Linear Elastic Analysis of a Spanner
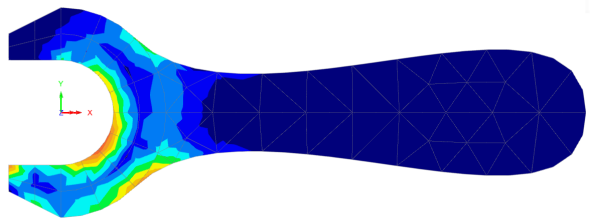
For LUSAS version:	21.1
For software product(s):	All (except LT versions)
With product option(s):	None.

Description

A linear static analysis is to be carried out on the spanner shown.

The spanner is supported as though it were being used to turn a nut and is loaded with a constant pressure load along the top edge of the handle.

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

Keywords

2D, Inplane, Plane stress, Linear, Elastic, Regular Meshing, Irregular Meshing, Copy, Rotate, Mirror, Transformation, Groups, Deformed Shape, Contour Plot, Principal Stress Vector Plot, Graph Plotting, Slice Section Results.

Objectives

The required output from the analysis is:

- ❑ A plot of the deformed shape.
- ❑ A plot of the equivalent stresses in the spanner.
- ❑ A graph showing the variation in equivalent stress where the handle meets the jaws.

Associated Files

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

- ❑ **spanner_modelling.lvb** carries out automated modelling of the example.

Modelling : Features

This section covers the definition of the features (Points, Lines and Surfaces) which together form the geometry of the spanner.

- ❑ The symmetry of the spanner will be used by firstly defining one half and then mirroring about a horizontal centreline.
- ❑ One half of the jaws will be defined by three Surfaces using 3 different methods. One Surface will be defined simply by its bounding coordinates, a second by sweeping a Line through a rotational transformation and a third by copying the second Surface using a pre-defined rotation.
- ❑ One half of the handle will be defined using one Surface. It will be bounded by three Lines, one of which will be a cubic spline.
- ❑ Once the Surfaces have been defined, they will all be mirrored about the spanner centreline.
- ❑ The features which make up the spanner will be divided into two Groups, the jaws and the handle, to make the assignment of attributes easier.

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 **Spanner**

- Use the default User-defined working folder.

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

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

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

- Leave the Timescale units as **Seconds**

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

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

- Enter a Title of **spanner**

- Click the **OK** button.
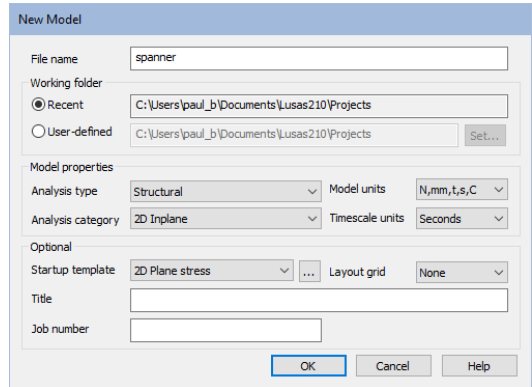
The Layers treeview will be populated with an initial set of drawing layers. The icons used signify whether a layer's display is 'on' or 'off', transparent or opaque, and drawn as a wireframe or solid.

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. The Undo button may be used to correct a mistake. The undo button allows any number of actions since the last save to be undone.

Modelling geometry

LUSAS Modeller is a feature-based modelling system. The model geometry is defined in terms of features (Points, Lines, Surfaces and Volumes) which are later meshed to generate the Finite Element Model ready for solution. The features form a hierarchy, with Points defining Lines, Lines defining Surfaces and Surfaces defining Volumes.

Geometry
> Surface   >
>> By Coords...

🖼 **Enter** coordinates of (**0, 20**), (**-20, 20**), (**-20, 30**) and (**0, 40**) to define the vertices of the Surface.

✏️ **Note.** Sets of coordinates are separated by commas or spaces. The **Tab** key is used to create new rows in the grid. The arrow keys are used to move between entries.

- Click the **OK** button to generate the Surface as shown.

✏️ **Note.** If the Z ordinate is omitted zero is assumed.

✏️ **Note.** Confirmation that the Surface has been created is given in the message window.

Enter Coordinates ✕

Grid style
☑ 3 columns

	X	Y	Z
1	0	20	
2	-20	20	
3	-20	30	
4	0 ⏷	40	

Local coordinate

Global coordinates ⌄

☐ Set as active local coordinate

[OK]   [Cancel]   [Help]

✏️ **Note.** Cartesian, cylindrical or spherical coordinates systems may be used to define models. In this example Cartesian coordinates will be used throughout.

The right-hand vertical Line will be used to create a Surface by sweeping the line through a rotation of 45 degrees clockwise.

- Select the Line by moving the cursor over the Line shown and clicking the left-hand mouse button.

The line will change colour to show it has been selected.

✏️ **Note.** Feedback on the items currently selected is provided on the right-hand side of the status bar at the bottom of the display.
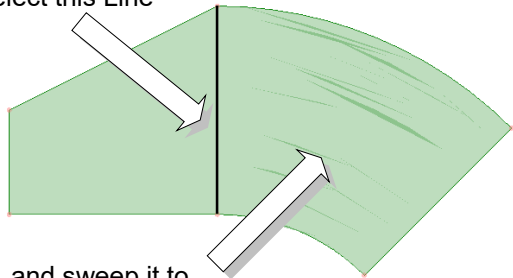
Select this Line

... and sweep it to form this Surface

Geometry
> Surface >
>> By Sweeping..
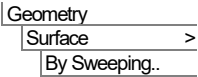
Select the **Rotate** option and enter **-45** for the angle of rotation about the **Z-axis**.

- Enter the attribute name as **Rotate 45 Clockwise**

- Click the **Save** button to save the dataset for potential re-use.

- Click the **OK** button to sweep the Line clockwise through 45 degrees about (0,0,0) to create a Surface.

Note. Clockwise angles are negative and anti-clockwise angles are positive.
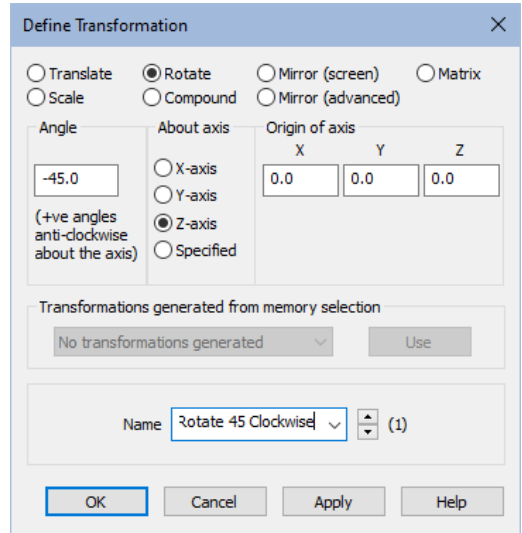
Whilst a suitable line on this newly created surface could be swept to create another surface, instead the surface just drawn will be copied to achieve the same result by rotating it through a 45 degree rotation clockwise.

