variable the advanced nonlinear parameters need to be set in the Incrementation section.

- Select the Incrementation **Advanced** button.
- Click the **Terminate on value of limiting variable** option.
- Set the **Point number** from the drop-down list to the Point selected earlier.
- Set the **Variable type** in the drop-down list as **V** to limit the displacement in the local Y direction.

- Set the **Value** as **10**
- Ensure the **Allow step reduction** option is ticked.
- Click the **OK** button to return to the load case dialog.
- Click the **OK** button to finish.

Save the model

File
Save



To save the model.



Note Geometric stiffening is not considered in this example because the nonlinear effects are predominantly due to yield in the material.

Running the Analysis : Nonlinear Material



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

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

During the analysis...

Apart from the increment and iteration, several parameters output are of special interest during the nonlinear analysis phase:

- ☐ **TLMDA (Total Load Factor)** The factor of load applied using the incrementation control is displayed here. It shows how the load application is progressing for a load control and an arc-length solution.
- ☐ **DTNRM (Displacement Norm)** The changes in this value indicate how well the problem is converging.

If the analysis is successful...

Analysis loadcase results are added to the  Treeview

In addition, these files will be created in the LUSASFiles32\<model_name> folder:



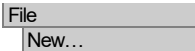
- ☐ **vnotch_nonlinear.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ☐ **vnotch_nonlinear.mys** this is the LUSAS results file which is loaded automatically into the  Treeview to allow results processing to take place.

If the analysis fails...

If errors are listed that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully. You may download this file from the user area of the LUSAS website.



- ☐ **vnotch_nonlinear_modelling.lvb** carries out the complete nonlinear material modelling of the example.

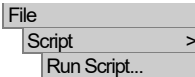


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **vnotch_nonlinear**
- 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 **vnotch_nonlinear_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Rerun the analysis to generate the results.

Viewing the Results : Nonlinear Material

This section covers a typical results processing session for a nonlinear analysis. In this session the following procedures will be carried out:

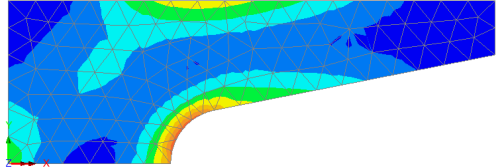
- ☐ A von Mises stress contour plot and associated yielded region plot will be drawn.
- ☐ A graph showing the displacement history of the notch opening through the analysis will be created.

Plotting Stress Contours

Analysis loadcase results are present in the  Treeview.

- In the  Treeview ensure that the **Contours** and **Mesh** layers are present and are turned **On**, and that **Geometry** and **Deformed Mesh** are turned **Off**.

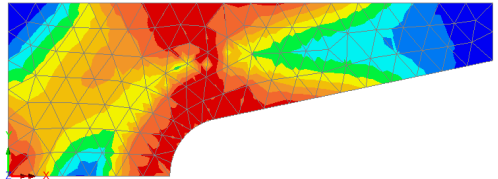
- In the  Treeview, right-click on the first load increment **Increment 1 Load Factor = 9** and select the **Set Active** option.
- Double click on the **Contours** entry to show its property dialog and select entity **Stress** and component equivalent stress **SE**
- Click the **Appearance** tab and for the **Classic** contour range, press the **Set** button. Ensure the **Maximum** stress contour value to be plotted is **300** so all stresses above yield will be drawn in red and click **OK**.
- Click the **OK** button again to display contours for the first load increment.



The display will show that the yield stress of 300Nmm^{-2} has not been reached after the first loading increment.

Changing the Active Results Loadcase

- In the  Treeview, right-click on the last load increment and select the **Set Active** option.

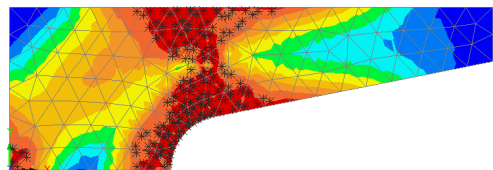


Contours for the selected increment will be displayed. The red area shows the peak von Mises stress which is being limited to 300N/mm^2 by the zero hardening slope as defined in the plastic material properties section for the model.