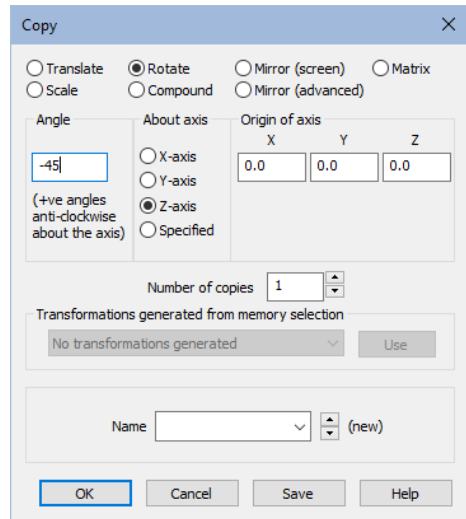- Select the previously drawn Surface by clicking on it with the left-hand mouse button.
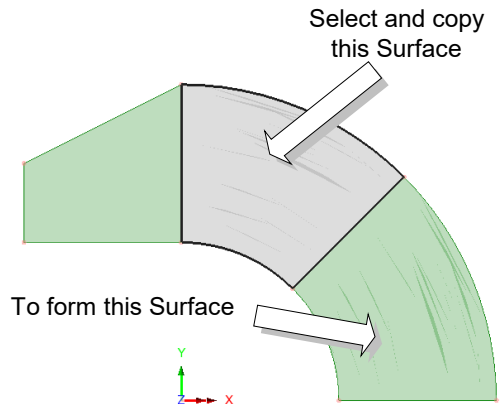
Geometry
| Surface | > |
| Copy... | |

Select the **Rotate** option. Enter an angle of **-45**

- Ensure that the number of copies is set to **1**

- Click the **OK** button to copy the Surface, rotating it clockwise through 45 degrees.

To define the top line of the handle of the spanner a cubic spline will be created.

Note. A cubic spline is a Line which passes through any number of Points. If required, the start and finish directions of the spline can be defined by specifying end-tangents (i.e. by specifying the directions of Lines at its ends).

In this example end-tangents are used to fix the start and finish directions of the spline so a construction Line must first be defined.

Geometry
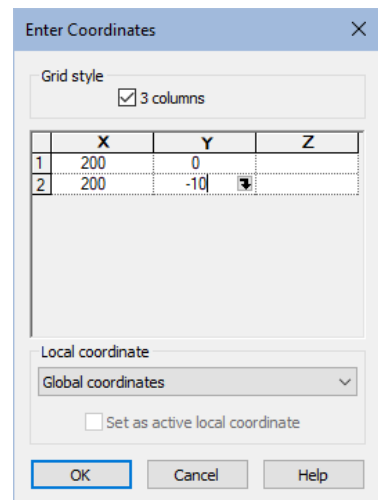| Line | > |
| By Coords... | |

Enter coordinates of (**200, 0**) and (**200, -10**) to define the construction Line.

- Click the **OK** button to generate a vertical Line away from the existing Surfaces.

This Line will be used to specify the finishing direction of the cubic spline.

Note. When selecting features to define a cubic spline it is very important that the correct features are selected in a particular order. The Lines that define the start and finish directions of the spline are to be selected first, followed by the Points that define the start and end positions of the spline.

Defining a cubic spline

- Select the arc shown by moving the mouse over the arc and click the left-hand mouse button. (This arc defines the direction in which the spline starts).

1. Select this arc

3. Select this Point

4. Select this Point

2. Select this Line

- Hold the **Shift** key down to add additional features to the selection

- Select the construction Line defined earlier by moving the mouse over the Line shown and click the left-hand mouse button. (This Line defines the direction in which the spline ends).

- Continue holding the **Shift** key down to further add to the selection. Select the Point on the end of the first arc selected. (This defines where the spline starts).

- Still holding the **Shift** key down, select the Point on the end of the construction Line. (This defines where the spline ends).

- Click the **OK** button to generate a default cubic spline to form the handle of the spanner, ensuring that both reverse options are off.

Geometry	
Line	>
Spline	>
Tangent to Lines...	
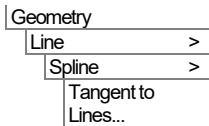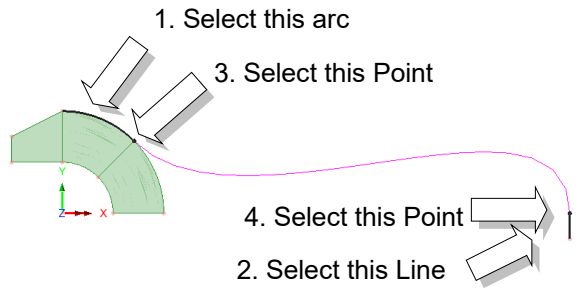
Note. Line directions can be seen by double-clicking on the Geometry entry in the Treeview and selecting the Line directions check box.

Once the spline is drawn correctly the construction Line is deleted.

- Drag a box around the construction Line by moving the mouse above and to the left of the Line.

Drag a box around the construction Line
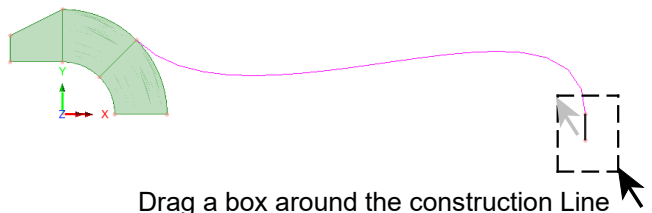
- Click the left-hand mouse button and holding it down move the mouse to the right and down so that a box is shown which completely encloses the Line as shown. Release the left-hand mouse button. LUSAS will highlight the selected features.

Edit
　Delete

[X] Delete the selected features.

- Click the **Yes To All** button to delete the selected Line and Points

Note. The **Delete** key can also be used to delete features. For a full list of keyboard shortcuts refer to the Online Help file, which can be accessed from the **Help > Help Topics** menu.

The centreline of the spanner can now be defined by joining the two unconnected points into a Line.
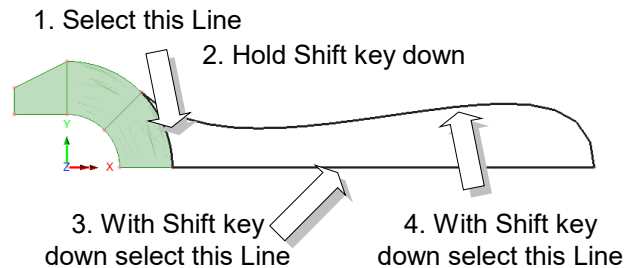
- Drag a box around the two Points on the centreline of the spanner as shown

Drag a box around these 2 Points
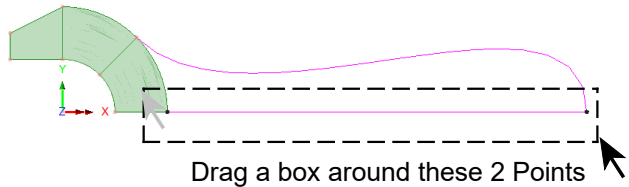
Geometry
　Line　　　>
　　By Points...

[/] A Line will be drawn between the points selected.

The Surface forming the handle of the spanner will now be defined by selecting the three Lines bounding the Surface.

- Select the 3 Lines which define the Surface of the handle in the order shown, ensuring the **Shift** key is held down to keep adding to the selection.

1. Select this Line

2. Hold Shift key down

3. With Shift key down select this Line

4. With Shift key down select this Line

Geometry
　Surface　　>
　　Lines...

[■] If prompted, press **OK** to draw the Surface formed by the 3 lines selected.

Note. Selecting the Lines in this anti-clockwise order ensures that the local element axes of the Surface will be suitable for the applied face loading that will be applied later in the example. Selecting the Lines by dragging a box around them would not necessarily produce the same Surface axes.
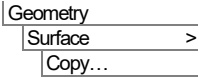
Mirroring model information

Half of the model has now been defined. This half can now be mirrored to create the whole model.

- Drag a box around the whole model to select all the features.

Note. An alternative to dragging a box around all the features to select them is to press the **Ctrl** and **A** keys at the same time.

Geometry
  Surface   >
    Copy…

Select **Mirror (screen)** and choose **Bottom** to mirror the selection about the lowermost line of that selection.