Yielded Material Plot

In addition to using contours to show yielded regions of the model, the spread of plasticity can be visualised using yield symbols.

- With no features selected click the right-hand mouse button in a blank part of the view window and select the **Values** layer.
- Select the **Stress** entity and **Yield** component at location **Gauss point** and click the **OK** button to display the yielded Gauss points with an asterisk.

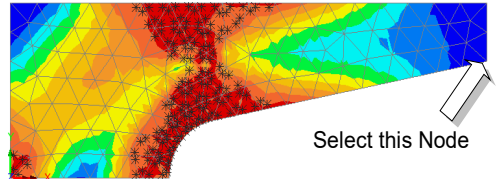


Note. By changing the active loadcase the spread of yield through the model can be viewed.

Displacement History Graph

To illustrate the nonlinear behaviour of the model a displacement history graph showing the displacement of a node on the notch against the total applied load factor is to be displayed.

- Select the node on the end of the notch as shown.



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 button is selected for the X axis results and click the **Next** button.
- Select **Displacement** from the entity drop down list for the component **DY**. The node number of the selected node will be shown. Click the **Next** button.

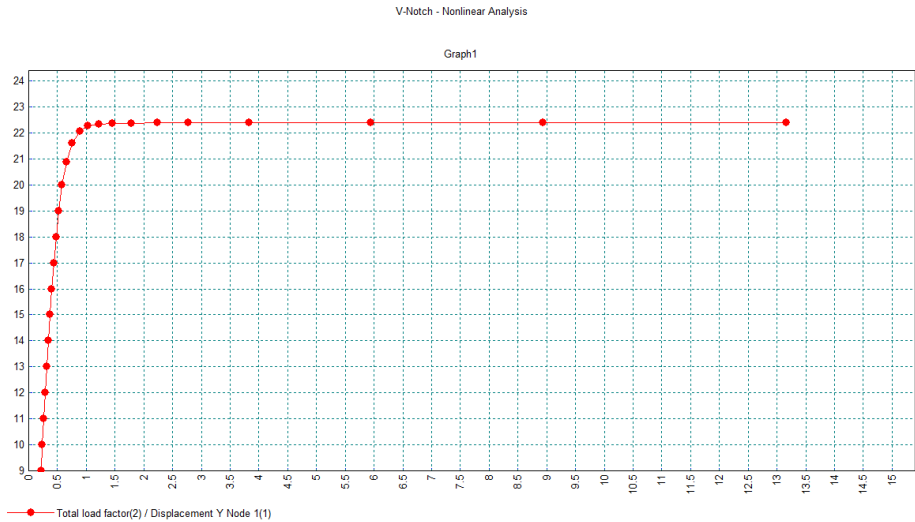
This defines the X axis results.

- Select **Named** results and click the **Next** button.
- From the drop-down list select **Total Load Factor** data and click the **Next** button.

This defines the Y axis results.

- Leave all title and axis fields blank and click the **Finish** button to create the graph in a new window and display the values used in an adjacent table.

Elasto-Plastic Analysis of a V-Notch



Note. The graph shows the progressive softening of the structural response as the load is increased. The load value corresponding to the flat section represents the limit of the load carrying capacity of the model.



Close the graph window.

Stage 3: Modelling : Contact Analysis (Linear Material)

For software product(s):	Any Plus version.
With product option(s):	Nonlinear.

This section details the changes required to the linear material model from the first part of the example to incorporate the geometry of a bolt which will be positioned and loaded to prise the notch arms apart. A nonlinear analysis is required because of the presence of slidelines and the boundary conditions used.

If the initial linear analysis was successful...



If the initial linear analysis was performed successfully open the model file **vnotch_linear.mdl** saved after completing the first part of this example. If carrying on from the previous stage of the example click **OK** to save the model (and any graphing data), then select **No** to not load a results file of the same name on top of this model.

- Enter the model file name as **vnotch_contact** and click the **Save** button.

Creating the starting model from a supplied file

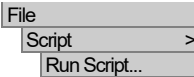


Alternatively, start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **vnotch_linear_contact**
- 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 create the model, select the file **vnotch_linear_modelling.lvb** that was downloaded and placed in a folder of your choosing.

Once created:



- Change the model description to **V-Notch - Contact Analysis (Linear Material)** and click **OK**
- In the  Treeview ensure that the **Geometry**, **Mesh** and **Attributes** layers are present and are turned on.

Deassigning the Loading

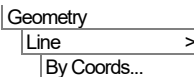
The initial pressure loading is no longer required. In the nonlinear contact model the loading will be applied as a prescribed displacement to the centre-line of the bolt.

- In the  Treeview click the right-hand mouse button on the loading attribute **Face Load 1** and select the **Deassign > From all** option.

Modelling the Bolt



Note. At this stage the bolt will be defined separated from the notch specimen. This will make assigning the attributes, especially the slidelines, easier. Once the attributes are assigned, the bolt will be moved to its starting position before the analysis is run.



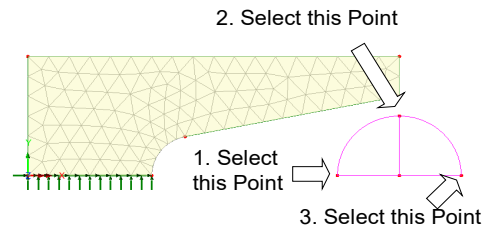
Enter coordinates of **(75,0)**, **(90,0)** and **(105,0)** and click **OK** to define 2 horizontal Lines on the bolt centreline.



Enter coordinates of **(90,0)** and **(90,15)** and click **OK** to define a vertical Line on the bolt centreline.

An arc is to be drawn to form the upper half of the bolt.

- Select the 3 Points in the order shown.
- Click **OK** to accept the values present on the dialog and the arc will be drawn.



This arc will now be split into 2 new arcs.

- Select the arc just drawn
- Enter the number of divisions as **2**
- Click **OK** to create 2 arcs and delete the original arc.

Surfaces will now be defined on the bolt.

- Select the left-hand arc and then the 2 Lines as shown.

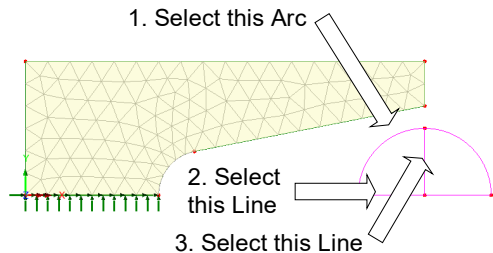


The left-hand Surface of the bolt will be drawn.

- Select the right-hand arc and then the other 2 Lines to create the Surface for the other half of the bolt.



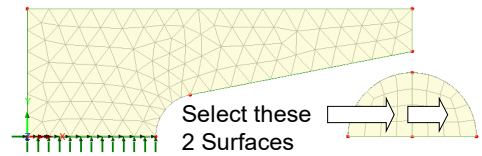
The right-hand Surface of the bolt will be drawn.



Meshing the Bolt

- In the  Treeview double-click on the mesh attribute **Plane Stress (TPM6)**
- Change the element shape to **Quadrilateral**, select **Regular Mesh**, and change the attribute name to **Plane Stress Quads**.
- Click the **OK** button to create a new mesh dataset in the  Treeview.

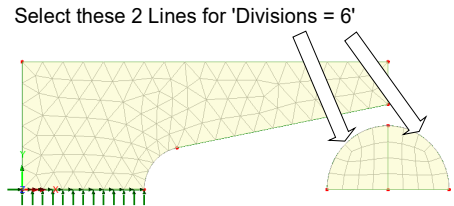
- Select both Surfaces defining the bolt and drag and drop the surface mesh dataset **Plane Stress Quads** from the  Treeview onto the selected features.



Modifying the mesh on the bolt

A new line mesh is required for the lines representing the surfaces of the bolt.