Note. Items selected for mirroring about the screen axes are represented by black lines on the dialog, with the effect of the mirroring setting chosen being shown in default feature colours, above, below, to the left, or to the right of the selection.
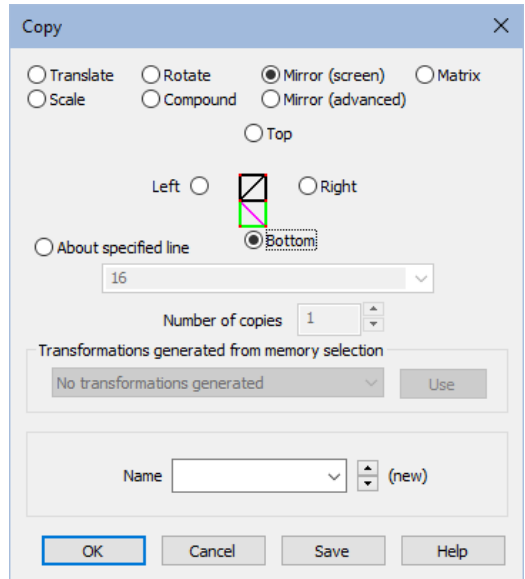
Note. Right-clicking on the view with geometry selected opens a menu with shortcuts to **Delete**, **Move**, **Copy** and **Sweep**.

This has completed the modelling of the spanner geometry.
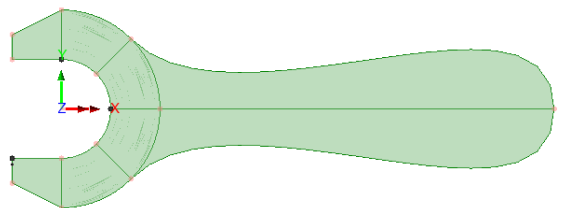
Using Groups

Tip. Model features can be grouped together to make assignment and viewing of model attributes easier.

In this example the Surfaces defining the jaws of the spanner will form one Group. The Surfaces defining the handle of the spanner will form another Group.

- Drag a box around the Surfaces representing the jaws of the spanner.

Drag a box to select the Jaw features

Geometry	
Group	>
New Group	

This adds a New Group entry to the Treeview for the features selected.
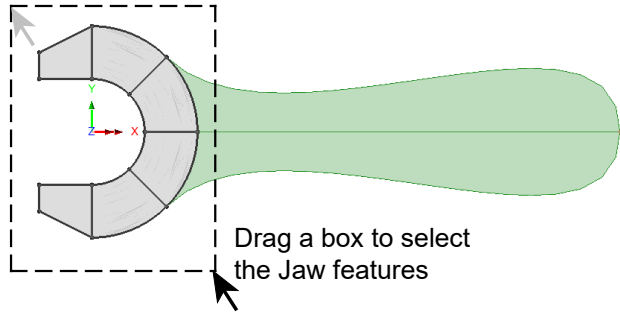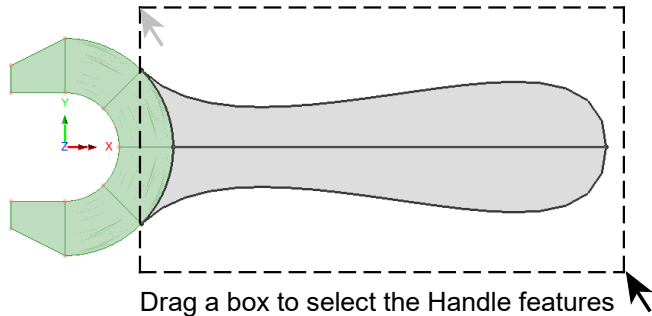
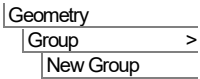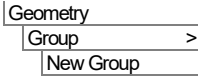- Enter the group name as **Jaws** and click **OK** to finish defining the group.

- Drag a box around the Surfaces representing the handle of the spanner.

Geometry	
Group	>
New Group	

This adds a New Group entry to the Treeview for the features selected.

Drag a box to select the Handle features

- Enter the group name as **Handle** and click **OK** to finish defining the group.

Modelling features recap

The geometry features that form the 2-Dimensional representation of the spanner have been defined. In this section the following topics have been covered:

❑ How to define Points, Lines and Surfaces.

❑ How to define Lines by specifying the coordinates of their Points and by joining existing Points.

❑ How to define Surfaces by their bounding coordinates, by sweeping, and by copying.

❑ How to define a cubic spline.

❑ How to select features by using the Shift Key to add to a previous selection.

❑ How to select features by dragging a box around features to be selected.

❑ How to select all features in a model by pressing the Control and A keys.

❑ How to rotate and mirror model features.

❑ How to define groups of features.

Modelling : Attributes

The Attributes ⬥ treeview is populated with potentially useful basic line mesh, geometric and support entries appropriate to the settings made on the new model dialog. Where applicable these can be assigned directly to selected features in a model or be edited as required. Otherwise, specific attributes will need to be defined.

For this example, the following attributes need to be defined:

❑ The finite element **mesh** to be used.

❑ The **thickness** of the spanner.

❑ The **material** from which the spanner is made.

❑ The **supports** to be used.

❑ The **loading** on the spanner.

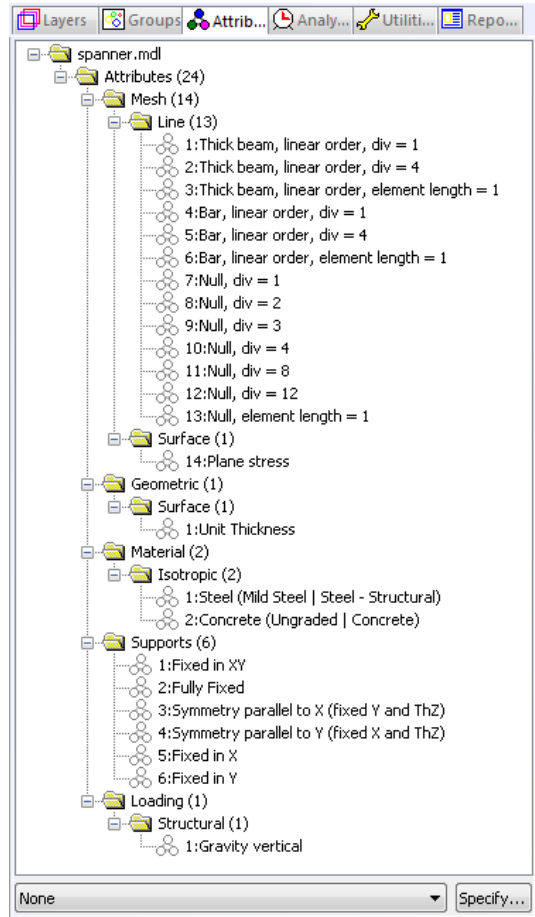## Defining and assigning model attributes

Note. LUSAS Modeller works on a define and assign basis where attributes are first defined, then assigned to features of the model. This can be done either in a step-by-step fashion for each attribute or by defining all attributes first and then assigning all, in turn, to the model.

❑ Attributes are first defined and subsequently displayed in the 🔵 Treeview as shown. Unassigned attributes appear as grey images.

❑ Attributes are then assigned to features by dragging an attribute dataset from the 🔵 Treeview onto previously selected features. Assigned attributes are then shown coloured.