- In the  Treeview double click the Line mesh entry **Null, div = 4**. Change the number of divisions to **6**. Change the name to be **Null, div = 6** and click OK.
- Select the 2 arcs at the top of the bolt as shown.
- Drag and drop the line mesh attribute **Null, div = 6** from the  Treeview onto the selected Lines.



The mesh will be redrawn as shown.

Geometric Properties

- In the  Treeview double click on the geometry attribute **Thickness=5**
- Change the thickness to **10** and change the attribute name to **Thickness=10**
- Click the **OK** button to create a new geometric mesh attribute in the  Treeview.
- Select both Surfaces of the bolt and drag and drop the geometry attribute **Thickness=10** from the  Treeview onto the selected features. Click **OK**.

Material Properties

- With the bolt selected, drag and drop the material attribute **Steel (N mm)** from the  Treeview onto the selected features. Click **OK**.

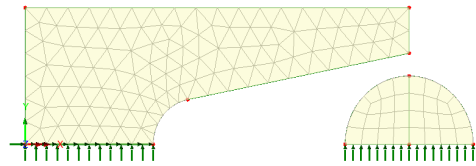


Note. At this stage the material model is linear elastic and does not include plasticity effects. The addition of plasticity is included at the next stage of the example.

Support Conditions

A roller support is required at the bolt centreline to restrain in the Y direction only.

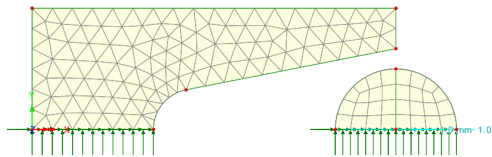
- Select the 2 horizontal Lines on the bolt centreline.
- Drag and drop the support attribute **Fixed in Y** from the Treeview onto the selected lines.
- Click **OK** to assign the support dataset to the selected Lines.



Loading Conditions

An incremental prescribed displacement will be used. At this stage a negative unit displacement will be applied. The magnitude of each increment is controlled later using nonlinear control parameters.

- Select the **Displacement, velocity, acceleration** option.
- Select the **Prescribed Displacement** option and click **Next**
- Select the **Incremental** option, choose the **Fixed** option in **X** and enter a displacement of **-1**
- Enter the attribute name as **Prescribed Load 1** and click **Finish**
- With the 2 horizontal Lines on the bolt centreline selected, drag and drop the loading dataset **Prescribed Load 1** from the Treeview onto the selected features.
- Click **OK** to assign the loading to **Analysis 1** and **Loadcase 1** with a factor of **1**



Slideline Definition

Slidelines automatically model components having dissimilar meshing patterns and can also model any frictional contact between interacting components. Slidelines are to be applied to the contacting Lines of the notch and the bolt.

- Ensure the **Primary Stiffness Scale** and **Secondary Stiffness Scale** values are set to **1** and leave the remaining values as defaults.
- Enter the slideline attribute name as **Slideline 1** and click **OK** to add the slideline attribute to the Treeview.

Attributes
Loading...

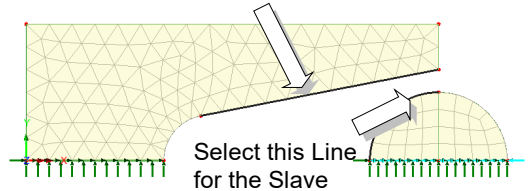
Attributes
Slideline...

The Coulomb friction coefficient defaults to zero, which will define a standard no friction slideline.

- Select only the inclined Line of the notch.

Select this Line for the Master

- Drag and drop the slideline attribute **Slideline 1** from the  Treeview onto the selection.



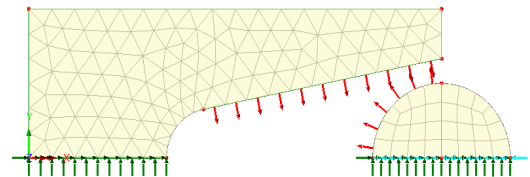
- Ensure the **Primary** option is selected and click **OK** to accept the default orientation.