Tip. Useful commands relating to the manipulation of attributes can be accessed by selecting an attribute in the 🔵 Treeview, then clicking the right-hand mouse button to display a shortcut menu.

```
Layers | Groups | Attrib... | Analy... | Utiliti... | Repo...
spanner.mdl
  Attributes (24)
    Mesh (14)
      Line (13)
        1:Thick beam, linear order, div = 1
        2:Thick beam, linear order, div = 4
        3:Thick beam, linear order, element length = 1
        4:Bar, linear order, div = 1
        5:Bar, linear order, div = 4
        6:Bar, linear order, element length = 1
        7:Null, div = 1
        8:Null, div = 2
        9:Null, div = 3
        10:Null, div = 4
        11:Null, div = 8
        12:Null, div = 12
        13:Null, element length = 1
      Surface (1)
        14:Plane stress
    Geometric (1)
      Surface (1)
        1:Unit Thickness
    Material (2)
      Isotropic (2)
        1:Steel (Mild Steel | Steel - Structural)
        2:Concrete (Ungraded | Concrete)
    Supports (6)
      1:Fixed in XY
      2:Fully Fixed
      3:Symmetry parallel to X (fixed Y and ThZ)
      4:Symmetry parallel to Y (fixed X and ThZ)
      5:Fixed in X
      6:Fixed in Y
    Loading (1)
      Structural (1)
        1:Gravity vertical
None                                    ▼  Specify...
```

Defining the mesh

The spanner will be meshed using both regular and irregular Surface meshes. A regular mesh is to be used for the jaws of the spanner. An irregular mesh will be used for the handle.
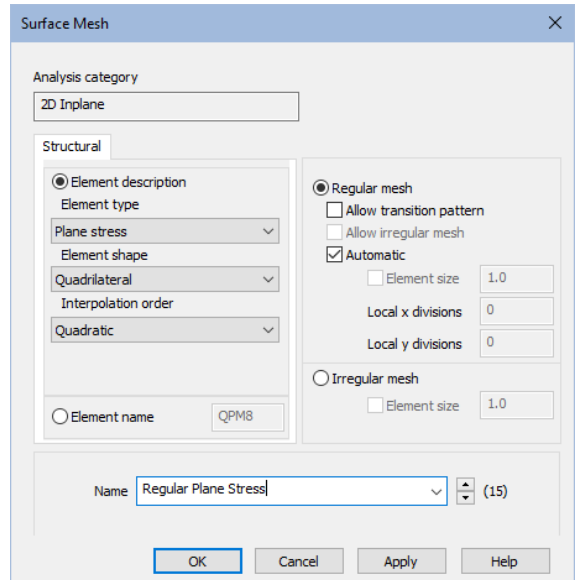
The number of elements in the regular Surface mesh will be controlled by defining line meshes on the Lines defining the boundary of the Surfaces. The number of elements in the irregular Surface mesh will be controlled by specifying an ideal element size. Other methods of controlling mesh density are also available.

Defining a regular surface mesh

- Select **Plane stress** elements, which are **Quadrilateral** in shape with a **Quadratic** interpolation order.

- Ensure that the **Regular Mesh** button is selected and **Allow transition pattern** is <u>not</u> selected.

- Enter the attribute name as **Regular Plane Stress**

- Click the **OK** button to add the Surface mesh dataset to the Treeview.
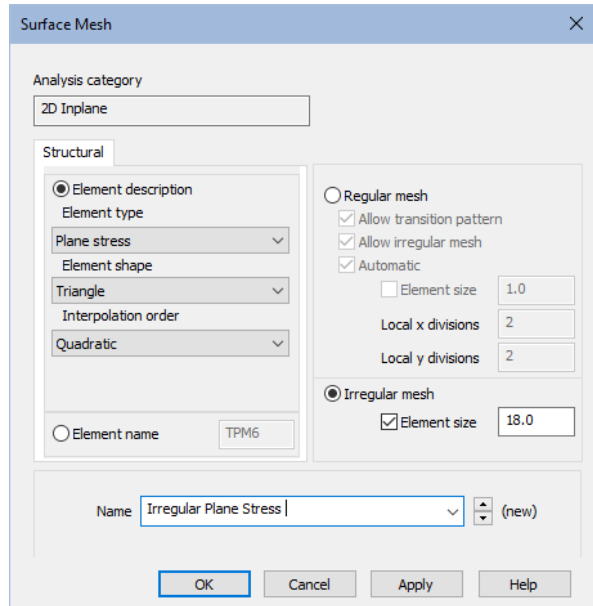
The mesh will be assigned to the model shortly.

Surface Mesh

Analysis category
2D Inplane

Structural

◉ Element description
Element type
Plane stress
Element shape
Quadrilateral
Interpolation order
Quadratic

○ Element name   QPM8

◉ Regular mesh
☐ Allow transition pattern
☐ Allow irregular mesh
☑ Automatic
☐ Element size   1.0
Local x divisions   0
Local y divisions   0
○ Irregular mesh
☐ Element size   1.0

Name Regular Plane Stress   (15)

OK   Cancel   Apply   Help

Controlling mesh density

The lines defining the spanner's geometry currently have 4 divisions per line by default, but in this example only 2 divisions are required for the Lines defining the jaws. This can be done by either changing the default number of divisions per Line or by making use of Line mesh attributes. In this example, the default number of divisions will be changed.

- Select the **Meshing** tab and set the **Default Line divisions** to **2**.

- Click the **OK** button.

Defining an irregular surface mesh

Attributes
| Mesh | > |
| Surface… |

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

- Select the **Irregular mesh** button.

- Select Element Size and then specify an element size of **18.** (This defines the maximum side length of the elements on the lines defining the perimeter of the spanner.)

- Enter the attribute name as **Irregular Plane Stress**.

Surface Mesh

Analysis category

2D Inplane

Structural

- ⦿ Element description
  Element type
  Plane stress
  Element shape
  Triangle
  Interpolation order
  Quadratic

- ○ Element name   TPM6

- ○ Regular mesh
  ☑ Allow transition pattern
  ☑ Allow irregular mesh
  ☑ Automatic
  ☐ Element size   1.0
  Local x divisions   2
  Local y divisions   2
- ⦿ Irregular mesh
  ☑ Element size   18.0

Name   Irregular Plane Stress   ▼ ⬍ (new)

[OK]   [Cancel]   [Apply]   [Help]

- Click the OK button to add the Surface mesh attribute to the 🔴Treeview.

The mesh will be assigned to the model shortly.

Defining the thickness

So far the spanner has been defined in two dimensions. In order to give the spanner its thickness geometry attributes will be used. The jaws of the spanner are 15mm thick whilst the handle is 10mm thick. So two geometry attributes are required.

Attributes
| Geometric | > |
| Surface… |

- Enter the thickness as **15**

- Enter the Attribute name as **Thickness=15**

- Click the **Apply** button to add the geometry attribute to the 🔴 Treeview.

Geometric Surface

		Value
Thickness	t	15.0
Eccentricity	ez	0.0

Name   Thickness=15   ▼ ⬍ (new)

[OK]   [Cancel]   [Apply]   [Help]
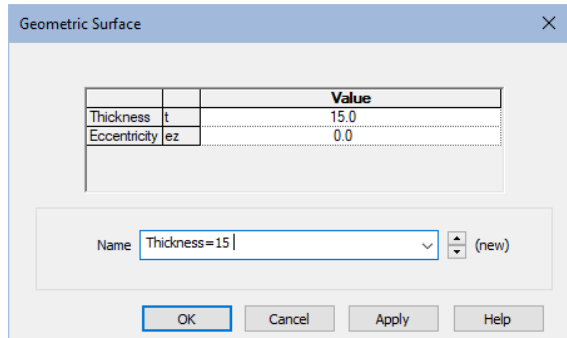
> **Note.** The Apply button allows information for another attribute to be entered using the same dialog.

- Change the thickness to **10**

- Change the attribute name to **Thickness=10** and click the **OK** button to add the additional geometry attribute to the 🔴 Treeview.

Assigning a surface mesh and thickness to the jaws

The Surface mesh and geometry attributes defined previously can now be assigned to the relevant features of the spanner. As an alternative to selecting features by dragging a box around them, the previously named Groups can be used.

- In the 🗐 Treeview, click the right-hand mouse button on the group name **Jaws** and select the **Select Members** option from the menu. If you already have some features selected click **Yes** to deselect them first.