- Select only the left arc of the bolt.

- Drag and drop the slideline attribute **Slideline 1** from the  Treeview onto the selection.

- Select the **Secondary** option and click the **OK** button to accept the default orientation.

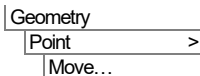
- To visualise the slidelines assigned to the model click the right-hand mouse button on the slideline attribute name in the  Treeview and select **Visualise Primary Assignments** and **Visualise Secondary Assignments**



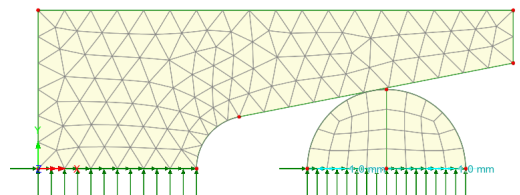
- Repeat the process above to deselect the slideline visualisation.

All assignments are now complete, and the bolt can be moved into a starting position just adjacent to the notch ready for the analysis.

- Drag a selection box around the bolt to select all its features.



-  Enter a translation of **-24** in the **X** direction.



- Leave the attribute name blank and click the **OK** button to move the bolt into position.

This moves the bolt to within touching distance of the notch in readiness for solving the model. Nonlinear control settings will be defined to move the bolt further to the left as the solution proceeds.

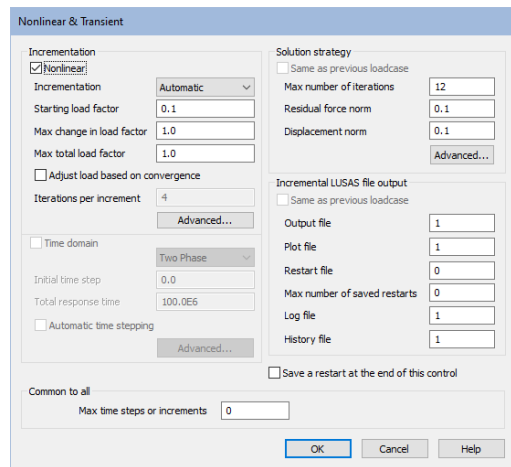
Nonlinear Analysis Control

A bolt displacement of 1mm is to be specified. This will be done in 10, 0.1mm increments using nonlinear control properties. The nonlinear analysis control parameters are applied as properties of the loadcase.

- In the  Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Nonlinear and Transient** from the **Controls** menu.

The Nonlinear & Transient dialog will appear.

- Set **Nonlinear** incrementation with **Automatic** control.
- Set the **Starting load factor** as **0.1**
- Set the **Maximum change in load factor** as **1.0**
- Set the **Maximum total load factor** as **1.0**
- Deselect the option to **Adjust load based upon convergence**
- Click the **OK** button to finish the definition of the nonlinear parameters.



The dialog box is titled "Nonlinear & Transient". It contains several sections: "Incrementation" with a checked "Nonlinear" checkbox and a dropdown set to "Automatic"; "Starting load factor" (0.1), "Max change in load factor" (1.0), and "Max total load factor" (1.0); an unchecked "Adjust load based on convergence" checkbox; "Iterations per increment" (-4); "Time domain" set to "Two Phase"; "Initial time step" (0.0), "Total response time" (100.0E6), and an unchecked "Automatic time stepping" checkbox; "Solution strategy" with an unchecked "Same as previous loadcase" checkbox; "Max number of iterations" (12), "Residual force norm" (0.1), and "Displacement norm" (0.1); "Incremental LUSAS file output" with an unchecked "Same as previous loadcase" checkbox; "Output file" (1), "Plot file" (1), "Restart file" (0), "Max number of saved restarts" (0), "Log file" (1), and "History file" (1); and a "Common to all" section with "Max time steps or increments" (0). At the bottom are "OK", "Cancel", and "Help" buttons.

Running the Analysis : Contact Analysis (Linear Material)



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

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

If the analysis is successful...

Analysis loadcase results are added to the  Treeview

In addition, these files will be created in the LUSASFiles32\<model_name> folder:



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

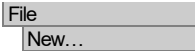
- ❑ **vnotch_linear_contact.mys** this is the LUSAS results file which is loaded automatically into the  Treeview to allow results processing to take place.

If the analysis fails...

If errors are listed that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully. You may download this file from the user area of the LUSAS website.



- ❑ **vnotch_linear_contact_modelling.lvb** carries out the modelling of the example.

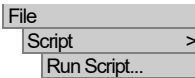


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **vnotch_linear_contact_modelling**
- 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 **vnotch_linear_contact_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Rerun the analysis to generate the results.

Viewing the Results - Contact Analysis (Linear Material)

Analysis loadcase results are present in the  Treeview. For a nonlinear analysis the results for the last load increment are set to be active by default.

Equivalent Stress Contour Plots

- In the  Treeview, right-click on the first load increment **Increment 1 Load Factor = 0.1** and select the **Set Active** option.
- If present in the  Treeview turn off the display of the **Mesh**, **Geometry** and **Attributes** layers.
- Ensure the **Deformed Mesh** layer is present and its display is On.

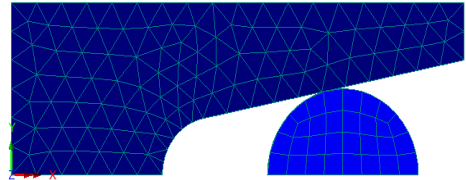
- Press the **Deformations** button in the  Treeview and select **Specify factor** with a value of **1**.



Note. When viewing results for contacting components it is important that the deformed mesh is plotted using a specified factor of 1 rather than a specified magnitude otherwise the components appear to contact incorrectly.

- If necessary, and with no features selected click the right-hand mouse button in a blank part of the View window and select the **Contours** option to add the contours layer to the  Treeview and display the contour properties.
- On the contour properties dialog select entity **Stress** and component equivalent stress **SE**.
- Click the **Appearance** tab and for the **Classic** contour range, press the **Set** button. Set the **Maximum** stress contour value to be plotted as **300** so all stresses above yield will be drawn in red and click **OK**.
- Click the **OK** button to display the contours and contour key for the first load increment.

The contour arrangement shows that the first load increment does not induce any stresses in the arms of the notch since contact has not yet taken place.

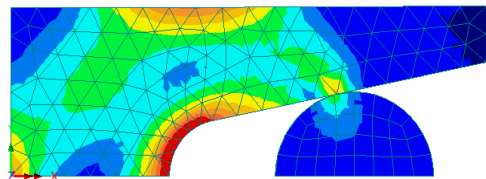


Changing the Active Results Loadcase

- In the  Treeview right-click on the last load increment for load factor 1.0 and select the **Set Active** option.

The contour plot for the final increment will be displayed.

From this stress plot where the bolt displacement is 1mm, it can be seen from the contour key that the material in the root of the notch is stressed to levels above the yield value of the material.





Note. By investigating the other load increments it can be seen that after the third load increment the bolt begins to induce stresses in the notch and that the load is transferred from the bolt to the specimen via the slidelines.

Stage 4: Modelling : Contact Analysis (Nonlinear Material)

For software product(s):	Any Plus version.
With product option(s):	Nonlinear.

Creating the Model



Files. The geometry of the notch lug is the same as that defined for the geometrically nonlinear contact analysis model. It may be recovered by one of two methods.

If the initial linear analysis was successful...



If the linear contact analysis was performed successfully re-open the model file **vnotch_contact.mdl** and select **No** to not load a results file of the same name on top of this model.

- Enter a model file name of **vnotch_nonlinear_contact** and click the **Save** button.

Creating the starting model from a supplied file



Alternatively, 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 **vnotch_nonlinear_contact**
- 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 **vnotch_linear_contact_modelling.lvb** that was downloaded and placed in a folder of your choosing.

File
Open...

File
Save As...

File
New...

File
Script >
Run Script...

Changing the model description

- Change the model description to **V-Notch - Contact Analysis (Nonlinear Material)** and click **OK**
- In the  Treeview ensure the **Geometry** and **Mesh** layers are present and displayed.