- Drag and drop the Surface mesh attribute **Regular Plane Stress** from the 🔴 Treeview onto the selected Surfaces. Modeller will confirm the mesh assignment for each Surface in the text window.
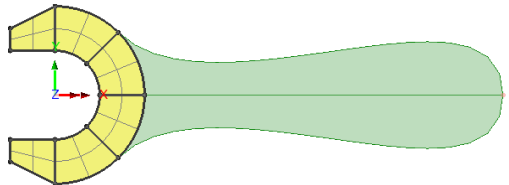
The element mesh for the jaws of the spanner will be drawn.

- Drag and drop the Surface geometry attribute **Thickness=15** from the 🔴 Treeview onto the selected Surfaces. The elements of the jaws remain selected.

- Click the left-hand mouse button in a blank part of the View Window to deselect any previously selected model features. This shows that the geometric assignment has been visualised by default.

Assigning surface mesh and thickness to the handle

- In the 🗐 Treeview, click the right-hand mouse button on the Group name **Handle** and select the **Select Members** option from the menu.

All Surfaces forming the group **Handle** will be selected.

- Drag and drop the Surface mesh attribute **Irregular Plane Stress** from the 🔴 Treeview onto the selected Surfaces defining the Handle.

LUSAS will draw the irregular element mesh for the handle of the spanner.

Note. The text output window will show messages relating to the radius of curvature for two of the elements created. These can be ignored for this example.
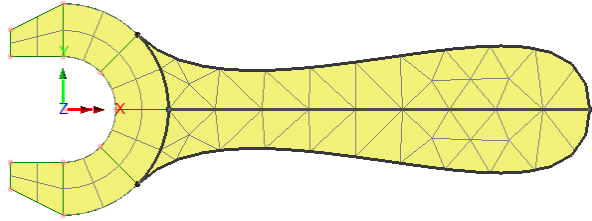
- Drag and drop the Surface geometry attribute **Thickness=10** from the Treeview onto the selected Surfaces. Again, confirmation of the assignment is provided in the text window.

- Click the left-hand mouse button in a blank part of the View Window to deselect any previously selected model features.

Assigning the material

The material required for this example is provided by default when using the startup templates specified when creating the model.

- Right-hand click the material attribute **Steel (Mild Steel | Steel - Structural)** from the Treeview and select **Assign to all** to apply the mild steel attribute to the whole model.

Visualising model attributes

Note. Any attributes (i.e. geometry, material, supports etc.) assigned to the model can be checked visually to ensure that the correct item has been assigned to the correct part of the model. For example:

- Click the left-hand mouse button in a blank part of the View Window to deselect any previously selected model features.

- In the Treeview, click the right-hand mouse button on the **Thickness=15** material attribute name and select the **Visualise Assignments** option from the dialog.

All features to which the **Thickness=15** attribute is assigned will be visualised.

- To turn-off the visualisation of the assignments, click the right-hand mouse
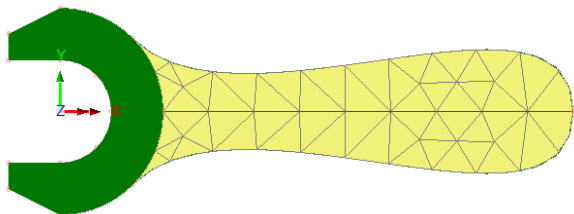
button on the **Thickness=15mm** material attribute name in the ⚛ Treeview and select the **Visualise Assignments** option again from the dialog.

📝 **Note.** This method can be used at any time during this example to check that selected attributes have been correctly assigned to the model.

Manipulating layers

At any time, the layers displayed in the 🗗 Treeview may be re-ordered, hidden, removed or be re-added. As an example:

- Select the **Geometry** entry in the 🗗 Treeview and drag it down to sit beneath the Mesh entry. In the view window the transparent geometry layer can now be seen drawn on top of the mesh.

- Click the right-hand mouse button on the **Mesh** entry in the 🗗 Treeview and deselect the **Display** option. The mesh will be hidden from the display.

Fleshing

Visualisation of assigned geometric attributes can also be seen using the fleshing option.

⚡ Select the fleshing on/off button to turn-on geometric property visualisation.

⊕ Select the isometric button to see the geometric visualisation on the elements.

↻ Select the dynamic rotate button to spin the spanner around within the view window.

⚡ Select the fleshing on/off button to turn-off the geometric visualisation.

🏠 Select the Home button to return the model to the default view.

↖ Reset to normal cursor mode.
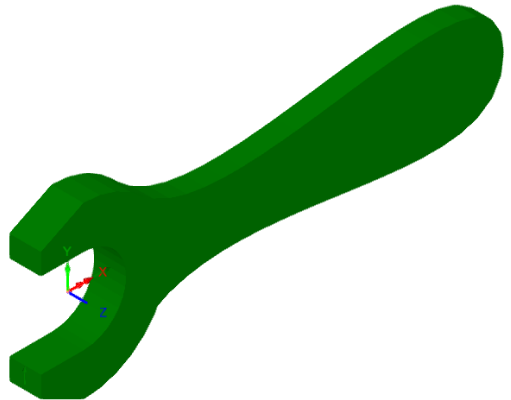
Supports

When using one of the standard startup templates, LUSAS provides the more common types of support by default. These can be seen in the 🔵 Treeview. Two support attributes are required, one which restrains movement in the X direction, and one which restrains movement in the Y direction.

Assigning the supports

- Select the Point on the centreline of the spanner as shown to assign the support **Fixed in X**.

Select these 2 Points for support 'Fixed in Y'

- Drag and drop the support attribute **Fixed in X** from the 🔵Treeview onto the selected point.

Select this Point for support 'Fixed in X'

- Click the **OK** button to assign the support to the Point selected for all the loadcases in analysis 1.
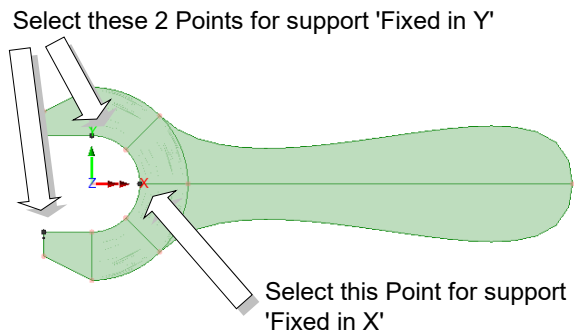
The support will be visualised using an arrow symbol.

- Select the 2 Points shown to assign the support **Fixed in Y**. Hold the **Shift** key down to add the second point to the initial selection.

- Drag and drop the support attribute **Fixed in Y** onto the selected points.

- Click the **OK** button to assign the support to the points selected.

- The supports will be visualised using arrow symbols.

- Click the left-hand mouse button in a blank part of the view window to deselect any previously selected model features.

Defining the loading

A pressure load is to be distributed evenly along the top edge of the handle.

Attributes	
Loading…	>

- Select **Distributed loads**, then **Face** and click **Next**

- On the Face loading dialog enter a load of **0.1** in the **y** direction.

- Enter the attribute name as **Face Load of 0.1**.

- Click the **Finish** button to add the loading attribute to the Treeview.

Assigning the loading

- Select the Line on the top edge of the spanner handle.

- Drag and drop the loading attribute **Face Load of 0.1** from the Treeview onto the selected Line.

- Click the **OK** button to assign the loading to the Line selected.
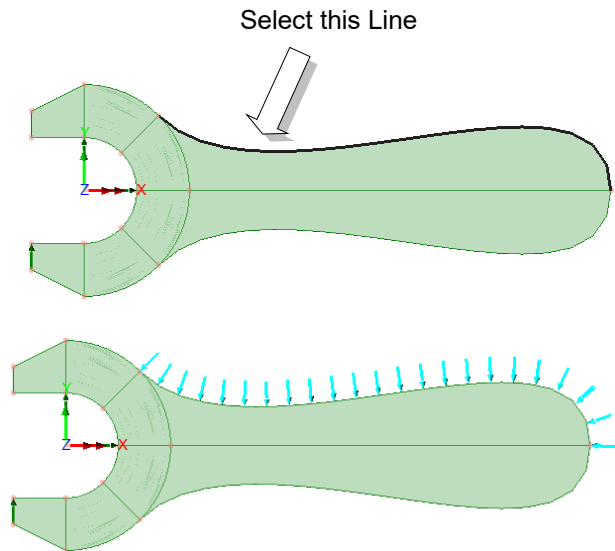
Select this Line

Note. If the loading is displayed in the opposite direction to that shown the Surface forming the top half of the handle may be reversed as follows:

- Select the Surface defining the top-half of the spanner by dragging a box around it.