Modifying the Geometry

The linear material attribute needs to be modified to include plastic properties.

- In the  Treeview double-click on the material attribute **Steel (N mm)**
- Select the **Plastic** check box and on the Plastic tab enter an **Initial uniaxial yield stress** of **300**
- Select the **Hardening** check box, select the **Hardening gradient** check box and enter a **Slope** of **0** and a **Plastic strain** of **1000** in the first line of the table.
- Click the **OK** button to overwrite the existing material definition.



Note. The perfectly plastic assumption is an initial simplification that would in practice be replaced by a more detailed description of the hardening behaviour of the material. This would typically involve specifying several hardening gradients (i.e nonlinear hardening) and the strain limits to which each slope applies.

Nonlinear Analysis Control

The nonlinear control properties required for this section of the analysis are already specified in the current model. However, the bolt displacement of 1mm is to be increased to 2.5mm by specifying 25, 0.1mm increments in the nonlinear section of the load case dialog.

- In the  Treeview double-click on **Nonlinear and Transient** to edit the existing parameters.
- Set the **Max total load factor** to **2.5**
- Click the **OK** button to return to the view window.

Save the model



Save the model.

File
Model Properties...

File
Save

Running the Analysis : Contact Analysis (Nonlinear Material)



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

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

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, these files will be created in the LUSASFiles32\<model_name> folder:



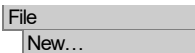
- ☐ **vnotch_nonlinear_contact.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ☐ **vnotch_nonlinear_contact.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If errors are listed that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully.



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

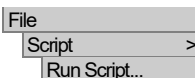


Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **vnotch_nonlinear_contact_modelling**
- 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 **vnotch_nonlinear_contact_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Rerun the analysis to generate the results.

Viewing the Results : Contact Analysis (Nonlinear Material)

Analysis loadcase results are present in the  Treeview.

Preparing to animate the spread of yielded material

As an alternative to viewing results individually for each loadcase, the change of stress and the spread of yielded material due to the increasing load increments can be animated instead. Stress contours and yield symbols will be plotted for selected load increments.

- If present in the  Treeview, turn off the display of the **Mesh**, **Geometry** and **Attributes** layers. Then ensure the **Deformed Mesh** and **Contour** layers are present and displayed.
- In the  Treeview, ensure the Deformations factor is set to **1** and click the **OK** button.



Note. When animating nonlinear loadcases it is important that the deformed mesh is plotted using a specified factor of 1 and not using a fixed screen size value of magnitude otherwise the deformed mesh for each load increment would be drawn the same.

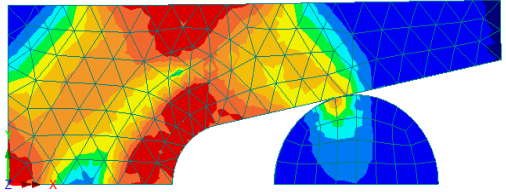
- In the  Treeview, set the last load increment (for load factor 2.5) active
- Double click the **Contours** layer in the  Treeview.

The contour plot properties will be displayed.

- Select **Stress** contour results of Equivalent stresses **SE**
- Click the **Appearance** tab and for the **Classic** contour range, press the **Set** button. Ensure the **Maximum** stress contour value to be plotted is **300** so all stresses above yield will be drawn in red and click **OK**.
- Click the **OK** button.

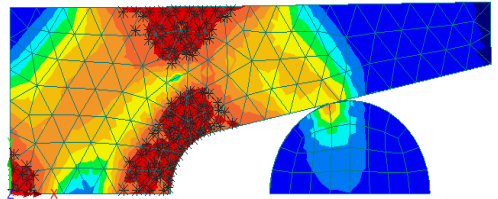
The contour plot and contour summary for load increment 25 will be displayed showing the region of yielded material has spread across the whole arm of the model.

- If necessary, change the layer display order to display the deformed mesh on top of the contour plot by moving the **Deformed mesh** to follow **Contours** in the  Treeview.
- With no features selected click the right-hand mouse button in a blank part of the view window and select **Values** to add the values layer to the  Treeview.



The values layer properties dialog will be displayed.

- Select **Stress** values of component **Yield**
- Click the **OK** button to display yield symbols for load increment 25 to show areas of the model that have yielded.



Animating the results

Tools
Animation Wizard...

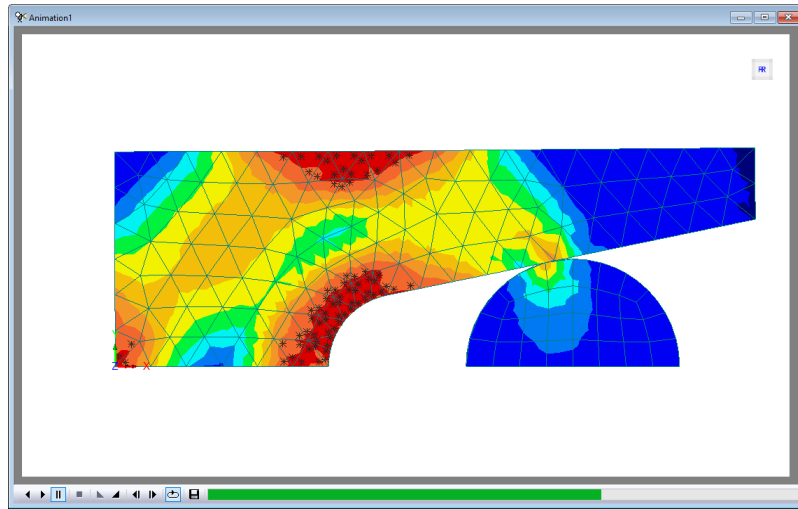
- Select the **Load history** option and click the **Next** button.
- Select the **Specified** option.

The list of available load cases for selection will appear.



Note. With so many load increments there is no need to animate the whole sequence. The selected load cases can be filtered to reduce the number to be animated.

- Enter a **Step** value of 4
- Select the first load increment, hold down the **Shift** key and select the last load increment.
- Click the  button to add the first and every 4th load increment to the included panel.
- Click the **Finish** button to create the animation sequence and display the animation in a new window.



Note. The buttons at the bottom of the window may be used to slow-down, speed-up, pause, step through frame by frame, stop the animation, or save it as a AVI file.

Saving Animations

Animations may be saved for replay in LUSAS at any time or saved for display in other windows animation players.

- Ensure the animation window is the active window.
- Enter **vnotch_nonlinear_contact** for the animation file name. An **.avi** file extension is automatically appended to the file name when it is saved.
- Click **Save** to create the animation file.

This completes the example.

Discussion: Use of the Multiple Analysis Facility

This example, as written, uses individual models to carry out the various linear and nonlinear analyses, ignoring and then considering the contact effects in stages.

Note that when the model geometry is unchanged between different analyses, the use of the multiple analysis facility within LUSAS Modeller would do away with the need to create separate models, or maintain “clone” copies of a model, in order to switch between and create results from linear static analysis and other analysis types.

For this worked example the linear and nonlinear material analyses could be set-up and solved within one model, and the contact analyses could be solved in another model. The procedure described below would be used to extend the initial linear analysis to additionally include a nonlinear material, and carry out a nonlinear analysis, all within a single model.

With the linear analysis section of the example completed, or with the linear analysis model loaded:

1. Create a new Structural analysis called Analysis 2 that inherits all the base analysis assignments. Note that loading assignments are NOT inherited between analyses. So:
2. Assign the loading **Face Load 1** to the line representing the inside of the V-notch for loadcase 2 within Analysis 2.
3. Define a nonlinear material and assign it to the model geometry for Analysis 2 (the linear material assignment previously made to the same geometry will still be valid for Analysis 1)
4. Define a nonlinear analysis control for loadcase 2 within Analysis 2.
5. Solve the model.
6. After a successful solve, nonlinear loadcase increments will be added to the Analysis 2 entry within the  Treeview.