This will reverse the Surface axes and hence the direction of the loading.

Geometry
Surface   >
Reverse...
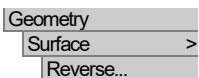
Saving the model

File
Save

🖫 To save the model file.

Modelling attributes recap

In this section, the attributes of the model were defined and assigned to the features.

❏ A regular Surface mesh with quadrilateral plane stress elements was defined and assigned to the jaws of the spanner. An irregular Surface mesh with triangular plane stress elements and a fixed element size was defined and assigned to the handle of the spanner.

❑ Two geometry attributes were used to specify the spanner jaws and handle thickness.

❑ A material attribute specifying the properties of steel was assigned to all Surfaces.

❑ Two support attributes were assigned in order to simulate the spanner being used to turn a nut and a structural face load was applied to the top edge of the handle.

❑ Attributes assigned to the model were checked visually for correct assignment.

The model definition is now complete. The next step in the process is to run an analysis to solve the problem.

Running the Analysis

Open the **Solve Now** dialog and press **OK** to run 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::

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

❑ **spanner.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. Note that a common mistake made when using LUSAS Modeller for the first time is to forget to assign particular attribute data (geometry, mesh, supports, loading etc.) to the model.

Rebuilding a Model

If errors are listed that for some reason cannot be corrected, 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.

❑ **spanner_modelling.lvb** carries out automated modelling of the example.

File
New…

Enter a file name of **Spanner**

- Use the default **User-defined** working folder.

- Ensure an Analysis type of **2D Inplane** is set.

- Click the **OK** button.

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

File
Script   >
    Run Script…
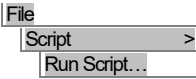
To create the model, open the file **spanner_modelling.lvb** that was downloaded and placed in a folder of your choosing.

Rerun the analysis to generate the results.

Viewing the Results

In this section the results produced by the analysis of the spanner will be viewed. There are several ways to do this in LUSAS, allowing you to choose the most appropriate way to present your results. For this example:

- ❑ A plot of the deformed mesh will be displayed and superimposed upon the undeformed shape for comparison.
- ❑ The principal stress vectors will be plotted.
- ❑ The von Mises stress contours for averaged stress values will be displayed.
- ❑ Peak values of von Mises stress will be marked.
- ❑ A graph will be produced showing the variation of stress along a slice section through the handle of the spanner.

Selecting the results to be viewed

Analysis loadcase results are present in the Treeview inside the Analysis 1 entry as signified by the results bitmap .
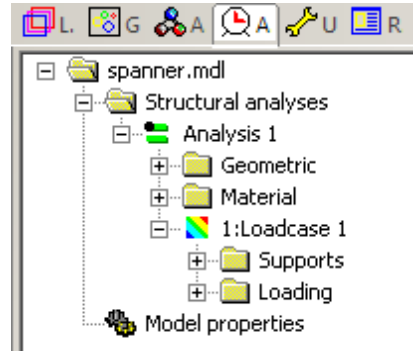
Deformed mesh plot

After solving, a deformed mesh is displayed.

The deformed mesh is drawn with a blue outline and a blue (transparent) fill colour.
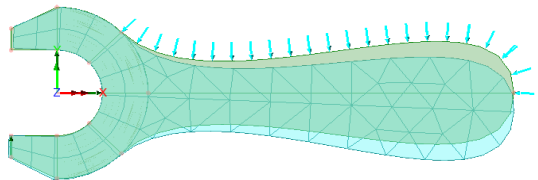
To view only the deformed mesh the Geometry and Attribute layers can be hidden:

- Click the right-hand mouse button on the **Geometry** entry in the Treeview and deselect the **Display** option. The geometry will be hidden from the display.

- Repeat to turn off the display of the **Attributes** layer.

Principal stress vector plots

Principal stresses can be plotted as vectors with different colours being used to signify tension and compression.

The mesh layer is no longer required and will now be hidden.

- Click the right-hand mouse button on the **Mesh** entry in the Treeview and deselect the **Display** option.

To add the Vectors layer:

- Click the right-hand mouse button in a blank part of the view window and select the **Vectors** option to add the vectors layer to the Treeview.

The vector properties dialog will be displayed.

- Select **Stress** vector results of **Principal** stresses from the entity drop down list.

- Click the **OK** button to display the vector plot with tension vectors shown in red and compression vectors shown in blue.
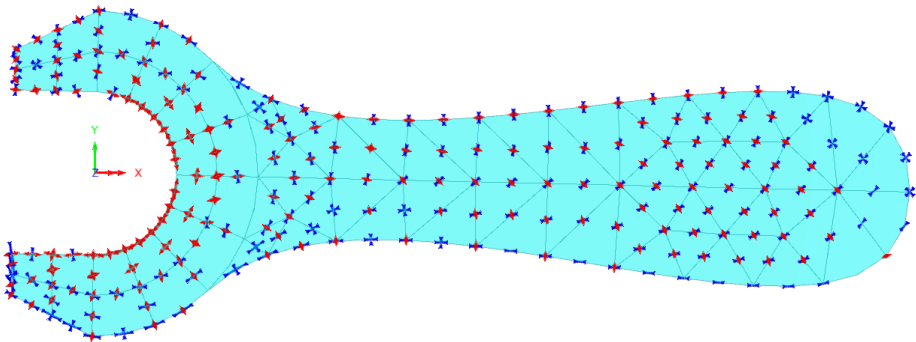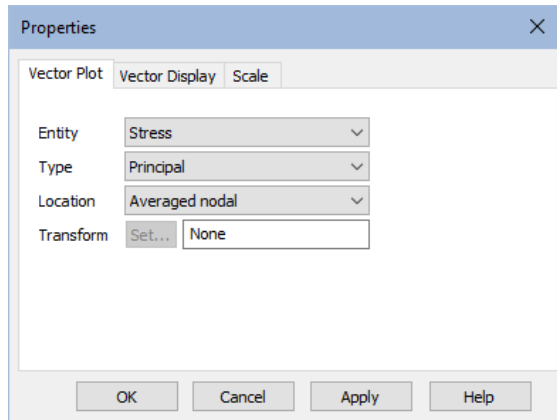
Creating new view windows

As an alternative to hiding, adding or removing layers from the ⬚ Treeview for each type of results to be displayed in a single view window, multiple view windows can be defined. New views can be created using default settings, or existing views can be saved before being applied to new views.

Window
  New Window

This creates a new view window with default view layers of Mesh, Geometry, and Attributes. In the ⬚ Treeview, a new folder for **spanner.mdl Window 2** will appear.

- In the Window 2 folder in the ⬚ Treeview, turn-off the **Geometry**, **Attributes** and **Deformed** mesh layers

- In the 🕐 Treeview, Loadcase 1 will be set active by default.

Using page layout mode

The model was created using a Working Mode view which allows a model of any size to be created. Results could be viewed using this mode of operation, but in order to allow additional information to be added without obscuring the model, Page Layout Mode could be used instead.

> View
> > Page Layout Mode...

The view window will resize to show the model view fitted on a piece of paper.

> File
> > Page Setup...

- Ensure that the **Landscape** option is selected and that left, right, top and bottom page **Margins** of **60, 10, 10, 10** respectively are set.

- Click the **OK** button.

Von Mises stress contour plot

Contours of von Mises Stress (Equivalent Stress) may be plotted as lines or as colour-filled contour ranges. To display stress contours the contour layer needs to be added to the ▣ Treeview.

- With no features selected, click the right-hand mouse button in a blank part of the active window and select the **Contours** option to add the contours layer to the ▣ Treeview.

The contour plot properties dialog will be displayed.

- Ensure that **Stress** contour results are selected from the Entity drop down list, equivalent stresses **SE** from the Component list, and **Averaged nodal** are selected from the Display list.

- On the same dialog select the **Appearance** tab and ensure that **Classic** contours are selected.

- Click the **OK** button to display the contour plot of equivalent stress along with the contour key.
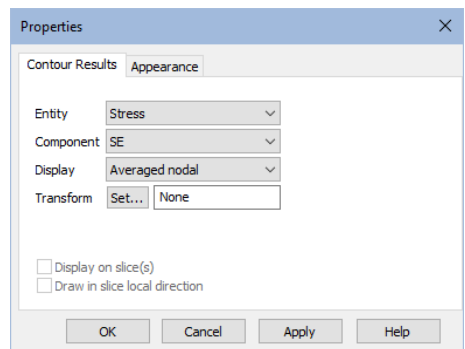
The display of the contour key is controlled by the Annotation layer. The contour key is not shown on the images that follow.

Order of layers in the Treeview

The order of the layers in the ⬚ Treeview only governs which colour filled regions of layers obscure other colour filled regions of other layers as seen in the view window.
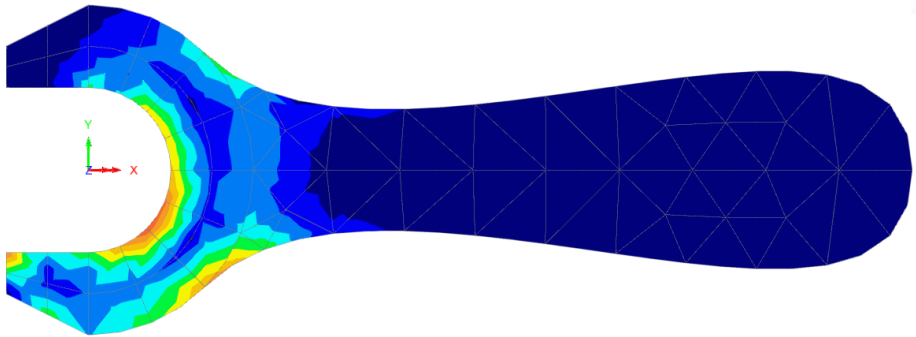
For example, if it was required to see the colour filled mesh layer obscure the contours layer, the mesh layer needs to be moved down the ⬚ Treeview list to a position after the contour layer.

- In the ⬚ Treeview select the **Mesh** layer within Window 2 and drag and drop the Mesh layer name on top of the contour layer name. The mesh layer name will then follow the contour name and the colour filled mesh layer will then be displayed on top of the contour layer.

Note. This ordering of layer names can also be done using the right-hand mouse button and selecting the **Move Down** option from the context menu presented.

The contour layer is required to be visible, so:
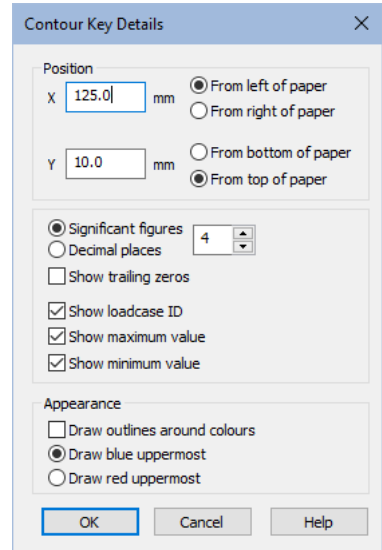
- In the ⬚ Treeview move the **Contours** layer beneath the Mesh layer to return the view to its previous state.

Moving the contour key and setting significant figures

The contour key can be moved graphically by selecting any text or contour key colouring and dragging the key to a new position.

To specify a precise location:

- In the ⬚ Treeview double-click Contours to open the properties dialog.

- Select the **Appearance** tab and press the **Set...** button next to the **Contour key** option.

- Set the X position to **125**mm **From left of paper**, and the Y position to **10**mm **From top of paper**.

- Limit the Significant figures to **4** and press **OK** to return to the contour properties dialog.

- Press **OK** to save changes and press **OK** again to see the contour key positioned centrally near the top of the page.
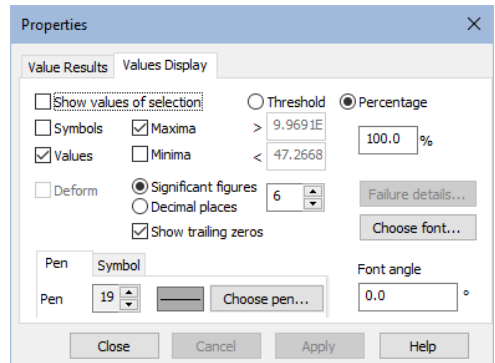
Marking peak values

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

The values properties dialog will be displayed.

- Ensure **Stress** contour results from the Entity drop down list, equivalent stresses **SE** from the Component list, and **Averaged nodal** from the Location list are selected.

- On the same dialog, select the **Values Display** tab and see that, by default, **Maxima** is chosen to display the top **100**% (so all) of results.

- Click the **OK** button to see the values added to the contour plot.
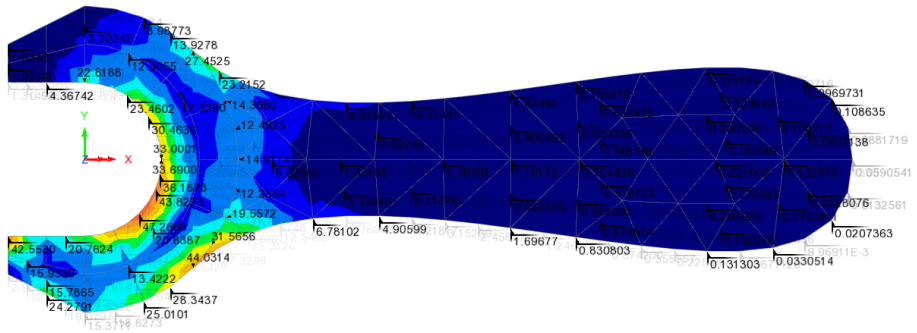
📝 **Note.** Values, by default, are only displayed if they do not clash with other values, labels or text visible in the model view window, with higher results values taking priority over lower ones, and larger negative values take priority over lesser ones. In situations where clashes occur the clashing text is made transparent. The display of results values takes priority over labels, which take priority over Annotation. Values are also displayed with a location symbol to show where they relate to on a model. Enlarging the view of a model will progressively make previously clashing text visible.

Peak values can also be isolated by restricting the percentage of results plotted.

- In the 🔲 Treeview double click the **Values** layer.

The values properties dialog will be re-displayed.

- Select the **Values Display** tab and ensure that the **Maxima** option is chosen to display the top **1**% of results. Deselect the **Minima** option.
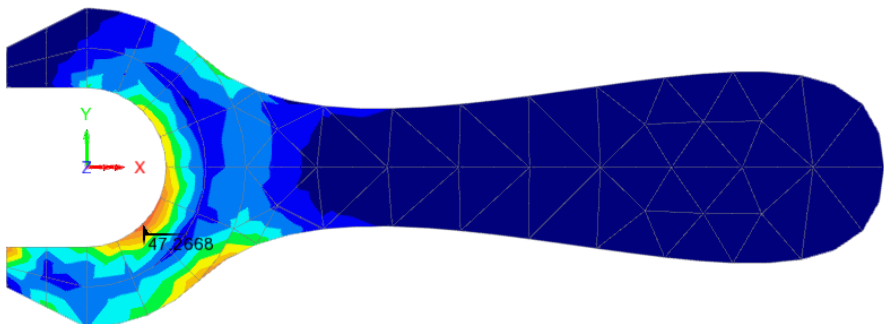
Saving view windows

View layers and settings defined for a view can be saved for use with other view windows and for use within reports.

28

- Accept the default settings presented and press the **OK** button to create a Saved view entry 🖼 '**View 1**' in the Utilities 🦴 treeview.

To update a view with the settings from a saved view:

- Create a New Window using the **Window > New Window** main menu item.

- Click on the Utilities 🦴 treeview and drag and drop the Saved view entry 🖼 for **View1** onto the newly created view window. This updates the newly created view to show those layers and settings saved with View 1.

Note. It is very useful to name a saved view with a name that reflects the contents of the view. For example, saved view names of 'Contours of SE', or 'Maximum displacement, or 'Isometric view' might be used, when relevant.

Creating a slice section of results

This requires working mode and not page layout mode.

Section slicing of the model to create graphs of results must be done with the model lying orthogonal to the screen.

- 🏠 Press the Home button to ensure that the model is viewed correctly.

- Ensure the **Snap to grid** button is selected and a grid size of **10** is specified.

- Click the **OK** button.
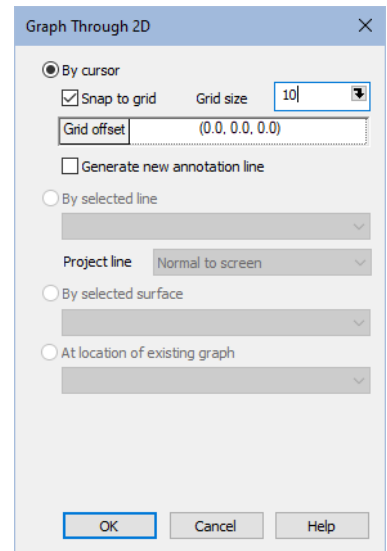
- With reference to the following image, click and drag the at the X=40 location to define a section slice through the handle of the spanner where it joins the jaws.
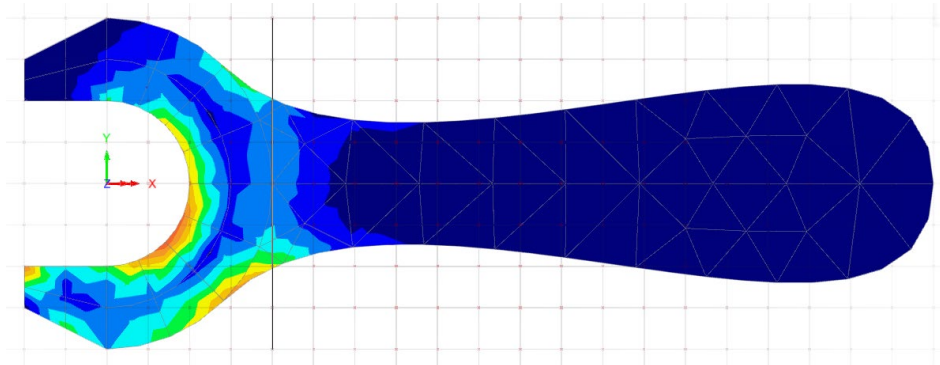
The loadcase and extent of the slice must now be defined:

- Ensure that **Loadcase 1** and an extent of **Visible model** are selected and Click the **Next** button.

This leads directly on to creating a graph using the graph wizard:

Selecting the slice data results to be plotted

In this example a graph is to be plotted of the variation in stress through the specified section of the spanner. The X axis results of distance through the spanner have been defined by the section slice. The Y axis results now need to be specified.

- On the Slice data dialog, ensure **Stress -** contour results from the Entity drop down list, and equivalent stresses **SE** from the Component list are selected.

- Click the **Next** button to define the Y axis results.
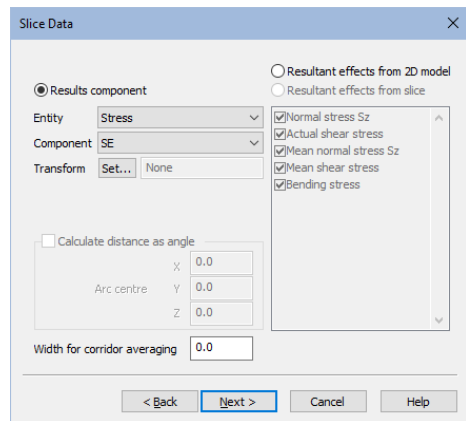
Title information for the graph is now to be added:

- Enter the graph title as **Stress at neck section**

- Enter the X axis label as **Distance along section**

- Enter the Y axis label as **Equivalent Stress (N/mm2)**

- Deselect the **Show symbols** button.

- Click the **Finish** button to display the graph in a new window and show the values used in an adjacent table.
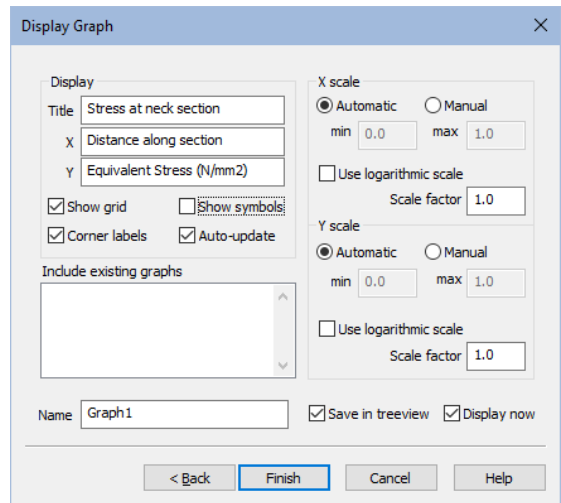
Note. If the graph title or axes labels are left unspecified Modeller will use default names.
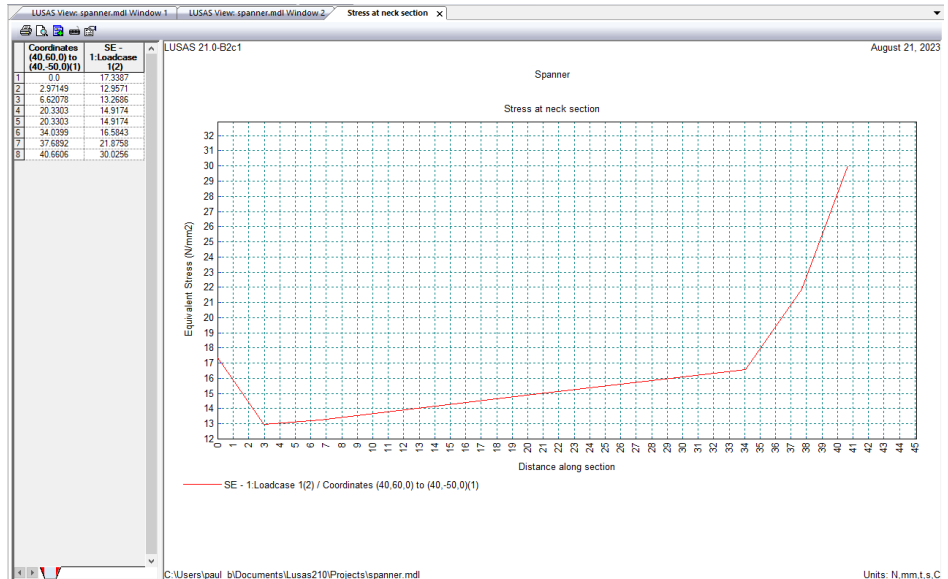
Note. The properties of the graph may be modified by clicking on the graph with the right-hand mouse button and selecting the **Edit Graph Properties** option.

Close the graph window and return the view window.

To exit from the Graph Through 2D option click the **Select Any** cursor button. The grid will be removed.

This completes the example.